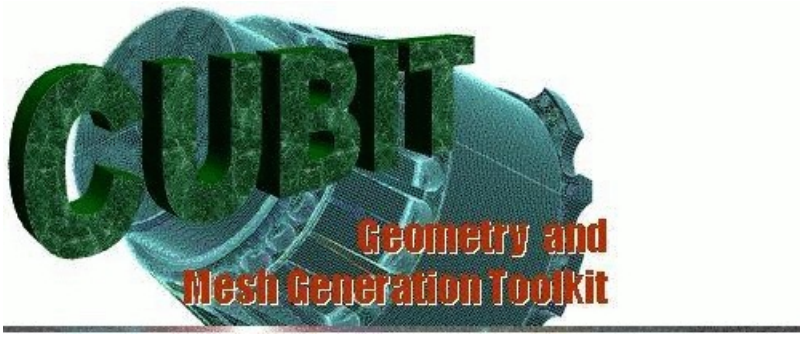
SAND Number: SAND2025-13033W



CUBIT® 17.06 User Documentation

Introduction - A quick overview of some of the main features and goals of the CUBIT Mesh Generation Toolkit, licensing and distribution, hardware requirements, and where to go for help.

Environment Control - A description of the CUBIT user environment, including using the graphical user interface, session control, command line syntax, journal files, graphics, entity picking, saving and restoring etc..

Geometry - A description of CUBIT's geometry features including building geometry from scratch, manipulating geometry in CUBIT, importing and exporting geometry formats, etc...

Mesh Generation - A description of CUBIT's mesh generation capabilities, including how to mesh geometry, meshing and smoothing schemes, setting sizes and intervals, importing a mesh, etc...

Finite Element Model - How to set up the finite element model for analysis, including defining boundary conditions, material properties, exporting the finite element model, etc.

Boundary Layer Meshing - How to set up boundary layers.

Immersive Topology Environment for Meshing (ITEM) - A description of Cubit's interactive meshing wizard including how to use the wizard, and a guide to geometry clean-up, setting up the finite element model, mesh generation in ITEM, etc.

Step-By-Step Tutorials

Appendix

Credits

Quick Reference

Official CUBIT Web Page

Sandia National Laboratories



Sandia National Laboratories is a multi-mission laboratory managed and operated by National Technology & Engineering Solutions of Sandia, LLC., a wholly owned subsidiary of Honeywell International, Inc., for the U.S. Department of Energy's National Nuclear Security Administration under contract DE-NA0003525.

Table of Contents

Introduction 18
Introduction 18
Cubit Mailing Lists 19
Hardware Requirements 20
How to Use This Manual 21
Key Features 22
Licensing and Distribution 24
Problem Reports and Enhancement Requests 25
Trademark Notice 26
Environment Control 27
Environment Control 27
session control 28
Session Control 28
Command Line Help 29
Command Syntax 30
Environment Commands 32
Environment Variables 35
Execution Command Syntax 37
Initialization Files 41
Interrupting Running Tasks 42
Saving and Restoring a Cubit Session 43
Starting and Exiting a CUBIT Session 45
recording and playback 47
Command Recording and Playback 47
Automatic Journal File Creation 48
Controlling Playback of Journal Files 50
Journal File Creation and Playback 52
Idless Journal Files 53
location direction specification 55
Location, Direction and Axis Specification 55
Drawing a Location, Direction, or Axis 56
Specifying an Axis 57
Specifying a Direction 59
Specifying a Location 62
Specifying a Location on a Curve 66
Specifying a Plane 69
listing information 74

Listing Information 74
List Cubit Environment 75
List Geometry 78
List Mesh 80
List Model Summary 81
List Special Entities 82
gui 83
Graphical User Interface 83
tree view 84
Model Tree 84
Power Tools 87
Geometry Power Tools 88
Meshing Tools 100
Mesh Quality Tools 102
Assembly Tool 105
Geometry Mesh Comparison Tool 107
Defeaturing Tool 109
Beams and Shells with the Geometry Power Tool 116
Machine Learning with the Geometry Power Tool 121
Slot Surface Preparation with the Geometry Power Tool 130
graphics window 133
Graphics Window 133
Viewing Curve Valence 134
Key Press Commands for the GUI 135
Right Click Commands for the GUI Graphics Window 137
Selecting Entities in the GUI 140
View Navigation in the GUI 142
drop down menus 145
Drop Down Menus 145
Options Menu 147
Undo Button 151
control panel 152
Command Panels 152
Command Panel Functionality 153
CUBIT Application Window 159
Command Line Workspace 162
Journal File Editor 165
Property Editor 167
Toolbars 170
Toolbar Customization 173

graphics window control 184
Graphics Window Control 184
Graphics Clipping Plane 185
Colors 187
Drawing and Highlighting Entities 190
Drawing Lines and Polygons 194
Entity Labels 196
Graphics Camera 198
Graphics Modes 200
Graphics Window Size and Position 202
Hardcopy Output 203
Graphics Lighting Model 204
Mesh Visualization 205
Miscellaneous Graphics Options 206
Mouse Based View Navigation: Zoom, Pan and Rotate 210
Saving Graphics Views 213
Updating the Display 214
Geometry, Mesh, and BC Entity Visibility 215
Command Line View Navigation: Zoom, Pan and Rotate 217
entity selection and filtering 219
Entity Selection and Filtering 219
Extended Selection Dialog 220
Command Line Entity Specification 227
Extended Command Line Entity Specification 230
Selecting Entities with the Mouse 236
Part Classification with Machine Learning 241
Training CAD Operations with Machine Learning 245
Select Visible Surfaces 248
Geometry 250
Geometry 250
model definitions 251
CUBIT Geometry Formats 251
ACIS Geometry Kernel 253
Mesh-Based Geometry 254
geom creation 258
Geometry Creation 258
primitive geometry 259
Creating Bricks 259
Creating Pyramids 260
Creating Frustums 261

Creating Toruses ... 262
Creating Cylinders ... 263
Geometric Primitives ... 264
Creating Prisms ... 266
Creating Spheres ... 267
bottom up creation ... 268
Bottom-Up Geometry Creation ... 268
Creating Volumes ... 269
Creating Curves ... 277
Creating Surfaces ... 284
Creating Vertices ... 292
transforms ... 294
Geometry Transforms ... 294
Align Command ... 295
Copy Command ... 297
Move Command ... 298
Reflect Command ... 300
Rotate Command ... 301
Scale Command ... 302
booleans ... 303
Geometry Booleans ... 303
Intersect ... 304
Subtract ... 305
Unite ... 306
decomposition ... 307
Geometry Decomposition ... 307
web cutting ... 308
Web Cutting with an Arbitrary Surface ... 308
Chop Command ... 309
Web Cutting with a Planar or Cylindrical Surface ... 310
Web Cutting by Sweeping Curves or Surfaces ... 312
Web Cutting with a Tool Body ... 315
Web Cutting ... 316
Web Cutting Options ... 317
splitting geometry ... 318
Split Curve ... 318
Split Periodic Surfaces ... 319
Split Surface ... 320
Splitting Geometry ... 333

Section Command 334
Separating Surfaces from Bodies 335
Separating Multi-Volume Bodies 336
cleanup and defeaturing 337
Geometry Cleanup and Defeaturing 337
Auto Healing 338
Healing 339
Spline Removal 340
What if Healing is Unsuccessful? 341
tweaking geometry 342
Tweaking Surfaces 342
Tweaking Vertices 350
Tweak Volume Bend 352
Tweaking Geometry 353
Tweaking Curves 354
Tweak Remove Topology 359
removing geometric features 361
Removing Surfaces 361
Removing Vertices 363
Removing Geometric Features 364
auto clean 365
Automatic Geometry Clean-up 365
Automatic Forced Sweepability 366
Automatic Surface Split 367
Automatic Small Curve Removal 368
Automatic Small Surface Removal 369
reduce 370
Reducing Geometry 370
Reduce Bolt Simplify 372
Reduce Bolt Fit Volume 373
Reduce Bolt Core 379
Reduce Bolt Spider 385
Reduce Spring 389
Reduce Thin Volumes 391
Reduce Thin RL 395
Reduce Slot Surface 398
Reduce Bolt Patch 402
Debugging Geometry 408
Finding Surface Overlap 409
Geometry Accuracy 412

Regularizing Geometry 413
Stitching Sheet Bodies 414
Trimming and Extending Curves 415
Validating Geometry 417
Blunt Tangency 418
imprint merge 421
Geometry Imprinting and Merging 421
Examining Merged Entities 422
Imprinting Geometry 423
Merge Tolerance 425
Merging Geometry 426
Using Geometry Merging to Verify Geometry 428
Unmerging 429
virtual geometry 430
Virtual Geometry 430
collapse geometry 431
Collapse Angle 431
Collapse Curve 434
Collapse Geometry 436
Collapse Surface 437
composite geometry 439
Composite Curves 439
Composite Geometry 440
Composite Surfaces 441
partitioned geometry 443
Removing Partitions 443
Using Mesh Intersections to Partition Surfaces 444
Partitioned Curves 446
Partitioned Surfaces 447
Partitioned Volumes 449
Partitioned Geometry 450
Deleting Virtual Geometry 451
Simplify Geometry 452
groups 455
Geometry Groups 455
Basic Group Operations 456
Groups in Graphics 458
Propagated Hex Groups 459
Quality Groups 465

Propagated Groups 466
Seeded Mesh Groups 467
Naming Convention for Propagated Hex Groups 469
attributes 471
Geometry Attributes 471
persistent attributes 472
Attribute Behavior 472
Attribute Commands 473
Attribute Types 474
Persistent Attributes 475
Using CUBIT Attributes 476
Entity IDs 477
Entity Names 479
metadata 482
Parts, Assemblies and Metadata 482
Importing and Exporting Metadata 483
Metadata Attributes 485
Working With Parts and Assemblies 488
import 492
Importing Geometry 492
Importing ACIS Files 493
Importing Facet Files 495
Importing FASTQ Files 499
Importing Granite Files 500
Importing IGES Files 501
Importing STEP Files 502
Importing SGM Files 504
export 505
Exporting Geometry 505
Exporting ACIS Files 506
Exporting Facet Files 507
Exporting IGES Files 508
Exporting STEP Files 509
Geometry Deletion 510
Geometry Orientation 511
Entity Measurement 513
Mesh Generation 515
Mesh Generation 515
Meshing the Geometry 517
interval assignment 518

Interval Assignment 518
Additional Interval Constraints 519
Vertex Sizing and Automatic Curve Biasing 521
Explicit Specification of Intervals 522
Explicit Specification of Intervals Using Interval Size 523
Automatic Specification of Intervals 524
Interval Matching 527
Interval Firmness 530
Relative Intervals 531
Mesh Interval Preview 532
Periodic Intervals 533
meshing schemes 534
Meshing Schemes 534
duplication 536
Copying a Mesh 536
conversion 539
HTet 539
QTri 541
THex 543
TQuad 545
traditional 546
Bias, Dualbias 546
Circle 550
Curvature 551
Equal 552
Hole 553
Mapping 554
Pave 556
Pentagon 560
Pinpoint 561
Polyhedron 562
Sphere 564
STransition 568
Stretch 570
Submap 571
Surface Vertex Types 574
Sweep 577
TetMesh 585
Tetprimitive 595

TriAdvance 596
TriDelaunay 597
TriMap 598
TriMesh 599
TriPave 607
TriPrimitive 608
parallel 609
pCamal 609
Sculpt 610
Sculpt Tech Brief 626
Sculpt Application 630
Parallel Meshing 633
Sculpt Mesh Transformation 634
Sculpt Adaptive Meshing 636
Sculpt Boundary Conditions 644
Sculpt Boundary Layers 653
Sculpt Sculpt Command Summary 657
Sculpt Mesh Improvement 659
Sculpt Input Data Files 672
Sculpt Mesh Type 681
Sculpt Output 692
Sculpt Process Control 696
Sculpt Bounding Box/Mesh Size 703
Sculpt Smoothing 712
Automatic Scheme Selection 717
free 720
Radialmesh 720
mesh quality assessment 725
Automatic Mesh Quality Assessment 725
Coincident Node Check 726
Controlling Mesh Quality 727
Metrics for Edge Elements 729
Metrics for Hexahedral Elements 730
Mesh Quality Assessment 733
Mesh Quality Example Output 734
Mesh Quality Command Syntax 736
Metrics for Quadrilateral Elements 738
Metrics for Tetrahedral Elements 741
Mesh Topology Check 744
Metrics for Triangular Elements 747

Metrics for Wedge Elements 749
Higher-order element types 750
Measuring Through Thickness 751
Finding Intersecting Mesh 752
mesh modification 756
Mesh Modification 756
mesh smoothing 757
Adjust Boundary Orthogonal 757
Centroid Area Pull 759
Condition Number 760
Edge Length Smoothing 762
Equipotential 763
Laplacian 764
Mean Ratio 765
Mesh Smoothing 766
Smart Laplacian 769
Untangle 770
Winslow 772
Align Mesh 773
Collapsing Mesh Edges 774
Creating and Merging Mesh Elements 776
Cleaning up a Mesh 779
Remeshing 781
Edge Swapping 784
Matching Tetrahedral Meshes 785
Mesh Coarsening 786
Mesh Refinement 788
Mesh Scaling 801
Block Repositioning 807
Node and Nodeset Repositioning 808
Mesh Pillowing 809
Mesh Column Operations 811
Removing Intersecting Mesh 814
mesh import 815
Importing a Mesh 815
Importing Fluent Files 816
Importing 2D Exodus Files 817
Importing Abaqus Files 818
Importing Exodus II Files 820

Importing Ideas Files 830
Importing Nastran Files 831
Importing Patran Files 832
adaptivity and sizing functions 833
Geometry Adaptive 833
Mesh Adaptivity and Sizing Functions 834
Bias Sizing Function 836
Constant Sizing Function 841
Curvature Sizing Function 842
Exodus II-based Field Function 843
Geometry Adaptive Sizing Function (Skeleton Sizing) 846
Interval Sizing Function 851
Inverse Sizing Function 852
Linear Sizing Function 853
Hyperbolic Tan Sizing Function 855
Free Meshes 858
Mesh Deletion 868
Mesh Validity 869
Skinning a Mesh 870
Lite Meshes 871
Creating Cohesive Elements 874
Finite Element Model 875
Finite Element Model 875
Element IDs 876
export 877
Export Mesh Association 877
Exporting Presto Files 878
Defining PARAMS for NASTRAN 879
Exporting ABAQUS files 880
Exporting an Exodus II File 881
Exporting the Finite Element Model 884
Exporting Fluent Grid Files 887
Transforming Mesh Coordinates 889
Coordinate Frames 890
Exporting GDF Files 891
exodus 892
Element Block Specification 892
Exodus II File Specification 903
Exodus II Model Title 904
Defining Materials 905

Exodus Boundary Conditions 908
Nodeset and Sideset Specification 910
non exodus 919
Cubit Initial Conditions 919
Using Constraints 920
Non-Exodus Boundary Conditions 921
Using CFD Boundary Conditions 923
Using Contact Surfaces 924
Using Loads 926
Miscellaneous Boundary Condition Commands 929
Using Restraints 931
Boundary Condition Sets 934
Boundary Layer Meshing 936
Boundary Layer Meshing 936
Step-by-Step Tutorials 942
Step-by-Step Tutorials 942
item 943
Item Tutorial Overview 943
ITEM Tutorial Step 1 944
ITEM Tutorial Step 2 945
ITEM Tutorial Step 3 947
ITEM Tutorial Step 4 952
ITEM Tutorial Step 5 954
ITEM Tutorial Step 6 957
ITEM Tutorial Step 7 961
ITEM Tutorial Step 8 962
ITEM Tutorial Step 9 966
power tools 968
Power Tools Tutorial Overview 968
Power Tools Tutorial Step 1 969
Power Tools Tutorial Step 2 972
Power Tools Tutorial Step 3 975
Power Tools Tutorial Step 4 978
Power Tools Tutorial Step 5 980
Power Tools Tutorial Step 6 983
Power Tools Tutorial Step 7 988
Power Tools Tutorial Step 8 992
Power Tools Tutorial Step 9 994
Power Tools Tutorial Step 10 995

Power Tools Tutorial Step 11 1001
decomposition 1005
Decomposition Tutorial 1005
Example 1. Sweeping multiple adjacent volumes 1010
Example 2. Interlocking rings 1012
Example 3. Webcutting using the sweep option 1014
Example 4. Using the Loft command 1016
Example 5. Multiple sweep directions 1019
Example 6. Employing Symmetry 1021
Example 7. Using virtual geometry in geometry decomposition 1028
Example 8. Sweeping volumes with narrow angles and surfaces 1033
gui 1040
GUI Basic Tutorial Overview 1040
GUI Basic Tutorial Step 1 1042
GUI Basic Tutorial Step 2 1044
GUI Basic Tutorial Step 3 1046
GUI Basic Tutorial Step 4 1048
GUI Basic Tutorial Step 5 1049
GUI Basic Tutorial Step 6 1051
GUI Basic Tutorial Step 7 1054
GUI Basic Tutorial Step 8 1057
GUI Basic Tutorial Step 9 1059
GUI Basic Tutorial Step 10 1061
GUI Basic Tutorial Step 11 1063
command line 1064
CL Basic Tutorial Overview 1064
CL Basic Tutorial Step 1 1066
CL Basic Tutorial Step 2 1067
CL Basic Tutorial Step 3 1068
CL Basic Tutorial Step 4 1069
CL Basic Tutorial Step 5 1071
CL Basic Tutorial Step 6 1072
CL Basic Tutorial Step 7 1074
CL Basic Tutorial Step 8 1075
CL Basic Tutorial Step 9 1077
CL Basic Tutorial Step 10 1080
CL Basic Tutorial Step 11 1081
Geometry Cleanup Process Flow 1082
ITEM 1084
Immersive Topology Environment for Meshing (ITEM) 1084

How to Use the ITEM Wizard 1087
Setting Up the Finite Element Model 1093
Defining the Geometric Model 1095
Generating a Mesh in ITEM 1097
Validating the Mesh in ITEM 1101
clean up 1102
Recognizing Nearly Sweepable Regions 1102
Blend Surfaces 1104
Clean Up the Geometry 1105
Resolving Problems with Conformal Assemblies 1107
Contact Surfaces 1111
Geometry Decomposition 1112
Forced Sweepability 1114
Bad geometry representation 1115
Determining an Appropriate Merge Tolerance 1116
Building a Sweepable Topology 1119
Small details in the model 1120
Determining the Small Feature Size 1124
Appendix 1126
Appendix 1126
alpha 1127
Alpha Commands 1127
Acis Geometry From Mesh 1128
Deleting Mesh Elements 1129
FeatureSize 1130
Importing Abaqus Files 1131
Importing MBG Files 1132
Exporting MBG Files 1133
Mesh Cutting 1134
Mesh Grafting 1139
Optimize Jacobian 1142
Randomize 1143
Refine Mesh Boundary 1144
Remove Tiny Edge Length 1145
Super Sizing Function 1147
Test Sizing Function 1148
Transition 1149
Triangle Mesh Coarsening 1151
Higher Order Element Metrics 1153

aprepro 1155
APREPRO 1155
Using APREPRO in CUBIT 1156
APREPRO Functions 1159
APREPRO Journaling 1168
python 1170
Python Interface 1170
Importing Cubit into Python 1171
Python Cubit Enhancement Scripts 1172
CubitInterface Namespace 1173
Entity Class Reference 1606
GeomEntity Class Reference 1610
Body Class Reference 1628
Volume Class Reference 1633
Surface Class Reference 1640
Curve Class Reference 1645
Vertex Class Reference 1658
CubitFailureException Class Reference 1660
InvalidEntityException Class Reference 1661
InvalidInputException Class Reference 1662
MeshErrorFeedback Class Reference 1663
AssemblyItem Class Reference 1665
CFD_BC_Entity Class Reference 1668
Dir Class Reference 1670
Loc Class Reference 1675
Navigation XML 1676
Periodic Space Filling Models (Tile) 1678
References 1680
Available Colors 1684
Element Numbering 1689
FASTQ 1692
FullHex vs NodeHex Representation 1695
ATO_to_Mesh 1696
Credits 1702
Credits 1702
Quick Reference 1703
Quick Reference 1703

SAND Number: SAND2025-13033W

CUBIT®

Geometry and Mesh Generation Toolkit

17.06 User Documentation

Michael Skroch, Steven J. Owen, Matthew L. Staten, Roshan W. Quadros, Byron Hanks, Brett Clark, Trevor Hensley, Corey Ernst, Randy Morris, Corey McBride, Clinton Stimpson, James Downer, Mark Richardson, Karl Merkley

Prepared by
Sandia National Laboratories
Albuquerque, New Mexico 87185 and Livermore, California 94550

Sandia National Laboratories is a multi-mission laboratory managed and operated by National Technology & Engineering Solutions of Sandia, LLC., a wholly owned subsidiary of Honeywell International, Inc., for the U.S. Department of Energy's National Nuclear Security Administration under contract DE-NA0003525.

Approved for public release; further dissemination unlimited.



Sandia National Laboratories



Introduction

- Key Features
- Hardware Requirements
- Licensing and Distribution
- Trademark Notice
- How to Use this Manual
- Cubit Mailing Lists
- Problem Reports and Enhancement Requests

Welcome to CUBIT, the Sandia National Laboratory automated mesh generation toolkit. CUBIT is a full-featured software toolkit for robust generation of two- and three-dimensional finite element meshes (grids) and geometry preparation. Its main goal is to reduce the time to generate meshes, particularly large hex meshes of complicated, interlocking assemblies. It is a solid-modeler based preprocessor that meshes volumes and surfaces for finite element analysis. Mesh generation algorithms include quadrilateral and triangular paving, 2D and 3D mapping, hex sweeping and multi-sweeping, tetrahedral meshing, and various special purpose primitives. CUBIT contains many algorithms for controlling and automating much of the meshing process, such as automatic scheme selection, interval matching, sweep grouping, and also includes state-of-the-art smoothing algorithms

The CUBIT environment is designed to provide the user with a powerful toolkit of meshing algorithms that require varying degrees of input to produce a complete finite element model. Many CUBIT users want to experiment with capabilities as soon as possible. Hence, CUBIT releases often contain algorithms which are not quite ready for production use. These features are listed in the Appendix, and are accessible to the user by specifying a developer flag.

The overall goal of the CUBIT project is to reduce the time it takes a person to generate an analysis model. Generating meshes for complex, solid model-based geometries requires a variety of tools. Many CUBIT tools are completely automatic, while others require user input. Usually, the automatic choices can be over-ridden by the user if necessary. Most meshing capabilities are integrated into the common CUBIT framework; there are also stand-alone tools like Verde. The user is encouraged to become familiar with all of the available tools, so that he can choose the right one for the job.

CUBIT Mailing Lists

CUBIT® users mailing list

We maintain a mailing list intended for our users to discuss CUBIT® issues and to use for a help resource. It also carries announcements that we feel are of interest to our user community. To subscribe to or unsubscribe from this list, send an email to cubit-support@sandia.gov from your organizational email address inserting **subscribe cubit** or **unsubscribe cubit** in the subject line.

To send mail to the list, use cubit@sandia.gov

CUBIT® Announcements mailing list

We also maintain a mailing list that will **only** broadcast announcements we feel will be of interest to our users. If you are already subscribed to our cubit@sandia.gov mailing list, you do NOT need to subscribe to the cubit-announce@sandia.gov list, as announcements will be echoed on it also. To subscribe to or unsubscribe from the announcements-only mailing list, send an email to cubit-support@sandia.gov from your organizational email address inserting **subscribe cubit-announce** or **unsubscribe cubit-announce** in the subject line.

Hardware Requirements

Cubit is available on the following platforms:

- Red Hat 8, 64 bit
- Windows 10, 64 bit

The Graphical User Interface version is available on all platforms.

For best results, local displays supporting OpenGL 3.2 or newer is recommended.

How to Use This Manual

This manual provides specific information about the commands and features of CUBIT. It is divided into chapters, which roughly follow the process in which a finite element model is created, from geometry creation to mesh generation to boundary condition application. Examples are provided in the tutorial chapter. Appendices contain advanced topics, alpha commands, summary of APREPRO functions, FASTQ reference, a troubleshooting guide, and references.

	Integrated in CUBIT are algorithms and tools, which are in a user-beware state. As they are further tested (often with the assistance of users) and improved, the tool becomes more stable and production-worthy. Since documentation of the tool is necessary for actual use, we have included the documentation of all available tools. However, a "hammer" icon is placed next to some capabilities as a warning.
	Certain portions of this manual contain information that is vital for understanding and effectively using CUBIT. These portions are highlighted with a "key" icon.

Key Features

Geometry Creation, Modification, and Healing

CUBIT usually relies on the ACIS solid modeling kernel for geometry representation; there is also mesh-based geometry. Other solid model kernels are planned. Geometry is imported or created within CUBIT. Geometry is created bottom-up or through primitives. CUBIT can also read STEP, IGES, and FASTQ files and convert them to the ACIS kernel. SolidWorks, AutoCAD, and some other commercial CAD systems can write SAT files directly.

Once in CUBIT, an ACIS model is modified through Booleans, or tweaking curves and surfaces. Without changing the geometric definition of the model, the topology of the model may be changed using virtual geometry. For example, virtual geometry can be used to composite two surfaces together, erasing the curve dividing them.

Sometimes, an ACIS model is poorly defined. This often happens with translated models. The model can be healed inside CUBIT.

Non-Manifold Topology

Typical assembly meshes require contiguous mesh across multiple parts in an assembly. CUBIT accomplishes this by taking the two touching surfaces of neighboring volumes, and merging them into a single surface. There will be only one mesh of the surface, and both volume meshes will share that surface mesh. (In contrast, some meshing packages keep two surfaces, and take steps to ensure their mesh connectivity and positions match.)

These shared surfaces are called non-manifold topology. Geometric models are usually imported into CUBIT as manifold (non-shared) models; then, surfaces which pass a geometric and topological comparison are "merged". A similar technique is used to merge model edges and vertices across parts. These comparisons are performed automatically, and can optionally be restricted to subsets of the model (to allow representations of such features as slide lines).

Geometry Decomposition

Solid models often require decomposition to make them amenable to hexahedral meshing. CUBIT contains a wide variety of tools for interactive geometry decomposition, and a capability for performing automatic geometry decomposition is also under development.

Mesh Generation

CUBIT contains a variety of tools for generating meshes in one, two and three dimensions. While the primary focus of CUBIT is on generating unstructured quadrilateral and hexahedral meshes, algorithms are also available for structured mesh generation and triangle/tetrahedral mesh generation. Several algorithms for generating mixed hex-tet meshes are also being developed.

Boundary Conditions

CUBIT uses different boundary conditions for EXODUS-II format and Non-Exodus formats such as ABAQUS, for importing and exporting mesh data. EXODUS represents boundary conditions on meshes using Element Blocks, Nodesets, and Sidesets. Element Blocks are used to group elements by material type. Nodesets are used to group nodes.

Other analysis programs can apply nodal boundary conditions to these sets, such as enforced displacement or nodal temperature values. Sidesets are used to group sides of elements, such as faces of hexes or edges of quads. Other analysis programs can apply face-based and edge-based boundary conditions to these sets, for example pressure or heat flux.

Using Element Blocks, Nodesets and Sidesets, a mesh and boundary conditions can be specified in an analysis-independent manner. Typically this specification is combined with an additional data file which designates the specific type of boundary condition (temperature, displacement, pressure, etc.), along with boundary condition values.

Non-Exodus export formats such as Abaqus support more specific boundary condition sets. These sets may include displacements, temperatures, forces, heatflux, pressure, or contact pairs.

Element Types

CUBIT supports a wide variety of element types, including 1d, 2d, and 3d elements of various orders. Each block has a unique element type. The element type is specified after the block is created, and after mesh generation (recommended). Higher order nodes are generated when the element type is specified. Higher order nodes are projected to curved geometry, depending on the user-settable node constraint flag.

Graphics Display Capabilities

CUBIT uses the VTK package for its graphics and rendering engine. CUBIT can display geometric and mesh entities in several modes, including hidden line, shaded, transparent or wireframe modes. CUBIT supports screen picking of geometric and mesh entities, as well as mouse-controlled view transformations like rotate, pan, and zoom. VTK takes advantage of hardware acceleration on most supported platforms. Image files of any displayed image can also be generated. CUBIT can also be run without graphics, to allow execution in batch mode or over slow network connections.

Graphical User Interface

A full graphical user interface (GUI) with the standard look and feel consistent with major platforms is available on all supported Cubit platforms. The GUI version can improve productivity, making new users aware of the wide range of CUBIT capabilities, and freeing new and experienced users from having to remember esoteric syntax. The GUI and non-GUI versions create and play back identical journal files, making it easier to switch from one environment to the other.

Command Line Interface

In the command line interface, commands are specified by text rather than mouse clicks. Commands can be entered interactively or in batch mode by playing back a journal file. The command line interface is available in the GUI through a window. The non-GUI version supports graphical picking and echoing to the command line, and also mouse-driven view transformations, but no menus and dialog boxes. The command line and GUI dialog boxes support the APREPRO preprocessor, which allows parameterization of input. The non-GUI version is available on all platforms, including Windows.

Licensing, Distribution and Installation

Please refer to https://cubit.sandia.gov/licensing for information on licensing and distribution.

Problem Reports and Enhancement Requests

CUBIT bugs, problem reports and enhancement requests should be reported via the Help Desk Portal or sent to cubit-help@sandia.gov or cubit-dev@sandia.gov. The CUBIT production meshing team or development team will review the email quickly. Users should expect some type of response within two days. Bugs are usually entered by a developer into CUBIT's bug tracking system.

Trademark Notice

ACIS™ is a proprietary format developed by Spatial Corporation.

Granite is a proprietary format developed by Parametric Technology Corporation

All other trademarks are the property of their respective owners.

Environment Control

- Session Control
- Graphical User Interface
- Command Recording and Playback
- Graphics Window Control
- Entity Selection and Filtering
- Location, Direction, and Axis Specification
- Listing Information

The CUBIT user interface is designed to fill multiple meshing needs throughout the design to analysis process. The user interface options include a full graphical user interface, a modern command line interface as well as no-graphics and batch mode operation. This chapter covers the interface options as well as the use of journal files, control of the graphics, a description of methods for obtaining model information, and an overview of the help facility.

Session Control

- Starting and Exiting a CUBIT Session
- Execution Command Syntax
- Initialization Files
- Environment Variables
- Command Syntax
- Command Line Help
- Environment Commands
- Saving and Restoring a CUBIT Session
- Interrupting Running Tasks

This section provides an overview to session control in CUBIT. This includes information on starting and exiting a CUBIT session, running CUBIT in batch mode, initialization files, how to enter commands, file manipulation, changing the working directory, memory manipulation and more. Much of your ability to use CUBIT effectively depends on mastery of concepts in this section. Even experienced users will find it useful to review this section periodically.

Command Line Help

In addition to the documentation you are currently viewing, CUBIT can give help on command syntax from the command line. For help on a particular command or keyword, the user can simply type **help <keyword>**. If the user is uncertain of the keyword, an asterisk ***** may be added to the end of the entered characters and help for all keywords that start with the entered characters will be printed. In addition, if the user has typed part of a command and is uncertain of the syntax of the remainder of the command, they can type a question mark **?** and help will be printed for the sequence of keywords currently entered. It is important to note that if the user has typed the keywords out of order, then no help will be found. If the user is not sure of the correct order of the keywords, the ampersand **&** key will search on all occurrences of whatever keywords are entered, regardless of the order. The results of this type of command are shown in the following listing.

CUBIT>help degenerate*
Help for words: degenerate*.
set Block Mixed Element Output { OFFSET | Degenerate }
set Degenerates [on|off]

CUBIT> volume 3 label ?
Completing commands starting with: volume, label.
Help not found for the specified word order.

CUBIT> volume 3 label &
Help for words: volume & label
Label Volume [on | off | name [only|id] | id | interval | size | scheme | merge | firmness]

CUBIT> label volume 3 ?
Completing commands starting with: label, volume.
Label Volume [on|off|name [only|ids]|ids|interval|size|scheme|merge|firmness]

Command Syntax 

The execution of CUBIT is controlled either by entering commands from the command line or by reading them in from a journal file. The commands can be either Cubit commands or Python statements. Similarly, a journal file may contain both Cubit commands and Python statements. By default, a command is interpreted as a Cubit command when starting the processing of a journal file or at the command line. A **#!python** statement can be given to indicate the subsequent commands are to be interpreted as Python statements. A **#!cubit** will similarly indicate the subsequent commands are to be interpreted as Cubit commands.

Throughout this document, each function or process will have a description of the corresponding CUBIT command; in this section, general conventions for command syntax will be described. The user can obtain a quick guide to proper command format by issuing the **`<keyword> help`** command; see Command Line Help for details.

CUBIT commands are described in this manual and in the help output using the following conventions. An example of a typical CUBIT command is:

```
Volume <range> Scheme Sweep [Source [Surface]
<range>] [Target [Surface] <range>] [Rotate
{on | OFF}]
```

The commands recognized by CUBIT are free-format and abide by the following syntax conventions.

1. Case is not significant.
2. The "#" character in any command line begins a comment. The "#" and any characters following it on the same line are ignored. Although note that the "#" character can also be used to start an Aprepro statement. See the Aprepro documentation for more information.
3. Commands may be abbreviated as long as enough characters are used to distinguish it from other commands.
4. The meaning and type of command parameters depend on the keyword. Some parameters used in CUBIT commands are:

   Numeric: A numeric parameter may be a real number or an integer. A real number may be in any legal C or FORTRAN numeric format (for example, 1, 0.2, -1e-2). An integer parameter may be in any legal decimal integer format (for example, 1, 100, 1000, but not 1.5, 1.0, 0x1F).

   String: A string parameter is a literal character string contained within single or double quotes. For example, 'This is a string'.

   Filename: When a command requires a filename, the filename must be enclosed in single or double quotes. If no path is specified, the file is understood to be in the current working directory. After entering a portion of a filename, typing a '?' will complete the filename, or as much of the filename as possible if there is more than one possible match.

   A filename parameter must specify a legal filename on the system on which CUBIT is running. The filename may be specified using either a relative path (**../cubit/mesh.jou**), a fully-qualified path (**/home/jdoe/cubit/mesh.jou**), or no path; in the latter case, the file must be in the working directory (See Environment Commands for details.) Environment variables and aliases may also be used in the filename specification; for example, the C-Shell shorthand of

referring to a file relative to the user's login directory (**~jdoe/cubit/mesh.jou**) is valid.

Toggle: Some commands require a "toggle" keyword to enable or disable a setting or option. Valid toggle keywords are "**on**", "**yes**", and "**true**" to enable the option; and "**off**", "**no**", and "**false**" to disable the option.

5. Each command typically has either:

   * an action keyword or "verb" followed by a variable number of parameters. For example:

   ```
   Mesh Volume 1
   ```

   Here **Mesh** is the verb and **Volume 1** is the parameter.

   * or a selector keyword or "noun" followed by a name and value of an attribute of the entity indicated. For example:

   ```
   Volume 1 Scheme Sweep Source 1 Target 2
   ```

   Here **Volume 1** is the noun, **Scheme** is the attribute, and the remaining data are parameters to the **Scheme** keyword.

The notation conventions used in the command descriptions in this document are:

- The command will be shown in a format that **looks like this**:
- A word enclosed in angle brackets (**<parameter>**) signifies a user-specified parameter. The value can be an integer, a range of integers, a real number, a string, or a string denoting a filename or toggle. The valid value types should be evident from the command or the command description.
- A series of words delimited by a vertical bar (**choice1 | choice2 | choice3**) signifies a choice between the parameters listed.
- A toggle parameter listed in **ALL CAPS** signifies the default setting.
- A word that is not enclosed in any brackets, or is enclosed in curly brackets (**{required}**) signifies required input.
- A word enclosed in square brackets (**[optional]**) signifies optional input which can be entered to modify the default behavior of the command.
- A curly bracket that is inside a square bracket (e.g. **[Rotate {on|OFF}]**) is only required if that optional modifier is used.

Environment Commands

- Working Directory
- File Manipulation
- CPU Time
- Comment
- History
- Error Logging
- Determining the CUBIT Version
- Echoing Commands
- Digits Displayed

Working Directory

The working directory is the current directory where journal files are saved. To list the current directory type

```
pwd
```

The current path will be echoed to the screen. By default, the current directory is the directory from which CUBIT was launched. The command to change the current directory is

```
cd "<new_path>"
```

The new path may be an absolute reference, or relative to the current directory. The <TAB> key will complete unique file references.

File Manipulation

A helpful addition is the ability to do a directory listing of a directory. The command for this is

```
ls ['<file_name>']

or

dir ['<file_name>']
```

Note also that you can delete files from the command line. The command for this is

```
Delete File ['<file_name>']
```

The file name may include the wildcard character *, but not the wildcard character ?, since the ? is used for command completion. File deletion from the command line can also be disabled. If deletions are set to **off** files cannot be deleted from the cubit command line.

```
Set Deletions [ON|Off]
```

The **mkdir** command is used to create a new directory. The syntax for this command is:

```
Mkdir "<directory_name>"
```

This creates a new directory with the specified name and path. The command accepts an absolute path, a relative path, or no path. If a relative path is specified, it is relative to the current working directory, which can be seen by typing 'pwd' at the cubit command prompt. If no path is specified, the new directory is created in the current working directory.

The command succeeds if the specified directory was successfully created, or if the specified directory already exists. The command fails if

the new directory's immediate parent directory does not exist or is not a directory.

CPU Time

At times it is important to see how much cpu time is being used by a command. One function available to do this is the timer command. The syntax for this command is:

```
Timer [Start|Stop]
```

The start option will start a CPU timer that will continue until the stop command is issued. The elapsed time will be printed out on the command line. If no arguments are given, the command will act like a toggle.

Comment

This keyword allows you to add comments without affecting the behavior of CUBIT.

```
Comment ['<text_to_print>'] [<aprepro_var>]
[<numeric_value>]
```

The comment command can take multiple arguments. If an argument is an unquoted word, it is treated as an aprepro variable and its value is printed out. Quoted strings are printed verbatim, and numbers are printed as they would be in a journal string. For example:

```
CUBIT> #{x=5}
CUBIT> #{s="my string"}
CUBIT> comment "x is" x "and s is" s

User Comment: x is 5 and s is my string

Journaled Command: comment "x is" x "and s is" s
```

History

This command allows you to display a listing of your previous commands.

```
History <number_of_lines>
```

For example, if you type history 10, the most recent 10 commands will be echoed to the input window.

Error Logging

```
[set] Logging Errors {Off | On File
'<filename>'[Resume]}
```

This setting will allow users to echo error messages to a separate log file. The resume option will allow output to be appended to existing files instead of overwriting them. For more information on CUBIT environment settings see List Cubit Environment.

Determining the CUBIT Version

To determine information on version numbers, enter the command **`Version`**. This command reports the CUBIT version number, the date and time the executable was compiled, and the version numbers of the ACIS solid modeler and the VTK library linked into the executable. This information is useful when discussing available capabilities or software problems with CUBIT developers.

Echoing Commands

By default, commands entered by the user will be echoed to the terminal. The echo of commands is controlled with the command:

```
[Set] Echo {On | Off}
```

Digits Displayed

CUBIT uses all available precision internally, but by default will only print out a certain number of digits in order for columns to line up nicely. The user can override that with the "set digits" command:

```
Set Digits [<num_to_list=-1>]
```

If the digits are set to -1, then the default number of digits for pretty formatting are used. If the digits are set to a specific number, such as 15, more digits of accuracy can be displayed. This may be useful when checking the exact position and size of geometric features.

The number of digits used for listing positions, vectors and lengths can be listed using the following command:

```
List Digits
```

Examples:

CUBIT> set digits 6

Coordinates and lengths will be listed with up to 6 digits.

CUBIT> set digits 20

For this platform, max digits = 15. Coordinates and lengths will be listed with up to 15 digits.

CUBIT> set digits -1

To reset digits to default, use 'set digits -1'

The number of coordinate and length digits listed will vary depending on the context.

Environment Variables

CUBIT can interpret the following environment variables. These settings are only applicable to the Command Line Version of CUBIT and do not apply to the Graphical User Interface. See also the CUBIT_STEP_PATH and CUBIT_IGES_PATH environment variables. See also the CUBIT_DIR, HOMEDRIVE and HOMEPATH settings.

DISPLAY	The graphics window or GUI will pop-up on the specified X-Window display. This is useful for running CUBIT across a network, or on a machine with more than one monitor. Unix only.
CUBIT_OPT	Execution command line parameter options. Any option that is valid from the command line may be used in this environment variable. See Execution Command Syntax.
CUBIT_Journal	Specifies path and name to use for journal file. The specified path may contain the following %-escape sequences: %a - abbreviated weekday name %A - full weekday name %b - abbreviated month name %B - full month name %d - date of the month [01,31] %H - hour (24-hour clock) [00,23] %I - hour (12-hour clock) [01,12] %j - day of the year [1,366] %m - month number [1,12] %M - minute [00,59] %n - replaced with the next available number between 01 and 999. %p - "a.m." or "p.m." %S - seconds [00,61] %u - weekday [1,7], 1 is Monday %U - week of year [00,53] %w - weekday [0,6], 0 is Sunday %y - year without century [00,99] %Y - year with century (e.g. 1999) %% - a '%' character The default value is "cubit%n.jou". This creates journal files in the current directory named "cubit00.jou", "cubit01.jou", "cubit02.jou", etc. To keep the same naming scheme but create the files the /tmp directory, set CUBIT_JOURNAL to "/tmp/cubit%n.jou" To create journal files in directories according to the day of the week, first create directories named "Mon", "Tues", etc. CUBIT will not create them for you. Next set CUBIT_JOURNAL to "%a/%n.jou". This will create journal files named "01.jou" through "999.jou" in the appropriate directory for the current day of the week.

Execution Command Syntax

Cubit is available as a GUI application, a command line application or as a module that can be loaded in Python. The GUI application can be accessed by running **bin/claro** and the command line application can be accessed by running **bin/cubit**. On Linux, an additional **cubit** script is provided which can start either **bin/claro** or **bin/cubit** depending on whether the **-nogui** option is given. Cubit can be loaded into Python if the **bin** folder is given to **sys.path** or if it is given in the **PYTHONPATH** environment variable.

When running in batch mode, it is strongly recommended to run **cubit**.

The applications can be run as rollows:

```
bin/cubit [options and args] [journalFile(s)]

bin/claro [options and args] [journalFile(s)]
```

Input journalFiles or scripts can be given after all other options are given. These scripts can be in either Cubit syntax or in Python syntax. The syntax is Python if the script has a **.py** file extension. For **.jou** files or files with other extensions, the **#!cubit** or **#!python** comments in the file indicate how to interpret the syntax.

Command options for the command line are:

cubit
  -help (Print this summary)
  -Include <$val> (Specify a journal file)
  -workingdir <$val> (Directory to use as working directory)
  -input $val (Playback commands in file $val)
  -solidmodel <$val> (Read .sat, .cub or .exo from file $val)
  -nogeom (Read .exo from -solidmodel $val without creating geometry)
  -lite (Read .exo from -solidmodel $val in lite mode)
  -fastq <$val> (Read FASTQ file $val)
  -initfile <$val> (Read $val as initialization file instead of $HOME/.cubit)
  -batch (Batch Mode - No Interactive Command Input)
  -nographics (Do not display graphics windows)
  -nogui (Do not display graphical user interface)
  -noinitfile (Do not read .cubit file)
  -noecho (Do not echo commands to console)
  -nojournal (Do not write journal file)
  -nodeletions (Do not allow file deletions)
  -journalfile <$val> (Name of journal file, will be overwritten)
  -restore [$val] (Name of restore file (default = cubit_geom.save.sat))
  -maxjournal [$val] (Maximum number of journal files to write)
  -warning [$val] (Warning Messages On/Off)
  -information [$val] (Informational Messages On/Off)
  -debug <$val> (Set specified flags on, e.g. 1,3,7-9 enables 1,3,7,8,9))
  -display <$val> (Specify display to be used for graphics window)
  -driver <$val> (Specify the type of driver to be used for graphics display)
  -nooverwritecheck (Do not perform file export overwrite check)
  -nobanner (Suppress printing of startup information)
  -version (Prints version information)
  -log <$val> (Copy all output to specified file)
  -python_version <version> (The major version of python to use: 2 or 3
  APREPRO variable pair (Quoted name value pair)

Each of these is optional. If specified, the quantities in square brackets, **[$val]**, are optional and the quantities in angle brackets, **<$val>**, are required.

Options are summarized in more detail below:

-help	Print a short usage summary of the command syntax to the terminal and exit.
-workingdir	Set the working directory to be used at startup. Journal files will be written to this directory.
-initfile <$val>	Use the file specified by **<$val>** as the initialization file instead of the default set of initialization files. See Initialization Files
-noinitfile	Do not read any initialization file. This overrides the default behavior described in Initialization Files
-solidmodel <$val>	Read the ACIS solid model geometry or .cub file information from the file specified by **<$val>** prior to prompting for interactive input.
-batch	Specify that there will be no interactive input in this execution of CUBIT. CUBIT will terminate after reading the initialization file, the geometry file, and the **input_file_list.**
-nographics	Run CUBIT without graphics. This is generally used with the -batch option or when running CUBIT over a line terminal.
-nogui	Run CUBIT without the graphical user interface.
-display	Sets the location where the CUBIT graphics system will be displayed, analogous to the **-display** environment variable for the X Windows system. Unix only.
-driver <type>	Sets the **<type>** of graphics display driver to be used. Available drivers depend on platform, hardware, and system installation. Typical drivers include *X11* and *OpenGL*.

-nojournal	Do not create a journal file for this execution of CUBIT. This option performs the same function as the Journal Off command. The default behavior is to create a new journal file for every execution of CUBIT.
-journalfile <file>	Write the journal entries to **<file>**. The file will be overwritten if it already exists.
-maxjournal <$val>	Only create a maximum of **<$val>** default journal files. Default journal files are of the form cubit#.jou where # is a number in the range 01 to 999.
-nodeletions	Turn off the ability to delete files with the delete file '<filename>' command.
-nooverwritecheck	Turn off the file overwrite check flag. Files that are written may then overwrite (erase) old files with the same name with no warning. This is typically useful when re-running journal files, in order to overwrite existing output files. See the **set File Overwrite Check [ON|off]** command.
-restore	Restore the specified filename (or "cubit_geom") mesh and ACIS files, e.g. cubit_geom.save.g and cubit_geom.save.sat.
-noecho	Do not echo commands to the console. This option performs the same function as the Echo Off command. The default behavior is to echo commands to the console.
-debug <$val>	Set to "on" the debug message flags indicated by **<$val>**, where **<$val>** is a comma-separated list of integers or ranges of integers, e.g. 1,3,8-10.
-information {on|off}	Turn **{on|off}** the printing of information messages from CUBIT to the console.
-warning {on|off}	Turn **{on|off}** the printing of warning messages from CUBIT to the console.
-Include <path>	Allows the user to specify a journal file from the command line.

-fastq <file>	Read the mesh and geometry definition data in the FASTQ file **<file>** and interpret the data as FASTQ commands. See T. D. Blacker, FASTQ Users Manual Version 1.2, SAND88-1326, Sandia National Laboratories, (1988). for a description of the FASTQ file format.
<input_file_list>	Input files to be read and executed by CUBIT. Files are processed in the order listed, and afterwards interactive command input can be entered (unless the -batch option is used.) The input files can be in either Cubit syntax or in Python syntax.
-log <file>	Copies all output to the specified file.
-python_version <version>	The major version of python to use: 2 or 3.
<variable=value>	APREPRO variable-value pairs to be used in the CUBIT session. Values can be either doubles or character type (character values must be surrounded by double quotes.). Command options can also be specified using the **CUBIT_OPT** environment variable. (See Environment Variables .)

Passing Variables into a CUBIT Session

To pass an aprepro variable into a CUBIT Session, start cubit with the variable defined in quotes i.e. cubit **"some_var=2.3"**

Initialization Files

CUBIT can execute commands on startup, before interactive command input, through initialization files. This is useful if the user frequently uses the same settings.

The following files are played back in order, if they exist, at startup:

> **$(cubit install directory)/.cubit.install**
> **$HOME/.cubit**
> **$(current working directory)/.cubit**

The **$(cubit install directory)** is determined by the location the program is installed. On Linux and Windows, it'll be the bin directory of the installation and on macOS it'll be the Cubit.app/Contents/MacOS directory.

$HOME is an environment variable pointing to the location of the user's home directory. On Windows, the **HOMEDRIVE** and **HOMEPATH** environment variables will be used instead of the **HOME** environment variable.

The **$(current working directory)** is determined by where the user starts the program itself.

If the **-initfile <filename>** option is used on the command that starts cubit, then the other init files are skipped and only the specified filename is played back.

These files are typically used to perform initialization commands that do not change from one execution to the next, such as turning off journal file output, specifying default mouse buttons, setting geometric and mesh entity colors, and setting the size of the graphics window.

Interrupting Running Tasks

Many operations in the command line version of CUBIT can be interrupted using **<Control>-C**. Pressing **<Control>-C** will attempt to interrupt the running process as soon as feasible, returning the user back to the command line. Not all operations may be interrupted, and many can only be interrupted at certain stages. Any current tasks are canceled as soon as it is feasible to do so, including playback of journal files. The playback of a journal file is always stopped, even if the current running task cannot be interrupted. The journal file will stop at the next opportunity, when the current task is completed. Interrupted journal files may be resumed at the next command. See the section titled Controlling Playback of Journal Files for more information on controlling playback of journal files.

The **GUI** has a cancel button that can be used to interrupt the current command. The cancel button will turn red when a command can be interrupted. The cancel button has an 'x' on it, and is located on the status bar, which is at the bottom of the application.

Saving and Restoring a Cubit Session

There are currently two ways to save/restore a model in CUBIT. A file can be saved with either the Exodus or CUBIT File method. The method of choice is determined by a set command. The CUBIT method is the default.

```
Set Save [exodus|CUBIT] [Backups <number>]
```

CUBIT File Method

- New
- Open
- Save
- Import
- Export

The CUBIT file is a binary cross-platform compatible file for the storage of a Cubit model that is compact in size and efficient to access. It includes both the geometry and the associated mesh, groups, blocks, sidesets, and nodesets. Mesh and geometry are restored from the Cubit file in exactly the same state as when saved. For example, element faces and edges are persistent, as well as mesh and geometry ids. The Graphical User Interface version of CUBIT also provides a toolbar with direct access to file operations using the CUBIT File method described here.

New

Creates a new blank model with default name, closing the current model. The New command essentially acts like the reset command.

Open '<filename>'

Opens an existing *.cub file, closing the current model.

Save

A default file name is assigned when CUBIT is started (in very much the same way the journal files are assigned on startup) in the form cubit01.cub, for example. The current model filename is displayed on the title bar of the CUBIT window. Typing save at any time during your session will save the current model to the assigned *.cub file. The *.cub file includes the *.sat file and the mesh. Groups, blocks, sidesets and nodesets are also saved within the *.cub file. To change the name of the current model, or to save the model's current geometry to a different file, use the save as command. Note that 'save <file.cub>' is NOT a valid command.

```
Save

Save As 'filename.cub' [Overwrite]
```

The set file overwrite command can be toggled on and off to allow overwriting when using the save as command. The command is defaulted to not allow overwriting.

```
Set File Overwrite [On|OFF]
```

A backup file is created by default, allowing access to previous states of the model. The backup files are named *.cub.1, *.cub.2... The user can set the total number of backups created per model with the following command (the default number of backups is 99,999):

```
Set Save Backups <number>
```

As soon as the number of model backups reaches the maximum, the lowest numbered backup file will be removed upon subsequent backup creation.

To check on the status of a 'set' command, type in the command in question without any options. For example, to check which save method is currently toggled, type:

```
Set Save
```

Import

Appends a *.cub file to an existing model.

```
Import Cubit 'filename.cub'
[merge_globally]
```

Export

In addition to saving an entire model, one can use the export command to save only a portion of a model. The geometry and associated mesh, groups, blocks, sidesets and nodesets are exported. Only bodies or free surfaces, curves or vertices can be exported to a Cubit file.

```
Export Cubit 'filename.cub' entity-list
```

Starting and Exiting a CUBIT Session

The following commands are used to control CUBIT execution.

Starting the Session

Cubit contains both a GUI and a command line version of the application. Additionally, Cubit can be imported into a Python session with Cubit functions being available. If you have not yet installed CUBIT, instructions for doing so can be found in Licensing, Distribution and Installation. For details on how to start the application and a list of startup options see the Execution Command Syntax section of this document. CUBIT can also be run with initialization files or in batch mode.

Windows File Association

Windows users have the option to associate .cub, .sat, and .jou files with CUBIT. This means that double-clicking on one of these files will open it automatically in CUBIT. This option is available during the installation process

Exiting the Session

The CUBIT session can be discontinued with either of the following commands

```
Exit

Quit
```

Resetting the Session

A reset of CUBIT will clear the CUBIT database of the current geometry and mesh model, allowing the user to begin a new session without exiting CUBIT. This is accomplished with the command

```
Reset [Genesis | Block | Nodeset | Sideset |
QA_Records]
```

A subset of portions of the CUBIT database to be reset can be designated using the qualifiers listed. Advanced options controlled with the Set command are not reset.

QA Records are stored in exodus, genesis, or cub files. If your file contains an excessive amount of qa records and you don't need them, it is beneficial to reset them for faster file I/O.

You can also reset the number of errors in the current Cubit session, using the command

```
Reset Errors <value>
```

which will set the error count to the specified value, or zero if the value is left blank.

Abort Handling

In the event of a crash, Cubit will attempt to save the current mesh as "crashbackup.cub" in the current working directory just before it exits.

To disable saving of the crashbackup.cub file set an environment variable

CUBIT_NO_CRASHSAVE equal to true. Or, use the following command:

```
Set Crash Save [On|Off]
```

This command will turn on or off crashbackup.cub creation during a crash on a per-instance basis. To minimize the effects of unexpected aborts, use Cubit's automatic journaling feature, and remember to save your model often.

Command Recording and Playback

Sequences of CUBIT commands can be recorded and used as a means to control CUBIT from ASCII text files. Command or "journal" files can be created within CUBIT, or can be created and edited directly by the user outside CUBIT.

- Journal File Creation & Playback
- Controlling Playback of Journal Files
- Automatic Journal File Creation
- IDless Journal Files

Automatic Journal File Creation

Controlling Automatic Journal File Creation

By default, CUBIT automatically creates a journal file each time it is executed. The file is created in the current directory, and its name begins with the word "cubit", followed by a number starting with cubit01.jou and continuing up to a maximum of cubit999.jou. It is recommended that the user keep no more than around 100 journal files in any directory, to avoid using up disk space and causing confusion. To that end, when the journal name increments to more than cubit99.jou, a warning will be given on startup telling the user that there are at least 99 journal files, and to please clean out unused files. If the user has up through cubit999.jou, then the user is warned that there are too many journal files in the current directory, and cubit999.jou will be re-used, destroying the previous contents.

When starting cubit, the choice of journal file name will depend on the existence of other cubitXX.jou files. Cubit will fill in gaps, starting with the lowest number. For example, if there are already journal files with names cubit01.jou, cubit02, jou, and cubit04.jou, then Cubit will use cubit03.jou as the current journal file.

Journal file names end with a ".jou" extension, though this is not strictly required for user-generated journal files. If no journaling is desired, the user may start CUBIT with the -nojournal command line option or use the command :

```
[Set] Journal {Off | On}
```

Turning journaling back on resumes writing commands to the same journal file.

Most CUBIT commands entered during a session are journaled; the exceptions are commands that require interactive input (such as Zoom Cursor), some graphics related commands, and the **Playback** command.

Recording Graphics Commands

All graphics related commands may be enabled or disabled with the command:

```
Journal Graphics {On | Off}
```

The default is **Journal Graphics Off** .

Recording Entity IDs and Names

When an entity is specified in a command using its name, the command may be journaled using the entity name, or by using the corresponding entity type and id. The method used to journal commands using names is determined with the command:

```
Journal Names {On | Off}
```

The default is **Journal Names On** .

If an entity is referred to using its entity type and id, the command will be journaled with the entity type and id, even if the entity has been named.

Recording APREPRO Commands

APREPRO commands may be echoed to the journal file using the

following command

```
[set] Journal [Graphics|Names|Aprepro|Errors]
[on|off]
```

See APREPRO Journaling for more information.

Recording Errors

The default mode for CUBIT is to not journal any command that does not execute successfully. To turn this mode off and echo all commands to the journal file, regardless of the success status, use the following command:

```
Journal Errors {On|OFF}
```

If a command did not execute successfully and the journal errors status is ON, then the unsuccessful command will be written as a comment to the file. For example an unsuccessful command might look like the following in the journal file

```
## create brick x 10 x 10 z 10
```

Since CUBIT recognizes this as erroneous syntax, it will issue an error when the command is issued, but will still write the command to the journal file as a comment, prefixing the command with "##".

This option may be useful when tracking or documenting program errors.

Controlling Playback of Journal Files

The following commands control the playback of Journal Files:

> **Stop**
>
> **Pause**
>
> **Sleep <duration_in_seconds>**
>
> **Resume [<n>]**
>
> **Where**
>
> **Next [<n>]**

The playback of a journal file can be interrupted in three ways. Pressing **ctrl-c** while the journal file is playing will halt playback of the journal file. (This only works in the command line version of CUBIT. See Interrupting Running Tasks for more information). Alternately, if the **stop** or **pause** commands are encountered in the journal file and CUBIT is reading commands from a terminal (as opposed to a redirected file), playback of the journal file will halt after that command.

The **sleep** command pauses execution for the specified number of seconds. It can be used to build a delay into journal files during presentations.

In the command line version of CUBIT you can resume playback of a journal file with the **resume** command. If playback was interrupted because **ctrl-c** was pressed, it will resume at the next command after the one that was interrupted. If playback stopped because of a **stop** or **pause** command in the journal file, it will resume at the next line after the **stop** or **pause** command. If the file was paused because of a **sleep** command in the file, it will resume automatically after the specified duration.

If journal files that are playing back contain **playback** commands themselves, there may be multiple current journal files. The **where** lists all current journal files and where the journal files have paused. Each line contains the stack position (a number), the filename and the current line in the file. Unless CUBIT is running in batch mode, the first line is always **<stdin>**. This just means that CUBIT will return to the command prompt after the top-most journal file has completed.

The remaining portion of any active journal file may be skipped by specifying the stack position (first number on each line of the output from the **where** command) of the file where you want to resume. Any remaining commands in active journal files with lower stack positions will be skipped.

The **next** command steps through interrupted journal files line-by-line. The argument to the **next** command is the number of lines to read before halting playback again. If no number is specified, the command will advance one line.

Journal playback can also be set to stop automatically when it encounters an error during playback. The command syntax is:

```
Set Stop Error {On|OFF}
```

Setting the stop error to "on" will cause the file to halt for each error. The setting is turned off by default.

Journal File Creation and Playback

Recording a Session

Command sequences can be written to a text file, either directly from CUBIT or using a text editor. CUBIT commands can be read directly from a file at any time during CUBIT execution, or can be used to run CUBIT in batch mode. To begin and end writing commands to a file from within CUBIT, use the command

```
Record '<filename>'

Record Stop
```

Once initiated, all commands are copied to this file after their successful execution in CUBIT.

Replaying a Session

To replay a journal file, issue the command

```
Playback '<filename>'
```

Journal files are most commonly created by recording commands from an interactive CUBIT session, but can also be created using automatic journaling or even by editing an ASCII text file.

Commands being read from a file can represent either the entire set of commands for a particular session, or can represent a subset of commands the user wishes to execute repeatedly.

Two other commands are useful for controlling playback of CUBIT commands from journal files. **Playback** from a journal file can be terminated by placing the **Stop** command after the last command to be executed; this causes CUBIT to stop reading commands from the current journal file. Playback can be paused using the Pause command; the user is prompted to hit a key, after which playback is resumed.

Journal files are most useful for running CUBIT in batch mode, often in combination with the parameterization available through the APREPRO capability in CUBIT. Journal files are also useful when a new finite element model is being built, by saving a set of initialization commands then iteratively testing different meshing strategies after playing that initialization file.

Idless Journal Files

Journal files can also be created without reference to entity IDs. The purpose of this command is to enable journal files created in earlier versions of CUBIT to be played back in newer versions of CUBIT. Using the "IDless" method, commands entered with an entity ID will be journaled with an alternative way of referring to the entity. Changes in CUBIT or ACIS often lead to changes in entity IDs. For example, a webcut may result in volume 3 on the left and volume 4 on the right. In another version of CUBIT, those entity IDs may be swapped (4 on the left and 3 on the right). Playing an IDless journal file makes the actual ID of an entity irrelevant. The syntax for this command is:

```
[set] Journal IDless {on|off|reverse}
```

The **on** option will enable idless journaling, and commands will be journaled without entity IDs. For example, "mesh volume 1" may be journaled as "mesh volume at 3.42 5.66 6.32 ordinal 2".

Selecting the **off** option will cause commands to be journaled in the traditional manner (i.e., as they are entered).

The **reverse** option allows you to convert idless journal files back into an ID-based journal file where the new journal file will reflect current numbering standards for IDs.

If you issue the command **Journal IDless** without any additional options, then the current status of ID journaling is printed. At startup, this should be "off".

The most likely scenario for converting older journal is to use the record command during playback. The following is an example.

```
journal idless on
record "my_idless.jou"
playback "my_journal.jou"
record stop
journal idless off
```

To record an *idless* journal file back into an *id-based* journal file you might use the following sequence.

```
journal idless reverse
record "new_id_based.jou"
playback "my_idless.jou"
record stop
journal idless off
```

Note: IDless conversions of APREPRO expressions are partially supported.

When IDless mode is set to **ON**, APREPRO functions such as Vx(id), that take an ID as an argument, are converted to use (x, y, z, ord) as arguments such as Vx(x, y, z, ord), where (x, y, z) is the center point coordinates and ord is the ordinal value. The ordinal values, 1..n, identifies each entity in a set of n entities that have a common center point. An entity's ordinal value is based on its creation order with respect to the other entities within the same set.

When IDless mode is set to **REVERSE** (using the above example) Vx(x, y, z, ord) will be converted to Vx(id). Outside these APREPRO functions, APREPRO expressions are not modified when converting a journal file to or from its IDless form. Hence, expressions reduced to an entity ID, such as in the command "volume {x} size 10," are not modified. Therefore, when moving a journal file from one version of CUBIT to another, it may be necessary to manually update IDs in APREPRO expressions.

Location, Direction and Axis Specification

- Specifying a Location
- Specifying a Location on a Curve
- Specifying a Direction
- Specifying an Axis
- Specifying a Plane
- Drawing a Location, Direction, or Axis

Many commands require that a location or a direction be specified. Although entering the three floating point numbers required to uniquely define a vector is perfectly acceptable, it may be more convenient to specify the direction or location with respect to existing entities in the model.

For example, the following commands might be used for creating straight curves using location and direction specification described here:

```
Create Curve [From] Location {options}
Location {options}

Create Curve [From] Location {options}
Direction {options} Length <val>
```

Drawing a Location, Direction, or Axis

Some commands require you to specify a location on a curve (i.e., webcutting with a plane normal to a curve). This location can be previewed with the following options:

1. A fraction along the curve from the start of the curve, or optionally, from a specified vertex on the curve.
2. A distance along the curve from the start of the curve, or optionally, from a specified vertex on the curve.
3. An xyz position that is moved to the closest point on the given curve.
4. The position of a vertex that is moved to the closest point on the given curve.

```
Draw Location On Curve <curve id> {Fraction
<f> | Distance <d> | Position <xval><yval>
<zval> | Close_To Vertex <vertex_id>} [[From]
Vertex <vertex_id> (optional for 'Fraction' &
'Distance')]
```

Some commands require a specified axis (such as webcut with a cylinder) and it is sometimes advantageous to view an axis before modifying geometry. To draw a preview of an axis use the following command:

```
Draw Axis {options}
```

Some commands require a specified location or point (such as create curve spline) and it is sometimes advantages to view a location before modifying or creating geometry. To draw a preview of a location use the following command:

```
Draw Location {options} [color <color_name>]
[no_flush]
```

Some commands require a direction. To draw a preview of a direction, use the following command:

```
Draw Direction {options} [from] [location
{options}]
```

Similar commands for drawing lines and polygons may also be useful.

Specifying an Axis

Some commands require a specified axis (such as webcut with a cylinder) and it is sometimes advantageous to view an axis before modifying geometry. An axis is simply a vector with a specified origin. The following options determine an axis specification:

- Last
- Direction {options} [Origin [Location] {options}] [Length <val>] [Angle <val>]
- Surface <id>
- Curve <id>
- [Revolve [About] Axis {options} Angle <val>]]

Last

```
Last
```

The last option recalls the last axis used in an axis command. The last axis does not carry over from CUBIT session to CUBIT session.

Specify a direction and a location

```
Direction {options} [Origin [Location]
{options}] [Length <val>] [Angle <val>]
```

To specify an axis simply specify a vector (a direction) and an origin (a location). Notice that the command requires the axis direction first because the origin defaults to 0 0 0 when not specified. An example of specifying an axis to draw a location using the swing command is as follows:

```
draw location 1 0 0 swing about axis direction
z ang 45
```

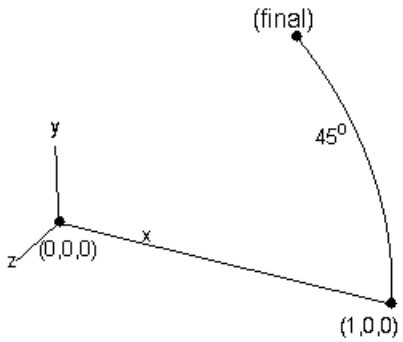


Figure 1 - Swinging a point about the z-axis

The location 1 0 0 was swung 45 degrees about an axis defined by a vector in the z direction and an origin at 0 0 0.

Specify a surface

```
Surface <id>
```

If a surface is specified, it must be a cone type surface. The axis of the cone surface is used. If a non-cone type surface is specified, an error will result.

Specify a curve

```
Curve <id>
```

If a curve is specified, it must be an arc, helix, or spline of constant curvature. All other curve types will result in an error.

Option to revolve an axis about an axis

```
[Revolve [About] Axis {options} Angle <val>]
```

To revolve one axis around another use the revolve keyword. The following example revolves the first axis (defined by the y-axis and origin) around the second axis (defined by the z-axis and origin) by 45 degrees and draws the result.

```
draw axis direction y revolve axis direction z
angle 45
```

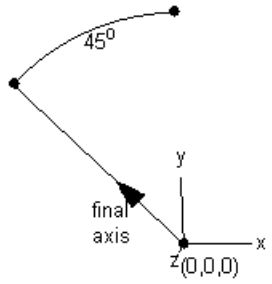


Figure 2 - Revolving an axis about another axis

Previewing an Axis

Sometimes it is helpful to preview an axis before using it in a command. An axis may be previewed using the Draw command. The options for previewing an axis are the same as the ones described above.

```
Draw Axis {options}
```

Specifying a Direction

Some commands require a specified a direction or vector for the command. A direction is basically a xyz vector in the model. The following options determine a direction specification:

- [Vector] <xval yval zval>
- Last
- X|Y|Z|Nx|Ny|Nz
- [On] | [Tangent] [At] Curve <id> {location on curve options}
- [On] | [Normal] [At] Surface <id> [Location {options}]
- [From] { Location {options} | {Node|Vertex} <id> }[Project] {Location {options} | [Entity] {Node|Vertex|Curve|Surface} <id> }
- [Rotate {options}]
- [Cross [With] Direction {options}]
- [Reverse]

Vector (XYZ values)

```
[Vector] <xval yval zval>
```

The most basic way to specify a direction is to just give the vector x-y-z components of the direction. The given vector need not be a unit vector. The following three commands simply draw a direction in the x-direction (1, 0, 0) as the Vector keyword is optional and unit vectors are not required:

```
draw direction vector 1 0 0
draw direction 1 0 0
draw direction 10 0 0
```

Last Direction Used

```
Last
```

The last option recalls the last direction used in a command. For example, if the following command is entered after the above vector commands a direction location would be drawn in the x-direction (1, 0, 0).

```
draw direction last
```

Last directions do not carry over from CUBIT session to CUBIT session. The last direction defaults to (1, 0, 0) if no direction has been used during the session.

Positive or Negative X,Y,Z Direction Vectors

```
X|Y|Z|Nx|Ny|Nz
```

The x|y|z|nx|ny|nz options assign the x direction, y direction, z direction, negative x direction, negative y direction and negative z direction respectively.

On Curve Tangent

```
[On] | [Tangent] [At] Curve <id> {location on
curve options}
```

The curve option simply finds a tangent vector on a curve. Note that the **on**, **tangent** and **at** keywords are optional, as well as the location on the curve. If no location is specified, the tangent at the start vertex of the curve is found. See the section above, Specifying a Location on a Curve, for details on how to specify where along the curve the tangent vector is

found.

```
draw direction curve 1
draw direction on curve 1
draw direction tangent at curve 1
draw direction tangent at curve 1 distance 3
draw direction tangent at curve 1 fraction .5
draw direction tangent at curve 1 distance 2
reverse
```

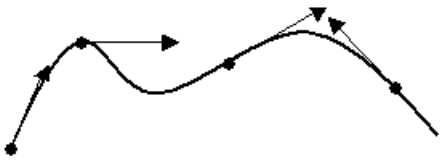


Figure 1 - Tangents to a Curve

On Surface Normal

```
[On] | [Normal] [At] Surface <id> [Location
{options}]
```

The surface option simply finds a normal vector on a surface. Note that the "on", "normal" and "at" keywords are optional, as well as the location on the surface. If no location is specified, the normal vector at the center of the surface is found. If a location is specified, the location is projected to the surface, then the normal vector is found.

```
draw direction on surface 1
draw direction on surface 1 location 1 2 0
```

From Location

```
[From] {Location {options} | Node|Vertex <id>}
[Project] {Location {options} | [Entity]
{Node|Vertex|Curve|Surface} <id>}
```

The from location option finds a direction that is from one location to another or from a location to an entity. If the second specification is an entity, the first location is projected to the entity to find the direction.

```
draw direction from vertex 1 vertex 2
draw direction from location on curve 1
fraction .5 surface 3
```

Note that when using an entity for the second specification, the Project and Entity keywords are generally optional. However, it is sometimes necessary to remove ambiguity from the previous location specification. For example, the following will not parse correctly:

```
draw direction location on curve 1 distance 2
surface 3
```

In this case, the location on the curve is parsed as a distance 2.0 from surface 3. Instead, the desired behavior is to find the location on curve 1 as a distance of 2.0 along the curve from the start of the curve, and project it to surface 3 to find the direction. The following commands (all equivalent) achieve this behavior:

```
draw direction location on curve 1 distance 2
project surface 3
draw direction location on curve 1 distance 2
entity surface 3
draw direction location on curve 1 distance 2
project entity surface 3
```

Rotate

```
[Rotate {options}]
```

The rotate option allows you to rotate the direction about another vector. You can string together as many rotations as necessary. For example:

```
draw direction 1 0 0 rotate about z 135 rotate
about curve 1 angle 50
```

Options that can be used with rotate are as follows:

```
{Ax|X|Ay|Y|Az|Z [Angle] <angle>} | { {[About]
| Towards} Direction {options} Angle <val> }
[Rotate (options)] [Origin (location)]
```

Ax, Ay, Az (or X,Y,Z) angles can be entered in any order. The optional specification of another rotate keyword in the options indicated that multiple nested rotations are permitted.

Cross

```
[Cross [With] Direction {options}]
```

The cross option allows you to find the vector cross product of the direction with another direction.

Reverse

```
[Reverse]
```

This keyword simply reverses the direction specification.

Previewing a Direction

Sometimes it is helpful to preview a direction before using it in a command. A direction may be previewed using the Draw command. The direction options are described above. See Specifying a Location for a list of location options.

```
Draw Direction {direction_options} [Location
(location_options)]
```

Specifying a Location

Some commands require a specified location or point (such as create curve spline) for the command. A location is basically an x-y-z position in the model. The following options determine a location specification:

- [Position] <xval yval zval>
- Last
- [At] {Node|Vertex} <id_list>
- [On] Curve <id_list> [location on curve options]
- [On] Surface <id_list> [Close_To | At Location {options} | CENTER]
- [On] Plane <options> [Close_To | At Location {options}]
- Center Curve <id_list>
- Extrema {Curve|Surface|Volume|Body|Group} <range> [Direction] {options} [Direction {options}] [Direction {options}]
- Fire Ray Location {options} Direction {options} At {Body|Volume|Surface|Curve|Vertex} <ids> [Maximum Hits <val>] [Ray Radius <val>]
- Between { Location <options> Location <options>} | { Location <options> Project {Curve|Surface} <id> } [Stop] [Fraction <val>] }
- [Move [all] {<xval yval zval> | {Dx|X|Dy|Y|Dz|Z} <val> | Direction {options} Distance <val>}]
- [Swing [all] [About] Axis {options} Angle <ang>]
- Multiple Location Specification

Position (XYZ values)

```
[Position] <xval yval zval>
```

The most basic way to specify a location is to just give the xyz values of the location. In this case the following two commands both draw a location at the coordinates (1, 2, 3), as the Position keyword is optional:

```
draw location position 1 2 3
draw location 1 2 3
```

Last Location Used in a Command

```
Last
```

The last option recalls the last location used in a command. For example, if the following command is entered after the above position commands a location would be drawn at the position (1, 2, 3).

```
draw location last
```

Last locations do not carry over from CUBIT session to CUBIT session. The last location defaults to (0, 0, 0) if no location has been used during the session.

Node or Vertex

```
[At] {Node|Vertex} <id_list>
```

Referring to a node or vertex simply returns the coordinates of that node or vertex. The command can also handle multiple locations where multiple locations are needed to complete the command string. The following draws a location at the coordinates of Vertex 5:

```
draw location vertex 5
```

On a Curve

Various options are available to specify a location on a curve. See the section Specifying a Location On a Curve for details.

On a Surface

```
[On] Surface <id_list> [Close_To | At Location
{options} | CENTER]
```

If a surface is used to specify a location without other options, the geometrical center of the surface is found (the center keyword is optional - the default). Otherwise, you can specify another general location and that location is projected to the surface. For example, the following command will draw the location that is position (5,0,0) projected to surface 1:

```
draw location on surface 1 location 5 0 0
```

Any valid location options listed on this page can be used to specify the location that is projected to the surface.

On a Plane

```
[On] Plane <options> [Close_To | At Location
{options}]
```

A location can be defined at the closest point on a plane to a location. See Specifying a Plane for plane options.

Center

```
Center Curve <id_list>
```

Finds the center of an arc - an error is returned if the curve is not an arc.

Extrema

```
Extrema {Curve|Surface|Volume|Body|Group}
<range> [Direction] {options} [Direction
{options}] [Direction {options}]
```

The extrema option returns the location of the maximum value, on the specified entity or group, in the specified direction. For example, the following places a vertex on a surface at the point of maximum y-axis value.

```
create vertex location extrema surf 1
direction y
```

Fire Ray

The **fire ray** command allows a user to identify a location, or set of locations, on an object by firing a ray at the object and determining the intersections. A ray can be fired at a list of bodies, volumes, surfaces, curves, or vertices. The fire ray command is:

```
Fire Ray Location {options} Direction
{options} At
{Body|Volume|Surface|Curve|Vertex} <ids>
[Maximum Hits <val>] [Ray Radius <val>]
```

The location options are described on this page. The direction options are described under Specifying a Direction. The user can specify the maximum number of hits that he wishes to receive back from the command. If this value is omitted, the command will return all intersections found. When firing a ray at a curve, a ray radius must be used. The ray radius is the distance from the curve the ray must be to be

considered a "hit." If no ray radius is used, the geometry engine default is used.

Between

```
Between {Location <options> Location <options>
} | {Location <options> Project
{Curve|Surface} <range>} [Stop] [Fraction
<val>]}
```

The between option finds a location that is between two locations or a location and an entity. An optional fraction can be given to specify the fractional distance from the first location to the second location or entity. For example, the following will draw a location at (5, 0, 0):

```
draw location between location 0 0 0 location
10 0 0
```

The following will draw a location at (2.5, 0, 0) - 25% of the distance from (0, 0, 0) to (10, 0, 0):

```
draw location between location 0 0 0 location
10 0 0 fraction .25
```

The second item can be an entity:

```
draw location between location 0 0 0 vertex 2
draw location between location 0 0 0 surface 1
```

In the second case, location (0, 0, 0) is projected to surface 1, then the location that is between (0, 0, 0) and the projected location is found.

Of course, any valid location can be used in the command. In the following example a location at the top center of the brick is found:

```
brick x 10
draw location between location bet vert 3 vert
2
location bet vert 8 vert 5
```

The first location is between vertices 3 and 2, and the second location is between vertices 8 and 5.

Note: you can "swing" a location about an axis, "rotate" a direction about another direction, "revolve" an axis about another axis and "spin" a plane about an axis. The only reason Cubit needs to use different keywords for each entity type is because the Cubit command language does not support expressions (as in using parentheses). The keyword **stop** is also used in the location/direction/axis/plane parsing as a partial workaround to this limitation. Using this stop keyword will aid in parsing out extended location specifications. Insert a stop after the first location to let the parser know that where the specifications begin and end.

Move

```
Move [All] { <xval yval zval> |
{Dx|X|Dy|Y|Dz|Z} <val> | Direction {options}
Distance <val> }
```

Any location can be optionally moved either a xyz distance or a certain distance in a given direction. As many moves as desired can be strung together. For example, the following will return a location at (5, 0, 0):

```
draw location 0 0 0 move 5 0 0
```

These examples add another move that basically moves the location (5, 0, 0) in a direction 45 degrees up and to the right a distance of 10 (all three commands are equivalent - see sections on directions and rotations):

```
draw location 0 0 0 move 5 0 0 move
{10*sind(45)} {10*sind(45)} 0
draw location 0 0 0 move 5 0 0 move direction
1 1 0 distance 10
draw location 0 0 0 move 5 0 0 move direction
1 0 0 rotate about 0 0 1 angle 45 dist 10
```

Swing

```
Swing [All] [About] Axis {options} Angle <ang>
```

Any location can be "swung" (rotated) about an axis by a certain angle. (See the section on specifying an axis for the axis syntax). As with moves, multiple swings can be strung together. The following example rotates the location (2.5, 5, 5) thirty degrees about an axis defined by Curve 11. Note that the right-hand rule is used to determine the direction of the swing about the axis.

```
draw location 2.5 5 5 swing about axis curve
11 angle 30
```

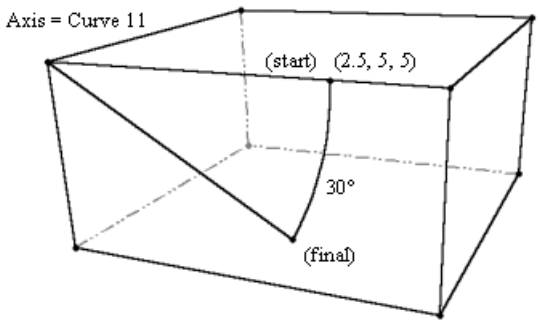


Figure 1 - Swinging a Location

Multiple Location Specification

```
Location {options} Location {options}...
```

Multiple location specifications can be used in a single command. For example, the following command uses several locations to create a spline curve at points (0,0,0), (1,2,3), (4,5,6), and (7,8,9).

```
create curve spline location 0 0 0 location 1
2 3 location 4 5 6 location 7 8 9
```

Previewing a Location

Sometimes it is advantageous to preview a location before using it in a command. A location can be previewed with the Draw command. All of the options that can be used to specify locations in a command can be used to preview locations as well. See above for a description of these options. The command syntax is:

```
Draw Location {options}
```

Specifying a Location on a Curve

Some commands require you to specify a location on a curve (i.e., webcutting with a plane normal to a curve). The following are the options for specifying a location (or locations in the case of the segment option) on a curve:

- {MIDPOINT|Start|End}
- Center
- Fraction <val 0.0 to 1.0> [From Vertex <id> | Start|End]
- Distance <val> [From {Vertex|Curve|Surface} <id> | Start | End]
- {{Close_To|At} Location {options} | Position <xval><yval><zval> | {Node|Vertex} <id>}
- Extrema [Direction] {options} [Direction {options}] [Direction {options}]
- Segment <num_segs>
- Crossing {Curve|Surface} <id_list> [Bounded|Near]}
- Previewing a Location

Center

```
center
```

The center option helps in identifying the location at the center of a given arc. Example: create vertex center curve 3

Start, Midpoint, or End

```
{ MIDPOINT | Start | End |
```

These options simply specify the location that is the midpoint, start or end point of a curve. By default, the midpoint is the understood location unless another location is specified.

Fraction

```
Fraction <val 0.0 to 1.0> [From Vertex <id> |
Start|End] |
```

The fraction option simply finds the location that is a fractional distance along the curve. By default, the fraction references the start of the curve; however, you can optionally specify which vertex to reference from.

Distance

```
Distance <d> [From {Vertex|Curve|Surface} <id>
| Start | End ] |
```

The distance option not only can find a location that is a certain distance along the curve from the start or end of the curve, but can also find a location (or locations if there is more than one solution) on a curve that is a specified distance from another curve or a surface. If the From Curve option is used both curves must lie in the same plane.

```
draw location on curve 13 distance 7 from
curve 2
```

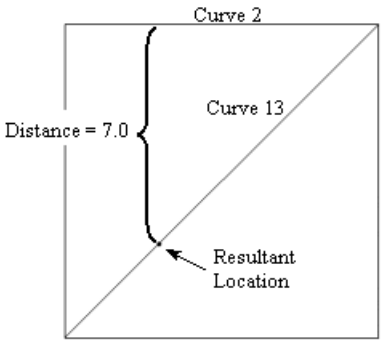


Figure 1 - Location on a Curve a Distance from Another Curve

{Close_To|At} Location

```
{{Close_To|At} Location {options} | Position
<xval><yval><zval> |{Node|Vertex} <id>} |
```

These options take a location closest to the location on the curve.

Extrema

```
Extrema [Direction] {options} [Direction
{options}] [Direction {options}]
```

The extrema option finds the maximum value location along a curve in a specified direction. For example:

```
create vertex location on curve 1 extrema ny
```

Creates a vertex on curve 1 at the location where the y axis value of the curve is at a minimum.

Segment

```
Segment <num_segs>
```

The segment option finds locations spaced evenly along the curve such as to break the curve into equal length "segments" (of course the curve is not modified). You must specify a minimum of two segments (if two segments were specified a location would be found at the center of the curve). The following example results in 4 locations:

```
draw location on curve 1 segment 5

create vertex on curve 1 segment 5
```



Figure 2 - Five Segments on a Curve

Crossing

```
Crossing {Curve|Surface} <id_list>
[Bounded|Near]}
```

The crossing option finds locations at the intersection of the curve and another curve or surface. By default, the curve(s) and surface are extended to infinity and the intersections are calculated; if the bounded option is specified only intersections that lie on the bounded entities will

be returned. The near option is valid only for two linear curves. If near is specified the nearest location between the two linear curves will be returned.

Previewing a Location on a Curve

A location on a curve can be previewed with the Draw command. All of the options that can be used for specifying a location on a curve can be used to preview a location on a curve. See above for a description of these options. The command syntax is:

```
Draw Location On Curve <curve id> {options}
```

Specifying a Plane

Some commands require a specified plane (such as sweep curve target) for the command. The following options determine a plane specification:

- {Location|Vertex|Node} <origin> Direction <normal>
- {Location|Vertex|Node} <origin> Direction <vec on plane> Direction <vec on plane>
- {Location|Vertex|Node} <2 locations> Direction <vector on the plane>
- {Location|Vertex|Node} <3 locations>
- Surface <id> [at location <loc>]
- [Normal To] Curve <id> [loc on curve options]
- Direction <Normal> Coefficient <val>
- Arc Curve <id>
- Linear Curve <id> <id>
- X|Xplane|Yz|Zy|Y|Yplane|Zx|Xz|Z|Zplane|Xy|Yx
- Last

The following options apply to all of the plane specifications listed above:

- [Offset <val>]
- [Move { <xval yval zval> | {Dx|X|Dy|Y|Dz|Z} <val> | Direction {options} [Distance <val>]]
- [[To] Location {options}]
- [Spin [About] Axis {options} Angle <ang>]]

Location and Normal Vector

```
{Location|Vertex|Node} <origin> Direction
<normal>
```

The first way to specify a plane is to specify a starting point and a direction vector:

```
draw plane location 1 2 3 direction 0 1 1
draw plane vertex 1 direction tangent at curve
1
```

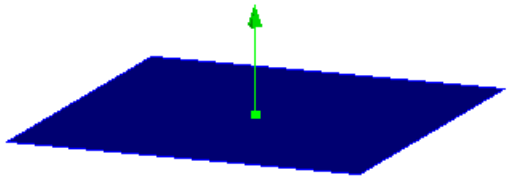


Figure 1. Specifying a plane with a location and surface normal

To see the options for location specification, see Specifying a Location. Direction options can be found at Specifying a Direction.

Location and Two Vectors on the Plane

```
{Location|Vertex|Node} <origin> Direction <vec
on plane> Direction <vec on plane>
```

It is also possible to select an origin point and 2 direction vectors on the plane.

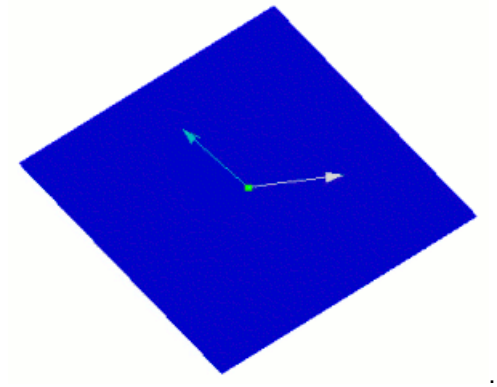


Figure 2. Specifying a plane with a point and 2 in-plane vectors

Two Locations and Vector on the Plane

```
{Location|Vertex|Node} <2 locations> Direction
<vector on the plane>
```

You can also specify 2 locations and 1 direction on the plane to define the plane.

```
draw plane vertex 1 2 direction 0 1 1
```

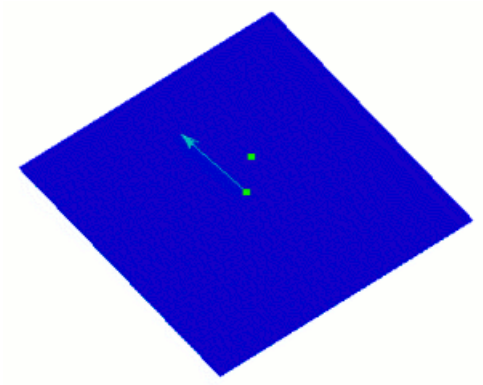


Figure 3. Specifying 2 locations and 1 direction on the plane

Three Points on the Plane

```
{Location|Vertex|Node} <3 locations>
```

A plane can be defined by three locations, vertices, or nodes. The locations are specified using Location Specification.

```
draw plane vertex 1 2 3
draw plane vertex 1 2 location 3 4 5
```

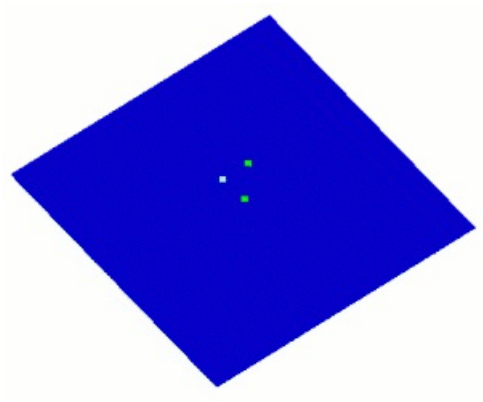


Figure 4. A plane specified by three points

Plane defined by a Surface

```
Surface <id> [At Location <loc>]
```

The surface option uses and existing surface to define the plane. If it is not a planar surface, the optional location specifier can be used to find the tangent plane of a specific point on the surface.

```
draw plane surface 1 at location 4 0 0
```

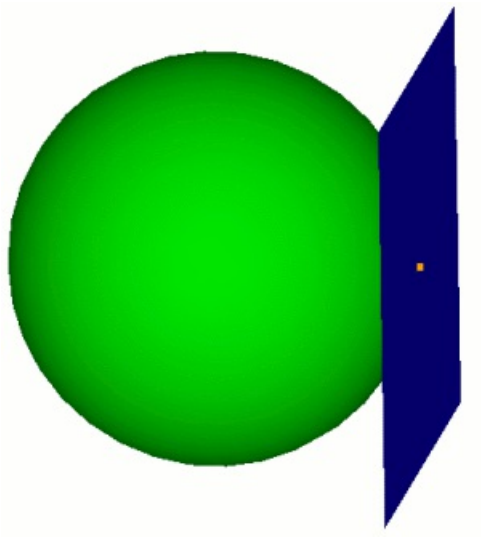


Figure 5. Specifying a Tangent plane to a Surface

Plane Normal to a Curve

```
[Normal To] Curve <id> [loc on curve options]
```

The Normal to Curve option allows you to define a plane by using an existing curve. The direction of the curve will define the surface normal of the new plane. The optional location argument specifies which point to use on the curve if it is not a straight curve. If no location is specified the plane will originate at the midpoint of the curve. See Specifying a Location on a Curve for more information on location options.

```
brick x 10
cylinder radius 3 z 12
subtract body 2 from 1
webcut body 1 xplane
draw plane normal to curve 30
```

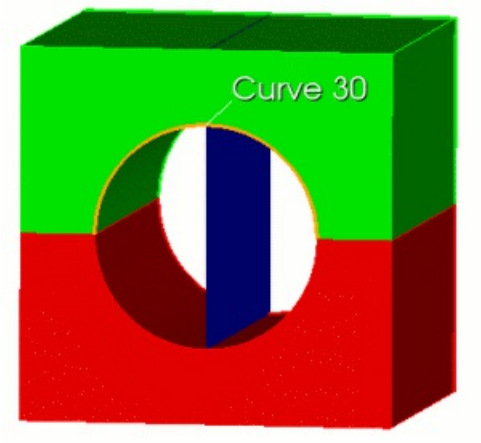


Figure 6. Draw Plane Normal to Curve

Plane Defined by a Non-linear curve

```
Arc Curve <id>
```

A plane can be defined by a single curve, provided that curve is not linear.

```
cylinder height 12 radius 3
draw plane arc curve 2
```

Plane Defined by a two linear curves

```
Linear Curve <id> <id>
```

A plane can be defined by a two linear curves, provided that the curves are not co-linear.

```
brick x 10
draw plane linear curve 2 3
```

Normal Vector and Coefficient

```
Direction <Normal> Coefficient <val>
```

The direction and coefficient option allows you to specify a plane based on a vector and an offset from the origin. The Coefficient argument specifies how far to offset the plane from the origin

```
draw plane direction 1 2 3 coefficient 3
```

Coordinate Plane

```
X|Xplane|Yz|Zy|Y|Yplane|Zx|Xz|Z|Zplane|Xy|Yx
```

A plane can be defined from any coordinate plane or combination thereof. The coordinate planes will pass through the origin unless optional specifiers are included.

```
draw plane xplane
webcut volume 1 plane xz
```

Last Location Used

```
Last
```

The last option will return the plane most recently used in a command. Last locations do not carry over from CUBIT session to CUBIT session. The last location defaults to (0, 0, 0) if no location has been used during the session.

The following options apply to all of the plane specification methods described above.

- `[Offset <val>]`
- `[Move {<xval yval zval>| {Dx|X|Dy|Y|Dz|Z} <val> | Direction {options} [Distance <val>]]`
- `[[To] Location {options}]`
- `[Spin [About] Axis {options} Angle <ang>]]`

A offset value will offset the plane in the direction of the surface normal.

The move option will displace the plane in the specified directions by the specified distance. The direction options are outlined on Specifying a Direction.

The location option will move the plane to a specified location without rotating it. See Specifying a Location for location options.

The spin option will rotate the plane around an axis. See Specifying an Axis for axis options.

Previewing a Plane

The ability to preview a plane prior to creating the plane or using it in a command is possible with the following commands:

```
Draw Plane (options) [Graphics |
{[Intersecting] {Body|Volume} <id_range>] [
[Extended] {Percentage|Absolute} <val>]}]
[Color 'color_name']
```

The options for specifying a plane are described above in the section on Plane Specification. By default, the commands draw the plane just large enough to intersect the bounding box of the entire model with minimum surface area. Optionally, you can give a list of bodies to intersect for this calculation. You can also extend the size of the surface by either a percentage distance or an absolute distance of the minimum area size. The default color is blue, but you can specify a different one. See the Appendix of the CUBIT Users Guide for available colors in CUBIT.

Preview a Cylindrical Plane

The ability to preview a cylindrical plane is possible with the following command:

```
Draw Cylinder Radius <val> Axis {x|y|z|Vertex
<id_1> Vertex <id_2> | <xyz values>} [Center
<x_val> <y_val> <z_val>] [[Intersecting] Body
<id_range>] [Extended Percentage|Absolute
<val>] [Color 'color_name']
```

The cylinder is defined by a radius and the cylinder axis. The axis is specified as a line corresponding to a coordinate axis, the normal to a specified surface, two arbitrary points, or an arbitrary point and the origin. The center point through which the cylinder axis passes can also be specified.

By default, the commands draw the cylinder just large enough to just intersect the bounding box of the entire model. Optionally, you can give a list of bodies to intersect for this calculation. You can also extend the length of the cylinder by either a percentage distance or an absolute distance of the cylinder length. The default color is blue, but you can specify a different one. See the Appendix of the CUBIT Users Guide for available colors in CUBIT.

Listing Information

The **List** commands print information about the current model and session. There are five general areas: *Model Summary, Geometry, Mesh, Special Entities,* and *CUBIT Environment*. The descriptions of these areas includes example output based on the model generated by a journal file listed below. The model consists of a 1x2x3 brick meshed with element size 0.1.

- List Model Summary
- List Geometry
- List Mesh
- List Special Entities
- List CUBIT Environment

Journal File Used for List Examples

```
brick x 1 y 2 z 3
body 1 size 0.1
mesh volume 1
block 1 volume 1
nodeset 1 surface 1
sideset 1 surface 2
group "my_surfaces" add surface 1 to 3
surface 2 name "BackSurface"
surface 3 name "BottomSurface"
surface 1 name "FrontSurface"
surface 4 name "LeftSurface"
surface 5 name "RightSurface"
surface 6 name "TopSurface"
```

List Cubit Environment

The user may list information about the current CUBIT environment such as message output settings, memory usage, and graphics settings.

Message Output Settings

There are several major categories of CUBIT messages.

- **Info** (Information) messages tell the user about normal events, such as the id of a newly created body, or the completion of a meshing algorithm.
- **Warning** messages signal unusual events that are potential problems.
- **Error** messages signal either user error, such as syntax errors, or the failure of some operation, such as the failure to mesh a surface.
- **Echo** messages tell the user what was journaled.
- **Debug** messages tell developers about algorithm progress. There are many types of Debug messages, each one concentrating on a different aspect of CUBIT.

By default, Info, Warning, Error, and Echo messages are printed, and Debug messages are not printed. Information, Error, Warning, Debug, and Echo message printing can be turned on or off (or toggled) with a set command. Debugging output can also be redirected to a file. Current message printing settings can be listed.

```
List {Echo|Info|Errors|Warning|Debug}

Set Echo [On|Off]

Set {Info|Warning} [On|Off] [logging]

Set Error [On|Off]

[Set] Debug <index> [On|Off]

[Set] Debug <index> File <'filename'>

[Set] Debug <index> Terminal
```

Message flags can also be set using command line options:

```
-warning {on|off}

-information {on|off}
```

Debug flags can be enabled from the command line with

```
-debug <setting>
```

where **<setting>** is a comma-separated list of integers or ranges of integers denoting which flags to turn on. E.g., to set debug flags 1, 3, and 8 to 10 on, the syntax is **-debug 1,3,8-10**.

Logging Output to a File

Output from CUBIT can be redirected to a log file, and the current state of logging can be listed.

```
[Set] Logging {Off|On File <'filename'>
[Resume]}

List Logging
```

If logging is enabled, by default any output to the console or command window will also go into the logging file. The **resume** option will append to the logfile, if it exists, instead of emptying the file. If the logfile doesn't already exist, it will be created.

Output of information and warning messages to the logging file can be controlled independent of console output settings by adding the **logging** option to the **set {info|warning} [on|off] logging** command.

Default Block Creation

```
Set Default Block
{ON|off|Volume|Surface|Curve]}

List Default Block
```

The **set Default Block** command will toggle whether or not default blocks are written during the export operation if no other blocks have been specified. The **List Default Block** command lists the geometric entity types for which blocks will automatically be generated at export.

Journaling Settings

```
List Journal
```

The **List Journal** command lists which types of CUBIT commands will be journaled and the file to which the journaled commands are being written.

Exodus Export Title

```
List Title

Title "<title_string>"
```

The **List Title** command will list the title to be written to an Exodus file on export. To assign a title to an Exodus file, use the **Title** command.

Listing Current Settings

```
List Settings
```

The **List Settings** command lists the value of all the message flags, journal file and echo settings, as well as additional information. The first section lists a short description of each debug flag and its current setting. Other message settings are listed next, followed by some flags affecting algorithm behavior.

Sample output

```
CUBIT> list settings
Debug Flag Settings (flag number, setting, output to, description):
 1  OFF  terminal           Debug Graphics toggle for some debug options.
 2  OFF  terminal           Whisker weaving information
 3  OFF  terminal           Timing information for 3D Meshing routines.
 4  OFF  terminal           Graphics Debugging (DrawingTool)
 5  OFF  terminal           FastQ debugging
 6  OFF  terminal           Submapping graphics debugging
 7  OFF  terminal           Knife progress whisker weaving information
 8  OFF  terminal           Mapping Face debug / Linear Programming debug
 9  OFF  terminal           Paver Debugging
.
.
.
echo              = On
info              = On
journal           = On
journal graphics  = Off
journal names     = On
journal aprepro   = On
journal file      = 'cubit11.jou'
```

```
warning           = On

logging           = Off
recording         = Off
keep invalid mesh = Off
default names     = Off
default block    = Volumes
catch interrupt   = On
name replacement character = '_', suffix character = '@'
Matching Intervals is fast, TRUE;
multiple curves will be fixed per iteration.
Note in rare cases 'slow', FALSE, may produce better meshes.
Match Intervals rounding is FALSE;
intervals will be rounded towards the user-specified intervals.
```

Graphical Display Information

```
List View
```

List view prints the current graphics view and mode parameters; See Graphics Window .

Memory Usage Information

Users are encouraged to use Unix commands such as `top' to check total CUBIT memory use. Developers may check internal memory usage with the following command:

```
List Memory [`<object type>']
```

Without an object type, the command prints memory use for all types of objects.

List Geometry

The following commands list information about the geometry of the model.

> **List Names [Group|Body|Volume|Surface|Curve|Vertex|All]**
>
> **List {Group|Body|Volume|Surface|Curve|Vertex} <range> [Ids]**
>
> **List {geom_list} [Geometry|Mesh [Detail]]**
>
> **List {Group|Body|Volume|Surface|Curve|Vertex} <range> {X|Y|Z}**

The first command lists the names in use, and the entity type and id corresponding to each name. Specifying **all** lists names for all types; other options list names for a specific entity type. The names for an individual entity can be obtained by listing just that entity. Sample output from the list names surface command is shown below. This output shows that, for example, Surface 2 has the name ` BackSurface '.

```
______Name______    __Type__    Id      _Propagated_
     BackSurface    Surface     2            No
   BottomSurface    Surface     3            No
    FrontSurface    Surface     1            No
     LeftSurface    Surface     4            No
    RightSurface    Surface     5            No
      TopSurface    Surface     6            No
```

List Names Example

The second command provides information on the number of entities in the model and their identification numbers. If a range is given then detailed information is given on each entity in that range, unless the **ids** option is also given. If the **ids** option is used, just a list of ids is printed. This list can be very useful for large models in which several geometry decomposition operations have performed. Sample output from the list surface command is shown below.

```
CUBIT> list surface ids
The 6 surface ids are 1 to 6.

CUBIT> list surf ids
The 108 surface ids are 192 to 266, 268 to 271, 273
to 301.
```

List Surface [range] Ids' Examples

The **<range>** can be very general using the general entity parsing syntax. Using a **<range>** gives a brief synopsis of the local connectivity of the model, e.g. one can list the ids of the surfaces containing vertex 2; as shown in the listing below.. An intermediately detailed synopsis can be obtained by placing the range of entities in a group, then listing the group.

```
CUBIT> list surface in vertex 2 ids
The 3 entity ids are 1, 5, 6.

CUBIT> group "v2_surfs" equals surface in vertex 2
CUBIT> list v2_surfs Group Entity 'v2_surfs' (Id =
3)
It owns/encloses 3 entities: 3 surfaces.
Owned Entities:              Mesh Scheme   Interval:
  Edge
_____Name____    Type_____Id +is meshed    Count
      Size
FrontSurface   Surface    1         map+      1 H
      0.1
```

```
  TopSurface    Surface     6          map+       1 H
       0.1
RightSurface    Surface     5          map+       1 H
       0.1
```

Using 'List' for Querying Connectivity.

The third command provides detailed information for each of the specific entities. This information includes the entity's name and id, its meshing scheme and how that scheme was selected, whether it is meshed and other meshing parameters such as smooth scheme, interval size and count. The entity's connectivity is summarized by a table of the entity's subentities and a list of the entity's superentities. Also, the nodesets, sidesets, blocks, and groups containing the entity are listed.

Specifying **geometry** will additionally list the extent of the entity's geometric bounding box, the geometric size of the entity, and depending on entity type, other information such as surface normal. See also the **list {entities} x** command below. If multiple volumes, surfaces, or curves are selected, it will list the total volume, area, or length of all entities, and the total geometric bounding box. If multiple volumes are selected, the centroid listed will be the composite centroid of the all of the volumes.

If a volume is recognized as a primitive (cylinder, brick, etc.), specifying **geometry** will list data for defining the primitive.

- Cylnder data is RCC: followed by 7 doubles: x,y,z coordinates of bottom center, x,y,z components for vector from bottom center to top center, and radius.
- Brick data is BOX: followed by 12 doubles: x,y,z coordinates of first corner, and 3 sets of x,y,z components for vectors from the first corner to each adjacent corner of the brick.

Specifying **mesh** will additionally list the number of mesh entities of each type interior to the entity and on bounding subentities. **Mesh detail** will list the ids of the mesh entities as well, following the format of the **list ids** command above.

The fourth command lists the entities sorted by either the x, y, or z coordinate of their geometric center. For example, in a large, basically cylindrical model centered around z-axis, it is useful to list the surfaces of a volume sorted by z to identify the source and target sweeping surfaces.

List Mesh

The following commands list mesh entity information.

```
List {Hex|Face|Edge|Node} <id_range>

List {Hex|Face|Edge|Node} <id_range> IDs
```

For both of these commands, the range can be very general, following the general entity parsing syntax. The first command provides detailed information. For an entity, the information includes its id, owning geometry, subentities and superentities. For a hex, the Exodus Id is also listed. For a node, its coordinates are listed. The second command just lists the entity ids, and is usually used in conjunction with complex ranges.

List Model Summary

The following commands print identical summaries of the model: the number of entities of each geometric, mesh, and special type

```
List Model

List Totals
```

The following output is generated from the **list model** command.

```
CUBIT> list model

Model Entity Totals:
      Geometric Entities:
             0 assemblies
             0 parts
             2 groups
             1 bodies
             1 volumes
             6 surfaces
            12 curves
             8 vertices
      Mesh Entities:
          6000 hexes
             0 pyramids
             0 tets
          7876 faces
             0 tris
          9854 edges
          7161 nodes
      Special Entities:
             1 element blocks
             1 sidesets
             1 nodesets

Journaled Command: list model
```

List Special Entities

```
List {special_type} <range>
```

Special entities include (element) blocks, sidesets and nodesets (representing boundary conditions). Like the **list geometry** and **list mesh** commands, if no range is specified then the number of entities of the given type is summarized. Otherwise, listing a special entity prints the mesh and geometry it contains.

(Some special entities are of interest mainly to developers and are not described here, e.g. whisker sheets, and whisker hexes.)

Graphical User Interface

- CUBIT Application Window
- Control Panel
- Graphics Window
- Model Tree
- Power Tools
- Property Editor
- Command Line Workspace
- Journal File Editor
- Toolbars
- Drop-Down Menus

The graphical user interface (GUI) can improve user productivity. It provides an easy way to control CUBIT without learning command syntax. Many geometry commands are faster and easier with the GUI. The underlying GUI components are constructed using a cross-platform development environment. As such, the GUI will behave similarly across all platforms supported by Cubit, yet each GUI will make use of platform specific widgets.

The GUI is built on top of the CUBIT command line. This means that GUI actions are translated to a CUBIT command-line string and journaled. Users familiar with command-line syntax can enter the same text in the GUI command-line window. Journal files can be created and played back in both environments with the same results. Although many things are faster and easier in the GUI, experienced users often use a combination of command line text and GUI button operations.

The discussion of the Graphical User Interface and its features is based on the basic windows contained within the CUBIT GUI Application Window. These are outlined in the subtopics listed above.

A full graphical user interface (GUI) with the standard look and feel consistent with major platforms is available on all supported Cubit platforms. The GUI version can improve productivity, making new users aware of the wide range of CUBIT capabilities, and freeing new and experienced users from having to remember esoteric syntax. The GUI and non-GUI versions create and play back identical journal files, making it easier to switch from one environment to the other.

Model Tree

The model tree provides a complete graphical hierarchical representation of the parent child relationship of all geometric entities. The tree is populated as the model is constructed by Cubit. In addition to showing a hierarchy of geometric entities, the tree also shows active Groups, Boundary Layers, and active Boundary Condition entities.

The tree works directly with the graphics window and picking. Selecting an entity in the tree will select the same entity in the graphics window. Selecting an entity in the graphics window will highlight the tree entry if that entry is currently visible. If an entity's visibility is turned off, the icon next to that entity in the geometry tree will disappear.

If the tree entry is not visible the user may press the Find button located at the bottom of the tree. The first occurrence of the selected entity will be shown on the tree.

Virtual entities have a small (v) after the name to indicate that they are virtual entities.

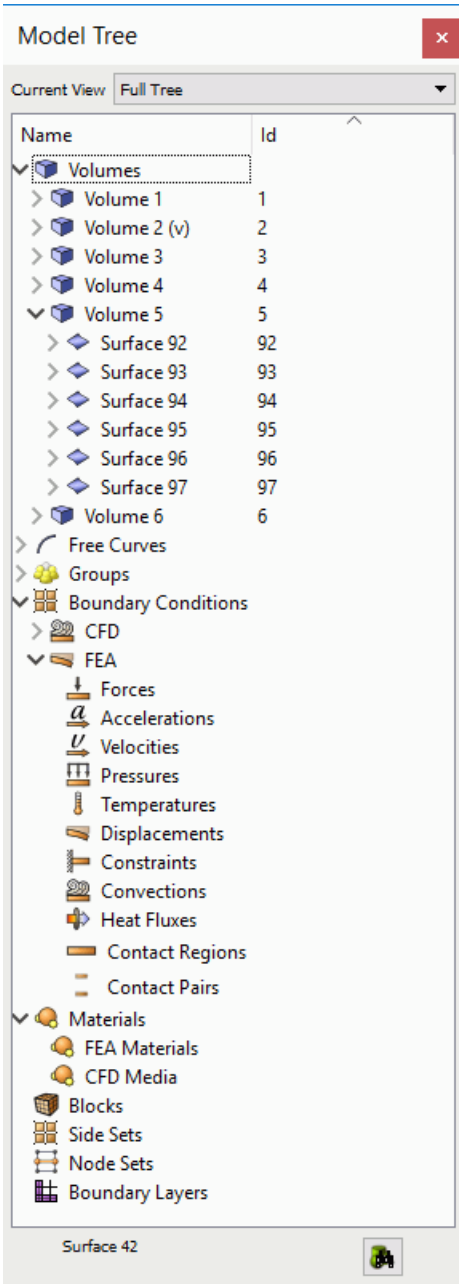


Figure 1. Geometry Tree Window

Drag and Drop

The Tree View window supports drag and drop of geometric entities into existing boundary condition sets. To create boundary conditions, see the Materials and Properties menu on the main control panel, or right-click on one of the boundary condition labels and select the "Create New" option from the context menu. Geometric entities or groups can be added to blocks, nodesets, or sidesets by dragging and dropping inside the tree view window.

Picked Group

The current selections in the graphics window can be added to a "picked group" by selecting the "Add to Picked Group" from the Right click menu. Selections can also be added to the picked group by dragging and dropping onto the group from the geometry tree window. The picked group can be substituted into any commands that use groups. To remove an item from the picked group, use the "Remove from Group" option in the right click menu in the geometry tree or from the graphics window.

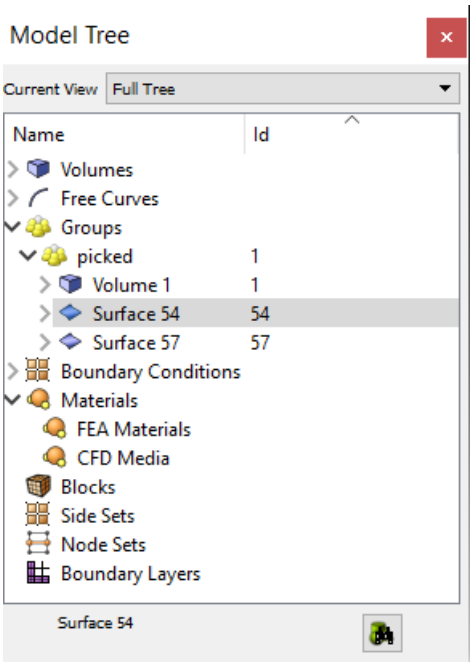


Figure 2. Picked Group

Right-Click Menu Functions

The geometry tree's context menu is sensitive to the type of item and the number of items selected. Functions that apply to the item type and number of selected items are available from the context menu. These include the following:

- **Zoom To** - Available for all geometric entities
- **Rotate About** - Change the center of rotation to the centroid of the entity without zooming
- **Fly-In** - Animated zoom feature
- **Locate** - Labels the selected entity in the graphics window
- **Draw** - Draw this entity by itself.
- **Isolate** - Similar to Draw command, but the display will not be refreshed with a graphics reset. To redisplay the model, select All Visible from the graphics window right-click menu.
- **Transparency On/Off** - Toggles transparency mode
- **Visibility On/Off** - Toggles visibility

- **Rename** - Allows you to rename entities from the tree. Clicking on a highlighted entity in the tree will do the same thing. This will also work for boundary condition entities (blocks, nodesets and sidesets)
- **Mesh** - Mesh selected entity at current settings.
- **Delete Mesh** - Available for meshed entities
- **Reset Entity** - Deletes mesh, and returns all settings to default values.
- **Delete** - Available when Volumes and Groups are selected.
- **Measure** - Available when two entities are selected or 1 curve is selected
- **Refresh Full Tree** - Used to return to main tree
- **Collapse Tree** - Available when entities are selected.
- **View Descendants/Ancestors** - Show this entity's individual hierarchy. Use the Refresh Full Tree option to return to main tree view.
- **View Neighbors** View adjacent entities. Use the Refresh Full Tree option to return to the main tree view.
- **Create New Volume** - Available when the user right-clicks over the Volumes (parent) label. Opens the geometry-volume-create panel
- **Import Geometry** - Available when the user right-clicks over the Volumes (parent) label. Opens import dialog.
- **Create New Group** - Available when the user right-clicks over the Groups (parent) label.
- **Clean Out Group** - Available when groups are selected. Removes all entities from group.
- **Remove from Group** - Available when groups are selected. Removes selected entity from the group.
- **Add Selected to Block/Nodeset/Sideset** - Add the selected entity in the graphics window to the chosen block, nodeset, or sideset in the geometry tree.
- **Delete Selected from Block/Nodeset/Sideset** - Delete the selected entity in the graphics window from the chosen block, nodeset, or sideset in the geometry tree.
- **Create New Block/Sideset/Nodeset** - Available when the user right-clicks over the respective Boundary Conditions (parent) label.
- **Create New <boundary condition>** - Available when highlighting desired boundary condition in the tree including CFD and FEA boundary conditions.
- **Draw Block/Sideset/Nodeset** - Draws the selected block/nodeset/sideset on top of existing entities
- **Draw Sideset/Nodeset Only** - Draws the selected nodeset/sideset independent of other entities
- **Delete Selected Boundary Condition** - Deletes any selected boundary conditions
- **Draw Selected Boundary Condition** - Draws selected boundary condition by itself
- **Draw Selected Boundary Condition (Add)** - Draws multiple boundary conditions
- **List Selected Boundary Condition** - Lists information about selected boundary conditions in the command line window
- **Remove from Block/Sideset/Nodeset** - Removes selected entity from the specified block, sideset or nodeset
- **Cleanup (Tets)** - Issues cleanup command for selected block. Only applicable for blocks composed of tet elements
- **Remesh (Tets)** - Issues remesh command for selected block. Only applicable for blocks composed of tet elements
- **List Info** - List information about selected entity in the output window.

Power Tools

The power tools contain useful tools to help users through the mesh generation process. The Immersive Topology Environment for Meshing, also known as ITEM. This panel contains a wizard-like environment which guides the user through the mesh generation process through a series of panels and diagnostics. The geometry repair and analysis tools contains diagnostics and tools for analyzing and repairing geometry, although many of these can now be found in the ITEM environment as well. The mesh quality and meshing power tools aid in mesh generation and verification. The geometry and mesh comparison tool identifies correlation between existing geometry and mesh. The defeaturing tool assists users with defeaturing geometry in a more automated fashion. The assemblies tool help users manage assemblies, parts and related metadata.

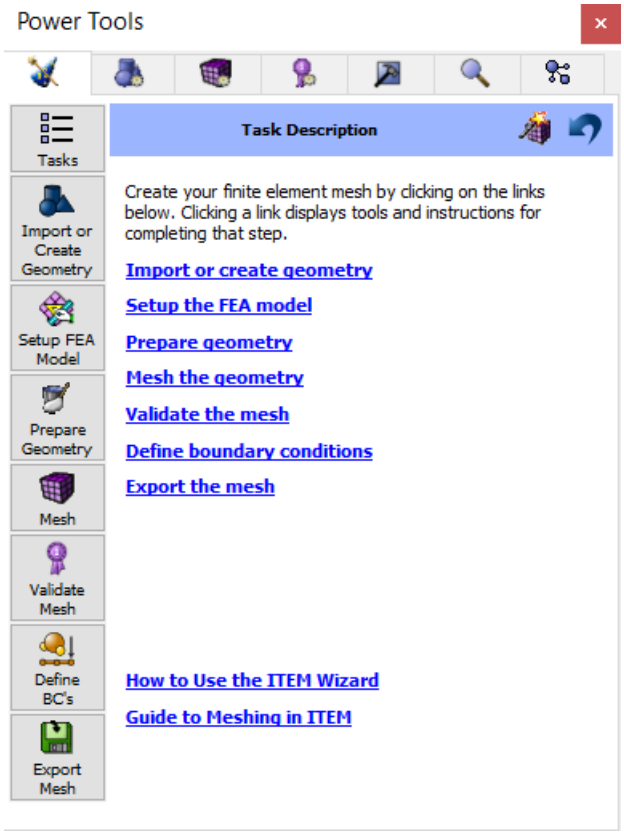


Figure 1. Power Tools Window

- Immersive Topology Environment for Meshing (ITEM)
- Geometry Analysis and Repair Tools
  - Machine Learning Tools
- Meshing Tools
- Mesh Quality Tools
- Defeaturing Tool
- Geometry/Mesh Comparison Tool
- Assemblies Tool

To familiarize yourself with the power tools environment (excluding ITEM), we recommend that you try the power tools tutorial.

To familiarize yourself with ITEM wizard, we recommend that you try the ITEM tutorial.

Geometry Power Tools

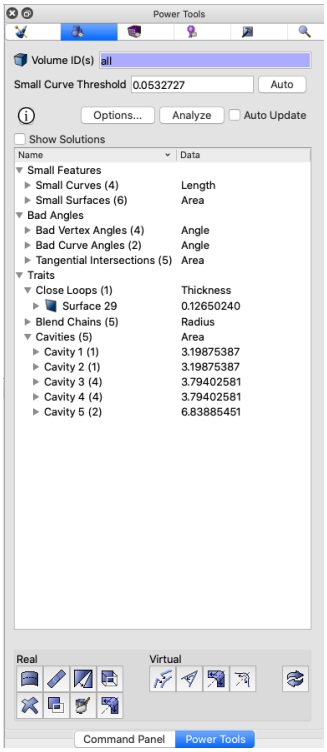


Figure 1. Geometry power tools panel

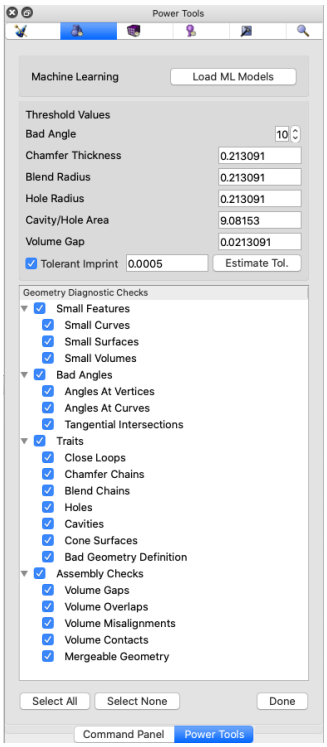


Figure 2. Geometry power tools options panel

The geometry power tools, shown in Figure 1. are located on the Tree View window under the blue geometry tab. The Geometry Power Tool provides several diagnostic tests to identify and repair problems in your CAD model prior to meshing including machine learning-based diagnostics and solutions.

Diagnostic tests include:

- Small Features: Small curves and surfaces and volumes
- Bad Angles: Near tangent angles at curves and vertices
- Geometric Traits: Identifies common geometric traits such as holes, blends, chamfers, etc.
- Assembly Checks: Gaps, overlaps and misalignments between multiple parts
- Beams and Shells: Tools for beam and shell modeling
- Tetmesh Poor Quality Predictions: Machine learning tool to predict poor quality tet mesh
- Part Classification: Machine learning tool to classify parts into common mechanisms

This tool analyzes geometry for various characteristics that may affect meshing outcomes and aid in simplification and defeaturing. It also contains a powerful toolkit of geometry modification methods to fix these problems. Many of the common geometry clean-up tools are available from this tool without the need to search through the command panels for relevant operations.

The geometry power tool includes a window that lists results from geometry analysis in a tree format. In addition, a solution window can be displayed that will display specific suggested geometry solutions for the currently selected entity.

Suggested Usage

The following is a suggested workflow for using the geometry power tool:

1. **Enter volumes to analyze:** Enter or pick the volume IDs you wish to analyze in the field labeled **Volume ID(s)**. By default, all volumes will be analyzed. For large or complex assemblies, consider selecting only a few volumes at a time to avoid long analysis times.
2. **Enter a small curve threshold:** The value entered in the field labeled **Small Curve Threshold** defines the basis for what is considered "small" for most geometry tests. If Cubit already has more than one volume defined, a default value for small curve threshold will be computed as **0.25*mesh_size**. To update the default **small curve threshold** for the current volumes, select the **Auto** button. If no mesh size is currently defined, an autosize factor of 2.5 will be used to compute a mesh size. (Equivalent to **vol all size auto factor 2.5**)
3. **Select diagnostics to perform:** Selecting the **Options...** button will display a list of available diagnostics grouped by category, as shown in Figure 2. By default all diagnostics are selected. Some diagnostics may not apply to specific geometry, or may only need to be run once per geometry. To avoid long analysis times, select only diagnostics that are relevant for your current problem scope. Clicking on the box by each test will select or deselect it. Categories of diagnostics may also be selected or deselected in a similar manner. All diagnostics may be selected or deseleted using the **Select All** and **Select None** buttons at the bottom of the panel. Threshold values used for some of the diagnostics can also be entered, including bad angle, chamfer thickness, blend or hole radius, cavity area and volume gap thresholds. Details on each of the diagnostics are described below. Select the **Done** button to return to the main Geometry power tool panel.
4. **Analyze the geometry:** Click the **Analyze** button to initiate an analysis of the selected diagnostics. The time taken for analysis will vary based on the number and complexity of volumes and the diagnostics selected.
5. **Select an entity to examine:** Once analysis is complete, the results will appear in the main window of the geometry power tool panel in the form of an expandable lists categorized by the selected diagnostics. Items in the list correspond to the selected tests. Expanding a list will display an ordered sub-list of geometry entities that have been identified by the test. Selecting one or more entities in one of the lists will also highlight the entities in the graphics window. Use shift-click or command/ctrl-click to select multiple entities in the list. Use the context menu (right click) to zoom or fly in, locate, draw or other methods to graphically examine the selected entities.
6. **Choose a geometry repair solution:** Multiple methods are provided for choosing and selecting a relevant geometry repair solution:
    - **Context Menu:** Right clicking on an entity in the list will reveal a list of options that are normally relevant for the selected entity type. (See Figure 3.) For example, selecting the **Remove Surface...** menu item will bring up the Remove Surface command panel pre-populated with the relevant entity. To execute the same operation on many entities at once, first select all relevant entities in the list.
    - **Show Solutions:** Selecting the **Show Solutions** check box at the top of the results window will display an additional window, (See Figure 4.) populated with relevant operations for the currently selected entity. Selecting a solution will display a preview of the operation in the graphics window. Double clicking the solution will execute the solution. A right click on the solution will show a context menu revealing the following options:
        - **Execute:** Execute the selected solution (same as double click).

- **Show More Solutions:** Add additional solutions computed for attached entities if they exist. For example, if a small curve is selected, this option will include additional solutions in the window for its attached surfaces and vertices.
- **Open Command Panel Operation:** Depending on the type of solution selected, the relevant command panel will appear pre-populated with the options called for in the solution. This provides the option to further customize the solution if the precise desired command is not displayed.

- **Command Panel Buttons:** The buttons at the bottom of the geometry power tool will display a specific geometry command panel. This can be useful if many similar operations are to be performed on different entities. A description of each is provided below.

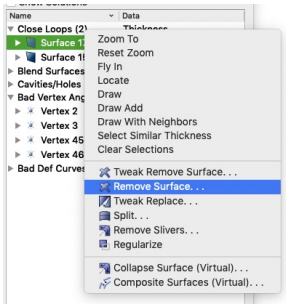


Figure 3. Geometry entity context menu in power tool.

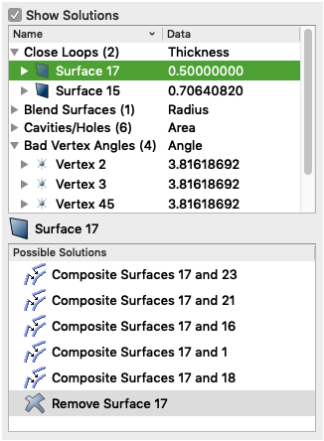


Figure 4. Entitiy-specific solutions displayed in geometry power tool.

Geometry Analysis Tools

The geometry power tools, contain various diagnostic tests that can be run on geometry to diagnose potential problems for mesh generation and defeaturing. To display a list of tests, click on the **Options...** button. The panel shown in Figure 2. will appear. Select or deselect the desired options from the window before performing an analysis. To avoid long analysis times, select only tests that are relevant for your current problem scope. Cubit will also save the current test selections between runs. The geometry analysis tests are summarized below:

Small Features

Small features may be necessary and desirable in a model, but many times they are the result of poor geometry construction, or they may just not be important to the analysis. The small features tests look for small curves, small surfaces, and small volumes. These tests rely on the user-defined **small curve threshold** value defined at the top of the Geometry power tool.

- **Small Curves** - Small curves, including zero-length curves such as hardpoints, are compared directly against the **small curve threshold** value, and identified if they are less than or equal to the given value.
- **Small Surfaces** - Small surfaces are identified based on area and hydraulic radius. Surface areas that are less than the square of the

current **mesh_size** are identified as small. For surfaces where the hydraulic radius, defined as **4*surface_area/perimeter**, is less than the **small curve threshold** are also identified as small.
- **Small Volumes** - Small volumes are identified by their hydraulic radius, defined as **6*volume/surface_area**.

Bad Angles

Small geometric angles at vertices and curves can sometimes over-constrain the resulting mesh resulting in poor element quality. These tests are controlled by the **Bad Angle** threshold value defined at the top of the Geometry power tool Options panel.

- **At Vertices** - For vertices, the angle formed by two attached curves is measured. Vertices where angles less than the **Bad Angle** threshold. Figure 5 shows an example of a bad angle where the resulting tet or hex mesh may result in poor mesh quality. Note that depending on how the angle is measured at the vertex, the small angle may be also be identified if **360.0-angle** is less than the **Bad Angle** threshold. The **blunt tangency** or **collapse angle** operations are useful for removing bad angles at curves.

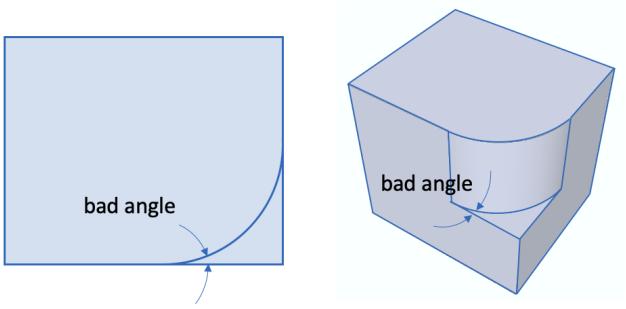


Figure 5. Bad Angle at Vertex Example

- **At Curves** - For curves, the angle formed by two attached surfaces is measured. Similar to vertices, curves where angles less than the **Bad Angle** threshold are identified.
- **Tangential Intersections** - A tangential intersection is formed when two parallel surfaces share an edge and have a 180 degree angle between them. The tangential intersection test is looking for the condition where two surfaces that meet tangentially share a common edge, and each of the surfaces has another edge which resides on a third face and forms a small angle. In the example shown in Figure 6., Surface 1 and Surface 2 are tangential to each other and share a common edge. Both Surface 1 and 2 have another edge which resides on Surface 3 and forms a small angle at the vertex common to all three surfaces.

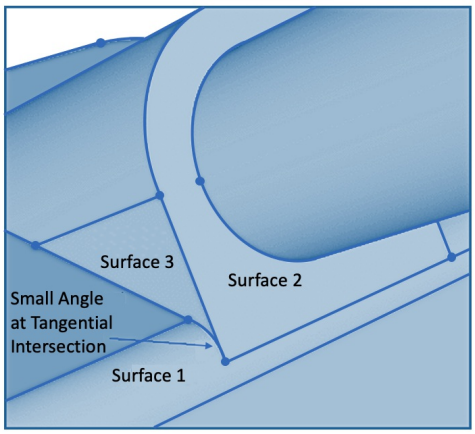


Figure 6. Tangential Intersection Example

Traits

The tests in the **Traits** category, group entities according to a specific characteristic of the geometry such as its thickness or radius. Use the threshold values at the top of the Geometry power tools Options panel to set limits on values used to control entities returned from these tests. Geometry Traits include the following:

- **Close Loops** - Close loops are identified by two curves on a single surface for which the shortest distance between them is less than the current **mesh_size**. Surfaces identified as close loops are ordered based on the minimum thickness of the surface between the loops. These surfaces and their immediate neighbors are often candidates for the **remove surface** command.

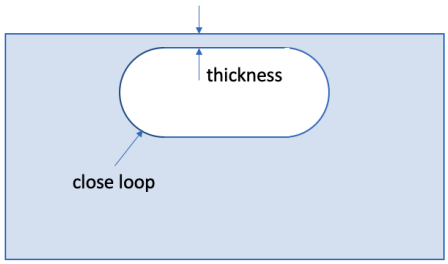


Figure 7. Close Loop Example

- **Chamfer Chains** - A chamfer surface can be identified as a narrow strip where its angle to neighboring surfaces is about 45 degrees as shown in Figure 7. Chamfers often occur as a chain or connected set of surfaces and are grouped together in the power tool as a collection of surfaces that can be expanded and examined individually. Chamfer chains are ordered based on the narrow thickness of the surfaces illustrated in Figure 8. Setting the **Chamfer Thickness** threshold in the Options panel will control which chamfer chains will be identified. The default value for **Chamfer Thickness** threshold is the current **mesh_size** Since chamfers with small thickness can effect the resulting size of the elements the **remove surface** option is often used to eliminate them.

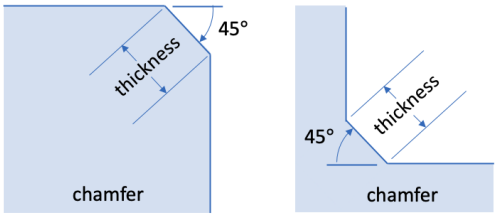


Figure 8. Chamfer Examples

- **Blend Chains** - A blend surface serves as a smooth transition between two neighboring surfaces, such as a fillet as shown in Figure 8. Blends are identified as surfaces having a constant radius along one of its parametric directions. Blends often occur as a chain or connected set of surfaces and are displayed as a collection of surfaces in the power tool that can be expanded and examined individually. Enter a **Blend Radius** threshold value at the top of Geometry power tools options panel to control the maximum radius of curvature for surfaces returned from this test. The default value for **Blend Radius** threshold is the current **mesh_size**. Resulting blend surfaces are ordered based upon their minimum radius of curvature. Blend chains can be candidates for the **remove surface blend_chain** or **split surface** commands.

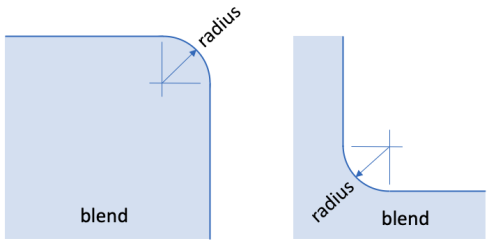


Figure 9. Blend Examples

- **Holes** - Holes are a special category of **Cavity** (see below). They are collections of surfaces that are bounded by curves where the exterior angle is greater than 180 degrees and at least one of the surfaces have a radius of curvature less than the **Hole Radius** threshold. Figure 10 illustrates a hole that is comprised of a cylindrical surface and a planar circular surface. Resulting hole collections of surfaces are ordered based upon their cylindrical radius. Holes can be candidates for the **remove surface cavity** command.

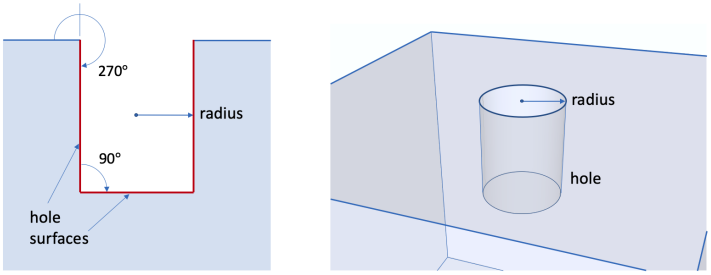


Figure 10. Hole Example

- **Cavities** - Small cavities in a volume may be candidates for removal from the geometry. A cavity is defined as a collection of surfaces bounded by curves with an external angle greater than 180 degrees. Enter the **Cavity Area** threshold value at the top of the Geometry power tools Options panel. This value controls the maximum total surface area for a cavity identified from this diagnostic test. Since cavities may consist of many individual surfaces, the resulting ordered list displayed in the power tool includes sub-lists of surfaces that can be expanded and examined individually. Surfaces contained with cavities or holes can be candidates for the **remove surface cavity** command which will remove all surfaces in the cavity sumiluatneously.

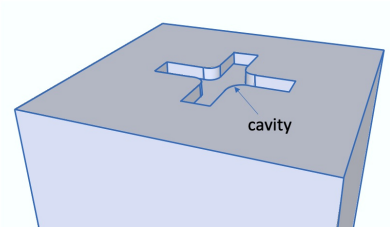


Figure 11. Cavity Example

- **Cone Surfaces** - Cones are defined as any surface comprising exactly two curves where one of the curves is of zero length. Cone surfaces can often cause difficulty for surface meshing, and should be removed when possible. Surfaces identified as cones are ordered based on their surface area. Cone surfaces are good candidates for the **tweak surface cone** command.

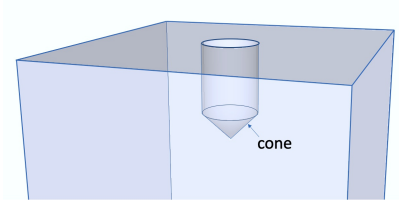


Figure 12. Cone Example

- **Bad Geometry Definition** - Cubit uses third party libraries, such as ACIS from Spatial, Inc. for much of its geometric modeling capabilities. The bad geometry definition check calls internal validation routines in these libraries, when available, to check for errors in geometry definition. Entities indetified as "bad geometry" are usually candidates for the **heal volume** command. If the third party library does not provide validation capabilities, this check will not return anything. Note: ACIS is a trademark of Spatial.

Assembly Checks

Check the interactions between multiple volumes. Here we check for overlaps, gaps and misalignments between nearby volumes. It will also identify volumes that are in contact as well as entities that are ready for merging.

For assemblies of volumes, it is important to identify if volumes will be connected (imprinted and merged) are in contact, or separated by some distance. The **Assembly Checks** provide diagnostics and solutions to validate and resolve these interactions.

The **Gaps**, **Overlaps** and **Misalignments** diagnostics normally identify undesirable conditions that must be resolved prior to **imprint** and **merge**. Once resolved, the **Volume Contacts** and **Mergable Geometry** can be used to validate connections before and after imprinting and merging.

The Options panel also provides a way to estimate or manually set an imprint tolerance. Entities closer than this tolerance will be considered mergable when used with the tolerant imprint command. When the **Tolerant Imprint** checkbox is selected in the Options panel, the diagnostic tests that identify gaps, overlaps and misalignments will also use the specified tolerance when computing issues.

- **Volume Gaps** - Lists volume pairs that are separated by a distance smaller than the **Volume Gap** tolerance specified in the Options panel, but are not in contact or overlapping. Gaps can result in parts that are not correctly merged and will not share nodes between volumes when meshed. Expanding a volume pair in the list will display individual surface pairs where gaps exist between the volumes. Figure 13 illustrates a gap between two volumes. Gaps can be visualized using the **Draw Volume Gap** context menu, also shown in Figure 13, where the surfaces that are within the gap tolerance are displayed in red. The **tweak surface replace** command can sometimes be used to correct overlaps.

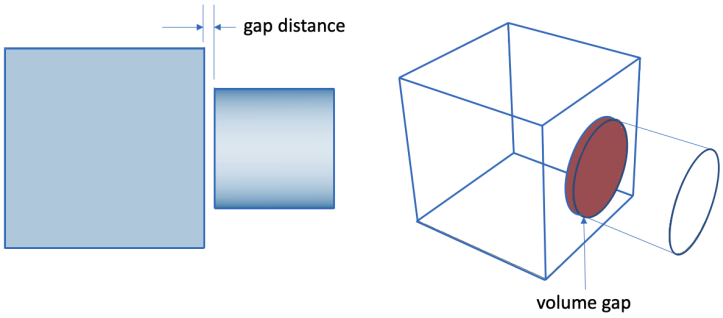


Figure 13. Volume Gap Example

- **Volume Overlaps** - Lists volume pairs that are overlapping. Figure 14. shows an example of a volume overlap. Overlapping volumes can result in sliver surfaces and bad element quality if they are not resolved prior to imprinting and merging. Overlaps can be displayed with the context menu item, **Draw Volume Overlap** which displays the overlap region in red. The **remove overlap** command or **tweak surface replace** commands can often be used to correct overlaps.

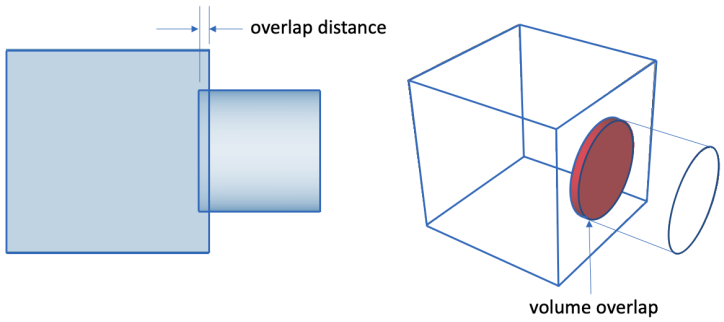


Figure 14. Volume Overlap Example

- **Volume Misalignments** - Misalignments are caused when neighboring volumes touch without overlap, but a small distance between neighboring vertices, curves or surfaces is identified. Figure 15 shows an example of a misalignment. Misalignments can result in sliver surfaces and bad element quality if not resolved prior to imprinting and merging. The **Volume Misalignments** diagnostic test will list pairs of volumes that are misaligned. Expanding a volume pair will reveal entity to entity misalignments that were detected between the pair. Three categories of misalignments will be displayed, namely: vertex-vertex, vertex-curve and vertex-surface ordered by their misalignment distance. These indicate entity pairs that are closer than the **Volume Gap** tolerance that is set in the Options panel. The **tweak surface replace** command can often be used to correct misalignments.

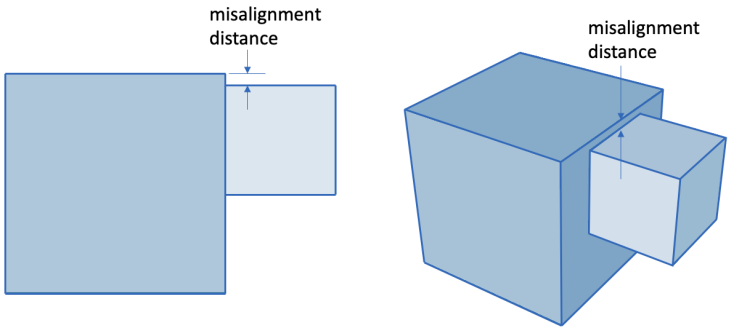


Figure 15. Volume Misalignment Example

- **Volume Contacts** - Volumes that have surfaces in contact but not merged are displayed with this diagnostic test. This provides a way to distinguish volumes that are merged from those that are not and validate whether a contact state should exist between neighboring volumes. This list will include volume pairs that are touching including those that have been identified by the **Volume Misalignment** diagnostic test. Expanding a volume pair will reveal pairs of surfaces on different volumes that are in contact. If the contact state is not correct, normally an **imprint** and **merge** operation should be performed
- **Mergable Geometry** - Pairs of entities on neighboring volumes that are co-located are identified by this diagnostic test. This is normally used to verify that the expected set of surfaces are coincident prior to merging. Mergeable geometry pairs of surfaces, curves and vertices are displayed in this list. Lower order entities (ie. curves and vertices) will not be displayed if its parent geometry

(ie. surface) is identified as mergeable. In most cases, lower order entities identified by this diagnostic indicate the existence of overlaps or misalignments and should be resolved before imprint and merging. Note that the default merge tolerance of 1e-6 is used to determine if entities are mergeable unless the **Tolerant Imprint** checkbox is selected in the Options panel and a user defined tolerance is set.

Geometry Repair Tools

The geometry repair tool buttons appear at the bottom of the Geometry Power Tool. Selecting one of these buttons will bring up the relevant command panel. Tools included in this panel have proven useful for geometry repair and defeaturing.

 **Split Surface Button**

The split surface tool is used to split a surface into two surfaces. This is useful for blend surfaces, for example, where splitting a surface may facilitate sweeping. To select a surface for splitting, click on the surface in the tree view. To select multiple surfaces in the window, hold the CTRL key* while selecting surfaces (surfaces must be attached to each other). Then press the split surface button to bring up the Control Panel window with the ids of selected surfaces in the text input window. The split surface menu is located on the Control Panel under Geometry-Surface-Modify. You must press the Apply button for the command to be executed. You can also bring up the Split Surface menu by selecting surfaces in the tree view and selecting **Split** from the right click menu.

***Note:** For Mac computers, use the command key (or apple key) to select multiple entities

 **Heal Button**

The healing function in Cubit is used to improve ACIS geometry that has been corrupted during file import due to differences in tolerances, or inherent limitations in the parent system. These errors may include: geometric errors in entities, gaps between entities, and the absence of connectivity information (topology). To heal a volume, select the volume in the geometry repair tree view. Then press the heal button. You may also press the heal button without a geometry selected in the window, and enter it later. The Control Panel window will come up under the Geometry-Volume-Modify option with the selected volume id highlighted. If no entity is selected, or if another entity type is selected, the input window will be blank. You can also open the healing control panel by selecting **Heal** from the right click menu in the geometry power tools window.

 **Tweak Button**

The tweak command is used to eliminate gaps between entities or simplify geometry. The tweaking commands modify geometry by offsetting, replacing, or removing surfaces, and extending attached surfaces to fill in the gaps. Tweaking can be applied to surfaces, and it can be applied to curves with a valence no more than 2 at each vertex. It can also be applied to some vertices. To tweak a surface, select the surface in the tree view. The Geometry-Surface-Modify control panel will appear with the selected surface id in the input window.

Tweaking is available for curves. Tweaking a curve creates a blended or chamfered edge between two orthogonal surfaces. The curve option is located on the Geometry-Curve-Modify panel under the Blend/Chamfer pull-down option.

Tweaking is also available for some vertices. Tweaking a vertex creates a

chamfered or filleted corner between three orthogonal surfaces. The vertex option is located on the Geometry-Vertex-Modify panel under the Tweak pull-down menu.

Note: Only curves with valence 2 or less at each vertex are candidates for tweaking. Any other curve will cause the Geometry-Surface-Modify menu to appear.

 **Merge Button**

The merge command is used to merge coincident surfaces, curves, and vertices into a single entity to ensure that mesh topology is identical at intersections. Unlike other buttons on the geometry repair panel, the merge button acts as an "Apply" button itself. All geometry that is listed under "mergeable entities" will be merged.

 **Remove Button**

The remove button is used to simplify geometry by removing unnecessary features. To use the remove feature, click on the surface(s) in the Tree View. Right click and select the Remove Option, or click the Remove icon on the toolbar. The Control Geometry-Surface-Modify control panel will appear, with the surface ids in the input window. The Remove control panel can also be accessed from the right-click menu in the Geometry Power Tools window. Select options and press apply.

 **Regularize Entity Button**

The regularize button is used to remove unnecessary topology. Regularizing an entity will essentially undo an imprint command.

 **Remove Slivers**

The remove slivers button is used to remove surfaces with less than a specified surface area. When ACIS removes a surface it extends the adjoining surfaces to fill the gap. If it is not possible to extend the surfaces or if the geometry is bad the command will fail.

 **Auto Clean Geometry**

The auto clean button is used to perform automatic cleanup operations on selected geometry. These automatic cleanup operations include forcing sweepable configurations, automatically removing small curves, automatically removing small surfaces, and automatically splitting surfaces.

 **Composite Button**

The composite button is used to combine adjacent surfaces or curves together using virtual geometry . Virtual geometry is a geometry module built on top of the ACIS representation. Surfaces may be composited to simplify geometry in order to facilitate sweeping and mapping algorithms by removing constraints on node placement. It is important to note that solid model operations such as webcut, imprint, or booleans, cannot be applied to models that have virtual geometry. Both curves and surfaces may be composited.

 **Collapse Angle Button**

The collapse angle button uses virtual geometry to collapse small angles. This is accomplished by partitioning and compositing surfaces in a way so that the small angle gets merged into a larger angle. Pressing the collapse button on the geometry power tools will open the collapse menu under Geometry-Vertex-Modify control panel. This panel can also be

opened by selecting **Collapse** from the right click menu in the Geometry Tools window.

 **Collapse Surface Button**

Pressing this button will open the collapse surface panel on the main control panel. The collapse surface function uses virtual geometry to eliminate small surfaces on the model to improve mesh quality. It is most useful for blend surfaces.

 **Collapse Curve Button**

Pressing this button will open the collapse curve panel on the main control panel. The collapse curve command is used to eliminate small curves using virtual geometry.

 **Reset Graphics Button**

The reset graphics button will refresh the graphics window display.

Note: Pressing most of the geometry tool buttons on the panel will only bring up applicable command panels on the Control Panel. You must press the Apply button on the Control Panel to execute the command.

Context (Right Click) Menu

The following right click menu options are available from the geometry power tool's main window when a geometry entity or category is selected. Figure 3. shows an example of a context menu. Specific options depend on the type of entity or category.

Test Categories

- **Select All** - Selects all entities in the category
- **Draw All** - Draw all entities in the category
- **Draw All Add** - Draw all entities in the category without first clearing the display
- **Locate All** - Labels all entities in the category in the graphics window. Refresh screen to hide.
- **Expand All** - Expand all categories to show sub-lists of entities
- **Collapse All** - Collapse all categories to hide sub-lists of entities

Entity Visualization Options

- **Zoom To**- Zoom to selected entity in the graphics window
- **Reset Zoom** - Reset graphics window zoom
- **Fly-in** - Animated zoom
- **Locate** - Labels the selected entities in the graphics window. Refresh screen to hide.
- **Draw** - Displays only selected entities by themselves.
- **Draw Add** - Adds the selected entity to the display without clearing.
- **Draw with Neighbors** - Displays only selected entities with all attached neighbors
- **Select Similar ...** - Selects other entities in the same category that have the same geometry characteristic. For example, area, loop thickness, blend radius, angle at vertex, etc.
- **Clear Blend Chain** - Available in Blend category. Selects surfaces in the same blend chain as the selected surface.
- **Clear Cavity/Hole** - Available in Cavity/Hole category. Selects surfaces in the same cavity or hole collection as the selected surface.
- **Clear Selections** - Clears all highlighted entities and reset graphics

Cubit Solution Options

Each of the following menu options are available based on the category and entity type selected. In each case they will open the relevant command panel pre-populated with the entity selected. Select multiple entities prior to selecting the context menu item below to execute the command on multiple entities simulaneously.

- **Tweak Remove Surface...**
- **Remove Surface...**
- **Tweak Replace...**
- **Split...**
- **Remove Slivers...**
- **Remove Blend...**
- **Remove Cavity...**
- **Blunt Tangency...**
- **Heal Owning Body...**
- **Merge Selected**
- **Merge All**
- **Unite Volumes**
- **Regularize**
- **Collapse Curve (Virtual)...**
- **Collapse Surface (Virtual)...**
- **Composite Curves (Virtual)...**
- **Composite Surfaces (Virtual)...**
- **Collapse Angle at Vertex (Virtual)...**

Meshing Tools

The meshing power tool provides a tool for determining whether a geometry can be meshed using autoscheme, or if it requires its scheme to be set explicitly. This tool is designed to help guide users through geometry decomposition process by providing a convenient way to see which geometries need further modification or decomposition prior to meshing.

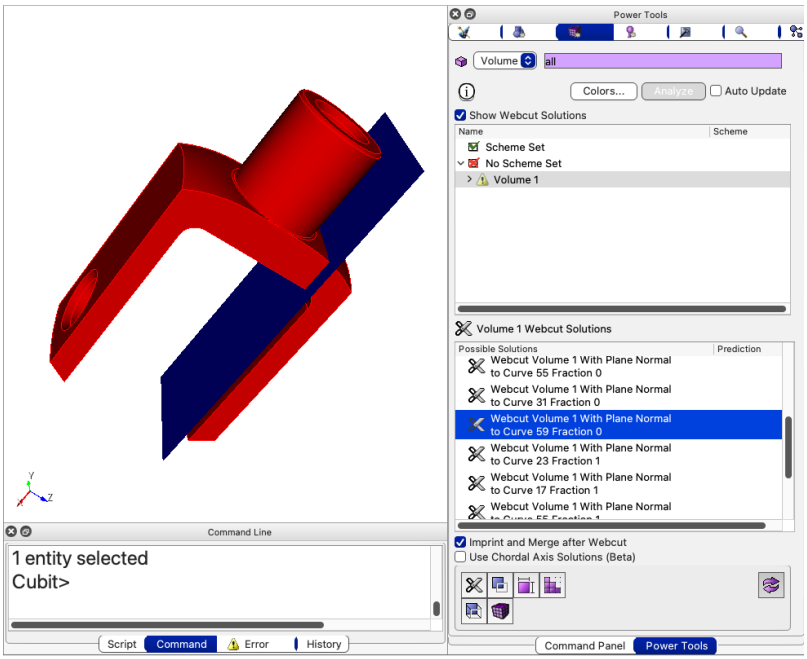


Figure 1. Meshing Power Tool showing the solution window with a preview of a webcut solution.

Entity Specification- The meshing power tool works for volumes or surfaces.

Colors Button - Opens the **Tools>Options** dialog to change the visualization colors of surface schemes for the meshing tool

Show Webcut Solutions - When selected the webcut solutions window will be displayed and populated with custom webcut solutions based on the selected volume.

Analyze Button - The Analyze button issues the autoscheme command for all selected volumes and surfaces and populates a two lists: **Scheme Set** and **No Scheme Set**.

Auto Update Checkbox - Used with the webcut solutions, when selected, after executing a webcut, the model will automatically update to determine meshability of the active volumes.

Output Tree - The output from the meshing tool is displayed in tree format. Geometry is divided into "Scheme Set" and "Scheme Not Set" divisions. The geometry is listed under these nodes. If autoscheme was successful, its assigned scheme is also displayed.

Solutions Window - A solutions window that displays potential webcuts for a selected volume is also available. Display the solutions window by selecting the **Show Webcut Solutions** checkbox at the top of the panel. When a volume is selected, a list of potential webcuts will be displayed. The solutions can be previewed by selecting them and double-clicking will execute the webcut. To further customize the webcut, a right-click on the solution will provide access to the relevant webcut command panel,

pre-populated with necessary parameters.

Imprint and Merge After Webcut Checkbox - Available when the Solutions window is displayed, when selected, the resulting webcut operations will also perform an imprint and merge operation.

Toggle Visibility Button - The meshing tool displays entities as red or green in the graphics window. Green means that they are currently meshable using the autoscheme. Red means that they require their scheme to be set explicitly. Turning this capability off will return the volumes and surfaces to their original colors.

Meshing Tools Buttons - Several meshing tools are available to the user from this window. Depending on the entity selected, these are also available from the right-click context menu, and they are described below.

Right Click Context Menu

- **Zoom To** - Zoom in on this element in the graphics window
- **Draw** - Draw this entity by itself in the graphics window
- **Locate** - Locates and labels entity in the graphics window
- **Rotate About** - Issues Rotate about command for selected entity
- **Visibility On/Off** - Toggle visibility
- **Reset Graphics**- Reset graphics display
- **Set Size** - Opens the Mesh/Entity/Interval panel on the control panel where you can set interval sizes for the selected geometry
- **Set Scheme** - Opens the Mesh/Entity/Mesh panel on the control panel where you can set a scheme for the selected entities
- **Set Vertex Type** - Available when surfaces are selected. Opens the Mesh/Surface/Mesh panel to set vertex types.
- **Imprint/Merge**- Opens the Geometry/Entity/Merge panel on the control panel. If you have entities selected in the tree window it will input them to the imprint/merge command.
- **Webcut** - Opens the Geometry/Volume/Webcut panel on the control panel. If a volume is selected in the meshing tool window it will input it in the webcut panel.
- **Color Surfaces** - Color surfaces based on their schemes. You can change the default colors by selecting the Options button.
- **Restore Colors** - Restores colors on selected entity or entity type
- **Mesh** - Meshes the selected entities (bypassing control panel)
- **Delete Mesh** - Deletes the mesh on selected entities
- **Unmerge** - Unmerges selected entities
- **View Descendants** - Opens a list of child entities and their meshing schemes. Press Analyze to return.
- **View Ancestors**- Opens a list of parent entities and their meshing schemes. Press Analyze to return.
- **View Neighbors**- Opens a list of bordering entities and their meshing schemes. Press Analyze to return.

Mesh Quality Tools

The mesh quality tool is located in the entity tree window under the quality tab. The Mesh Quality Tool works on meshed entities to analyze mesh quality based on selected metrics. Output from the mesh quality analysis can be visualized using color-coded scales. The mesh quality tool also contains tools to improve mesh quality including smoothing, refinement, node merging, mesh validation, deleting mesh elements, and repositioning nodes.

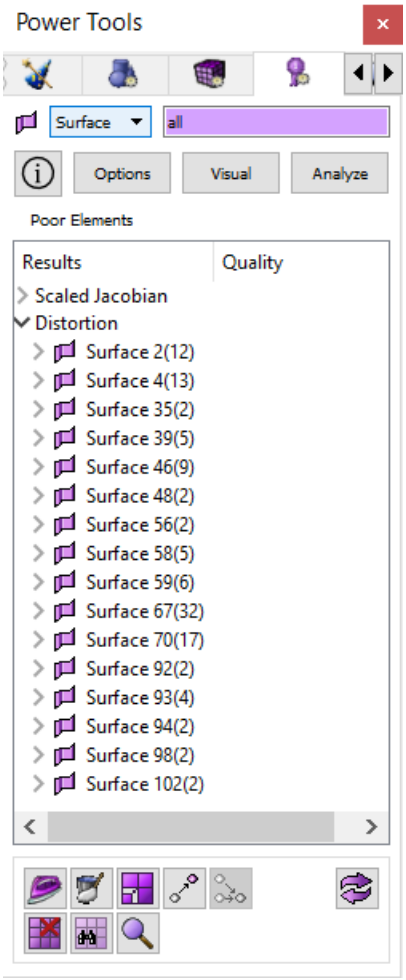


Figure 1. Mesh Quality Tools

Entity Type - The mesh quality tools can only be applied to mesh entities including volumes, surfaces, hexahedra, quadrilaterals, triangles, or tetrahedra.

Help Button - Opens context specific help for this topic.

Options Button - Clicking on this button will show the Tools>Option menu dialog that allows users to manually enter metric range settings. The settings are persistent between sessions. For a description of quality metrics and default ranges click on one of the following links:

- Metrics for Hexahedral Elements
- Metrics for Quadrilateral Elements
- Metrics for Tetrahedral Elements
- Metrics for Triangular Elements

Visual Button - Clicking on this button will open the Mesh/Entity/Quality command panel specific to the entity selected. To visualize elements in the graphics window based on a color-coded quality scale, you must

select the entities to visualize and check the "Display Graphical Summary" check box. Once that box is selected, you must also make sure the "Draw Mesh Elements" option is selected. Then press the Apply button

Analyze Button - This button starts the quality processing based on the metrics/filters selected.

Output Window/Tree - The failed elements are shown in the tree under the heading "Poor Elements". For each metric/filter the output will be listed in a tree format with the following nodes.

1. The top node on the tree is the name of the metric.
2. The next node under is the owning volume or surface when volumes or surfaces are analyzed.
3. The next node will be categories or groups of elements. Possible categories are:
    - All Above Threshold - represents all mesh elements above the quality threshold upper range
    - All Below Threshold - represents all mesh elements below the quality threshold lower range
    - Top "n" - This will expand into a list, up to 50 elements long, of the worst offending elements above the upper threshold range.
    - Bottom "n" - This will expand into a list, up to 50 elements long, of the worst offending elements below the lower threshold range.
4. At the lowest level of the tree are mesh elements.

The mesh elements can be sorted by quality or by numeric order. To change the way items are sorted, click on the headings. The right-click or context menu will show various remedies depending on what is selected. Performing an operation on a parent node will perform the same operation on all of the child nodes.

Mesh Quality Tool Buttons

The buttons on the bottom of the mesh quality tool window are some of the tools you may use to improve mesh quality and include.

- **Smooth Button** - Opens the Mesh>Entity>Smooth panel
- **Refine Button** - Opens the Mesh>Entity>Refine panel
- **Move Node** - Opens the Mesh>Node>Move Node panel
- **Merge Node** - Opens the Mesh>Node>Merge Node panel
- **Delete Mesh Element** - Deletes selected mesh entity
- **Validate Mesh** - Issues the validate mesh command
- **Check Coincident Nodes** - Issues the check coincident nodes command.
- **Refresh Graphics**

Right-Click Context Menu Items

- **Draw** - issues a draw command for any tree node below the metric name.
- **Color Code** - Issues a 'quality .... draw mesh' command for any tree node below the metric name
- **Locate** - Issues Locate for volume/surface/hex/quad/tet/tri. The locate command will draw and label selected entities in the graphics window.
- **Fly-In** - Issues Fly-in for volume/surface/hex/quad/tet/tri. The fly-in command is an animated zoom feature.
- **Zoom to** - Issues Zoom command for volume/surface/hex/quad/tet/tri
- **Rotate About** - Issues Rotate About command for volume/surface/hex/quad/tet/tri
- **Vis on/off** - Issues visibility on/off for volume/surface

- **Smooth** - Issues generic smooth command for volume/surface/hex/tet
- **Smooth Surface Parent** - issues a smooth surface command for the surface parents of selected quads and tris.
- **Delete Mesh** - issues delete mesh propagate command for vol/surf
- **Delete Elements** - issues delete element command for mesh entities in all categories except 'all'
- **Validate mesh** - validates selected volume or surface
- **Check Coincident Nodes** - checks for coincident nodes on volume or surface
- **Smooth Panel** - brings up the correct smooth panel depending on what's selected
- **Smooth Surface Panel** - bring up the smooth surface panel with correct surface ids for selected quads and tris
- **Merge Node Panel** - brings up the panel to merge nodes
- **Move Node Panel** - brings up the panel to move nodes
- **Reset Graphics** - resets the display

Assembly Tool

In Cubit versions prior to 15.4 assembly data was managed on the model tree. Beginning with Cubit version 15.4 a power tool for assemblies is available.

A detailed, command-based discussion of assemblies and metadata can be found here.

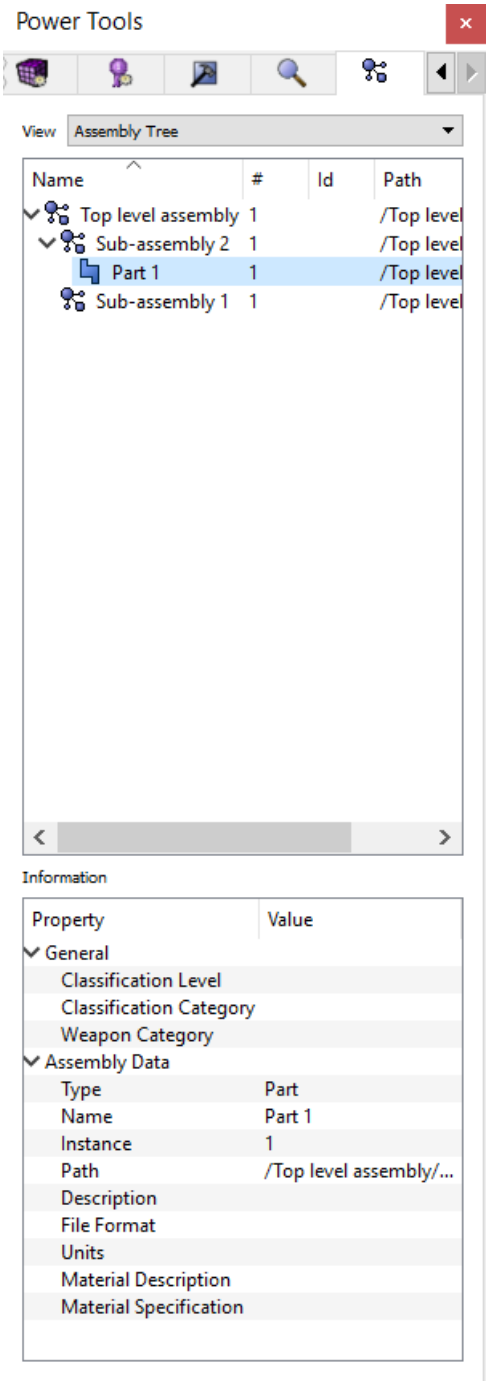


Figure 1 - Assembly Power Tool

Like all other power tools in Cubit, the user is encouraged to experiment with the various options found in context menus. The context menus are specific to the entity type selected.

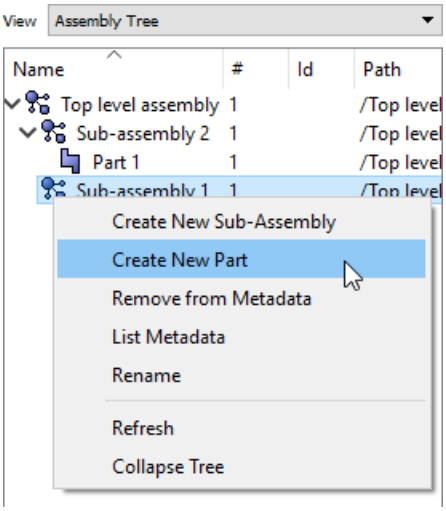


Figure 2 - Assembly Context Menu

Metadata Attributes

Users have access to the following attributes:

- **Type** - The metadata type: Assembly, Sub-Assembly or Part
- **Name** - The name for the assembly or part. This can be edited from the property window
- **Instance** - The numeric value associated with the part or assembly
- **Path** - The absolute path of the part or assembly
- **Description** - The description of the part or assembly
- **Material Description** - The name or description of the material of which this part is composed
- **Material Specification** - The formal specification number of the material of which this part is composed
- **File Format** - The name of the file system containing the original version of this entity
- **Units** - The unit system of this part or assembly

Another View

At the top of the tool is a pull down menu. This allows the user to change from the full assembly view to a parts view.

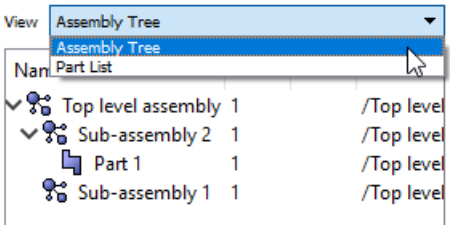
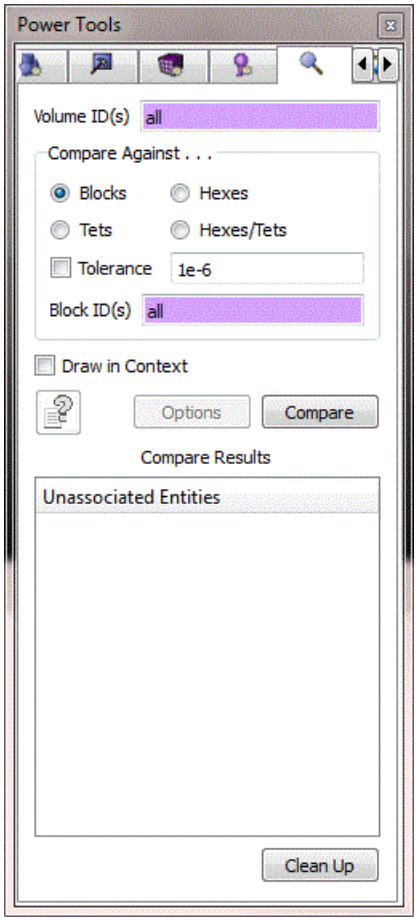


Figure 3 - Pull down Menu

Geometry/Mesh Comparison Tool

The Geometry/Mesh Comparison Tool tries to find geometry and mesh that do not correspond. The typical use is to import a geometry file and then import a mesh file that is associated with the geometry. The comparison tool will locate mesh that does not correspond to the geometry. The tool will also show geometry that does map to any mesh.



The user selects the volumes for the comparison, then selects the mesh entities for the comparison. A default comparison **tolerance** value of 1e-6 will be used unless otherwise specified. No additional setup is required. Select the "**Compare**" button to generate results.

Unassociated entities will be displayed in one of two categories:

1) Mesh elements not associated with any volume

2) Partially meshed volumes

Clicking on the labels in the tree will cause the entities to be drawn in the graphics window. If "**Draw Without Refreshing**" is selected, the draws will be additive. If "**Draw Without Refreshing**" is not selected, the previous draw will be removed when the current drawn entities are shown.

The underlying Cubit command for the tool is the following:

> **Compare volume <id range> {block <id range> | hex <id range> | tet <id range> [tolerance <value>]**

The command will create three types of groups that contain non-

corresponding mesh and/or geometry. The group named "**mesh_with_no_volume**" contains hexes or tets that cannot be associated with any volume. The groups named "**No_meshed_Volume_***" contain the curves of a volume (for display purposes) that is completely void of any hexes or tets. Lastly, the groups named "**Partially_meshed_Volume_***" contain hexes or tets, faces or tris, and curves of volumes that could only be partially associated with mesh. The group is created with these entities so that the user can see the partially meshed regions of the volume.

Defeature Tool

The Defeature Tool is capable of removing small irrelevant curves and surfaces. These small curves and surfaces are one of the main sources of low quality elements and meshing failures. Sliver surfaces and curves generally exist at fillets, chamfers, and sliver surfaces at misalignments in imprinted assembly models.

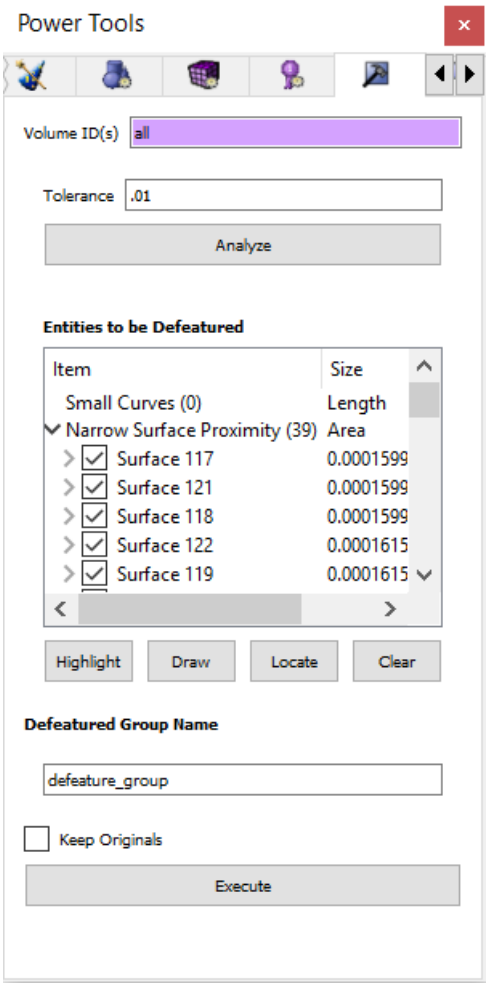


Figure 1 - Defeature Power Tool

Defeaturing small curves and surfaces involves three main steps:

1. Analyze the model to automatically detect small curves and surfaces.
2. Manually deselect, if needed, detected small curves and surfaces.
3. Execute the defeature tool to remove small curves and surfaces.

Step 1 requires specifying volume ids (e.g. all) and a tolerance (e.g. 0.6) as shown in Figure 1. Clicking “Analyze” button will automatically find small curves and surfaces in the volumes specified. Figure 2 shows the highlighted small curves and surfaces with the label information. Figure 3 shows a zoom view of a small surface.

In Step 2 the user is allowed to deselect entities by unchecking entities from the list “Entities to be Defeatured”. Users can also use “Highlight”, “Draw”, and “Locate” buttons to examine the automatically detected entities (see Figure 2).

In Step 3 actual defeaturing is performed by clicking the “Execute” button (see Figure 5). Figure 4 shows the zoom view of a defeatured volume.

Defeatured volumes are created in a new user specified group (by default in “defeature_group”) as shown in Figure 6. Only the volumes that have small curves and surfaces will be defeatured. Also, by default old original volumes are deleted and new defeatured volumes (child entities) will use the corresponding old ids. Please use the option “Keep Originals” if you want to have both old original and new defeatured volumes.

NOTE:

1. The new defeatured volumes are in MBG format. That is defeatured volumes are facet based instead of NURBS based ACIS volumes. Therefore, it is highly recommended to perform NURBS based operations such as webcut and imprint before calling defeature.

Command Syntax:

```
Set tolerant mesh mbg only
```

This command forces the mesh to associate with new defeatured volume. Currently, this command must be called before calling the defeature command below.

```
Defeature curve_length <value> [Curve <ids>]
[Curve <ids>] surface_prox2d <value> [Surface
<ids> ] [group <id>] [keep]
```

curve_length <value>: Curves with length less than or equal to <value> are automatically detected as candidate for defeaturing if auto_identify is specified. Otherwise, [Curve <ids>] must be specified.

surface_prox2d <value>: Surfaces with narrow region between opposing bounding curves are automatically detected as candidate for defeaturing if auto_identify is specified. The 2d proximity <value> specified in detecting surfaces containing narrow regions. If auto_identify is not specified, then [Surface <ids>] must be specified.

group <id>: Defeatured volumes are added to the group id specified.

keep: If keep argument is specified original entities are kept along with new defeatured volumes. If keep argument is not specified, then original entities are deleted and new defeatured volumes and its subentities (surfaces, curves, and vertices) will use the ids of original volumes.

Preserving Critical Geometric Entities

Before defeaturing the geometry, the user may wish to specify geometry that will be preserved during defeaturing. The below given "Fix" keyword is used to preserve any entity. The user may specify a volume, surface, curve, or vertex to fix.

```
Mesh Tolerant Fix
[Volume|Surface|Curve|Vertex] <range>
```

To reverse the effects of fixing a geometric entity, the user may "free" an entity using the following syntax

```
Mesh Tolerant Free
[Volume|Surface|Curve|Vertex] <range>
```

Example for fixing geometric entities:

reset

brick x 10

brick x .1

move vol 2 x 5

unite all

mesh tolerant fix surf all

mesh tolerant fix curve all

Defeature curve_length .2 curve 31 29 27 26 24 32 13 30 17 28 22 25

surface_prox2d .2 surface 13 14 15 16 12

Sample Journal File:

Even though the defeature tool is mainly intended to driven by the GUI, it can be used via command line. Without the GUI, it will be harder to provide the list of small curves and surfaces to the defeature command. Here is a sample journal file:

import simple assembly

import acis 'assembly11a.sat'

perform any ACIS based operations such as webcutting and imprinting first

imprint all

merge all

enable the developer only command

set developer on

force the mesh to associate with defeatured MBG volumes

set tolerant mesh mbg only

create a new group to store defeatured volumes

group 'defeatured_vols' add volume all

perform actual defeaturing by specifying the volume ids, tolerance, and small curve/surf ids.

defeatured volumes will be placed in the user specified group id and original entities can be

kept along with new defetured volume using “keep” option.

defeature volume all curve_length 0.3 curve 107 103 102 100 88 85 82 80 9 6 4 2 214 212 211 210 203 200 199 197 188 187 185 183 170 167 164 162 234 232 227 225 254 253 252 251 249 248 243 242 272 271 270 269 265 264 259 258 288 287 286 285 281 280 275 274 304 303 302 301 297 296 291 290 312 311 307 306 surface_prox2d 0.3 surface 47 48 50 51 41 43 40 42 2 4 1 3 111 112 118 120 121 122 124 126 128 129 130 132 134 135 136 138 140 141 142 144 81 82 83 84 88 89 90 91 94 95 96 97 100 101 102 103 group 2 keep

del any old original volumes if you don’t want it anymore

delete vol 1 to 11

enable visibility of only defeatured vols

vol all vis off

vol all in group 2 vis on

set scheme to tetmesh

vol all in group 2 scheme tetmesh

set mesh size

vol all in group 2 size 1

mesh defeatured vols

mesh vol all in group 2

disable developer only command

set dev off

Figures

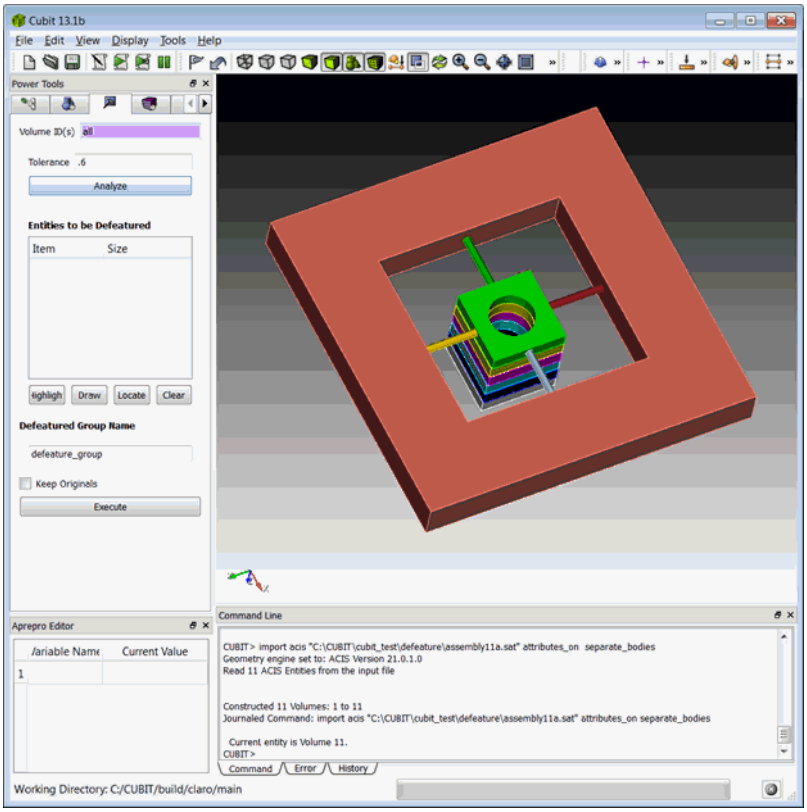


Figure 1: Specify Volume ID and Tolerance before clicking "Analyze"

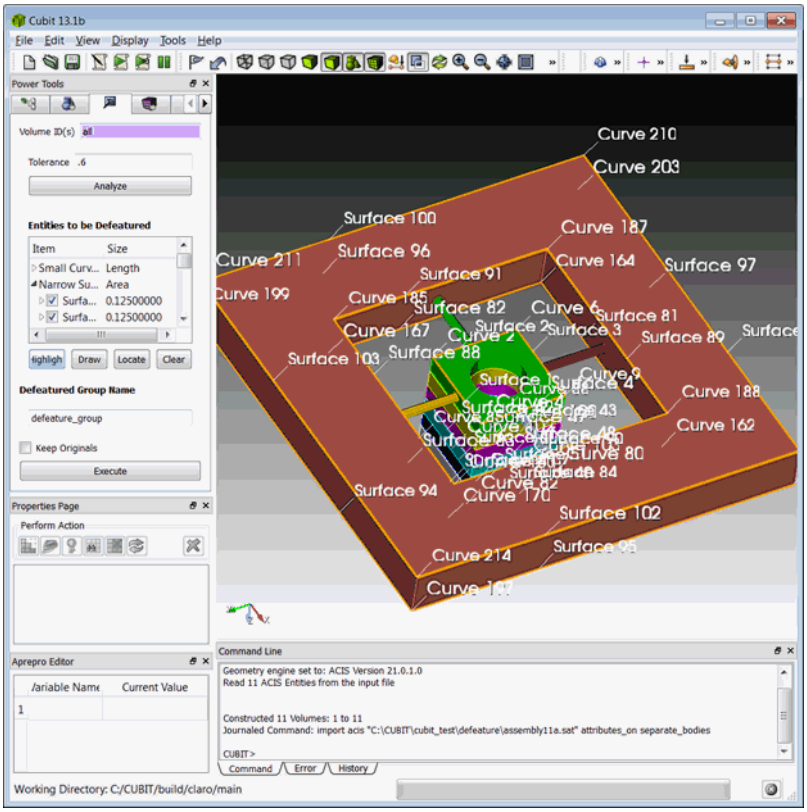


Figure 2: Use “Highlight”, “Draw”, and “Locate” to visualize small curves and surfaces

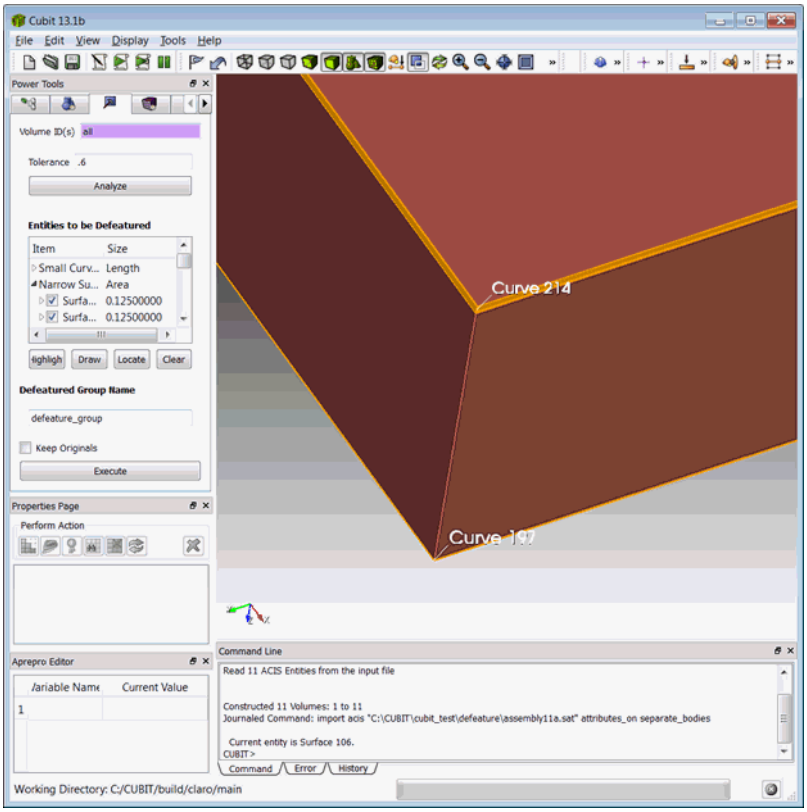


Figure 3: Zoom view of a small curve and surface

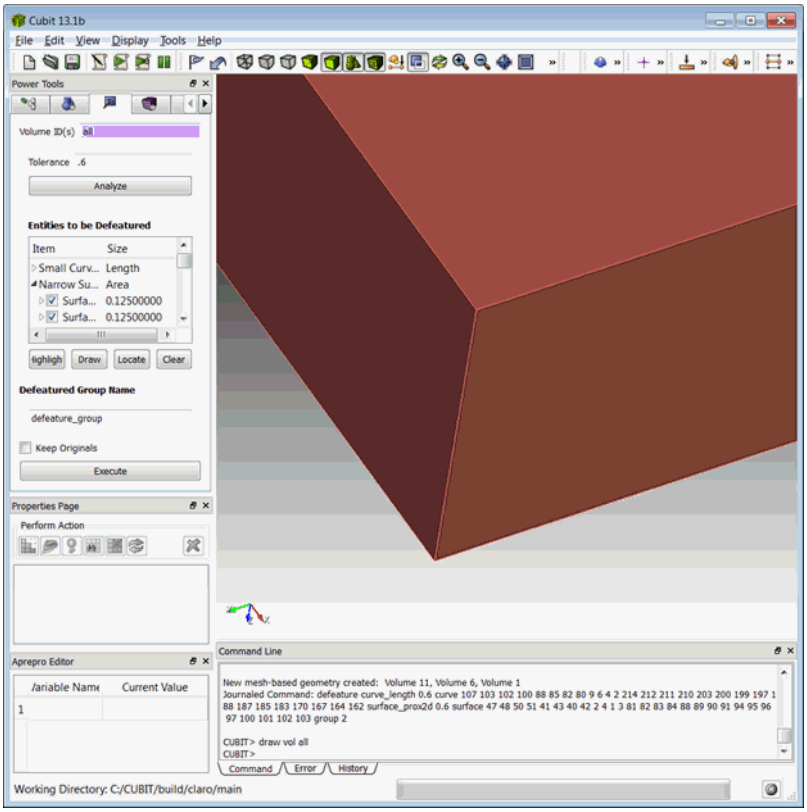


Figure 4: Zoom view of defeatured volume

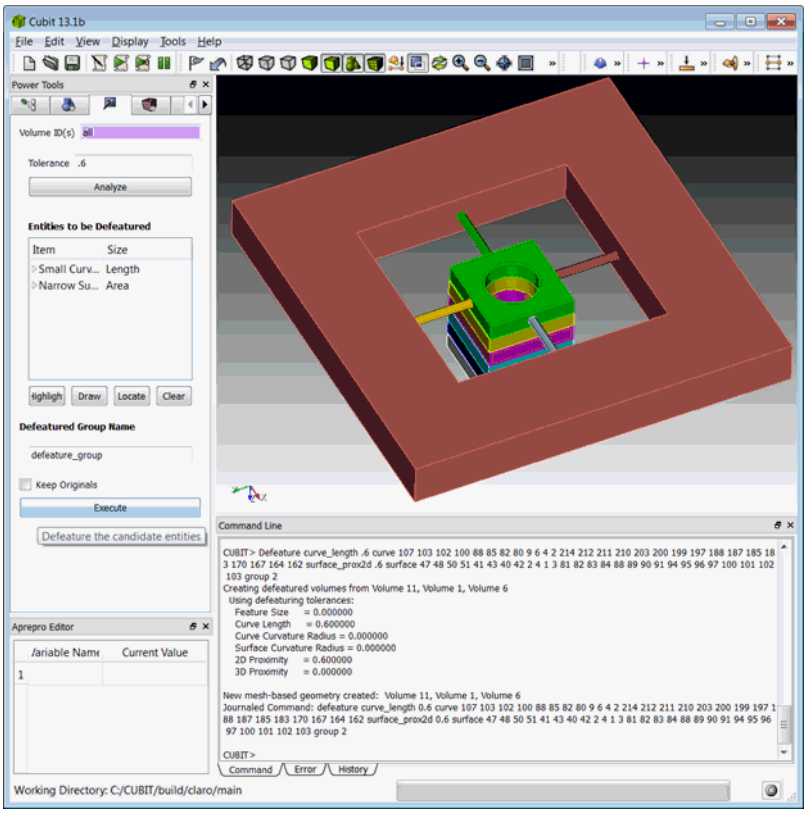


Figure 5: Click Execute button to Defeature automatically/manually selected entities

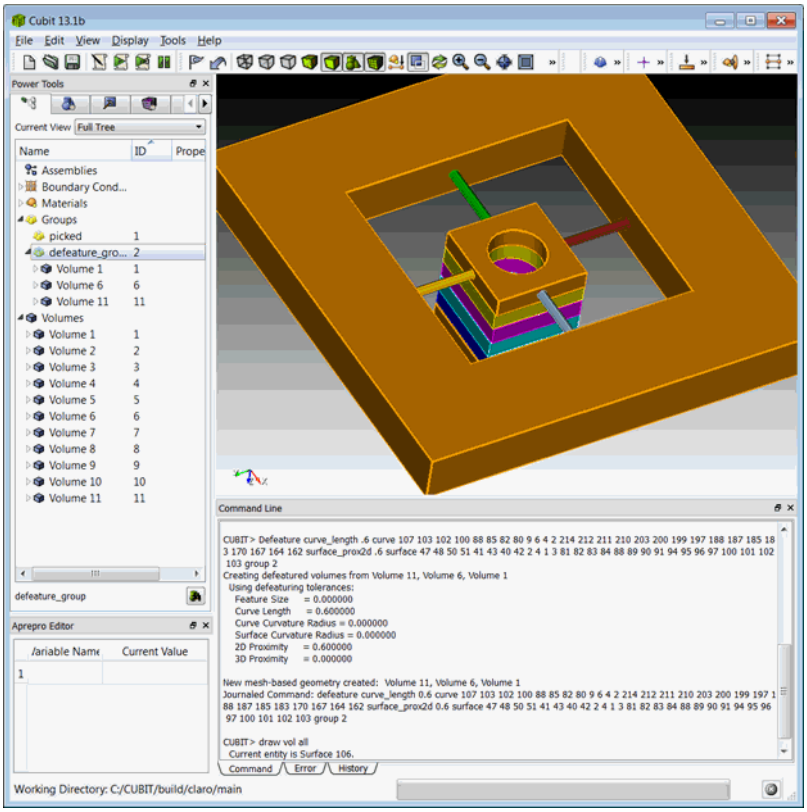


Figure 6: New defeature_group contains defeatured volumes in MBG format

Beams and Shells with the Geometry Power Tool

This page describes the beam and shell modeling tools that are part of Cubit's Geometry Power Tool. Geometry simplification is often required when modeling assemblies comprised primarily of thin volumes. Shell finite elements are often used rather than full 3D hex or tet elements. The task for analysts in this case is to reduce the 3D set of thin volumes to a set of connected sheet bodies where a mesh of triangles or quadrilaterals can be applied. This procedure can be managed with the Geometry Power Tool if the Beam and Shell diagnostic is selected.

Background

The Geometry Power Tool in Cubit provides a series of diagnostic checks on your model used to defeature or simplify a CAD model prior to meshing. Clicking the **Analyze** button will perform the selected diagnostic tests and display an expanding tree listing geometric entities that are identified by each test. Once identified, suggested solutions can be easily previewed and executed.

The Beam and Shell diagnostics work best in conjunction with the machine learning models. As such, it is recommended that if using these tools, that the **Load ML Models** button first be selected using the procedure described in the page Machine Learning with the Geometry Power Tool.

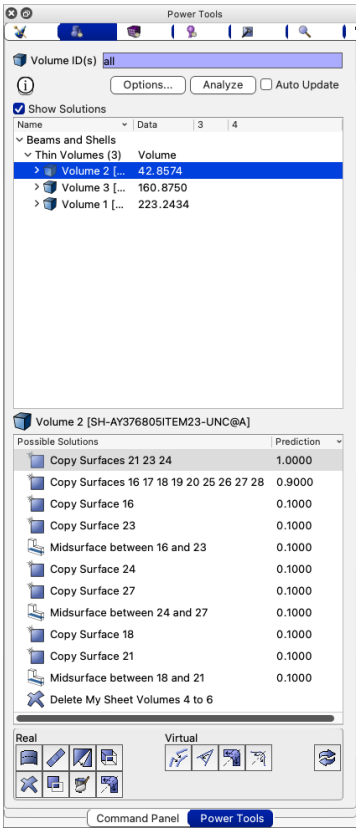


Figure 1. Thin Volumes diagnostic displayed with Solution window. Reduce solutions show for volume in figure 3.

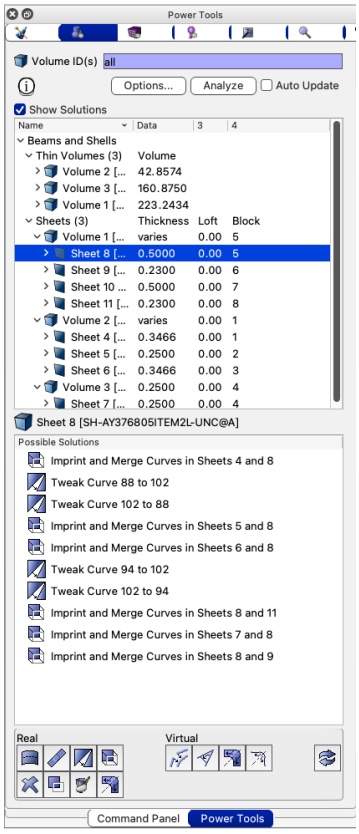


Figure 2. Sheets diagnostic displayed with Solution window. Solutions for connections at one of the sheets are displayed.

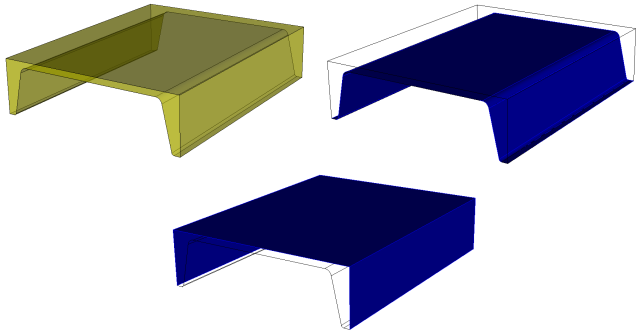


Figure 3. Two reduce options for the same volume

Thin Volumes

This diagnostic manages the reduction of thin volumes to their 2D or sheet body equivalent. Its primary use is to preview and execute custom reduction operations on selected thin volumes when using the Show Solutions checkbox. The default mode for this diagnostic is to simply list all active volumes. If the machine learning models have been loaded, this diagnostic will be limited to volumes identified as "thin" for its classification category. Figure 1 shows the power tool with an example of a simple 3 volume assembly, where volumes are listed in ascending order based on their geometric volume.

Thin Volume Solutions

If **Show Solutions** is checked, selecting a volume in the list will reveal

custom solutions for reducing the selected volume to one or more sheet bodies. Figure 3 shows an example of a C-shape volume with its **reduce** solutions displayed in figure 1 in the **Solutions** window. Selecting a solution, will display a preview of the reduction operation. In this example, the first two solutions show the reduced representation previewed in blue. Double clicking on any of the solutions will execute the solution using the corresponding reduce thin command. Note that reducing a 3D thin volume will automatically set up an association from the volume to its child sheet body(s).

Deleting Reduced Solutions

If a reduce solution (sheet bodies originating from the volume) is already defined, a **delete** operation will also be available in the custom solution. Double-clicking the **delete** solution will delete the sheet body(s) and remove it as one of its sheet body children.

Machine Learning Confidence

If the machine learning models are loaded, a *Confidence* value will be displayed with each custom reduce option. This is a predicted suitability score based on existing training data. The solutions will be listed with the highest confidence listed first. Predictions are based based on existing training data and can be updated using the Reinforcement Learning tool. They can also be modified to reflect user preferences using the right-click menu option when the solution is selected using either the **Maximize Confidence** or **Minimize Confidence** options.

Sheets

This diagnostic will list all volumes defined as sheet bodies. It is most useful when used with the **reduce thin** operations, as sheet bodies will automatically be grouped based on their 3D thin volume parent. If a sheet body was not created with a **reduce thin** command, it will be grouped under a generic "orphan" category.

Sheet Attributes

Additionally, when using the **sheets** diagnostic, three new columns for data entry will appear which are used for displaying and updating the **thickness**, **loft** and **block** for the selected sheet body. When using a **reduce thin** operation, the thickness and loft values for the new sheet body will be computed and displayed. Blocks will also automatically be created containing the new sheet body with the thickness and loft assigned as attributes.

Clicking on the entries for thickness, loft or block for a given sheet or volume will permit editing of these values. Once edited, the appropriate commands will be issued to update the block attributes. Changing the block ID will also update or create a new block if necessary.

Sheet attributes can also be exported in the form of a spread sheet by right-clicking on the sheet bodies or volumes that should be exported and selecting "Export Sheet Data..."

Sheet Solutions

Custom solutions displayed will be **tweak** or **imprint/merge** operations that will attempt to connect nearby sheet bodies. For example, when selecting a solution for **tweak**, a preview similar to figure 4 may be displayed. Double-clicking will execute the **tweak**, extending the surfaces to touch as shown in figure 5.

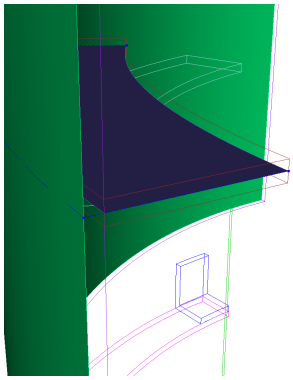


Figure 4.

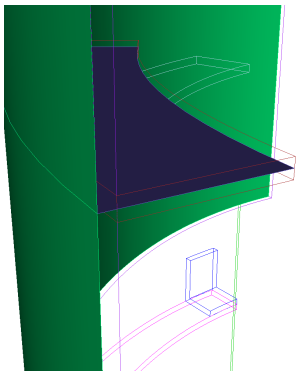


Figure 5.

Right-Click Menu Options

The beam and shell diagnostics provide many of the same visualization options available with the other diagnostics. A few additional tools, useful for shell modeling, are also provided and described here:

Draw Sheets [Add]

Draws the selected sheets in the current graphics display mode (usually smooth shade) and its parent volumes in wire frame. If right-clicking the "sheet" or "thin volume" categories in the diagnostic window, all sheets and their parents will be displayed.

Draw Sheets with Thickness [Add]

Draws the sheets similar to above, but also displays the sheet as a 3D transparent volume extrusion, where the thickness of the extrusion is defined by the current assigned thickness attribute. The transparent extruded volume will also reflect the current loft attribute for the assigned block. Note that this is similar this is similar to using the command line **draw shell volume**. This tool is useful for visually validating the currently assigned loft and thickness values.

Select/Locate Merged Curves

Selecting this tool will highlight or locate all curves from the selected sheets that are merged with a neighbor. This is useful for checking that the intended connections have been made. Note that this option will only select merged curves on sheet bodies that are children of neighboring 3D thin volumes. (It is assumed that sheet bodies on the same parent volume will be merged)

Select/Locate Unmerged Curves

Similar to the previous option, but finds connections that have not yet been made and highlights the closest curve. This is useful for identifying sheet bodies that are not yet connected but should be.

Delete My Sheets

Deletes child sheet bodies from a parent 3D thin volume and removes its association.

Export Sheet Data...

Brings up a file-picker dialog and exports the current sheet attribute data to a .csv file. If right-clicking the "Sheets" or "Thin Volumes" category

headers, all sheet data will be exported. Otherwise, only the selected sheets will be exported.

Maximize/Minimize Confidence

Available when right-clicking on a reduce command in the solution window. If the machine learning models are loaded, a confidence value will be displayed next to each thin volume reduce command in the custom solution window. This indicates a *suitability* score (0 to 1), with the solutions ordered highest confidence to lowest. In many cases choosing the highest confidence predicted by the ML supervised learning tool is sufficient, however since the predictions are based only on existing training data, or models it has seen before, it is possible that the highest confidence solution is not desirable. If this occurs, the **maximize** or **minimize confidence** right-click menu options can be used.

Selecting one of these options will effectively add or replace existing training data to augment the machine learning model. Executing one of these commands will trigger *retraining*, the ML model will be reloaded and a new predicted confidence displayed. Subsequent operations that use the same or similar reduction will now be updated to reflect the new confidence prediction.

The primary use case for **maximize** and **minimize confidence** is following reinforcement learning. If the RL methods are unable to identify the best solution for a given situation, the user can interveen and provide their own preferences to the learning model.

Auto Reduce Thin Volumes...

Available when right-clicking on the "Thin Volumes" diagnostic category. Brings up the command panel for the **reduce thin auto** command populated for all active thin volumes. This command provides an automatic method for reducing multiple thin volumes simultaneously while maintaining connections.

Reinforcement Learning...

Available when right-clicking on the "Thin Volumes" diagnostic category. Brings up the command panel for the **reduce thin RL** command populated for all active thin volumes. This command uses machine learning with reinforcement learning methods to build a knowledge base of reduction solutions with their suitability. Use RL in conjunction with the **maximize** and **minimize confidence** options to inject customized user preferences into the RL solutions. It can also be used for predicting an ordered sequence of Cubit commands (journal file) for reducing and connecting all active thin volumes.

Machine Learning with the Geometry Power Tool

This page describes the machine learning tools that are part of Cubit's Geometry Power Tool for tet mesh quality prediction and part classification.

Background:

The Geometry Power Tool in Cubit provides a series of diagnostic checks on your model used to defeature or simplify a CAD model prior to meshing. Clicking the **Analyze** button will perform the selected diagnostic tests and display an expanding tree listing geometric entities that are identified by each test. Once identified, suggested solutions can be easily previewed and executed.

This new capability enhances the Geometry Power Tool using state-of-the-art machine learning (ML) methods to predict meshing outcomes, suggest solutions as well as classify certain common part types.

Enable Machine Learning:

Use the following procedure to access the new ML capabilities:

1. Navigate to the Geometry Power Tool by clicking on the Geometry icon in the power tools window as shown in Figure 1.

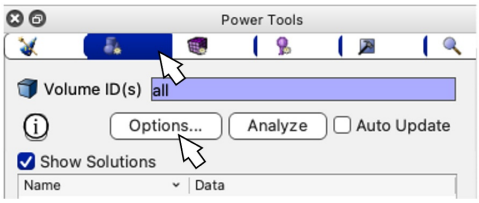


Figure 1. Navigation to Power Tool and Options panel.

2. Select the **Options...** button shown in Figure 1.
3. The panel shown in Figure 2. should appear. Click the button labelled **Load ML Models**
4. After the machine learning models are loaded, a variety of diagnostic options become accessible, as illustrated in Figure 3. Available diagnostics encompass "Reduce for Simulation," "Tetmesh Poor Quality Predictions," and classifications for "Parts" and "Surfaces." Furthermore, settings specific to ML-based diagnostics will also be made available for fine-tuning.

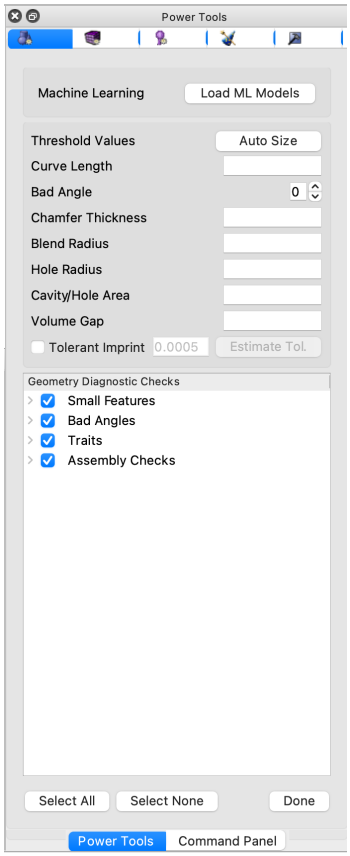


Figure 2. Options panel without ML models loaded

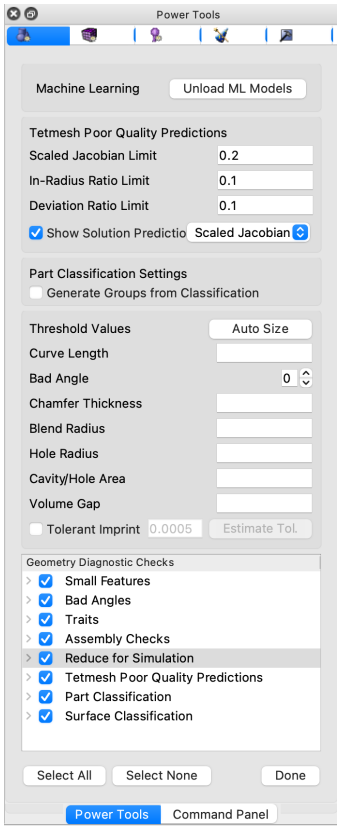


Figure 3. Options panel after loading ML models.

Geometry Analysis Tools

The following describe the additional diagnostics available when ML is enabled in the Geometry Power Tools.

1. Reduce for Simulation
2. Tetmesh Poor Quality Predictions
3. Part Classification
4. Surface Classification

Reduce for Simulation

This diagnostic category leverages machine learning to pinpoint

geometric entities crucial for targeted physics simulations. It facilitates the examination, visualization, and contextual simplification into proxy geometry or boundary conditions suitable for simulation. This is achieved through the reduce command suite within Cubit. Supported categories include:

- Fasteners
- Beams and Shells
- Slot Surfaces

Fasteners

The "Fasteners" category builds upon the "bolt" section found in the Part Classification tool, streamlining the management of connections within assemblies. This diagnostic is especially useful given the tedious nature of managing fastener connections and the varying proxy requirements for simulation across different physics.

Procedure for Using the Fasteners Diagnostic Tool

1. **Open the Options Panel:** Start by opening the options panel in the Geometry Power Tool.
2. **Enable Reduce for Simulation:** Locate and check the "Reduce for Simulation" checkbox within the options panel.
3. **Select Categories:**
    - Expand the categories under "Reduce for Simulation".
    - Select the categories relevant to your project (Clamped Members, All Bolts, Pilot Holes). Avoid selecting categories that are not required to reduce unnecessary computation time.
4. **Close the Options Panel:** After making your selections, click the "Done" button to proceed.
5. **Optimize Analysis (Optional):**
    - If analyzing only a portion of the model, you can reduce computation time by entering the volume IDs of the portions of the assembly you are interested in.
6. **Initiate Analysis:**
    - Click the "Analyze" button to start the diagnostic process.
    - Note: The analysis might take a significant amount of time for larger models due to the machine learning classification process involved.
7. **Review Results:**
    - Upon completion, the tool will display up to three categories based on the findings, such as identified fasteners or pilot holes, if any.

Bolt Categorization Methods

1. **Clamped Members:**
    - Groups bolts by the members they fasten, naming clamped members according to their ID (e.g., "Volume 1 to 2").
    - A context menu provides options for visualizing, zooming, fly-in, drawing, and locating the clamped members, as well as selecting and visualizing all bolts clamping the two members.
    - Right-click menu options to open command panel for reduction operations, allowing operation on all bolts in clamped members simultaneously
    - Expanding the list under the clamping members' name shows individual bolts, with right-click options available for visualization.
    - Selecting an individual bolt populates the solutions window (see **Show Solutions** checkbox) with custom solutions. Choose to preview or execute directly, or access a command panel for additional options.
    - Right-click option to switch the display to show clamped member names based on their volume or block name (e.g.,

"PartA to PartB") rather than ID (e.g., "Volume 1 to 2"").

2. **All Bolts:**
    - Categorizes bolts by name, derived from step or ACIS file definitions, facilitating operations on bolts with similar characteristics.
    - Selection and visualization of all bolts with a common name are supported, with listing all bolts without further categorization if names are not provided.
3. **Pilot Holes:**
    - Lists concentric hole pairs for potential fastener application, useful in cases where CAD models do not provide fasteners or they have been removed.
    - Groups pilot holes according to their clamped members, with options for visualizing and operating on all pilot holes simultaneously.
    - Selecting an individual pilot hole populates the solutions window (see **Show Solutions** checkbox) with custom solutions. Choose to preview or execute directly, or access a command panel for additional options.

Below is an illustration of the Geometry Power Tool, showcasing the three categories and sample geometry with highlighted clamped members and associated bolts.

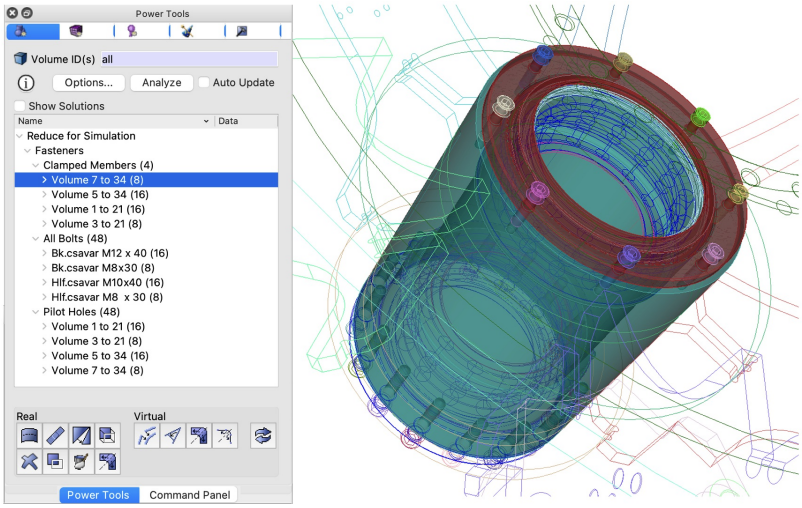


Figure 2. Fasteners diagnostic shows clamped members with their associated bolts.

Beams and Shells

Geometry simplification is often required when modeling assemblies comprised primarily of thin volumes. Shell finite elements are often used rather than full 3D hex or tet elements. The task for analysts in this case is to reduce the 3D set of thin volumes to a set of connected sheet bodies where a mesh of triangles or quadrilaterals can be applied. This procedure can be managed with the Geometry Power Tool if the Beam and Shell diagnostic is selected. See Beams and Shells with the Geometry Power Tool for details.

Slot Surfaces

The Slot Surfaces diagnostic is a new addition to the geometry power tool for preparing slot surfaces in Electromagnetic (EM) modeling. A slot surface is used to identify potential pathways where EM radiation can potentially exit. This diagnostic will help manage slot surfaces and prepare them for the application of boundary conditions and analysis. This diagnostic utilizes Machine Learning (ML) to predict the most likely slot surfaces. See Slot Surface Preparation with the Geometry Power Tool for details.

Tetmesh Poor Quality Predictions

The solid model used as the basis for a tetrahedral mesh may contain small features or angles that can lead to poor mesh quality or very small elements. This diagnostic will provide a list of entities (vertices, curves, surfaces) that are predicted to result in poor quality tet elements sorted by their predicted mesh quality metric. This allows the user to quickly focus on regions of the model that will result in the worst mesh quality and apply geometry operations to improve the meshing outcome without having to mesh.

Three different mesh quality metrics are used for the basis of the predictions: Scaled Jacobian, In-radius and Deviation. Entities identified by these diagnostics will be sorted according to their ML-predicted metric starting with the worst quality entity. The three edit fields at the top of the Options panel control the quality limit. For example, a **Scaled Jacobian Limit** of 0.2 will identify all geometry entities predicted to have nearby tet elements whose Scaled Jacobian is less than 0.2. For our purposes, "nearby tets" are defined as those within two edge lengths of the geometric entity.

- **Poor Scaled Jacobian** - Select this option if you would like to view mesh quality predictions based on Scaled Jacobian. Scaled Jacobian is a value between -1.0 and 1.0, where 1.0 is ideal and -1.0 is completely inverted. This value is a function of the angles at the vertices of the tet so that values less than or close to zero should be avoided for analysis.
- **Poor In-Radius** - Select this option if you would like to view mesh quality predictions based on In-Radius. The in-radius is defined as the radius of the largest inscribed sphere in a tetrahedron. We are interested in a measure of how closely the tet elements meet the user-prescribed mesh size. Since in-radius is an absolute distance, we use the in-radius ratio defined as **in-radius/ideal-inradius**. The ideal in-radius is defined as the radius of an ideal tet (Scaled Jacobian = 1.0) whose edge lengths are precisely the user-specified mesh size. Therefore, a value of 1.0 meets exactly the mesh quality requirement, while a value less than 1.0 is smaller than prescribed. To determine the ideal in-radius current mesh size defined on the local volume will be used. if not set, then a target of **4*small curve threshold** will be used.
- **Poor Deviation** - Select this option if you would like to view mesh quality predictions based on Deviation. Deviation is defined as the distance from the midpoint of a triangular face at the boundary to its closest point on the geometry. A large value for deviation indicates that the mesh does not conform well or deviates significantly from the geometry. As deviation is also an absolute distance, we define the deviation ratio as deviation/mesh size. A value close to 0.0 for deviation ratio is better as it indicates the triangular faces very nearly match the geometry. A larger value closer to 1.0 indicates the local deviation from the geometry is on the order of the mesh size. To determine the deviation ratio, the current mesh size defined on the local volume will be used. If not set, then a target mesh size of **4*small curve threshold** will be used.

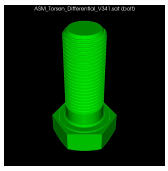
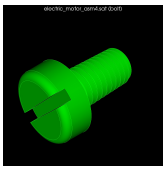
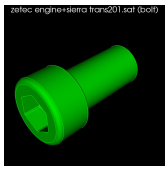
Context Menus: To display all entities in a specific category, select the diagnostic category title and use the right click context menu to choose **Draw**. Expanding the list will display individual entities that can be visualized using standard draw tools such as **Zoom To**, **Fly In**, **Draw Owning Volume**, **Draw with Neighbors**, etc. When one or more entities are selected in the list, a context menu also provides access to selected command panel options. When multiple entities are selected, the context menu can be used to invoke command panels to operate on multiple entities simultaneously.

Part Classification

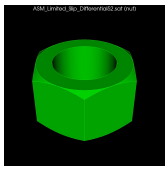
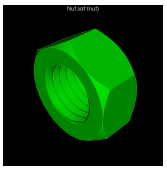
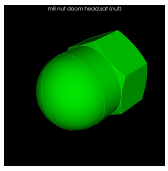
This diagnostic provides the ability to classify volumes according to

several common part types. When selected, characteristic geometric features of each volume will be computed and ML methods will be used to determine the part classification. Each volume will be placed into a list based on its most probable categorization. Volumes in each classification will be ordered based upon a **Confidence** metric. A confidence of 1.0 indicates a 100% confidence of categorization. Confidence values closer to 0.5 are less likely to be categrized correctly, but ML indicated a higher probablility than other part types. The current part categories are illustrated below showing a few examples from each category. In addition to these Cubit's classify command can manage custom creation and editing of part categories.

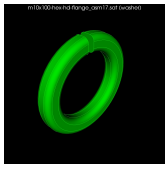
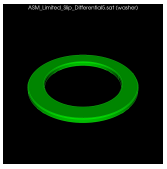
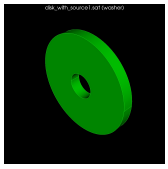
Bolts

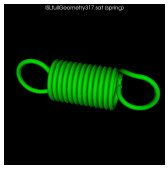
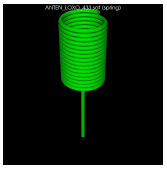
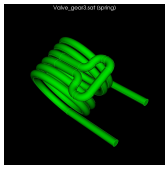
Nuts

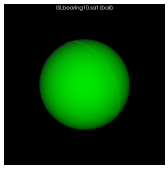
Washers

Springs

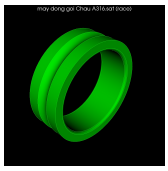




Balls

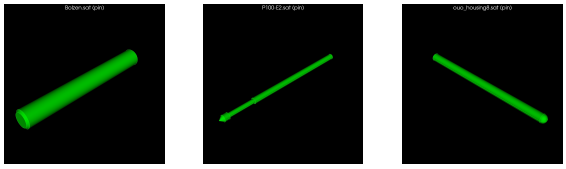


Bearing Races

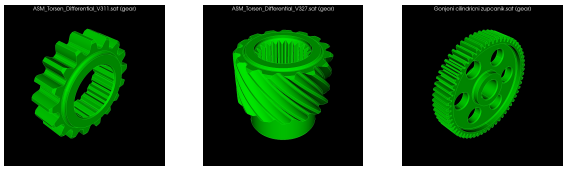




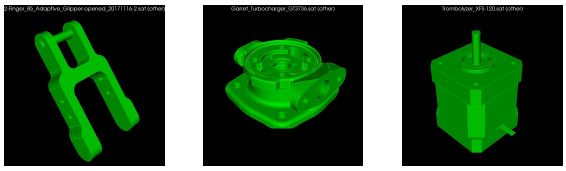
Pins



Gears



Other Parts



Note that the **Other Parts** category simply includes parts that cannot be categorized with sufficient confidence in any of the other categories.

Context Menus: To display all volumes in a specific category, select the part category title and use the right click context menu and choose **Draw**. Expanding the list will display individual volumes that can be visualized using standard draw tools such as **Zoom To**, **Fly In**, **Draw Overlapping Volumes**, **Draw Nearby Volumes**, etc. The context menu also provides access to command panel options such as **Delete** or **Reduce**. When multiple volumes are selected, the context menu can be used to invoke command panels to operate on multiple volumes simultaneously.

To view the volume features or the confidence values computed by ML, use the right click context menu when a part/volume is selected and choose either **List ML Features** or **List ML Predictions** menu option.

Surface Classification

This diagnostic provides the ability to classify surfaces. Surface categories provided with Cubit are currently limited to two categories: **slots** and **non_slots**. The limited categories are intended to be exemplars for additional user customized categires that can be defined using Cubit classify command. Similar to part classification, when selected, characteristic geometric features of each surface will be computed and ML methods will be used to determine its categorization. Each surface will be placed into a list based on its most probable categorization. Surfaces in each classification will be ordered based upon a Confidence metric. A confidence of 1.0 indicates a 100% confidence of categorization. Confidence values closer to 0.5 are less likely to be categrized correctly, but ML indicated a higher probablility than other part types.

Solution Predictions

Tetmesh Poor Quality Predictions

Once entities have been identified, custom solutions may be displayed by checking the **Show Solutions** check box above the list of entities. When

checked, the Solution window will appear as shown in Figure 4. When an entity is selected in the upper window, an ordered list of solutions will appear in the Solution window. Each solution has an associated metric prediction so that the solution predicted by ML to provide the best possible meshing result will appear at the top of the list.

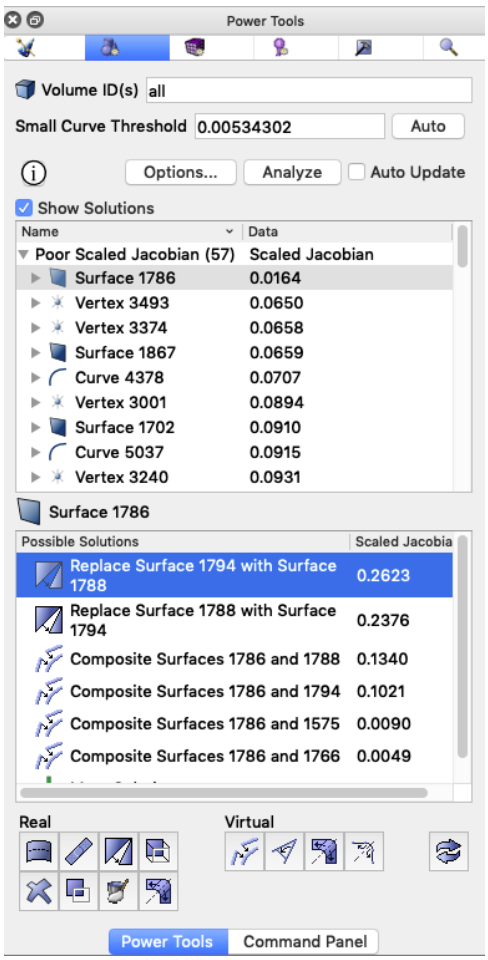


Figure 4. Geometry Power Tool showing Scaled Jacobian quality predictions as well as an ordered list of possible solutions.

For example, in Figure 4. Surface 1786 is selected which has a predicted minimum Scaled Jacobian metric at the surface of 0.0164. In most cases this metric would be unacceptable for analysis. To correct for this, the first solution in the list is a **Replace Surface** command that is predicted to result in a Scaled Jacobian of 0.2623. Selecting the solution will display a preview of the operation and double clicking will execute the solution. Other solutions may also be previewed and considered, however those with lower predictions would most likely be avoided.

Context Menus: A context menu is available when selecting a solution. This also provides an option for execution of the solution as well as additional visualization options. It also provides direct access to the appropriate command panel if further customization of the command is desired.

Controlling Solution Predictions: When selecting an entity from one of the metric categories (Scaled Jacobian, In-Radius, Deviation), the predicted value and ordering of the solutions will be based on the corresponding category. Note that solutions may also be generated for other diagnostics (ie. small curves, close loops, blend surfaces, etc.). When ML has been enabled, solutions will also be ordered these categories based upon the predicted tetmesh outcome. The metric used for ordering solutions can be customized using the **Options** panel shown in Figure 3. Use the **Show Solution Predictions** check box to toggle the use of ML-predictions to order solutions. When solution predictions are enabled, a drop-down selector will allow selection of one of the three

metrics to prioritize solutions.

Part Classification

When selecting a volume from a classification category, selected solutions will also be displayed. Custom solutions based upon a specific part type are still in development and will expand in future versions of the geometry power tool. In particular, the reduce family of commands are available when one or more volumes classified as a "bolt" or "spring" are selected.

Slot Surface Preparation with the Geometry Power Tool

This page describes the slot surfaces diagnostic tool that is part of Cubit's Geometry Power Tool. The Slot Surfaces diagnostic is a new addition to the geometry power tool for preparing slot surfaces in Electromagnetic (EM) modeling. A slot surface is used to identify potential pathways where EM radiation can potentially exit. This diagnostic will help manage slot surfaces and prepare them for the application of boundary conditions and analysis. This diagnostic utilizes Machine Learning (ML) to predict the most likely slot surfaces.

Background

The Geometry Power Tool in Cubit provides a series of diagnostic checks on your model used to defeature or simplify a CAD model prior to meshing. Clicking the **Analyze** button will perform the selected diagnostic tests and display an expanding tree listing geometric entities that are identified by each test. Once identified, suggested solutions can be easily previewed and executed.

The Slot Surfaces diagnostics work best in conjunction with the machine learning models. As such, to use this diagnostic the **Load ML Models** button must first be selected using the procedure described in the page Machine Learning with the Geometry Power Tool.

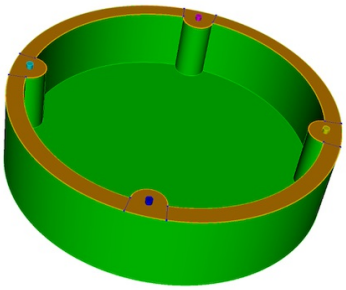


Figure 1. Example volume showing slot surface highlighted.

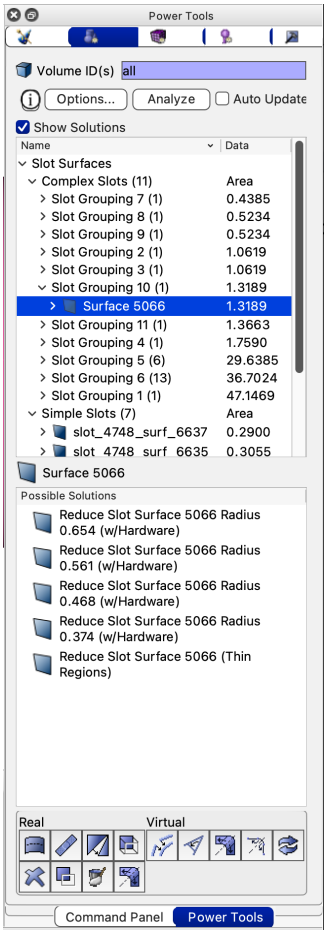


Figure 2. Slot Surfaces diagnostic displayed with Solution window.

Slot Surfaces

The Slot Surfaces diagnostic is only available when the ML Models have been previously loaded in the Options panel. Once the ML models are loaded, select the Slot Surfaces diagnostic from the list of diagnostic tools. When this diagnostic is selected, the analyze button will identify all surfaces that are identified as slots. The surfaces will be separated into two groups: simple and complex.

1. **Simple Slots:** Simple slots have four sides and can be used directly for representing slots without further decomposition.
2. **Complex Slots:** Complex slots can be comprised of groups of adjacent surfaces or can be a single surface that may require additional decomposition. For example, a gasket-type surface between two volumes that may be held together with bolts or another type of fastener. Each grouping of surfaces in the complex category can be expanded and individual surfaces selected. Selecting a surface will display potential reduce operations in the solution window.

Solutions Window

If the **Show Solutions** checkbox is selected, a list of potential decomposition strategies will be listed in the solutions window. The decomposition strategies will be variations of the **reduce surface slot** command. Selecting a solution will preview where cuts will occur and double-clicking will execute the cuts. If fasteners are present, the ML

tools should identify them and use them in the **reduce surface slot** commands presented in the solutions window.

Right-Click Menu Options

The slot surfaces diagnostics provide many of the same visualization options available with the other diagnostics. A few additional tools, useful for shell modeling, are also provided and described here:

Export Slot Data

This option supports the setup of a Morph input deck and will execute the command **export slot data**. Morph is an external meshing tool developed at Sandia. Morph supports slot surfaces for modeling EM, but requires named surfaces and curves to identify the correct topology. When selected, a file browser will appear where a filename can be specified. A text file will then be written with the necessary information needed for Morph input for slot surfaces.

List Slot Data

Executes the command **export slot data preview**. Similar to the **Export Slot Data...** menu option, rather than writing a text file, the information will be echoed to the Cubit output window.

Reduce Slot Surface

Available when selecting a solution in the solutions window. The command panel for reducing surface slots will appear. This panel provides options to input the hardware and radius information, automatically populating the necessary fields based on the slot geometry. Additionally, users have the ability to specify grouping and naming options for the decomposed slot surfaces.

Graphics Window

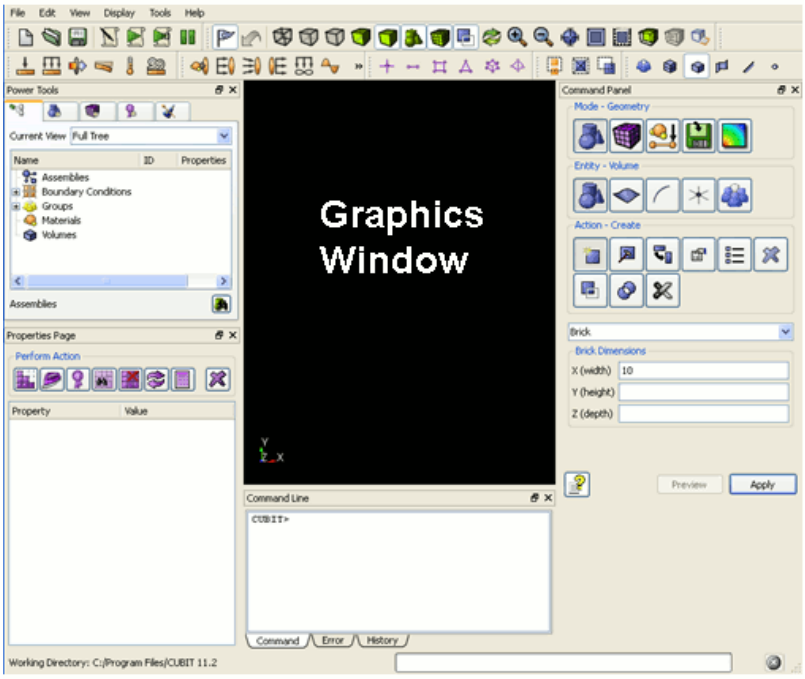


Figure 1. Graphics Window

The graphics window is used to view and select entities. Select one of the options below:

- View Navigation
- Selecting Entities
- Key Press Commands
- Right Click Commands for the GUI Graphics Window
- Viewing Curve Valence

Viewing Curve Valence

To view your model based on a color-coded curve valence scale, click on the curve valence button on the Display Toolbar. Curve valence refers to the number of surfaces attached to each curve. Curves with exactly two surfaces attached are shown in blue. Curves with exactly one surface are shown in red. Curves with more than two attached surfaces are shown in white.

This tool is useful for quickly visualizing merged/unmerged topology. Merged curves will usually have a valence > 2, while unmerged curves typically have a valence of 2. Curves with a valence of 1 may indicate a floating surface.

Key Press Commands for the GUI

Several commands have a key press shortcut. These commands will be executed with respect to the currently selected entities; see the following table:

Shortcut Key	Command
l	List information about the current entity to the output window.
i	Toggle the visibility of the selected entity (make invisible or visible).
e	Echo entity id to command line.
Tab	Select the next entity.
Shift Tab	Select the previous entity.
0	Toggle picking of vertices.
1	Toggle picking of curves.
2	Toggle picking of surfaces.
3	Toggle picking of volumes.
4	Toggle picking of groups.
Shift **0**	Toggle picking of mesh nodes
Shift **1**	Toggle picking of mesh edges.
Shift **2**	Toggle picking of mesh faces.
Shift **3**	Toggle picking of mesh hexes.
F5	Refresh graphics window
Shift **S**	Activate/inactivate graphics clipping plane

Right Click Commands for the GUI Graphics Window

Clicking the Right mouse button in the graphics window will bring up a menu. One of two menus will appear, depending on whether an entity is currently selected.

With Entity Selected

When an entity is selected, the options available will depend upon the type of entity selected. The following describes the menu options and when they are available.

Entity Selections

Menu Option	Description	Entity Type(s)
Select Other...	Brings up a dialog with alternate entity selections	All
Select Similar Volume	Selects other volumes with the same geometric volume and number of surfaces	Volumes
Select Similar Surface	Selects other surfaces with the same area and number of curves	Surfaces
Select Blend Chain	Selects other surfaces in the same blend chain that have the same radius of curvature	Blend surfaces
Select Cavity	Selects other surfaces in the same cavity collection. Surfaces bounded by curves where the external angle is greater than 180 degrees.	Cavity surfaces
Select Similar Curves	Selects other curves with the same length	Curves
Select Enclosure	Selects all connected surfaces to the current selection(s) on the same volume	Surfaces
Select Between	If at least two surfaces (or collections of surfaces) are selected, all connected surfaces situated between the two surfaces will be selected.	Surfaces
Select Similar Curves	Selects other curves with the same length	Curves
Select by Name	Selects other entities with the same name. Includes entities with the same name prefix prior to the "@" character if it exists.	Geometry entities
Pick Extended...	Brings up dialog for extended selection. Related entites can be selected using custom criteria defined from a python script.	Surfaces

Entity Visualization

Menu Option	Description	Entity Type(s)
Zoom To	Zoom to the selected entity	All

Rotate About	Center of rotation is changed to the centroid of the selected entity(s)	All
Locate	Text label with line pointing to entity is displayed	All
Draw	Clears the graphics window and draws the selected entity(s)	All
Draw Elements	Brings up a secondary menu with options to Draw specific 3D mesh entity types (without geometry).	Meshed Volumes
Draw Normal	Displays an arrow from the center of the selected surface in the direction of its normal. Also colors surface indicating orientation.	Geometry Entities
Draw With Neighbors	Clears the graphics window and displays the selected entity(s) along with entities sharing a common vertex	Geometry Entities
Draw With Nearby Volumes	Draws the selected volume(s) along with volumes that are within a small distance	Volumes
Draw Volume Overlap	Draws the curves in the selected volume(s) as well as any volume it overlaps. The region shared between the overlapping volumes will be displayed in red	Volumes
Isolate	Turn off visibility for all entities except the selected entity(s)	Geometry Entities
Visibility Off	Turn off the visibility of the selected entity(s)	Geometry Entities

Entity Operations

Menu Option	Description	Entity Type(s)
Add to Group/BC	Opens a dialog box for adding the selected entity to an existing boundary condition or group	All
Remove from Group/BC	Opens a dialog box for removing the selected entity from an existing boundary condition or group	All
Mesh	Mesh the selected entities using the current meshing scheme and size settings	Unmeshed Geometry Entities
Delete Mesh	Deletes the mesh from the selected entities using the "propagate" option. (Deletes mesh entities on lower order entities.)	Meshed Geometry Entities
Show Quality	Displays the mesh quality of the selected entities to the output window.	Meshed Geometry Entities
Measure	When a single entity is selected, displays to the output window the geometric length, area or volume. When two entities are selected, displays the closest distance between the entities.	Geometry Entities
List Information	Show a menu with additional options for listing information about the selected entities. **List Basic Info:** Lists connectivity and associated lower order entities. **Geometry:** Basic info plus additional geometric information such as centroid, bounding box, surface area etc. **List Mesh Info:** Basic info plus additional information about the associated mesh.	All

Save Selection As...	Brings up a File Browser to save the currently selected entities as a cub file.	Geometry Entities
Delete	Deletes the selected geometry entities	Free Geometry Entities
Remove	Removes a surface using the **remove surface** command.	Surfaces in Volumes

Without Entity Selected

Menu Option	**Description**	**Entity Type(s)**
Refresh Display	Clears temporary graphics a refreshes display of current geometry and mesh.	All
Draw Mesh	Displays any existing mesh entities without geometry.	Meshed Entities and Free Mesh
Reset Zoom	Reset graphics viewing options to original settings.	All
All Visible	Reset visibility status to ON for all entities and display.	All
Display Options...	Opens the Display Options Dialog to set graphics window characteristics such as background color, lighting, triad options etc.	All

Selecting Entities in the GUI

Geometry, mesh entities, and boundary conditions can be selected with the left mouse button directly in the graphics window. Before selecting any entity, however, the correct selection mode must be chosen. This dictates which entity types will be available for selection in the graphics window. The *Select* Toolbars, which are located above the graphics window by default, are used to change the entity selection modes.



Figure 1. The Selection Toolbars for Geometry and Mesh Entities

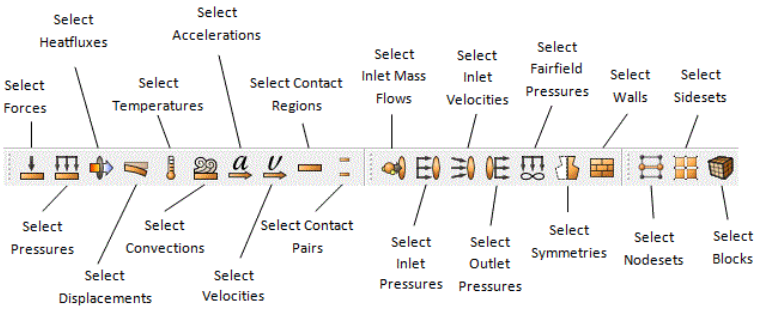


Figure 2. The Selection Toolbar for Boundary Conditions

Figures 1 and 2 shows the selection toolbars. Selecting one of the entity selection modes will only permit selection of that particular entity type within the graphics window. These selections will not override a Pick Widget in the command panel.

If both volume and surface entities are picked on the select toolbar, a single click will select the surface while a double click will select the volume. More detailed information on selecting and specifying entities can be found in Entity Selection and Filtering .

Pre-Selection

When the mouse cursor is over an entity type that has been selected from the Pick toolbar, that entity will become highlighted. This is called **pre-selection** and is used as a graphical guide to show which entity will be picked when the mouse button is clicked.

Graphics pre-selection may slow down your graphics speed for large models. You can disable pre-selection from the Tools->Options dialog box.

Polygon, Circle and Box Select

The polygon/circle/box selection feature allows you to select entities by drawing a box, circle or polygon on the screen. To create a box or circle selection, press and hold the <CTRL> button* while clicking and dragging the left mouse button. Release the left mouse to complete the box or circle select. To create a polygon selection, press and hold the <CTRL>* button while clicking and dragging the left mouse button. Click the left mouse button to create another side for the polygon. Press either of the other buttons to close the polygon and complete the selection. Only entities that are in active selection mode will be selected. To change between the polygon, circle or box method, press the *Toggle Between Polygon/Box/Circle Select* button on the Select Toolbar. Clicking the *Toggle Selected Enclosed/Extended* button will toggle between Enclosed

Selection and Extended Selection. Enclosed selection will only select entities that are fully enclosed within the bounding box, circle or polygon. Extended selection will select entities that are either fully OR partially enclosed within the bounding box. Toggling the the *Select X-Ray* will select entities that are hidden behind other entities. X-ray selection will only apply to smoothshade and hiddenline graphics modes.

***Note:** For Mac computers use the command (or apple) button for polygon or box select.

View Navigation in the GUI

There are two different default paradigms for view navigation: Cubit command line and Cubit GUI. The user is allowed to customize the mouse settings as desired. Mouse settings in the GUI are modified by accessing the **Tools** pull-down menu, then select **Options**. The Mouse Settings dialog is shown below (See Mouse-Based Navigation for the command line version).

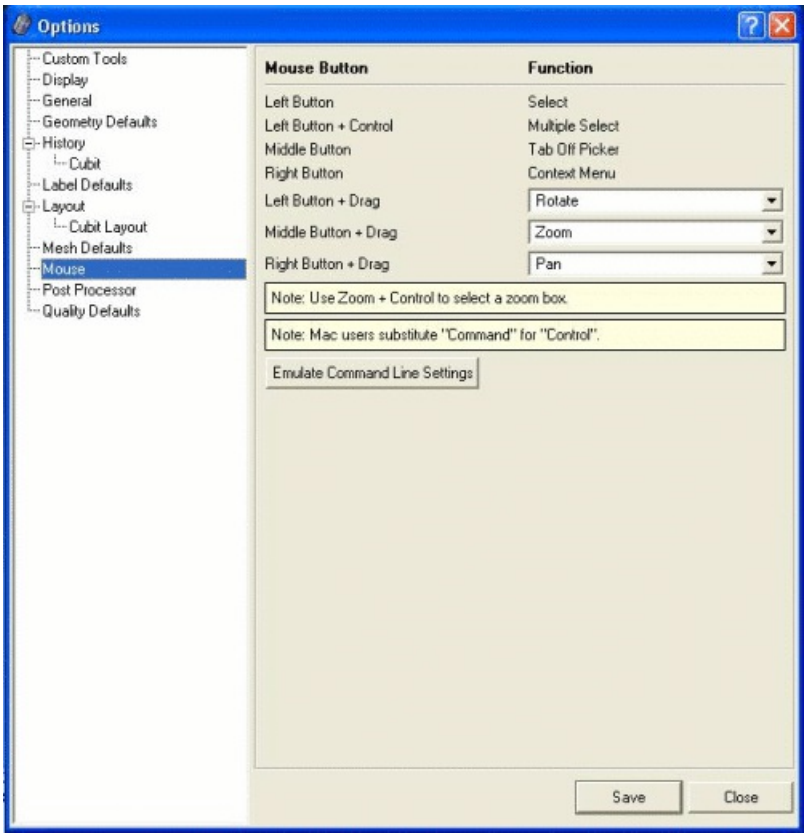


Figure 1. Mouse Settings Dialog

Rotations

Where the cursor is in the graphics window will dictate how the view will be rotated. If the cursor is outside of an imaginary circle, shown in Figure 2, the view will be rotated in 2d, around an axis normal to the screen. If it is inside the circle, as in Figure 3, the rotations will be in 3d, about the current view spin center. The spin center can be changed to any x-y-z location. The most common way is by zooming to an entity, which changes the spin center to the centroid of that entity. The "view at" command will change the spin center without zooming:

> **View at vertex 3**

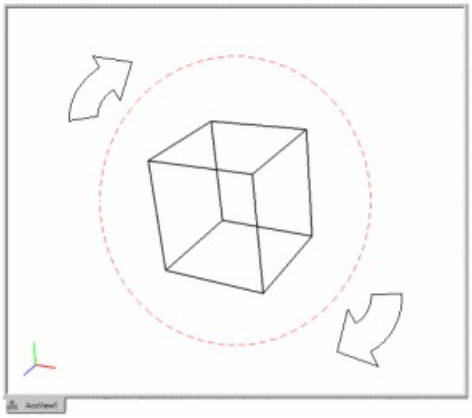


Figure 2. With the mouse pointer outside the circle the view is rotated about an axis normal to the screen

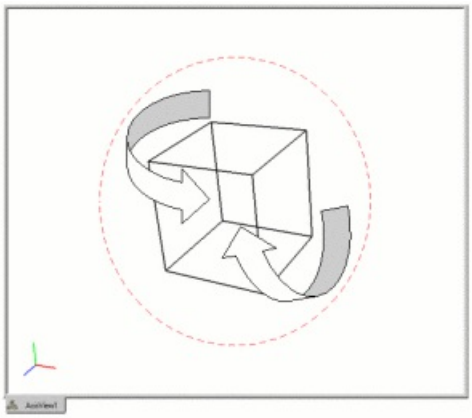


Figure 3. With the mouse pointer inside the circle the view is rotated about the current spin center

Zooming

To zoom, press the appropriate buttons or keys and move the cursor vertically, as shown in Figure 4. The wheel on a wheel mouse will also zoom.

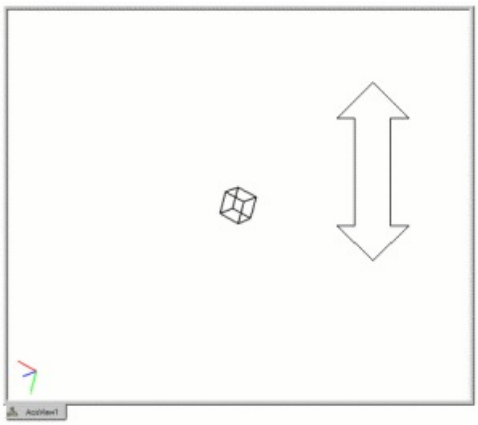


Figure 4. Move the mouse pointer vertically to zoom in and out

Panning

To pan, press the appropriate buttons or keys and move the cursor horizontally or vertically, as shown in Figure 5.

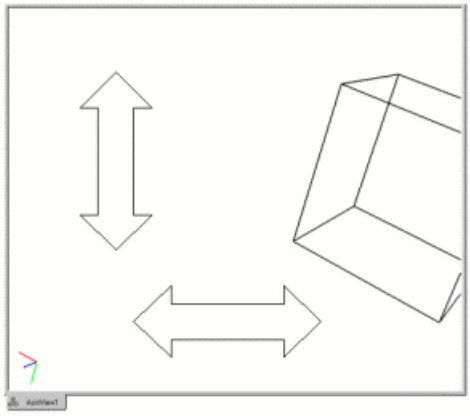


Figure 5. Move the mouse pointer horizontally or vertically to pan the view

Drop Down Menus

The Cubit Drop-Down Menus, located at the top of the Cubit Application Window provide access to capabilities such as file management, checkpoints, display manipulation, journaling, system setup, component management, window management, and help.

Cubit (Mac Only)

This menu contains the Preferences dialog box, also called the Options dialog box on other platforms. It also contains the About Cubit menu and the Quit Cubit option. It is only available on Mac computers.

File

This menu provides common file operations, including importing and exporting of geometry and meshimport and export. A list of recently saved or imported files is also provided, allowing a quick way to import current or recent work. Non-Mac users can also exit and reset the program from this menu (These options are found under the Cubit tab for Mac Users).

Edit

This menu only provides a way to enable the Undo feature of the system. If Undo is enabled, one level of Undo is available to the user.

View

The View Menu lists all available toolbars and windows in the current CUBIT session. Selecting a toolbar or window will make it visible. Deselecting a toolbar or window will hide it. You can also hide an undocked window or toolbar by clicking on the small "x" in the upper right corner. For more information on docking and undocking toolbars, see CUBIT Application Window.

Display

The Display Menu controls display options for the graphics window. These options are explained below:

- **View Point** - Controls the camera view point. Choices are front, back, top, bottom, right, left and isometric views.
- **Render Mode** - Controls visibility modes, including: wireframe, true hidden, hidden line, transparent, and shaded.
- **Geometry** - Controls geometry visibility
- **Mesh** - Controls mesh visibility
- **Graphics Composite** - Controls the visibility of composited entities in the graphics window.
- **Refresh** - Updates the graphics display
- **Background** - Changes the background color
- **Zoom In** - Enlarges the model in graphics window
- **Zoom Out** -Shrinks the model in graphics window
- **Zoom To Fit** - Enlarges or shrinks model in the graphics window so it fills the whole screen
- **Toggle Perspective** - When this option is selected, the entities in the graphics display window are drawn in perspective mode.
- **Toggle Scale** - Turns on or off a graphical scale that can be drawn in the graphics window to obtain a bearing on model or part sizes.
- **Toggle Clipping Plane** - Turns on or off the graphics clipping plane
- **Toggle Clipping Plane Manipulation** - Turns on or off manipulation of the graphics clipping plane

- **Show Curve Valence** - Turns on or off the curve valence highlighting

Tools

The Tools Menu contains access to GUI-specific tools and options. These options are explained below.

- **Journal Editor** - Opens journal file editor. The Journal Editor is used to write, edit, play, and save journal files. It can also be used to create and edit Python scripts. A built-in translator will convert between the two files types.
- **Play Journal File** - Plays a specified journal file. You can browse through files and folders on your computer to select the journal file to play.
- **Options** - Opens the Option dialog box. This dialog box controls all of the preferences for the GUI including display colors and widths, mouse settings, journal file options, mesh and geometry defaults, and general layout preferences. MAC users can find this menu under the Cubit tab.
- **Components** - Opens the Components dialog box. This window is used to load and unload external and internal components.

Help

- **Tip of the Day** - Open the tip of the day box.
- **Cubit Tutorials** - Opens a menu of step-by-step tutorials for Cubit.
- **Cubit Manual** - Menu to bring up on-line searchable documentation (this document).
- **About** - Menu to show the current version number and trademark information. Mac users can find the version number under the About Cubit menu in the Cubit drop-down.

Options Panel

To change program preferences in the Graphical User Interface select: **Tools > Options**. On a Mac OS, the equivalent panel is called **Preferences**, and is available from **Cubit > Preferences** The options panel includes a series of Tabs, containing categories of global settings to customize the Cubit environment. The following categories are available:

- Command Panels
- Display
- General
- Geometry Defaults
- History and Cubit Journalling
- Label Defaults
- Layout
- Mesh Defaults
- Mouse Settings
- Post Processor
- Quality Defaults

Command Panels

This menu controls how command panels are displayed and managed, including which style of button hierarchy is displayed.

- **Command Panel Behavior**
  - Automatic Input Field Focus -
  - Automatic Panel Reset
- **Panel Warnings**
  - Hide Mesh Warning - Hide the warning dialog that appears when trying to mesh entities that already contain a mesh.
- **Navigation Hierarchy**
  - Classic (Restart Required) - Toggle between entity-first hierarchy (Classic) and action-first hierarchy (Non-classic). This changes the arrangement and navigation of panels in the geometry mode of the command panel. The classic navigation arranges the icons so that entities are first selected, followed by an action. For the non-Classic mode, the action is first selected followed by the entity type. A restart is required to see changes in the command panel navigation hierarchy.
- **Boundary Condition Modules** - Disable/Enable the display of GUI tools used for managing commercial FEA and CFD model preparation. These options are available only in non-classic navigation hierarchy. Uncheck the **Classic** checkbox to enable these options.
  - FEA (Restart Required) - Enable or Disable the FEA GUI tools
  - CFD (Restart Required) - Enable or Disable the CFD GUI tools

Display Preferences

This menu controls entity display features for the graphics window which include the following:

- Display Triad in Graphics Window
- Enable Pre-Selection
- Background Color
- Perspective Angle
- Line Width

- Highlight Line Width
- Text Size
- Ambient Intensity
- Ambient Color
- Light Intensity
- Light Color

General Preferences

This menu controls general program options including the following:

- **Prompt for Unsaved Application Data** - When this is checked and the user opens a new .cub file or exits the application with unsaved changes, a dialog box will pop up asking if they want to save changes first. The user can uncheck this option to prevent that dialog box from appearing. This is checked by default.
- **Prompt for Unsaved Journal Data** - When this button is checked and the user closes the journal file editor with unsaved changes the program will prompt to save the changes. The user can uncheck this button to prevent the dialog box from appearing. It is checked by default.
- **Change to Script Directory for Playback** - When this option is checked, Claro will change the working directory to the directory the script is in when the script/journal file is run. When the script is finished, Claro will change the directory back to the previous one. This is useful when using relative paths in a journal file. When the option is unchecked, Claro won't change the directory when a journal file is run in which case the user may have to manually change the working directory when their journal file has relative paths.
- **Prompt When Translating from Python -** When checked, if the user translates a python script to a cubit journal file, the journal editor will warn them that commands may be lost. When unchecked, the journal editor will not issue the warning. There is a checkbox on the warning dialog that sets this option as well.
- **Default Syntax** - Sets the default syntax to use when creating a new journal file in the editor. The Cubit option is only available when the cubit component is loaded.
- **Show Startup Splash Screen** - Option to hide the startup splash screen on opening Claro.

Geometry Defaults

This menu controls the geometry defaults.

- Vertex Size
- Use Silhouette on Geometry
- Silhouette pattern

The user can also change the default geometry engine to one of the following:

- ACIS
- Facets

The faceting tolerance can also be controlled from this menu to change the way facets are drawn in the graphics window.

History Preferences

This menu controls the input window history and journal file options. These include:

- **Maximum Number of Commands** - The max number of commands kept in the current command history.
- **Maximum Number of Lines** - Maximum number of lines in input

window.
- **Journal Command History** - Whether to use a journal file to save command history. Default is to use a journal file.
- **Output Log** - When this option is checked, you can save error log to a separate output file.

Label Defaults

This menu controls the geometry and mesh entity labels in the graphics window.

- Text Size
- Label Geometry and Mesh Entities Toggles- Choose label visibility for each type of geometry or mesh entity

Layout Preferences

This menu option controls input window formatting and control panel docking options.

- **Font for command line workspace**
- **Font size for command line workspace**
- **Reset Window Layout Button** - Used to reset GUI windows to their default positions

Also included in the layout preferences is a list of available windows with a checkbox to show/hide each window.

Cubit Layout Settings

This menu controls the layout of Cubit specific buttons and tabs on the GUI.

- **Show script tab** - Shows the script tab on the command line window
- **Use Labels on Buttons**- Option to apply a label to each button on the control panel
- Preferred Location (currently under construction)

Mesh Defaults

- **Node Size**
- **Element Shrink**
- **Mesh Line Color** - The same as "Color Lines" command.
- **Default Element Type** - Tet/Tri or Hex/Quad
- **Surface Scheme Coloring** (used in Meshing Power Tool) - This option allows you to select different colors for surface schemes when visualized using the meshing power tools.

Mouse Settings

This menu controls mouse button controls. Pressing the **Emulate Command Line Settings** button will cause all of the settings to simulate mouse controls in the command line version of CUBIT. For a detailed description of mouse settings see the **View Navigation-GUI** page.

Post Processor Settings

Post Processor Executable Directory - Option to browse for post processor executable directory.

Quality Defaults

This menu controls quality defaults for different quality metrics. For a

description of the different quality metrics see the respective pages:

- Hexahedral metrics
- Quadrilateral metrics
- Tetrahedral metrics
- Triangular metrics

Undo Button

Cubit has an undo capability. To enable the Undo feature click on the "Enable Undo" button on the Toolbar.

 **Enable Undo Button**

Alternatively to turn undo on and off, the following command may be used in the command line:

undo {on|off}

The Undo capability is implemented for geometry commands including webcutting, geometry creation, transformations, and booleans. Multiple undos are also allowed. The commands will be undone in reverse order of their execution.

Limitations

- The undo button is not currently enabled for most meshing commands

Command Panels

The Command Panels provide a graphical means of accessing almost all of the CUBIT functionality. The main CUBIT Command Panel is divided into seven modes. Each of these modes pertains to a major component of the CUBIT application. To view information about each of the tools in the Control Panel select the help icon on each panel to access context specific help.

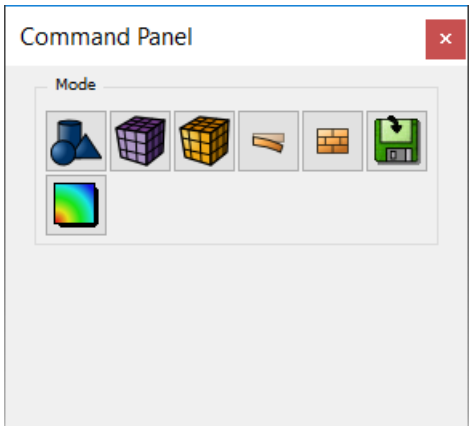


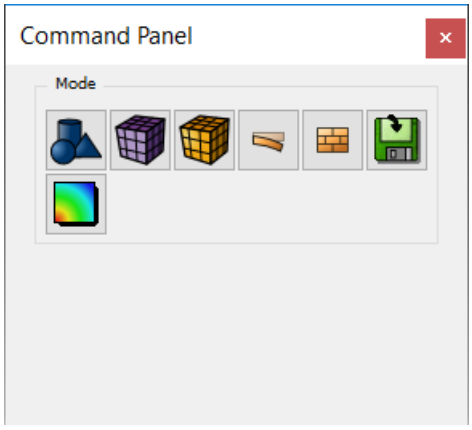
Figure 1. The CUBIT Control Panel

A brief description of the functionality of the Control Panel window follows.

Control Panel Functionality

Command Panel Functionality

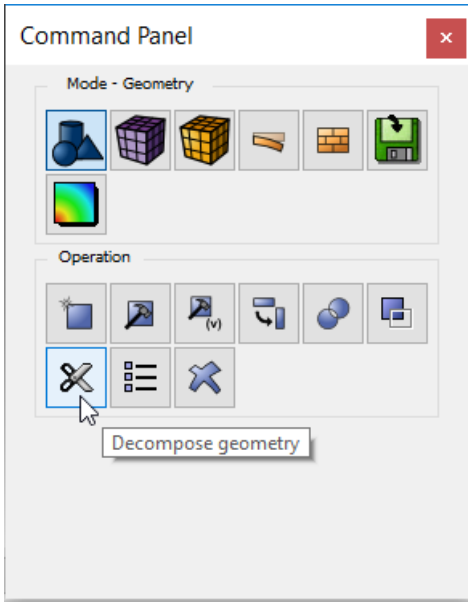
The Command Panel is arranged first by mode on the top row of buttons. Modes are arranged by task.



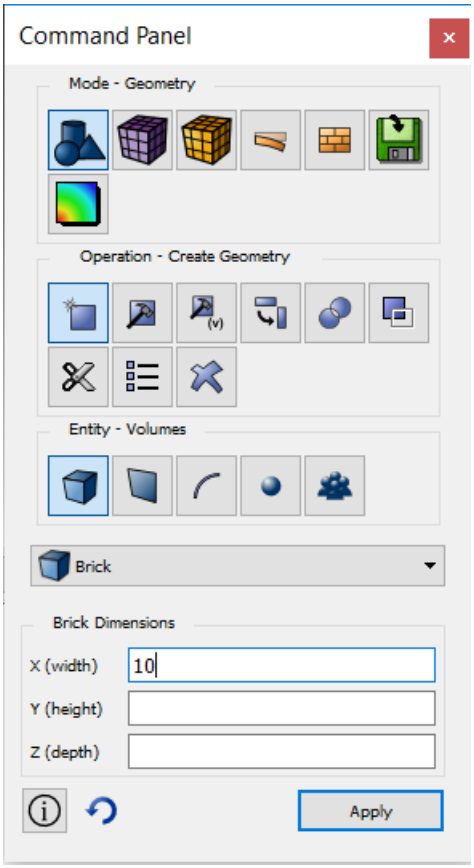
The modes are:

- Geometry
- Mesh
- Analysis Groups and Materials
- FEA Boundary Conditions
- CFD Boundary Conditions
- Analysis Setup (Export)
- Post-meshing Tool Launch

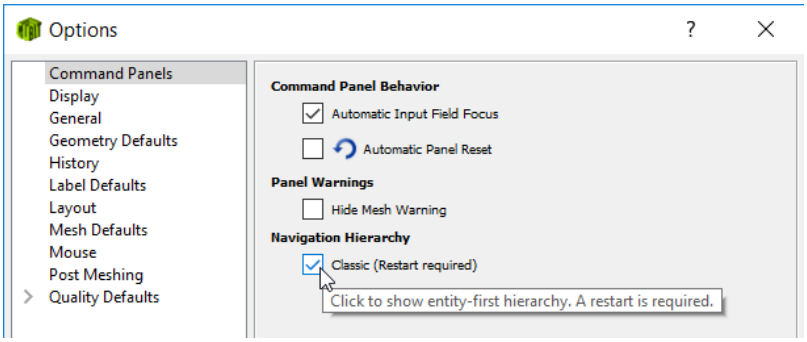
All of the geometry related tasks, for instance, can be found under the Geometry mode. When a mode is selected, a second row of buttons becomes available. The second row of buttons shown depends on the selected mode. For example, if Geometry is selected, the second row of buttons will contain operations that can be performed on geometry, such as creation, modification, decomposition, boolean, merging, deleting, and so forth.



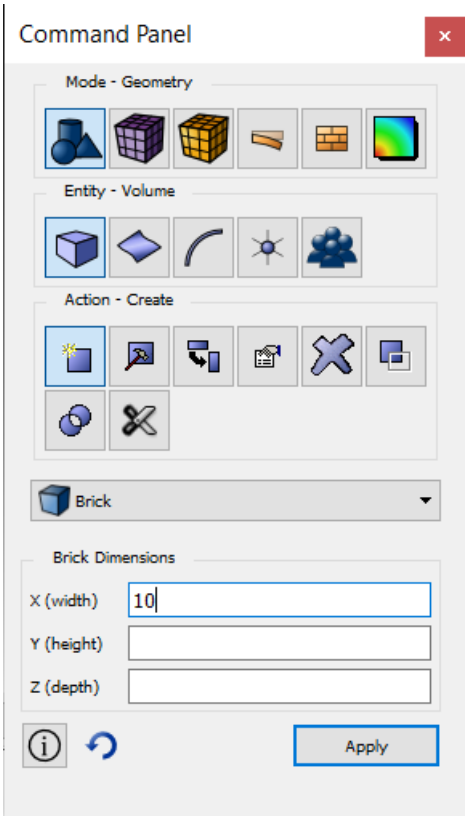
Selecting an operation will cause a row of geometry entity buttons to be displayed. Specific, entity-based operations will be shown after selecting the entity type, as shown below.



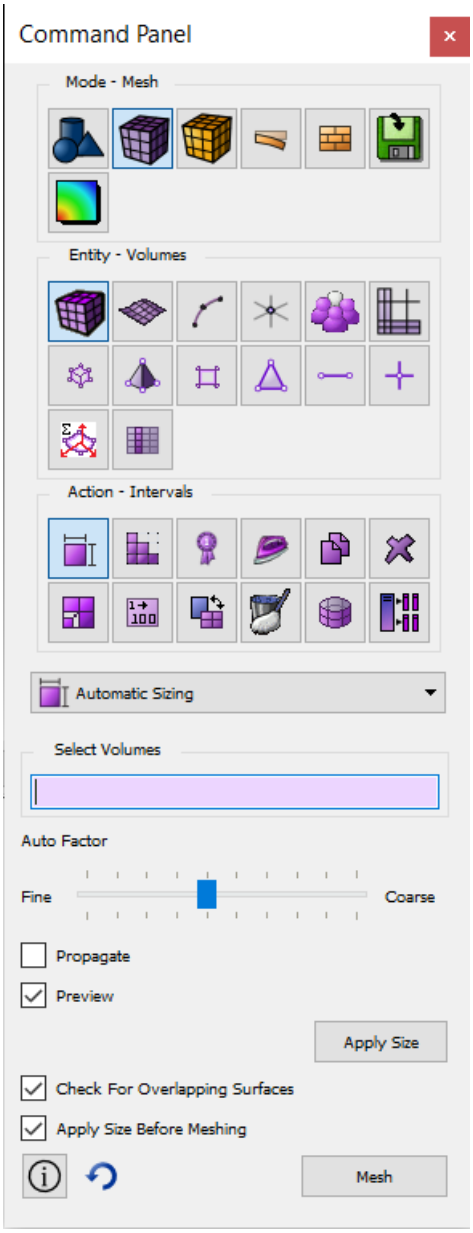
Note: This hierarchy is different than previous versions of Cubit. In previous versions the second row of buttons was geometry entity types, such as volume, surface, curve, vertex, and group. If a user wishes to use the 'classic' hierarchy for geometry, an option can be selected in Tools/Options/Command Panels.



The 'classic' hierarchy's button hierarchy for creating a solid brick is the following:



For all other modes, such as mesh, analysis groups, and so forth, the 'classic' button hierarchy remains.



All command panels are constructed similarly. Each abstracts a set of Cubit commands. Options are selected using check boxes, radio buttons, combo boxes, edit fields, and other standard GUI widgets. Each command panel includes an Apply button. Pressing the Apply button will generate a command to Cubit. Nothing happens until and unless the Apply button is pressed.

> **Note: The edit fields are free form, which means the user may enter any valid string into the fields. Any string that is valid for the command line is valid for the command panel edit fields.**

Where possible, default values are placed into edit fields. At any time, with the cursor placed over a blank portion of the command panel, the user may right-click to select Reset Data which will clear all fields and replace default values.

ID Input Entry Methods

The *ID Input Fields* provide a location where Geometric IDs, required for the current command, can be entered. IDs can be entered in several ways:

Simple Keyboard entry

ID numbers can be entered directly in the field. Each ID must be separated with a space. Select the field first before typing.

Graphical selection

IDs can be entered automatically by selecting entities directly in the Graphics Window. The current entity available for selection is based on the current entity selection mode. In some cases, not all entities of the current entity selection mode will be available for picking. The program may automatically filter the applicable entities based on the context of the current command

Geometry Tree selection

IDs may be entered by selecting the corresponding geometric entity from the geometry tree. To select multiple entities use the **<ctrl>** key.

Ranges

A range of IDs may be typed into the field. For example:

> **1 to 5**

will automatically enter all IDs from 1 to 5 inclusive in the field. Keywords such as **all** and **except** can also be used. Any range that can be entered directly on a CUBIT command line can also be used in the ID input field. See Command Line Entity Specification for a description of the syntax.

As Part of Other Entities

Syntax can be entered in the *ID Input field* that will specify an entity based upon its topological relationship to other entities For example, if a **Vertex** Selection Type Button was highlighted, entering

> **in surf 1**

will automatically enter all vertices in surface 1 into the *Input Field*. CUBIT has a rich set of syntax rules for specifying entities based upon topology relationships. See Command Line Entity Specification for a description.

In Groups

Entities that are part of groups may be specified in the ID Input Field. For example, if the Vertex Selection Type Button is highlighted, entering:

> **in picked**

will automatically enter all vertices in the picked group into the active *ID Input Field*.

Dragged and Dropped

Entities can be dragged and dropped into the *ID Input Field* from the Tree View window.

Right-Click Context Menu for ID Input Fields

When the right mouse button is selected while in an *ID Input Field*, the following menu options will appear:

- **Done Selecting** - Enters current selection and removes cursor from selection window
- **Select Other** - Displays selection dialog
- **Select All** - Selects all available entities and puts "select all" in input window
- **Highlight -** Highlight the current selection
- **Zoom To** - Zooms to current entity in the selection field within the graphics window

- **Rotate About** - Change center of rotation to the center of selected entity
- **Draw** - Draws the entities listed in the input field within the graphics window
- **Isolate** - Turns visibility off for all entities other than the selected entities. Similar to draw command, but entities remain hidden with a graphics refresh. Select **All Visible** in the graphics window to turn visibility back on.
- **Visibility Off** - Removes the current entity from the input window and hides it on the graphics screen
- **Mesh** - Mesh the listed entities using either an assigned scheme or a default scheme where none is assigned
- **Delete Mesh** - Deletes mesh on all entities listed in the input window
- **Reset Entity** - rehighlights the entities listed in the input field within the graphics window
- **List Info** - Displays a sub menu of choices including basic, geometry, and mesh. Selecting the basic option will list schemes, visibility, and interval assignments. The geometry option will add information about the geometry and geometry engine. The mesh option will list information about mesh entities.
- **Delete** - Deletes the current geometric object in the input window.

Value Fields

Integer and real values pertinent to the command are entered in this window. Input placed in parenthesis { } will be evaluated when the command is executed. For example:

```
{10*0.02}
```

is valid input. Additionally, any APREPRO syntax is valid in the *Value Field*, including mathematical functions and boolean operations. See the section, APREPRO for a description of syntax.

Advancing Pickwidgets

Some command panels have several id input fields such as the Mesh>Hex>Create panel. A convenience feature implemented for such panels is an advancing pickwidget feature. Pressing the middle mouse button after selecting an entity will advance to the next id input field.

CUBIT Application Window

The default CUBIT Application Window is shown in the following image.

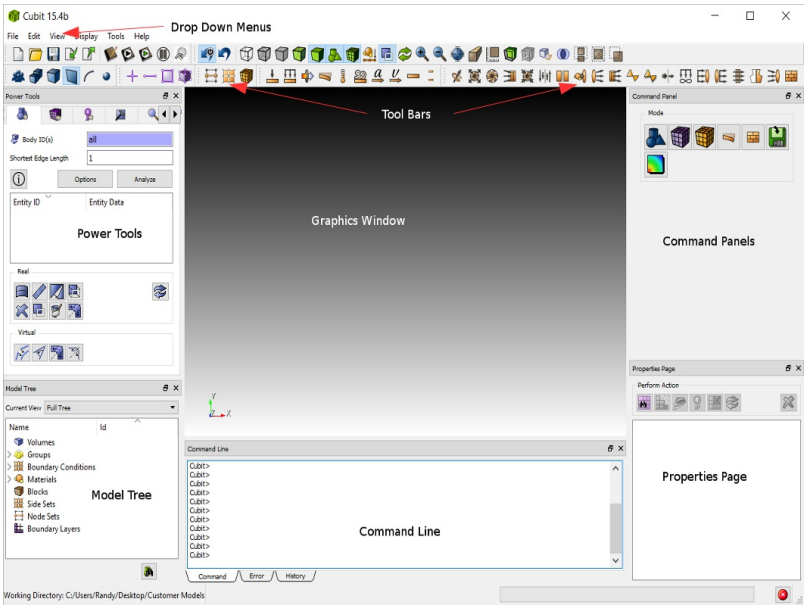


Figure 1. The CUBIT Application Window

Graphics Window- The current model will be displayed here. Graphical picking and view transformations are done here.

Power Tools - Geometry tree hierarchy view, geometry analysis and repair tool, meshing tool, meshing quality tool, and ITEM Wizard.

Property Editor - The Property Editor lists attributes of the current entity selection. Some of these properties can be edited from the window.

Command Panel - Most Cubit commands are available through the command panels. The panels are arranged topologically, by mode.

Command Line Workspace - The command line workspace contains both the cubit command and error windows. The command window is used to enter cubit commands and view the output. The error window is used to view cubit errors.

Drop Down Menus - Standard file operations, Cubit setup and defaults, display modes, and other functionality is available in the pull-down menus.

Toolbars - The most commonly used features are available by clicking toolbar icons.

Context Sensitive Help in the GUI

The Graphical User Interface has a context-sensitive help system. To obtain help using a specific window or control panel, press F1 when the focus is in the desired window. It may be necessary to click inside a text box to switch focus to a particular window. If no context specific help is available, it will open the cubit help documentation where you can search for a particular topic.

Customizing the Application Window

All windows in the CUBIT Application can be *Floated* or *Docked*. In the

default configuration, all windows are docked. When a window is *docked* the user can click on the area indicated below.

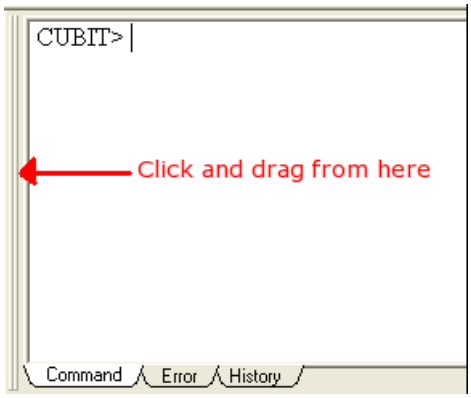


Figure 2. A docked window. Click and drag to float.

By dragging with the left mouse button held down, the window will be un-docked from the Application Window. Dragging the window to another location on the Application Window and releasing the mouse button will cause it to dock again in a new location. The bounding box of the window will automatically change to fit the dimensions of the window as it is dragged. Releasing the mouse button while the window is not near an edge will cause the window to Float. To stop the window from automatically docking, hold the CONTROL key down while dragging.

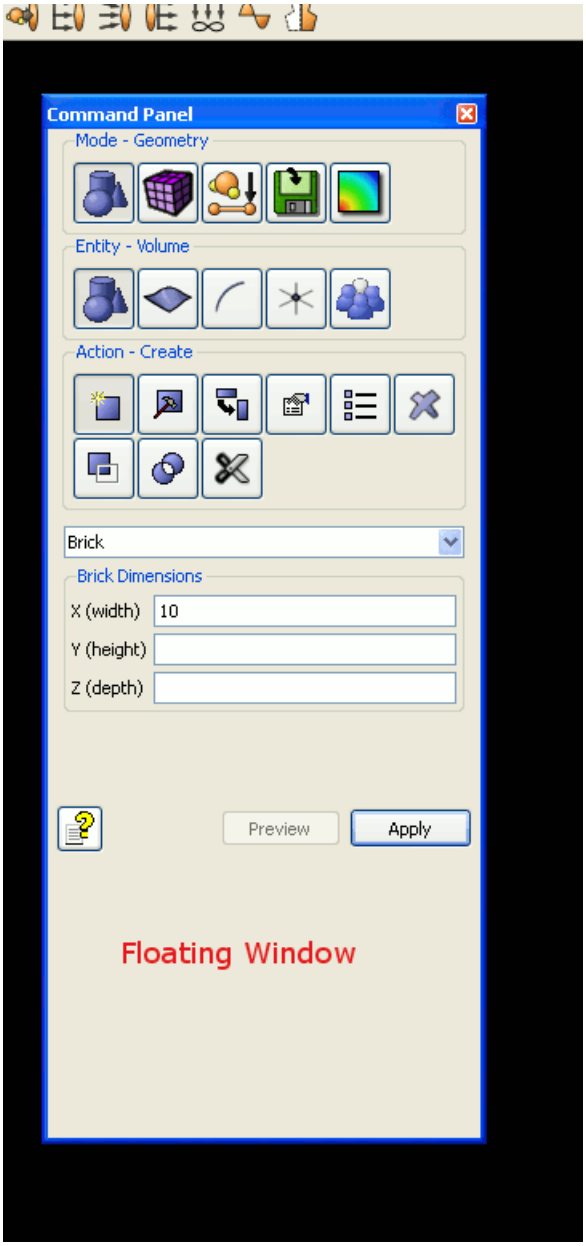
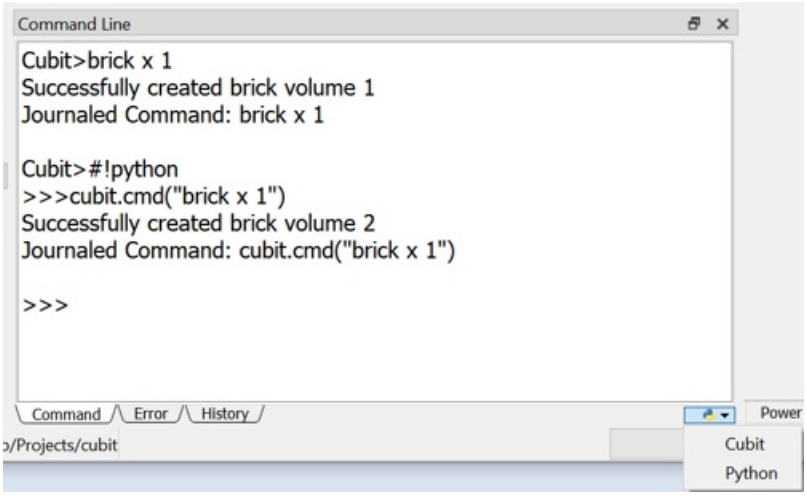


Figure 3. A Floating Window

When a window is *floating*, as shown in Figure 3, it is possible to dock it by clicking the title bar of the window and dragging it to its new docked location.

Note: Double clicking on the title bar of an floating window will cause the window to *redock* in its last docked position.

Command Line Workspace



The Command Line Workspace is the interface for command interaction between the user and the CUBIT application. The user can enter commands into this window as if they were using the command line version of CUBIT. Journaled commands will be echoed to this screen, even if they were not typed in manually. Thus, if the user wants to know what the command sequence for a particular action on the GUI is, they can watch for the "Journaled Command:" line to appear. In addition, this screen will contain important informational and error messages. The command window has the following three tabs:

1. Command
2. Error
3. History

Command Window

The command line workspace emulates the environment in the command line version of Cubit. Commands can be entered directly by typing at the **CUBIT>** or **>>>** prompt. This window also prints out error messages, informational messages, and journaled commands.

Entering Commands

To enter commands in the command line workspace, the command window must be active. Activate the command window by clicking anywhere inside the window. Commands are typed in at the **CUBIT>** or **>>>** prompt. Cubit commands are accepted with the **CUBIT>** prompt and Python statements are accepted at the **>>>** prompt. If you do not remember the specific Cubit command sequence you can type **help** and the name of the command phrase. The input window will show all of the commands that contain that word or phrase. Alternatively, if you know how a command starts, but do not remember all of the options, you can type **?** at the end of the command to show all possible command completions. See Command Syntax for an explanation of command syntax rules.

Command Mode

The command window can be in either a Cubit command mode or a Python statement mode. Switching modes can be done with **#!cubit** to switch to the Cubit command mode or with a **#!python** to switch to the Python statement mode. A convenience button is also available in the lower right corner of the command window to switch modes with a click

of the mouse button.

Python Statements

The Python interpreter in Cubit works as though you were entering lines at the Python command prompt. This means a blank line is interpreted as the end of a block. If you want to add whitespace for clarity, you could add a # mark for a comment on any white line that is in a loop or a class.

A second python script can be included and executed by either using the **play** Cubit command or by using a Python **import** statement.

The interface between cubit and Python is the 'cubit' object. Among many methods, this object has a method called **cmd** which takes as an argument a Cubit command string. Thus the following command can be issued in the Command window at the **>>>** prompt, which will create a cube with sides 10 units long.

```
cubit.cmd("create brick x 10")
```

The following script is a simple example that illustrates using loops, strings, and integers in Python.

```
%>for i in range(4):
. . x=i*3
. . for j in range(4):
. .    y=j*3
. .    for k in range(4):
. .      z=k*3
. .      mystr="create vertex x "+str(x)+" y
"+str(y)+" z "+str(z)
. .      cubit.cmd(mystr)
```

This simple script will create a grid of vertices four wide. Scripts can be more advanced, even creating customized windows and toolbars. For a complete list of python/cubit interface commands see the Appendix.

Repeating Commands

Use the **Up** and **Down** arrow keys on the keyboard to recall previously executed commands.

Commands can be repeated in other ways as well.

- Hitting the enter key while the cursor is on a previous command line will copy that command to the current prompt.
- The command window supports copy and paste for repeating commands.

Focus Follows Cursor

Beginning with version 13.0, Cubit includes a 'focus follows cursor' option for the command window. The option can be enabled and disabled from the Tools/Options/General options panel. The setting is persistent between sessions and is disabled by default.

Please note, the **focus follows cursor** behavior is available only in the command window. All other windows or widgets require the user to *click* the mouse in order to grab focus.

Error Window

The error window is located in the Command Line Workspace under the Error tab. If there are errors, a warning icon will appear on the tab. The icon will disappear when you open the window to view errors. The error window only displays the error output, which can make it easier to find and read the error output. The command that caused the error will be printed along with the error information. If the command was from a

journal file, the file name and number will be printed next to the command.

History Window

The history window lists the last 100 commands. The number of commands listed can be configured in the options dialog on the History page. You can re-run the commands in the history window using the context menu. You can also clear the history using the context menu.

Docking and Undocking the Input Window

The command window can be undocked by clicking and dragging the left edge. If it is floating it can be redocked by double-clicking the solid blue bar. By default, it will always be redocked in the bottom of the application window. To change the size of the floating window, click and drag the edge of the window. To change the height of the docked window, click and drag the top edge or right edge.

Journal File Editor

The Journal File Editor is a built-in, multi-document text editor that can read, edit, play, and translate CUBIT journal files and Python Scripts. To open the journal file editor, select the  icon on the File Tools toolbar, or from the Tools Menu.

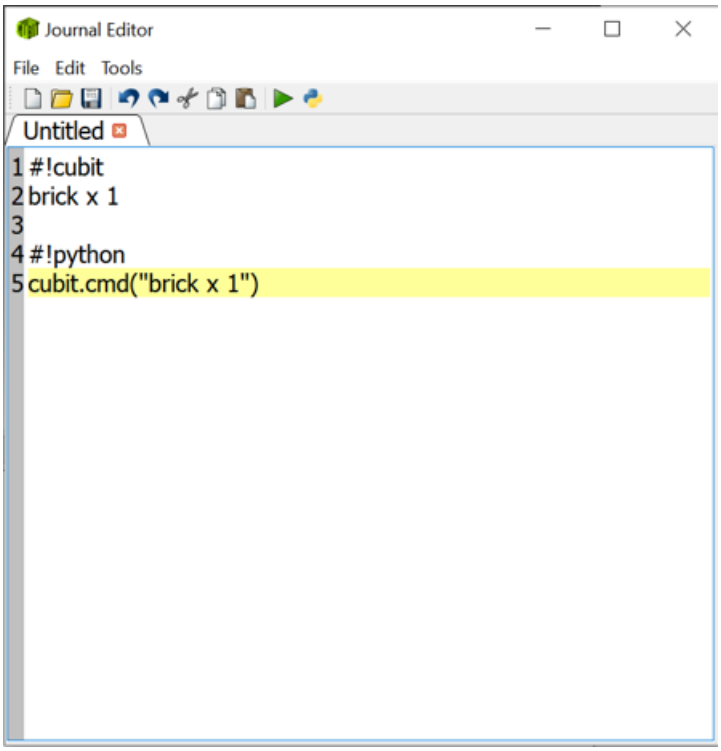


Figure 1. The Journal File Editor

The Journal File Editor can be used to create a new Python or Cubit command script. By default, a new journal file will be in Cubit command syntax. Enter the commands in the order you want them executed. You can play the commands all at once using the play button on the toolbar. You can also play a few commands at a time. Select the commands you want to play. Then, right click and select the "Play Selected" menu item. If you have a Cubit script, you can convert it to a Python script by hitting the Python toolbar button. If you have a few lines you want to convert to Python, select them, and right click to get the popup menu and choose "Translate Selected to Python"

The Journal File Editor can also be used to edit an existing journal file. Use the File > Open menu item to open the file you want to edit. You still have all the command play options with an existing journal file.

You can import commands entered in the Command Line Workspace. The File > Import menu gives the option to import commands from the history tab. Only the current commands shown in the history tab will be imported. Some of the commands you previously entered might not show up if you have the recommended text trimming turned on. Text trimming improves the application's performance for speed and memory. It will trim off the oldest text in the window when a size limit is reached. To get all the command from your current session, make sure that command journaling is turned on.

The Journal File editor can be used to edit multiple files at the same time. Each document is displayed in its own tab. The tab shows the journal file's syntax and name. If you close the Journal File Editor with unsaved

data, it will prompt you to save changes for each of the modified journal files you have open.

Journal Editor Toolbar

The Journal Editor's Toolbar provides quick access to several important functions.



- **New** - Creates a new journal file. The new journal file is placed in a new tab.
- **Open** - Used to select a journal file to open.
- **Save** - Saves the current journal file.
- **Undo** - Undo the last text change.
- **Redo** - Redo the last text change, after Undo.
- **Cut** - Standard text cut operation
- **Copy** - Standard text copy operation
- **Paste** - Standard text paste operation
- **Play Journal File** - Plays the entire journal file
- **Translate to Python** - Translates the current Cubit commands in the journal file to Python scripts.

Other Functionality Available in the Journal Editor

The context ('right-click') menu in the journal editor includes several additional functions, including:

- **Play Selected** - Play commands in the selected lines
- **Translate Selected to Python** - Translate the selected commands to Python statements by enclosing each command in **cubit.cmd()**
- **Comment Selected Lines** - Highlight any text, select 'comment selected lines', and the highlighted lines will be commented.
- **Uncomment Selected Lines** - Highlight any text, select 'uncomment selected lines', and the highlighted lines will be uncommented.
- **Clear** - select this menu item to clear the contents of the journal file.
- **Find** - Selecting 'find' from the context menu, or from the edit menu, will bring up a dialog enabling the user to find text in the journal file. Options are available to do case-sensitive searches, change search direction, and so forth.

Property Editor

The Property Editor is a window that lists properties about the current entity selection. Some of the properties, like CUBIT ID, entity type, or geometry engine, are listed for reference only. Other attributes, like name, or mesh intervals, color, mesh scheme, or smooth scheme can be edited from the window. The Property Editor is located on the left panel in the GUI. The highlighted entity/entities in the graphics window are listed in the property editor window. The Property Editor also lists information about selected mesh entities, boundary conditions, and assemblies. Selecting an object from the Tree View will also open the object in the property editor.

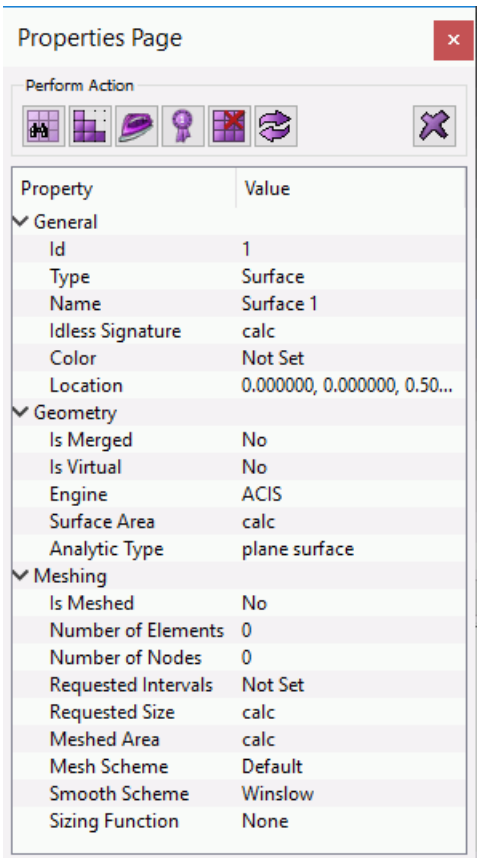


Figure 1. Property Editor Window

The row of buttons on the top of the editor are shortcuts to common commands. These include:

>

- Meshes the selected entity/entities at their current interval and scheme settings
- Smooth selected entity using the current smoothing scheme
- Preview mesh intervals on selected entity
- Delete mesh on specified entity (do not propagate to lower order entities)
- Reset entity to default settings and delete mesh
- Calculates volumes and surface areas
- Delete current entity

Editing Entity Attributes from the Property

Editor

The Property Editor provides a convenient way to change attributes on entities. . Some of the fields cannot be changed, some can be edited from an input field, and others are edited by selecting from a list, or by opening the corresponding window from the Control Panel.

If multiple entities are selected, the attributes that are similar to both entities will be shown. Changing an attribute from the property editor will change that attribute on both entities. If multiple entities are selected the total volume, surface area, and length of all entities will be shown.

Below is a summary of properties listed for each attribute type.

General Attributes

- **Entity ID** - CUBIT ID for geometry or boundary condition element
- **Entity Type** - Geometric type such as Volume, Surface, Curve, Vertex
- **Name** - Name by which the entity can be referred to from within CUBIT instead of using its ID. The entity name can be edited from this window.
- **Color** - Opens a dialog box with available colors. A color name can also be input directly into the text field. See Appendix for a list of available colors.

Geometry Attributes

- **Is Merged** - Returns "Yes" if this entity is merged
- **Is Virtual** - Returns "Yes" if this entity is a virtual entity
- **Location** - Returns the location of specified vertex.
- **Geometry Engine** - ACIS or Mesh-Based Geometry
- **Volume** - The volume of the specified body
- **Surface Area** - Surface area of selected surface
- **Analytic Type** - Returns the analytic type of entity (such as cone, sphere, etc)
- **Length** - Length of selected curve

Meshing Attributes

- **Is Meshed** - Returns "Yes" if the entity is already meshed
- **Number of Elements** - Similar to "List Totals" command
- **Requested Intervals** - Number of requested mesh intervals on element. This can be edited from this window. The number must be an integer
- **Requested Size** - Requestd interval size for element. Clicking on box will open the interval specification panel on the control panel. The interval size can also be entered manually in the text box.
- **Meshed Volume** - The meshed volume may be slightly different than the actual element volume due to the mesh approximation on curved surfaces.
- **Meshed Area** - The meshed area may be slightly different than the actual surface area due to mesh approximation on curved edges.
- Length of Meshed Edges - Combined total of mesh edge lengths on curve
- **Mesh Scheme** - The mesh scheme for this entity. This can be changed from the property editor by selecting from the drop-down list.
- **Smooth Scheme** - The smooth scheme for this entity. This can be changed from the property editor by selecting from the drop-down list.

Boundary Condition Attributes

- **ID** - Boundary condition ID. This is an arbitrary user-defined ID that is exported with the finite element model. This value can be edited

from the property editor

- **Name** - A user-defined name that is included in the metadata for that object. This value can be edited from the property editor.
- **Description** - A user-defined description that is included in the metadata for that object. This value can be edited from the property editor.
- **Color** - Opens a dialog box with available colors. A color name can also be input directly into the text field. See Appendix for a list of available colors.
- **Element Type** - The finite element type for this block, nodeset, or sideset.
- **Element Count** - The total number of elements for this block or sideset
- **Node Count** - Total number of nodes (available for nodesets only)
- **Attribute Count and Attributes**- The attributes represent material specification data that is associated with the element block. These values can be changed in the property editor. You can specify up to 10 attributes per block.

Toolbars

The CUBIT toolbars provide an effective way for accessing frequently used commands.

Below is a brief description of each of the available toolbars. To view a description of the function of each tool, hold the mouse over the tool in the CUBIT Application to display tool tips.

Users may customize and share toolbars.

File

Provides CUBIT (*.cub) file operations. This toolbar also includes Journal File operations.



Figure 1. File Toolbar

From left to right, the tool buttons are as follows:

- **New** - Issues a Reset command to begin a new session
- **Open** - Displays file open dialog
- **Save** - Displays file save dialog
- **Import** - Displays import dialog
- **Export** - Displays export dialog
- **Journal Editor** - Opens the journal editor
- **Play Journal File** - Plays a journal file
- **Play ID-Less Journal File** - Plays a journal file in Id-less mode
- **Pause** - Pause the playback of a journal file
- **Custom Toolbar Editor** - Opens the custom toolbar editor

Display

Controls the display mode, checkpoint undo, zoom, perspective clipping plane, and curve valence display options in the Graphics Window.



Figure 2. Display Toolbar

From left to right, the tool buttons are as follows:

- **Undo On/Off** - Toggle undo functionality on/off
- **Undo** - Undo last operation
- **Wireframe** - View in wireframe mode
- **True Hidden Line** - View in true hidden line mode
- **Hidden Line** - View in hidden line mode
- **Transparent** - View in transparent mode
- **Shaded** - View in shaded mode
- **View Geometry** - Toggle geometry display on/off
- **View Mesh** - Toggle mesh display on/off
- **View BC** - Toggle boundary condition display on/off
- **Graphics Composite** - Toggle graphics composite line display on/off
- **Refresh** - Refresh the graphics display
- **Zoom In**
- **Zoom Out**
- **Zoom to Fit**
- **Toggle Perspective**

- **Scale** - Toggle display of the scale
- **Clipping Plane** - Toggle display of the clipping tool
- **Clipping Manipulation** - When clipping plane is active, toggle clipping tools
- **Curve Valence** - Display curve valence in the graphics window
- **Overlapping Surface** - Locate (point to) overlapping surfaces in the model
- **Enclosed/Extended** - Toggle between enclosed and extended selection mode
- **X-Ray** - Toggle x-ray selection mode on/off
- **Selection Shape** - Toggle between box, sphere, polygon selection box

Select

Controls the Entity Selection Mode for picking or selecting entities.

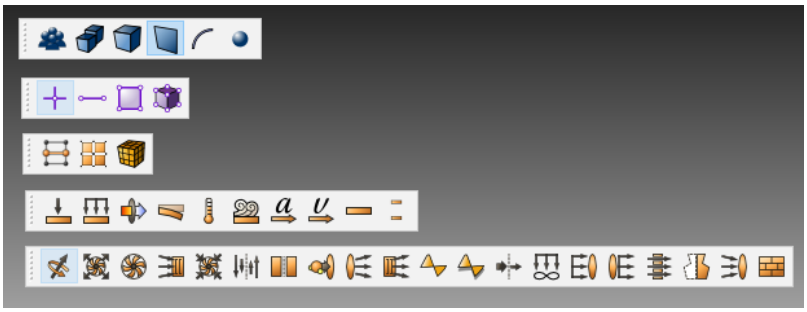


Figure 3. Select Toolbars

From top to bottom:

- Geometry Selection Filters
  - Groups
  - Bodies
  - Volumes
  - Surfaces
  - Curves
  - Vertices
- Mesh Selection Filters
  - Node
  - Edge
  - 2d (Triangle, Quadrilateral)
  - 3d (Hexahedral, Tetrahedral, Pyramid, Wedge)
- Boundary Condition and Material Containers
  - Nodeset
  - Sideset
  - Block
- FEA Boundary Conditions
  - Force
  - Pressure
  - Heatflux
  - Displacement
  - Temperature
  - Convection
  - Acceleration
  - Velocity
  - Contact Regions
  - Contact Pairs
- CFD Boundary Conditions
  - Axis
  - Exhaust Fan
  - Fan
  - Inlet Vent
  - Intake Fan

- Interface
- Interiors
- Inlet Mass Flow
- Outflows
- Outlet Vents
- Periodics
- Periodic Shadows
- Porous Jump
- Farfield Pressure
- Inlet Pressure
- Outlet Pressure
- Radiator
- Symmetries
- Inlet Velocities
- Walls

Toolbar Customization

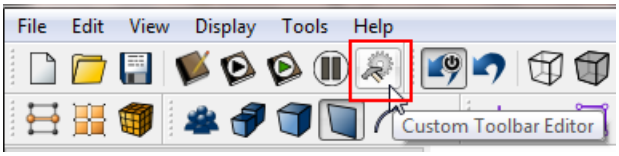
For many years Cubit has provided users with the ability to create custom tool buttons. These custom buttons launch pre-defined journal or Python scripts. With the release of Cubit 15.4 this capability has been expanded.

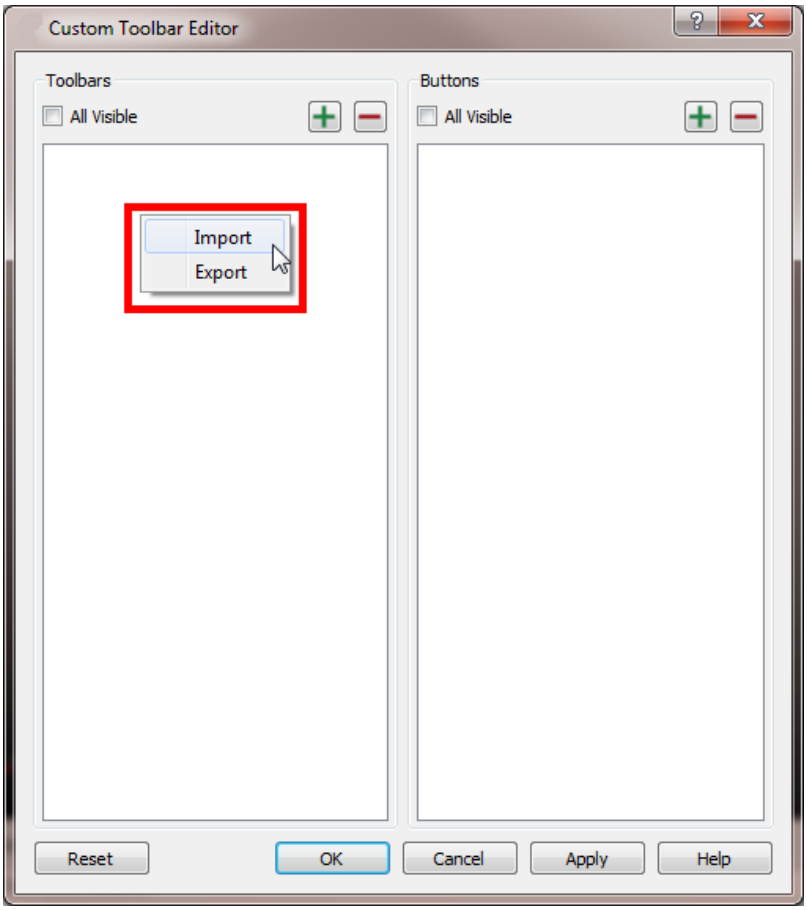
Menu

- Importing an Existing Toolbar
- Exporting a Toolbar
- Creating a new Toolbar
- Creating a Command Panel Button
- Creating a Journal File Button
- Creating a Python Script Button
- Creating a Basic Tool Button
- Modifying an Existing Toolbar

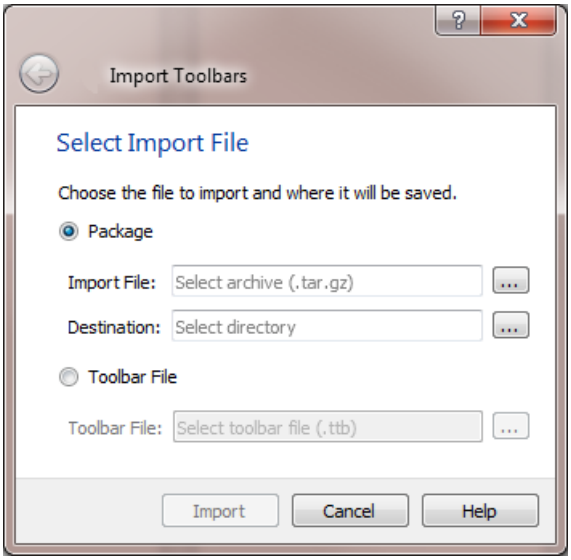
Importing an Existing Toolbar

- Locate and press the Custom Toolbar Editor button located on the File Tools button bar. This will launch the Custom Toolbar Editor.

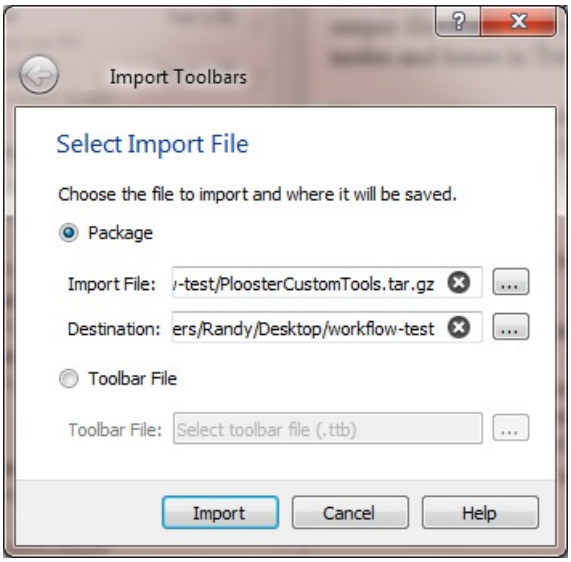




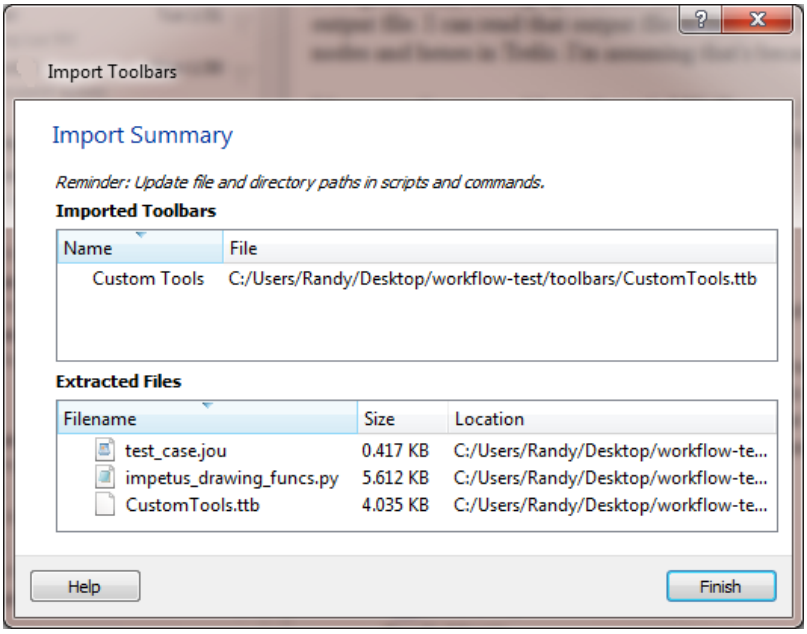
- Select Import from the context menu



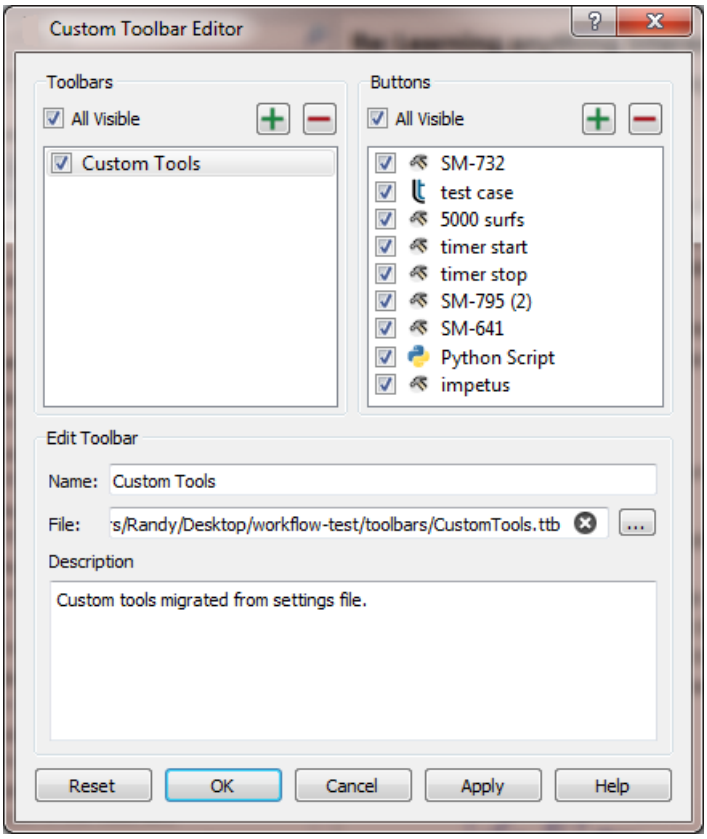
From this dialog a user may import an **entire package** containing multiple toolbars or a **single** toolbar. In this example we will import an entire package containing multiple toolbars.



- After selecting **import**, an import summary is shown.



- Select **Finish**

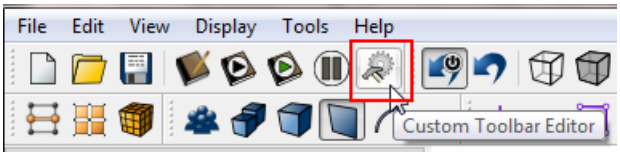


- Select **OK** to finish the import

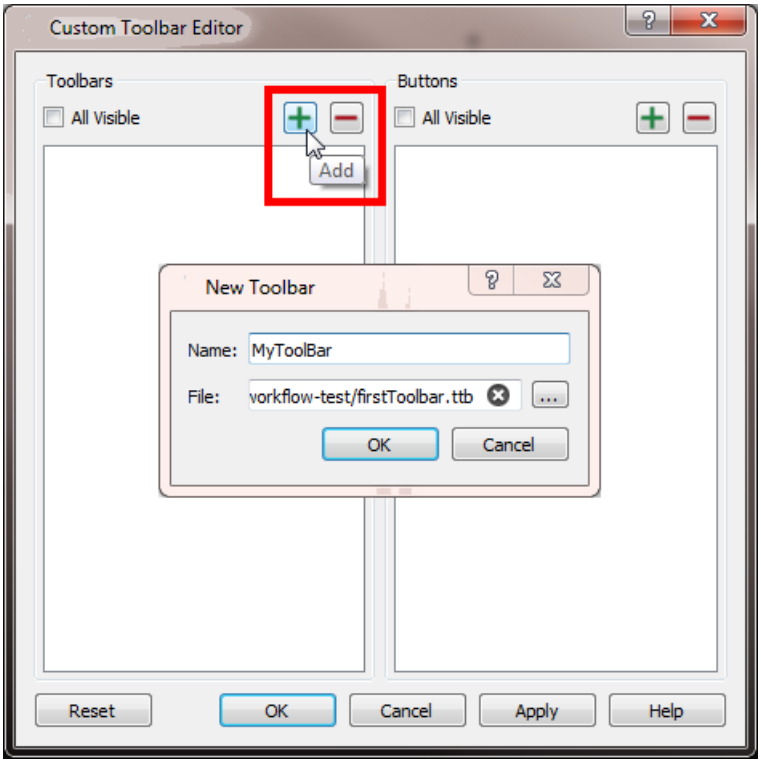
The new toolbar and buttons will be displayed as the last toolbar on the GUI. It is a docking window so it can be moved and placed anywhere on the GUI.

Creating a New Toolbar

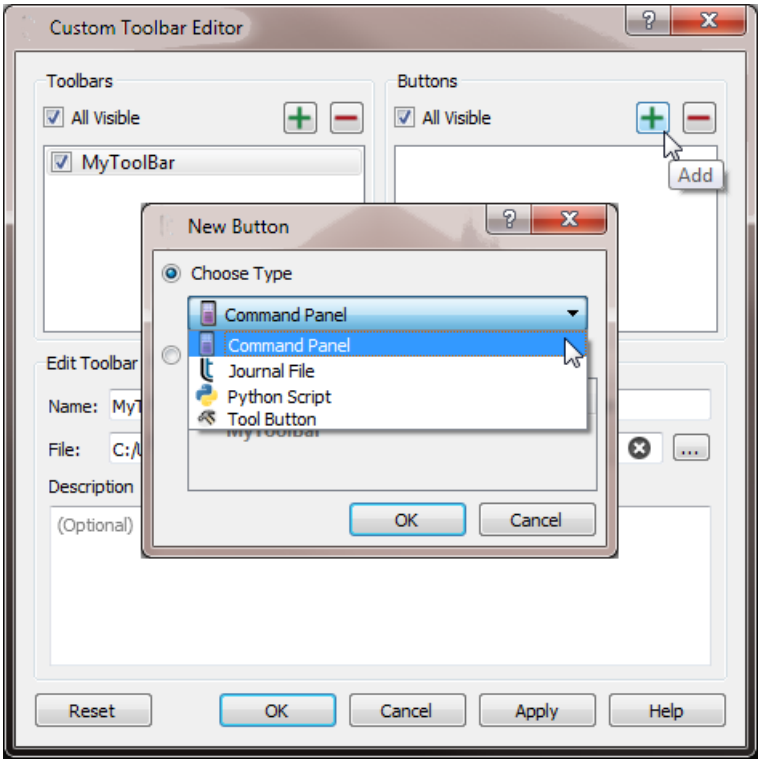
- Locate and press the Custom Toolbar Editor button on the File Tools button bar. This will launch the Custom Toolbar Editor.



- Press the **Add** button
- Name the new toolbar and press **OK**



- Press the **Add** button in the Buttons area



A user may define 4 different types of toolbar buttons

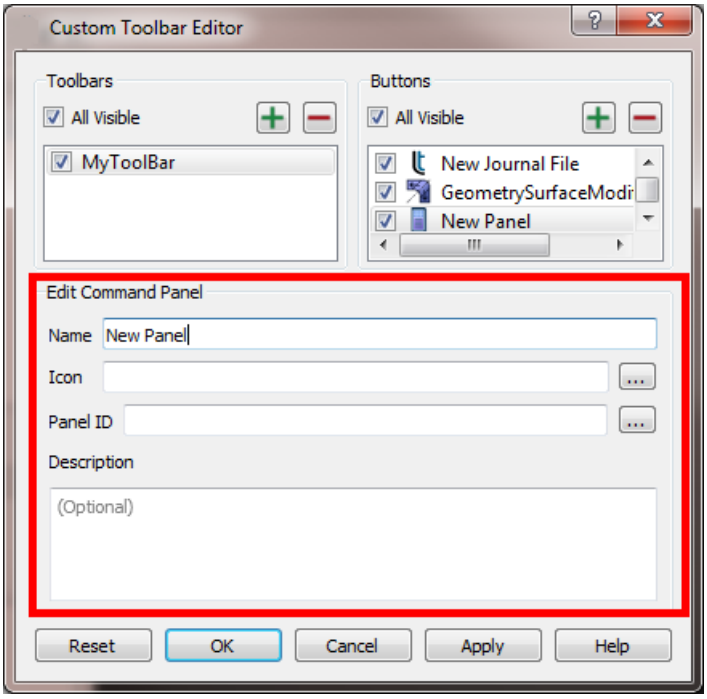
- Command Panel
- Journal File
- Python Script
- Tool Button

Creating a Command Panel Button

A **Command Panel Button** enables users to launch a command panel with the push of a button. A command panel button can be defined one of three ways:

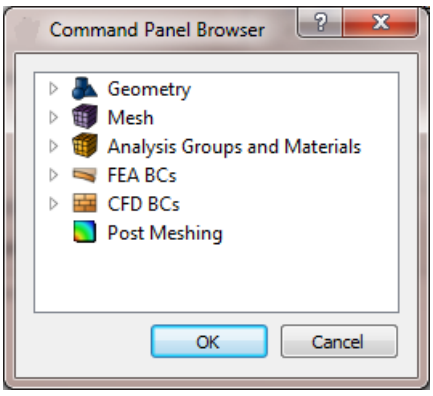
Use the definition dialog

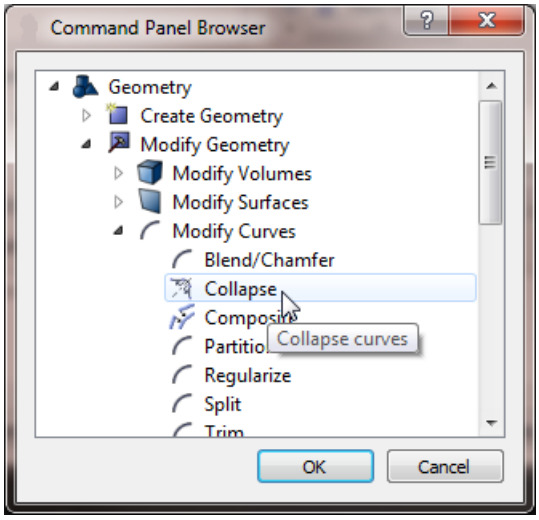
- Select **Command Panel** from the New Button type pulldown menu
- Press **OK**
- Complete the dialog indicating
  - the name of the button
  - the icon to use
  - the panel ID of the command panel to show -- **see instructions for find the panel ID (below)**
  - an optional description of the command panel
- Press **OK** to save the definition and exit the dialog
- Or, press **Apply** to save the definition



To find the Command Panel ID:

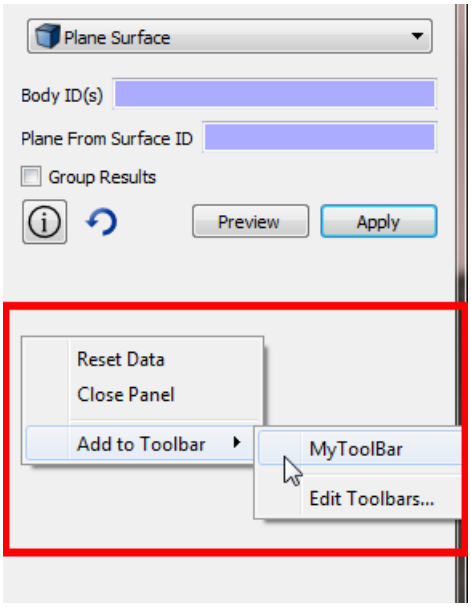
- Press the browse button next to the Panel ID edit field to launch the **Command Panel Browser**
- Navigate the browser to locate the desired command panel
- Select the desired command panel
- Press **OK** to make the selection
- The Panel ID will be shown in the Panel ID edit field





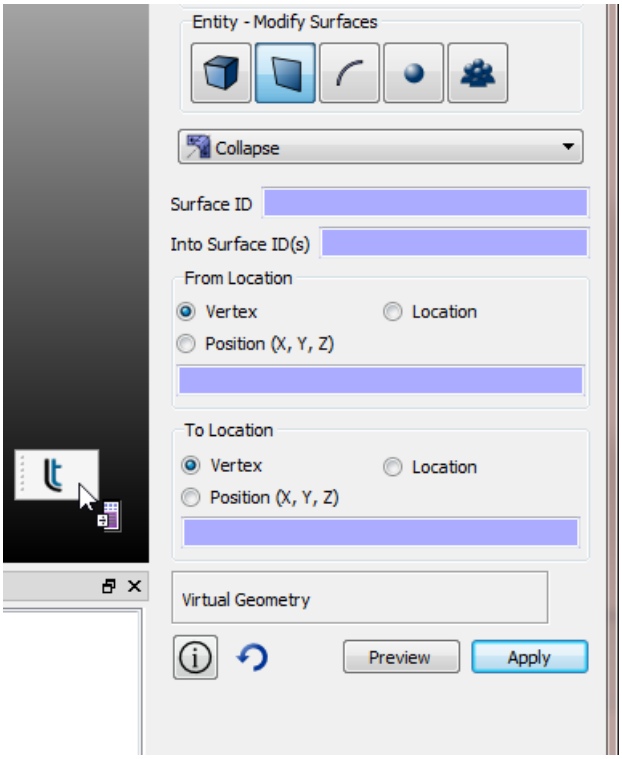
Use the context menu on a command panel

- Show the context menu on a command panel
- Select **Add to Toolbar**
- Select the toolbar to which this command panel will be added
- an icon representing the command panel will be added to the selected toolbar



Drag a command panel onto the toolbar

- Using the mouse, "drag" the command panel onto the desired toolbar
- an icon representing the command panel will be added to the selected toolbar
- In the image below, the Surface Collapse command panel is being dragged onto a toolbar



- The resulting toolbar looks like the following

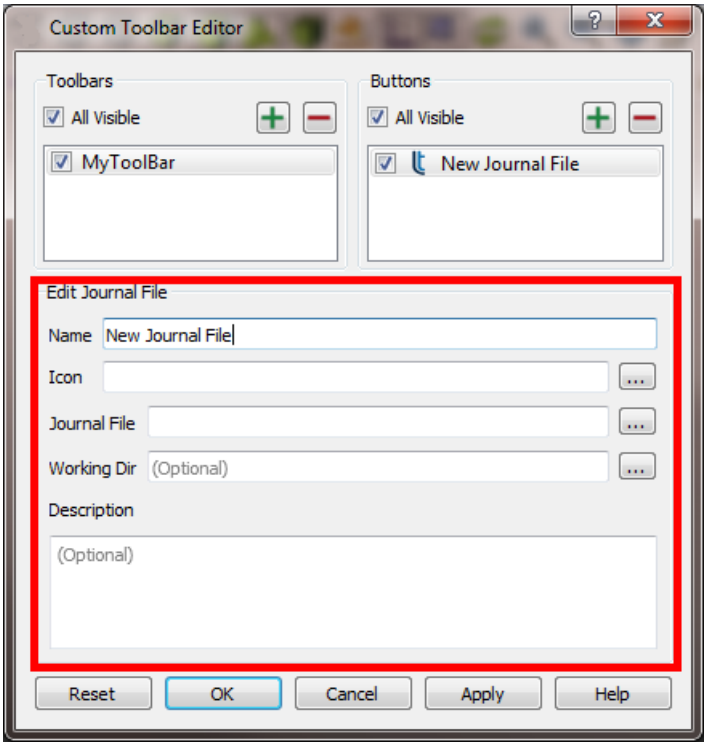


All command panels include a context menu which can be accessed by clicking on an empty place in the command panel and using the mouse to show the menu.

Creating a Journal File Button

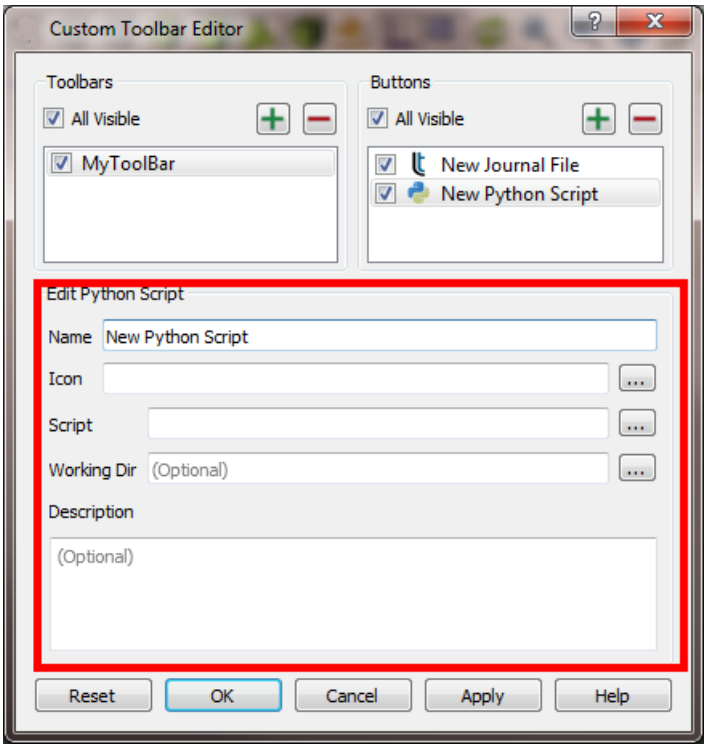
A Journal File Button will launch a journal file when pressed. The journal file may reside anywhere on the file system. A journal file button is defined by:

- Select **Journal File** from the New Button type pulldown menu
- Press **OK**
- Complete the dialog indicating
  - the name of the button
  - the icon to use
  - the name of the journal file to play
  - an optional working directory
  - an optional description of the journal file
- Press **OK** to save the definition and exit the dialog
- Or, press **Apply** to save the definition



Creating a Python Script Button

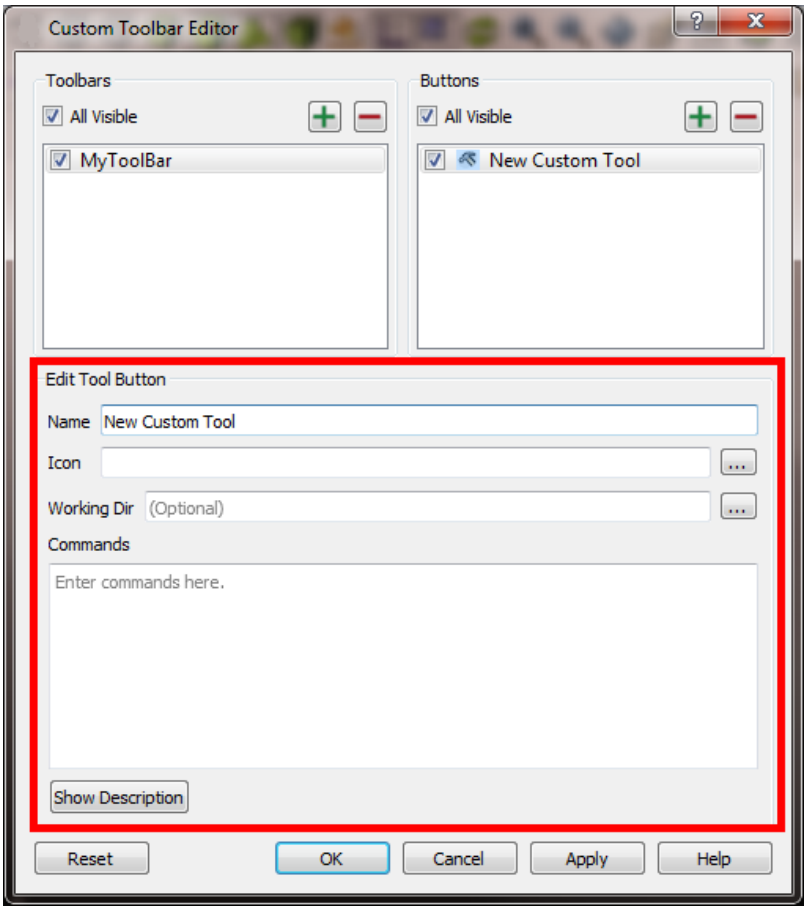
- Select **Python Script** from the New Button type pulldown menu
- Press **OK**
- Complete the dialog indicating
  - the name of the button
  - the icon to use
  - the name of the Python script to execute
  - an optional working directory
  - an optional description of the Python script
- Press **OK** to save the definition and exit the dialog
- Or, press **Apply** to save the definition



Creating a Basic Tool Button

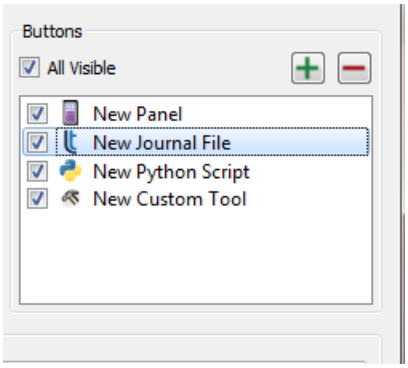
The "Basic" too button has been available to users for many years. It contains a set of commands that execute when the user presses the button.

- Select **Tool Button** from the New Button type pulldown menu
- Press **OK**
- Complete the dialog indicating
  - the name of the button
  - the icon to use
  - an optional working directory
  - the commands to execute (these are the same commands used in any journal file)
  - an optional description of the commands
- Press **OK** to save the definition and exit the dialog
- Or, press **Apply** to save the definition



Modifying an Existing Toolbar

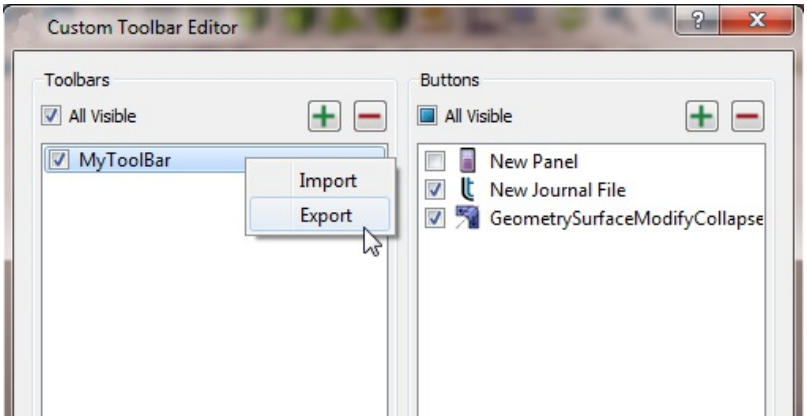
- In the Custom Toolbar Editor select the toolbar to modify
  - Press the Add (green plus-sign) button to add a new button
  - Press the Delete (red minus-sign) to remove a button
  - Select the check box to hide or show the button
  - **Change the button order** by selecting a button in the Buttons dialog and dragging to a new position
  - Any other parameter may be modified using the Edit Tool Button dialog
  - Press **OK** to save the definition and exit the dialog
  - Or, press **Apply** to save the definition



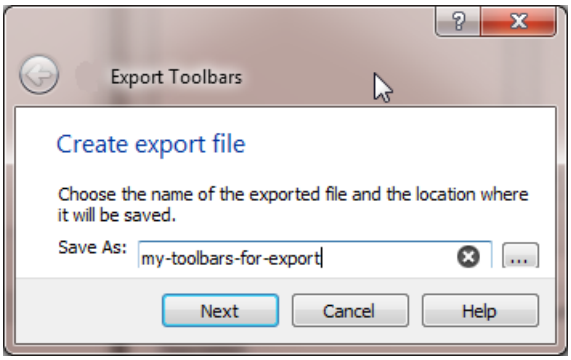
Exporting a Toolbar

A user may want to share a toolbar, or a set of toolbars, with another user. This is easily accomplished.

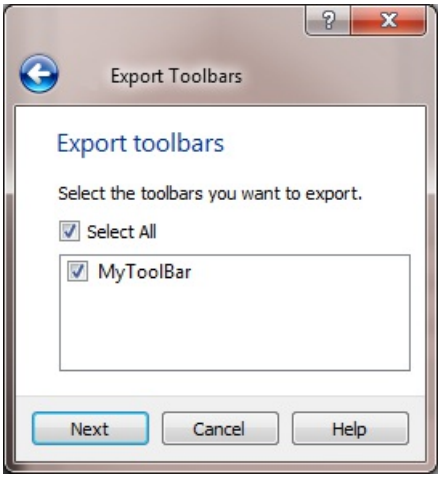
- Launch the Custom Toolbar Editor dialog by selecting the  icon.
- Or, select the **Edit** item from the toolbar's context menu
- Select **Export** from the context menu



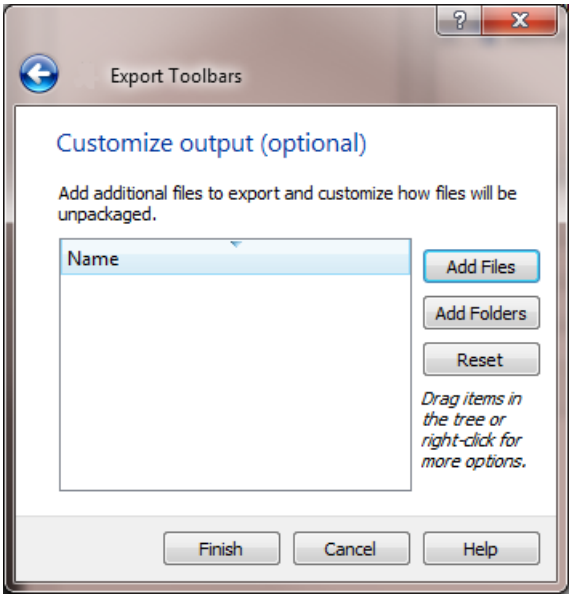
- Provide a file name to the Export Toolbars dialog. The file extension will be appended automatically
- The file type will be **.tar.gz**
- Click **Next** on the dialog



- In the next dialog select the toolbars to be included in the export
- Click **Next** on the dialog



- Optionally add files or folders that contain journal files or Python scripts referenced by tool buttons
- Click **Finish** in the dialog
- Look for the **.tar.gz** file in the designated folder



Graphics Window Control

The graphics display windows present a graphical representation of the geometry and/or the mesh. The quality and speed of rendering the graphics, the visibility, location and orientation of objects in the window, and the labeling of entities, among other things, can all be controlled by the user.

Unless the **-nographics** option was entered on the command line, a graphics window with a black background and an axis triad will appear when CUBIT is first launched. The geometry and mesh will appear in this window, and can be viewed from various camera positions and drawn in various modes (wire frame, hidden line, smooth shade, etc.). This section will discuss methods for manipulating the graphics with the mouse and for controlling the appearance of entities drawn in the graphics window.

All geometry, mesh, and simulation objects created in CUBIT are put into the view automatically. Visibility, color and various other attributes of entities in the view can be controlled individually. In addition, CUBIT can also optionally show entities in a temporary view mode independent of their visibility. Drawing of items in temporary mode can be added to the regular view mode to customize the appearance. The overall view is controlled by various attributes like graphics mode, camera position, and lighting, to further enhance the graphics functionality.

The graphics view relies on OpenGL to render the scene which leverages the graphics hardware of the system. CUBIT requires the graphics hardware to support OpenGL version 3.2 or newer. If OpenGL 3.2 is not available, CUBIT will attempt to fall back to a software based implementation bundled with CUBIT. This software based approach may not perform as well as a hardware-based approach.

The following items discuss the various graphics capabilities available in CUBIT:

- Command Line View Navigation: Rotate Zoom and Pan
- Mouse Based View Navigation: Rotate Zoom and Pan
- Updating the Display
- Graphics Modes
- Drawing and Highlighting Entities
- Drawing Locations, Lines and Polygons
- Mesh Visualization
- Graphics Clipping Plane
- Entity Labels
- Colors
- Geometry and Mesh Entity Visibility
- Graphics Camera
- Graphics Lighting Model
- Graphics Window Size and Position
- Saving Graphics Views
- Hardcopy Output
- Miscellaneous Graphics Options

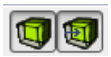
Graphics Clipping Plane

The graphics clipping plane feature allows the user to temporarily cut parts of the model away to help visualize the interior of a geometry or mesh. The command syntax is:

```
Graphics Clip {On|Off} [ Plane <plane> |
[Location <location>] [Direction <direction>]]

Graphics Clip Manipulation {On|Off}
```

The GUI tool bar buttons to enable and manipulate the Graphics Clipping Plane are shown below:



The first command activates the graphics clip manipulation tools in the graphics window. The keyboard shortcut "Shift-S" while the graphics window is active will also activate the clipping plane. The manipulation of the clipping plane is controlled as follows:

- **Red Line** - Clicking and dragging the left mouse on plane bounded by a red tube moves the plane along the arrow
- **Center Ball** - Clicking and dragging the left mouse on the center ball moves the origin of the rotation plane
- **Arrow** - Clicking and dragging the left mouse button on the arrow head or tail changes the direction on which the plane moves
- **Right Mouse Button** - Clicking and dragging the right mouse button on any part of the window resizes it
- **Middle Mouse Button** - Clicking and dragging the middle mouse button on the red plane moves both the center of rotation and the cutting plane
- **White Bounding Border** - Clicking and dragging the left mouse on the white bounding border moves the whole widget

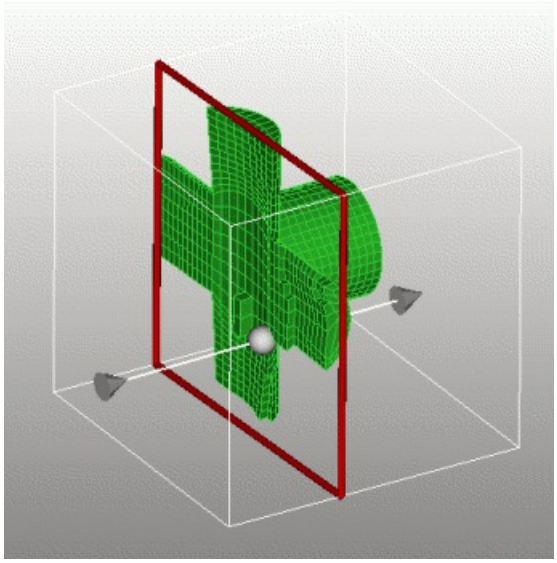


Figure 1. Graphics Clipping Plane

The second command turns on/off the visibility of manipulation widget in the graphics window. The clipping plane is still active, but the controls are hidden. The normal mouse-based view navigation controls apply.

Examples

brick x 10

sphere rad 1
graphics clip on location -2 0 0
rotate -45 about y
#shows the sphere inside the brick

brick x 10
cylinder rad 2 z 12
subtract 2 from 1
mesh vol 1
quality vol 1 draw mesh
graphics clip on
#shows the mesh quality on interior elements

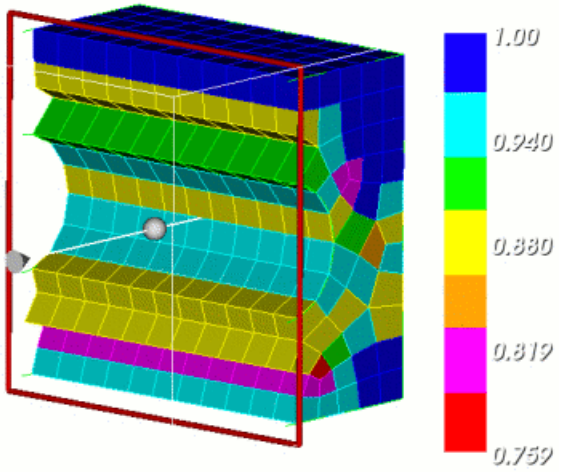


Figure 2. Viewing mesh quality of interior elements

Colors

Specifying Colors in Commands

There are multiple ways to refer to a color in a command. They are

1. **<Color_Name>**
2. **User "name"**
3. **ID <id>**
4. **RGB**
5. **RGBA**
6. **Default**
7. **Highlight**

The first option uses the name of a pre-defined color as listed in the Available Colors Appendix. This option may not be used for user-defined colors. An example of a pre-defined color assignment is given below:

```
color volume 1 lightblue
```

The second option is used with user-defined colors only. Include the name of the user-defined color in quotes. Pre-defined colors will not work with this command.

```
color volume 1 user "mycolor"
```

The third option allows you to identify a pre-defined color by its ID. The color IDs are also listed in the Available Colors appendix. This option is rarely used.

```
color volume 1 id 5
```

The fourth option allows you to specify a color by R, G and B values ranging between 0.0 and 1.0. This option can be used to specify colors not found on the color table.

```
color volume 1 rgb 0.7 0.4 0.35
```

The fifth option allows you to specify a color by R, G, B, and A values ranging between 0.0 and 1.0. This option allows you to specify an Alpha component allowing for transparency. 1.0 is opaque and 0.0 is transparent. This option can be used to specify colors not found on the color table.

```
color volume 1 rgba 0.7 0.4 0.35 0.4
```

The default option is used to set an entity's color to its default value. The default color may also be specified in drawing commands, but the command's behavior will be the same as if the color option had not been included at all.

```
color volume 1 default
```

The seventh option refers to the current highlight color.

```
draw curve 1 tangent color highlight
```

User-Defined Colors

CUBIT has a palette of 92 pre-defined colors, listed in the Appendix under Available Colors. Users may also define their own colors in addition to those defined by CUBIT. Each color is defined by a name and by its RGB components, which range from 0 to 1. Adding user defined colors to the color table is an alternative to using RGB or RGBA color specifications.

To define an additional color, use either of the commands

```
Color Define "<name>" RGB <r g b>

Color Define "<name>" R <r> G <g> B <b>.
```

This is done with the command

```
Color Release "<color_name>"
```

Color names can be listed with the command

```
Help Color
```

They are also listed in the appendix of this manual, along with their RGB definitions. To view a chart of color names and IDs, including those for user-defined colors, use the command

```
Draw Colortable
```

Assigning Colors

Colors may be assigned to all geometric entities, and to some other objects as well. To assign a color to an entity or other object, use one of the following commands.

```
Color Axis Labels {<color_name>| id <color_id>
| rgb <r> <g> <b> }

Color Background {<color_name>| id <color_id>
| rgb <r> <g> <b>} [<color_name2>|id
<color_id2>| rgb <r> <g> <b>]

Color Block <block_id_range>{<color_name> | id
<color_id> | rgb <r> <g> <b>}

Color Body <body_id_range> [Geometry|Mesh]
{<color_name>| id <color_id> | rgb <r> <g> <b>
| Default}

Color Curve <curve_id_range> [Geometry|Mesh]
{<color_name>| id <color_id> | rgb <r> <g> <b>
| Default}

Color Group <group_id_range> [Geometry|Mesh]
{<color_name>| id <color_id> | rgb <r> <g> <b>
| Default}

Color Highlight {<color_name>| id <color_id>|
rgb <r> <g> <b>}

Color Lines <color_spec>

Color NodeSet <id_range> { <color_name> | id
<color_id> | rgb <r> <g> <b> | Default }

Color SideSet <id_range>{ <color_name> | id
<color_id> | rgb <r> <g> <b> | Default }

Color Surface <surface_id_range>
[Geometry|Mesh] {<color_name>|| rgb <r> <g>
<b> Default}

Color Title {<color_name>|id <color_id>| rgb
<r> <g> <b> }

Color Volume <volume_id_range> [Geometry|Mesh]
{<color_name>| id <color_id> | rgb <r> <g> <b>
| Default}
```

Including the Mesh keyword will change the color of the mesh belonging to the specified entity, without changing the color of the entity geometry itself. Conversely, including the Geometry keyword will change the

itself. Conversely, including the Geometry keyword will change the geometry color without changing the mesh color. Including both keywords is identical to including neither keyword.

Colors are inherited by child entities. If you explicitly set the color for a volume, for example, all of its surfaces will also be drawn in that color. Once you assign a color to an entity, however, it will remain that color and will no longer follow color changes to parent entities. To make an entity follow the color of its parent after having explicitly set another color, use Default as the color name in the color command.

Colors can also be assigned to nodesets, sidesets, and element blocks. These colors do not take effect, however, unless the nodeset, sideset, or element block is drawn with a Draw command.

The background color and the color used to draw highlighted entities can be changed to any color.

By default, the axes are labeled with a white X, Y, and Z, indicating the three primary coordinate directions. If the background is changed to white, these labels are impossible to read; the color used to draw axis labels can be changed to any color. Changing the axis label color will change the text color for both the model axis and the triad (corner axis).

When several entity types are labeled, it can become difficult to determine which labels apply to which entities. To help distinguish which entities are being referred to by the labels, you may want to change the color of labels for specific entity types.

When a meshed surface is drawn in a shaded graphics mode, the mesh edges are not drawn in the same color as the surface. This is to prevent confusion between mesh edges and geometric curves, and to make the mesh edges more visible. The color used to draw mesh edges in this situation is known as the line color, and is gray by default; this color can be changed to any color.

Assigning Global Colors

Colors may be assigned globally also. To assign a global color, use one of the following commands. Global color assignment is useful if one desires all entities to appear the same.

```
Color Global {<color_name>| id <color_id> |
rgb <r> <g> <b> | default}

Color Global Surface {<color_name>| id
<color_id> | rgb <r> <g> <b> | default} Curve
{<color_name>| id <color_id> | rgb <r> <g> <b>
| default} Vertex {<color_name>| id <color_id>
| rgb <r> <g> <b> | default}
```

The first command assigns the desired color to all geometry entities. The color may be enter by color name or color id. The default option resets colors to the default value.

The second command assigns the desired colors to surfaces, curves and vertices. All three value must be entered. For example, users my select global colors for surface and vertex and specify that curves have default colors.

Drawing, Locating, and Highlighting Entities

In order to effectively visualize the model, it is often necessary to draw an entity by itself, or several entities as a group. This is easily done with the command

```
Draw {Entity specification} [Color
<color_spec>] [Zoom] [Add]
```

where Entity specification is an entity list as described in Command Line Entity Specification. This command clears the display before drawing the specified entity or entities. Specification of a color will draw those entities in that color. This will not permanently change the color of the entity. The zoom option will zoom in on the selected entities after drawing them in the graphics window. If the **add** option is specified, the display is not cleared, and the given entity is added to what is already drawn on the screen. The entities specified in this command are drawn regardless of their visibility setting (see Geometry and Mesh Entity Visibility for more details about visibility).

Entities may also be drawn by selecting them with the mouse and then typing Ctrl-D while the mouse is in the graphics window. This will clear the screen and then draw only those entities that are currently selected.

Entities can be highlighted using the command

```
Highlight {Entity specification}
```

This command highlights the specified entities in the current display with the current highlight color. Highlighting can be removed using the command

```
Graphics Clear Highlight
```

To return to the normal display of the entire model, type Display.

The **Locate** command will label and point to the specified entity or location in the graphics window. The command syntax is:

Locate <entity_list> [<string>]

Locate location <options> [<string>]

For example, suppose you have an idless reference to a curve of:

Curve (at 5 5 0 ordinal 1)

You can find the curve with the following command:

locate location 5 5 0

Supplying an optional string will draw the text as the label instead of the entity type and id.

Additionally, the visibility of individual entities, or sets of entities, can be controlled with the following visibility commands.

```
{Vertex|Curve|Surface|Volume|Body|Group}
<range> [Geometry|Mesh] Visibility {on|off}

Edge [Visibility] {on|off}

{Mesh|Geometry} [Visibility]{on|off}
```

Drawing Other Objects

In addition to the common geometry, mesh and genesis entities, other objects may be drawn with variations of the Draw command. As with the other Draw commands, typing Display after drawing these objects will restore the scene to its normal display.

Displaying Entity Orientation

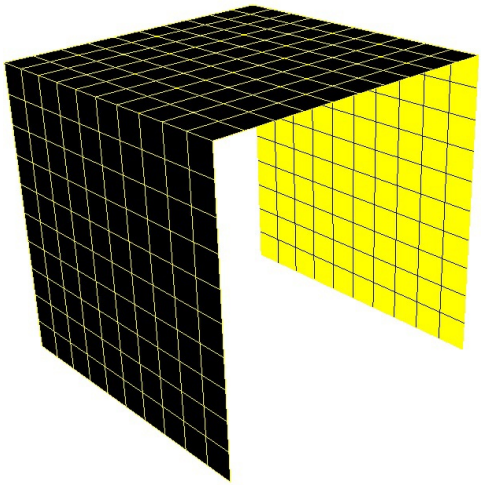
The normal to one or more surfaces, mesh faces, or mesh triangles may be drawn with the command

```
Draw {Surface | Face | Tri} <id_range> Normal
[Length <length>] [Face | Tri] Color <color>
[Add]
```

Surface normal command colors the surfaces using two different colors. The surface exposed to the positive half space (i.e, along the direction of normal), will always be colored black. The surace exposed to the negative half space will be colored using the specified <color>.

If the Face or Tri qualifier is included in the Draw Normal command, the normals for all faces or tris that belong to the specified surface are drawn.

Arrow representing the normal will be displayed if "Length" is specified



The forward, or tangent, direction of a curve can be drawn with the command:

The forward, or tangent, direction of a curve can be drawn with the command:

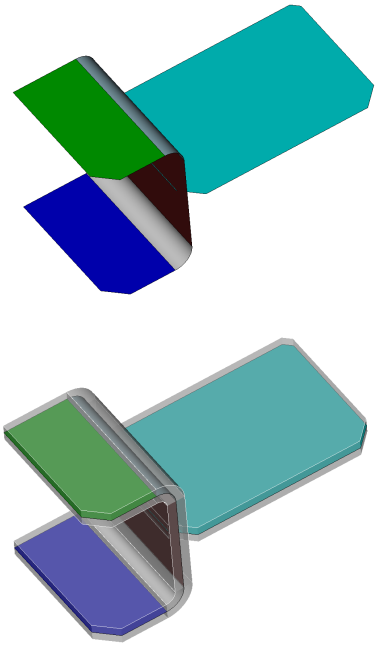
```
Draw Curve <id_range> Tangent [Length
<length>][Color <color_spec>]
```

If a color is not specified, the tangent is drawn in the same color as the curve.

Displaying Sheet Volume Thickness

For volumes that can be identified as a **sheet**, an offset 3D volume can be displayed.

```
Draw Volume <ids> Thickness <value> [Loft
<value>] [Include_Normal] [Color <color>]
[Transparent] [Add]
```



Above figures show example sheet volumes (left) and the same sheet volumes displayed with their thickness (right) using the command:

```
draw volume all thickness 4 loft 0.5 color
white transparent add
```

The **thickness** value is a positive floating number that indicates the thickness of the offset volume in a direction normal to the sheet volume. The **loft** option should be a value between 0 and 1 indicating where the preview volume will be displayed with respect to the sheet volume. A loft value of 0 will display the volume offset in the direction of the sheet volume surface normal. A value of 1 will reverse the direction of the offset. A value of 0.5 will display the volume centered on the sheet volume (as shown in the figure above). A **color** using one of Cubit's standard color keywords should also be specified. The **transparent** option can also be used to display the offset volume in transparent mode. Transparency can also be useful when using the **add** option so that both the sheet volumes and their offset volumes can be visualized together. The **include_normal** will display an arrow at the center of the sheet volume's surfaces indicating the normal direction of each surface

Volume Sources and Targets

Once the source and target surfaces have been set on a volume that will be meshed with the sweep algorithm, the source and target may be visually identified with the command

```
Draw Volume <volume_id_range> [Source][Target]
[Length <size>]
```

If the Source keyword is included, the normal of the source surface or surfaces will be drawn in green into the specified volume. If the Target keyword is included, the normal of the target surface or surfaces will be drawn in red into the specified volume.

Model Axis

The model axis may be drawn with the command

```
Draw Axis [Length <length>]
```

The axis is drawn as three lines beginning at the model origin, one line in each of the three coordinate directions. The length of those lines is determined by the length parameter, which defaults to 1.

Surface Isoparameter Lines

Isoparameter lines may be drawn on surfaces in the model using the command

```
Draw Surface <surface_id_range> Isoparametric
[Number <number>| [u <number>] [v <number>]]
```

If you specify the Number of lines, then the number of u- and v-parameter lines will be equal. You may specify instead a number of lines for each of the u and v parameters. The u-parameter lines will be drawn in red and the v-parameter lines will be drawn in blue.

Surface Overlap

The overlapping regions between two surfaces may be drawn with the command

```
Draw Surface <id> <id>Overlap [Add]
```

This command will draw the curves of each of the surfaces in green, and the portion of the surfaces that overlap in red. The Add keyword will draw the overlapping surfaces on top of the current graphics display. Without the Add keyword, the display will only show the specified surfaces and their overlapping regions.

Volume Overlap

The overlapping region between two volumes may be drawn with the command

```
Draw Volume <id> <id> Overlap [Add]
```

This command will draw the input volumes in transparent mode and draw the volume(s) of intersection as red, shaded solids. The **Add** keyword will draw the results on top of the current graphics display. Without the **Add** keyword, the display will only show the specified volumes along with the intersection volume(s).

Geometry Preview

Several options are available for previewing geometry without actually generating it. This is typically used in conjunction with webcutting and surface creation. The following Draw commands can be used for previewing geometry:

> Draw Location On Curve
>
> Draw Location
>
> Draw Direction
>
> Draw Line
>
> Draw Polygon
>
> Draw Axis
>
> Draw Plane
>
> Draw Cylinder

Drawing Locations, Lines and Polygons

In some cases it may be useful to simply draw a location, line or polygon to the screen to help visualize some aspect of the model. Locations, Lines and polygons are not geometry or mesh entities and are only visible until a refresh or display command is issued.

Drawing Locations

```
Draw Location {options}... [color
<color_name>][no_flush]
```

A single point or series of points may be drawn to the graphics window using this command. Any number of locations may be specified that will be drawn to the graphics window as single points. Options for specifying a location are described in the section Specifying a Location. The optional **color** argument allows for a custom color to be used. The available color definitions are located in the appendix. Other options for drawing locations and directions are also available dscribed in the section Drawing a Location, Direction, or Axis.

Drawing Lines

```
Draw Line Location {options} Location
{options} ... [color <color_name>][no_flush]
```

A straight line or series of segments may be drawn to the graphics window using this command. Any number of locations may be specified that will be connected with a line. Options for specifying a location are described in the section Specifying a Location. The optional **color** argument allows for a custom color to be used. The available color definitions are located in the appendix.

Drawing Polygons

```
Draw Polygon Location {options} Location
{options} Location {options} ... [color
<color_name>][no_flush]
```

A filled polygon may be drawn to the graphics window using this command. Any number of locations may be specified as vertices. At least three locations must be specified. Locations for vertices can be described using any of the standard location options described in Specifying a Location. The optional **color** argument allows for a custom color to be used for the fill. The available color definitions are located in the appendix.

Buffered Drawing

The optional **no_flush** argument for both the **draw location**, **draw line** and **draw polygon** commands may also be used when many simultaneous **draw** commands are being issued. This prevents the graphics from being drawn after each command is issued, which can be very inefficient. Instead the draw commands are buffered and sent all at once to be drawn. The following command:

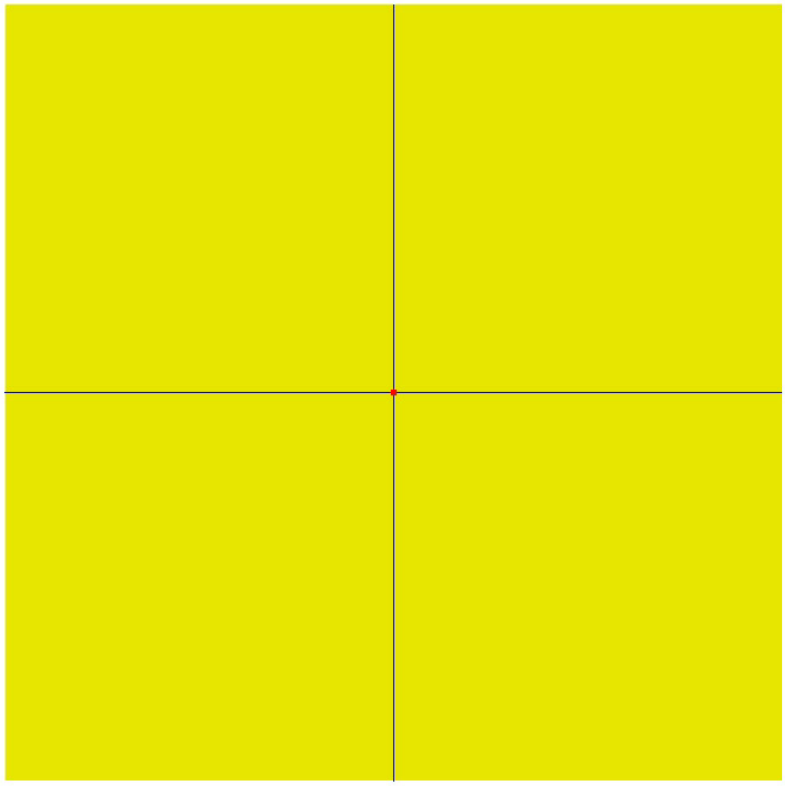
```
graphics flush
```

can be used to force a draw following a series of commands that use the **no_flush** option.

Example

The following is a simple example that will draw the figure below using cubit commands

```
draw polygon location pos -1 -1 0 location pos 1 -1 0
location pos 1 1 0 location pos -1 1 0 color yellow no_flush
draw line location pos -1 0 0 location pos 1 0 0 color blue
no_flush
draw line location pos 0 -1 0 location pos 0 1 0 color blue
no_flush
draw location pos 0 0 0 color red no_flush
graphics flush
```



Entity Labels

Most entities may be labeled with text that is drawn at the centroid of the entity.

Mesh entities can be labeled with their ID number or their Element ID. Element ID labels are only valid after putting the mesh entities into a block.

Geometric entities can be labeled with their ID number or with other information.

Labels for groups of entity types can be turned on or off.

The following commands will accomplish this.

```
Label [On|Off|Name
[Only|ID]|ID|Interval|Size|Merge|Firmness]

Label All [On|Off|Name
[Only|ID]|ID|Interval|Size|Merge|Firmness]

Label Body [On|Off| Name [Only|ID]
|ID|Interval|Size| Merge |Firmness]

Label Curve [On|Off|Name [Only|ID] |ID|
Interval| Size| Merge| Firmness]

Label {Hex|Tet|Face|Tri|Edge}
[On|Off|ElementId]

Label Element [On|Off]

Label Geometry [On|Off|Name [Only|ID] |ID|
Interval| Size| Merge| Firmness]

Label Mesh [On|Off]

Label Node [On|Off|ElementId|SphereId]

Label Surface [On|Off|Name [Only|ID] |ID|
Interval| Scheme| Size| Merge| Firmness]

Label Vertex [On|Off|Name [Only|ID]
|ID|Interval| Size| Merge| Firmness]

Label Volume [On|Off|Name [Only|ID] |ID
|Interval| Size |Scheme |Merge |Firmness]
```

The meaning of each of each label type is listed below. Note that some label types don't make sense for every entity type.

On - The same as IDs.

Name - Name of the entity, if the entity has been named. Default name otherwise.

Name Only - If the entity has been named, use the name as the label. Otherwise, don't use a label.

Name IDs - If the entity has been named, use the name as the label. Otherwise, use the ID as the label.

Interval - The number of intervals set on the entity.

Firmness - Same as interval, but followed by a letter indicating the firmness of the interval setting (see the Mesh Generation chapter for description of firmness settings.)

Merge - Whether or not the entity is mergeable. Note that this is sometimes not clear, because, for example, a curve may show that it isn't mergeable because one of its owning surfaces may be unmergeable, while another owning surface may be mergeable.

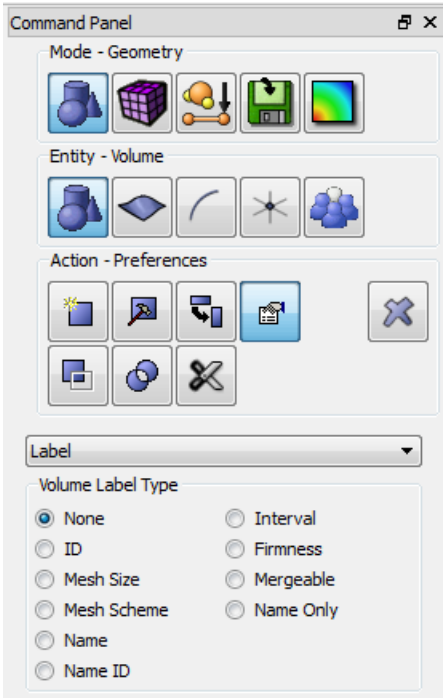
Size - The mesh size set on this entity.

ElementId - The Global Element Id of each element. Will only be labeled for hexes, tets, tris, etc. which are in a block.

SphereId - The id of the sphere element associated with this node, if there is one. A sphere element is only associated with a node if the node (or it's geometry owner) is put into a block.

Note: Three dimensional entity types such as body will have their labels displayed in the center of the entity. Thus, in the **smooth shade** and **hidden line** graphics modes the labels will be hidden

The GUI includes command panels to manipulate the labels settings for any given entity type. The command panel for the Volumes labels settings is shown below as an example:



Graphics Camera

One way to change what is visible in the graphics window is to manipulate the camera used to generate the scene. A scene camera has attributes described below, and depicted graphically in Figure 1. The values of these camera attributes determine how the scene appears in the graphics window.

These view settings may be accessed in the GUI via the **Display/View Point** menu.

Position (From) - The location of the camera in model coordinates.

View Direction (At) - The focal point of the camera in model coordinates.

Up Direction (Up) - The point indicating the direction to which the top of the camera is pointing. The Up point determines how the camera is rotated about its line of sight.

Projection - Determines how the three-dimensional model is mapped to the two-dimensional graphics window.

Perspective Angle - Twice the angle between the line of sight and the edge of the visible portion of the scene.

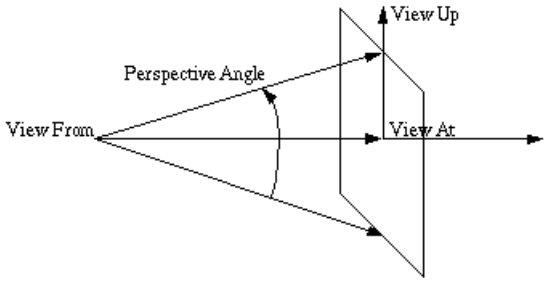


Figure 1: Schematic of From, At, Up, and Perspective Angle

The camera can be moved to one of several predefined orientations using the command

```
View {Front | Back | Top | Bottom | Right |
Left | Iso}
```

At any time, the camera can be moved back to its original position and view using the command

```
View Reset
```

To see the current settings of these attributes, use the command

```
List View
```

The current value of the view attributes will be printed to the terminal window, along with other useful view information such as the current graphics mode and the width of the current scene in model coordinates.

Camera Attributes can be changed using the Rotate, Zoom and Pan commands, or directly as follows.

Changing Camera Attributes Directly

Camera attributes are most easily modified using interactive mouse

manipulation (see Mouse-Based View Navigation) or using the rotate, pan and zoom commands. However, the camera attributes can also be modified directly with the following commands:

```
From <x y z>

At <x y z>

At
{Body|Volume|Surface|Curve|Vertex|Hex|Tet|Wedge|Tri|Face|Node}
<id_list>

Up <x y z>

Graphics Perspective <On|Off>

Graphics Perspective Angle <degrees>
```

If **graphics perspective** is **on**, a perspective projection is used; if graphics perspective is off, an orthographic projection is used. With a perspective projection, the scene is drawn as it would look to a real camera. This gives a three-dimensional sense of depth, but causes most parallel lines to be drawn non-parallel to each other. If an orthographic projection is used, no sense of depth is given, but parallel lines are always drawn parallel to each other.

In a perspective view, changing the **perspective angle** changes the field of view by changing the angle from the line of sight to the edge of the visible scene. The effect is similar to a telephoto zoom with a camera. A smaller perspective angle results in a larger zoom. This command has no effect when graphics perspective is off.

The GUI tool bar button for changing the graphics perspective mode is as follows:



Graphics Modes

By default, the scene is viewed as a smoothshaded model. That is, only curves and edges are drawn, and surfaces are transparent. Surfaces can be drawn differently by changing the graphics mode:

```
Graphics Mode {Wireframe | Hiddenline |
Smoothshade | Transparent  } [Geometry | Mesh
| Highlight]
```

The GUI tool bar buttons for manipulating the graphics modes are as follows:



Examples and a brief description of each mode are shown below


WireFrame - Surfaces are invisible. (This mode can also be accessed by typing **'wireframe'** at the command prompt.)


HiddenLine - Surfaces are not drawn, but they obscure what is behind them, giving a more realistic representation of the view. (This mode can also be accessed by typing **'hiddenline'** at the command prompt.)


SmoothShade - Surfaces are filled and shaded. Shaded colors are interpolated across the entire surface using the graphics lighting model. This produces the most realistic results. (This mode can also be accessed by typing **'shaded'** at the command prompt.)


Transparent - Renders surfaces as semi-transparent shaded images, allowing objects to shine-through from behind. Is not supported on all platforms, and generally requires advanced graphics hardware. (This mode can also be accessed by typing **'transparent'** at the command prompt.)

This determines what pattern is used to draw lines behind surfaces (e.g. dotted, dashed, etc.; click here for a list of valid line patterns).

Displaying Using the Element Facets

There is another option that is similar to a graphics mode, set with the command

```
Graphics Use Facets [On|Off]
```

This command determines how shaded and filled surfaces are drawn when they are meshed. If Graphics Use Facets is on, the mesh facets (element faces) are used to render the model. This is particularly helpful for curved surfaces which may cut through some of the mesh faces. A comparison of graphics facets on and off is shown below.

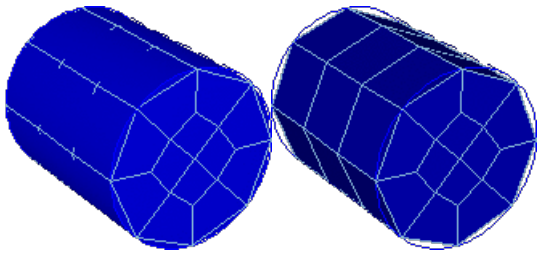


Figure 1. A meshed cylinder shown with graphics facets off (left) and graphics facets on (right); note how geometry facets on the curved surface obscure mesh edges when facets are off.

Displaying Composite Surface Lines

Composite surfaces are surfaces that have been joined together using virtual geometry. By default, the underlying surfaces are marked with dashed lines. To toggle this setting so that underlying surfaces are not shown, use the following command:

```
Graphics Composite {On|Off}
```

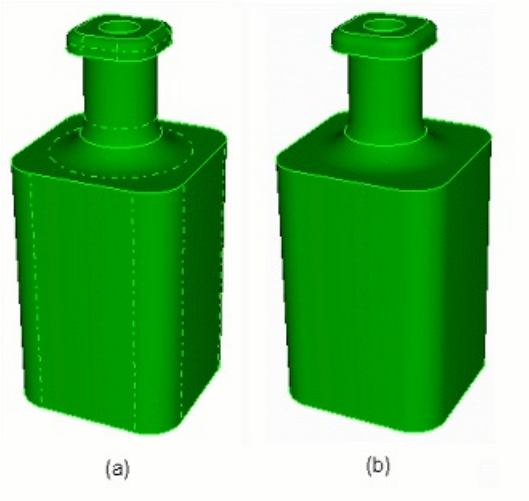


Figure 2. A part shown with (a) composite surfaces displayed (b) composite surfaces not displayed

The GUI tool bar button for toggling the display of graphics composites is as follows:



Graphics Window Size and Position

By default in the command line version, CUBIT will create a single graphics window when it starts up (to run CUBIT without a graphics window, include -nographics on the command line when launching CUBIT.) The graphics window position and size is most easily adjusted using the mouse, like any other window on an X-windows screen. However, the size of the graphics window can also be controlled using the following commands:

```
Graphics WindowSize <width_in_pixels>
<height_in_pixels>

Graphics WindowSize Maximum

Graphics WindowSize Minimum
```

After using the **Graphics WindowSize Maximum** and **Graphics WindowSize Minimum** commands, the previous window size can be restored by using the command

```
Graphics WindowSize Restore
```

The position of the graphics window can also be controlled using the **Graphics WindowLocation** command.

```
Graphics WindowLocation <x> <y>
```

The <x> and <y> coordinates refer to the distance in pixels from the upper left hand corner of the monitor.

In addition, on Unix workstations, the graphics window size and position can be controlled by placing the following line in the user's .Xdefaults file:

```
cubit.graphics.geometry XxY+xpos+ypos
```

where the **X** and **Y** are window width and height in pixels, respectively, and **xpos** and **ypos** are the offsets from the upper left hand corner.

Using Multiple Windows

You can use up to ten graphics windows simultaneously, each with its own camera and view. Each window has an ID, from 1 to 10, shown in the title bar of the window. Commands that control camera attributes apply to only one window at a time, the active window. Currently, the display lists of all windows are identical.

The following commands are used to create, delete, and make active additional graphics windows. These commands are also valid in the GUI (by typing at the command line prompt.)

```
Graphics Window Create [ID]

Graphics Window Delete <ID>

Graphics Window Active <ID>
```

Hardcopy Output

CUBIT's Graphical User Interface provides the capability to print the contents of the graphics window directly to a printer. Use **File/Export/Screen Shot** to access this functionality.

In addition, a command line option is provided for dumping the contents of the graphics window to postscript or image files.

The command for generating hardcopy output files is:

```
Hardcopy '<filename>' {jpg | png | bmp | pnm |
tiff | eps} [Window <window_id>]
```

Each of these options saves the view in the specified window (or the current window), to the specified file, in the format indicated. The file can then be sent to a printer or inserted into another document.

Screen Capture Programs

It should also be noted that many commercial applications are available for capturing screen images. In many cases, these applications may be more convenient for interactively capturing and saving a portion of the screen than the **Hardcopy** command discussed above. On UNIX platforms, the XV utility written by John Bradley is a good choice. In some cases this utility or its equivalent may be included with your system software. For Windows users, the *Print Screen* button will send a copy of the screen to the clipboard which can then be pasted into a paint program.

Graphics Lighting Model

For shaded graphics display modes, the lighting model controls the intensity of the highlights and shadows for objects displayed in the graphics window. CUBIT offers two commands for controlling the lighting model.

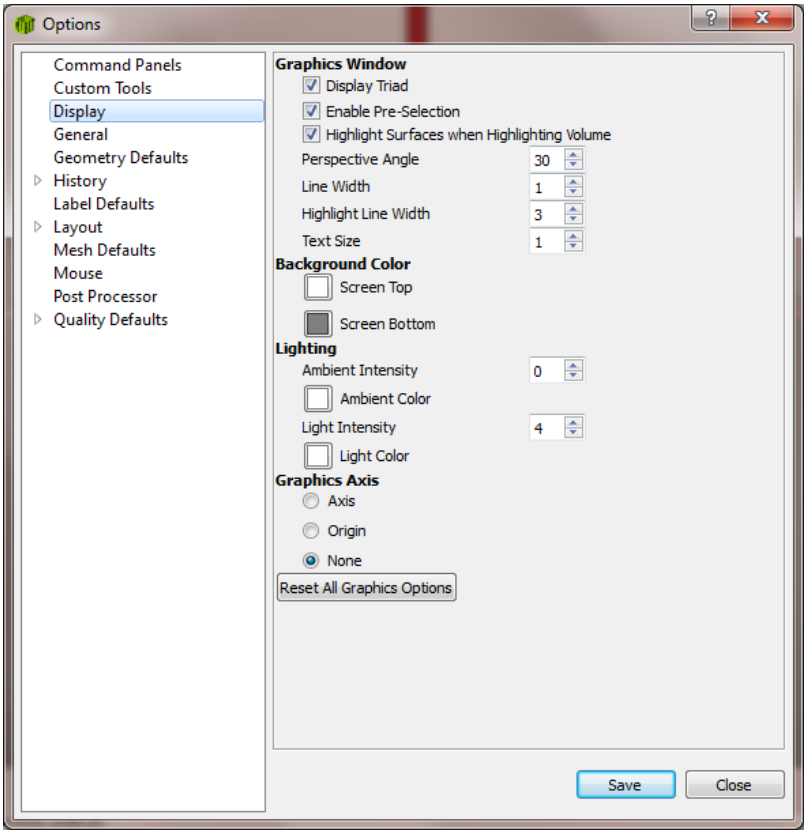
```
Graphics Ambient Intensity {<intensity> | <r g
b>}

Graphics Light Intensity {<intensity> | <r g
b>}
```

The **ambient** intensity is the light available in the environment. There is no particular direction to the light source. In contrast, the **light** intensity is the effect of a simulated light source placed at the viewer's line of sight. The **light** intensity affects the intensity of the highlights and shadows, while the **ambient** intensity affects the brightness of the objects in the overall scene.

An **intensity** value from 0 to 1 can be used, where 0 represents no light and 1 represents maximum. Alternatively **r g b** color components can be used. This changes the color of the directional or ambient light source, affecting the resulting color of the objects in the model.

The GUI Options panel for manipulating these settings is found under Tools/Options and is shown below:



Mesh Visualization

A volume mesh can be viewed one layer at a time using a visualization tool known as mesh slicing. This tool divides the elements of one or more volumes into axis-aligned layers, and then allows the mesh to be displayed one layer at a time. Mesh slicing is especially useful to view the quality of swept meshes that are axis aligned.

Notes on Mesh Slicing

Mesh slicing is only intended to be a rough visualization tool. Because the average mesh edge length is used to determine the thickness of each layer, a layer may be more than one element deep. Unstructured meshes, meshes with large variations in edge length, and non-axis-aligned meshes will be more difficult to visualize with this tool.

Mesh Slicing Command

Mesh slicing can be started either by entering a keypress in the graphics window, which slices the mesh of the entire model, or by entering the command

```
Graphics Slice {Body | Volume} <id_range> Axis
{X | Y | Z}
```

which slices only the bodies or volumes indicated, with a plane along the axis specified.

Key presses in the graphics window which control mesh slicing are summarized in the following table.

Key	Action
X,Y or Z	Initiate mesh slicing using the X, Y or Z plane
K	Move the slicing plane in the positive coordinate direction
J	Move the slicing plane in the negative coordinate direction
S	Toggles drawing single or multiple slice layers in the view
Q	Exit from mesh slicing mode

See Graphics Clipping Plane for instructions on clipping the graphics using the GUI clipping plane.

Miscellaneous Graphics Options

In addition to the commands discussed above, there are several other graphics system options in Cubit that can be controlled by the user.

They include:

- Silhouette Lines
- Line Width
- Highlight Line Width
- Text Size
- Point Size
- Graphics Status
- Graphics Scale
- Model Axis
- Corner Axis
- Resetting the Graphics
- Shrink
- Facet Tolerance

Silhouette Lines

Some shapes, such as cylinders, are drawn with silhouette lines; these lines don't represent true geometric curves, but help visualize the shape of a surface. Silhouette lines can be turned on or off with the command

```
Graphics Silhouette [On|Off]
```

The pattern used to draw silhouette lines can be set using the command

```
Graphics Silhouette Pattern [Solid | Dashdot |
Dashed | Dotted | Dash_2dot | Dash_3dot |
Long_dash | Phantom]
```

Line Width

This option controls the width of the lines used in the **wireframe**, **shaded**, **transparent**, **hiddenline** and **truehiddenline** displays. The default is 1 pixel wide. The command to set the line width is

```
Graphics LineWidth <width_in_pixels>
```

Highlight Line Width

This option controls the width of the lines used when highlighting an entity. Setting this to a width greater than the global line width often makes it easier to locate highlighted entities. If this setting has not been changed, the line width set in the command above is used. After using this command, it is necessary to refresh the graphics by either typing "display" or clicking the Refresh Graphics button. The command to set the highlighting line width is

```
Highlight LineWidth <width_in_pixels>
```

Text Size

This option controls the size of text drawn in the graphics window. The size given in this command is the desired size relative to the default size. After using this command, it is necessary to refresh the graphics by either typing "display" or clicking the Refresh Graphics button. The command to set the text size is

```
Graphics Text Size <size>
```

Point Size

This option controls the size of points drawn in the graphics window, such as vertices or heads of vectors; alternatively, the size of points representing nodes or vertices can be set independently of the global point size. The commands to set the point sizes are

```
Graphics Point Size <size>

Graphics [Node|Vertex] Point Size <size>
```

Graphics Status

All graphics commands can be disabled or re-enabled with the command

```
Graphics {On|Off}
```

While graphics are off, changes in the model will not appear in the graphics window, and all graphics commands will be ignored. When graphics are again turned on, the scene will be updated to reflect the current state of the model.

Graphics Scale

A graphical scale can be drawn in the graphics window within the viewing area to obtain a bearing on model or part sizes. The command to turn the graphical scale on and off is:

```
Graphics Scale [On|Off]
```

Model Axis

The model axis may be drawn in the scene at the model origin. The axis is controlled with the command

```
Graphics Axis [Type <AXIS | Origin>] [On|Off]
```

The command is used to specify whether the model axis is visible, and to determine how the axis is drawn. If you include Type Axis , the axis will be drawn as three orthogonal lines; if you include Type Origin, the axis will be drawn as a circle at the model origin.

Corner Axis (Triad)

By default, an axis appears in the corner of the graphics window. This corner axis, also called the triad, can be disabled or re-enabled with the command

```
Graphics Triad [On | Off]
```

Resetting the Graphics

Many of the graphic options can be reset back to default values with the command:

```
Graphics Reset
```

The graphic options set to defaults are:

- ambient and spot light intensity
- background color
- text size
- graphics mode

- silhouetting
- point size
- view type (Perspective)

In addition, this command also:

- centers the view on all visible entities (Zoom Reset)
- turns all labeling off
- turns vertex visibility off
- turns mesh and geometry visibility on
- moves the graphics camera back to its original position (View Reset)

Shrink

The shrink graphics attribute allows you to view the elements shrunken about their centroid. This is useful for viewing 3D meshes, permitting viewing of interior elements. It may also be useful for visually inspecting the mesh for missing elements. To use the shrink option use:

> **graphics shrink <value>**
> **draw hex <range>**
> **draw tet <range>**
> **etc...**

where **value** is a number between 0 and 1. One (1) will shrink the elements to a point, while zero (0) will not shrink the elements. The following figures illustrate the effect of element shrink on a hex mesh.

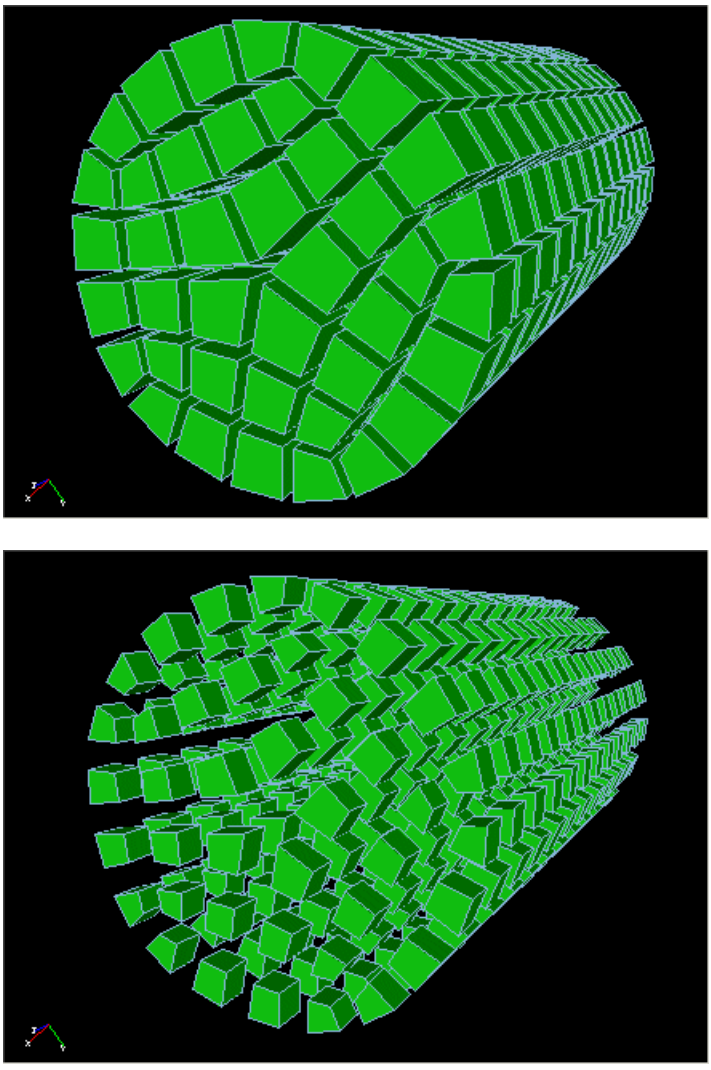


Figure 1. Top: shrink=0.2, Bottom: shrink=0.5

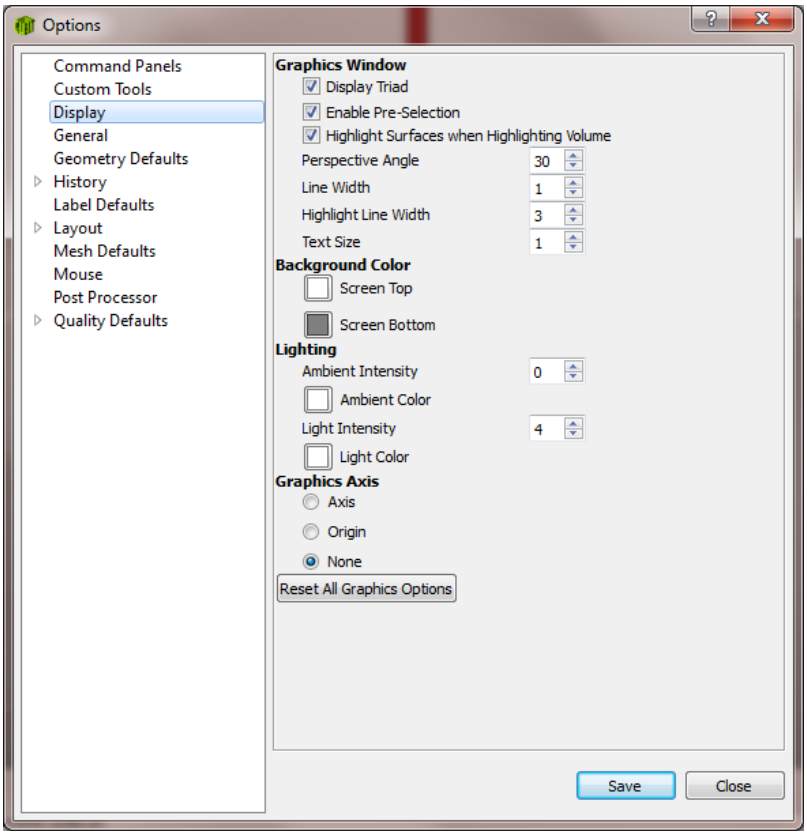
Facet Tolerance

The graphics tolerance commands change the way that facets are drawn in the graphics window. It does not affect the underlying geometry, just the graphics display. It can be useful to change the facet tolerance on large models if the refresh speed is slow.

```
Graphics Tolerance [ [ANGLE|Distance]
<val>|Default ]
```

Specifying an **angle** will change the maximum allowable angle between neighboring facets. The **distance** option will set a maximum distance between adjacent facets. Increasing either of these numbers will result in coarser facets. The **default** option will return values to their default settings.

The GUI Options panel for manipulating these settings is found under Tools/Options and is shown below:



Mouse Based View Navigation: Zoom, Pan and Rotate

The mouse can be used to navigate through the scene using various view transformations. These transformations are accomplished by clicking a mouse button in the graphics window and dragging, sometimes while holding a modifier key such as Shift or Control. When run with graphics on, CUBIT is always in mouse mode; that is, mouse-based transformations are always available, without needing to enter a CUBIT command.

Mouse-based view transformations are accomplished by placing the pointer in the graphics window and then either holding down a mouse button and dragging, or by clicking on a location in the graphics window. Some functions also require one or more modifier keys to be held down; the modifier keys used in CUBIT are Shift Shift and Control Ctrl. Each of the available view transformations has a default binding to a mouse button-modifier key combination. This binding can be changed by the user if desired. Transformations and button mappings are summarized in the following table.

Note: These settings are applicable only to the UNIX command line version of CUBIT. For a description of the Graphical User Interface Mouse Operations see GUI View Navigation.

The bindings are based on the following mouse button definitions:

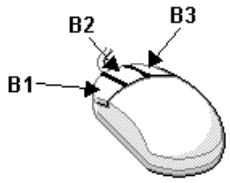


Figure 1. Default Mouse Function Mappings for the Command Line

Table 1. Mouse Function Bindings for Zoom, Pan, and Rotate

Function	Description	Binding
Rotate	Rotates the scene about the camera axis. Dragging the mouse near the center of the graphics window will rotate the camera's X- or Y-axis; dragging near the edge of the window will rotate about the Z-axis (i.e. about the camera's line of sight). Type a u in the graphics window to see the dividing line between the two types of rotation.	**B1**
Zoom	Zooms the scene in or out by clicking the mouse in the graphics window and dragging up or down. If the mouse has a wheel, the wheel will also zoom.	**B2**
Pan	"Drags" the scene around with the mouse	**B3**
Navigational Zoom	Zooms the scene by moving both the camera and its focal point forward.	Shift **B2**
Telephoto Zoom	Zooms the scene by decreasing the field of view.	Shift Ctrl **B2**
Pan Cursor	Click on new center of view	Shift **B3**

Changing the View Transformation Button Bindings

The default mapping of functions to mouse buttons, described in the **Default Mouse Function Mappings** table above, can be modified. There are two ways to assign a function to a button/modifier combination.

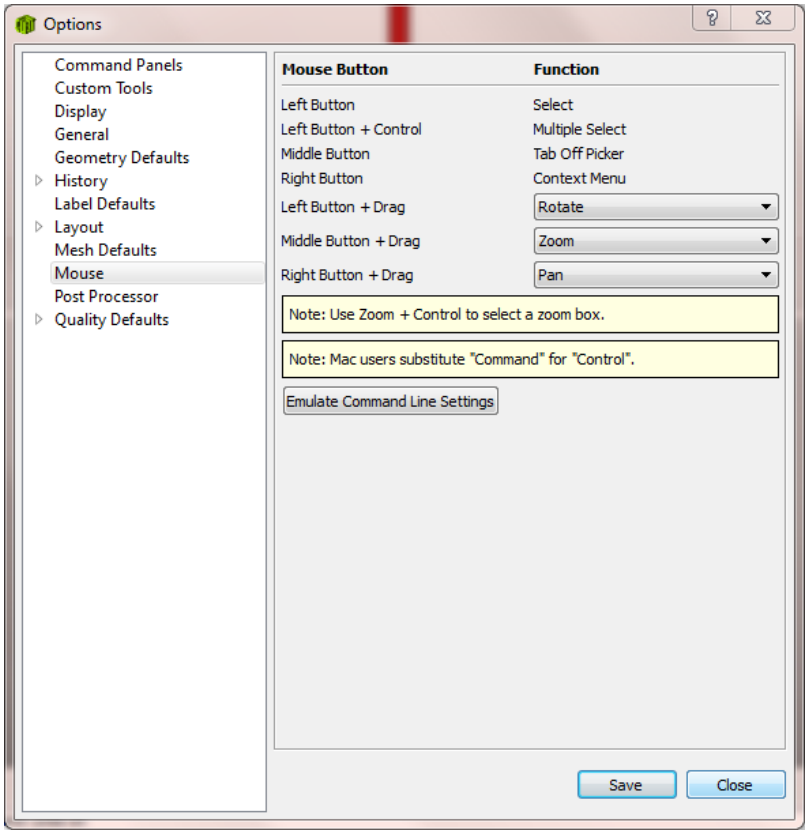
First, you can use the command

```
Mouse Function <function_id> Button <1|2|3>
[Shift][Control]
```

Type **Help Mouse Function** to see a list of function IDs that may be used in this command.

Second, you can assign functions interactively. To do so, first put the pointer into a graphics window and then hit the F key. On-screen instructions will lead you through the rest of the process.

The GUI Options panel for managing the mouse bindings can be found at **Tools/Options/Mouse**, and is as follows:



Saving and Restoring Views

After performing view transformations, it may be useful to return to a previous view. A view is restored by setting the graphics camera attributes to a given set of values. The following keys, pressed while the pointer is in the graphics window, provide this capability:

> **`V`** - Restores the view as it was the last time Display was entered.
>
> **`F1 to F12`** - These function keys represent 12 saved views. To save a view, hold down the Control key while pressing the function key. To restore that view later, press the same function key without the Control key.

Note: In the Graphical User Interface version the **F1**, **F2** and **F3** keys are

used as an alternate form of dynamic viewing, therefore the ability to save views is not currently supported in the GUI.

You can also save a view by entering the command

```
View Save [Position <1-12>] [Window <window_id>]
```

The current view parameters will be stored in the specified position. If no position is specified, the view can be restored by pressing V in the graphics window. If a position is specified, the view can be restored with the command

```
View Restore Position <1-12> [Window
<window_id>]
```

These commands are useful in as entries in a .cubit startup file. For example, to always have **F1** refer to a front view of the model, the following commands could be entered into a .cubit file:

```
From 0 1

At 0

Up 0 1 0

Graphics Autocenter On

View Save Position 1
```

The first three commands set the orientation of the camera. The fourth command ensures that the model will be centered each time the view is restored. The final command saves the view parameters in position 1. The view can be restored by pressing **F1** while the cursor is in a graphics window.

Additionally, you can change the 'gain' on the mouse movements by changing the mouse gain setting, via the command:

```
Mouse Gain <value>
```

where a value of 3 would be 3X as sensitive to mouse movements, and a value of 0.5 would be half as sensitive.

```
Set ReverseZoom {on|off}
```

Another user preference, the direction of 'zooming' obtained by using the mouse can be 'flipped', by toggling the reversezoom setting.

Saving Graphics Views

The current graphics view can be saved and restored using the following commands:

```
View Save Position <n>

View Restore Position <n>
```

When you save a view, you save the camera settings in effect at the time the command is issued. When you restore the view, the camera is returned to the saved position, orientation, and field of view.

If autocenter is on at the time you save the view, then restoring the view will automatically adjust the camera settings to center on the entire model and fit the entire model on the screen, a lot like "zoom reset." You turn autocenter on by typing "graphics autocenter on."

Example of how to **save a top view**:

```
at 0

from 0 1 0

up 1 0

graphics autocenter on

view save position 3
```

Use this command to restore that view:

```
view restore position 3
```

The view will then be looking down the y-axis, with the x-axis to the top and the z-axis to the right. The model will be centered in the view and zoomed so that everything just fits into the graphics window. This is true even if the model is not centered on the origin.

If autocenter is off when the "view save" command is issued, the camera is not adjusted to fit the scene into the graphics window. Instead, it is placed exactly where it was at the time the "save" command was issued.

Note that many graphics commands, such as "at", "from", and "up", do not change what appears in the graphics window until a "display" command is issued. They do, however, take immediate effect internally, and they do affect what is saved by the "view save" command.

In the command line version of CUBIT, you can save a view by holding down the shift key and pressing one of the function keys (F1-F12). Each function key corresponds to a different saved view. A total of 12 views can be saved. A view can be restored at a later time by pressing the appropriate function key WITHOUT holding down the shift key.

It may be useful to save views in your cubit file so that they are available every time you run CUBIT. Use CUBIT to save front, top, and side views in positions 1, 2, and 3. If views are saved in your cubit file, it is convenient to add a "view reset" command after the views have been saved. Then the graphics will initially appear as they would if the view commands had not been included in your cubit file.

Updating the Display

Among the most common graphics-related commands is:

> **Display**

This command clears all highlighting and temporary drawing, and then redraws the model according to the current graphics settings. The GUI tool bar button for executing this command is:



Two related commands are:

```
Graphics Flush

Graphics Clear
```

Graphics Flush redraws the graphics without clearing highlighting or temporary drawing. **Graphics Flush** is useful when a previously executed command modified the graphics and didn't update the screen and the user wishes to update the display. The **Graphics Clear** command clears the graphics window without redrawing the scene, leaving the window blank.

NOTE: Although most changes to the model are immediately reflected in the graphics display, some are not (for graphics efficiency). Typing **Display** will update the display after such commands. **Ctrl-R** will also update the display as long as the mouse is in the graphics window.

Prevent Graphics From Updating

For especially large models, it may take excessively long to update the display after an action has been performed. To prevent the graphics from automatically updating, use the following command:

```
Graphics Pause
```

This command prevents the graphics window from being updated until the next time the **Display** command is issued**.**

NOTE: The **Plot** command is synonymous to the **Display** command, and either can be used with identical results**.**

Geometry, Mesh, and BC Entity Visibility

The visibility of geometry, mesh, BC and Genesis entities can be turned on or off, either individually or globally. After visibility is turned off, the associated entities will remain invisible until visibility is turned on again.

The command to control global visibility is:

```
{Mesh|Geometry|BC} { [Visibility] [on|off] }
```

This command sets the global visibility on or off for *all* mesh, geometry, or BC entities, respectively. Turning off BC visibility also affects Genesis entities such as blocks, sidesets, and nodesets. Global visibility settings take precedence over the visibility set on individual entities. By default, Mesh and Geometry visibility is on, and BC visibility is off.

Global visibility of geometry, mesh, and BC entities can also be controlled from these tool bar buttons in the GUI (from left to right):



The command to control the individual visibility of geometry entities is:

```
{ {Body|Curve|Surface|Volume|Vertex} <range> }
[Mesh][Geometry] Visibility [On|Off]
```

If the **Mesh** keyword is included, only the visibility of the *mesh* belonging to the specified geometric entity is affected. Similarly, if the **Geometry** keyword is included, only the visibility of the *geometry* is affected. If neither keyword is included, the command is identical to including both keywords.

Invisibility of geometry is inherited; visibility is not. For example, if a volume is invisible, its surfaces are also invisible unless they also belong to some other visible volume. As another case, if the volume is visible, but a surface is set to invisible, the surface will not follow its parent's visibility setting, but will remain invisible.

If vertex visibility is turned on, the vertices of the geometry become visible. The default for vertex visibility is off. The default for all other geometry entities is on.

The commands to control visibility of edges and nodes are:

```
Edge [Visibility] [On|Off]

Node [Visibility] [On|Off]
```

These commands set the global visibility on or off for *all* edges or nodes, respectively. If edge visibility is off, mesh edges will not be drawn when mesh faces are drawn. Edge visibility is on by default; node visibility is off by default. Face visibility is always on when mesh visibility is on.

The command to control the individual visibility of genesis entities is:

```
{Block|Nodeset|Sideset} <range> visibility
[{on|off}]
```

Genesis entities and boundary conditions are best viewed with geometry and mesh visibility off and BC visibility on.

Entity visibility for individual geometry and Genesis entities can also be controlled via context (right-click) menus in the Tree and in the graphics window.

Entities that are not visible can still be drawn temporarily using the "draw" command to display one or more specific entities.

Command Line View Navigation: Zoom, Pan and Rotate

Commands used to affect camera position or other functions are listed below. All rotation, panning, and zooming operations can include the **Animation Steps** qualifier, makes the image pass smoothly through the total transformation. Animation also allows the user to see how a transformation command arrives at its destination by showing the intermediate positions.

Rotation

```
Rotate <degrees> About [Screen | Camera |
World] {X | Y | Z} [Animation Steps
<number_steps>]

Rotate <degrees> About Curve <curve>
[Animation Steps <number_steps>]

Rotate <degrees> About Vertex <vertex_1>
Vertex <vertex_2> [Animation Steps
<number_steps>]
```

Rotation of the view can be specified by an angle about an axis in model coordinates, about the camera's "**At**" point, or about the camera itself. Additionally rotations can be specified about any general axis by specifying start and end points to define the general vector. The right hand rule is used in all rotations.

Plain degree rotations are in the Screen coordinate system by default, which is centered on the camera's **At** point. The **Camera** keyword causes the camera to rotate about itself (the camera's From point). The **World** keyword causes the rotation to occur about the model's coordinate system. Rotations can also be performed about the line joining the two end vertices of a curve in the model, or a line connecting two vertices in the model.

Panning

```
Pan [{Left|Right} <factor1>] [{Up|Down}
<factor2>] [Screen | World ] [Animation Steps
<number_steps>]
```

Panning causes the camera to be moved up, down, left, or right. In terms of camera attributes, the **From** point and **At** point are translated equal distances and directions, while the perspective angle and up vector remain unchanged. The scene can also be panned by a factor of the graphics window size.

Screen and World indicate which coordinate system **<factor>** is in. If **Screen** is indicated (the default), **<factor>** is in screen coordinates, in which the width of the screen is one unit. If **World** is indicated, **<factor>** is expressed in the model units.

Zooming

```
Zoom Screen <factor> [Animation Steps
<number_steps>]

Zoom <x_min> <y_min> <x_max> <y_max>
```

```
[Animation Steps <number_steps>]

Zoom {Group | Body | Volume | Surface | Curve
| Vertex | Hex | Tet | Face | Tri | Edge |
Node} <id_range> [Animation Steps
<number_steps>] [Direction {options}]

Zoom cursor [click|drag][animation steps
<number>]

Zoom Reset
```

Zoom Screen will move the camera **<factor>** times closer to its focal point. The result is that objects on the focal plane will appear **<factor>** times larger.

Zooming on a specific portion of the screen is accomplished by specifying the zoom area in screen coordinates; for example, **Zoom 0 .25 .25** will zoom in on the bottom left quarter of the screen.

Zooming on a particular entity in the model is accomplished by specifying the entity type and ID after entering **Zoom**. The image will be adjusted to fit bounding box of the specified entity into the graphics window, and the specified entity will be highlighted. You can specify a final direction to look at when zooming by using the direction option.

To center the view on all visible entities, use the **Zoom Reset** command.

The GUI tool bar buttons for controlling zoom in, zoom out, and zoom reset are as follows:



Entity Selection

- Command Line Entity Specification
- Extended Command Line Entity Specification
- Selecting Entities With the Mouse
- Extended Selection Dialog
- Part Classification with Machine Learning
- Training CAD Operations with Machine Learning
- Select Visible Surfaces

CUBIT Entity specification is a means of selecting objects or groups of objects. Entities can be selected from the command line using entity specification parameters, or directly in the graphics window using the mouse. This chapter describes these methods of entity selection.

Extended Selection Dialog

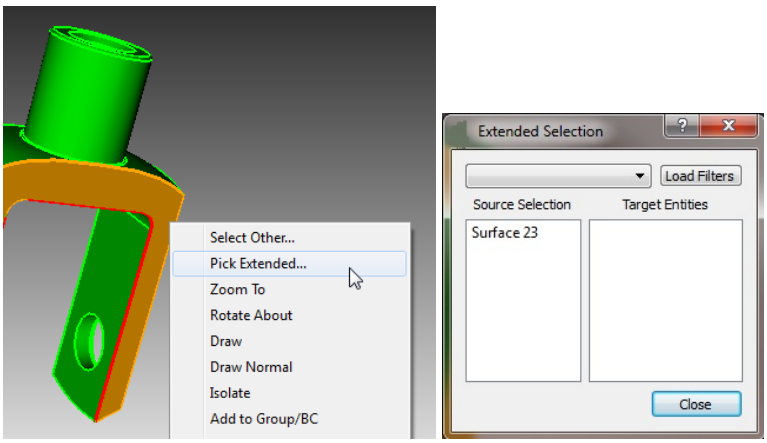
Selecting entities in the graphics window can sometimes be complicated. The Extended Selection Dialog leverages the combination of Python and the CubitInterface class to give users a very powerful mechanism for creating and managing custom selection filters.

Accessing the Dialog

The dialog is accessible any time a geometry entity is selected in the graphics window. Consider this workflow:

Launch the Dialog

- Select a geometric entity
- Get the context menu and select "Pick Extended...". The Extended Selection dialog will be shown with the selected entity (or entities) listed in the **Source Selection** window.

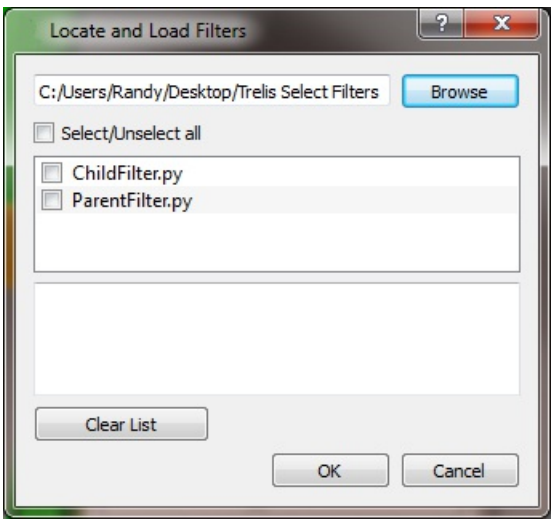


When the dialog is first shown, no filters are available.

Load Existing Filters

Load existing filters by:

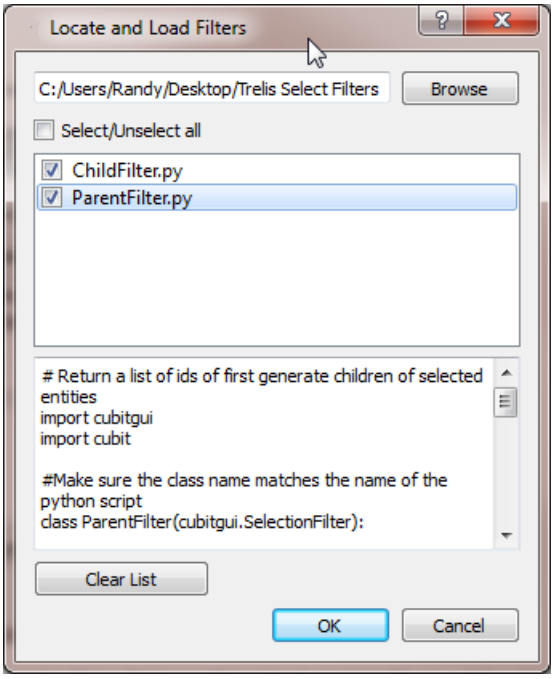
- Press the "**Load Filters**" button
- The **Locate and Load Filters** dialog will be shown



The very first time this dialog is shown, the path to the filters folder will be

blank and no filters will be shown in the filter list. Press the "**Browse**" button and select the folder that contains the custom filters. This folder can be anywhere on the file system. Cubit will remember the location and use it during subsequent sessions. The folder may be changed at any time.

A list of custom filters, written in Python, will be shown. Select any given filter to examine its contents. Check all filters to be included in the menu for Extended Filters. Then press "**OK**".

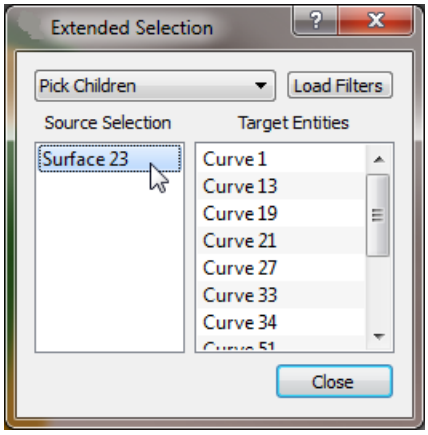


At this point, all of the filters selected in the Locate and Load Filters dialog will be available for use in the Extended Selection dialog.

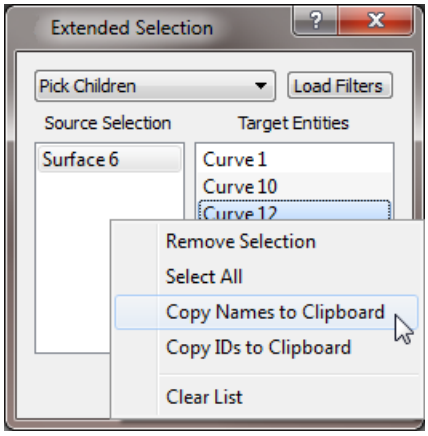


Use a Filter

Use the pull down menu to select a filter. Click on an entity in the **Source Selection** list. Geometric entities that fit the filter criterion will be shown in the **Target Entities** list.



- Make selections in the **Target Entities** list
- If a pickwidget is active, selected IDs will be copied into the pickwidget
- Access the context menu in the **Target Entities** list for other options, including copying entity names or IDs to the clipboard.



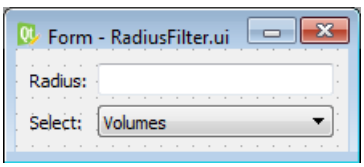
Dragging from Target Entities to Source Selection

Depending on the nature of the selection filter it may be useful to 're-seed' the **Source Selection** with an item from **Target Entities**. Simply drag an item or items from **Target Entities** into the **Source Selection** list.

Creating Parameterized Filters

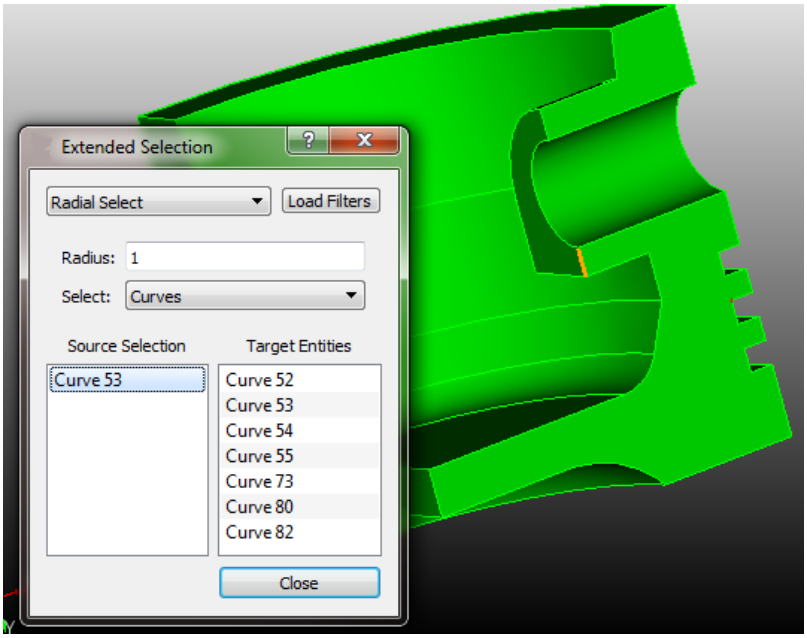
Parameterized filters will require a user interface which will be added into the extended selection filter dialog. The user interface can be made using Qt's Designer, which is a free tool that ships with the Qt toolkit. The Qt Designer tool produces an XML file that will be read by Cubit and automatically included in the Extended Selection Filter dialog.

For example, if we wanted to create a selection filter that would select all entities of a certain type within a certain radius of a source entity, we would require a user interface that captures the desired radius and the desired entity type to be selected. An image of that user interface is shown below. The image was copied directly from Qt Designer.



Notice two input fields: 1) a Line Edit to capture the desired selection radius and 2) a Combo Box that contains "Volumes", "Surfaces", "Curves", "Vertices" to specify the target entity selection type. The

extended selection filter that contains this custom interface is shown below. The example shows a selection of all curves within 1 unit of the source selection.



Writing Custom Python Filters

The class CubitInterface is used by the GUI to drive Cubit and access its database. You can read about the Python Interface used by Cubit for more details. Suffice to say, all of the functions and data included in CubitInterface are available to Python programmers.

Extended Selection custom filters are written in Python. Follow the instructions below, save the filters on the file system, then load the filters as explained above.

A Simple Example Filter

This first example shows a filter that will return a list of first generation children of the selected entities. No user input is required and no additional user interface is necessary.

ChildFilter.py

```
1  # Return a list of first generation children of selected entities
2  import cubitgui
3  import cubit
4
5  #Make sure the class name matches the name of the python script
6  class ChildFilter(cubitgui.SelectionFilter):
7
8    # This name will be shown in the Extended Selection dialog filter menu
9    def display_name(self):
10     return "Pick Children"
11
12   # This is the filter to be executed
13   def run_filter(self):
14     # Get whatever is in the Source Selection list
15     source_types = self.get_source_types()
16     source_ids = self.get_source_ids()
17     self.clear_target_selections()
18     for st, id in zip(source_types, source_ids):
19       if st == "body":
20         relativetype = "volume"
21       else:
22         if st == "volume":
23           relativetype = "surface"
24         else:
25           if st == "surface":
26             relativetype = "curve"
27           else:
28             if st == "curve":
29               relativetype = "vertex"
30             else:
31               if st == "vertex":
32                 relativetype = "volume"
33               else:
34                 return
35       relative_ids = cubit.get_relatives(st, id, relativetype)
36       for rid in relative_ids:
37         self.add_target_selection(relativetype, rid)
38
```

- The filter's File Name and Class Name must match. In this example, the file name is 'ChildFilter.py' and the class name is "ChildFilter".
- LINE 2: import cubitgui -- this is the Python module that owns the base class (SelectionFilter) from which this filter class is derived.
- LINE 3: import cubit -- this is the Python module that grants access to all of the CubitInterface functions.
- LINE 6: When defining the class do the following:
  - Ensure the class name matches the file name
  - Derive the class from cubitgui.SelectionFilter
- LINE 9: Implement the 'virtual' function "display_name". Return a string that represents the name of the filter. It is this name that will be shown in the Extended Selection dialog's filter menu.
- LINE 13: Implement the 'virtual' function "run_filter. This is the actual filter.
- LINE 15: The function get_source_types() returns a list of the types selected in the "Source Selection" list of the Extended Selection dialog.
- LINE 16: The function get_source_ids() returns a list of the ids selected in the "Source Selection" list of the Extended Selection dialog.
- LINE 17: The function clear_target_selections() clears the Target Entities list of the Extended Selection dialog.
- LINE 18: The example filter begins manipulating data to suit its needs
- LINE 35: The function get_relatives(...) is a member of CubitInterface. It returns a list of ids of a specified type
- LINE 37: The function add_target_selection(<type> <id>) adds one instance of an entity-type/id to the Target Entities list.

The Selection Filter Class

As mentioned above, the custom Python filters must implement a class and that class must be derived from the SelectionFilter class. The SelectionFilter class is available in the Cubit SDK. The SDK is available to any user.

A Sample Filter that includes Additional User Interface

In order to include user input into an extended selection dialog, two additional things must happen:

1. The developer must create the user interface definition file (.ui file) using Qt Designer.
2. An additional function must be implemented in the class definition of the Python filter. The additional function is called **get_ui_file()**.

Qt UI objects supported by the extended selection filter include:

- **QLineEdit** - std::string get_line_edit_value (object-name)
- **QComboBox** - std::string get_combo_box_value (object-name)
- **QRadioButton** - bool get_radio_button_value (object-name)
- **QCheckBox** - bool get_check_box_value (object-name)
- **QSpinBox** - int get_spin_box_value (object-name)
- **QSlider** - int get_slider_value (object-name)

As the example Python code shows, the Qt objects are referenced by name. These code snippets below are not complete. Complete examples and a video tutorial are available from www.csimsoft.com.

```
class RadiusFilter(cubitgui.SelectionFilter):
    def display_name(self):
        return "Radial Select"

    def get_ui_file(self):
        script_dir = os.path.dirname(os.path.realpath(__file__))
        return script_dir + "/RadiusFilter.ui"

    def run_filter(self):
        # Get the radius and selection type from the ui
        try:
            radius = float(self.get_line_edit_value("leRadius"))
        except ValueError:
            print("Warning: Could not convert radius to decimal value")
            return

        combo_text = self.get_combo_box_value("cbSelectionType")
        if combo_text == "Volumes":
            selection_type = "volume"
        elif combo_text == "Surfaces":
            selection_type = "surface"
        elif combo_text == "Curves":
            selection_type = "curve"
        elif combo_text == "Vertices":
            selection_type = "vertex"
        else:
            return

        self.clear_target_selections()

        # Get the center of our current selection
        source_types = self.get_source_types()
        source_ids = self.get_source_ids()
```

```
center = [0.0, 0.0, 0.0]
for st, id in zip(source_types, source_ids):
    geom_center = cubit.get_center_point(st, id)
    center[0] += geom_center[0]
    center[1] += geom_center[1]
    center[2] += geom_center[2]

num_sources = len(source_ids)
center[0] /= num_sources
center[1] /= num_sources
center[2] /= num_sources

min_coord = [center[0] - radius, center[1] - radius, center[2] - radius]
max_coord = [center[0] + radius, center[1] + radius, center[2] + radius]
range_str = "with x_coord >= {0} and x_coord <= {1} ".format(min_coord[0], max_coord[0])
range_str += "and y_coord >= {0} and y_coord <= {1} ".format(min_coord[1], max_coord[1])
range_str += "and z_coord >= {0} and z_coord <= {1}".format(min_coord[2], max_coord[2])

potential_selection = cubit.parse_cubit_list(selection_type, range_str)
if len(potential_selection) == 0:
    return

# Go through the potential selections and verify that it is within the radius
for target_id in potential_selection:
    geom_center = cubit.get_center_point(selection_type, target_id)
    dist_squared = 0.0
    for index in range(0, 3):
        dist_squared += (geom_center[index] - center[index])**2

    if dist_squared <= radius**2:
        self.add_target_selection(selection_type, target_id)
```

Command Line Entity Specification

CUBIT identifies objects in the geometry, mesh, and elsewhere using ID numbers and sometimes names. IDs and names are used in most commands to specify which objects on which the command is to operate.

These objects can be specified in CUBIT commands in a variety of ways, which are best introduced with the following examples (the portion of each command which specifies a list of entities is shown in blue):

General ranges: Surface 1 2 4 to 6 by 2 3 4 5 Scheme Pave

Combined geometry, mesh, and genesis entities: Draw Sideset 1 Curve 3 Hex 2 4 6

Geometric topology traversal: Vertex in Volume 2 Size 0.3

Mesh topology traversal: Draw Edge in Hex 32

Genesis entity traversal: Draw Block in Hex 32

***All* keyword**: ListBlock all

***Expand* keyword**: my_curve_group expand Scheme Bias Factor 1.5

***Except* keyword**: List Curve 1 to 50 except 2 4 6

In addition to the examples above, there is an extended parsing capability that allows entities to be specified by a general set of criteria. See Extended Entity Specification for details. The following is a simple example of an extended entity specification:

By Criteria: Draw Curve With Length > 3

Types of Entity Range Input

The types of entity range input available in CUBIT can be classified in 4 groups:

1. **General range parsing**

   Entity IDs can be entered individually (volume 1), in lists (volume 1 2 3), in ranges (volume 3 to 7), and in stepped ranges (volume 3 to 7 step 2). The word **all** may also be used to specify all entities of a given type.

   An ID range has the form **<start_id> to <end_id>**. It represents each ID between start_id and end_id, inclusive.

   A stepped ID range has the form **<start_id> To <end_id> {Step|By} <step>**. It represents the set of IDs between **start_id** and **end_id**, inclusive, which can be obtained by adding some integer multiple of step to start_id. For example, **3 to 8 step 2** is equivalent to **3 5 7**.

   The various methods of specifying IDs can be used together. For example:

   draw surface 1 2 4 to 6 vertex all

2. **Topological traversal**

   Topological traversal is indicated using the "in" and "common_to" identifiers, can span multiple levels in a hierarchy, and can go either up or down the topology tree. For example, the following

entity lists are all valid:

vertex in volume 3

volume in vertex 2 4 6

surface common_to volume 2 3

curve common_to surface 2 3

curve 1 to 3 in body 4 to 8 by 2

If ranges of entities are given on both sides of the "in" identifier, the intersection of the two sets results. For example, in the last command above, the curves that have ids of 1, 2 or 3 and are also in bodies 4, 6 and 8 are used in the command.

Topology traversal is also valid between entity types. Therefore, the following commands would also be valid:

draw node in surface 3

draw surface in edge 362

draw hex in face in surface 2

draw node in hex in face in surface 2

draw edge in node in surface 2

draw face common_to volume 1 2

3. **Exclusion**

   Entity lists can be entered then filtered using the "except" identifier. This identifier and the ids following it apply only to the immediately preceding entity list, and are taken to be the same entity type. For example, the following entity lists are valid:

   curve all except 2 4 6

   curve 1 2 5 to 50 except 2 3 4

   curve all except 2 3 4 in surface 2 to 10

   curve in surface 3 except 2 (produces empty entity list!)

   Entity names can also be used to specify the exclusion list. For example:

   curve all except pivot_1

   When using mulitple names to specify the exclusion list it is necessary to use the "in" keyword with parentheses. For example:

   curve all except curve in (pivot_1 top_left)

   In the above example, all curves are in the entity list except the curve named "pivot_1" and the curve named "top_left".

4. **Group expansion**

   Groups in CUBIT can consist of any number of geometry entities, and the entities can be of different type (vertex, curve, etc.). Operations on groups can be classified as operations on the group itself or operations on all entities in the group. If a group identifier in a command is followed immediately by the `expand' qualifier, the contents of the group(s) are substituted in place of the group identifier(s); otherwise the command is interpreted as an operation on the group as a whole. If a group preceding the `expand' qualifier includes other groups, all groups are expanded in a recursive fashion.

For example, consider group 1, which consists of surfaces 1, 2 and curve 1. Surfaces 1 and 2 are bounded by curves 2, 3, 4 and 5. The commands in Table 1, illustrate the behavior of the `expand' qualifier.

Table 1. Parsing of group commands; Group 1 consists of Surfaces 1-2 and Curve 1; Surfaces 1 and 2 are bounded by Curves 2-5.

Command	Entity list produced
Curve in Group 1	Curve 1
Curve in group 1 expand	Curves 1, 2, 3, 4, 5

The `expand' qualifier can be used anywhere a group command is used in an entity list; of course, commands which apply only to groups will be meaningless if the group id is followed by the `expand' qualifier.

Precedence of "Except" and "In"

Several keywords take precedence over others, much the same as some operators have greater precedence in coding languages. In the current implementation, the keyword "Except" takes precedence over other keywords, and serves to separate the identifier list into two sections. Any identifiers following the "Except" keyword apply to the list of entities excluded from the entities preceding the "Except". Table 2 shows the entity lists resulting from selected commands.

Table 2. Precedence of "Except" and "In" keywords; Group 1 consists of Surfaces 1-2 and Curve 1.

Command	Entity list produced
Curve all except 1 in Group 1	(All curves except curve 1)
Curve all except 2 3 4 in Surf 2 to 10	(All curves except 2, 3, 4)

In the first command, the entities to be excluded are the contents of the list "[Curve] 1 in Group 1", that is the intersection of the lists "Curve 1" and "Curve in Group 1"; since the only curve in Group 1 is Curve 1, the excluded list consists of only Curve 1. The remaining list, after removing the excluded list, is all curves except Curve 1.

In the second command, the excluded list consists of the intersection of the lists "Curve 2 3 4" and "Curve in Surf 2 to 10"; this intersection turns out to be just Curves 2, 3 and 4. The remaining list is all curves except those in the excluded list.

Placement in CUBIT Commands

In general, anywhere a range of entities is allowed, the new parsing capability can be used. However, there can be exceptions to this general rule, because of ambiguities this syntax would produce. Currently, the only exception to this rule is the command used to define a sideset for a surface with respect to an owning volume.

Extended Command Line Entity Specification

In addition to basic entity specification, entities may be specified using an extended expression. An extended expression identifies one or more entities using a set of entity criteria. These criteria describe properties of the entities one wishes to operate upon.

Extended Parsing Syntax

The most common type of extended parsing expression is in the following format:

```
{Entity_Type} With {Criteria}
```

Entity_Type is the name of any type of entity that can be used in a command, such as Curve, Hex, or SideSet. Criteria is a combination of entity properties (such as Length), operators (such as >=), keywords (such as Not), and values (such as 5.3) that can be evaluated to true or false for a given entity. Here are some examples:

```
curve with length <1

surface with is_meshed = false

node with x_coord > 10 And y_coord > 0
```

Keywords

These are the keyword defined by extended parsing

Keyword	Description
All, To, Step, By, Except, In, Common_To, Expand, Include	These keywords are used the same way as in basic entity specification. For example: **draw surface all** **draw surface 1 to 5 step 2 curve 1 to 3 in body 4 to 8 by 2** **draw hex in face in surface 2** **draw face common_to volume 1 2** **draw node in hex in face in surface 2 curve 1 2 5 to 50 except 2 3 4** **draw volume 10 include similar** **list block in hex 10**
Not	Not flips the logical sense of an expression - it changes true to false and false to true. For example: **draw surface with not is_meshed**

Of	The "of" operator is used to get an attribute value for a single entity, such as "length of curve 5". Only attributes that return a single numeric value may be used in an "of" expression. There must be only one entity specified after the "of" operator, but it can be identified using any valid entity expression. An example of a complete command which includes the "of" operator is: **list curve with length < length of curve 5 ids**
And, Or	These logic operators determine how multiple criteria are combined. **draw surface with length > 3 or with is_meshed = false**
< > <= >= = <>	These relational operators compare two expressions. You may use = or == for "equals". <> means "not equal". For example: **draw surface with x_max <= 3** **draw volume with z_max <>12.3**
Tolerance	In the case of = or == you may include a tolerence parameter for these relational operators rather than needing to write a more lengthy inequality with >= and <=. For example: **draw curve with length = 5 tolerance 1e-3**
+ - * /	These arithmetic operators work in the traditional manner. **draw surface with length * 3 + 1.2 > 10**
( )	Parentheses are used to group expressions and to override precedence. When in doubt about precedence, use parentheses. **draw surface with length > 3 and (with is_meshed = false or x_min > 1)**

Functions

The following functions are defined. Not all functions apply to all entities. If a function does not apply to a given entity, the function returns 0 or false.

Keyword	Description
ID	the ID of an entity
Length	The length of a curve or edge
Area	The area of a surface.

Volume	The volume of a volume.
Exterior_Angle	Works for curves with an exterior angle greater than (>), less than (<), or equal to (=) a given angle in degrees. This is used if you want to do some operation, such as refinement, on all the reentrant curves or curves with surfaces that form a certain angle.
Radius	Radius of an arc or blend surface.
Normal	Compares the normal of a surface to a vector. **draw surface with normal 1 0 0** The given vector is normalized to a unit vector for comparison. The two vectors are compared using the magnitude of the difference between them. A default tolerance of 1e-6 is used unless specified. **draw surface with normal 1 0 0 tolerance 0.001**
Type	Compares the element type of a block. **select block with type "tetra10"** The element type needs to be in quotes. The generic element type can be used to select any order of that element type. For example: hex => hex8, hex20, hex27
Is_Duplicate	Whether a Volume has a duplicate (exact copy of itself at the same location)
Is_Meshed	Whether a geometric entity has been meshed or not
Is_Spline	Whether a geometric entity is defined using a NURBS representation. Otherwise the entity has an analytic representation.
Is_Blend	Whether a geometric surface is a blend. Blends have a constant principal radius of curvature and meet two or more adjoining surfaces at an angle of approximately 180 degrees. "Fillets" are examples of blend surfaces.
Is_Chamfer	Whether a geometric surface is a chamfer. Chamfers are thin surfaces bounded by surfaces where the exterior angle is approximately 45 or 225 degrees.
Is_Plane	Whether a geometric surface is planar.
Is_Periodic	Whether a geometric surface is periodic, such as a sphere or torus.

Is_Sheetbody	A geometric entity is a sheetbody if it is a collection of surfaces that do not form a solid.
Element_Count	The number of elements owned by this geometric or exodus entity. Only elements of the same dimension as the geometric entity are counted (number of hexes in a volume, number of faces on a surface, etc.). For an exodus entity, all contained mesh entities are counted.
Dimension	The topological dimension of an entity (3 for volumes, 2 for surfaces, etc.).
X_Coord, Y_Coord, Z_Coord	The x, y, or z coordinate of the entity's bounding box center point projected onto the entity.
X_Min, Y_Min, Z_Min	The x, y, or z coordinate of the minimum extent of the entity's bounding box
X_Max, Y_Max, Z_Max	The x, y, or z coordinate of the maximum extent of the entity's bounding box
Is_Merged	Whether a geometry entity has a merge flag on. All geometric entities have one set by default.
Is_Virtual	A flag that specifies whether an entity is virtual geometry. An entity is virtual if it has at least one virtual (partition/composite) topology bridge.
Has_Virtual	An entity "has_virtual" if it is virtual itself, or has at least one child virtual entity
Is_Real	An entity "is_real" if it has at least one real (non-virtual) topology bridge.
Num_Parents	Used to specify geometry entities with a specified number of parent entities. May be used to find "free curves" where num_parents=0 or non-manifold curves where num_parents>2.
Is_Free	Used to specify geometry entities without parent entities. May be used to find free curves and vertices where num_parents=0 or free surfaces that do not form a solid (sheet bodies).
Block_Assigned	Used to specify elements which have been assigned to a block. This is also useful to find elements NOT assigned to a block by using "not block_assigned".

Has_Scheme	Used to specify geometry entities which have been assigned a specified scheme. The scheme name is specified with the keyword string used when setting the scheme. Wildcards can also be used when specifying the scheme name. For example, **draw surface with has_scheme '*map'** will draw surfaces with scheme map or submap.
Similar	Used with the **include** keyword. Compares a list of geometry entities and adds additional entities that are classified as similar. Implemented for **curves**, **surfaces** and **volumes**. Similar is defined as the same geometric length, area or volume using a tolerance of 0.1 percent, as well as the same number of child entities. For example, **"draw volume 10 include similar"** will draw all volumes with the same geometric volume and number child surfaces as volume 10.
Cavity	Used with the **include** keyword. Compares a list of surface entities and adds additional adjacent surfaces that are part of the same cavity. Implemented only for **surfaces**. A cavity is defined as the collection of surfaces bounded by curves where the exterior angle is greater than 180 degrees. For example, **"draw surface 10 include cavity"** will draw surface 10 along with the cavity to which it belongs.
Hole	Used with the **include** keyword. Compares a list of surface entities and adds additional adjacent surfaces that are part of the same hole. Implemented only for **surfaces**. A hole is a special case of a cavity that includes at least one cylindrical surface. For example, **"draw surface 10 include hole"** will draw surface 10 along with the hole to which it belongs.
Blend_Chain	Used with the **include** keyword. Compares a list of surface entities and adds additional adjacent surfaces that are part of the blend_chain. Implemented only for **surfaces**. A blend_chain is defined as the collection of attached surfaces that have the same minimum radius of curvature. For example, **"draw surface 10 include blend_chain"** will draw surface 10 along with the blend_chain to which it belongs.
Chamfer_Chain	Used with the **include** keyword. Compares a list of surface entities and adds additional adjacent surfaces that are part of the chamfer_chain. Implemented only for **surfaces**. A chamfer_chain is defined as the collection of attached surfaces that have the same width and are bounded by exterior angles of 135 and/or 225 degrees. For example, **"draw surface 10 include chamfer_chain"** will draw surface 10 along with the chamfer_chain to which it belongs.

Continuous	Used with the **include** keyword. Compares a list of curve or surface entities and adds additional adjacent entities that are continuous. Implemented for **curves** and **surfaces**. For surfaces, **continuous** is defined as the collection of attached surfaces that are bounded by curves where the exterior angle is approximately 180 degress (tolerance = 15 degrees). For curves, **continuous** is defined as the collection of attached curves that are bounded by vertices where the angle is approximately 180 degrees (tolerance = 15 degrees). For example, **"draw surface 10 include continuous"** will draw surface 10 along with other surfaces within the same continuous collection.
Nearby	Used with the **include** keyword. It will include additional geometry entities of the same dimension that are within a small distance of the specified entities. Implemented for **curves**, **surfaces** and **volumes**. The distance tolerance is automatically defined relative to the size of the entity(s). An example use case would be, **"draw volume 10 include nearby"** which will draw volume 10 along with volumes that are within a small distance of volume 10.

Precedence

For complicated expressions, which entities are referred to is influenced by the order in which portions of the expression are evaluated. This order is determined by precedence. Operators with high precedence are evaluated before operators with low precedence. You may always include parentheses to determine which sub-expressions are evaluated first. Here all operators and keywords listed from high to low precedence. Items listed together have the same precedence and are evaluated from left to right.

```
(, ) Expand Not *, / +, - <, >, <=, >=, <>, =
And, Or Except In Of With
```

Because of precedence, the following two expressions are identical:

```
curve with length + 2 * 2 > 10 and length <=
20 in my_group

expand(curve with (((length + (2*2)) > 10
)and( length <= 20 ))) in ( my_group expand )
```

Selecting Entities with the Mouse

The following discussion is applicable only to the command line version of CUBIT. See GUI Entity Selection for a description of interactive entity selection with the Graphical User Interface. Also refer to Extended Selection Dialog to learn how to use Python scripts to create extensive selection capabilities.

Many of the commands in CUBIT require the specification of an entity on which the command operates. These entities are usually specified using an object type and ID (see Entity Specification) or a name. The ID of a particular entity can be found by turning labels on in the graphics and redisplaying; however, this can be cumbersome for complicated models. CUBIT provides the capability to select with the mouse individual geometry or mesh entities. After being selected, the ID of the entity is reported and the entity is highlighted in the scene. After selecting the entities, other actions can be performed on the selection. The various options for selecting entities in CUBIT are described below, and are summarized in Table 1:

Table 1. Picking and key press operations on the picked entities

Key	Action
ctrl + B1	Pick entity of the current picking type.
shift + ctrl + B1	Add picked entity of the current picking type to current picked entity list.
tab	Query-pick; pick entity of current picking type that is below the last-picked entity.
n	Lists what entities are currently selected.
l	Lists basic information about each selected entity. This is similar to entering a List command for each selected entity.
g	Lists geometric information about the selection. As if the List Geometry command were issued for each entity. If there are multiple entities selected, a geometric summary of all selected entities is printed at the end, including information such as the total bounding box of the selection.
i	Makes the current selection invisible. This only affects entities that can be made invisible from the command line (i.e. geometric and genesis entities.)
s	Draws a graphical scale showing model size in the three coordinate axes. This is a toggle action, so pressing the 's' key again in the graphics window will turn the scale off.
ctrl + z	Zoom in on the current selection.
e	Echo the ID of the selection to the command line.
a	Add the current selection to the picked group. Only geometry will be added to the group (not mesh entities). If a selected entity is already in the picked group, it will not be added a second time.
r	Remove the current selection from the picked group. If a selected entity was not found in the picked group, this command will have no effect.
ctrl + r	Redisplays the model.

`c`	Clear the picked group. The picked group will be empty after this command.
`m`	Lists what entities are currently in the picked group.
`d`	Display and select the entities in the picked group.
`ctrl + d`	Draws the entity that is selected.

Details of selecting entities with a mouse are outlined in the following items:

- Entity Selection
- Query Selection
- Multiple Selected Entities
- Information about the Selection
- Picked Group
- Substituting the Selection into Commands
- Select Commands

Entity Selection

Selecting entities typically involves two steps:

1. Specifying the type of entity to select

Clicking on the scene can be interpreted in more than one way. For example, clicking on a curve could be intended to select the curve or a mesh edge owned by that curve. The type of entity the user intends to select is called the picking type. In order for CUBIT to correctly interpret mouse clicks, the picking type must be indicated. This can be done in one of two ways. The easiest way to change the picking type is to place the pointer in the graphics window and enter the dimension of the desired picking type and an optional modifier key. The dimension usually corresponds to the dimension of the objects being picked:

`Table 2. Picking Modes in Graphics Window`

Number	Default pick	Number +shift pick
0	vertices	nodes
1	curves	edges
2	surfaces	all 2D elements
3	volumes	all 3D elements
4	bodies	

If a Shift modifier key is held while typing the dimension, the picking type is set to the mesh entity of corresponding dimension, otherwise the geometry entity of that dimension is set as the picking type. For example, typing 2 while the pointer is in the graphics window sets the picking type so that geometric surfaces are picked; typing Shift-1 sets the picking type so that mesh edges are picked. To differentiate between picking "tris" or "quads" use "**pick face**" or "**pick tri**"

The picking type can also be set using the command

> **Pick <entity_type>**

where entity_type is one of the following: Body , Volume , Surface , Curve , Vertex , Hex , Tet , Face , Tri , Edge , Node , or DicerSheet .

2. **Selecting the entities**

To select an object, click on the entity (this command can be mapped to a different button and modifiers, as described in the section on Mouse-Based View Navigation). Clicking on an entity in this manner will first de-select any previously selected entities, and will then select the entity of the correct type closest to the point clicked. The new selection will be

highlighted and its name will be printed in the command window.

Query Selection

If the highlighted entity is not the object you intended to selected, press the Tab key to move to the next closest entity. You can continue to press tab to loop through all possible selections that are reasonably close to the point where you clicked. Shift-Tab will loop backwards through the same entities.

Multiple Selected Entities

To select an additional entity, without first clearing the current selection, hold down the control key while clicking on an object. You can select as many objects as you would like. By changing the picking type between selections, more than one type of entity may be selected at a time. When picking multiple entities, each pick action acts as a toggle; if the entity is already picked, it is "unpicked", or taken out of the picked entities list.

To select entities using rubberband, hold the control key, click and drag to enclose the entities to select. Different rubberband shapes are available to use: box, circle and polygon. A toolbar button is provided to toggle between the different shapes.

Information About the Selection

When an entity is selected, its name, entity type, and ID are printed in the command window. There are several other actions which can then be performed on the picked entity list. These actions are initiated by pressing a key while the pointer is in the graphics window. Table 1 summarizes the actions which operate on the selected entities.

Picked Group

There is a special group whose contents can be altered using picking. This group is named picked , and is automatically created by CUBIT. Other than its relationship to interactive picking, it is identical to other groups and can be operated on from the command line. Like other groups, both geometric and mesh entities can be held in the picked group. Table 1 lists the graphics window key presses used with the picked group.

Note: It is important to distinguish between the current selection and the picked group contents. Clicking on a new entity will select that entity, but will not add it to the picked group. De-selecting an entity will not remove an entity from the picked group.

Substituting Selection into Other Commands

There are three ways to use mouse-based selection to specify entities in commands.

1. The Selection Keyword

You may refer to all currently selected entities by using the word selection in a command; the picked type and ID numbers of all selected entities will be substituted directly for selection . For example, if Volume 1 and Curve 5 are currently selected, typing

```
Color selection Blue
```

is identical to typing

```
Color Volume 1 Curve 5 Blue
```

Note that the selection keyword is case sensitive, and must be entered as all lowercase letters.

2. Echoing the ID of the Selection

Typing an e into a graphics window will cause the ID of each selected entity to be added to the command line at the current insertion point. This is a convenient way to use entities of which you don't already know the name or ID.

As an added convenience, the picking type can be set based on the last word on the command line using the ` key. Note that this is not the apostrophe key, but rather the left tick mark, usually found at the upper-left corner of the keyboard on the same key as the tilde (~). For example, a convenient way to set the meshing scheme of a cylinder to sweep would be as follows:

```
Volume (hit `, select cylinder, hit e) Scheme
Sweep Source Surface (hit `, select endcap,
hit e) Target (select other endcap, hit e)
```

The result will be something similar to

```
Volume 1 Scheme Sweep Source Surface 1 Target
2
```

Notice that you must use the word Surface in the command, or ` will not select the correct picking type.

3. Using the Picked Group in Commands

Like other groups, the picked group may be used in commands by referring to it by name. The name of the picked group is picked. For example, if the contents of the picked group are Volume 1 and Volume 2, the command

```
Draw picked
```

is identical to

```
Draw Volume 1 Volume 2
```

Note that picked is case sensitive, and must be entered as all lowercase letters.

Select Commands

Creating and Modifying Selections

The following commands may be used to create a new selection or modify the current seelction.

```
Select <entity_list> [add|remove]
```

This command selects the specified entities. If the **add** option is specified, the entities are added to the current selection. If the **remove** option is specified, the entities are removed from the current selection. If neither is specified, the current selection is replaced with the specified entities.

```
Select None
```

This command clears the selection.

```
Select Seed {face <ids>|tri <ids>}
feature_angle <angle>
```

This command creates a new selection based on a seed face and feature

angle. It finds all the neighboring faces with a feature angle that is less than the specified angle and adds them to the selection.

Rubberband Selection Control

The following commands control the behavior of rubberband selection.

```
Select Occluded {on|off}
```

When turned on, the selection will include entities that are occluded, or hidden behind other entities, in the current graphics view.

```
Select Partial {on|off}
```

When turned on, the selection will include all entities that touch the rubberband. When turned off, the selection will include only entities that lie completely within the rubberband region.

```
Select Rubberband Shape {box|polygon|circle}
```

Choose the rubberband shape to be box, polygon, or circle. If **polygon** is selected, the shape of the polygon is defined by the left mouse button clicks in the graphics windows. To end defining the polygon shape and make the selection, click the right mouse button.

Part Classification with Machine Learning

Predicting Part Classification
Categorizing Parts into New Categories
Training
Listing Categories
Resetting Categories
User Training Data
Reclassification
Volume Selection by Category

In a complex assembly containing tens or hundreds of volumes and surfaces, it is sometimes useful to classify or identify entities according to a predefined category. Cubit uses machine learning methods to categorize volumes or surfaces into one of a predefined set of categories. Custom categories, defined by the user, can also be set up used for classification. The **classify** command is primarily used to identify and control part classification using the Cubit command line. In addition, the machine learning tools encapsulated in the geometry power tool in Cubit's graphical user interface provides an extensive set of tools for classifying and applying geometric solutions.

Syntax:

classify {volume|surface <ids>} [confidence] [features [importance]]

classify {volume|surface <ids>} "<string>" [export_acis]

classify train

classify list

classify reset ["<string>"]

classify user path ["<string>"]

classify aggregate [model "<string>" category "<string>"]

reclassify {volume <ids>} "<string>"

Predefined Fixed Part Classification Categories

Cubit currently provides the following set of predefined categories for classification:

Volumes

1. bolt
2. spring
3. washer
4. nut
5. insert
6. pin
7. ball
8. race
9. gear
10. thin
11. other

Surfaces

1. slot

2. non_slot

Discussion

Part and Surface classification uses machine learning methods to identify entities based on characteristic geometric features of the volume or surface. These methods utilize a fixed set of training data which has been provided with your Cubit installation. This training data may be augmented by the user to refine the existing categories or add new categories. When using any of the above **classify** commands for the first time, a short pause may occur while the fixed training data from the Cubit installation and the user training data are loaded into the program.

Predicting Part and Surface Classification

The command **classify {volume|surface <ids>}** is used to identify a volume or surface based upon the set of categories from both fixed and user training data. When executed, various geometric features of the specified entity(s) are computed and a prediction made as to the most likely categorization based upon the existing training data. Depending upon the number of entities to classify and the complexity of the surfaces or volumes, this command may take a few seconds to complete. The result will be a string printed to the output window of the predicted category, such as:

> **Volume 1 (Solid) is "spring"**

or

> **Surface 1 is "slot"**

The optional **confidence** argument may be used to list the confidence values for each of the existing categories for both fixed and user defined. The confidence is a numerical value from 0 to 1 computed by the machine learning method indicating the confidence level of the classification for each category. The highest numerical value of confidence is selected as the predicted category.

The **features** and **importance** arguments are primarily used for diagnostics to examine the resulting geometric features and their relative significance to the prediction. The features are a list of about 50 scalar values that are computed for each surface or volume that are used in the machine learning methods and the importance is the relative weight given to each feature when making the prediction.

Categorizing Entities into New Categories

If the existing set of fixed categories is insufficient, users can create new categories to augment the fixed categories. In addition, if a predicted category for a given surface or volume is incorrect or insufficient, training data for the fixed categories may also be added. To add training data to a category, use the **classify {volume|surface <ids>} "<string>"** command, where <string> is either an existing category name or a new category. Use quotation marks to distinguish the category name. For example:

> **classify volume 1 "spring"**

When executed, the features of the given surface or volume will be computed and written to disk as a new set of training data. User training data defined in this way is written to an application directory which is persistent so it can be used in subsequent runs of Cubit.

In most cases the more examples of a category that can be provided, the more accurate will be subsequent predictions. For example, if a single

volume of new category **"widget"** is added, all volumes that are identical to the initial example will most likely be predicted as **"widget"**, however small variations of the volume may not. For this reason, providing as many varying examples of a **"widget"** as possible is advantageous when setting up a new category. Providing identical examples of the same volume to the **classify** command is also not detrimental, as duplicates are automatically identified and filtered out. Note that when categorizing a new surface or volume, the output window will indicate whether new training data was added or whether the volume represents duplicate data.

The **export_acis** option can be used to automatically write an acis **.sat** file of the specified volume(s). If used, the file(s) will be written to the user training directory titled with the name of the category. Each acis file will contain a single volume and named using the category name and a unique incremented integer ID. While primarily used for debugging, they can also be used as a visual representation of the current user categories. (Not currently supported for surfaces)

Training

Whenever new training data is added, the classification models must be retrained. Although retraining happens automatically any time training data is added or deleted when ussing Cubit commands, the **train** command is useful for forcing a retrain should additional training data be added manually to the training directories on disk. In addition to rerunning the training process, it will print to the output window the number of supporting volumes used for each category when computing classification predictions.

Listing Categories

All existing classification categories may be printed to the output window using the **classify list** command. For user defined categories or fixed categories that have been augmented, a **(U)** will follow the name of the category.

In addition to the category names, the **classify list** command will also print the path to both the fixed training data and the user training data on disk.

Resetting Categories

To remove an existing user defined category use the command **classify reset ["<string>"]** where **<string>** is a a category name in quotations. This will remove all user training data for the given category from disk and retrain the classification models without the category. If a fixed category is specified, the user-defined training data defined in the category will be removed, keeping only the fixed data. If this command is used without a category, all user training data will be removed.

User Training Data

When a new category is created or a fixed category is added to, the resulting training data is written to a default application directory that is specific to a platform. For example, for Mac and Linux OS the directory will be located at:

> **/Users/<user_name>/Library/Application Support/Cubit/ml**

To display both the current user training data directory and the fixed training data directory, use the **classify list** command. While the fixed training data directory canot be changed, it may be worthwhile to change the user training directory. To change the user training directory, use the

command **classify user path "<path string>"** where **<path string>** is the full path to a writable directory on disk in quotes. Changing the user training directory may be useful to temporarily use training data from another source or to ignore all user training data without removing it.

Using the command, **classify user path** without a path specification will set the user training data directory back to its default for the platform.

Aggregate Classification Data

To add (aggregate) user-defined classification training data to Cubit's existing data, use the **Classify Aggregate** command. If a specific model and category are not specified, all user-defined data will be aggregated.

Reclassification

At times, it may be necessary to remove a surface or volume from its current category classification, or move the surface or volume from one category to another. Use the **reclassify {volume|surface <ids>} "<string>"** command to update the training data for one or more volumes where **<string>** is the name of a category. If the training data for the given volume is currently present in another category, it will be removed from its current category and added to the new specified category. If no category is specified, it will be removed from its current category without adding it to another category. Training data from the fixed training data cannot be removed or modified.

Surface or Volume Selection by Category

It may be useful to identify all surfaces or volumes based on their predicted classification category. To do so, use the syntax **with category "<string>"** in conjunction with other Cubit commands that accept IDs. A few example uses this syntax might include:

```
group "mysprings" add volume with category "spring"

draw volume with category "bolt"

delete volume with category "insert"
```

In the first example, a group named **mysprings** is created and all volumes with the predicted category of **spring** added to it. The second example would draw all volumes that are classified as **bolt**, and the third example would delete all volumes that are classified as **insert**.

Note that in order to identify surfaces or volumes based on their predicted category, features for all surfaces or volumes in the model will be computed. For assemblies with hundreds or thousands of parts, this may be time consuming. Also note that a similar capability is available in the geometry power tool machine learning tools which will list the predicted category for each volume in the model in separate drop-down lists and build cubit groups from each category if requested.

Training CAD Operations with Machine Learning

Regression Models
Predicting CAD Operation Outcomes
Assigning new CAD Operation Training Data
Training CAD Operations
Listing Regression Models
Resetting Regression Models
User Training Data

Syntax:

predict copy_surface {volume <ids> surface <ids>} [features [importance]]

predict midsurface {volume <ids> surface <ids>} [features [importance]]

learn copy_surface {volume <ids> surface <ids>} label <value> [export_acis]

learn midsurface {volume <ids> surface <ids>} label <ids> [export_acis]

learn train [<string>]

learn reset ["<string>"]

learn user path ["<string>"]

Regression Models

Some of the ML models used in Cubit are used for recommending CAD operations that can be used to resolve a particular geometry or topology issue, known as *regression* models. They differ from classification ML models described, in that they predict a specific numerical outcome for a given operation. Cubit currently supports two types of regression models based on the expected output or ground truth label: *Mesh Quality* and *Suitability*.

Mesh Quality

These models predict a mesh quality outcome, including scaled Jacobian, scaled In-radius and scaled deviation. The intent of the mesh quality models is to recommend a solution from a range of different CAD operations based on the predicted best mesh quality outcome. Each of the mesh quality regression models have a corresponding Cubit command that they are used for. The currently supported mesh quality ML models include the following:

1. vertex_no_op
2. curve_no_op
3. surface_no_op
4. remove_surface
5. tweak_replace_surface
6. composite_surfaces
7. collapse_curve
8. remove_topology_curve
9. virtual_collapse_curve
10. remove_topology_surface
11. blunt_tangency

12. remove_cone
13. collapse_angle
14. remove_blend
15. remove_cavity

Mesh quality models are mostly used for driving defeaturing of CAD models. They are currently used in Cubit's Geometry Power Tool diagnostics to predict where potential mesh quality issues will appear on the FE mesh before meshing and to provide a recommendation for CAD operations that can be used to resolve the issue based on a predicted mesh quality outcome.

Suitability

The *suitability* models will predict a heuristic value from 0 to 1 based on how well the outcome is expected to be successful within the context of a given assembly. Suitability models include the following:

1. copy_surface
2. midsurface

The *suitability* models are currently limited to making recommendations for the corresponding **reduce thin volume** commands. They are also used as the primary user knowledge base for Cubit's reinforcement learning procedure for reducing thin volumes. The following sections outline how the regression models can be managed using Cubit's command line tools. At present, only the suitability models are supported for predicting and training user training data.

Predicting CAD Operation Outcomes

A suitability score will be displayed at the command line with the following commands:

Learn copy_surface volume <id> surface <ids>

Learn midsurface volume <id> surface <ids>

For these commands, the **copy_surface** and **midsurface** ML regression models will be used to predict a suitability score. These reflect the suitability of using the respective reduce volume thin commands in the current context. The volume ids and surface ids used in the **Learn** commands should be the same as those used in the equivalent reduce thin commands.

Adding New CAD Operation Training Data

Adding additional user training data is the same as prediction (above) except that a **label** value is provided, where a value of zero represents low or infeasible condition and 1.0 is high or most preferred solution. In this case, features and labels are appended to the existing user training data so that subsequent ML predictions will use this data to determine suitability outcomes.

Learn copy_surface volume <id> surface <ids> label <value>

Learn midsurface volume <id> surface <ids> label <value>

Training CAD Operations

A training operation is automatically triggered anytime new training data is added. Training can also be invoked from a command by using the following command.

Learn Train ["<string>"]

Learn Train ["<string>"]

The **<string>** value should be either one of **copy_surface** or **midsurface**. Mesh Quality models are currently not supported. If the **<string>** is absent, then all available regression models will be trained.

Listing Regression Models

To list all available regression models use the following command:

Learn List ["<string>"]

If the **<string>** is absent, then all available regression models will be listed.

Resetting Regression Models

To remove all use training data for a given regression model, the following command can be used:

Learn Reset ["<string>"]

If the **<string>** is absent, then all available regression models will be reset.

User Training Data

Learn User Path ["<path_string>"]

When a new training datum is written, it will write to a default application directory that is specific to a platform. For example, for Mac and Linux OS the directory will be located at:

```
/Users/<user_name>/Library/Application Support/Cubit/ml
```

To display both the current user training data directory and the fixed training data directory, use the **Learn List** command. While the fixed training data directory cannot be changed, it may be worthwhile to change the user training directory. To change the user training directory, use the command **Learn user path "<path_string>"** where **<path_string>**> is the full path to a writable directory on disk in quotes. Changing the user training directory may be useful to temporarily use training data from another source or to ignore all user training data without removing it.

Using the command, **Learn User Path** without a path specification will set the user training data directory back to its default for the platform. Changing the user path using the **Learn** command will also change the path for the **Classify** command.

Select Visible Surfaces

The following commands select surfaces based on their visibility within a model. The rendering visibility of a surface, such as by turning visibility off or using a graphics clipping plane, does not affect these commands.

Shared Options:

All the following command have these optional parameters that can be specified.

> [threshold <threshold>]

This value is the confidence level used to filter selected surfaces. Surface with a confidence value below the threshold will not be selected. The confidence value is calculated based on the visible amount of a surface compared to it percentage of total surface area in a model. If only a little of a surface is seen but that surface comprises a large percentage of the total surface area of the model then the calculated confidence value will be low.

> [verbose]

This option causes the confidence value of all seen surfaces to be printed out.

> [exclude [holes] <exclude cavity max area>]

This option excluded surface that are part of a hole from the selection. Surfaces with a surface area below the <exclude cavity max area>] value will not be selected.

Commands

> select visible surfaces [threshold <threshold>] [exclude [holes] <exclude cavity max area>] [verbose]

This command selects all surfaces visible on the screen.

> select visible cavity [threshold <threshold>] [exclude [holes] <exclude cavity max area>] [verbose]

This command selects all surfaces visible around the current camera location.

> select visible cavity volume <id> [threshold <threshold>] [exclude [holes] <exclude cavity max area>] [verbose]

This command selects all surface visible from the centroid of the specified volume.

> select visible cavity surface <ids> path [threshold <threshold>] [exclude [holes] <exclude cavity max area>] [verbose]

This command selects all surfaces visible from three location along the bath between the centroid of the two specified surfaces.

> select visible cavity surface <ids> [threshold <threshold>] [exclude [holes] <exclude cavity max area>] [verbose]

The command selects all the surfaces visible from the combined centroid of the provided surfaces.

> select visible cavity location <options> [path] [threshold <threshold>] [exclude [holes] <exclude cavity max area>] [verbose]

This command behaves differently depending on the number of locations provided.

With one location specified this command behaves like the “select visible cavity” command.

With two locations specified this command behaves like the “select visible cavity surface <ids> path“.

With three or more location specified this command behaves like the “select visible cavity surface <ids>” command.

```
select visible exterior [threshold <threshold>] [exclude [holes] <exclude cavity max area>] [verbose]
```

This command selects all the surfaces visible on the exterior of the model.

Geometry

- CUBIT Geometry Formats
- Geometry Creation
- Geometry Transforms
- Geometry Booleans
- Geometry Decomposition
- Geometry Cleanup and Defeaturing
- Geometry Imprinting and Merging
- Virtual Geometry
- Geometry Orientation
- Geometry Groups
- Geometry Attributes
- Entity Measurement
- Parts, Assemblies, and Metadata
- Geometry Deletion

CUBIT usually relies on the ACIS solid modeling kernel for geometry representation; there is also mesh-based geometry. Other solid model kernels are planned. Geometry is imported or created within CUBIT. Geometry is created bottom-up or through primitives. CUBIT imports ACIS SAT files. CUBIT can also read STEP, IGES, and FASTQ files and convert them to the ACIS kernel. SolidWorks, AutoCAD, and some other commercial CAD systems can write SAT files directly.

Once in CUBIT, an ACIS model is modified through booleans. Without changing the geometric definition of the model, the topology of the model may be changed using virtual geometry. For example, virtual geometry can be used to composite two surfaces together, erasing the curve dividing them.

Sometimes, an ACIS model is poorly defined. This often happens with translated models. The model can be healed inside CUBIT.

CUBIT Geometry Formats

- ACIS
- Mesh-Based Geometry

Setting the Geometry Kernel

The geometry kernel can be switched between ACIS and Mesh-Based Geometry from the command line using the following command:

```
Set Geometry Engine {Acis|Facet}
```

The geometry engine will automatically be set when importing a model.

Terms

Before describing the functionality in CUBIT for viewing and modifying solid geometry, it is useful to give a precise definition of terms used to describe geometry in CUBIT. In this manual, the terms topology and geometry are both used to describe parts of the geometric model. The definitions of these terms are:

Topology: the manner in which geometric entities are connected within a solid model; topological entities in CUBIT include vertices, curves, surfaces, volumes and bodies.

Geometry: the definition of where a topological entity lies in space. For example, a curve may be represented by a straight line, a quadratic curve, or a b-spline. Thus, an element of topology (vertex, curve, etc.) can have one of several different geometric representations.

Topology

Within CUBIT, the topological entities consist of vertices, curves, surfaces, volumes, and bodies. Each topological entity has a corresponding dimension, representing the number of free parameters required to define that piece of topology. Each topological entity is bounded by one or more topological entities of lower dimension. For example, a surface is bounded by one or more curves, each of which is bounded by one or two vertices.

Bodies and Volumes

A CUBIT Body is defined as a collection of other pieces of topology, including curves, surfaces and volumes. The use of Body is not required, and is in fact deprecated in favor of using Volume. Bodies may still be used for grouping volumes, but it is suggested to use Groups instead.

Although a Body may contain groups of Surfaces or Volumes, for most practical purposes within the CUBIT environment, a single Volume or Surface will belong to a single Body. For typical three-dimensional models, this means that there should be one Body for every Volume in the model, where the default Body ID is the same as the Volume ID. For this reason, in many instances the term Volume and Body are used interchangeably, although it is more consistent to always refer to Volumes and Volume IDs, and only use Bodies when absolutely necessary.

Non-Manifold Topology

In many applications, the geometry consists of an assembly of individual parts, which together represent a functioning component. These parts

parts, which together represent a functioning component. These parts often have mating surfaces, and for typical analyses these surfaces should be joined into a single surface. This results in a mesh on that surface which is shared by the volume meshes on either side of the shared surface. This configuration of geometry is loosely referred to as **non-manifold topology**.

Bounding Box Calculations

Bounding box calculations are used for many routines and subroutines in Cubit. These calculations are done using a faceted representation by default. To use the default modeling engine for more accurate (and longer) calculations change the **Facet Bbox** setting.

```
Set Facet BBox [ON|Off]
```

There are also various settings to control the accuracy of bounding box calculations based on point lists.

```
Set Tight [[Bounding] [Box]
[{Surface|Curve|Vertex} {on|off}]]
```

If surfaces are used, surface facet points will be included in the point list used to calculate the tight bounding box. This will include vertices and points on the curves. This is the default implementation.

If curves are used, curve tesselation points will be included in the point list used to calculate the tight bounding box. This includes the vertices on the ends of the curves. One use for this is to find a more accurate tight bounding box, since curve tessellations are typically more fine than surface tessellations. However, in practice, it is recommended to just use surface tessellations. One special case is if the user sends in a list of curves as the criteria for the tight bounding box, the curve tessellations are always used, even if this parameter is false.

If vertices are used, vertex points will be included in the point list used to calculate the tight bounding box. In extremely large models, it could be advantageous to just use vertices. So the user would turn off both the surface and curve flags. One special case is if the user sends in a list of curves as the criteria for the tight bounding box, the curve tessellations are always used, even if the curve parameter is false and this parameter is true.

ACIS Geometry Kernel

ACIS is a proprietary format developed by Spatial Technologies. CUBIT incorporates the ACIS third party libraries directly within the program. The ACIS third party libraries are used extensively within CUBIT to import, export and maintain the underlying geometric representations of the solid model for geometry decomposition and meshing. There are many ways to get geometry into the ACIS format. ACIS files can be exported directly from several commercial CAD packages, including SolidWorks, AutoCAD, and HP PE/SolidDesigner. Third party ACIS translators are also available for converting from native formats such as Pro Engineer. CUBIT also uses the ACIS libraries for importing IGES and STEP format files.

Importing and creating geometry using the ACIS geometric modeling kernel currently provides the widest set of capabilities within CUBIT. All geometry creation and modification tools have been designed to work directly on the ACIS representation of the model.

Mesh-Based Geometry

In contrast to the ACIS format, Mesh-Based Geometry (MBG) is not a third party library and has been developed specifically for use with CUBIT. Most of CUBIT's mesh generation tools require an underlying geometric representation. In many cases, only the finite element model is available. If this is the case, CUBIT provides the capability to import the finite element mesh and build a complete boundary representation solid model from the mesh. The solid model can then be used to make further enhancement to the mesh. While the underlying ACIS geometry representation is typically non-uniform rational b-splines (NURBS), Mesh-Based Geometry uses a *facetted* representation. Mesh-Based Geometry can be generated by importing either an Exodus II format file or a facet file.

- Creating Mesh-Based Geometry Models
- Improving Mesh-Based Geometry Models for Meshing
- Meshing Mesh-Based Models
- Exporting Mesh-Based Geometry

Many of the same operations that can be done with traditional CAD geometry can also be done with mesh-based geometry. While all mesh generation operations are available, only some of the geometry operations can be used. For example, the following can be done with geometric entities that are mesh-based:

- Geometry Transformations
- Merging
- Virtual Geometry Operations

Some operations that are not yet available with mesh-based geometry include:

- Booleans
- Geometry Decomposition
- Geometry Clean-Up

Creating Mesh-Based Geometry Models

Mesh based geometry models can be created in one of two ways

- Importing Exodus II files
- Importing facet files

While both of these methods create geometry suitable for meshing, there are some significant differences:

Exodus II files

Exodus II contains a mesh representation that may include 3D elements, 2D elements, 1D elements and even 0D elements. It may also contain deformation information as well as boundary condition information. The import mesh geometry command is designed to decipher this information and create a complete solid model, using the mesh faces as the basis for the surface representations. Exodus II is most often used when a solid model that has previously been meshed requires modification or remeshing. Importing an Exodus II file will generate both geometry and mesh entities, assigning appropriate ownership of the mesh entities to their geometry owners. Deleting the mesh and remeshing, refining or smoothing are common operations performed with an Exodus II model.

Facet files

The facet file formats supported by CUBIT are most often generated

from processes such as medical imaging, geotechnical data, graphics facets, or any process that might generate discrete data. Importing a facet file will generate a surface representation only defined by triangles. If the triangles in the facet file form a complete closed volume, then a volume suitable for meshing may be generated. In cases where the volume may not completely close or may not be of sufficient quality, a limited set of tools has been provided. In addition to the standard meshing tools provided in CUBIT, it is also possible to use the triangle facets themselves as the basis for an FEA mesh.

Improving Mesh-Based Geometry Models for Meshing

In many cases, the triangulated representations that are provided from typical imaging processes are not of sufficient quality to use as geometry representations for mesh generation. As a result, CUBIT provides a limited number of tools to assist in cleaning up or repairing triangulated representations.

1. Using tolerance on STL files

Stereolithography (STL) files, in particular, can be problematic. The import mechanism for STL provides a **tolerance** option to merge near-coincident vertices.

2. Using the stitch option on AVS and facet files

The stitch option on the import facets|avs command provides a way to join triangles that otherwise share near-coincident vertices and edges. This is useful for combining facet-based surfaces to generate a water-tight model.

3. Using the improve option on facet files.

The improve option on the **import facets** command will collapse short edges on the boundary of the triangulation. This option improves the quality of the boundary triangles.

4. Smoothing faceted surfaces.

Individual triangles in a faceted surface representation may be poorly shaped. Just like mesh elements may be smoothed, facets may also be smoothed in CUBIT using the following command

```
Smooth <surface_list> Facets [Iterations
<value>] [Free] [Swap]
```

To use this command, the surface cannot be meshed. Facet smoothing consists of a simple Laplacian smoothing algorithm which has additional logic to make sure it does not turn any of the triangles in-side out. It also determines a local surface tangent plane and projects the triangle vertices to this plane to ensure the volume will not "shrink". The **iterations** option can be used to specify the number of Laplacian smoothing operations to perform on each facet vertex (The default is 1).

The **free** option can be used to ignore the tangent plane projection. Used too much, the **free** option can collapse the model to a point. One of two iterations of this option may be enough to clean up the triangles enough to be used for a finite element mesh.

The **swap** option can be used to perform local edge swap operations on the triangulation. The quality of each triangle is assessed and edges are swapped if the minimum quality of the triangles will improve.

5. Creating a thin offset volume

Offset surfaces may be generated from an existing facet-based surface. This would be used in cases where a thin membrane-like volume might

be required where only a single surface of triangles is provided. This command may be accomplished by using the standard **create body offset** command

The result of this command is a single body with an inside and outside surface separated by a small distance which is generally suitable for tet meshing. This command is currently only useful for small offsets where self-intersections of the resulting surface would be minimal. It is most useful for bodies that may be initially composed of a single water-tight surface.

6. Creating volumes from surfaces

A mesh-based geometry volume can be created from a set of closed surfaces. This can be accomplished in the same manner as the standard create body surface command

```
Create Body Surface <surface_id_range>
```

This command is limited to surfaces that match triangles edges and vertices at their boundary. The command will internally merge the triangles to create a water-tight model that would generally be suitable for tet meshing.

Meshing Mesh-Based Models

Mesh-Based models may be meshed just like any other geometry in CUBIT by first setting a scheme, defining a size and using the mesh command. This standard method of mesh generation can be somewhat time consuming and error prone for complex facet models with thousands of triangles. CUBIT also provides the option of using the facets themselves as a surface triangle mesh, or as the input to a tetrahedral mesher. This may be accomplished with one of two options:

```
Mesh <entity_list> From Facets
```

This command will generate triangular finite elements for each facet on the surface. If the **entity_list** is composed of one or more volumes, then the tetrahedral mesh will automatically fill the interior. This method is useful when further cleanup and smoothing operations are needed on the triangles after import.

```
Import Facets <filename> Make_elements
```

The make_elements on the import facets command will generate the triangular finite elements on the surface at the time the facets are read and created. This option is useful if no further modifications to the facets are necessary.

Creating triangular finite elements in this manner can greatly speed up the mesh generation process, however it is limited to non-manifold topology. If the triangular elements are to be used for tetrahedral meshing (i.e. all edges of the triangulation should be connected to no more than two triangles)

Exporting Mesh-Based Geometry

Mesh-Based geometry models and their mesh may be exported by one of the following methods:

- Exporting to an Exodus II File
- Exporting to a facet file

Exodus II

Exporting to an Exodus II file saves the finite element mesh along with any boundary conditions placed on the model. It will not save the individual facets that comprise the mesh-based geometry surface

representation. Importing an Exodus II file saved in this manner will regenerate the surfaces only to the resolution of the saved mesh.

Facet files

CUBIT also provides the option to save just the surface representation to a facet or STL file. The following commands can be used for saving facet or STL files:

```
Export Facets 'filename' <entity_list>
[Overwrite]

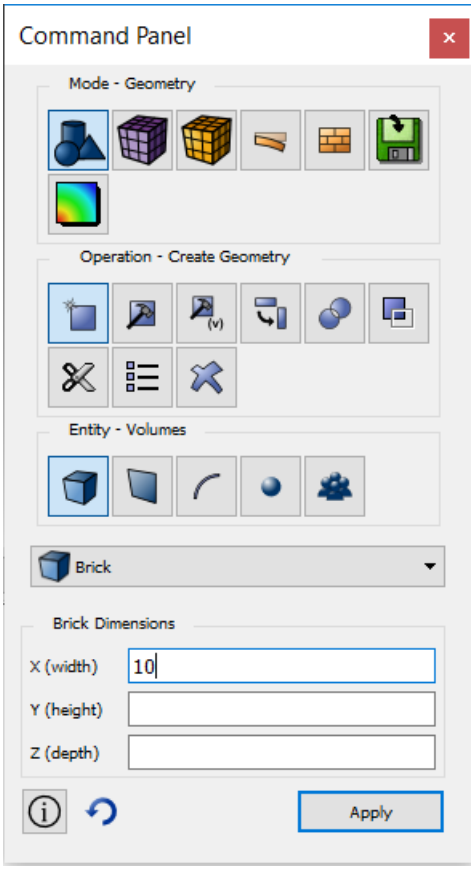
Export STL [ASCII|Binary] 'filename'
<entity_list> [Overwrite]
```

These commands provide the option of saving specific surfaces or volumes to the facet file. If no entities are provided in the command, then all surfaces in the model will be exported to the file. The **overwrite** option forces a file to overwrite any file of the same name in the current working directory.

Geometry Creation

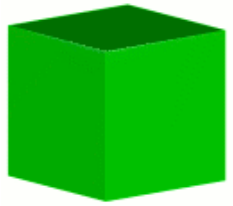
There are three primary ways of creating geometry for meshing in CUBIT. First, CUBIT provides many geometry primitives for creating common shapes (spheres, bricks, etc.) which can then be modified and combined to build complex models. Secondly, geometry can be imported into CUBIT. Finally, geometry can be defined by building it from the "bottom up", creating vertices, then curves from those vertices, etc. Two of these three methods for creating geometry in CUBIT will be described in detail in this section.

All of these geometry creation commands have been expressed in the GUI's command panels. To navigate to the volume creation command panels, for example, select "Mode-Geometry", then "Entity-Volume", then "Action-Create", as shown below. Other geometry creation command panels are available for each geometry type.



- Bottom-Up Geometry Creation
- Geometric Primitives

Creating Bricks



The brick is a rectangular parallelepiped.

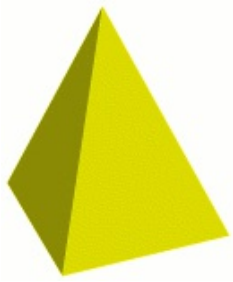
Command

```
[Create] Brick {Width|X} <width> [{Depth|Y}
<depth>] [{Height|Z} <height>] [Bounding Box
{entity_type} <id_range>] [Tight] [[Extended]
{Percentage| Absolute} <val>]]
```

Notes

- A cubical brick is created by specifying only the width or x dimension.
- A brick can be specified to occupy the bounding box of one or more entities, specified on the command line.
- If the **Tight** option is specified with **Bounding Box**, the result is the smallest brick that can contain the entities specified, which is the default behavior of the Bounding Box option.
- If the **Extended** option is specified with **Bounding Box**, the result is a brick that is extended from a "tight" brick by the input percentage or absolute value.
- If a bounding box specification is used in conjunction with any of the other parameters (X, Y or Z), the parameters specified override the bounding box results for that or those dimensions.

Creating Pyramids



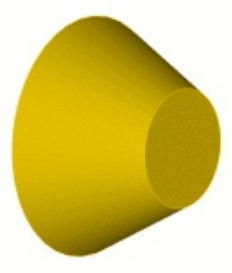
A pyramid is a general n-sided prism.

Command

```
[Create] Pyramid [Height|Z] <z-height> Sides
<nsides> Radius <radius> [Top <top-x-radius>]

[Create] Pyramid [Height|Z] <z-height> Sides
<nsides> [Major [Radius] <x-radius> Minor
[Radius] <y-radius> ] [Top <top-x-radius>]
```

Creating Frustums



A frustum is a general elliptical right frustum, which can also be thought of as a portion of a right elliptical cone.

Command

```
[Create] Frustum [Height|Z] <z-height> Radius
<x-radius> [Top <top_radius>]

[Create] Frustum [Height|Z] <z-height> Major
Radius <radius> Minor Radius <radius> [Top
<top_radius>]

[Create] Frustum Volume <id> [Axis <options>]
```

Notes

- If used, Major Radius defines the x-radius and Minor Radius the y-radius.
- If used, Top Radius defines the x-radius at the top of the frustum; the top y radius is calculated based on the ratio of the major and minor radii.
- A frustum can also be created to bound a volume. An optional axis for the frustum can be specified. If not specified, the axis will be determined from a tight bounding box.

Creating Toruses



The torus command generates a simple torus

Command

```
[Create] Torus Major [Radius] <major-radius>
Minor [Radius] <minor-radius>

[Create] Torus Volume [ID]
```

Notes

- **Minor Radius** is the radius of the cross-section of the torus; **Major Radius** is the radius of the spine of the torus.
- The **minor radius** must be less than the **major radius**.
- Giving a volume as input will create a torus that matches a toroidal surface in the input volume. The input volume must have a toroidal surface for this command to work.

Creating Cylinders



The cylinder is a constant radius tube with right circular ends.

Command

```
[Create] Cylinder [Height|Z] <val> Radius
<val>

[Create] Cylinder [Height|Z] <val> Major
Radius <val> Minor Radius <val>

[Create] Cylinder Volume <id> [Axis <options>]
```

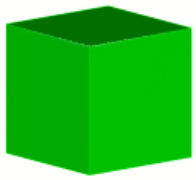
Notes

- A cylinder may also be created using the frustum command with all radii set to the same value.
- Specifying major and minor radii can produce a cylinder with an oval cross section.
- A cylinder that tighty bounds a volume can also be created. An optional axis for the cylinder can be specified. If not specified, the axis will be determined from a tight bounding box.

Geometric Primitives

The geometric primitives supported within CUBIT are pre-defined templates of three-dimensional geometric shapes. Users can create specific instances of these shapes by providing values to the parameters associated with the chosen primitive. Primitives available in CUBIT include the brick, cylinder, torus, prism, frustum, pyramid, and sphere. Each primitive, along with the command used to generate it and the parameters associated with it, are described next. For some primitives, several options can be used to generate them, and are described as well.

The following Primitives can be generated with CUBIT:



Brick



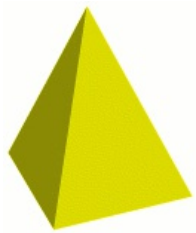
Cylinder



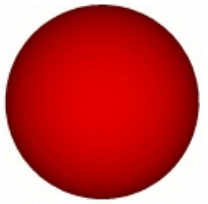
Prism



Frustum



Pyramid



Sphere



Torus

General Notes

- Primitives are created and given an ID equal to one plus the current highest body ID in the model.
- Primitive solids are created with their centroid at the origin or the world coordinate system.
- For primitives with a Height or Z parameter, the axis going through these primitives will be aligned with the Z axis.
- For primitives with a Major Radius and a Minor Radius, the Major Radius will be along the X axis, the Minor Radius along the Y axis.
- For primitives with a Top Radius, this radius will be that along the X axis; the Y axis radius will be computed using the Major, Minor and Top Radii given.

Creating Prisms



The prism is an n-sided, constant radius tube with n-sided planar faces on the ends of the tube.

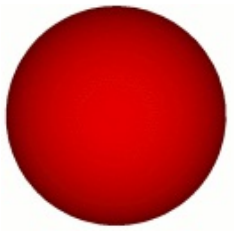
Command

```
[Create] Prism [Height|Z] <z-val> Sides
<nsides> Radius <radius>
```

Notes

- The radius defines the circumradius of the n-sided polygon on the end caps.
- If a major and minor radius are used, the end caps are bounded by a circum-ellipse instead of a circumcircle.
- The number of sides of a prism must be greater than or equal to three. A prism may also be created using the pyramid command with all radii set to the same value.
- If the **Extended** option is specified with **Bounding Box**, the result is a brick that is extended from a "tight" brick by the input percentage or absolute value.
- If a bounding box specification is used in conjunction with any of the other parameters (X, Y or Z), the parameters specified override the bounding box results for that or those dimensions.

Creating Spheres



The sphere command generates a simple sphere, or, optionally, a portion of a sphere or an annular sphere.

Command

```
[Create] Sphere Radius <radius> [Xpositive]|
[Xnegative] [Ypositive]|[Ynegative]
[Zpositive]|[Znegative] [Delete] [Inner
[Radius] <radius>]
```

Notes

- If Xpositive/Xnegative, Ypositive/Ynegative, and/or Zpositive/Znegative are used, a sphere which occupies that side of the coordinate plane only is generated, or, if the delete keyword is used, the sphere will occupy the other side of the coordinate plane(s) specified. These options are used to generate hemisphere, quarter sphere or a sphere octant (eighth sphere).
- If the inner radius is specified, a hollow sphere will be created with a void whose radius is the specified inner radius.

Bottom-Up Geometry Creation

CUBIT supports the ability to create geometry from a collection of lower order entities. This is accomplished by first creating vertices, connecting vertices with curves and connecting curves into surfaces. Currently only ACIS bodies or volumes may not be constructed by stitching a set of surfaces together, and only in a certain number of cases; however surfaces may also be swept or rotated to create bodies or volumes. Existing geometry may be combined with new geometry to create higher order entities. For example, a new surface can be created using a combination of new curves and curves already extant in the model. Commands and details for creating each type of geometry entity are given below.

The following describes each of the basic entities that can be generated with CUBIT using the bottom-up approach

- Creating Vertices
- Creating Curves
- Creating Surfaces
- Creating Bodies

Creating Volumes

Currently, CUBIT can create volumes:

1. from surfaces by sweeping a single surface into a 3D solid,
2. by offsetting an existing volume,
3. by extending one or more surfaces or sheet bodies
4. by sweeping a curve around an axis,
5. by stitching together surfaces that can form a closed volume,
6. by lofting from one surface to another surface, or
7. by thickening a surface body.

Sweeping of planar surfaces, belonging either to two- or three-dimensional bodies, is allowed, and some non-planar faces can be swept successfully, although not all are supported at this time. The following methods for generating volumes are described:

- Sweep Surface Along Vector
- Sweep Surface About Axis
- Sweep Surface Along Curve
- Sweep Surface Perpendicular
- Sweep Surface to a Volume
- Offset
- Sheet extended from surface
- Sweep Curve About Axis
- Stitch Surfaces Together
- Loft Surfaces Together
- Thicken Surfaces
- Sweep Surface
- Sweep Surface along Direction
- Sweep Surface along Helix

There are five forms of the sweep command; the syntax and details for each are given below. Common options for first four forms are:

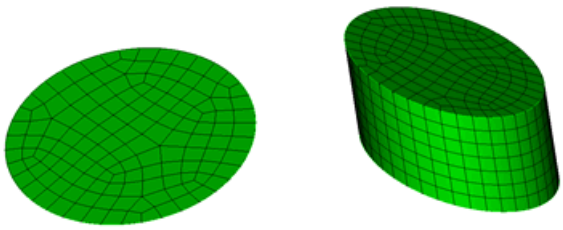
> **draft_angle**: This parameter specifies the angle at which the lateral faces of the swept solid will be inclined to the sweep direction. It can also be described as the angle at which the profile expands or contracts as it is swept. The default value is 0.0.
>
> **draft_type**: This parameter is an ACIS-related parameter and specifies what should be done to the corners of the swept solid when a non-zero draft angle is specified. A value of 0 is the default value and implies an extended treatment of the corners. A value of 1 is also valid and implies a rounded (blended) treatment of the corners.
>
> **anchor_entity**: The default behavior for the sweep command is to move the source surface along a path to create a new 3D solid. The anchor_entity option instructs the sweep to leave the source surface in its original location.
>
> **include_mesh**: This option will sweep the source surface and existing mesh into a meshed 3D solid. The mesh size is automatically computed using the Default auto interval specification.

The sweep operations have been designed to produce valid solids of positive volume, even though the underlying solid modeling kernel library that actually executes the operation, ACIS, allows the generation of solids of negative volume (i.e., voids) using a sweep.

1. Sweep Surface Along Vector: Sweeps a surface a specified distance along a specified vector. Specifying the distance of the sweep is

optional; if this parameter is not provided, the face is swept a distance equal to the length of the specified vector. The include_mesh option will create a volumetric mesh if the surface is already meshed as shown below. The keep option will keep the original surface while creating the volume.

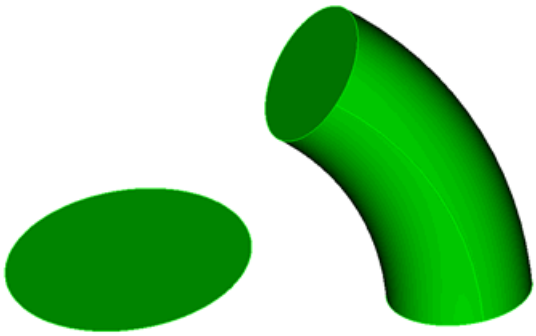
```
Sweep Surface {<surface_id_range>} Vector
<x_vector y_vector z_vector> [Distance
<distance_value>] [switchside]  [Draft_angle
<degrees>] [Draft_type <0|1>][rigid]
[anchor_entity][include_mesh] [keep] [merge]
```



Surface mesh swept along a vector

2. Sweep Surface About Axis: Sweeps a surface about a specified vector or axis through a specified angle. The axis of revolution is specified using either a starting point and a vector, or by a coordinate axis. This axis must lie in the plane of the surfaces being swept. The steps parameter defaults to a value of 0 which creates a circular sweep path. If a positive, non-zero value (say, n) is specified, then the sweep path consists of a series of n linear segments, each subtending an angle of [(sweep_angle) / (steps-1)] at the axis of revolution. The include_mesh option will create a volumetric mesh if the surface is already meshed as shown below. The keep option will keep the original surface while creating the volume.

```
Sweep Surface {<surface_id_range>} Axis
{<xpoint ypoint zpoint xvector yvector
zvector>|Xaxis|Yaxis|Zaxis} Angle <degrees>
[switchside]  [Steps <number_of_sweep_steps>]
[Draft_angle <degrees>] [Draft_type <0|1>]
[rigid][anchor_entity][include_mesh] [keep]
[merge]
```



Surface swept around an axis of 50 degree angle



Specifying multiple surfaces that belong to the same body will not work as expected, as ACIS performs the sweep operation in place. Hence, if a range of surfaces is provided, they ought to each belong to different bodies.

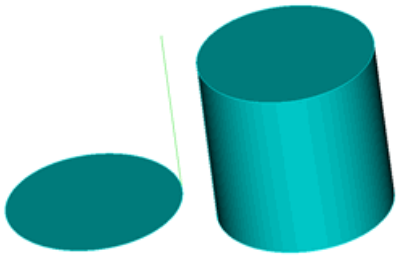
3. Sweep Surface Along Curve: This command allows the user to

sweep a planar surface along a curve:

```
Sweep Surface <surface_id_range> Along Curve
<curve_id> [Draft_angle <degrees>] [Draft_type
<0 | 1 | 2>][rigid][anchor_entity]
[include_mesh] [keep] [individual] [merge]
```

One of the ends of the curve must fall in the plane of the surface and the curve cannot be tangential to the surface. The relationship between the surface orientation and the guide curve is maintained through out the sweep. If the "rigid" option is specified the orientation of the surface is kept static throughout the sweep. Sweep along curve also supports an additional draft type "2" which implies a "natural" extension of the corners from their curves.

The include_mesh option will create a volumetric mesh if the surface is already meshed as shown below. The keep option will keep the original surface while creating the volume. If multiple curves to sweep along are specified, the individual option creates an individual or separate volume along each curve.

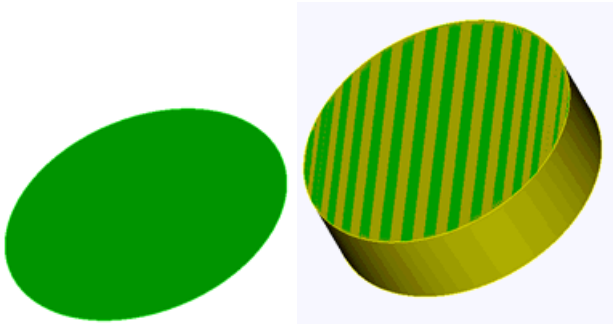


Volume generated by sweeping a surface along a reference curve

4. Sweep Surface Perpendicular: This command allows the user to sweep a planar surface perpendicular to the surface:

```
Sweep Surface <surface_id_range> Perpendicular
Distance <distance> [Switchside] [Draft_angle
<degrees>] [Draft_type <integer>]
[anchor_entity][include_mesh] [keep] [merge]
```

The sweeping plane must be planar in order to determine the sweep direction. The switchside option will reverse the direction of the sweep.



The original surface is retained with the 'keep' option. A new volume is created by sweeping the surface along the surface normal.

The include_mesh option will create a volumetric mesh if the surface is already meshed as shown below. The keep option will keep the original surface while creating the volume.

5. Sweep Surface to a Volume: This command allows users to sweep a surface to a volume.

```
Sweep Surface <surface_id_range> Target
{Volume|Body} <id> [Direction {options}]
[Plane {options}]
```

The direction keyword can be used to control the direction of sweep. Without it, Cubit will determine the sweep direction (usually normal to the sweeping surface). The plane option can be used to define a stopping plane.

6. Offset: The following command creates a body offset from another body or set of surfaces at the specified distance. The new surfaces are extended or trimmed appropriately. A positive distance results in a larger body; a negative distance in a smaller body.

```
Create Body Offset [From] Body <id_range>
Distance <value>

Create Sheet Offset From Surface <id_list>
Offset <val> [Surface <id_list> Offset <val>]
[Surface <id_list> Offset <val> ...] [Preview]
```

Using the second form of the command, the sheet body can be created from a list of surfaces, and the surfaces may offset by different distances. This command currently requires the original surfaces to be on solid bodies.

This option is also available for limited cases for facet-based surfaces.

7. Sheet Extended from Surface: The following command creates a body offset from another body or set of surfaces at the specified distance. The new surfaces are extended or trimmed appropriately. A positive distance results in a larger body; a negative distance in a smaller body.

```
Create Sheet Extended From Surface <id_list>
[Intersecting <entity_list>] [Extended
{Percentage|Absolute} <val>] [Preview]
```

This command allows multiple surfaces to be extended at the same time. Optionally, you can give a list of bodies to intersect for this calculation. You can also extend the size of the surface by either a percentage distance or an absolute distance of the minimum area size. The plane can be previewed with the preview option. Figure 1 shows a set of surfaces being created using the extended absolute option.

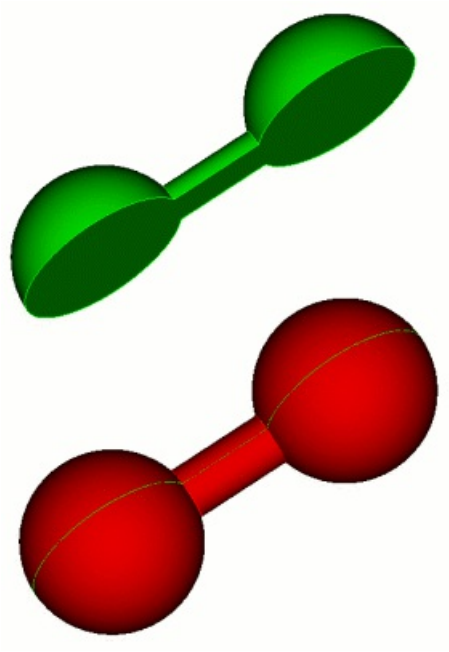


Figure 1. Sheet created from extending multiple surfaces

8. Sweep Curve About Axis: Sweeps a curve or set of curves about a given axis through a specified angle. The axis is specified the same as in the Sweep Surface About Axis command. The steps, draft_angle, and draft_type options are the same as are described above. To create the solid, the make_solid option must be specified, otherwise a surface will be created, rather than a solid. If the rigid option is specified, then the curve or set of curves will remain oriented as originally oriented, rather than rotating about the axis.

```
Sweep Curve <curve_id_range> {Axis <xpoint
ypoint zpoint xvector yvector
zvector>|Xaxis|Yaxis|Zaxis} Angle <degrees>
[Steps <Number_of_sweep_steps>] [Draft_angle
<degrees>] [Draft_type <integer>] [Make_solid]
[Rigid]
```

9. Stitch Surfaces Together: A body can be created from various surfaces that form a closed volume with command below. The geometry must be ACIS-type geometry (i.e. imported from IGES, STEP or fastq files) This option is also available for limited cases for facet-based surfaces.

```
Create {Body|Volume} Surface
<surface_id_range> [HEAL|Noheal] [Keep]
[Sheet]
```

The **heal** option will attempt to close small gaps in the surface; the **noheal** option disables this behavior. The **keep** option preserves the original surfaces.

All of the surfaces must form a closed water-tight volume for this command to succeed unless the **sheet** option is specified.

The **sheet** option allows for the creation of an open body. If the set of surfaces form a closed volume a sheet body is created instead of a volume.

In situations where the boundaries are not exactly within tolerance, the following command may be more effective:

```
Stitch {Body|Volume} <id_range>
```

```
[tolerance <value>] [no_tighten_gaps]
```

10. Loft Surfaces Together: A body can be "lofted" between two surfaces to form a new body. Surfaces from solid bodies and sheet bodies may be used to create a loft body. In order to create the loft body, two surfaces coincident to the input surfaces are created. The loft body is extruded along the shortest path between the corresponding vertices that define the shapes of the two surfaces. If guide curves are used the loft body is extruded along the guide curves. This new body is solid. The surfaces used to create the loft body are unchanged.

```
Create {Body|Volume} Loft Surface <ids> [guide
curve <id_list> [global_guides]]
[Takeoff_factors <one value per surface in
order>=.001] [Takeoff_vector Surface <id>
{direction options}] [match vertex <ids>]
[closed] [preview] [show_matching_curves]
```

Note: Source surface ids must be specified in lofting order.

Go to Location, Direction, and Axis Specification to see the direction command description.

The following options are available for lofting:

- **Guide curve:** Multiple curves may be specified to guide the loft. The curves must touch the curves of each source surface. If the global_guides option is specified the guides curves are applied in a global nature.
- **Takeoff_factors:** Takeoff factors control how strongly the loft follows the takeoff vectors. When specifying takeoff factors one value must be specified for each source surface.
- **Takeoff_vector:** The takeoff vector controls the tangency or direction of the loft for each surface. The default takeoff vector for each surface is the normal at the surface centroid. If manually specified, one takeoff vector must be specified for each surface.
- **Match vertex:** This option guides the loft in how to match the vertices of the source surfaces. Multiple match vertex sets may be specified. When specifying match vertices, one vertex id from each source surface must be specified. The match vertices must be specified in loft order.
- **Closed:** This option atempts to create a toroidal solid. The last source surface is lofted to the first source surface.
- **Preview:** This option will preview the linking curves of the final solid.
- **Show_matching_curves:** This option will preview how the vertices of the source surfaces will be matched.

Lofting can be used to split a body in order to create a more structured mesh. Figure 2 below shows a single volume swept from a large paved surface. Figure 3 shows this same volume after surfaces defined on the source and target surfaces have been used to create a loft body. This original body was chopped with the loft body. The resulting two bodies were merged. The yellow volume was swept as the volume in Figure 2 was but the purple volume was submapped, producing a much more structured mesh overall.

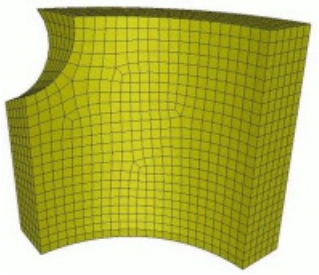


Figure 2. Mesh before loft. Single swept volume with a large paved face.

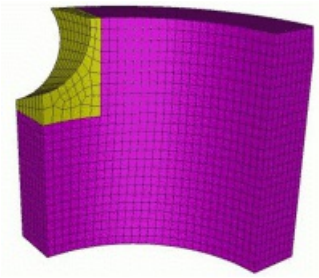


Figure 3. Mesh after loft. The yellow volume is paved and the purple volume is submapped.

11. Thicken Surfaces: A surface body can be thickened to create a volume body. The surface can be thickened in both directions using the "both" keyword, thickened in the direction of surface normal using a positive depth, or thickened in the opposite direction using a negative depth. To thicken multiple surfaces, all surface normals must be consistent.

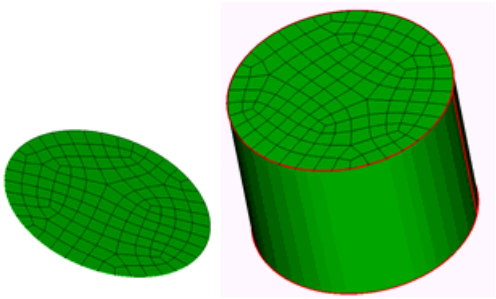
```
Thicken [Volume|BODY] <id> Depth <depth>
[Both]
```

12. Sweeping a Surface to a Plane: Sweeps a surface normal to a plane and towards the plane until the swept surface reaches the plane. See plane options for ways to describe a plane.

Sweep surface <id> target plane <options>

13. Sweep Surface along a Direction: Sweep a surface along a direction to create a volume. See direction options for ways to specify a direction.

```
Sweep Surface <surface_id_range> Direction
(options)      [switchside] [draft_angle
<degrees>] [draft_type <integer>] [rigid]
 [anchor_entity] [include_mesh] [keep] [merge]
```



Surface extruded along -X direction without 'include_mesh' option

14. Sweep Surface along Helix: Sweep a surface along a helix, where the helix is defined by an axis, **thread_distance** (distance between turns

in axis direction), axis, and handedness (**right_handed** or **left_handed**.

```
Sweep {Surface|Curve} <id_range> Helix    {axis
<xpoint ypoint zpoint xvector yvector zvector>
 | xaxis | yaxis | zaxis} thread_distance <val>
angle <val>    [RIGHT_HANDED|left_handed]
[anchor_entity] [include_mesh] [keep] [merge]
```

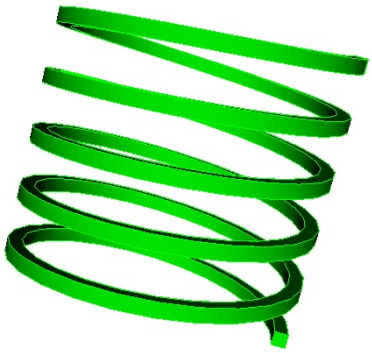
***** Specifying multiple Surfaces that belong to the same Body can cause the creation of invalid Bodies and is discouraged. *****

axis = axis about which to create the sweep

thread_distance = distance between each 360 degree segment of the helix

angle = number of degrees in rotation of the helix

handedness = right-handed or left- handed threads



Helical Sweep

Creating Curves

Curves are created by specifying the bounding lower-order topology (i.e. the vertices) and the geometry (shape) of the curve (along with any parameters necessary for that geometry). There are several forms of this command:

- Straight
- Parabolic, Circular, Ellipse
- Spline
- Copy
- Combine Existing Curves
- Arc Three
- Arc End Vertices and Radius
- Arc Center Vertex
- Arc Center Angle
- From Vertex Onto Curve
- Offset
- From Mesh Edges
- Close_To
- Surface Intersection
- By Projection
- Helix
- Tangents

1. Straight: The first form of the command creates a straight line or a line lying on the specified surface. If a surface is used, the curve will lie on that surface but will not be associated with the surface's topology.

```
Create Curve [Vertex] <vertex_id> [Vertex]
<vertex_id> [On Surface <surface_id>]
```

Straight curves can be created using an axis. The syntax is as follows:

```
Create Curve Axis {options}
```

The length of the axis must be specified. Go to Location, Direction, and Axis Specification to see the axis command description.

Additionally, several connected straight curves can be created with a single command. The syntax for the polyline command is as follows:

```
Create Curve Polyline Location {options}
Location {options} ...
```

Notice that two or more locations are used to define a polyline. See Location, Direction, and Axis Specification for the location command description.

2. Parabolic, Circular, Ellipse: The parabolic option creates a parabolic arc which goes through the three vertices. The circular and ellipse options create circular and elliptical curves respectively that go through the first and last vertices.

```
Create Curve [Vertex <vertex_id> [Vertex]
<vertex_id> [[Vertex] <vertex_id>
[Parabolic|Circular|ELLIPSE [start angle
<val=0>] [stop angle <val=90>]]]
```

If **'ellipse'** is specified, Cubit will create an ellipse assuming the vectors between vertices (1 and 3) and (2 and 3) are orthogonal. v1-v3 and v2-v3 define the major and minor axes of the ellipse and v3 defines the center

point. These vectors should be at 90 degrees. If not, Cubit will issue a warning indicating the vertices are not sufficient to create an ellipse and will then default to creating a spiral.

The angle options will specify what portion of the ellipse to create. If none are specified, **start angle** will default to 0 and **stop angle** to 90 and the ellipse will go from vertex 1 to vertex 2; if the vertices are free vertices they will be consumed in the ellipse creation. **Start angle** tells Cubit where to start the ellipse -- the angle from the first axis (v1 - v3) specified. **Stop angle** tells Cubit where to end the ellipse -- the angle from the first axis. The angle follows the right-hand rule about the normal defined by (v1 - v3) X (v2 - v3).

3. Spline: The spline form of the command creates a spline curve that goes through all the input vertices or locations. To create a curve from a list of vertices use the syntax shown below. The **delete** option will remove all of the intermediate vertices used to create the spline leaving only the end vertices.

```
Create Curve [Vertex] <vertex_id_list>
[Spline] [Delete]
```

Additionally, spline curves can be created by inputting a list of locations. Where the spline will pass through all of the specified locations. The syntax is shown below:

```
Create Curve Spline {List of locations}
```

See Location, Direction, and Axis Specification to view the location specification syntax.

4. Copy: This command actually copies the geometric definition in the specified curve to the newly created curve. The new curve is free floating.

```
Create Curve From Curve <curve_id>
```

5. Combine Existing Curves: This command creates a new curve from a connected chain of existing ACIS curves.

```
Create Curve combine curve <id_list> [delete]
```

6. Arc Three: The following command creates an arc either through 3 vertices or tangent to 3 curves. The *Full* qualifier will cause a complete circle to be created.

```
Create Curve Arc Three {Vertex|Curve}
<id_list> [Full]
```

7. Arc End Vertices and Radius: The following command creates an arc using two vertices, the radius and a normal direction. The *Full* qualifier will cause a complete circle to be created.

```
Create Curve Arc Vertex <id_list>
Radius <value> Normal {<x> <y> <z> |
{direction options} [Full]
```

Go to Location, Direction, and Axis Specification to see the direction command description.

8. Arc Center Vertex: The next form of the command creates an arc using the center of the arc and 2 points on the arc. The arc will always have a radius at a distance from the center to the first point, unless the *Radius* value is given. Again, the *Full* qualifier will cause a complete circle to be created.

```
Create Curve Arc Center Vertex <center_id>
<end1_id> <end2_id>
[Radius <value>] [Full]
[Normal {<x> <y> <z> | {direction options}]
```

Go to Location, Direction, and Axis Specification to see the direction command description.

Note: Requires 3 Vertices - first is the center, the other two are the end points of the arc. A normal direction is required when the three points are colinear. Otherwise a normal direction is optional.

9. Arc Center Angle: This form of the command creates an arc using the center position of the arc, the radius, the normal direction and the sweep angle.

```
Create Curve Arc Center {<x=0> <y=0> <z=0> |
{location options}
Radius <value>
Normal {<x> <y> <z> | {direction options}
Start Angle <value=0> Stop Angle <value=360>
```

Go to Location, Direction, and Axis Specification to see the location and direction command descriptions.

10. From Vertex Onto Curve: The following command will create a curve from a vertex onto a specified position along a curve. If none of the optional parameters are given, the location on the curve is calculated as using the shortest distance from the start vertex to the curve (i.e., the new curve will be normal to the existing curve).

```
Create Curve From Vertex <vertex_id> Onto
Curve <curve_id> [Fraction <f> | Distance <d>
| Position <xval><yval><zval> | Close_To
Vertex <vertex_id> [[From] Vertex <vertex_id>
(optional for 'Fraction' & 'Distance')]] [On
Surface <surface_id>]
```

Note: Default = Normal to the Curve

11. Offset: The next command creates curves offset at a specified distance from a planar chain of curves. The direction vector is only needed if a single straight curve is given. The offset curves are trimmed or extended so that no overlaps or gaps exist between them. If the curves need to be extended the extension type can be *Rounded* like arcs, *Extended* tangentially (the default -straight lines are extended as straight lines and arcs are extended as arcs), or extended *naturally*.

```
Create Curve Offset Curve <id_list> Distance
<val> [Direction <x> <y> <z>]
[Rounded|EXTENDED|Natural]
```

Note: Direction is optional for offsets of individual straight curves only

In all cases, the specified vertices are not used directly but rather their positions are used to create new vertices.

12. From Mesh Edges: This commands creates a curve from an existing mesh given a starting node and an adjacent edge.

```
Create Curve From Mesh Node <id> Edge <id>
[Length <val>]
```

The adjacent edge indicates which direction to propagate the curve. The curve will be composed of mesh edges up to the specified length. If no length is specified the curve will propagate as far as the boundary of the mesh. Figure 1 shows a example of a curve generated from the mesh.

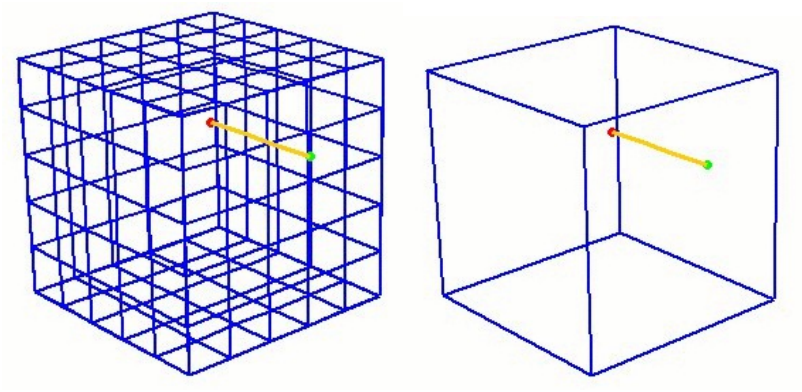


Figure 1. Example of curve created from mesh

The underlying geometry kernel used for this command is Mesh-Based geometry. The new curve will also be meshed with the edges it was propagated through. A related command for assigning mesh edges directly to a mesh block is the Rebar command. See Element Block Specification for more details.

Note: Full hexes or full tets must be used to propagate the curves through the interior of volume.

13. Close_To This option takes two geometric entities and creates the shortest possible curve between the two entities at the location where the two entities are the closest. The two entities may NOT intersect. If two vertices are given, the command will create a straight line between the two vertices.

```
Create Curve Close_To
{Vertex|Curve|Surface|Volume|Body} <id_1>
{Vertex|Curve|Surface|Volume|Body} <id_2>
```

14. Surface Intersection The following command creates curves at surface intersections. Multiple curves can be created from a single command.

```
Create Curve Intersecting Surface <id_list>
```

15. By Projection The project command projects curves, or the curves of a surface another a single surface or multiple surfaces of a volume or body. The command syntax is as follows:

```
Project { Curve <id_list> | Surface <id_list>
} Onto Surface <surface_id> [Imprint
[Keepcurve] [Keepbody]] [Trim]

Project { Curve <id_list> | Surface <id_list>
} Onto {Body <id> | Volume <id>}
[Target_surface <id_list>] [Imprint
[Keepcurve] [Keepbody]]
```

The first form of the command takes a list of curves or surfaces, and a projection surface. If a list of curves is given, the result will be the

creation of a set of free curves on top of the projection surface. If a list of surfaces is given, the result will be the same as selecting the curves of the surface (i.e. a group of free curves on the projecting surface).

The second form will imprint the list of curves (or curves of the surface(s)) onto the surfaces of the specified bodies or volumes. The **Target_surface** option helps when the projection is ambiguous, for example projecting curves onto a thin-walled volume where the projection could be to either side, as shown in Figure 2.

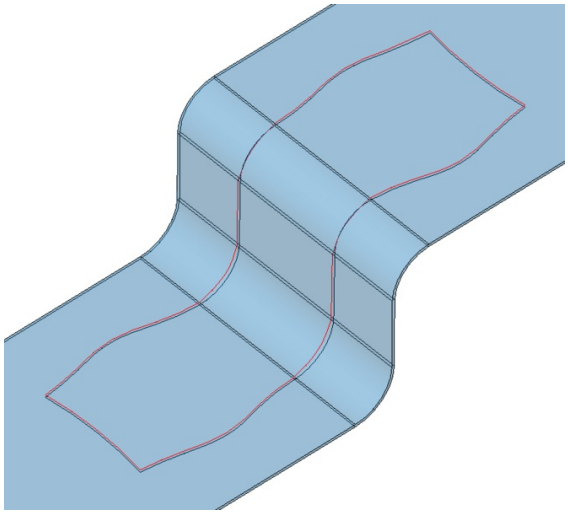


Figure 2. Example of projecting curves to specific side of thin-walled volume.

The imprint option will imprint the resulting projected curves onto the projection surface. If this option is NOT given, the new curves will lie coincident to the surface, but will not be part of the surface. Imprinting changes the topology of the projection surface. Keepcurve option retains the new curves as both free curves, and curves in the projection surface. The keepbody option retains the original body under the new imprinted body. When projecting curves, the trim option will cause the curve to be trimmed to the target surface.

16. Creating a Helix: This command will create a helical curve. The command syntax is as follows:

```
Create Curve Helix { axis <xpoint ypoint zpoint
xvector yvector zvector> | xaxis | yaxis |
zaxis } location (options) thread_distance
<value> angle <value> [RIGHT_HANDED |
left_handed]
```

axis = axis about which to create the helix

location (options) = starting point of the helix

thread_distance = distance between each 360 degree segment of the helix

angle = number of degrees in rotation of the helix

handedness = right-handed or left- handed threads

17. Tangents: This command will create a spline curve by specifying the end points and the tangents at those points. The command syntax is as follows:

```
create curve tangent vertex <id> vertex <id>
[start direction (options)] [end direction
(options)]
```

```
create curve tangent start location (options)
end location (options) start direction
(options) end direction (options)
```

Both forms of the command can be broken into two parts: the points and the tangent directions. The first point is associated with the start direction.

The first form of the command takes an existing vertex and an optional tangent direction. If the direction is not specified, it will be taken from the tangent at the endpoint of the connected curve. If the vertex is not connected to a curve, or it is connected to multiple curves, the direction must be specified. If the vertex is not connected to a curve, it will be incorporated into the new curve. Otherwise, a new vertex will be created for the new curve.

The second form of the command takes a location and a tangent direction. The directions must be specified and vertices will be created for the locations.

The two command forms can also be mixed. A curve can be created from an existing vertex and a specified location.

Examples:

```
create curve tangent start location 0 0 0 end location 1 1 0 start
direction 1 0 0 end direction 1 0 0
```

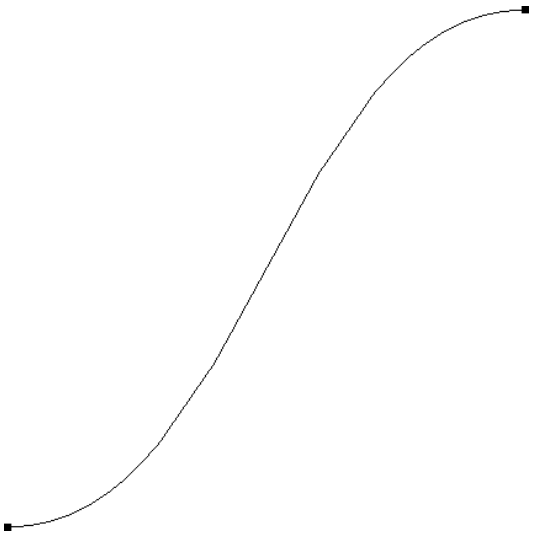


Figure 3. Create tangent curve with locations and tangent directions

```
create curve tangent vertex 1 vertex 3

merge vertex all
```

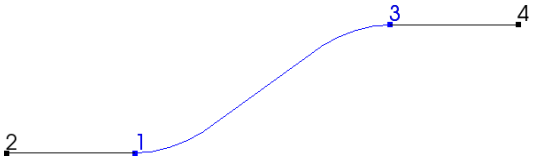


Figure 4. Create tangent curve by specifying vertices on curves. Tangents are extracted from the curve at the vertex location.

Go to Location, Direction, and Axis Specification to see the location and direction command description.

Creating Surfaces

There are two major ways to create surfaces in CUBIT. First, surfaces can be created in CUBIT by fitting an analytic or spline surface over a set of bounding curves. In this case, the curves must form a closed loop, and only one loop of curves may be supplied. The second method, is by sweeping a curve about an axis, along a vector, or along another curve. The result of these surface creation commands is a "sheet body" or a body that has zero measurable volume (it does however have a volume entity). This body may be decomposed with booleans and special webcutting commands or it may be used as a tool to decompose other bodies. Booleans can be used to cut holes out of these surfaces.

The following options may be used for creating a surface in CUBIT.

- Bounding Curves
- Bounding Vertices or Nodes
- Copy
- Extended Surface
- Planar Surface
- Net Surface
- Offset
- Skinning
- Sweeping of Curves
- Midsurface
- Weld Profile
- Meshed Entities
- Circular Surface
- Parallelogram
- Ellipse
- Rectangle

1. Bounding Curves: The first form of this command produces an analytic or spline surface fit to cover the bounding curves.

```
Create Surface Curve <curve_id_1> <curve_id_2>
<curve_id_3>...
```

Another version of this command creates a surface from a set of bounding curves that all lie on one surface. If the curves are selected they must lie on the surface, and they must create a closed loop. The **On Surface** option forces the surface to match the geometry of the underlying surface exactly.

```
Create Surface Curve <id_list> On Surface
<surface_id>
```

2. Bounding Vertices or Nodes: The second form of this command uses vertices to fit an analytic spline surface. The **On Surface** option creates the surface from a set of nodes and vertices that all lie on one surface and restrains the surface to match the geometry of the underlying surface. The project option will project the nodes or vertices to the specified surface.

```
Create Surface [Node|Vertex| <id_list> [On
Surface <surface_id> {Project} ]
```

3. Copy: The next form creates a surface using the same geometric description of the specified surface. The new surface will be a stand-alone sheet body that is geometrically identical to the user supplied

surface.

```
Create Surface From Surface <surface_id>
```

4. Extended Surface: The fourth form of the command creates a surface that is extended from a given surface or list of surfaces. The specified surface's geometry is examined and extended out "infinitely" relative to the current model in CUBIT (i.e. extended to just beyond the bounding box of the entire model). The given surfaces are extended as shown in the table.

```
Create Surface Extended From Surface
<surface_id>
```

Table 1. Surface Extension Results

Surface Type	Resulting Extended Surface
Spherical	Shell of Full Sphere
Planar	Plane of infinite size relative to model
Toroidal	Shell of Full Torus
Conical, cone, cylinder...	Shell of outside conic axially aligned with given conic of infinite height relative to model
Spline	Surface is extended to extents of the spline definition. This may not be any further than the surface itself, so caution should be used here.

Multiple surfaces can be offset at the same time to form a sheet body, by using the Create Sheet Extended from Surface command.

5. Planar Surface: The following commands create ***planar*** surfaces. The first passes a plane through 3 vertices, the second uses an existing plane, the third creates a plane normal to one of the global axes, and the fourth creates a plane normal to the tangent of a curve at a location along the curve. By default, the commands create the surface just large enough to intersect the bounding box of the entire model with minimum surface area. Optionally, you can give a list of bodies to intersect for this calculation. You can also extend the size of the surface by either a percentage distance or an absolute distance of the minimum area size. The plane can be previewed with the command Draw Plane [with]... (where the rest of the command is the same as that to create the surface).

```
Create Planar Surface [With] Plane Vertex
<v1_id> [Vertex] <v2_id> [Vertex] <v3_id>
[Intersecting] Body <id_range>] [Extended
Percentage|Absolute <val>]

Create Planar Surface [With] Plane Surface
<surface_id> [Intersecting] Body <id_range>]
[Extended Percentage|Absolute <val>]

Create Planar Surface [With] Plane
{Xplane|Yplane|Zplane} [Offset <val>]
[Intersecting] Body <id_range>] [Extended
Percentage|Absolute <val>]
```

```
Create Planar Surface [With] Plane Normal To
Curve <curve_id>{Fraction <f>| Distance <d> |
Position <xval><yval><zval> | Close_to vertex
<vertex_id>} [[From] Vertex <vertex_id>
(optional for 'fraction' & 'distance')]
[Intersecting] Body <id_range>] [Extended
Percentage|Absolute <val>]
```

6. Net Surface: *Net surfaces* can be created with two different commands. A net surface passes through a set of curves in the u-direction and a set of curves in the v-direction (these u and v curves would looked like a mapped mesh). The first form of the command uses curves to create the net surface. The curves must pass within tolerance of each other to work. The second form uses a mapped mesh to create the surface. The mapped mesh can be of a single surface or a collection of mapped or submapped surfaces that form a logical rectangle. By default net surfaces are healed to take advantage of any possible internal simplification.

```
Create Surface Net U Curve <id_list> V Curve
<id_list> [Tolerance <value>] [HEAL|Noheal]

Create Surface Net [From] [Mapped] Surface
<id_list> [Tolerance <value>] [HEAL|Noheal]
```

A suggested geometry cleanup method is to use a virtual composite surface to map mesh a set of complicated surfaces then create a net surface from this mesh. Then the original surfaces can be removed with the *noextend* option and the new net surface combined back onto the body.

7. Offset: The following command creates surfaces ***offset*** from existing surfaces at the specified distances.

```
Create Surface Offset [From] Surface <id_list>
Distance <val>
```

The surface offset command will only translate the existing surfaces, without extending or trimming them. An alternate form of the command for sheet bodies will maintain connections between surface by extending or trimming as they are offset, shown in Figure 1. On the left, the surfaces are offset using the surface offset command. On the left, the surface is created by using the "sheet" version of the command.

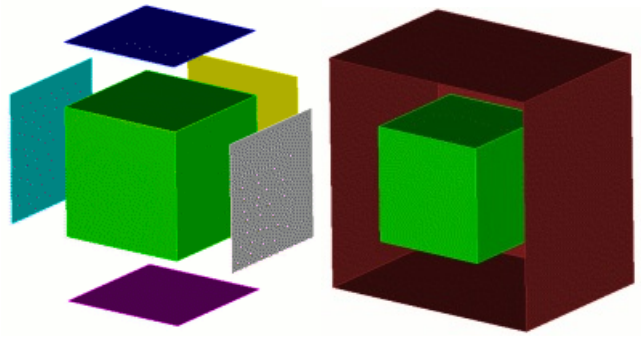


Figure 1. Offsetting surfaces to form individual surfaces or sheet bodies

8. Skinning: The following command creates a ***skin*** surface from a list of curves. An example of a skin surface is to create a surface through a set of parallel lines.

```
Create Surface Skin Curve <id_list> [pair
tolerance <val>
```

The **pair tolerance** option indicates vertices on curves to skin will be paired up with a given tolerance. If a pair cannot be found for a vertex, a new vertex is inserted to split a curve and make a pair. This option is useful to control creation of lateral curves in the new surface and to avoid skew for meshes created on those surfaces.

9. Sweeping of Curves: A curve or a set of curves can be swept along a path to create new surfaces. The path may be specified as an axis and angle, a vector and distance, by indicating another curve or set of contiguous curves, or by specifying a target plane. The following commands show the options available:

```
Sweep Curve <curve_id_range> { Axis <xpoint
ypoint zpoint xvector yvector zvector> | Xaxis
| Yaxis | Zaxis } Angle <degrees> [Steps
<Number_of_sweep_steps>] [Draft_angle
<degrees>] [Draft_type <integer>] [Make_solid]
[Include_mesh] [Keep][Rigid]

Sweep Curve <curve_id_range> Vector <xvector
yvector zvector> [Distance <distance>]
[Draft_angle <degrees>] [Draft_type <integer>]
[Include_mesh] [Keep] [Rigid]

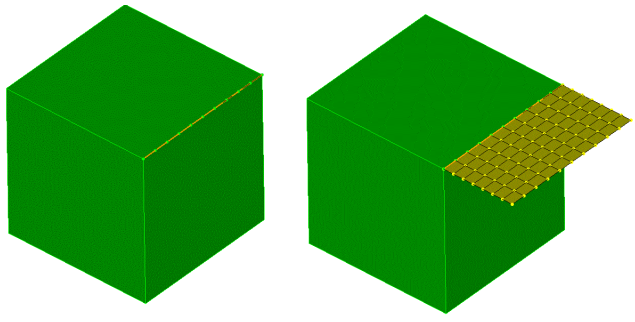
Sweep Curve <curve_id_range> Along Curve
<refcurve_id_range> [Draft_angle <degrees>]
[Draft_type <integer>] [Include_mesh] [Keep]
[Rigid]

Sweep Curve <curve_id_range> Target Plane
<options>

Sweep Curve <curve_id_range> Target
{Volume|Body} <id> Direction {options} [Plane
<options>] [Unite]
```

In the first command, the steps options provides a way of faceting the sweep, so instead of a smooth round sweep, there are facets to the surface. The **make_solid** option closes the newly-created surface to the axis, so that a solid is created instead of a surface.

In the above commands, the include_mesh option will create a surface mesh if the curve is already meshed (see figure below). The keep option will keep the original curve while creating the surface.



The **sweep curve target plane** command sweeps a curve until it hits a target plane. The options for the target plane are described under Specifying a Plane.

The last command sweeps a curve to a target volume or body and can only be used on sheet bodies. Use the direction keyword to specify the sweep direction and the plane keyword to specify a stopping plane. The **unite** keyword will unite the sheet bodies after sweeping

The other options are as follows:

draft_angle: determines how much drafting in of the surface is desired

draft_type:

0 => extended (draws two straight tangent lines from the ends of each segment until they intersect)

1 => rounded (create rounded corner between segments)

2 => natural (extends the shapes along their natural curve) ***

rigid: normally the curve will rotate to maintain its original orientation to the sweep path. The rigid option disallows this rotation.

10. Midsurface: Multisurfaces may be created midway between pairs of surfaces using the following command:

```
Create Midsurface {Body|Volume} <id> Surface
<id11> <id12> ... <idN1> <idN2>
```

where N denotes the number of pairs of surfaces. An even number of surfaces must be specified, and the command will group them by pairs in the order in which they are provided. The resulting surface will be trimmed by the specified body or volume <id>. This replaces the *Create Midplane* command in previous versions of CUBIT.

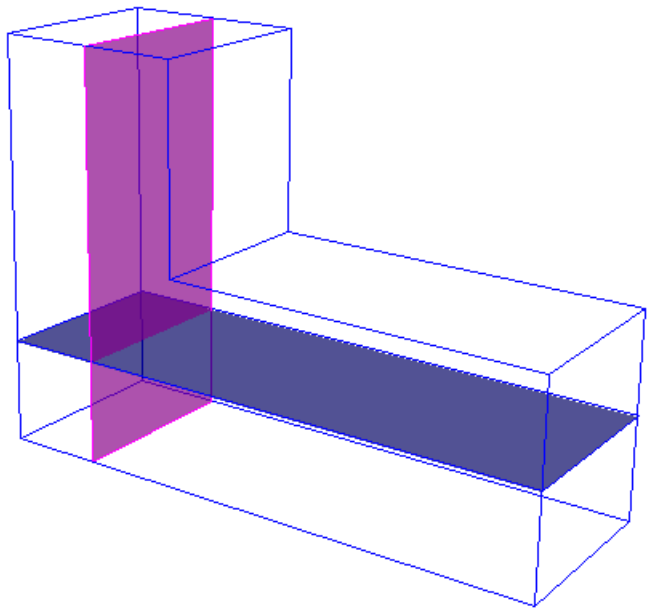


Figure 2. Multisurface created with the Create Midsurface command

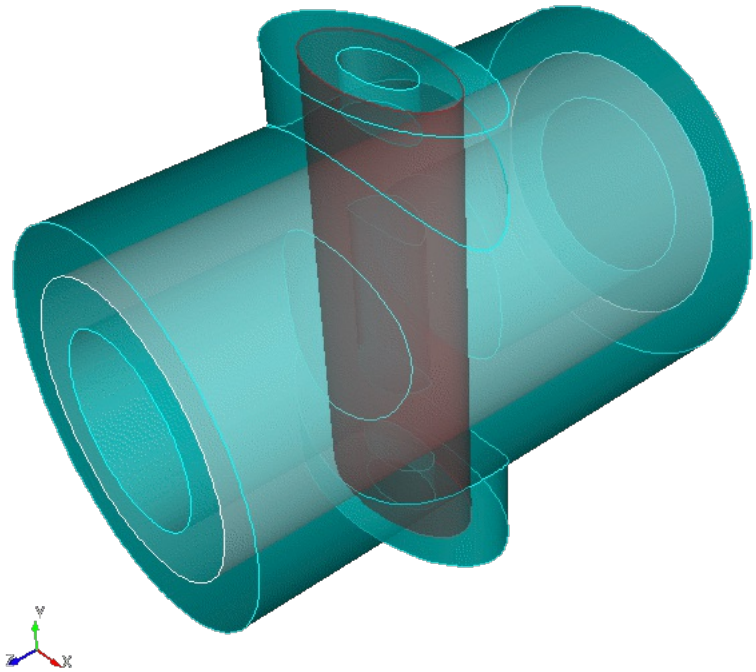


Figure 3. Midsurface created from 2 pairs of cylindrical surfaces

Midsufaces can also be extracted without surface pair specification if the resulting surface is a single sheet of surfaces (no T intersections). The following is the command syntax for automatic midsurface extraction:

```
Create Midsurface {Body|Volume} <id_range>
Auto [Delete] [Transparent] [Thickness] [Limit
<lower_bound> <upper_bound>] [Preview]
```

Figure 4 shows a simple auto midsurface example. The command for the example is:

```
create midsurface volume 1 auto delete
```

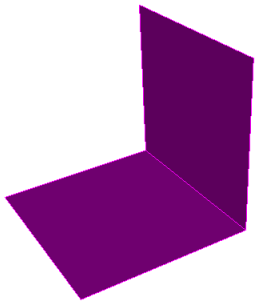


Figure 4. Midsurface created from a volume

The command option descriptions are listed below.

Auto enables the automatic mid-surface algorithm. Turning Auto off requires the user to specify a single surface pair to create a mid-surface.

Transparent shows the successfully midsurfaced volumes as transparent in the graphics display

Thickness applies a 2D property to the created mid-surface geometry.

Limit search range gives the algorithm a range to find surface pairs within.

11. Weld Profile: Surfaces may be created by specifying a weld profile using the following command:

```
Create Surface Weld [Root] Location {options}
Weld Surface <id_list> Length <val> [<val2>]
```

Weld surfaces can be used to create a simulated welded joint by sweeping the surface along the root curve and uniting the new body to the model. An example of the command is illustrated below. For a detailed description of the location specifier see Location Direction, and Axis Specification.

```
create surface weld root location vertex 25
weld surface 13 14 length 2
```

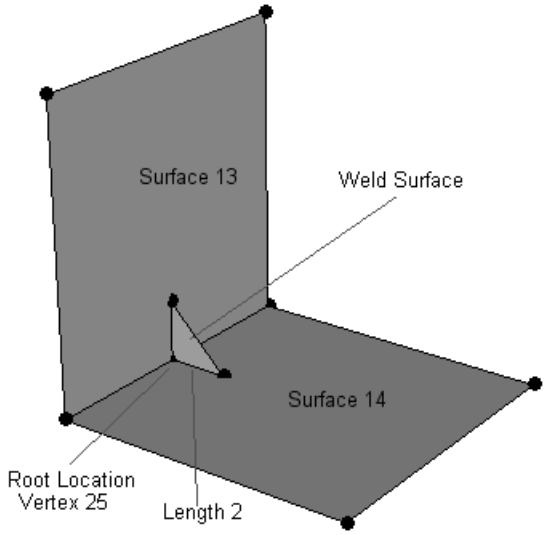


Figure 5. Weld Profile surface with length and root specifications

12. Creating A Surface From Mesh Entities: Surfaces may be created from the boundaries of meshed volumes, surfaces, and/or from individual quadrilateral mesh elements. The individual option makes it so you can enter multiple surfaces at once, and not have them merged together into a larger surface, but instead retain their own original boundaries. The optional tolerance value allows the user to specify a tolerance to which the resulting surface should be fit. The default value is 0.001. If surface creation fails, increasing the tolerance value can help.

```
Create Acis [From] {Surface <id_range> |
Volume <id_range> | Face < id_range>
[Individual]} [Tolerance <value>]
```

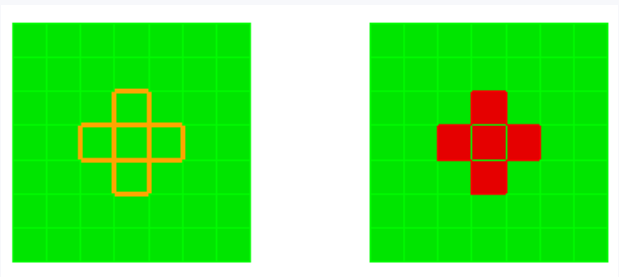


Figure 6. Acis Surface created from a Set of Quadrilaterals

13. Creating a Circular Surface: This command creates a 2D circular surface. The surface will be centered at the origin and on the z-plane if a

plane option is not specified.

```
create surface circle radius <value> {xplane|yplane|ZPLANE}
```

This command creates a 2D circular surface by specifying three vertices; the first vertex will be the center of the surface, the second vertex will be used to define the radius of the surface, and the third vertex will assist in defining the plane that the surface will lie in.

```
create surface circle center vertex <v1_id> <v2_id> <v3_id>
```

This command creates a 2D circular surface by forming a circular curve through three points.

```
create surface circle vertex <v1_id> <v2_id> <v3_id>
```

14. Creating a Parallelogram: This command creates a 2D parallelogram surface, centered at the origin, by specifying three corner vertices. These vertices will form three consecutive corners of the parallelogram surface.

```
create surface parallelogram vertex <v1_id> v2_<id> <v3_id>
```

15. Creating an Ellipse: This command creates a 2D elliptical surface, centered at the origin, by specifying at least a major radius. On an x-y plane this radius will be the radius along the x-direction. The minor radius will be the radius along the y-direction. By default, the surface will lie in the z-plane.

Create Surface Ellipse major radius <value> [minor radius <value>] [xplane|yplane|ZPLANE]

This command creates a 2D elliptical surface using three vertices. The first two vertices define the major and minor radii of the ellipse surface. The third point defines the center of the ellipse. It is important to note that a line from v1_id to v3_id must be orthogonal to a line from v2_id to v3_id, otherwise the command will fail.

Create Surface Ellipse vertex <v1_id> <v2_id> <v3_id>

16. Creating a Rectangle: This command creates a rectangular surface centered at the origin. If only a width value is specified, the surface will be a square. On an x-y plane, the width value is the x-direction and the height is the y-direction. By default, the surface will lie in the z-plane.

Create Surface rectangle width <value> [height <value>] [xplane|yplane|ZPLANE]

Creating Vertices

The basic commands available for creating new vertices directly in CUBIT are:

- XYZ location
- On Curve - Fraction
- On Curve - General
- From Vertex
- At Arc
- At Intersection

1. XYZ location: The simplest form of this command is to specify the XYZ location of the vertex. It can also be created lying on a curve or surface in the geometric model by specifying the curve or surface id; the position of the vertex will be the point on the specified entity which is closest to the position specified on the command. With all of these commands, the user is able to specify the color of the vertex.

```
Create Vertex <x><y><z> [On [Curve | Surface]
<id>] [Color <color_name>]
```

2. On Curve - Fraction: A vertex can be positioned a certain fraction of the arc length along a curve using the second form of the command.

```
Create Vertex On Curve <id> Fraction <0.0 to
1.0> [Color <color_name>]
```

Vertex 3 in the following example was created with this command:

```
create vertex on curve 1 fraction 0.25 from vertex 1
```

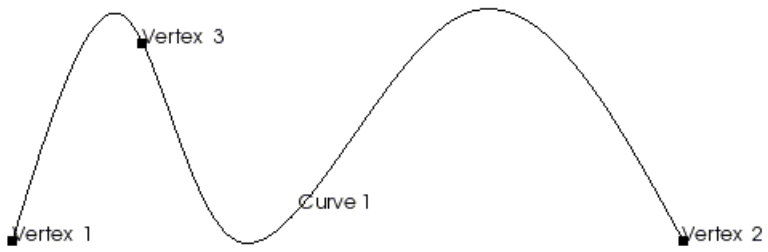


Figure 1. Create Vertex a Fraction of the length of a Curve

3. On Curve - General: A more general purpose form of the command is also available for creating vertices on curves:

```
Create Vertex On Curve <id_list> { MIDPOINT |
Start | End | Fraction <val 0.0 to 1.0> [From
Vertex <id> | Start|End] | Distance <val>
[From {Vertex|Curve|Surface} <id> | Start|End]
| {{Close_To|At} Location {options} | Position
<xval><yval><zval>|{Node|Vertex} <id>} |
Extrema [Direction] {options} [Direction
{options}] [Direction {options}] | Segment
<num_segs> | Crossing {Curve|Surface}
<id_list> [Bounded|Near] } [Color
<color_name>]
```

It allows the vertex to be created at a fractional distance along the curve,

at an actual distance from one of the curves ends, at the closest location to an xyz position or another vertex, or at a specified distance from a vertex, curve or surface. You can also preview the location first with the command Draw Location On Curve (where the rest of the command is identical to the Create Vertex form).

4. From Vertex: Create a vertex from an existing vertex.

```
Create Vertex from Vertex <id_list> [ On
{Curve|Surface} <id> ] [Color <color_name>]
```

If 'on curve|surface' option is used, the vertex is positioned on that curve or surface. When the 'on curve|surface' is not used, the new vertex is positioned on the existing vertex.

5. At Arc: Another form simply creates vertices at arc or circle centers.

```
Create Vertex Center Curve <id_list> [Color
<color_name>]
```

6: At Intersection: The last form creates vertices at the intersection of two curves. If the *bounded* qualifier is used, the vertices are limited to lie on the curves, otherwise the extensions of the curves are also used to calculate the intersections. The *near* option is only valid for straight lines, where the closest point on each curve is created if they do not actually intersect (resulting in two new vertices).

```
Create Vertex AtIntersection Curve <id1> <id2>
[Bounded] [Near] [Color <color_name>]
```

Geometry Transforms

- Align
- Copy
- Move
- Scale
- Rotate
- Reflect

Bodies can be modified in CUBIT using transform operations, which include align, copy, move, reflect, restore, rotate, and scale. With the exception of the copy operation, transform operations in CUBIT do not create new topology, rather they modify the geometry of the specified bodies. ACIS, Mesh Based Geometry and Virtual Geometry representations may be transformed. If the geometric entity has been meshed, the nodes of the mesh will be transformed along with the geometry. To transform the nodes of a mesh as they are written to the Exodus II mesh file without modifying their location within CUBIT, see Transforming Mesh Coordinates.

Align Command

The align command is a combination of the rotate and move commands. This transformation is useful for aligning surfaces in preparation for geometry decomposition and aligning models for axis-symmetric analysis. If the **[include_merged]** option is used, all entities that are merged with the specified volume will be included in the align transformation also.

The first align command will transform the specified volumes by computing a transformation that aligns the source axis or surface with the target axis or surface such that the source axis centroid or surface centroid is coincident with the target axis origin or surface centroid and the surface normal(s) and/or axis(s) are pointing either in the same or opposite direction (depending on their initial alignment). The source surface need not be in the specified volumes. If the **[reverse]** option is specified, the resulting alignment is flipped 180 degrees.

An optional second axis or surface source-target pair can be specified. This will result in an additional rotational alignment: The specified volumes will be rotated about the first target axis or surface normal such that the second target axis or surface is at the same angle about the first target axis or surface normal.

```
Align Volume <id_range>
     Source {Axis {options}|Surface
<surface_id>}
     Target {Axis {options}|Surface
<surface_id>}
     [Source {Axis {options}|Surface
<surface_id>}
     Target {Axis {options}|Surface
<surface_id>}]
     [reverse] [include_merged] [preview]
```

The image below shows the two alignments that occur when the additional source-target pair is specified.

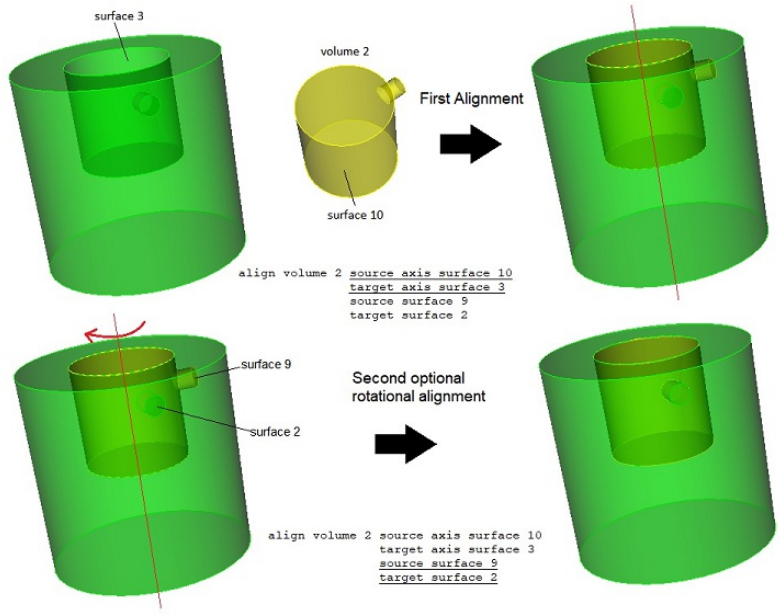


Figure 1 - Aligning with optional rotation

The second form of the align command either aligns a face of a volume or two vertices (forming a direction) with the xy, yz, and xz planes or the

x, y, and z axes. If the **[reverse]** option is specified, the resulting alignment is flipped 180 degrees.

```
Align Volume <id_range> {Surface <surface_id>|
Vertex <vertex_id>} {{X|Y|Z Axis}|{XY|XZ|YZ
plane}} [reverse] [include_merged] [preview]
```

The third form of the command is a rotational alignment, where the specified entities are rotated about the specified axis, where the angle of rotation is the angle between the first and second locations with respect to the axis.

```
Align Volume <id_range> Location {options}
with Location {options} about Axis {options}
[include_merged] [preview]
```

The fourth form of the command uses vertex pairs to define the transformation. The first pair of vertices define a translation so that the source and target vertices are coincident (i.e.,the source is moved to the target). The optional second pair of vertices define a rotation such that the source and target vertices are all collinear after the transformation. The optional third pair of vertices define a rotation such that all the source and target vertices will be coplanar.

```
{Body|Volume|Surface|Curve|Vertex|Group}
<id_range> Align Using Vertex <src_id>
<tgt_id>
[Collinear Vertex <src_id> <tgt_id> [Coplanar
Vertex <src_id> <tgt_id>] ] [include_merged]
```

Copy Command

The copy command copies an existing entity to a new entity without modifying the existing entity. A copy can be made of several entities at once, and the resulting new entities can be translated or rotated at the same time. The commands for copying entities are:

```
Vertex <range> Copy [Move [X <dx>] [Y <dy>] [Z
<dz>]] [Preview]

Vertex <range> Copy [Move <direction_options>
[Distance <val>]] [Preview]

{Body|Volume|Surface|Curve|Vertex|Group}
<range> Copy Move [X <dx>] [Y <dy>] [Z <dz>]
[Nomesh] [Repeat <value>] [Preview]

{Body|Volume|Surface|Curve|Vertex|Group}
<range> Copy Move <direction_options>
[Distance <val>] [Nomesh] [Repeat <value>]
[Preview]

{Body|Volume|Surface|Curve|Vertex|Group}
<range> Copy Reflect {X|Y|Z} [Nomesh]
[Preview]

{Body|Volume|Surface|Curve|Vertex|Group}
<range> Copy Reflect [Vertex <v1_id> [Vertex]
<v2_id] [Nomesh] [Preview]

{Body|Volume|Surface|Curve} <range> Copy
Reflect <x> <y> <z> [Nomesh] [Preview]

{Body|Volume|Surface|Curve} <range> Copy
Rotate <angle> About {X|Y|Z} [Repeat <value>]
[Nomesh] [Preview]

{Body|Volume|Surface|Curve} <range> Copy
Rotate <angle> About <x> <y> <z> [Nomesh]
  [Repeat <value>] [Repeat <value>] [Preview]

{Body|Volume|Surface|Curve} <range> Copy Scale
<scale> | X <val> Y <val> Z <val> [About
Vertex <id>] [Nomesh] [Repeat <value>]
 [Preview]
```

If the copy command is used to generate new entities, a copy of the original mesh generated in the original entity will also be copied directly onto the new entity unless the **nomesh** option is used.

Several of the commands include the **Repeat** token. If that token is used the command will repeat itself **value** times.

This is currently limited to copies that do not interact with adjacent geometry through non-manifold topology. For details on mesh copies, see the Mesh Duplication documentation.

Move Command

The move command moves a body, volume, free surface, free curve or free vertex by a specified offset. The command syntax is:

```
Vertex <id_range> [Move [X <dx>] [Y <dy>] [Z
<dz>]] [Copy] [Preview]

Vertex <id_range> Move <direction_options<
[Distance <val>] [Copy] [Preview]

{Body|Volume|Surface|Curve|Vertex|Group}
<id_range> [Move [X <dx>] [Y <dy>] [Z <dz>]]
[Copy [Nomesh]] [Preview]

{Body|Volume|Surface|Curve|Vertex|Group}
<id_range> Move <direction_options> [Distance
<val>] [Copy [Nomesh]] [Preview]
```

where <dx> <dy> <dz> and <distance> represent relative offsets in the major axis directions. If the copy option is specified, a copy is made and the copy is moved by the specified offset. The nomesh option will copy and move only the geometry.

These forms of the Move command will only work on free surfaces and free curves. To move a curve or surface that is part of a higher-order entity, the **Move {entity} ...** command is used.

Moving Other Geometric Entities

It is also possible to move bodies by specifying one of its child entities. For example, a body can by moved by specifying one of its curves. However, if a lower-order entity is moved, the parent body and all related entities will also be moved. The commands for moving bodies using a child entity are given below. Alternatively, the tweak command can be used to move curves and surfaces without moving the parent body.

```
Move {Vertex|Curve|Surface|Volume|Body|Group}
<id_range> [Midpoint] Location <x> [<y> [<z>]]
[Include_Merged] [Preview]

Move {Vertex|Curve|Surface|Volume|Body|Group}
<id_range> Location [Midpoint] [X <val>] [Y
<val>] [Z <val>] [Except [X] [Y] [Z]]
[Include_Merged] [Preview]

Move {Vertex|Curve|Surface|Volume|Body|Group}
<id_range> Normal to Surface <id> Distance
<val> [Include_Merged] [Preview]

Move {Vertex|Curve|Surface|Volume|Body|Group}
<id_range> [Midpoint] General Location
<location_options> [Except [X] [Y] [Z]]
[Include_Merged] [Preview]
```

The first form of the command will move the entity to an absolute location. If moving a group, the centroid of the group is moved to that location. The second form will move the entity by a relative distance in any of the xyz axis directions. "Except" is used to preserve the x, y, or z plane in which the center of the entity lies. The third form of the command will move the body along an axis defined by the outward-facing surface normal of another surface. The fourth form of the command uses general location parsing to move the entity.

Moving Bodies Relative to Other Geometric Entities

It is also possible to move bodies relative to other geometric entities in the model. The following command takes as arguments two geometric entities. The first entity is the one to move. The second entity is where it will be moved. In both cases, the midpoints of the specified entity are used to determine the distance and direction of the move. In the case of groups, centroids are used. "Except" is used to preserve the x, y, or z plane in which the center of the entity lies.

```
Move {Vertex|Curve|Surface|Volume|Body|Group}
<id_range> [Midpoint] Location
{Vertex|Curve|Surface|Volume|Body|Group} <id>
[Midpoint] [Except [X] [Y] [Z]]
[Include_Merged] [Preview]
```

Moving Merged Entities

The easiest way to move merged entities is by adding the *include_merged* keyword to the command. All entities that are merged with the specified entities will move together.

The only other way that merged entities can be moved is by including each of the merged entities in the entity list.

Move Undo

The Undo option allows a user to reverse the most recent move. This command will only work for the **Move {entity}** commands, and not the **{Entity} Move** commands. The syntax is:

```
Move Undo
```

Reflect Command

The reflect command mirrors the body about a plane normal to the vector supplied. The reflect command will destroy the existing body and replace it with the new reflected body, unless the copy option is used.

```
{Body|Volume|Surface|Curve|Vertex|Group}
 <range> [Copy] Reflect <x-comp> <y-comp> <z-
comp>

{Body|Volume|Surface|Curve|Vertex|Group}
 <range> [Copy] Reflect {X|Y|Z}
```

Rotate Command

The rotate command rotates a body about a given axis without adding any new geometry. If the Angle or any Components are not specified they are defaulted to be zero. The commands to rotate a body or bodies are:

```
Body <range> [Copy] Rotate <angle> About
{X|Y|Z} [Preview]

Body <range> [Copy] Rotate <angle> About <x-
comp> <y-comp> <z-comp> [Preview]

Rotate
{Body|Volume|Surface|Curve|Vertex|Group}
<id_range> about {X|Y|Z|<xval> <yval> <zval>}
Angle <val> [Include_Merged] [Preview]

Rotate
{Body|Volume|Surface|Curve|Vertex|Group}
<id_range> About Vertex <id> Vertex <id> Angle
<val> [Include_Merged] [Preview]

Rotate
{Body|Volume|Surface|Curve|Vertex|Group}
<id_range> About Normal of Surface <id> Angle
<val> [Include_Merged] [Preview]

Rotate
{Body|Volume|Surface|Curve|Vertex|Group}
<id_range> About Origin <xval> <yval> <zval>
Direction <xval> <yval> <zval> Angle <val>
[Include_Merged] [Preview]
```

If the copy option is specified, a copy is made and rotated the specified amount.

Rotating Merged Entities

The easiest way to rotate merged entities is by adding the *include_merged* keyword to the command. All entities that are merged with the specified entities will rotate together.

The only other way that merged entities can be rotated is by including each of the merged entities in the entity list.

Scale Command

The **scale** command resizes an entity (body, volume, surface, or curve) by a scaling factor. The scaling factor may be a constant, or may differ in the x, y, and z directions. The entity chosen will be scaled about the point or vertex indicated. If no point or vertex is entered, it will be scaled about the origin. Any mesh on the object will be scaled too, unless the **nomesh** keyword is used.

The command to scale entities is:

```
{Body|Volume|Surface|Curve} <id_range> Scale
{<scale> | x <val> y <val> z <val>} [About
{<x> <y> <z> | Vertex <id>}] [Nomesh] [Copy
[Repeat <value>] [Group_Results]] [Preview]
```

If the **copy** option is specified, a copy of the entity is made and scaled the specified amount. Use the **repeat** option to create multiple copies.

Geometry Booleans

- Intersect
- Subtract
- Unite

CUBIT supports boolean operations of intersect, subtract, and unite for bodies.

An automatic function associated with webcutting operations is regularizing geometry which can be turned off or back on with the following command:

```
Set Boolean Regularize [ON | off]
```

Intersect

The intersect command generates one or more new volumes composed of the space that is shared by the volumes being intersected. There are two forms of the command. The first form is:

```
Intersect {Volume|[Body]} <range> [Keep]
[Preview]
```

In this form, all volumes in the range will be intersected, and the resulting intersection volume(s) will contain all intersections of all input volumes. The original volumes will be deleted. The second form is:

```
Intersect {Volume|[Body]} <id> [With {Volume|
[Body]} <range>] [Keep] [Preview]
```

In this form, the first volume is intersected with all volumes in the range specified using the **with** keyword. Intersections between volumes in the range are not included in the output. The original volume will be deleted and the other volumes updated with the intersection results.

The **keep** option results in the original volumes used in the intersect being kept.

If the **Preview** option is included in the command, the input volumes will not be modified. The computed intersection volume will be drawn as a red, shaded solid. For best results change the graphics mode to transparent or hidden line so the intersection is visible. Otherwise the intersection volume will be hidden by the volumes being intersected.

Subtract

The subtract operation subtracts one body or set of bodies from another body or set of bodies. The order of subtraction is significant - the body or bodies specified before the **From** keyword is/are subtracted from bodies specified after **From**. The new body retains the original body's id. If any additional bodies are created, they will be given the next highest available ids. The **keep** option simply retains all of the original bodies. The command is:

```
Subtract [Volume|BODY] <range> From
[Volume|BODY] <range> [Imprint] [Keep]
```

The imprint option imprints the subtracted bodies onto the resultant body.

Remove Overlap

The **Remove Overlap** command is a simplified form of the **subtract** operation above. This command takes exactly two volumes as arguments and uses the **modify** argument to define the volume where material is to be removed.

```
Remove Overlap Volume <id1> <id2> modify
[Volume <id> | Smaller | Larger]
```

The **Smaller** and **Larger** options are an alternate method of specifying which volume from **<id1>** or **<id2>** where material will be removed. When these argument are used, the geometric volume is measured for both volumes and the smaller or larger volume repectively is used as the **modify** entity. When both volumes have exactly the same volume, the smaller or larger entity ID is used.

Unite

The unite operation combines two or more bodies into a single body. The original bodies are deleted and the new body is given the next highest body ID available, unless the **keep** option is used. The commands are:

```
Unite [Volume|BODY] <range> [With
[Volume|BODY] <range>] [Keep]

Unite Body {<range> | All} [Keep]

Unite Body {<range> | All} [Include_mesh]
```

The second form of the command unites multiple bodies in a single operation. If the all option is used, all bodies in the model are united into a single body. If the bodies that are united do not overlap or touch, the two bodies are combined into a single body with multiple volumes.

The unite command allows sheet bodies to be united with solid bodies. To disable this capability you can turn the following setting off:

```
Set Unite Mixed {ON|Off}
```

Geometry Decomposition

Geometry decomposition is often required to generate an all-hexahedral mesh for three-dimensional solids, as fully automatic all-hex mesh generation of arbitrary solids is not yet possible in CUBIT. While geometry booleans can be used for decomposition (and are the basis of the underlying implementation of advanced decomposition tools described here), CUBIT has a webcut capability specially tuned for decomposition. It is also useful to split periodic surfaces to facilitate quad and hex meshing.

- Web Cutting
- Splitting Geometry
- Section Command
- Separating Multi-Volume Bodies
- Separating Surfaces From Bodies

Web Cutting with an Arbitrary Surface

An arbitrary "sheet" surface can also be used to web cut a body. This sheet need not be planar, and can be bounded or infinite. The following commands are used:

```
Webcut {blank} with sheet {body|surface} <id>
[webcut_options]

Webcut {blank} with sheet extended [from]
surface <id> [webcut_options]
```

In its first form, the command uses a sheet body, either one that is pre-existing or one formed from a specified surface. Note that in this latter case the (bounded) surface should completely cut the body into two pieces. Sheet bodies can be formed from a single surface, but can also be the combination of many surfaces; this form of web cut can be used with quite complicated cutting surfaces.

Extended sheet surfaces can also be used; in this case, the specified surface will be extended in all directions possible. Note that some spline surfaces are limited in extent, and so these surfaces may or may not completely cut the blank.

Chop Command

The **chop** command works similarly to a web cut command, but is faster. Given two bodies, the command will find the intersection of the two bodies, and divide the main body into a body that lies outside the intersection, and a body that lies inside the intersection. The tool body will be deleted, unless the **keep** option is specified. The syntax of the command is:

```
Chop [Volume|BODY] <id> with [Volume|BODY]
<id> [keep] [nonreg]
```

The **nonreg** option results in the bodies being non-regularized.

Web Cutting with a Planar or Cylindrical Surface

The commands used to **web cut** with a planar or cylindrical surface in CUBIT are:

- Coordinate Plane
- Planar Surface
- Plane from 3 Points
- Plane Normal to Curve
- General Plane Specification
- Cylindrical Surface
- Cone Surface

Coordinate Plane

In the command's simplest form, a coordinate plane can be used to cut the model, and can optionally be offset a positive or negative distance from its position at the origin.

```
Webcut {Volume|Body|Group} <id_range> [With]
Plane {xplane|yplane|zplane} [Offset <val>]
[rotate <theta> about x|y|z <xval> <yval>
<zval> [center <xval> <yval> <zval>]]
webcut_options
```

The cutting plane can be rotated about a user-specified axis using the **rotate** option. The center of rotation can be moved by using the **center** option.

Planar Surface

An existing planar surface can also be used to cut the model; in this case, the surface is identified by its ID as the cutting tool.

```
Webcut {Volume|Body|Group} <id_range> [With]
Plane Surface <surface_id> webcut_options
```

Plane from 3 Points

Any arbitrary planar surface can be used by specifying three vertices that define the plane, and can optionally be offset a positive or negative distance from this plane.

```
Webcut {Volume|Body|Group} <id_range> [With]
Plane Vertex <vertex_1> [Vertex] <vertex_2>
[Vertex] <vertex_3> [Offset <value>]
webcut_options
```

The plane to be used for the web cut can be previewed with the preview option in the general webcut options.

Plane Normal to Curve

The next command allows a user to specify an infinite cutting plane by specifying a location on a curve. The cutting plane is created such that it is normal to the curve tangent at the specified location.

```
Webcut {Volume|Body|Group} <id_range> [With]
Plane Normal To Curve <curve_id>
{Position <xval><yval><zval> | Close_To Vertex
<vertex_id>} webcut_options
```

```
Webcut {Volume|Body|Group} <id_range> [With]
Plane Normal To Curve <curve_id>
{Fraction <f> | Distance <d>} [[From] Vertex
<vertex_id>] webcut_options
```

The position on the curve can be specified as:

1. A fraction along the curve from the start of the curve, or optionally, from a specified vertex on the curve.
2. A distance along the curve from the start of the curve, or optionally, from a specified vertex on the curve.
3. An xyz position that is moved to the closest point on the given curve.
4. The position of a vertex that is moved to the closest point on the given curve.

The point on the curve can be previewed with the Draw Location On Curve command and the plane to be used for the web cut can be previewed with the preview option in the general webcut options.

General Plane Specification

A webcut plane can be defined using the general plane specification options in the Specifying a Plane section of the documentation.

```
Webcut {Volume|Body|Group} <id_range> [With]
General Plane {options} webcut_options
```

Cylindrical Surface

Finally, a semi-infinite cylindrical surface can be used by specifying the cylinder radius, and the cylinder axis. The axis is specified as a line corresponding to a coordinate axis, the normal to a specified surface, two arbitrary points, or an arbitrary point and the origin. The "center" point through which the cylinder axis passes can also be specified.

```
Webcut {Volume|Body|Group} <range> [With]
Cylinder Radius <val> Axis {x|y|z|normal of
surface <id>| vertex <id_1> vertex <id_2>|
<x_val> <y_val> <z_val>>} [center <x_val>
<y_val> <z_val>] webcut_options
```

Cone Surface

A semi-infinite cone surface can be used by specifying the cone outer radius, and the cone inner radius. The axis is specified as a location first of where the outer radius is applied and the second location of where the inner radius is applied.

```
Webcut {Volume|Body|Group} <ids> [With] cone
radius <val> <val> location {options} location
{options} [Imprint] [Merge] [group_results]
[preview]
```

Web Cutting by Sweeping Curves or Surfaces

Webcutting with sweeping creates a swept tool body in the same step as the web cut operation. There are 4 general ways to **web cut** with sweeping:

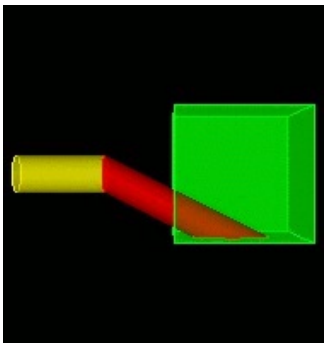
- Web Cutting by Sweeping a Surface Along a Trajectory
- Web Cutting by Sweeping a Surface About an Axis
- Web Cutting by Sweeping a Curve(s) Along a Trajectory
- Web Cutting by Sweeping a Curve(s) About an Axis

Web Cutting by Sweeping a Surface Along a Trajectory

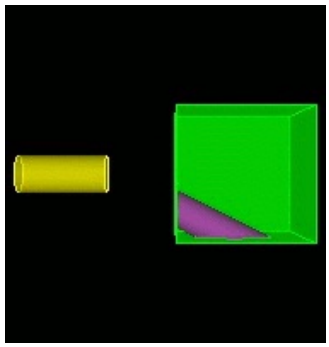
This command allows one or more surfaces to be swept, creating a volume that is used for the web cut. If more than one surface is specified, the surfaces must contain coincident curves. The surfaces are swept along a direction and some distance or perpendicular and some distance or along a curve. For best results the curve to sweep the surface along should intersect one of the surfaces. The **through_all** option will sweep the surfaces along the trajectory far enough so as to intersect all input bodies. The **stop surface <id>** option is used to identify a surface at which the sweep will stop. If using this option when sweeping along a curve, the sweep will stop at the first place possible. The **up_to_next** option indicates that the user wants to web cut with only the first water tight volume that forms as a result of the intersection between sweep and union of all blank bodies. The **[Outward|Inward]** options specify a sweeping direction that is either INTO the volume or OUT from the volume.

```
Webcut {Volume|Body|Group} <range> Sweep
Surface <id_range> {Vector <x> <y> <z>
[Distance <distance>] | Along Curve <id>}
[Through_all | Stop Surface <id> | Up_to_next
] [webcut_options]

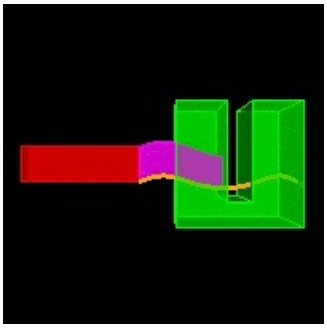
Webcut {Volume|Body|Group} <id> Sweep Surface
<id_range> Perpendicular {Distance <distance>
| Through_all | Stop Surface <id>}
[OUTWARD|Inward] [webcut_options]
```



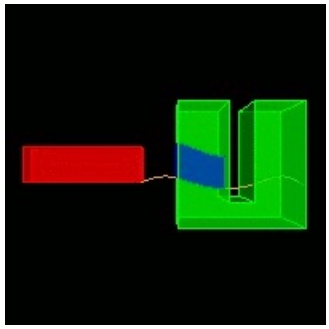
sweeping a surface in a direction



resultant web cut



along a curve to a stop surface



resultant web cut

Figure 1. Examples of web cutting with swept surfaces

Web Cutting by Sweeping a Surface About an Axis

This command allows a one or more surfaces to be swept, creating a volume that is used for the web cut. If more than one surface is specified, the surfaces must contain coincident curves. The surface is swept about a user-defined axis or about one of the x y z coordinate axes and a specified angle. The **stop surface <id>** option is used to identify a surface at which the sweep will stop. The **up_to_next** option indicates that the user wants to web cut with only the first water tight volume that forms as a result of the intersection between sweep and union of all blank bodies. For these 2 options to work correctly the user must specify an angle large enough for the rotation to traverse the **stop surface** or the **up_to_next** surface.

```
Webcut {Volume|Body|Group} <id> Sweep Surface
<id_range> {Axis <xpoint ypoint zpoint xvector
yvector zvector> | Xaxis | Yaxis | Zaxis }
Angle <degrees> [Stop Surface <id> |
Up_to_next] [webcut_options]
```

Web Cutting by Sweeping a Curve(s) Along a Trajectory

This command allows a curve(s) to be swept, creating a surface that is used for the web cut. If multiple curves are specified, they must share vertices and form a continuous path. The curve(s) is swept along a direction and some distance or along another curve. If sweeping a curve(s) along another curve, for best results the curve(s)-to-swept and the curve to sweep along should intersect at some point. The **stop surface <id>** option is used to identify a surface at which the sweep will stop. If using this option when sweeping along a curve, the sweep will stop at the first place possible. The **through_all** option will sweep the curve(s) along the trajectory far enough so as to intersect all input bodies. For the web cut to be successful, the swept curve(s) must completely traverse a portion of a blank body(s), cutting off a complete piece of the blank body(s). Option **through_all** should not be used when defining the web cut with a vector and a distance or along a curve.

```
Webcut {Volume|Body|Group} <id> Sweep Curve
<id_range> {Vector <x> <y> <z> [Distance
<distance>| Along curve <id>] } [Through_all |
Stop Surface <id>] [webcut_options]
```

Web Cutting by Sweeping a Curve(s) About an Axis

This command allows a curve to be swept, creating a surface that is used for the web cut. If multiple curves are specified, they must share vertices and form a continuous path. The curve(s) is swept about a user-defined axis or about one of the x y z coordinate axes and a specified angle. For the web cut to be successful, the swept curve(s) must completely traverse a portion of a blank body(s), cutting off a complete piece of the blank body(s). The **stop surface <id>** option is used to identify a surface at which the sweep will stop. For this option to work correctly the user must specify an angle large enough for the rotation to traverse the **stop surface**.

```
Webcut {Volume|Body|Group} <id> Sweep Curve
<id_range> {Axis <xpoint ypoint zpoint xvector
yvector zvector> | Xaxis | Yaxis | Zaxis }
Angle <degrees> [Stop Surface <id>]
[webcut_options]
```

Web Cutting using a Tool or Sheet Body

Any existing body in the geometric model can be used to cut other bodies; the command to do this is:

```
Webcut {blank} tool [body] <id>
[webcut_options]
```

This simply uses the specified tool body in a set of boolean operations to split the blank into two or more pieces.

Another form of the command cuts the body list with a temporary sheet body formed from the curve loop. This is the same sheet as would be created from the command Create Surface Curve <id_list>.

```
Webcut {Body|Group} <id_range> [With] Loop
[Curve] <id_range> NOIMPRINT|Imprint]
[NOMERGE|Merge] [group_results]

Webcut {Volume|Body|Group} <id_range> [With]
Bounding Box {Body|Volume|Surface|Curve|Vertex
<id_range>} [Tight] [[Extended]
{Percentage|Absolute} <val>] [{X|Width} <val>]
[{Y|Height} <val>] [{Z|Depth} <val>]]
NOIMPRINT|Imprint] [NOMERGE|Merge]
[group_results]
```

The final form of this command cuts a body with the bounding box of another entity. This bounding box may be tight or extended.

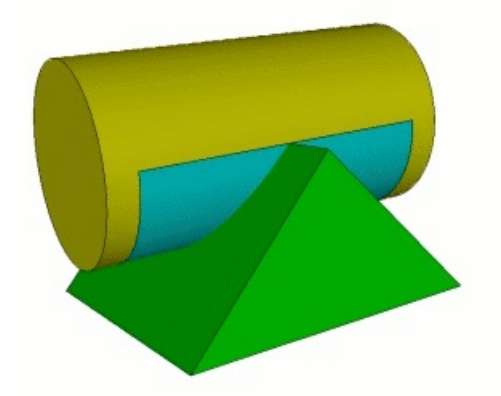


Figure 1. Cylinder cut with bounding box of prism.

Web Cutting

The term "web cutting" refers to the act of cutting an existing body or bodies, referred to as the "blank", into two or more pieces through the use of some form of cutting tool, or "tool". The two primary types of cutting tools available in CUBIT are surfaces (either pre-existing surfaces in the model or infinite or semi-infinite surfaces defined for web cutting), or pre-existing bodies.

The various forms of the web cut command can be classified by the type of tool used for cutting. These forms are described below, starting with the simplest type of tool and progressing to more complex types.

- Web Cutting Using the Chop Command
- Web Cutting Using Planar or Cylindrical Surface
- Web Cutting with Arbitrary Surface
- Web Cutting Using Tool or Sheet Body
- Web Cutting by Sweeping Curves or Surfaces
- Web Cutting Options

General Notes

The primary purpose of web cutting is to make an existing model meshable with the hex meshing algorithms available in CUBIT. While web cutting can also be used to build the initial geometric model, the implementation and command interface to web cutting have been designed to serve its primary purpose. Several important things to remember about web cutting are as follows:

- The geometric model should be checked for integrity (using imprinting and merging) before starting the decomposition process. This makes the checking process easier, since there are fewer bodies and surfaces to check. Once the model passes that initial integrity check, it is rare that decompositions using web cut will result in a model that does not also pass the same checks.
- The use of the Imprint option can in cases save execution time, since it limits the scope of the imprint operations and thereby works faster. The alternative is performing and Imprint All on the pieces of the model after all decompositions have been completed; this operation has been made much faster in more current releases of CUBIT, but will still take a noticeable amount of time for complicated models.
- While the web cut commands make it very simple to cut your model into very many pieces, we recommend that the user restrict the decomposition they perform to only that necessary for meshability or for obtaining an acceptable mesh. Having more volumes in the model may simplify individual volumes, but may not always result in a higher quality mesh; it will always increase the run time and complexity of the meshing task.
- When the web cut command is executed the associated geometry will be regularized. This behavior can be changed, see geometry booleans.
- Web cutting volumes will automatically separate parent bodies as well. This behavior can also be changed, see Separating Multi-Volume Bodies.
- If a geometric entity got split after the webcut operation, then the notesets/sidesets/blocks applied on that initial geometric entity will be carried over to the split entities.

The Decomposition Tutorials and the Power Tools Tutorial contain some examples that demonstrate the use of web cutting operations.

Web Cutting Options

The following options can be used with all web cut commands:

[NOIMPRINT|Imprint [include_neighbors]]: In its default implementation, web cutting results in the pieces not being imprinted on one another; this option forces the code to imprint the pieces after web cutting. The include_neighbors option will also imprint adjacent bodies.

[NOMERGE|Merge]: By default, the pieces resulting from an imprint are manifold; specifying this option results in a merge check for all surfaces in the pieces resulting from the web cut.

[Group_results]: The various pieces resulting from the previous command are placed into a group named `webcut_group'.

[Preview]: This option will preview the web cutting plane without executing the command.

Split Curve

The Split Curve command will split a curve without the need for geometry creation (unlike imprinting). The syntax is shown below.

```
Split Curve <id> [location on curve options]
[Merge] [Preview]
```

To split a curve, simply specify a location or a location on curve (see location specification). Using the **Preview** keyword will draw the splitting location on the curve. The **Merge** keyword will merge any topology that contains the newly created vertex.

Split Periodic Surfaces

Solids which contain periodic surfaces include cylinders, torii and spheres. Splitting periodic surfaces can in some cases simplify meshing, and will result in curves and surfaces being added to the volume. The command used to split periodic surfaces is:

```
Split Periodic Body <id_range|all>
```

This command splits all periodic surfaces in a body or bodies.

Split Surface

The Split Surface command divides one or more surfaces into multiple surfaces. The command results are similar to imprint with curve. However, curve creation is not necessary for splitting surfaces. Three primary forms of the command are available.

- Split Across
- Split Extend
- Split (Automatically)
- Split Skew

The first form splits a single surface using locations while the second splits by extending a surface hard-line until it hits a surface boundary. The split automatic splits either a single surface or a chain of surfaces in an automatic fashion.

Split Across

Two forms of Split Across are available:

```
Split Surface <id> Across [Pair] Location
<options multiple locs> [Preview [Create]]

Split Surface <id> Across Location <multiple
locs> Onto Curve <id> [Preview] Create]]
```

This command splits a surface with a spline projection through multiple locations on the surface. See Location, Direction, and Axis Specification for a detailed description of the location specifier. Figure 1 shows a simple example of splitting a single surface into two surfaces. A temporary spline was created through the three specified locations (Vertex 5 6 7), and this curve was used to split the surface.

```
split surface 1 across location vertex 5 6 7
```

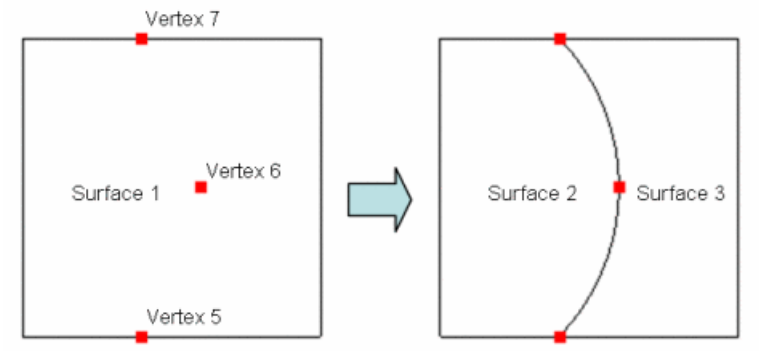


Figure 1 - Splitting Across with Multiple Locations

The **Pair** keyword will pair locations to create multiple surface splitting curves (each defined with two locations). An even number of input locations is required. Figure 2 shows an example:

```
split surface 1 across pair vertex 5 7 6 8
```

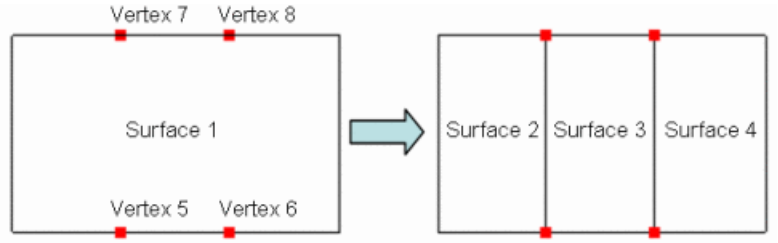


Figure 2 - Splitting Across with Pair Option

The **Preview** keyword will show a graphics preview of the splitting curve. If the **Create** keyword is also specified, a free curve (or curves) will be created - these are the internal curves that are used to imprint the surfaces.

The **Onto Curve** format of the command takes one or more locations on one side of the surface and projects them onto a single curve on the other side of the surface. Figure 3 shows an example:

```
split surface 1 across vertex 5 6 onto curve 4
```

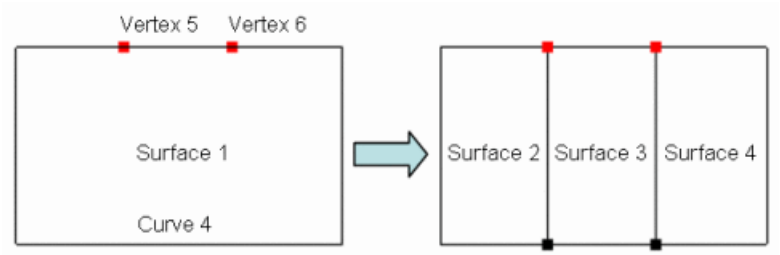


Figure 3 - Splitting Across with Onto Curve

Split Extend

The Split Extend function can be called with the following command:

```
Split Surface <id_list> Extend [Vertex
<id_list> | AUTO] [Preview [Create]]
```

With the following settings:

```
Set Split Surface Extend Normal {on|OFF}

Set Split Surface Extend Gap Threshold <val>

Set Split Surface Extend Tolerance<val>
```

This command splits a surface by extending a surface hard-line until it hits a surface boundary. Figure 4 shows a simple example of extending a curve. The hard-line curve was extended from the specified vertex until it hit the surface boundary.

```
split surface 1 extend vertex 2
```

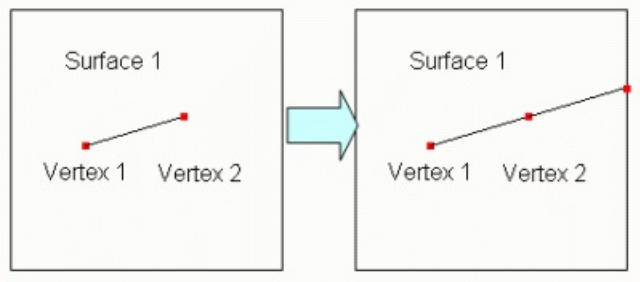


Figure 4 - Splitting by Extending Hard-line

The **auto** keyword will search for all hard-lines and extend them according to the Split Surface Extend settings. Figure 5 shows an example:

```
split surface 1 extend auto
```

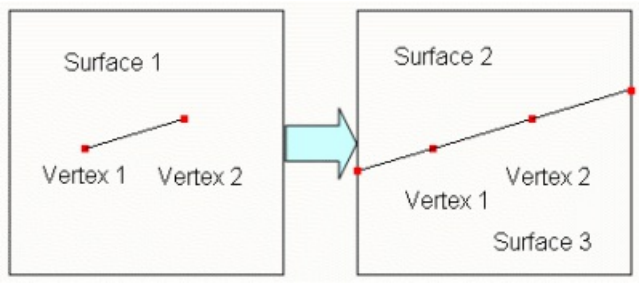


Figure 5 - Splitting by Extending with Auto Option

The **preview** keyword will show a graphics preview of the splitting curve. If the **create** keyword is also specified, a free curve (or curves) will be created - these are the internal curves that are used to imprint the surfaces.

The **normal** setting can be turned on or off. When it is on, Cubit will attempt to extend the hard-line so that it is normal to the curve it will intersect. An example of this is in Figure 6:

```
set split surface normal on
split surface 1 extend vertex 2
```

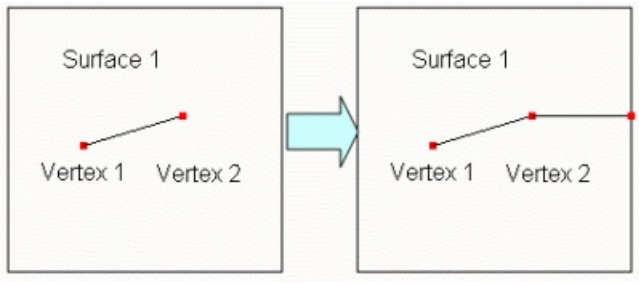


Figure 6 - Splitting by Extending a Hard Line with Normal Setting ON

Cubit uses the **gap threshold** to decide whether or not to extend a hard-line when the user specifies *auto*. If the distance between a vertex on a hard-line and the curve it will hit is greater than the gap threshold, then Cubit will not extend that hard-line. The default value is INFINITY, and can be set to any value. To reset the value back to INFINITY, set the gap threshold to -1.0. **Note: This setting only applies when using the keyword *auto***. An example of using the gap threshold is shown in Figure 7:

```
set split surface gap threshold 2.0
split surface 1 extend auto
```

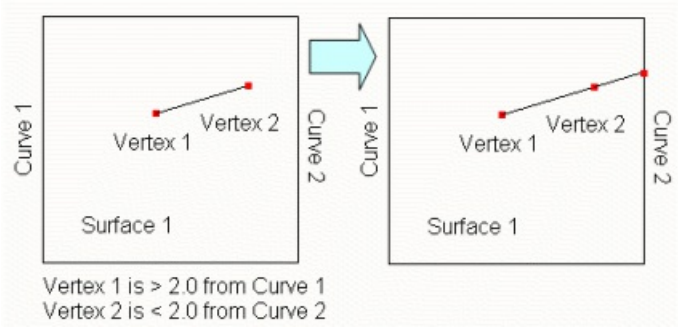


Figure 7 - Extending Hard-lines with Gap Threshold = 2.0. (Notice Vertex 1 was not extended because it exceeded the gap threshold)

The **tolerance** setting can be used to avoid creating short curves on the surface boundary. If Cubit tries to extend a hard-line that comes within

tolerance of a vertex, it will instead snap the extension to the existing vertex. An example of this is shown in Figure 8:

```
set split surface tolerance 1.0
split surface 1 extend vertex 2
```

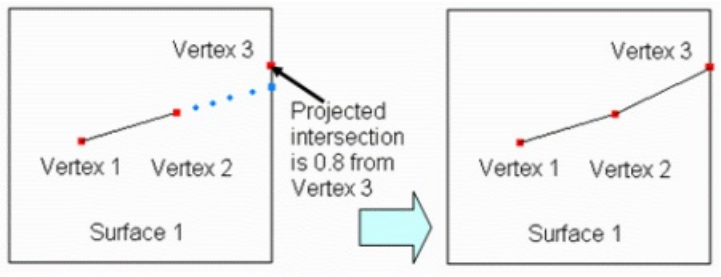


Figure 8 - Extending Hard-lines with Tolerance (Notice the extension snapped to Vertex 3)

Split (Automatically)

This form of the command splits a single surface or a chain of surfaces in an automatic fashion. It is most convenient for splitting a fillet or set of fillets down the middle - oftentimes necessary to prepare for mesh sweeping. These surfaces cannot have multiple curve loops.

```
Split Surface <id_list> [Corner Vertex
<id_list>] [Direction Curve <id>]
[Segment|Fraction|Distance <val> [From Curve
<id>]] [Through Vertex <id_list>] [Parametric
<on|OFF>] [Tolerance <val>] [Preview [Create]]
```

- Logical Rectangle
- Split Orientation
- Corner Vertex <id_list>
- Direction Curve <id>
- Segment|Fraction|Distance <val> [From Curve <id>]
- Through Vertex <id_list>
- Parametric <on|OFF>
- Tolerance <val>
- Preview [Create]
- Settings (Tolerance, Parametric, Triangle)

The volume shown in Figure 9 was quickly prepared for sweeping by splitting the fillets and specifying sweep sources as shown (with the sweep target underneath the volume). The surface splits are shown in blue.

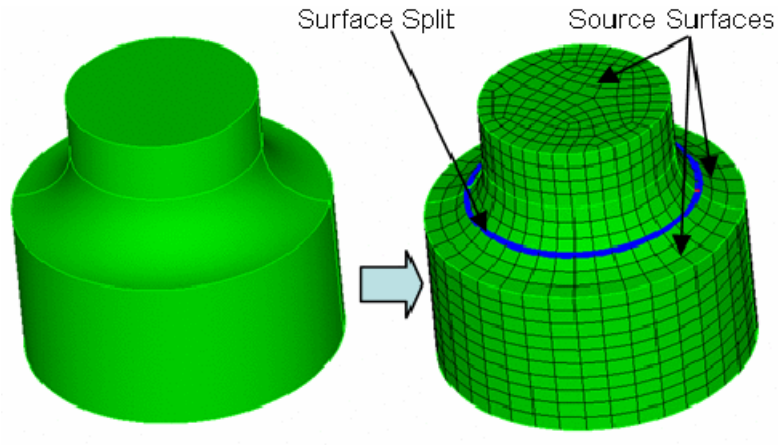


Figure 9 - Splitting Fillets to Facilitate Sweeping

Each surface is always split with a *single* curve along the length of the

surface (or multiple single curves if the Segment option is used). The splitting curve will either be a spline, arc or straight line.

Logical Rectangle

The Split Surface command analyzes the selected surface or surface chain to find a *logical rectangle*, containing four logical sides and four logical corners; each side can be composed of zero, one or multiple curves. If a single surface is selected (with no options), the logical corners will be those closest to 90 and oriented such that the surface will be ***split parallel to the longest aspect ratio*** of the surface. If a chain of surfaces is selected, the logical corners will include the two corners closest to 90 on the starting surface of the chain and the two corners closest to 90 on the ending surface of the chain (***the split will always occur along the chain***).

In Figure 10, the logical corners selected by the algorithm are Vertices 1-2-5-6. Between these corner vertices the logical sides are defined; these sides are described in Table 1. The default split occurs from the center of Side 1 to the center of Side 3 (parallel to the longest aspect ratio of the surface), and is shown in blue.

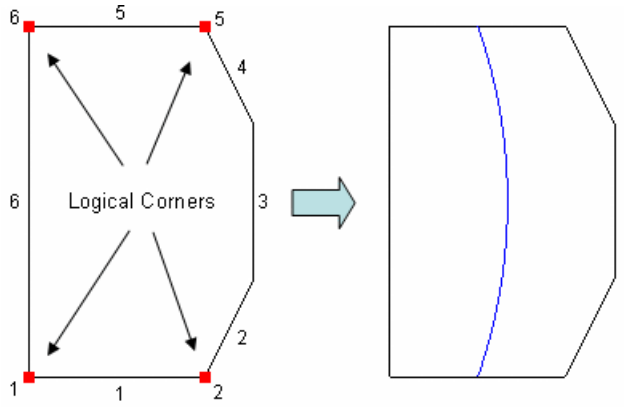


Figure 10 - Split Surface Logical Properties

Table 1. Listing of Logical Sides for Figure 10

Logical Side	Corner Vertices	Curve Groups
1	1-2	1
2	2-5	2,3,4
3	5-6	5
4	6-1	6

Figure 11 shows a surface along with 2 possibilities for its logical rectangle and the resultant splits.

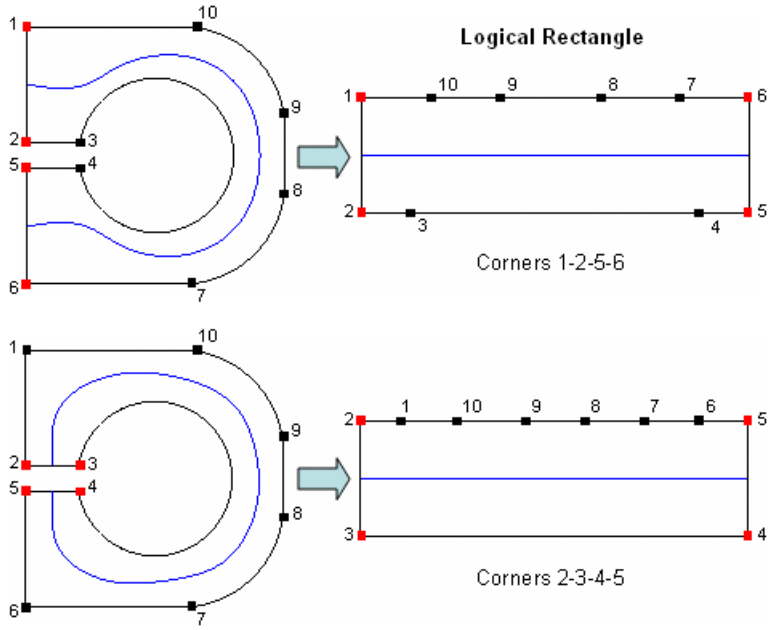
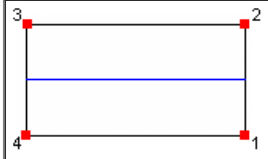
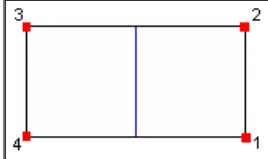
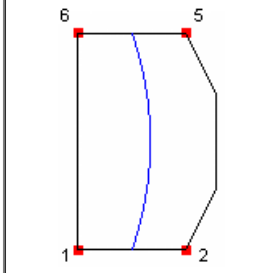
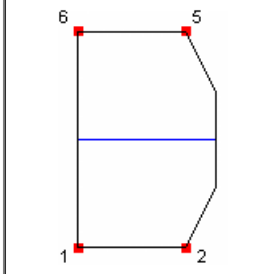
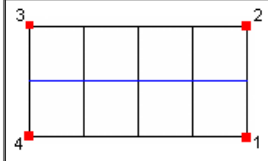
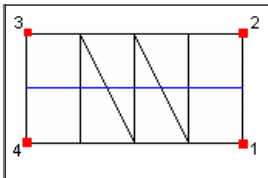
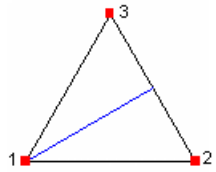
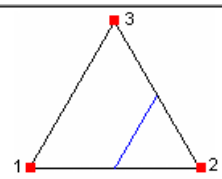
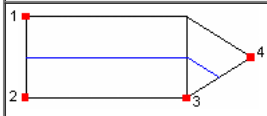
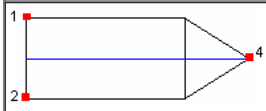
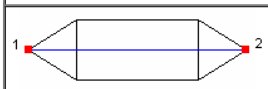
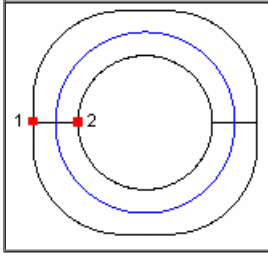


Figure 11 - Different Possible Logical Rectangles for Same Surface

Table 2 shows various surfaces and the resultant split based on the automatically detected or selected logical rectangle. Note that surfaces are always traversed in a counterclockwise direction.

Table 2 - Sample Surfaces and Logical Rectangles

Surface(s) (Resultant Split in Blue)	Ordered Corners (to form the *Logical Rectangle*)
	1-2-3-4 (using aspect ratio)
	4-1-2-3 (user selected)
	1-2-5-6
	2-5-6-1
	1-2-3-4 (split is always along the chain)

	1-2-3-4 (notice triangular surfaces along the chain)
	1-1-2-3 (note side 1 of the logical rectangle is collapsed; side 3 is from vertex 2 to 3)
	1-2-2-3 (note side 2 of the logical rectangle is collapsed)
	1-2-3-4
	1-2-4-4
	1-1-2-2
	1-1-2-2 (selected automatically)

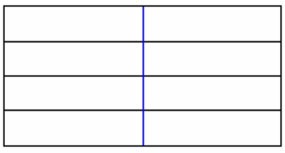
Split Orientation

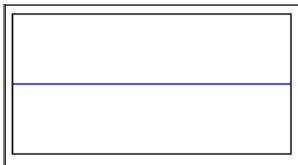
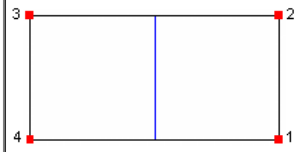
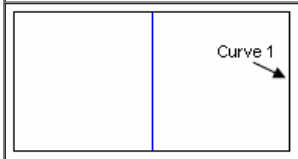
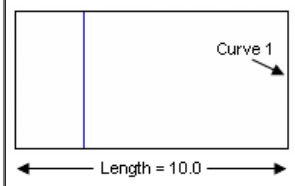
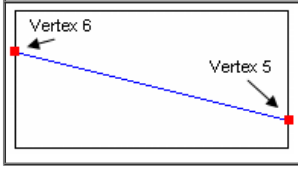
If a chain of surfaces are split, the surfaces will always be split along the chain. The command will not allow disconnected surfaces.

For a single surface, the split direction logic is a bit more complicated. If no options are specified, the surface aspect ratio determines the split direction - the surface will be split parallel to the longest aspect ratio side through the midpoint of each curve. This behavior can be overridden by the order the Corner vertices are selected (the split always starts on the side between the first two corners selected), the Direction option, the From Curve option, or the Through Vertex list.

Table 3 shows examples of the various split orientation methods. These options are explained in more detail in the sections below.

Table 3 - Split Orientation Methods

Surface Example	Split Orientation Method
	Multiple surfaces are *always* split along the chain

	Parallel to longest surface aspect ratio (default)
	Corner Vertex 4 1 2 3 (split always starts on side 1 of the logical rectangle)
	Direction Curve 1
	From Curve 1 Fraction .75 or From Curve 1 Distance 7.5
	Through Vertex 5 6

Corner Specification

The **Corner** option allows you to specify corners that form *logical rectangle* the algorithm uses to orient the split on the surface. When analyzing a surface to be split, the software automatically selects the corners that are closest to 90. The Preview option displays the automatically selected corners in red. Sometimes incorrect corners are chosen, so you must specify the desired corners yourself. The split always starts on the side between the first two corners selected and finishes on the side between the last two corners selected. Figure 12 shows a situation where the user had to select corners to get the desired split.

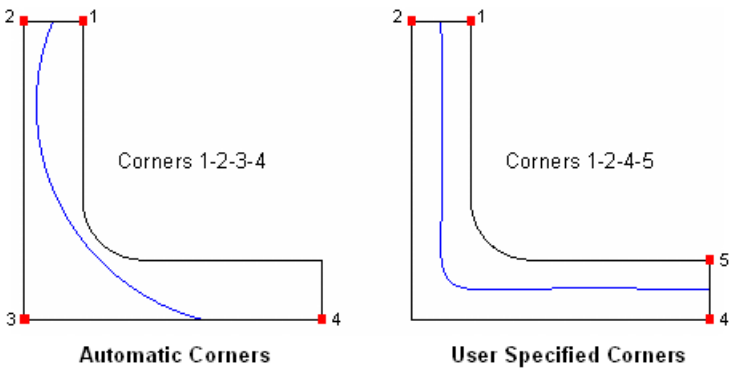
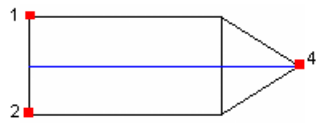
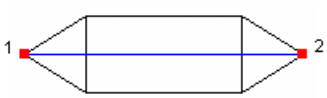


Figure 12 - Selecting the Desired Corners

The split can be directed to the tip of a triangular shaped surface by selecting that corner vertex twice (at the start or end of the corner list) when specifying corners, creating a zero-length side on the logical rectangle. A shortcut exists whereas if you specify only 3 corner vertices, the zero-length side will be directed to the first corner selected. If you

specify only 2 corner vertices, a zero-length side will be directed to both the first and second corner you select. Table 4 shows these examples. Note the software will automatically detect triangle corners based on angle criteria - the corner selection methods for zero-length sides explained in this section need only be applied if the angles are outside of the thresholds specified in the Set Split Surface Auto Detect Triangle settings.

Table 4 - Selecting Corners to Split to Triangle Tips

Surface	Corner Specification
	1-2-4-4- or 4-4-1-2 or 4-1-2 (shortcut method)
	1-1-2-2 or 2-2-1-1 or 1-2 or 2-1 (shortcut method)

Direction

The **Direction** option allows you to conveniently override the default split direction on a single surface. Simply specify a curve from the logical rectangle that is parallel to the desired split direction. If Corners are also specified, the Direction option will override the split orientation that would result from the specified corner order. The Direction option is not valid on a chain of surfaces. Figure 13 shows an example.

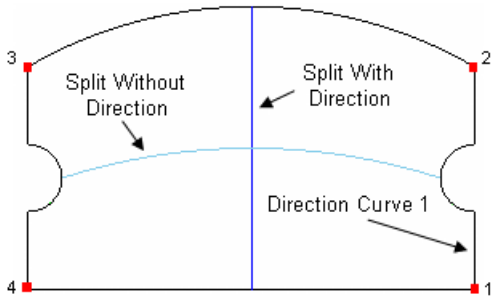
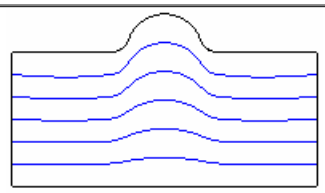


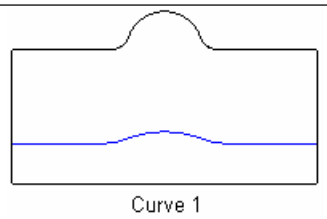
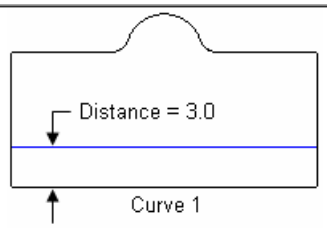
Figure 13 - Direction Specification Overrides Corner Order

Segment|Fraction|Distance

The **Segment** option allows you to split a surface into 2 or more segments that are equally spaced across the surface. The **Fraction** option allows you to override the default 0.5 fractional split location. The **Distance** option allows you to specify the split location as an absolute distance rather than a fraction. By specifying a **From Curve**, you can indicate which side of the logical rectangle to base the segment, fraction or distance from (versus a random result). Table 5 gives examples of these options.

Table 5 - Segment, Fraction, Distance Examples

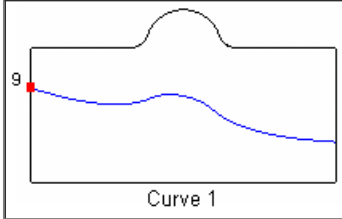
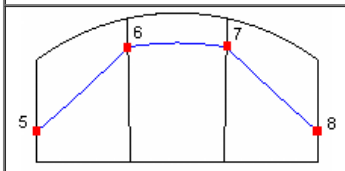
Surface	Command Options
	Segment 6 From Curve 1

	Fraction .3 From Curve 1
	Distance 3 From Curve 1

Through Vertex

The **Through Vertex** option forces the split through vertices on surface boundaries perpendicular to the split direction. Use this option if the desired fraction is not constant from one end of the surface to another or if a split would otherwise pass very close to an existing curve end resulting in a short curve. Through vertices can be used in conjunction with the Fraction option - the split will linearly adjust to pass exactly through the specified vertices. It is not valid with the Segment option. The maximum number of Through Vertices that can be specified is equal to the number of surfaces being split plus one. The selected vertices can be free, but must lie on the perpendicular curves. Table 6 gives several examples.

Table 6 - Through Vertex Examples

Surface(s)	Command Options
	Fraction .3 From Curve 1 Through Vertex 9
	Through Vertex 5 6 7 8

Parametric

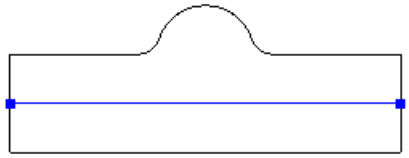
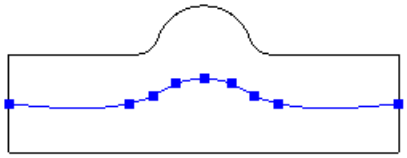
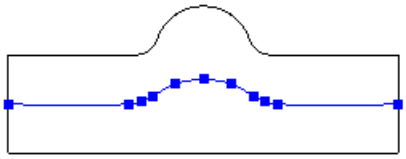
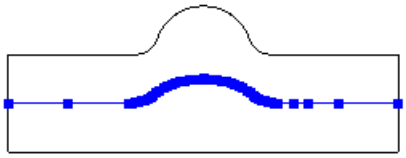
By default, split locations are calculated in 3D space and projected to the surface. As an alternative, split locations can be calculated directly in the surface parametric space. In rare instances, this can result in a smoother or more desirable split. The command option **Parametric {on|Off}** can be used to split the given surfaces in parametric space. Alternatively, the default can be overridden with the Set Split Surface Parametric {on|OFF} command.

Tolerance

A single absolute tolerance value is used to determine the accuracy of the split curves. A smaller tolerance will force more points to be interpolated. The tolerance is also used when detecting an analytical curve (e.g., an arc or straight line) versus a spline. A looser tolerance will result in more analytical curves. The default tolerance is 1.0. The command option **Tolerance <val>** can be used to split the given surfaces using the given tolerance. Alternatively, the default tolerance can be overridden with the Set Split Surface Tolerance <val> command.

It is recommended to use the largest tolerance possible to increase the number of analytical curves and reduce the number of points on splines, resulting in better performance and smaller file sizes. The Preview option displays the interpolated curve points. Table 7 shows the effect of the tolerance for a simple example.

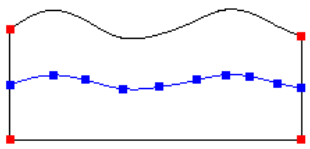
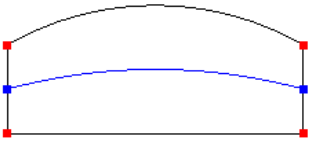
Table 7 - Effect of Tolerance on Split Curve

Surface	Tolerance
	2.0
	1.0
	0.5
	0.01

Preview

The **Preview** keyword will show a graphics preview (in blue) of the splitting curve (or curves) and the corner vertices (in red) selected for the logical rectangle. The curve preview includes the interpolated point locations that define spline curves. Note that if no points are shown on the interior of the curve, it means that the curve is an analytical curve (line or arc). If the **Create** keyword is also specified, a free curve (or curves) will be created - these are the internal curves that are used to imprint the surfaces. Table 8 shows some examples.

Table 8 - Graphics Preview

Surface	Curve Type
	Spline
	Arc (no preview points shown on interior of curve)

Settings

This section describes the settings that are available for the automatic split surface command. To see the current values, you can enter the command **Set Split Surface**, optionally followed by the setting of interest (without specifying a value).

```
Set Split Surface Tolerance <val>
```

This sets the default tolerance for the accuracy of the split curves. See the Tolerance section for more information.

```
Set Split Surface Parametric {on|OFF}
```

This sets the default for whether surfaces are split in 3D (default) or in parametric space. See the Parametric section for more information.

```
Set Split Surface Auto Detect Triangle
{ON|off}

Set Split Surface Point Angle Threshold <val>

Set Split Surface Side Angle Threshold <val>
```

The split surface command automatically detects triangular shaped surfaces as explained in the section on Corners. This behavior can be turned off with the setting above. Two thresholds are used when detecting triangles - the **Point Angle** threshold and the **Side Angle** threshold, specified in degrees. Corners with an angle below the Point Angle threshold are considered for the tip of a triangle (or the collapsed side of the logical rectangle). Corners within the Side Angle threshold of 180 are considered for removal from the logical rectangle. In order for a triangle to actually be detected, corners for both the point and side criteria must be met. The default Point Angle threshold is 45, and the default Side Angle threshold is 27. Figure 14 provides an illustration.

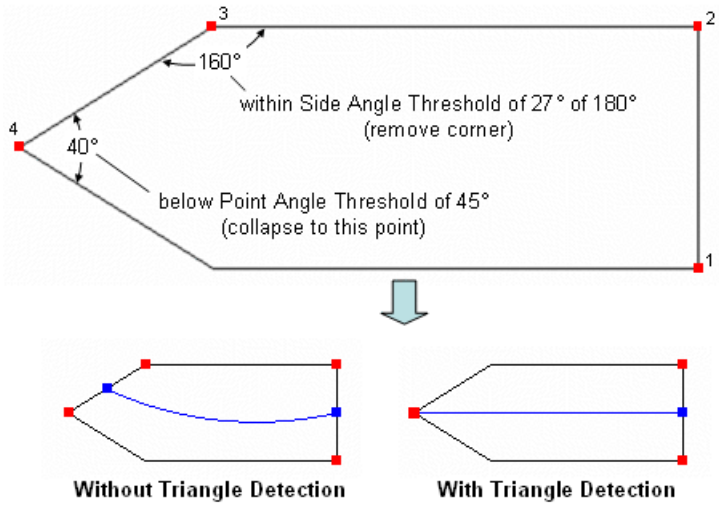


Figure 14 - Triangle Detection Settings

Split Skew

The Split Skew function can be called with the following command:

```
Split Surface <id_list> Skew [Preview]
[Create]
```

This command will split a surface or list of surfaces in a logical way to reduce the amount of skew in a quadrilateral mesh. This function uses the control skew algorithm to determine where to make these logical splits. Users should note that Split Skew can only be utilized effectively on surfaces that lend themselves to a structured meshing scheme. These surfaces cannot have multiple curve loops. Figure 15 shows a simple example of a surface being split.

```
split surface 1 skew
```

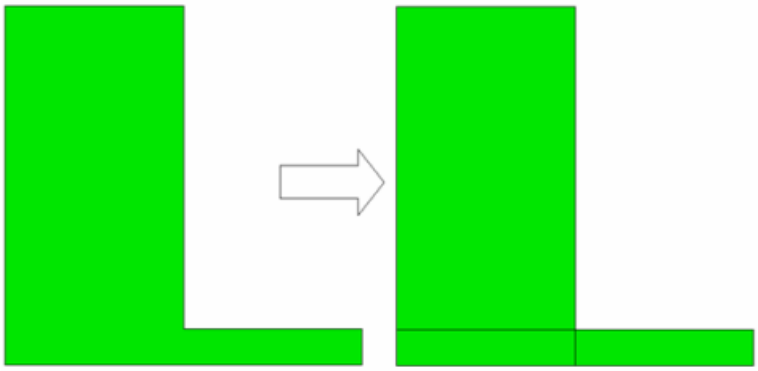


Figure 15. Split Skew applied to an L-shaped surface

The **Preview** keyword will show a graphics preview of the splitting curves. If the **Create** keyword is also specified, free curves will be created.

Splitting Geometry

The Split command divides curves or surfaces into multiple entities. The command results are similar to imprinting. However, vertex and/or curve creation is not necessary for the split command.

- Split Curve
- Split Surface
- Split Periodic Surfaces

Section Command

This command will cut a body or group of bodies with a plane, keeping geometry on one side of the plane and discarding the rest. The syntax for this command is:

```
Section {Body|Group} <id_range> [With]
{Xplane|Yplane|Zplane} [Offset <value>]
[NORMAL|Reverse] [Keep]

Section {Body|Group} <id_range> With Surface
<id> [NORMAL|Reverse] [Keep]
```

In the first form, the specified coordinate plane is used to cut the specified bodies. The offset option is used to specify an offset from the coordinate plane. In the second form, an existing (planar) surface is used to section the model. In either case, the reverse keyword results in discarding the positive side of the specified plane or surface instead of the other side. The keep option results in keeping both sides; the section command used with this option is equivalent to webcutting with a plane.

Separating Surfaces from Bodies

The separate surface command is used to separate a surface from a sheet body or a solid body. The command is:

```
Separate Surface <range>
```

Separating a surface from a solid body will create a "hole" in the solid body. Thus the solid body will become a sheet body. The newly separated surface will be also sheet body, but it will have a different id. Multiple surfaces can be separated from a body at the same time, but each separated surface will result in a distinct sheet body, as if the command had been performed on each surface individually.

Separating Multi-Volume Bodies

The separate and split commands are used to separate a body with multiple volumes into a multiple bodies with single volumes. The commands are:

```
Separate {Body|Volume} <id_range|all>
```

and

```
Split {Body|Volume} <id_range|all>
```

Only very rarely will either of these commands be needed. They are provided for the occasional instance that a multi-volume body is found. These commands are interchangeable.

Another related command allows the user to control the separation of bodies after webcutting. In most instances the user will want to separate bodies after webcutting. One reason to possibly have this option turned off is to be able to keep track of all the volumes during a webcut. Setting this option to "off" keeps all volumes in the same body. But the more common approach is to name the original body and allow naming to keep track of volumes. This setting is on by default. The syntax is:

```
Set Separate After Webcut [ON|Off]
```

Geometry Cleanup and Defeaturing

Frequently, models imported from various CAD platforms either provide too much detail for mesh generation and analysis, or the geometric representation is deficient. These deficiencies can often be overcome with small changes to the model. Several tools are provided in CUBIT for this purpose.

The following describes the features available in CUBIT for clean up and defeaturing

- Healing
- Tweaking Geometry
- Removing Geometric Features
- Automatic Geometry Clean-up
- Regularizing Geometry
- Finding Surface Overlap
- Validating Geometry
- Debugging Geometry
- Geometry Accuracy
- Trimming and Extending Curves
- Stitching Sheet Bodies
- Blunting Tangencies (Removing small angles)
- Defeaturing Tool
- Reducing Geometry

Auto Healing

Healing is sometimes needed to fix models that have geometric problems. Geometry created in another modeling system and translated into an ACIS model may be imprecise due to the inherent limitations of the parent system, or due to limitations of data transfer through neutral file formats. This leads to problems such as gaps between entities and the absence of connectivity information (topology). Healing can fix these problems. The general steps to healing are:

- **Unhook** - All surfaces of the body are unhooked so they are stand-alone.
- **Stitch** - Connectivity information (Topology) my be absent (models may have free faces), or incomplete (solids may have open shells). Stitching rebuilds the topology of a model.
- **Simplify** - Models often contain curves and surfaces that appear analytic, but are actually represented as splines. Simplification of such geometry to analytics results in a reduction of data size and speed improvements in geometric operations.
- **Tighten Gaps** - The gap between boundary curves and the surface definition might be large. This operation attempts to reduce this gap using ACIS' tolerant geometry.

Autohealing makes these steps automatic with the following command:

```
Healer Autoheal Body <id_range> [Keep]
```

The **keep** option will retain the original body, putting the resulting healed body in a new body.

Healing

Healing is an optional module that detects and fixes ACIS models.

It is possible to create ACIS models that are not accurate enough for ACIS to process. This most often happens when geometry is created in some other modeling system and translated into an ACIS model. Such models may be imprecise due to the inherent numerical limitations of their parent systems, or due to limitations of data transfer through neutral file formats. This imprecision can also result when an ACIS model is created at a different tolerance from the current tolerance settings. This imprecision leads to problems such as geometric errors in entities, gaps between entities, and the absence of connectivity information (topology). Since ACIS is a high precision modeler, it expects all entities to satisfy stringent data integrity checks for the proper functioning of its algorithms. Therefore, if such imprecise models must be processed by an ACIS based system, "healing" of such models is necessary to establish the desired precision and accuracy.

The following sections describe how to use the Healing capability in ACIS and CUBIT to analyze and heal defective ACIS geometry.

- Validating/Analyzing Geometry
- Auto Healing
- Geometry Simplification
- What if Healing is Unsuccessful?

Geometry Simplification

Spline geometry of the surfaces and curves of an entity can be replaced with geometrically equivalent analytic surfaces and curves. The following command attempts to do this:

```
Healer Simplify {volume <ids>|body <ids>}
[{simplifytol} <value>] [keep]
```

Use **simplifytol** to specify a tolerance for the simplification operation. The default tolerance is 1e-6. Note: Even if the spline entity is within this tolerance to the analytic form, simplification still may not be possible.

The **keep** option will retain the original body and generate a new body containing analytic surfaces.

```
Execute Filter Curve Geometry_type Spline
```

What if Healing is Unsuccessful?

The ACIS healing module is under continued development and is improving with every release. However, there will often be situations where healing is unable to fully correct the geometry. This might be okay, as meshing is rarely affected by the small inaccuracies healing addresses. However, boolean operations on the geometry can fail if the bad geometry must be processed by the operation (i.e., a webcut must cut through a bad curve or vertex).

Here are some possible methods to fix this bad geometry:

- Return to the source of the geometry (i.e., Pro/ENGINEER) and increase the accuracy. Re-export the geometry.
- Remove the offending surface from the body (using the remove surface command), then construct new surfaces from existing curves and combine the body back together.
- Composite the surfaces over the bad area, mesh and create a net surface from the composite, remove the bad surfaces and combine.
- Export the geometry as IGES, import the IGES file into a new model and look for double surfaces or surfaces that show up at odd angles using the find overlap commands. Delete and recreate surfaces as needed and combine the surfaces back together into a body.

Contact the development team (cubit-dev@sandia.gov) if you need further help with fixing bad geometry.

Tweaking Surfaces

The following options of the Tweak Surface command are available. Command syntax and examples follow below.

- Tweaking a Surface Using an Offset
- Tweaking a Surface by Moving
- Tweaking Surfaces to Target Surfaces
- Removing a Surface
- Tweaking a Conical Surface
- Tweaking Doublers to Target Surface
- Removing Holes and Slots from Sheet Bodies
- Removing Fillets from Sheet Bodies
- Changing the Taper of Surfaces

Tweaking a Surface Using an Offset

```
Tweak Surface <id_list> Offset <val> [Surface
<id_list> Offset <val>] [Surface <id_list>
Offset <val> ...] [Keep] [Preview]
```

The **Tweak Offset** form of the command offsets an existing set of surfaces and extends the attached surfaces to meet them. A positive offset value will offset the surface in the positive surface normal direction while a negative value will go the other way. Different offsets may be specified for each surface. Figure 1 shows a simple example of offsetting. Note that you can also offset whole groups of surfaces at once. The keep option will retain the original surfaces and curves.

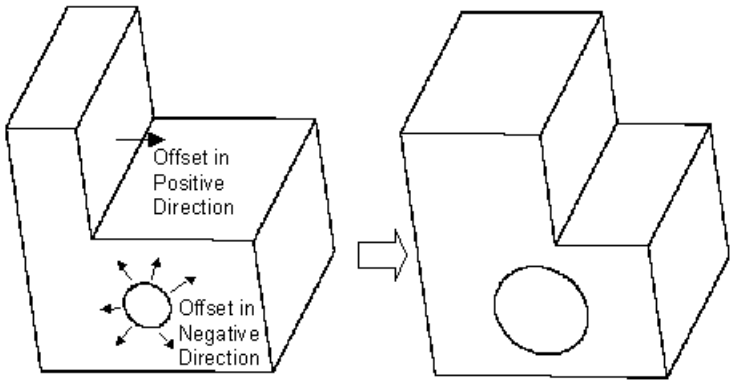


Figure 1. Tweak Offset

Tweaking a Surface by Moving

The **Tweak move** form of the command simply moves the given surfaces along a vector direction. The direction can be specified either absolutely or relative to other geometry entities in the model (from entity centroid to location). Note that when moving a surface for tweak, the surface is moved and the surface and the adjoining surfaces are extended or trimmed to match up again. So, for example, moving a vertically oriented planar surface in the vertical direction will have no effect. In this example, if you move the surface 10 in the x and 5 in the y the effect will be to move it simply 10 in the x. You can also use this form of the command to move a protrusion around - just be sure to specify all of the surfaces on the protrusion for moving. The last form of the command can be used to move a surface along another surface's normal.

```
Tweak Surface <id_range> Move
{Vertex|Curve|Surface|Volume|Body} <id>
```

```
Location {Vertex|Curve|Surface|Volume|Body}
<id> [Except [X][Y][Z]] [Keep] [Preview]

Tweak Surface <id_range> Move
{Vertex|Curve|Surface|Volume|Body} <id>
Location <x_val> <y_val> <z_val> [Except [X]
[Y][Z]] [Keep][Preview]

Tweak Surface <id_range> Move <dx_val>
<dy_val> <dz_val> [Keep] [Preview]

Tweak Surface <id_range> Move Direction
<options> Distance <val> [Keep] [Preview]

Tweak Surface <id_range> Move Normal To
Surface <id> Distance <val> [Except [X][Y][Z]]
[Keep][Preview]
```

Tweaking Surfaces to Target Surfaces

The **Tweak target** form of the command actually replaces the given surfaces with a copy of the new surfaces, then extends and trims surfaces to match up. This can be useful for closing gaps between components or performing more complicated modifications to models. The command syntax is:

```
Tweak {Curve|Surface} <id_list> Target
{Surface <id_list> [Limit Plane (options)]
[EXTEND|noextend] | Plane (options)} [keep]
[preview]

Tweak Surface <id_list> Replace [With] Surface
<id_list> [Keep] [Preview]
```

The **plane** option allows a plane to be specified instead of target surface(s). If a target surface is supplied, the user can also use a **limit plane** if he wishes. A limit plane is a plane that the tweak will stop at if the tweaked surface does not completely intersect the target surface. The limit plane must be used with the extend option. See the help for Specifying a Plane for the options available to define a plane.

Single target surfaces are automatically extended so that the tweaked body will fully intersect the target. Unfortunately, extending multiple target surfaces can sometimes result in an invalid target, so the option is given to tweak to unextended targets with the **noextend** option. In this case, the tweaked body must fully intersect the existing targets for success. If you experience a failure when tweaking to multiple targets or the results are unexpected, it is recommended to try the **noextend** option (**NOTE:** Tweaking to multiple targets is only implemented in the ACIS geometry engine). It is recommended to always **preview** before using the tweak target commands.

Figure 2 shows a simple example.

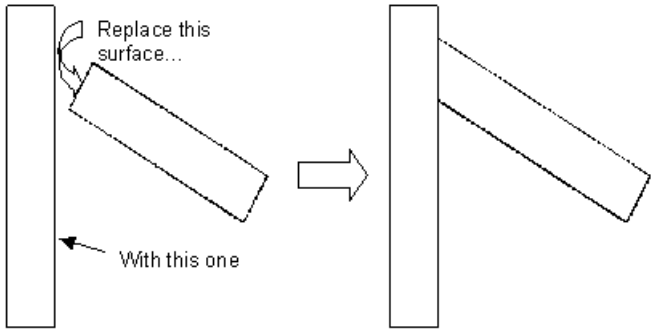


Figure 2. Tweak Surface Target (Viewed directly from the side)

Removing a Surface

The **Tweak remove** command allows you to remove surfaces from a model by extending the adjacent surfaces to fill in the resulting gaps. It is identical to the **Remove Surface** command. See Removing Surfaces for a description of the command options.

```
Tweak Surface <id_list> Remove [Blend_Chain]
[Cavity] [EXTEND|Noextend] [Keepsurface]
[Keep] [Preview]
```

Tweaking a Conical Surface

The **Tweak cone** form of the command is used to replace a conical projection with a flat circular surface. This command is useful for simplifying bolt holes. The command syntax is.

```
Tweak Surface <id_range> Cone [Preview]
```

The following is a simple example illustrating the use of the tweak surface cone command.

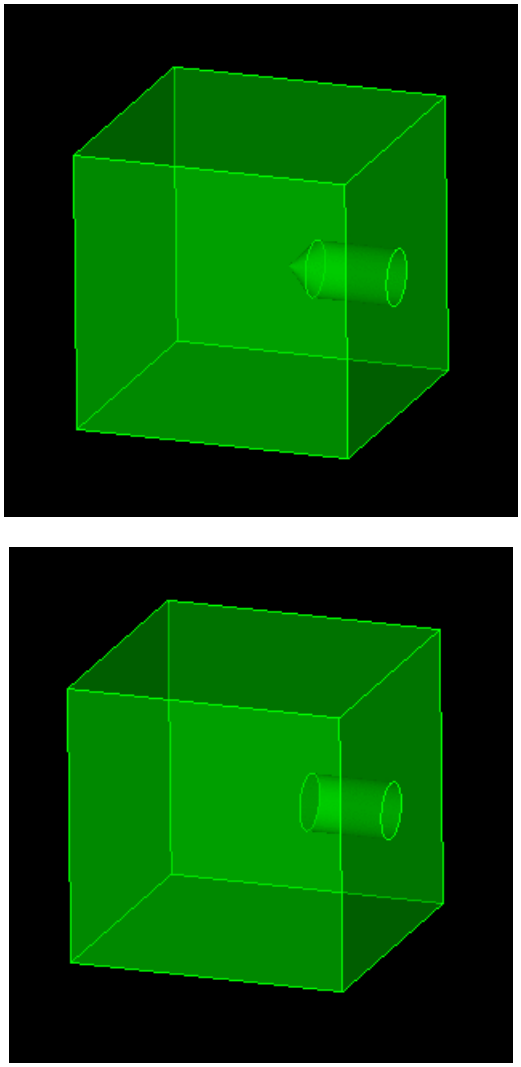


Figure 3. Conical bolt hole before and after tweaking

Tweaking Doublers to Target Surfaces

The Tweak Doubler form of the command takes a specified surface and creates drop-down surfaces either normal to the doubler surface or by a

user specified vector to a target surface. This can be helpful in creating surfaces for weld elements between midsurfaced geometry. The resulting surfaces do not create a bounding volume, and do not imprint themselves onto the target surface. The command syntax is:

```
Tweak Surface <id_list> Doubler Surface
<id_list> {[Limit Plane (options)]
[EXTEND|noextend]} [Internal] [Direction
(options)] [Thickness] [Preview]
```

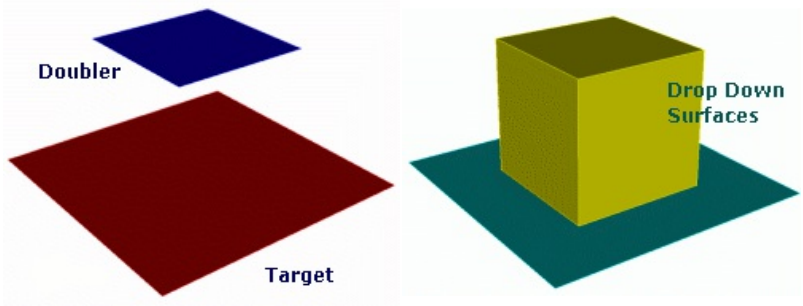
The **plane** option allows a plane to be specified instead of target surface(s). If a target surface is supplied, the user can also use a **limit plane** if he wishes. A limit plane is a plane that the tweak will stop at if the tweaked surface does not completely intersect the target surface. The limit plane must be used with the extend option. See the help for Specifying a Plane for the options available to define a plane.

Single target surfaces are automatically extended so that the tweaked body will fully intersect the target. Unfortunately, extending multiple target surfaces can sometimes result in an invalid target, so the option is given to tweak to unextended targets with the **noextend** option. In this case, the tweaked body must fully intersect the existing targets for success. If you experience a failure when tweaking to multiple targets or the results are unexpected, trying the noextend option is recommended.

If the doubler surface has a thickness property value, you can propagate that thickness value to the newly created drop-down surfaces by using the **thickness** flag.

It is recommended to always preview before using the tweak doubler commands.

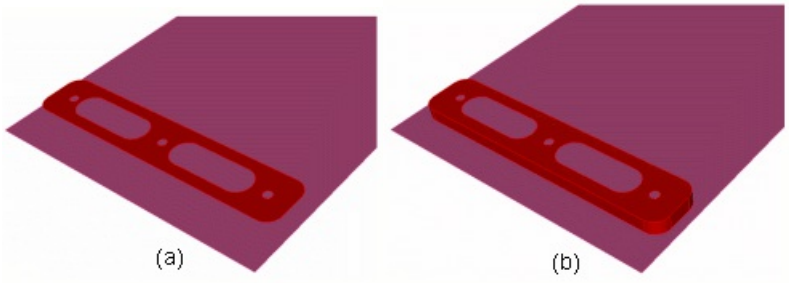
NOTE: This function only works for ACIS geometry.



Geometry **Output**

Figure 3. Extending a doubler surface to target

The **internal** option will also include internal curves when the surface is extended (see Figure 4c). The **direction** option will create a skewed surface along the given direction (see Figure 4d).



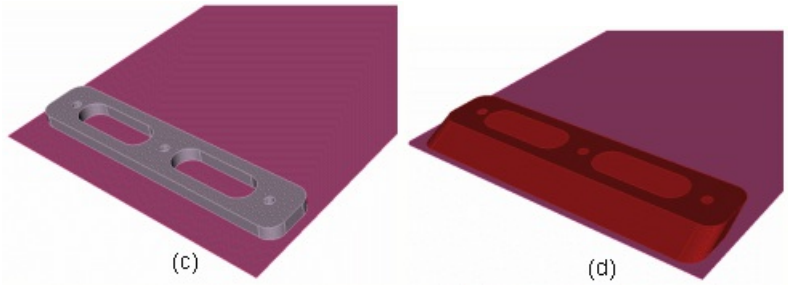


Figure 4. Explanation of tweak doubler options (a) Original surfaces (b) No option flags used (c) Internal option used - notice internal curves dropped down (d) Direction flag - notice skew

Removing Holes and Slots from Sheet Bodies

The **Tweak Hole/Slot Idealize** command takes a specified sheet body(s) and searches for either holes or slots (or both) which meet the user's input parameters. This can be helpful in removing small holes or slots quickly and efficiently from midsurfaced bodies where such level of detail isn't required. The command syntax is:

```
Tweak Surface <id_list> Idealize {[Hole Radius
<val>] [Slot Radius <val> Length <val>]}
[Exclude Curve <id_list>] [Preview]
```

Below is a diagram showing the different parameters available for input by the user.

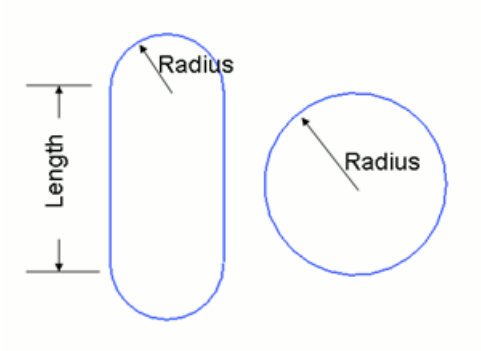


Figure 5. Input parameters for tweak surface idealize command

#Hole Removal Example
tweak surface 13 idealize hole radius 6

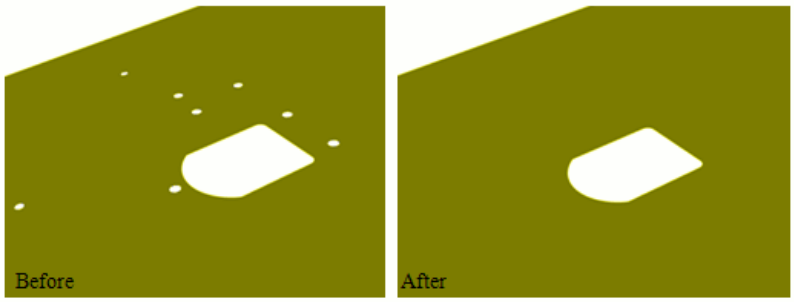


Figure 6. Example of hole removal using tweak surface idealize command

The **exclude** option allows the user to specify individual curves that should not be deleted, even if they meet the search criteria for removal. Figure 7 shows another hole removal example where several curves were

excluded.

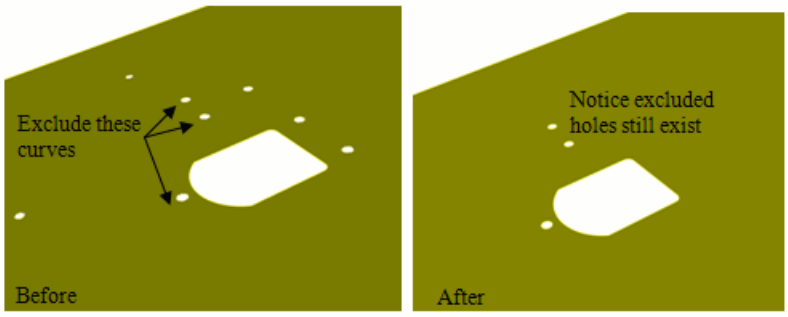


Figure 7. Example of hole removal using exclude option

Note: This feature is for ACIS geometry

It is recommended to always **preview** before using the tweak command. Preview will highlight all curves slated to be removed if the command is executed.

Removing Fillets from Sheet Bodies

The **Tweak Fillet Idealize** command takes a specified sheet body(s) and searches for either internal or external fillets (or both) which meet the users' radius parameter. This can be helpful in removing fillets quickly and efficiently from midsurfaced bodies where such level of detail isn't required. The command syntax is:

```
Tweak Surface <id_list> Idealize Fillet Radius
<val> {[Internal] [External]} [Exclude Curve
<id_list>] [Preview]
```

#Fillet Removal Example
tweak surface 13 idealize fillet radius 6 internal

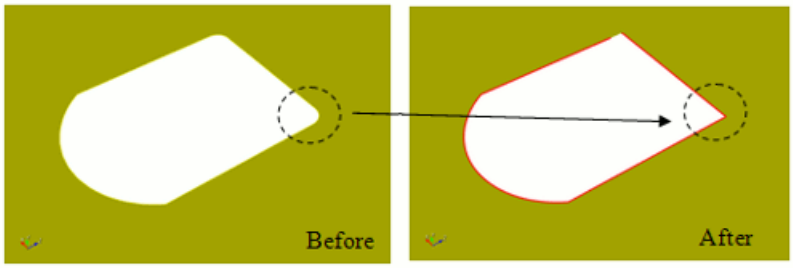


Figure 8. Example of fillet removal using tweak surface idealize command

Note: This feature is for ACIS geometry

It is recommended to always **preview** before using the tweak command. Preview will show the result if the command is executed.

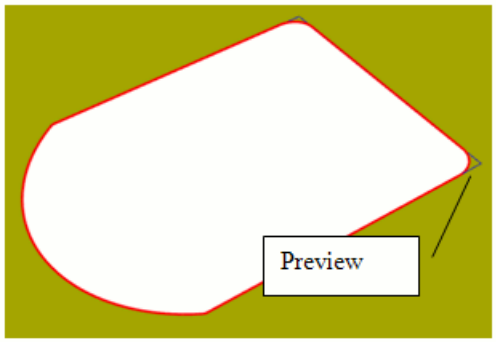


Figure 9. Preview of the tweak surface idealize command

Changing the Taper of Surfaces

The taper or angle of a surface can be made shallower or steeper using the taper surface command. Several surfaces can be tapered at once to get a smooth transition. The command syntax is:

```
taper Surface <id_list> angle <val> {from
plane <options> | about curve <id> | about
vertex <id_1><id_2>} [keep] [preview]
```

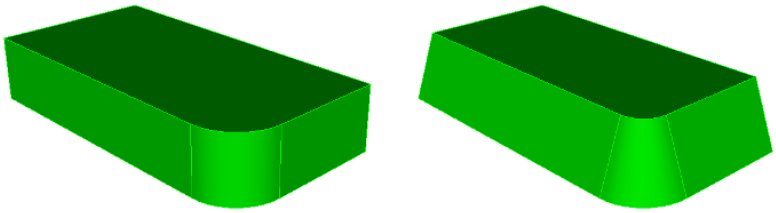


Figure 10. taper surface 1 7 6 angle -20 from plane surface 3

The **keep** and **preview** options are helpful when figuring out the correct command setup.

The **from plane** form of the command is the most basic form of the command. The plane normal defines the direction used to set the angle for the surface. The origin of the plane determines how a surface is tapered. If the plane intersects the middle of a surface, the surface will be rotated in on the top and out on the bottom. If it intersects the bottom, the top of the surface will be rotated in and the bottom will stay fixed. If it intersects the top, the bottom of the surface will rotate out and the top will stay fixed.

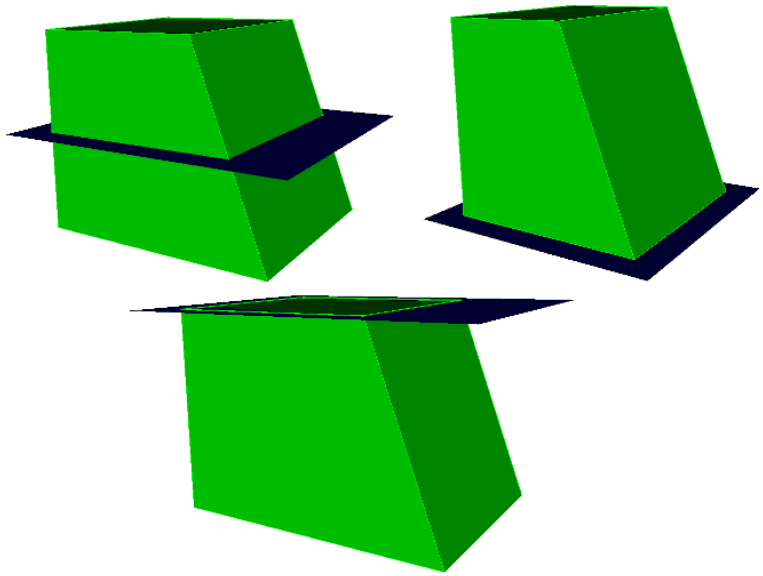


Figure 11. A cube with one surface tapered 20 degrees with a plane in the middle, bottom, and top.

The **about curve** form of the command can be used to rotate a surface about a curve. The axis of rotation is determined using the tangent direction of the curve at its starting point. That axis crossed with the normal of the surface at the starting point determine the direction used to set the surface angle. In order to work, the curve must be part of one of the surfaces being changed. Since the direction is based on the curve sense, the final direction is not always obvious. Running the command with the **preview** option first can help determine what angle to apply.

The **about vertex** form of the command is similar to the **about curve** form. The axis of rotation is a vector from the first vertex to the second. That axis crossed with the normal of the surface at the starting point determine the direction used to set the surface angle. In order to work, the vertices must be part of one of the surfaces being changed.

Tweaking Vertices

The Tweak Vertex command can be used to do the following:

- Tweaking a Vertex With a Chamfer
- Tweaking a Vertex With a Non-Equal Chamfer
- Tweaking a Vertex With a Fillet Radius

Tweaking a Vertex With a Chamfer

```
Tweak Vertex <id_range> Chamfer Radius <value>
[Keep] [Preview]
```

This form of the command creates a chamfered corner at the specified vertex. Can be use on volumes or free surfaces. The 'keep' option creates another volume on which the tweak is applied; the original volume remains unmodified.

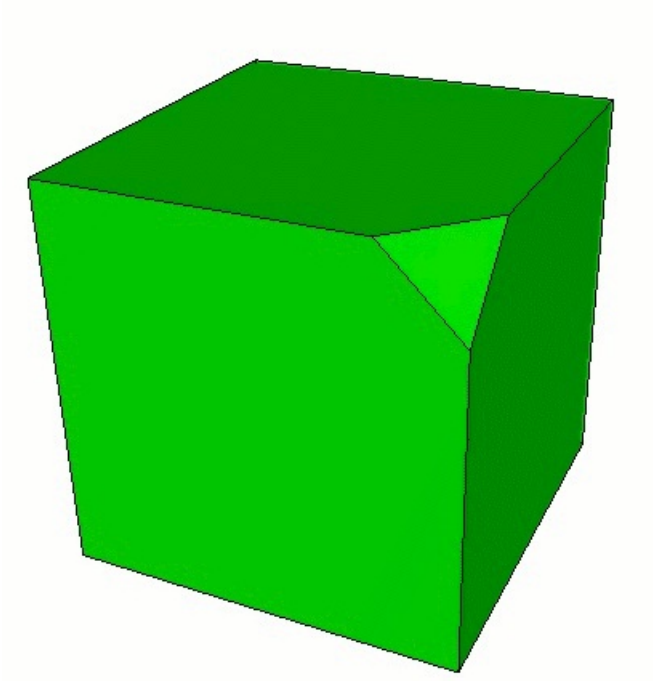


Figure 1. Tweak Vertex Chamfer

Tweaking a Vertex With a Non-Equal Chamfer

```
Tweak Vertex <id_range> Chamfer Radius <value>
[Curve <id> Radius <value> Curve <id> Radius
<value> Curve <id>] [Keep] [Preview]
```

This next form of the command creates a non-equal chamfered corner at the specified vertex. Can only be used on vertices of volumes. The 'keep' option creates another volume on which the tweak is applied; the original volume remains unmodified.

Tweaking a Vertex With a Fillet Radius

```
Tweak Vertex <id_range> Fillet Radius <value>
[Keep] [Preview]
```

This command replaces a vertex with a filleted radius. The command can only be used on free surfaces. The 'keep' option creates another volume

on which the tweak is applied; the original free surface remains unmodified.

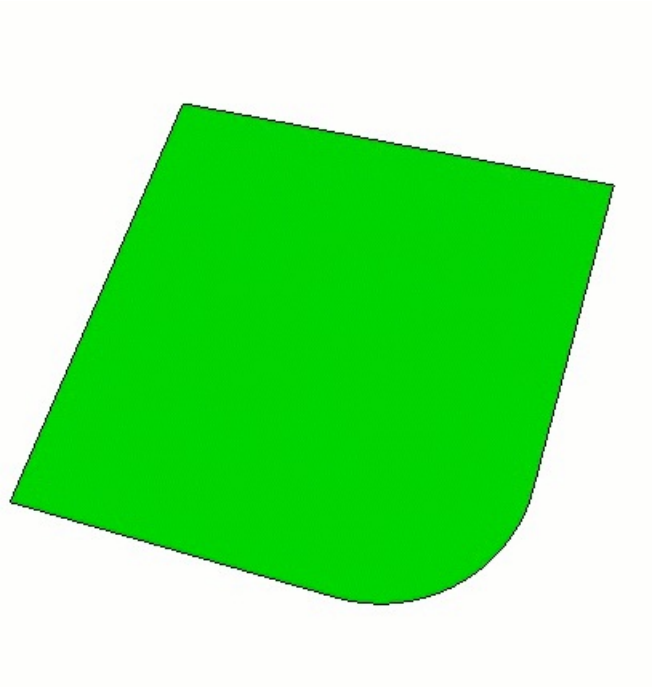


Figure 2. Tweak Vertex Fillet

Tweak Volume Bend

Entity bending bends a solid model around a given axis. In any bending operation, some material is stretched while other material is compressed, but the topology of the model is maintained. The command syntax is:

```
Tweak {Volume|Body} <id_list> Bend Root
<location_options> Axis <direction_vector>
Direction <direction_vector> Radius <val>
angle <val> [Preview] [Keep] [Center_bend]
[Location <options>]
```

Root and **axis** determine location for the bend. **Direction** determines direction of the bend. **Radius** and **angle** determine how much to bend. **Center_bend** will bend both sides of the volume around the bend location instead of one side. **Location** can be used to select only specific parts of a volume to bend.

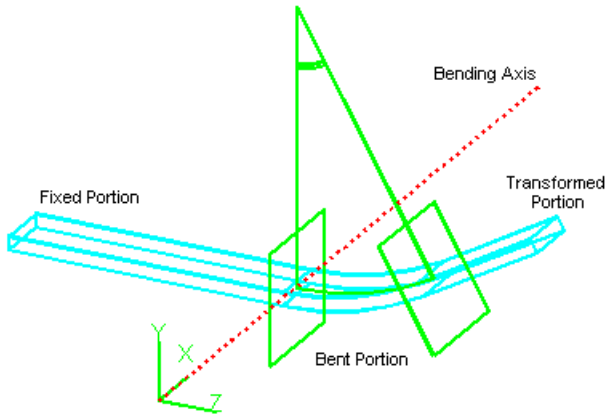


Figure 1. Bending a volume

#Ex: Bend parts of a body specified by the location option
create brick width 11 height 1
create brick width 1 depth 10 height 10
create brick width 1 depth 10 height 10
create brick width 1 depth 10 height 10
move body 2 general location position -3 5 0
move body 3 general location position 0 5 0
move body 4 general location position 3 5 0
subtract body 2 from body 1
subtract body 3 from body 1
subtract body 4 from body 1
tweak volume 1 bend root 0 0 0 axis 1 0 0 direction 0 0 -1 radius 1 angle 3.14
location vertex 39 47

Tweaking Geometry

- Tweaking Vertices
- Tweaking Curves
- Tweaking Surfaces
- Tweak Remove Topology
- Tweak Volume Bend

The tweaking commands modify models by moving, offsetting or replacing surfaces, curves, or volumes while extending the adjoining surfaces to fill the resulting gaps. This is useful for eliminating gaps between components, simplifying geometry or changing the dimensions of an object.

Tweaking Curves

The following options of the Tweak Curve command are available. Command syntax and description follow below.

- Create a Chamfer or Fillet
- Tweaking a Curve Using an Offset Distance
- Removing a Curve
- Tweaking a Curve Using a Target Surface, Curve, or Plane
- Tweaking a Pair of Curves to a Corner

Create a Chamfer or Fillet

The Tweak Curve Chamfer or Fillet command is used to fillet or chamfer a curve. The radius value is the radius of the fillet arc or chamfer cut distance. The command syntax is:

```
Tweak Curve <id_range> {Fillet|Chamfer} Radius
<value> [Keep] [Preview]
```

In addition to creating chamfers of a single cut distance, the chamfer can be specified be two values. The syntax is:

```
Tweak Curve <id_list> Chamfer Radius <val1>
[<val2>] [Keep] [Preview]
```

Figure 1 shows a brick ('br x 10') chamfered with two different cut distances ('Tweak Curve 1 2 Chamfer Radius 2 4').

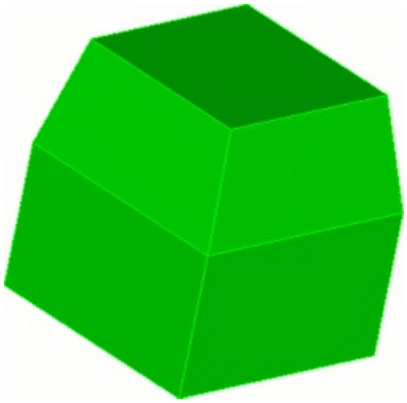


Figure 1 Chamfer with two different distances

Individual curves can also be filleted with different start and finish radius values. The syntax is:

```
Tweak Curve <id> Fillet Radius <val1> [<val2>]
[Keep] [Preview]
```

Figure 2 shows a brick ('br x 10') filleted with different start and end radius values (‘Tweak Curve 1 2 Chamfer Radius 2 4’).

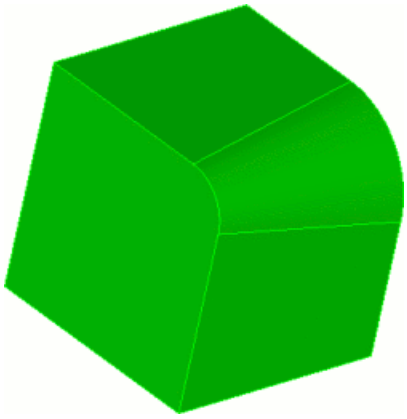


Figure 2. Fillet with two different radii

For all Tweak Fillet and Tweak Chamfer variations, the keep option prevents the destruction of the original geometry after the operation and the preview option temporarily displays the new geometry configuration without actually changing the geometry.

Tweaking a Curve Using an Offset Distance

```
Tweak Curve <id_list> Offset <val> [Curve
<id_list> Offset <val>] [Curve <id_list>
Offset <val> ...] [Keep] [Preview]
```

Tweaking curves a specified distance offsets the existing curves and extends the attached surfaces to meet them. A positive offset value will enlarge the surface while a negative value will decrease the area of the attached surface. Different offset values can be specified for each curve. The keep option prevents the destruction of the original geometry after the operation. The preview option temporarily displays the new geometry configuration without actually changing the geometry. Figure 3 shows an example of offsetting a curve a specified distance.

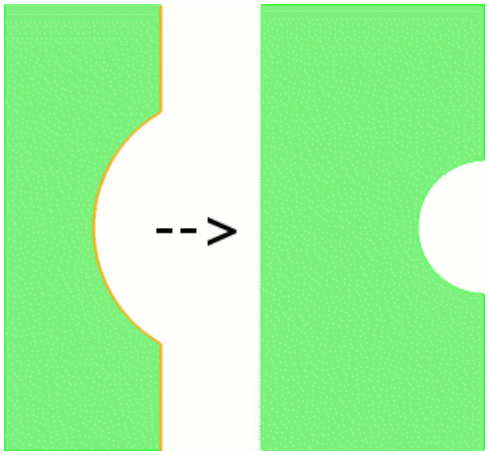


Figure 3 Offsetting a set of curves a specified distance

Removing a Curve

```
Tweak Curve <id_list> Remove [Keep] [Preview]
```

Similar to the Tweak Curve Remove command, the tweak curve remove function removes a specified curve from a sheet body. Figure 4 shows a simple example of removing a curve from a sheet body.

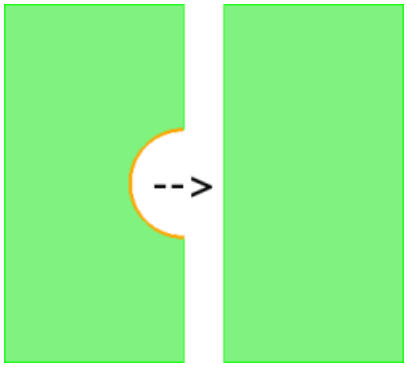


Figure 4. Removing a curve from a sheet body

The keep option prevents the destruction of the original geometry after the operation. The preview option temporarily displays the new geometry configuration without actually changing the geometry.

Tweaking a Curve Using Target Surfaces, Curves, or Plane

Use Tweak Curve Target to offset a curve to a specified surface, plane or curve. Figure 5 shows an example of tweaking a curve to several surfaces.

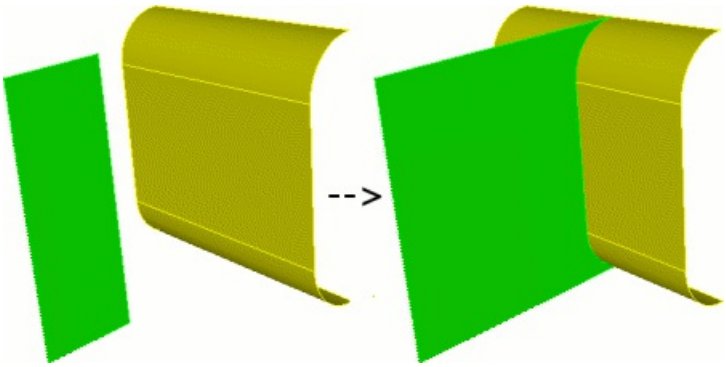


Figure 5 Tweaking a curve to multiple target surfaces

Similarly, a target plane can be specified using the Plane specification syntax. The Tweak Curve syntax is:

```
Tweak Curve <id_list> Target {Surface
>id_list> [Limit Plane (options)]
[EXTEND|Noextend] | Plane (options)}
[Max_area_increase <val>] [Keep] [Preview]

Tweak Curve <id_list> Target Curve <id_list >
[EXTEND|Noextend] [Max_area_increase <val>]
[Keep] [Preview]
```

If a target surface is supplied, the user can also use a **limit plane** if he wishes. A limit plane is a plane that the tweak will stop at if the tweaked curve does not completely intersect the target surface. The limit plane must be used with the extend option. See the help for Specifying a Plane for the options available to define a plane.

It should be noted that if the source and target surfaces are from the same body the resulting geometry will be automatically stitched. Single target surfaces are automatically extended so that the tweaked body will fully intersect the target. Unfortunately, extending multiple target surfaces can sometimes result in an invalid target, so the option is given to tweak to non-extended targets with the **noextend** option. In this case, the tweaked body must fully intersect the existing targets for success. If you experience a failure when tweaking to multiple targets or the results are

unexpected, it is recommended to try the noextend option (**NOTE:** Tweaking to multiple targets is only implemented in the ACIS geometry engine). If a value for the **max_area_increase**keyword is given, Cubit will not perform the tweak if the resulting surface area increases by more than the specified amount. The keyword expects a percentage to be entered (i.e. '50' for 50%). It is recommended to always **preview** before using the tweak target commands.

For all tweak target variations, the **keep** option prevents the destruction of the original geometry after the operation and the preview option temporarily displays the new geometry configuration without actually changing the geometry.

Although it may not be intuitive curves can also serve as the target geometry. Figure 6 shows an example of extending a curve to another curve.



Figure 6 Tweaking a curve to a target curve

Notice that the source curve actually extends to the target curve as if the target were a surface.

Tweaking a Pair of Curves to a Corner

When creating mid-surface geometry it is often useful to extend surfaces to form a corner. To handle this specific but common case use the tweak corner command.

```
Tweak Curve <id> <id> Corner [Preview]
```

Figure 7 shows a typical tweak corner example. Notice that surfaces are extended/trimmed to intersect at a corner.

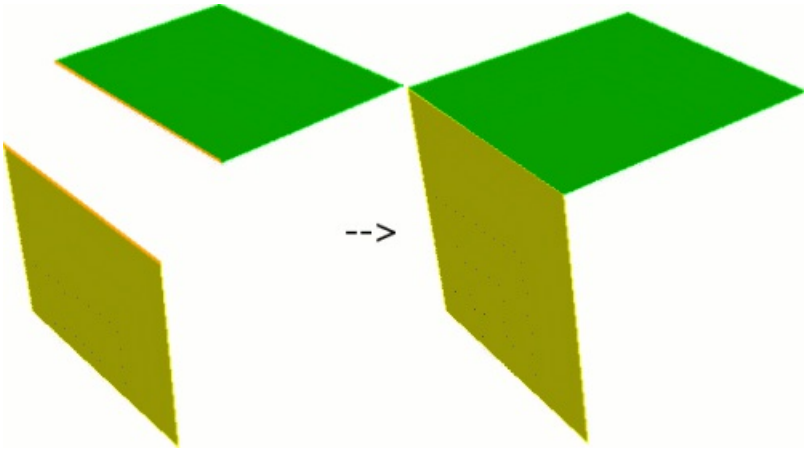


Figure 7. Tweaking two curves to a corner

The preview option temporarily displays the new geometry configuration without actually changing the geometry.

Tweak Remove Topology

The **Tweak Remove Topology** command removes curves and surface from a model and replaces them with new topology. The reconstruction of the new topology and the stitching of it into the model is done using real solid modeling kernel operations. This command is intended to be used on small curves and surfaces in the model. The command tries to find small curves/surfaces neighboring the specified topology and includes these neighbors in the removal process. Thus, the command can often be used to remove networks of small features just by specifying a single curve or surface.

```
Tweak Remove_Topology {Surface <id_range> |
Curve <id_range> | Surface <id_range> Curve
<id_range>} Small_curve_size <val>
Backoff_distance <val>
```

The **small_curve_size** is input by the user, and is used to calculate the small curves and surfaces. The **backoff_distance** value specifies how far away from the original topology cuts are made to cut out the old topology and stitch in the new topology. The removed topology is replaced by simplified topology where possible often resulting in a dimension reduction of the original topology. Extraneous curves that are introduced during the cutting and stitching process are regularized out if possible using the solid modeling kernel regularize functionality or are composited out using virtual geometry if the regularization is not possible.

Note: This command is currently only implemented for ACIS and Catia models.

Example

```
reset
set attribute on
import acis "test10.sat"
separate body all
set attribute off
Auto_clean Volume 1 Split_narrow_regions Narrow_size
2.2
tweak remove_topology curve 19 small_curve_size .21
backoff 1.5
```

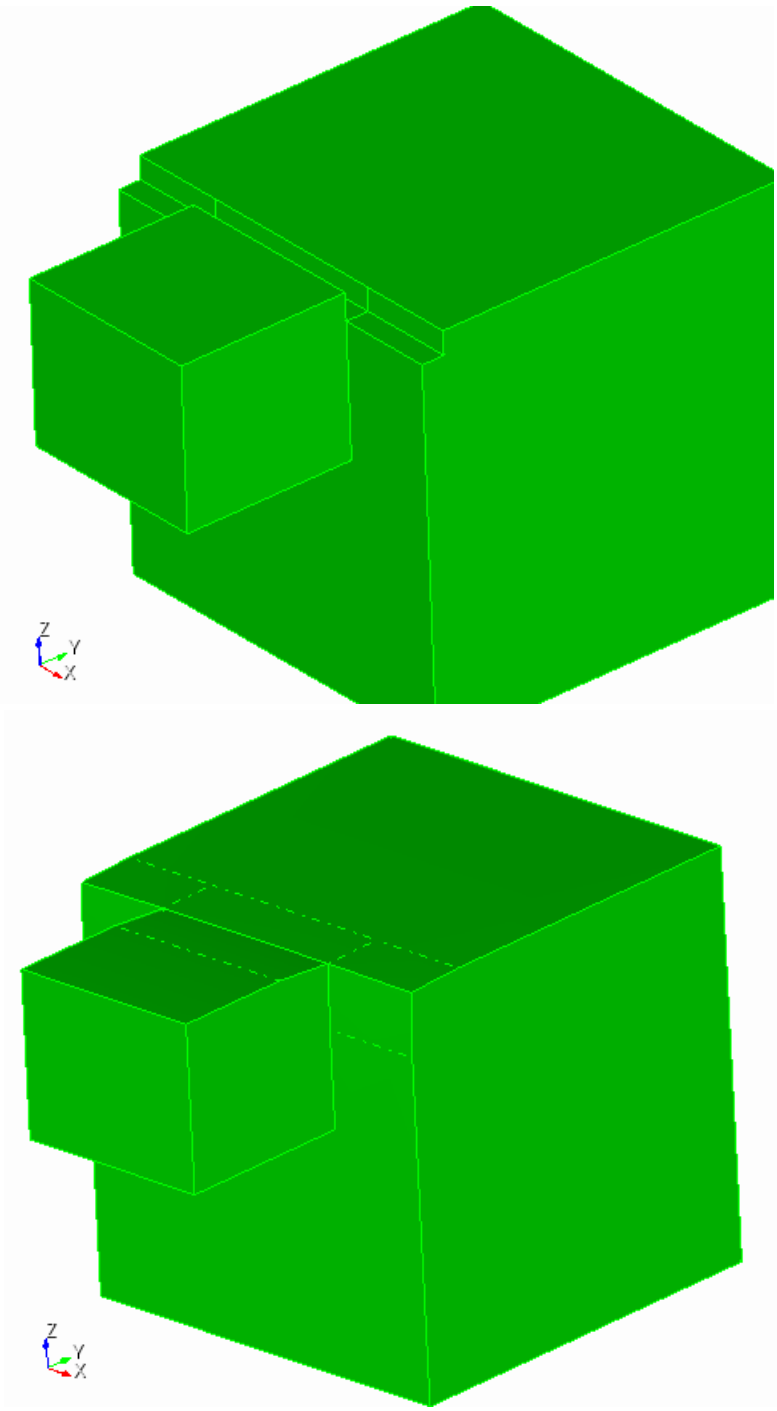


Figure 1. Tweak Remove Topology command

Removing Surfaces

- Remove Sliver Surfaces

The remove surface command removes surfaces from bodies. By default, it attempts to extend the adjoining surfaces to fill the resultant gap. This is a useful way to remove fillets and rounds and other features such as bosses not needed for analysis. See Figure 1 for an example of this process. The syntax for this command is:

```
Remove Surface <id_range> [Blend_Chain]
[Cavity] [EXTEND|Noextend] [Keepsurface]
[Keep] [TOGETHER|Individual|connected_sets]
```

The **noextend** qualifier prevents the adjoining surfaces from being extended, leaving a gap in the body. This is sometimes useful for repairing bad geometry - the surface can be rebuilt with surface from curves or a net surface, etc.., then combined back onto the body.

The **keep** option will retain the original body and put the results of the remove surface in a new body. The **keepsurface** option will retain the surface which was removed.

By default, the **TOGETHER** option removes the surfaces in a single call to the solid modeler. The **individual** option will remove surfaces one-by-one instead of as a group. If one removal fails, the rest are still attempted. Without the **individual** option, no surface is removed unless they are all able to be removed together. The **connected_sets** option sorts the surfaces into sets of connected surfaces, operating on each set separately, allowing the remove operation to succeed on some sets if it fails on others.

The **blend_chain** option will not only remove the selected surface but will also remove any surfaces belonging to the same blend chain. A blend is a non-planar surface such as a fillet that has a constant radius of curvature in at least one of its principal parametric directions. The blend chain includes all connected surfaces that share a common radius of curvature.

The **cavity** option can be used to remove all surfaces defining a hole or cavity. A cavity is defined as the collection of surfaces bounded by curves where the exterior angle is greater than 180 degrees. Designating the cavity option will automatically include all surfaces that are part of the cavity to which the surface belongs. If the surface does not belong to a cavity, this option will be ignored.

This command is identical to the Tweak Surface Remove command.

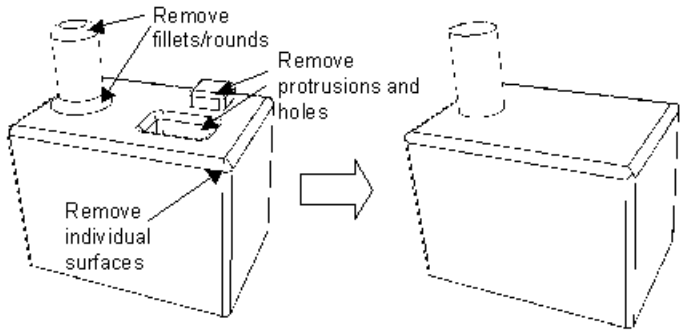


Figure 1. Remove Surface Example

Remove Sliver Surface

This command uses the ACIS remove surface capability on surfaces that have area less than a specified area limit. When ACIS removes a surface it extends the adjoining surfaces and intersects them to fill the gap. If it is not possible to extend the surfaces or if the geometry is bad the command will fail. The syntax for this command is:

```
Remove Slivers Body <id_range>
[EXTEND|Noextend] [Keepsurface] [Keep]
[Arealimit [<double>]]

Default Arealimit = 0.1
```

The **noextend**, **keepsurface** and **keep** options operate as for the remove surface command. The **arealimit** option allows the user to set the area below which surfaces will be removed.

Removing Vertices

At times you may find that you have an extraneous vertex in your model. This would be a vertex connected to two and only two curves. This stray vertex can cause unwanted mesh artifacts, due to the fact that a mesh node MUST lie on this vertex, thereby disallowing the possibility of movement for better quality. Fortunately there is a relatively easy way of getting rid of this stray vertex using the tweak surface command.

```
Tweak Surface <id> Replace With Surface
<same_id>
```

Note that you are replacing a surface with itself. In doing so, the geometry engine will do an intersection check on that surface, and should realize that the vertex doesn't need to be there.

Removing Geometric Features

- Vertex Removal
- Curve Removal
- Surface Removal

Cubit has the ability to remove surfaces, curves, and vertices in an effort to simplify the model for meshing. For example, surface removal extends adjacent surfaces to those being removed to fill in the gap where the removed surfaces were. Curve removal replaces a short, sliver curves with a vertex. And vertex removal can be facilitated with tweaking if it is connect to two curves of the same geometric type.

Automatic Geometry Clean-up

The automated geometry clean-up commands are used to automatically clean up geometry in preparation for meshing. These commands are built in to the ITEM interface, but they can also be used on their own. They include:

- Automatic Forced Sweepability
- Automatic Small Curve Removal
- Automatic Small Surface Removal
- Automatic Surface Split

Automatic Forced Sweepability

In some cases, a volume can be "forced" into a sweepable configuration by compositing surfaces on the linking surfaces. The automatic forced sweep command will attempt to automatically composite linking surfaces together to create a sweepable topology. This command can be useful in cases where there are many linking surfaces that prohibit sweepability and are not needed to define the mesh. It is assumed that the user has assigned the source and target surfaces for the sweep prior to calling this function. CUBIT will try to composite linking surfaces together to get rid of problems such as 1) non-submappable linking surfaces, 2) interior angles between curves of a surface that deviate far from multiples of 90 degrees, and 3) surfaces with curves smaller than the small curve size, if a small curve size is specified. This command is incorporated into the ITEM GUI, but is also available from the command line using the following command syntax.

```
Auto_clean Volume <id_range>
Force_sweepability [Small_curve_size <val>]
```

The **small_curve_size** qualifier is an optional argument. If a curve size is specified, the command will try to remove surfaces with curves smaller than this size by compositing the surface with adjacent surfaces.

Example

The following cylinder has been webcut and had surface splits so that it is not sweepable. The split surface command has also introduced 3 small curves on the surfaces. After the source and target surfaces are set, the force sweepability command is issued to automatically composite neighboring surfaces to make the volume sweepable and remove the small curves. The results are shown in the image below.

```
auto_clean volume 1 force_sweepability
small_curve_size .7
```

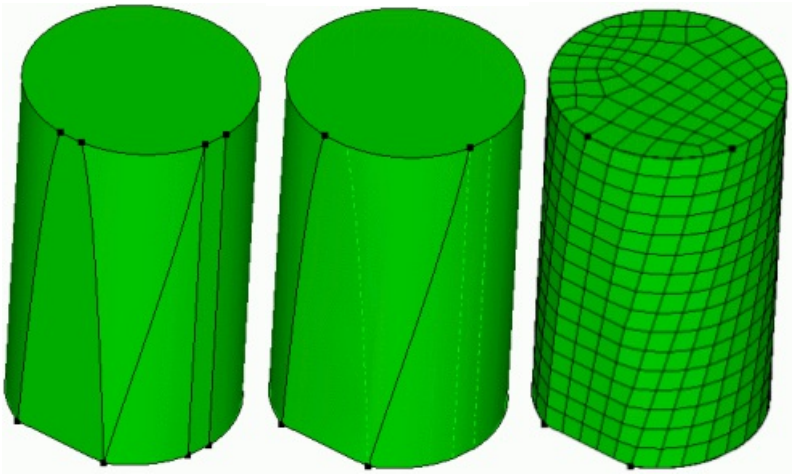


Figure 1. Linking surfaces are composited to force a sweepable volume topology

Automatic Surface Split

This auto clean command will attempt to automatically split narrow regions of surfaces. In this context, any surface that contains a portion that narrows down to a small angle is considered a narrow region. The command will use the split command from the underlying solid modeling kernel. The user specifies a size that defines what it narrow. This command also propagates the splits to neighboring narrow surfaces. This command is usually used as a preprocessor to the "tweak remove_topology" command but can also be used on its own.

```
Auto_clean Volume <id_range>
Split_narrow_regions Narrow_size <val>
```

Example

The model has a surface that necks down to a narrow region. This surface also has some neighboring narrow surfaces to which the splits are propagated.

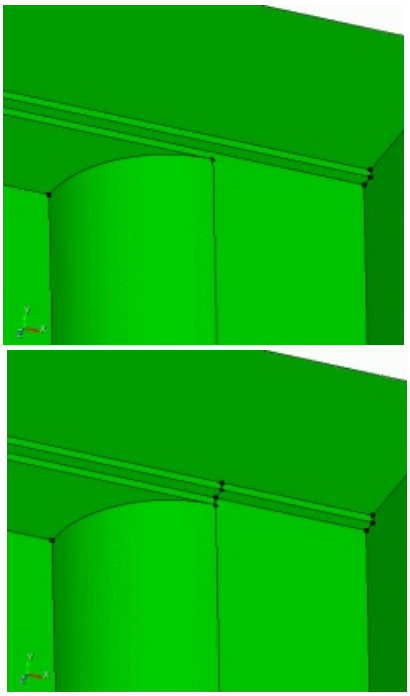


Figure 1. Automatic small and narrow surface removal on a cylinder

Automatic Small Curve Removal

The automatic small curve removal command uses composites and collapse curves commands to automatically remove small curves from a volume. This is useful for removing small or unnecessary details from a model to facilitate meshing algorithms. The user enters a small curve size. Any curve smaller than this specified size will be removed. This command is issued from the ITEM toolbar. More information can be found by reading the section entitled Small Details in the Model in the ITEM documentation. This command can also be called from the command line. The syntax of this command is:

```
Auto_clean Volume <id_range> Small_curves
Small_curve_size <val>
```

Note: The automatic curve removal should be used with caution, as the user has little control over how curves are removed.

Example:

The cylindrical model has 3 small curves just less than 0.7. The remove small curves command will remove two of the small curves by compositing two neighboring surfaces and the third using the collapse curve functionality.

```
auto_clean volume 1 small_curves
small_curve_size .7
```

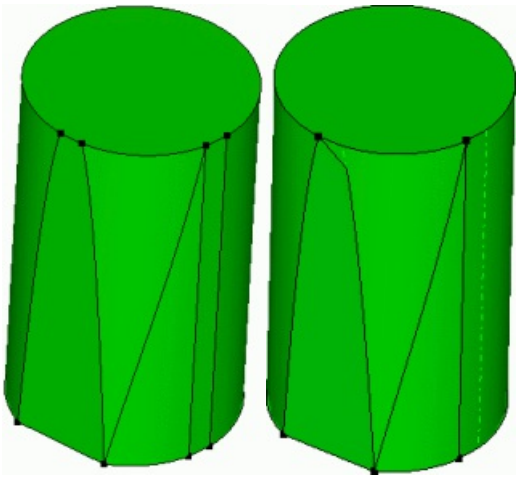


Figure 1. Automatic small curve removal on a cylinder

Automatic Small Surface Removal

This auto clean command will attempt to remove small and narrow surfaces from the model by compositing them with neighboring surfaces. The user specifies a small curve size value. This value is used in two different ways. First, a small area is calculated as the small curve size squared. This value is used to compare against when looking for small surfaces. The small curve size is also used to identify surfaces that are narrower than the small curve size.

```
Auto_clean Volume <id_range> Small_surfaces
Small_curve_size <val>
```

Example

The cylindrical model has 2 small surfaces and a few narrow surfaces. The surfaces are composited to remove these.

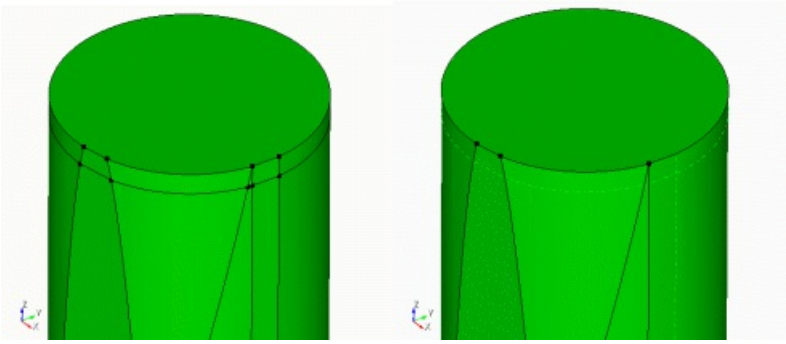
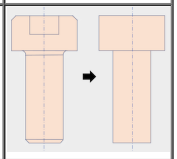
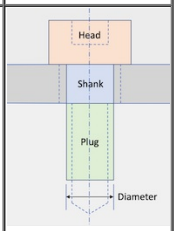
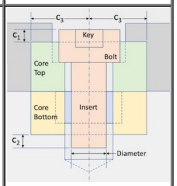
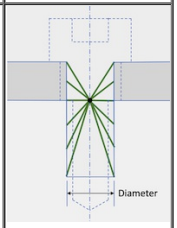
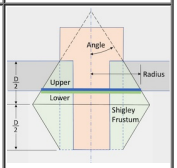
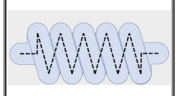


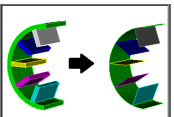
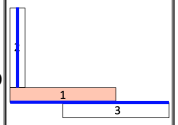
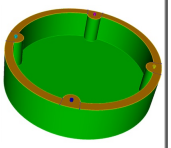
Figure 1. Automatic small and narrow surface removal on a cylinder

Reducing Geometry

The **reduce** options provide automatic defeaturing and simplification solutions for specific classes of geometry. These options are most often used as methods for rapidly modifying geometry representing fasteners and springs to representations that can be readily used in analysis. For example, the CAD representation of a bolt may include threads, fillets, chamfers, cavities and may also overlap surrounding geometry. The **reduce** option can automatically defeature, webcut and modify surrounding geometry as well as mesh and apply boundary conditions according to a designated *recipe*. A limited number of reduce recipes are currently supported for bolts, including the following:

- Reduce Simplify
- Reduce Bolt Fit Volume
- Reduce Bolt Core
- Reduce Bolt Spider
- Reduce Spring
- Reduce Thin Volumes
- Reduce Thin Volumes with Reinforcement Learning
- Reduce Slot Surface

Reduce Recipe	Command Syntax	Description	
Reduce Bolt (Simplify)	**reduce volume <id_list>**	Simplify/Defeature bolt geometry only	
Reduce Bolt Fit Volume	**reduce volume <id_list> bolt fit_volume**	Simplify bolt and fit to surrounding geometry	
Reduce Bolt Core	**reduce volume <id_list> bolt core**	Simplify bolt and insert surrounding core geometry	
Reduce Bolt Spider	**reduce volume <id_list> bolt spider**	Remove bolt and replace with spider mesh	
Reduce Bolt Patch	**reduce volume <id_list> bolt patch**	Remove bolt and replace with a circular sideset patch	
Reduce Spring	**reduce volume <id_list> spring**	Creates curve(s) following center of spring cross-section	

Reduce Thin Volume	**reduce volume <id_list> thin auto**	Reduces the thin 3D volumes to a connected set of 2D shell surfaces	
Reduce Thin Volume RL	**reduce volume <id_list> thin RL**	Uses machine learning and reinforcement learning to reduce thin 3D volumes to a connected set of 2D shell surfaces. See also Beam and Shell Modeling with the Geometry power Tool	
Reduce Slot Surface	**reduce surface <id> slot**	Provides decomposition solutions for Electromagnetics simulation for decomposing and preparing slot surfaces for analysis. See also Slot Surface Preparation with the Geometry power Tool	

The **reduce** command is often used in conjunction with the Geometry Power tool and the machine learning classification methods. The classification tool can group volumes according to commonly recognized shapes such as bolt, nut, washer, spring, etc.

Reduce (Simplify)

The simplest form of the **Reduce** command. This option will simplify and defeature volumes without affecting the surrounding geometry.

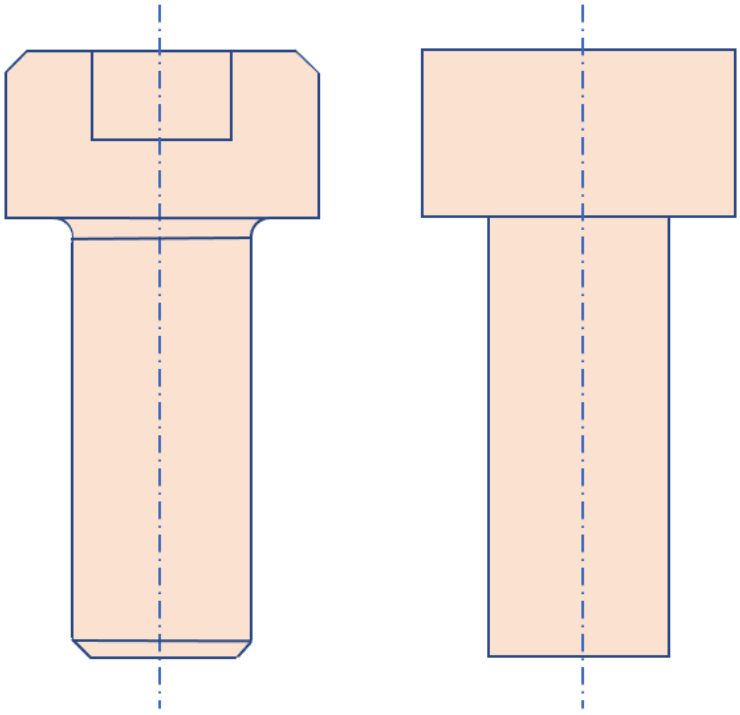


Figure 1. Example before and after of the Reduce command

Syntax:

> **Reduce volume {<ids>} [preview]**

Discussion:

The **reduce** command without additional options will perform simplification and defeaturing operations on the given volumes. This includes automatic chamfer, blend, cavity and small surface removal. While intended primarily for the bolt and fastener use case, it can also be useful for other volume types, however results may vary depending on complexity of the geometry.

volume ids: Specify the ids of the volumes to be reduced. The Geometry Power Tool classification diagnosic can be used for identifying volumes as "bolts".

preview: optional argument to display a preview of the reduce operation without execution of the reduction. This option will display the reduced volume in blue with the surrounding volumes displayed in wireframe.

Reduce Bolt Fit_Volume

The **Reduce Bolt** command is intended to prepare a volume identified as a **bolt** for analysis by simplifying its geometry, fitting to overlapping geometry, creating blocks and groups, etc. The **fit_volume** option can also modify the surrounding hole geometry or optional **insert** which may be at the bolt shaft. In addition to simplifying the bolt and insert geometry, this operation can remove any overlap and fit the shaft geometry to its surrounding volume or insert. See also the **reduce bolt core** option which adds the ability to generate a core region surrounding the bolt.

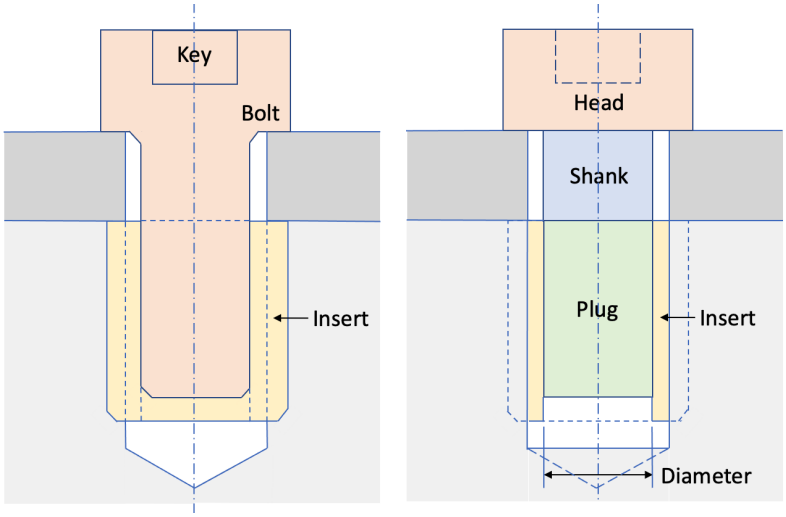


Figure 1. Example before and after of the Reduce Bolt Fit_Volume command. Also shows optional insert geometry at the bolt shaft.

Syntax:

> **Reduce {volume <ids>} bolt fit_volume [insert {volume <ids>}] [diameter {<value>|auto}] [align_axis] [tight_fit] [adjust_hole_diameter] [simplify_hole] [remove_key] [merge] [webcut [{Head|Shank|BOTH}]] [mesh] [mesh_size <value>] [bolt_block_id {<value>|Default}] [increment_bolt_block_id] [bolt_block_name {<string>|Default}] [head_block_id {<value>|Default}] [increment_head_block_id] [head_block_name {<string>|Default}] [shank_block_id {<value>|Default}] [increment_shank_block_id] [shank_block_name {<string>|Default}] [plug_block_id {<value>|Default}] [increment_plug_block_id] [plug_block_name {<string>|Default}] [insert_block_id {<value>|Default}] [increment_insert_block_id] [insert_block_name {<string>|Default}] [preview]**

Discussion:

CAD representations of a bolt assembly can often define overlapping volumetric regions or leave gaps between the bolt and its surrounding geometry. It may also include an optional insert, a cylindrical part surrounding the bolt shaft which can also overlap the bolt and surrounding geometry. The fit_volume option is intended to manage this overlap or gap by adjusting the hole or insert diameter and/or removing void space below the bolt.

In addition to removing overlap or gap, the bolt can be automatically webcut into three parts: head, shank and plug as

shown in figure 1. The resulting webcut volumes can in turn be assigned to blocks and the diameter of the shank can be altered. The plug can also be merged or contiguously meshed with the fastened geometry.

The following outlines the options for the reduce bolt fit_volume command.

volume ids: Specify the ids of the volumes to be reduced. The Geometry Power Tool classification diagnosic can be used for identifying volumes as "bolts".

Optional Arguments

insert {volume <ids>}: An insert is a cylindrical shaped volume which may be surrounding the shaft of the bolt. The top of the insert is usually flush with the lower volume (light grey volume in figure 1). When using the insert option with the ID of the insert volume, the reduce command will automatically simplify the insert geometry, removing any fillets, rounds or small features on the insert. Any overlap or gap between the shaft and surrounding geometry will be removed so that the insert will fit flush with the shaft and bolt hole. The tight_fit and adjust_hole_diameter options cannot be used when including an insert. If the insert option is not used, any existing insert geometry that may be present at the bolt shaft will be ignored.

diameter {<value>|auto}: Use the diameter option to alter the diameter of the bolt shank to a specific diameter indicated by <value>. The auto option will change the diameter of the bolt to exactly match the diameter of its hole. If the diameter option is not used, the existing diameter of the bolt will be used.

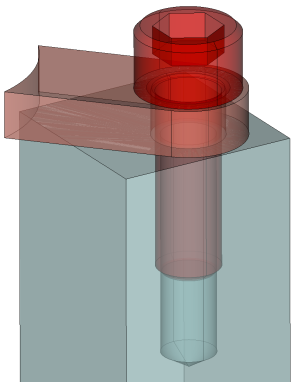


Figure 2. Bolt geometry prior to reduce operation showing nearby volumes.

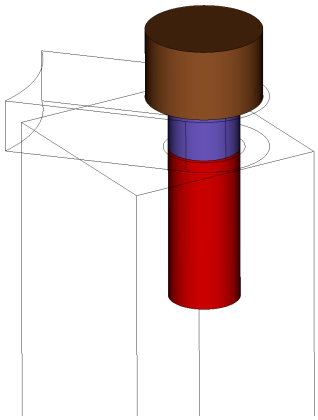


Figure 3. Bolt after reduce operation using fit_volume, webcut and block assignment options.

align_axis: Use this option if the center line of the bolt does not exactly match its hole center line. This option will automatically transform and rotate the bolt volume so that the center line of the hole and bolt are aligned and the bolt head is in contact with the upper volume.

tight_fit: This option is normally used when the bolt shaft is overlapping the lower geometry as shown in figure 2. The tight_fit option will perform a boolean subtract operation to ensure that the bolt exactly fits the shaft geometry to the lower volume, removing any gaps and overlaps surrounding or below the bolt. Figure 3

shows the result of the tight_fit option. This option cannot be used if an insert volume is also specified. See figure 4. for an example illustrating the tight_fit option.

adjust_hole_diameter: This option is normally used to adjust the diameter of the surrounding hole to match that of the bolt shaft, removing any gap or overlap. In contrast to the tight_fit option, only the hole diameter is tweaked without removing any existing void space below the bolt. This option cannot be used if an insert volume is also specified. See figure 4 for an example illustrating the the adjust_hole_diameter option.

simplify_hole: Bolt holes can sometimes include a fillet at the lip of the hole and/or a conical shape indentation at its base. The simplify_hole option will automatically remove these features from the hole leaving a flat base and sharp corner at the lip. Simplify_hole is usually used with the adjust_hole_diameter to ensure the shank fits flush with the hole surfaces and the hole is simplified. Simplify_hole has no effect on the lower volume if the tight_fit option is used. It can however simplify the upper volume if it contains any fillets or chamfers at its lip.

remove_key: The bolt geometry can often include a hexagonal shaped hole at the top of the bolt that fits an allen key. By default, this key hole will not be removed. Including the remove_key will ensure this hole is removed from the bolt geometry. If the mesh option is used, a hex mesh can be generated in most cases either with or without the key removed. Hex meshing will most likely be unsuccessful if the diameter of the key hole exceeds that of the bolt shaft. As a consequence, remove_key option may be necessary to facilitate meshing.

webcut [{Head|Shank|BOTH}]: The bolt can be cut into up to 3 different volumes as shown in figure 1. Using the optional Head or Shank options a single webcut can be executed separating just the head from the shank or just the shank from the plug respectively. If no arguments are used to the webcut option, or the both option is used, both webcuts will be performed resulting in three volumes. The location of the webcut on the shank will be where the bolt exits the lower volume.

merge: When the merge option is used. the plug portion of the bolt will be imprinted and merged with the hole surface(s) in the lower volume. In addition, the plug, shaft and head will be merged together. If an insert volume is specified, the exterior insert surface will be imprinted and merged with the hole surface(s) and the bolt plug will be imprinted and merged with the interior surface(s) of the insert. If the merge option is not used, all volumes created from the reduce command will not be imprinted or merged.

mesh: This option will attempt to generate a swept mesh on the bolt and insert geometries as part of the reduce command. If the allen key hole is present in the bolt head, a webcut will be performed on the bolt geometry to facilitate meshing. If meshing is unsuccessful, consider using the remove_key option.

mesh_size <value>: Specifies the mesh size to be used with the mesh option. If not specified, an automatic size will be determined.

Block ID and name assignment:When the webcut option is used, the resulting volumes may be assigned to a block. The following options may be used to assign to blocks based on an ID or a block name:

- **head_block_id {<value>|Default}, head_block_name {<string>|Default}:**
- **shaft_block_id {<value>|Default}, shaft_block_name {<string>|Default}:**

- **plug_block_id {<value>|Default}, plug_block_name {<string>|Default}:**

The blocks may be defined by either a block_name or block_id. If the block does not yet exist, a new one will be created. The Default option will automatically select an id or name.

If webcut is not used, the resulting reduced bolt volume may still be assigned to a block id using either the bolt_block_id {<value>|Default} or bolt_block_name {<string>|Default} options.

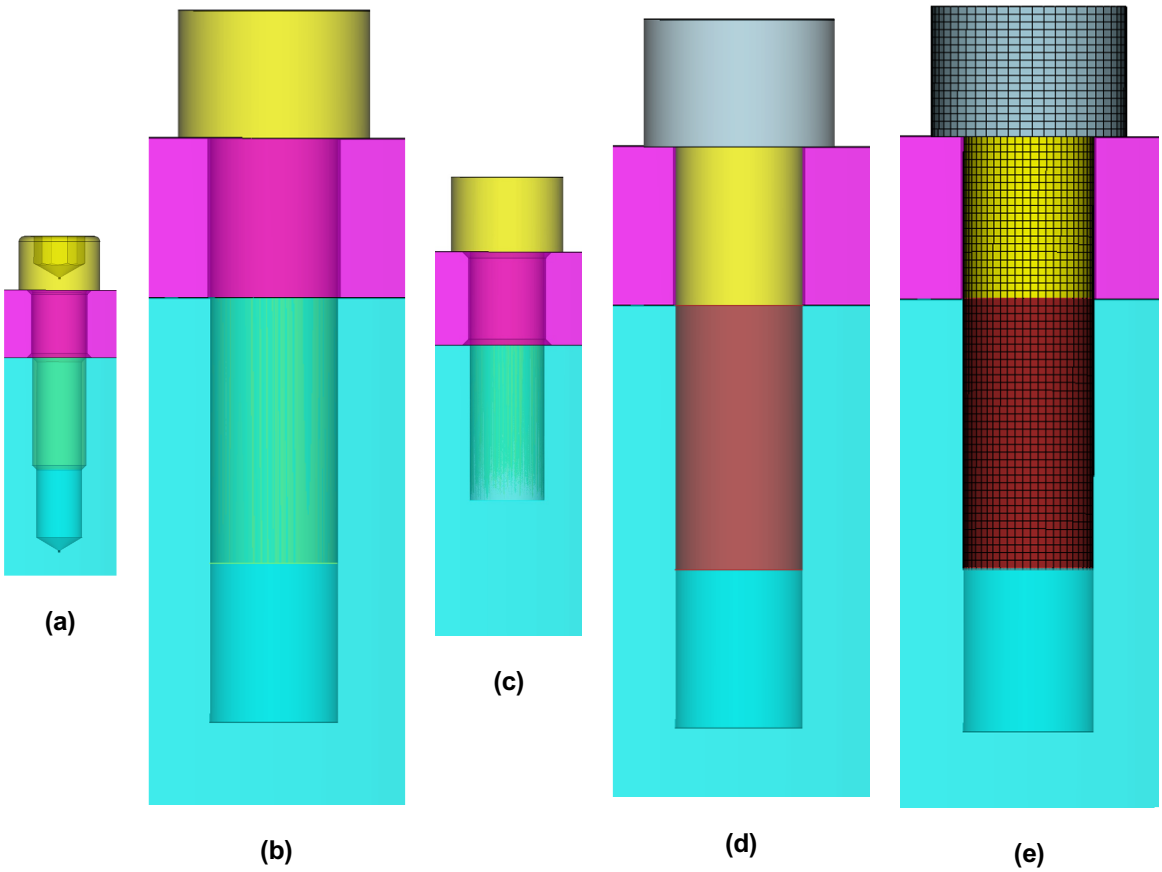
Each of the block ID specifications for bolt, plug, shaft and head also have a corresponding optional increment argument. When assigning new webcut volumes to blocks, the block ID can be automatically generated by incrementing from a specified block_id. For example, if head_block_id is defined as 100, and the increment_head_block_ids option is used, each new head volume generated will be assigned to a new unique block id starting with 100, followed by 101, 102, 103, etc.

If an **insert** volume is specified, **insert_block_id**, **insert_block_name** and **increment_insert_block_id** may also be used in a similar manner to set up block information on the insert volume.

preview: optional argument to display a preview of the **reduce bolt fit_volume** operation without execution of the reduction. This option will display the reduced bolt geometry in blue with the surrounding volumes displayed in wireframe.

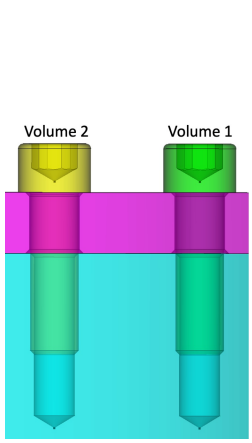
Examples

The following figures illustrate variations on the **fit_volume** option for the **reduce bolt** command. The original CAD geometry is pictured on the left with results on the right.

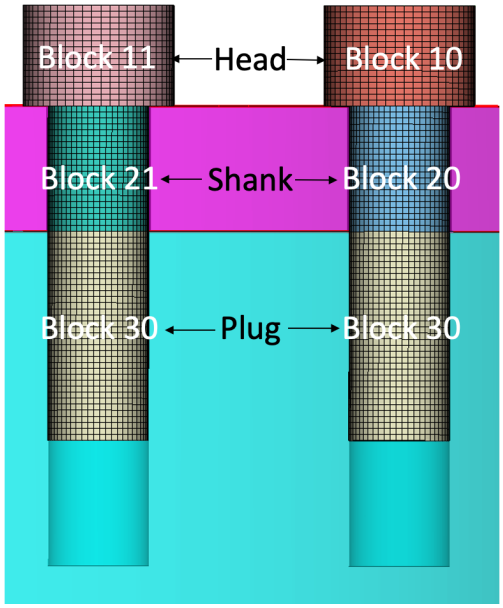


Initial CAD geometry	reduce volume 2 bolt fit_volume adjust_hole_diameter simplify_hole remove_key merge	reduce volume 2 bolt fit_volume tight_fit remove_key merge	reduce volume 2 bolt fit_volume remove_key adjust_hole_diameter simplify_hole webcut merge	reduce volume 2 bolt fit_volume remove_key adjust_hole_diameter simplify_hole webcut merge mesh

Figure 4. Example of four different variations of syntax for the reduce bolt fit_volume command on a single bolt. In this case, the bolt (volume 2) overlaps the cyan colored lower volume. Note the difference between the adjust_hole in figures (b), (d) and (e) and tight_fit (c) options. In both cases, volume overlap is removed, however adjust_hole_diameter only adjusts the hole diameter to match the bolt shaft, but tight_fit also removes any void space below the bolt. Also note in figure (d), three separate volumes are created from the bolt geometry when the webcut option is used. In figure (e), the bolt has been meshed with default sizing using Cubit's sweep tool



(a)

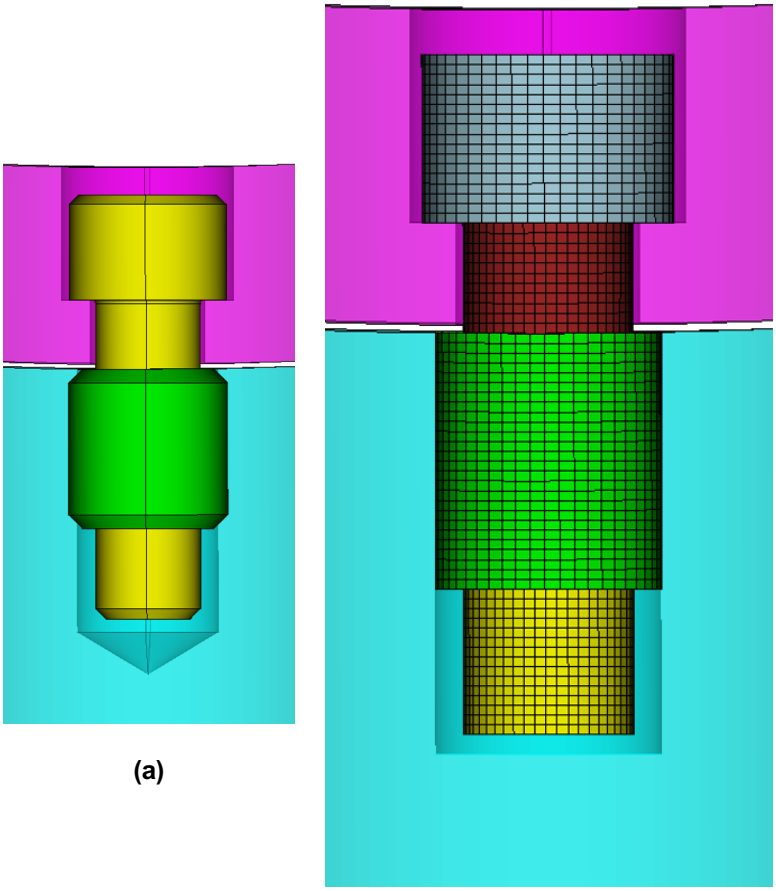


(b)

Initial CAD geometry	reduce volume 1, 2 bolt fit adjust_hole_diameter simplify_hole remove_key webcut merge mesh head_block_ID 10 increment_head_block_ID shank_block_ID 20 increment_shank_block_ID plug_block_ID 30

Figure 5. Example of using the reduce bolt fit_volume command to reduce multiple bolts with a single command while specifying blocks. In this case the block IDs are specified for head, shank and plug volumes. Note that the increment_head_block_ID and increment_shank_block_ID options are used, but the increment_plug_block_ID option is not. The result is that the block IDs on the heads (block 10, 11) and shanks (blocks 20, 21) are

incremented with each new bolt, while the block defined on all plugs will remain the same (block 30) without incrementing.



(a)

(b)

Initial CAD geometry

reduce volume 2 bolt fit_volume
insert volume 1 simplify_hole webcut
merge mesh

Figure 6. Example of using the reduce bolt fit_volume command with the insert option. In this case, volume 1 is specified as the insert and volume 2 is the bolt. Note that both the bolt and insert volumes have been simplified and the insert volume imprinted and merged with both the hole surfaces and the bolt plug.

Reduce Bolt Core

The **Reduce Bolt** command is intended to prepare a volume identified as a **bolt** for analysis by simplifying its geometry, fitting to overlapping geometry, creating blocks and groups, etc. For the **core** option of the **reduce** command, a cylindrical geometry surrounding the bolt will be webcut from the surrounding geometry. The core geometry is often used to define a higher resolution hex mesh than the surrounding geometry to better capture potential failure conditions. The **core** option can also manage corresponding "insert" volumes surrounding the shaft of the bolt. The **reduce bolt core** command is similar to the **reduce bolt fit_volume** command except that the **core** option adds the ability to define a core region surrounding the bolt.

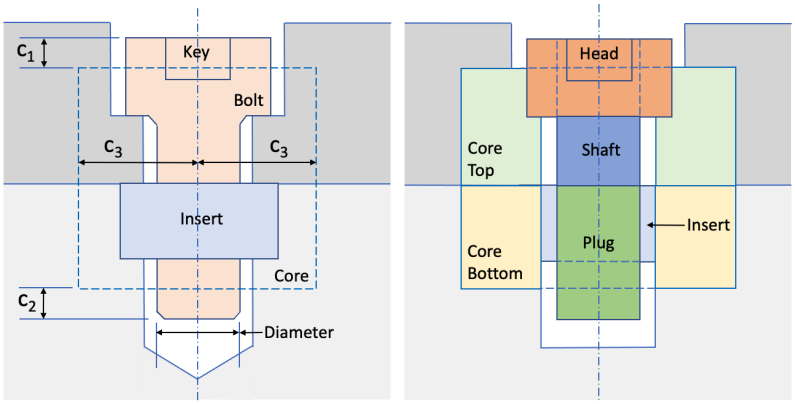


Figure 1. Example before and after of the Reduce Bolt Core command

Syntax:

> **Reduce volume {<ids>} bolt core [insert volume {<ids>}] [c1 <value>] [c2 <value>] [c3 <value>] [diameter {<value>|auto}] [align_axis] [tight_fit] [adjust_hole_diameter] [simplify_hole] [remove_key] [merge] [webcut [{Head|Shank|BOTH}]] [mesh] [mesh_size <value>] [bolt_block_id {<value>|Default}] [bolt_block_name {<string>|Default}] [increment_bolt_block_id] [head_block_id {<value>|Default}] [head_block_name {<string>|Default}] [increment_head_block_id] [shank_block_id {<value>|Default}] [shank_block_name {<string>|Default}] [increment_shank_block_id] [plug_block_id {<value>|Default}] [plug_block_name {<string>|Default}] [increment_plug_block_id] [insert_block_id {<value>|Default}] [insert_block_name {<string>|Default}] [increment_insert_block_id] [core_top_block_id {<value>|Default}] [core_top_block_name {<string>|Default}] [increment_bolt_core_top_id] [core_bottom_block_id {<value>|Default}] [core_bottom_block_name {<string>|Default}] [increment_core_bottom_block_id] [preview]**

Discussion:

The dimensions of the core geometry are defined relative to the bolt that it surrounds. Two core volumes are normally generated where the cylindrical core geometry is subtracted from the top and bottom volumes as illustrated in figure 1. In addition to generating the core volumes, the bolt can be automatically webcut into three parts: head, shank and plug. If an insert volume is present, it can be automatically modified to remove overlap and merged to the shank and hole surfaces. In addition the

diameter of the bolt shank can be altered, the hole fit to the shank diameter and the the allen key cavity optionally removed. The resulting volumes can in turn be hex meshed and assigned to blocks.

An example of the **reduce bolt core** operation is illustrated in figures 2 and 3. In this example, an "insert" volume is present that is overlapping the hole geometry. The resulting geometry from the operation is shown in figure 3.

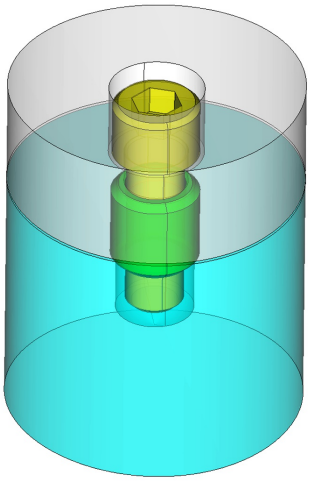


Figure 2. Bolt and insert geometry prior to reduce bolt core operation showing nearby volumes.

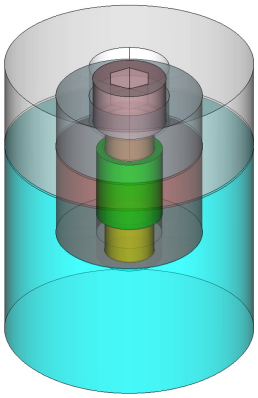


Figure 3. Bolt, insert and core volumes after the reduce operation

The **reduce bolt core** operation also provides the option to assign the resulting volumes to blocks and mesh the volumes at a specified size. Figure 4 illustrates the resulting default blocks and mesh denerated from the example shown in figure 3.

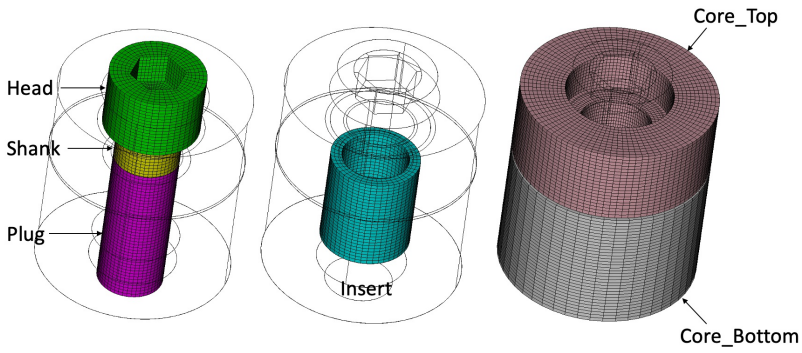


Figure 4. Example blocks and mesh automatically generated from the Reduce Bolt Core command

The following outlines the options for the **reduce bolt fit_volume** command.

volume ids: Specify the ids of the volumes to be reduced. The Geometry Power Tool classification diagnosic can be used for identifying volumes as "bolts".

Optional Arguments

insert {volume <ids>}: An insert is a cylindrical shaped volume which may be surrounding the shaft of the bolt. The top of the insert is usually flush with the lower volume (light grey volume in figure 1). When using

the **insert** option with the ID of the insert volume, the **reduce** command will automatically simplify the insert geometry, removing any fillets, rounds or small features on the insert. Any overlap or gap between the shaft and surrounding geometry will be removed so that the insert will fit flush with the shaft and bolt hole. The **tight_fit** and **adjust_hole_diameter** options cannot be used when including an **insert**. If the **insert** option is not used, any existing insert geometry that may be present at the bolt shaft will be ignored.

c1 <value>, c2 <value>, c3 <value>: Dimensions relative to the bolt defining the size of the core geometry as illustrated in Figure 1.

1. **c1 <value>**: Distance from the top of the bolt head to the top of the core geometry.
2. **c2 <value>**: Distance from the bottom of the bolt shaft to the bottom of the core geometry.
3. **c3 <value>**: Radius of core relative to the center line of the bolt.

diameter {<value>|auto}: Use the diameter option to alter the diameter of the bolt shank to a specific diameter indicated by <value>. The auto option will change the diameter of the bolt to exactly match the diameter of its hole. If the diameter option is not used, the existing diameter of the bolt will be used.

align_axis: Use this option if the center line of the bolt does not exactly match its hole center line. This option will automatically transform and rotate the bolt volume so that the center line of the hole and bolt are aligned and the bolt head is in contact with the upper volume.

tight_fit: This option is normally used when the bolt shaft is overlapping the lower geometry as shown in figure 2. The **tight_fit** option will perform a boolean subtract operation to ensure that the bolt exactly fits the shaft geometry to the lower volume, removing any gaps and overlaps surrounding or below the bolt. Figure 3 shows the result of the **tight_fit** option. This option cannot be used if an **insert** volume is also specified. See figure 4. for an example illustrating the **tight_fit** option.

adjust_hole_diameter: This option is normally used to adjust the diameter of the surrounding hole to match that of the bolt shaft, removing any gap or overlap. In contrast to the **tight_fit** option, only the hole diameter is tweaked without removing any existing void space below the bolt. This option cannot be used if an **insert** volume is also specified. See figure 4 for an example illustrating the the **adjust_hole_diameter** option.

simplify_hole: Bolt holes can sometimes include a fillet at the lip of the hole and/or a conical shape indentation at its base. The **simplify_hole** option will automatically remove these features from the hole leaving a flat base and sharp corner at the lip. **Simplify_hole** is usually used with the **adjust_hole_diameter** to ensure the shank fits flush with the hole surfaces and the hole is simplified. **Simplify_hole** has no effect on the lower volume if the **tight_fit** option is used. It can however simplify the upper volume if it contains any fillets or chamfers at its lip.

remove_key: The bolt geometry can often include a hexagonal shaped hole at the top of the bolt that fits an allen key. By default, this key hole will not be removed. Including the **remove_key** will ensure this hole is removed from the bolt geometry. If the **mesh** option is used, a hex mesh can be generated in most cases either with or without the key removed. Hex meshing will most likely be unsuccessful if the diameter of the key hole exceeds that of the bolt shaft. As a consequence, **remove_key** option may be necessary to facilitate meshing.

webcut [{Head|Shank|BOTH}]: The bolt can be cut into up to 3 different volumes as shown in figure 1. Using the optional **Head** or **Shank** options a single webcut can be executed separating just the head from the shank or just the shank from the plug respectively. If no arguments are used to

the **webcut** option, or the **both** option is used, both webcuts will be performed resulting in three volumes. The location of the webcut on the shank will be where the bolt exits the lower volume.

merge: When the **merge** option is used. the plug portion of the bolt will be imprinted and merged with the hole surface(s) in the lower volume. In addition, the plug, shaft and head will be merged together. If an insert volume is specified, the exterior insert surface will be imprinted and merged with the hole surface(s) and the bolt plug will be imprinted and merged with the interior surface(s) of the insert. If the **merge** option is not used, all volumes created from the **reduce** command will not be imprinted or merged.

mesh: This option will attempt to generate a swept mesh on the bolt and insert geometries as part of the **reduce** command. If the allen key hole is present in the bolt head, a webcut will be performed on the bolt geometry to facilitate meshing. If meshing is unsuccessful, consider using the **remove_key** option.

mesh_size <value>: Specifies the mesh size to be used with the **mesh** option. If not specified, an automatic size will be determined.

Block ID and name assignment:When the **webcut** option is used, the resulting volumes may be assigned to a block. The following options may be used to assign to blocks based on an ID or a block name:

- **head_block_id {<value>|Default}, head_block_name {<string>|Default}:**
- **shaft_block_id {<value>|Default}, shaft_block_name {<string>|Default}:**
- **plug_block_id {<value>|Default}, plug_block_name {<string>|Default}:**

The blocks may be defined by either a **block_name** or **block_id**. If the block does not yet exist, a new one will be created. The **Default** option will automatically select an id or name.

If **webcut** is not used, the resulting reduced bolt volume may still be assigned to a block id using either the **bolt_block_id {<value>|Default}** or **bolt_block_name {<string>|Default}** options.

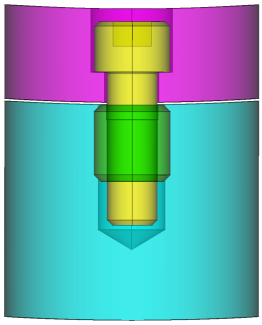
Each of the block ID specifications for bolt, plug, shaft and head also have a corresponding optional **increment** argument. When assigning new webcut volumes to blocks, the block ID can be automatically generated by incrementing from a specified block_id. For example, if **head_block_id** is defined as **100**, and the **increment_head_block_ids** option is used, each new **head** volume generated will be assigned to a new unique block id starting with **100**, followed by **101**, **102**, **103**, etc.

If an **insert** volume is specified, **insert_block_id**, **insert_block_name** and **increment_insert_block_id** may also be used in a similar manner to set up block information on the insert volume.

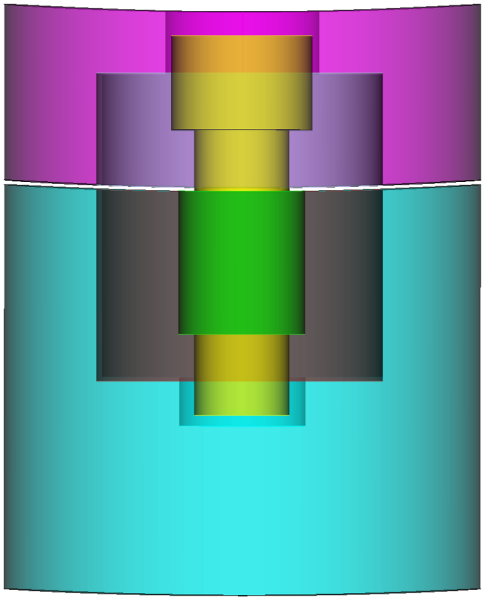
preview: optional argument to display a preview of the **reduce bolt fit_volume** operation without execution of the reduction. This option will display the reduced bolt geometry in blue with the surrounding volumes displayed in wireframe.

Examples

The following figures illustrate variations on the **core** option for the **reduce bolt** command.



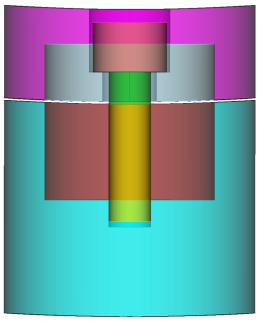
(a)



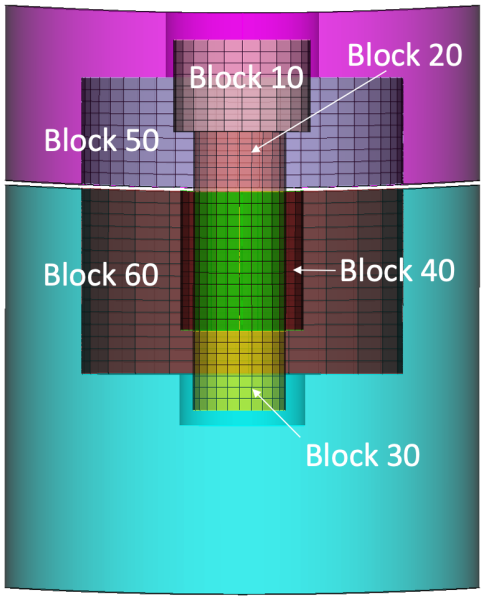
(b)

Initial CAD geometry

reduce volume 2 bolt core insert volume 1 c1 0.936345 c2 0.936345 c3 3.4749 simplify_hole remove_key merge



(c)



(d)

reduce volume 2 bolt core c1 1 c2 1 c3 4 diameter 2 adjust_hole_diameter simplify_hole remove_key webcut merge

reduce volume 2 bolt core insert volume 1 c1 1 c2 1 c3 4 diameter 2 simplify_hole remove_key webcut merge mesh mesh_size 0.3 head_block_ID 10 shank_block_ID 20 plug_block_ID 30 insert_block_ID 40 core_top_block_ID 50 core_bottom_block_ID 60

Figure 5. Examples of using the reduce bolt core command where (a) is the original geometry. In (b), an insert volume is included and the dimensions of the core are specified. Figure (c) does not include the insert and modifies the dimensions of core using the c1, c2 and c3 options. In addition, the diameter of the bolt shaft has been modified from its original diameter and the bolt cut into head, shank and plug volumes. Figure (d) again includes the insert and also specifies a mesh and custom mesh size to be used. In addition, note that block IDs for each of the resulting volumes have been specified. The resulting block IDs are also illustrated in figure (d).

Reduce Bolt Spider

The **Reduce Bolt** command is intended to prepare a volume identified as a **bolt** for analysis by replacing the bolt geometry with a set of mesh edges or *beams* and generating blocks containing the mesh. There are currently three different options for the spider command as illustrated in figures 1 to 3.

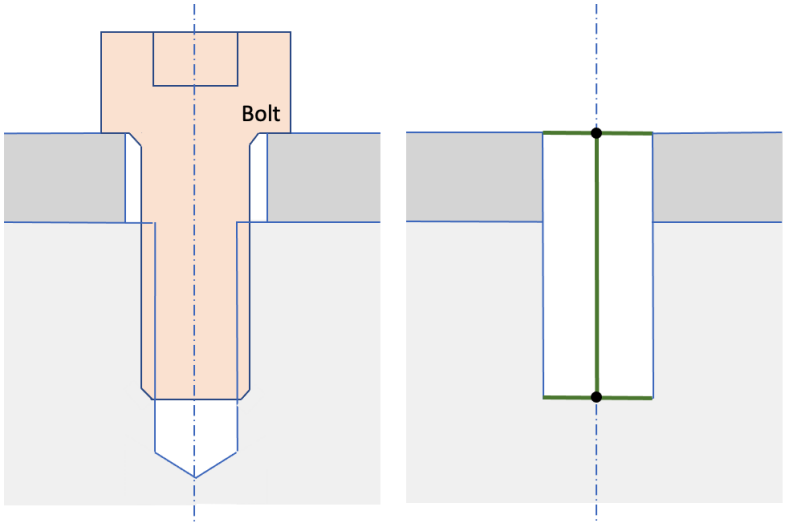


Figure 1. Example before and after of the Reduce Bolt Spider Wagon_Wheel command

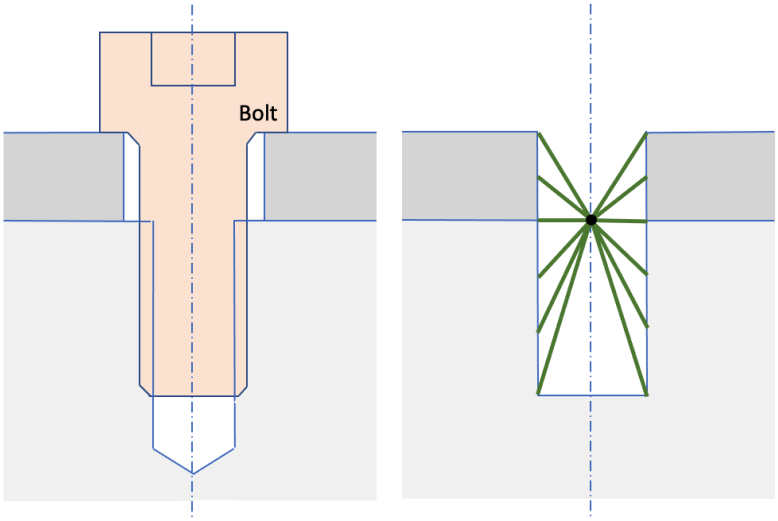


Figure 2. Example before and after of the Reduce Bolt Spider J2G command

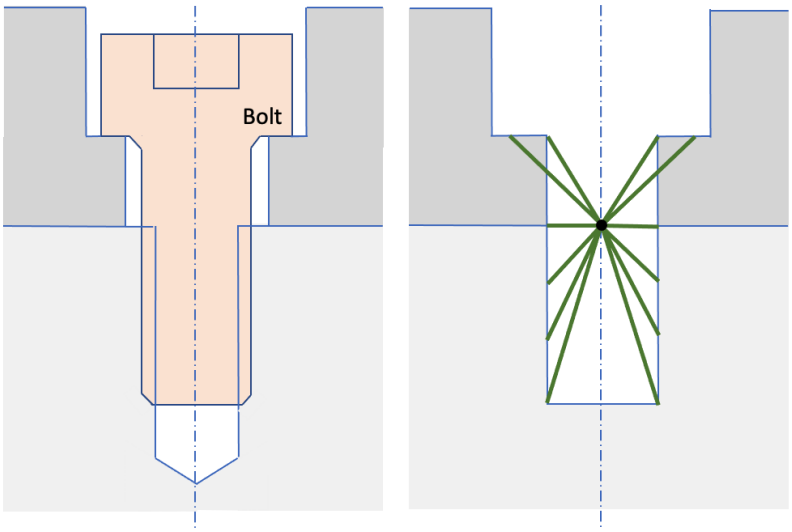


Figure 3. Example before and after of the Reduce Bolt Spider Countersink command

Syntax:

```
Reduce {volume <ids>|{upper surface <ids>lower surface
<ids>}} bolt spider {wagon wheel|j2g|countersink}
[diameter <value>] [mesh] [mesh_size]
[spider_block_id {<value>|Default}] [spider_block_name
{<string>|Default}] [increment_spider_block_id]
[rebar_block_id {<value>|Default}] [rebar_block_name
{<string>|Default}] [increment_spider_block_id]
[upper_spider_block_id {|Default}]
[increment_upper_spider_block_id]
[upper_spider_block_name {|Default}]
[lower_spider_block_id {|Default}]
[increment_lower_spider_block_id]
[lower_spider_block_name {|Default}] [preview]
[preview]
```

Discussion:

The **spider** options will also simplify the surrounding geometry at a bolt hole, including removing any blends or chamfers. It will also simplify the hole so in the lower volume to which the bolt is fastened will fit exactly to the bolt geometry or specified diamater.

The **spider** options will not, by default, generate the beam mesh as shown in figures 1 to 3, but instead define blocks to which the beam elements will be added when the mesh is generated or the **mesh** option is used. The following describes the options for the **reduce bolt spider** command.

Bolt holes can be defined by specifing the cylindrical bolt volume or by specifying the upper and lower surfaces of the bolt holes. The latter is helpful if bolt volumes don't exist in the model but the holes do.

volume ids: Specify the ids of the volumes to be reduced. The Geometry Power Tool classification diagnosic can be used for identifying volumes as "bolts".

upper/lower bolt hole surface ids: Specify the upper and lower surfaces of the bolt hole. Multiple upper/lower pairs can be specified.

{wagon wheel|j2g|countersink}: One of these three methods must be specified for the **spider** option.

1. **wagon wheel**: Shown in figure 1., a center beam element is generated and assigned to the **rebar** block. In addition, beams are generated extending from the ends of the rebar beam radially

connecting the top and bottom circular curves of the hole. The radial beam elements are assigned to the **spider** block.
2. **j2g**: Shown in figure 2., a center node is generated and assigned to the **rebar** block. In addition, beams are generated extending from the the rebar node radially connecting all nodes of the cylindrical surfaces of the hole. The radial beam elements are assigned to the **spider** block.
3. **countersink**: Shown in figure 3., similar to the **j2g** option, however beam elements are also generated connecting the outer radius of bolt with the center rebar node.

Optional Arguments

diameter <value>: Use the **diameter** option to alter the diameter of the resulting hole. If no value is specified for **diameter**, the existing diameter of the bolt will be used.

mesh: This option can be used to generate the beam mesh as part of the **reduce** operation. Since the resolution of the beam mesh depends on the nodes defined on the hole geometry, the surfaces of the hole will also be meshed using a mapped meshing scheme. Use the **mesh_size** option to control the resolution of the beam mesh. When the hole surfaces are meshed, the beam elements will also be generated and assigned to the existing **spider** and **rebar** blocks. Note that if the **mesh** option is not used, an empty **spider** and **rebar** block will be generated.

mesh_size: Use the **mesh_size** option to control the resolution of the beam mesh. If no **mesh_size** has been defined, if a mesh size has been defined on the bolt volume, it will be used, otherwise an automatic default size will be computed and used for meshing.

The **spider** and **rebar** blocks may be defined by either a **block_name** or **block_id**. If the block does not yet exist, a new one will be created. The **Default** option will automatically select an id or name. The default names for the **spider** and **rebar** blocks are **Spider_Block** and **Rebar_Block** respectively.

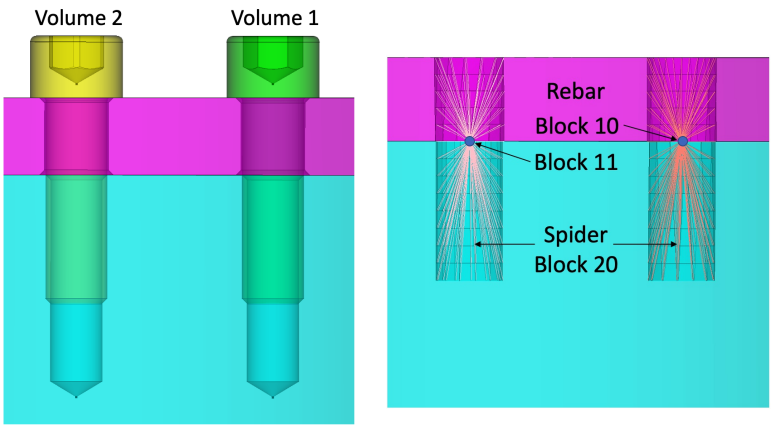
When building new blocks for each bolt volume, the block IDs can be automatically generated by incrementing from a specified block_id. The **increment_spider_block_id** and **increment_rebar_block_id** options are used for this prupose. For example, if **rebar_block_id** is defined as **100**, and the **increment_rebar_block_ids** option is used, each new **rebar** generated will be assigned to a new unique block id starting with **100**, followed by **101**, **102**, **103**, etc.

Upper and lower portions of the resulting spider can be placed into separate blocks with the **upper_spider_*** and **lower_spider_*** options. (Default is to put the spider joint into a single block.) Names and ids of these upper and lower blocks can be controlled with ***_upper_*** and ***_lower_*** forms of the block name/id parameters detailed in the preceeding paragraph.

preview: optional argument to display a preview of the spider operation without execution of the reduction. This option will display the proposed beam mesh with the surrounding volumes displayed in wireframe.

Examples

The following figure illustrates the **spider** option for the **reduce bolt** command.



(a) (b)

Initial CAD geometry

```
reduce volume 1 2 bolt spider
J2G mesh mesh_size 3.0
spider_block_ID 10
increment_Spider_Block_ID
rebar_block_ID 20
```

Figure 4. Examples of using the reduce bolt spider j2g command where (a) is the original geometry of two bolts overlapping the lower volume (cyan color). In (b), the spider j2g option has been used which replaces the bolts with a beam mesh. Note that the bolt hole has been modified to fit the shaft of the original bolts. In this case, the rebar blocks are comprised of the central nodes from which the beam elements radiate. Since the increment_rebar_block_id option is used, block 10 is defined where volume 1 existed and block 11 at volume 2. For the spider block, since the increment_spider_block_id was not used, all beam elements were placed into block 20. Note that since the mesh option was used, the hole surfaces were meshed and the spider block beam elements were generated. Otherwise, a subsequent mesh operation performed on the lower (cyan) volume would also generate the beam elements and assign them to their appropriate blocks.

Reduce Spring

The **Reduce Spring** command is intended to prepare a volume identified as a **spring** for analysis by creating one or more free curves that follow the spring's path at the middle of its cross-section. The new curve(s) can be meshed with beam elements and used in place of 3D elements to represent the spring.

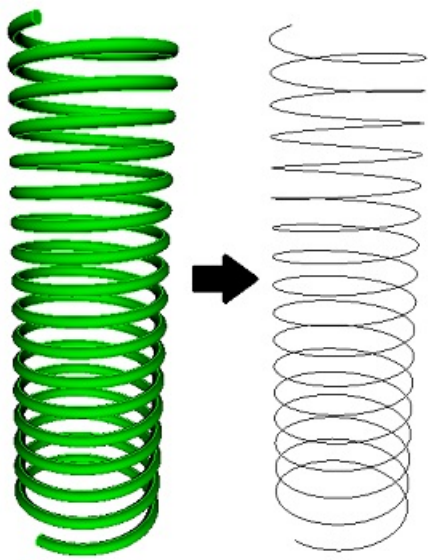


Figure 1. Example before and after of the Reduce Spring command

Syntax:

Reduce Volume <ids> Spring [combine] [mesh [size {<value>}]] [keep]
[block_id {<value>|Default}] [increment_block_id] [block_name {<string>|Default}] [preview]

Discussion:

volume ids: Specify the ids of the volumes to be reduced. The Geometry Power Tool classification diagnosic can be used for identifying volumes as "springs".

combine: By default, the curves resulting from the **reduce spring** command will be based upon the initial surface geometry of the spring. This may result in multiple curves of varying lengths connected by vertices. The **combine** option reduces the spring to a single free curve. This is helpful if the default operation would otherwise result in short curves that would produce unacceptable beam elements.

mesh: Use the **mesh** option to automatically mesh the resulting curve(s) with beam elements. The optional **size** argument can be used to specify a target length for the beam elements along the curve(s). If a **size** is not specified, an automatic size is assigned.

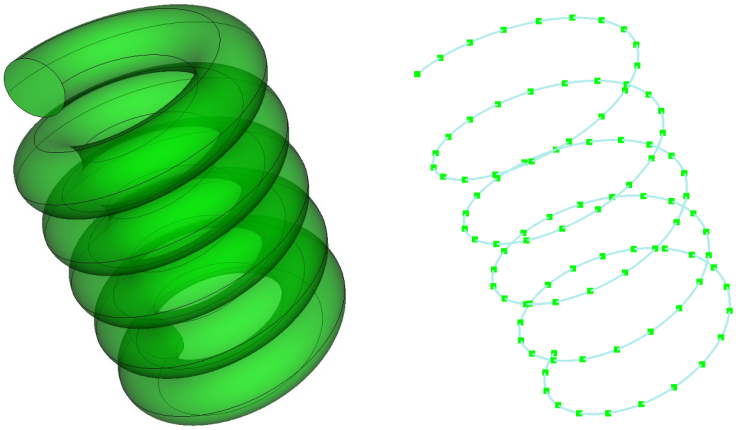
keep: Optionally retain the initial spring geometry after the reduce command is completed. If **keep** is not used, the spring volume will be deleted when a valid set of mid-curves has been produced.

Block ID and name assignment:When the **block_id** or **block_name** options are used, the resulting curves and mesh may be assigned to a block. The block may be defined by either a **block_name** or **block_id**. If the block does not yet exist, a new one will be created. The **Default** option will automatically select an id or name.

increment_block_id:When reducing multiple spring volumes in a single **reduce** command, it may be desirable to change the block ID assignment with each spring. When assigning new curves or beam elements to blocks, the block ID can be automatically generated by incrementing from a specified **block_id**. For example, if **block_id** is defined as **100**, and the **increment_block_ids** option is used, each new set of mid-curves and their associated beam elements generated for a unique spring volume will be assigned to a new block id starting with **100**. The next spring volume will be block **101**, followed by **102**, **103**, etc.

Examples

The following figure illustrates the **spring** option for the **reduce** command.



(a) **(b)**

Initial CAD geometry

reduce volume 1 spring combine mesh size 0.001 block_ID Default

Figure 2. Example of using the reduce spring command where (a) is the original geometry of a spring. In (b), the spring option has been used which in this case replaces the spring volume with a single free curve meshed with beam elements. A new block is generated and assigned the resulting curve and beam elements and the original spring geometry is deleted leaving only the free curves in place of the volume.

Reduce Thin Volumes

The **Reduce Thin Volumes** commands simplifies a 3D thin volume into a connected set of sheet bodies. The **reduce thin** commands are frequently used with the intention of generating shell finite elements. Note that these commands will perform similar geometric operations to the **surface copy** and **midsurface** commands, but will also keep track of **thickness** and **loft**; attributes necessary for building a full representation for shell finite element analysis. The resulting sheet bodies will also automatically generate blocks with these attributes and will maintain their association to their original 3D parent geometry.

The **Thin Auto** version (illustrated in figure 1) automatically reduces a set of 3D thin volumes into a connected set of sheet bodies based on an internal geometric reasoning algorithm resulting in sheet bodies that are connected at shared (merged) curves. It will also automatically build thickness, loft and block information. The **copy** and **midsurface** versions of the command are more prescriptive, and offer more control so that specific required reductions can be performed. The **copy** option can be used to prescribe a few required reductions in an assembly prior to automatically reducing the remainder with the **reduce thin auto** command.

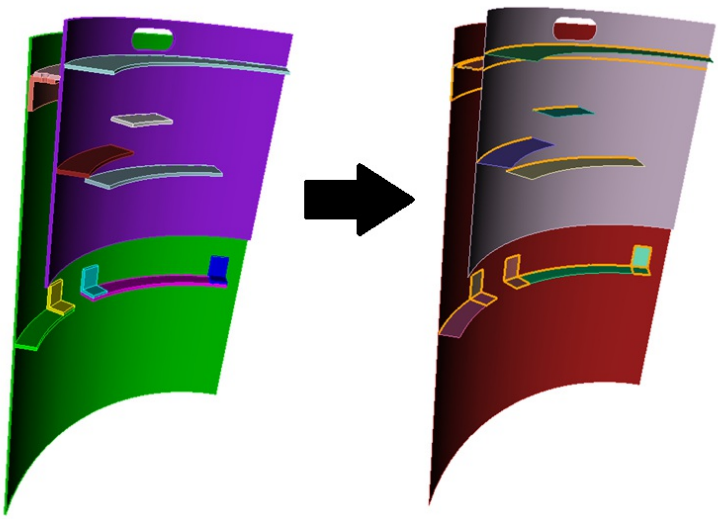


Figure 1. Reducing a set of 3D thin volumes. Merged curves are highlighed.

Syntax:

Reduce Volume <volume ids> Thin Auto [Sort] [Preview]

Reduce Volume <volume ids> Thin Copy <surface ids> loft factor <values> thickness <values>...[Combine] [Delete] [Preview]

Reduce Volume <volume id> Thin Midsurface surface <id1, id2> loft factor <value> thickness <value> [Combine] [Delete] [Preview]

Thin Auto

To use the **Thin Auto** command, the 3D thin volumes must be touching so that if the user imprinted and merged them, they would be connected at merged surfaces. This routine traverses the set of 3D volumes, finding the best 2D reductions that preserve connections between the 3D volumes. This routine also creates shell element blocks containing the resulting 2D shell surfaces. Each block also is attributed with appropriate thickness and loft factor values corresponding to the parent 3D volumes. These **thickness** and **loft** factor values are the first and second block

attributes respectively. Finally the command creates an internal association between the original 3D parent volume and the 2D reduction so that the user is able to visually inspect and validate the reduction. See the **draw shell volume** command below for more details.

volume ids: Specify the ids of the 3D volumes to be reduced. These volumes should not already be imprinted and merged. However, users should be certain that the volumes do indeed imprint and merge successfully since the routine does this internally to ultimately connect the resulting 2D shell surfaces. If a user wants to prescribe a specific reduction solution, it can be done manually using the **Reduce Thin Copy** command. To include the manually reduced volumes in the **auto** operation, the parent 3D volumes (not the generated 2D reductions) should be specified along with the other 3D volumes being reduced.

sort: If the **sort** option is specified, the surfaces of all 2D reductions containing the same base name will be added to the same block. This is done so that copies of the same volume remain together. These volumes should be identical in nature, with the same thickness. For example, if the user has a repeated part in the model with the base name 'bracket', all 2D reductions with that base name, like 'bracket@A', 'bracket@B', 'bracket@C',... will have all their surfaces added to the same block. If the **sort** option is not used, the surfaces of each 2D reduction will be in their own block. In either case, the next available block id is used.

It is also important to note that only 3D thin volumes that have two logical sides can be automatically reduced. For example, a piece of sheet metal, bent or pressed into most shapes has two logical sides. 3D thin volumes with t-junctions do not have two logical sides. See figure 2 below for some examples.

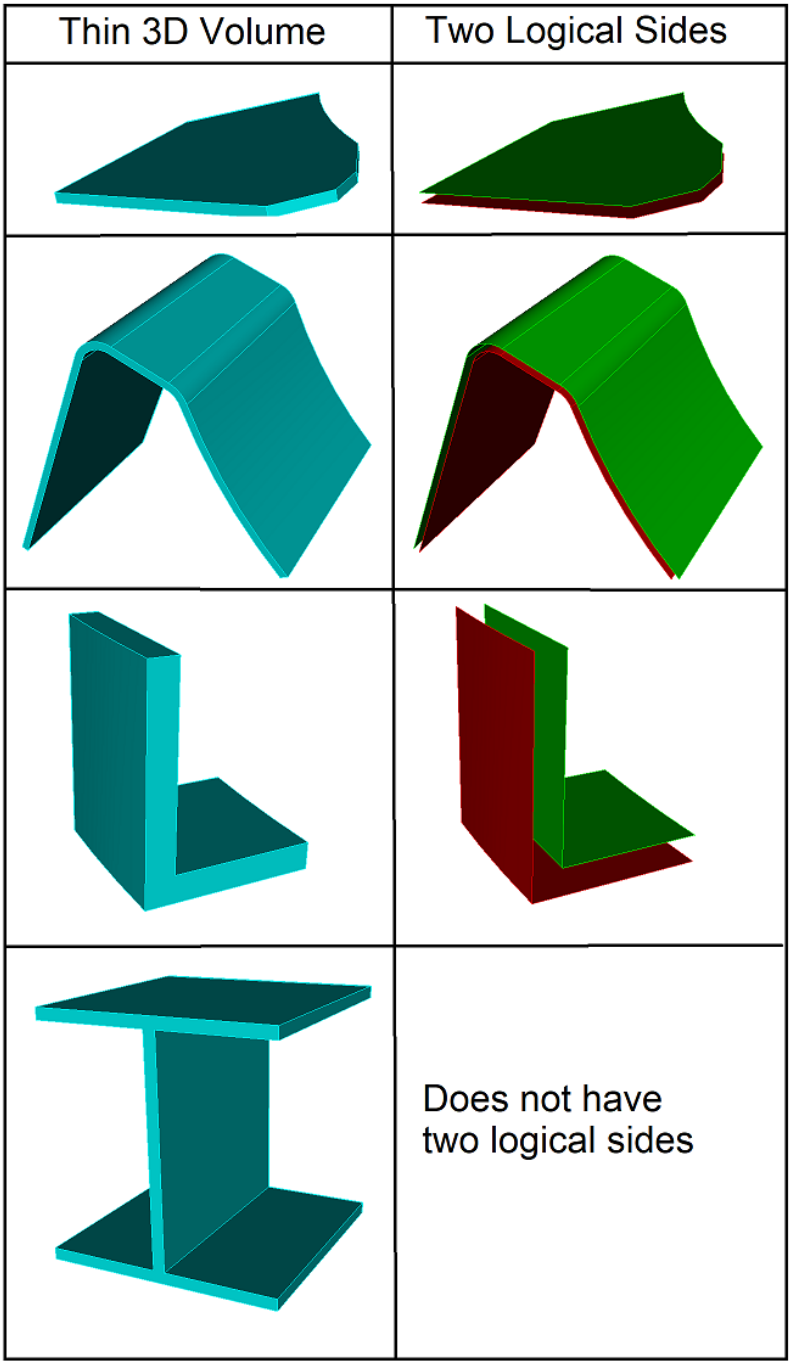


Figure 2. Logical sides of thin volumes.

Thin Copy and Midsurface

Using this form of the command, the user defines the reduction by specifying the surfaces the thin 3D volume will be reduced to. Unlike the **auto** form, nothing is done to make adjacent reductions contiguous. Each surface must be specified with a corresponding **loft** factor and **thickness** value. The **combine** option reduces a volume into a multi-surface sheet body. Otherwise, each surface will result in a single-surface sheet body. **delete** option deletes the original 3D thin volume(s).

Note that sheet bodies created using the \textbf{reduce thin midsurface} option, are currently not supported in the **auto** option. If the intention is to initially prescribe a few reductions prior to using the **auto** command, avoid using the **midsurface** command.

Draw Shell Volume

To visually inspect that the 2D reductions correctly approximate the original 3D thin volumes, the command below can be used. The 3D

volume is drawn in wireframe mode, 2D reduction in shaded mode, and the element block of the 2D reduction in shaded or optionally transparent mode.

Draw Shell Volume <ids> [Color<color_spec>] [Transparent] [Add]

Exporting Shell Data

To export information to a csv file detailing the reductions, the command below can be used:

Export Shell Data <string> [Volume<ids>] [Brief] [Overwrite]

By default, the following columns are written for each 3D volume specified. If no volumes are specified, the data for all 3D thin volumes reduced are exported. For each 3D thin volume, the following columns of data are exported:

sheet body ID	sheet body name	parent volume ID	parent volume name	thickness	loft	block

If the **brief** option is used, only the following columns of data are exported:

block	sheet base name	thickness	loft

Reduce Thin Volumes with Reinforcement Learning

The **Reduce Thin Volumes RL** command, similar to the **Reduce Thin Volumes *Auto*** simplifies a 3D thin volume into a connected set of sheet bodies. The **reduce thin volume** commands are frequently used with the intention of generating shell finite elements. In contrast to the **auto** option, the **RL** option uses machine learning methods to build a persistent knowledge base to improve its choice of reduce operations. If an assembly has not yet been *learned*, the method may initially predict a relatively poor solution. As the RL method is run, it builds training data, effectively learning the most suitable reduce solutions for a given configuration of thin volumes. In general, the more diverse problems encountered, the better the outcomes of the RL predictions.

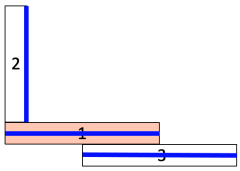


Figure 1.

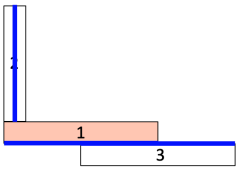


Figure 2.

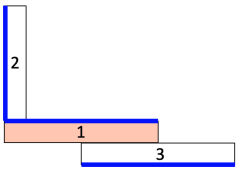


Figure 3.

When generating a shell model of a 3D set of thin volumes, the user often must navigate many tools to reduce the volumes and then establish connections. The sequence of commands to build such a model, for anything other than a trivial case, can become unwieldy. Figures 1-3 show one such example where multiple different solutions can be generated for this simple idealized model of the thin volumes. It should be apparent that some solutions may be acceptable, while others are not. The RL tool assists in developing a knowledge base of assembly states and how best to resolve them when encountered.

The **RL** option can be used to generate a sequence of Cubit commands, which can be exported to a journal file. The user can then validate the result, edit and annotate the journal for archival purposes. The commands generated for reduction will be of the form **reduce thin copy** or **reduce thin midsurface**. Note that these commands will perform similar geometric operations to the **surface copy** and **midsurface** commands, but will also keep track of **thickness** and **loft**; attributes necessary for building a full representation for shell finite element analysis. The resulting sheet bodies will also automatically generate blocks with these attributes and will maintain their association to their original 3D parent geometry.

Syntax:

Reduce Volume <volume ids> Thin RL [Number Iterations <value>] [Initialize Random [<value>]] [Stopping Criteria <value>] [Learning Interval <value>] [Journal Result <string>] [Delete] [Preview]

Preparation

Similar to the **Reduce Thin Auto**, to use the **Reduce Thin RL** command, for best results, the 3D thin volumes should be touching so that if the user imprinted and merged them, they would be connected at merged surfaces. This may involve using checking for gaps and overlaps

and resolving prior to using this tool. Some cleanup and defeaturing of the volumes may also be necessary, such as removing chamfers, rounds or other small features that may not be relevant to the final FEA model.

Strategies

The RL tool allows for learning and exploration of the state space so that a knowledge base (training data) can be established. It can also be used for faster prediction of reduction operations once a sufficient knowledge base has been established. In all cases the user is able to validate and experiment with the result using the journal file produced.

Predict Mode: This mode is normally used when sufficient training data has been established. By setting the argument **Number Iterations = 1**, the RL method will build the sequence of reduce commands based on predictions made from the existing training data.

Learning Mode: This is normally used when working with a new assembly that may be significantly different than previous models encountered. The user will typically run the RL methods for multiple iterations (**Number Iterations > 1**), allowing the method to gather information about the state space of the assembly. Depending on the complexity of the assembly, this may take anywhere from a few minutes to several hours to gather sufficient data. At each iteration, a single *average reward* value is computed and displayed at the command line that represents the completeness of the resulting solution. This value is a heuristic measurement of how well the resulting sheet bodies connect as well as other factors including potential introduction of small curves and narrow surfaces.

To operate in *learning mode*, RL will use various measures to determine when to terminate learning:

1. **Stopping Criteria:** A value between zero and one. If the average reward meets or exceed this value. Default value is 0.999.
2. **Number Iterations:** Maximum number of iterations to perform. Default value is 100.
3. **No Additional Unique Solutions:** In some cases, espcially for smaller models, the RL procedure will have attempted all possible combinations of reduce operations, and may terminate prior to reaching the stopping criteria or number of iterations.
4. **User Abort:** If the process appears to be taking too long, the use may hit the red (x) in the Cubit GUI to abort the command prior to reaching stopping criteria or number of iterations. If this occurs, whatever the RL method has learned to date will be retained.

User Influenced Learning: The RL tool provides an ideal method for efficiently establishing a lot of training data in a short amount of time. The *learning*, however is built on internal predefined heuristics that generally apply to most assembly states. The user can over-ride these preferences by adding their own training data. The most convenient way to do that is through the Cubit Geometry Power Tool, using the **Thin Volumes** diagnostic. This diagnostic will display the current predicted confidence value for a given **reduce** command in the solution window. This can be influenced by the user by using the **maximize** or **minimize confidence** right-click options. These options are also available from the Cubit command line using the **Learn** series of commands.

Arguments

Number Iterations <value>: Maximum number of RL iterations to perform before termination. Default is 100.

Initialize Random {<value>}: Used to initialize the state to random **reduce** solutions. Normally the state is initialized by predicting the best solution based on the current training data. Random initialization can be used for maximum exploration of the state space, and is most useful for

models that have not been seen before. It can also be used to experiment with known solutions to see if alternative solutions can be derived. The options **<value>** is a *seed* that can be used to guide the internal random number generator.

For maximum exploration of the state space in learning mode, the **Initialize Random** may also used. Normally, the starting point for the RL method will be a prediction based on the current knowledge base (training data). The Initialize Random options, ignores any current

Stopping Criteria <value>: A value between zero and one. The RL method terminates if the average reward meets or exceed this value. Default value is 0.999.

Learning Interval <value>: Integer value that indicates how often to update the machine learning model. For example, a learning interval of 5 will update the training model with information it has learned at every fifth RL iteration. Following the update, the state rewards will be reinitialized based on predictions from the ML model to begin the next iteration. Default for **learning interval** is 100. If the default is used, normally the ML model is only updated after the final iteration.

Journal Result "<string>": If a filename is provided using this option, a journal file with that name will be written to the current working directory with the resulting sequence of commands from the *best* RL iteration. The resulting journal file will be annotated with comments indicating reward values and success or failure of connections.

Delete: Parent volumes will be deleted after the sheet bodies have been generated. This is generally not recommended as the association between 3D solid thin volumes and their resulting child sheet body(s) is maintained behind the scenes in Cubit. Visualization and management of sheet data, such as block attributes is maintained through this association, as well as the **Export Shell Data** command.

Preview: If the preview option is used, the final best solution will not be executed, but rather a graphical preview of the result will be displayed. The preview will also include visual cues for connections that have been maintained and those that do not. In addition, if the **journal result** is used, the journal file will also be produced. This option is often used to preview and validate the result, prior to running the resulting journal once the solution is established as acceptable.

Reduce Slot Surface

The **Reduce Slot Surface** is used to decompose slot surfaces in Electromagnetic (EM) modeling for easier application of boundary conditions.

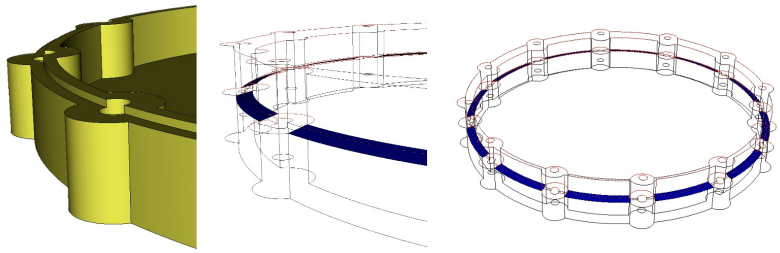


Figure 1.Example preview of slot surfaces, shown in blue (right) constructed from the original geometry (left).

Syntax:

Reduce Surface <ids> slot [radius {<value>|Factor <value>|SHIGLEY [angle {<value>}]}] [inner {curve <ids>}] [outer {curve <ids>}] [hardware volume <ids>] [[create] skin] [group <string>] [group_edges <string>] [group_inner_edges <string>] [group_outer_edges <string>] [group_inner_surfaces <string>] [group_outer_surfaces <string>] [name <string>] [name_edges <string>] [name_inner_edges <string>] [name_outer_edges <string>] [name_inner_surfaces <string>] [name_outer_surfaces <string>] [make_free_curves] [preview]

Discussion:

The **Reduce Surface Slot** command is engineered for the preparation of models for electromagnetic (EM) simulations, facilitating the creation of slots—pathways allowing EM radiation to traverse. This command is versatile, designed for compatibility with both Cubit quad and tri meshers and the Morph mesher. It plays a vital role in generating surface meshes essential for EM simulations.

Prerequisites and Best Practices:

Before invoking the **Reduce Surface Slot** command, adhere to these best practices to ensure optimal model preparation:

1. **Defeaturing and Simplification:** Simplify the model by defeaturing or removing unnecessary details that may complicate splitting procedure in the **reduce slot** command.

2. **Enclosure Integrity:** Verify that the enclosure region is fully closed. Use the **remove surface** commands as necessary to seal any gaps or holes.

3. **Imprint and Merge:** For mismatched mating surfaces above and below the slot pathways, perform an **imprint** and **merge** operation to ensure that the upper and lower slot surfaces align perfectly.

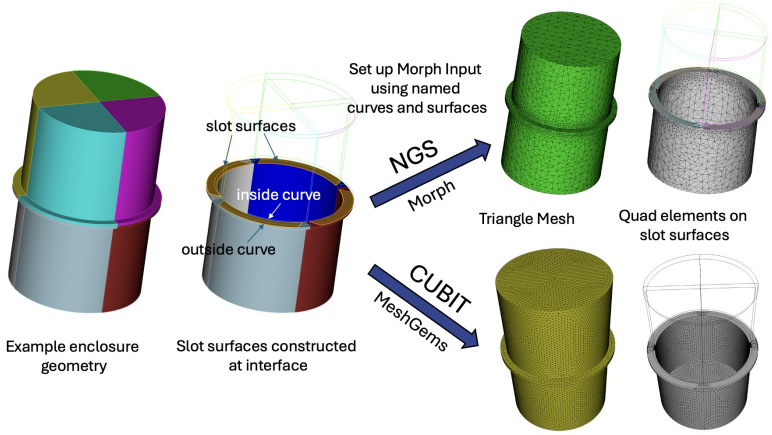


Figure 2.Slot Preparation Workflow: Identification and division of slot surfaces via the reduce surface slot command. The images display the distinct meshing results with Cubit or Morph.

Common Steps for Cubit and Morph Workflows:

1. **Adjust Options:** Customize the command settings according to the provided command syntax description.

2. **Invoke the Command:** Execute the Reduce Surface Slot command to begin the process of defining and splitting the slot surfaces. It's also worth noting that achieving the desired outcome may require some trial and error, with a **preview** option available to visualize the slot pathways before finalizing.

Cubit Workflow:

For Cubit, the focus is on direct mesh generation:

1. **Mesh Preparation:** Unlike Morph, Cubit does not require specific naming of surfaces and curves for mesh generation.

2. **Quad Meshes on Slot Surfaces:** Explicitly generate quad meshes on the slot surfaces using Cubit commands for assigning intervals, setting schemes and meshing.

3. **Tri Meshes for Remaining Surfaces:** Apply tri meshers for the rest of the model, setting mesh size, scheme, and then meshing directly in Cubit.

Morph Workflow:

For Morph, the emphasis is on preparing data for meshing in an external process:

1. **Surface and Curve Naming:** Important for Morph, where naming conventions are essential for the mesher to recognize and process the model correctly. Names can be explicitly identified in the command or the default naming convention may be used.

2. **Export Patch Data:** Utilize the **export slot data** command to provide Morph with the necessary information for quad and tri mesh generation.

Command Syntax Description:

surface ids: Specifies the slot surfaces designated for decomposition. Multiple surfaces can be included in the selection, provided they form a coherent, continuous loop. These surfaces are allowed to incorporate disruptions like holes, which accommodate hardware or fasteners, ensuring component cohesion. Importantly, these surfaces must delineate the region intended for the electromagnetic radiation pathway.

hardware volume ids: Identifies the volumes of fasteners that intersect

with the slot surface. The centroid of each fastener, as it appears on the slot surface, guides the determination of cut locations. This parameter is optional; in its absence, the command automatically seeks out holes, inferring potential fastener centroids based on the centers of these holes.

radius: Specifies the distance from the centers of fasteners on the slot surface to where cuts will be made. The radius can be defined using one of three methods:

1. **Explicit Definition:** Directly sets the radius to a specific value.
2. **Factor:** Determines the radius as a multiplier of the bolt or hole dimensions, applicable when bolts or holes are present.
3. **Shigley Method:** Calculates the radius based on a specified angle, the bolt head radius, and the thickness of the clamped member. This method is applicable only in the presence of hardware fasteners.

The process aims to prevent the creation of sliver geometries. However, users should exercise caution to prevent cuts that produce overly small curves or surfaces. A tolerance of 10% of the specified radius is applied, allowing cuts to be adjusted to the nearest vertex to mitigate the risk of slivers or tangent conditions.

group: Used to create and name a group containing all resulting decomposed surfaces. The options **group_edges**, **group_inner_edges**, **group_outer_edges**, **group_inner_surfaces** and **group_outer_surfaces** are used to create and name groups of cutting curves, inner curves, and outer curves, respectively.

name: Similar to the group option, but individual names are assigned to the resulting decomposed surfaces and curves. The names of decomposed surfaces, side curves, inner curves, outer curves, inner surfaces and outer surfaces can be defined using the respective options. If not specified, a default naming convention will be used. Note that these names will be used in the Morph input deck and can be written using the **export slot data** command

make_free_curves: A specialized option to generate free curves at the boundaries of the decomposed slot surfaces. Grouping and naming options also apply to the free curves.

create skin: This function constructs a sheet body through a lofting process that spans from inner to outer curves. It is applicable to any slot surface, offering significant utility when dealing with multiple surfaces that do not form coplanar paths. This feature is particularly valuable for defining surfaces for quad meshing, effectively outlining the electromagnetic (EM) pathway. An alternative to having this command create a skin surface is for the user to create a skin surface and tweaking nearby surfaces to match the skin surface. If done this way, the tweak operation will preserve any bolt holes, and a **reduce surface slot** operation can be performed afterwards.

preview: Displays a blue preview of the slot paths without performing the actual cutting operations as well as a wire frame outline of the owning volume

Export Slot Data:

The **Export Slot Data** command is used to list or export the information required for a Morph input deck. It can either provide a preview in the output window or export the data to a text file if a filename is specified. This command assumes the use of default naming conventions. The syntax is as follows:

Export Slot Data [surface] [full] [free_curves] [overwrite]

Export Slot Data [surface] [full] [free_curves] preview

There are two different forms of the command. The first form exports the data to a file, while the second form provides a preview in the output window.

Optional arguments:

- **surface :** This optional argument allows you to specify the surfaces for which you want to export Morph slot data. If not specified, data for all surfaces with the expected naming convention will be written or previewed.
- **full:** In addition to the slot data, this option writes out the full Morph input deck. Otherwise, it will only write the section of the input deck related to slots.
- **free_curves:** Use this option when the **make_free_curves** option is used on the reduce slot command. It ensures that the curves are named appropriately, taking into account the existence of free curves in the model.
- **overwrite:** When exporting to a file, this option allows automatic overwrite of any file with the same name.

Reduce Bolt Patch

The **Reduce Bolt** command generates a proxy representation of a **bolt** for analysis. It replaces the bolt geometry with two concentric circular surfaces centered on the bolt axis, separating the connected volumes where sidesets are automatically applied.

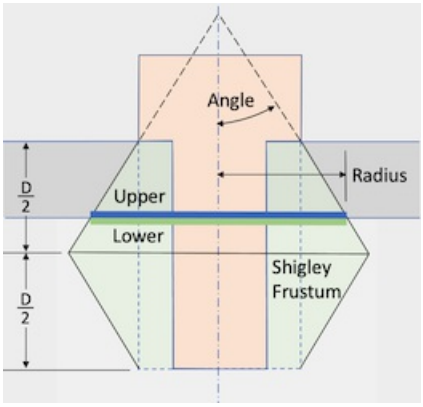


Figure 1. Example of the Reduce Bolt Patch command. The Shigley frustum is used to calculate the diameter of the sidesets positioned between the volumes.

Syntax:

```
reduce {volume <ids>|upper surface <ids> lower surface
<ids>} {bolt|hole} patch
  [contact upper {surface <ids>}] [contact lower {surface
<ids>}]
  [radius {<value>|Factor <value>|SHIGLEY [angle
{<value>}] [grip {AUTO|bolt|washer}]}]
  [mesh] [mesh_size <value>] [mesh_scheme <string>]
[no_simplify_hole]
  [{FILL|no_fill}] [{DELETE|no_delete}] [array_name
<string>]
  [sideset_naming_convention <value>]
  [upper_patch_sideset_id {<value>|Default}]
  [increment_upper_patch_sideset_id]
  [start_upper_sideset_id {<value>|Default}]
  [upper_patch_sideset_name {<string>|Default}]
  [lower_patch_sideset_id {<value>|Default}]
  [increment_lower_patch_sideset_id]
  [start_lower_sideset_id {<value>|Default}]
  [lower_patch_sideset_name {<string>|Default}]
[name_patch_surfaces]
  [preview]
```

Discussion:

The **Reduce Bolt Patch** command is often used in structural dynamic applications to replace bolts or bolt holes with concentric circular sidesets attached to the upper and lower volumes. If the specified radius exceeds surface bounds, clipping occurs. The resulting circular surfaces are automatically assigned to customizable sidesets. The patch radii are determined using the default Shigley frustum angle, as illustrated in Figure 1. Users can modify the Shigley angle or set the diameter explicitly as an absolute value or a factor of the shank radius. The command also supports automatic meshing of the sidesets with a user-defined scheme and size.

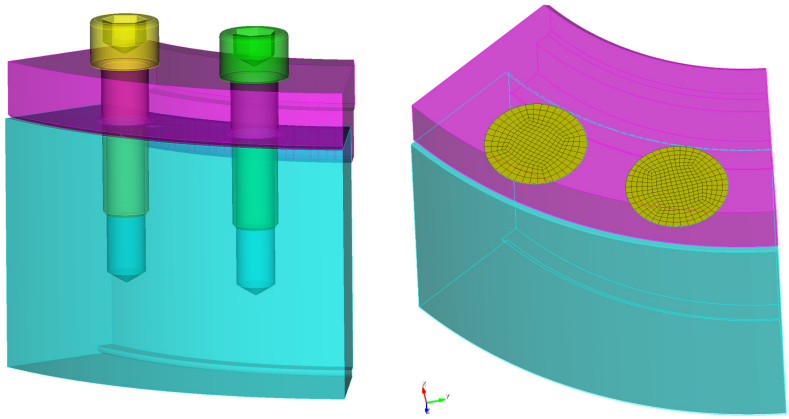


Figure 2. Example of before and after where the reduce bolt patch command has been applied to two bolts. This example uses the mesh and default fill and delete options.

The following outlines the options for the **reduce bolt patch** command.

You can input either the ID of the bolt to be reduced or the IDs of holes intended for fasteners if no bolt volume is present. Choose between the **bolt** or **hole** options based on your specific needs.

bolt

volume <ids>: Specify the ids of the volumes to be reduced. The Geometry Power Tool classification diagnosic can be used for identifying volumes as "bolts".

hole

upper surface <ids> lower surface <ids>: Specify hole IDs for fastener reduction, including at least one surface from both upper and lower holes. Cubit checks axis alignment and matches multiple holes if specified. For more than two fastened volumes, Cubit supports up to three; designate the topmost with **>upper surface** and the rest with **lower surface**.

Optional Arguments

contact upper {surface <ids>} contact lower {surface <ids>}: In rare cases, the **reduce patch** command may not auto-identify upper and lower contact surfaces. If this occurs, you can manually specify these surfaces to resolve the issue. Note: This option is only available for a single bolt or hole set.

radius {<value>|Factor <value>|SHIGLEY [angle {<value>}] [grip {AUTO|bolt|washer}]}

radius <value>: The **Reduce Bolt Patch** command allows you to directly set the radius by specifying a floating-point value. Use this option to set an absolute value for the sideset radius on the contact surfaces. This value determines the radius of the resulting imprinted circle where the sideset is defined.

radius factor <value>: The **radius factor** option allows you to define the radius of the resulting imprinted circle as a multiple of the shank bolt's radius. If no bolt is present and only the hole is specified, the radius will be a multiple of the upper hole's radius.

radius shigley angle <value>: The **radius shigley angle** option is the default method for determining the radius. It employs a Shigley angle, as outlined in SAND2008-0371 (Figure 1). If no radius specification is provided, the Shigley angle defaults to 30 degrees to compute the radius. This radius is determined by the intersection of the frustum with the surface between the upper and lower volumes.

grip {AUTO|bolt|washer}: Use this option in combination with the **SHIGLEY** radius method. It specifies whether the Shigley cone calculation is based on the bolt head or the washer outer radius.

- **AUTO** (default): If a washer is present in the bolt–washer assembly, its outer radius is used; otherwise, the bolt head radius is used. This follows the usual convention of basing Shigley calculations on the washer when available.
- **BOLT**: Always use the bolt head radius, ignoring any washer present.
- **WASHER**: Always use the washer outer radius. If no washer is present, an error is issued.

Because the washer radius is normally larger than the bolt head radius, specifying **WASHER**—or using **AUTO** when a washer is present—will typically yield a larger circular patch radius.

The circular patch size can vary based on several factors, such as the thickness of the upper volume, the radius of the bolt head, and the length of the bolt. For more comprehensive details, refer to SANDIA REPORT SAND2008-0371, “Guideline for Bolted Joint Design and Analysis: Version 1.0.”

Note: The **radius shigley angle** option is applicable only when a bolt is specified. If no bolt is present and only holes are specified, and no radius is given, the **radius factor** option becomes the default. In such cases, a default factor of 2.5 will be used to compute the radius.

The **Reduce Bolt Patch** command imprints concentric circular patches onto the surfaces of the upper and lower volumes. If the surfaces on the upper and lower volume do not extend beyond the specified radius, the resulting surfaces will be clipped accordingly.

mesh: Option to specify whether to include meshing as part of the command.

mesh_size <value>: Optionally specify a target mesh size when using the **mesh**

mesh_scheme <string>: Specify a target meshing scheme. The following meshing schemes can be used for this operation:

- **circle** - Uses Cubit's circle scheme. If the **no_fill** option is used, the **hole** scheme will automatically be used. Meshes with quads.
- **pave** - Uses Cubit's paving scheme to mesh with quads at the specified size.
- **tri** - Uses triangle meshing at the specified size.

FILL|no_fill: The **FILL|no_fill** option controls hole treatment. By default setting, **FILL** removes the hole entirely, along with any features like fillets. The **no_fill** option retains the hole but simplifies it, removing fillets and creating an annulus centered on the hole's axis. The annulus diameter is set by one of the **radius** options.

DELETE|no_delete: The default behavior of this command is to delete the bolt as part of the operation. However, users have the flexibility to choose whether or not to delete the bolt.

no_simplify_hole: The **reduce surface patch** command, coupled with the **no_fill** option, is designed to streamline the meshing process by automatically eliminating chamfers, blends, and conical elements from hole geometries. However, the **no_simplify_hole** switch overrides this behavior, preserving all existing features of the hole's geometry. This can be particularly useful in scenarios where the default simplification process might lead to command failure due to complex hole configurations. Activating this option allows for the continuation of the operation without the standard simplification step.

array_name: This option requires a string value that will be employed as

a prefix for all resulting sidesets which are created by the command. When the **name_patch_surfaces** option is simultaneously activated, this prefix is also applied to the names of any generated surfaces. Therefore, the prefix ensures consistent naming conventions across sidesets and, conditionally, surfaces, streamlining subsequent identification and processing. Note that if the **array_name** is not explicityly specified, the name prefix "Bolt_Patch" will be prepended to the name of each of the sideset and surface names.

Sideset IDs and name assignment: The following options may be used to assign the resulting circular surfaces based on an ID or a sideset name:

- **upper_patch_sideset_id {<value>|Default} [increment_upper_patch_sideset_id][start_upper_sideset_id {<value>|Default}][upper_patch_sideset_name {<string>|Default}]**
- **lower_patch_sideset_id {<value>|Default} [increment_lower_patch_sideset_id][start_lower_sideset_id {<value>|Default}][lower_patch_sideset_name {<string>|Default}]**

Specify sidesets using either **sideset_name** and/or **sideset_id**. If not existing, a new one will be created. Using **Default** selects the next available ID(s) or names them as "Upper_Patch" and "Lower_Patch" if unspecified.

Optional **increment** and **start_id** arguments are available for both upper and lower sideset ID specifications. When used, these options automatically generate incrementing sideset IDs. For example, if **lower_patch_sideset_id** is set to **100** and **increment_lower_patch_sideset_id** is used, new bolt patches will have unique sideset IDs starting from 100 (e.g., 101, 102, 103). If both **upper_patch_sideset_id** and start_upper_sideset_id (or their lower counterparts) are used with **increment**, the new patches may be added to the existing sideset and a new, incrementing sideset, effectively adding the same patch to two different sidesets.

name_patch_surfaces: This option allows you to name the surfaces created by the **reduce patch** command. You can use either default or custom names. Default names are "Upper_Patch" and "Lower_Patch". If you specify names using **upper_patch_sideset_name** or **lower_patch_sideset_name**, those names will instead be applied to the resulting circular surfaces. This is helpful for referencing in future scripts or journal commands.

Sideset Naming Convention

Default Naming: Sidesets are automatically named to reflect the connections between components. For instance, for clamped volumes with names "PartA" and "PartB" generates sideset names such as "PartA_to_PartB_123" and "PartB_to_PartA_123." These names encapsulate the parts involved and a unique identifier for the fastener's volume, "_123" in this case. In this example, the name "PartA_to_PartB_123" signifies the sideset on PartA that denotes its interface with PartB, and similarly, "PartB_to_PartA_123" is assigned to PartB to mark its connection with PartA. When blocks are designated and the clamped volumes are assigned, the naming convention defaults to using block names rather than volume names.

Default Naming Format:

```
array_name + "_" + [NameA] + "_" + [NameB] + "_" + [Volume ID of Bolt]
```

where **NameA** designates the name of the volume or block the sideset is attached to and **NameB** designates the volume or block the sideset is in

contact with.

Naming Examples:

If **array_name** is set to "Connect", the volume ID of the bolt is 3, and the clamped volume names are "PartA" and "PartB" the resulting sideset names for the resulting sidesets would be:

- `Connect_PartA_to_PartB_3`
- `Connect_PartB_to_PartA_3`

Alternative Default Naming: Activating the option with **sideset_naming_convention** set to **1** changes the naming scheme. Sidesets begin with a "Bolt_Patch" prefix (assuming no **array_name** is given), then add "_Lower" or "_Upper" to indicate their position relative to the bolt's bearing surface, and conclude with a numeric identifier for the bolt volume. This convention offers an alternative to the default, which is applied automatically without needing explicit selection.

Alternative Naming Format:

```
array_name + "_" + [Volume ID of Bolt] + "_Upper/Lower"
```

Naming Examples:

If **array_name** is set to "Connect" and the volume ID of the bolt is 3, the resulting sideset names for the upper and lower portions would be:

- `Connect_3_Upper`
- `Connect_3_Lower`

This naming applies to surfaces as well when the **name_patch_surfaces** option is activated. In cases where a single bolt connects more than two volumes, the naming includes the individual connections. For example, a bolt fastening three volumes would yield names such as:

- `Connect_1_3_Upper`
- `Connect_1_3_Lower`
- `Connect_2_3_Upper`
- `Connect_2_3_Lower`

Here, the first number represents the index of the connection at the same bolt, and the second number is the volume ID of the bolt.

Overriding the Default:

Be aware that these default naming conventions can be superseded by the explicit naming options, **upper_patch_sideset_name** and **upper_patch_sideset_name**, provided in the previous section of this documentation.

Export Patch Data Command

This section outlines the use of the **export patch data** command, which allows users to save or preview information derived from the **reduce bolt patch** process. This command facilitates the generation of data necessary for Morph input decks or simulation purposes, including the creation of a .csv file. Below is the detailed syntax for utilizing this command.

> **Export Patch Data <filename> [{csv|morph|BOTH}] [surface <ids>] [lower name <name>] [upper name <name>] [sideset_naming_convention <value>] [full] [overwrite]**
>
> **Export Patch Data [surface <ids>] [lower name <name>]**

> **[upper name <name>] [sideset_naming_convention <value>] [full] preview**

The command is available in two variations: one for writing the output to a file and another for previewing the morph patch data in the output window. Both variations require the prior use of the **name_patch_surfaces** option within the **reduce bolt patch** command to identify and process the surfaces correctly.

Command Options and Arguments

- **csv|morph|BOTH**: Specifies the output format. By default, both CSV and Morph input data are written to a file. If both are selected, the base filename provided is used for the Morph input file, with a .csv extension appended for the CSV file. Selecting **csv** generates only a CSV file, including a header with named patch surfaces listed as pairs, the distance between these pairs, and the x-y-z coordinates of the contact pairs' center. Choosing **morph** outputs only the Morph input data, containing essential information for the Morph tet meshing tool.
- **surface <ids>**: Allows specification of one or more surfaces for output. If omitted, the command writes data for any surfaces meeting the sideset patch data criteria.
- **sideset_naming_convention <value>**: Defines the naming convention used in the reduce bolt patch command. It is crucial to use the same convention as previously defined. A value of zero or not using this option defaults to the convention outlined above. A value of **1** indicates an alternative naming convention.
- **[lower name <name>] [upper name <name>]**: Applicable when **sideset_naming_convention=1** is used. These options assume a naming convention for upper and lower patches based on the specified strings.
- **full**: When used, additional data necessary for a comprehensive definition of the Morph input deck is included. Without this option, only basic patch data is written.

Debugging Geometry

The following command checks for inconsistencies in the CUBIT topological model, by checking the specified entities and all child topology and/or comparing to solid model topology:

```
Geomdebug Validate [compare] <entity_list>
```

This command checks for:

- Consistent CoFace senses
- Loops are closed/complete
- Consistent CoEdge senses
- Correct vertex order on curves w.r.t. parameterization
- Correct tangent direction of curves w.r.t. parameterization

Related Commands:

```
Geomdebug Vertex <vertex_id>

Geomdebug Curve <curve_id>

Geomdebug Surface <surface_id>

Geomdebug body <body_id>

Geomdebug Containment {Curve | Surface} <id>
{Location (options) | Node <id_list>}
```

The following command prints info about GeometryEntities owned by specified entity:

```
Geomdebug Geometry <entity_list> [interval
<n>] [index <n>] [TEXT] [GRAPHIC] [attributes]
```

The following command lists (TopologyBridge) topology for specified entity:

```
Geomdebug solidmodel <entity_list> [index <n>]
[depth<n>|up<n>|down<n>]
```

The following command lists GroupingEntities.

```
Geomdebug GPE <entity_list>
```

Finding Surface Overlap

The surface overlap capability finds surfaces that *overlap* each other, with the capability to specify a distance and angle range between them. This is useful for debugging geometry imprinting and merging problems, as well as for finding gaps in large assembly models. Finding overlapping geometry is done using the command:

```
Find [Surface] Overlap [{Body|Surface|Volume}
<id_list> [Filter_Sliver]
```

If a list of entities is not specified, all bodies in the model are checked. By default the command does not check the surfaces within a given body against each other; rather, it only checks surfaces between bodies. This can be overridden by inputting a surface list (i.e. **find overlap surface all**), or with a setting (see below).

The **filter_sliver** option will remove false positives from the list by weeding out sliver surfaces that have a merged curve between them. The following pictures is an example of a sliver surface.

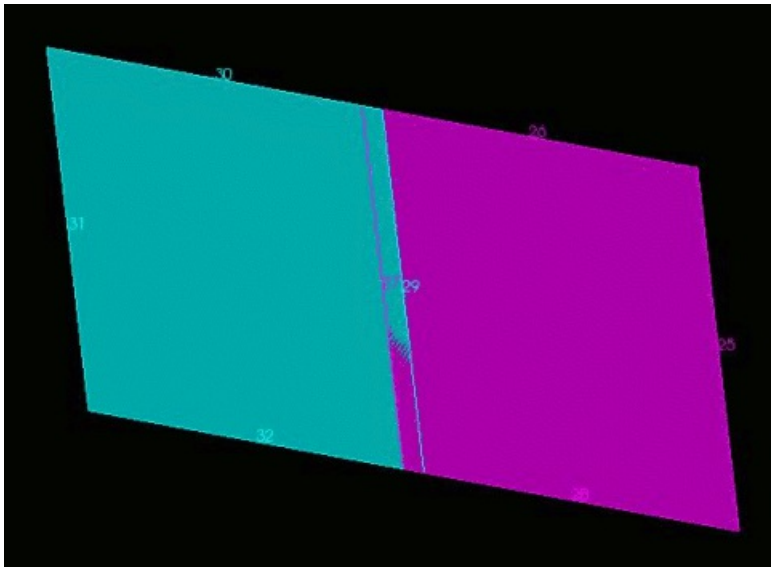


Figure 1. Example of a sliver surface

If curves 27 and 29 are merged before you run the find overlapping surface checkthe user will get the two surfaces in the picture as an overlapping surface pair. However, if the *filter_sliver* keyword is used, Cubit will not find the two surfaces to be overlapping.

Facetted Representation

This command works entirely off of the facetted surface representation of the model (the facetted representation is what you see in a shaded view in the graphics). There are inherent advantages and disadvantages with this method. The biggest advantage is avoidance of closest-point calculations with NURBS based geometry, which tends to be slow. This method also eliminates possible problems with unhealed ACIS geometry. The disadvantage is working with a less accurate (i.e., facetted) representation of the geometry. To circumvent problems with this facetted geometry, various settings can be used to control the algorithm. For example, you might consider using a more accurate facetted representation of the model - see below.

Find Overlap Settings

Various settings are used to control the precision and handling of overlaps during the find overlap process. A listing of the settings that find overlap uses is printed using the command:

```
Find [Surface] Overlap Settings
```

These settings, and the commands used to control them, are described below.

Facet - Absolute/Angle - The angular tolerance indicates the maximum angle between normals of adjacent surface facets. The default angular tolerance is 15 - consider using a value of 5 . This will generate a more accurate facetted representation of the geometry for overlap detection. This can be particularly useful if the overlap command is not finding surface pairs as you would expect, particularly in "curvy" regions. Note however that the algorithm will run slower with more facets. The distance tolerance means the maximum actual distance between the generated facets and the surface. This value is by default ignored by the facetter - consider specifying a reasonable value here for more accurate results.

```
Set Overlap [Facet] {Angle|Absolute} <value>
```

Gap - Minimum/Maximum - the algorithm will search for surfaces that are within a distance from the minimum to maximum specified. The default range is 0 to 0.01. Testing has shown this to be about right when searching for coincident surfaces. Gaps can be found by using a range such as 3.95 to 5.05.

```
Set Overlap {Minimum|Maximum} Gap <value>
```

Angle - Minimum/Maximum - the algorithm will search for surfaces that are within this angle range of each other. The default range is 0.0 to 5.0 degrees. Testing has shown that this range works well for most models. It is usually necessary to have a range up to 5.0 degrees even if you are looking for coincident surfaces because of the different types of faceting that can occur on curvy type surfaces. For example, for the case of a shaft in a hole, the facets of the shaft usually won't be coincident with the facets of the hole, but may be offset by a certain distance circumferentially with each other. The 5 degree max angle range will account for this. If you find that the algorithm is not finding coincident surfaces when it should, you can increase the upper range of this value. Note that this parameter is useful also for finding plates coming together at an angle.

```
Set Overlap {Minimum|Maximum} Angle <value>
```

Normal - this setting determines whether to search for surfaces whose normals point in the same direction as each other (**same**), away from each other (**opposite**) or either (**any**). The default is ANY, but it may be useful to limit this search to *opposite*, as this would be the usual case for most finds.

```
Set Overlap Normal {ANY|opposite|same}
```

Tolerance - two individual facets must overlap by more than this area for a match to be found. Consider the two cylindrical curves at the interface of the shaft and the block in Figure 2. Note that some of the facets actually overlap, even though the curves will analytically be coincident. You can filter out false matches by increasing the overlap tolerance area. The default value for this setting is 0.001.

```
Set Overlap Tolerance <value>
```

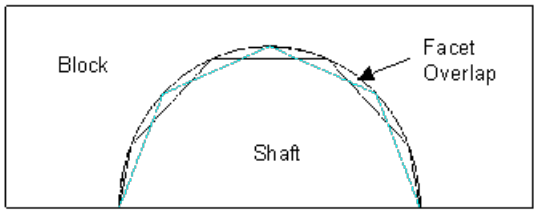


Figure 2. Possible false find due to overlap (tolerance will prevent finding match)

Group - the surface pairs found can optionally be placed into a group. The name of the group defaults to "overlap_surfaces".

```
Set Overlap Group {on|OFF}
```

List - by default the command lists out each overlapping pair - this can be turned off using the command:

```
Set Overlap List {ON|off}
```

Display - by default the command clears the graphics and displays each overlapping pair - this can be turned off using the command:

```
Set Overlap Display {ON|off}
```

Body - by default the command will not search for overlapping pairs within bodies - only between different bodies. Turn this setting on to search for pairs within bodies. Note however that this will slow the algorithm down.

```
Set Overlap [Within] {Body|Volume} {on|OFF}
```

Imprint - If on, Cubit will imprint the overlapping surfaces that it finds together. This will often force imprints that just imprinting bodies together will miss. For each pair of overlapping surfaces, the containing body of one surface is imprinted with the individual curves of the other surface, until the resulting surfaces no longer overlap.

```
Set Imprint {on|OFF}
```

Geometry Accuracy

The accuracy setting of the ACIS solid model geometry can be controlled using the following command:

```
[set] Geometry Accuracy <value = 1e-6>
```

Some operations like imprinting can be more successful with a lower accuracy setting (i.e., 0.1 to 1e-5). However, it is not recommended to change this value. ***Be sure to set it back to 1e-6 before exporting the model or doing other operations as a higher setting can corrupt your geometry***.

Regularizing Geometry

The regularize command removes unnecessary topology, which in effect reverses the imprint operation. This can help clean up the model from extra features that are unnecessary for the geometric definition of the model. The following command regularizes the model:

```
Regularize
Body|Group|Volume|Surface|Curve|Vertex <range>
[keep {curve <ids>|vertex <ids>}]
```

The **keep** option allows the user to specify curves and vertices that should not be removed during the regulairze operation.

If you are frequently using web-cutting or other boolean operations to decompose your geometry, it may be convenient to always generate regularized geometry. To set creation of regularized geometry during boolean operations use the following command:

```
Set Boolean Regularize [ON | off]
```

Stitching Sheet Bodies

The stitch command stitches together the specified sheet bodies into either a larger sheet body or a solid volume(s). The tolerance value can be used when these sheet bodies don't line up exactly along the edges. This is common for IGES and STEP models. Only manifold stitching is performed, i.e., edges will be shared with no more than two surfaces.

```
Stitch {Body|Volume} <id_range> [Tolerance
<value>] [No_simplify] [No_tighten_gaps]
[Restricted]
```

This command has three stages to it:

1. **Stitch the surfaces together along overlapping edges** Normally IGES and some STEP files do not contain topological information that links surfaces together to share bounding curves. Stitching is an operation that builds up this topological information.
2. **Simplify geometry** The command replaces splines with analytics where possible.
3. **Tighten up gaps (inaccuracies) between the sheet bodies** The command will build the geometry necessary to tighten the gaps in the model.

When the stitch operation completes, a print statement lets the user know if the resulting body is not a closed, solid body.

The user can choose to omit the second and third options of stitching with the **no_simplify** and **no_tighten_gaps** options respectively. This may be necessary in very large or complex models, where the regular approach fails, or takes an inordinate amount of time.

The **restricted** option limits the stitching operation to the boundary curves of the specified sheet bodies or volumes. All non-boundary curves are ignored by the stitch algorithm. This functionality is intended as a performance enhancement.

Trimming and Extending Curves

Curves can be trimmed or extended with the following command:

```
Trim Curve <id> AtIntersection {Curve|Vertex
<id>} Keepside Vertex <id> [near]
```

Curves can be trimmed or extended where they intersect with another curve or at a vertex location. When trimming to another curve, the curves must physically intersect unless they both are straight lines in which case the **near** option is available. With the **near** option the closest intersection point is used to the other line - so it is possible to trim to a curve that lies in a different plane. When trimming to a vertex, if the vertex does not lie on the curve, it is projected to the closest location on the curve or an extension of the curve if possible.

The **Keepside** vertex is needed to determine which side of the curve to keep and which side to throw away. This vertex need not be one of the curve's vertices, nor does it need to lie on the curve. However, if it is not on the curve it will be projected to the curve and that location will determine which side of the curve to keep.

If the curve is part of a body or surface, it is simply copied first before trimming/extending. If it is a free curve a new curve is created and the old curve is removed. The figures below show several examples of trimming/extending curves.

Trimming a Curve

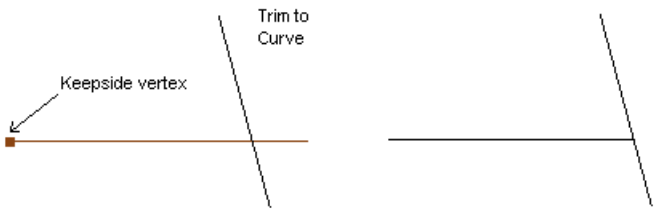


Figure 1. Trimming a Curve to an Intersecting Curve

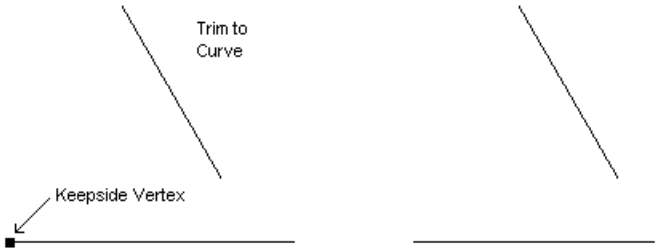


Figure 2. Trimming a Curve to a Non-Intersecting Curve Using the Near Option

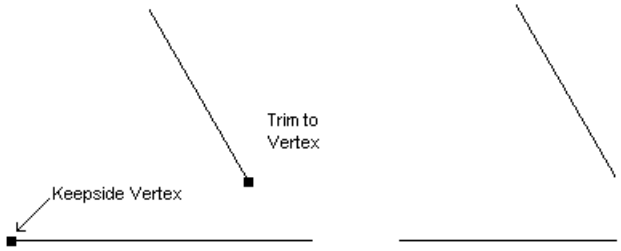


Figure 3. Trimming a Curve to a Vertex

Extending a Curve

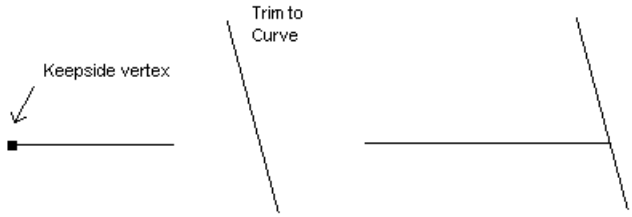


Figure 4. Extending a Curve to An Intersecting Curve

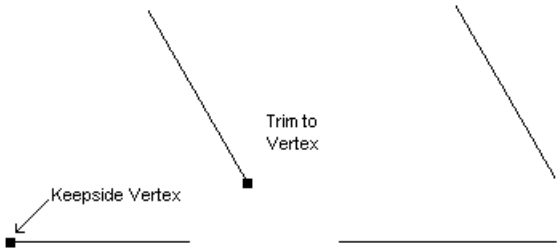


Figure 5. Extending a Curve to a Non-Intersecting Vertex Using the Near Option

Validating/Analyzing Geometry

Detailed checks of geometry and topology can be performed using the validate command:

```
Validate
{Body|Volume|Surface|Curve|Vertex|Group}
<id_range> [level val=<value=20>] [verbose]

Healer Analyze Body <id_range> [level val=
<value=20>] [verbose]
```

These two commands traverse the given entity's topological tree structure, checking for bad data or invalidities along the way. Both commands do exactly the same thing. More extensive diagnostics can be ran by specifying a higher check level. The check levels are value multiples of 10 between 0 and 70 inclusive. Use the **level** option of the command to specify the level (how extensive) of the validation. The **verbose** option prints additional details of any problems.

where **value** is one of the following:

10 = Fast error checks

20 = Level 10 checks plus slower error checks (default)

30 = Level 20 checks plus D-Cubed curve and surface checks

40 = Level 30 checks plus fast warning checks

50 = Level 40 checks plus slower warning checks

60 = Level 50 checks plus slow edge convexity change point checks

70 = Level 60 checks plus face/face intersection checks

```
Validate {Volume|Surface|Curve|Vertex} <range> Mesh
```

The **Validate {...} mesh** command performs a connectivity check of the mesh elements to determine the validity of the mesh.

The validate command can also check for consistent surface normals and return a list of offending surfaces. The syntax for the command is as follows:

```
Validate [Body] <body_id> Normal [Reference
[Surface] <surface_id>] [Reverse]
```

Using the "reference" keyword, a reference surface is compared to the normal consistency of all other specified surfaces. Inconsistent surfaces can be reversed using the "reverse" keyword.

Blunt Tangency

The blunt tangency commands are used to eliminate small angles in the model caused by fillets. The operation 'blunts' the tangency, or small angle, using two different approaches. The first replaces the tangency with a pair of surfaces, essentially moving the vertex to a location with a larger angle. The second approach creates a surface with a larger radius through a tweak operation, increasing the sharp angle at the vertex.

```
Blunt tangency vertex <id> [remove_material]
[composite] [angle <value>] [depth <value>]
[preview]
```

```
Blunt tangency tweak vertex <ids> [angle
<value>] [offset surface <id>] [tolerance
<value>] [preview]
```

As shown in Figure 1, the **depth** parameter controls the depth of the surfaces that replace the tangency, while the **angle** parameter controls the resultant angle of the new tangency. Figure 2 demonstrates the behavior of the **remove_material** option which removes material instead of adding it.

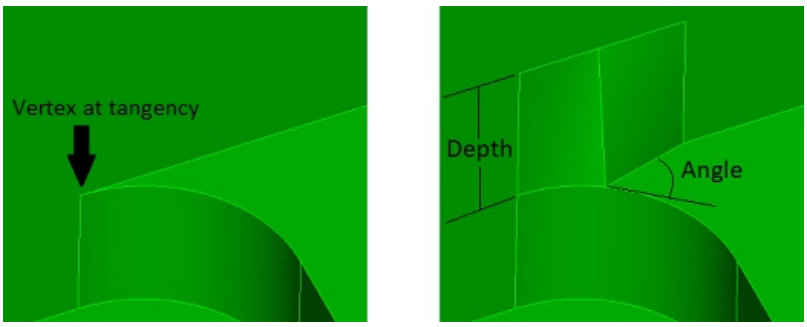


Figure 1. Blunt Tangency Operation



Figure 2. Remove Material Option

The **composite** option will composite the surfaces created in the blunt tangency operation with the adjacent larger surface, from which they were cut out. See figure 3.

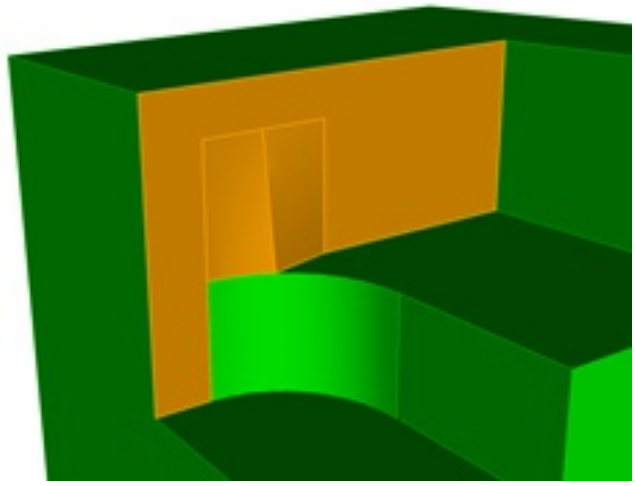


Figure 3. Blunt Tangency with Composite Option. Highlighted surface is resulting composite surface.

The second form of the command replaces the surface with the radius at the vertex with a larger radius surface, increasing the angle. The command tweaks the surface that results in the least amount of material removed (Figure 4) or added (Figure 5). The **angle** option works the same as in the first form of the command. The **offset surface** option specifies which surface is tweaked (offset), as sometimes the surface that results in the smallest material added/removed is not correct. The **tolerance** option prevents small curves from being created, snapping to adjacent vertices within the tolerance (Figure 6) or possibly modifying an adjacent surface (Figure 7).

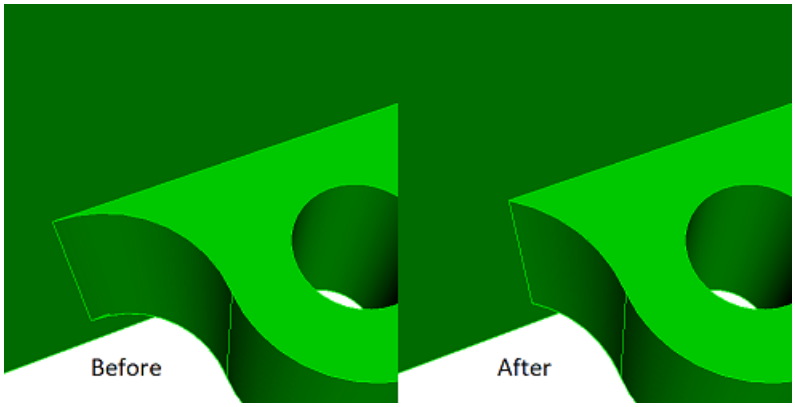


Figure 4. Blunt Tangency Tweak -- Material Removed

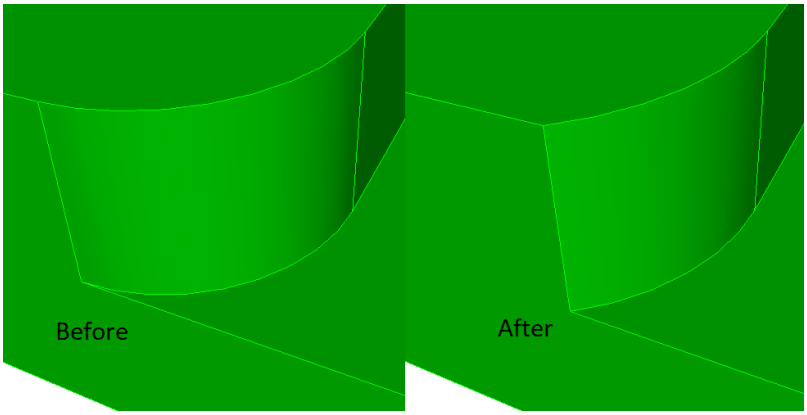


Figure 5. Blunt Tangency Tweak -- Material Added

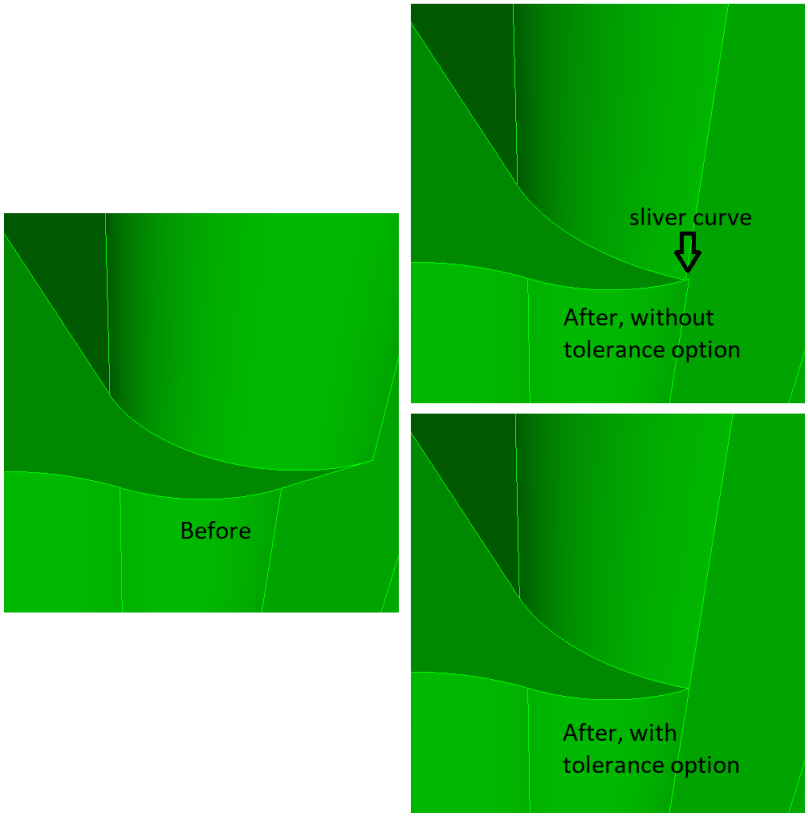


Figure 6. Blunt Tangency Tweak -- Tolerance

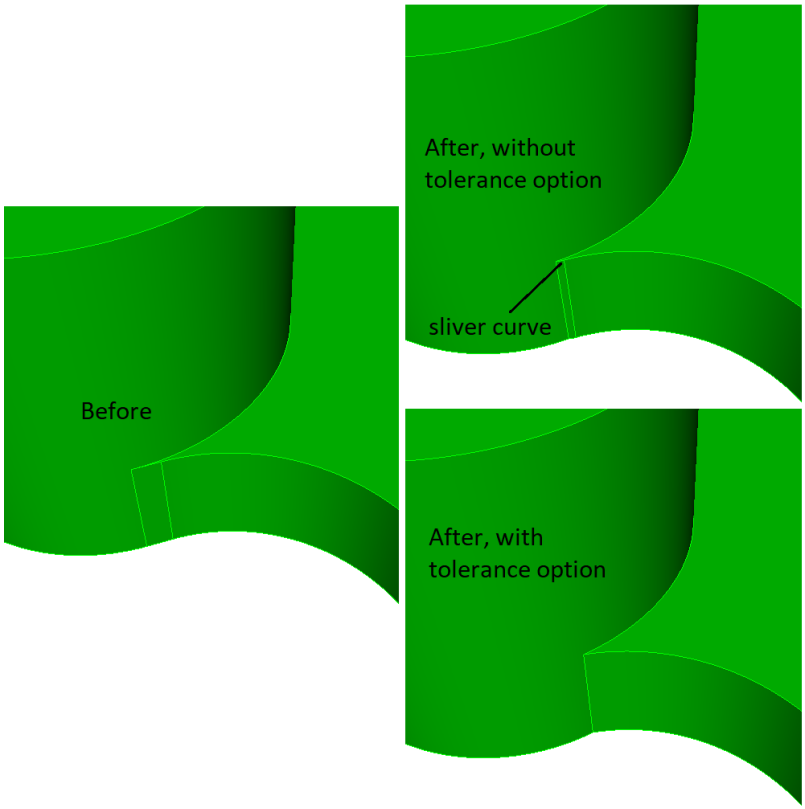


Figure 7. Blunt Tangency Tweak -- Tolerance

Geometry Imprinting and Merging

- Imprinting Geometry
- Merging Geometry
- Examining Merged Entities
- Merge Tolerance
- Unmerging
- Using Geometry Merging to Verify Geometry

Geometry is created and imported in a manifold state. The process of converting manifold to non-manifold geometry is referred to as "geometry merging", since it involves merging multiple geometric entities into single ones. When importing mesh-based geometry, the merging step can be automatic. Imprinting is a necessary step in the merging process, which ensures that entities to be merged have identical topology.

Examining Merged Entities

There are several mechanisms for examining which entities have been merged. The most useful mechanism is assigning all merged or unmerged entities of a specified type to a group, and examining that group graphically. This process can be used to examine the outer shell of an assembly of volumes, for example to verify if all interior surfaces have been merged. To put all the merged entities of a given type into a specified group, use the command:

```
Group {<`name'>|<id>} add [Surface | Curve |
Vertex] with Is_merged
```

To put all the unmerged entities of a given type into a specified group, use the command:

```
Group {<`name'>|<id>} add [Surface | Curve |
Vertex] with Is_merged=0
```

Entities can also be labeled in the graphics according to the state of their merge flag. See the Preventing geometry from merging section for information on controlling the merge flag. To turn merge labeling on for a specified entity type, use the command

```
Label {Vertex | Curve | Surface} Merge
```

Imprinting Geometry

To produce a non-manifold geometry model from a manifold geometry, coincident surfaces must be merged together (See Geometry Merging); this merge can only take place if the surfaces to be merged have like topology and geometry. While various parts of an assembly will typically have surfaces, which coincide geometrically, an imprint is necessary to make the surfaces have like topology. There are three types of imprinting:

- Regular Imprinting
- Tolerant Imprinting
- Mesh-Based Imprinting

To preview which surfaces can or should be imprinted, or to force imprints that the regular imprint command misses, the Find Overlap command can be used.

Regular Imprinting

The commands used to imprint bodies together are:

```
Imprint [Volume|BODY] <range> [with
[Volume|BODY] <range>] [Keep]
```

A body can also be imprinted with curves, vertices or positions, and surfaces can be imprinted with curves. It is useful to imprint bodies or surfaces with curves to eliminate mesh skew, generate more favorable surfaces for meshing, or create hard lines for paving. Imprinting with a vertex or position can be useful to split curves for better control of the mesh or to create hard points for paving. Impriting a vertex onto a volume allows for a tolerance to be specified, snapping the vertex to the closest location on the volume that is within tolerance.

```
Imprint Body <body_id_range> [with] Curve
<curve_id_range> [Keep]

Imprint Body <body_id_range> [with] Vertex
<vertex_id_range> [tolerance <value>] [Keep]

Imprint {Volume|Body} [with] Position <coords>
[position <coords> ... ]

Imprint Surface <surface_id_range> [with]
Curve <curve_id_range> [Keep]
```

An **Imprint All** will imprint all bodies in the model pairwise; bounding boxes are used to filter out imprint calls for bodies which clearly don't intersect.

```
Imprint [Body] All
```

Tolerant Imprinting

Normal imprinting may be ineffective for some assembly models that have tolerance problems, generating unwanted sliver entities or missing imprints altogether. Tolerant imprinting is useful for dealing with these tolerance challenged assemblies. To determine coincident and overlap entities, tolerant imprinting uses the merge tolerance. The commands also include an optional **tolerance** value that will be used for the purposes of the single command. Specifying an optional **tolerance** value will not change the default, system tolerance value.

A limitation of tolerant imprinting is that it cannot imprint intersecting surfaces onto one another, as normal imprinting can. Tolerant imprinting

imprints only *overlapping* entities onto one other.

```
Imprint Tolerant {Body|Volume} <range>
[tolerance <value>]
```

Tolerant imprinting can also be used to imprint curves onto surfaces, provided that the tolerance between surface and curve(s) falls within the merge tolerance. The 'merge' option will merge the owning volume of the specified surface with all other volumes that share any curves with this surface.

```
Imprint Tolerant Surface <id> with Curve
<id_range> [merge]  [tolerance <value>]

Imprint Tolerant Surface <id> <id> with Curve
<id_range> [merge]  [tolerance <value>]

Imprint Tolerant Surface <id> <id> [tolerance
<value>]
```

The second form of the command imprints the specified bounding curves of one surface onto another surface and vice versa. Any specified curves that are not bounding either of the two specified surfaces will not be imprinted. The 'merge' option will merge all the volumes sharing any curve of these two surfaces, after the imprint.

It is recommended that normal imprinting be used when possible and tolerant imprinting be used only when normal imprinting fails.

Mesh-Based Imprinting

Another form of the imprint command,

```
Imprint Mesh {Body | Volume} <id_list>
```

uses coincident mesh entities and virtual geometry to create imprints. See the Partitioned Geometry section for more information on this command.

Imprint Settings

After imprint operations, an effort is made to remove sliver entities: sliver curves and surfaces. Previously, all curves in participating bodies less than 0.001 were removed. Newer versions of Cubit changed this because there might be times when the user wants sliver curves/surfaces to be generated during an imprint operation. In order to give the user more control over the cleanup of these sliver entities after imprint operations, a command was implemented so that the user can set an 'imprint sliver cleanup tolerance'. The default tolerance for curves is the merge tolerance 0.0005. The default tolerance for surfaces is a suitable tolerance chosen internally based on the bounding box of the entity. Sliver surfaces are removed whose maximum gap distance among the long edges is smaller than the tolerance and who have at most three long edges. A long edge is an edge whose length is greater than the specified tolerance.

```
Set {Curve|Surface} Imprint Cleanup Tolerance
<value>
```

Merge Tolerance

Geometric correspondence between entities is judged according to a specified absolute numerical tolerance. The particular kind of spatial check depends on the type of entity. Vertices are compared by comparing their spatial position; curves are tested geometrically by testing points 1/3 and 2/3 down the curve in terms of parameter value; surfaces are tested at several pre-determined points on the surface. In all cases, spatial checks are done comparing a given position on one entity with the closest point on the other entity. This allows merging of entities which correspond spatially but which have different parameterizations.

The default absolute merge tolerance used in CUBIT is 5.0e-4. This means that points which are at least this close will pass the geometric correspondence test used for merging. The user may change this value using the following command:

```
Merge Tolerance <val>
```

If the user does not enter a value, the current merge tolerance value will be printed to the screen. There is no upper bound to the merge tolerance, although in experience there are few cases where the merge tolerance has needed to be adjusted upward. The lower bound on the tolerance, which is tied to the accuracy of the solid modeling engine in CUBIT, is 1e-6.

Finding Nearly Coincident Entities

These commands find vertex-vertex, vertex-curve and vertex-surface pairs whose separation is within the specified tolerance range. If a tolerance range isn't specified the default will be from merge tolerance to 10*merge tolerance. It is useful for determining if you need to expand merge tolerance to accomodate sloppy geometry.

```
Find Near Coincident Vertex Vertex
{Body|Volume} <id_range> [low_tol <value>]
[high_tol <value>]

Find Near Coincident Vertex Curve
{Body|Volume} <id_range> [low_tol <value>]
[high_tol <value>]

Find Near Coincident Vertex Surface
{Body|Volume} <id_range> [low_tol <value>]
[high_tol <value>]
```

Merging Geometry

The steps of the geometry merging algorithm used in CUBIT are outlined below:

1. Check lower order geometry, merge if possible
2. Check topology of current entities
3. Check geometry of current entities
4. If both entities are meshed, check topology of meshes.
5. If geometric topology, geometry, and mesh topology are alike, merge.

Thus, in order for two entities to merge, the entities must correspond geometrically and topologically, and if both are meshed must have topologically equivalent meshes. The geometric correspondence usually comes from constructing the model that way. The topological correspondence can come from that process as well, but also can be accomplished in CUBIT using Imprinting.

If both entities are meshed, they can only be merged if the meshes are topologically identical. This means that the entities must have the same number of each kind of mesh entity, and those mesh entities must be connected in the same way. The mesh on each entity need not have nodes in identical positions. If the node positions are not identical, the position of the nodes on the entity with the lowest ID will be used in the resulting merged mesh.

There are several options for merging geometry in CUBIT.

Merge geometry automatically

```
Merge All [Group|Body|Surface|Curve|Vertex]
[group_results][tolerance <value>]
```

All topological entities in the model or in the specified bodies are examined for geometric and topological correspondence, and are merged if they pass the test.

If a specific entity type is specified with the Merge all, only complete entities of that type are merged. For example, if Merge all surface is entered, only vertices which are part of corresponding surfaces being merged; vertices which correspond but which are not part of corresponding surfaces will not be merged. This command can be used to speed up the merging process for large models, but should be used with caution as it can hide problems with the geometry.

Test for merging in a specified group of geometry

```
Merge {Group|Body|Surface|Curve|Vertex}
<id_range>[With
{Group|Body|Surface|Curve|Vertex} <id_range>]
[group_results] [force] [tolerance<value>]
```

All topological entities in the specified entity list, as well as lower order topology belonging to those entities, are examined for merging. This command can be used to prevent merging of entities which correspond and would otherwise be merged, e.g. slide surfaces.

Force merge specified geometry entities

```
Merge Vertex <id> with Vertex <id> Force

Merge Curve <id> with Curve <id> Force
```

```
Merge Surface <id> with Surface <id> Force
```

This command results in the specified entities being merged, whether they pass the geometric correspondence test or not. This command should only be used with caution and when merging otherwise fails; instances where this is required should be reported to the CUBIT development team.

Preventing geometry from merging

```
Body <id_range> Merge [On | Off]

Volume <id_range> Merge [On | Off]

Surface <id_range> Merge [On | Off]

Curve <id_range> Merge [On | Off]

Vertex <id_range> Merge [On | Off]
```

These commands provide a method for preventing entities from merging. If merging is set to off for an entity, merging commands (e.g. "merge all") will not merge that entity with any other.

Other Merge Commands

```
Set Merge Test BBox {on|OFF}
```

This is an additional test for merging to see if a pair of surfaces should merge. First, it creates a bounding box for each surface by summing individual bounding boxes of each of the surface's curves. A comparison is then made to see if these two bounding boxes are within tolerance. This can help to weed out any potential incorrect merges that can result from non-tight bounding boxes.

```
Set Merge Test InternalSurf {on|OFF|spline}
```

This is an extra check when merging surfaces. A point on one surface, closest to its centroid is found. Another point, closest to this point is found on the other surface. If these two points are not within merge tolerance, the two surfaces will not be merged. If set to **on**, all surface types will be included in this check. If set with the **spline** option, then splines are only checked this way; analytic surfaces are excluded. This is another check to prevent incorrect merges from occurring.

Using Geometry Merging to Verify Geometry

Geometry merging is often used to verify the correctness of an assembly of volumes. For example, groups of unmerged surfaces can be used to verify the outer shell of the assembly (see Examining Merged Entities.) There is other information that comes from the **Merge all** command that is useful for verifying geometry.

In typical geometric models, vertices and curves which get merged will usually be part of surfaces containing them which get merged. So, if a **Merge all** command is used and the command reports that vertices and curves have been merged, this is usually an indication of a problem with geometry. In particular, it is often a sign that there are overlapping bodies in the model. The second most common problem indicated by merging curves and vertices is that the merge tolerance is set too high for a given model. In any event, merged vertices and curves should be examined closely.

Unmerging

The unmerge command is used to reverse the merging operation. This is often in cases where further geometry decomposition must be done.

```
Unmerge {all|<entity_list> [only]}
```

Un-merging an entity means that the specified geometric entity and all lower-order (or child) entities will no longer share non-manifold topology with any other entities. For example, if a body is unmerged, that body will no longer share any surfaces, curves, or vertices with any other body.

```
[Set] Unmerge Duplicate_mesh {On|OFF}
```

If any meshed geometry is unmerged, the mesh is kept as necessary to keep the mesh of higher-order entities valid. For example, if a surface shared by two volumes is to be unmerged and only one of the volumes is meshed, the surface mesh will remain with whichever surface is part of the meshed volume.

When unmerging meshed entities, the default behavior of the code is that the placement if the mesh is determined by the following rules:

- If neither entity has meshed parent entities, the mesh is kept on one of the two entities.
- If one entity has a meshed parent entity, the mesh is kept on that entity.
- If both entities have meshed parents, the mesh is kept on one and copied on the other.

If **unmerge duplicate_mesh** is turned on, the rules described above are overwritten and whenever a meshed entity is unmerged the mesh is always copied such that both entities remain meshed.

To get back to the default behavior, turn **unmerge duplicate_mesh** off.

Virtual Geometry

- Composite Geometry
- Partitioned Geometry
- Collapsing Geometry
- Simplify Geometry
- Deleting Virtual Geometry

The Virtual Geometry module in CUBIT provides a way to modify the topology of the model without affecting the underlying ACIS geometry representation and without making changes to the actual solid model. Virtual Geometry includes the capability to composite or partition geometry as well as creates new virtual geometric entities. Virtual Geometry operations are most often used as a tool for adjusting the geometry to allow mapping, sub-mapping or sweeping mesh generation schemes to be applied.

The advantage to using Virtual Geometry is that all operations are reversible. With standard geometry modification commands, changes are made to the underlying geometry representation and cannot be changed once effected. With virtual geometry, the original solid model topology can be easily restored. This is useful when geometry modifications are made in order to apply a particular meshing scheme. Virtual geometry can be applied and later removed once the part has been meshed.

Collapse Angle

The collapse command allows the user to collapse small angles using virtual geometry. The command syntax is:

```
Collapse Angle at Vertex <id range> [angle
<degrees>] [Curve <id1> [Arc_length <length>]]
[Curve <id2> [Arc_length <length> | Same_size
| Perpendicular | Tangent]] [Composite_vertex
<angle>] [Preview]
```

The collapse angle command is used to eliminate small angles at vertices, where curves meet at a tangential point. The command will split each curve at a specified distance (δ1 and δ2) as shown in Figure 1, and create two new vertices along those curves. The remaining small angle will be composited into its neighboring surface using virtual geometry. One of two methods may be used for specifying options for the **collapse angle** command: **(1) Simple**, and **(2) Complete** as described below:

(1) Simple

Curves are not specified for this option. Instead, Cubit will automatically identify the smallest angle at the vertex and collapse it using the **tangent** option described below (see Figure 4). An optional **angle** option may also be specified that controls the arc length (δ1) along curve C1 where the curve will be split. If not specified a value of 30 degrees will be used. For the **simple** option of this command, a range of vertices may be used. The **complete** version of the command requires exactly one vertex and two curves.

(2) Complete

The complete options of the command allow you to specify which curves and where to split each curve. You must input a distance for the first curve (δ1), but the second location can be determined based on the length and direction of the first curve.

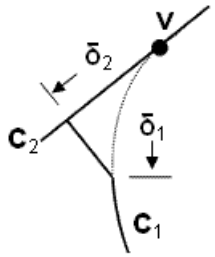


Figure 1. Collapse angle syntax

The **arclength** option will split each curve at a specified distance δ1 and δ2, (See Figure 1) measured from the vertex. You must input at least one **arclength** for each of the options listed below.

The **same_size** option will split curve 2 so that the two resulting curves, δ1 and δ2, are the same length as shown in Figure 2.

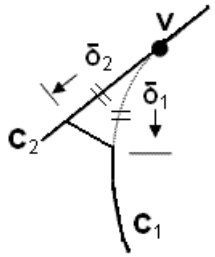


Figure 2. Collapse angle using the same_size option

The **perpendicular** option will split curve 2 so it is perpendicular to the split location on curve 1, as shown in Figure 3.

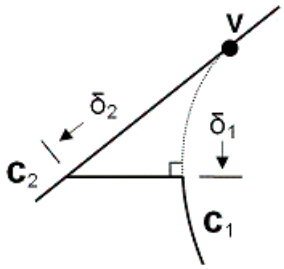


Figure 3. Collapse angle using the perpendicular option

The **tangent** option will split curve 2 where a line tangent to curve 1 at the split location intersects curve 2, as shown in Figure 4.

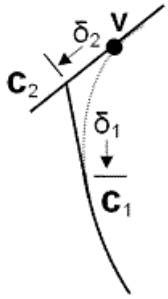


Figure 4. Collapse angle using the tangent option

The **composite_vertex** option automatically composites resulting surfaces if there are only two curves left at the vertex, and the angle is less than a specified tolerance.

The **preview** option will preview composited surface before applying changes.

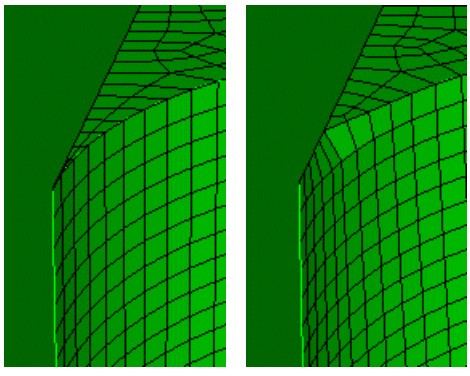


Figure 5. An example of a meshed surface that is generated after

using the collapse angle command.

Collapse Curve

The collapse curve command allows the user to collapse small curves using virtual geometry. It is intended to be used in cases where removing a small curve to simplify topology will facilitate meshing. The operation can be thought of as reconnecting curves from one vertex on the small curve to the other vertex. If the user doesn't specify which vertex to keep during the operation CUBIT will choose one of the vertices. The operation is performed using virtual partitions and composites on the curves and surfaces surrounding the small curve. The command syntax is:

```
Collapse Curve <id> [Vertex <id>] [Ignore]
[Real_split]
```

The **vertex** keyword allows the user to specify which vertex on the small curve to keep during the operation or in other words which vertex to "collapse to". Depending on the surrounding topological configuration some vertices cannot currently be chosen so if the user specifies a vertex to collapse to that results in a complex topological configuration that CUBIT can't currently handle the user will be notified and encouraged to pick a different vertex. If the user doesn't specify a vertex CUBIT will attempt to choose the "best" vertex to keep based on surrounding topology and geometry. Currently, the collapse curve command only handles curves where the vertex that is NOT retained has a valence of 3 or 4.

The **ignore** keyword allows the user to specify whether or not small portions of surfaces that are partitioned off of one surface and composited with a neighboring surface during the collapse curve operation are considered when evaluating the new composite surface. By specifying the **ignore** option the user tells CUBIT that these small surfaces will be ignored in future evaluations of the composite surface. This can be beneficial in cases where the small surface makes a sharp angle with the neighboring surface it is being composited with. These first derivative discontinuities of composite surfaces can make it difficult for the meshing algorithms to proceed and ignoring the small surfaces during evaluation can help remedy this problem. By default the small surfaces will not be ignored.

The **real_split** option tells CUBIT to use the solid modeling kernel's (ACIS) split surface functionality to do the splitting rather than using virtual partitioning. The result is that you only have virtual composites at the end and no virtual partitions. The main advantage of using this option is that the solid modeling kernel's split operation is often more reliable than the virtual partition.

Figure 1 shows a typical example where the collapse curve command should be used to simplify the topology for meshing.

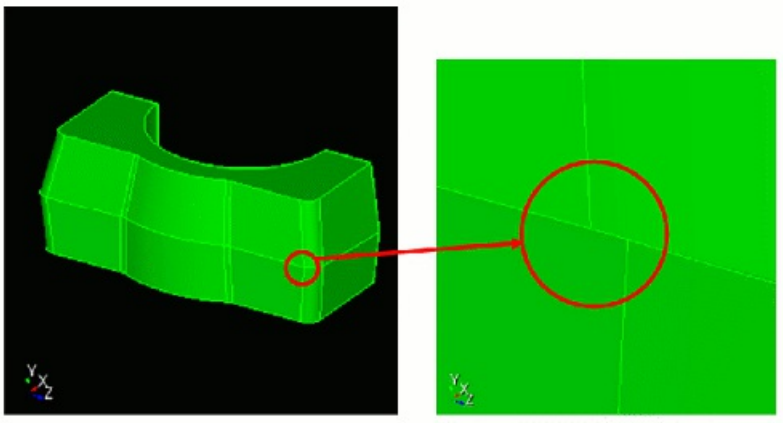


Figure 1. Example where the collapse curve operation is needed.

Figure 2 shows the above example after collapsing the small curve

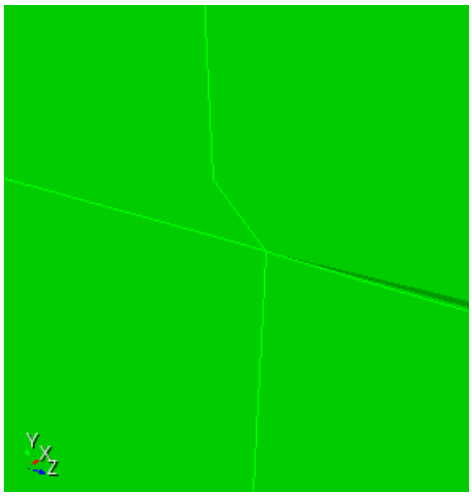


Figure 2. Above example after collapsing the small curve.

Collapse Geometry

The collapse geometry commands use virtual geometry to tweak small angles and curves to improve meshability of geometry models. The following options for collapsing geometry are available:

- Collapse Angle
- Collapse Curve
- Collapse Surface

Collapse Surface

The collapse surface command allows the user to remove surface boundaries from the model. This is accomplished by splitting the surface at two given locations and combining it into two adjacent surfaces using virtual geometry operations. The command syntax is:

```
Collapse Surface <id> Across Location1
Location 2 With Surface <id_list> [Preview]
```

The locations option can use any of the general Cubit location commands. However, the vertex and curve options are among the most useful location options. For example, the command

collapse surface 15 across vertex 128 curve 40 with surface 26 117

would split surface 15 by the line that is formed between vertex 128 and the midpoint of curve 40. It would then composite the two parts of surface 15 that are adjacent to surfaces 26 and 117. The result is that three surfaces have been reduced to two.

The collapse surface command is most useful in removing blended surfaces (i.e. fillets and chamfers) from a model. For example, Figure 1 below shows a set of highlighted surfaces on a bracket. By collapsing all these surfaces the model shown in Figure 2 is created. Collapsing the surfaces for this model simplifies the model and allows for the creation of a higher quality mesh.

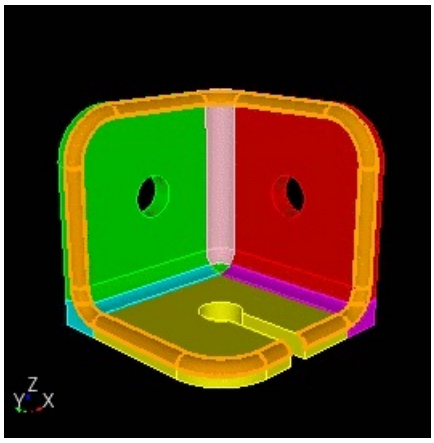


Figure 1. Bracket with chamfered edges.

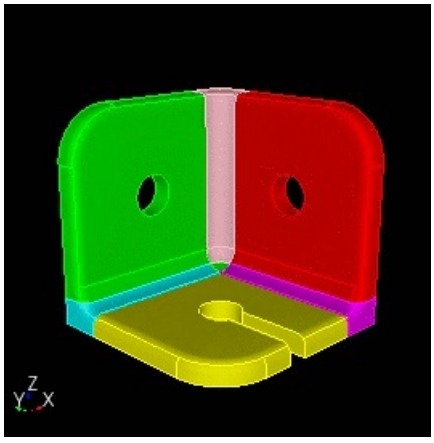


Figure 2. Bracket after highlighted edges have been collapsed

Composite Curves

The full command for the creation of composite curves is:

```
Composite Create Curve <id_range> [Keep Vertex
<id_list>] [Angle <degrees>]
```

The additional arguments provide two methods to prevent vertices from being removed from the model or *composited* over. The first method, **keep vertex** explicitly specifies vertices which are not to be removed. This option can also be used to control which vertex is kept when compositing a set of curves results in a closed curve.

The **angle** option specifies vertices to keep by the angle between the tangents of the curves at that vertex. A value less than zero will result in no composite curves being created. A value of 180 or greater will result in all possible composites being created. The default behavior is an empty list of vertices to keep, and an angle of 180 degrees.

Composite Geometry

- Composite Curves
- Composite Surfaces

The virtual geometry module has the capability to combine a set of connected curves into a single composite curve, or a set of connected surfaces into a single surface. The general purpose is to suppress or remove the child geometry common to those entities being composited. For example, compositing a set of curves suppresses the vertices common to those curves, thus removing the constraint that a node must be placed at that vertex location.

The basic form of the command to create composites is:

```
Composite Create {Surface|Curve} <id_list>
```

This command will composite as many surfaces (or curves) as possible, in many cases creating multiple composites.

The entities combined to create the composite must either all be unmeshed or all be meshed. A meshed composite surface can not be removed unless the mesh is first deleted.

Care should be taken when compositing over large C^1 discontinuities as it may cause problems for the meshing algorithms and may result in poor quality elements. C^1 discontinuities are corners or abrupt changes in the surface normal.

The command to remove a composite is:

```
Composite Delete {Surface|Curve} <id>
```

Composite Surfaces

The general command for composite surface creation is:

```
Composite Create Surface <id_range> [Angle
<degrees>] [Nocurves] [Keep [Angle <degrees>]
[Vertex <id_list>]]
```

Related Commands

```
Graphics Composite {on|off}
```

The **angle** argument prevents curves from being removed from the model or *composited* over. Composites will not be generated where the angle between surface normals adjacent to the curve is greater than the specified angle.

When a composite surface is created, the default behavior is to also to composite curves on the boundary of the new composite surface.

Curves are automatically composited if the angle between tangents at the common vertex is less than 15 degrees. The **nocurves** option can be used to prevent any composite curves from being created.

The **keep** keyword can be used to change the default choice of which curves to composite. The arguments following the **keep** keyword behave the same as for explicit composite curve creation. The **nocurves** and **keep** arguments are mutually exclusive.

Controlling the Surface Evaluation Method for Composite Surfaces

It typically takes longer to mesh a single composite surface than to mesh the surfaces used in the creation of the composite. To improve speed, composite surfaces use an approximation method to evaluate the closest point to a trimmed surface. However, this evaluation method may give poor results for composites of highly convoluted surfaces.

The virtual geometry module provides a way to change the way surfaces are evaluated using the following command:

```
Composite Closest_pt Surface <id>
{Gme|Emulate}
```

The default behavior is to use the **emulate** method, as it is typically considerably faster. Specifying the **gme** option will force the specified composite surface to use the exact calculation of the closest point to a trimmed surface, as provided by the solid modeler. The **gme** option, however, can be considerably slower.

Composite Determination

The **composite create surface** command is non-deterministic in some circumstances. When three or more adjacent surfaces are to be composited, all the surfaces may not be able to be composited into a single surface as illustrated in Figure 1. In this case different subsets of the surfaces may be composited and the command will choose arbitrary subsets to composite. As an example, there are three surfaces A, B, and C, all adjacent to each other. The common curve between A and B is AB, the common curve between B and C is BC, and the common curve between A and C is CA. If the curve BC cannot be removed, either due to the angle specified in the composite command, or because there is a fourth surface, D, also using that curve, the command will arbitrarily

choose to either composite A and B or A and C.

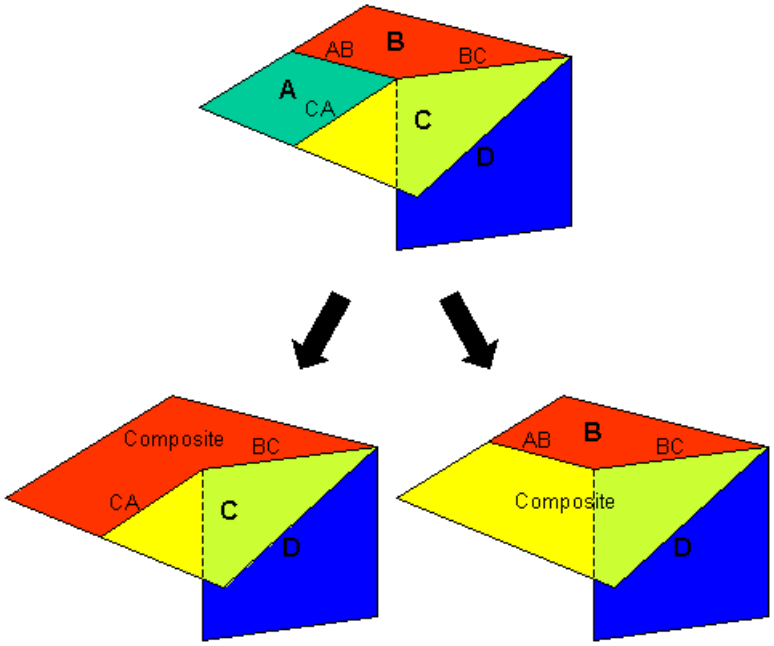


Figure 1. In some cases, the program will make a determination of which surfaces to composite.

Removing Partitions

There are two commands used to remove partitions:

```
Partition Merge {Curve|Surface|Volume}
<id_list>
```

The command combines existing partitions where possible. This command is similar to the composite create command. The difference is that this command is special-cased for partitions, and will result in more efficient geometric evaluations. If all the partitions of a real solid model entity are merged, such that there is only one partition remaining, the virtual geometry will be removed, and the original solid model geometry will be restored to the model.

The CUBIT delete command can also be used for removing partitions. See Deleting Virtual Geometry for a description of its use.

Using Mesh Intersections to Partition Surfaces

To assist in various mesh editing tasks such as joining, a *mesh-based imprinting* capability is provided. The command

```
Imprint Mesh {Body | Volume} <id_list>
```

determines imprint locations using the mesh on the surfaces of the specified bodies or volumes. Regions of coincidence between the surfaces is determined by searching for coincident nodes in the mesh of the surfaces. Virtual geometry is then used to partition the surfaces and curves at the boundary of these regions of coincident mesh.

The **imprint mesh** functionality differs from a normal geometric imprint in the following ways:

- The location of the imprint is determined from coincidence of mesh nodes.
- The mesh remains intact through the imprint operation.
- Virtual geometry is used to create the imprint.
- The imprinting can be done on all types of geometry (including mesh-based geometry, merged geometry, and virtual geometry.)

The following is a trivial example of this capability. The following commands create two meshed blocks:

```
brick width 10
brick width 6
body 2 move x 8
volume 1 2 size 1
mesh volume 1 2
```

Figure 1 shows the results of these commands.

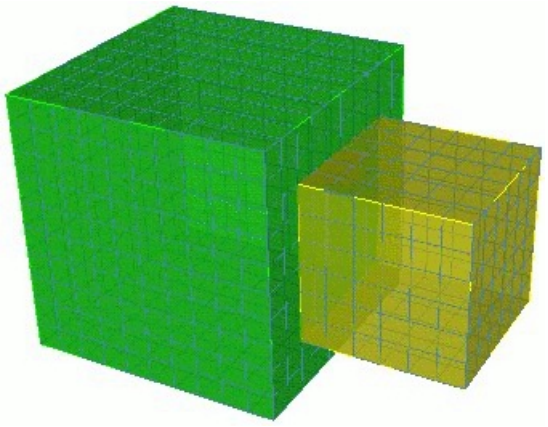


Figure 1. Two adjacent meshed volumes. The coincident meshes will form the basis of the imprint operation.

The mesh of the blocks can be joined by first doing a mesh-based imprint and then merging:

```
imprint mesh body 1 2
merge body 1 2
```

Figure 2. shows the results of the imprint operation. A meshed surface is created at the interface between the two meshed volumes. The nodes on the new surface are shared by the neighboring hexahedra of both volumes.

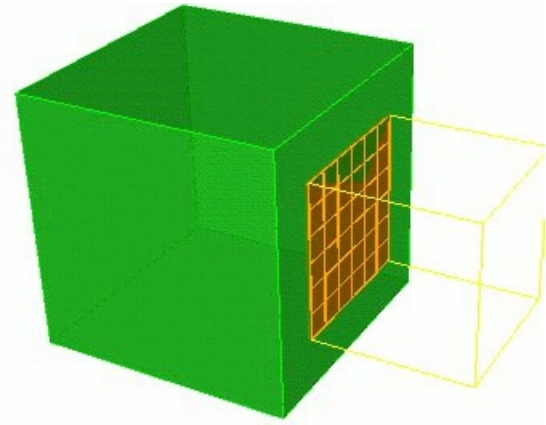


Figure 2. The imprinted surface. Adjacent volume meshes joined at the interface surface.

Partitioned Curves

There are four methods for specifying locations at which to partition curves:

```
Partition Create Curve <curve_id> {Fraction
<fraction_list> | Position <xpos> <ypos>
<zpos> | [with] <vertex_list> | <node_list> }
```

The first two forms of the command create additional vertices and use those vertices to split a curve. The third form of the command uses existing vertices to split the curve. The fourth form of the command uses existing nodes to split the curve.

Using the **fraction** option, vertices are created at the specified fractions along the curve (in the range [0,1].) Subsequently, the curve is split at each vertex, resulting in n+1 new curves, where n is the number of fraction values specified.

Using the **position** option, vertices are created at the closest location along the curve to each of the specified position. Subsequently, the curve is split at each vertex, resulting in n+1 new curves, where n is the number of positions specified.

If the **node** option is used, meshed curves may be partitioned. The specified nodes must lie on the curve to be partitioned. The curve is split at each node specified, and any other mesh entities are divided appropriately amongst the curve partitions.

Partitioned Surfaces

There are several forms of the command to partition a surface. A surface may be partitioned using hard points, curves, polylines, mesh edges, mesh faces or mesh triangles.

- Partitioning with Vertices or Nodes
- Partitioning with Curves
- Partitioning with Mesh Edges
- Partitioning with Mesh Faces or Triangles

Partitioning with Vertices and Nodes

Partitioning with Hard Points

There are two methods of partitioning a surface using vertices and nodes. The first method is to create a set of hard points using nodes, vertices, or coordinates that constrain the mesh to particular points on the surface. The syntax is:

```
Partition Create Surface <id> Vertex <id_list>
[Individual]

Partition Create Surface <id> Node <id_list>
[Individual]
```

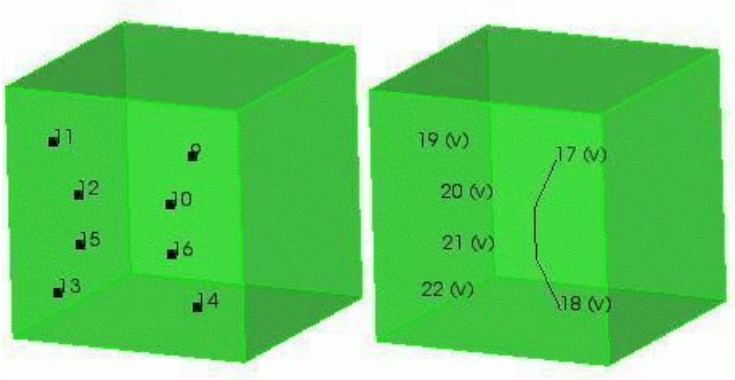
Partitioning with Polylines

The second method is to define a polyline using a set of vertices or coordinates. This method splits the surface using a polyline defined by the a list of positions specified as either coordinate triples, or existing vertices. The polyline is projected to the surface to define the curve for splitting the surface. If only one position is specified a zero-length curve with a single vertex will be created The syntax is identical to above WITHOUT the individual option.

```
Partition Create Surface <id> Vertex <id_list>

Partition Create Surface <id> Position <x> <y>
<z> [[Position] <x> <y> <z> ...]
```

In the following simple example, the surface is partitioned using both methods. On the left half of the object, the surface is partitioned using the individual option (vertices 11 12 15 13). On the right half, a polyline is used (vertices 9 10 16 14). All of the free vertices can then be deleted, leaving the virtual curves shown in the second picture. Vertices 19 20 21 and 22 are all zero-length curves. The small 'v' in parentheses is to indicate that it is virtual geometry. The resulting mesh is shown in the third picture. Notice that the polyline constrains the entire curve to the mesh, while the hardpoints constrain only that individual point.



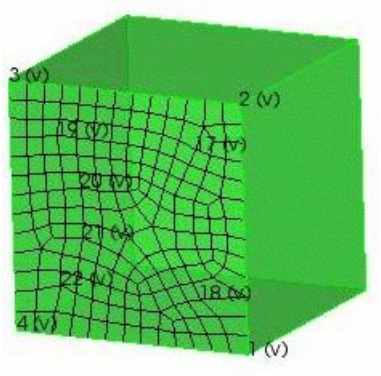


Figure 1. Partitioning a Surface Using Vertices

Partitioning with Curves

This form of the command splits the existing surface into several surfaces by creating curves that approximate the projection of the specified existing curves onto the surface. The syntax is:

```
Partition Create Surface <id> Curve <id_list>
```

Partitioning with Mesh Edges

Meshed surfaces may be partitioned with mesh edges. The specified mesh edges must be owned by the surface to be partitioned. The shape of the curve(s) used to split the surface is specified by a set of mesh edges.

If the split location is specified by a series of mesh edges, and the specified mesh edges form a closed loop, the node option may be used to control which node the vertex is created at.

```
Partition Create Surface <id> Edge <id_list>
[Node <node_id>]
```

Partitioning with Faces or Triangles

Surfaces may also be partitioned by specifying a list of triangles or faces (quads). The boundary of the list will automatically be detected and new curves and vertices created at the appropriate locations. Curves are created from the mesh edges and used to split the surface. The surface mesh is split and assigned to the appropriate surface partitions.

```
Partition Create Surface <id> Face|Tri
<id_list>
```

Partitioned Volumes

To partition a volume by giving a center and radius:

```
Partition Create Volume <id> Center [Location]
{options} Radius <val>
```

This command splits the existing volume into two volumes. All volume elements that lie within the specified radius of the specified center location are identified, and the exterior faces of these elements are used to create a surface and partition the volume. The center can be specified with any of the location options.

Figure 1 shows an example of a partitioned volume. A cube that has been map meshed is partitioned using a center at one of its vertices. The result is two distinct volumes with a surface separating the two. The interface surface is composed of the faces of the interior hex elements.

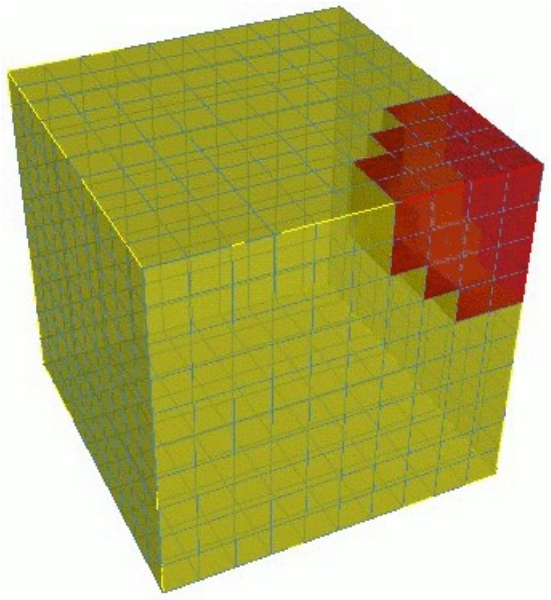


Figure 1. A partitioned volume

This command may be useful for separating small regions of a meshed volume so that remeshing or mesh improvement may be performed locally.

Partitioned Geometry

Partitioning provides a method to introduce additional topology into the model, to better constrain meshing algorithms. This is accomplished by splitting, or partitioning, existing curves or surfaces.

- Partitioned Curves
- Partitioned Surfaces
- Partitioned Volumes
- Using Mesh Intersections to Partition Surfaces
- Removing Partitions

Deleting Virtual Geometry

Removing Virtual Geometry

The following command removes all lower-order virtual geometry from the specified entities.

```
Virtual Remove <entity_list>
```

Examples:

```
virtual remove surface 5
```

Removes all composite and partition curves from surface 5.

```
virtual remove body all
```

Remove all virtual geometry from all bodies.

For removing individual virtual entities, see the sections of the documentation for each type of virtual entity:

- Composite curves
- Composite surfaces
- Partition curves
- Partition surfaces

Using The Delete Command With Composites

If the general **delete** command is invoked for a composite surface, the composite surface will be removed, and the original surfaces used to define the composite will be restored to the model. The defining surfaces are NOT also deleted. As with any other non-virtual surfaces, the **delete** command will fail if the composite has a parent volume.

To delete composite surfaces with a parent volume, the composite delete command can be used. The behavior is analogous for composite curves.

If the **delete** command is used on a volume containing a composite surface or curve, or on a surface containing a composite curve, the entire volume or surface will be deleted, including the original entities used to define the composite, as those entities are also children of the entity being deleted.

Using the Delete Command With Partitions

It is recommended that the delete command not be used with partitions, as it may break subsequent usage of the merge and delete forms of the partition command for other partitions of the same real geometry entity. However, if the delete command is used for partitions, the behavior is to delete the specified partition, and when the last partition of the real geometry is deleted, to restore the original geometry.

The **delete** command can also be used on parents of partitions. For example, a volume containing partitioned surfaces, or a surface containing partitioned curves can be deleted. In this case, the specified entity will be deleted along with all of its children, including the partition entities, and the original entities that were partitioned.

Simplify Geometry

Simplifying topology by compositing individually selected surfaces is often a tedious and time-consuming task. The simplify command addresses the tedium by automatically compositing surfaces and curves based on selected criteria between neighboring entities. Figure 1 shows a typical example of simplify command usage ('simplify volume 1 angle 15').

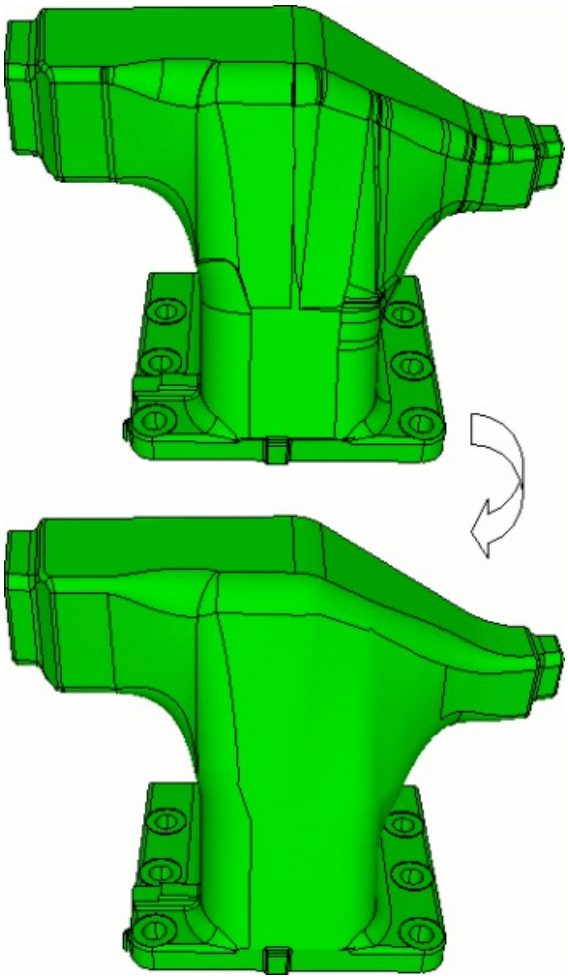


Figure 1. Typical Simplify command usage

The command syntax and discussion items are shown below.

```
Simplify {Volume|Surface|Curve} <range>
[Angle< value >] [Respect {Surface <id_range>
| Curve <id_range> | Vertex <id_range>|
Imprint | Fillet}] [Preview]
```

Feature Angle

Feature angle is defined as the angle between the normals of adjacent surfaces at the shared curve. At each facet point along the curve, a normal is computed for each surfaces. If at any point the angle between the two normals is greater than the **feature_angle**, the surfaces are not composited. If all angles between normals are less than **feature_angle**, the two surfaces are composited (assuming any other specified criteria are met). Feature angle is always used as the criteria and if an angle is not specified the value is set to 15 degrees.

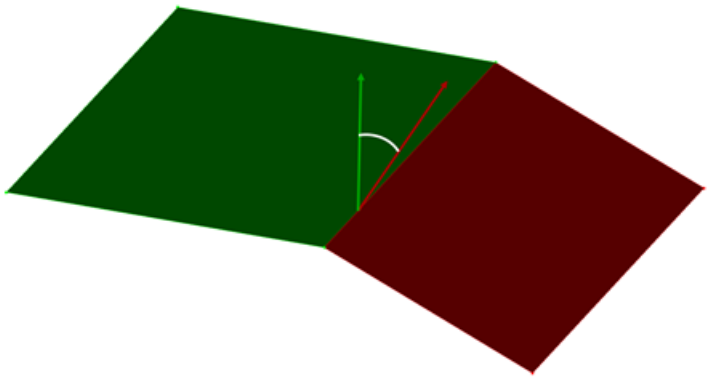


Figure 2. Feature Angle

Automatically Compositing Curves

The simplify command can also be used to automatically composite curves using an angle tolerance. Curves will be composited together only if they are explicitly specified in this command, and not as the result of two surfaces being composited.

Respecting Vertices, Curves and Surfaces

Surfaces, curves, and vertices can be specified to prevent geometry features from automatically being composited. Figure 2 show an example of respecting a surface (‘simplify vol 1 angle 15 respect surf 289’).

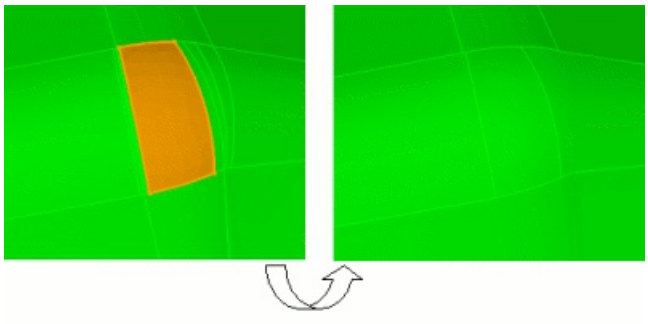


Figure 3. Respecting a surface

For complex geometries, it is often useful to preview the simplify command and then add any respected geometry to the command respect lists.

Respecting Imprints

Curves created by imprints can automatically be respected by the simplify command. Figure 3 shows an example of geometry with split fillets.

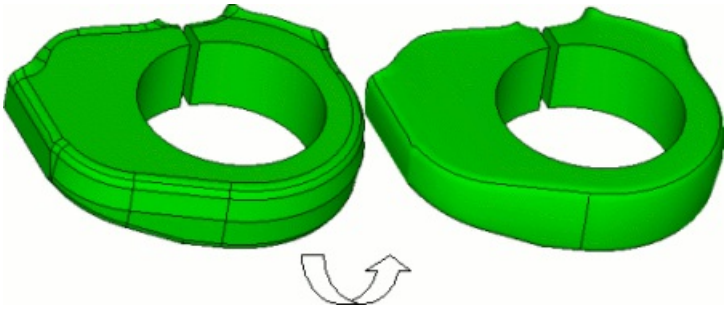


Figure 4. Respecting imprint geometry

Notice that in the split curves are respected by the Simplify command (‘simplify vol 1 angle 40 respect imprint’).

Other Options

The *preview* option shows what curves are respected without compositing any surfaces. It should also be pointed out that multiple respect specifications can be chained together. For example:

> **Simplify volume 1 angle 15 respect curve 1 respect imprint respect fillet preview**

Groups

Groups are collections of geometry and mesh entities. Groups can contain other groups. Groups provide a powerful capability for organizing and performing operations on multiple entities with minimal input. Groups can collect entities according to various criteria, such as position, size, or whether they are meshed. Performing an operation on a group is usually the same as performing that operation on all the entities in that group. The following describes the Group operations available in CUBIT:

- Basic Group Operations
- Groups in Graphics
- Propagated Hex Groups
- Quality Groups
- Group Propagation

There are several utilities in CUBIT which use groups as a means of visualizing output. These utilities are described elsewhere, but listed here for reference:

- Webcut results
- Merged and unmerged entities
- Sweep groups
- Interval matching
- Disassociated Meshes
- Importing ACIS, IGES, STEP, Free Meshes

Basic Group Operations

Geometry Groups

The common syntax to create or modify a group is to give it a name and a collection of entities:

```
Group ["name"] Equals <list of entities>
```

The *Equals* operation assigns the group to contain the given list. If the group already existed, its prior contents are overwritten. (Once a group is created, it can be refered to by id as well.) Here, entities can be geometry entities, mesh entities, or both. For example, the command,

```
group "Exterior" equals surface 1 to 2, curve 3 to 5
```

will create the group named (names are case sensitive). Any command taking entities can also take a group: e.g., **mesh Exterior**, **list Exterior**, or **draw Exterior**.

Geometry entities may specified by name as well. E.g.,

```
group 'Interior' add surface with name 'bill' 'john' 'fred'
```

will place the surfaces named 'bill' 'john' and 'fred' in the group Interior.

Wildcards (*) can also be used with names. To add all surfaces with the substring 'bob' in their name, use the command:

```
group 'interior' equals surface with name '*bill*'
```

There are a variety of operations for modifying the contents of a group, such as adding or removing entities. These can also be used to create a group from scratch by using a new name.

```
Group ["name" | <id>] Add <entity list>
```

```
Group ["name" | <id>] Remove <entity list>
```

```
Group ["name" | <id>] Xor <entity list>
```

Group Booleans

Groups may also be created from the contents of existing groups by set booleans: Group A boolean B preposition C. These operations will also overwrite the contents of existing groups. The **intersect** command will create a new group that contains elements common to both groups: A = B ∩ C. The **unite** command collects entities that exist in either group: A = B ∪ C. The **subtract** command collect the entities in one group but not the other: A = C ∖ B.

```
Group {<'name'>|<id>} Intersect Group <id>
with Group <id>

Group {<'name'>|<id>} Unite Group <id> with
Group <id>

Group {<'name'>|<id>} Subtract Group <id> from
Group <id>
```

Group Copy and Transform

The contents of a group can be copied and transformed (e.g. moved), or simply transformed. (The group itself is not copied, only the contents.)

This works only for groups of geometry entities, or groups of free mesh elements without associated geometry. Groups can collect free mesh entities in much the same way as a geometric entity. If the optional **copy** keyword is provided, the entities in the group will be copied first, and the copy will be transformed. Geometric and mesh groups behave differently. For geometry groups, if the group contains geometric entities that are meshed, the mesh will also be copied and transformed, unless the optional **nomesh** keyword is specified. You can copy a single surface of a volume, but you cannot move just it, because geometric entities cannot be transformed unless the parent entity is also. In addition, all merged partners must be contained in the group. However, any geometric entity can be copied and transformed. For mesh groups, transforming the nodes will implicitly transform the elements containing those nodes. The **copy** keyword is ignored if the group contains mesh entities. These types of mesh groups can only contain free mesh entities; for copying and transforming mesh associated to geometry, use a geometry group. (In the following, recall that an existing group name can always be used in place of 'Group <id>'.)

```
Group <id> [Copy [nomesh]] [Move <dx> <dy>
<dz>]

Group <id> [Copy [nomesh]] [Move {x|y|z}
<distance>...]

Group <id> [Copy [nomesh]] [Move <direction>
[distance]]

Group <id> [Copy [nomesh]] [Reflect {x|y|z}]

Group <id> [Copy [nomesh]] [Reflect <x> <y>
<z>]

Group <id> [Copy [nomesh]] [Rotate <angle>
About {x|y|z}]

Group <id> [Copy [nomesh]] [Rotate <angle>
About <x> <y> <z>]

Group <id> [Copy [nomesh]] [Rotate <angle>
About Vertex <Vertex_id1> <Vertex_id2>]

Group <id> [Copy [nomesh]] [Scale <scale> | x
<val> y <val> z <val>]
```

Deleting Groups

Groups can be deleted with the following command:

```
Delete Group <id range> [Propagate]
```

The option **propagate** will also delete any contained groups, recursively. That is, if group A contains group B and group B contains group C, then **Delete Group A Propagate** will delete groups A, B, and C.

Cleaning Out Groups

You can remove all of the entities in a group via the **cleanout** command:

```
Group <group_id_range> Cleanout
[Geometry|Mesh] [Propagate]
```

By default all entities will be removed - optionally you can cleanout just geometry or mesh entities. As in delete, the **propagate** option will cleanout the group specified and all of its contained groups recursively.

Groups in Graphics

In the GUI version of CUBIT, groups may be picked with the mouse.

When displaying a group containing hexes, only the outside skin of the hexes will be displayed.

Propagated Hex Groups

- Starting on a Surface
- Starting on a Face

Propagated hex groups are a way of grouping hexes from a hex mesh using sweep-type criteria. For example, creating a group containing all hexes between two specified mesh faces.

Note: the first examples below are based on first executing these commands:

```
brick width 10
volume 1 size 1
mesh volume 1
```

Propagated Hex Group Starting on a Surface

Starting on a surface can end at a surface or can end after the number of times the user specifies.

- Ending at a Surface
- Number of Times
- Ending at a Surface with Multiple
- Number of Times with Multiple
- Ending at a Surface with Direction
- Number of Times with Direction

Ending at a Surface

```
Group ['name' | <id>] Add Hex Propagate
Surface <id> Target Surface <id>
```

Example

```
group 2 add hex propagate surface 1 target
surface 2
```

Result: Group 2 will be created containing 1000 hexes

Number of Times

```
Group ['name' | <id>] Add Hex Propagate
Surface <id> Times <number>
```

Example

```
group 2 add hex propagate surface 1 times 4
```

Result: Group 2 will be created containing 400 hexes

Both methods, ending at surface or number of times, can be used with the "multiple" option which will create several groups depending upon the multiple number specified.

Ending at a Surface with Multiple

```
Group ['name' | <id>] Add Hex Propagate
Surface <id> Target Surface <id> Multiple
<number>
```

Example

```
group 2 add hex propagate surface 1 target
surface 2 multiple 2
```

Result: Five groups will be created and stored with their respective ids of multiple 2, these groups will be stored in the parent group, Group 3, and Group 3 will be stored in the grand parent group, Group 2.

Number of Times with Multiple

```
Group ['name' | <id>] Add Hex Propagate
Surface <id> Times <number> Multiple <number>
```

Example

```
group 2 add hex propagate surface 1 times 10
multiple 5
```

Result: Two groups will be created and stored with their respective ids of multiple 5, these two groups will be stored in the parent group, Group 3, and Group 3 will be stored in the grand parent group, Group 2.

If number of times is specified and the direction is ambiguous, the surface direction or the node direction can be specified to direct the propagation. If the end surface is specified, only a node direction can be specified to direct the propagation. When specifying the node direction, the node has to be picked such that when the hexes are propagated, the picked node lies in these propagated hexes. If that node is never reached while propagating, the direction is not found and zero hexes will be included in the specified group.

Note: for the examples below, the result can be seen by executing these commands:

brick x 10
vol 1 size 1
brick width 10
body 2 move 10
volume all size 1
merge all
mesh volume all

Ending at Surface with Direction

```
Group ['name' | <id>] Add Hex Propagate
Surface <id> Times <number> Direction Node
<id>
```

Example

```
group 2 add hex propagate surface 6 target
surface 12 direction node 1530
```

Result: Group 2 will be created containing 400 hexes

Note: The direction command and the multiple command can be combined (i.e. group 2 add propagate surface 6 times 4 multiple 2 direction node 1530)

Number of Times with Direction

```
Group ['name' | <id>] Add Hex Propagate
Surface <id> Times <number> Direction [surface
<id> | node <id>]
```

Example

```
group 2 add hex propagate surface 6 times 4
```

```
direction surface 4

group 2 add hex propagate surface 6 times 4
direction node 1530
```

Result: group 2 will be created containing 400 hexes

Propagated Hex Group Starting on a Face

When starting on a face, the propagation method can end at a surface, end at a face or can end after the number of times the user specifies:

- Ending at a Surface
- Ending at a Face
- Number of Times
- Ending at a Surface with Multiple
- Ending at a Face with Multiple
- Number of Times with Multiple
- Ending at a Face with Direction
- Ending at a Surface with Direction
- Number of Times with Direction

Ending at a Surface

```
Group ['name' | <id>] Add Hex Propagate
[Source] Face <id range> Target Surface <id>
```

Example

```
group 2 add hex propagate face 1 11 21 target
surface 2
```

Result: Group 2 will be created containing 30 propagated hexes (10 layers of 3 hexes)

Ending at a Face

```
Group ['name' | <id>] Add Hex Propagate
[Source] Face <id> Target Face <id>
```

Example

```
group 2 add hex propagate face 1 target face
1721
```

Result: Group 2 will be created containing 5 propagated hexes (5 layers of 1 hex)

Note: Ending at a face requires starting at one face at one time, but ending at surface allows multiple start faces

Number of Times

```
Group ['name' | <id>] Add Hex Propagate
[Source] Face <id range> Times <number>
```

Example

```
group 2 add hex propagate face 2 times 4
```

Result: Group 2 will be created containing 4 propagated hexes (4 layers of 1 hex)

All of these methods, ending at surface, end at a face or number of times, can be used with the "multiple" option which will create a

grandparent (top-level), parent (mid-level, contained within the grandparent) and child (bottom level, contained within the parent) groups. The child groups will contain each hex layer (specified number of layers per child group), all organized into a single parent group, which is organized underneath the group ID given to the command. Subsequent propagation commands could then be executed adding to the grandparent group, but creating a new parent and child groups. This way multiple propagation "sets" can be stored in one grandparent group, if desired.

Ending at a Surface with Multiple

```
Group ['name' | <id>] Add Hex Propagate
[Source] Face <id> Target Surface <id>
Multiple <number>
```

Example

```
group 2 add hex propagate face 1 target
surface 2 multiple 1
```

Result: Ten groups will be created and stored with their respective ids, one for each layer of hexes. These groups will be stored in the parent group, Group 3, and Group 3 will be stored in the grand parent group, Group 2. A subsequent propagation command could be executed adding to group 2 (the grandparent), which would create a single group contained in group 2 (the parent), containing the hex layer groups (the children).

Ending at a Face with Multiple

```
Group ['name' | <id>] Add Hex Propagate
[Source] Face <id> Target Surface <id>
Multiple <number>
```

Example

```
group 2 add hex propagate face 1 target face
1721 multiple 1
```

Result: 5 groups will be created and stored with their respective ids, one for each layer of hexes. These groups will be stored in the parent group, Group 3, and Group 3 will be stored in the grand parent group, Group 2. A subsequent propagation command could be executed adding to group 2 (the grandparent), which would create a single group contained in group 2 (the parent), containing the hex layer groups (the children).

Number of Times with Multiple

```
Group ['name' | <id>] Add Hex Propagate
[Source] Face <id> Times <number> Multiple
<number>
```

Example

```
group 2 add hex propagate face 1 times 10
multiple
```

Result: Two groups will be created and stored with their respective ids, these two groups will be stored in the parent group, Group 3, and Group 3 will be stored in the grand parent group, Group 2.

If the end surface or end face is ambiguous, a node direction can be specified to direct the propagation. When specify the node direction, the node has to be picked such that when the hexes are propagated, the picked node lies in these propagated hexes. If that node is never reached while propagating, the direction is not found and zero hexes will be

included in the specified group.

Ending at Face with Direction

```
Group ['name' | <id>] Add Hex Propagate
[source] Face <id> Target Face <id> Direction
Node <id>
```

Example

```
group 2 add hex propagate face 1721 target
face 1 direction node334
```

Result: group 2 will be created containing 6 hexes

Ending at Surface with Direction

```
Group ['name' | <id>] Add Hex Propagate
[Source] Face <id range> Target Surface <id>
Direction Node <id>
```

Example

```
group 2 add hex propagate face 1 target
surface 2 direction node 334
```

Result: group 2 will be created containing 10 hexes

Note: The direction command and the multiple command can be used together (i.e. group 2 add propagate face 1721 end face 1 multiple 2 direction node 334)

If number of times is specified and the direction is ambiguous, a surface direction or a node direction can be specified to direct the propagation. The node direction has the same condition as when ending at a surface or face and that is it must lie in the propagated hexes.

Number of Times with Direction

```
Group ['name' | <id>] Add Hex Propagate
[Source] Face <id> Times <number>Direction
[surface <id> | node <id>]
```

Example

```
group 2 add hex propagate face 110 times 4
direction surface 2

group 2 add hex propagate face 1 times 4
direction node 269
```

Result: group 2 will be created contained 4 hexes

Note: The direction command and the multiple command can be used together. (i.e. group 2 add propagate face 1721 times 4 multiple 2 direction surface 1)

Naming Convention for Propagated Hex Groups

A special naming convention can be used for the propagated hex groups, best described by an example.

The following command will create a hierarchy of logically named groups, as follows.

```
group 'W1P1T1' add propagate surf 1 end surf 2
```

```
multiple 1
```

The hierarchy looks like this:

```
W1
    W1P1
        W1P1T1
        W1P1T2
        W1P1T3
        ...
        W1P1T10
```

Where W1P1 is contained within W1, and W1P1T1, W1P1T2, etc.. are contained within W1P1.

The software simply looks for numerical numbers in the group name and parses out the correct grandparent, parent and child names from the substrings. There must be exactly 3 substrings in the group name, each ending with an integer for the command to work properly.

A subsequent command:

```
group 'W1P2T1' add propagate surf 3 end surf 5
multiple 1
```

will add a parent group to W1, called W1P2, and the subsequent child groups:

```
W1
    W1P1
        W1P1T1
        W1P1T2
        W1P1T3
        ...
        W1P1T10
    W1P2
        W1P2T1
        W1P2T2
        W1P2T3
        ...
        W1P2T10
```

Quality Groups

Groups can also be formed from the hexes or faces obtained from the quality command. Each group formed using quality can be drawn with its associated quality characteristics {i.e. jacobian low .2 high .3} automatically.

```
Group {<'name'>|id} {Add|Equals|Remove|Xor}
Quality { Hex | Tet | Face | Tri | Volume |
Surface | Group } <id_range> { quality metric
name (default is SHAPE) } [ High <value> ] [
Low <value> ] [ Top <number> ] [ Bottom
<number>]
```

The following example illustrates the use of quality groups:

```
group 2 add quality volume 1 jacobian
```

In this case, if the meshed brick from the section Propagated Hex Groups is used, Group 2 will be created and it will contain 1000 hexes with quality characteristics.

The quality metric names can be found in the Quality Assessment section of the documentation.

Propagated Groups

Creating propagated groups is a mechanism for joining groups of elements that meet specific criteria. For hex groups it might be grouping hexes from a hex mesh using sweep-type criteria. For surface elements, it might be grouping faces or tris into sidesets based on angle criteria.

- Propagated Hex Groups
- Propagated Surface Groups using the Seed Method

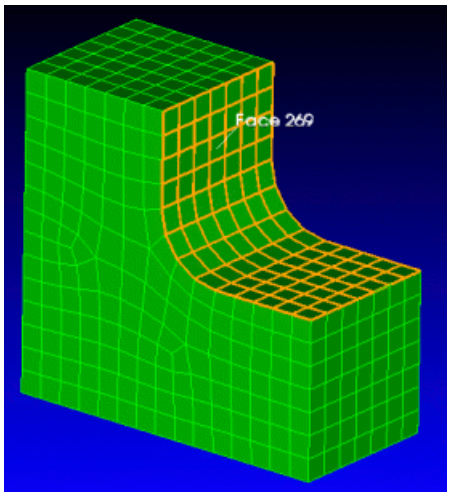
Seeded Mesh Groups

It is also possible to automatically group surface mesh elements based on feature angles. Given a seed element, the algorithm will loop over all adjacent elements and create groups of elements whose surface normals are similar, or which fall within a certain radius. The command syntax is:

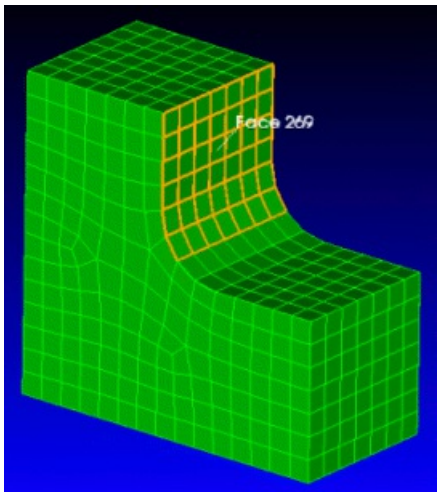
```
Group {<'name'>|<id>} {Add|Equals|Remove|Xor}
Seed <mesh_entities> {Feature_angle <angle>
[Divergence]|Depth <number>}
```

The seed element may be a quad, tri, or node element. There are two methods of angle comparison for this command. The feature angle option will compare angles of the each element to its adjacent elements by comparing surface normals. In the case of nodes, the seed node surface normal will be the average of the adjacent faces or tris. Nodes will be added if their attached faces meet the angle requirements. The divergence option will compare angles to the original seed element's surface normal. The depth option will add elements within a certain radius.

The following figures illustrate the use of the seed method to create mesh groups using the feature angle and divergence methods.



CUBIT> group 'mygroup1' add seed face 269 feature_angle 45



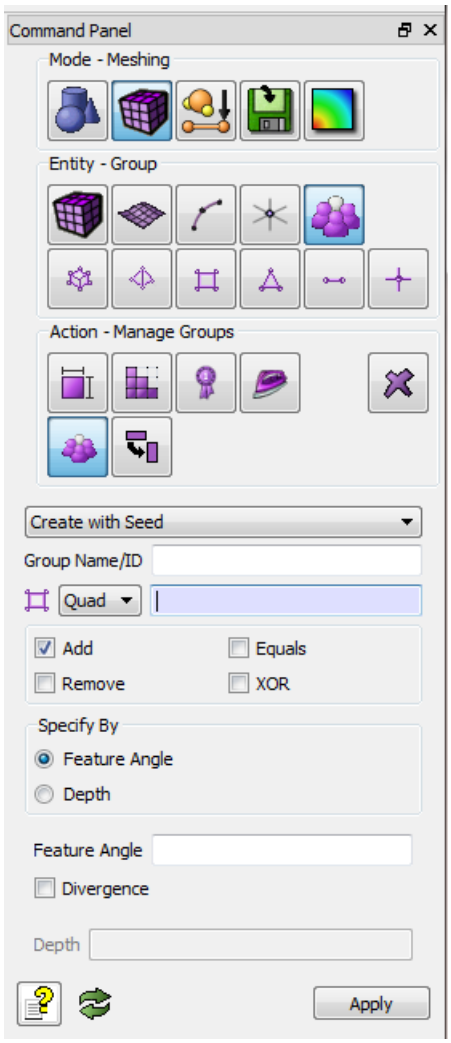
CUBIT> group 'mygroup2' add seed face 269 feature_angle 45 divergence

The seed method of creating groups is particularly useful for creating

groups on free meshes for the purpose of assigning nodesets and sidesets.

The GUI command panel for this command is found by selecting

"Mode-Meshing", "Entity-Group", "Action-Manage Groups", then "Create with Seed." The command panel is shown below:



Naming Convention for Propagated Hex Groups

A special naming convention can be used for the propagated groups, best described by an example.

The following command will create a hierarchy of logically named groups, as follows.

```
group 'W1P1T1' add propagate surf 1 end surf 2
multiple 1
```

The hierarchy looks like this:

```
W1
    W1P1
        W1P1T1
        W1P1T2
        W1P1T3
        ...
        W1P1T10
```

Where W1P1 is contained within W1, and W1P1T1, W1P1T2, etc.. are contained within W1P1.

The software simply looks for numerical numbers in the group name and parses out the correct grandparent, parent and child names from the substrings. There must be exactly 3 substrings in the group name, each ending with an integer for the command to work properly.

A subsequent command:

```
group 'W1P2T1' add propagate surf 3 end surf 5
multiple 1
```

will add a parent group to W1, called W1P2, and the subsequent child groups:

```
W1
    W1P1
        W1P1T1
        W1P1T2
        W1P1T3
        ...
        W1P1T10
    W1P2
        W1P2T1
        W1P2T2
        W1P2T3
        ...
        W1P2T10
```

Geometry Attributes

Each geometric topological entity has specific information attached to it. These attributes specify aspects of the entity such as the color that entity is drawn in and the meshing scheme to be used when meshing that entity. This section describes those geometry attributes that are not described elsewhere in this manual.

- Entity Names
- Entity IDs
- Persistent Attributes

Attribute Behavior

In this context, attributes are defined as data associated directly with a particular geometry entity. In CUBIT's implementation of attributes, these data can occupy one of three "states" at any given time: they can be stored in data fields on CUBIT's geometry entities; they can be stored in an intermediate representation, using CUBIT's attribute objects; or they can exist only on the ACIS objects. When they are stored on ACIS objects, those attributes are written to and read from disk files with the geometry. This mechanism allows CUBIT-specific information to be stored and retrieved with the geometry data. By default, attribute data is not stored with geometry. To enable the use of attributes, use the commands described in the following sections.

Attribute Commands

Most non-CUBIT-developer uses of attributes will be to use all or none of the attributes. Therefore, the most common command to enable and disable the use of attributes is:

```
Set Attribute {On|Off}
```

When this option is on, all defined attributes will be saved with the geometry when the user enters the Export Acis command.

When a geometry is imported into CUBIT, any attributes defined on that geometry and recognized as CUBIT attributes are imported and put into an intermediate representation (that is, this information is not assigned directly to the geometry entities). To find out which attributes are defined on a given set of entities, use the following command:

```
   List [<entity_list>] Attributes [Type
<attribute type>] [All] [Print]
```

If no entities are entered, attribute information for all the geometric entities defined in CUBIT is printed.

The **Type** option can be used to list information about a specific attribute type; values for are the same as those in the previous table.

If the **All** option is entered, information about all attribute types will be printed, even if there are none of those attributes defined for the specified entities.

If the **Print** option is entered, the information stored in each attribute will be printed; this command is usually used only by CUBIT developers.

Control By Attribute Type or Geometric Entity

Attributes can be enabled or disabled by attribute type, to allow the use of only user-specified attribute types. To turn on or off specific attributes, use the command:

```
Set Attribute <attribute type> {On|Off}
```

where **<attribute type>** is one of the types shown in the previous table.

Attributes can also be controlled to automatically write (update) and read (actuate) to/from solid model files automatically, using the command:

```
Set Attribute <attribute_type>
Auto  {Actuate|Update} {On|Off}
```

Finally, attributes can be manually written to and read from the geometric entities, and removed from cubit entities, using the command

```
{geom_list} Attribute {All|Attribute_type}
{Actuate|Remove|Update|Read|Write}
```

where **geom_list** is a list of geometry entities. This command is recommended only for developers' use.

Attribute Types

The attribute types currently implemented in CUBIT are shown below.

Attribute Types	Description
Color	Entity Color
Composite vg	Used to restore composite virtual topology
Genesis entity	Membership in boundary conditions (block, sideset, nodeset)
Id	Entity Id
Mesh container	Handle to mesh defined for the owner
Mesh scheme	Meshing scheme (e.g. paving, sweeping, etc.)
Name	Entity name
Partition vg	Used to restore partition virtual topology
Smooth scheme	Smoothing scheme (e.g. Laplacian, Condition Number)
Unique Id	Unique entity id, used to cross-reference other entities
Vertex type	Used to define mesh topology at vertex for mapping/submapping
Virtual vg	Used to store virtual geometry entity(ies) defined on an entity

Persistent Attributes

Typical data assigned to topological entities during a meshing session might include intervals, mesh schemes, group assignments, etc. By default, most of this data is lost between CUBIT sessions, and must be restored using the original CUBIT commands. Using CUBIT's persistent attributes capability, some of this data can be saved with the solid model and restored automatically when the model is imported into CUBIT.

- Attribute Behavior
- Attribute Types
- Attribute Commands
- Using CUBIT Attributes

Using CUBIT Attributes

A typical scenario for using CUBIT attributes would be as follows.

> Construct geometry, merge, assign intervals, groups, etc. (i.e. normal CUBIT session)
>
> Enable automatic use of attributes using the command:
>
> ```
> Set Attribute On
> ```
>
> Export acis file (see Export Acis command).
>
> Subsequent runs:
>
> Enable automatic reading and actuating of attributes:
>
> ```
> set attribute on
> ```
>
> Import ACIS file (see Import Acis command)

Used in this manner, geometry attributes allow the user to store some data directly with the geometry, and have that data be assigned to the corresponding CUBIT objects without entering any additional commands.

Entity IDs

Topological entities (including groups) are assigned integer identification numbers or ids in CUBIT in ascending order, starting with 1 (one). Each new entity created within CUBIT receives a unique id within the topological entity type. This id can be used for specifying the entity in CUBIT commands, for example "draw volume 3".

There is a separate id space for each type of topological entity. For example, all mesh nodes are given ids from 1 to *n*, where *n* is an integer greater than or equal to the number of nodes in the model. Likewise, all hexahedra are given ids from 1 to *m*, where *m* is an integer greater than or equal to the number of hexahedra in the model.

Element Ids

Each mesh entity (hex, tet, face, tri, edge, node, etc.) may also have a *Global Element ID* from an id space which is used for all mesh entities. A mesh entity is only assigned a *Global Element ID* if it is in a block, and is the global id that will be assigned to the element during Exodus export. The *Global Element ID* provides a single id space across all the different element types.

Gaps in ID space

After working with a model for some time, various operations will cause gaps to be left in the numbering of the geometric & mesh entities. The **compress ids** commands can be used to eliminate these gaps:

```
Compress [ids] [all] [Retainmax] [Sort]

Compress [Ids] [All]
{Group|Body|Volume|Surface|Curve|Vertex|Element|Hex|Tet|Face|Edge|Node}
[Retainmax]
```

Typing **compress** with no options or **compress all** will compress the ids of all entities; otherwise, the entity type for which ids should be compressed can be specified. The **retainmax** argument will retain the maximum id for each entity type, so that entities created subsequent to this command will receive ids greater than that value. If the **sort** qualifier is included, the new id of each entity will be determined by its size and location. Small entities are given a lower id than large entities. Entities that are the same size are sorted by their location, with lower x coordinate, then y, then z leading to a lower id. For example, two vertices are always the same size, so they are sorted based on the lowest x coordinate. If they are equal, then lowest y coordinate, etc. If two entities are found to have the same size and location, they are sorted according to their previous ids. This option can be used to restore ids in translated models in a manner which leads to more persistence than purely random id assignment.

Renumbering IDs

The renumber command can be used to change the id numbers assigned to meshed entities.

```
Renumber
{Node|Edge|Tri|Face|Hex|Tet|Wedge|Element}
<id_range> Start_id <id> [Uniqueids]
```

Any valid range specification can be used to specify the source ids. There is no requirement that the ids being renumbered are consecutively numbered. The new id numbers will be consecutive beginning at the

specified start id. For the command to be successful there can be no existing ids within the effective range of the start id. If the resultant destination range is not free of id numbers, the command will fail with an appropriate error.

Using the **uniqueids** keyword will result in the elements to be renumbered such that no element shares the same ID.

For convenience, all elements and nodes in a block can be renumbered with a single command:

```
Renumber block <id_range> [node_start_id <id>]
[elem_start_id <id>] [localids]
```

By default, the *Global Element ID* is renumbered with the **renumber block** command. If **localids** is specified, the hex, tet, face, tri, or edge id is renumbered instead.

Volume ID

The volume id command is used to renumber a single volume.

```
Volume <old_id> Id <new_id>
```

This command replaces the volume's old_id with the new_id if no other is using the new_id number. Entity renaming only works for volumes; it does not work for nodes, curves or surfaces.

Entity Names

By default, geometric entities in CUBIT are referenced using an entity type (e.g. Surface, Volume) and an id, for example "draw surface 1". However, geometric entities can also be assigned names, to simplify working with specific entities. Once a name is assigned to an entity, that name can be used in any CUBIT command in place of the entity type and number. For example, if surface 1 were named 'mysurf1', the command above would be equivalent to "draw mysurf1". Also, since entity names are saved with the geometry, this also provides a means for persistent identifiers for geometric entities. Names can be added or removed using the following commands.

```
{Group|Body|Volume|Surface|Curve|Vertex} {Name
| Rename} {`<entity_name>'| Default}

{Group|Body|Volume|Surface|Curve|Vertex}
Remove Name {`<entity_name>'| All | Default}
```

The name of each topological entity appears in the output of the List command. In addition, topological entities can be labeled with their names (see label command). A list of all names currently assigned and their corresponding entity type and id (optionally filtered by entity type) can be obtained with the command

```
List Names
[{Group|Body|Volume|Surface|Curve|Vertex|All}]
```

Notes:

- In a merge operation, the surviving entity is given the name(s) of the deleted entity.
- A geometric entity may have multiple names, but a particular name may only refer to a single entity.

Case-Insensitive Names

Entity names in CUBIT are case-insensitive. This means that there is no difference between the name 'mysurf1' and MySurf1'. The case of names will be preserved as assigned. For example, a volume named 'MyVolume' will display the name 'MyVolume' but can be referenced by any case version of the name, 'MYVOLUME', 'mYvOlume', etc. Case-sensitivity can be toggled on/off with the command:

```
[Set] Case Sensitive Names [on|OFF]
```

Valid and Invalid Names

Although any string may be used as an entity name, only valid names may be used directly in commands. A name is valid if it begins with a letter or underscore ("_"), followed by any combination of zero or more letters, digits, or the characters ".", "_", or "@". If an attempt is made to assign an invalid name to an entity, CUBIT will generate a valid version of the invalid name by replacing invalid characters with an underscore. Then both the valid and invalid versions of the name are assigned to the entity. For example, assigning the name "123#" to a volume will result in the volume having two names, "123#" and "_23_". The valid name can be used directly in commands (mesh _23_), while the invalid name can only be referenced using a longer, less direct syntax (mesh volume with name "123#").

Reconciling Duplicate Names

When an attempt is made to assign the same name to two different entities, a suffix is added to the name of the second entity to make it unique. The suffix consists of the "@" character followed by one or more letters or numbers. For example, the following commands will result in volumes 1 to 3 having the names "hinge", "hinge@A", and "hinge@B", respectively:

```
volume 1 name "hinge"
volume 2 name "hinge"
volume 3 name "hinge"
```

To prevent this automatic "fixing" of names, the *Fix Duplicate Names* flag may be switched to off. If the user attempts to assign a duplicate name while the flag is set to off, the name will remain unchanged.

```
Set Fix Duplicate Names [ON|Off]
```

Automatic Name Creation

CUBIT provides an option for automatically assigning names to entities upon entity creation. This option is controlled with the command:

```
Set Default Names {On|OFF}
```

When this option is on, entities are assigned default names consisting of a geometry type concatenated with the entity id, for example 'cur1', 'surf26', or 'vol62'.

Automatic Name Propagation

CUBIT automatically propagates names through webcuts. If an entity that has been assigned the name "Gear" is split through webcuts, the resulting bodies are named "Gear" and "Gear@A". Try the following example.

```
br x 10
volume 1 name "Cube"
webcut volume 1 xplane
webcut volume 1 2 yplane
webcut volume 1 2 3 4 zplane
label volume name
```

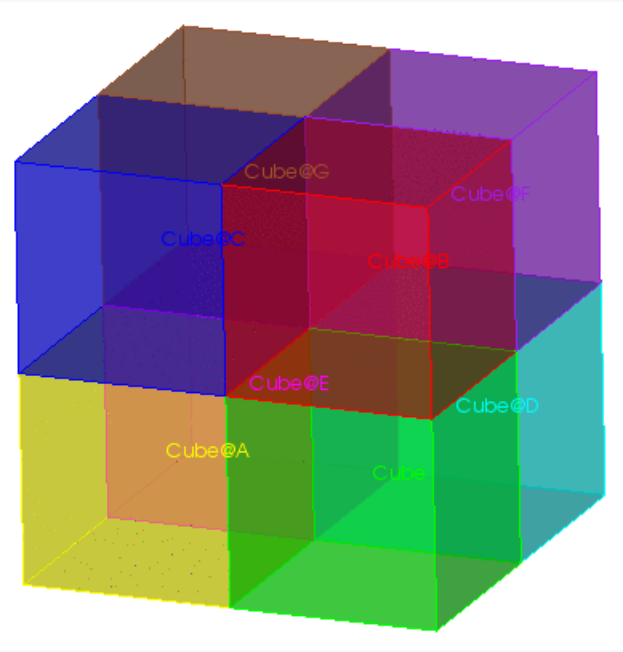


Figure 1. Name Propagation through Webcuts

You can operate on these propagated names using wildcards such as:

```
mesh volume with name 'Cube*'
block 1 volume with name 'Cube*'
```

Naming Merged Entities

When entities that have the same base name, such as "platform" and "platform@A", are merged, the resulting entities is assigned both names. The **set merge base names on** command tells Cubit that in this situation, it should merge the names too. The command syntax is:

```
Set Merge Base Names [On|OFF]
```

For example:

```
brick x 10
vol 1 copy move 10
surf 6 name 'platform'
surf 10 name 'platform'
```

Surface 10 actually is named platform@A, since we don't want duplicate names

```
merge all
list surf 6
```

You see that surface 6 has both 'platform' and 'platform@A' as names. Now, for the contrasting example

```
brick x 10
vol 1 copy move 10
surf 6 name 'platform'
surf 10 name 'platform'
set merge base names on
merge all
list surf 6
```

You see that surface 6 has only 'platform' as its name.

Parts, Assemblies, and Metadata

Overview of Parts, Assemblies and Metadata

A geometric model may be organized into a hierarchy of assemblies, sub-assemblies, and parts. These parts and assemblies can be assigned certain attribute values. The parts, assemblies, and associated attributes are referred to as DART Metadata, or simply metadata. Metadata can be imported from files, or can be created within CUBIT. Metadata can be exported to both mesh and geometry files.

Although useful in its own right, the primary purpose of CUBIT's metadata capabilities is to enable interoperability with the set of applications participating in the DART project (see the Sandia Analysis Workbench Wiki page at https://dart.sandia.gov/wiki/display/SAW/DartMetadataPlugin). DART interoperability enables CUBIT to preserve assembly relationships and material data through the analysis process.

This section describes the procedures for importing, manipulating and exporting metadata within CUBIT.

- Working with Parts and Assemblies
- Metadata Attributes
- Importing and Exporting Metadata

Importing and Exporting Metadata

Metadata can be imported from and exported to a file. In most cases metadata will be imported and exported with a data file such as a SAT file or a genesis file. CUBIT is also compatible with DART artifacts, including artifact dependency tracking.

- Importing Metadata
- Exporting Metadata
- Importing and Exporting DART Artifacts

Importing Metadata

Parts and assemblies can be created and associated with geometry by importing a DART Metadata file along with a geometry file, using the XML option of the import command. At this time the only two geometry formats which support metadata import are STEP and ACIS:

```
Import {Step|Acis} "<filename>". . . [XML "
<xml_filename>"]
```

To successfully associate the contents of the geometry file with the parts described in the metadata, the XML file must follow the DART Metadata 3.0 XML schema found at https://dart.sandia.gov/wiki/display/SAW/DartMetadataPlugin, and the geometry file must contain extra DART data. A suitable STEP file and a corresponding metadata file can be exported from Pro/E using the Pro/E (Creo) extension, see the Metadata plugin page for details. A SAT file and corresponding metadata file can be obtained by exporting them from CUBIT using the XML option of the export command.

Exporting Metadata

Some export commands include an XML option. Including this option in the export command instructs CUBIT to write out a DART metadata file, in addition to the traditional data file. The metadata file includes the data required to enable interoperability with other DART-compliant applications.

The only geometry export command which supports the XML option is ACIS export:

```
Export Acis "<acis_filename>" [XML
"<xml_filename>"]
```

When an ACIS file exported with metadata, the specified XML file includes a description of the assembly hierarchy as it appears in CUBIT.

Metadata can also be written to an XML file when exporting mesh. The only mesh export command which supports the XML option is genesis export:

```
Export {Genesis|Mesh} "<mesh_filename>" [XML
'<xml_filename>']
```

The XML file generated during mesh export includes the same information in a geometry metadata file, but also includes mesh-related data such as mappings between parts and element blocks, and includes any block, nodeset, or sideset names or descriptions which have been defined.

Importing and Exporting DART Artifacts

The DART project has defined a specific way to package data files with corresponding metadata files. A correctly packaged set of data files with a corresponding metadata file is called an *artifact*. An artifact's metadata file is always located in the same directory as the primary data file, and is always named *artifact.dta*.

Within the DART environment, dependencies between artifacts may be tracked by placing tracking information into metadata files. CUBIT supports automated artifact dependency tracking. Tracking information in an input metadata file is automatically reflected in any output metadata file written by CUBIT.

If input is correctly packaged as an artifact, CUBIT can automatically locate and read the metadata file corresponding to a particular input data file. To have CUBIT do this, select the "Import as Artifact" checkbox in the Open File dialog.

CUBIT can also package output as an artifact. To do so, select the "Export as Artifact" checkbox in the export dialog box.

When importing or exporting artifacts using the command line, include the *XML* option in the import or export command, specifying the xml file called *artifact.dta* in the same directory as the main data file.

For dependency tracking purposes, it may be necessary to import an artifact's metadata file by itself. For example, it may be necessary to import an artifact consisting of an IGES file. Since the *Import IGES* command does not support the *XML* option, the metadata file must be imported separately. To do so, use the command:

```
Import XML "<xml_filename>"
```

When working with correctly packaged artifacts, the XML filename will always be artifact.dta.

Metadata Attributes

Each part and assembly has several attributes, including its name and description. In addition, there are several attributes which do not describe any particular part or assembly. The “global” attributes describe the assembly tree as a whole, or the metadata as a whole.

These sections describe how to view and edit metadata attributes.

- Part and Assembly Metadata Attributes
- Viewing Part and Assembly Metadata Attributes
- Modifying Part and Assembly Metadata Attributes
- Viewing and Modifying Global Metadata Attributes

Part and Assembly Metadata Attributes

Each part and assembly has several attributes. Some attributes apply to both parts and assemblies, while other attributes apply to only parts. The attributes are listed in the following table:

Attribute Name	**Attribute Description**	**Applies To:**	
		Part	**Assembly**
Name	Name of Part or Assembly	**x**	**x**
Description	Description of Part or Assembly	**x**	**x**
Instance	Instance Number	**x**	**x**
File	The name of the file containing the original version of this entity. Often a reference to a PDM system.	**x**	**x**
Units	The unit system of this part or assembly.	**x**	**x**
Material_Description	The name or description of the material of which this part is composed.	**x**	
Material_Specification	The formal specification number of the material of which this part is composed.	**x**	
Density	The density of the material of which this part is composed. Setting it to a non-positive value will clear the attribute, as if there were no value assigned	**x**	

	assigned.		
Material_Volume	The volume of the region enclosed by this part. The material_volume is not calculated from the volumes associated with the part. It will often differ from the actual volume enclosed by this part's associated geometric volumes, and can also be manually set to any non-negative value. Setting it to a non-positive value will clear the attribute, as if there were no value assigned.	**x**	
Elemental_Composition	A string value describing the composition of the material, typically expressed as percentages of given elements.	**x**	

Viewing Part and Assembly Metadata Attribute Values

The easiest way to view a part or assembly's metadata attribute values is to select the item in the entity tree. The item's metadata attributes are listed in the property page.

A part or assembly's metadata attribute values can also be viewed using the **Metadata List** command:

```
Metadata List [<attribute_name>]
{Part|Assembly} "<path>"
```

The attribute_name should be one of the attribute names in the table above. If no attribute name is included in the command, all metadata attributes are listed.

Metadata attributes can also be listed based on a volume.

```
Metadata List [<attribute_name>] Volume <id>
```

This volume-based command works just like the part-based command, but lists the metadata for the part with which the volume is associated.

Modifying Metadata Attributes

A part or assembly's metadata attributes can be modified in the property page. Simply select the part or assembly in the entity tree, then click in the appropriate text field in the property page.

A part or assembly's metadata attributes can also be modified using the **Metadata Modify** command:

```
Metadata Modify <attribute> "new value"
{Part|Assembly} "<path>"
```

where **attribute** is one of the attributes listed in the table above. The specified attribute value will be changed to **new_value**.

There is also a volume-based version of the **Metadata Modify**

There is also a volume-based version of the **Metadata Modify**

command:

```
Metadata Modify <attribute> "new_value" Volume
<id>
```

The volume-based command works just like the part-based command, operating on the part with which the volume is associated. Note that if the specified volume is not associated with a part, a new part will be created and associated with the volume.

Viewing and Modifying Global Metadata

There are several attributes which do not describe any particular part or assembly. These "global" attributes describe the metadata as a whole:

Attribute Name	**Description**
Classification_Level	The level of sensitivity of the metadata. Usually one of the following: • Secret • Confidential • Unclassified
Classification_Category	The classification category. Usually one of the following: • Not Restricted • Restricted Data (RD) • Formerly Restricted Data (FRD) • National Security Information (NSI)
Weapon_Category	Sigma 1 through Sigma 15

Global metadata values can be viewed using the **Metadata List** command:

```
Metadata List <attribute_name>
```

Global metadata values can be modified using the **Metadata Modify** command:

```
Metadata Modify <attribute_name> "new_value"
```

For both commands, **attribute_name** should be one of the attribute names in the table above.

Working With Parts and Assemblies

Volumes can be organized into a hierarchical tree of parts, assemblies, and sub-assemblies. Assemblies may contain parts and other assemblies. Parts, on the other hand, may not contain sub-entities.

Each part and assembly has a name and an optional description. Other attributes may also be assigned, such as a material specification or a link to an entry in a PDM system. See Metadata Attributes.

The relationship between the geometric model and the assembly is determined by associating parts with volumes. A single part can be associated with any number of volumes, including zero volumes. A volume, however, can be associated with only one part.

As volumes are modified, CUBIT automatically maintains the appropriate relationships with parts. If a volume is associated with a part, and that one volume is split into multiple volumes through a webcut or some other operation, each of the resulting volumes is automatically associated with the original volume's part. Copying a volume will also result in the new volume being associated with the same part as the original volume.

- Identifying Parts and Assemblies
- Creating Parts and Assemblies
- Deleting Parts and Assemblies
- Associating Parts with Volumes
- Viewing All Assembly Information at Once
- Assemblies Tool in Power Tools

Identifying Parts and Assemblies

A part or assembly is identified by its assembly path. An assembly path is much like a directory path in a file system. It consists of the name of each ancestor in the assembly tree, separated by a forward slash. For example, a part named "p1" contained within the top-level assembly "a1" would be identified by the path "/a1/p1". If the part "p2" is part of the assembly "a2", and "a2" is a sub-assembly of "a1", then "p2" has the path "/a1/a2/p2".

More than one part or assembly may have the same name. To differentiate between parts or assemblies with the same name and path, each part also has an instance number. If two entities have the same name, they will not have the same instance number. For example, two parts named "p1" may be "p1 instance 1" and "p1 instance 2".

Instance numbers may be incorporated into assembly paths by placing the instance number in angled braces after a part or assembly name. For example, "p1 instance 3" is identified in a path as "p1<3>". Other examples of instance numbers in assembly paths include "/a1<1>/a2<1>/p1<3>" and "/a1/a2<1>/p1". Assembly paths are always allowed to incorporate instance numbers, but are only required to include as many instance numbers as it takes to avoid ambiguity. Note that some commands do accept ambiguous paths, selecting a random entity which matches the path.

Most commands which accept assembly paths also allow the path to be followed by an "instance" command option (for example, metadata list part "/a1/p1" instance 3). The instance option always refers to the instance number of the last item in the path (p1 in the example).

Creating Parts and Assemblies

Parts and assemblies can be created using the following commands:

```
Metadata Create {Assembly|Part}
"<absolute_path>" [Instance <instance>]
```

If the **instance** option is not included, CUBIT will assign an appropriate instance number to the new entity. If the instance option IS included, an entity with the specified name and instance number must not already exist or the command will fail.

Note that the path must be absolute, identifying each ancestor of the new entity. Any ancestors of the new entity which do not already exist are automatically created.

Deleting Parts and Assemblies

To delete a part or an assembly, use the Metadata Remove command:

```
Metadata Remove {Part "<path>" | Assembly
"<path>" [propagate]}
```

This will remove the specified part or assembly. If the **propagate** option is specified when removing an assembly, all contained parts and subassemblies will be removed automatically before the assembly itself is removed. Otherwise, assemblies will only be removed if they have no contents.

It is also possible to remove all parts and assemblies that have no association with geometric volumes in the model:

```
Metadata Clean
```

This can be extremely useful when importing geometry which has been simplified with metadata which has not been simplified. For example, eMatrix currently writes out the full assembly hierarchy even when exporting a simplified representation of the geometry.

Associating Parts with Volumes

The relationship between the geometric model and the assembly is determined by associations between parts and volumes. As stated previously, a part may be associated with any number of volumes, while a volume may be associated with only one part. The easiest way to associate a volume with a part is to use the entity tree in the user interface. Drag a volume in the tree onto a part in the tree, and the volume and part are now associated. Since a volume can only be associated with one part at a time, any previous association between that volume and a part is removed.

Part-to-volume associations can be created on the command line using the **Metadata Modify Path** command:

```
Metadata Modify Path "<part_path>" Volume
<ids>
```

The specified volume or volumes will be associated with the part specified by part_path. Any volumes already associated with the specified part will retain their association with the part.

Associations can be removed using the **Metadata Remove** command:

```
Metadata Remove Volume <ids>
```

After the Metadata Remove command has been issued, the specified volumes are no longer associated with any part.

The set of volumes associated with a given part can be modified using the **Metadata Replace** command:

```
Metadata Replace Part "<part_path>" Volume
<ids>
```

When the Metadata Replace command is issued, all associations the part may have had with any volumes are removed. New associations are then created with the specified volume or volumes.

Viewing All Assembly Information at Once

Once an assembly tree is created, all assemblies, parts, and part-to-volume associations can be viewed using the command:

```
Metadata List Tree
```

This will print the names of all parts and assemblies in the output window, along with the IDs of the volumes associated with each part.

It is also possible to view all parts, their properties, and their volume associations using a spreadsheet application such as Microsoft Excel. This is done by generating a file using the command:

```
Export Part_List "<filename>" [OverWrite]
```

This command writes an XML file in a format that Excel can convert to a spreadsheet. To do this, simply import the XML file into Excel as an XML List. The data can then be sorted and filtered by any of the parts' properties.

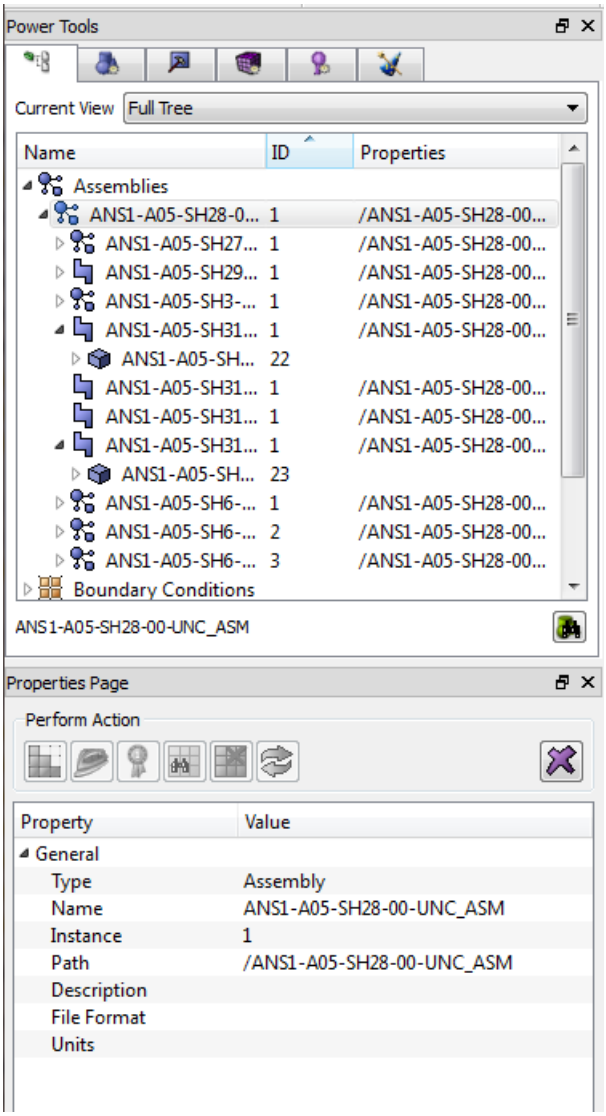
The **Export Part_List** command is particularly useful for identifying parts which are not correctly associated with parts. Among the fields that can be filtered is the **is-part** field. This field is FALSE for each volume that is not associated with a part. Filtering on this value will show a list of all volumes that are not associated with any part. The **volume-ids** field will show the ID of each unassociated volume, and the **volume-name** field will show each unassociated volume's name, if any.

It is equally easy to identify parts that are not associated with volumes. Display only those rows with a blank value in the **volume-ids** field to see a list of parts that have no associated volume.

Similar methods can be used to identify missing materials information. Fields can also be sorted to group the parts by material.

Metadata in the GUI

Metadata may be displayed and manipulated in the GUI. The tree view includes a category for metadata. The category is labelled "Assemblies" in the tree view. Users are able to drag volumes into parts on the tree. Also, selecting an Assembly or Part on the tree will cause the attributes for the entity to be displayed in the property page where further data manipulation is enabled.


Power Tools
Current View
Full Tree
Name
ID
Properties
Assemblies
ANS1-A05-SH28-0... 1 /ANS1-A05-SH28-00...
ANS1-A05-SH27... 1 /ANS1-A05-SH28-00...
ANS1-A05-SH29... 1 /ANS1-A05-SH28-00...
ANS1-A05-SH3-... 1 /ANS1-A05-SH28-00...
ANS1-A05-SH31... 1 /ANS1-A05-SH28-00...
ANS1-A05-SH... 22
ANS1-A05-SH31... 1 /ANS1-A05-SH28-00...
ANS1-A05-SH31... 1 /ANS1-A05-SH28-00...
ANS1-A05-SH31... 1 /ANS1-A05-SH28-00...
ANS1-A05-SH... 23
ANS1-A05-SH6-... 1 /ANS1-A05-SH28-00...
ANS1-A05-SH6-... 2 /ANS1-A05-SH28-00...
ANS1-A05-SH6-... 3 /ANS1-A05-SH28-00...
Boundary Conditions
ANS1-A05-SH28-00-UNC_ASM
Properties Page
Perform Action
Property
Value
General
Type Assembly
Name ANS1-A05-SH28-00-UNC_ASM
Instance 1
Path /ANS1-A05-SH28-00-UNC_ASM
Description
File Format
Units

Importing Geometry

- Importing ACIS Models
- Importing FASTQ Models
- Importing STEP Files
- Importing IGES Files
- Importing SGM Files
- Importing Facet Files
- Other Formats

Other Formats

Internally, CUBIT represents geometry as either ACIS solid model geometry or mesh-based geometry. CUBIT can import ACIS geometry in the native "sat" file format. CUBIT can also import STEP and IGES files and internally converts them into ACIS solid model geometry. For compatibility with Sandia legacy applications, CUBIT can import FASTQ input decks to create ACIS geometry, as well. CUBIT also contains experimental support for using SGM for representing geometry.

If you have geometry that has been created in another format, such as in SolidWorks, you will need to translate that geometry into something that Cubit can read. Many solid modeling packages have an Export ACIS .sat command, which is probably the easiest way of translating your model. If you do not have that option, there are some other possibilities.

- Try a different file format, such as STEP or IGES.
- As a last resort, contact the Cubit team. They might have other options for importing your file.

See Also

> **Importing a Mesh**

Importing ACIS Files

The command used to read an ACIS file is:

```
Import Acis '<acis_filename>' [No_bodies]
[No_surfaces] [No_curves][No_vertices][Group
{'<name>'|<id>}] [Binary|Ascii] [Show_Each]
[Sort] [XML '<xml_filename>'] [Attributes_On]
[Separate_Bodies] [merge_gloabally] [Heal]
```

The **import ACIS** command is the primary mechanism for generating geometry within CUBIT. ACIS parts can be generated and saved with CUBIT, but in most cases are developed within a 3rd party CAD package and exported for use in CUBIT. CUBIT provides the capability to import ACIS solid models and make modifications to them so they can be meshed. CUBIT incorporates the commercial ACIS libraries developed and maintained by Spatial Inc. for reading and writing ACIS format files. IGES and STEP format files can also be imported and exported to/from CUBIT using the Spatial's libraries.

Import Options

It is possible to include *free* entities (vertices, curves and surfaces) in the file. The default operation is to read all entities in the file whether they are included as part of a body or are free. By using any of the options **no_bodies, no_surfaces, no_curves,** or **no_vertices**, the user may exclude certain types of *free* entities.

The **group** option of the import command will allow the user to create a group for each set of imported geometry. The newly created group can later be accessed using the name or id specified with the group option.

The import capability of ACIS files supports both the ASCII format (.sat) and binary format (.sab). When importing, the filename extension will determine the default file type, be it ASCII or binary. A (.sat) extension will default to ASCII, while a (.sab) extension will default to binary. If you use a different file extension you can specify the type with the **[binary|ascii]** option. Binary files can be significantly faster but are not guaranteed to be upward compatible, nor cross-platform compatible. Therefore, it is recommended that models be archived in ASCII format.

Normally the numerical IDs of the geometric entities contained in the ACIS model are used directly within CUBIT. The sort option provides the capability to compress the IDs read from the ACIS file. The **sort** option does the same thing as the compress ids sort command, but combines it with the import command to remove a step in the process.

The **show_each** option is a graphics option that applies to how the volumes are shown as they are imported. If there are multiple volumes in the file, the graphics display will be updated between each volume during import.

The **xml** option will read assembly information and other metadata from an XML file in the DART metadata XML format. See the metadata documentation and the Analyst's Home Page for details.

The **attributes_on** option will enable attribute support for the file. Attributes include properties like entity color, entity id, and meshing scheme. Including the attributes option will only affect the current import. The settings will be restored to their previous settings after importing.

To retain any possible merge information when importing an ACIS file use the **attribute_on** command.

The **separate_each** option creates a separate body for each volume that is imported, preventing multi-volume bodies from being imported.

When importing, the use may specify the scope of the merge using **merge_globally**. The default behavior is to merge within the scope of the file being imported. With the **merge_globally** option, imported entities will merge with anything, including entities already in the Cubit session that have merge attributes on them.

Use the **heal** option to heal the entities when importing.

Case-Insensitive Entity Names

Entity names in Cubit are case-insensitive. This means that there is no difference between 'myvolume' and MyVolume'. When importing entity names as attributes any names that are case-sensitive duplicates will be made unique. This is done by appending one or more underscores, '_', followed by a number 1 to 9 or a letter A to Z. For example, two names 'myvolume' and 'MyVolume' encountered in that order will result in the names 'myvolume' and MyVolume_1'. Case-sensitivity can be toggled on/off with the command:

```
[Set] Case Sensitive Names [on|OFF]
```

Importing ACIS files at startup

ACIS files can also be imported using the **"-solid"** option when starting CUBIT from the UNIX command prompt. (See Execution Command Syntax for details.) Note that the filename must be enclosed in single or double quotes. This command will create as many bodies within CUBIT as there are bodies in the input file.

See also Exporting ACIS Files.

Importing Facet Files

CUBIT provides the capability to import a model composed of facets to create geometry. The command to import facets from a file is:

```
Import [Facets|AVS] ''<filename>"
[Feature_Angle <angle>] [LINEAR||Spline]
[MERGE|No_merge] [Make_elements] [Stitch]
[Improve]

Import STL ''<filename>" [Feature_Angle
<angle>] [Surface_Feature_Angle <angle>]
[LINEAR|Gradient|Quadratic|Spline]
[MERGE|No_merge] [Make_elements] [Stitch]

Import OBJ ''<filename>" [Feature_Angle
<angle>] [Surface_Feature_Angle <angle>]
[Make_elements]
```

Facets are simply triangles that have been stitched together to form surfaces. Faceted geometry representations are commonly used for graphics, bio-medical, geotechnical and many other applications that output a discrete surface representation. Upon import, the resulting geometry representation is Mesh-Based Geometry. Figure 1. shows an example of a faceted model and the resulting geometry created in CUBIT.

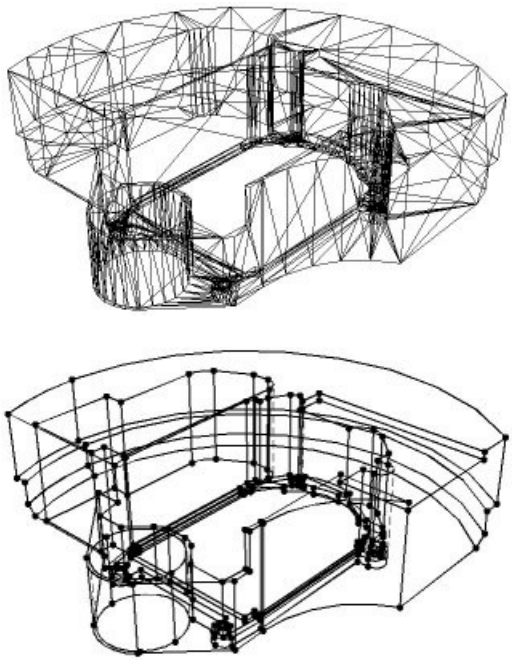


Figure 1. Example of faceted model and the resulting solid model created in CUBIT from the facets.

For convenience, the import facet command currently supports three different formats, facet, AVS and STL

- **Facet format**: The facet file format is a simple ASCII file that contains vertex coordinates and connectivities. The facet file format is described below.
- **AVS format**: The AVS format is a general geometry format that can support a variety of polygonal shapes. In CUBIT's implementation of the AVS import, it will support only triangles.
- **STL format**: Perhaps the most common format in the industry is Stereolithography (STL). CUBIT supports both ASCII and binary

forms of the STL format. While the STL format is adequate for graphics and visualization, it can be problematic for geometry applications such as CUBIT. Each triangle in the STL format is represented independently. This means that multiple definitions of a single vertex are included in the file. CUBIT will attempt to merge duplicate vertices to form a water-tight surface. In cases where the vertex locations may not correspond exactly, an optional **tolerance** argument may be used on the import command. The **tolerance** option is used only for STL format files.

- **OBJ format**: A geometry definition file format first developed by Wavefront Technologies for its Advanced Visualizer animation package. The file format is open and has been adopted by other 3D graphics application vendors. Cubit only supports importing OBJ files that contain tessellations with polygonal faces.

Facet File Format

The format for the ASCII facet file is as follows

```
n m
id1 x1 y1 z1
id2 x2 y2 z2
id3 x3 y3 z3
.
.
.
idn xn yn zn
fid1 id<1> id<2> id<3> [id<4>]
fid2 id<1> id<2> id<3> [id<4>]
fid3 id<1> id<2> id<3> [id<4>]
.
.
.
fidm id<1> id<2> id<3> [id<4>]
```

Where:

```
n = number of vertices
m = number of facet
id<i> = vertex ID of vertex i
x<i> y<i> z<i> = location of vertex i
fid<j> = facet ID of facet j
id<1> id<2> id<3> = IDs of facet vertices
[id<4>] = optional fourth vertex for quads
```

As noted above, the facets can be either quadrilaterals or triangles. Upon import, the facets serve as the underlying representation for the geometry. By default, the facets are not visible once the geometry has been imported. To view the facets, use the following command:

```
draw surf <id range> facets
```

Feature Angle

The **feature angle** option is used to specify the angle at which surfaces will be split by a curve or where curves will be split by a vertex. 180 degrees will generate a surface for every facet, while 0 degrees will define a single, unbroken surface from the shell of the mesh. The default angle is 135 degrees. This feature is identical to the feature angle option available when importing Exodus II files.

For the stl format, it is possible to independently control the feature angle for surfaces and curves. If **Surface_Feature_Angle** is specified, it controls the angle at which surfaces will be split by a curve, while **Feature_Angle** controls the angle at which curves will be split by a vertex. If **Surface_Feature_Angle** is not specified, **Feature_Angle** will control the angle for both surfaces and curves.

Smooth Curves and Surfaces

This option permits the use of a higher order approximation of the surface when remeshing/refining the resulting geometry. Default is to use the original facets themselves as the curve and surface geometry representation. If the facet model to be imported is to represent geometry with curved surfaces, it may be useful to apply this option. If the Spline option is selected, it will use a 4th order B-Spline approximation to the surface [Walton,96]. More information on using smooth approximation of the facets is available in Importing an Exodus II File.

Merge

This option allows the user to either merge or not merge the resulting surfaces. The default option is to merge adjacent surfaces. This results in non-manifold topology, where neighboring surfaces share common curves. The **no_merge** option, adjacent surfaces will generate distinct/separate curves.

Make elements

This option creates mesh elements from each of the facets on the facet surface.

Stitch

The stitch option is used with the facet or avs format files to try to merge vertices and triangles that are close. Figure 2 shows an example of where this might be employed. The model on the left contains facets that are not connected between the red and blue groups. In this case, the surfaces will not be water-tight, even though the vertices on the boundary between the two groups may be coincident. The **stitch** option attempts to eliminate the extra edge and vertex between the groups to form the model on the right. This option can be useful when importing facet files for 3D meshing. CUBIT's 3D meshing algorithms require a water-tight (closed) set of surfaces.

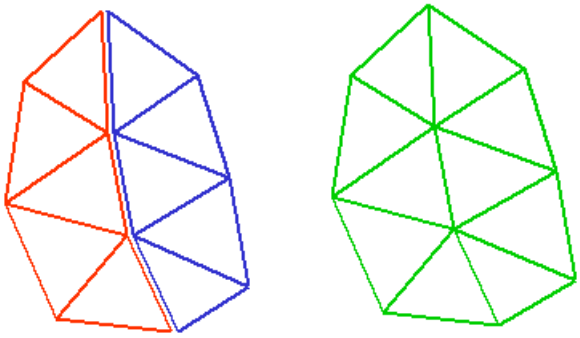
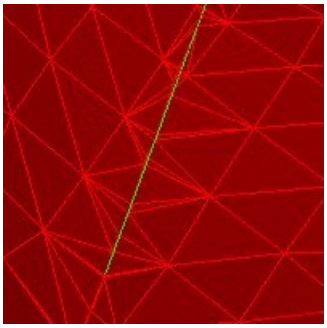


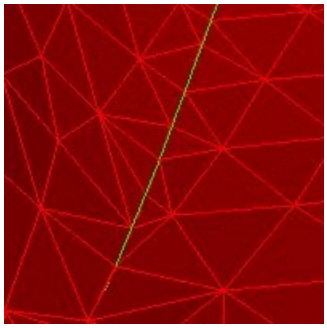
Figure 2. Example use of the stitch option on import.

Improve

The **improve** option will collapse short edges on the boundary of the triangulation that are less than 30% the length of the average edge length in the model. In some cases, short edges are the result of discrete boolean operations on the triangulation which may result in edges that are of negligible length. This option is particularly useful for boundaries where multiple surfaces come together at an edge. Figure 3. shows an example of where the improve option improved the quality of the triangles at the boundary. This option is especially useful if the facets themselves will be used for the FEA mesh.



Triangles near a boundary that have not been used the improve option



The same set of triangles where improve option has collapsed edges

Figure 3. Example use of the improve option

Importing FASTQ Files

CUBIT can read a FASTQ file and convert it into an ACIS model:

```
Import Fastq '<fastq_filename>'
```

Note that the filename must be enclosed in single or double quotes.

FASTQ is an older, 2d meshing tool; (Blacker 88.) FASTQ files are a series of commands much like a CUBIT journal file. All FASTQ commands are fully supported except for the "Body" command (it is unnecessary and ignored), the "corn" (corner) line type, and some of the specialized mapping primitive "Scheme" commands. Standard mapping, paving, and triangle primitive scheme commands are handled. The pentagon, semicircle, and transition primitives are not handled directly, but are meshed using the paving scheme. The FASTQ input file may have to be modified if the Scheme commands use any non-alphabetic characters such as `+', `(`, or `)'. Circular lines with non-constant radius are generated as a logarithmic decrement spiral in FASTQ; in CUBIT they will be generated as an elliptical curve.

Since a FASTQ file by definition will be defined in a plane, it must be projected or swept to generate three dimensional geometry. CUBIT supports sweeping options to convert imported FASTQ geometries into volumetric regions.

Importing Granite Files

As of version 13.0, native Granite models are no longer supported.

Importing IGES Files

The ACIS IGES translator provides bi-directional functionality for data translation between ACIS and the IGES (Initial Graphics Exchange Specification) format.

The commands to import IGES files are:

```
Import Iges '<iges_filename>' [No_bodies]
[No_surfaces] [No_curves] [No_vertices] [Group
{'<name>'|<id>}] [Nofreesurfaces]
[HEAL|noheal] [Logfile ['filename'] [Display]]
[Show_Each] [Sort]
```

Import Options

It is possible to include free entities (vertices, curves and surfaces) in the file. Default operation is to read all entities in the file whether they are included as part of a body or are free. By using any of the options **no_bodies, no_surfaces, no_curves**, or **no_vertices**, the user may exclude certain types of *free* entities.

The **group** option of the import command will allow the user to create a group for each set of imported geometry. The newly created group can later be accessed using the name or id specified with the group option.

The **nofreesurfaces** option will automatically convert free surfaces to bodies. By default this option is off.

By default, bodies are automatically healed when imported - if this causes problems, you can disable this option by using the **noheal** argument.

The **logfile** option specifies a file where informational messages generated during import of the STEP file will be written. The display option will display the file.

The **show_each** option is a graphics option that applies to how the volumes are shown as they are imported. If there are multiple volumes in the file, the graphics display will be updated between each volume during import.

Normally the numerical IDs of the geometric entities contained in the ACIS model are used directly within CUBIT. The **sort** option provides the capability to compress the IDs read from the ACIS file. The sort option does the same thing as the compress ids sort command, but combines it with the import command to remove a step in the process.

Note that the IGES import and export functionality might not be available on all 64-bit platforms.

See also Exporting IGES Files.

Importing STEP Files

The ACIS STEP translator provides bi-directional functionality for data translation between ACIS and the file format standard STEP AP203.

STEP AP203 is an international standard which defines a neutral file format for representation of configuration control design data for a product.

The command used to import a STEP file are:

```
Import Step '<step_filename>' [No_bodies]
[No_surfaces] [No_curves] [No_vertices]
[HEAL|Noheal] [Logfile ['filename'] [Display]]
[Show_Each] [Group {'<name>'|<id>}] [Sort]
[XML '<xml_filename>']
```

Import Options

- It is possible to include free entities (vertices, curves and surfaces) in the file. The default operation is to read all entities in the file whether they are included as part of a body or are free. By using any of the options **no_bodies**, **no_surfaces**, **no_curves**, or **no_vertices**, the user may exclude certain types of free entities.
- By default, bodies are automatically healed when imported - if this causes problems, you can disable this option by using the **noheal** argument.
- The **logfile** option specifies a file where informational messages generated during import of the STEP file will be written. The display option will display the file.
- The **show_each** option is a graphics option that applies to how the volumes are shown as they are imported. If there are multiple volumes in the file, the graphics display will be updated between each volume during import.
- The **group** option of the import command will allow the user to create a group for each set of imported geometry. The newly created group can later be accessed using the name or id specified with the group option.
- Normally the numerical IDs of the geometric entities contained in the STEP model are used directly within CUBIT. The **sort** option provides the capability to compress the IDs read from the STEP file. The sort option does the same thing as the compress ids sort command, but combines it with the import command to remove a step in the process.
- The **xml** option will read assembly information and other metadata from an XML file in the DART metadata XML format. See the metadata documentation and the Analyst's Home Page for details.
- If the geometric entities in the STEP file have names, they will be read in.
- Beginning with version 13.0, Cubit will read assembly information embedded in the imported STEP file. No additional arguments are required. The resultant assembly/part structure will be displayed in the GUI's main entity tree.

Import Settings

By default, names on bodies in STEP files are not read in. To change this, the following command is available:

```
[set] Read Step Body Names [on|OFF]
```

Exporting a STEP file from Pro/Engineer

To export a STEP file from Pro/ENGINEER, from the Export STEP

Dialog, Press Options.

In the file step_config.pro add the following:

```
STEP_EXPORT_FORMAT AP203_CD.
```

Also be sure your export option is set to Solids. If the geometry has problems in CUBIT, you may need to increase the geometry accuracy in Pro/ENGINEER.

See also Exporting STEP Files.

Importing SGM Files

CUBIT contains experimental support for importing SGM and STEP files using the Scalable Geometric Modeler (SGM) to represent the geometry.

The commands to import SGM files are:

```
Import SGM '<filename>' [restore_ids]
[read_colors]
```

Import Options

The **Import SGM** command can import either .sgm files or .step files. If given a .sgm file, the **restore_ids** option may be used to restore ids as they were in a previous SGM session. Restoration of IDs is not supported if a model was previously imported into SGM. The **read_colors** option is available if one wants to read colors found in the file. If **read_colors** is not given, colors will be automatically assigned by CUBIT based on the ID of the volumes.

Other CUBIT capabilities are limited when it comes to working with models imported into SGM. Visualization, list commands, and machine learning classification will work, but meshing, export and other operations are not yet supported.

Exporting Geometry

Geometry can be exported from CUBIT in a variety of formats, including the ACIS ".sat" and ".sab" formats as well as in more portable exchange formats like STEP and IGES.

- Exporting ACIS Files
- Exporting STEP Files
- Exporting IGES Files
- Exporting Facet Files

Exporting ACIS Files

Geometry can be exported from within CUBIT to the ACIS "sat" (ASCII) and "sab" (binary) formats. These formats can be used to exchange geometry between ACIS-compliant applications. The command used to export geometry is:

```
Export Acis [Debug] 'filename'
[<geometry_entity_list>] [Binary|Ascii]
[Current] [Overwrite]
```

The filename should be enclosed in single or double quotes. By convention, binary and ASCII ACIS files use the .sab and .sat filename extensions, respectively. If a geometry entity list is not specified, the entire ACIS model is exported. A geometry entity list is specified in the same format used for other CUBIT commands (See Entity Specification). Note that the model is saved as manifold geometry, and will have that representation when imported back into CUBIT (See Non-Manifold Topology and Geometry Merging.)

When exporting, the filename extension will determine the default file type, either ASCII or binary. A .sat extension will default to ASCII; a .sab extension will default to binary. If you use a different file extension you can specify the type with the **[binary|ascii]** option (with an unsupported extension exporting will default to ASCII but importing requires the type to be specified). Binary files can be significantly faster but are not guaranteed to be upward compatible nor cross-platform compatible (although testing has determined compatibility between NT and HP/UX).

In the GUI version, the **current** option will set the default filename for autosave (cntrl-S or File->Save (auto inc)) to the imported filename. Also, the filename is then set in the window titlebar.

When exporting with the "**file overwrite**" option on, the software will check to see if the file exists already, and if it does, exporting will fail in the command line version or ask to confirm the overwrite in the GUI version of CUBIT. The **overwrite** option will override this option and overwrite the file. The "file overwrite" option defaults to ON in the GUI version, OFF in the command line version.

When exporting, you can set the version of the Acis geometry. This allows backwards compatibility to previous versions of Cubit or other Acis-based applications. The command to change the Acis geometry engine version is:

```
Set Geometry Version [version_number]
```

where **version_number** can be one of the following:106, 107, 201, 300, 301, 401, 402, 403, 500, 501, 502, 503, 600, 601, 602, 603, 700, 701, 702, 703, 704, 705, 800, 1007, 1100, 1200, 1300, 1400, 1500, 1600, 1700, 1800, 1900, 2100, 2200, 2401, 2502. Note that you cannot set a version number that is higher than that of your current engine. For example, Cubit 6.0 was based on Acis 6.2, so you cannot set a geometry version of 700.

To retain any merging information during export of an ACIS file, **set attribute on** and **set attribute off** need to be used before and after the **export acis** command.

See also Importing ACIS Models.

Exporting Facet Files

Facet files may be exported directly, or by converting from an ACIS representation. The syntax for exporting facet files is:

```
Export Facets 'filename' <entity_list>
[Overwrite]
```

The overwrite function allows you to overwrite an existing facet file.

STL facet files may be generated from geometry or from a triangle mesh. The syntax for exporting to the STL format is:

```
Export STL [ASCII|binary] 'filename'
[<entity_list>] [tri <id_range>] [angle=15]
[mesh|fast] [sidesets|sideset<ids>]
[Overwrite]
```

The **[entity_list]** option is a list of geometric entities (bodies, volumes, or surfaces). By default, the graphics facets for the geometric entities will be written to the STL file. The **[angle]** keyword specifies the dihedral angle used during facet generation. By default, a "water-tight" set of graphics facets is exported for solid volumes. If a water-tight set of facets is not of interest to the user and performance is more important, the **fast** option can be used. It generates a faceting that does not take the extra step of ensuring water-tightness. To export the triangle mesh on the geometric entities, instead of the graphics facets, specify the **[mesh]** keyword. Note that STL export of quad meshes is not supported.

The **[sidesets]** or **[sideset <ids>]** options will export all or the specified sidesets respectively in your model as surface designation for any or all triangles in the file. If present, one sideset will be generated for each surface designation in the STL file. Following is an example surface designation in an STL file. It would appear following all triangles.

```
surface 1
  0 1 2 3 4 5 6 7 8 9
  10 11 12 13 14 15 16 17 18 19
  20 21 22 23
endsurface 1
```

The id following the surface designation will be used as the sideset ID.

Alternatively, a list of mesh triangles can be specified for export. If neither geometry entities nor mesh are specified, all volumes and sheet bodies are written out.

Exporting IGES Files

The ACIS IGES translator provides bi-directional functionality for data translation between ACIS and the IGES (Initial Graphic Exchange Standard) format. The command to export IGES files is:

```
Export Iges 'filename'
[<geometry_entity_list>] [Solid] [Logfile
['filename'] [Display]] [Overwrite]
```

As with ACIS file export, you can specify which individual entities to export. If unspecified, all ACIS entities are exported.

The **logfile** option is used to save information regarding the conversion to IGES format. This information saved to a file with the name specified by the user, or named 'iges_export.log' by default. When running the GUI version of CUBIT, the logfile can be displayed in a dialog window by using the **display** option.

The **solid** option allows solid volumes to be exported as Manifold Solid B-Rep Objects (MSBO). Without this option, the iges file is simply a collection of stand-alone surfaces.

The overwrite option works the same as with ACIS file export.

See Importing IGES Files for information on setting up the IGES import and export functionality.

Note that the IGES import and export functionality might not be available on all 64-bit platforms.

Exporting STEP Files

CUBIT can export geometry to the STEP format, an emerging standard for storing geometry and other information. The STEP AP203 and STEP AP214 standards are supported. It is recommended to use AP214 for exchange of geometry information with CUBIT. The command used to export a STEP file is:

```
Export Step 'filename'
[<geometry_entity_list>]  [Logfile
['filename'] [Display]] [Overwrite]
```

As with ACIS file export, you can specify which individual entities to export. If unspecified, all ACIS entities are exported.

The **logfile** option is used to save information regarding the conversion to STEP format. This information saved to a file with the name specified by the user, or named 'step_export.log' by default. When running the GUI version of CUBIT, the logfile can be displayed in a dialog window by using the **display** option.

The overwrite option works the same as with ACIS file export.

If bodies, volumes, surfaces, curves, or vertices have names, they will be written into the STEP file.

See Importing STEP Files for information on setting up the STEP import and export functionality.

Geometry Deletion

Geometry can be deleted from the model using the following command:

```
Delete [Body | Surface | Curve | Vertex]
<id_range>
```

Any type of Body can be deleted, whether it is based on solid model geometry or another representation. Other entities (Surface, Curve, Vertex) can be deleted when they are "free", i.e. when they are not contained in an entity of higher topological order (Body, Surface or Curve, respectively); this type of geometry is often created from the lowest order topology up.

Geometry Orientation

The orientation of surface and curve geometry is the direction of the normal and tangent vectors respectively.

Each surface has a forward (or top) side. The evaluation of the surface normal at any point on the surface will return a vector at that point, orthogonal to the surface and directed towards the forward side of the surface. The mesh faces generated on each surface will have the same normal direction as their owning surface.

Each curve has a forward direction and a corresponding start and end vertex. The direction of the curve is from start to end vertex. The evaluation of the tangent vector of the curve at any point along the curve will result in a vector that is both tangent to the curve and pointing in the forward direction of the curve (towards the end vertex along the path of the curve.) The mesh edges created on each curve will be oriented in the same direction as their owning curve. The exported nodes and edges of a curve mesh will be written in the order they occur along the path of the curve.

Higher-dimension geometry has uses lower-dimension geometry with an associated sense (forward or reversed) for each lower-dimension entity. For example, a volume as a sense for each surface used to bound the volume. If the surface normal points outside the volume, then the volume uses the surface with a forward sense. If the surface normal points into the interior of the volume, the volume uses the surface with a reversed sense. Similarly a surface is bounded by a set of curves forming a loop such that the direction of the loop and the sense of each curve results in a cycle that is counter-clockwise around the surface normal.

Adjusting Orientation

By default, a surface is oriented so that its normal points OUT of the volume of which it is a part. For a merged surface (a surface which belongs to more than one volume) or a free surface (a surface that belongs to no volume, also known as a sheet body), the orientation of the surface is arbitrary. The orientation of a surface influences the orientation of any elements created on that surface. All surface elements have the same orientation as the surface on which they are created. The following commands are available to adjust the normal-direction for a surface:

```
Surface <id_range> Normal Opposite

Surface <id_range> Normal Volume <id>
```

The orientation of a surface can be flipped from its current orientation by using the "Opposite" keyword. The orientation of a merged surface can be set to point OUT of a specific volume by specifying that volume in the "Volume" keyword.

Occasionally, volumes will be created "inside-out". The command:

```
Reverse {Body|Volume|Surface} <id_range>
```

will turn a given volume, surface, or body inside out. This should be equivalent to reversing the normals on all the surfaces. This shouldn't be encountered very often, as it is a very rare condition.

The following commands are available to adjust the tangent direction of a curve:

```
Curve <id_range> Tangent Opposite

Curve <id_range> Tangent {Forward|Reverse}
```

```
Surface <id>

Curve <id_range> Tangent {Start|End} Vertex
<id>
```

The first command reverses the tangent direction of the curve. The second command sets the tangent direction such that it is used by a specific surface with a specified sense. The third command sets the tangent direction of the curve such that the curve starts or ends with the specified vertex. For the latter two forms of the command, the curve must be adjacent to the specified surface or vertex.

The below command can be used to change the orientation of multiple curves at once. With the direction option, the curve will be oriented along the specified direction. With the location option, the vertex closest to the give location becomes the start vert in the oriented curve. The curve orientation can be reversed using the opposite argument. Also, a vertex id can be specified to make it the start vertex in the oriented curve.

```
Curve <id_range> Orient Sense {direction
(options)|location (options)|vertex
<id_range>} [Opposite]
```

The above command is useful in changing the orientation of multiple curves at once using various options described. This becomes helpful, e.g., when bias is applied on multiple curves. By default, bias depends on the orientation of the curve, i.e., bias begins at start vertex.

Entity Measurement

To output various properties of entities, the following **Measure** command options are available.

- Measure Between
- Measure Small
- Measure Angle
- Measure Void
- Measure Volume
- Measure Surface

Measure Between

```
Measure Between { { Vertex|Curve|Surface
|Volume|Node} <id1> | Location <options> |
Plane <options> | Axis <options> } With {
{Vertex|Curve|Surface|Volume|Node} <id2> |
Location <options> | Plane <options> | Axis
<options> }

Measure Between {Surface|Curve} <id1 >
[Surface|Curve] <id2> [Node]

Measure Between
{Vertex|Curve|Surface|Volume|Node|Edge|Face|Tri|Hex|Tet}
<id1> With
{Vertex|Curve|Surface|Volume|Node|Edge|Face|Tri|Hex|Tet}
<id2>
```

The **Measure Between** command outputs the distance from one entity, location, plane, or axis to the next. The two entities in the command should be separated by the word "with". The result will always be the minimum distance between entities. For example, measuring between two spheres will output the minimum distance between them, not the distance between centroids. The example shown below will output the minimum distance between vertex 1 and surface 2.

```
measure between vertex 1 surface 2
```

The second form of the command is just for surfaces or curves and contains the **Node** argument. This argument attempts to measure between corresponding nodes on a pair of surfaces or curves. The command tries to determine a one-to-one mapping of nodes between the pair. It returns the greatest distance between any two nodal pairs, least distance between any two nodal pairs, and average distance between all of the nodal pairs. The mapping algorithm works best on surfaces if they are parallel.

The last form of the command measures between any geometry or mesh entities. The measurement to the mesh entities is to their center (i.e. the averaged vector location of all of the nodes belonging to the mesh entity).

With 2 entities selected in the graphics window, the user can right click one of the entities and measure the distance between the entities.

Measure Small

```
Measure Small {Length|Area|Volume|All}
{Body|Surface} <id_list>
```

The **Measure Small** command locates all of the lengths, areas, or volumes smaller than the **Measure Small Tolerance** setting. Entities meeting the small tolerance criteria are listed in the output window and

typically highlighted in the view port. The following two commands set the small tolerance to 0.1 and output all of the curves within body 1 with lengths at or below the small tolerance.

```
set measure small tolerance 0.1

measure small length body 1
```

Measure Angle

```
Measure Angle { Direction <options> | Plane
<options> | Axis <options> } With { Direction
<options> | Plane <options> | Axis <options> }
```

The **Measure Angle** command displays the interior angle between the two entered entities. When a plane and a direction are specified, the angle between the direction vector and its projection into the plane is displayed. The measured angle represents the distance between the orientations of entities, and does not require the entities to intersect. Angles of model features can be measured by using the various options associated with the Direction, Planes, and Axis commands.

```
measure angle direction tangent curve 1 with plane surf 1
```

Measure Void

```
Measure Void [Face | Tri] <range>[No_Checks]
```

The **Measure Void** command takes a closed list of quadrilaterals or triangles and calculates the volume of the internal region defined by the given list of elements. This command assumes that the normals on the given elements are consistently ordered. If the normals are pointing away from the interior of the void, the reported volume may be negative. This command will check to ensure that the given elements do form a closed, manifold shell, otherwise an error is reported. Common uses will be to calculate the volume of an internal void for use in determining bulk element properties for a thermal analysis.

Rather than issuing an error, the **no_checks** option does not check for closure of the faces and will compute a void volume regardless of their watertightness. This is useful if faces are all touching, but may not have complete topological closure.

Measure Volume

```
Measure Volume <range> [Overlap | Shell]
```

The **Measure Volume** command prints summary information about the specified volumes and surfaces of these volumes, such as average volume, minimum volume, angles, average surface area, etc. If the **shell** option is specified information about the shells of the volumes is additionally printed. The **overlap** option does not print any summary information, but only reports pairs of intersecting volumes.

Measure Surface

```
Measure Surface <range>
```

The **Measure Surface** command prints summary information about the specified surfaces and curves of these surfaces, such as average area, minimum area, angles, minimum curve length, etc.

Mesh Generation

- Meshing the Geometry
- Interval Assignment
- Meshing Schemes
- Mesh Quality Assessment
- Mesh Modification
- Mesh Validity
- Mesh Adaptivity and Sizing Functions
- Mesh Deletion
- Free Meshes
- Skinning a Mesh
- Creating Cohesive Elements

The methods used to generate a mesh on existing geometry are discussed in this chapter. The definitions used to describe the process are first presented, followed by descriptions of interval specification, mesh scheme selection, and available curve, surface, and volume meshing techniques. The chapter concludes with a description of the mesh editing capabilities, and the quality metrics available for viewing mesh quality.

Element Types

For each entity topology-type in the model geometry, CUBIT can discretize the entity using one, or several, types of basic elements, for each order entity in the geometry (vertex, curve, etc.). CUBIT uses a basic element designator to describe the corresponding entity, or entities, in the mesh, and a given geometric topology entity can be discretized with one, or several, of basic elements types in CUBIT. For example, a geometric surface in CUBIT is discretized into a number of faces, where faces is the basic element designator for surfaces. These faces can consist of two types of basic elements, quadrilaterals or triangles. The basic element designators corresponding to each type of geometric entity, along with the types of basic elements supported in CUBIT, are summarized in the table below.

For each basic element, CUBIT also supports several element type definitions, whose use depends on the level of accuracy desired in the finite element analysis. For example, CUBIT can write both linear (4-noded) and quadratic (8- or 9-noded) quadrilaterals. The element type definition is specified after meshing occurs, as part of the boundary condition specification. See Finite Element Model Definition for a description of that process and the various element types available in CUBIT.

Each mesh entity is associated with a geometric entity which "owns" it. This associativity allows the user to mesh, display, color, and attach attributes to the mesh through the geometry. For example, setting a mesh attribute on a surface affects all faces owned by that surface.

Mesh Generation Process

Starting with a geometric model, the mesh generation process in CUBIT consists of four primary steps:

Set interval size and count for individual entities or groups

The size or interval is always applied to a specific geometric entity. For example:

volume 1 size 2.0

Set mesh schemes

CUBIT supports numerous meshing schemes for meshing solid model entities. For example:

volume 1 scheme sweep

Generate the mesh for the model

Use the mesh command to generate the mesh on a specified geometric entity. For example:

mesh volume 1

Inspect mesh for quality and suitability for targeted analysis

CUBIT provides various quality metrics for the user to verify the suitability of the mesh for analysis. The quality command can be used to check the elements generated on a specific geometric entity. For example:

quality volume 1

There are also mechanisms for improving mesh quality locally using smoothing and local mesh topology changes and refinement. For complex models, this process can be iterative, repeating all of the steps above.

The mesh for any given geometry is usually generated hierarchically. For example, if the mesh command is issued on a volume, first its vertices are meshed with nodes, then curves are meshed with edges, then surfaces are meshed with faces, and finally the volume is meshed with hexes. Vertex meshing is of course trivial and thus the user is given little control over this process. However, curve, surface, and volume meshing can be directly controlled by the user. Each of the steps listed are described in detail in the following sections.

Geometry Entity Type	Basic Element Designator	Basic Element(s) In CUBIT
Vertex	Node	Node
Curve	Edge	Edge
Surface	Face	Quadrilateral, Triangle
Volume (or Body)	Element	Hexahedron, Tetrahedron, Pyramid, Wedge

Meshing the Geometry

After assigning interval or sizing attributes to a geometric entity and a meshing scheme is applied, the geometry is ready to be meshed. To mesh a geometric entity, use the command:

```
Mesh <entity> <id_range> [GLOBAL|Individual]
```

The **<entity>** to be meshed may be any one of the following:

Body
Volume
Surface
Curve
Vertex

The **Global** and **Individual** options affect how the constraints are gathered for interval matching. With the Global option, the interval constraint equations are calculated from all entities in the entity list. The Individual option calculates the interval constraint equations from each entity individually. The Global option is the default.

Default Scheme and Interval Selection

If either interval settings or schemes have not already been set on the entities being meshed, CUBIT will do its best to automatically set one or both of these attributes. See Auto Scheme Selection and Auto Specification of Intervals for a description of how CUBIT chooses these attributes. In cases where the automatic scheme selection algorithm fails to select a scheme for the geometry, the meshing operation will fail. In this case explicit specification of the meshing scheme and/or further geometry decomposition may be necessary.

Continuing Meshing After a Mesh Failure

Frequently when meshing large assemblies containing a number of volumes, the mesh command can be applied to a group of volumes with the same mesh command. Typically, if a mesh failure is detected, the meshing operation will continue to mesh the remaining volumes specified at the command line. The following command permits the user to override this feature to discontinue meshing additional volumes and return to the command line immediately after a mesh failure is detected:

```
Set Continue Meshing [ON|Off]
```

The default for this command is **ON**.

Turning this setting **OFF** is useful when meshing assemblies where a meshing failure of one volume would adversely affect the meshing of adjoining volume(s). This occurs frequently when meshing a sweep group using the sweep scheme.

Interval Assignment

- Interval Firmness
- Setting Interval Sizes
  - Explicit Specification of Intervals
  - Explicit Specification of Intervals Using Interval Size
  - Periodic Intervals
  - Relative Intervals
  - Automatic Specification of Intervals
  - Vertex Sizing and Automatic Curve Biasing
- Additional Interval Constraints
- Match Intervals Solver
- Examining Intervals
  - Listing Intervals
  - Mesh Preview

"Intervals" means the number of mesh edges on a curve (or across a periodic surface). "Interval matching" is the process of assigning intervals that are close to the user-desires, and satisfy the constraints imposed by the various quad and hex meshing algorithms (e.g., equal intervals on opposite sides of a mapped surface). The desired number of intervals (goals) are set on geometric entities by either specifying the interval count or size. "Soft" user-set and auto-set mesh sizes and interval counts are goals. Any hard counts are part of the constraints. The constraints are determined by the meshing schemes, as well as the vertex types (corners) of surfaces, and the edge-types of volumes, and sweep directions. The user may set additional constraints, including upper and lower bounds.

Additional Interval Constraints

Curve Minimum | Maximum

Rather than specifying a hard interval count, which may overconstrain the interval matcher, the user can specify an upper and lower bound that is acceptable. Typical uses are sweeping a complex assembly where the normal compromises lead to too many (or few) intervals on specific curves, or thin layers.

```
{Group|Body|Volume|Surface} <range>
{Interval | Size | Periodic Interval}
{[Lower]|Upper}
Bound {On|Off|<bound>}
```

```
Curve <range> {Interval | Size}
{[Lower]|Upper}
Bound {On|Off|<bound>}
```

Loop Minimum | Maximum

The user can specify the minimum and maximum number of intervals on a loop. This is mostly used for small loops, such as holes drilled in a plate, to ensure they have at least 4 or 6 intervals. If no entties are specified, then the setting is applied globally to all current and future loops in the model.

```
set [{group|volume|surface|curve} <range>]
interval loop minimum {<count>|default}
```

```
set [{group|volume|surface|curve} <range>]
interval loop maximum {<count>|default}
```

```
list [{group|volume|surface|curve} <range>]
interval loop [minimum] [maximum] [default]
```

Manual Constraints

General Constraints

```
Curve <curve_id_range> Interval
{Equal_to|Greater_than_equal|Less_than_equal}
[Curve] <curve_id_range> [ Extra <intervals>]
```

```
Curve <curve_id_range> Interval
{Equal_to|Greater_than_equal|Less_than_equal}
Extra <intervals>
```

These sets a constraint that the interval matcher resolves when it is run. E.g., the command "curve 2 3 greater_than_equal curve 4 5 extra 4" stores the inequality constraint "c2 + c3 >= c4 + c5 + 4" in the interval matcher. While this can resolve quality issues, it is also an easy way to make the interval matching problem infeasible.

Sets of Curves with the Same Intervals

Interval **same** is a two way constraint that is resolved immediately. If the user subsequently changes the interval on a curve in the set, then the other curves are changed immediately. One problems is if the user hard sets an interval on one curve and then sets a size on another, the hard set interval on the other curve is not changed.

```
Curve <range> Interval {Same|Different}
```

```
List Curve [ <curve_id_range> ] Interval Same
```

Specifying that curves have the "same" intervals stores them in a set. More curves may be added to an existing set, and sets merged, by future commands. The current contents of the affected sets are printed after each command. A curve may be removed from a set by specifying that its intervals are "different."

Even

The user can also constrain the parity of intervals on curves:

```
{Curve|Surface|Volume} <range> Interval {Even
| Odd}
```

If **Even** is specified, then during subsequent interval setting commands and during interval assignment, curves are forced to have an even number of intervals. If the current number of intervals is odd, then it is increased by one to be even. If **Odd** is specified then intervals may be either even or odd. Setting intervals to even is useful in problems where adjoining faces are paved one by one without global interval assignment.

Vertex Sizing and Automatic Curve Biasing

Sizes can now be specified on vertices to control biasing along curves. If a curve has a bias scheme the vertex sizes will be honored, even if it is inherited from parent geometry.

Set a size on a vertex with the following command:

vertex <id> size <size>

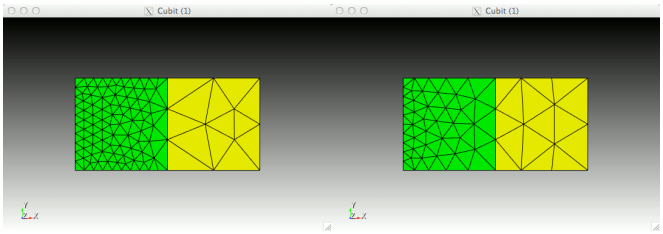
Bias can be turned on with:

curve <id> scheme bias

For tri/tet meshing, curve biasing is on by default to generate higher quality tri/tet meshes. Not only is the difference noticeable when setting sizes on vertices, but it is also noticeable when setting various sizes on connected curves, surfaces, or volumes. To turn curve biasing off issue the following command:

curve<id> scheme equal

In the following examples, the surfaces have been given sizes. In the first graphic auto bias is not enabled. In the second graphic auto bias is enabled.



When auto bias is enabled sizes on vertices are respected. If a size hasn't been directly set on a vertex the size is inherited from the parent(s). If there are multiple parents the inherited size is averaged. In the examples shown above the sizes of the vertices attached to both surfaces was an average of the two surface sizes. That affected the biasing while curve meshing.

Explicit Specification of Intervals

The density of mesh edges along curves is specified by setting the actual number of intervals or by specifying a desired interval size. The number of intervals can be explicitly set curve by curve, or implicitly set by specifying the intervals on a surface or volume containing that edge. For example, setting the intervals for a volume sets the intervals on all curves in that volume.

The command to specify the number of intervals at the command line is:

{Curve|Surface|Volume|Body|Group} <range>
Interval <intervals>

When setting interval counts for surfaces, volumes, bodies and groups, an interval's firmness of ***soft*** is assigned to the owned curves. When setting the interval count for a curve, a firmness of ***hard*** is assigned.

The user can scale the current intervals with the following commands. Scaling is done on an entity by entity basis.

{Curve|Surface|Volume|Body|Group} <range>
Interval Factor <factor>

Explicit Specification of Intervals Using Interval Size

The number of intervals along curves can be specifying by setting a desired interval size. The interval size can be explicitly set curve by curve, or indirectly set by specifying the interval size on a surface or volume containing that curve. The size for an entity is determined with the following method. If the entity has a size explicitly set then that size is used. Otherwise the entity averages the size determined for its parents. If an entity doesn't have any parents then a size is automatically calculated from all of the geometry in the model. If the auto size functionality is turned off then a default size of 1.0 is used. Some meshing algorithms may calculate a different default size.

For example, Suppose you have two volumes that share a face and corresponding curves. If the size on volume one is set to 1.0 and the size on volume two is set to 3.0 then the size for the common face will be set to 2.0. The size for the remaining faces on volume one and two will be 1.0 and 3.0 respectively. The size for the common curves will be set to 2.0.

The command to specify the interval size at the command line is:

> **{Curve|Surface|Volume|Body|Group} <range> [Interval] Size <interval_size>**

Interval sizes set directly on an entity are given the type “user_set”. Interval sizes determined from parents or automatically calculated are give the type “calculated”.

When interval matching or meshing the interval count for each curve is computed by dividing the curve's arc length by the specified interval size. Interval counts calculated in this manner are considered to have a default firmness of soft.

The user can scale the current intervals or size with the following commands. Scaling is done on an entity by entity basis.

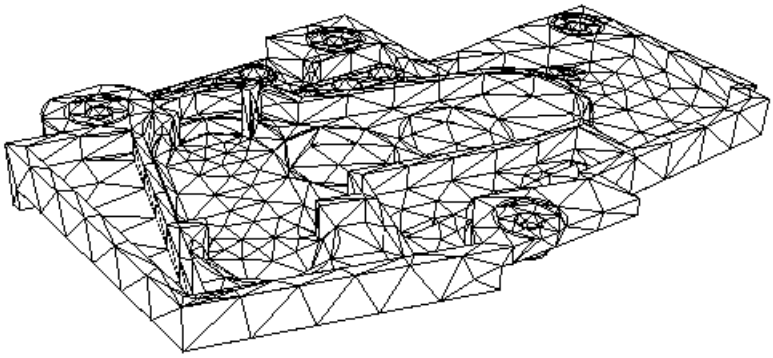
> **{Curve|Surface|Volume|Body|Group} <range> [Interval] Size Factor <factor>**

Automatic Specification of Interval Size

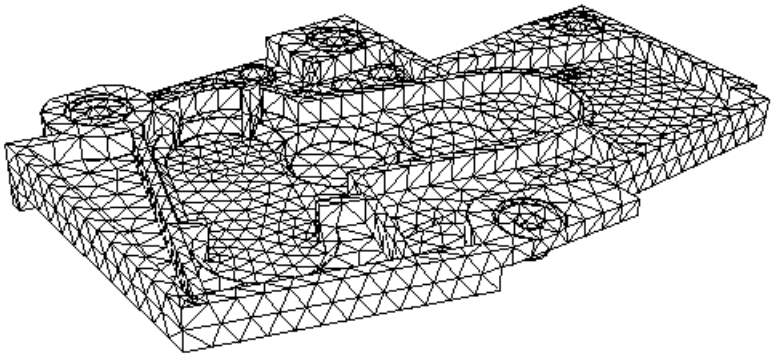
In addition to specifying intervals explicitly based on a known count or size, CUBIT is able to compute interval sizes automatically based on characteristics of the model geometry. The following automatic interval size setting command can be used:

{geom_list} Size Auto [Factor <factor>] [Individual] [Propagate]

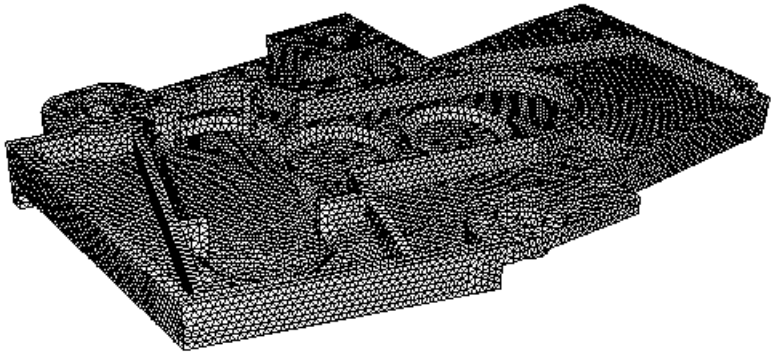
Vertices are not valid in the **geom_list** for this command. Automatic interval size assignment works by examining the geometric characteristics of the entities in the **geom_list** and assigning a heuristic size to the entities and their child entities. The factor may be a floating point number between 1.0 and 10.0, where 1.0 represents a fine interval size and 10.0 represents a coarse size. Figure 1 shows an example of different auto size specification on a CAD model.



(a) auto size factor = 7.0



(b) auto size factor = 5.0



(c) auto size factor = 1.0

The user may assign the interval size to be the arc length of the smallest curve contained in the specified entity or entities using the following command:

{geom_list} Size Smallest Curve

Vertices are not allowed in the **geom_list** for this command. This command assigns a ***soft*** interval firmness.

Automatic Interval Size Specification

An automatic interval size with an auto size factor of 5 will automatically be computed and applied to any curve for which the following is true:

1) Intervals have not been explicitly defined by the user for a curve or its owning entities.

2) An Interval size has not been explicitly defined by the user for a curve and it is not possible to determine an interval size from its owning entities.

This automatic interval size is based upon all the geometry in the model. The automatic interval size specifications can be overridden easily by specifying another auto size factor or an explicit interval size.

If an auto size factor of 5 is undesirable for most meshing operations, the default factor may be changed by using the following command:

Set Auto Size Default <value>

where **value** is a number from 1 to 10. This will be the default auto size factor used when either a factor has not been specified on the size auto command or when an automatic interval size specification is used.

In previous versions of CUBIT a default interval of 1 was assigned to all entities. If this behavior is still desired, the following command may be used to enforce this condition:

Set Default Autosize [ON|off]

Maximum Spanning Angle on Arcs

On many CAD models, arcs or small holes require that a finer mesh be specified around these entities in order to maintain reasonable mesh quality. To facilitate this, the user may specify the maximum angle an element edge may span on an arc. To change or list the maximum arc span, use the following commands

Set Maximum Arc_Span <angle>

List Maximum Arc_Span

The angle parameter must be a positive value less than 360. The maximum arc span setting will only be used if there is not already a user defined interval set on the arc, and if the interval setting produces mesh edges which exceed the maximum spanning angle. Figure 2 shows the effect of three different maximum arc_span settings on a small hole using the pave scheme.

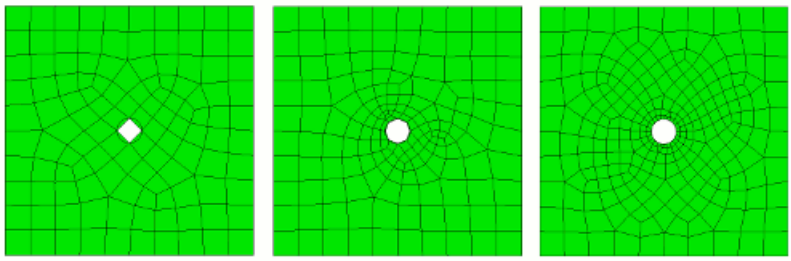


Figure 2. Maximum arc_span settings of 90, 45 and 15 degrees respectively.

Default arc span setting: In addition to setting an automatic size factor, if there are otherwise no user-defined interval sizes defined on an arc and no **maximum arc_span** has been set by the user when a tetrahedral mesh or triangle mesh is defined, a maximum spanning angle of **60 degrees** will be used. Removing the use of the arc_span setting can be accomplished with the following:

```
Set Maximum Arc_Span Default
```

Note that once interval sizes have been defined when the entity has been meshed, it may be necessary to reset the interval settings **(reset {geom_list})** to use a new maximum arc span setting when remeshing.

Interval Matching

Each meshing scheme in CUBIT imposes constraints on the intervals assigned to the curves bounding the surface or volume. For example, map meshing a surface requires that intervals on opposite sides are equal. Meshing any surface with quadrilaterals, regardless of scheme, requires an even number of intervals on its boundary. For connected surfaces and assemblies, these interval constraints must be resolved globally to ensure that each surface and volume will be meshable with the assigned scheme. Beyond satisfying these constraints, the number of intervals (mesh size) should be what the user wants, or as close to it as possible. The global solution technique implemented in CUBIT is referred to as "interval matching."

Interval matching is automatically performed by the **mesh** command before generating elements. Invoking interval matching manually is useful if the user wishes to mesh only some entities but wants to ensure it will be possible to mesh all entities later: e.g. match intervals on all the volumes, but just mesh surface 3 for now. Interval matching can also be called to check whether the assigned schemes, corners, edge types, and intervals are compatible.

The command syntax for manually matching intervals is the following:

```
Match Intervals {Surface|Volume|Body|Group}
<range>
```

Here the entity list can be any mixed collection of groups, bodies, volumes, surfaces and curves.

Troubleshooting

If the quality of the interval solution is poor, try changing the interval count on individual entities, or, e.g., changing some schemes from map to pave to provide more freedom, and rerunning interval matching.

There may be no interval solution. To improve the chances of finding a solution, at least as a test, try setting intervals to soft firmness. Even with soft intervals, a solution might not exist if the mapping corners or sweep directions do not line up right.

If there is no solution, the following command may help in determining the cause (this is not yet supported by IIA in Cubit 15.6):

```
Match Intervals {Surface|Volume|Body|Group}
<range> [Seed Curve <range>]
[Assign Groups [Only|Infeasible]] [Map|Pave]
```

Specifying **Assign Groups** will create groups that contain independent subproblems of the global problem. Specifying **Assign Groups Only** will group independent subproblems, but the algorithm will not attempt to solve these subproblems. **Assign Groups Infeasible** will put each independent subproblem with no solution into specially named groups. Often poor corner choices and surface meshing schemes will be illuminated this way. If **Map** or **Pave** is specified, then only subproblems involving mapping or paving constraints will be considered. If a **Seed Curve** is specified, then only those subproblems containing that curve will be considered.

Interval Solver Version

For consistent results running old journal files, the user may want to use the old solver.

```
set interval version {<cubit version
number>|default}

list interval version
```

Setting version >= 15.6 uses IIA (2020 solver) and enables small-loop bounds. Using a prior version uses BBIA (1997 solver). Both solvers solve the problem

```
min f(x,g)
s.t. Ax=b
x in [lo,hi]
A, x, b integer
```

but the objective f is slightly different so the solution may be different. General problems of this form are known to be difficult because the solution must be integer valued, and neither solver is guaranteed to find the global minimum.

Here x are the intervals and internal variables; b are the bounds on those variables; A and b are the constraints, including hardsets; g are the goals, the user-desired number of intervals for each curve. The objective f models the desire to minimize the maximum relative change in mesh size. In mathematical language, for the new IIA solver f is the maximum lexicographic vector of the ratio achieved:goal if achieved>goal, else goal:achieved. The older solver uses an approximation to IIA's f.

IIA Solver >= Cubit 15.6

Cubit 15.6 in 2020 introduced "Incremental Interval Assignment (IIA)" based on integer linear algebra. It robustly and quickly finds an integer solutions to Ax=b, and tries to improve it by adding integer vectors from A's nullspace to satisfy the bounds and come close to the goals.

IIA solutions tend to be slightly coarser than BBIA, often one interval less, and closer to the user goals and the global optimum. IIA is faster than BBIA, dramatically faster (6000x) in some cases involving submapping and hard-sets. Please send the Cubit team any example which takes longer than 1 second.

BBIA Solver < Cubit 15.6

The solution method that Cubit introduced in 1996 was to use linear programming to find a floating point solution to Ax=b, then use branch-and-bound rounding to try to find a nearby integer solution. The first part is fast and robust, the second part slow and error prone. Linear programming is limited to linear objective functions, so the goal:achieved ratio is only approximated and extreme size differences can give unexpected compromises.

For BBIA, advanced users may wish to experiment with the following:

```
Set Match Intervals Rounding {on|off}

Set Match Intervals Fast {on|off}

Set Match Intervals Delta <interval_difference
= 0.>
```

If **set match intervals rounding** is set to **on**, the intervals will be rounded to the nearest integer. If the setting is **off**, the intervals will be rounded toward the user specified intervals.

If **set match intervals fast** is set to **off** a single curve will be fixed per branch and bound iteration. Note in rare cases this may produce better meshes, but will generally be slower. If set **on**, multiple curves will be fixed per iteration.

Set match intervals delta affects the convergence tolerance for the optimization. Here **delta** means the difference between a curve's goal and assigned intervals. A larger value makes matching intervals faster, but the quality of the solution may be worse. The default is 0.0. Hint: try 1.0.

Interval Firmness

Before describing the methods used to set and change intervals, it is important that the user understand the concept of interval firmness. An interval firmness value is assigned to a geometry curve along with an interval count or size; this firmness is one of the following values:

> **hard**: interval count is fixed and is not adjusted by interval size command or by interval matching
>
> **soft**: current interval count is a goal and may be adjusted up or down slightly by interval matching or changed by other interval size commands.
>
> **default**: default firmness setting, used for detecting whether intervals have been set explicitly by the user or by other tools

Interval firmness is used in several ways in CUBIT. Each curve is assigned an interval firmness along with an interval count or size. Commands and tools which change intervals also affect the interval firmness of the curves. Those same commands and tools which change intervals can only do so if the curves being changed have a lower-precedence interval firmness. The firmness settings are listed above in order of decreasing precedence. For example, some commands are only able to change curves whose interval firmness is soft or default ; curves with hard firmness are not changed by these commands.

More examples of interval setting commands and how they are affected by firmness are given in the following sections.

A curve's interval firmness can be set explicitly by the user, either for an individual curve or for all the curves contained in a higher order entity, using the command:

```
{geom_list} Interval {Default | Soft | Hard}
```

All curves are initialized with a firmness of default. Any command that changes intervals (including interval assignment) upgrades the firmness to at least soft.

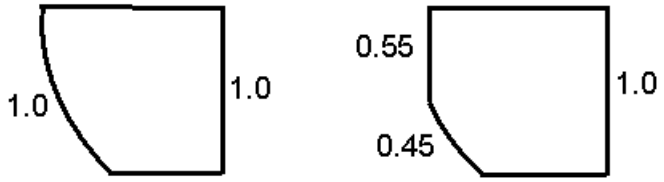
Precedence

If a size is specified multiple times for a single entity, the following precedence is used:

- The highest firmness command takes precedence.
  Hard commands include "curve <id> interval <val>", and "{geometry_list} interval hard" will fix the size at the current size.
- Within a given firmness, the last-issued command takes precedence.
  For example, if the user commands "surface 1 size 1" then "volume 1 size 2", and surface 1 is part of volume 1, then surface 1 will have a size of 2.

Relative Intervals

If the user needs fine control over mesh density, then for curvy or slanted sides of swept geometries, it is often useful to treat curves as if they had a different length when setting interval sizes. For example, the user may wish to specify that a slanting side curve and a straight side curve have the same "relative" length, despite their true length as shown in the following figure. These are not interval matching constraints; interval matching may change intervals so that the user-specified ratio does not hold exactly.



The relative lengths of curves are set with the following command:

```
{geom_list} Relative Length <size>
```

The following command is used to assign intervals proportional to these lengths:

```
{geom_list} Relative Interval <base_interval>
```

For a curve with relative length x, setting a relative interval of y produces xy intervals, rounded to the nearest integer.

Mesh Interval Preview

It is sometimes useful to view the nodal locations/intervals on curves graphically before meshing (which can take considerably more time). The command to do this is:

```
Preview Mesh
{Body|Volume|Surface|Curve|Vertex} <id_range>
[Hard] [color <color>]
```

To clear the display of the temporary nodes, simply issue a "**display**" command. The purpose of the **hard** option is that only curves that have an interval firmness of hard will be previewed.

Periodic Intervals

The number of intervals on a periodic surface, such as a cylinder, in the dimension that is not represented by a curve is usually set implicitly by the surface size.

However, periodic intervals and firmness can be specified explicitly by the following commands:

```
Surface <range> Periodic Interval <intervals>

Surface <range> Periodic Interval
{Default|Soft|Hard}
```

Meshing Schemes

Meshing schemes in CUBIT can be divided into four broad categories.

- Traditional Meshing Schemes
- Free Meshing Schemes
- Conversional Meshing Schemes
- Duplication Meshing Schemes

In addition, Cubit supports two parallel meshing applications, pCamal and Sculpt

- Parallel Meshing

If no scheme is selected, Cubit will attempt to assign a scheme using the automatic scheme selection methods.

- Automatic Scheme Selection

Traditional Meshing Schemes

Traditional meshing schemes are used to apply a mesh to an existing geometry using the methods described in Meshing the Geometry (i.e. setting a scheme, applying interval sizes, and meshing). Traditional meshing schemes are available for all geometry types.

- Bias, Dualbias
- Circle
- Curvature
- Equal
- Hole
- Mapping
- Pave
- Pentagon
- Pinpoint
- Polyhedron
- Sphere
- STransition
- Stretch
- Submap
- Sweep
- Tetmesh
- Tetprimitive
- Tridelaunay
- TriAdvance
- Trimap
- Trimesh
- Tripave
- Triprimitive

Free Meshing Schemes

Free meshing schemes will create a free-standing mesh without any prior existing geometry. The final mesh will have mesh-based geometry.

- Radialmesh

Conversional Meshing Schemes

Conversional meshing schemes are used to convert an existing mesh into a mesh of different element type or size. For example, the THex scheme will convert a tetrahedral mesh into a hexahedral mesh.

- HTet
- QTri
- THex
- TQuad

Duplication Meshing Scheme

The duplication meshing scheme is used to copy an existing mesh from

one geometry onto another similar geometry.

- Copy

General Meshing Information

Information on specific mesh schemes available in CUBIT is given in this section. The following sections have important meshing-related information as well, and should be read before applying any of the mesh schemes described below.

In most cases, meshing a geometric entity in CUBIT consists of three steps:

- Set the interval number or size for the entity (See Interval Assignment.)
- Set the scheme for the object, along with any scheme-specific information, using the scheme setting commands described below.
- Mesh the object, using the command:

```
Mesh {geom_list}
```

This command will match intervals on the given entity, then mesh any unmeshed lower order entities, then mesh the given entity.

After meshing is completed, the mesh quality is automatically checked (see Mesh Quality Assessment), then the mesh is drawn in the graphics window.

The following table classifies the meshing schemes with respect to their applicable geometry.

Curves	Surfaces	Volumes
Bias/Dualbias	Circle	Copy
Copy	Copy	HTet
Curvature		Mapping
	Hole	Polyhedron
Equal	Mapping	Sphere
Pinpoint		Submap
Stretch	Pave	Sweep
	Pentagon	TetMesh, TetINTRIA
	Polyhedron	Tetprimitive
	QTri	THex
	Submap	
	TriDelaunay	
	Triprimitive	
	TriMap	
	TriMesh	
	TriAdvance	
	TriPave	
	STransition	

Copying a Mesh

Applies to: Curves, Surfaces, Volumes

Summary: Copies the mesh from one entity to another

Syntax:

```
Curve <range> Scheme Copy source Curve <range>
[Source Percent [<percentage> | auto]] [Source
[combine|SEPARATE]] [Target
[combine|SEPARATE]] [Source Vertex <id_range>]
[Target Vertex <id_range>]]

Surface <id> Scheme Copy Source Surface <id>
Source Curve <id> Target Curve <id> Source
Vertex <id> Target Vertex <id> [Nosmoothing]
[mirror]

Volume <range> Scheme Copy [Source Volume]
<id> [[Source Surface <id> Target Surface
<id>] [Source Curve <id> Target Curve <id>]
[Source Vertex <id> Target Vertex <id>]]
[Nosmoothing]

Copy Mesh Curve <id> Onto Curve
<curve_id_range> [Source Node <starting node
id> <ending node id>] [Source Percent
[<percentage>|auto]] [Source Vertex
<id_range>] [Target Vertex <id_range>]

Copy Mesh Surface <surface_id> Onto Surface
<surface_id> Source Vertex <id> Target Vertex
<id> Source Curve <id> Target Curve <id>
[interior (pair vertex <id> <id>) ...]
[smooth] [mirror] [preview]

Copy Mesh Volume <volume_id> Onto Volume
<volume_id> [Source Vertex <vertex_id> Target
Vertex <vertex_id> [Source Curve <curve_id>
Target Curve <curve_id>] [Nosmoothing]
```

Related Commands:

```
Set Morph Smooth {on | off}
```

Discussion:

If the user desires to copy the mesh from a surface, volume, curve, or set of curves that has already been meshed, the copy mesh scheme can be used. Note that this scheme can be set before the source entity has been meshed; the source entity will be meshed automatically, if necessary, before the mesh is copied to the target entity. The user has the option of providing orientation data to specify how to orient the source mesh on the target entity. For example, when copying a curve mesh, the user can specify which vertex on the source (the source vertex) gets copied to which vertex on the target (the target vertex). If you need to reference mesh entities for the copy, use the **Copy Mesh** commands. If no orientation data is specified, or if the data is insufficient to completely determine the orientation on the target entity, the copy algorithm will attempt to determine the remaining orientation data automatically. If conflicting, or inappropriate, orientation data is given, the algorithm attempts to discard enough information to arrive at a proper mesh orientation.

Curve mesh copying has certain options that allow the copying of just a

section of the source curves' mesh. These options are accessed through the extra keyword options. The **percent** option allows the user to specify that a certain percentage of the source mesh be copied--in this context the auto keyword means that the percentage will be calculated based on the ratio of lengths of the source and target curves. The **combine** and **separate** keywords relate to how the command line options are interpreted. If the user wishes to specify a group of target curves that will each receive an identical copy of a source mesh, then the **target separate** option should be used (this is the default). If, however, the user wishes the source mesh to be spread out along the range of target curves, then the **target combine** option should be used. The source curves are treated in a similar fashion.

Surface mesh copying with multiple holes in the surface may require matching up interior pair vertices. This will be required if the algorithm cannot match them up spatially. Interior pair vertices can be specified with the option **Interior pair vertex <id> <id> ...**

Volume mesh copying depends on the surface copying scheme. Because of this, the target volume must not have any of its surfaces meshed already.

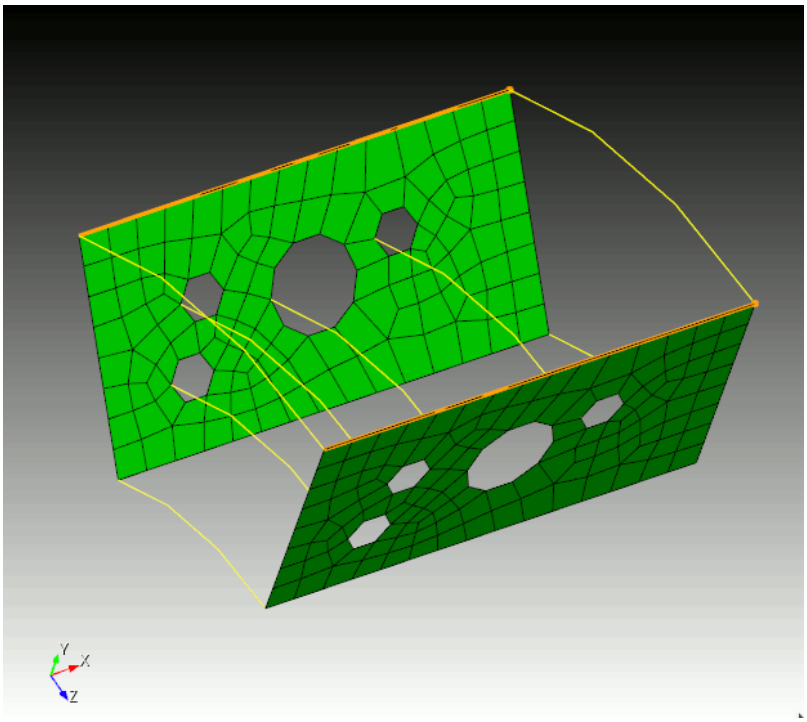
An exact copy of the mesh may not always happen. Dissimilar geometry or smoothing may cause inexact copies. If the geometry is similar, the smoothing option may be turned off to get an exact copy of the mesh, by either specifying **Nosmoothing** or by omitting **Smooth**. If the geometry is dissimilar, the user may set the morph smoothing flag on, which will activate a special smoother that will match up the meshes as closely as possible.

Example:

As an example, the following copy is done with the command

copy mesh surf 23 onto surf 14 source curve 1 source vertex 1 target curve 24 target vertex 20

The source and target vertices match up, and are highlighted, while the source and target curves match up and are highlighted. Matching the source and target curves/vertices help define the orientation.



HTet

Applies to: Volumes

Summary: Converts an existing hex mesh into a conforming tetrahedral mesh.

Syntax:

```
HTet Volume <range> {UNSTRUCTURED |
structured}
```

Discussion:

Unlike other meshing schemes in this section, The HTet command requires an existing hexahedral mesh on which to operate. Rather than setting a meshing scheme for use with the mesh command, the HTet command works after an initial hex mesh has been generated.

Two methods for decomposing a hex mesh into tetrahedra are available. Set the method to be used with the optional arguments unstructured and structured. The unstructured method is the default. Figure 1 shows the difference between the two methods:

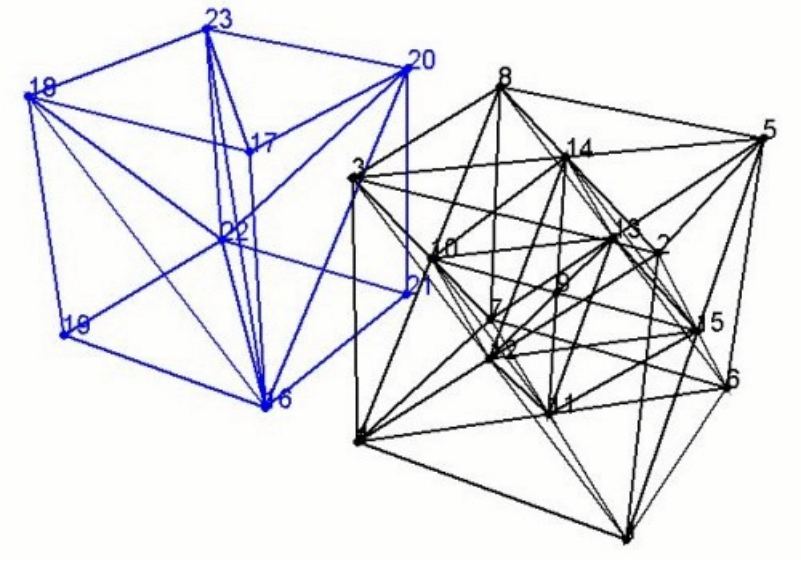


Figure 1. Left: Unstructured method creates 6 tets per hex. Right: Structured method creates 28 tets per hex

Unstructured

This method creates 6 tetrahedra for every hexahedra. No new nodes will be generated. The orientation of the 6 hexahedra will be based upon the element node numbering, as a result orientations may change if node numbering changes. This method is referred to as unstructured because the number of tetrahedra adjacent each node will be relatively arbitrary in the final mesh. Tetrahedral element quality is generally sufficient for most applications, however the user may want to verify quality before performing analysis.

Structured

With this approach, 28 tetrahedra are generated for every hexahedra in the mesh. This method adds a node to each face of the hex and one to the interior. Although this method generates significantly more elements, the orientation and quality of the resulting tetrahedra are more consistent.

Each previously existing interior node in the mesh will have the same number of adjacent tetrahedra.

QTri

Applies to: Surfaces

Summary: Meshes surfaces using a quadrilateral scheme, then converts the quadrilateral elements into triangles.

Syntax:

```
Surface <range> Scheme Qtri [Base Scheme
quad_scheme>]

QTri { Surface <range> | Face <range> }

Set QTri Split [2|4]

Set QTri Test {Angle|Diagonal}
```

Discussion:

QTri is used to mesh surfaces with triangular elements. The surface is, first, meshed with the quadrilateral scheme, and, then, the generated quads are split along a diagonal to produce triangles. The first command listed above sets the meshing scheme on a surface to QTri. The second form sets the scheme and generates the mesh in a single step.

In the first command, the user has the option of specifying the underlying quadrilateral meshing scheme using the base scheme <quad_scheme> option. If no base scheme is specified, CUBIT will automatically select a scheme. For non-periodic surfaces, the base scheme will be set to scheme pave. For periodic surfaces, the base scheme will be set to scheme map.

Generally, the second command, Qtri Surface <range>, is used on surfaces that have already been meshed with quadrilaterals. If, however, this command is used on a surface that has not been meshed, a base scheme will automatically be selected using CUBIT's auto-scheme capabilities. The user can over-ride this selection by specifying a quadrilateral meshing scheme prior to using the qtri command (using the Surface <range> Scheme <quad_scheme> command). QTri may also be performed on quadrilateral elements on a surface or a subset of quadrilateral elements on a surface. To split existing quadrilaterals, the QTri command can be given a list of faces.

In addition to the default 2 tris per quad, the set qtri split command may alter the QTri scheme so that it will split the quad into 4 triangles per quad. Where the 4 option is used, an additional mesh node is placed at the centroid of each quad.

There are two methods that may be used to calculate the best diagonal to use for splitting the quadrilateral elements: angle or diagonal. The angle measurement uses the largest angle, while the diagonal option uses the shortest diagonal. The largest angle measurement will be more accurate but takes more time.

Also, the QTri scheme is used in the TriMesh command as a backup to the TriAdvance triangle meshing scheme.

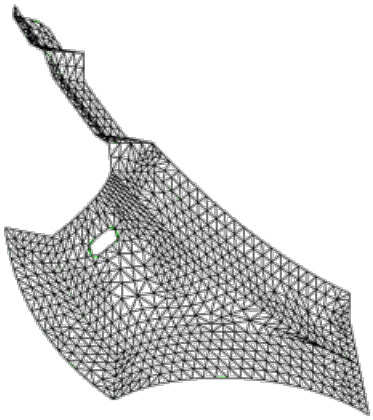


Figure 1. Surface meshed with scheme QTri

THex

Applies to: Volumes

Summary: Converts a tetrahedral mesh into a hexahedral mesh.

Syntax:

```
THex Volume <range>
```

Discussion:

The THex command splits each tetrahedral element in a volume into four hexahedral elements, as shown in Figure 1. This is done by splitting each edge and face at its midpoint, and then forming connections to the center of the tet.

When THexing merged volumes, all of the volumes must be THexed at the same time, in a single command. Otherwise, meshes on shared surfaces will be invalid. An example of the THex algorithm is shown in Figure 2.

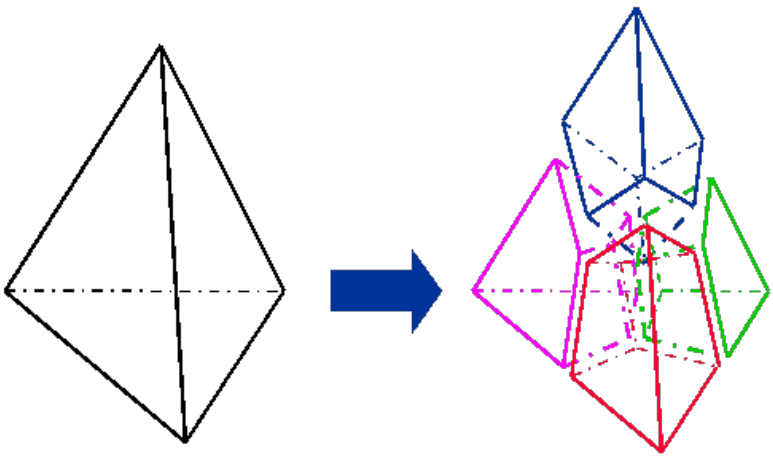
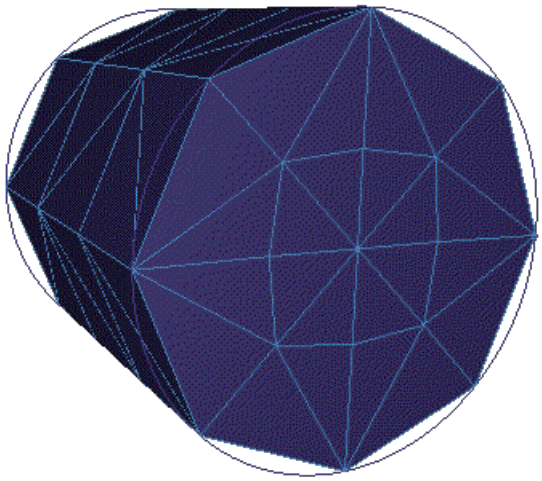


Figure 1. Conversion of a tetrahedron to four hexahedra, as performed by the THex algorithm.

.



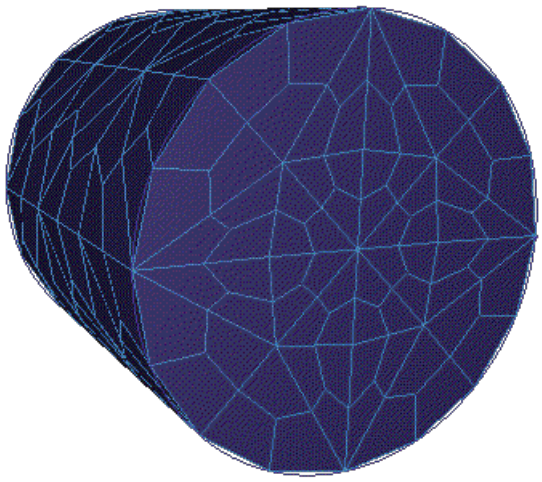


Figure 2. A cylinder before and after the THex algorithm is applied.

TQuad

Applies to: Surfaces

Summary: Converts a triangular surface mesh into a quadrilateral mesh.

Syntax:

```
TQuad Surface <range>
```

Discussion:

The TQuad command splits each triangular surface element in four quadrilateral elements, as shown in Figure 1. This is done by splitting each edge at its midpoint, and then forming connections to the center of the triangle. The result is the same as using the THex algorithm, but only applies to surfaces. In general it is better to use a mapped or paved mesh to generate quadrilateral surface meshes. However, the TQuad scheme may be useful for converting facet-based triangular meshes to quadrilateral meshes when remeshing is not possible.

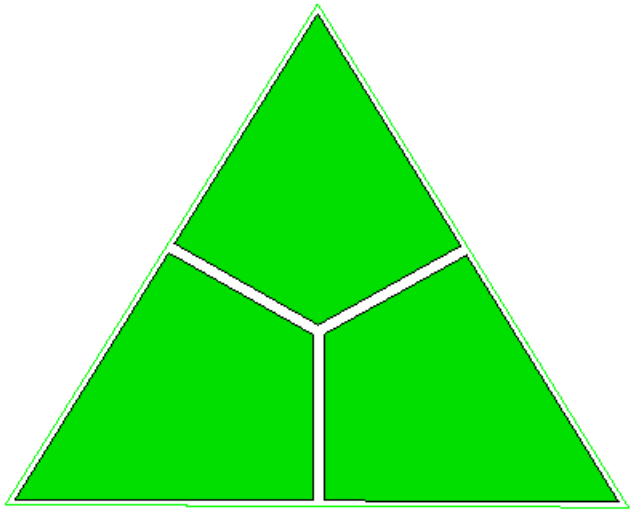


Figure 1. A triangle split into 3 quads using the TQuad scheme

Bias, Dualbias

Applies to: Curves

Summary: Meshes a curve with node spacing biased toward one or both curve ends.

Syntax:

```
Curve <range> Scheme Bias

Curve <range> Scheme Bias
{Factor|First_Delta|Fraction} <double> [Start
Vertex <range>] [preview]

Curve <range> Scheme Dualbias
{Factor|First_Delta|Fraction} <double>
[preview]

Curve <range> Scheme Bias Fine Size <double>
{Coarse Size <double> | Factor <double>}
[Start Vertex <range>] [preview]

Curve <range> Scheme Dualbias Fine Size
<double>
{Coarse Size <double> | Factor <double>}
[preview]

Curve <range> Scheme Multi_bias Start <size>
[Fraction <value> <size>]... End <size>
[Start Vertex <id>][Respect_intervals]
[preview]
```

Related Commands:

```
Curve <range> Reverse Bias

Set Maximum Interval <int>

See also Surface Sizing Function Type Bias

See also Curve Scheme Stretch
```

The main differences between scheme bias and stretch are the following: scheme stretch does not use strict geometric series for node placement. If you specify scheme bias or dualbias using the "fine size" form, the interval count will be hard-set to a value that fills in the curve.

Auto Bias

When using the command '**curve <range> scheme bias**' with no additional parameters, an auto setting will be enabled by default for tet and tri meshing. This scheme honors sizes at a curve's vertices and that vertex size will be used to create a biased edge mesh. For example, two volumes with different sizes set on the volumes are merged. The size at the vertices (averaged from sizes on the parent entities) will be used to create the biased edge mesh.

A user can set a size on a vertex with the following command:

```
Vertex <id> Size <size>
```

More Discussion:

The Bias and DualBias schemes space the curve mesh unequally, placing more nodes towards (or away from) the ends of the curve according to a geometric progression. The ratio of successive edges is the "factor," which may be greater than or less than one. For bias, the

series starts at the first vertex of the curve, or the "start vertex" if specified. For dualbias, the series starts at both ends of the curve and meets in the middle.

The command behaves differently depending on which set of parameters are specified. There are three basic variables: the interval count, the bias factor, or the first edge size. The curve length is a given, fixed quantity. The user can specify any two of these variables, and the third will be automatically determined.

If the "{Factor|First_Delta|Fraction}" form is specified, then the interval count is taken as a given. The interval count is whatever was specified previously by an interval count or size command (see Interval Assignment). If "Factor" is specified, then the first edge size will be automatically chosen so that the geometric progression of edges "fit" onto the curve. If "first_delta" is specified, then the first edge length is exactly that absolute value, and the "factor" is automatically chosen. If "fraction" is specified, then the first edge length is the curve length times that fraction, and again the "factor" is automatically chosen.

If the "fine size" is specified, then the first edge length is exactly that absolute value. If the "factor" is specified, then the interval count is automatically chosen. If an approximate coarse size is specified, then this also determines the factor, and again the interval count is automatically chosen. If a surface sizing function type bias is used, then the curves of the surface are sized using similar formulas.

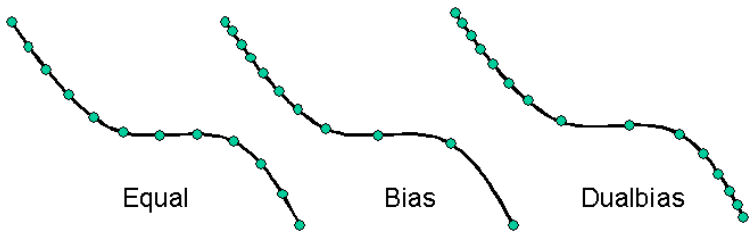
If no start or end vertex is specified, the curve's start vertex is used as the starting point of the bias. (A curve's start vertex can be identified by listing the curve from the "CUBIT>" prompt.)

If a curve needs to have its nodes distributed towards the opposite end, it can be easily edited using the reverse bias command. Reversing the curve bias using this command is equivalent to setting a bias factor equal to the inverse of the original bias factor. Reversing the bias can be performed on both meshed and unmeshed curves.

The maximum interval setting allows the user to set a maximum number of intervals on any bias curve. This value is doubled for a curve with a dualbias scheme. It can be easy to accidentally specify a very large number of intervals and this setting allows the user to place an upper limit the number of intervals.

The preview option will allow the user to preview mesh size and distribution on the curve before meshing.

The following figure shows the result of meshing edges with equal, bias and dualbias schemes.



Multi Bias

The multi-bias scheme allows several biases to be created on a single curve by specifying desired sizes at multiple locations along the curve (see Figure 1 below). The "start" and "end" sizes must be specified, and any number of fraction-size pairs may be specified, where the fraction value is between 0 - 1. If "Start Vertex" is given, the specified vertex is considered to have the fraction value of 0, with the opposite vertex having the fraction value of 1. If "respect_intervals" is not specified, the scheme will choose an appropriate number of intervals for the curve based on the given sizes.

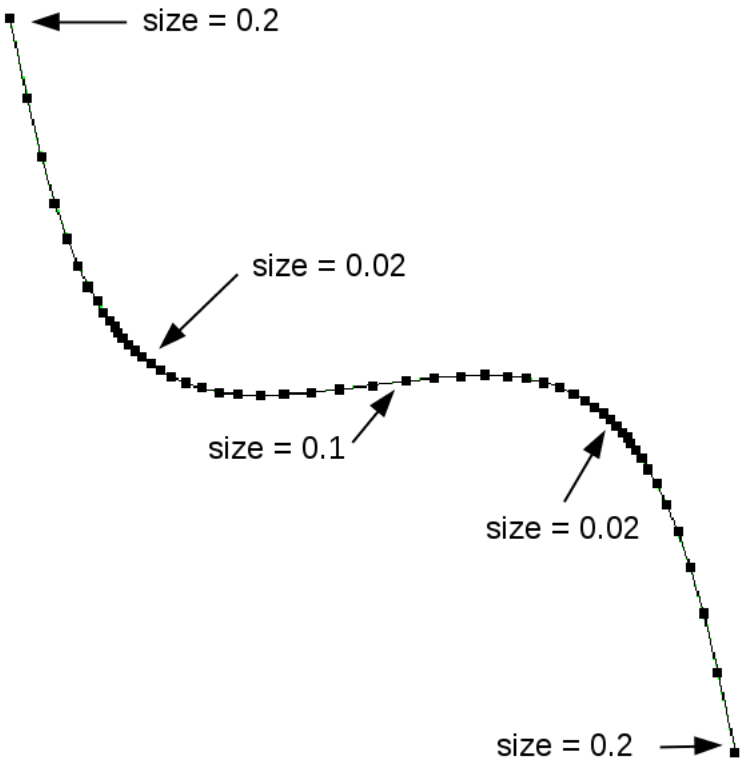


Figure 1. Curve with scheme multi_bias.

If the "respect_intervals" option is given, the multi-bias scheme will try to get as close to the desired sizes as possible, but will always respect the number of intervals set on the curve and adjust sizing as necessary (see Figure 2).

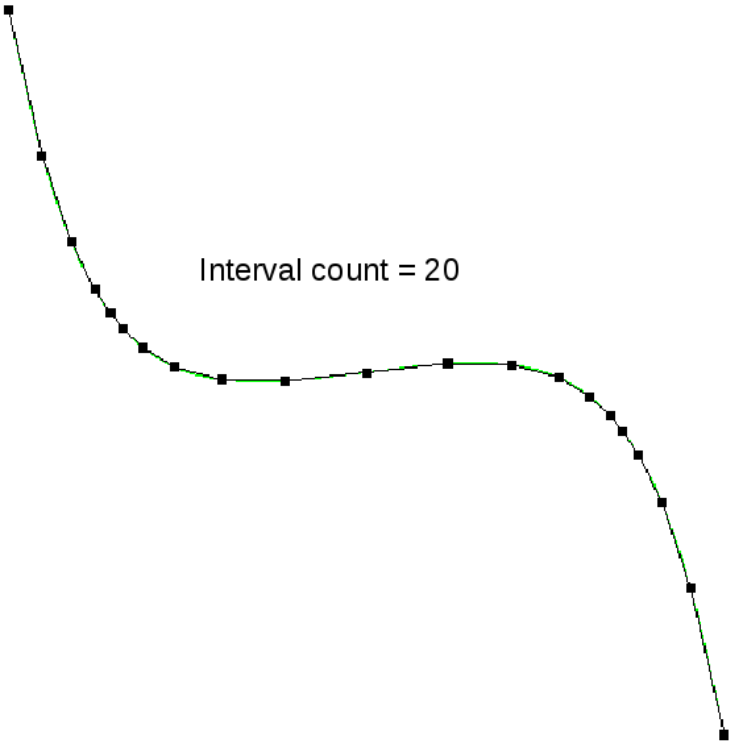


Figure 2. Same curve as in Figure 1, but with the respect_intervals option. Note the areas of relatively dense mesh correspond to the dense mesh in the original.

Circle

Applies to: Surfaces

Summary: Produces a circle-primitive mesh for a surface

Syntax:

```
Surface <range> Scheme [Sector] Circle
[Interval <int>] [fraction <double>]
```

Discussion:

The Circle scheme is used in regions that should be meshed as a circle. A "circle" consists of a single loop of bounding curves containing an even number of intervals. Thus, the circle scheme can be applied to circles, ellipses, ovals, and regions with "corners" (e.g. polygons). The bounding curves should enclose a convex region. Non-planar bounding loops can also be meshed using the circle primitive provided the surface curvature is not too great. The mesh resembles that obtained via polar coordinates except that the cells at the "center" are quadrilaterals, not triangles. See Figure 1 for an example of a circle mesh. Radial grading of the mesh may be achieved via the optional [intervals] input parameter. The Fraction option has the range 0 < fraction < 1 and defaults to 0.5. Fraction determines the size of the inner portion of the circle mesh relative to the total radius of the circle. The sector option was added to revert to legacy behavior which is not recommended.

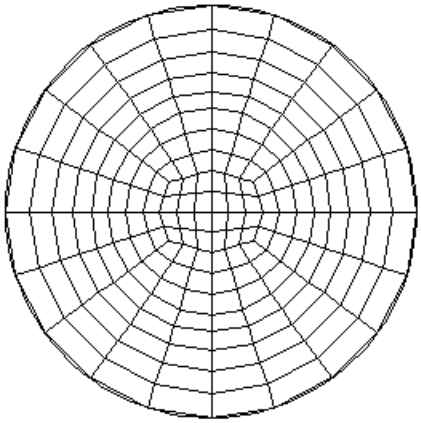


Figure 1. Circle Primitive Mesh

Curvature

Applies to: Curves

Summary: Meshes curves by adapting the interval size to the local curvature.

Syntax:

```
Curve <range> Scheme Curvature <double>
```

Discussion:

The value of <double> controls the degree of adaptation. If zero, the resulting mesh will have nearly equal intervals. If greater than zero, the smallest intervals will correspond to the locations of largest curvature. If less than zero, the largest intervals will correspond to the locations of largest curvature. The default value of <double> is zero. Straight lines and circular arcs will produce meshes with near-equal intervals. The method for generating this mesh is iterative and may sometimes not converge. If the method does not converge, either the <double> is too large (over-adaptation) or the number of intervals is too small. Currently, the scheme does not work on periodic curves.

Equal

Applies to: Curves

Summary: Meshes a curve with equally-spaced nodes

Syntax:

```
Curve <range> Scheme Equal
```

Discussion:

See Interval Assignment for a description of how to set the number of nodes or the node spacing on a curve.

Hole

Applies to: Annular Surfaces

Summary: Useful on annular surfaces to produce a "polar coordinate" type mesh (with the singularity removed).

Syntax:

```
Surface <surface_id_range> Scheme Hole
[Rad_intervals <int>] [Bias <double>] [Pair
Node <id> With Node <id>]
```

Discussion:

A polar coordinate-like mesh with the singularity removed is produced with this scheme. The azimuthal coordinate lines will be of constant radius (unlike scheme map) The number of intervals in the azimuthal direction is controlled by setting the number of intervals on the inner and outer bounding loops of the surface (the number of intervals must be the same on each loop). The number of intervals in the radial direction is controlled by the user input, rad_intervals (default is one).

A bias may be put on the mesh in the radial direction via the input parameter bias. The default bias of 0 gives a uniform grading, a bias less than zero gives smaller radial intervals near the inner loop, and a bias greater than zero gives smaller radial intervals near the outer loop.

The correspondence between mesh nodes on the inner and outer boundaries is controlled with the pair node "<loop node-id> with node <loop node-id>" construct. One id on the inner loop and one id on the outer loop should be given to connect the two nodes by a radial mesh line. Not choosing this option may result in sub-optimal node pairings with possible negative Jacobians. To use this option, mesh the inner and outer curve loops and then determine the mesh node ids.

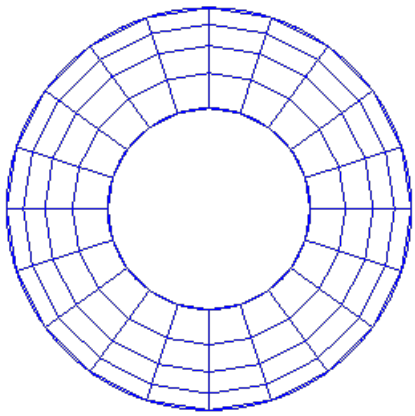


Figure 1. Example of Hole Scheme

Mapping

Applies to: Surfaces, Volumes

Summary: Meshes a surface/volume with a structured mesh of quadrilaterals/hexahedra.

Syntax:

```
Volume <range> Scheme Map
```

```
Surface <range> Scheme Map [Direction
{Options}]
```

Discussion:

A structured mesh is defined as one where each interior node on a surface/volume is connected to 4/6 other nodes. Mappable surfaces contain four logical sides and four logical corners of the map; each side can be composed of one or several geometric curves. Similarly, mappable volumes have six logical sides and eight logical corners; each side can consist of one or several geometric surfaces. For example, in Figure 1 below, the logical corners selected by the algorithm are indicated by arrows. Between these vertices the logical sides are defined; these sides are described in Table 1.

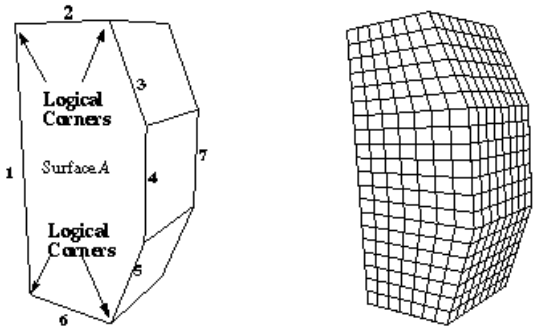


Figure 1. Scheme Map Logical Properties

Table 1. Listing of Logical Sides

Logical Side	Curve Groups
Side 1	Curve 1
Side 2	Curve 2
Side 3	Curve 3, Curve 4, Curve 5
Side 4	Curve 6

Interval divisions on opposite sides of the logical rectangle are matched to produce the mesh shown in the right portion of Figure 1. (i.e. The number of intervals on logical side 1 is equated to the number of intervals on logical side 3). The process is similar for volume mapping except that a logical hexahedron is formed from eight vertices. Note that the corners for both surface and volume mapping can be placed on curves rather than vertices; this allows mapping surfaces and volumes with less than four and eight vertices, respectively. For example, the mapped quarter cylinder shown in Figure 2 has only five surfaces.

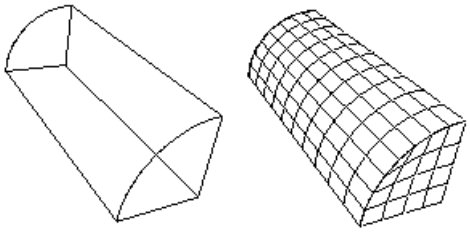


Figure 2. Volume Mapping of a 5-surfaced volume

The mapper works on a bicubic interpolation of the points on the boundary to represent the surface. There may be times that those points may not be on the surface exactly if the surface is not suitable for bicubic interpolation. The Mapping Constraint flag tells the mapper to relax the nodes to the geometry or not.

```
Set Mapping Constraint {ON|off}
```

When on, the mapping constraint relaxes the node to the nearest point on the geometry. In some situations, the nearest point might be incorrect for the intent of the mesh. To help the mesher find the correct location, a projection direction may optionally be specified for surfaces.

```
Surface <range> Scheme Map [Direction
{Options}]
```

If a projection direction is specified, the nodes are moved to the geometry in a straight line along the given direction. The direction can be specified using any of the direction options.

Pave

Applies to: Surfaces

Summary: Automatically meshes a surface with an unstructured quadrilateral mesh.

Syntax:

```
Surface <range> Scheme Pave Related Commands:

[Set] Paver Diagonal Scale <factor (Default =
0.9)> [set] Paver Grid Cell <factor (Default =
2.5)>[set] Paver LinearSizing {Off | ON}
Surface <range> Sizing Function Type ...

[Set] Paver Smooth Method {DEFAULT | Smooth
Scheme | Old}

[Set] Paver Cleanup {ON|Off|Extend}
```

Discussion:

Paving (Blacker, 91; White, 97) allows the meshing of an arbitrary three-dimensional surface with quadrilateral elements. The paver supports interior holes, arbitrary boundaries, hard lines, and zero-width cracks. It also allows for easy transitions between dissimilar sizes of elements and element size variations based on sizing functions. Figure 1 shows the same surface meshed with mapping (left) and paving (right) schemes using the same discretization of the boundary curves.

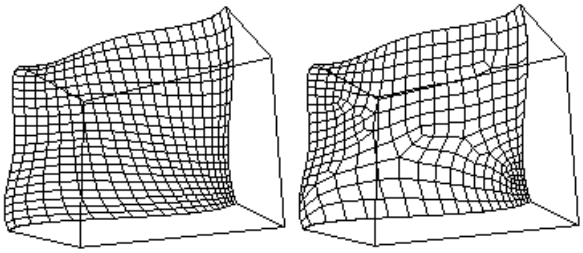


Figure 1. Map (left) and Paved (right) Surface Meshes

Element Shape Improvement

When meshing a surface geometry with paving, clean-up and smoothing techniques are automatically applied to the paved mesh. These methods improve the regularity and quality of the surface mesh. By default the paver uses its own smoothing methods that are not directly-callable from CUBIT. Using one of CUBIT's callable smoothing methods in place of the default method will sometimes improve mesh quality, depending on the surface geometry and specific mesh characteristics. If the paver produces poor element quality, switching the smoothing scheme may help. This is done by the command:

```
[set] Paver Smooth Method {DEFAULT | Smooth
Scheme | Old}
```

When the "Smooth Scheme" is selected, the smoothing scheme specified for the surface will be used in place of the paver's smoother. See "Mesh Smoothing" for more information about the available smoothing schemes in CUBIT.

Controlling Flattening of Elements

The smoothers flatten elements, such as inserted wedges, that have two edges on the active mesh front. In meshes where this "corner" is a real corner, flattening the element may give an unacceptable mesh. The following command controls how much the diagonal of such an element is able to shrink.

```
[set] Paver Diagonal Scale <factor (Default =
0.9)>
```

The range of for the scale factor is 0.5 to 1.0. A scale factor of 1.0 will force the element to be a parallelogram as long as it is on the mesh front. A value of 0.5 will allow the diagonal to be half its calculated length. The element may became triangular in shape with the two sides on the mesh front being collinear.

Controlling the Grid Search for Intersection Checking

The paver divides the bounding box of a surface into a number of cells based on the average length of an element. It uses these cells to speed intersection checking of new element edges with the existing mesh. If both very long and very short edges fall in the same area, it is possible that a long edge which spans the search region is excluded from the intersection check when it does intersect the new element. The following command allows the user to adjust the size of the grid cells.

```
[set] Paver Grid Cell <factor (Default = 2.5)>
```

The grid cell factor is a multiplier applied to the average element size, which then becomes the grid cell size. The surface's bounding box is divided by this cell size to determine the number of cells in each direction. A larger cell size means each cell contains more nodes and edges. A smaller cell size means each cell has fewer nodes and edges. A larger cell size forces the intersection algorithm to check more potential intersections, which results in long paver times. A smaller cell size gives the intersection algorithm few edges to check (faster execution) but may result in missed intersections where the ratio of long to short element edges is great. Increase this value if the paver is missing intersections of elements.

Controlling the Paver Sizing Function

The paving algorithm will automatically select a "linear" sizing function if the ratio the largest element to the smallest is greater than 6.0 and no other sizing function is specified for the surface. This is usually desirable. When it is not, the user can change this behavior with the command:

```
[set] Paver LinearSizing {Off | ON}
```

Setting paver linear sizing to "off" will keep the default behavior. The size of the element will be based on the side(s) of the element on the mesh front. For a discussion of sizing functions, including how to automatically set up size transitions, see Adaptive Meshing.

Controlling Paver Cleanup

The paver uses a mesh clean-up process to improve mesh quality after the initial paving operation. Clean-up applies local connectivity corrections to increase the number of interior mesh nodes that are connected to four quadrilaterals. Sometimes it fails to improve the mesh. The following command allows the user to control some aspects of the clean-up process.

```
[Set] Paver Cleanup {ON|Off|Extend}
```

The default option is to clean-up the mesh. The off option will turn clean-up off and may give an invalid mesh. The extend option enables a non-local topology replacement algorithm. The command without any option will list the current setting.

The extend option attempts to group several defective nodes in a region that may be replaced with a template that has fewer defects. The images below show a mesh before and after using this option.

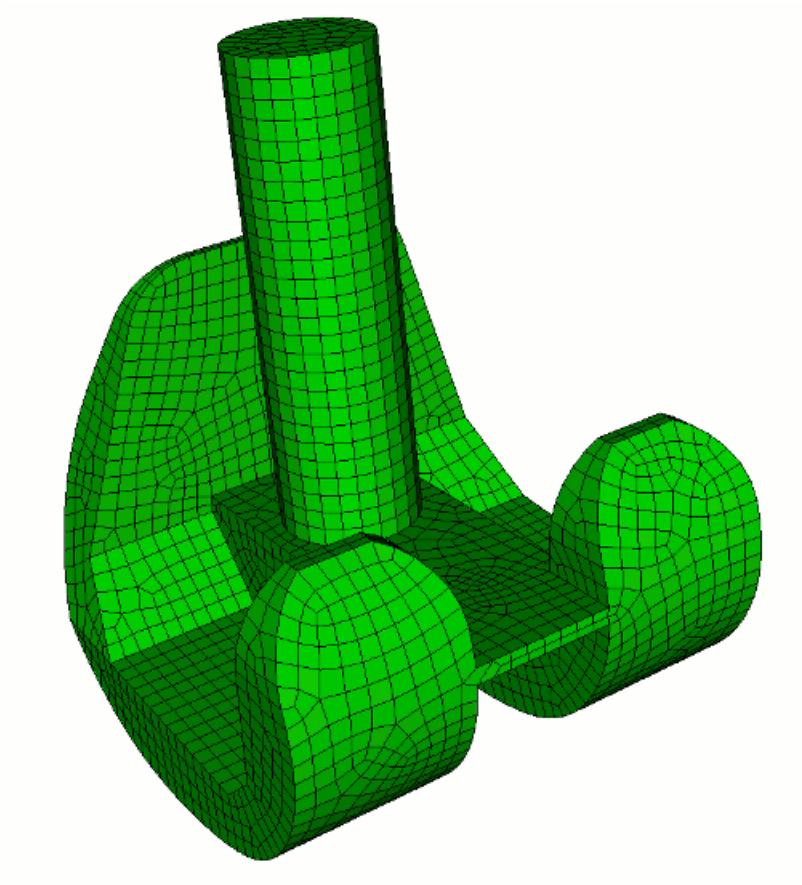


Figure 2. Paved mesh before using cleanup extend

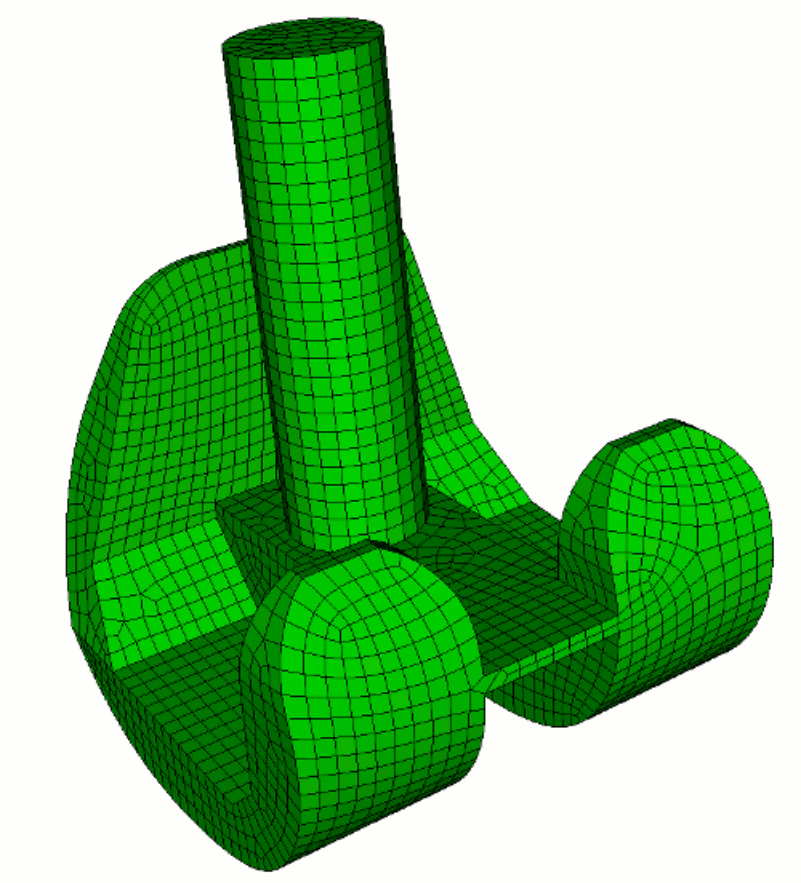


Figure 3. Paved mesh after using cleanup extend

Pentagon

Applies to: Surfaces

Summary: Produces a pentagon-primitive mesh for a surface

Syntax:

```
Surface <range> Scheme Pentagon
```

Discussion:

The pentagon scheme is a meshing primitive for 5-sided regions. It is similar to the triprimitive and polyhedron schemes, but is hard-coded for 5 sided surfaces.

The pentagon scheme indicates the region should be meshed as a pentagon. The scheme works best if the shape has 5 well-defined corners; however shapes with more corners can be meshed. The algorithm requires that there be at least 10 intervals (2 per side) specified on the curves representing the perimeter of the surface. In addition, the sum of the intervals on any three connected sides must be at least two greater than the sum of the intervals on the remaining two sides. Figure 1 shows two examples of pentagon meshes.

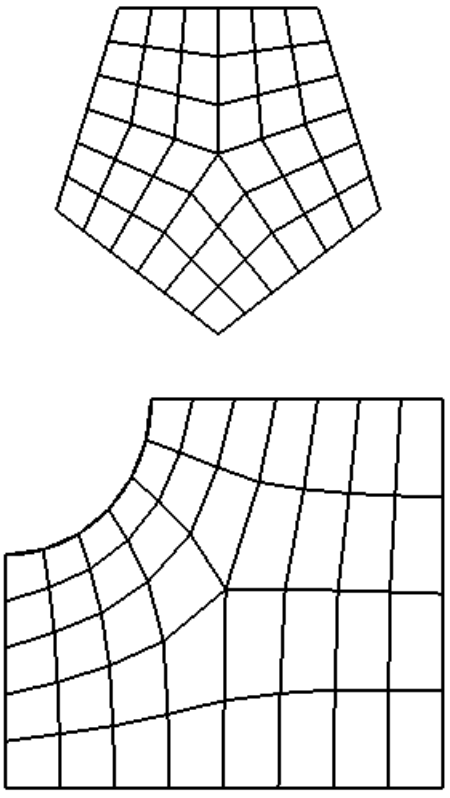


Figure 1. Examples of Pentagon Scheme Meshes

Pinpoint

Applies to: Curves

Summary:Meshes a curve with node spacing specified by the user.

Syntax:

```
Curve <range> Scheme Pinpoint Location <list
of doubles>
```

Discussion:

The **Pinpoint** scheme allow the user to specify exactly where on a curve to place nodes. The list of doubles are absolute positions, measured from the start vertex. The user can enter as many as needed, and they do not need to be in numerical order. Below is an example of a curve that has been meshed using the following scheme:

```
curve 2 scheme pinpoint location 1 4 5 6 6.2
6.4 6.6 9:
```

Polyhedron

Applies to: Surfaces and Volumes.

Summary: Produces an arbitrary-sided block primitive mesh for a surface or volume.

Syntax:

```
Volume <range> Scheme Polyhedron

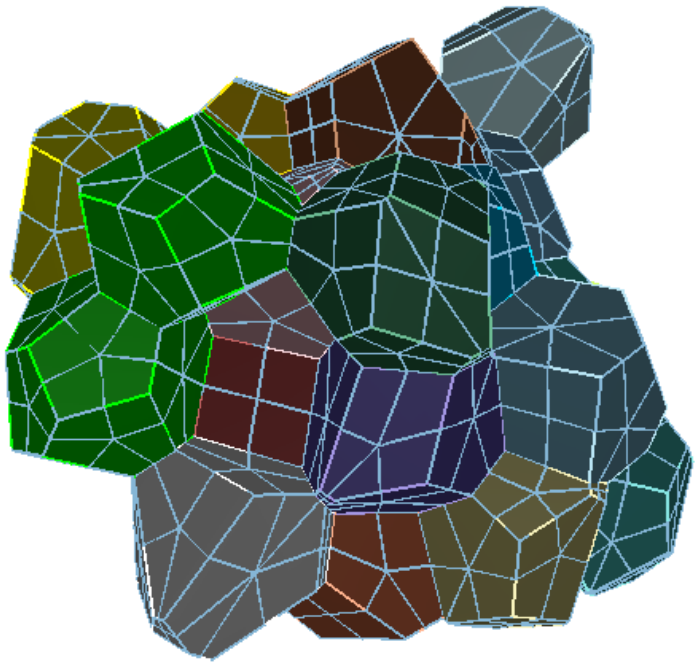
Surface <range> Scheme Polyhedron
```

Discussion:

The polyhedron scheme is a meshing primitive for 2d and 3d n-sided regions. This is similar to the triprimitive , tetprimitive, and pentagon schemes, except rather than 3, 4, or 5 sides, it allows an arbitrary number of sides. The scheme works best on convex regions. Surfaces must have only one loop, and each vertex must be connected to exactly two curves on the surface (e.g., no hardlines). Volumes must have only one shell, each vertex must be connected to exactly three surfaces on the volume, and each surface should be meshed with scheme polyhedron. There are some interval assignment requirements as well, which should be automatically handled by CUBIT.

If the polyhedron scheme is specified for the volume, then the surfaces of the volume are automatically assigned scheme polyhedron as well, unless they were hard-set by the user. Schemes should be specified on all volumes of an assembly prior to meshing any of them. Scheme polyhedron attaches extra data to volumes; if Cubit is behaving strangely, the user may need to explicitly remove that data with a **reset volume all**, or similar command.

Scheme polyhedron was designed for assemblies of material grains, where each volume is roughly a Voronoi region, and the assembly is a periodic space-filling model (tile). Figure 1 shows two examples of polyhedron meshes.



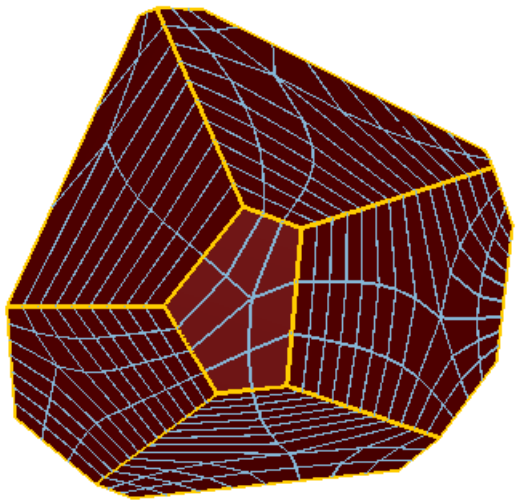


Figure 1. Examples of Polyhedron Scheme Meshes

Sphere

Applies to: Volumes topologically equivalent to a sphere and having one surface.

Summary: Generates a radially-graded hex mesh on a spherical volume.

Syntax:

```
Volume <range> Scheme Sphere [Graded_interval
<int>] [Az_interval <int>] [Bias <val>]
[Fraction <val>] [Max_smooth_iterations
<int=2>]
```

Discussion:

This scheme generates a radially-graded mesh on a spherical volume having a single bounding surface. The mesh is a straightforward generalization of the circle scheme for surfaces. The mesh consists of an inner region and an outer region. The inner region is a mapped mesh of a cube and the outer region contains fronts that trasition from the cube surface to the sphere surface. The following describes the parameters that control the sphere mesh.

Graded_interval:

The number of intervals in the outer region from the inner mapped mesh to the surface of the sphere is controlled by the **graded_interval** input parameter. Azimuthal mesh lines in the outer portion of the sphere will have approximately constant radius. If **graded_interval** is not specified, a default number of intervals will be computed based on the interval size value assigned to the sphere volume.

Az_interval:

The number of azimuthal intervals around the equator is controlled by the **az_interval** input parameter. To maintain symmetry, the **az_interval** will be rounded to the nearest multiple of 8.

If **az_interval** is not specified, a default number of intervals will be computed either based on the the the interval value or on the mesh size value assigned to the volume. If the interval value is set (**volume 1 interval 40**, for example), the interval value will be used to define the number of azimuthal intervals. Otherwise, the mesh size will be used as the approximate size for elements on the inner mapped mesh.

Bias:

The **bias** parameter controls the amount of radial grading in the outer region of the mesh from the inner mapped mesh to the sphere surface. A **bias = 1** will results in equal size intervals, while a **bias < 1** will generate smaller intervals towards the sphere interior and a **bias > 1** will generate smaller elements towards the sphere surface. If the **bias** parameter is not specified, a default **bias** will be computed so that element size gradually increases from the inner mapped mesh to the sphere surface. The default **bias** value will also be based on the interval size assigned to the sphere volume as it attempts to maintain approximately isotropic elements throughout the sphere.

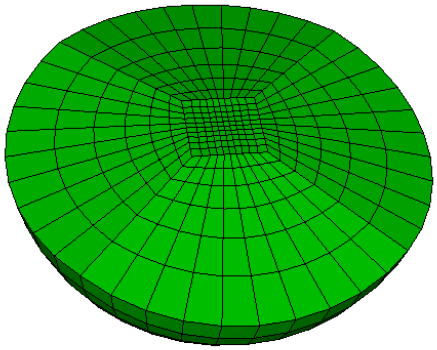
Fraction:

The **fraction** parameter (between 0 and 1) determines what fraction of the sphere is occupied by the inner mapped mesh. The **fraction** is defined as ratio of the diagonal of the cube containing the mapped mesh to sphere's diameter. The default value for **fraction** is 0.5. Interval sizes in the inner mapped mesh are normally constrained by the **az_intervals**.

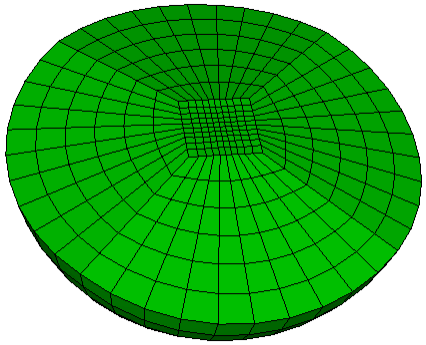
If **az_intervals** are not specified, element sizes in this region will be based upon the interval size assigned to the sphere volume.

Max_smooth_iterations:

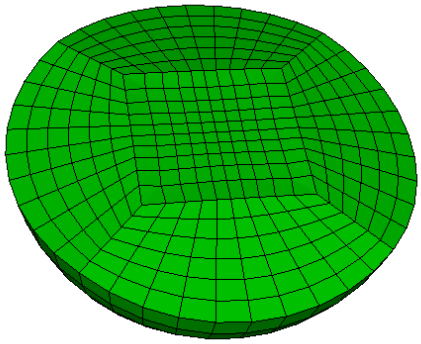
The **Max_smooth_iterations** parameter determines the number of smoothing iterations following initial definition of the sphere mesh. By default, the number of smoothing interations is set to 0, which will result in a symmetric mesh. Note that smoothing can improve the quality of the mesh, however, it may disturb the bias and fraction. When bias and fraction are critical then smoothing iterations should be set to 0.



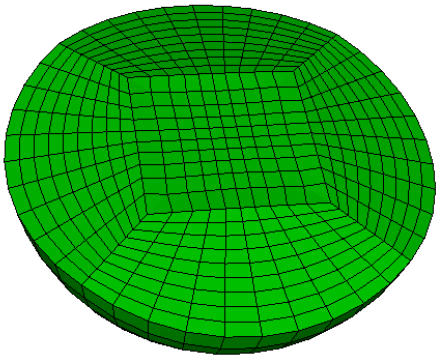
SPHERE MESH: fraction 0.3 graded_interval 6 az_interval 40 bias 0.8 max_smooth_iterations 0



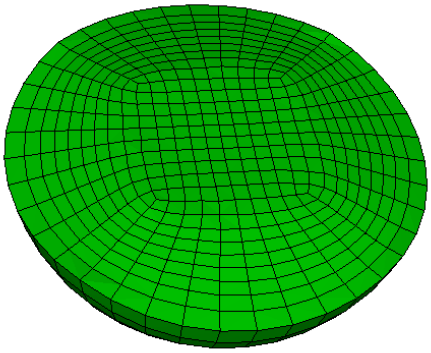
BIAS (uniform): fraction 0.3 graded_interval 6 az_interval 40 bias 1.0 max_smooth_iterations 0



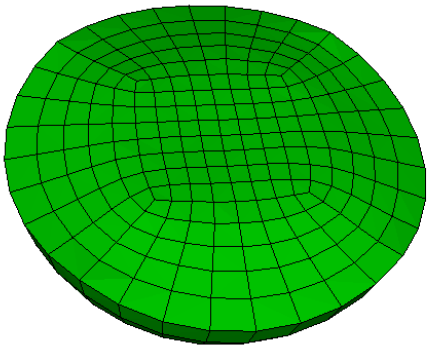
FRACTION: fraction 0.7 graded_interval 6 az_interval 40 bias 1.0 max_smooth_iterations 0



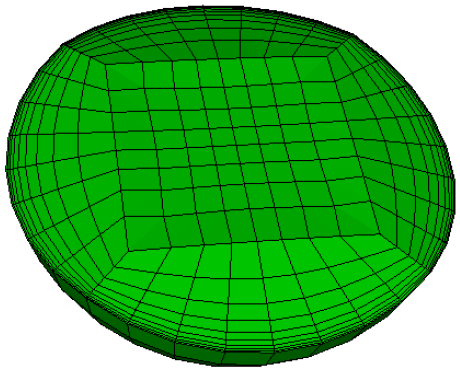
INTERVAL: fraction 0.7 graded_interval 9 az_interval 40 bias 1.0 max_smooth_iterations 0



SMOOTHING: fraction 0.7 graded_interval 9 az_interval 40 bias 1.0 max_smooth_iterations 2



AZIMUTHAL (mesh coarseness): fraction 0.7 graded_interval 5 az_interval 32 bias 1.0 max_smooth_iterations 2



BIAS (graded): fraction 0.9 graded_interval 9 az_interval 32 bias 1.5 max_smooth_iterations 0

STransition

Applies to: Surfaces

Summary:

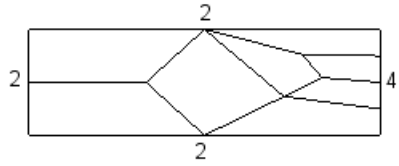
Produces a simple transitional mapped mesh.

Syntax:

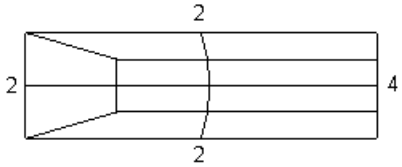
```
Surface <surface_id_range> Scheme STransition
[Triangle] [Coarse]
```

Discussion:

The STransition scheme transitions a mesh from one element density to another across a surface. This scheme is particularly helpful when the Paving scheme produces a poor mesh. The following two figures show a specific case where the STransition scheme may offer an improvement.

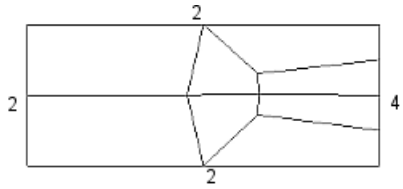


Pave scheme



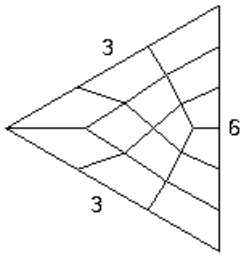
STransition scheme

The **coarse** option forces the mesh to transition to a coarser mesh in the first layer.



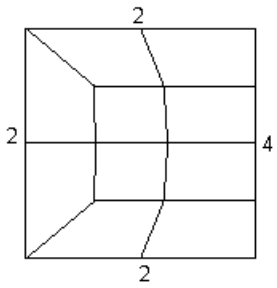
STransition scheme with coarse option

For triangular surfaces, the STransition scheme with the **triangle** option will produce similar results when compared to the Triprimitive scheme. However, STransition is capable of handling more varied interval settings. The following triangle fails when using the Triprimitive scheme but succeeds with the STransition scheme.

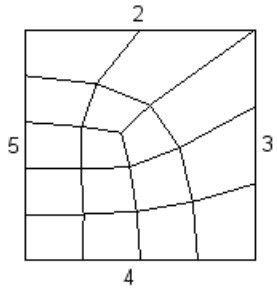


STransition scheme on a triangular surface with intervals set to 3, 3, and 6.

The figures below show the STransition meshing scheme response to different shapes and interval settings.

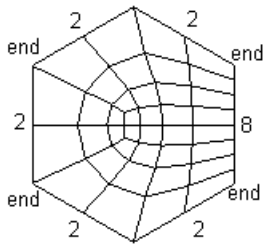


STransition scheme on a rectangular surface with three intervals set to 2 and one set to 4.



STransition scheme on a rectangular surface with intervals set to 2, 3, 4, and 5.

The user also has the option of specifying END or SIDE surface vertex types.



STransition scheme on a hexagon surface with five intervals set to 2, one interval set to 8, and user specified endpoints.

Note, that the Centroid Area Pull smoothing algorithm sometimes gives better results than the default Winslow smoothing algorithm for STransition meshes.

Stretch

Applies to: Curves

Summary: Permits user to specify the exact size of the first and/or last edges on a curve.

Syntax:

```
Curve <range> Scheme Stretch [First_size
<double>]  [Last_size <double>] [Start Vertex
<id>]

Curve <range> Scheme Stretch [Stretch_factor
<double>] [Start Vertex <id>]
```

Related Commands:

> Scheme Bias and Dualbias.

Discussion:

This scheme allows the user to specify the exact length of the first and/or last edge on a curve mesh. Intermediate edge lengths will vary smoothly between these input values. Reasonable values for these parameters should be used (for example, the sizes must be less than the total length of the curve). If last_size is input, first_size must be input also. If stretch_factor is input, neither first_size nor last_size can be input. This scheme does not currently work on periodic curves.

Submap

Applies to: Surfaces, Volumes

Summary: Produces a structured mesh for surfaces/volumes with more than 4/6 logical sides

Syntax:

```
{Surface|Volume} <range> Scheme Submap
```

Related Commands:

```
{Surface|Volume} <range> Submap Smooth <on|off>
```

Discussion:

Submapping (Whiteley, 96) is a meshing tool based on the surface mapping capability discussed previously, and is suited for mesh generation on surfaces which can be decomposed into mappable subsurfaces. This algorithm uses a decomposition method to break the surface into simple mappable regions. Submapping is not limited by the number of logical sides in the geometry or by the number of edges. The submap tool, however is best suited for surfaces and volumes that are fairly blocky or that contain interior angles that are close to multiples of 90 degrees.

An example of a volume and its surfaces meshed with submapping is shown in Figure 1.

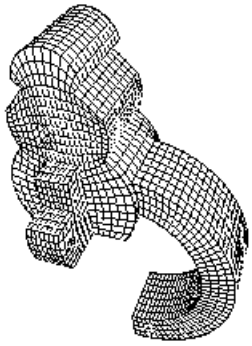


Figure 1. Quadrilateral and Hexahedral meshes generated by submapping

Like the mapping scheme, submapping uses vertex types to determine where to put the corners of the mapped mesh (See Surface Vertex Types). For surface submapping, curves on the surface are traversed and grouped into " logical sides " by a classification of the curves position in a local "i-j" coordinate system.

Volume submapping uses the logical sides for the bounding surfaces and the vertex types to construct a logical "i-j-k" coordinate system, which is used to construct the logical sides of the volume. For surface and volume submapping, the sides are used to formulate the interval constraints for the surface or volume.

Figure 2 shows an example of this logical classification technique, where the edges on the front surface have been classified in the i-j coordinate system; the figure also shows the submapped mesh for that volume.

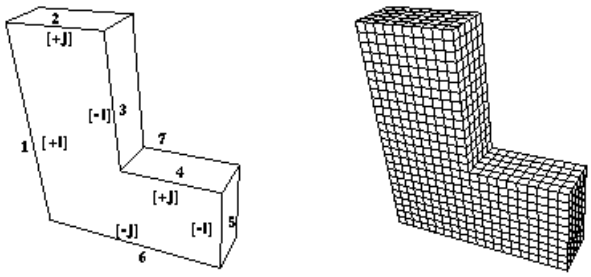


Figure 2. Scheme Submap Logical Properties

In special cases where quick results are desired, submap cornerpicking can be set to OFF. The corner picking will be accomplished by a faster, but less accurate algorithm which sets the vertex types by the measured interior angle at the given vertex on the surface. In most cases this is not recommended.

```
Set Submap CornerPicking {ON|off}
```

In special cases where 4 corners will be selected for a four-sided mapped region, but the region has more than 4 reasonable locations for the 4 corners, one may choose between the submapping or mapping corner picking algorithms to determine the 4 locations for 4 corners. In most cases this is not recommended. The following commands may be used.

```
Set Cornerpicking_MapAsSubmap {on|OFF}

Set Cornerpicking_SubmapAsMap {on|OFF}

List Cornerpicking_MapAsSubmap

List Cornerpicking_SubmapAsMap
```

After submapping has subdivided the surface and applied the mapped meshing technique mentioned above, the mesh is smoothed to improve mesh quality. Because the decomposition performed by submapping is mesh based, no geometry is created in the process and the resulting interior mesh can be smoothed. Sometimes smoothing can decrease the quality of the mesh; in this case the following command can turn off the automatic smoothing before meshing:

```
{Surface|Volume} <range> Submap Smooth
<on|off>
```

Surface submapping also has the ability to mesh periodic surfaces such as cylinders. An example of a periodic surface meshed with submapping is shown in Figure 3. The requirement for meshing these surfaces is that the top and bottom of the cylinder must have matching intervals.

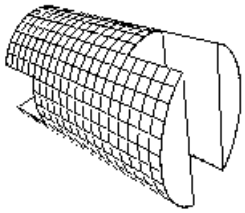


Figure 3. Periodic Surface Meshing with Submapping

For periodic surfaces, there are no curves connecting the top and bottom of the cylinder. Setting intervals in this direction on the surface can be done by setting the periodic interval for that surface (see Interval Assignment). No special commands need to be given to submap a periodic surface, the algorithm will automatically detect the fact that the

surface is periodic. Currently, periodic surfaces with interior holes are not supported.

Surface Vertex Types

- Surface Vertex Commands
- Listing and Drawing Vertex Types
- Triangle Vertex Types
- Adjusting the Automatic Vertex Type Selection Algorithm
- Volume Curve Types

Several meshing algorithms in CUBIT "classify" the vertices of a surface or volume to produce a high quality mesh. This classification is based on the angle between the edges meeting at the vertex, and helps determine where to place the corners of the map, submap or trimesh, or the triangles in the trimap or tripave schemes. For example, a surface mapping algorithm must identify the four vertices of the surface that best represent the surface as a rectangle. Figure 1 illustrates the vertex angle types for mapped and submapped surfaces, and the correspondence between vertex types and the placement of corners in a mapped or submapped mesh.

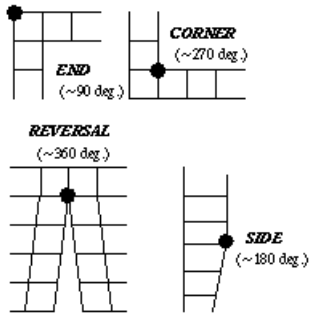


Figure 1. Angle Types for Mapped and Submapped Surfaces: An End vertex is contained in one element, a Side vertex two, a Corner three, and a Reversal four.

The surface vertex type is computed automatically during meshing, but can also be specified manually. In some cases, choosing vertex types manually results in a better quality mesh or a mesh that is preferable to the user. Vertex types can be specified directly as End, Side, Corner, or Reversal, or can be specified by giving the desired interior angle as 90, 180, 270, or 360, respectively.

Vertex types have a firmness, just as meshing schemes do. Automatically selected vertex types are **soft**, while user-set vertex types are **hard**.

Surface Vertex Commands

Vertex types are set using the following commands:

```
Surface <surface_id> [Vertex <vertex_id_range>
[Loop_index <int>]] Type
{End|Side|Corner|Reversal}

Surface <surface_id> [Vertex
[<vertex_id_range> [Loop_index <int>]] Angle
<value>

Surface <surface_id> [Vertex <vertex_id_range>
[Loop_index <int>]] Type {Default|Soft|Hard}
```

If no vertices are specified, the command is applied to all vertices of each surface.

Note that a vertex may be connected to several surfaces and its

classification can be different for each of those surfaces.

The influence of vertex types when mapping or submapping a surface is illustrated in Figure 2. There, the same surface is submapped in two different ways by adjusting the vertex types of ten vertices.

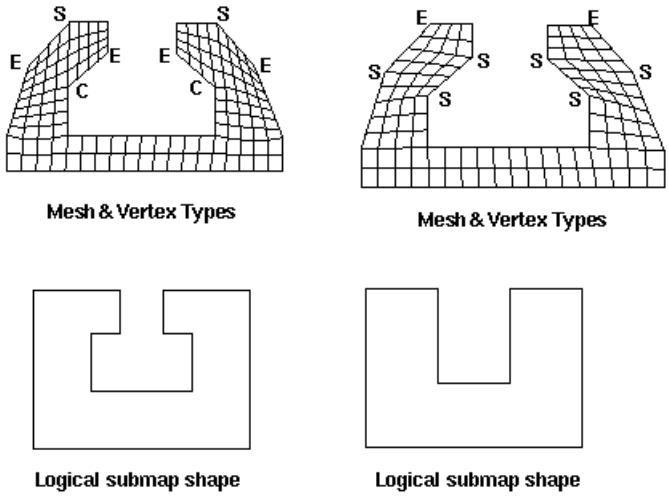


Figure 2. Influence of vertex types on submap meshes; vertices whose types are changed are indicated above, along with the mesh produced; logical submap shape shown below.

The **loop_index** is an advanced option used only for vertices where the boundary of a single surface passes through the same vertex more than once. This case is not common. If no loop index is specified for such a vertex, the specified vertex type is assigned to all occurrences of the vertex. The loop index for a specific occurrence of a vertex can be determined by listing the surface (**list surface <id>**) to show the list of curves in each loop bounding the surface, with the start and end vertex listed for each curve. The loop index begins at zero for the first curve in the first loop, and is incremented by one for subsequent curves through the last curve in the last loop. The loop index values corresponding to a specific vertex will be the loop index of each curve whose start vertex is the desired vertex.

Listing and Drawing Vertex Types

Listing a surface lists the types of the vertices. The vertex type settings may also be drawn with the following commands:

```
Draw Surface <surface_id_range> {Vertex
Angle|Vertex Type}
```

Triangle Vertex Types

For a surface that will be meshed with scheme trimap or tripave, the user may specify the angle below which triangles are inserted:

```
Surface <surface_id_range> Angle <angle>
```

The user may also set whether to add a triangle at a particular vertex:

```
Surface <surface_id> [Vertex <vertex_id_range>
[Loop_index <int>]] Type
{Triangle|Nontriangle}
```

Adjusting the Automatic Vertex Type Selection Algorithm

The user may specify the maximum allowable angle at a corner with the following command:

```
Set {Corner|End} Angle <degrees>
```

The user may also give greater priority to one automatic selection criteria over the others by changing the following absolute weights. The **corner weight** considers how large angles are at corners. The **turn weight** considers how L-shaped the surface is. The **interval weight** considers how much intervals must change. The **large angle weight** affects only auto-scheme selection: surfaces with a large angle will be paved instead. Each weight's default is 1 and must be between 0 and 10. The bigger a weight the more that criteria is considered.

```
Set Corner Weight <value>

Set Turn Weight <value>

Set Interval Weight <value>

Set Large Angle Weight <value>
```

An illustration of a mesh produced by the submapping algorithm is shown in Figure 2. The meshes produced by submapping on the left and right result from adjusting the vertex types of the eight vertices shown.

Volume Curve Types

When sweeping, a 2.5 dimensional meshing scheme, curves perpendicular to the sweep direction can have a type with respect to the volume. These types are usually automatically selected. The following commands are useful:

```
Draw Volume <surface_id_range> {Curve
Angle|Curve Type}

List Volume <volume_id> Curve Type

Volume <volume_id> [Curve <curve_id_range>]
Type {End|Side|Corner|Reversal}

Volume <volume_id> [Curve <curve_id_range>]
Type {Default|Soft|Hard}
```

Sweep

Applies to: Volumes

Summary: Produces an extruded hexahedral mesh for 2.5D volumes.

Syntax:

```
Volume <range> Scheme Sweep [Source [Surface] <range>]
[Target [Surface] <range>]
[Propagate bias]
[Sweep smooth {auto | smart affine | linear | residual |
winslow} ]
[Sweep transform {LEAST SQUARES | Translate}] [Autosmooth
target {ON|off} ]

Volume <range> Scheme Sweep Vector <xval yval zval>

Volume <range> autosmooth target [off|ON]
fixed imprints [on|OFF]
smart smooth [ON|off] tolerance <val 0.0 to 1.0=0.2>
nlayers <val >=0=5>
```

Related Commands:

```
Set Multisweep [On|Off]

Multisweep Smoothing {ON|Off}

Multisweep Volume <range> Remove

Volume <range> Redistribute Nodes {ON|off}

[Set] Legacy Sweeper {On|Off}
```

Discussion:

The sweep algorithm can sweep general 2.5D geometries and can also do pure translation or rotations. A 2.5D geometry is characterized by source and target surfaces which are topologically similar. The hexahedral mesh is swept (extruded) between source and target along a single logical axis. Bounding the swept hexahedra between source and target surfaces, are the linking surfaces. Figures 1 and 2 show examples of source, target and linking surfaces.

Command Options: The user can specify the source and target surfaces. The user can also specify a geometric vector approximating the sweep direction, and let CUBIT determine the source and target surfaces. The user can specify just the source surfaces, and let cubit guess the target, or "scheme auto" can also be used.

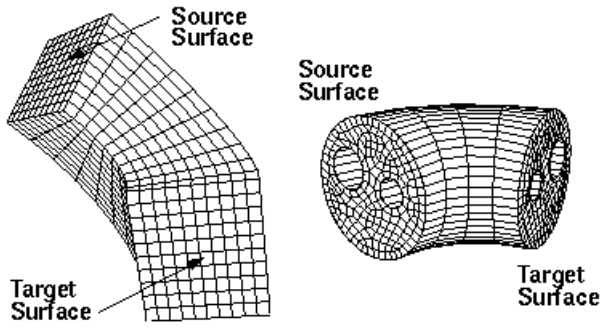


Figure 1. Sweep Volume Meshing

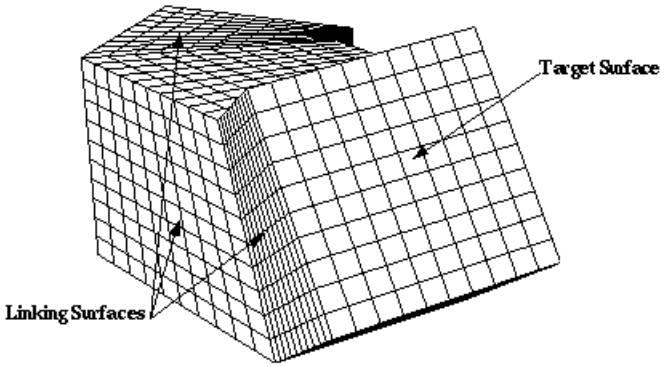


Figure 2. Multiple Linking Surface Volume Meshing with Scheme Sweep

In general, the procedure for using the sweep scheme is to first mesh the source surfaces. Any surface meshing scheme may be employed. Figure 1 displays swept meshes involving mapped and paved source surfaces. Linking surfaces must have either mapping or submapping schemes applied. The sweep algorithm can also handle multiple surfaces linking the source surface and the target surfaces. An example of this is shown in Figure 2. Note that for the multiple- linking-surface meshing case, the interval requirement is that the total number of intervals along each multiple edge path from the source surface to the target surface must be the same for each path. Once the appropriate mesh is applied to the source surface and intervals assigned, the **mesh** command may be issued.

In many cases auto-scheme selection can simplify this process by recognizing sweepable geometries and automatically select source and target surfaces. If the source and target surfaces are not specified, CUBIT attempts to automatically select them. CUBIT also automatically sets curve and vertex types in an attempt to make the mesh of the linking surfaces lead from a source surface to a target surface. These automatic selections may occasionally fail, in which case the user must manually select the source/target surfaces, or some of the curve and vertex types. After making some of these changes, the user should again set the volume scheme to sweep and attempt to mesh. In some cases of 1-1 sweeps, the source and target are swapped. Precedence for which surface to use as the source is {meshed, merged, specified as source}. If the user wants to avoid swaps and enforce that a particular surface is the source, then they can mesh that prior to sweeping.

Occasionally the user must also adjust intervals along curves, in addition to the usual surface interval matching requirements. For a given pair of source/target surfaces, there must be the same number of hexahedral layers between them regardless of the path taken. This constrains the number of edges along curves of linking surfaces. For example, in Figure 1 right, the number of intervals through the holes must be the same as along the outer shell.

Propagate bias Option: The propagate bias option attempts to preserve the source bias by propagating bias mesh schemes from the curves of the source surface to the curves of the target surface. It also propagates bias from one linking curve to all other linking curves.

Sweep transform Option: Swept meshes are created by projecting points between the source and target surfaces using affine transformations and then connecting them to form hexahedra. The method used to calculate the affine transformations is set using the sweep transform option.

> **Least squares**: If the least squares option is selected then affine transformations between the source and target are calculated using a least squares method.

translate: If the translate option is selected then a simple translate affine transformation is calculated based upon the centroid of the source and target.

Sweep smooth Option: *Note: This option is available only in Legacy mode. The command '**set legacy sweeper on|off** controls the mode. Legacy mode is OFF by default.*

To ensure adequate mesh quality, optional smoothing schemes are available to reposition the interior nodes. The sweep tool permits five types of smoothing that are set with the following command prior to meshing a volume whose mesh scheme is sweep:

Linear: If this option is selected, no layer smoothing is performed. The node positions are determined strictly by the affine transformation from the previous layer. Good quality swept meshes can be constructed using “linear” provided the volume geometry and meshed linking surfaces permit the volume mesh to be created by a translation, scaling, and/or rotation of the source mesh. Volumes for which this is nearly true may also produce acceptable quality with “linear”. As one would expect, this option generates swept meshes more quickly than the other sweep smooth options. This option is rarely needed since the next option produces better results with little time penalty.

Smart affine: The “smart affine” option does minimal smoothing of the interior nodes. Affine transformations are used to project the source and target surfaces to the middle surface of the volume. The position of the middle surface nodes is the average of the projected nodes from the source and target surfaces. The error in projecting from source and target is computed, and this error is linearly distributed back to the source and target.

Residual: The “residual” method is often used for meshing volumes that cannot be swept with the “smart linear” method. It tends to produce better quality meshes than the “smart linear” method while running faster than the Winslow-based smoother. The sweeping algorithm uses an affine transformation to calculate the interior nodes’ positions, but the mesh on the linking surface determines the positions of the nodes on the boundary of the layer. For the “residual” method, CUBIT calculates corrective adjustments for interior nodes using the “residuals” from boundary nodes. The “residual” is defined as the distance between the boundary node’s position (as determined by the surface mesh) and the boundary node’s ideal position (as determined by the affine transformation of the previous layer). Cubit computes the residual forward from the source and backward from the target to get best the possible node position.

Winslow: Smooth scheme “winslow” smooths each layer using a weighted, elliptic smoother. The weights are computed from the source mesh; they help maintain any biased spacing that occurs on the source mesh. For example, one might want to use the “winslow” option if the source was a biased mesh that was created using scheme circle. The biasing of the outer elements of the source mesh may be destroyed if one of the other smooth options is used. The interior nodes are initially place using the residual method.

AUTO: This is the default for the sweep smooth option. “auto” causes the Sweeper to automatically choose between “smart affine” and “residual.” Auto will choose “off” if the layer needs little or no smoothing or “residual” if it needs smoothing. Scheme “auto” does not guarantee that no negative Jacobians are produced. This option produces acceptable results in most cases. If it fails to produce a quality mesh, then choose one of the other sweep smooth options.

> If none of these smooth schemes result in adequate mesh quality, one can consider trying one of the volume smoothing schemes such as condition number or mean ratio.

Autosmooth target Option and Command

During sweeping, a quad mesh is placed on each source surface. Then the collection of nodes & quads from all the source surfaces is projected onto the target surface. The **autosmooth target** command or sweep command options control the placement of the nodes onto the target surface.

```
Volume <range> autosmooth target [off|ON]
fixed imprints [on|OFF]
smart smooth [ON|off] tolerance <val 0.0 to
1.0=0.2>
nlayers <val >=0=5>
```

Issuing the command “Volume <id> autosmooth target off”, or using these options in the sweep command, will project the source nodes onto the target without any subsequent smoothing to improve quality. The result is that the relative placement of the nodes on the target will be as close to identical as possible to the relative placement of the node on the sources. This should be used when sweeping models that are very thin, and smoothing of the target could result in significant skew introduced in the thin layers in the sweep. Axisymmetric models might also want to turn OFF the autosmooth target so that the nodes are identically placed on the symmetry plane surfaces.

Issuing the command “Volume <id> autosmooth target on”, or using it as an option in the sweep command, will call a surface smoother after the initial projection of the nodes onto the target in order to improve surface element quality. This smoothing does not consider hex element quality, only quality of the target surface mesh. This command will smooth all nodes on the target surface. Adding the **“fixed imprint on”** keyword onto the command will cause the target nodes which are projections of source nodes on source curves and vertices to remain fixed during smoothing. Only target nodes, which are projections of source surface nodes will be smoothed. The “**smart smooth on**” option provides further control to the user. If “smart smooth” is turned on, target surface smoothing will only move nodes which are within “nlayers” of a target surface quad element that has a scaled Jacobian quality measure less than the specified “tolerance” value.

Multisweep

While the basic sweeping algorithm requires a *single* target surface, the sweeping algorithm can also handle *multiple* target surfaces. The multisweep algorithm works by recognizing possible mesh and topology conflicts between the source and target surfaces and works to resolve these conflicts through the use of the virtual geometry capabilities in CUBIT. Figure 4 shows some examples of volumes which have been meshed with the multisweep algorithm.

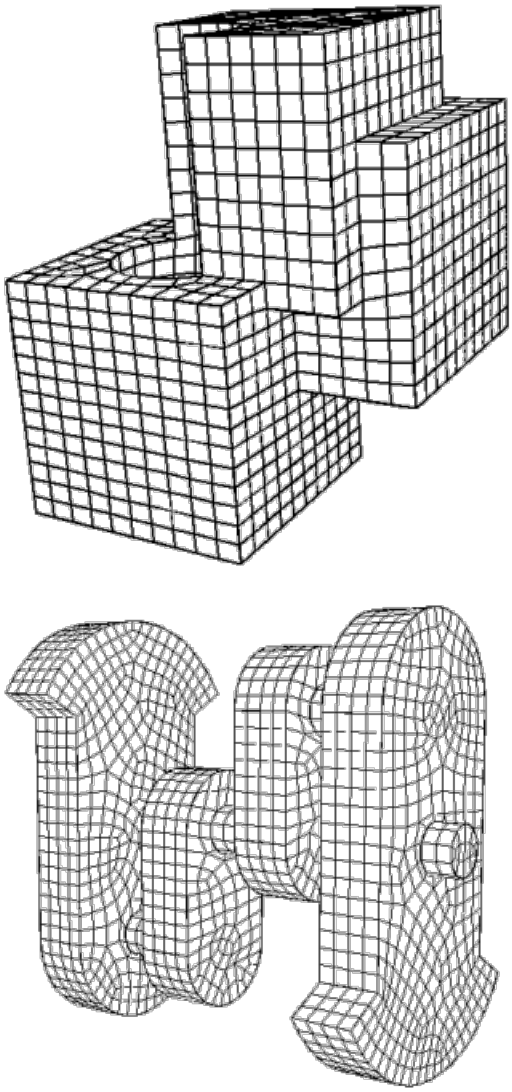


Figure 4. Examples of Multisweep meshes.

Linear: If this option is active and/or target surfaces are omitted from the scheme setting command, CUBIT will determine source and target surfaces (See Automatic Scheme Selection). Sweeping can be further automated using the "sweep groups" command.

- Limitations: Not all geometries are sweepable. Even some that appear sweepable may not be, depending on the linking surface meshes. Highly curved source and target surfaces may not be meshable with the current sweep algorithm.

Grouping Sweepable Volumes

Swept meshing relies on the constraint that the source and target meshes are topologically identical or the target surface is unmeshed. This results in there being dependencies between swept volumes connected through non-manifold surfaces; these dependencies must be satisfied before the group of volumes can be meshed successfully. For example, if the model was a series of connected cylinders, the proper way to mesh the model would be to sweep each volume starting at the top (or bottom) and continuing through each successive connected volume.

With larger models and with models that contain volumes

that require many source surfaces, the process of determining the correct sweeping ordering becomes tedious. The sweep grouping capability computes these dependencies and puts the volumes into groups, in an order which represents those dependencies. The volumes are meshed in the correct order when the resulting group is meshed.

To compute the sweep dependencies, use the command:

```
Group Sweep Volumes
```

This will create a group named "sweep_groups", which can then be meshed using the command:

```
Mesh sweep_groups
```

In some automated meshing systems, the source and target surfaces are named using a naming pattern. For example, all source surfaces might be given names "xxx.source" and all target surfaces might be named "xxx.target". This allows the automated setting of the sweep direction based on predetermined names rather than ids. The following command is used to set the source and targets based on the naming pattern.

```
Set {Source|Target} Surface Pattern
'<pattern>' [Include Volume Name]
```

The **pattern** is checked against all surfaces in the model using a simple case-sensitive substring match. All surfaces which contain that string of letters in their name will be designated as either a source or target surface, depending on which option the user specifies. For example:

```
br x 10
surface 1 name 'brick.top'
surface 2 name 'brick.bottom'
set source surface pattern 'top'
set target surface pattern 'bottom'
volume 1 scheme sweep
list volume 1 brief
```

Node Redistribution

```
Volume <range> redistribute nodes
{ON|off}
```

With redistribute set to ON, the boundary nodes of a mappable surface are moved until the spacing between the nodes are equivalent on the two opposing curves. In other words, the parametric values of the nodes lying on the two opposite curves are matched.

Redistribute option ON will assist in avoiding the skewness of the mapped mesh. In the below examples, the linking surfaces are meshed using mapped scheme, and with redistribute option ON, the skewness is significantly avoided (see figures (4) and (5)).

Note:

1. Redistribute option ON will affect all mapped surfaces, not just the linking surfaces of a swept volume. Even though the example below shows a swept volume, the command can be used independent of the sweeping command. That is, it

can be used while meshing surface models that contain mappable surfaces.

2. If the linking surfaces of a swept mesh contain submappable surfaces, then the affect of redistribute option ON is generally not seen. The current implementation is restricted to mappable surfaces only and doesn't handle submappable surfaces. In the future, we should be able to easily extend the redistribute option to submappable surfaces.

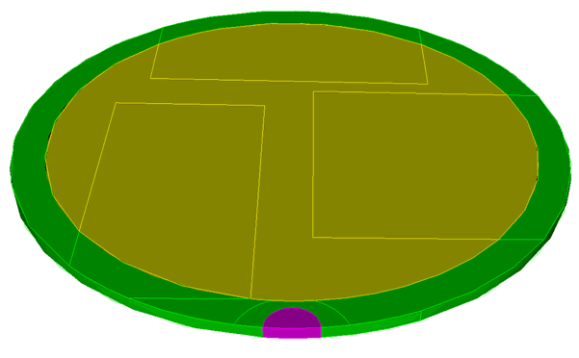


Figure 1 - Linking surfaces of a many-to-one sweepable solid (shown in green) is mappable

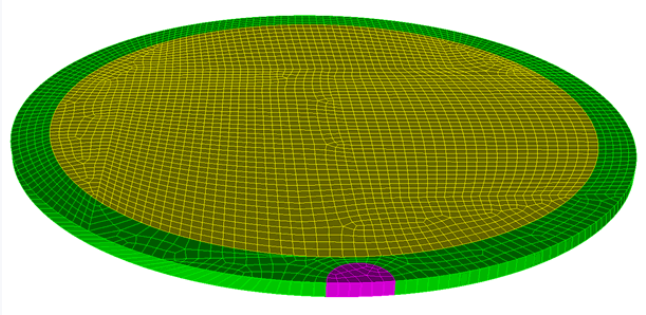


Figure 2 - Highly skewed elements on the linking mapped surface with 'redistribute nodes OFF'

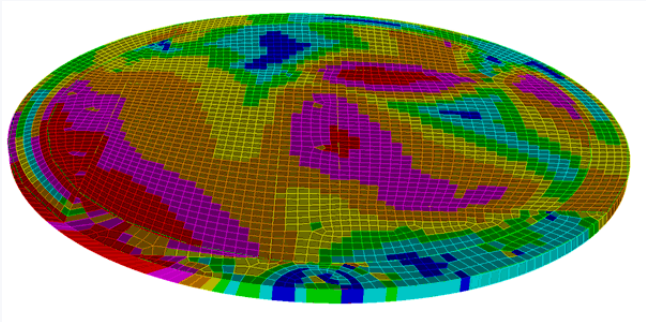


Figure 3 - Quality of mesh with 'Redistribute Nodes OFF'

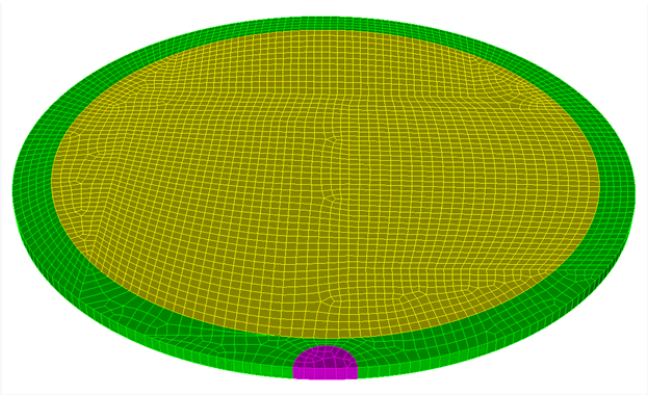


Figure 4 - High skew on the linking mapped surface can be avoided with 'Redistribute Nodes ON'

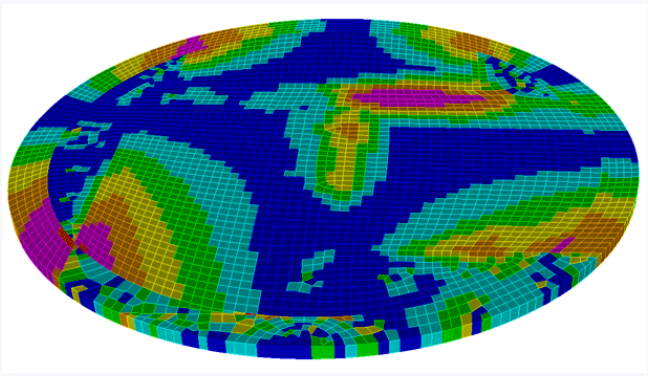


Figure 5 - Quality of mesh with 'Redistribute Nodes ON'

TetMesh

Applies to: Volumes

Summary: Automatically meshes a volume with an unstructured tetrahedral mesh.

Syntax:

```
Volume <range> Scheme TetMesh [Proximity Layers {on[<num_layers>]|OFF}] [Geometry Approximation Angle <angle>]
```

Related Commands:

```
[Set] Tetmesher Add mid_edge_nodes {on|OFF}

[Set] Tetmesher Optimize Surface mid_edge_nodes {on|OFF}

[Set] Tetmesher Anisotropic layers {on|OFF [<layers=2>]}

[Set] Tetmesher Boundary Recovery {on|OFF}

[Set] Tetmesher HPC {ON|off} [Threads <value=4>]

[Set] Tetmesher HPC minimum size [<size>]

[Set] Tetmesher Interior Points {ON|off}

[Set] Tetmesher Optimize { { [Level <level> [Overconstrained Edges {on|OFF}] [Overconstrained Tetrahedra {on|OFF}] [Sliver {on|OFF}] } | Default }

[Set] Trimesher Surface Gradation <value>

[Set] Trimesher Volume Gradation <value>

[Set] Trimesher Geometry Sizing {ON|off}

[Set] Trimesher Split Overconstrained Edges {on|OFF}

THex Volume All

Volume <volume_id> Tetmesh Respect {Face|Tri|Edge} <range>

Volume <volume_id> Tetmesh Respect Node <range> [Size <size>]

Volume <volume_id> Tetmesh Respect Clear

Volume <volume_id> Tetmesh Respect File '<filename>'

Volume <volume_id> Tetmesh Respect Location (options)

Tetmesh Tri <range> [Make {Block|Group} [<id>]]

Tetmesh Tri <range> {Add|Replace} {Block|Group} <id>
```

```
Volume <id_range> Tetmesh growth_factor <value
1.0 to 10.0 = 1.05>
```

Discussion

The **TetMesh** scheme fills an arbitrary three-dimensional volume with tetrahedral elements. The surfaces are first triangulated with one of the triangle schemes (TriMesh, TriAdvance or TriDelaunay) or a quadrilateral scheme with the quadrilaterals being split into two triangles (QTri). If a meshing scheme has not been applied to the surfaces, the TriMesh scheme will be used.

Included in Cubit is a third party software library for generating tetrahedral meshes called MeshGems. This is a robust and fast tetrahedral mesher developed by the French laboratory INRIA and distributed by Distene. It utilizes an algorithm for automatic mesh generation based upon the Voronoi-Delaunay method. Figure 1 shows a CAD model meshed with the **TetMesh** scheme, with the TriMesh scheme used to mesh the surfaces.

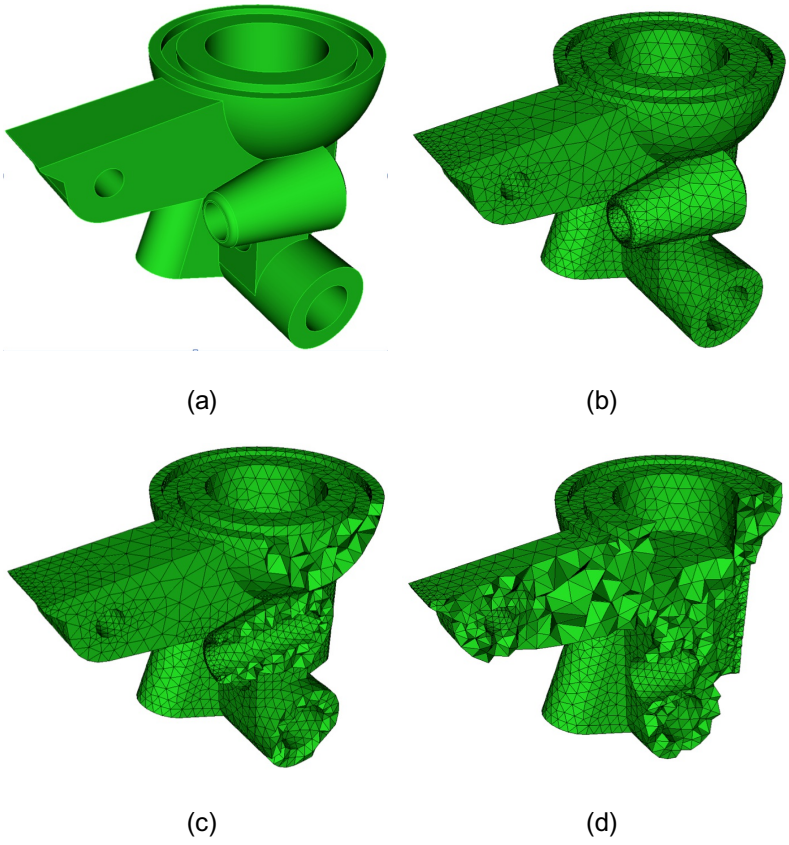


Figure 1. Tetrahedral mesh generated with the TetMesh scheme using default settings. (a) Initial CAD geometry (b) CAD model with surface mesh generated with TriMesh scheme. (c) and (d) Cut-away views of the interior tetrahedral mesh

The **TetMesh** scheme is usually very good at generating a mesh with its default settings. In most cases no adjustments to default settings are necessary. Using the size assigned to the volume, either assigned explicitly or defined with an auto size, the **TetMesh** scheme will attempt to maintain the assigned size, except where features smaller than the specified size exist. In this case, smaller tets will automatically be generated to match the feature size. The tet mesher will then generate a smooth gradation from the small tets used to capture features, to the size specified on the volume. This effect is shown in figure 1 where internal transitions in tetrahedra size can be seen. User defined sizes and intervals can also be assigned to individual surfaces and curves for more specific control of element sizes.

A **sizing function** can also be used with the **TetMesh** scheme to control element sizes, however the algorithm used for meshing surfaces will automatically revert to the **TriAdvance** scheme. This is because the **TetMesh** scheme provides built-in capabilities for adaptively controlling the element sizes based on geometry. More details can be found in Geometry Adaptive Sizing for TriMesh and TetMesh Schemes

When using the **TetMesh** and **TriMesh** schemes, recommended practice is to mesh all surfaces and volumes simultaneously. This provides the greatest flexibility to the algorithms to determine feature sizes and their effect on neighboring surfaces and volumes.

TetMesh Scheme Options

The **Tetmesh** options described below can be set to adjust the default behavior of the tet mesher. Scheme options are assigned independently to each volume as part of the **scheme tetmesh** command.

```
Proximity Layers {on[<num_layers>]|OFF}
```

In some thin regions of the model, it may be necessary to ensure a minimum number of element layers through the thickness to better capture physical properties. Using the **proximity layers** setting, the specified minimum **num_layers** of tetrahedra will be placed in thin regions, even if the tetrahedra sizes drop below the size assigned to the volume. The default setting for **proximity layers** is **OFF** where element sizes will not be affected in thin regions.

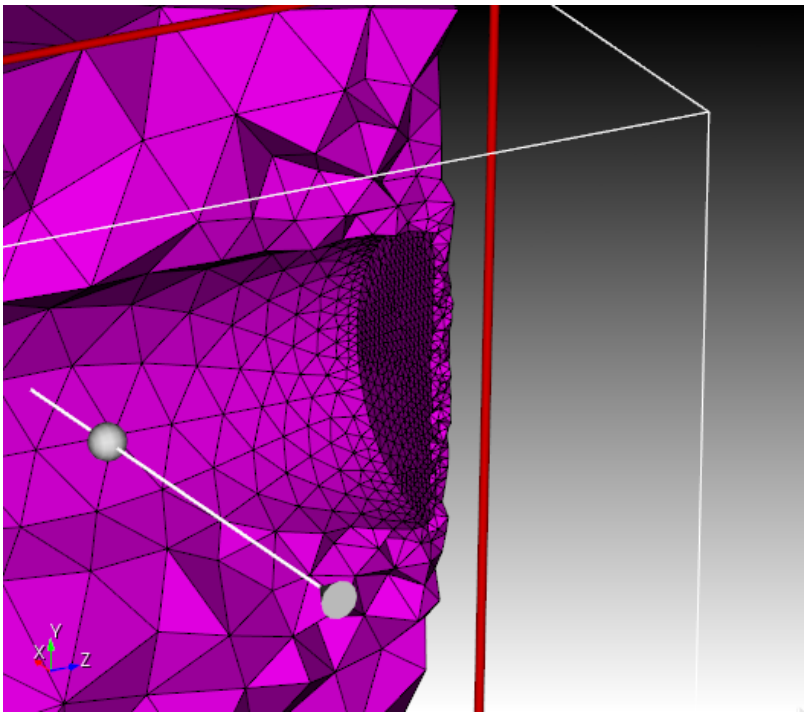


Figure 2. Demonstrates the effect of using proximity layers on a cut-away section of a volume. Note the layers of smaller tets placed in the thin region.

```
Geometry Approximation Angle <angle>
```

For non-planar CAD surfaces, an approximation must always be made to capture the curved features using the linear faces of the tetrahedra. When a **geometry approximation angle** is specified, the tet mesher will adjust element sizes on curved surfaces so that the linear edges of the tetrahedra will deviate no greater than the specified **angle** from the

geometry. Figure 3 illustrates how the geometry approximation angle is determined. If the red curve represents the geometry and the black segments represent the mesh, the angle **θ** is the angle between the tangent plane at point **A** and the plane of a triangle at **A**. **θ** represents the maximum deviation from the geometry that the mesh will attempt to capture. As shown in figure 2(b), a smaller geometry approximation angle will normally result in more elements, but it will more closely approximate the actual geometry. The default approximation angle is 15 degrees.

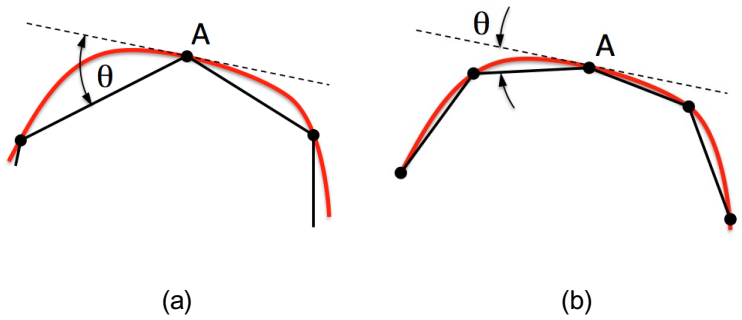


Figure 3. The geometry approximation angle θ is shown as the maximum deviation between the tangent plane at A and the plane of a triangle at A.

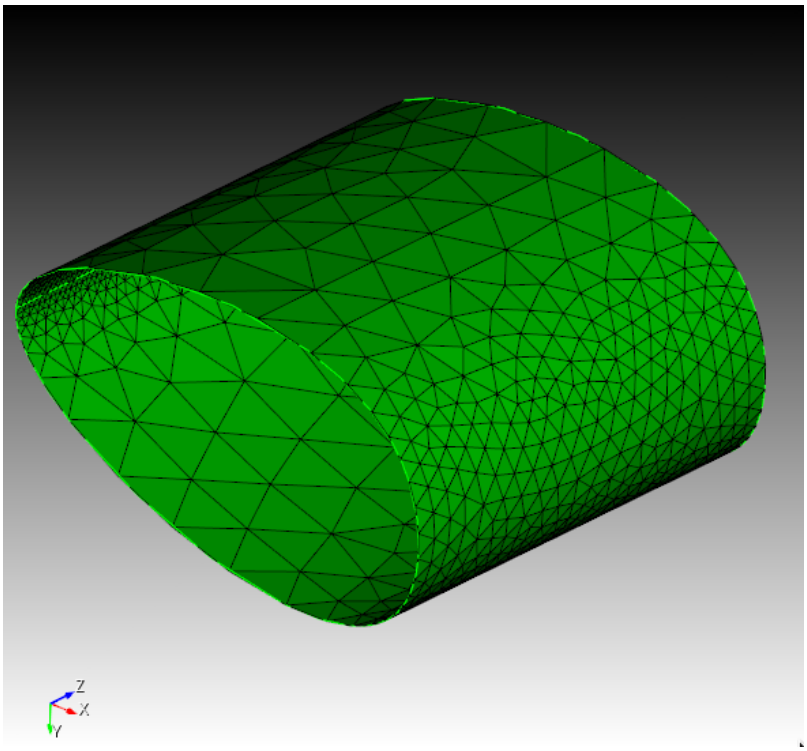


Figure 4. Demonstrates the effect of the geometry approximation angle set on the volume. Triangle sizes on the interior of surfaces will be adjusted to better capture curvature.

Global Tetmesher Options

The user may set options that control the operation of the tet-meshing algorithms. These tetmesher options are global settings and apply to all tetmeshes generated when the scheme is set to **TetMesh** until the option is changed by the user.

```
[Set] Tetmesher Add mid_edge_nodes {on|OFF}
```

If the triangle mesh given to tetmeshing has quadratic (mid-edge) nodes, tetmeshing can automatically create quadratic edge nodes while generating the tets if this option has been turned on. By performing this step during tetmeshing, these nodes can be placed optimally by the Meshgems tetmesher, improving element quality. If triangle or face elements have been set to be 'respected' in the tetmesh, they must also have quadratic edge elements or meshing will fail. The default value for this option is off. If set to off, quadratic edge nodes will be placed after meshing, exactly half way along the linear edge.

```
[Set] Tetmesher Optimize Surface
mid_edge_nodes {on|OFF}
```

If the triangle mesh given to tetmeshing has quadratic (mid-edge) nodes, tetmeshing can also automatically optimize the locations of these mid edges nodes during the tetmeshing operation to achieve improved quality. create quadratic edge nodes while generating the tets if this option has been turned on. By performing this step during tetmeshing, these nodes can be placed optimally by the Meshgems tetmesher, improving element quality. If triangle or face elements have been set to be 'respected' in the tetmesh, they must also have quadratic edge elements or meshing will fail. The default value for this option is off. If set to off, quadratic edge nodes will be placed after meshing, exactly half way along the linear edge.

```
[Set] Tetmesher Anisotropic Layers {on|OFF
[<layers=2>]}
```

The **Anisotropic Layers** setting attempts to place the specified number of layers of tetrahedra through thin regions of the volume while respecting the volume mesh size in the thick direction. The default number of layers is two. This option is currently under development and can sometimes generate high aspect ratio tetrahedra. The number of layers generated can sometimes exceed the number of layers specified..

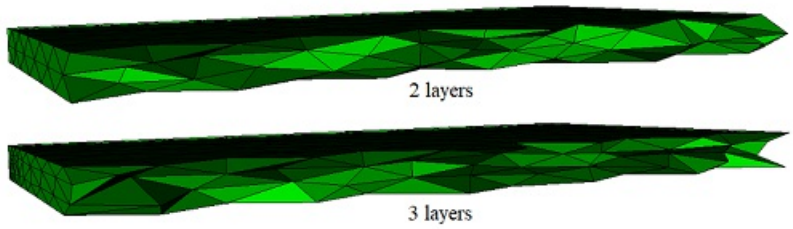


Figure 5. Anisotropic Volume Meshing

```
[Set] Tetmesher Boundary Recovery {on|OFF}
```

The **TetMesh** scheme includes a specialized module known as *Boundary Recovery*. Normally if the quality of the surface mesh is good, the boundary recovery module is not used and the resulting tet mesh will conform exactly to the triangles defined on the surfaces without additional processing. In some cases where the surface mesh contains triangles that are of poor quality (ie. highly stretched or sliver shaped triangles) the tet mesher is unable to generate sufficiently good quality elements. When this occurs, the boundary recovery module is automatically invoked. This module does additional processing to temporarily modify boundary triangles so that reasonable quality tets may be inserted. The boundary adjustment is done as an intermediate phase and in most cases the

boundary triangulation remains unchanged following meshing. The **TetMesh** scheme in Cubit will automatically invoke the boundary recovery module if the minimum surface mesh quality drops below a condition number of 0.2. However, if the the boundary recovery option is set to **ON**, the tet mesher will use the boundary recovery module regardless of surface mesh quality. Turning this setting **ON** will normally increase the time to generate the mesh, but may result in improved mesh quality. The default setting is **OFF**.

```
[Set] Tetmesher HPC {ON|off} [Threads
<value=4>]
```

This option turns on or off MeshGems-Tetra HPC, the multithread or distributed tetrahedral volume mesh generator. The MeshGems-Tetra HPC software is an automatic multithread or distributed tetrahedral mesh generator based on the constrained VORONOI-DELAUNAY method. Using the **threads** option, one can specify the maximum number of threads MeshGemsTetra HPC will use to generate the mesh. The effective number of threads used will be determined by the number of parallel subdomains used, the default of 4, and the maximum of 8. If HPC is off, the older serial tetmesher MeshGems-Tetra is used. The default setting is **ON**.

```
[Set] Tetmesher HPC minimum size [<size>]
```

Sets the minimum edge length in tetmeshing, when using Distene's MeshGems-Tetra HPC.

```
[Set] Tetmesher Interior Points {ON|off}
```

Infrequently, the user desires a model with as few interior points as possible. The **Interior Points** command allows the user to enable or disable, or turn **OFF** the insertion of interior points. If interior points are disabled, the tetmesher will attempt to mesh the volume using only the exterior points. This may not be possible and a few points will be inserted to allow tet-meshing to complete. The default setting is **ON**, meaning that interior points will be inserted according to the specified element size.

```
[Set] Tetmesher Optimize Level <level>
```

The **Tetmesher Optimize Level** command allows the user to control the degree of optimization used to automatically improve element quality following the initial generation of tetrahedra. The optimization level is an integer in the range 0 to 6, which represent how aggressively the algorithm will attempt to improve element quality by automatically adjusting element connectivity and smoothing. The integers 0 to 6 can also be represented as **none (0), light (1), medium (2), standard (3), strong (4), heavy (5), and extreme (6)**. Greater values will result in greater computation time, however may result in improved mesh quality. The default is 3 or standard optimization.

```
[Set] Tetmesher Optimize Overconstrained Edges
{on|OFF}
```

This option controls the splitting of overconstrained edges. An edge is considered overconstrained when it connects two surface nodes but does not belong to the surface. This condition may not be desirable for some FEA analysis. Splitting edges can useful to guarantee two elements through the thickness. When using MeshGems-Tetra, this option cannot be used by itself; it must be used with the **optimize**

tetrahedra option. If using MeshGems-Tetra HPC, it can be used by itself. The default for **optimize overconstrained edges** is **OFF**.

```
[Set] Tetmesher Optimize Overconstrained
Tetrahedra {on|OFF}
```

In some cases, the default mesh generated with the **TetMesh** scheme may result in cases where more than one triangle face of a single tetrahedra lies on the same geometric surface. This condition may not be desirable for some FEA analysis. The default for **optimize overconstrained tetrahedra** is **OFF**.

```
[Set] Tetmesher Optimize Sliver {on|OFF}
```

A sliver tetrahedra is one in which the four nodes of the tet are nearly co-planar. Sliver tets are a common occurrence when using the Delaunay method, but are normally removed by standard optimization. In some cases, sliver tets may still remain even after optimization. To facilitate removal of all sliver-shaped tets, the **optimize sliver** option may be set to **ON**. In this event, additional processing will be done on the mesh to attempt to identify and remove all sliver-shaped tets from the mesh. Since this step may take additional time, and in most cases is not needed, the default setting is **OFF**.

```
[Set] Tetmesher Optimize Default
```

The **Tetmesher Optimize Default** command restores the default optimization values: level = 3 (standard), overconstrained edges = off, overconstrained tetrahedra = off, and sliver = off.

Using tets as the basis of an unstructured hexahedral mesh

Tet meshing can be used to generate hexahedral meshes using the **THex** command. Each of the tetrahedron can be converted into 4 hexes, producing a fully conformal hexahedral mesh, albeit of poorer quality. These meshes can often be used in codes that are less sensitive to mesh quality and mesh directionality. The **THex** command requires that all tets in the model be converted to hexahedra with the same command.

Conforming the tetmesh to internal features

In some cases it is necessary for the finite element mesh to conform to internal features of the model. The tetmesh scheme provides this capability provided the **tetmesh respect** command has been previously issued to define the features that will be respected.

```
Volume <volume_id> Tetmesh Respect
{Face|Tri|Edge} <range>

Volume <volume_id> Tetmesh Respect Node
<range> [Size <size>]
```

The tetmesh respect command allows the user to specify mesh entities that will be part of a tetrahedral mesh. These faces, triangles, edges, or nodes are inside the volume since all surface mesh features will appear in the final tetrahedral mesh by default. These mesh entities specified to be respected can be generated from other meshing commands on free vertices, curves, or surfaces.

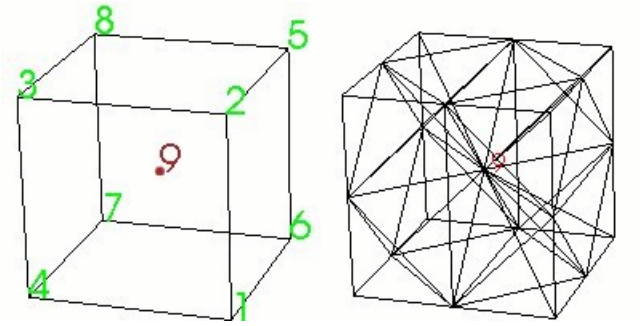


Figure 2. Example of using tetmesh respect to ensure node 9 is captured in the tetmesh.

Figure 2 is an example of using the **tetmesh respect** command to enforce a node at the center of a cube. Node 9 in this example was generated by first creating a free vertex at the center location and meshing the vertex. (mesh vertex 9). The following commands would then be used to generate the tetmesh that respected node 9.

```
volume 1 scheme tetmesh
tetmesh respect node 9
mesh volume 1
```

The **tetmesh respect** command can also be used to enforce multiple mesh entities. To accomplish this, the **tetmesh respect** command may be issued multiple times. For example, If node 12 and a triangle 2 inside volume 3 was to appear in the volumetric mesh, the following commands could be used:

```
volume 3 scheme tetmesh
volume 3 tetmesh respect node 12
volume 3 tetmesh respect tri 2
mesh volume 1
```

The **tetmesh respect** command can also be given a **size** value with a node. When given a size, the generated tet elements surrounding the node will have sizes matching the given size. This may be useful to provide refinement at given locations within a tet mesh.

Unlike the **tetmesh respect** command described above, the **tetmesh respect file** and **tetmesh respect location** commands do not require underlying geometry.

```
Volume <volume_id> Tetmesh Respect File
'<filename>'

Volume <volume_id> Tetmesh Respect Location
(options)
```

These two commands create mesh data that only the tetmesher knows about. Thus, to respect a point at (1.0, 0.0, -1.0) in your model, enter the command

```
volume 1 tetmesh respect location 1 0 -1
```

This is much simpler than creating the vertex, meshing it, and then respecting it.

If the model has many points that must be respected, use the file version of the command. First generate a file with all of the points, edges, and triangles that should be respected. The format of the file is the format used by the facet file. Now, use the following command to respect all of the information in the file for the given volume.

```
volume 2 tetmesh respect file 'my_points.facet'
```

Finally, the following command is used to remove the respected data from an entity.

```
Volume <volume_id> Tetmesh Respect Clear
```

The tetmesh respect clear command is the only way to remove respected data from a volume without deleting the volume. Unfortunately, it removes all respected data from the volume. Therefore, if the model has a lot of data to be respected, it is best to put it in a file or keep a journal file that can be edited.

Controling the gradation of the mesh size inside the volume

```
Volume <id_range> Tetmesh growth_factor <value
1.0 to 10.0 = 1.05>
```

The **growth_factor** option controls how fast the tetrahedra sizes can change when transitioning from small to larger sizes within the volume. For example a value of 1.5 will attempt to limit the ratio between 2 adjacent tetrahedral edges. Valid values for gradation should be greater than or equal to 1.0 and usually less than 2 or 3. The larger the value, the faster the transition is. Likewise, values closer to 1.0 will result in a more uniform mesh. The default setting for **growth_factor** is 1.05, allowing for a somewhat slow transition between sizes within a volume. The size at the interior of a volume can be controlled using the **Volume <range> [Interval] Size <interval_size>** command.

Gradation of the triangles on the surfaces can also be controlled independently using the global settings **[set] trimesher surface gradation** and **[set] trimesher volume gradation**.

Generating a Tetmesh from a Skin of Triangles

```
Tetmesh Tri <ids> [growth_factor <value>]
[Make {Block|Group} [<id>]]

Tetmesh Tri <ids> [growth_factor <value>]
{Add|Replace} {Block <id>|Group <id>}
```

The **Tetmesh Tri** command generates a tetrahedral mesh from the list of triangles entered. The triangles must form a closed surface. The command fails if they do not. The list of triangles may be a skin, and thus a command such as **tetmesh tri in block 1** would be acceptable, should block 1 be a previously defined skin.

The first command form has optional arguments. If the **make** option and its arguments are present, then the specified block or group will contain the generated tet elements. The command fails with the make option if the specified block or group already exists. If the block or group id is omitted, the next available block or group id is used.

The second command form has two options, **add** and **replace**. Each option requires specifying an existing block or group. If the block or group does not exist, the command fails. The **add** option appends the tet elements to the block or group. The **replace** option removes any existing mesh from the block or group before adding the tet elements.

The **growth_factor** option helps control the transition from small to larger sizes within the mesh. The value specified will be the approximate ratio in the size of adjacent tets going from the boundary into the interior of the mesh. For example, a **growth_factor** of 1.0 will give near-constant sizing, while a **growth_factor** of 1.3 allows approximately 30% growth in each layer of adjacent tets.

Tetprimitive

Applies to: Volumes

Summary: Meshes a 4 "sided" object with hexahedral elements using the standard tetrahedron primitive.

Syntax:

```
Volume <range> Scheme Tetprimitive [Combine
Surface <range>] [Combine Surface <range>]
[Combine Surface <range>] [Combine Surface
<range>]
```

Discussion:

The tetprimitive scheme is used to create a hexahedral mesh in a volume which fits the shape of a tetrahedral primitive. The **Tetprimitive** scheme assumes that each of the four surfaces have been meshed with the triprimitive, or similar, meshing scheme. If more than four surfaces form the tetrahedron geometry, the surfaces forming a logical side can be combined using the **combine** option.

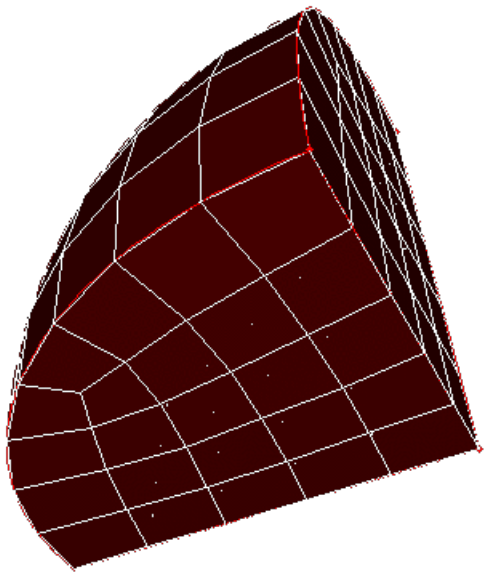


Figure 1. Sphere octant hex meshed with scheme Tetprimitive, surfaces meshed using scheme Triprimitive

TriAdvance

Applies to: Surfaces

Summary: Automatically meshes surface geometry with triangle elements.

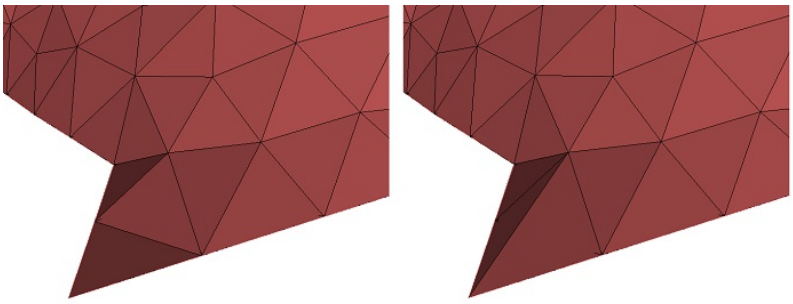
Syntax:

```
Surface <range> Scheme TriAdvance
```

Discussion:

The triangle meshing scheme TriAdvance fills an arbitrary surface with triangle elements. It is an advancing front algorithm which allows holes in the surface and transitions between dissimilar element sizes. It can use a sizing function like the pave scheme if one is defined for the surface. Future development will add hard lines to this scheme's capabilities. You specify this scheme for a surface by giving the command:

To insure that coplanar (two-tris-sharing-three-node condition) or nearly coplanar tris at sharp geometric vertices are not generated using this scheme, a corrective edge-swap is automatically done.



Without edge swap correction

With edge swap correction

TriDelaunay

Applies to: Surfaces

Summary: Automatically meshes parametric surface geometry with triangle elements.

Syntax:

```
Surface <range> Scheme TriDelaunay
```

Discussion:

The scheme TriDelaunay is a parametric meshing algorithm. It can be run in two modes. The default mode (**asp**) combines the Delaunay [Watson,81] criterion for connecting nodes into triangles with an advancing-front approach for inserting nodes into the mesh. This method maximizes the number of regular triangles in the mesh but does not guarantee the minimum angle quality of the triangles. A guaranteed quality (**gq**) mode can be used for planar surfaces (*only*). This mode refines the initial Delaunay configuration by placing points at the centroids of the worst triangles until the mesh has an acceptable density. To switch between the two modes, use the following setting command.

```
[Set] Tridelaunay point placement {gq |
guaranteed quality | asp}
```

TriDelaunay can also utilize a sizing function if one is defined for the surface.

Note: This algorithm is unstable for periodic surfaces which include a singularity point, E.G. spheres with poles, cone tips and some types of toruses. Use scheme TriMesh, TriAdvance or QTri to mesh non-parametric or periodic parametric surfaces.

TriMap

Applies to: Surfaces

Summary: Places triangle elements at some vertices, and map meshes the remaining surface.

Syntax:

```
Surface <range> Scheme Trimap
```

Related Commands:

```
Surface <range> Vertex <range> Type
{Triangle|Notriangle}
```

Discussion:

Some surfaces contain bounding curves which meet at a very acute angle. Meshing these surfaces with an all-quadrilateral mesh will result in a very skewed quad to resolve that angle. In some cases, this is a worse result than simply placing a triangular element to resolve that angle. This scheme resolves this situation by placing a triangular element in these tight corners, and filling the remainder of the surface with a mapped mesh.

The algorithm can automatically compute whether a triangular element is necessary, along with where to place that element. To override the choice of where triangular elements are used, the following command can be issued:

```
Surface <range> Vertex <range> Type
{Triangle|Notriangle}
```

TriMesh

Applies to: Surfaces

Summary: Automatically meshes surface geometry with triangle elements using the third part *meshgems* tool.

Syntax:

```
Surface <range> Scheme TriMesh [Geometry Approximation Angle <angle>] [Meshgems] [Minimum Size <value>]
```

Related Commands:

```
[Set] Trimesher Minimum Size <value>

[Set] Trimesher Surface Gradation <value>

[Set] Trimesher Volume Gradation <value>

[Set] Trimesher Geometry Sizing {ON|off}

[Set] Trimesher Clean Discrete {on|OFF}

[Set] Trimesher Discrete Composites {on|OFF}

[Set] Trimesher Split Overconstrained Edges {on|OFF}

[Set] Trimesher Ridge Angle {<value=100>}

[Set] Trimesher Anisotropic layers {on|OFF [<layers=2>]}

[Set] Trimesher Surface Proximity {on|OFF [<ratio=1>]}
```

Discussion:

The **TriMesh** scheme fills a surface of arbitrary shape with triangle elements. The **TriMesh** scheme serves as the default method for meshing the surfaces of volumes for the TetMesh scheme.

Included in Cubit is a third party software library for generating triangle meshes called MeshGems. This is a robust and fast triangle mesher developed and distributed by Distene. Figure 1 shows a CAD model where surfaces have been meshed with the **TriMesh** scheme. The triangle mesh was then used as input to the TetMesh scheme.

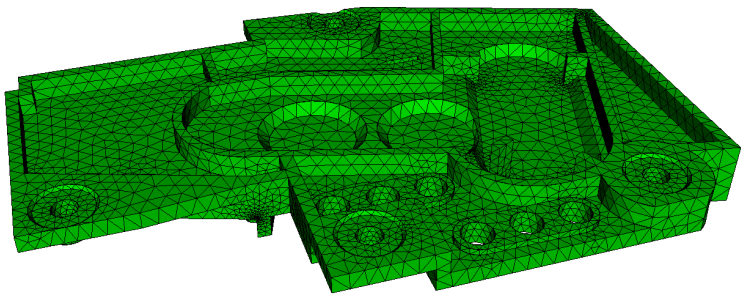


Figure 1. Triangle meshes generated with the TriMesh scheme using default settings on the surfaces of a CAD model.

The **TriMesh** scheme is usually very good at generating a mesh with its default settings. In most cases no adjustments to default settings are

necessry. Using the size assigned to the surface, either assigned explicitly or defined with an auto size, the **TriMesh** scheme will attempt to maintain the assigned size, except where features smaller than the specified size exist. In this case, smaller triangles will automatically be generated to match the feature size. The triangle mesher will then generate a smooth gradation from the small triangles used to capture features, to the size specified on the surface. This effect is shown in figure 1 where the transitions in triangle sizes can be seen. If no size is specified on the surface, it will use the size that was set on its parent volume. User defined sizes and intervals can also be assigned to individual curves for more specific control of element sizes.

Although rare, if meshing fails when using the **TriMesh** scheme, Cubit will automatically attempt to mesh the surface with the **TriDelaunay** scheme. Subsequent mesh failures will also attempt meshing with the **TriAdvance** and **QTri** schemes.

A **sizing function** can also be used with the **TriMesh** scheme to control element sizes, however the algorithm used for meshing will automatically revert to the **TriAdvance** scheme. This is because the MeshGems algorithm provides built-in capabilities for adaptively controlling the element sizes based on geometry. More details can be found in Geometry Adaptive Sizing for TriMesh and TetMesh Schemes

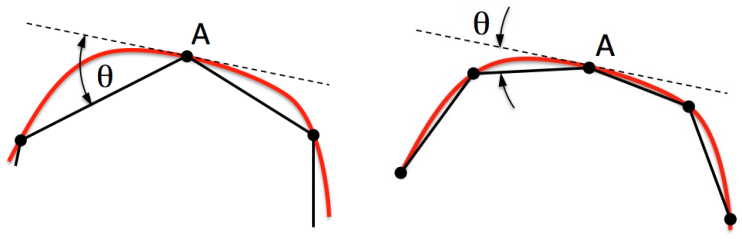
When using the **TriMesh** and **TetMesh** schemes, recommended practice is to mesh all surfaces and volumes simultaneously. This provides the greatest flexibility to the algorithms to determine feature sizes and their effect on neighboring surfaces and volumes.

TriMesh Scheme Options

The **TriMesh** options described below can be set to adjust the default behavior of the tri mesher. Scheme options are assigned independently to each surface as part of the **scheme TriMesh** command. Note that the options described here will apply only if the **TriMesh** scheme is used. **TriDelaunay** and **TriAdvance** schemes will not utilize these options when meshing.

```
Geometry Approximation Angle <angle>
```

For non-planar CAD surfaces and non-linear curves, an approximation must always be made to capture the curved features using the linear edges of the triangle. When a **geometry approximation angle** is specified, the triangle mesher will adjust triangle sizes on curved boundaries so that the linear edges of the triangle will deviate from the geometry by no greater than the specified **angle**. Figure 2 illustrates how the geometry approximation angle is determined. If the red curve representes the geometry and the black segments represent the mesh, the angle **θ** is the angle between the tangent plane at point **A** and the plane of a triangle at **A**. **θ** represents the maximum deviation from the geometry that the mesh will attempt to capture. As shown in figure 2(b), a smaller geometry approximation angle will normally result in more elements, but it will more closely approximate the actual geometry. The default approximation angle is 15 degrees.



(a) (b)

Figure 2. The geometry approximation angle θ is shown as the maximum deviation between the tangent plane at A and the plane of a triangle at A.

Note that the **geometry approximation angle** is also effective in controlling the element size on the interior of surfaces as illustrated in figure 3. This is most useful when used in conjunction with the **TetMesh** Scheme where smaller tets will be placed in regions of higher curvature.

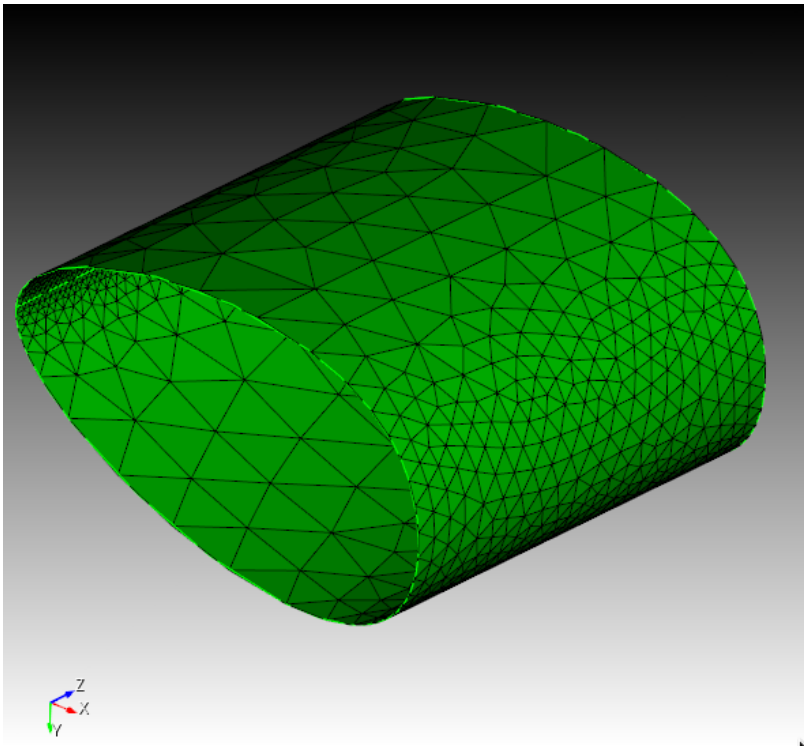


Figure 3. Demonstrates the effect of the geometry approximation angle to better capture surface curvature on the interior of surfaces.

```
Minimum Size <value>
```

By specifying a **minimum size**, the tri mesher will attempt to prevent creating elements smaller than this specified size. It should be noted that there may still be a small number of elements with a size slighly less that this value; it is not an exact setting.

The **MeshGems** option will use only the MeshGems triangle mesher on the specified surfaces. It will not revert upon failure to the **TriDelaunay** or **TriAdvance** schemes.

Global Trimesher Gradation Options

The user may set options that control the gradation of the tri-meshing algorithms. These trimesher options are global settings and apply to all trimeshes generated when the scheme is set to **TriMesh** until the option is changed by the user.

The **minimum size** setting controls the smallest edge length generated

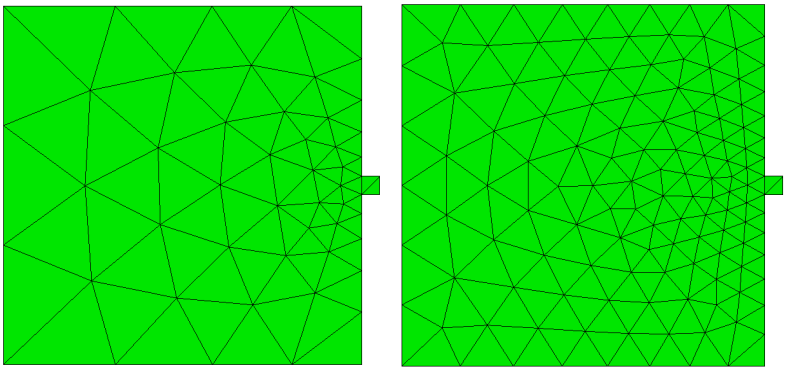
during triangle meshing.

```
[Set] Trimesher Minimum Size <value>
```

The global **gradation** options control how fast the triangle sizes can change when transitioning from small to larger sizes. For example a value of 1.5 will attempt to limit the change in element size of adjacent triangles to no greater than a factor of 1.5. Valid values for gradation should be greater than 1.0 and usually less than 2 or 3. The larger the value, the faster the transition resulting in fewer total elements. Likewise, values closer to 1.0 can result in significantly more elements, especially when small features are present. The default setting for **gradation** is 1.3. Gradation can be controlled for both surfaces and volumes.

```
[Set] Trimesher Surface Gradation <value>
```

Surface gradation will control the growth of triangles where element size has been determined by bounding curves. For example, Figure 4 shows a small feature where element sizes have been determined locally by the length of the small curves. A gradation is applied so that triangle sizes increase away from the small feature. A surface gradation of 1.3 is shown on the left, while a surface gradation of 1.1 is shown on the right.



(a) (b)

Figure 4. Demonstrates the effect of changing the default gradation, where (a) is the default gradtion of 1.3, compared with (b) using a gradation of 1.1. Note that both images use the same interval size setting for the surface.

```
[Set] Trimesher Volume Gradation <value>
```

Volume gradation will control the growth of triangles where element size has been determined by the proximity of other nearby surfaces. For example, Figure 5a and 5b shows a brick with a small void where the surface meshes are generated with the **TriMesh** scheme. The surface gradation has been adjusted to a large number so its effect is negligible. The small element size determined for the void is propagated to the exterior surfaces. The resulting gradation of the nearby triangles on the surface is determined by the **trimesh volume gradation** setting.

Note that the **trimesh volume gradation** command is different than the growth factor control setting. The **trimesh volume gradation** controls the gradation of triangles on the surface due to nearby features where small tets will exist, whereas the **volume <range> tetmesh growth_factor** command controls the gradation of the interior tet elements.

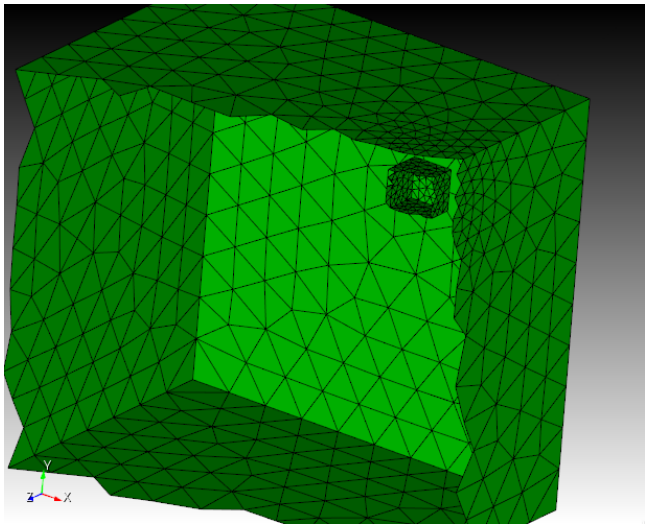


Figure 5a. An example of a cut-away mesh with a volume gradation, where the small size on the interior void propagates to the exterior surfaces

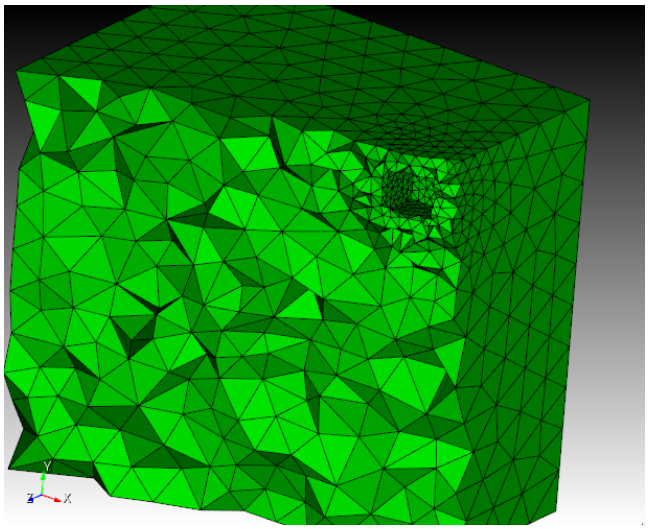


Figure 5a. An example of a cut-away mesh with a volume gradation, where the small size on the interior void propagates to the exterior surfaces

```
[Set] Trimesher Geometry Sizing {ON|off}
```

The global **Geometry Sizing** setting can be toggled on or off. If set to **on**, the element size will be influenced by the **geometry approximation angle**. If set to **off**, **geometry approximation angle** will not be involved in the computation of element size. See geometry approximtion angle for more information.

```
[Set] Trimesher Discrete Composites {on|OFF}
```

The option **Discrete Composites** forces trimeshing to convert the composite into a discrete (faceted) representation, from which the trimesher generates a mesh. The default behavior is to use the underlying geometry, if the composite has it, to generate a mesh. Using the underlying geometry is typically a more robust and reliable approach, not dependent on good faceting. However, it should be noted that composites with mesh curves will be triangle meshed with the discrete

method, as this is not supported with the default method at this time but shortly will be.

```
[Set] Trimesher Clean Discrete {on|OFF}
```

The option **Clean Discrete** performs a step to 'clean' the faceting given to the trimesher for discrete surfaces (mesh-based geometry and composites), previous to triangle meshing. The 'cleaning' is actually a meshing of the facets which typically generate a better, more optimal set of facets representing the surface. This 'clean' step uses geometry approximation, with angle of 8 degrees to generate the new facets. The option is OFF by default.

```
[Set] Trimesher Ridge Angle {<value=100>}
```

The **ridge_angle** setting is only used when meshing discrete surfaces (composites or facet/mesh-based geometry surfaces). It is the threshold for deteremining when to preserve lines (ridges) defined in the discrete surface. Lines in the discrete surface having a dihedral angle larger than **ridge_angle** will be preserved in the generated triangle mesh. In Figure 7, the composite surface has a dihedral angle of 17° at its ridge. In the first image **ridge_angle** is set to less than 17° so the ridge is preserved. In the second image **ridge_angle** is set to greater than 17° so the ridge is not preserved.

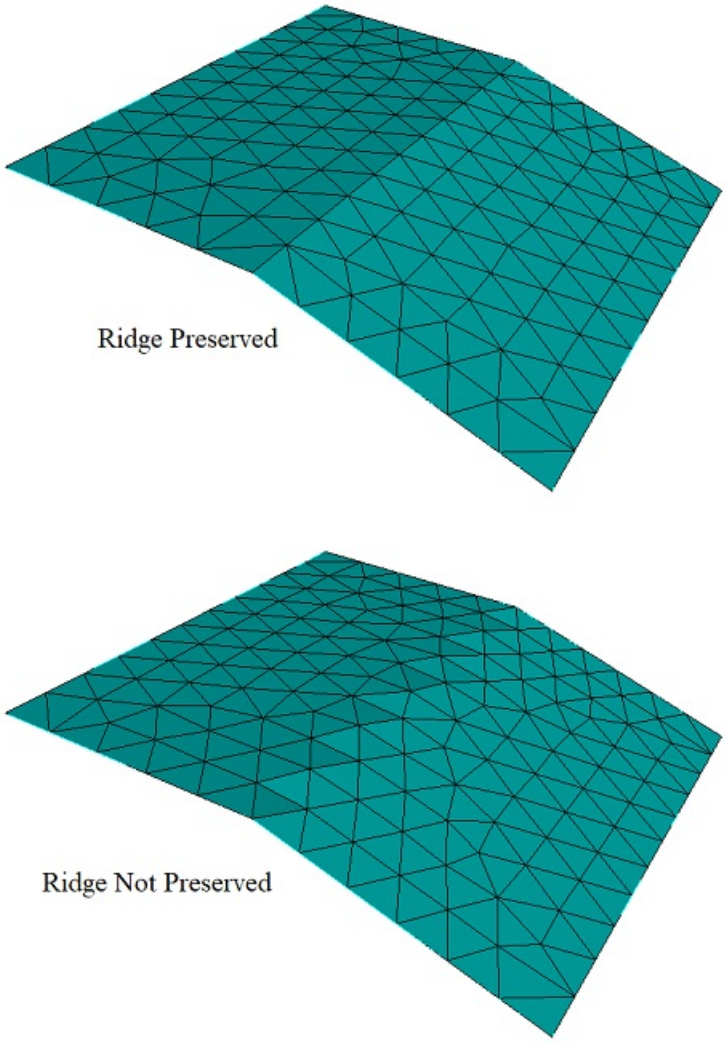


Figure 6. Ridge Angle Setting

```
[Set] Trimesher Split Overconstrained Edges
```

```
{on|OFF}
```

The global **Split Overconstrained Edges**, if set to **on**, splits edges owned by the surface, but with both nodes on curves. This feature can help when two elements through the thickness of the mesh is desired. Figure 7 shows the effect of this option.

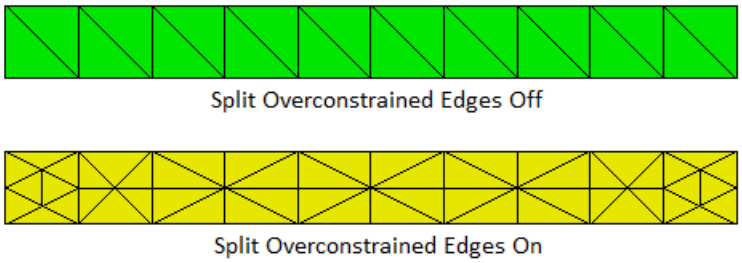


Figure 7. Split overconstrained edges

```
[Set] Trimesher Anisotropic Layers {on|OFF
[<layers=2>]}
```

The **Anisotropic Layers** setting attempts to place the specified number of layers triangle through thin regions of the surface while respecting the surface mesh size in the thick direction. The default number of layers is two. This option is currently under development and can sometimes generate ill-formed triangles. The number of layers generated can sometimes exceed the number of layers specified..

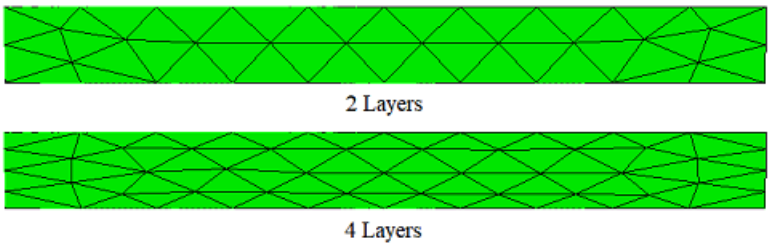


Figure 8. Anisotropic Surface Meshing

```
[Set] Trimesher Surface Proximity {on|OFF
[<ratio=2>]}
```

The **Surface Proximity** setting will add refinment to thin regions of surfaces. The ratio option can be used to multiply a scale factor to a size computed from proximity. By default, Surface Proximity is not enabled and if enabled, the default ratio is 1.0.

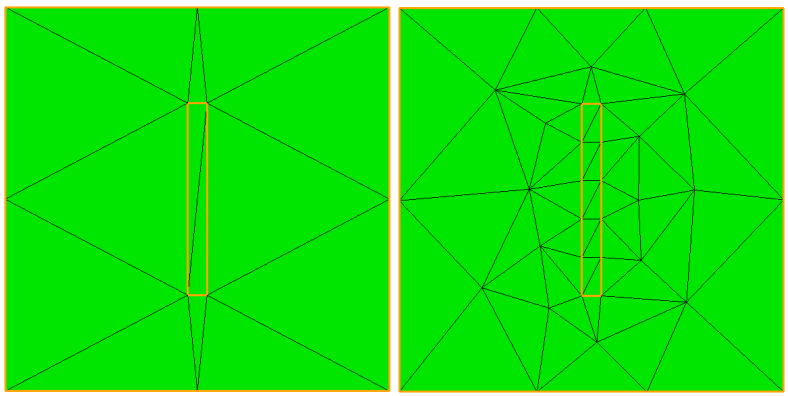


Figure 9. Surface Proximity disabled on the left, and enabled on the right

TriPave

Applies to: Surface

Summary: Places triangle elements at some vertices, and paves the remaining surface.

Syntax:

```
Surface <range> Scheme Tripave
```

Related Commands:

```
Surface <range> Vertex <range> Type
{triangle|notriangle}
```

Discussion:

Similar to the trimap algorithm, but uses paving instead of mapping to fill the remainder of the surface with quadrilaterals.

The algorithm can automatically compute whether a triangular element is necessary, along with where to place that element. To override the choice of where triangular elements are used, the following command can be issued:

```
Surface <range> Vertex <range> Type
{triangle|notriangle}
```

TriPrimitive

Applies to: Surfaces

Summary: Produces a triangle-primitive mesh for a surface with three logical sides

Syntax:

```
Surface <range> Scheme Triprimitive [SMOOTH |
nosmoothing]
```

Discussion:

The triprimitive scheme indicates that the region should be meshed as a triangle. A surface may use the triprimitive scheme if three "natural", or obvious, corners of the surface can be identified. For instance, the surface of a sphere octant (shown in the figure below) is handled nicely by the triprimitive scheme. The algorithm requires that there be at least 6 intervals (2 per side) specified on the curves representing the perimeter of the surface and that the sum of the intervals on any two of the triangle's sides be at least two greater than the number of intervals on the remaining side. The following figure illustrates a triprimitive mesh on a 3D surface.

By default, the triprimitive algorithm will smooth the mesh with an iterative smoothing scheme. This smoothing can be disabled by using the "nosmoothing" option with this command. The quality of the mesh will often be significantly degraded by disabling smoothing, but in certain cases the unsmoothed mesh may be preferred.

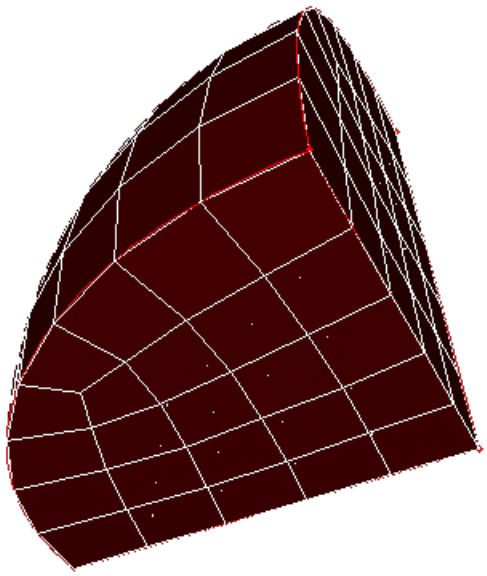


Figure 1. Surfaces meshed with scheme Triprimitive

pCamal

pCamal is an application written and maintained by the Cubit development team. It is designed to work in a distributed computing environment to generate 3D hex elements of a sweep mesh. It first uses the serial Cubit application to generate the 2D quad elements. These elements are written to a file that can then be used by pCamal to generate the most time consuming and memory intensive portion of the mesh: the 3D hex elements. The following describes how to set up the necessary inputs to pCamal using Cubit's sweeping command.

To set up for pCamal, first use the **parallel meshing** setting:

```
Set Parallel Meshing {on|OFF}
```

You would then use the sweep scheme and mesh your 3D volumes as normal. When Cubit performs the mesh operation on a volume that has a **sweep** scheme applied when the **parallel meshing** option is **ON**, only the surface entities will be meshed, leaving the hex elements for pCamal. Surfaces will be meshed with appropriate source, target and linking surface designations.

Exporting a Parallel Mesh for pCAMAL

The following command can be used for exporting a mesh in exodus format for use with pCAMAL

```
Export Parallel "<filename>" [Block <id_list>]
[Overwrite] [Processor <number>]
```

The options are the same as those for the **export genesis** command except for the addition of the processor option.

The processor option allows the user to specify the number of processors that will be used to mesh the volume with the pCAMAL option. This same option exists in the pCAMAL application and is more often used there since the number of available processors is known then rather than when the output file is created in Cubit.

If the processor option is given, Cubit attempts to balance the number of sweepable volumes to run on *N* processors by converting many-to-one sweeps to one-to-one sweeps, subdividing the sweep volume along its sweep direction, or partitioning the source surface of a one-to-one sweep if the number of source quads is much larger than the number of layers.

To determine if you are currently in parallel meshing mode you may list the current status using the List Parallel command.

```
List Parallel Meshing
```

Note: pCamal is not currently distributed with the current release of Cubit. Contact the Cubit developers if you are interested in obtaining a copy of the executable for linux operating systems.

Sculpt

Sculpt is a separate parallel application designed to generate all-hex meshes on complex geometries with little or no user interaction. Sculpt was developed as a separate application so that it can be run independently from Cubit on high performance computing platforms. It was also designed as a separable software library so it can be easily integrated as an in-situ meshing solution within other codes. Cubit provides a front end command line and GUI for the Sculpt application. The command will build the appropriate input files based on the current geometry and can also automatically invoke Sculpt to generate the mesh and bring the mesh back to Cubit.

- Preparing to Use Sculpt
- Sculpt Command
- Controlling the Execution of Sculpt
- Sculpt Help Command
- Sculpt Path Command
- Sculpt Examples
- Sculpt Technical Description
- Sculpt Application Documentation

Preparing to Use Sculpt

Platforms

Sculpt is available for Windows, Mac and Linux operating systems.

Sculpt Installation

Sculpt is a stand-alone executable, separate from Cubit. In order for Cubit to start up Sculpt, it must be on your system and accessible to Cubit. The default installation of Cubit should install files in the correct locations for this to occur. Check with Cubit support if it did not come with your installation or you are not able to locate it or any of its supporting applications.

To run Sculpt from Cubit, four executable files are needed:

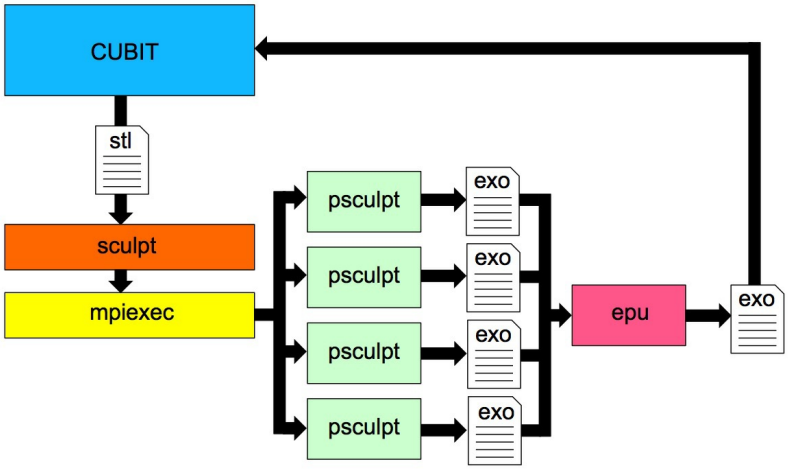
- **sculpt:** Application that controls start-up of **mpiexec** and **psculpt**. Main entry point from Cubit, that checks for the existence and compatibility of either the system **mpiexec** application or will use a local cubit instalation of **mpiexec**.
- **psculpt:** The main mpi-based Sculpt application. Requires **mpiexec** to run.
- **mpiexec:** Standard application available on most linux-based operating systems for starting up mpi-based applications on multiple processors. This should be available with your Cubit installation, but is also available from open-mpi.org
- **epu:** Used for combining multiple exodus files, generated with Sculpt, into a single exodus file. This executable is optional, but is useful for importing the resulting mesh into Cubit for viewing. It is part of the SEACAS tool suite developed by Sandia National Laboratories and is also included with your Cubit installation. It can also be obtained in open source form from sourceforge.net.

To view the current path to these executables that Cubit will use, issue the following command from the Cubit command window

```
Sculpt Path List
```

See the **Sculpt Path** Command for more info on setting and customizing these paths.

The following image illustrates the process flow when the **sculpt** command is used in Cubit.



For the Sculpt meshing process, a set of files, including a facet-based stl file are written to disk. The **sculpt** application is then started up which in turn invokes **mpiexec** to start up multiple instances of **psculpt** in parallel. **psculpt** then performs the meshing and writes one exodus file per processor. These files can then be combined using **epu** and then imported back into Cubit for viewing.

Setting your Working Directory

When using the **Sculpt** command in Cubit, several temporary files will be written to the current working directory. Because of this, it is important to set your working directory before using Sculpt to a desired location where you want these files placed.

Sculpt Command

The **Sculpt Command** in Cubit invokes the sculpt application. You can designate the target geometry, which can be existing volumes or blocks within the current Cubit run or imported from external files (e.g., STL, diatom, or microstructure files). The command syntax for preparing a model for Sculpt is as follows:

```
Sculpt [parallel]

Geometry Options

{[volume <ids>] [block <ids>] [stl_file "
<filename.stl>"] [diatom_file "
<filename.diatom>"] [input_vfrac "
<filename.e>"] [input_micro "<filename.tec>"]
[input_cart_exo "<filename.e>"] [input_spn "
<filename.spn>"]}

[spn_xyz_order "<0,1,..5>"] [input_stitch "<filename.st>"]
[stitch_field "<field_name>"] [stitch_timestep
{<timestep_value>|first|last}] [stitch_timestep_id
<timestep_ID>] [stitch_info]

Process Options

[processors <value>] [fileroot "
<rootfilename>"] [{OVERWRITE|no_overwrite}]
[absolute_path] [{EXECUTE|no_execute}]
[{COMBINE|no_combine}] [{IMPORT
[unique_genesis_ids]|no_import}]
[{SHOW|no_show}] [{CLEAN|no_clean}]
[{gen_input_file <string>|no_gen_input_file}]
[{debug <value>] [quiet] [preview]
```

Grid Options

```
[{size <value>|autosize <value>}] [box
{align|location <options>|expand <value>}]
[{xintervals|nelx} <value> {yintervals|nely}
<value> {zintervals|nelz} <value>] [input_mesh
"<filename.g>"] [input_mesh_material <value>]
[input_mesh_pamgen "<filename.pam>"]
```

Mesh Options

```
[void <value>] [void_block <value>]
[separate_void_blocks] [stair <value>] [htet
<value>] [htet_material <value>] [htet_method
<value>] [periodic <value>]
```

Smoothing Options

```
[smooth <value>] [csmooth <value>]
[num_laplace <value>] [max_opt_iters <value>]
[opt_threshold <value>] [curve_opt_thresh
<value>] [max_pcol_iters <value>]
[pcol_threshold <value>] [max_gq_iters
<value>] [gq_threshold <value>] [max_deg_iters
<value>] [deg_threshold <value>]
[geo_smooth_max_deviation <value>]
```

Improve Options

```
[pillow <value>] [pillow_surfaces]
[pillow_curves] [pillow_curve_layers <value>]
[pillow_curve_thresh <value>]
[pillow_boundaries] [pillow_smooth_off]
[defeature <value>] [min_vol_cells <value>]
[defeature_bbox] [defeature_iters <value>]
[capture <value>] [capture_angle <value>]
[capture_side <value>] [thicken_material
<value>...] [thicken_void <value>] [remove_bad
<value>] [wear_method <value>]
[crack_min_elem_thickness <value>]
[temp_use_sipe_depth <value>] [min_num_layers
<value>]
```

Adapt Options

```
[adapt_type <value>] [adapt_threshold <value>]
[adapt_levels <value>] [adapt_material
<value>...] [adapt_export]
[adapt_non_manifold] [adapt_load_balance]
```

Boundary Condition Options

```
[gen_sidesets <value>] [material_name
<value>... <string>...] [sideset_name
<value>... <string>...] [nodeset_name
<value>... <string>...] [sideset_definition
<value>... <string>...] [nodeset_definition
<value>... <string>...] [free_surface_sideset
<value>...] [match_sidesets <value>...]
[match_ss_nodeset <value>...]
```

Output Options

```
[exodus <string>] [large_exodus] [xtranslate
<value>] [ytranslate <value>] [ztranslate
<value>] [xscale <value>] [yscale <value>]
[zscale <value>] [volfrac_file <string>]
[quality <string>] [write_geom] [write_mbg]
[compare_volume] [compute_ss_stats]
```

Controlling the Execution of Sculpt in Cubit

The following command options can be used to control the execution of Sculpt from within Cubit when used with the **sculpt** command. Follow the links above for other options that control he behavior of the sculpt algorithms and methods.

volume <ids> | block <ids>

List of volumes or blocks to include in the mesh. One file containing a faceted representation (STL) per volume will be generated and saved in the current working directory to be used as input to Sculpt. Each volume will be treated as a separate material within sculpt and a conforming mesh will be generated where volumes touch. If the Block command is used, one file per block will be used. Each block represents a separate material in Sculpt.

fileroot '<root filename>'

Root of file names for output. When the **sculpt** command is executed, Cubit will generate multiple files in the working directory used for input to the Sculpt application. The '<root filename>' will be used as the basis for naming these files.

processors <value>

Specify the number of processors that MPI will use to execute the Sculpt application. If not specified, the maximum number of available processors on the local machine will be used up to a maximum of 16.

OVERWRITE | no_overwrite

By default, Cubit will overwrite an existing set of files with the same '<root filename>'. To over-ride, use the **no_overwrite** option.

absolute_path

By default, Cubit will write the relative path names of files used in the .run and .diatom files. To force absolute path names to be written, use the **absolute_path** option

EXECUTE | no_execute

By default, Cubit will attempt to run sculpt in parallel on the machine Cubit is currently running on. To generate just the required input to run Sculpt at a later time or on another machine, use this option. A file of the form <root filename>.run will be generated in the current working directory. (for example "model.run"). Executing the .run file from the linux command line should run sculpt in parallel. It can also be used to run sculpt on a cluster where a Cubit executable may not be available.

size <value> | autosize <value>

autosizesizeThe option is the absolute cell size for the Cartesian grid and is the same as the cell_size option in sculpt. The option is a value between 0 and 10. It represents a model independent size where 1 is the small size and 10 is large. This is the same scaling factor used in Cubit's auto sizing but is divided by ten. A size value will be computed from the selected autosize and used as the absolute cell size for the base Cartesian grid.

box location <options>

When using a Cartesian grid as the overlay grid definition, the **location** options define the bounds of the Cartesian grid. The first Location <option> defines the minimum Cartesian coordinate of the grid and the second, the maximum. The <options> can be any valid method for defining a coordinate location in cubit. In most cases the **position** option can be used. The default is computed as an enclosing bounding box with 2.5 additional cells on each side.

Note that the **xmin, ymin, zmin, xmax, ymax, zmax** options can also

be used for specifying the bounds of the Cartesian grid from the command line.

COMBINE | no_combine

If the **no_combine** option is used, following execution of Sculpt, the resulting exodus meshes will not be combined using the **epu seacas** tool. Otherwise the default will automatically combine the meshes generated by each processor into a single mesh. Note that epu should be installed on your system and the path to epu defined using the **sculpt path** command. Epu is a code developed by Sandia National Laboratories and is part of the SEACAS tool suite. It combines multiple Exodus databases produced by a parallel application into a single Exodus database. The epu program should be included with distributions of Cubit beginning with Version 15.0.

IMPORT | no_import

no_import If the option is used, following execution of Sculpt, the result will be not be imported into Cubit as a free mesh. The default **IMPORT** option will automatically import the mesh that was generated in Sculpt. If the **no_combine** option has been used, then multiple free meshes will be imported with duplicate nodes and faces at processor domain boundaries. Otherwise a single free mesh, the result of the **epu** code, will be imported. Note that the resulting mesh will not be associated with the original geometry, however Block (material) definitions will be maintained. In addition, a separate group will be generated for each imported mesh (One per processor). The default will automatically import the mesh following mesh generation in Sculpt.

SHOW | no_show

If the **no_show** option is used, while the external Sculpt process is running, no ouput from the Sculpt application will be displayed to the command window. Otherwise, the default **SHOW** is used and output from the Sculpt application will be echoed to the Cubit command window. This option is only effective if the **no_execute** is not used.

CLEAN | no_clean

no_cleanCLEANsculptclean If the option is used, temporary files generated during the command will be deleted. This includes any exodus mesh files, .stl, .diatom, .log and .run files. The default for this option is , therefore, use the option to keep any temporary files generated as part of the current Sculpt run.

gen_input_file <file name> | no_gen_input_file

An input file with the given file name will be generated when the command is executed. This is a text file containing all sculpt options used in the command. The input file is intended to be used for batch execution of sculpt. To run sculpt from the operating system command line you would use the **-i** option. For example: **sculpt -i myinputfile.i -j 4** where **myinputfile.i** is the name of the input file specified with the **gen_input_file** option and **-j 4** is the number of processors to use.

debug <value> The debug option is used only as a developer debugging tool. It will set the debug processor and sleep upon execution to allow a debugger to be attached to the process.

quiet

The **quiet** option used as an option in the **sculpt** command will suppress output from the sculpt application to the Cubit output window.

preview

When used with the **sculpt** command, the **preview** option will print the contents of the sculpt input file to the Cubit output window, based on the

currently defined options in the **sculpt** command. It will not execute sculpt.

Sculpt Help Command

Help about any of the Sculpt options can be printed to the output window using the following command:

```
Sculpt [parallel] print_help [<"sculpt_option">]
```

where <"sculpt_option"> is any valid sculpt application option listed above.

Sculpt Path Command

The command for letting Cubit know where the Sculpt and related applications are located is:

```
  Sculpt [Parallel] Path
 [List|Psculpt|Epu|Mpiexec]
```

This command defines the path to **psculpt**, **epu** and **mpiexec** on your system. In most cases, however, these paths should be automatically set provided Sculpt was successfully installed with your Cubit installation. The **list** option will list the current paths that Cubit will use for these tools. If an alternate path to these executables is desired, it is recommended that this command be used in the .cubit initialization file so that it wont be necessary to define these parameters every time Cubit is run.

Sculpt Mesh Quality Control

In most cases, the Sculpt tool can be used without adjusting default values. Depending on the characteristics of the geometry to be meshed, the default values may not yield adequate mesh quality. Upon completion, Sculpt reports to the command line, a summary of the mesh that was generated. This includes a summary of the mesh quality. Care should be taken to review this summary to ensure the minimum mesh quality is in a range suitable for analysis.

The element metric used for computing mesh quality in Sculpt is the Scaled Jacobian. This is a value between -1 and 1 that is a relative measure of the angles at the element's nodes. A value of 1 indicates a perfect 90 degree angle between each of its edges. In most cases a value less than zero, or negtive Jacobian element, indicates an unusable mesh. Sculpt's default settings try to achieve a minimum Scaled Jacobian of 0.2, which is normally usable in most analysis. The following discussion provides several options for adjusting the model or Sculpt parameters to help improve mesh quality.

1. **Locating poor mesh quality:** When the Sculpt mesh has been imported back into CUBIT it is a good idea to display the element quality. You can do this with variations of the following commands:

   ```
     quality hex all scaled jacobian
   quality hex all draw mesh
   ```

   Identify regions where hexes are poor quality and zoom in to these regions.
2. **Modifying the geometry:** Zooming in to poor quality elements may reveal that the mesh does not adequately represent the underlying geometry. In some cases you may find that small features, or small gaps between parts may be on the order of the size of the Sculpt cell size. If these features are not important to the analysis, you may consider using Cubit's geometry modification tools to remove features or close small gaps.
3. **Modifying the cell size:** In cases where small geometric features or gaps are important to the simulation, it may be necessary to use

a smaller base cell size. Use the size or autosize input parameters or increase the numbers of intervals. Normally to adequately capture a feature you would want the cell size to be no greater than about 1/3 to 1/2 the size of the smallest feature you would want to represent in the simulation.

4. **Turning on Pillowing for multiple materials:** For models that have more than one material that share an interface, unless the geometry is precisely aligned with the global axis, it is usually a good idea to turn on pillowing. Pillowing automatically inserts an additional layer of hexes at interface boundaries to improve mesh quality. Without pillowing may notice inverted or poor quality elements at curve interfaces where 2 or more materials meet.
5. **Modifying smoothing parameters:** Sculpt includes a tiered approach to smoothing to improve element quality. It starts by applying smoothing to all nodes in the mesh and progressively restricts the smoothing operations to only those nodes that fall below a user-defined scaled Jacobian threshold. Default numbers of iterations and thresholds for each smoothing phase have been tuned for general use, however it may be worthwhile to adjust these parameters. The three smoothing phases include:
    - **Laplacian Smoothing:** Applied to all elements. Very inexpensive fast approach to improve quality, but can result in degraded element quality if applied to excess. A fixed default of 2 iterations is applied to all hexes. Increasing the num_laplace parameter can improve some cases, especially convex shapes
    - **Optimization Smoothing:** Applied only to elements who's scaled Jacobian falls below the opt_threshold parameter (default 0.6) and their surrounding elements. This approach uses a more expensive optimization technique to improve regions of elements simultaneously. The max_opt_iters parameter can control the maximum number of iterations applied (default is 5). Iterations will terminate, however, if no further improvement is detected. Because this method optimizes node locations simultaneously, neighboring nodes with competing optimum can sometimes limit mesh quality.
    - **Spot Optimization:** Also known as *parallel coloring*, is applied only to elements who's element quality falls below the pcol_threshold parameter (default 0.2). This technique is the most expensive of the techniques, but focusses only on nodes that are immediately adjacent to poor quality hexes. Each node is smoothed independently of its neighbors, and may require a high number of iterations using the max_pcol_iters to achieve desired results. Increasing the pcol_threshold and max_pcol_iters may yield improved results.

   Observing the mesh quality output to the command line following each smoothing iteration can provide some insight on the effect of modifying smoothing parameters.
6. **Creating degenerate hexes:** Some geometries will not permit a usable mesh with a traditional all-hex mesh. Sculpt includes the option to automatically and selectively collapse element edges to improve low-quality elements. The max_deg_iters and the deg_threshold values are used to control the creation of degenerates. Degenerate elements are treated as standard hex elements, but use repeated nodes in the eight-node connectivity array.
7. **Creating hex-dominant mesh** Another option for avoiding mesh quality issues is to generate a few tet elements in the mesh using the htet option. With this option you can specify a scaled Jacobian threshold value below which hexes will be converted to tet elements. The interface between hex and tet elements is managed by an automatically defined set of nodesets and sidesets that describe where multi-point constraints will be applied.
8. **Defeaturing** The defeature option does an initial filter on the cells of the base grid and attempts to reassign the material ID for cells that meet certain criteria. These are cases where a small grouping

of cells form a small volume, or where protrusions exist that would otherwise be difficult or impossible to mesh with good quality elements. By reassigning the cells in these locations, in many cases it will allow the mesh to be acceptable. This operation may result in small changes to the boundary or surface definitions, however usually small enough to still be a reasonable approximation.

Sculpt Examples

- Basic Sculpt
- Size and Bounding Box
- Meshing the Void
- Automatic Sideset Definition
- Running Sculpt Stand-Alone
- Meshing Multiple Materials With Sculpt

The following examples use this simple geometry. Execute these commands prior to performing the example **Sculpt** command line operations

```
sphere rad 1
sphere rad 1
vol 2 mov x 2
cylinder rad 1 height 2
vol 3 rota 90 about y
vol 3 mov x 1
unite vol all
```

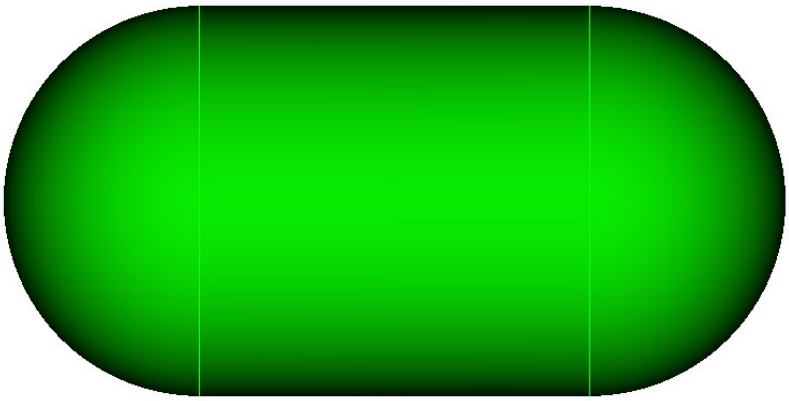


Figure 1. Geometry created from the above commands and used for the following examples.

Basic Sculpt

This example illustrates use of Sculpt with all default options. So that we can view the result, we will also use the **overwrite**, **combine** and **import** options.

```
sculpt volume 1
draw block all
```

The result of this operation is shown in Figure 2. For this example, behind the scenes, Cubit built an input file for Sculpt, ran it on 4 processors, combined the resulting 4 meshes, and subsequently imported the resulting mesh into Cubit. Note that Volume 1 remains "unmeshed" and we have created a free mesh that is not associated with a volume. The result of any Sculpt command is always an unassociated free mesh.

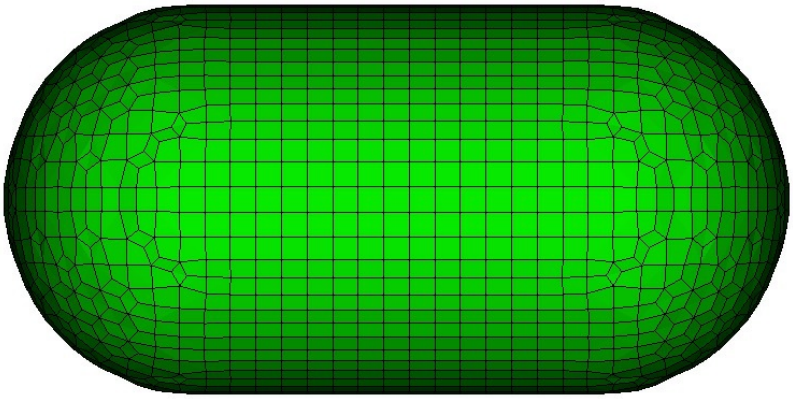


Figure 2. Free mesh generated from sculpt command

Size and Bounding Box

This example illustrates the use of the **size** and **box** options

```
delete mesh
sculpt volume 1 size 0.1 box location position
-1.5 0 -1.5 location position 1 1.5 0
draw block all
```

In this case we have used the **size** option to define the base cell size for the grid. We have also used the **box** option to define a bounding box in which the mesh will be generated. Any geometry falling outside of the bounding box is ignored by Sculpt. Figure 3 shows the mesh generated with this command.

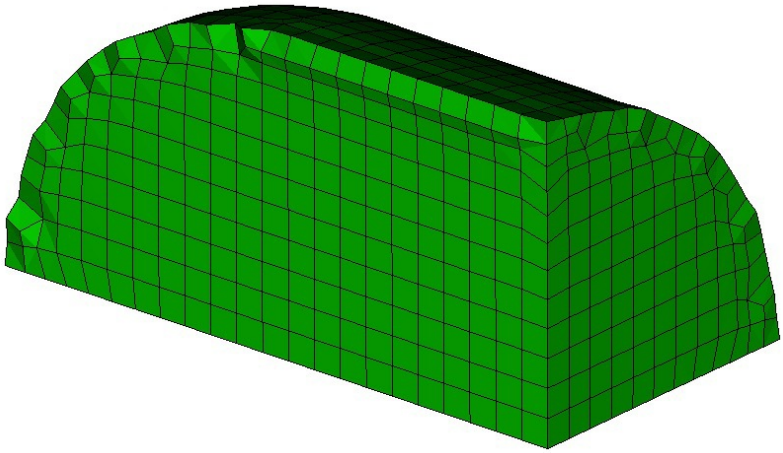


Figure 3. Sculpt "box" option limits the extent of the generated mesh.

Meshing the Void

In this example we illustrate the use of the **void** option:

```
delete mesh
sculpt volume 1 size 0.1 box location position
-1.5 0 -1.5 location position 1 1.5 0 void 1
draw block all
```

The result is shown in figure 4. Notice that this example is precisely the same as the last with the exception of the addition of the **void** option. Mesh is generated in the space surrounding the volume out to the extent of the bounding box. In this case, an additional material block is defined and automatically assigned an ID of 2. The nodes and element faces at the interface between the two blocks are shared between the two materials.

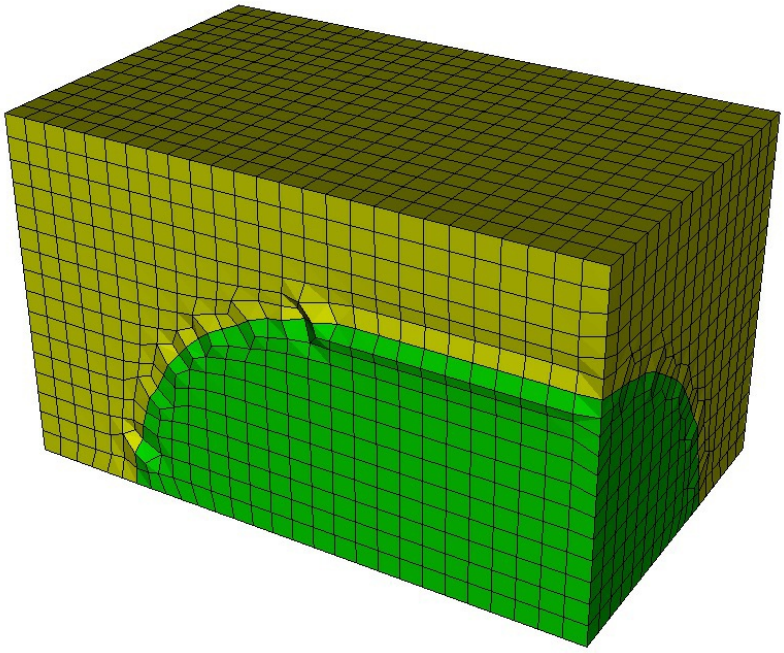


Figure 4. Sculpt "void" operation generates mesh outside the volume.

Automatic Sideset Definition

In this example we illustrate the use of the **gen_sidesets** option.

Generating sidesets on the free mesh with Cubit: Sideset boundary conditions can be manually created on the resulting free mesh from Sculpt using the standard **Sideset <sideset_id> Face <id_range>** syntax. The **Group Seed** command is also useful in grouping faces based on a feature angle to be used in a single sideset.

Generating sidesets in Sculpt: Sculpt also provides several options for defining sidesets as part of the Sculpt run. The following illustrates one option:

```
delete mesh
sculpt volume 1 size 0.1 box location position
-1.5 0 -1.5 location position 1 1.5 0 void 1
gen_sidesets 2
list sideset all
draw sideset all
```

Once again we use the same syntax but add the **gen_sidesets 2** option to automatically generate a series of sidesets. The **list** command should reveal that 10 sidesets were defined for this example with IDs 1 to 10. Figure 5 shows the result of the **draw** command showing all of the sidesets in different colors. Note that for the **gen_sidesets 2** option, sidesets are created with the following criteria:

- Interfaces between materials
- Exterior surfaces
- Surfaces at the domain boundary

See the gen_sidesets option above for a description of other options for generating sidesets in Sculpt.

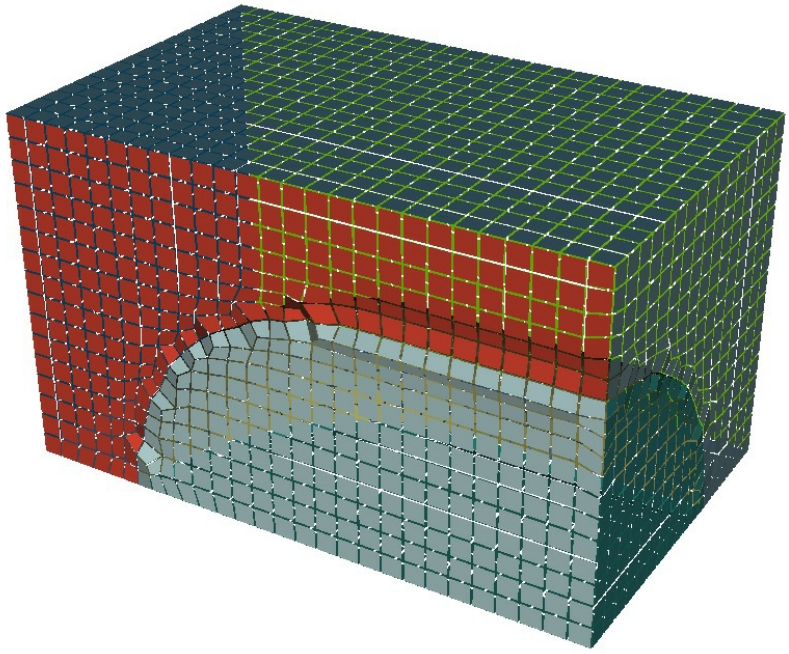


Figure 5. Automatic sidesets created using Sculpt

Running Sculpt Stand-Alone

This example illustrates how to set up the files necessary to run Sculpt as a stand-alone process. This can be done on the same desktop machine or moved to a larger cluster machine more suited for parallel processing.

Begin by setting your working directory to a location that is convenient for placing example files

```
cd "path/to/my/sculpt/examples"
```

Next we issue the basic **sculpt** command to mesh the volume

```
delete mesh
sculpt volume 1 processors 8 fileroot "bean"
over no_execute no_clean
```

In this case, we used the **no_execute** option which does not invoke the Sculpt application. Instead it will write a series of files to the working directory. The **fileroot** option defines the base file name for the files that will be written; in this case **"bean"**. We also use the **processors** option to set the number of processors to be used to 8. Finally, since the default **clean** option will remove temporary files after execution of sculpt, we use the **no_clean** option to ensure they will persist.

To see the files that Cubit placed in the working directory, bring up a terminal window on your desktop and change directories to the current working directory (ie. **cd path/to/my/sculpt/examples**). A directory listing should reveal 3 files as shown in Figure 6.

```
examples — bash — 82×6
sjowen@sajn2009-137:~/sculpt/examples$ ls -l
total 1408
-rw-r--r--  1 sjowen  SANDIA\Domain Users     156 Nov  7 11:21 bean.diatom
-rwxr-xr-x  1 sjowen  SANDIA\Domain Users     318 Nov  7 11:21 bean.run
-rw-r--r--  1 sjowen  SANDIA\Domain Users  711256 Nov  7 11:21 bean_Volume_1.stl
sjowen@sajn2009-137:~/sculpt/examples$
```

Figure 6. Directory listing of files written from Cubit

The following describes the purpose of each of the resulting files:

- **bean.diatom:** *Diatoms* is a file format used by Sandia's CTH and Alegra analysis programs that includes a rich constructive solid geometry definition. A series of directives for constructing and

orienting primitives to build a complete solid model can be used. Included in the Diatom description is an STL import option. While any standard Diatom description may be used as input to Sculpt, for Cubit's purposes, only the STL option is used. This file contains a listing of all STL files that will be used as input to Sculpt.

- **bean.run:** The *.run* file contains the unix command line for running sculpt. Note that the file permissions have been set to execute to allow this file to be used as a unix script. Figure 7 shows the .run file for this example. Note that the command uses **mpiexec** and the **psculpt** executables, along with their full path. These paths may need to be edited when running on a different machine. It also includes the default parameters for setting the sizes, bounding box and smoothing parameters that have been computed by Cubit.

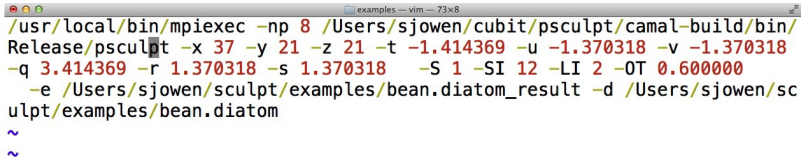


Figure 7. Unix command line for running Sculpt generated by Cubit

- **bean_Volume_1.stl:** The STL file is a copy of the geometric model. In our case, it is a representation of the cylinder and sphere object we have been working with. The STL format is a set of triangles that describe the surfaces of the object. One STL file will be generated for each Volume. If the Block option is used, then one file for each Block would be created.

To run sculpt on the same machine, from the terminal window in your current working directory you would issue the following command:

```
./bean.run
```

If Sculpt is to be run on a different machine, copy the files in the working directory to the other machine and issue the same command. Remember to change the path to the **mpiexec** and **psculpt** executables to match those on the new machine. For running on cluster machines that have scheduling of resources, check with your system administrator for how to submit a job for running.

After running Sculpt, Figure 8 shows the resulting files that would be written to the current working directory.

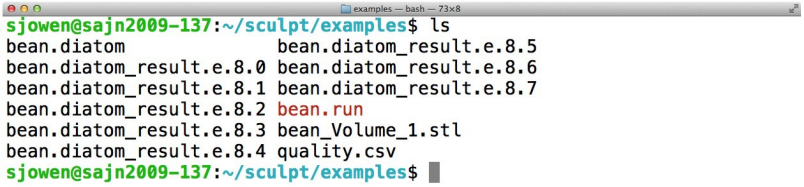


Figure 8. 8 Exodus files were generated and placed in working directory

Note that 8 exodus files have been generated, 1 from each processor. These files can be used by themselves or used as-is for use in a simlation, or they can be combined into a single file. The exodus files produced by Sculpt include all appropriate parallel communication information as defined by the **Nemesis** format. Nemesis is an extension of Sandia's Exodus II format that also includes appropriate parallel communication information.

To combine the resulting exodus files into a single file, we can use the **epu** tool. Epu should be included in your Cubit distribution, but may require you to set up appropriate paths for it to be recognized. To run epu on this model, use the following command from a unix terminal window:

```
epu -p 8 bean.diatom_result
```

The result should be a single file with the name **bean.diatom_result.e**. The mesh in this file can then be imported into Cubit. Switch back to your Cubit application and from the command line type the following command:

```
import mesh "bean.diatom_result.e" no_geom
```

The result should be the same mesh we generated previously that is shown in Figure 2.

Meshing Multiple Materials With Sculpt

This example illustrates using Sculpt to mesh models with multiple materials. To begin with, we will modify our current model by adding some additional volumes. Use the following commands:

```
delete mesh
cylinder rad 0.5 height 3
cylinder rad 0.5 height 3
vol 5 mov x 2
```

The resulting geometry should look like the image in Figure 9.

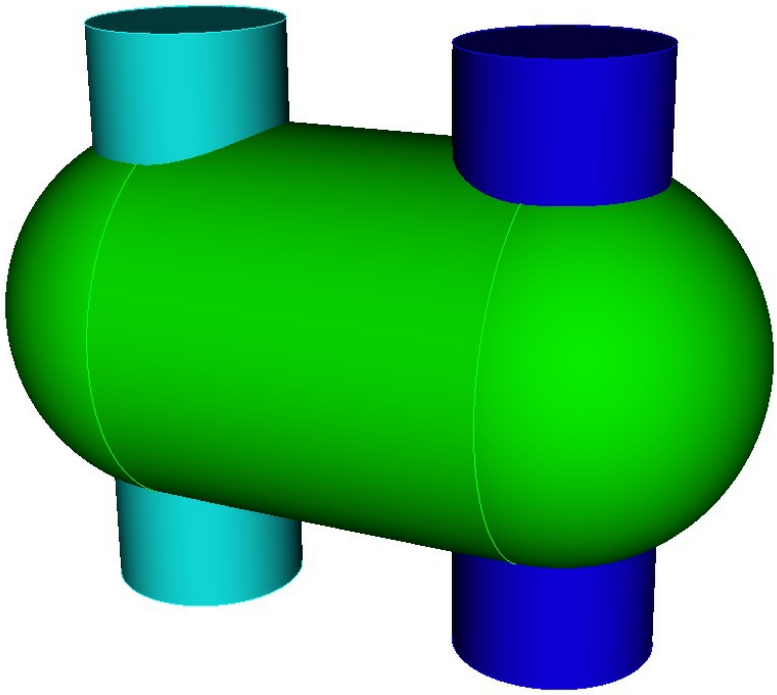


Figure 9. Geometry used to demonstrate multiple materials with Sculpt

Use this geometry to generate a mesh using Sculpt.

```
sculpt volume all size 0.075
draw block all
```

The resulting mesh should look like the image in Figure 10.

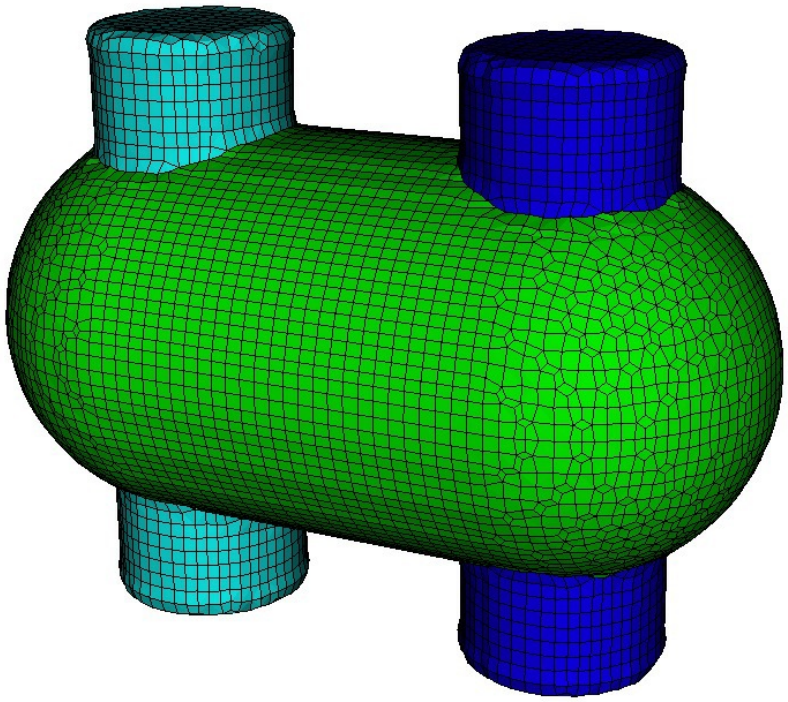


Figure 10. Mesh generated on multiple materials

Notice that one mesh block per volume was created. We should also note that no boolean operations were performed prior to building the mesh with Sculpt. In fact, volumes 4 and 5 were significantly overlapping volume 1. This would be an invalid condition for normal Cubit meshing operations. Figure 11 shows a cut-away image of the mesh using the Clipping Plane tool.

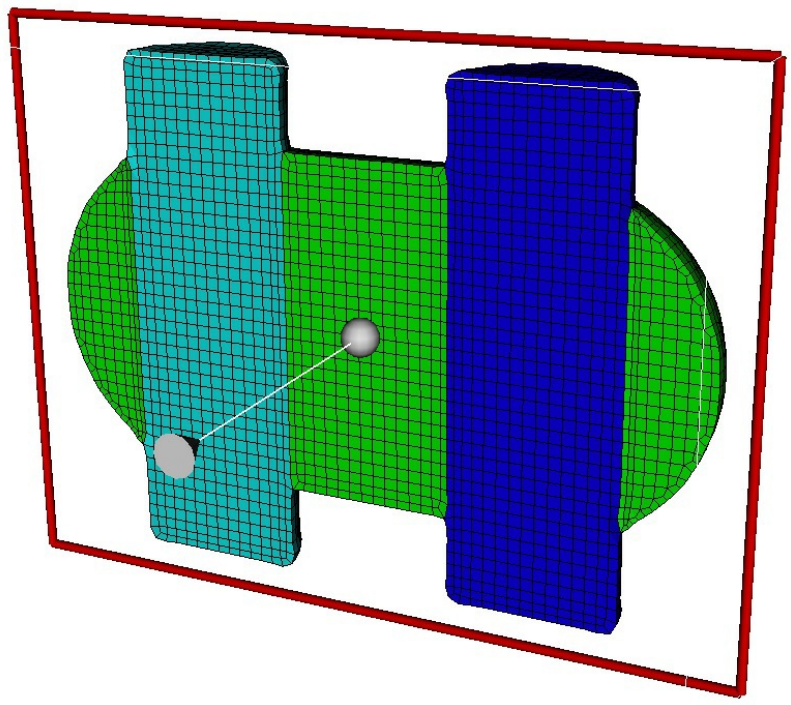


Figure 11. Cut-away of mesh generated on multiple materials

We should also note that imprint/merge operations typically needed, were also not required. While it is usually best to avoid overlaps to avoid ambiguities in the topology, Sculpt is able to generate a mesh giving precedence to the most recently defined materials. Merging is performed strictly by geometric proximity. Volumes closer than about one half the user input size will normally be automatically merged.

Next, we will examine the mesh quality of the free mesh. The following command will display a graphical representation of the Scaled Jacobian metric.

```
quality hex all scaled jacobian draw mesh
```

The result is shown in Figure 12. Note the elements (colored red) at the interface between materials are unacceptable for simulation. This is caused by the Sculpt algorithm projecting nodes to a common curve interface shared by the materials.

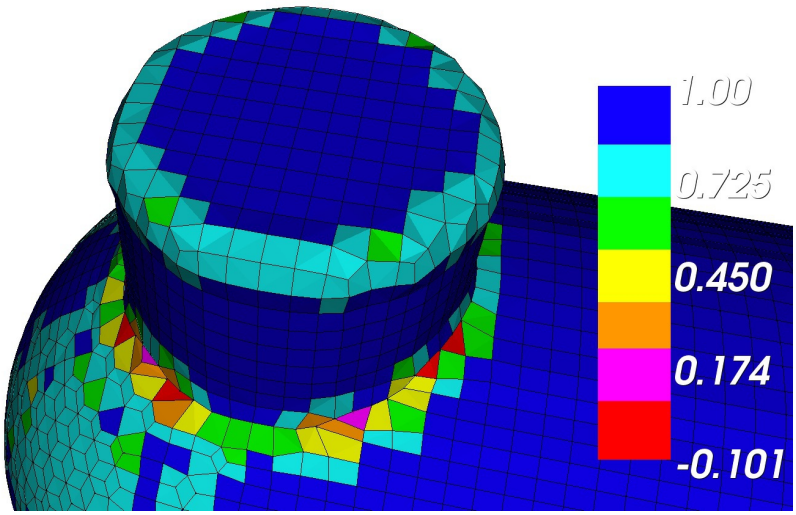


Figure 12. Mesh quality of multi-material mesh.

In most cases, the poor element quality at material interfaces can be improved by using the **pillow** option. Adding this option will direct Sculpt to add an additional layer of elements surrounding each surface. To see the result of pillowing, issue the following commands:

```
delete mesh
sculpt volume all size 0.075 over combine
import pillow 1
quality hex all scaled jacobian draw mesh
```

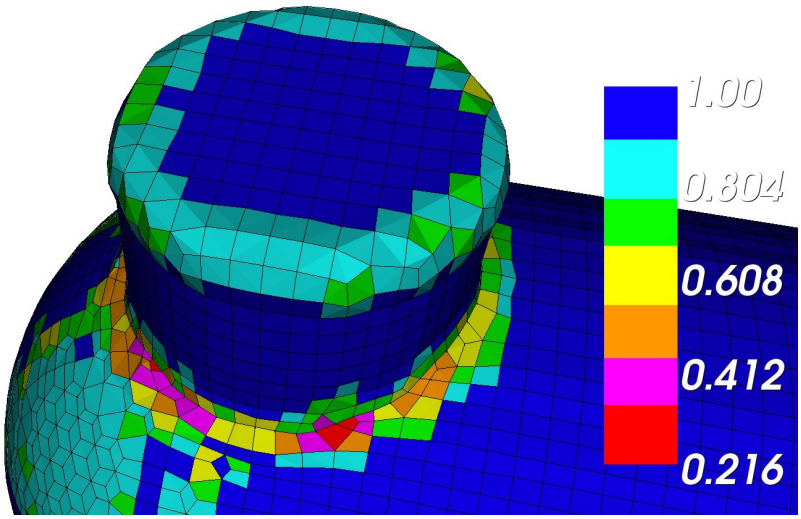


Figure 13. Mesh quality of multi-material mesh using pillow option

The resulting mesh is showed in Figure 13. Note the improved mesh quality at the shared curve interface. A closer look at the mesh, shown in Figure 14. reveals the additional layer of hexes surrounding each surface that allows for improved mesh quality when compared with Figure 11. When generating meshes with multiple materials that must share common interfaces, the **pillow** option is usually recommended.

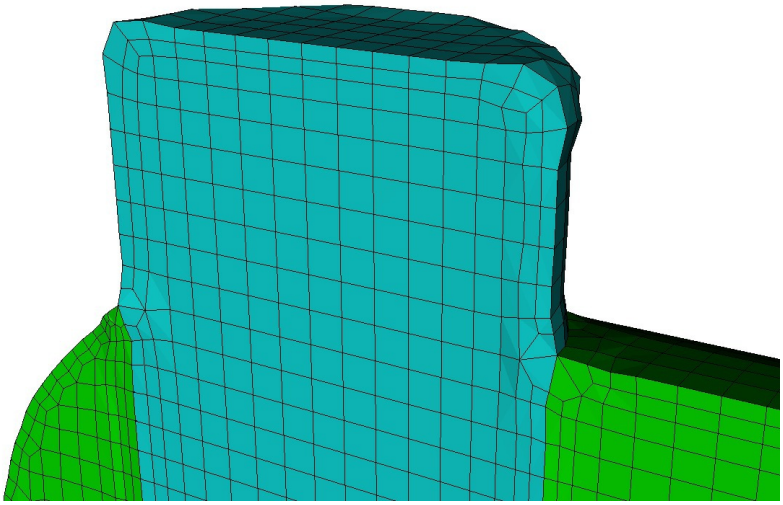


Figure 14. Cutaway of mesh reveals the additional layer of hexes surrounding each surface when the pillow option is used.

Sculpt Technical Description

This document provides a brief technical overview of the Sculpt application, a separate companion application to Cubit designed to generate all-hex meshes of complex geometries. Details on command arguments to Sculpt may be found here. Also information for using Cubit to set up input for Sculpt may be found here.

The method for generating an all-hex mesh employed by Sculpt is often referred to in the literature as an *overlay-grid* or *mesh-first* method. This differs significantly from the algorithms employed by Sweeping and Mapping, which are classified as *geometry-first* methods. Mapping and Sweeping start with the geometry, carefully fitting logical groupings of hexes to conform to a recognized topology. In contrast, the Sculpt method begins with a base Cartesian grid encompassing the geometry which is used as the basis for the mesh. Geometric features are carved or sculpted from the Cartesian grid and boundaries smoothed to create the final hex mesh. The obvious benefit of the Sculpt (*mesh-first*) method over Mapping and Sweeping (*geometry-first*) methods is there is no need to decompose the geometry into mappable or sweebable components, a process that can often be very time consuming, tedious and sometimes impossible. Input to Sculpt can be any geometry regardless of features and complexity.

The basic Sculpt procedure is illustrated in figure 1. Beginning with a Cartesian grid as the base mesh, shown in figure 1(a), a geometric description is imposed. Nodes from the base grid that are near the boundaries are projected to the geometry, locally distorting the nearby hex cells (figure 1(b)). A pillow layer of hexes is then inserted at the surfaces by duplicating the interface nodes on either side of the boundaries and inserting hexes (figures 1(c) and (d)). While constraining node locations to remain on the interfaces, smoothing procedures can now be employed to improve mesh quality of nearby hexes (figure 1(e)).

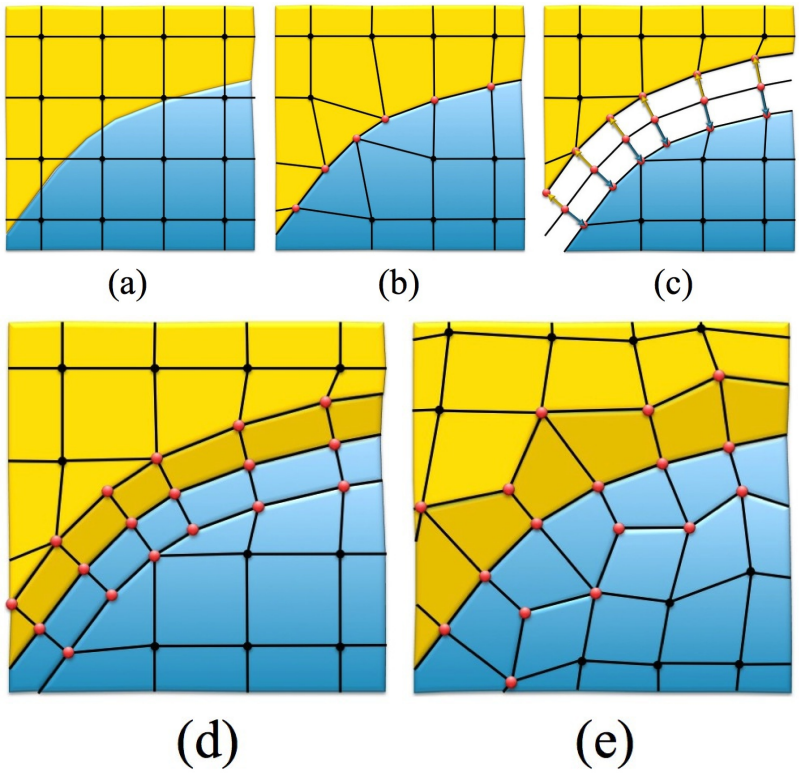


Figure 1. The procedure for generating a hex mesh using the Sculpt overlay grid method

Sculpt is limited to capturing geometric features with the available

resolution of the selected base mesh. Because of this, care should be taken in selecting an appropriate cell size. In addition, no attempt is made by the Sculpt procedure to capture sharp exterior features. Figure 2 shows an example of a sculpt mesh of a CAD model. Note that exterior corner features are rounded, however the effect of sharp feature capture becomes less pronounced as resolution increases as demonstrated in figures 3(a) and (b).

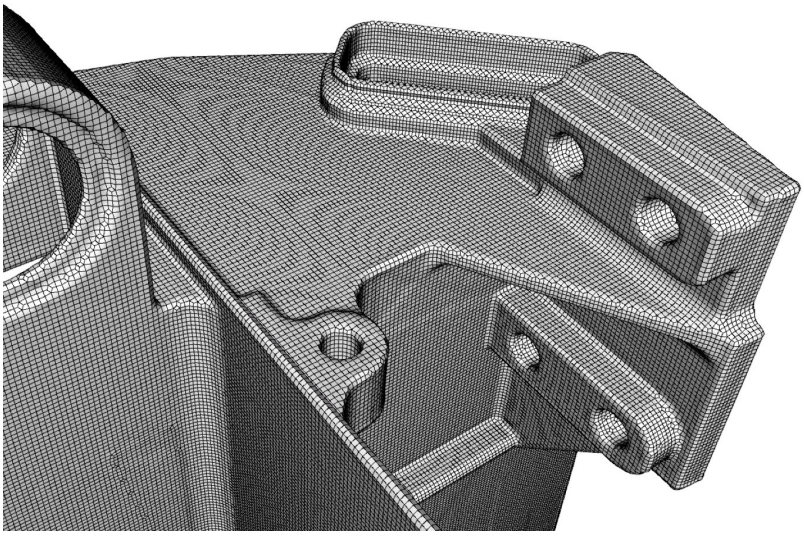
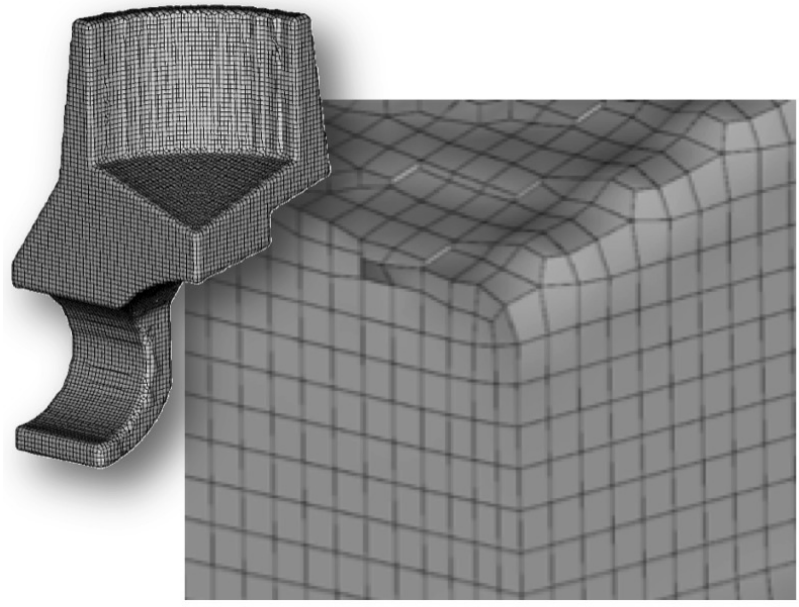
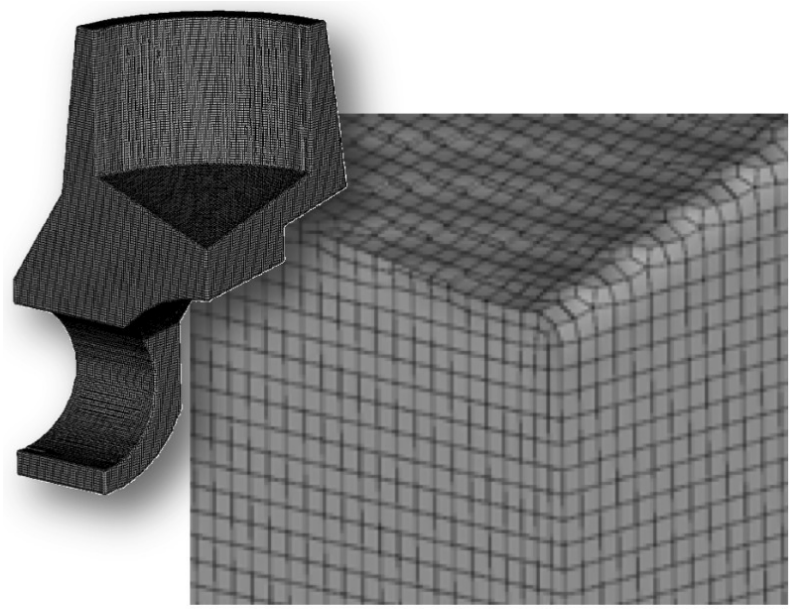


Figure 2. Hex mesh generated using the Sculpt overlay grid procedure



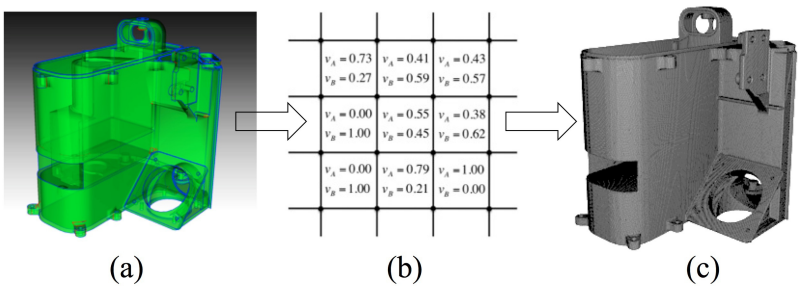
(a)



(b)

Figure 3. Examples of the same model meshed at two different resolutions showing a cutaway view of the mesh.

Another aspect of model preparation for computational simulation involves geometry cleanup and simplification. In most cases, geometry-first methods, such as Sweeping, require an accurate non-manifold boundary representation before mesh generation can begin. Small, sometimes unseen gaps, overlaps and misalignments can result in sliver elements or mesh failure. Tedious manual geometry simplification and manipulation is often required before meshing can commence. Sculpt, however employs a solution that avoids much of the geometry inaccuracy issues inherent in CAD design models. Using a faceted representation of the solid model, a voxel-based volume fraction representation is generated. Figure 4 illustrates the procedure where a CAD model serving as input (figure 4(a)) is processed by a procedure that will generate volume fraction scalar data for each cell of an overlay Cartesian grid (figure 4(b)). One value per material per cell is computed that represents the volume fraction of material filling the cell. A secondary geometry representation is then extracted using an interface tracking technique from which the final hex mesh is generated (figure 4(c)). While similar to its initial facet-based representation, the new secondary geometry description developed from the volume fraction data results in a simplified model that tends to wash over small features and inaccuracies that are smaller than the resolution of the base cell size.



(a) (b) (c)

Figure 4. A representation of the procedure used to generate a hex mesh with Sculpt using Volume Fractions.

While acknowledging some loss in model fidelity in this new volume-fraction based geometric model, the advantage and time-savings to the analyst of being able to ignore troublesome geometry issues is enormous. At the same time it may be important to understand what the additional discrete approximations will make to solution accuracy and employ relevant engineering judgement in the use of this technology.

References

The following technical papers, written by the author of Sculpt, describe the Sculpt procedure in more depth. These papers were presented at the International Meshing Roundtable and are external links to pdf documents.

Parallel Hex Meshing from Volume Fractions: Describes the basic algorithms and mathematics used in the Sculpt procedure.

Parallel Smoothing for Grid-Based Methods: A brief description of the smoothing procedures used in Sculpt.

Validation of Grid-Based Hex Meshes with Computational Solid Mechanics: Describes a study where computational results from Sculpt meshes are compared with Sweep meshes using the Sierra Solid Mechanics Tool as a comparison.

A Template-Based Approach for Parallel Hexahedral Two-Refinement: Describes the refinement procedures used for generating adapted Sculpt meshes.

Sculpt Application

This page describes the Sculpt application, a separate companion application to Cubit designed to run in parallel for generating all-hex meshes of complex geometry. Sculpt was developed as a separate application so that it can be run independently from Cubit on high performance computing platforms. It was also designed as a separable software library so it can be easily integrated as an in-situ meshing solution within other codes. As installed with Cubit, Sculpt can be set up and run directly from Cubit, in a batch process from the unix command line or from a user-defined input file. This documentation describes the input file and command line syntax for the Sculpt Application when running in batch mode. See this page for using Cubit to set up input for Sculpt. A brief technical description of Sculpt may also be found here.

- Sculpt System Requirements
- Running Sculpt
- Sculpt Command Summary
  - Process Control
  - Input Data Files
  - Output
  - Overlay Grid Specification
  - Mesh Type
  - Boundary Conditions
  - Adaptive Meshing
  - Smoothing
  - Mesh Improvement
  - Mesh Transformation
  - Boundary Layers
- Sculpt Examples

Sculpt System Requirements

Sculpt is currently built for windows, linux and mac operating systems. Current supported OS versions should be the same as those supported by Cubit. It is designed to take advantage of 64 bit multicore and distributed memory computers, using open-mpi as the basis for parallel communications.

Running Sculpt

Sculpt can be run using one of two excutables:

psculpt

requires the use of **mpiexec** to start the process. Number of processors to use is specified by the -np argument to mpiexec. psculpt and its input parameters are also used as input to mpiexec. For example:

```
mpiexec -np 8 psculpt -stl myfile.stl -cs 0.5
```

If appropriate system paths have not been set, you may need to use full paths when referring to mpiexec and psculpt.

sculpt

This application assumes that mpiexec is included in the standard CUBIT installation directory. The number of processors to use is specified by the -j option. For example:

```
sculpt -j 8 -stl myfile.stl -cs 0.5
```

If the -j option is not used, sculpt will default to a single processor for execution. The -mpi option can also be used with the sculpt application to indicate a specific mpi installation that is not included with CUBIT. For example:

```
sculpt -j 8 -mpi /path/to/mpiexec -stl myfile.stl -cs 0.5
```

If the path specified by the -mpi option does not exist or the mpi version is incompatible, sculpt will attempt to use the local CUBIT-installed mpiexec or else the system mpiexec in the PATH environment.

Sculpt Examples

The following illustrate simple use cases of the Sculpt application. To use these examples, copy the following stl and diatom files to your working directory

```
brick1.stl
brick2.stl
bricks.diatom
```

Example 1

```
sculpt -j 4 -stl brick1.stl -cs 0.5
```

Runs sculpt with 4 processors with geometry input from brick1.stl. Uses a base Cartesian cell size of 0.5. The bounding box and all other parameters will be defaulted. The result should be the 4 exodus files:

```
brick1.stl_results.e.4.0
brick1.stl_results.e.4.1
brick1.stl_results.e.4.2
brick1.stl_results.e.4.3
```

These files can be combined into a single file using the SEACAS tool **epu**

```
epu -p 4 brick1.stl_results
```

The result of this operation should be a single file:

```
brick1.stl_results.e
```

To view the resulting mesh in Cubit, use the import free mesh command. For example:

import mesh "brick1.stl_results.e" no_geom

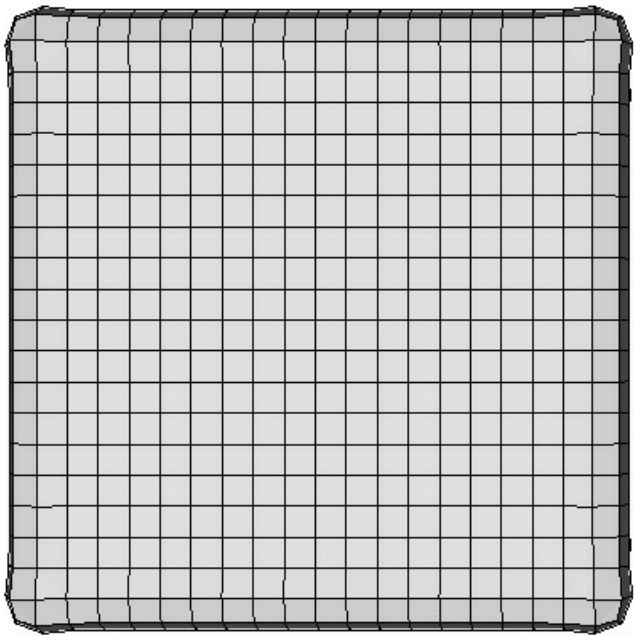


Figure 1. Example 1 mesh

Example 2

```
mpiexec -np 4 psculpt -x 46 -y 26 -z 26 -t -6.5 -u -6.5 -v -6.5 -q 16.5 -r 6.5 -s 6.50 -d bricks.diatom
```

In this case we use **mpiexec** to start 4 processes of **psculpt**. We explicitly define the number of Cartesian intervals and the dimensions of the grid. Rather than using the -stl option, we use the -d option which allows us to specify the diatom file, bricks.diatom. This file allows us to specify multiple stl files, where each one represents a different material. In this case we use both brick1.stl and brick2.stl, which are called out in bricks.diatom.

We can use similar commands as used in Example 1 to combine and import the free mesh into Cubit for display.

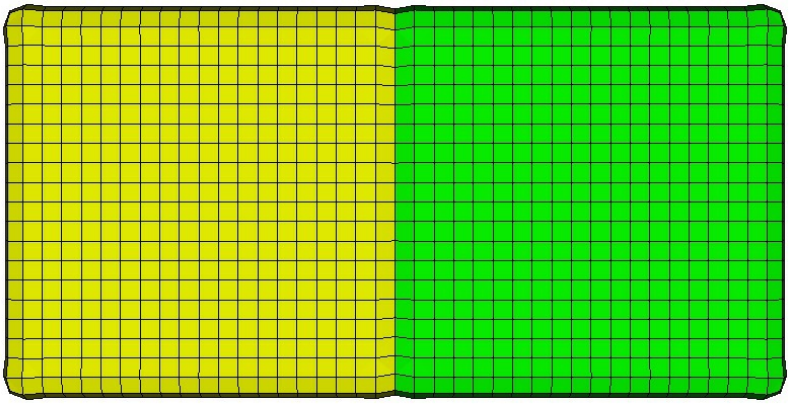


Figure 2. Example 2 mesh

Parallel Meshing

Cubit has been designed as a serial application, using a single CPU to generate its meshes. In some cases, where memory or time constraints are critical, parallel meshing may be necessary. Cubit currently provides a few separate applications designed to run in parallel either on a desktop or on massively parallel cluster machines. In these cases, Cubit can be used as a pre-processor to manipulate geometry and set up for meshing, however the actual meshing procedure is performed as a separate process or on another machine. The following two parallel meshing applications are available:

- pCamal
- Sculpt

A separate application for parallel refinement is also available:

- STK_Adapt

Sculpt Mesh Transformation

Sculpt options for applying transformations to the mesh following mesh generation. For cases where the initial geometry description may not be at the desired scale or bounds, the transformation options provide the ability to apply transformations to the node locations following the mesh generation procedure. This can be effective for microstructure models, where the size and location may be defined by the given intervals of the data.

```
Mesh Transformation          -tfm     --transform
  --xtranslate               -xtr  <arg> Translate final mesh coordinates in X
  --ytranslate               -ytr  <arg> Translate final mesh coordinates in Y
  --ztranslate               -ztr  <arg> Translate final mesh coordinates in Z
  --xscale                   -xsc  <arg> Scale final mesh coordinates in X
  --yscale                   -ysc  <arg> Scale final mesh coordinates in Y
  --zscale                   -zsc  <arg> Scale final mesh coordinates in Z
```

Sculpt Command Summary

Translate Mesh Coordinates in X

```
Command: xtranslate     Translate final mesh coordinates in X

Input file command:   xtranslate <arg>
Command line options: -xtr <arg>
Argument Type:        floating point value
```

Command Description:

Translate all mesh coordinates written to Exodus file by X delta distance.

Translate Mesh Coordinates in Y

```
Command: ytranslate     Translate final mesh coordinates in Y

Input file command:   ytranslate <arg>
Command line options: -ytr <arg>
Argument Type:        floating point value
```

Command Description:

Translate all mesh coordinates written to Exodus file by Y delta distance.

Translate Mesh Coordinates in Z

```
Command: ztranslate     Translate final mesh coordinates in Z

Input file command:   ztranslate <arg>
Command line options: -ztr <arg>
Argument Type:        floating point value
```

Command Description:

Translate all mesh coordinates written to Exodus file by Z delta distance.

Scale X Mesh Coordinates

```
Command: xscale     Scale final mesh coordinates in X

Input file command:   xscale <arg>
```

```
Command line options: -xsc <arg>

Argument Type:        floating point value
```

Command Description:

Scale all mesh X coordinates written to Exodus file by given factor

---

Scale Y Mesh Coordinates

```
Command: yscale     Scale final mesh coordinates in Y

Input file command:   yscale <arg>
Command line options: -ysc <arg>
Argument Type:        floating point value
```

Command Description:

Scale all mesh Y coordinates written to Exodus file by given factor

---

Scale Z Mesh Coordinates

```
Command: zscale     Scale final mesh coordinates in Z

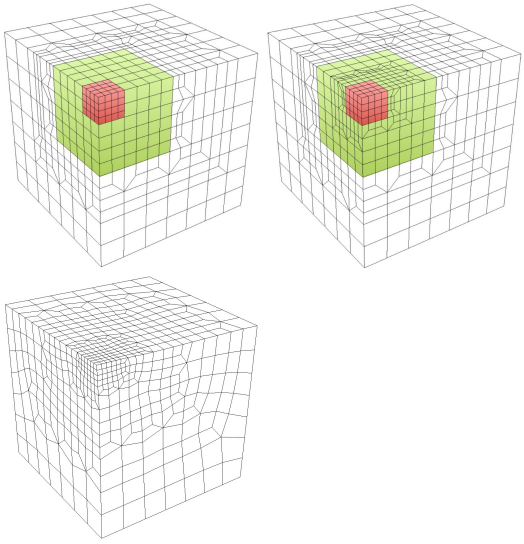
Input file command:   zscale <arg>
Command line options: -zsc <arg>
Argument Type:        floating point value
```

Command Description:

Scale all mesh Z coordinates written to Exodus file by given factor

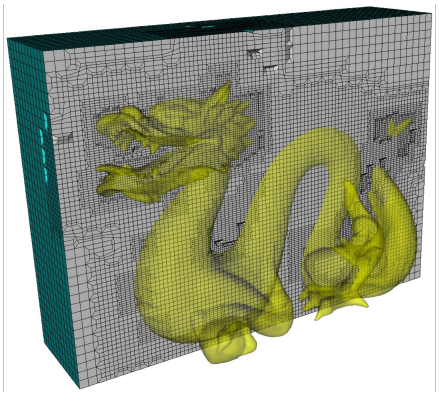
Sculpt Adaptive Meshing

Sculpt options for specifying adaptive meshing. Sculpt uses an initial overlay Cartesian grid that serves as the basis for the all-hex mesh. The default mesh size will roughly follow the constant size cells of the overlay grid. The adaptivity option allows the user to automatically split cells of the Cartesian grid based on geometric criteria, resulting in smaller cells in regions with finer details. The adapted grid is then used as the basis for the Sculpt procedure.

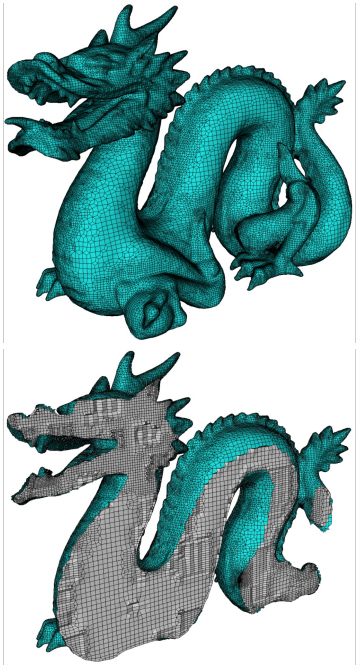


Adaptive mesh begins with constant size coarse Cartesian grid. Cells are recursively split based on geometry criteria and transitions added between levels. Projections and smoothing are performed to improve element quality.

Three options are used for controlling the adaptivity in sculpt: adapt_type, adapt_levels and adapt_threshold. The adapt_type option controls the method and geometric criteria used for deciding which cells to split in the grid, while the adapt_levels option controls the the maximum number of times any one cell can be split. Depending upon the adapt_type selected, the adapt_threshold is used as the specific geometric threshold value at which the decision is made to split any given cell.



Initial cut-away view of adapted grid from dragon model before performing Sculpt operations.



The final mesh of the dragon model and cutaway view of the mesh is shown with up to 4 levels of adaptive refinement.

Adaptive Meshing	-adp --adapt	
--adapt_type	-A	<arg> Adaptive meshing type
--adapt_threshold	-AT	<arg> Threshold for adaptive meshing
--adapt_levels	-AL	<arg> Number of levels of adaptive refinement
--adapt_material	-AM	<arg> Info for adapting material
--adapt_export	-AE	Export exodus mesh of refined grid
--adapt_non_manifold	-ANM	Refine at non-manifold conditions
--adapt_load_balance	-ALB	Adaptive parallel load balancing
--adapt_memory_stats	-AMS	Write memory usage stats for adaptivity

Sculpt Command Summary

Adaptive Refinement Type

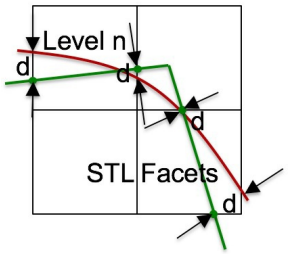
```
Command: adapt_type     Adaptive meshing type

Input file command:   adapt_type <arg>
Command line options: -A <arg>
Argument Type:        integer (0, 1, 2,...7)
Input arguments: off (0)
                 facet_to_surface (1)
                 surface_to_facet (2)
                 surface_to_surface (3)
                 vfrac_average (4)
                 coarsen (5)
                 vfrac_diff (6)
                 vfrac_difference (6)
                 resample (7)
                 material (8)
                 region (12)
```

Command Description:

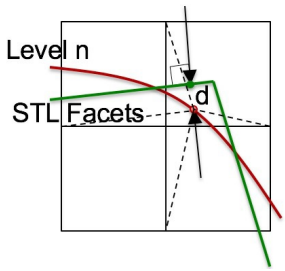
This option will automatically refine the mesh according to a user-defined criteria. Without this option, a constant cell size will be assumed everywhere in the model. To build the mesh, Sculpt uses an approximation to the exact geometry of the CAD model by interpolating mesh surfaces from volume fraction samples in each cell of the Cartesian grid. In general, the, higher the resolution of the Cartesian grid, the more sampling is done and the more accurate the mesh will represent the initial geometry. The adapt_type selected will control the criteria used for refining the mesh. If the criteria is not satisfied, the refinement will continue until a threshold indicated by the adapt_threshold parameter is satisfied everywhere, or the maximum number of levels (adapt_levels) is reached. The following criteria for refinement are available:

- **off (0):** Cartesian grid is defined only by **nelx**, **nely**, and **nelz** or **cell_size** which is used as the basis for the sculpt mesh. No refinement will be performed.
- **facet_to_surface (1):** This option will evaluate every location where an edge in the Cartesian grid intersects a triangle of the STL model and measures the closest distance to the approximated geometry. The cells adjacent to intersecting edges where the measured distance is greater than the **adapt_threshold** will be identified for uniform refinement. This is done for each refinement level where a new approximated geometry is then computed based upon the finer resolution grid. The refinement will continue until all measured distances are less than the adapt_threshold, or the maximum number of levels (**adapt_levels**) is reached. This option can only be used if input comes from an STL file. Microstructures and diatoms are currently not supported.



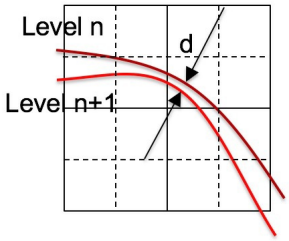
Distance from STL Facet to Approximated Geometry. The distance d is measured between the facets (green) where they cross the edges of the grid, to the closest point on the interpolated geometry. If d > adapt_threshold then the cell is split.

- **surface_to_facet (2):** This criteria is similar to **facet_to_surface** (1) except that the locations selected for sampling are chosen from the vertices representing the approximated surfaces. The closest distance measured to any of the facets in the STL model is used as the criteria for refinement. Those cells at vertices where the distance measured exceeds the **adapt_threshold** are identified for refinement. This option is generally faster than 1, but may miss features if the initial resolution of the grid is too coarse. This option can also only be used if input geometry comes from an STL file. Microstructures and diatoms are currently not supported.



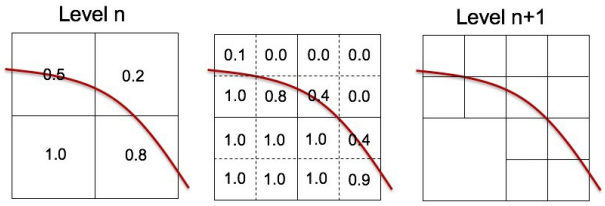
Distance from Approximated Geometry to STL Facet. The distance d is measured between points on the interpolated geometry corresponding to a projected point on the grid, to its closest point on one of the STL facets. If d > adapt_threshold then the cell is split.

- **surface_to_surface (3):** This criteria will test each cell to compute the local interpolated surface for the cell and compare with the surface interpolated for its eight subdivided child cells. If the distance between these two approximated surfaces is greater the the user defined **adapt_threshold**, then the cell will be uniformly refined. This option can be used with STL and diatom input geometry, but not with Microstructures.



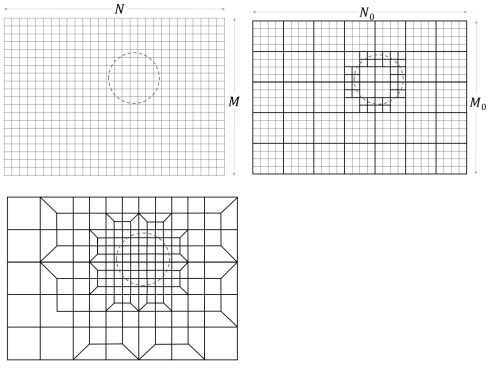
Distance Between Child and Parent Approximated Geometry. After computing the interpolated geometry for level n and level n+1, d is the distance between the two geometry representations. Cells where d > adapt_threshold are split.

- **vfrac_average (4):** Each cell of the Cartesian grid is tested to determine if it should be subdivided into eight cells. The volume fraction of the parent cell is compared with the average volume fraction of its eight child cells. If the absolute difference between the average child volume fraction and its parent volume fraction is greater than the user defined **adapt_threshold** then the cell is uniformly refined. The **adapt_threshold** for this case should be a number between 0 and 1. A smaller number will be more sensitive to changes in geometry, usually resulting in more refinement at interfaces.



Difference of Cell Volume Fraction. Volume fractions are evaluated for the 8 child cells of a cell at level n. This example shows where one or more of the volume fractions at level n+1 of the lower left cells does not differ by more than a threshold d, so it remains unsplit.

- **coarsen (5):** Given a dense set of data on a Cartesian Grid, Sculpt will begin at a coarse resolution and refine to capture changes in the data. It uses the adapt_levels option to determine the coarseness of the initial grid. For example, a dense grid of LxMxN cells will begin with an initial resolution of L/2^a x M/2^a x N/2^a, where a is the user defined adapt_levels value. Cells will be identified for refinement if the volume fraction of any material in a cell is greater than the user defined **adapt_threshold** and less than 1.0-adapt_threshold. This option is available only for **input_spn** and **input_micro** formats. It is most useful for cases where very dense data is initially provided which would be too fine to serve as an FEA mesh. This method will effectively coarsen the mesh on the interior and exterior of solids, but maintain a fine resolution at geometry boundaries.



Refine to Dense Data (Coarsening). Initial grid at resolution N X M is coarsened to N0 X M0 based on the

adapt_levels value. Coarse cells are then split similar to criteria in adapt_type = 4.

- **vfrac_difference (6):** Each cell of the Cartesian grid is tested to determine if it should be subdivided into eight cells. The volume fraction of the parent cell is compared with each of the volume fractions of its eight child cells. If the absolute difference between any of the child volume fractions and its parent volume fraction is greater than the user defined **adapt_threshold** then the cell is uniformly refined. The **adapt_threshold** for this case should be a number between 0 and 1. A smaller number will be more sensitive to changes in geometry, usually resulting in more refinement at interfaces.

- **resample (7):** Input volume fraction data from microstructure file formats (**input_micro, input_cart_exo and input_spn**) is down sampled, averaging volume fraction data across multiple cells according to the **adapt_levels** set. To resample at 1/2 resolution, use **adapt_levels = 1**, For 1/4 resolution, use **adapt_levels = 2**, etc. For example, if the input data has a resolution of 100X100X100, **adapt_levels = 1** will result in effectively a 50X50X50 grid resolution where each 2X2X2 group of cells in the original data is averaged into a single larger cell. Similarly, adapt_type = 2 will result in a 25X25X25 grid resolution, where each 4X4X4 group of cells makes up a single cell in the resample. Default **adapt_levels** is 2.

- **material (8):** Refinement will be done in cells where the predominant volume fraction is a user defined material ID. Must be used with the **adapt_material** option to indicate a material to be refined.

To maintain a conforming mesh, transition elements will be inserted to transition between smaller and larger element sizes. Default for the **adapt_type** option is **off (0)** (or that no adaptive refinement will take place).

In all cases the initial Cartesian grid defined by xint, yint and zint or the **cell_size** value will be used as the basis for refinement and will define the approximate largest element size in the mesh.

Adaptive Refinement Threshold

```
Command: adapt_threshold     Threshold for adaptive meshing

Input file command:   adapt_threshold <arg>
Command line options: -AT <arg>
Argument Type:        floating point value >= 0.0
```

Command Description:

This value controls the sensitivity of of the adaptivity. The value used should be based upon the adapt_type:

- **facet_to_surface (1)**

  surface_to_facet (2)

  surface_to_surface (3)

  For these options, the **adapt_type** selected represents an absolute distance between surfaces or facets. Where the distance exceeds **adapt_threshold** the nearby cell or cells will be identified for refinement. The smaller this number the more sensitive will be the adaptation and greater the resulting number of elements. If not specified, the **adapt_threshold** will be determined as follows:

```
adapt_threshold = 0.25 * cell_size / adapt_levels^2
```

- **vfrac_average (4)**

  coarsen (5)

  The **adapt_threshold** value in this case represents the maximum difference in volume fraction between a parent cell and the average of its eight child cells. This value should be between 0.0 and 1.0. The smaller the number, the more sensitive will be the adaptation and the greater the number of resulting elements. A default **adapt_threshold** of 0.01 is used if not specified.

- **material (8)**

  Refinement occurs if volume fraction of material in a cell exceeds the threshold value. A separate **threshold** can also be set for each **adapt_material** overriding the the global value for threshold.

Note that the user defined adapt_threshold may not be satisfied everywhere in the mesh if the value defined for adapt_levels is exceeded.

Number of Adaptive Levels

```
Command: adapt_levels     Number of levels of adaptive refinement

Input file command:   adapt_levels <arg>
Command line options: -AL <arg>
Argument Type:        integer >= 0
```

Command Description:

The maximum number of levels of adaptive refinement to be performed. One level of refinement will split each Cartesian grid cell identified for uniform refinement into eight child cells. Two levels of refinement will split each cell again into eight, resulting in sixty-four child cells, three levels into 256, and so on. The maximum number of subdivision per cell is give as:

```
number of cells = 8^adapt_levels
```

The minimum edge length for any cell will be given by:

```
min cell edge length = cell_size / adapt_levels^2
```

The actual number of refinement levels used will be determined by whether all cells meet the **adapt_threshold**, or the adapt_levels value is exceeded. The default adapt_levels is 2. Note that setting the **adapt_levels** more than 4 or 5 can result in long compute times.

Info for Adapt Material

```
Command: adapt_material     Info for adapting material

Input file command:   adapt_material <arg>
Command line options: -AM <arg>
Argument Type:        5 optional values
```

Command Description:

Specifies material information to be used with the **adapt_type = material** option. This option expects up to 5 ordered values representing the following:

1. **material id**: ID of material to be refined
2. **levels**: (optional) Number of levels to uniformly refine within the specified material. If not specified or **levels** <= 0, the global value

for **adapt_levels** will be used.
3. **threshold**: (optional) Threshold used to determine whether to uniformly refine. Refinement occurs if volume fraction of material in a cell exceeds the threshold value. If not specified or **threshold** <= 0, the global value for **adapt_threshold** will be used.
4. **expansion distance**: (optional) Absolute distance beyond the specified material that will be uniformly refined. If not specified or **expansion distance** = 0, uniform refinement will be restricted to the material. A negative integer value for **expansion distance** will be interpreted as the number of layers of cells beyond the material that will be refined.
5. **region**: (optional) specify a region id as defined by an **adapt_region** specification. Refinement will be limited to the adapt_region overlapping the specified region. If not specified or **region** <= 0, the full material domain will be refined.

This option may be used multiple times in the same input, once per material to be adapted.

Export Refined Cartesian Grid

```
Command: adapt_export     Export exodus mesh of refined grid

Input file command:   adapt_export
Command line options: -AE
```

Command Description:

Export an exodus mesh containing the refined Cartesian grid. Interface reconstruction, boundary layer insertion and smoothing have not yet been applied to this mesh. It is the base mesh used as input to Sculpt. One file per processor will be exported in the form "vfrac_adapt.e.x.x". The exodus mesh produced will also contain the computed volume fractions for each material present in the model represented as element variables.

This option is primarily used for debugging the refinement option. However the mesh produced with this option can be used as the base mesh when used with the **input_mesh** option. For example, instead of Cartesian grid options, the input mesh may be specified as **input_mesh = vfrac_adapt.e.1.0**. Sculpt will use the refined mesh and the volume fraction element variables to build the final mesh.

Adapt Cells at Non-manifold Nodes

```
Command: adapt_non_manifold     Refine at non-manifold conditions

Input file command:   adapt_non_manifold
Command line options: -ANM
```

Command Description:

If refinement results in a non-manifold condition at a node, the surrounding cells will be identified for refinement. In some cases, using this option will result in a closer match to geometry for thin layers or small features. Using this option will normally result in more elements at material interfaces. Note that in all cases non-manifold conditions will be resolved even without this option in a subsequent step, however without this option, the resulting solution may not match geometry as accurately.

The **adapt_non_manifold** option is off by default. It is currently only implemented for adapt_type that use an STL geometry definition. (**adapt_type = 1,2,3**)

Adaptive Parallel Load Balancing

```
Command: adapt_load_balance     Adaptive parallel load balancing

Input file command:   adapt_load_balance
Command line options: -ALB
```

Command Description:

Do adaptive load balancing during refinement. Uses Zoltan's Recursive Coordinate Bisection (RCB) algorithm to repartition parallel domains following each adaptive level.

Default (OFF), refinement, will use roughly spatially equal volume domains on each processor. This can cause significant imbalance in work between processors, sometimes causing failure, especially where adpativity is localized. When using **adapt_load_balance**, Sculpt will repartition the parallel domains after each adaptivity level so that each processor maintains approximetly equal numbers of elements to equalize work and memory among processors.

Write Memory Usage Stats for Adaptivity

```
Command: adapt_memory_stats     Write memory usage stats for adaptivity

Input file command:   adapt_memory_stats
Command line options: -AMS
```

Command Description:

When the **adapt_type** option is enabled, it allows users to monitor the memory usage and availability during the adaptivity procedures. By setting this option, both minimum and maximum memory usage for the current processor arrangement will be recorded. This information can be particularly useful when building meshes with many levels of refinement or when there are significant variations in the spatial distribution of refinement regions.

Sculpt Boundary Conditions

Sculpt options for specifying the methods for generating nodesets, sidesets and blocks on the mesh. Several automatic methods for generating nodesets and sidesets are provided in Sculpt using the gen_sidesets option. Where multiple blocks are required, Block IDs are normally defined using the material ID in the diatom file. Each STL file can be associated with a different block ID. If the mesh_void option is used, the ID for the block of elements in the void region can be set using the void_mat option.

For other input formats such as volume fraction microstructure data or Cartesian Exodus files, the Block IDs are defined by the individual formats.

```
Boundary Conditions           -bc     --boundary_condition
  --void_mat                  -VM   <arg> Void material ID (when mesh_void=true)
  --separate_void_blocks      -SVB        Separate void into unique block IDs
  --material_name             -mn   <arg> Label Material (Block) with Name
  --sideset_name              -sn   <arg> Label Sideset with Name
  --nodeset_name              -nn   <arg> Label Nodeset with Name
  --sideset                   -sid  <arg> User Defined Sideset
  --nodeset                   -nid  <arg> User Defined Nodeset
  --gen_sidesets              -SS   <arg> Generate sidesets
  --free_surface_sideset      -FS   <arg> Free Surface Sideset
  --match_sidesets            -mss  <arg> Sidesets ids of matching pairs
  --match_sidesets_nodeset    -msn  <arg> Nodeset defining match_sidesets
```

Sculpt Command Summary

Void Material ID

```
Command: void_mat     Void material ID (when mesh_void=true)

Input file command:   void_mat <arg>
Command line options: -VM <arg>
Argument Type:        integer > 0
```

Command Description:

When the **mesh_void** option is used, this value is the material (block) ID assigned to all elements in the void region. If void_mat option is not used, the material ID of elements in the void region will be the maximum material ID in the model + 1. Note that the void_mat may be the same as an existing material in another part of the model.

Separate Void Blocks

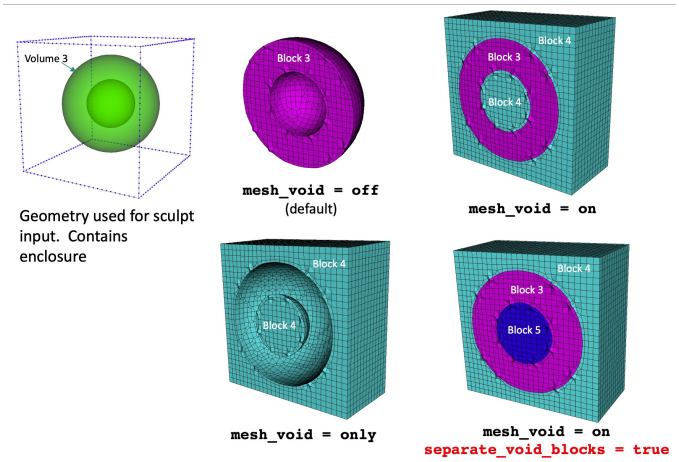
```
Command: separate_void_blocks     Separate void into unique block IDs

Input file command:   separate_void_blocks
Command line options: -SVB
```

Command Description:

When the **mesh_void** option is used, the default case will generate a single block ID for all elements in the void region. Turning this option **ON** will separate the elements into unique block IDs where the elements are contiguous. For example, elements in a void region completely enclosed and interior to a volume will be assigned a different block ID to those exterior to the volume. The **void_mat** option can be used to specify the ID of the first void block. Subsequent blocks will be numbered incrementally from the first void mat ID. If **void_mat** is not defined, the next highest IDs not used by a material will be used for the void block

IDs.



Example of the effect of using **separate_void_blocks** with the **mesh_void** option

---

Material Name

```
Command: material_name     Label Material (Block) with Name

Input file command:   material_name <arg>
Command line options: -mn <arg>
Argument Type:        integer and string
```

Command Description:

Optionally assign a name to a material. Specify a valid material ID followed by a name. This option can be used multiple times to label one or more materials. The material name will be assigned as the block name to be written to the output exodus file. A valid integer representing a material defined in the input should be used. For example, the name will be associated with the material ID identified in the diatom file for a given package. The command will be ignored if a valid matching material is not found.

Note: When using the **input_mesh** option and **input_mesh_material = blocks**, block names may be included in the input exodus mesh as part of the exodus block description. If the **material_name** option is used and exodus blocks names are included in the input file, the specified material name in the sculpt input will be used.

Sideset Name

```
Command: sideset_name     Label Sideset with Name

Input file command:   sideset_name <arg>
Command line options: -sn <arg>
Argument Type:        integer and string
```

Command Description:

Optionally assign a name to a sideset. Specify a valid sideset ID followed by a name. This option can be used multiple times to label one or more sidesets. For example:

```
sideset_name 10 left_faces
sideset_name 20 right_faces
sideset_name 30 bottom_faces
sideset_name 40 top_faces
```

The assigned sideset name will be written to the output exodus file. A

valid integer representing a sideset to be generated should be used. The sideset ID should refer to an existing sideset defined using **gen_sidesets** or **sideset** options.

Note: When using the **input_mesh** option and **input_mesh_material = blocks**, sideset names may be included in the input exodus mesh as part of the exodus sideset description. If the **sideset_name** option is used and exodus sideset names are included in the input file, the specified sideset name in the sculpt input will be used.

Nodeset Name

```
Command: nodeset_name     Label Nodeset with Name

Input file command:   nodeset_name <arg>
Command line options: -nn <arg>
Argument Type:        integer and string
```

Command Description:

Optionally assign a name to a nodeset. Specify a valid nodeset ID followed by a name. This option can be used multiple times to label one or more nodesets. For example:

```
nodeset_name 10 left_nodes
nodeset_name 20 right_nodes
nodeset_name 30 bottom_nodes
nodeset_name 40 top_nodes
```

The assigned nodeset name be will written to the output exodus file. A valid integer representing a nodeset to be generated should be used. The nodeset ID should refer to an existing sideset defined using **gen_sidesets** or **nodeset** options.

Note: When using the **input_mesh** option and **input_mesh_material = blocks**, nodeset names may be included in the input exodus mesh as part of the exodus nodeset description. If the **nodeset_name** option is used and exodus nodeset names are included in the input file, the specified nodeset name in the sculpt input will be used.

User Defined Sideset

```
Command: sideset     User Defined Sideset

Input file command:   sideset <arg>
Command line options: -sid <arg>
Argument Type:        integer and string
```

Command Description:

Define a sideset (collection of mesh faces) based on a user-defined specification. Requires one integer representing a unique sideset ID and a character string representing a geometry specification. Current supported geometry specifications include: **xmin, xmax, ymin, ymax, zmin, zmax**. For a mesh defined with an axis-aligned orientation this option will place sidesets at the bounding box boundaries. For example:

```
sideset 10 xmin
sideset 20 xmax
sideset 30 ymin
sideset 40 ymax
```

Note: See also option **gen_sidesets = RVE**. The gen_sidesets option will automatically define sidesets at boundaries for Cartesian base meshes.

User Defined Nodeset

```
Command: nodeset     User Defined Nodeset

Input file command:   nodeset <arg>
Command line options: -nid <arg>
Argument Type:        integer and string
```

Command Description:

Define a nodeset (collection of nodes) based on a user-defined specification. Requires one integer representing a unique nodeset ID and a character string representing a geometry specification. Current supported geometry specifications include: **xmin, xmax, ymin, ymax, zmin, zmax**. For a mesh defined with an axis-aligned orientation this option will place nodesets at the bounding box boundaries. For example:

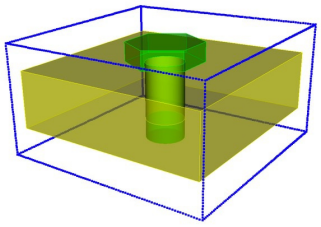
```
    nodeset 10 xmin
    nodeset 20 xmax
    nodeset 30 ymin
    nodeset 40 ymax
```

Generate Sidesets

```
Command: gen_sidesets     Generate sidesets

Input file command:   gen_sidesets <arg>
Command line options: -SS <arg>
Argument Type:        integer (0, 1, 2, 3, 4, 5)
Input arguments: off (0)
                 fixed (1)
                 variable (2)
                 geometric_surfaces (3)
                 geometric_sidesets (4)
                 rve (5)
                 input_mesh_and_stl (6)
                 input_mesh_and_free_surfaces (7)
                 rve_variable (8)
                 input_mesh (9)
```

Command Description:



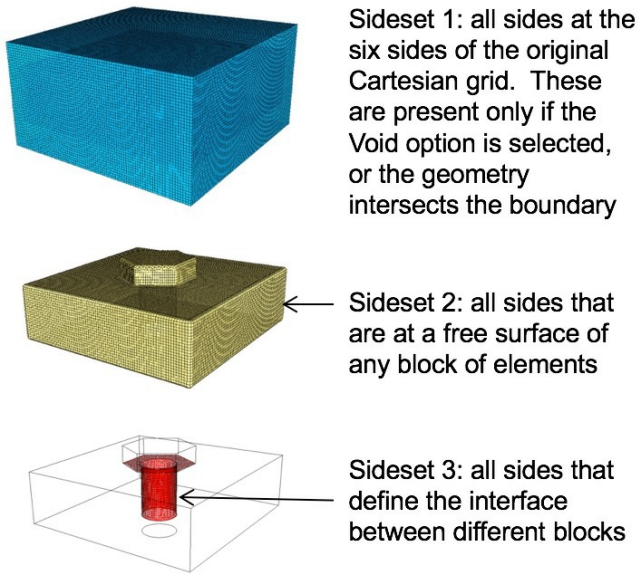
Geometry used in sideset examples below.

Generate exodus sidesets using one of the following options:

off (0): No sidesets will be generated

fixed (1): Exactly 3 sidesets will be generated according to the following:

- Sideset 1: All sides at the domain boundary. Sides will only be present in this sideset if the model intersects the enclosing bounding box or the void option is used.
- Sideset 2: All sides at the model boundary. Any side on the model that is not interior will be included. This should represent a full enclosure of the model if it does not intersect the domain boundary.
- Sideset 3: All sides at material interfaces. Includes sides on the interior where adjacent blocks are different.

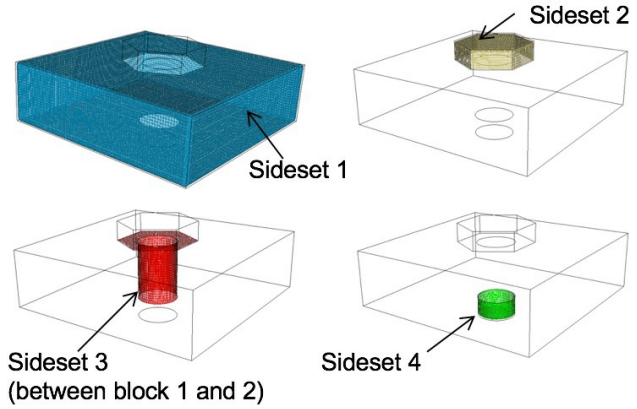


Example of fixed(1) sidesets.

variable (2): A variable number of sidesets will be generated with the following characteristics:

- Surfaces at the domain boundary
- Exterior material surfaces
- Interfaces between materials

Unlike Fixed sidesets, grouping of sides will be contiguous. A separate sideset will be generated for each set of contiguous sides.



Example of variable(2) sidesets.

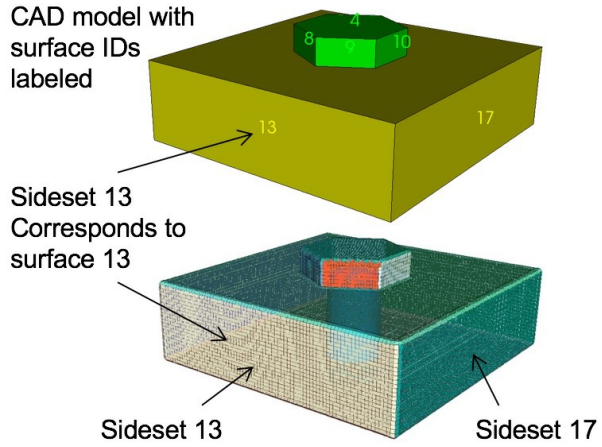
geometric_surfaces (3): Sidesets will be generated according to imported surface ID information. STL files may include an optional surface designation for any or all triangles in the file. Surface information may be written automatically from Cubit based on geometric surface IDs or sideset IDs. See the cubit sculpt parallel sideset option for more details. Alternatively, use the "export stl ..." command with the "sidesets" option to export all sidesets in a Cubit model as surface information. If present, one sideset will be generated for each surface designation in the STL file. Following is an example surface designation in an STL file. It would appear following all triangles.

```
surface 1
  0 1 2 3 4 5 6 7 8 9
  10 11 12 13 14 15 16 17 18 19
  20 21 22 23
endsurface 1
```

The id following the surface designation will be used as the sideset ID. Up to 10 triangle IDs, per line may be assigned to the surface. Triangle IDs are assigned based on order they appear in the STL file. Any number of surfaces may be defined. For this option, the assumption is that all triangles included in the STL files will be included in at least one surface

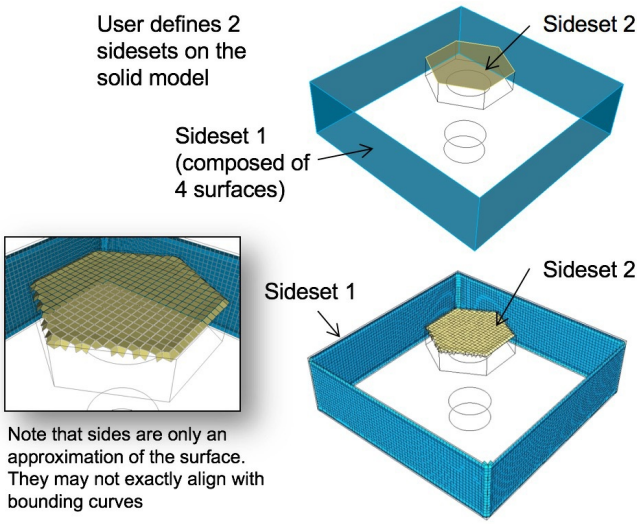
designation.

In this example there a 17 sidesets generated.
One for each surface in the CAD model



Example of all geometric surfaces (3) defining sidesets.

geometric_sidesets (4): Similar to **geometric_surfaces**, except that only a portion of the triangles may be designated as sideset surfaces. This option is useful when using Cubit to identify specific surfaces as sidesets.

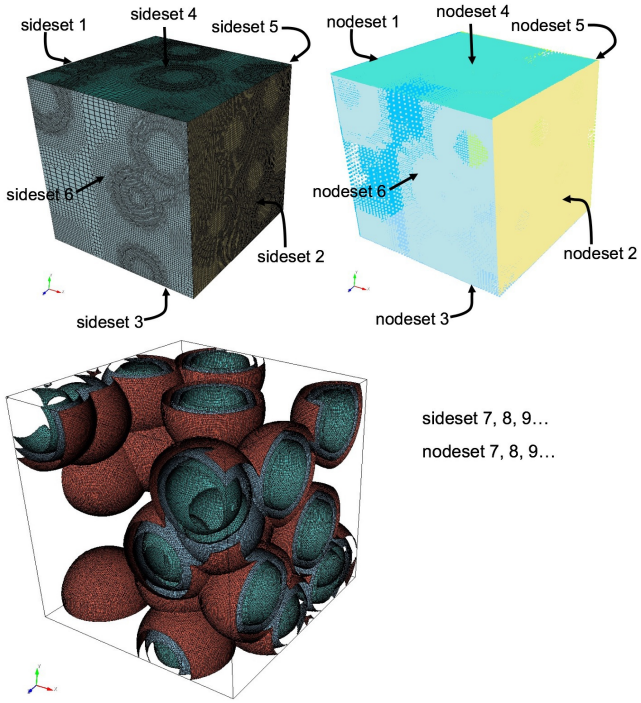


Example of selected geometric sidesets (4) in Cubit defining sidesets in Sculpt.

RVE (5): When using the full bounding box, such as representative volume elements (RVE) for microstructures, the nodesets and sidesets with IDs 1 to 6 are reserved for the six faces of the bounding box. They are assigned as follows:

Nodeset/Sideset ID	Contains nodes/faces
1	on minimum X domain boundary
2	on maximum X domain boundary
3	on minimum Y domain boundary
4	on maximum Y domain boundary
5	on minimum Z domain boundary
6	on maximum Z domain boundary

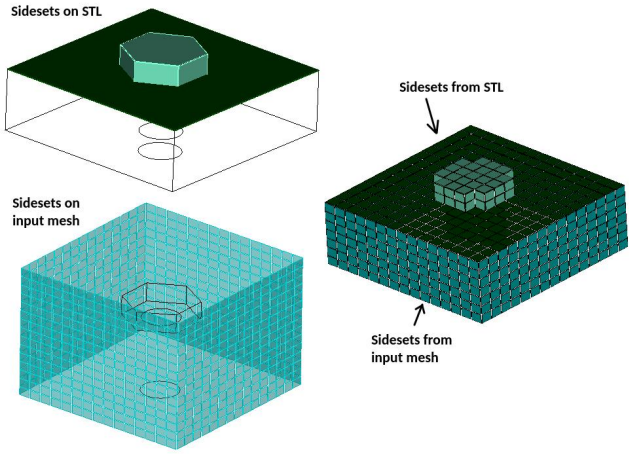
In addition, a nodeset and sideset will be generated on interior surfaces for each unique pair of adjacent material IDs. One final nodeset will also be generated along interior curves at all internal triple junctions (curves where at least 3 surfaces share a common curve).



Example of automatically defined sidesets at domain boundaries of an RVE and at all interface surfaces between materials.

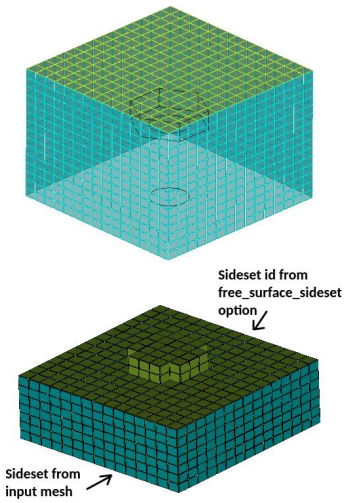
input_mesh (9): Used with the **input_mesh** option where an exodus file is used as the base grid. Only sidesets and nodesets defined in the input exodus mesh are transferred to the output mesh.

input_mesh_and_stl (6): Used with the **input_mesh** option where an exodus file is used as the base grid. Sidesets and nodesets defined in the input exodus mesh are transferred to the output mesh if the surface is not an interior surface. Sidesets defined in the augmented STL input file are transferred to the output mesh for interior surfaces. See also the **free_surface_sideset** option for prescribing a sideset on interior surfaces cut by the STL definition when using the **input_mesh** option.



Example of sidesets defined in the input mesh and corresponding domain boundary sidesets in the output mesh.

input_mesh_and_free_surfaces (7): Used with the **input_mesh** option where an exodus file is used as the base grid. Sidesets and nodesets defined in the input exodus mesh are transferred to the output mesh if the surface is not an interior surface. Sidesets defined in the **free_surface_sideset** option are used to define sidesets for interior surfaces.



Example of sidesets defined in the input mesh and corresponding domain boundary sidesets in the output mesh.

rve_variable (8): Nodesets 1-6 and Sidesets 1-6 are defined at the boundaries as described in the **gen_sidesets = rve (5)** option. With the **rve_variable** option, additional nodesets and sidesets at material interfaces on the interior of the mesh are defined similar to the **gen_sidesets = variable (2)** option. Grouping of interior sides in a sidesets will be contiguous, where a separate sideset will be generated for each unique set of contiguous sides. Nodesets will be generated in a similar manner.

Free Surface Sidesets

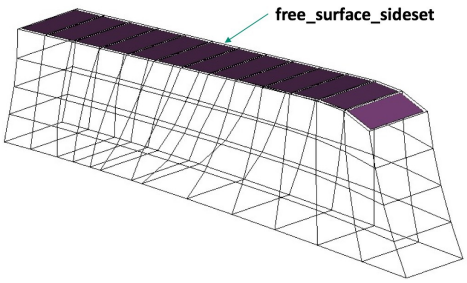
```
Command: free_surface_sideset     Free Surface Sideset

Input file command:   free_surface_sideset <arg>
Command line options: -FS <arg>
Argument Type:        integer(s) >= 0
```

Command Description:

Given exodus sidesets are treated as interior surfaces for STL projection.

Used with the **input_mesh** option when using an exodus mesh as the base grid. This may be useful if the **capture** option is enabled and some of the STL surfaces are close in proximity to the boundaries of the input exodus mesh. When close in proximity, sculpt will by default not project those boundary nodes to the STL surface but keep them on the domain boundary. If a list of sideset IDs are given here, the sideset faces will be projected to the STL. The sideset IDs should refer to sidesets that are defined in the specified input_mesh exodus file.



Example of free_surface_sideset defined on the top surface faces of an input mesh

---

Match Sideset Ids

```
Command: match_sidesets     Sidesets ids of matching pairs

Input file command:   match_sidesets <arg>
Command line options: -mss <arg>
Argument Type:        integer(s) >= 0
```

Command Description:

If used with an unstructured base grid (input mesh), this option allows the user to define a crack in the input mesh, where the faces of each vertical side (wall) of the crack are each in a different sideset. The faces at the bottom of the crack share a common edge (V-bottom) or face (square-bottom). Sculpt will match or equalize the volume fractions of the bottom cells on either side of the crack. This produces a uniform, higher quality mesh at the crack. The sidesets must be specified in a pairwise order. This option must be used with the **--input_mesh** (-im) option.

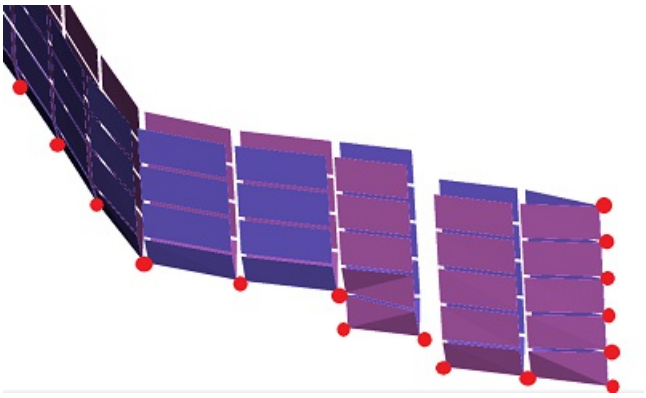
Match Sideset Nodeset Id

```
Command: match_sidesets_nodeset     Nodeset defining match_sidesets

Input file command:   match_sidesets_nodeset <arg>
Command line options: -msn <arg>
Argument Type:        integer >= 0
```

Command Description:

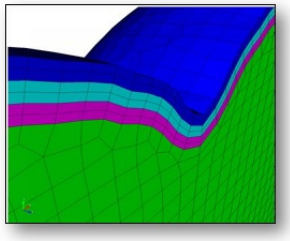
If using the match_sidesets option, this option is used to explicitly define the seam between match_sideset pairs, instead of deriving it. If deriving, the seam is where sideset boundaries share common edges, at the bottom of a crack. But at times the seam can be inadvertently found along the crack wall, if the sideset have been swapped. Explicitly specifing the seam through a nodeset prevents this.



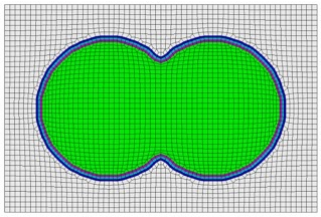
Specifying match_sideset_nodeset to explicitly define crack bottom.

Sculpt Boundary Layers

Sculpt options for defining boundary layers in the mesh. Boundary layers are thin hex layers that can be defined at surfaces, extending either inward or outward from a material. The user may specify the number and thickness of the hex layers as well as the material ID of the layers. Layer thicknesses should normally be "thin" with respect to the size of the cells. Layers will not intersect, so should be defined on surfaces where nearby layers will not overlap. Boundary layers are specified based upon a material ID, where hex layers will be placed at surfaces where the material interfaces with other materials, or at free surfaces.



Example of boundary layers.



Boundary layers defined at the surfaces of a material.

Boundary Layers	-bly	--boundary_layer
--begin	-beg	<arg> Begin specification blayer or blayer_block
--end	-zzz	<arg> End specification blayer or blayer_block
--material	-mat	<arg> Boundary layer material specification
--num_elem_layers	-nel	<arg> Number of element layers in blayer block
--thickness	-th	<arg> Thickness of first element layer in block
--bias	-bi	<arg> Bias of element thicknesses in blayer block

Sculpt Command Summary

Boundary Layer Begin

```
Command: begin     Begin specification blayer or blayer_block

Input file command:   begin <arg>
Command line options: -beg <arg>
Argument Type:        blayer, blayer_block
```

Command Description:

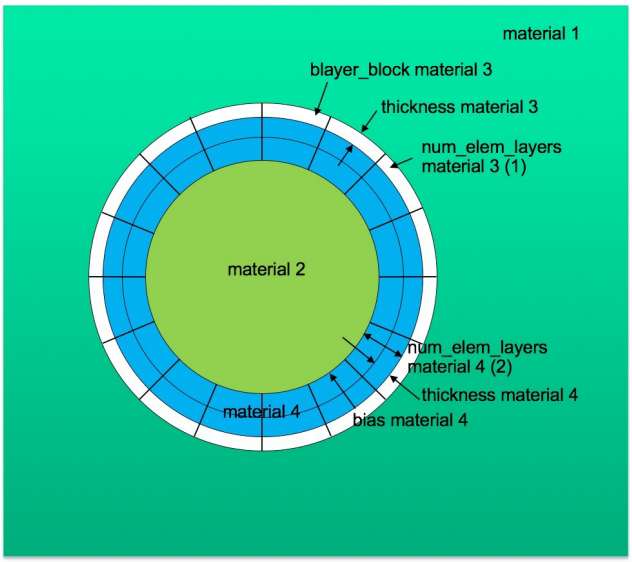
Defines the beginning of a specification block. Must be closed with "end" argument. Currently supports the following specifications:

blayer
Defines a boundary layer specification. Layers of hex elements are placed at the interface of a given material. Valid argumnts used within a blayer specification include: material, and begin blayer_block.

blayer_block
Defines a set of element layers within a given blayer definition that share a common material ID. Valid arguments used within a blayer_block specification include: material, num_elem_layers, thickness and bias.

Example:
The following example shows a boundary layer specification in a sculpt input file. In this example, two boundary layer blocks are defined at the interface of materials 1 and 2. Two material blocks with ID 3 and 4 are generated with 1 and 2 element layers respectively.

```
BEGIN BLAYER
  MATERIAL = 1 2
  BEGIN BLAYER_BLOCK
    MATERIAL = 3
    NUM_ELEM_LAYERS = 1
    THICKNESS = 0.1
  END BLAYER_BLOCK
  BEGIN BLAYER_BLOCK
    MATERIAL = 4
    NUM_ELEM_LAYERS = 2
    THICKNESS = 0.2
    BIAS = 1.3
  END BLAYER_BLOCK
END BLAYER
```



Example schema for boundary layers corresponding to input file below.

Boundary Layer End

```
Command: end     End specification blayer or blayer_block

Input file command:   end <arg>
Command line options: -zzz <arg>
Argument Type:        blayer, blayer_block
```

Command Description:

Defines the end of a specification block. Must be preceded with "begin" argument. Currently supports arguments blayer and blayer_block.

Boundary Layer Material

```
Command: material     Boundary layer material specification

Input file command:   material <arg>
Command line options: -mat <arg>
Argument Type:        integer > 0
```

Command Description:

Defines a material ID in a boundary layer specification. When used within a BLAYER specification, it references one or two existing materials in the

input where boundary layers will be generated. If a single material is specified, hex layers will be generated at all interfaces of the designated material with any adjacent material. If two material IDs are specified, layers will be generated only at interfaces where the two materials are adjacent.

In most cases, the material ID(s) in the BLAYER specification refer to material IDs defined in the diatom file for specific geometry inserts such as STL files or diatom primitives. It can also be defined as the void material ID (VOID_MAT) or a material in a volume fraction description such as input_vfrac, input_micro, input_cart_exo or input_spn.

When used within a BLAYER_BLOCK specification, it refers to a new block that will be generated for which all elements in the blayer_block will be assigned. Normally it refers to a unique material ID that is not already referenced in the input. Where the material ID is already used, elements in the blayer block will be added to the existing material.

A material ID must be defined for both a BLAYER and BLAYER_BLOCK. This value does not have a default.

Number of Element Layers in Boundary Layer

```
Command: num_elem_layers     Number of element layers in blayer block

Input file command:   num_elem_layers <arg>
Command line options: -nel <arg>
Argument Type:        integer > 0
```

Command Description:

Number of element layers to be defined within a BLAYER_BLOCK specification. num_elem_layers must be defined for all BLAYER_BLOCKs.

Boundary Layer Thickness

```
Command: thickness     Thickness of first element layer in block

Input file command:   thickness <arg>
Command line options: -th <arg>
Argument Type:        floating point value
```

Command Description:

Thickness of the first layer defined in a BLAYER_BLOCK. Value is an absolute distance. No default is provided and must be defined for all BLAYER_BLOCKs

---

Boundary Layer Bias

```
Command: bias     Bias of element thicknesses in blayer block

Input file command:   bias <arg>
Command line options: -bi <arg>
Argument Type:        floating point value
```

Command Description:

Bias factor applied to additional layers of a BLAYER_BLOCK. Used in conjunction with the THICKNESS parameter (thickness of first layer) it defines a multiplier for the thickness for subsequent element layers defined within the same BLAYER_BLOCK. Default BIAS is 1.0 and is

optional.

Sculpt Command Summary

Following is a listing of the available input commands to either sculpt or psculpt. When used from the unix command line, commands may be issued using the short form argument, designated with a single dash(-), or with the longer form, designated with two dashes (--). When used in an input file, only the long form may be used, omitting the two dashes (--)

Process Control	-pc	--process
--num_procs	-j	<arg> Number of processors requested
--input_file	-i	<arg> File containing user input data
--debug_processor	-D	<arg> Sleep to attach to processor for debug
--debug_flag	-dbf	<arg> Dump debug info based on flag
--quiet	-qt	Suppress output
--print_input	-pi	Print input values and defaults then stop
--version	-vs	Print version number and exit
--threads_process	-tpp	<arg> Number of threads per process
--iproc	-ip	<arg> Number of processors in I direction
--jproc	-jp	<arg> Number of processors in J direction
--kproc	-kp	<arg> Number of processors in K direction
--build_ghosts	-bg	Write ghost layers to exodus files for debug
--vfrac_method	-vm	<arg> Set method for computing volume fractions
Input Data Files	-inp	--input
--stl_file	-stl	<arg> Input STL file
--diatom_file	-d	<arg> Input Diatom description file
--input_vfrac	-ivf	<arg> Input from Volume Fraction file base name
--input_micro	-ims	<arg> Input from Microstructure file
--input_cart_exo	-ice	<arg> Input from Cartesian Exodus file
--input_spn	-isp	<arg> Input from Microstructure spn file
--spn_xyz_order	-spo	<arg> Ordering of cells in spn file
--compress_spn_ids	-csp	<arg> Compress IDs from SPN file
--input_stitch	-ist	<arg> Input from Stitch file
--stitch_timestep	-stt	<arg> Timestep in Stitch file to read
--stitch_timestep_id	-stn	<arg> Timestep ID in Stitch file to read
--stitch_field	-stf	<arg> Field in Stitch file to read
--stitch_info	-sti	List header info for Stitch file
--lattice	-l	<arg> STL Lattice Template File
Output	-out	--output
--exodus_file	-e	<arg> Output Exodus file base name
--large_exodus	-le	<arg> Output large Exodus file(s)
--volfrac_file	-vf	<arg> Output Volume Fraction file base name
--quality	-Q	<arg> Dump quality metrics to file
--export_comm_maps	-C	Export parallel comm maps to debug exo files
--write_geom	-G	Write geometry associativity file
--write_mbg	-M	Write mesh based geometry file <beta>
--compare_volume	-cv	Report vfrac and mesh volume comparison
--compute_ss_stats	-css	Report sideset statistics
Overlay Grid Specification	-ovr	--overlay
--nelx	-x	<arg> Num cells in X in overlay Cartesian grid
--nely	-y	<arg> Num cells in Y in overlay Cartesian grid
--nelz	-z	<arg> Num cells in Z in overlay Cartesian grid
--xmin	-t	<arg> Min X coord of overlay Cartesian grid
--ymin	-u	<arg> Min Y coord of overlay Cartesian grid
--zmin	-v	<arg> Min Z coord of overlay Cartesian grid
--xmax	-q	<arg> Max X coord of overlay Cartesian grid
--ymax	-r	<arg> Max Y coord of overlay Cartesian grid
--zmax	-s	<arg> Max Z coord of overlay Cartesian grid
--cell_size	-cs	<arg> Cell size (nelx, nely, nelz ignored)
--align	-a	Automatically align geometry to grid
--bbox_expand	-be	<arg> Expand tight bbox by percent
--input_mesh	-im	<arg> Input Base Exodus mesh
--input_mesh_blocks	-imb	<arg> Block ids of Input Base Exodus mesh
--input_mesh_material	-imm	<arg> Material definition with input mesh
--input_mesh_pamgen	-imp	<arg> Input Base mesh defined by Pamgen
--join_parallel	-jp	<arg> Join parallel files
Mesh Type	-typ	--type
--stair	-str	<arg> Generate Stair-step mesh
--mesh_void	-V	<arg> Mesh void
--trimesh	-tri	Generate tri mesh of geometry surfaces
--tetmesh	-tet	<arg> Under Development
--deg_threshold	-dg	<arg> Convert hexes below threshold to degenerates
--max_deg_iters	-dgi	<arg> Maximum number of degenerate iterations
--htet	-ht	<arg> Convert hexes below quality threshold to tets
--htet_method	-hti	<arg> Method used for splitting hexes to tets
--htet_material	-htm	<arg> Convert hexes in given materials to tets
--htet_transition	-htt	<arg> Transition method between hexes and tets
--htet_pyramid	-htp	<arg> Local transition pyramid
--htet_tied_contact	-htc	<arg> Local transition tied contact

```
  --htet_no_interface          -htn  <arg> Local transition none

  --periodic                   -per        Generate periodic mesh
  --check_periodic             -cp   <arg> Check for periodic geometry
  --check_periodic_tol         -cpt  <arg> Tolerance for checking periodicity
  --periodic_axis              -pax  <arg> Axis periodicity is about
  --periodic_nodesets          -pns  <arg> Nodesets ids of primary/secondary (leading/trailing) nodesets

Boundary Conditions            -bc     --boundary_condition
  --void_mat                   -VM   <arg> Void material ID (when mesh_void=true)
  --separate_void_blocks       -SVB        Separate void into unique block IDs
  --material_name              -mn   <arg> Label Material (Block) with Name
  --sideset_name               -sn   <arg> Label Sideset with Name
  --nodeset_name               -nn   <arg> Label Nodeset with Name
  --sideset                    -sid  <arg> User Defined Sideset
  --nodeset                    -nid  <arg> User Defined Nodeset
  --gen_sidesets               -SS   <arg> Generate sidesets
  --free_surface_sideset       -FS   <arg> Free Surface Sideset
  --match_sidesets             -mss  <arg> Sidesets ids of matching pairs
  --match_sidesets_nodeset     -msn  <arg> Nodeset defining match_sidesets

Adaptive Meshing               -adp    --adapt
  --adapt_type                 -A    <arg> Adaptive meshing type
  --adapt_threshold            -AT   <arg> Threshold for adaptive meshing
  --adapt_levels               -AL   <arg> Number of levels of adaptive refinement
  --adapt_material             -AM   <arg> Info for adapting material
  --adapt_export               -AE        Export exodus mesh of refined grid
  --adapt_non_manifold         -ANM       Refine at non-manifold conditions
  --adapt_load_balance         -ALB       Adaptive parallel load balancing
  --adapt_memory_stats         -AMS       Write memory usage stats for adaptivity

Smoothing                      -smo    --smoothing
  --smooth                     -S    <arg> Smoothing method
  --csmooth                    -CS   <arg> Curve smoothing method
  --laplacian_iters            -LI   <arg> Number of Laplacian smoothing iterations
  --max_opt_iters              -OI   <arg> Max. number of parallel Jacobi opt. iters.
  --opt_threshold              -OT   <arg> Stopping criteria for Jacobi opt. smoothing
  --curve_opt_thresh           -COT  <arg> Min metric at which curves won't be honored
  --max_pcol_iters             -CI   <arg> Max. number of parallel coloring smooth iters.
  --pcol_threshold             -CT   <arg> Stopping criteria for parallel color smooth
  --max_gq_iters               -GQI  <arg> Max. number of guaranteed quality smooth iters.
  --gq_threshold               -GQT  <arg> Guaranteed quality minimum SJ threshold
  --geo_smooth_max_deviation   -GSM  <arg> Geo Smoothing Maximum Deviation

Mesh Improvement               -imp    --improve
  --pillow                     -p    <arg> Set pillow criteria (1=surfaces)
  --pillow_surfaces            -ps         Turn on pillowing for all surfaces
  --pillow_curves              -pcv        Turn on pillowing for bad quality at curves
  --pillow_boundaries          -pb         Turn on pillowing at domain boundaries
  --pillow_curve_layers        -pcl  <arg> Number of elements to buffer at curves
  --pillow_curve_thresh        -pct  <arg> S.J. threshold to pillow hexes at curves
  --pillow_smooth_off          -pso        Turn off smoothing following pillow operations
  --capture                    -c    <arg> Project to facet geometry <beta>
  --capture_angle              -ca   <arg> Angle at which to split surfaces <beta>
  --capture_side               -sc   <arg> Project to facet geometry with surface ID
  --defeature                  -df   <arg> Apply automatic defeaturing
  --min_vol_cells              -mvs  <arg> Minimum number of cells in a volume
  --defeature_bbox             -dbb        Defeature Filtering at Bounding Box
  --defeature_iters            -dfi  <arg> Maximum Number of Defeaturing Iterations
  --thicken_material           -thm  <arg> Expand a given material into surrounding cells
  --thicken_void               -thv  <arg> Insert void material to remove overlap
  --micro_expand               -me   <arg> Expand Microstructure grid by N layers
  --micro_shave                -ms         Remove isolated cells at micro. boundaries
  --remove_bad                 -rb   <arg> Remove hexes with Scaled Jacobian < threshold
  --wear_method                -wm   <arg> Method for removing void at free surface
  --crack_min_elem_thickness   -cmet <arg> Minimum element thickness in crack
  --min_num_layers             -mnl  <arg> Minimum number of layers to keep using wear_method=2

Mesh Transformation            -tfm    --transform
  --xtranslate                 -xtr  <arg> Translate final mesh coordinates in X
  --ytranslate                 -ytr  <arg> Translate final mesh coordinates in Y
  --ztranslate                 -ztr  <arg> Translate final mesh coordinates in Z
  --xscale                     -xsc  <arg> Scale final mesh coordinates in X
  --yscale                     -ysc  <arg> Scale final mesh coordinates in Y
  --zscale                     -zsc  <arg> Scale final mesh coordinates in Z

Boundary Layers                -bly    --boundary_layer
  --begin                      -beg  <arg> Begin specification blayer or blayer_block
  --end                        -zzz  <arg> End specification blayer or blayer_block
  --material                   -mat  <arg> Boundary layer material specification
  --num_elem_layers            -nel  <arg> Number of element layers in blayer block
  --thickness                  -th   <arg> Thickness of first element layer in block
  --bias                       -bi   <arg> Bias of element thicknesses in blayer block
```

Sculpt Mesh Improvement

Sculpt options for modifying the mesh to improve mesh quality.

Automatic smoothing provides an effective method for improving element quality. However there may be some cases that cannot be improved with smoothing alone. The options included in this section will apply changes to the underlying hex mesh or to the volume fraction data to increase the opportunity for smoothing to produce a good quality mesh.

```
Mesh Improvement                -imp     --improve
  --pillow                      -p    <arg> Set pillow criteria (1=surfaces)
  --pillow_surfaces             -ps         Turn on pillowing for all surfaces
  --pillow_curves               -pcv        Turn on pillowing for bad quality at curves
  --pillow_boundaries           -pb         Turn on pillowing at domain boundaries
  --pillow_curve_layers         -pcl  <arg> Number of elements to buffer at curves
  --pillow_curve_thresh         -pct  <arg> S.J. threshold to pillow hexes at curves
  --pillow_smooth_off           -pso        Turn off smoothing following pillow operations
  --capture                     -c    <arg> Project to facet geometry <beta>
  --capture_angle               -ca   <arg> Angle at which to split surfaces <beta>
  --capture_side                -sc   <arg> Project to facet geometry with surface ID
  --defeature                   -df   <arg> Apply automatic defeaturing
  --min_vol_cells               -mvs  <arg> Minimum number of cells in a volume
  --defeature_bbox              -dbb        Defeature Filtering at Bounding Box
  --defeature_iters             -dfi  <arg> Maximum Number of Defeaturing Iterations
  --thicken_material            -thm  <arg> Expand a given material into surrounding cells
  --thicken_void                -thv  <arg> Insert void material to remove overlap
  --micro_expand                -me   <arg> Expand Microstructure grid by N layers
  --micro_shave                 -ms         Remove isolated cells at micro. boundaries
  --remove_bad                  -rb   <arg> Remove hexes with Scaled Jacobian < threshold
  --wear_method                 -wm   <arg> Method for removing void at free surface
  --crack_min_elem_thickness    -cmet <arg> Minimum element thickness in crack
  --min_num_layers              -mnl  <arg> Minimum number of layers to keep using wear_method=2
```

Sculpt Command Summary

Pillow

```
Command: pillow     Set pillow criteria (1=surfaces)

Input file command:   pillow <arg>
Command line options: -p <arg>
Argument Type:        integer (0, 1, 2, 3)
Input arguments: off (0)
                 surfaces (1)
                 curves (2)
                 domain_boundaries (3)
                 surfaces_no_smoothing (100)
                 curves_2_layers (212)
                 curves_3_layers (213)
                 curves_4_layers (214)
                 curves_5_layers (215)
                 curves_2_layers_no_smoothing (202)
                 curves_3_layers_no_smoothing (203)
                 curves_4_layers_no_smoothing (204)
                 curves_5_layers_no_smoothing (205)
```

Command Description:

For models that have more than one material that share an interface, unless the geometry is precisely aligned with the global axis, it is usually a good idea to turn on pillowing. Pillowing automatically inserts an additional layer of hexes at interface boundaries to improve mesh quality. Without pillowing you may notice inverted or poor quality elements at curve interfaces where 2 or more materials meet.

The pillow option will generate an additional layer of hexes at surfaces as a means to improve element quality near curve interfaces. This is intended to eliminate the problem of 3 or more nodes from a single hex face lying on the same curve. Use one or more of the following options to set up pillowing:

- **pillow_surfaces:** Pillow around all surfaces
- **pillow_curves:** Pillow bad quality at curves
- **pillow_boundaries:** Pillow at domain boundaries
- **pillow_curve_layers:** Number of element layers to buffer curves
- **pillow_smooth_off:** Turn OFF smoothing following pillow operations

See help on the above options for more information

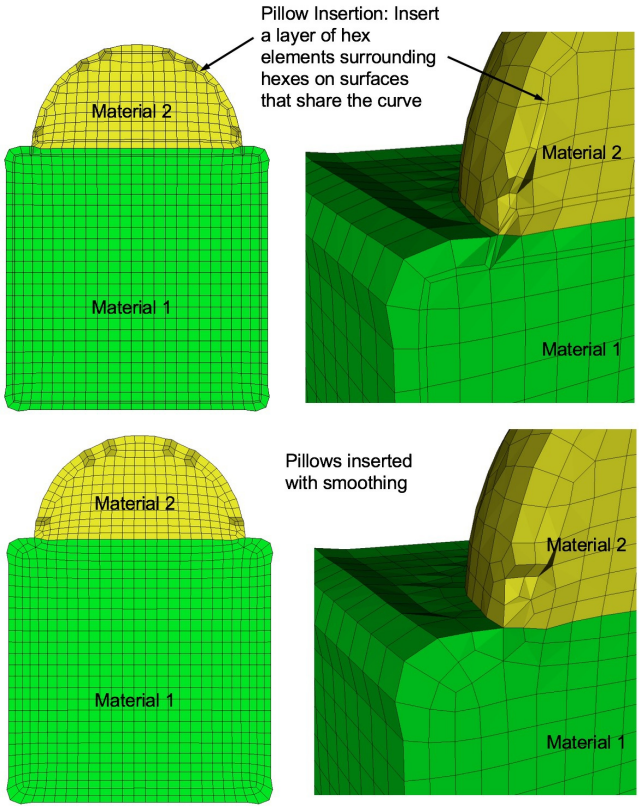
Pillow All Surfaces

```
Command: pillow_surfaces     Turn on pillowing for all surfaces

Input file command:   pillow_surfaces
Command line options: -ps
```

Command Description:

Pillow option to insert a layer of hexes surrounding each internal surface in the mesh. Where two volumes share a common interface is defined as a surface. All hexes that have at least one of its faces on a surface are defined as the "shrink set" of hexes. A separate shrink set is defined for each unique surface. Hexes in the set are shrunk away from their hex neighbors not in the shrink set. A layer of hexes is then inserted surrounding all hexes in each set. This enforces the condition where no more than one hex edge will lie on any single curve thus allowing more freedom for the smoother to improve element quality.



Example of surface pillowing, before and after smoothing

Surface pillowing is off by default. If both **pillow_curves** and **pillow_surfaces** options are used, curve pillowing will be performed before surface pillowing.

See the **pillow** option for more information on setting additional options for pillowing.

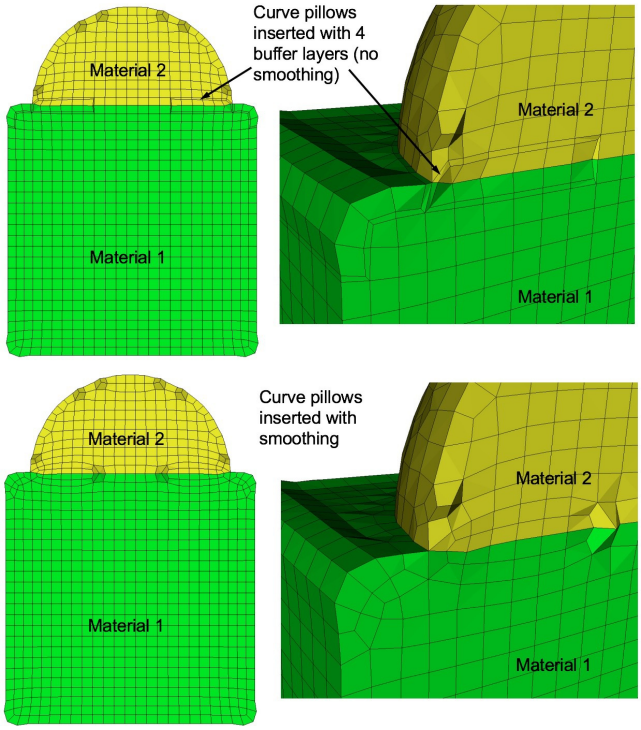
Pillow Bad Quality at Curves

```
Command: pillow_curves     Turn on pillowing for bad quality at curves

Input file command:   pillow_curves
Command line options: -pcv
```

Command Description:

Pillow option to selectively pillow hexes at curves. Only hexes that have faces with 3 or more nodes on a curve will be pillowed. Additional buffer layers of hexes beyond the poor quads at the curves will be included in the pillow region. The number of buffer layers beyond the curve can be controlled with the **pillow_curve_layers**, where the default will be 3 layers.



Example of curve pillowing with four pillow_curve_layers, before and after smoothing

Curve pillowing is off by default. If both **pillow_curves** and **pillow_surfaces** options are used, curve pillowing will be performed before surface pillowing.

See the **pillow** option for more information on setting additional options for pillowing.

Pillow at Domain Boundaries

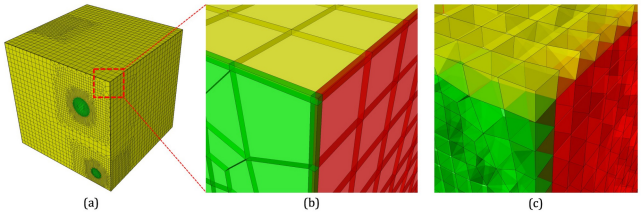
```
Command: pillow_boundaries     Turn on pillowing at domain boundaries

Input file command:   pillow_boundaries
Command line options: -pb
```

Command Description:

Pillow option to insert pillow layers at domain boundaries of the initial Cartesian grid definition. One layer of hexes is inserted on each of the six faces of the Cartesian Domain. This option is useful where the void option is used to generate a mesh in the full Cartesian grid and where the adapt option has been used. Without this option, it is likely that hexes with two faces on the same domain boundary will occur if the adaptation extends to the boundary. Turning on the **pillow_boundaries** option

should correct for these cases.



Example of pillowing at boundaries on a microstructure RVE. (b) before smoothing (c) after smoothing

Boundary pillowing is off by default. The **pillow_boundaries** option may be used in the same input as **pillow_surfaces** or **pillow_curves**. The **pillow_boundaries** option must also be used with the **mesh_void** option to ensure hexes will exist at the Cartesian domain boundary. See the **pillow** option for more information on setting additional options for pillowing.

Number of Element Layers to Buffer Curves

```
Command: pillow_curve_layers     Number of elements to buffer at curves

Input file command:   pillow_curve_layers <arg>
Command line options: -pcl <arg>
Argument Type:        integer > 0
```

Command Description:

Used for setting the number of buffer hex layers when the **pillow_curves** option is used. When **pillow_curves** is used a shrink set is formed from hexes that would otherwise have two or more edges on the same curve. This value will control the extent to which neighboring hexes will be included in the shrink set. The default **pillow_curve_layers** is 3. Setting this value lower will localize the modifications to the hex mesh, whereas, more layers will extend the region that is affected in correcting the poor quality at curves.

Scaled Jacobian Threshold for Curve Pillowing

```
Command: pillow_curve_thresh     S.J. threshold to pillow hexes at curves

Input file command:   pillow_curve_thresh <arg>
Command line options: -pct <arg>
Argument Type:        floating point value (-1.0->1.0)
```

Command Description:

Used for setting the quality threshold for pillowing hexes at curves. When determining hexes to include in the shrink set, the **pillow_curves** option will look for hexes with more than two nodes of a hex on the same curve. If this condition is satisfied, it will test the mesh quality of quads on the adjacent surfaces that share the common curve. If at least 3 nodes are on a common curve and the Scaled Jacobian of any of the attached quads falls below the, **pillow_curve_thresh** scaled Jacobian metric, then the associated hexes will be included in the shrink set.

Default for **pillow_curve_thresh** is 0.3. Increasing this value will tend to increase the total number of hexes added to the mesh, but may result in better mesh quality after smoothing. Lowering this value may reduce the number of additional hexes but could potentially result in more hexes with poor or bad Scaled Jacobian metrics.

Turn OFF Smoothing Following Pillow Operations

```
Command: pillow_smooth_off     Turn off smoothing following pillow operations

Input file command:   pillow_smooth_off
Command line options: -pso
```

Command Description:

Controls the smoothing following pillow operations. To maximize element quality at pillowed hexes, smoothing is always performed after inserting the hex layers. The smoothing step may be omitted if **pillow_smooth_off** is set. This option can be useful for visualizing the pillow layers that have been inserted, but in most cases will generate poor quality or inverted elements.

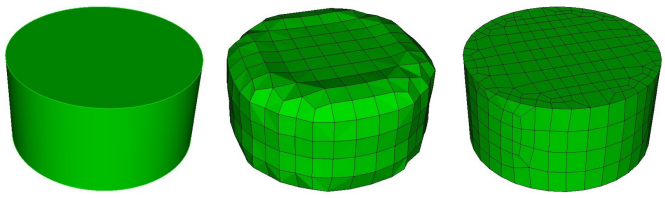
Capture

```
Command: capture     Project to facet geometry <beta>

Input file command:   capture <arg>
Command line options: -c <arg>
Argument Type:        integer (0, 1, 2)
Input arguments: off (0)
                 on (1)
                 external_surfaces (2)
                 projections_only (3)
                 feature_angle_smooth (4)
                 topology_smooth (5)
```

Command Description:

This is an experimental option still in development. Nodes at the surfaces of a default sculpt mesh will not necessarily exactly lie on the geometric surfaces prescribed by the input STL geometry. While this characteristic can provide additional flexibility for defeaturing and element quality, there are cases where a more exact surface representation may be desired. The capture option attempts to address this by extracting sharp features and/or projecting nodes to the facet geometry.



Simple example illustrating the effect of the **capture = 5** option. Options **smooth = to_geometry** and **pillow_curves = true** are also used for this example.

Several options are currently being studied as possible solutions. They include the following:

0 = (off) Capture option is off. No attempt is made at capturing sharp features.

1 = (on) STL geometry is used as basis for feature capture. A user defined feature angle is used (capture_angle) to first generate groups of facets from the STL geometry based on capture_angle. Topological curves are defined based on projections to closest surface facets and edges. With default smoothing option, the surface nodes will be projected to the closest STL surfaces as a final step before exporting the exodus mesh. Consider using **smooth = to_geometry** option.

2 = (exterior_surfaces) Only exterior surfaces are captured. Uses the same procedure as described in capture = 1, except that interior surfaces (those with two adjacent volumes), will be ignored in the capture and projections stage.

3 = (projections_only) For this option, additional topology based on feature angle is not extracted. Only the final projection of surface nodes to the STL facets is done. Note that this option is useful for organic shapes that do not have sharp features, or where sharp features should be ignored.

4 = (feature_angle_smooth) This option uses the procedure outlined in **capture = 1**, except that the **smooth = to_geometry** is used by default. Note that **capture = 1** used with **smooth = to_geometry** should be identical to this option.

5 = (topology_smooth) Curve topology is defined similar to **capture = 1**, except that element face topology is first determined based on closest assigned facet. Curve topology is then extracted based on adjacent element face associativity. Surface node projections are only done for nodes that have unambiguous neighbor associativity. This provides for a tolerant approach to resolving topology that may result in defeaturing. (i.e. where the STL facet topology may be locally more complex than can be resolved by the prescribed resolution). This option also uses the **smooth = to_geometry** option as default for smoothing. Also note that **capture = 5** it is only currently available for serial execution (j=1)

Capture Angle

```
Command: capture_angle     Angle at which to split surfaces <beta>

Input file command:   capture_angle <arg>
Command line options: -ca <arg>
Argument Type:        floating point value (0 -> 360)
```

Command Description:

This is an experimental option still in development. Feature angle for capture option.

Capture Side

```
Command: capture_side     Project to facet geometry with surface ID

Input file command:   capture_side <arg>
Command line options: -sc <arg>
Argument Type:        integer > 0
```

Command Description:

Similar to the capture option, the capture_side option will project nodes to the initial triangle facets, however projections will be limited only to surface nodes closest to the surface ID specified by the argument. Note that the input STL file can identify and group facets according to a surface ID. However surface IDs are utilized only when using the gen_sidesets option with arguments 3 and 4. When using Cubit, the STL file written when using the sculpt parallel command with sideset options 3 and 4 will include surface identification for surfaces in the STL file. A workflow for using the capture_side option might include the following:

1. Generate or import CAD model into Cubit.
2. Identify and group selected surfaces into a single sideset with unique ID.
3. In the Sculpt GUI set the sideset generation option to 4.
4. Turn on the option: Export Run Files Only.

5. In your working directory edit the input file (.i).
6. Add the option capture_side = <id> to the input file.
7. Run sculpt in batch using the input file to control execution.

The result should be a mesh where surface nodes closest to the surfaces identified by the unique sideset ID will lie precisely on their closest surface.

Defeature

```
Command: defeature     Apply automatic defeaturing

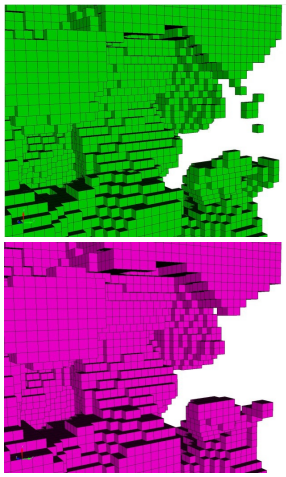
Input file command:   defeature <arg>
Command line options: -df <arg>
Argument Type:        integer (0, 1, 2, 3)
Input arguments: off (0)
                 filter (1)
                 collapse (2)
                 filter_and_collapse (3)
```

Command Description:

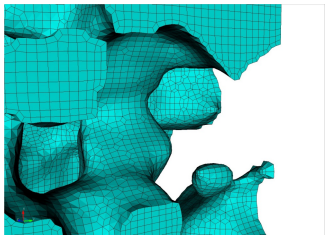
Option to automatically detect and remove small features. Primarily used for defeaturing microstructure data, however can be used with any input format. The following options are available:

- **off (0):** No defeaturing performed (default)
- **filter (1):** Filters the Cartesian grid data so that groupings of cells of a common material with less than **min_vol_cells** will be reassigned to the predominant neighboring material. If the **min_vol_cells** argument is not specified, the minimum number of cells in a volume will be set to 5. This has the effect of removing small volumes that would otherwise be generated. This option will also remove protrusions, where a cell surrounded on 4 or 5 sides by another material ID will be reassigned to the predominant neighboring material. This option is available with multiple processors.

  See also the **defeature_iters** and **defeature_bbox** options for additional control of the **defeature = filter** option. The **compare_volume** option can also be used to validate that changes made to material volumes are within acceptable limits.

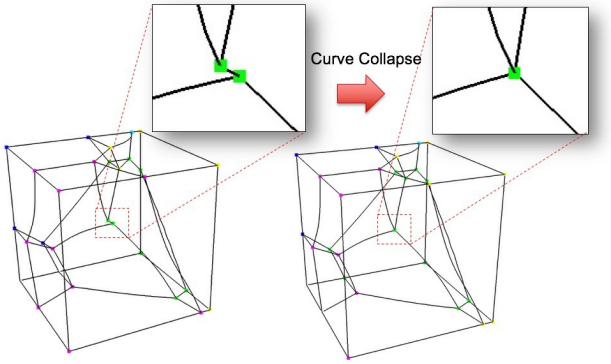


Example grid cells before and after defeaturing has been applied



Final mesh after using defeaturing.

- **collapse (2):** Curve and surface collapses are performed. This option is only available when used with the **trimesh** option. After geometry has been extracted and built from the volume fraction data curves containing exactly one mesh edge are collapsed into a single vertex. Surfaces that are identified with exactly 2 curves, each of which have 2 mesh edges are collapsed into a single curve. Only available as serial option (-j 1)



Example collapsing of small curve on microstructure model when using defeature=2 and trimesh option

- **filter_and_collapse (3):** Performs both option **filter (1)** and **collapse (2)** on a **trimesh**. Only available as serial option (-j 1)

Minimum Number of Cells in a Volume

```
Command: min_vol_cells     Minimum number of cells in a volume

Input file command:   min_vol_cells <arg>
Command line options: -mvs <arg>
Argument Type:        integer >= 0
```

Command Description:

When used with **defeature** options **filter (1)** or **filter_and_collapse (3)**, specifies the minimum number of cells below which a volume will be eliminated. The cells of small volumes will be absorbed into the predominant material of the neighboring cells. If not specified and defeature options **filter (1)** or **filter_and_collapse (3)** are used, the **min_vol_cells** value will be set to 5.

Defeature at Bounding Box

```
Command: defeature_bbox     Defeature Filtering at Bounding Box

Input file command:   defeature_bbox
Command line options: -dbb
```

Command Description:

The **defeature_bbox** option is used in conjunction with **defeature = filter (1)**. It is used to modify the defeature filter criteria at cells that are

immediately adjacent to the Cartesian grid's domain boundary. It is most effective for microstructure data but can be used with any input format. The **defeature = filter (1)** option will remove protrusions identified by cells that are surrounded on 4 or 5 sides by another material. For cells that are at the domain boundary, cells will have missing adjacent cells on at least one face. If the **defeature_bbox=true** option is used, the missing adjacent cells are considered a different material and counted in the 4 or 5 surrounding cells with a different material. In contrast, the **defeature_bbox=false** option will not count the missing adjacent cells. Using the **defeature_bbox=true** has the effect of more aggressively modifying cells at the domain boundaries to avoid protrusions. The default for this option is **defeature_bbox=false**. It will be ignored if **defeature = filter (1)** is not used.

Maximum Number of Defeature Iterations

```
Command: defeature_iters     Maximum Number of Defeaturing Iterations

Input file command:   defeature_iters <arg>
Command line options: -dfi <arg>
Argument Type:        integer >=0
```

Command Description:

Used with the **defeature** option. Controls the maximum number of iterations of defeature filtering that will be performed. Setting this value greater than the default of 10 can be useful for very noisy data where a significant number of iterations will need to be performed to resolve the geometry.

When performing non-manifold resolution, the defeature state of some of the cells may be effected. As a result, the defeaturing and non-manifold resolution procedures are performed in a loop until no further changes can be made. The **defeature_iters** sets the maximum number of defeature and non-manifold resolution procedures that will be performed. Note that if defeaturing reaches the maximum iteration value without completely resolving all non-manifold conditions, that subsequent sculpt procedures may not succeed. Set this value higher to allow the defeaturing and non-manifold resolution to run to completion. The **stair = 1** option can be used to interrogate the model to see where non-manifold conditions may still exist.

Thicken a material

```
Command: thicken_material     Expand a given material into surrounding cells

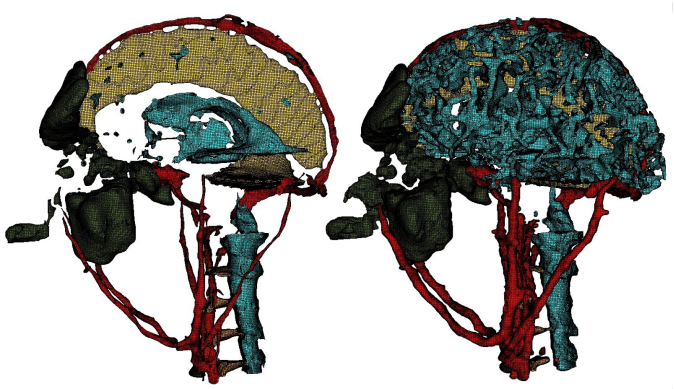
Input file command:   thicken_material <arg>
Command line options: -thm <arg>
Argument Type:        integer >= 0 floating point value (0.0->1.0)
```

Command Description:

Add additional cells at the boundary of a given material. Takes two input values, a material and a volume fraction between 0 and 1. This option is useful for noisy input data that may not form contiguous volumes. Thickening a material may close small gaps making the material continuous. To perform the thicken operation, cells in adjacent materials are removed and reassigned to the indicated material. This option requires both a valid material ID and volume fraction value, where the volume fraction represents the amount of material to be added to each neighboring cell. For example:

thicken material = 1 0.2
thicken_material = 2 0.5

each neighboring cell to material 1 will change approximately 20 percent of its volume to be material 1. Other materials present in the cell will be decreased accordingly to maintain a sum of 1.0 for each cell. Additional material is accumulated in neighboring cells from each adjacent cell it shares with material 1, so that if for example a neighbor cell shares faces with three cells of material 1, it will add 0.6 (0.2 X 3) of material 1 volume fraction to the neighbor. If more than one **thicken_material** option is used, the thicken operation will be performed in the order they appear in the input. For the above example, material 1 would first be thickened, followed by material 2. If materials 1 and 2 are adjacent, thickening in this case, material 2 would take precedence, potentially removing cells from material 1 at their interface.



Bitmap input is used on a Cartesian base grid to generate the mesh for complex head and brain anatomy. Left: Some of the materials prior to applying the thicken_material option. Right: After applying the thicken_material option.

Thicken void material

```
Command: thicken_void     Insert void material to remove overlap

Input file command:   thicken_void <arg>
Command line options: -thv <arg>
Argument Type:        floating point value (0.0->1.0)
```

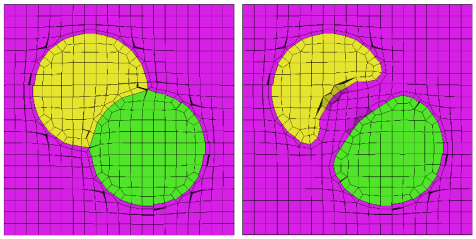
Command Description:

Add additional void material when non-void material is detected as touching or immediately adjacent. Takes one input value, a volume fraction normally about 1.0 that indicates the quantity of volume fraction inserted at each node if the input grid where non-void material adjacency is detected. A value of 1.0 indicates void material equal to the volume of one cell will be added at the nodes, reducing the volume fractions of other materials present in adjacent cells. Smaller input values will generate a smaller gap between materials, but can run the risk of materials bleeding into one another.

This option is useful when it is known that non-void materials in the model should not touch, instead should have a gap where they would otherwise touch or overlap. For example:

thicken_void = 1.0

each node where its adjacent cells have two or more non-void materials present will have additional void material added. In this case, if 8 adjacent cells are assumed, a contribution of 1/8 void volume fraction will be added to each adjacent cell to the node. Other materials present in the cells will be decreased accordingly to maintain a sum of 1.0 for each cell.



Left: Initial mesh without **thicken_void**. Right: Mesh with **thicken_void=1**. Void material (Magenta elements) is inserted between the yellow and green materials to ensure separation.

Microstructure Expansion

```
Command: micro_expand     Expand Microstructure grid by N layers

Input file command:   micro_expand <arg>
Command line options: -me <arg>
Argument Type:        integer >= 0
```

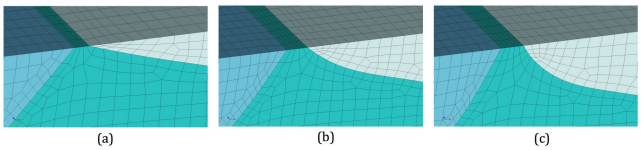
Command Description:

This option expands the Cartesian grid by a specified number of layers. It can be used with any of the following input options:

```
--input_micro
--input_cart_exo
--input_spn
```

In some cases the interior material interfaces may intersect the domain boundaries at small acute angles. When this occurs it may be difficult or impossible to achieve computable mesh quality at these intersections. To address this problem, one or more layers of hexes may be added to the Cartesian grid. The volume fractions from cells at the boundary are copied to generate additional layers. This has the effect of increasing the angle of intersection for any material interfaces intersecting the domain boundary. Usualy a value of 1 or 2 is sufficient to sufficiently improve quality.

Note that the resulting mesh in the expanded layers serves only to improve mesh quality and will only duplicate existing data at the boundaries. It may not reflect the actual material structure within the expansion layers.



(a) Initial mesh (b) One expansion layer added (c) Two expansion layers added

---

Microstructure Shave

```
Command: micro_shave     Remove isolated cells at micro. boundaries

Input file command:   micro_shave
Command line options: -ms
```

Command Description:

This option potentially modifies the outermost layer of Cartesian cells of a microstructures file. It will identify isolated cells where the assigned material is unique from all of its surrounding cells at the boundary. When

this occurs, the cell material is reassigned to the dominant nearby material.

This option is useful if it is noted that a cell structure just barely grazes the exterior planar boundary surface. Poor quality elements can often result with this condition. The micro_shave option will, in effect, remove material from the cell structure, but will result in better quality elements by removing the intersection region with the boundary.

micro_shave can be used with any of the following input options:

```
    --input_micro
    --input_cart_exo
    --input_spn
```

Remove bad elements below threshold

```
Command: remove_bad     Remove hexes with Scaled Jacobian < threshold

Input file command:   remove_bad <arg>
Command line options: -rb <arg>
Argument Type:        floating point value -1.0 >= 1.0
```

Command Description:

Remove hexes below the specified scaled Jacobian metric.

Method for removing void at free surface

```
Command: wear_method     Method for removing void at free surface

Input file command:   wear_method <arg>
Command line options: -wm <arg>
Argument Type:        integer (0, 1, 2)
Input arguments: cell (0)
                 cell (1)
                 sheet (2)
```

Command Description:

This option defines how the mesh removes void elements that are at the **free_surface_sideset**. Used with the **free_surface_sideset**, **input_mesh** and **capture=5** options. Normally with the default **wear_method = cell (0)** option, elements in the input_mesh outside of the specified **free_surface_sideset** are removed when the percent of its volume (volume fraction) lying outside of the free_surface_sideset exceeds 50 percent. If the **wear_method = sheet (2)** option is set, volume fraction of continuous layers of elements (sheets) that lie at the **free_surface_sideset** can be removed. The entire sheet of elements will be retained or removed based on the total volume fraction of the elements in the sheet.

The **wear_method = sheet (2)** option is most useful when used with a swept **input_mesh** where the **free_surface_sideset** is approximately orthogonal to the sweep direction. It can especially improve mesh quality when using the **capture=5** option where small geometric features, such as cracks, are encountered near the **free_surface_sideset**. See also the **crack_min_elem_thickness** option, to control how cracks are captured at the free_surface_sideset.

Minimum element thickness of crack

```
Command: crack_min_elem_thickness     Minimum element thickness in crack
```

```
Input file command:   crack_min_elem_thickness <arg>
Command line options: -cmet <arg>
Argument Type:        floating point value
```

Command Description:

Defines the minimum allowed thickness of the elements resolving the side of a crack. Used with the **wear_method = sheet (2)** and **match_sideset** options to potentially improve mesh quality at cracks near the **free_surface_sideset**. Cracks are normally identified using the **match_sideset** and **match_sidesets_nodset** options. The distance from the bottom of the crack to the **free_surface_sideset** is measured to determine the element thickness. If the thickness is below the specified **crack_min_elem_thickness** value, the crack walls are merged together at this location. If a **crack_min_elem_thickness** is not specified, cracks near the **free_surface_sideset** will only be collapsed when the surrounding volume fraction of the sheet drops below 50 percent.

Minimum number of layers to keep using wear_method=2

```
Command: min_num_layers     Minimum number of layers to keep using wear_method=2

Input file command:   min_num_layers <arg>
Command line options: -mnl <arg>
Argument Type:        integer >= 0
```

Command Description:

Defines the minimum number of swept layers of the reference mesh to retain when using sculpt with the **wear_method = sheet (2)**. These layers will be retained even if their volume fraction indicates that they should be discarded.

Sculpt Input Data Files

Options for specifying input files to Sculpt. Sculpt uses a method for representing geometry based upon volume fractions defined on a Cartesian or unstructured grid. Sculpt will accept facet-based (STL) or analytic (diatom) geometry, but will first convert the input geometry to the required volume fraction description before generating the hexahedral mesh. Various formats for volume fraction data can also be imported directly into Sculpt and used as the basis for hex meshing. The following formats for geometry are currently supported in Sculpt:

- STL (stl_file)
- Diatom (diatom_file)
- Volume Fractions
  - Exodus element variables (import_vfrac)
  - Exodus blocks (import_cart_exo)
  - Ascii Voxel Data (import_spn)
  - Ascii Volume Fraction Data (import_micro)

```
Input Data Files              -inp     --input
  --stl_file                  -stl  <arg> Input STL file
  --diatom_file               -d    <arg> Input Diatom description file
  --input_vfrac               -ivf  <arg> Input from Volume Fraction file base name
  --input_micro               -ims  <arg> Input from Microstructure file
  --input_cart_exo            -ice  <arg> Input from Cartesian Exodus file
  --input_spn                 -isp  <arg> Input from Microstructure spn file
  --spn_xyz_order             -spo  <arg> Ordering of cells in spn file
  --compress_spn_ids          -csp  <arg> Compress IDs from SPN file
  --input_stitch              -ist  <arg> Input from Stitch file
  --stitch_timestep           -stt  <arg> Timestep in Stitch file to read
  --stitch_timestep_id        -stn  <arg> Timestep ID in Stitch file to read
  --stitch_field              -stf  <arg> Field in Stitch file to read
  --stitch_info               -sti        List header info for Stitch file
  --lattice                   -l    <arg> STL Lattice Template File
```

Sculpt Command Summary

STL File

```
Command: stl_file     Input STL file

Input file command:   stl_file <arg>
Command line options: -stl <arg>
Argument Type:        file name with path
```

Command Description:

File name of a single STL (facet geometry) file to be used as input. Either an stl_file or diatom_file designation should be included to run Sculpt. The stl_file option will support a single STL file. To use multiple STL files, where each file represents a different material, use the diatom_file file option where multiple file names may be specified.

It is recommended that STL files used as input to Sculpt be "water-tight". While in many cases non-watertight geometries will be successful, unexpected or incorrect results may result. It is recommended practice to use Cubit to first import the STL geometry and allow the sculpt parallel command to write a new STL geometry file for use in Sculpt. Cubit's sculpt parallel command will attempt to stitch and repair any triangle facets that are not completely closed. Other commercial tools are available for STL geometry that may be effective in repairing the geometry prior to use in Sculpt.

Diatom File

```
Command: diatom_file     Input Diatom description file

Input file command:   diatom_file <arg>
Command line options: -d <arg>
Argument Type:        file name with path
```

Command Description:

File name of a diatom file to be used as input to Sculpt. Both stl_file and diatom_file cannot be used simultaneously. A diatom file is a constructive solid geometry description containing primitives for generating a full geometric definition of the model. Diatoms are commonly used as input to Sandia's CTH and Alegra codes. Multiple STL files can also be defined in a Diatom file. The following is a simple example of a diatom file that would read 3 different STL files:

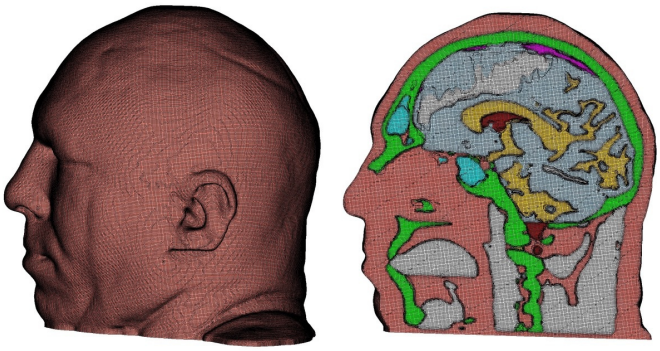
```
diatoms
  package 'blue_material'
    material 1
    insert stl
      file = 'blue_part1.stl'
    endinsert
    insert stl
      file = 'blue_part2.stl'
    endinsert
  endpackage
  package 'red_material'
    material 2
    insert stl
      file = 'red_part1.stl'
    endinsert
  endpackage
enddiatom
```

Note that the first two files, blue_part1.stl and blue_part2.stl belong to the same material. As a result, elements generated within the geometry of these files will belong to block 1. Likewise, the elements generated within the geometry of red_part1.stl will belong to block 2.

Bitmap Files

The Diatom format will also support bitmap files. These are binary files that set each cell either on or off for the specified material. The following is an example diatom specification for a bitmap file. Note that the bitmap specification includes nx, ny, nz dimensions for the size of the input file.

```
diatoms
  package 'Skull'
    material 1
    insert bitmap
      file = 'skull_bitmap_file'
      nx = 680
      ny = 408
      nz = 236
    endinsert
  endpackage
enddiatom
```



Example mesh generated with Diatom bitmap option

For a full description of the diatom format see the CTH or Alegra documentation.

D. A. Crawford, A. L. Brundage, E. N. Harstad, K. Ruggirello, R. G.

Schmitt, S. C. Schumacher and J. S. Simmons, "CTH User's Manual and Input Instructions, Version 10.3", CTH Development Project, Sandia National Laboratories, Albuquerque, New Mexico 87185, February 14, 2013

Input Volume Fraction File

```
Command: input_vfrac     Input from Volume Fraction file base name

Input file command:   input_vfrac <arg>
Command line options: -ivf <arg>
Argument Type:        base file name with path
```

Command Description:

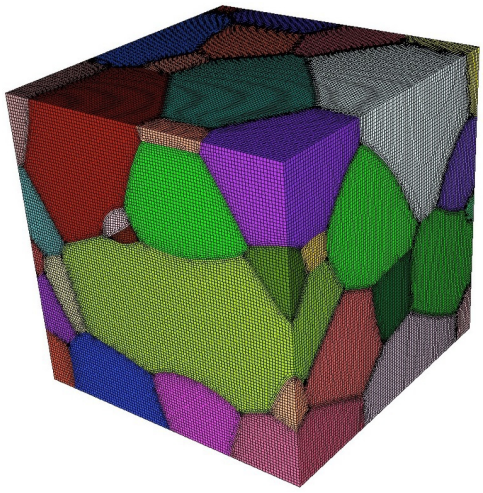
Sculpt can optionally take an exodus file containing volume fraction data stored as element variables. Normally the exodus file has initially been written using the --volfrac_file (-vf) option. Since the exodus file will be a Cartesian grid spread across multiple processors, the base filename for the parallel series of exodus files is used as the argument for this command. The input volume fraction file(s) would be used instead of an STL or diatom file. Since computing volume fractions from geometry can be time consuming, precomputing the volume fractions and reading them from a file can be advantageous if multiple meshes are to be generated from the same volume fraction data.

Input Microstructure File

```
Command: input_micro     Input from Microstructure file

Input file command:   input_micro <arg>
Command line options: -ims <arg>
Argument Type:        file name with path
```

Command Description:



Example all-hex mesh of microstructure

A microstructure file is an ascii text file containing volume fraction data for each cell of a Cartesian grid. The format for this file includes header information followed by data for each cell. The following is an example:

```
TITLE = triple line system
VARIABLES = x y z, phi_1, phi_2, phi_3
ZONE i = 2 , j = 2 , k = 2
0.0000       0.0000       0.0000      0.5000      0.5000      0.0000
1.0000       0.0000       0.0000      0.3333      0.3333      0.3334
0.0000       1.0000       0.0000      1.0000      0.0000      0.0000
1.0000       1.0000       0.0000      0.0000      1.0000      0.0000
0.0000       0.0000       1.0000      0.2000      0.4000      0.4000
```

```
1.0000        0.0000        1.0000        0.6000        0.1000        0.3000
0.0000        1.0000        1.0000        0.0000        0.0000        1.0000
1.0000        1.0000        1.0000        0.9000        0.0000        0.1000
```

The header information should contain the following:

TITLE: any descriptive character string

VARIABLES: a list of variables separated by spaces or commas. It should include x, y, z as the first three variable names. The remaining names are arbitrary. The number of variable names listed must correspond to the number of data values for each cell of the Cartesian grid.

ZONE: Specify the number of cells in the i, j and k directions (corresponding to x, y, and z respectively)

The body of the file will contain one line per cell of the grid. The first three values correspond to the centroid location of a cell in the grid. The remaining values represent volume fractions for the cell for each variable listed. The sum of the volume fractions for each individual cell should be 1.0

Currently this format assumes that cell sizes are exactly 1.0 x 1.0 x 1.0 and the minimum cell centroid location is always 0.0, 0.0, 0.0. This results in a Cartesian grid with minimum coordinate = (-0.5, -0.5, -0.5) and maximum coordinate = (i-0.5, j-0.5, k-0.5). If a size other than 1x1x1 is required consider using the scale and/or translate options.

Example usage of this command is as follows:

```
  sculpt -j 8 -ims my_micro_file.tec -p 1
```

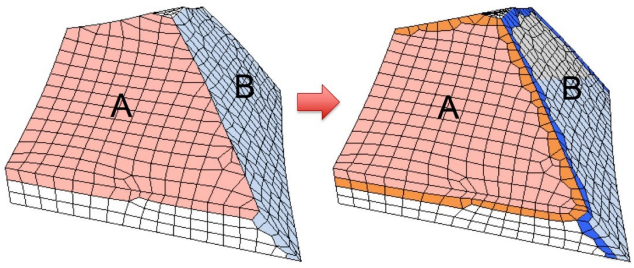
Smoothing: Sculpt will set automatic defaults for smoothing if user options have not been defined. These include:

```
--smooth 9 (surface smoothing option - no surface projection)
--csmooth 2 (curve smoothing option - hermite interpolation)
```

These options will generally provide a smoother curve and surface representation but may not adhere strictly to the volume fraction geometric definition. To over-ride the defaults, consider using the following options:

```
--smooth 8 (surface smoothing option - projection to interpolated surface)
--csmooth 5 (curve smoothing option - projection to interpolated curve)
```

Pillowing: For most 3D models it is recommended using pillowing since triple junctions (curves with at least 3 adjacent materials) will typically be defined where malformed hex elements would otherwise be generated. Surface pillowing (option 1) is usually sufficient to remove poor quality elements at triple junctions.



Pillows (hex layers) inserted at surfaces to improve element quality around curves. Note mesh quality at curve between surfaces A and B.

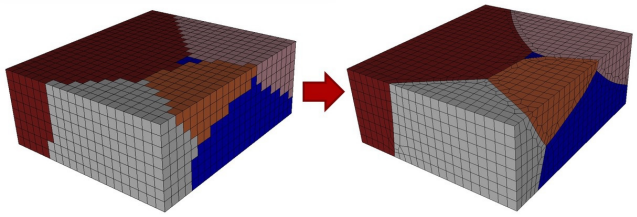
Input Cartesian Exodus File

```
Command: input_cart_exo     Input from Cartesian Exodus file

Input file command:   input_cart_exo <arg>
Command line options: -ice <arg>
Argument Type:        file name with path
```

Command Description:

An exodus mesh containing a Cartesian grid of elements can also be used as the source of a sculpt mesh. For this option the following conditions must be met:



Example Cartesian Exodus file and the resulting hex mesh.

1. A single (non-parallel) exodus II format file.
2. Contains only hex elements configured as a Cartesian grid.
3. All hex elements must be exactly equilateral cubes.
4. Each hex element has been assigned to exactly one block. (Any number of blocks may be defined in the file)

Provided these conditions are met, sculpt will treat each block as a separate material and generate a smooth conforming mesh between the materials. This option is useful for converting a stair-step mesh into a smooth conforming mesh. The resulting sculpt mesh will have the same dimensions as the original exodus mesh, but will add layers of hexes at material interfaces.

Example usage of this command is as follows:

```
  sculpt -j 8 -ice my_cartesian_file.e -p 1
```

Smoothing: Sculpt will set automatic defaults for smoothing if user options have not been defined. These include

```
--smooth 9 (surface smoothing option - no surface projection)
--csmooth 2 (curve smoothing option - hermite interpolation)
```

These options will generally provide a smoother curve and surface representation but may not adhere strictly to the volume fraction geometric definition. To over-ride the defaults, consider using the following options:

```
--smooth 8 (surface smoothing option - projection to interpolated surface)
--csmooth 5 (curve smoothing option - projection to interpolated curve)
```

Pillowing: For most 3D models it is recommended using pillowing since triple junctions (curves with at least 3 adjacent materials) will typically be defined where malformed hex elements would otherwise be generated. Surface pillowing (option 1) is usually sufficient to remove poor quality elements at triple junctions.

Input Microstructure SPN File

```
Command: input_spn     Input from Microstructure spn file

Input file command:   input_spn <arg>
Command line options: -isp <arg>
Argument Type:        file name with path
```

Command Description:

A .spn file is an optional method for importing volume fraction data into

sculpt for meshing. This format is a simple ascii text file containing one integer per cell of a Cartesian grid. Each integer represents a unique material identifier. Any number of materials may be used, however for practical purposes, the number of unique materials should not exceed more than about 50 for reasonable performance.

An example file containing a 3 x 3 x 3 grid with 2 materials may be defined as follows:

```
1 1 2 1 2 1 1 1 1
1 2 2 1 2 2 1 1 2
2 1 1 1 2 1 1 2 2
```

Any unique integer may be used to identify a material. All cells with the same ID will be defined as a continuous block with the same exodus block ID in the final mesh. All integers should be separated by a space or newline. The number of integers in the file should exactly correspond to the size of the Cartesian grid. The dimensions of the Cartesian grid must be specified on the command line as part of the input. The following is an example:

```
sculpt -j 8 -x 10 -y 24 -z 15 -isp "my_spn_file.spn" -p 1
```

The default order of the cells in the input file will be read according to the following schema:

```
for (i=0; i<nx; i++)
  for (j=0; j<ny; j++)
    for (k=0; k<nz; k++)
       // read next value from file
```

Where nx, ny, nz are the number of cells in each Cartesian direction. This ordering can be changed to nz, ny, nx using the spn_xyz_order option. The initial size of the Cartesian grid will be exactly nx X ny X nz with the minimum coordinate at (0.0, 0.0, 0.0). If a size other than the default is required, consider using the scale and/or translate options.

Smoothing: Sculpt will set automatic defaults for smoothing if user options have not been defined. These include:

```
--smooth 9 (surface smoothing option - no surface projection)
--csmooth 2 (curve smoothing option - hermite interpolation)
```

These options will generally provide a smoother curve and surface representation but may not adhere strictly to the volume fraction geometric definition. To over-ride the defaults, consider using the following options:

```
--smooth 8 (surface smoothing option - projection to interpolated surface)
--csmooth 5 (curve smoothing option - projection to interpolated curve)
```

Pillowing: For most 3D models it is recommended using pillowing since triple junctions (curves with at least 3 adjacent materials) will typically be defined where malformed hex elements would otherwise be generated. Surface pillowing (option 1) is usually sufficient to remove poor quality elements at triple junctions.

XYZ ordering of cells in SPN File

```
Command: spn_xyz_order     Ordering of cells in spn file

Input file command:   spn_xyz_order <arg>
Command line options: -spo <arg>
Argument Type:        integer (0 to 5)
Input arguments: xyz (0)
                 xzy (1)
                 yxz (2)
                 yzx (3)
                 zxy (4)
                 zyx (5)
```

Command Description:

This option is valid with the 'input_spn' option. The default order of the cells in the spn input file will be read according to the following schema:

```
    for (i=0; i<nx; i++)
      for (j=0; j<ny; j++)
        for (k=0; k<nz; k++)
           // read next value from file
```

If the spn file has the cells in a different order, use this option to specify the order. 0 (xyz) is the default.

Compress IDs in SPN file

```
Command: compress_spn_ids     Compress IDs from SPN file

Input file command:   compress_spn_ids <arg>
Command line options: -csp <arg>
Argument Type:        no argument
```

Command Description:

This option is valid with the **input_spn** or **input_stitch** options. The default will use the integers in the spn or stitch file as the final block IDs in the resulting mesh file. Turning this option ON will compress the IDs so that block IDs will start at 1 and be contiguous through the number of materials. If used, check the Sculpt output for a listing of the correspondance between the block IDs and the IDs used in the SPN file.

Input Stitch File

```
Command: input_stitch     Input from Stitch file

Input file command:   input_stitch <arg>
Command line options: -ist <arg>
Argument Type:        file name with path
```

Command Description:

Stitch is a new I/O system that has been added to Sandia's SPPARKS (Stochastic Parallel PARticle Kinetic Simulator) tool. It was specially developed for additive manufacturing (AM) simulations. Using Stitch, an output database for microstructure simulations is created incrementally by appending lattice sites much in the same way new material is added to an AM part. See the **dump stitch** (https://spparks.github.io/doc/dump.html) and **set stitch** (https://spparks.github.io/doc/set.html) commands in the SPPARKS docs for details.

Stitch Timestep

```
Command: stitch_timestep     Timestep in Stitch file to read

Input file command:   stitch_timestep <arg>
Command line options: -stt <arg>
Argument Type:        floating point value
Input arguments: first (-10)
                 last (-20)
```

Command Description:

Used with the **input_stitch** option to specify a floating point value timestep to extract from the stitch file. Either a **stitch_timestep** or **stitch_timestep_id** should be used (not both). If neither is specified, the

first timestep in the stitch file will be used. Keywords "first" or "last" may also be used in place of a floating value indicating the first or last timestep in the stitch file.

Stitch Timestep ID

```
Command: stitch_timestep_id     Timestep ID in Stitch file to read

Input file command:   stitch_timestep_id <arg>
Command line options: -stn <arg>
Argument Type:        integer > 0
```

Command Description:

Used with the **input_stitch** option to specify an integer ID representing the timestep to extract from the stitch file. Either a **stitch_timestep_id** or **stitch_timestep** should be used (not both). The **stitch_timestep_id** should be an integer, **N**, representing the **N**th timestep encountered in the stitch file. **stitch_timestep_id = 1** represents the first timestep in the file. If neither **stitch_timestep_id** or **stitch_timestep** is specified, the first timestep encountered in the stitch file will be used.

Stitch Field

```
Command: stitch_field     Field in Stitch file to read

Input file command:   stitch_field <arg>
Command line options: -stf <arg>
Argument Type:        character string
```

Command Description:

Used with the **input_stitch** option to specify the field to extract from the stitch file. If not specified, the first field in the stitch file will be used.

List Stitch header info

```
Command: stitch_info     List header info for Stitch file

Input file command:   stitch_info
Command line options: -sti
```

Command Description:

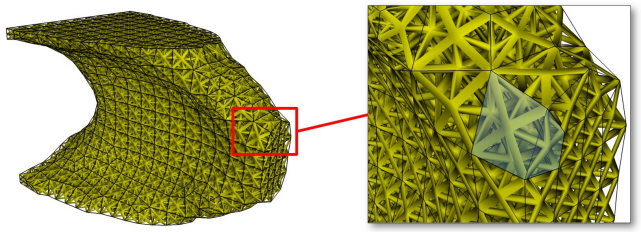
Used with the **input_stitch** option to list the header information for the given stitch file. If **stitch_info** is present (no arguments), Sculpt execution will stop after listing stitch information.

STL Lattice Template File

```
Command: lattice     STL Lattice Template File

Input file command:   lattice <arg>
Command line options: -l <arg>
Argument Type:        file name with path
```

Command Description:



Lattice geometry generated from exodus mesh.

Generate a lattice structure from a hex mesh. This command takes the name of an STL format template file which defines the lattice over a unit cube. To generate a valid lattice structure, the facets should be symmetric to the three coordinate planes. The lattice structure will be transformed and copied into each hex of the mesh. The result will be an STL file containing lattice geometry for the mesh.

This option currently requires the name of an exodus mesh on which to define the lattice. Use the --exodus_file (-e) option to specify its path. The current implementation is limited to one block, however if a second block is contained in the Exodus file it will be treated as a solid and stl facets will be generated at the skin of the block.

The name of the output STL file may also be defined by using the --stl_file (-stl) option. If no stl file is specified, the output will use the name of the input exodus file with the extension "_lattice.stl" appended.

In addition to the full lattice geometry, an additional file containing only the lattice from the first layer of hexes will be written. This may be useful in reducing the size of the STL file for visualization purposes only. The name of this file will be the name of the full STL geometry file with the extension ".vis.stl" appended.

The following is an example input file using the lattice option:

```
BEGIN SCULPT
  lattice = lattice_template.stl  $contains unit cube with triangles
  exodus_file = file.e $ hex mesh containing one or two element blocks
  stl_file = file.stl $ name of output stl file
END SCULPT
```

Note that this option is currently limited to serial execution (-j 1)

Sculpt Mesh Type

Sculpt options for specifying the type of mesh that will be generated. The default mesh type that will be produced from Sculpt is an unstructured all-hex mesh that will attempt to conform as closely as possible to the input geometry. Sculpt will normally generate its mesh on the interior of the input geometry, however with the **mesh_void** option, it can also generate the mesh on the exterior of the geometry, out to the extent of the user-defined Cartesian overlay grid.

In addition to the default hex mesh, other types of meshes may be produced. This includes the stair-step mesh where the cells of the Cartesian grid inside or intersecting the geometry are used directly as the mesh without projections or smoothing. A triangle mesh may also be generated, which can be used as the basis for a facet-based geometry representation. Other methods include the capabilities to generate a hex-dominant mesh with hexes and tets as well as the ability to include degenerate elements.

```
Mesh Type                    -typ     --type
  --stair                    -str  <arg> Generate Stair-step mesh
  --mesh_void                -V    <arg> Mesh void
  --trimesh                  -tri       Generate tri mesh of geometry surfaces
  --tetmesh                  -tet  <arg> Under Development
  --deg_threshold            -dg   <arg> Convert hexes below threshold to degenerates
  --max_deg_iters            -dgi  <arg> Maximum number of degenerate iterations
  --htet                     -ht   <arg> Convert hexes below quality threshold to tets
  --htet_method              -hti  <arg> Method used for splitting hexes to tets
  --htet_material            -htm  <arg> Convert hexes in given materials to tets
  --htet_transition          -htt  <arg> Transition method between hexes and tets
  --htet_pyramid             -htp  <arg> Local transition pyramid
  --htet_tied_contact        -htc  <arg> Local transition tied contact
  --htet_no_interface        -htn  <arg> Local transition none
  --periodic                 -per       Generate periodic mesh
  --check_periodic           -cp   <arg> Check for periodic geometry
  --check_periodic_tol       -cpt  <arg> Tolerance for checking periodicity
  --periodic_axis            -pax  <arg> Axis periodicity is about
  --periodic_nodesets        -pns  <arg> Nodesets ids of primary/secondary (leading/trailing) nodesets
```

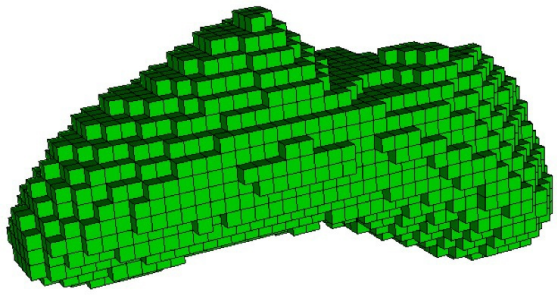
Sculpt Command Summary

Stair

```
Command: stair     Generate Stair-step mesh

Input file command:   stair <arg>
Command line options: -str <arg>
Argument Type:        integer (0, 1, 2, 3)
Input arguments: none (0)
                 off (0)
                 on (1)
                 full (1)
                 interior (2)
                 fast (3)
```

Command Description:



Example stair-step mesh on STL geometry.

The stair option generates a stair-step mesh where the cells of the Cartesian grid are used in the final mesh without projection or smoothing to the material interfaces. Cells selected from the Cartesian grid to be used in the final mesh will have volume fraction greater than 0.5. Several different options for the stair argument are available:

off (0): Stair option is off (default)

full (1): Stair-step mesh is generated, but additional processing is done to ensure material interfaces are manifold. This option may add or subtract cells from the basic mesh (where volume fraction > 0.5) to ensure no non-manifold connections between nodes and edges exist in the final mesh.

interior (2): The exterior boundary will be smooth while internal material interfaces will be stair-step. This option also ensures manifold connections between elements.

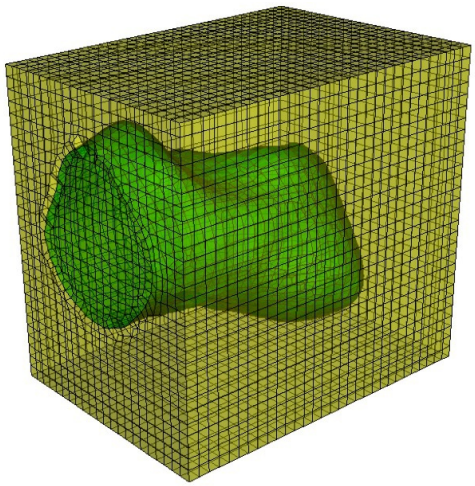
fast (3): Generates the final mesh based only on volume fraction criteria. No additional processing is done to ensure manifold connections between edges and nodes.

Mesh Void

```
Command: mesh_void     Mesh void

Input file command:   mesh_void <arg>
Command line options: -V <arg>
Argument Type:        true/false or only
Input arguments: off (0)
                 false (0)
                 on (1)
                 true (1)
                 only (2)
```

Command Description:



Mesh is generated in the void region surrounding the STL geometry.

The mesh_void accepts the following parameters:

off (0): No mesh is generated in the void region

on (1): Mesh is generated in the void region

only (2): Mesh is generated only in the void region and not in the material

If **mesh_void** option is set to on or only, then the void space surrounding the geometry will be treated as a separate material. Elements will be generated in the void to the extent of the Cartesian grid boundaries. If void_mat option is not used, the material ID of elements in the void region

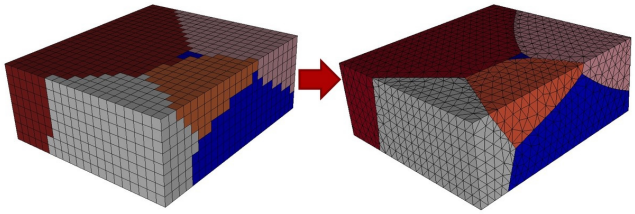
will be the maximum material ID in the model + 1. See also the **separate_void_blocks** option to separate the void elements into contiguous blocks.

Trimesh

```
Command: trimesh     Generate tri mesh of geometry surfaces

Input file command:   trimesh
Command line options: -tri
```

Command Description:



Trimesh generated from voxel microstructure data.

Generate a triangle mesh of the surface geometry. Surface geometry will be defined based on input grid resolution as well as user defined smoothing smoothing parameters. Resulting exodus mesh will contain only TRI elements. All TRI elements will be assigned to the same block in the exodus file.

This option is most often used in conjunction with the --write_geom option used to build a mesh-based geometry in Cubit. Use the following command in Cubit to import a Sculpt trimesh exodus file and s2g file (produced from --write_geom)

```
  import s2g <root filename>
```

See write_geom for more information on s2g files.

Tetmesh

```
Command: tetmesh     Under Development

Input file command:   tetmesh <arg>
Command line options: -tet <arg>
Argument Type:        none
Input arguments: off (0)
                 on (1)
                 true (1)
                 meshgems (2)
```

Command Description:

Under Development - uses space-filling tets as base grid. Size and extent is defined by bounding box options.

The **meshgems (2)** option uses a third party tet mesher to place interior tets. Triangle mesh is defined by splitting quads on surface. Both tetmesh options are currently only implemented for serial execution.

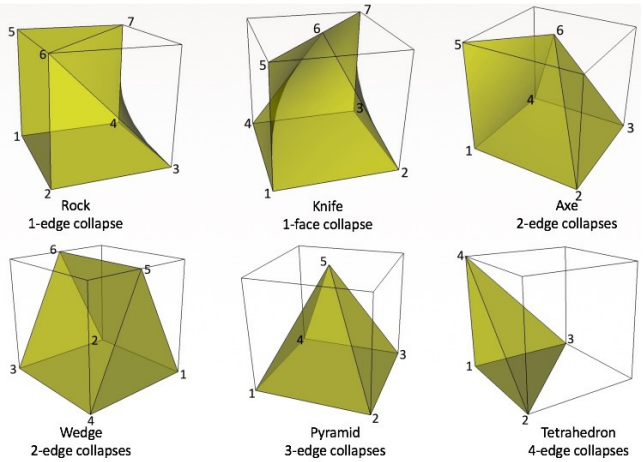
Degenerate (Edge Collapse) Threshold

```
Command: deg_threshold     Convert hexes below threshold to degenerates

Input file command:   deg_threshold <arg>
```

```
Command line options: -dg <arg>
Argument Type:        floating point value (-1.0 -> 1.0)
```

Command Description:



Examples of degenerates hexes where select edges have been collpased.

Some geometries will not permit a usable mesh with a traditional all-hex mesh. Sculpt includes the option to automatically and selectively collapse element edges to improve low-quality elements. The max_deg_iters and the deg_threshold values are used to control the creation of degenerates. Degenerate elements are treated as standard hex elements, but use repeated nodes in the eight-node connectivity array.

The deg_threshold value indicates scaled Jacobian threshold for edge collapses. Nodes at hexes below this threshold will be candidates for edge collapses, provided doing so will improve the minimum scaled Jacobian at the neighboring hexes. Default is -1.0.

Maxmimum Degenerate Iterations

```
Command: max_deg_iters     Maximum number of degenerate iterations

Input file command:   max_deg_iters <arg>
Command line options: -dgi <arg>
Argument Type:        integer >= 0
```

Command Description:

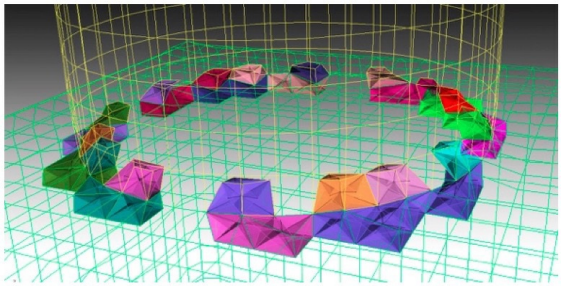
Maximum number of edge collapse iterations to perform to create degenerate hex elements. Default is 0. See also **deg_threshold**

HTet

```
Command: htet     Convert hexes below quality threshold to tets

Input file command:   htet <arg>
Command line options: -ht <arg>
Argument Type:        floating point value (-1.0 -> 1.0)
```

Command Description:



Tet elements generated where quality drops below threshold.

Automatically generate tets in place of poor quality elements. This option can be used to eliminate poor quality hex elements by replacing each hex that falls below the user defined Scaled Jacobian with 6 or 24 tets. The method used for splitting is controlled by the **htet_method** option. The default threshold value for **htet** is -1.0, which turns off the generation of all tets. A value of 1.0 will split all hexes into tets.

If a neighboring element is a hex, and will not be split, one may choose whether to use pyramid transitions or have hanging nodes. The default is to have hanging nodes with a tied contact condition being created. The transition type may be specified with the **htet_transition** command.

If tet blocks are created, their ids will be the material id plus an offset based on the maximum material id. Likewise, any pyramid blocks created will be offset as well, with their ids coming after hex block ids if there are no tets, or with their ids coming after tet blocks.

HTet Method

```
Command: htet_method     Method used for splitting hexes to tets

Input file command:   htet_method <arg>
Command line options: -hti <arg>
Argument Type:        integer (1, 2)
Input arguments: none (0)
                 structured (1)
                 unstructured (2)
```

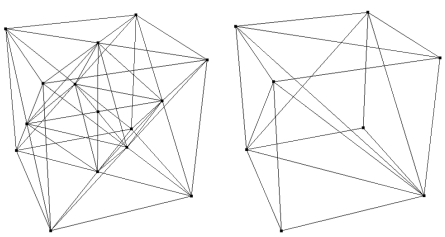
Command Description:

Specifies which method is used for splitting hexes into tets:

structured (0): Each hex is subdivided into 24 tets. Additional nodes are The 24 tets are formed by inserting one node at the center of each face and one on the interior.

unstructured (1): Each hex is subdivided into 6 tets. No additional nodes are inserted. Note that the unstructured method does not currently support the **htet_transition** options **pyramid** and **tied_contact**.

Default **htet_method** is **structured** (0).



Left: Structured **htet_method** subdivides each hex into 24 tets. Right: Unstructured **htet_method** subdivides each hex into 6 tets

HTet Material

```
Command: htet_material     Convert hexes in given materials to tets

Input file command:   htet_material <arg>
Command line options: -htm <arg>
Argument Type:        integer >= 0
```

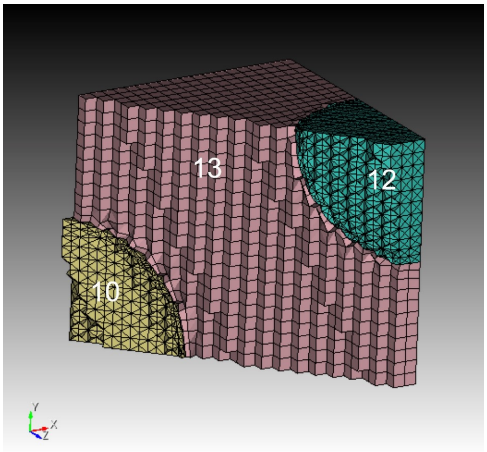
Command Description:

Generate tets in place hexes in a given material. This option can be given multiple times to specify multiple materials. Each hex in a material is replaced with 24 tets. The 24 tets are formed by inserting one node at the center of each face and one on the interior.

If an neighboring element is a hex, and will not be split, one may choose whether to use pyramid transitions or have hanging nodes. The default is to have hanging nodes with a tied contact condition being created. The transition type may be specified with the **htet_transition** command.

If tet blocks are created, their ids will be the material id plus an offset based on the maximum material id. Likewise, any pyramid blocks created will be offset as well, with their ids coming after hex block ids if there are no tets, or with their ids coming after tet blocks.

```
htet_material = 10
htet_material = 12
htet_transition = pyramid
htet_no_interface =  10 13
```



Simple example of the use of hybrid tet-hex capability using the above example input. Materials 10 and 12 use tet elements while 13 remains hexes. The default transition is to use pyramids, while the specific interface between 10 and 13 has no interface.

HTet Transition

```
Command: htet_transition     Transition method between hexes and tets

Input file command:   htet_transition <arg>
Command line options: -htt <arg>
Argument Type:        none/pyramid/tied_contact
Input arguments: none (0)
                 pyramid (1)
                 tied_contact (2)
```

Command Description:

When generating tets adjacent to hexes, the transition type between the two elements can be defined. Possible options are:

- **none (0):** No transition between hex and tet

- **pyramid (1):** Pyramid transition between hex and tet
- **tied_contact (2):** Tied contact condition between hex and tet

If pyramid transition is specified, the hex may be split into 1 pyramids and 20 tets, 2 pyramids and 16 tets, 3 pyramids and 12 tets, and so forth. The mesh will remain conformal if pyramid transition is specified.

A tied contact condition can be defined to ensure continuity of the neighboring tets and hexes. To facilitate this, one additional nodeset and sideset will be generated and output to the exodus file if the gen_sidesets = variable (2) option is specified. The sideset and nodeset will be identified with the following IDs:

Sideset 10000 = the set of hex faces that interface a set of 4 tets.

Nodeset 1000 = the set of nodes at the interface between hexes and tets. One node per face in Sideset 10000 will be included.

Local HTet Transition Pyramid

```
Command: htet_pyramid     Local transition pyramid

Input file command:   htet_pyramid <arg>
Command line options: -htp <arg>
Argument Type:        integer(s) >= 0
```

Command Description:

When generating tets adjacent to hexes, pyramid transitions can be specified for a given material or material interface. To specify a material interface, two material ids are given to specify pyramid transition between the two materials. To specify multiple materials or multiple material interfaces, this command may be used multiple times.

Local HTet Transition Tied Contact

```
Command: htet_tied_contact     Local transition tied contact

Input file command:   htet_tied_contact <arg>
Command line options: -htc <arg>
Argument Type:        integer(s) >= 0
```

Command Description:

When generating tets adjacent to hexes, tied contact transitions can be specified for a given material or material interface. To specify a material interface, two material ids are given to specify tied contact transition between the two materials. To specify multiple materials or multiple material interfaces, this command may be used multiple times.

Local HTet Transition None

```
Command: htet_no_interface     Local transition none

Input file command:   htet_no_interface <arg>
Command line options: -htn <arg>
Argument Type:        integer(s) >= 0
```

Command Description:

When generating tets adjacent to hexes, no transition can be specified for a given material or material interface. To specify a material interface, two material ids are given to specify no transition between the two materials. To specify multiple materials or multiple material interfaces,

this command may be used multiple times.

Generate Periodic Mesh

```
Command: periodic     Generate periodic mesh

Input file command:   periodic
Command line options: -per
```

Command Description:

Generates a periodic mesh for either Cartesian or unstructured mesh input. Ensures that resulting mesh nodes and faces are precisely matching on opposite sides of the mesh.

Unstructured mesh input: When used with the **--input_mesh** option opposite sides of the mesh must be identified using pairs of primary (leading) and secondary (trailing) nodesets using the **--periodic_nodesets** (-pns) option. Nodes in the nodeset pairs must be separated by a constant translation or rotation. If a rotation is used between primary (leading) and secondary (trailing) nodesets, the **--periodic_axis** (-pax) option must be used. If not used, then the transformation is assumed to be pure translation. Input geometry is assumed to be periodic with a period equal to that of the input mesh. Results from non-periodic geometry used with the **periodic** option may be unpredictable. The following is an example of an input file that uses the periodic option on an unstructured input mesh:

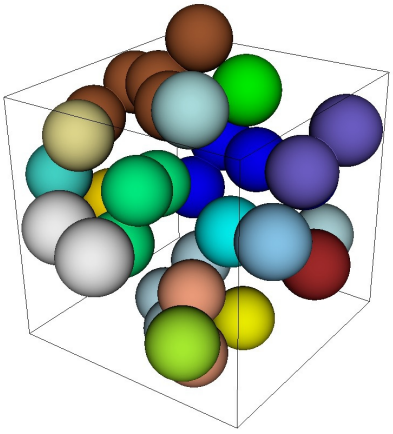
```
BEGIN SCULPT
   diatom_file = geometry_file.diatom
   input_mesh = input_exodus_file.g
   exodus_file = output_exodus_file
   smooth = to_geometry
   capture = 5
   capture_angle = 10
   free_surface_sideset = 1000
   gen_sidesets = input_mesh_and_free_surfaces
   periodic = true
   periodic_nodesets = 3224 3225
   periodic_axis = 0 0 0 0 1 0
END SCULPT
```

Cartesian grid input: This option is often used for computational materials modeling. Sculpt can generate a true periodic mesh in a representative volume element (RVE) where meshes on all opposite faces of the RVE will precisely match. When used with a Cartesian grid, the **--periodic_nodesets** and **--periodic_axis** options are ignored. The following is an example sculpt input file that utilizes the **--periodic** option on a Cartesian grid with geometry defined in a diatom file. It also utilizes the **--adapt_type** option to automatically refine and the **gen_sidesets = RVE** option to generate sidesets at the six RVE faces.

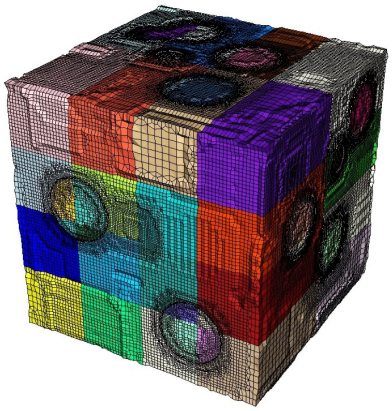
```
BEGIN SCULPT
   diatom_file = spheres_periodic.diatom
   xmin = -18.705510
   ymin = -18.705510
   zmin = -18.705510
   xmax = 18.705510
   ymax = 18.705510
   zmax = 18.705510
   nelx = 38
   nely = 38
   nelz = 38
   periodic = true
   defeature = 1
   min_vol_cells = 10
   adapt_type = vfrac_average
   adapt_levels = 2
   adapt_threshold = 0.00001
   gen_sidesets = RVE
   exodus_file = spheres_periodic
   mesh_void = true
END SCULPT
```

Geometry Requirements: In order to generate a valid periodic mesh, the input geometry must also be periodic and the bounding box parameters should span exactly one period of the geometry. To check the periodicity of the geometry and prescribed bounding box, see the **check_periodic** option.

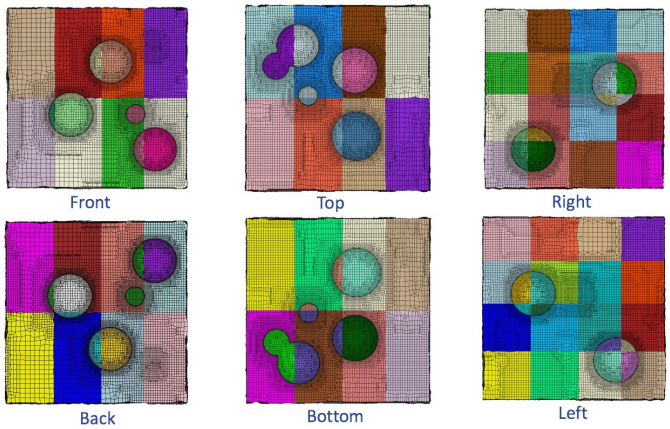
Note: The resulting mesh at the boundaries of the Cartesian grid (RVE) will not be projected to the planes of the bounding box. The result will be a "ragged" boundary in order to maintain periodicity between nodes on opposite sides of the mesh. Also note that results from the use of the **periodic** option may be undefined or unstable when used with non-periodic input geometry.



Periodic geometry used for example described in diatom file. RVE boundary shown with respect to the geometry.



Resulting periodic mesh generated from example input.



Six faces of the RVE from above example illustrating periodicity on a 32 processor decomposition. Note that top three images are a mirror image of the bottom three images.

Check for periodic geometry

```
Command: check_periodic     Check for periodic geometry

Input file command:   check_periodic <arg>
Command line options: -cp <arg>
Argument Type:        on, off, only
Input arguments: off (0)
                 on (1)
                 only (2)
```

Command Description:

When using the **periodic** option with a Cartesian base grid, the input geometry must be periodic with respect to the grid bounding box in order to meet the minimum requirements of a valid periodic mesh. The bounding box must span exactly one period in each dimension. If this requirement is not met, a valid mesh may still be generated, however, periodicity will not be guaranteed. The **check_periodic** option is used to check this requirement. See also **check_periodic_tol** to set the tolerance for checking periodicity.

Options:

- **ON:** The **check_periodic** option is ON by default to ensure periodicity is enforced. Sculpt will fail if the geometry and bounding box do not meet the requirements for periodicity.
- **OFF:** Turning this option OFF will by-pass this check and attempt to generate the mesh even if periodic requirements are not met.
- **ONLY:** The ONLY option will perform a check for periodic requirements and report diagnostics. An exodus file (or files) will be produced with the name "check_periodic.0.0.x.x". A stair-step mesh of the domain will be produced with an additional 2 blocks: 999 and 998. Block 999 shows master cells that are not matched. Block 998 shows paired ghost cells that should have the same volume fraction as those in block 999, but do not. Sculpt will immediately stop execution after producing the "check_periodic.0.0.x.x" mesh. Note that 2 additional layers on all sides of the Cartesian grid will be present in the mesh. These are used internally in Sculpt for parallel ghosting.

The **check_periodic** option is ignored if the **periodic** option is OFF or set to false.

Tolerance used for periodic check

```
Command: check_periodic_tol     Tolerance for checking periodicity

Input file command:   check_periodic_tol <arg>
Command line options: -cpt <arg>
Argument Type:        floating point value
```

Command Description:

Used on conjunction with the **check_periodic** option. It specifies a tolerance value when checking periodicity. Check periodic option checks the difference between computed volume fractions for cells on the overlay grid that are separated by exactly one period. The periodic tolerance is the allowable volume fraction difference between cells separated by one period. Default value is 1e-6.

Periodic Mesh Axis

```
Command: periodic_axis     Axis periodicity is about

Input file command:   periodic_axis <arg>
```

```
Command line options: -pax <arg>
Argument Type:        six floating point values
```

Command Description:

For an unstructured base grid, specifies an axis about which the nodes in the primary (leading) nodesets will be rotated about to produce the secondary (trailing) nodesets. Six floating point numbers are specified, the first three define the origin of the axis and the last three define the axis direction. This option must be used with **--periodic** (-per), **--periodic_nodesets** (-pns), and **--input_mesh** (-im) options. If the **--periodic** (-per) option is used without the --periodic_axis option, the transformation between primary (leading) and secondary (trailing) nodesets is assumed to be pure translation.

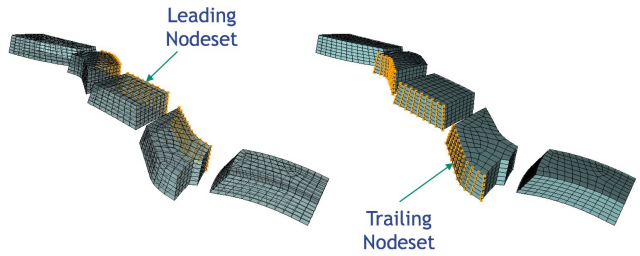
Periodic Nodeset Ids

```
Command: periodic_nodesets     Nodesets ids of primary/secondary (leading/trailing) nodesets

Input file command:   periodic_nodesets <arg>
Command line options: -pns <arg>
Argument Type:        integer(s) >= 0
```

Command Description:

For an unstructured base grid, specifies the primary-secondary (leading-trailing) nodeset pairs. Primary (leading) nodesets should be able to be translated or rotated about a specified axis to produce the nodes in the secondary (trailing) nodesets. Nodesets must be specified in pairs, where each primary (leading) nodeset corresponds to a single secondary (trailing) nodeset. Each nodeset pair must maintain an identical translation or rotation. If a rotation is used, the axis and origin of rotation must be specified with the **--periodic_axis** (-pax) option. This option should be used with **--periodic** (-per), **--periodic_nodesets** (-pns), and **--input_mesh** (-im) options.)



Unstructured input mesh used to generate periodic mesh. Matching primary (leading) and secondary (trailing) nodesets are defined in the exodus file.

Sculpt Output

Sculpt options for specifying output. The primary format for the hex meshes produced from Sculpt is Exodus II. One exodus file will be produced for each processor based upon the -j or num_procs argument. If required, the exodus files can be joined using the epu utility.

Other options for export include the ability to dump the volume fraction representation of the input geometry as well as the ability to write geometry files for use in Cubit.

```
Output                          -out     --output
  --exodus_file                 -e     <arg> Output Exodus file base name
  --large_exodus                -le    <arg> Output large Exodus file(s)
  --volfrac_file                -vf    <arg> Output Volume Fraction file base name
  --quality                     -Q     <arg> Dump quality metrics to file
  --export_comm_maps            -C           Export parallel comm maps to debug exo files
  --write_geom                  -G           Write geometry associativity file
  --write_mbg                   -M           Write mesh based geometry file <beta>
  --compare_volume              -cv          Report vfrac and mesh volume comparison
  --compute_ss_stats            -css         Report sideset statistics
```

Sculpt Command Summary

Exodus File

```
Command: exodus_file     Output Exodus file base name

Input file command:   exodus_file <arg>
Command line options: -e <arg>
Argument Type:        character string
```

Command Description:

The base file name of the resulting exodus mesh. Exodus files will be in the form <exodus_file>.e.<nproc>.<iproc>. For example, if the number of processors used is 3 and the exodus_file argument is "model" the following files would be written:

```
model.e.3.0
model.e.3.1
model.e.3.2
```

If no exodus_file argument is used, output files will be in the form <stl_file>_diatom_results.e.<nprocs>.<iproc>. For example, if the number of processors used is 3 and the stl_file (or diatom_file) is "model.stl", the following files would be written:

```
model_diatom_results.e.3.0
model_diatom_results.e.3.1
model_diatom_results.e.3.2
```

A full path may be used when specifying the base exodus file name, otherwise files will be placed in the current working directory. If the exodus_file option is not used, exodus files will be placed in the same directory as the input diatom or stl file.

Large Exodus Output

```
Command: large_exodus     Output large Exodus file(s)

Input file command:   large_exodus <arg>
Command line options: -le <arg>
Argument Type:        no argument
Input arguments: off (0)
                 false (0)
```

```
on (1)
true (1)
```

Command Description:

Generate output Exodus file(s) to allow IDs greater than 2^31 (2.14 Billion). This option should be used if the intent is to generate billions of elements in parallel on HPC platforms. This option will approximately double the size of output exodus files, so is normally only used for very large parallel applications.

Volume Fraction File

```
Command: volfrac_file     Output Volume Fraction file base name

Input file command:   volfrac_file <arg>
Command line options: -vf <arg>
Argument Type:        character string
```

Command Description:

Optionally generate exodus files containing a hex mesh of the Cartesian grid containing volume fraction data as element variables. This series of parallel exodus files can later be used as direct input to sculpt using the --input_vfrac (-ivf) command. If not specified, no volume fraction data files will be generated.

Quality

```
Command: quality     Dump quality metrics to file

Input file command:   quality <arg>
Command line options: -Q <arg>
Argument Type:        file name with path
```

Command Description:

Specify a filename to write mesh quality metrics. If the file already exists, metrics will be appended. Quality metrics and other details of the run will be written to this file. This option is currently off by default.

Export Communication Maps

```
Command: export_comm_maps     Export parallel comm maps to debug exo files

Input file command:   export_comm_maps
Command line options: -C
```

Command Description:

Used for debugging and verification. Exodus files of the mesh containing the communication nodes and faces at processor boundaries will be written as nodes and side sets. This provides a way to visually check the validity of the parallel communication maps.

Write S2G Geometry File

```
Command: write_geom     Write geometry associativity file

Input file command:   write_geom
Command line options: -G
```

Command Description:

An s2g (Sculpt to Geometry) file, with the pattern <fileroot>.s2g, will be produced when this argument is used where fileroot is the string specified by the --exodus_file or -e option. An s2g file includes geometry associativity for the exodus file that is written. If used with Cubit's "import s2g <fileroot>" a mesh-based geometry will be generated in Cubit with geometric entities prescribed by Sculpt through the s2g file.

When used with the --trimesh option, the s2g file can provide information to Cubit to build a set of mesh-based geometry volumes where only the surfaces are meshed. This is useful for using the tet meshing capabilities in Cubit to mesh the discrete geometry that was generated in Sculpt. For example, a tet mesh may be constructed from microstructures spn data (see import_spn) with the following workflow:

1. Run Sculpt to generate an exodus and s2g file. An example input file may look like the following:

```
begin sculpt
  import_spn = myfile.spn
  trimesh = true
  write_geom = true
  pillow = 1
end sculpt
```

2. Import the file into Cubit to generate a mesh based geometry:

```
import s2g myfile
```

3. Delete the triangle mesh, set sizes and mesh:

```
delete mesh
vol all scheme tetmesh
vol all size 2.0
mesh vol all
```

Note that the write_geom and trimesh options are still in development and will currently only work with a single processor (-j 1).

Write Mesh Based Geometry

```
Command: write_mbg     Write mesh based geometry file <beta>

Input file command:   write_mbg
Command line options: -M
```

Command Description:

An MBG (Mesh Based Geometry) file will be produced when this argument is used with the pattern <fileroot>.mbg, where fileroot is the string specified by the --exodus_file or -e option. An MBG file includes the surface and topology definition defined by sculpt as a result of the interface reconstruction process. It will correspond to the boundary of the 3D elements that are generated in the exodus file, or the surface elements generated with the --trimesh option.

An MBG file can be be imported into Cubit using the following Cubit command line options:

```
  import mbg "<fileroot>.mbg"
```

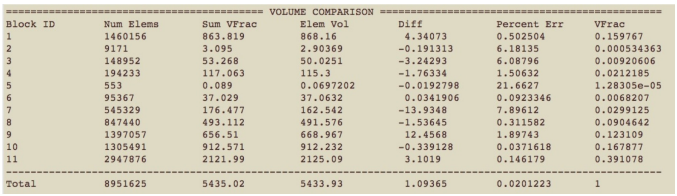
Report VFrac to Mesh Volume Comparison

```
Command: compare_volume     Report vfrac and mesh volume comparison

Input file command:   compare_volume
Command line options: -cv
```

Command Description:

A report will be generated and printed to the terminal following the mesh summary that compares the input volume fraction of the geometry with that of the final finite element mesh. If a volume fraction format is not used as input, the volume fractions will be computed on the refined base grid and used as comparison. Note that exact geometric volumes of the STL or analytic geometry are not used for comparison, rather the volume fraction approximation of the geometry on the refined Cartesian grid.



```
========================================= VOLUME COMPARISON =============================================
Block ID        Num Elems       Sum VFrac       Elem Vol        Diff            Percent Err     VFrac
1               1460156         863.819         868.16           4.34073        0.502504        0.159767
2               9171            3.095           2.90369         -0.191313       6.18135         0.000534363
3               148952          53.268          50.0251         -3.24293        6.08796         0.00920606
4               194233          117.063         115.3           -1.76334        1.50632         0.0212185
5               553             0.089           0.0697202       -0.0192798      21.6627         1.28305e-05
6               95367           37.029          37.0632          0.0341906      0.0923346       0.0068207
7               545329          176.477         162.542         -13.9348        7.89612         0.0299125
8               847440          493.112         491.576         -1.53645        0.311582        0.0904642
9               1397057         656.51          668.967          12.4568        1.89743         0.123109
10              1305491         912.571         912.232         -0.339128       0.0371618       0.167877
11              2947876         2121.99         2125.09          3.1019         0.146179        0.391078
------------------------------------------------------------------------------------------------------
Total           8951625         5435.02         5433.93          1.09365        0.0201223       1
```

Example output from the **compare_volume** command.

The following is a brief description of each column:

- **Block ID:** ID of material/block
- **Num Elems:** Number of hex elements assigned to block in final mesh
- **Sum VFrac:** Sum of input volume fraction for block. For STL or diatom geometry, approximates the volume fraction. For 3D image data (ie. bitmap, input_spn) sums the exact volume fraction input.
- **Elem Vol:** Sum of final mesh volume for block
- **Diff:** Absolute difference between input and output volume fractions for block. (Elem Vol - Sum VFrac)
- **Percent Err:** Percent error represented by Difference between input and output volume fractions for block
- **VFrac:** Total volume fraction represented by Elem Vol. for block. VFrac volume should sum to 1.0.

Report Sideset Statistics

```
Command: compute_ss_stats     Report sideset statistics

Input file command:   compute_ss_stats
Command line options: -css
```

Command Description:

A report will be generated and printed to the terminal following the mesh summary that displays the statitics for the surface areas of the sidesets in the final FEA mesh. This is often useful for microstructures use cases where it may be important to know the interface area between materials. This should be used with the **gen_sidesets** option to be effective.

Sculpt Process Control

Options for controlling the execution of Sculpt. Sculpt is a parallel application that uses MPI to distribute and build the hex mesh on multiple processors. The -j or num_procs option is normally used to specify the number of processors to use. Sculpt will write a separate exodus file for each processor, which can be joined into a single file using the epu utility. While any number of processors may be used, you would normally use a -j value less than or equal to the number of cores available on your hardware.

Sculpt options can be specified directly from the command line using the "short" commands, or from an input file where the longer forms of the commands are used. Since an input file can be commented and modified, it is generally the recommended method for running Sculpt.

Process Control	-pc	--process
--num_procs	-j	<arg> Number of processors requested
--input_file	-i	<arg> File containing user input data
--debug_processor	-D	<arg> Sleep to attach to processor for debug
--debug_flag	-dbf	<arg> Dump debug info based on flag
--quiet	-qt	Suppress output
--print_input	-pi	Print input values and defaults then stop
--version	-vs	Print version number and exit
--threads_process	-tpp	<arg> Number of threads per process
--iproc	-ip	<arg> Number of processors in I direction
--jproc	-jp	<arg> Number of processors in J direction
--kproc	-kp	<arg> Number of processors in K direction
--build_ghosts	-bg	Write ghost layers to exodus files for debug
--vfrac_method	-vm	<arg> Set method for computing volume fractions

Sculpt Command Summary

Number of Processors

```
Command: num_procs     Number of processors requested

Input file command:   num_procs <arg>
Command line options: -j <arg>
Argument Type:        integer > 0
```

Command Description:

The number of processors that Sculpt will use to generate the mesh. For a structured, Cartesian base grid, the domain will be automatically divided into roughly equal sized rectangular regions based on this value.

For unstructured input (see **--input_mesh**), to utilize more than one processor, the base mesh must first be docomposed into the same number of regions specified by **num_procs**. The **decomp** tool, part of the Sandia, **SEACAS** tool suite can be used to break up an exodus mesh into multiple regions suitable for sculpt input.

An independent mesh of a portion of the domain is generated on each processor. Continuity across processor boundaries is maintained with MPI (Message Passing Interface). Each processor will write a separate Exodus II file to disk containing its portion of the domain. The Sandia SEACAS tool, **epu** can be used to join parallel files into a single file if desired.

If not specified on the command line, the number of processors used will be 1.

For additional control on the arrangement of processor domains on a Cartesian base grid, see arguments **iproc, jproc, kproc.**

Input File

```
Command: input_file     File containing user input data

Input file command:   input_file <arg>
Command line options: -i <arg>
Argument Type:        file name with path
```

Command Description:

Rather than specifying a complicated series of arguments on the command line, an input file may also be used. An input file is a simple text file containing all arguments and parameters to be used in the current sculpt run. Input files are normally expected to have a ".i" extension. Arguments used in the input file are limited to the Long Names indicated for each command.

User comments can also be made anywhere in the file but must follow a "$" sign. The argument assignments that are intended to be read must be contained within a "begin sculpt" and "end sculpt" block. All arguments may use upper or lower case and can optionally use "=" between the command and its parameter. The following is an example input file:

```
BEGIN SCULPT
  stl_file = "mygeom.stl"
  cell_size = 0.5
  exodus_file = "mymesh"
  mesh_void = true
END SCULPT
```

The following is an example of using an input file with sculpt:

```
sculpt -j 4 -i myinput.i
```

Note that the number of processors (-j) should always be used on the command line and cannot be included in the input file. Relative or absolute paths for files may also be used.

Debug Processor

```
Command: debug_processor     Sleep to attach to processor for debug

Input file command:   debug_processor <arg>
Command line options: -D <arg>
Argument Type:        integer >= 0
```

Command Description:

Used for debugging. All processes will sleep until the designated process is attached to a debugger. Note: value of 0 corresponds to first processor, 1 to second, etc.

Debug Flag

```
Command: debug_flag     Dump debug info based on flag

Input file command:   debug_flag <arg>
Command line options: -dbf <arg>
Argument Type:        integer >= 0
Input arguments: off (0)
                 lost_nodes (1)
                 non-manifold (2)
                 defeature (3)
                 thickening (4)
                 initial-projections (5)
                 reversal (6)
                 gq-color (8)
                 gq-full (9)
```

```
gq-full (9)
hostname (10)
test_large_global_ids (11)
use_bbox_for_unstructured (12)
allow_input_mesh_sidesets (13)
```

Command Description:

Used for debugging. Set flag to dump specific info based on the following:

0 (off) Default, No debug output

1 (lost_nodes) Dump processor lost node info

2 (non-manifold) Export Non-manifold resolution state as exodus file after each inner and outer iteration.

3 (defeature) Export Defeature state as exodus file after each inner and outer iteration.

4 (thickening) Export the Thickened st:wqate as exodus file after each material has been thickened.

Guaranteed Quality:

5 (initial-projections) Turn off initial minimizer projection.

6 (reversal) Use Non-manifold reversal case

7 Combine debug_flag 5 and 6

8 (gq-color) Use guaranteed quality laplacian color smoothing

9 (gq-full) Combine debug_flags 5,6 and 8

10 (hostname) Display the host name for each processor

11 (test_large_global_IDs) starts the global node and element ID numbering at INT_MAX (2^31) for testing large_exodus option.

12 (use_bbox_for_unstructured) Use the xmin, ymin, ... options to limit the domain on an input_mesh

13 (allow_input_mesh_sidesets) Don't restrict use of gen_sidesets = 6, 7 or 9 if using parallel. (This may result in poor elements at proc boundaries).

Quiet

```
Command: quiet     Suppress output

Input file command:   quiet
Command line options: -qt
```

Command Description:

Suppress any output to the command line from Sculpt as it is running.

Print Input

```
Command: print_input     Print input values and defaults then stop

Input file command:   print_input
Command line options: -pi
```

Command Description:

Display all input parameters and defaults used in the current Sculpt run to the output window and then stop. No mesh (or volume fractions) will be

generated.

Version

```
Command: version     Print version number and exit

Input file command:   version
Command line options: -vs
```

Command Description:

Prints Sculpt version information and exits.

Threads Per Processor

```
Command: threads_process     Number of threads per process

Input file command:   threads_process <arg>
Command line options: -tpp <arg>
Argument Type:        integer > 0
```

Command Description:

This option is currently experimental and under development. Sculpt may use shared memory parallelism to improve performance. When built with the Kokkos library, some algorithms in sculpt will use shared memory parallel threads in addition to MPI distributed memory parallelism (MPI+X). Currently this option is implemented only for surface and volume Laplacian smoothing algorithms. This option may not be available requiring a custom build of sculpt to be used. Check with developers if you would like to use this option.

Number of processors in I

```
Command: iproc     Number of processors in I direction

Input file command:   iproc <arg>
Command line options: -ip <arg>
Argument Type:        integer > 0
```

Command Description:

Arguments iproc, jproc and kproc provide user control over the processor decomposition in I, J, and K directions respectively. iproc * jproc * kproc must equal the number of processors specified on the command line using the -j option. In some cases, where it is known that significant refinement will be done in a localized region of the domain, it may be worth manually specifying the number of cell layers, or intervals that each processor will contain. This should provide rough processor load balacing so that regions with an expected large number of elements will have a smaller physical domain, but roughly similar element counts. This may avoid single processor memory limitations on large HPC machines. To specify processor intervals, optionally, the exact interval count in I, J, K directions should be specified. The number of intervals specfied for each direction should be the value of **iproc**, **jproc**, and **kproc** respectively. In addition, the sum of the intervals in each direction should equal the user specified **nelx**, **nely** and **nelz** values respectively. For example, for a 24 processor (-j 24) arrangement with nelx = nely = 28 and nelz = 27, one possible arrangment of processors would be as follows:

```
nelx = 28
nely = 28
nelz = 27
iproc = 2 14 14
jproc = 2 14 14
```

```
kproc = 6 7 6 5 4 3 2
```

For this example the I and J directions are equally subdivided into 2 processors with 14 intervals in both directions. The K direction, however is subdivided into 6 processors, where the intervals are progressively thinner, starting with 7 intervals at the bottom (smallest z) and ending with 2 intervals at the top (largest z). Note that the number of intervals in any direction must always be at least 2 or greater. If processor intervals are omitted, Sculpt will attempt to roughly equally space the intervals in each direction. If at least one direction of intervals is detected in the input, then all directions should be specified. Note that this option is only available when using a Cartesian base grid specification and cannot be used with an unstructured base grid using the **input_mesh** option.

Number of processors in J

```
Command: jproc     Number of processors in J direction

Input file command:   jproc <arg>
Command line options: -jp <arg>
Argument Type:        integer > 0
```

Command Description:

Arguments iproc, jproc and kproc provide user control over the processor decomposition in I, J, and K directions respectively. iproc * jproc * kproc must equal the number of processors specified on the command line using the -j option. In some cases, where it is known that significant refinement will be done in a localized region of the domain, it may be worth manually specifying the number of cell layers, or intervals that each processor will contain. This should provide rough processor load balacing so that regions with an expected large number of elements will have a smaller physical domain, but roughly similar element counts. This may avoid single processor memory limitations on large HPC machines. To specify processor intervals, optionally, the exact interval count in I, J, K directions should be specified. The number of intervals specfied for each direction should be the value of **iproc**, **jproc**, and **kproc** respectively. In addition, the sum of the intervals in each direction should equal the user specified **nelx**, **nely** and **nelz** values respectively. For example, for a 24 processor (-j 24) arrangement with nelx = nely = 28 and nelz = 27, one possible arrangment of processors would be as follows:

```
nelx = 28
nely = 28
nelz = 27
iproc = 2 14 14
jproc = 2 14 14
kproc = 6 7 6 5 4 3 2
```

For this example the I and J directions are equally subdivided into 2 processors with 14 intervals in both directions. The K direction, however is subdivided into 6 processors, where the intervals are progressively thinner, starting with 7 intervals at the bottom (smallest z) and ending with 2 intervals at the top (largest z). Note that the number of intervals in any direction must always be at least 2 or greater. If processor intervals are omitted, Sculpt will attempt to roughly equally space the intervals in each direction. If at least one direction of intervals is detected in the input, then all directions should be specified. Note that this option is only available when using a Cartesian base grid specification and cannot be used with an unstructured base grid using the **input_mesh** option.

Number of processors in K

```
Command: kproc     Number of processors in K direction

Input file command:   kproc <arg>
Command line options: -kp <arg>
```

```
Command line options:  kp <arg>
Argument Type:         integer > 0
```

Command Description:

Arguments iproc, jproc and kproc provide user control over the processor decomposition in I, J, and K directions respectively. iproc * jproc * kproc must equal the number of processors specified on the command line using the -j option. In some cases, where it is known that significant refinement will be done in a localized region of the domain, it may be worth manually specifying the number of cell layers, or intervals that each processor will contain. This should provide rough processor load balacing so that regions with an expected large number of elements will have a smaller physical domain, but roughly similar element counts. This may avoid single processor memory limitations on large HPC machines. To specify processor intervals, optionally, the exact interval count in I, J, K directions should be specified. The number of intervals specfied for each direction should be the value of **iproc**, **jproc**, and **kproc** respectively. In addition, the sum of the intervals in each direction should equal the user specified **nelx**, **nely** and **nelz** values respectively. For example, for a 24 processor (-j 24) arrangement with nelx = nely = 28 and nelz = 27, one possible arrangment of processors would be as follows:

```
nelx = 28
nely = 28
nelz = 27
iproc = 2 14 14
jproc = 2 14 14
kproc = 6 7 6 5 4 3 2
```

For this example the I and J directions are equally subdivided into 2 processors with 14 intervals in both directions. The K direction, however is subdivided into 6 processors, where the intervals are progressively thinner, starting with 7 intervals at the bottom (smallest z) and ending with 2 intervals at the top (largest z). Note that the number of intervals in any direction must always be at least 2 or greater. If processor intervals are omitted, Sculpt will attempt to roughly equally space the intervals in each direction. If at least one direction of intervals is detected in the input, then all directions should be specified. Note that this option is only available when using a Cartesian base grid specification and cannot be used with an unstructured base grid using the **input_mesh** option.

Write the ghost layers for debug

```
Command: build_ghosts     Write ghost layers to exodus files for debug

Input file command:   build_ghosts
Command line options: -bg
```

Command Description:

If set, this option will dump the ghost hexes at the boundaries of processor domains to the exodus files. This is used only for debugging.

Volume Fraction Calculation Method

```
Command: vfrac_method     Set method for computing volume fractions

Input file command:   vfrac_method <arg>
Command line options: -vm <arg>
Argument Type:        integer (1, 2, 3)
Input arguments: cth (0)
                 cth (1)
                 r3d (2)
                 winding (3)
```

Command Description:

Sets the method used for computing volume fractions from geometry

input. Three options are currently available:

CTH (1): The default method. It uses the CTH third party library from Sandia Laboratories for approximating intersections using an adaptive ray firing method to determine inside-outside status of multiple locations within a grid cell. This method can be used with STL and all valid primitive types defined by the diatom format.

R3D (2): Uses the R3D third party library developed by Los Alamos Laboratories. Machine precision intersection calculations are performed to generate accurate volume fractions from the STL description. This method is valid for STL and diatom input packages specifying STL input files. Non STL format geometry defined in the diatom file will be ignored for this format.

Winding (3): Uses the fast winding number from the igl third party library, which should gracefully handle non-watertight stl files. R&D is in progress.

Sculpt Overlay Grid Specification

Sculpt options for setting up the overlay grid. Sculpt is an overlay-grid method that requires a base mesh that it will modify to generate the final mesh. The base mesh can be in the form of a Cartesian grid, but can also be any general unstructured hexahedral mesh defined in an exodus file (see the **input_mesh** option). Pamgen can also be used to generate an unstructured base mesh (see **input_mesh_pamgen**).

When an overlay Cartesian grid is used as the basis for the all-hex mesh that will be produced, the bounds and size of the cells defining the grid must be specified. The Cartesian grid can be defined in one of two ways:

1. Define the bounding box and number of intervals in each coordinate direction. (**xmin, ymin, zmin, xmax, ymax, zmax, nelx, nely, nelz**)
2. Define a **cell_size**. Sculpt will then automatically define the Cartesian grid coordinates and intervals by evaluating the bounding box of the input geometry and adding a small number of cells in each coordinate direction.

Other options for setting up the Cartesian base grid include **align** and **expand** which are normally used with the second method. The **align** option will automatically rotate the grid to best match the characteristic direction of the geometry rather than maintaining alignment with the global Cartesian directions. The **expand** option over-rides the default expansion of the Cartesian grid beyond the bounding box of the geometry and allow the user to specify a specific expansion percentage.

```
Overlay Grid Specification   -ovr     --overlay
  --nelx                     -x    <arg> Num cells in X in overlay Cartesian grid
  --nely                     -y    <arg> Num cells in Y in overlay Cartesian grid
  --nelz                     -z    <arg> Num cells in Z in overlay Cartesian grid
  --xmin                     -t    <arg> Min X coord of overlay Cartesian grid
  --ymin                     -u    <arg> Min Y coord of overlay Cartesian grid
  --zmin                     -v    <arg> Min Z coord of overlay Cartesian grid
  --xmax                     -q    <arg> Max X coord of overlay Cartesian grid
  --ymax                     -r    <arg> Max Y coord of overlay Cartesian grid
  --zmax                     -s    <arg> Max Z coord of overlay Cartesian grid
  --cell_size                -cs   <arg> Cell size (nelx, nely, nelz ignored)
  --align                    -a          Automatically align geometry to grid
  --bbox_expand              -be   <arg> Expand tight bbox by percent
  --input_mesh               -im   <arg> Input Base Exodus mesh
  --input_mesh_blocks        -imb  <arg> Block ids of Input Base Exodus mesh
  --input_mesh_material      -imm  <arg> Material definition with input mesh
  --input_mesh_pamgen        -imp  <arg> Input Base mesh defined by Pamgen
  --join_parallel            -jp   <arg> Join parallel files
```

Sculpt Command Summary

Number of Intervals X

```
Command: nelx     Num cells in X in overlay Cartesian grid

Input file command:   nelx <arg>
Command line options: -x <arg>
Argument Type:        integer > 0
```

Command Description:

Defines the number of intervals in the x direction of the base Cartesian grid used for defining the volume fraction definition and meshing For best results the intervals specified should result in approximately equilateral cells.

See also nely, nelz

Number of Intervals Y

```
Command: nely     Num cells in Y in overlay Cartesian grid

Input file command:   nely <arg>
Command line options: -y <arg>
Argument Type:        integer > 0
```

Command Description:

Defines the number of intervals in the y direction of the base Cartesian grid used for defining the volume fraction definition and meshing For best results the intervals specified should result in approximately equilateral cells.

See also nelx, nelz

Number of Intervals Z

```
Command: nelz     Num cells in Z in overlay Cartesian grid

Input file command:   nelz <arg>
Command line options: -z <arg>
Argument Type:        integer > 0
```

Command Description:

Defines the number of intervals in the z direction of the base Cartesian grid used for defining the volume fraction definition and meshing For best results the intervals specified should result in approximately equilateral cells.

See also nelx, nely

Xmin Bounding Box Range

```
Command: xmin     Min X coord of overlay Cartesian grid

Input file command:   xmin <arg>
Command line options: -t <arg>
Argument Type:        floating point value
```

Command Description:

Defines the minimum x coordinate of the bounding box or range of the Cartesian mesh to be used for meshing.

See also ymin, zmin, xmax, ymax, zmax.

Ymin Bounding Box Range

```
Command: ymin     Min Y coord of overlay Cartesian grid

Input file command:   ymin <arg>
Command line options: -u <arg>
Argument Type:        floating point value
```

Command Description:

Defines the minimum y coordinate of the bounding box or range of the Cartesian mesh to be used for meshing.

See also xmin, zmin, xmax, ymax, zmax.

Zmin Bounding Box Range

```
Command: zmin     Min Z coord of overlay Cartesian grid

Input file command:   zmin <arg>
Command line options: -v <arg>
Argument Type:        floating point value
```

Command Description:

Defines the minimum z coordinate of the bounding box or range of the Cartesian mesh to be used for meshing.

See also xmin, ymin, xmax, ymax, zmax.

Xmax Bounding Box Range

```
Command: xmax     Max X coord of overlay Cartesian grid

Input file command:   xmax <arg>
Command line options: -q <arg>
Argument Type:        floating point value
```

Command Description:

Defines the maximum x coordinate of the bounding box or range of the Cartesian mesh to be used for meshing.

See also xmin, ymin, zmin, ymax, zmax.

Ymax Bounding Box Range

```
Command: ymax     Max Y coord of overlay Cartesian grid

Input file command:   ymax <arg>
Command line options: -r <arg>
Argument Type:        floating point value
```

Command Description:

Defines the maximum y coordinate of the bounding box or range of the Cartesian mesh to be used for meshing.

See also xmin, ymin, zmin, xmax, zmax.

Zmax Bounding Box Range

```
Command: zmax     Max Z coord of overlay Cartesian grid

Input file command:   zmax <arg>
Command line options: -s <arg>
Argument Type:        floating point value
```

Command Description:

Defines the maximum z coordinate of the bounding box or range of the Cartesian mesh to be used for meshing.

See also xmin, ymin, zmin, xmax, ymax.

Cell Size

```
Command: cell_size     Cell size (nelx, nely, nelz ignored)

Input file command:   cell_size <arg>
Command line options: -cs <arg>
Argument Type:        floating point value
```

Command Description:

Defines a target edge size for the cells of the base Cartesian grid. Both interval and cell_size can not be specified simultaneously. If cell_size is used without a range specification, a bounding box of the geometry will be computed and used as the default range

Align

```
Command: align     Automatically align geometry to grid

Input file command:   align
Command line options: -a
```

Command Description:

The align option will attempt to orient the Cartesian grid with the main dimensions of the geometry. This is done by defining a tight bounding box around the geometry using an optimization procedure where the objective is to minimize the difference in volume between an enclosing box and the geometry. Using the align command will override any bounding box parameters previously entered and will build an "aligned" bounding box around the full geometry. It is currently only implemented for STL geometry and will ignore any other diatom definitions. Note that this option will also write temporary stl and diatom files to the working directory.

Bounding Box Expansion Factor

```
Command: bbox_expand     Expand tight bbox by percent

Input file command:   bbox_expand <arg>
Command line options: -be <arg>
Argument Type:        floating point value
```

Command Description:

Sculpt will measure a tight bounding box of the input model and expand the box by the specified percentage in x, y and z. Input value can be any positive or negative floating point value where 1.0 represents 100 percent expansion. If not specified, the default will add about 2.5 cell widths to the bounding box on each side. This option should be used with the cell_size option. It will be ignored if a specific bounding box has been defined (ie. xmin, ymin, etc...).

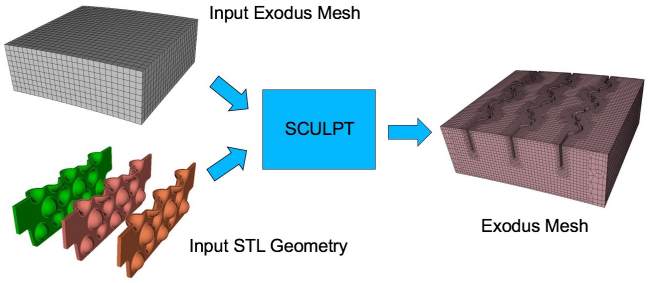
Input Base Exodus Mesh

```
Command: input_mesh     Input Base Exodus mesh

Input file command:   input_mesh <arg>
Command line options: -im <arg>
Argument Type:        file name with path
```

Command Description:

Option to import an Exodus file to use as the base mesh for Sculpt. Sculpt's meshing procedure requires a base mesh from which geometry is recovered and captured. The default base mesh is a Cartesian grid that is defined by specifying a bounding box and intervals. The **input_mesh** option permits a general hexahedral mesh to be used as the base mesh instead of a Cartesian grid. This option currently supports a serial and parallel Exodus files containing HEX8 elements with any number of blocks.



An exodus file is used as the base mesh for Sculpt and STL files describe the geometry to be sculpted.

The **input_mesh** option can also be used in parallel. Sculpt currently requires the mesh to be decomposed prior to running sculpt. The SEACAS **decomp** tool can be used to pre-process any exodus mesh to break it into multiple meshes ready for use in sculpt. SEACAS is an open source library available on github. For example, when using four processors with sculpt, you would use the following command:

```
decomp -p 4 simple-mesh.g
```

The result would be the four meshes:

```
simple-mesh.g.4.0
simple-mesh.g.4.1
simple-mesh.g.4.2
simple-mesh.g.4.3
```

Once the base mesh has been decomposed, Sculpt can be run. In this case, the **input_mesh** option would use the root **simple-mesh.g** as the argument.

```
input_mesh = simple-mesh.g
```

If the **-j 4** option is used, sculpt will look for 4 meshes in the current working directory with the appropriate root and extension.

Four different options are supported for describing the geometry when using the **input_mesh** option:

- **stl_file:** A single file containing a water-tight faceted description of the geometry. Note that only the portion of the STL file completely contained within the base mesh will be represented in the final mesh.
- **diatom_file:** May contain analytic descriptions of geometric primitives and/or references to multiple STL files.
- **input_spn:** The materials of the cells in the spn file are mapped onto the elements of the input mesh using inverse distance-weighted interpolation. As with the stl and diatom files, only the portion of the spn file completely contained within the base mesh will be represented in the final mesh. The **input_mesh_blocks** option can be used in conjunction with the spn_file option to limit the scope of the mapping of material from the spn file to the mesh file. If this options is used, only elements in the specified blocks will get mapped to. For more details, see the **input_mesh_blocks** option.
- **Element Variables:** The geometry may also be described by element variables in the Exodus file. Element variables should

represent material volume fractions where the sum of element variables for any one cell should be between 0.0 and 1.0. Any number of element variables may be used where each unique variable defined will describe an element block in the final Exodus mesh produced. if the sum of element variables is less than 1.0 for any one element, a void material will be assumed and removed from the base mesh unless the **mesh_void** option is used.

Limitations:

- An STL file and element variables cannot be used in the same input. If element variables are present in the Exodus file and an STL or Diatom file is used, the element variables will be ignored.
- If an input mesh is used, any Cartesian grid specifications will be ignored (ie. **nelx**, **xmin**, **xmax**).
- The **adapt_type** option will work only for an exodus input mesh that defines a mapped mesh. Adapt types **vfrac_average (4)** and **vfrac_difference (6)** are currently the only criteria supported with the **input_mesh** option.

Blocks of Input Base Exodus Mesh

```
Command: input_mesh_blocks     Block ids of Input Base Exodus mesh

Input file command:   input_mesh_blocks <arg>
Command line options: -imb <arg>
Argument Type:        integers > 0
```

Command Description:

This option is valid when specifying both **input_mesh** and **spn_file**. Using this option, the materials of the cells in the spn file are mapped onto only the elements of the specified blocks in the input_mesh file. The remaining blocks are treated as void. The behavior without this option maps the materials of the cells in the spn file onto elements of all blocks in the input_mesh file.

Material Definition with Input Mesh

```
Command: input_mesh_material     Material definition with input mesh

Input file command:   input_mesh_material <arg>
Command line options: -imm <arg>
Argument Type:        integers > 0
Input arguments: geometry (0)
                 blocks (1)
                 proximity_blocks (2)
                 geometry_and_blocks (3)
```

Command Description:

This option is valid when specifying an 'input_mesh' . Using this option, the material definition in the final mesh may be defined based on the material definitions on the geometry, or based on the block ids of the input mesh. For example, a diatom file defining geometry would have materials defined which are used to define the materials in the final mesh. The default is to use material definitions on the geometry. Possible options are:

- **geometry (0):** Material defined by geometry. Block definitions in input file will be ignored. Instead, only the geometry defined from the STL or diatom geometry definition will be respected.
- **blocks (1):** Material defined by blocks in input mesh. Both the geometry definition defined by the STL/diatom file and by the block definition in the input file will be used to construct the mesh. Block definitions used in the final mesh will be defined only by the

input_mesh blocks, ignoring any material definitions in the diatom file. Note that pillowing may be required to maintain mesh quality where interior intersecting interfaces between STL/diatom geometry and block geometry occur. When used with the **mesh_void** option, element block groupings are made only for the interior of the STL/diatom geometry. Elements in the void region are assigned to the designated **void_mat**

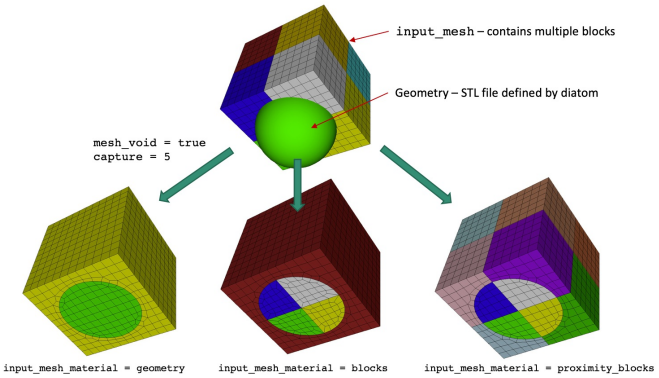
- **proximity_blocks (2):** Material defined by blocks in input mesh based on proximity method. Geometric interfaces in the final mesh will be defined based on the STL/diatom definition, similar to the **geometry (0)** method. Material/block assignment for elements is defined only by the proximity of individual elements contained within the regions defined by blocks in the input mesh. Block assignment is made by collecting all elements in the final mesh whose centroid falls within a given block in the input mesh. Note that this method avoids the interior intersections and subsequent poor mesh quality that may result from the **blocks (1)** method described above. As a result, interior interfaces may be non-continuous (stair-step). In order to avoid collisions between the material ID defined in the diatom file and those defined in the input_mesh, block IDs in the final mesh are assigned based on the following definition:

```
ID = (diatom_material_ID-1)*max_input_mesh_block_ID + input_mesh_block_ID
```

If the max_input_mesh_block_ID is less than the max_diatom_material_ID, then the following definition is used:

```
ID = (input_mesh_block_ID-1)*max_diatom_material_ID + diatom_material_ID
```

- **geometry_and_blocks (3):** Material defined by the stl geometry and the blocks in input mesh. Both the geometry definition defined by the STL/diatom file and by the block definition in the input file will be used to construct the mesh. Block definitions used in the final mesh will be defined by both the material IDs defined in the diatom file and the input_mesh blocks. Where stl geometry overlaps a material block defined in the input_mesh, the stl geometry material will take precedence.



Example of the resulting meshes using the different options for **input_mesh_material**. In this case, geometry is defined by an STL file (sphere) and **input_mesh** is a brick containing 8 different material blocks.

Input Base mesh defined by Pamgen

```
Command: input_mesh_pamgen     Input Base mesh defined by Pamgen

Input file command:   input_mesh_pamgen <arg>
Command line options: -imp <arg>
Argument Type:        file name with path
```

Command Description:

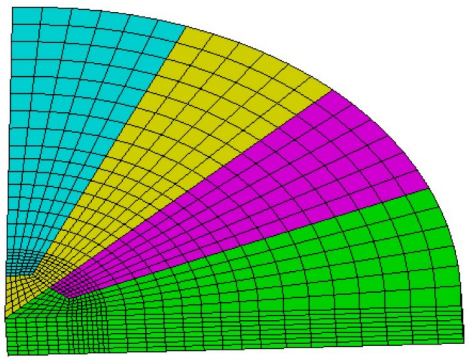
Option to use Pamgen to create a base mesh for Sculpt. Pamgen is an open source meshing tool developed at Sandia for generating hexahedral meshes from geometric primitives. In addition to being a stand-alone meshing solution, it is a parallel tool that is integrated as an inline meshing tool for Sandia's shock physics simulation tool, Alegra. Pamgen has also been integrated in Sculpt as a solution for automatically defining a base mesh.

The **input_mesh_pamgen** option permits a mesh defined my Pamgen input parameters to define the base mesh. A limited set of brick and cylinder primitives are supported by Pamgen. The name of an ascii file containing the pamgen mesh definition is used as the argument for this option. The following is a simple example of a pamgen mesh description. It generates a partial cylinder with a span of 90 degrees and height of 1.0. Other parameters allow for specific interval and sizing specifications as well as block/material identification.

```
mesh
    radial trisection
        trisection blocks, 2
        zmin -0.00075
        numz 1
            zblock 1 1. interval 8
        numr 3
            rblock 1 2.0 interval 8
            rblock 2 3.0 interval 8
            rblock 3 4.0 interval 8
        numa 1
            ablock 1 90. interval 24
    end
    set assign
        nodeset, ilo, 100
        block sideset, ihi, 45, 2
    end
end
```

For a full description of Pamgen and input parameters see the following document:

David M. Hensinger, Richard R. Drake, James G. Foucar, Thomas A. Gardiner, "Pamgen, a Library for Parallel Generation of Simple Finite Element Meshes", Sandia Report SAND2008-1933 (2008)



Base mesh generated by pamgen using the above input parameters. Colors represent 4 different processors when used in parallel mode.

Similar to the **input_mesh** option, the same geometry input options are available. They include **stl_file**, **diatom_file** and **input_spn**. See the **input_mesh** option for additional details and limitations.

Join Parallel Files

```
Command: join_parallel     Join parallel files

Input file command:   join_parallel <arg>
Command line options: -jp <arg>
Argument Type:        true/false
```

```
Input arguments: off (0)
                 false (0)
                 on (1)
                 true (1)
```

Command Description:

Import a set of files, one per processor, using the base name defined by the **import_mesh** option. When not used (default), the assumption of input exodus meshes is to include parallel (nemesis) data where parallel relationships between neighboring processors has already been established. This is normally done by using the SEACAS **decomp** tool to decompose a single exodus mesh into multiple files.

If the **join_parallel** option is used, sculpt assumes the parallel relationships are not included and will establish these relationships based on proximity of node locations at processor boundaries. Note that the file naming convention should follow the standard exodus parallel file naming convention. For example, a mesh spread across 4 files would be named:

```
brick.e.4.0
brick.e.4.1
brick.e.4.2
brick.e.4.3
```

This option is currently only implemented for axis aligned rectangular processor domains. This option can also be used to just stitch exodus files and dump resulting files without any additional sculpt operations. The following is an example of a sculpt input file that does simple stitching without any additional sculpt operations:

```
begin sculpt
  input_mesh = brick.e
  exodus_file = brick_ouput
  join_parallel  = true
  input_mesh_material = blocks
  stair = fast
end sculpt
```

Sculpt Smoothing

Sculpt options for specifying how the mesh will be smoothed following mesh generation.

Sculpt includes a tiered approach to smoothing to improve element quality. It starts by applying smoothing to all nodes in the mesh and progressively restricts the smoothing operations to only those nodes that fall below a user-defined scaled Jacobian threshold. Default numbers of iterations and thresholds for each smoothing phase have been tuned for general use, however it may be worthwhile to adjust these parameters. The three smoothing phases include:

- **Laplacian Smoothing:** Applied to all elements. Very inexpensive fast approach to improve quality, but can result in degraded element quality if applied to excess. A fixed default of 2 iterations is applied to all hexes. Increasing the num_laplace parameter can improve some cases, especially convex shapes.
- **Optimization Smoothing:** Applied only to elements who's scaled Jacobian falls below the opt_threshold parameter (default 0.6) and their surrounding elements. This approach uses a more expensive optimization technique to improve regions of elements simultaneously. The max_opt_iters parameter can control the maximum number of iterations applied (default is 5). Iterations will terminate, however, if no further improvement is detected. Because this method optimizes node locations simultaneously, neighboring nodes with competing optimum can sometimes limit mesh quality.
- **Spot Optimization:** Also known as parallel coloring, is applied only to elements who's element quality falls below the pcol_threshold parameter (default 0.2). This technique is the most expensive of the techniques, but focusses only on nodes that are immediately adjacent to poor quality hexes. Each node is smoothed independently of its neighbors, and may require a high number of iterations using the max_pcol_iters to achieve desired results. Increasing the pcol_threshold and max_pcol_iters may yield improved results.

Smoothing	-smo	--smoothing
--smooth	-S	<arg> Smoothing method
--csmooth	-CS	<arg> Curve smoothing method
--laplacian_iters	-LI	<arg> Number of Laplacian smoothing iterations
--max_opt_iters	-OI	<arg> Max. number of parallel Jacobi opt. iters.
--opt_threshold	-OT	<arg> Stopping criteria for Jacobi opt. smoothing
--curve_opt_thresh	-COT	<arg> Min metric at which curves won't be honored
--max_pcol_iters	-CI	<arg> Max. number of parallel coloring smooth iters.
--pcol_threshold	-CT	<arg> Stopping criteria for parallel color smooth
--max_gq_iters	-GQI	<arg> Max. number of guaranteed quality smooth iters.
--gq_threshold	-GQT	<arg> Guaranteed quality minimum SJ threshold
--geo_smooth_max_deviation	-GSM	<arg> Geo Smoothing Maximum Deviation

Sculpt Command Summary

Smooth

```
Command: smooth     Smoothing method

Input file command:   smooth <arg>
Command line options: -S <arg>
Argument Type:        integer (0, 1, 2, 3)
Input arguments: off (0)
                 default (1)
                 on (1)
                 fixed_bbox (2)
                 no_surface_projections (3)
                 to_geometry (4)
                 to_geom (4)
                 geo_smooth (5)
                 geometry_smoothing (5)
                 geo_smoothing (5)
```

Command Description:

Automatic adjustment of node locations following meshing to improve element quality. Controls the combined Laplacian and optimization smoothing procedures applied to volume and surface nodes (see csmooth for curve smoothing options) Uses the **laplacian_iters**, **max_opt_iters**, **opt_threshold**, **max_pcol_iters**, **pcol_threshold**, **mqx_gq_iters** and **gq_threshold** arguments to control the sensitivity and aggressiveness of the smoothing operations. In most cases, the default options for these parameters are sufficient, however increasing iterations or threshold values, while potentially causing longer run times, may result in improved mesh quality.

Smoothing will adjust the location of nodes on surfaces, projecting them to an approximated surface representation defined by interface reconstruction from volume fractions. In addition to turning smoothing **on** and **off**, the surface projection characteristics can be adjusted using the **bbox_fixed** and **no_surface_projections** options.

- **off (0):** No volume and surface smoothing is performed.
- **on/default (1):** (Default) Combined Laplacian/Optimization (Hybrid) smoothing both surface and volumes. Automatic boundary buffer layer improvement is performed at interior surfaces intersecting the domain boundary.
- **fixed_bbox (2):** Uses standard hybrid smoothing procedure (option 1), however nodes at the the domain boundaries will be projected to one of the six planes of the bounding box or unstructured **input_mesh**. This option turns off the automatic boundary buffer improvement.
- **no_surface_projections (3):** Uses the **fixed_bbox** method, however interior surfaces are not projected. This can result in smoother interior surface representations for microstructures models. This is effective in smoothing noisy surface data, but can potentially reduce overall volume. The **laplacian_iters** option will control the amount of smoothing that will occur, with higher numbers of iterations resulting in a smoother surface representation, but also resulting in more reduction in geometric volume. This method is default for microstructures file formats.
- **to_geometry (4):** When used with the **capture** option, smoothing will also move nodes to the closest geometry entity. It must currently be used with **capture** to ensure that curves and surfaces are first identified and associated with boundary mesh entities. This option will only work with STL or diatom input that contains STL geometry.
- **geo_smooth (5):** Similar to the **smooth = to_geometry** option, it is also intended to be used with the **capture** option. With the **geo_smooth** option, nodes are initially projected to the closest geometry entity. However, following the initial projection, Sculpt will utilize the Laplacian smoothing operation, similar to the **smooth = no_surface_projections** option, to effectively remove noise and local deviations in the geometry definition. The **laplacian_iters** option will control the amount of geometric smoothing that will occur. The optional command **geo_smooth_max_deviation** can also be used to control the maximum distance any individual node location can deviate from its original geometry definition.

Boundary Buffer Improvement: Sculpt's smoothing procedures will use an automatic boundary buffer improvement method. It will attempt to improve the quality of hexes where interior surfaces are close to tangent with the bounding box. This can result in nodes that may not lie precisely on the planes of the domain boundary. The **fixed_bbox (2)** and **no_surface_projections (3)** options will turn off the automatic boundary buffer improvement.

Curve Smoothing

```
Command: csmooth     Curve smoothing method

Input file command:   csmooth <arg>
Command line options: -CS <arg>
Argument Type:        integer (0, 1, 2, ...6)
Input arguments: off (0)
                 circle (1)
                 hermite (2)
                 average_tangent (3)
                 neighbor_surface_normal (4)
                 vfrac (5)
                 linear (6)
```

Command Description:

The csmooth option controls the smoothing method used on curves. In most cases the default should be sufficient, however it may be useful to experiment with different options. The default curve smoothing option is **vfrac (5)**. The following curve smoothing options are available:

- **off (0):** No curve smoothing will be performed.
- **circle (1):** Nodes projected to a fitted circle defined current node and its two neighbors.
- **hermite (2):** Nodes projected based on Hermite interpolation. Note that this method can only be used in serial (-j 1)
- **average_tangent (3):** Nodes projected based on average tangent of neighbors. Note that this method may not be parallel serial consistent.
- **neighbor_surface_normal (4):** Nodes projected based on neighboring surface normals and the resulting intersecting planes.
- **vfrac (5):** (Default) Nodes projected to initial curve interface defined from the original volume fraction data.
- **linear (6):** Nodes projected to the linear segment defined by the node and its two immediate neighbors.

Laplacian Iterations

```
Command: laplacian_iters     Number of Laplacian smoothing iterations

Input file command:   laplacian_iters <arg>
Command line options: -LI <arg>
Argument Type:        integer >= 0
```

Command Description:

Number of Laplacian smoothing iterations performed when Hybrid smoothing option is used. Default value is 2.

Maximum Optimization Iterations

```
Command: max_opt_iters     Max. number of parallel Jacobi opt. iters.

Input file command:   max_opt_iters <arg>
Command line options: -OI <arg>
Argument Type:        integer >= 0
```

Command Description:

Indicates the maximum number of iterations of optimization-based smoothing to perform. May complete sooner if no further improvement can be made. Default is 5

Optimization Threshold

```
Command: opt_threshold     Stopping criteria for Jacobi opt. smoothing

Input file command:   opt_threshold <arg>
Command line options: -OT <arg>
Argument Type:        floating point value (-1.0 -> 1.0)
```

Command Description:

Indicates the value for scaled Jacobian where Optimization smoothing will be performed. Elements with scaled Jacobian less than opt_threshold and their neighbors will be smoothed. Default value is 0.6

Curve Optimization Threshold

```
Command: curve_opt_thresh     Min metric at which curves won't be honored

Input file command:   curve_opt_thresh <arg>
Command line options: -COT <arg>
Argument Type:        floating point value (-1.0 -> 1.0)
```

Command Description:

Indicates the value for scaled Jacobian where if a node that falls on a curve has neighboring quads less than this value, then the smoothing will no longer honor the curve definition. Instead the optimization smoother will attempt to place the node to optimize the neighboring mesh quality, without regard for its placement on its owning curve.

Normally this value should be set close to zero to avoid too many nodes from floating off of their owning curves, however, if mesh quality is constrained by curve geometry, setting this value higher can help to avoid bad or poor quality elements. Default for this value is 0.1.

Maximum Parallel Coloring Iterations

```
Command: max_pcol_iters     Max. number of parallel coloring smooth iters.

Input file command:   max_pcol_iters <arg>
Command line options: -CI <arg>
Argument Type:        integer >= 0
```

Command Description:

Maximum number of spot smoothing (also known as parallel coloring) iterations to perform. May complete sooner if no further improvement can be made. Default is 100. See also pcol_threshold.

Parallel Coloring Threshold

```
Command: pcol_threshold     Stopping criteria for parallel color smooth

Input file command:   pcol_threshold <arg>
Command line options: -CT <arg>
Argument Type:        floating point value (-1.0 -> 1.0)
```

Command Description:

Indicates scaled Jacobian threshold for spot smoothing (also known as parallel coloring). A parallel coloring algorithm is used to uniquely identify and isolate nodes to be improved using optimization. Default is 0.2.

Maximum Guaranteed Quality Iterations

```
Command: max_gq_iters     Max. number of guaranteed quality smooth iters.

Input file command:   max_gq_iters <arg>
Command line options: -GQI <arg>
Argument Type:        integer >= 0
```

Command Description:

Maximum number of guaranteed quality smoothing iterations to perform. Guaranteed quality smoothing performs a constrained Laplacian smoothing algorithm to adjust node locations. If the result of a smoothing operation results in adjacent element quality falling below the specified **gq_threshold** value, then move distance is cut until minimum threshold is achieved or the metric is improved. To achieve parallel consistency, a parallel coloring methodology is employed. The **max_gq_iters** defines the maximum number of parallel color iterations employed. Default is 0 (off).

Note that guaranteed quality can be utilized in conjunction with other smoothing methods (Laplacian, Optimization and Parallel Coloring), however to be effective it is normally used independent from other smoothing. For example, to use guaranteed quality the following is suggested:

```
laplacian_iters = 0
max_opt_iters = 0
max_pcol_iters = 0
max_gq_iters = 100
```

Guaranteed Quality Threshold

```
Command: gq_threshold     Guaranteed quality minimum SJ threshold

Input file command:   gq_threshold <arg>
Command line options: -GQT <arg>
Argument Type:        floating point value (-1.0 -> 1.0)
```

Command Description:

Indicates scaled Jacobian threshold for guaranteed quality smoothing. Default is 0.2. see also **max_gq_iters**

Geo Smooth Max Deviation

```
Command: geo_smooth_max_deviation     Geo Smoothing Maximum Deviation

Input file command:   geo_smooth_max_deviation <arg>
Command line options: -GSM <arg>
Argument Type:        floating point value (>= 0.0
```

Command Description:

Used only in conjunction with the **smooth = geo_smooth** option. It controls the maximum distance any individual node can deviate from the geometry definition. If a smoothing operation computes a location that will move a node further than the prescribed **geo_smooth_max_deviation** value from the geometry, the node movement will be artificially limited by this value. If not specificied, no limitations will be placed on node movement due to smoothing operations when the **smooth = geo_smooth** is used. A value of zero (0) will constrain all nodes at interfaces to lie on the geometry, similar to the **smooth = to_geometry** option. When using this option, the maximum deviation from the geometry for any individual node will be reported in the MESH SUMMARY in the Sculpt ouput.

Automatic Scheme Selection

- Default Scheme Selection
- Automatic Scheme Selection General Notes
- Surface Auto Scheme Selection
- Volume Auto Scheme Selection

For volume and surface geometries the user may allow CUBIT to automatically select the meshing scheme. Automatic scheme selection is based on several constraints, some of which are controllable by the user. The algorithms to select meshing schemes will use topological and geometric data to select the best quad or hex meshing tool. Auto scheme selection will not select tet or tri meshing algorithms. The command to invoke automatic scheme selection is:

```
{geom_list} Scheme Auto
```

Specifically for surface meshing, interval specifications will affect the scheme designation. For this reason it is recommended that the user specify intervals before calling automatic scheme selection. If the user later chooses to change the interval assignment, it may be necessary to call scheme selection again. For example, if the user assigns a square surface to have 4 intervals along each curve, scheme selection will choose the surface mapping algorithm. However if the user designates opposite curves to have different intervals, scheme selection will choose paving, since this surface and its assigned intervals will not satisfy the mapping algorithm's interval constraints. In cases where a general interval size for a surface or volume is specified and then changed, scheme selection will not change. For example, if the user specified an interval size of 1.0 a square 10X10 surface, scheme selection will choose mapping. If the user changes the interval size to 2.0, mapping will still be chosen as the meshing scheme from scheme selection. If a mesh density is not specified for a surface, a size based on the smallest curve on the surface will be selected automatically.

Default Scheme Selection

If the user does not set a scheme for a particular entity and chooses to mesh the entity, CUBIT will automatically run the auto scheme selection algorithm and attempt to set a scheme. In cases where the auto scheme selection fails to choose a scheme, the meshing operation will fail. In this case explicit specification of the meshing scheme and/or further geometry decomposition may be necessary.

The default scheme selection in CUBIT, unless otherwise set, will attempt to set a quadrilateral or hexahedral meshing scheme on the entity. If tet or tri meshing will always be the desired element shape, the following command can be used:

```
Set Default Element [Tet|Tri|HEX|QUAD|None]
```

Setting the default element to **tet** or **tri** will bypass the auto scheme selection and always use either the triadvance or tetmesh schemes if the scheme has not otherwise been set by the user. The default settings of **quad** or **hex** will use the automatic scheme selection.

Previous functionality of CUBIT used a default scheme of map and interval of 1 for all surface and volume entities. For backwards compatibility and if this behavior is still desired, the **none** option may be used on the **set default element** command.

Auto Scheme Selection General Notes

In general, automatic scheme selection reduces the amount of user input. If the user knows the model consists of 2.5D meshable volumes, three commands to generate a mesh after importing or creating the model are needed. They are:

> **volume all size <value>**
>
> **volume all scheme auto**
>
> **mesh volume all**

The model shown in the following figure was meshed using these three commands (part of the model is not shown to reveal the internal structure of the model).

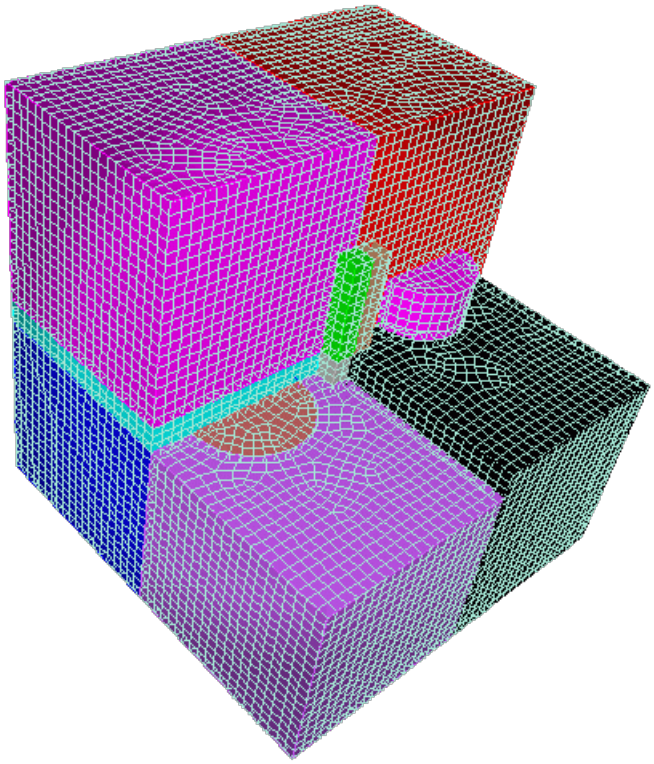


Figure 1. Non-trivial model meshed using automatic scheme selection

Scheme Firmness

Meshing schemes may be selected through three different approaches. They are: default settings, automatic scheme selection, and user specification. These methods also affect the scheme firmness settings for surfaces and volumes. Scheme firmness is completely analogous to interval firmness.

Scheme firmness can be set explicitly by the user using the command

```
{geom_list} Scheme {Default | Soft | Hard}
```

Scheme firmness settings can only be applied to surfaces and volumes.

This may be useful if the user is working on several different areas in the model. Once she/he is satisfied with an area's scheme selection and doesn't want it to change, the firmness command can be given to hard set the schemes in that area. Or, if some surfaces were hard set by the user, and the user now wants to set them through automatic scheme selection then she/he may change the surface's scheme firmness to soft or default.

Surface Auto Scheme Selection

Surface auto scheme selection (White, 99) will choose between Pave, Submap, Triprimitive, and Map meshing schemes, and will always result in selecting a meshing scheme due to the existence of the paving algorithm, a general surface meshing tool (assuming the surface passes the even interval constraint).

Surface auto scheme selection uses an angle metric to determine the vertex type to assign to each vertex on a surface; these vertex types are then analyzed to determine whether the surface can be mapped or submapped. Often, a surface's meshing scheme will be selected as Pave or Triprimitive when the user would prefer the surface to be mapped or submapped. The user can overcome this by several methods. First, the user can manually set the surface scheme for the "fuzzy" surface. Second, the user can manually set the "vertex types" for the surface. Third, the user can increase the angle tolerance for determining "fuzziness." The command to change scheme selection's angle tolerances is:

```
[Set] Scheme Auto Fuzzy [Tolerance] {value}
(value in degrees)
```

The acceptable range of values is between 0 and 360 degrees. If the user enters 360 degrees as the fuzzy tolerance, no fuzzy tolerance checks will be calculated, and in general mapping and submapping will be chosen more often. If the user enters 0 degrees, only surfaces that are "blocky" will be selected to be mapped or submapped, and in general paving will be chosen more often.

Volume Auto Scheme Selection

When automatic scheme selection is called for a volume, surface scheme selection is invoked on the surfaces of the given volume. Mesh density selections should also be specified before automatic volume scheme selection is invoked due to the relationship of surface and volume scheme assignment.

Volume scheme selection chooses between Map, Submap and Sweep meshing schemes. Other schemes can be assigned manually, either before or after the automatic scheme selection.

Volume scheme selection is limited to selecting schemes for 2.5D geometries, with additional tool limitations (e.g. Sweep can currently only sweep from several sources to a single target, not multiple targets); this is due to the lack of a completely automatic 3D hexahedral meshing algorithm. If volume scheme selection is unable to select a meshing scheme, the mesh scheme will remain as the default and a warning will be reported to the user.

Volume scheme selection can fail to select a meshing scheme for several reasons. First, the volume may not be mappable and not 2.5D; in this case, further decomposition of the model may be necessary. Second, volume scheme selection may fail due to improper surface scheme selection. Volume schemes such as Map, Submap, and Sweep require certain surface meshing schemes, as mentioned previously.

Radialmesh

Summary: Creates a free cylindrical mesh with precise node locations based on input radii, angles, and offsets, then creates mesh-based geometry to fit the mesh.

Syntax:

```
Create Radialmesh \
    NumZ <val> [Span <val>] \
         Zblock 1 [<offset val>] \
                  {Interval|Bias|Fraction|First
Size} <val> \
                  [{Interval|Bias|Fraction|Last
Size} <val>] \
         Zblock 2 [<offset val>] \
                  {Interval|Bias|Fraction|First
Size} <val> \
                  [{Interval|Bias|Fraction|Last
Size} <val>] \
         ... NumZ \

    NumR <val> {Trisection|Initial
Radius<val>} \
         Rblock 1 <offset radius val> \
                  {Interval|Bias|Fraction|First
Size} <val> \
                  [{Interval|Bias|Fraction|Last
Size} <val>] \
         Rblock 2 <offset radius val> \
                  {Interval|Bias|Fraction|First
Size} <val> \
                  [{Interval|Bias|Fraction|Last
Size} <val>] \
         ... NumR \

    NumA <val> [Full360] [Span <val>] \
         Ablock 1 [<offset angle val>] \
                  {Interval|Bias|Fraction|First
Angle} <val> \
                  [{Interval|Bias|Fraction|Last
Angle} <val>] \
         Ablock 2 [<offset angle val>] \
                  {Interval|Bias|Fraction|First
Angle} <val> \
                  [{Interval|Bias|Fraction|Last
Angle} <val>] \
         ... NumA
```

Discussion:

The purpose of the **radialmesh** command is to create a cylindrical mesh with precise node locations. Unlike all other meshing commands which place nodes using smoothing algorithms to optimize element quality, node locations for the radialmesh command are calculated based on the input radii, angles, and offsets. In addition, the radialmesh command does not mesh existing geometry. Rather, it creates a mesh based on the input parameters, after which a mesh-based geometry is created to fit the free mesh.

The radialmesh command requires input for the 3 coordinate directions (Z, radial, angular). The number of blocks in each direction is specified with the numZ, numR, and numA values in the command. Each block forms a new volume in the final mesh. All bodies in the mesh are merged to form a conformal mesh between blocks.

The Radialmesh command can create meshes which span any angle greater than 0.0 up to 360 degrees. In addition, meshes can model either a tri-section (see Figure 1), or a non-trisection mesh (see Figure 2).

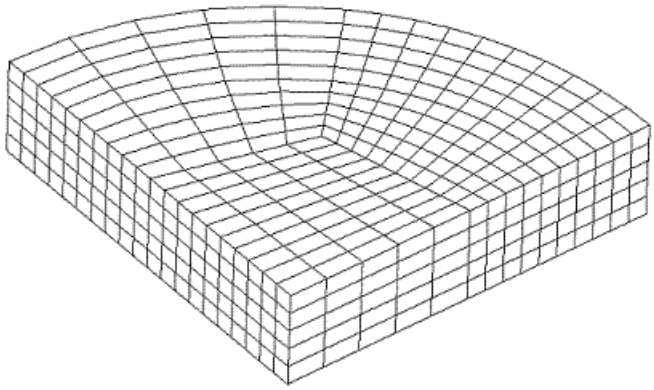


Figure 1. Tri-section Radialmesh

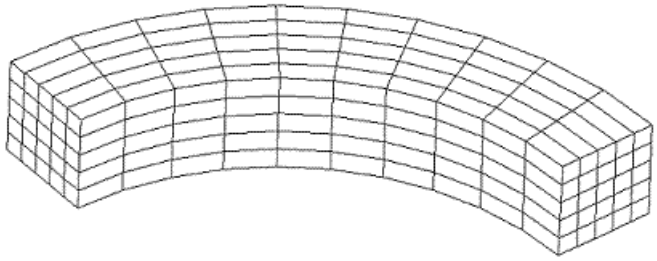


Figure 2. Non-tri-section Radialmesh

The command to generate the mesh in Figure 1 is:

```
create radialmesh \
      numZ 1 zblock 1 1 interval 5 \
      numR 3 trisection rblock 1 2 interval 5 \
                  rblock 2 3 interval 5 \
                  rblock 3 4 interval 5 \
      numA 1 span 90 ablock 1 interval 10
```

The command to generate the mesh in Figure 2 is:

```
create radialmesh \
      numZ 1 zblock 1 1 interval 5 \
      numR 1 initial radius 3 rblock 1 4 interval 5 \
      numA 1 span 90 ablock 1 interval 10
```

A mesh can span an entire 360 degrees by using the “full360” keyword. For example, the mesh in Figure 3 was generated with the following command:

```
create radialmesh numZ 1 zblock 1 1 interval 5 \
   numR 3 trisection rblock 1 1 interval 5 \
        rblock 2 2 interval 5 \
        rblock 3 3 interval 5 \
   numA 5 full360 span ablock 1 interval 5 \
        ablock 2 interval 5 \
        ablock 3 interval 5 \
        ablock 4 interval 5
```

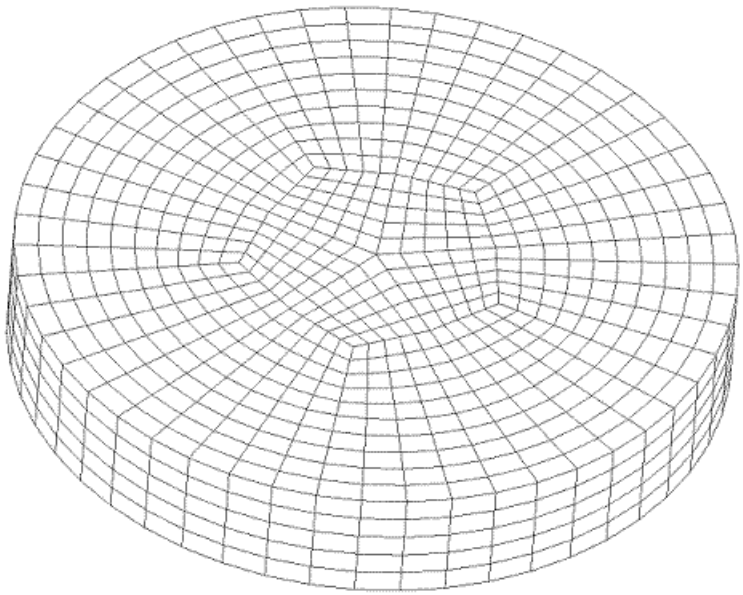


Figure 4. Radialmesh using full360 option

After the mesh is generated, the radialmesh command fits the mesh with mesh based geometry. The surfaces created to fit the mesh are given special names according to their location on the geometry. To see the names of the surfaces, issue the command **label surface name** after creating a radialmesh. Also, if you create a tri-section mesh, the edges on the center axis are given names. To see these names issue the command **label curve name** after creating a trisection Radialmesh.

The user can control the number of intervals and the spacing of these intervals using the optional parameters in each rblock, zblock and ablock. There are 11 combinations that these can be combined as listed below:

- **Interval Only**- Example: "interval 5." The block will be meshed with 5 equally spaced intervals.
- **First Size Only**- Example: “first size 2.5.” The block will be meshed with intervals of approximately 2.5 in length. The total number of intervals is internally calculated and depends on the overall block length.
- **Fraction Only**- Example: “fraction 0.3333.” The block will be meshed with intervals approximately 0.3333*overall block length.
- **Interval and Bias**- Example: “interval 5 bias 1.5.” There will be 5 intervals on the block, which each interval being 1.5 times the previous one. The length of each interval is calculated internally.
- **Interval and Fraction**- Example: “interval 5 fraction 0.25.” There will be 5 intervals on the block, the first being .25 of the length of the block with the remaining decreasing in size.
- **Interval and First Size**- Example: “interval 5 first size 0.2.” There will be 5 intervals on the block, the first being 0.2 in length. The remaining intervals will increase or decrease to fill the blocks length.
- **First Size and Last Size**- Example: “first size 0.2 last size 0.4.” The first interval will be 0.2 in length. The last interval will be 0.4 in length. The total number of intervals is internally calculated to allow for transition between the 2 specified sizes.
- **First Size and Bias**- Example “first size 0.2 bias 0.85.” The first interval will be 0.2 in length and the remaining intervals will scale by a factor of 0.85 from one to the next until the block is filled. The total number of intervals is internally calculated and depends on the overall block length.

- **Fraction and Bias**- Example “fraction 0.25 bias 1.25.” The first interval will be 0.25 of the overall block length and the remaining intervals will scale by a factor of 1.25 from one to the next until the block is filled. The total number of intervals is internally calculated and depends on the overall block length.

- **Interval and Last Size**- Example: “last size 1.5 interval 5.” The last interval will be 1.5 in length. The remaining intervals will scale up or down to fit 5 intervals in the block.

- **Last Size and Bias**- Example: “last size 2.0 bias 1.1.” The last interval will be 2.0 in length. The remaining intervals will scale by 1.1 until the block is filled. The total number of intervals is internally calculated and depends on the overall block length.

Figure 5 shows an example of a bias spaced mesh with the following command:

```
create radialmesh numZ 2 zblock 1 1 first size 0.2 \
        zblock 2 10 first size 0.2 last size 1.0 \
    numR 3 trisection rblock 1 1 interval 5 \
        rblock 2 2 first size .25 \
        rblock 3 5 first size .25 bias 2.0 \
    numA 1 span 90 ablock 1 interval 5
```

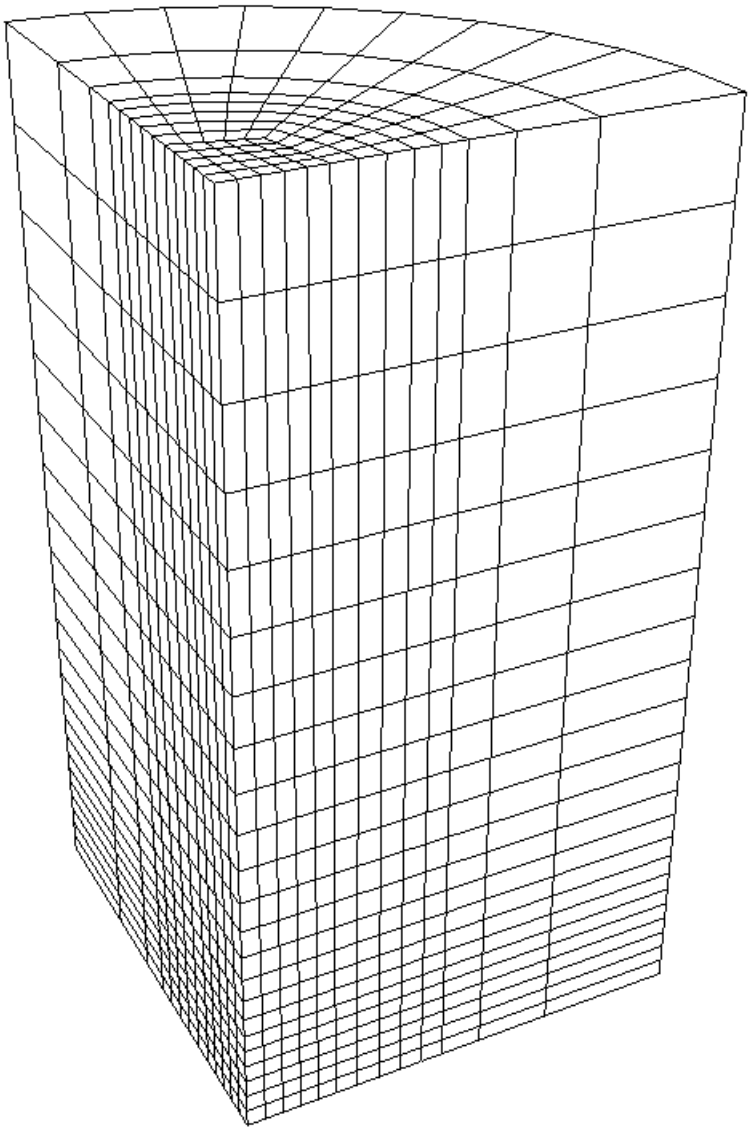


Figure 5. Radialmesh created with biased spacing

Automatic Mesh Quality Assessment

CUBIT performs an automatic calculation of mesh quality which warns users when a particular meshing scheme or other meshing operation has created a mesh whose quality may be inadequate. These warnings are supplied in case the user forgets to manually check the mesh quality.

CUBIT automatically calculates the SHEAR quality of hexahedral and quadrilateral elements and the SHAPE quality of tetrahedral and triangular elements. The SHEAR metric measures element skew and ranges between zero and one with a value of zero signifying a non-convex element, and a value of one being a perfect, right-angled element. The SHAPE metric also ranges between zero and one with a value of zero signifying a degenerate or inverted element and a value of one signifying a perfect, equilateral element. The quality of the mesh is then defined to be the minimum value of the shear metric for hexahedral and quadrilateral elements and the shape metric for tetrahedral and triangular elements, with the minimum taken over the elements in the mesh.

If the quality of the mesh is zero, the code reports *"ERROR: Negative Jacobian Element Generated"* to the command window. By default, if the quality of the mesh is positive but less than a certain threshold, the code reports *"WARNING: Poorly-Shaped Element Generated"* to the command window. Also reported in this case is the ID of the offending element, the value of its shear (or shape) metric, and the value of the threshold to which it was compared. The default value of the threshold parameter is 0.2. Users may change the threshold value by issuing the command

```
Set Quality Threshold <double=0.2>
```

The user may also change what type of message is printed in the case of a poor quality, but positive Jacobian mesh. This message can be printed as a warning (the default) or an error or can be turned off completely using the command

```
Set Print Quality { WARNING|Error|Off }
```

The above commands only affect the message generated for meshes with a quality greater than zero and less than the given threshold value; an error will always be generated for meshes with a quality of zero (that is, for meshes containing negative Jacobian elements).

Coincident Node Check

The ability to check for coincident nodes in the model is available in CUBIT. It uses an efficient octal hash tree to make the comparisons. The command is:

```
Quality Check Coincident Node [ In ]
[Group|Body|Volume|Surface|Curve|Vertex
<id_range> ] [ Merge [Delete] ] [
HIGHLIGHT|Draw [color <number>]] [List] [Into
Group [names|id] ]
```

If no entity list is given, the command works on all the nodes in the model. If an entity list is given, then it compares the nodes on those entities *with the rest of the nodes in the model*. By default the command highlights the coincident nodes in the graphics window and lists the total number of coincident nodes found. You can also have it clear the graphics and draw the nodes, and/or list the coincident node ids. Optionally, the coincident nodes found can be placed in a group.

If the model being operated on is from an imported universal file (i.e., no geometry exists in the model), you can merge the coincident nodes with the *merge* option. In this case *delete* allows you to delete the extra nodes (recommended). If you do not delete them they are placed into an output group.

You can control the tolerance used to check between nodes with the following setting (default = 1e-8):

```
set Node Coincident Tolerance [<value>]
```

Controlling Mesh Quality

If the quality of a model after meshing isn't acceptable, there are two options available to improve that quality. The user can ask for more smoothing, or delete the mesh and start over. There are some commands that the user can invoke before meshing the model which can help to improve mesh quality. Some of them are discussed here.

Skew Control

The philosophy behind the skew control algorithm is one of subdividing surfaces into blocky, four-sided areas which can be easily mapped. The goal of this subdivide-and-conquer routine is to lessen the skew that a mesh exhibits on submapped regions. By controlling the skew on these surfaces, the mesh of the underlying volume will also demonstrate less skew.

The commands for skew control are:

```
Control Skew Surface <surface_id_range>
[Individual]

Delete Skew Control Surface {surface_list}
[Propagate]
```

The keyword **Individual** is deprecated. Its purpose is to specify that surfaces should be processed without regards to the other surfaces in the given list. This is not necessary, and could lead to problems with the final mesh. When the command is entered, the algorithm immediately processes the surfaces, inserting vertices and setting interval constraints on the resulting subdivided curves. In this way, the mesh is more constrained in its generation, and the resulting skew on the model can be lessened. The only surfaces that can utilize this algorithm are those that lend themselves to a structured meshing scheme, although future releases might lessen this restriction.

The user also has the ability to delete the changes that the skew control algorithm has made. This is done by using the **delete skew control** command.

When the user requests the deletion of the skew control changes on a given surface, every curve on that surface will have the skew control changes deleted, even if a given curve is shared with another surface on which skew control was performed. If the user wishes to propagate the deletion of skew control to all surfaces which are affected by one (or more) particular surfaces, the keyword **propagate** should be used.

Propagate Curve Bias

When a bias mesh scheme is applied to a curve, this sometimes creates skewing of the surface mesh that is attached. Sometimes the user will want to ensure that the same bias is applied to curves on attached surfaces so that this skewing is minimized. The command for doing this is:

```
Propagate Curve Bias
[Surface|Volume|Body|Group <id_list>]
```

This command will search out all simply mappable surfaces in the input list, find which curves of those have a bias scheme set, and will propagate that bias across the mappable surfaces.

Adjust Boundary

```
Adjust Boundary {Surface|Group} <id_range>
[Angle <double>]
```

This command can be used to improve element quality for mapped or submapped surface meshes. Often, due to vertex positions, the curve meshing for a surface will lead to a poor quality surface mesh. This command can be used to adjust the curve meshes in an attempt to generate a better quality surface mesh. The command works by looking at the angle the mesh edges leave the boundary. In a perfect mapped or submapped mesh, the mesh edges will be orthogonal to the boundary, or will go off at 90 degree angles. The adjust boundary command looks at the deviation of the mesh edges, and if it is greater than the prescribed angle deviation, it will move the node location such that it is 90 degrees, if possible. The deviation angle by default is 5 degrees and can be changed by the user through the **[Angle <double>]** option in the command. In order to modify the curve meshes, the surface meshes are first deleted then later remeshed after the curve meshes have been repositioned and fixed. This command assumes that the volumes attached to the surface have not been meshed, if they have been, the command will return an error message. It should be noted that this command, while useful, may not always work due to interval constraints (i.e., you may need to change the intervals on the surface), or if the surfaces are not very blocky.

Metrics for Edge Elements

The metrics used for edge elements in CUBIT are summarized in the following table:

Function Name	Dimension	Full Range	Acceptable Range
Length	L^0	0 to inf	None

Quality Metric Definitions:

Length: Distance between beginning and ending nodes of an edge

Comments on Algebraic Quality Measures

1. The quality command for edge length only accepts edge elements as input; it does not accept geometry as input.
2. The length metric is currently only available for edge elements. Edge elements are created by default when curves and surfaces are meshed. Edge elements are not created for interior volume elements.

Metrics for Hexahedral Elements

The metrics used for hexahedral elements in CUBIT are summarized in the following table:

Function Name	Dimension	Full Range	Acceptable Range	Reference
Aspect Ratio	L^0	1 to inf	1 to 4	1
Skew	L^0	0 to 1	0 to 0.5	1
Taper	L^0	0 to +inf	0 to 0.4	1
Stretch	L^0	0 to 1	0.25 to 1	2
Diagonal Ratio	L^0	0 to 1	0.65 to 1	3
Dimension	L^1	0 to inf	None	1
Condition No.	L^0	1 to inf	1 to 8	5
Mass Increase Ratio	L^0	1 to inf	None	
Node Distance	L^1	-inf to inf	None	
Scaled Jacobian	L^0	-1 to +1	0.5 to 1	5
Shear	L^0	0 to 1	0.3 to 1	5
Shape	L^0	0 to 1	0.3 to 1	5
Relative Size	L^0	0 to 1	0.5 to 1	5
Shear & Size	L^0	0 to 1	0.2 to 1	5
Shape & Size	L^0	0 to 1	0.2 to 1	5
Timestep	Seconds	0 to inf	None	6
High Order Metrics				
Distortion	L^0	0 to 1	0.6 to 1	7
Element Volume	L^3	-inf to inf	None	1
Jacobian	L^3	-inf to inf	None	5

Hexahedral Quality Definitions

With a few exceptions, as noted below, Cubit supports quality metric calculations for linear hexahedral elements only. When calculating quality metrics, that only support linear elements, for a higher order hexahedral element, Cubit will only use the corner nodes of the element.

Aspect Ratio: Maximum edge length ratios at hex center.

Skew: Maximum |cos A| where A is the angle between edges at hex center.

Taper: Maximum ratio of lengths derived from opposite edges.

Stretch: Sqrt(3) * minimum edge length / maximum diagonal length.

Diagonal Ratio: Minimum diagonal length / maximum diagonal length.

Dimension: Pronto-specific characteristic length for stable timestep calculation. Char_length = Volume / 2 grad Volume.

Condition No. Maximum condition number of the Jacobian matrix at 8 corners.

Mass Increase Ratio: This metric stems from the global target time step and the element time step. The density required to fulfill the target time step (via mass scaling) divided by the block density is termed the mass increase ratio. Because the density within each element is constant, a ratio in the element density is equivalent to a ratio in the element mass.

This metric calculates the requisite density for each element to attain the prescribed target time step. If that density is greater than the defined density, the metric yields a value greater than one. This desired global time step is set by the user with the command:

```
[Set] Target Timestep <value>
```

As stated, this metric computes the element based timestep metric and consequently element blocks must be defined with material properties of Young's modulus, Poisson's ratio, and a target timestep must be set.

If this metric is computed in the context of a block ('quality block 1 mass increase ratio') an accompanying printout of the mass increase per block is given.

Node Distance: Minimum distance between any two adjacent corner nodes.

Scaled Jacobian: For linear elements the minimum Jacobian divided by the lengths of the 3 edge vectors.

Shear: 3/Mean Ratio of Jacobian Skew Matrix

Shape: 3/Mean Ratio of weighted Jacobian Matrix

Relative Size: Min(J, 1/J), where J is the determinant of weighted Jacobian matrix

Shear & Size: Product of Shear and Size Metrics

Shape & Size: Product of Shape and Size Metrics

Timestep: The approximate maximum timestep that can be used with this element in explicit transient dynamics analysis. This critical timestep is a function of both element geometry and material properties. To compute this metric on hexes, the hexes must be contained in an element block that has a material associated to it, where the materials poisson's ratio, elastic modulus, and density are defined.

High Order Elements

The preceding metrics will measure quality based only on the 8 corner nodes of the hexahedron. The following metrics also take into account the mid nodes.

Distortion: {min(|J|)/actual volume}*parent volume, parent volume = 8 for hex. Cubit also supports Distortion calculations for hex20 elements.

Element Volume: For linear hexes, the jacobian at hex center. For higher-order hexes, the hex is subdivided into sub-tets, the volumes of which are summed.

Jacobian: Minimum pointwise volume of local map at 8 corners at center of hex. Cubit also supports Jacobian calculations for hex27 elements.

References for Hexahedral Quality Measures

1. (Taylor, 89)
2. FIMESH code
3. Unknown
4. (Knupp, 00)
5. P. Knupp, Algebraic Mesh Quality Metrics for Unstructured Initial Meshes, to appear in Finite Elements for Design and Analysis.
6. Flanagan, D.P. and Belytschko, T., 1984, "Eigenvalues and Stable Time Steps for the Uniform Hexahedron and Quadrilateral," Journal of Applied Mechanics, Vol. 51, pp.35-40.
7. SDRC/IDEAS Simulation: Finite Element Modeling - User's Guide

Mesh Quality Assessment

- Metrics for Edge Elements
- Metrics for Triangular Elements
- Metrics for Quadrilateral Elements
- Metrics for Tetrahedral Elements
- Metrics for Hexahedral Elements
- Metrics for Wedge Elements
- Metrics for High Order Elements
- Mesh Quality Command Syntax
- Mesh Quality Example Output
- Automatic Mesh Quality Assessment
- Controlling Mesh Quality
- Coincident Node Check
- Mesh Topology Check
- Find Intersecting Mesh
- Measuring Number of Tets Through the Thickness

The `quality' of a mesh can be assessed using several element quality metrics available in CUBIT. Information about the CUBIT quality metrics can be obtained from the command

```
Quality Describe {Hex | Hexahedral | Tet |
Tetrahedral | Face | Quad | Quadrilateral |
Tri | Triangular}
```

which gives data on the quality metrics for each of the above element types. The following pages discuss the mesh quality assessment capabilities in CUBIT.

Mesh Quality Example Output

The typical summary output from the command **quality surface 24** is shown in Figure 1. Figure 2 shows the corresponding histogram. The colored element display resulting from the **command quality surface 1 draw `Skew'** is shown Figure 3. A color legend is also printed to the console as shown in Figure 4.

```
Surface 24 Quad quality, 292 elements:
---------------------------------------
  Function Name    Average    Std Dev     Minimum   (id)    Maximum  (id)
---------------  ---------  ---------  -------------  -------------
   Aspect Ratio 1.339e+00  3.374e-01  1.001e+00 (244)  3.662e+00 (132)
           Skew 1.848e-01  1.461e-01  7.986e-04 (212)  6.440e-01 (284)
         Taper  1.342e-01  9.397e-02  8.689e-03 (164)  5.500e-01 (133)
       Warpage  9.991e-01  4.465e-03  9.283e-01 (14)  1.000e+00 (82)
  Element Area  6.075e-04  4.725e-04  4.941e-05 (248)  2.202e-03 (274)
       Stretch  7.276e-01  1.233e-01  3.266e-01 (147)  9.587e-01 (161)
 Maximum Angle  1.099e+02  1.329e+01  9.079e+01 (82)  1.738e+02 (14)
 Minimum Angle  7.143e+01  1.185e+01  3.373e+01 (135)  8.955e+01 (82)
 Condition No.  1.250e+00  6.244e-01  1.003e+00 (161)  1.107e+01 (14)
      Jacobian  5.125e-04  4.273e-04  9.696e-06 (14)  1.918e-03 (274)
Scaled Jacobian 9.044e-01  1.104e-01  1.072e-01 (14)  9.999e-01 (82)
         Shear  9.045e-01  1.104e-01  1.072e-01 (14)  9.999e-01 (82)
         Shape  8.436e-01  1.314e-01  9.033e-02 (14)  9.966e-01 (161)
 Relative Size  3.036e-01  2.531e-01  3.226e-03 (248)  9.710e-01 (45)
Shear And Size  2.789e-01  2.361e-01  1.477e-03 (14)  9.389e-01 (45)
Shape And Size  2.609e-01  2.234e-01  1.245e-03 (14)  9.389e-01 (45)
    Distortion  8.118e-01  1.352e-01  9.654e-02 (14)  9.864e-01 (82)
```

Figure 1. Typical Summary for a Quality Command

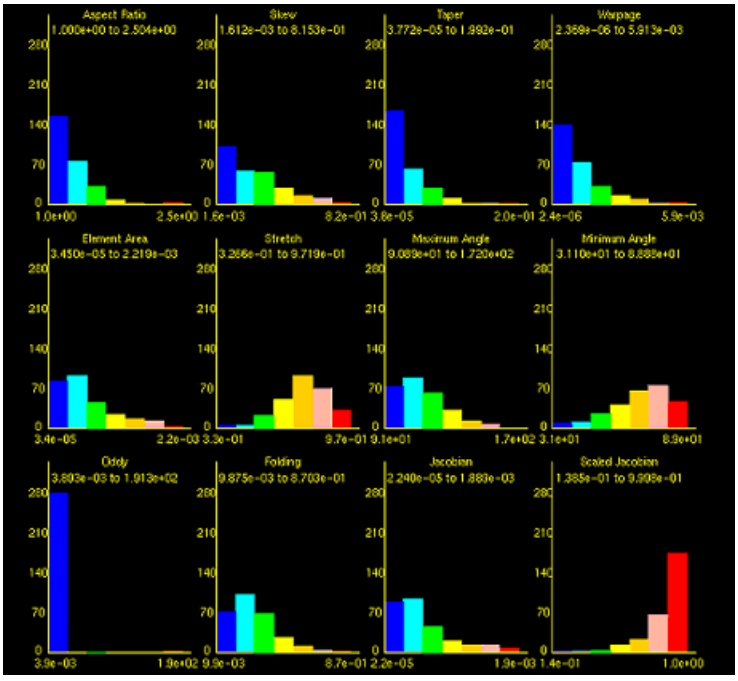


Figure 2. Histogram output from command "Quality Surface 24 Draw Histogram"

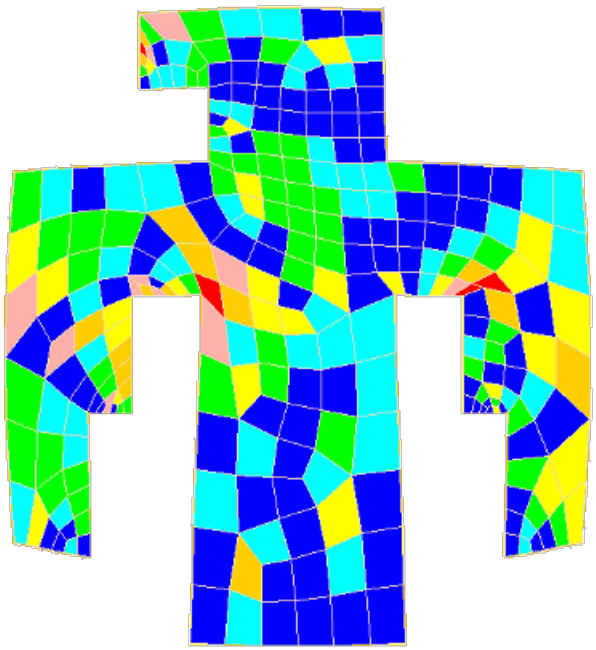


Figure 3. Graphical output of quality metric for command "Quality Surface 24 Skew Draw Mesh"

```
Surface 24 Quad quality, 280 elements:
    Skew ranges from 1.612e-03 to 8.153e-01 (280 entities)
    Blue ranges from 1.612e-03 to 1.178e-01 (102 entities)
    Cyan ranges from 1.178e-01 to 2.341e-01 (60 entities)
   Green ranges from 2.341e-01 to 3.503e-01 (58 entities)
  Yellow ranges from 3.503e-01 to 4.666e-01 (29 entities)
DkYellow ranges from 4.666e-01 to 5.828e-01 (15 entities)
    Pink ranges from 5.828e-01 to 6.990e-01 (12 entities)
     Red ranges from 6.990e-01 to 8.153e-01 (4 entities)
```

Figure 4. Legend for command "Quality Surface 1 Skew Draw Mesh"

Mesh Quality Command Syntax

The base command to view the quality of a mesh is the following:

```
Quality {geom_and_mesh_list} [metric name]
[quality options] [filter options]
```

Where the list contains surfaces and volumes and groups that have been meshed with faces, triangles, hexes, and tetrahedra; the list can also specify individual mesh entities or ranges of mesh entities.

If a specific metric name is given, only that metric or metrics are computed for the specified entities. Note that the metric given must be one which applies to the given entities. To see a list of quality metrics for individual entities see the Mesh Quality Assessment section and select the desired entity type: hexahedral, tetrahedral, quadrilateral, triangle. or edge

The metric name can also be more general than a specific metric. Four generalized options for metric name can be used:

Allmetrics: All of the metrics corresponding to the element type of the geom_and_mesh_list will be computed and reported.

Algebraic: All algebraic metrics corresponding to the element type of the geom_and_mesh_list will be computed and reported (e.g., Shape, Shear, Relative Size).

Robinson: All Robinson metrics corresponding to the element type of the geom_and_mesh_list will be computed and reported (e.g., Aspect Ratio, Skew, Taper).

Traditional: All the traditional Cubit metrics corresponding to the element type of the geom_and_mesh_list will be computed and reported (e.g., area, volume, angle, stretch, dimension).

If no metric name is supplied, the default metric is "**Shape**".

Quality Options

The quality options are:

Scope

```
[ Global | Individual ]
```

If the user specifies **individual**, one quality summary is generated for each entity specified on the command line. If the user specifies **global**, or specifies neither, then one quality summary is generated for each mesh element type.

Draw

```
[ Draw [Histogram] [Mesh] [Monochrome] [Add] ]
```

If the user specifies **draw histogram**, then histograms are drawn in a separate graphics window. The window contains one histogram for each quality metric. If the user specifies draw mesh, then the mesh elements are drawn in the default graphics window. A color-coded scale will appear in the graphics window. The histogram and mesh graphics are color coded by quality: a small metric value corresponds to red, a large metric value to blue and in-between values according to the rainbow. You can grab the side of color bar and resize it. The text gets smaller as the color bar width decreases. You can also grab in the middle of the color bar

and move it around. It can be repositioned to the bottom or top and it will automatically change orientations. See Figure 1.

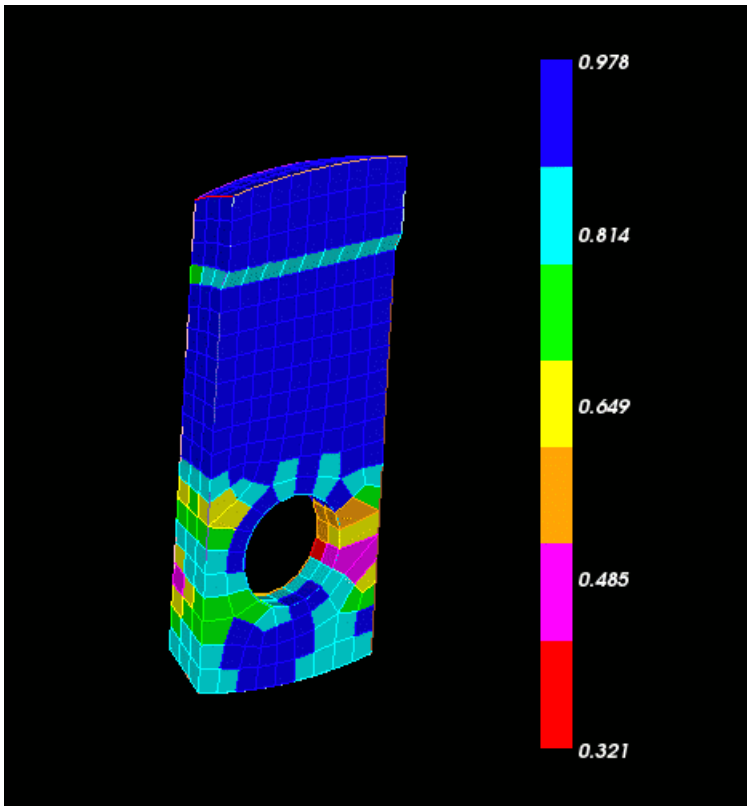


Figure 1. Quality Scale

If monochrome is specified, then the graphics are not color-coded. If add is specified, then the current display is not cleared before drawing the mesh elements.

List

```
[ List [Detail] [Id] [Verbose Errors] ]
[Geometry]
```

If the user specifies **List**, then the quality data is summarized in text form. **List Detail** lists the mesh elements by ascending quality metric. **List Id** lists the ids of the mesh elements. If **Verbose Errors** is specified, then details about unacceptable quality elements are printed out above the summaries. If **Geometry** is specified, then a list of the geometric entities that own the elements will be printed.

Filter

There are several options available to *filter* the output of the quality command, using the following filter options :

```
[High <value>] [Low <value>]
```

Discards elements with metric values above or below value; either or both can be used to get elements above or below a specified value or in a specified range.

```
[Top <number>] [Bottom <number>]
```

Keeps only number elements with the highest or lowest metric values. For example, " **Quality hex all aspect ratio top 10** " would request the elements with the 10 highest values of the aspect ratio metric.

Metrics for Quadrilateral Elements

The metrics used for quadrilateral elements in CUBIT are summarized in the following table:

Function Name	Dimension	Full Range	Acceptable Range	Reference
Aspect Ratio	L^0	1 to inf	1 to 4	1
Skew	L^0	0 to 1	0 to 0.5	1
Taper	L^0	0 to +inf	0 to 0.7	1
Warpage	L^0	0 to 1	0.9 to 1.0	NEW
Stretch	L^0	0 to 1	0.25 to 1	2
Minimum Angle	degrees	0 to 90	45 to 90	3
Maximum Angle	degrees	90 to 360	90 to 135	3
Condition No.	L^0	1 to inf	1 to 4	4
Deviation	L^2	0 to inf	None	
Jacobian	L^2	-inf to inf	None	4
Scaled Jacobian	L^0	-1 to +1	0.5 to 1	4
Shear	L^0	0 to 1	0.3 to 1	5
Shape	L^0	0 to 1	0.3 to 1	5
Relative Size	L^0	0 to 1	0.3 to 1	5
Shear & Size	L^0	0 to 1	0.2 to 1	5
Shape & Size	L^0	0 to 1	0.2 to 1	5
High Order Metrics				
Distortion	L^2	-1 to 1	0.6 to 1	6
Element Area	L^2	-inf to inf	None	1

Quadrilateral Quality Definitions

Aspect Ratio: Maximum edge length ratios at quad center

Skew: Maximum |cos A| where A is the angle between edges at quad center

Taper: Maximum ratio of lengths derived from opposite edges

Warpage: Cosine of Minimum Dihedral Angle formed by Planes Intersecting in Diagonals

Element Area: Jacobian at quad center

Stretch: Sqrt(2) * minimum edge length / maximum diagonal length

Minimum Angle: Smallest included quad angle (degrees).

Maximum Angle: Largest included quad angle (degrees).

Condition No. Maximum condition number of the Jacobian matrix at 4 corners

Jacobian: Minimum pointwise volume of local map at 4 corners & center of quad

Scaled Jacobian: For linear elements the minimum Jacobian divided by the lengths of the 2 edge vectors

Shear: 2/Condition number of Jacobian Skew matrix

Shape: 2/Condition number of weighted Jacobian matrix

Relative Size: Min(J, 1/J), where J is determinant of weighted Jacobian matrix

Shear and Size: Product of Shear and Relative Size

Shape and Size: Product of Shape and Relative Size

Distortion: {min(|J|)/actual area}*parent area, parent area = 4 for quad

Deviation: Absolute distance from quad centroid to its associated surface

Comments on Algebraic Quality Measures

Shape, Relative Size, Shape & Size, and Shear are algebraic quality metrics that apply to quadrilateral elements. Cubit encourages the use of these metrics since they have certain nice properties (see reference 5 below). The metrics are referenced to a square-shaped quadrilateral element, thus deviations from a square are measured in various ways.

Shape measures how far skew and aspect ratio in the element deviates from the reference element.

Relative size measures the size of the element vs. the size of reference element. If the element is twice or one-half the size of the reference element, the relative size is one-half. The reference element for the Relative Size metric is a square whose area is determined by the average area of all the quadrilaterals on the surface mesh under assessment

Shape and size metric measures how both the shape and relative size of the element deviate from that of the reference element.

The SHEAR metric is based on the condition number of the skew matrix. SHEAR is really just an algebraic skew metric but, since the word skew is already used in the list of quad quality metrics, Cubit has chosen to use the word 'shear.'

Shear = 1 if and only if quadrilateral is a rectangle.

The Robinson 'skew' metric equals the ideal (zero) if the quad is a rectangle. It also attains the ideal if the quad is a trapezoid, a kite, or even triangular!

References for Quadrilateral Quality Measures

1. (Robinson, 87)
2. FIMESH code.
3. Unknown.
4. (Knupp, 00)
5. P. Knupp, Algebraic Mesh Quality Metrics for Unstructured Initial Meshes, submitted for publication.
6. 6. SDRC/IDEAS Simulation: Finite Element Modeling--User's Guide

Details on Robinson Metrics for Quadrilaterals

The quadrilateral element quality metrics that are calculated are aspect ratio, skew, taper, element area, and stretch. The calculations are based on metrics described in (Robinson, 87). An illustration of the shape parameters is shown in Figure 1, below. The stretch metric is calculated by dividing the length of the shortest element edge divided by the length of the longest element diagonal.

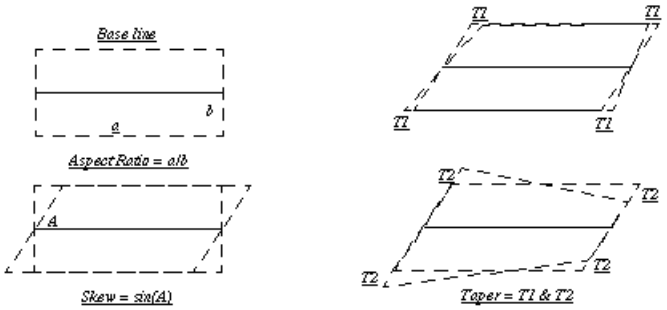


Figure 1. Illustration of Quadrilateral Shape Parameters (Quality Metrics)

Metrics for Tetrahedral Elements

The metrics used for tetrahedral elements in CUBIT are summarized in the following table:

Function Name	Dimension	Full Range	Acceptable Range	Reference
Aspect Ratio Beta	L^0	1 to inf	1 to 3	1
Aspect Ratio Gamma	L^0	1 to inf	1 to 3	1
Condition No	L^0	1 to inf	1 to 3	2
Node Distance	L^1	-inf to inf	None	
Scaled Jacobian	L^0	-1 to 1	0.2 to 1	2
Shape	L^0	0 to 1	0.2 to 1	3
Relative Size	L^0	0 to 1	0.2 to 1	3
Shape and Size	L^0	0 to 1	0.2 to 1	3
High Order Metrics				
Distortion	L^0	-1 to 1	0.6 to 1	
Element Volume	L^3	-inf to inf	None	1
Inradius	L^1	-inf to inf	None	None
Jacobian	L^3	-inf to inf	None	2
Scaled Jacobian	L^3	-1 to 1	0.2 to 1	
Normalized Inradius	L^0	-1 to 1	0.15 to 1	
Mean Ratio	L^0	0 to 1	0.2 to 1	
Mass Increase Ratio	L^0	1 to inf	None	
Timestep	Seconds	0 to inf	None	4

Tetrahedral Quality Definitions

With a few exceptions, as noted below, Cubit supports quality metric calculations for linear tetrahedral elements only. When calculating quality metrics, that only support linear elements, for a higher order tetrahedral element, Cubit will only use the corner nodes of the element.

Aspect Ratio Beta: CR / (3.0 * IR) where CR = circumsphere radius, IR = inscribed sphere radius

Aspect Ratio Gamma: Srms**3 / (8.479670*V) where Srms = sqrt(Sum(Si**2)/6), Si = edge length

Condition No.: Condition number of the Jacobian matrix at any corner

Inradius: For all tets but tetra10s, the radius of the smallest, fully contained sphere of the linear tet. For tetra10s, the mid-edge nodes are used to subdivide the tet into 12 linear sub-tets. The inradius is the smallest inradius of the 12 linear sub-tets * 2.3.

Jacobian: Minimum pointwise volume at any corner. Cubit also supports Jacobian calculations for tetra15 and tetra10 elements.

For tetra15 or tetra10 elements, all 15 or 10 nodes are included for the Jacobian calculation. For all other tet types, only the corner nodes are

considered.

Node Distance: Minimum distance between any two adjacent corner nodes.

Scaled Jacobian: For linear elements the minimum Jacobian divided by the lengths of 3 edge vectors

Shape: 3/Mean Ratio of weighted Jacobian Matrix

Relative Size: Min(J, 1/J), where J is the determinant of the weighted Jacobian matrix

Shape & Size: Product of Shape and Relative Size Metrics

High Order Elements

The preceding metrics will measure quality based only on the 4 corner nodes of the tetrahedron. The following metrics also take into account the mid nodes.

Distortion: {min(|J|)/actual volume}*parent volume, parent volume = 1/6 for tet. Cubit also supports Distortion calculations for tetra10 elements.

For tetra10 elements, the distortion metric can be used in conjunction with the shape metric to determine whether the mid-edge nodes have caused negative Jacobians in the element. The shape metric only considers the linear (parent) element. If a tetra10 has a non-positive shape value then the element has areas of negative Jacobians. However, for elements with a positive shape metric value, if the distortion value is non-positive then the element contains negative Jacobians due to the mid-side node positions.

Element Volume: For linear tets, (1/6) * Jacobian at corner node. For higher order tets, the tet is subdivided into sub-tets, the volumes of which are summed.

Normalized Inradius: Ratio of minimum subtet inner radius to tet outer radius (circumsphere). Subtets are defined by subdividing the tet into 12 smaller tets by using a common point at the centroid of the tet and the 6 mid-edge nodes as shown in Figure 1. The minimum in-radius of any of these 12 tets normalized by its parent outer-radius and a constant is used to determine this metric. The Normalized Inradius metric is also valid for linear elements, except that all mid-edge nodes are defined as the midpoint of their corner nodes.

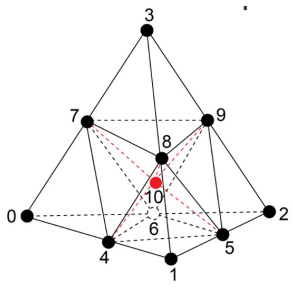


Figure 1. Subtet subdivision used for determining Normalized Inradius quality metric

Mean Ratio: General description: Mean ratio quality metric measures the deviation of a tetrahedral element from an equilateral tetrahedron through the root-mean-squared edge length. In this context, we employ a volume ratio. For a 4-node tet, the volume is compared to the cube of root-mean-squared length of the six edges. For the 10-node tet, 12 sub-tets are formed and minimum mean ratio of the 12 is returned. Unlike the

normalized inradius, the mean ratio is quite sensitive to a single, highly-elongated sub-tet. We note that for an equilateral 10-node tetrahedral element, there are two families of sub-tetrahedra. Sub-tets connected to the corner nodes or parent nodes of the 4-node tet have a mean ratio of 1 by construction. They are equilateral tets. The other family of sub-tets are not equilateral tets. These interior sub-tets connected entirely to mid-edge nodes are scaled such that all sub-tetrahedra have a mean ratio of 1 for an equilateral tet.

Mass Increase Ratio: This metric stems from the global target time step and the element time step. The density required to fulfill the target time step (via mass scaling) divided by the block density is termed the mass increase ratio. Because the density within each element is constant, a ratio in the element density is equivalent to a ratio in the element mass. This metric calculates the requisite density for each element to attain the prescribed target time step. If that density is greater than the defined density, the metric yields a value greater than one. This desired global time step is set by the user with the command:

```
[Set] Target Timestep <value>
```

As stated, this metric computes the element based timestep metric and consequently element blocks must be defined with material properties of Young's modulus, Poisson's ratio, and a target timestep must be set.

If this metric is computed in the context of a block ('quality block 1 mass increase ratio') an accompanying printout of the mass increase per block is given.

Timestep: The approximate maximum timestep that can be used with this element in explicit transient dynamics analysis. This critical time step is a function of both element geometry and material properties. To compute this metric on tets, the tets must be contained in an element block that has a material associated to it, where the materials poisson's ratio, elastic modulus, and density are defined.

Note that, for tetrahedral elements, there are several definitions of the term "aspect ratio" used in literature and in software packages. Please be aware that the various definitions will not necessarily give the same or even comparable results.

References for Tetrahedral Quality Measures

1. (Parthasarathy, 93)
2. (Knupp, 00)
3. P. Knupp, Algebraic Mesh Quality Metrics for Unstructured Initial Meshes, to appear in Finite Elements for Design and Analysis.
4. Flanagan, D.P. and Belytschko, T., 1984, "Eigenvalues and Stable Time Steps for the Uniform Hexahedron and Quadrilateral," Journal of Applied Mechanics, Vol. 51, pp.35-40.
5. SDRC/IDEAS Simulation: Finite Element Modeling - User's Guide

Mesh Topology Check

The ability to check for non-manifold topology among mesh entities is given with the following command.

```
Quality Check Topology [[Hex <range>] [Tet
<range>] [Face <range>] [Tri <range>]]
```

If no entity list is given, it will check the entire model. Multiple element types are also allowed. The command checks for non-manifold boundaries (edges) in the element set entered. For quads and tris the command lists and highlights all edges that have more than two tris or faces connected.

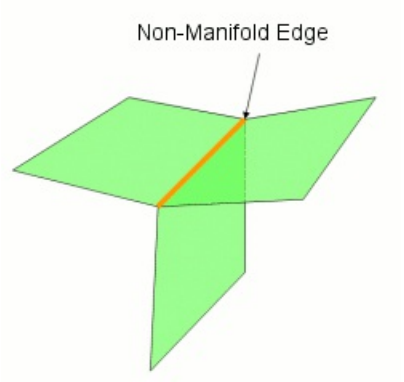


Figure 1. Topology check for quads and tris

For hexes and tets it looks for edges with two or more elements connected that do not share common faces.

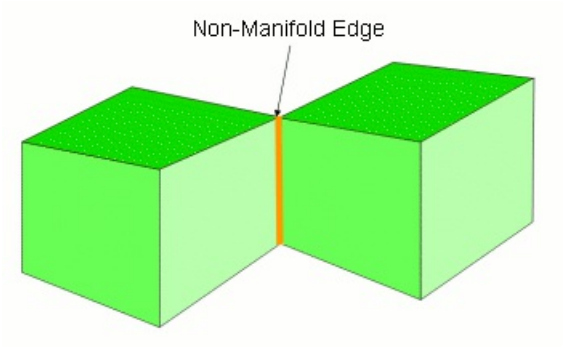


Figure 2. Topology check for hexes and tets

Additional topology checks fall into three categories:

- - model edges
- - coincident nodes
- - coincident quadrilateral(faces) or triangles

Model Edge Check

The model edge check will find edges with adjoining quadrilaterals or triangles whose angles between the surface normals exceed a specified value. The default angle is 40 degrees.

The following commands check for model edges:

```
Topology check model edge
{group|volume|surface|curve} <id_range> [angle
```

```
<value>] DRAW|nodraw|highlight] [BRIEF|verbose]
[RESULT GROUP[{<name>|{<id>}|nogroup]

Topology check model edge
{block|sideset|nodeset} <id_range> [angle
<value>] DRAW|nodraw|highlight] [BRIEF|verbose]
[RESULT GROUP[{<name>|{<id>}|nogroup]

Topology check model edge
{hex|tet|face|tri|edge} <id_range> [angle
<value>] DRAW|nodraw|highlight] [BRIEF|verbose]
[RESULT GROUP[{<name>|{<id>}|nogroup]
```

The optional **angle** parameter allows the user to specify a custom angle value against which the check will be performed. The default angle is **40 degrees**.

By default, the command will draw the model edges.

By default, very little information is output to the command line. The optional **verbose** parameter will output a list of the flagged model edges.

By default, the model edges will be written to the group **'model_edges'**. Optionally, the user may specify no grouping, or the user may specify the name or id of an existing group into which the model edges will be written. The contents of the existing group will be replaced by the model edges.

Interface Checks

Cubit will verify the interfaces between sections of a model. The existence of coincident nodes, for example, may not necessarily be an error in the model if the nodes are in sliding contact or are constrained by some type of multi-point constraint. The existence of coincident quadrilaterals or triangles may indicate that the model is not correctly joined.

The following commands check for coincident nodes.

```
Topology check coincident node
{group|volume|surface|curve|vertex} <id_range>
[tolerance <value>] DRAW|nodraw|highlight]
[BRIEF|verbose] [RESULT GROUP[{<name>|
{<id>}|nogroup]

Topology check coincident node
{block|sideset|nodeset} <id_range> [tolerance
<value>] DRAW|nodraw|highlight] [BRIEF|verbose]
[RESULT GROUP[{<name>|{<id>}|nogroup]

Topology check coincident node
{hex|tet|face|tri|edge|node} <id_range>
[tolerance <value>] DRAW|nodraw|highlight]
[BRIEF|verbose] [RESULT GROUP[{<name>|
{<id>}|nogroup]
```

The optional **tolerance** parameter allows the user to specify a custom tolerance value against which the check will be performed. The **default tolerance is 1.0 e-6**.

The default group name is **'coincident_nodes**.'

All other options behave similarly to those described above under Model Edge Check.

The following commands check for coincident quadrilaterals.

```
Topology check coincident quad
{group|volume|surface} <id_range> [tolerance
<value>] DRAW|nodraw|highlight] [BRIEF|verbose]
[RESULT GROUP[{<name>|{<id>}|nogroup]
```

```
Topology check coincident quad
{block|sideset|nodeset} <id_range> [tolerance
<value>] DRAW|nodraw|highlight] [BRIEF|verbose]
[RESULT GROUP[{<name>|{<id>}|nogroup]

Topology check coincident quad {hex|tet|face}
<id_range> [tolerance <value>]
DRAW|nodraw|highlight] [BRIEF|verbose] [RESULT
GROUP[{<name>|{<id>}|nogroup]
```

The default group name is '**coincident_quads**.'

All other optional parameters behave similarly to those described above.

The following commands check for coincident triangles.

```
Topology check coincident tri
{group|volume|surface} <id_range> [tolerance
<value>] DRAW|nodraw|highlight] [BRIEF|verbose]
[RESULT GROUP[{<name>|{<id>}|nogroup]

Topology check coincident tri
{block|sideset|nodeset} <id_range> [tolerance
<value>] DRAW|nodraw|highlight] [BRIEF|verbose]
[RESULT GROUP[{<name>|{<id>}|nogroup]

Topology check coincident tri
{hex|tet|face|tri} <id_range> [tolerance
<value>] DRAW|nodraw|highlight] [BRIEF|verbose]
[RESULT GROUP[{<name>|{<id>}|nogroup]
```

The default group name is '**coincident_tris**.'

All other optional parameters behave similarly to those described above.

Metrics for Triangular Elements

The metrics used for triangular elements in CUBIT are summarized in the following table:

Function Name	Dimension	Full Range	Acceptable Range	Reference
Maximum Angle	degrees	60 to 180	60 to 90	1
Minimum Angle	degrees	0 to 60	30 to 60	1
Aspect Ratio	L^0	1 to inf	1 to 3	
Aspect Ratio Alpha	L^0	1 to inf	1 to 3	
Condition No	L^0	1 to inf	1 to 1.3	2
Deviation	L^2	0 to inf	None	
Scaled Jacobian	L^0	-1 to 1	0.2 to 1	2
Relative Size	L^0	0 to 1	0.25 to 1	3
Shape	L^0	0 to 1	0.25 to 1	3
Shape and Size	L^0	0 to 1	0.25 to 1	3
High Order Metrics				
Distortion	L^2	-1 to 1	0.6 to 1	4
Scaled Jacobian	L^2	-1 to 1	0.2 to 1	
Element Area	L^2	0 to inf	None	1
Normalized Inradius	L^2	0 to 1	None	

Approximate Triangular Quality Definitions:

Maximum Angle: Maximum included angle in triangle

Minimum Angle: Minimum included angle in triangle

Aspect Ratio: Ratio of circumcirle to inradius

Aspect Ratio Alpha: Ratio of longest edge length to inradius

Condition No: Condition number of the Jacobian matrix

Deviation: Absolute distance from triangle centroid to associated surface

Scaled Jacobian: Minimum Jacobian divided by the lengths of 2 edge vectors

Relative Size: Min(J, 1/J), where J is determinant of weighted Jacobian matrix

Shape: 2/Condition number of weighted Jacobian matrix

Shape & Size: Product of Shape and Relative Size

Distortion: {min(|J|)/actual area}*parent area, parent area = 1/2 for triangular element

Element Area: (1/2) * Jacobian at corner node

Normalized Inradius: 4.0 * minimum_subtri_inradius / radius of circle containing three corner nodes

Comments on Algebraic Quality Measures

Relative Size, Shape, and Shape & Size are algebraic metrics, which have well behaved properties. Cubit encourages the use of these metrics over other metrics. These metrics are referenced to an ideal element which, in the case of triangular elements, is an equilateral triangle. Thus deviations from an equilateral triangle are measured in various ways by the algebraic metrics.
Relative size measures the size of the element vs. the size of reference element. If the element is twice or one-half the size of the reference element, the relative size is one-half. By default, the size of the reference element is the average size of all the elements that the quality command is currently evaluating.

The shape and size metric measures how both the shape and relative size of the element deviate from that of the reference element.

References for Triangular Quality Measures

1. Traditional.
2. Knupp, 2000.
3. P. Knupp, Algebraic Mesh Quality Metrics for Unstructured Initial Meshes, submitted for publication.
4. SDRC/IDEAS Simulation: Finite Element Modeling--User's Guide

Metrics for Wedge Elements

The metrics used for wedge elements in CUBIT are summarized in the following table:

Function Name	Dimension	Full Range	Acceptable Range
Aspect Ratio	L^0	1 to inf	1 to 3
Element Volume	L^3	-inf to inf	None
Condition No	L^0	1 to inf	1 to 3
Jacobian	L^3	-inf to inf	None
Scaled Jacobian	L^0	-1 to 1	0.2 to 1
Shape	L^0	0 to 1	0.2 to 1
Distortion	L^0	-1 to 1	0.6 to 1

Wedge Quality Definitions

With a few exceptions, as noted below, Cubit supports quality metric calculations for linear wedge elements only. When calculating quality metrics, that only support linear elements, for a higher order wedge element, Cubit will only use the corner nodes of the element.

Aspect Ratio: Maximum edge length ratios at the wedge center

Element Volume: Calculated by dividing the wedge into 11 tetrahedron and summing the volume of each.

Condition No.: Condition number of the Jacobian matrix at any corner

Jacobian: Minimum pointwise volume at any corner. Cubit also supports Jacobian calculations for Wedge21elements.

Scaled Jacobian: For linear elements the minimum Jacobian divided by the lengths of 3 edge vectors

Shape: 3/Mean Ratio of weighted Jacobian Matrix

Distortion: {min(|J|)/actual volume}*parent volume

Metrics supporting higher-order element types

The following tables details the quality metrics that support some higher order element types:

Edges

Function Name	Higher order types supported
Length	All

Triangles

Function Name	Higher order types supported
Distortion	tri6
Element Area	All
Normalized Inradius	tri6

Quadrilateral

Function Name	Higher order types supported
Distortion	quad8
Element Area	All

Tetrahedron

Function Name	Higher order types supported
Distortion	tetra10
Element Volume	All
Inradius	tetra10
Jacobian	tetra15
Normalized Inradius	tetra10
Mass Increase Ratio	tetra10
Mean Ratio	tetra10
Timestep	tetra10

Hexahedron

Function Name	Higher order types supported
Distortion	hex20
Element Volume	All
Jacobian	hex27

Measuring Number of Tets Through the Thickness

The ability to check the number of tets through the thickness is given with the following command.

```
Quality Surface <surf1_id> <surf2_id>
num_thru_thickness
```

Finding Intersecting Mesh

The **find mesh intersection** capability finds intersecting mesh between blocks, bodies, volumes, or surfaces. This command is useful for identifying cases where the geometry does not intersect but the mesh does. The command can find intersecting 2-dimensional mesh by specifying a list of surfaces, or 3-dimensional mesh by specifying blocks, bodies, or volumes. Surfaces that have mesh between them that *intersects* within a tolerance of 1e-6 are located. Finding surfaces with intersecting mesh is done using the command:

```
Find Mesh Intersection
{Block|Body|Surface|Volume} <id_list> [with
{Block|Body|Surface|Volume} <id_list>] [low
<value=0.0001>] [high <value>] [exhaustive]
[worst <num_worst>] [draw] [log]
[group<'name'>]
```

Finding Intersecting 2D Mesh

To find intersections between 2-dimensional mesh surfaces must be specified. If intersections are found, the surfaces containing the intersecting mesh are drawn (Figure 1) and the put into a group named 'surf_intersect', unless the user has specified another name using the **group <'name'>** option. Also, the ids of the intersecting surface pairs are printed to the terminal,

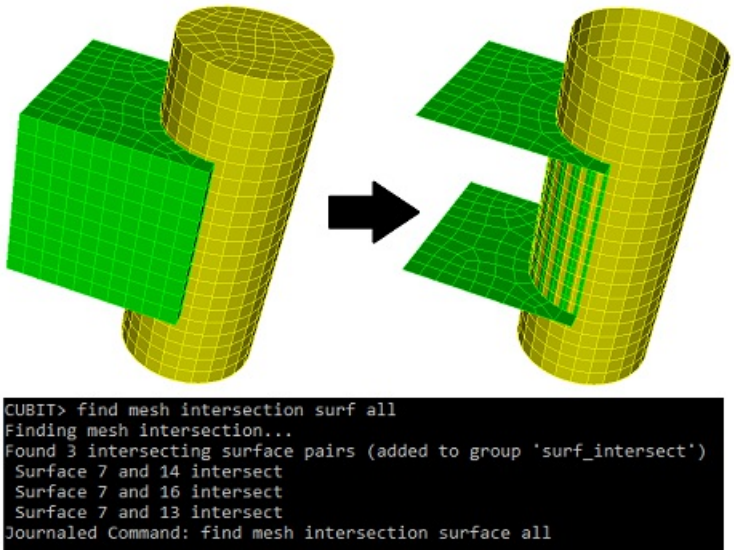


Figure 1. "Find mesh intersection surface all"

The **draw** option will draw the surfaces and intersecting mesh in wire frame mode, allowing the user to see exactly where on the surface the mesh intersection is (Figure 2).

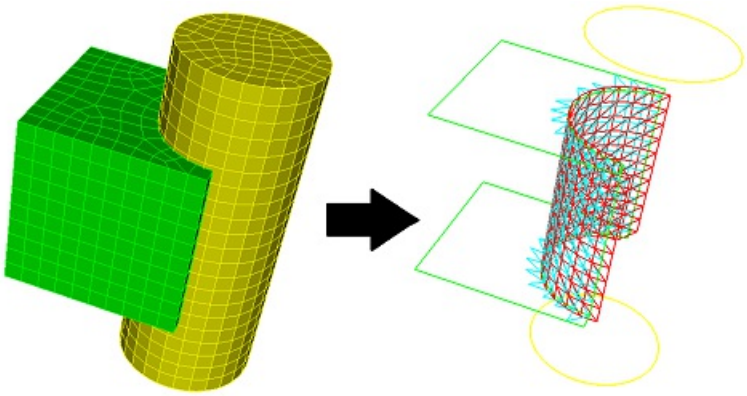


Figure 2. "Find mesh intersection surface all draw"

Facetted Representation

Detecting mesh intersections between surfaces works entirely off of the mesh, converting the mesh into triangular facets. (The facetted representation is what you see in a shaded view in the graphics). For example, a quad is split into two triangles. Higher order 2D elements are split into multiple triangles.

Drawing Mesh Intersection

The command below will draw the mesh intersection for only a pair of surfaces. The surfaces and intersecting mesh are drawn in wire frame mode, allowing the user to see exactly where on the surface the mesh intersection is.

```
Draw Surface <id> <id> mesh intersection [add]
[include_volume]
```

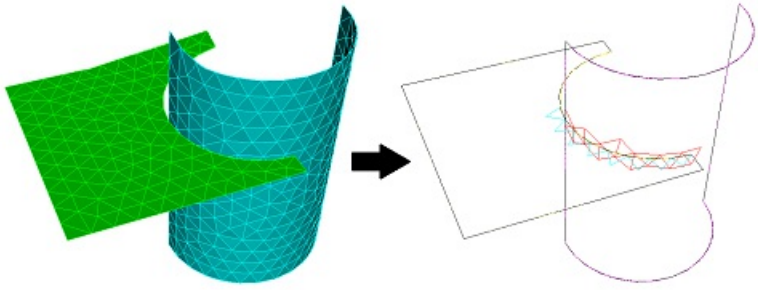


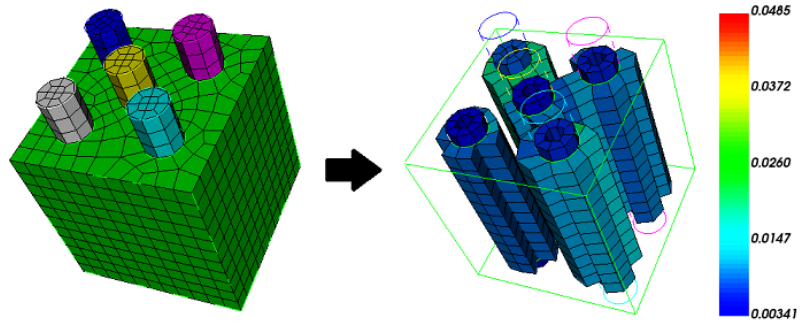
Figure 3. Draw Mesh Intersection

Finding Intersecting 3D Mesh

With 3-dimensional entities mesh element intersections can be located by specifying entities: blocks, bodies, or volumes. If intersections are found the intersecting elements are put into a group named 'mesh_intersect' unless the user has specified another name using the **group <'name'>** option. Data is printed to the terminal detailing the intersections. The largest intersection value is reported for each pair of intersecting entities (blocks, volumes, or bodies). This value is the fraction of an element's volume (which element belongs to the entity in the first column) that intersects elements belonging to the entity in the second column. See Figure 5 below. The information printed in columns from left to right is:

- the entities that have intersecting mesh
- the other entities with which that entity's mesh intersects
- the highest intersection value of an element belonging to the entity in column one

- the id of that element
- the number of elements of the entity in column one intersecting other elements of the entity in column two




```
CUBIT> find mesh intersection block  all
   Block     Intersecting with Block   % Worst Intersection   Element Id   #Elements
   -------   -----------------------   --------------------   ----------   ---------
   Block 5   Block 1                   0.048493               2006         104
   Block 4   Block 1                   0.033280               1754         104
   Block 3   Block 1                   0.030450               1538         104
   Block 1   Block 4                   0.024319               701          88
             Block 5                   0.021979               462          88
             Block 6                   0.012961               685          88
             Block 3                   0.012175               896          88
             Block 2                   0.011416               1228         88
   Block 2   Block 1                   0.020682               1430         104
   Block 6   Block 1                   0.020682               2198         104
Found 960 intersecting intersecting elements (added to group 'mesh_intersect')
```

Figure 5. "Find mesh intersection block all draw"

If the **with** option is used, the user specifies additional entities. The additional entities will not be reported in the first column of the output. This allows the user to focus on entities of interest without outputting too much data to the terminal. See Figure 6 below.

```
CUBIT> find mesh intersection block 1 with block all
   Block     Intersecting with Block   % Worst Intersection   Element Id   #Elements
   -------   -----------------------   --------------------   ----------   ---------
   Block 1   Block 4                   0.024319               701          88
             Block 5                   0.021979               462          88
             Block 6                   0.012961               685          88
             Block 3                   0.012175               896          88
             Block 2                   0.011416               1228         88
Found 440 intersecting intersecting elements (added to group 'mesh_intersect@A')
```

Figure 6. "Find mesh intersection block 1 with block all"

The **low** and **high** options set how much cumulative intersection should be detected. A low value of 0.1 would ignore elements that do not intersect more than 10% of their volume. Similarly, a high value of 0.5 would discard elements that intersect more than 50% of their volume. Both low and high can be used simultaneously. The default for the **low** value is 0.0001

The **exhaustive** option examines all elements for intersection. The default is to only examine elements with nodes on the boundary of the specified entities, anticipating that the intersections will occur mostly at boundaries.

The **worst** parameter limits the printout to the 'n' worst entities with elements of the highest intersections. The intersection fraction reported here is the cumulative intersection an element has with elements of all other entities in the check.

The **draw** option draws the intersecting elements using a color spectrum, red corresponding to high intersection and green to low. The color is according to cumulative intersection, as described in the **worst** option.

If the **log** option is specified, the output from the command will also be sent to a file named "mesh_intersection01.txt", with the number used in the file name incremented as needed. This becomes useful when you have hundreds of volumes with intersections.

Note:

- The shape of higher-order mesh elements is considered when computing intersection.
- 3D mesh intersection detection works on free mesh, as long as it has been placed into blocks.
- An intersection fraction greater than 100% is possible, when multiple elements in different entities intersect an element by more than its volume
- For removing mesh intersection, click here.

Mesh Modification

- Mesh Smoothing
- Mesh Refinement
- Mesh Scaling
- Mesh Pillowing
- Mesh Coarsening
- Mesh Cleanup
- Mesh Intersection Removal
- Node and Nodeset Repositioning
- Collapsing Mesh Edges
- Align Mesh
- Creating and Merging Mesh Elements
- Matching Tetrahedral Meshes
- Remeshing

After meshing is completed, it may be desirable to change features of the mesh without remeshing the whole volume. Mesh modification methods include tools for improving mesh quality, repositioning mesh elements, or changing mesh density. These methods can be applied on the whole model, or on small sections of the model without requiring remeshing the geometry, and without modifying the underlying geometry.

Adjust Boundary Orthogonal

Applies to: Surface Meshes

Summary: This smoother creates a near orthogonal grid and optionally will make an orthogonal grid if the geometry permits.

Syntax:

```
Adjust Boundary [Orthogonal] {Surface|Group}
<id_range> [Iterations <val>] [snap_to_normal
[curve <id>] [fixed curve <id>]]
```

Discussion:

Adjust Boundary Orthogonal iteratively applies the centroidal area pull algorithm with free boundary nodes. This approximates the affects of an elliptical smoothing algorithm. This algorithm works best with mapped meshes which have an element aspect ratio close to 1. The **snap_to_normal** option is not allowed for non-mapped meshes.

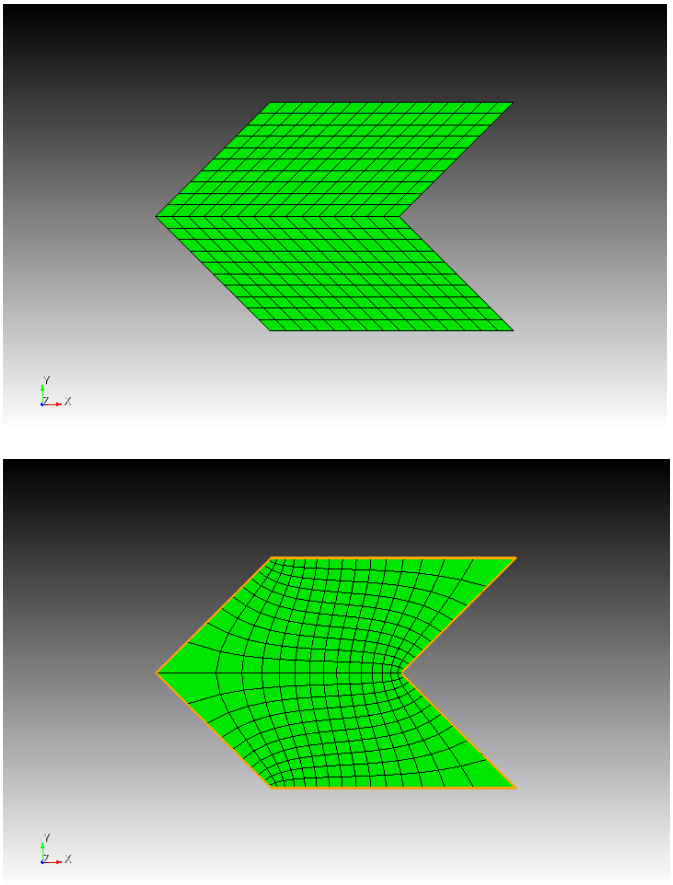


Figure 1. The affect of the "adjust boundary orthogonal surface 1" on a chevron shape. Note that the nodes are pulled into the acute angles and the edges at the boundary are pulled into a position that is closer to perpendicular at the boundary.

With some geometries with a mapped mesh it is possible to draw a line that is orthogonal to a boundary curve along the entire u or v direction of the mesh. In these cases, this command optionally allows the user to specify the option **snap_to_normal**. Nodal lines will be created normal to the first curve this is found that will allow perpendicular element edges to span the mesh. The user may optionally specify a curve that is used as the perpendicular basis for projecting the edges.

An edge may also be set as fixed so that a subsequent adjust boundary orthogonal will not affect that edge. If both **snap_to_normal** and **fixed** are set, the curve ids **MUST** be identical.

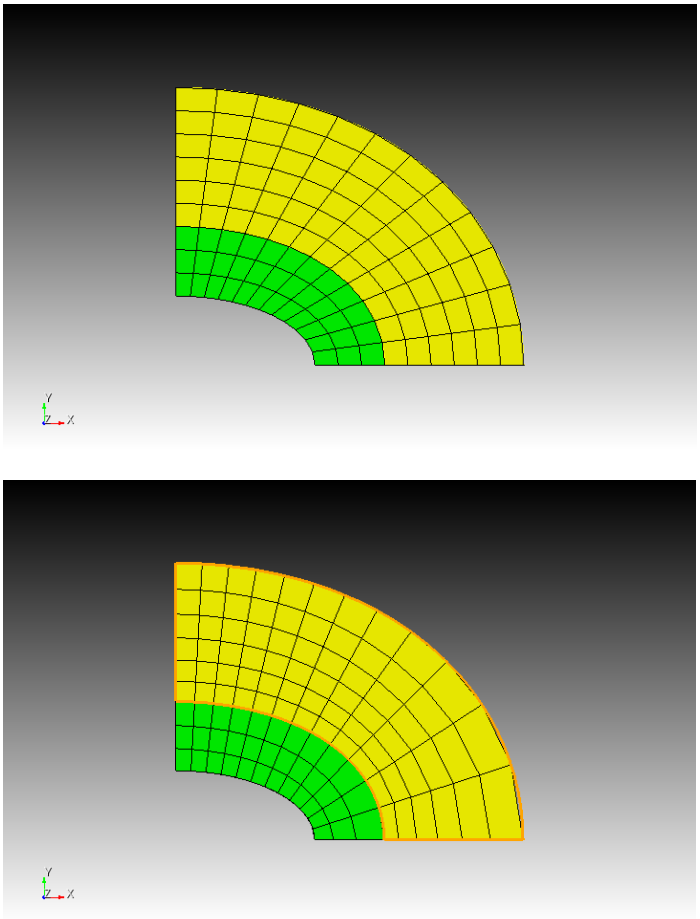


Figure 2. The affect of adjust boundary orthogonal with the snap to normal curve option is shown. The resulting mesh is orthogonal to the given boundary and projects straight through the mesh.

The following is an example of how to use this command to create the desired grid in Cubit. Note that to get the desired orthogonal grid the user must adjust the surfaces one at a time.

```
reset
create surface ellipse major radius 2 minor radius 1 zplane
imprint volume 1  with position 0 1 0
create curve offset curve 2  distance 1 extended
create curve offset curve 4  distance 2 extended
create surface skin curve 2 4
create surface skin curve 4 5
delete surface 1
merge all
surface all scheme map
mesh surf all
adjust boundary orthogonal surface 2  snap_to_normal curve 6
adjust boundary orthogonal surface 3  snap_to_normal curve 4  fixed
curve 4
```

Centroid Area Pull

Applies to: Surface Meshes

Summary: Attempts to create elements of equal area

Syntax:

```
Surface <range> Smooth Scheme Centroid Area
Pull [Free]
```

Discussion:

This smooth scheme attempts to create elements of equal area. Each node is pulled toward the centroids of adjacent elements by forces proportional to the respective element areas (Jones, 74).

The **free** option allows the nodes on the boundary (curves) to be moved during the smooth operation.

Condition Number

Applies to: Triangular or Quadrilateral Surface Meshes, Tetrahedral or Hexahedral Volume Meshes. Does not apply to Mixed Element Meshes.

Summary: Optimizes the mesh condition number to produce well-shaped elements.

Syntax:

```
Surface <surface_id_range> Smooth Scheme
Condition Number [beta <double=2.0>] [cpu
<double=10>]
```

Related Commands:

Untangle

Discussion:

The condition number smoother is designed to be the most robust smoother in Cubit because it guarantees that if the initial mesh is non-inverted then the smoothed mesh will also be non-inverted. The price exacted for this capability is that this smoother is not as fast as some of the other smoothers.

Condition Number measures the distance of an element from the set of degenerate (non-convex or inverted) elements. Optimization of the condition number increases this distance and improves the shape quality of the elements. Condition number optimization requires that the given mesh contain no negative Jacobians. If the mesh contains negative Jacobians and this command is issued, Cubit automatically calls the Untangle smoother and attempts to remove the negative Jacobians. If successful, condition number smoothing occurs next; the resulting mesh should have no negative Jacobians. If untangling is unsuccessful, condition number smoothing is not performed.

There is no "fixed/free" option with this command; boundary nodes are always held fixed.

The command above only sets the smoothing scheme; to actually smooth the mesh one must subsequently issue the command "smooth surface <surface_id_range>" or "smooth volume <volume_id_range>".

Stopping Criteria: Smoothing will proceed until the objective function has been minimized or until one of two user input stopping criteria are satisfied. To input your own stopping criterion use the optional parameters 'beta' and 'cpu' in the command above. The value of beta is compared at each iteration to the maximum condition number in the mesh. If the maximum condition number is less than the value of beta, the iteration halts. In Cubit condition number ranges from 1.0 to infinity, with 1.0 being a perfectly shaped element. Thus the smaller the maximum condition number, the better the mesh shape quality. The default value of the beta parameter is 2.0. The value supplied for the "cpu" stopping criterion tells the code how many minutes to spend trying to optimize the mesh. The default value is 10 minutes. Optimization may also be halted by using "control-C" on your keyboard.

To view a detailed report of the smoothing in progress issue the command "set debug 91 on" prior to smoothing the surfaces or volumes. You will get a synopsis of whether or not untangling is needed first and whether the stopping criteria have been satisfied. In addition the following printout information is given for each iteration of the conjugate gradient numerical optimization:

Iteration=n, Evals=m, Fcn=value1, dfmax=value2,

> **time=value3 ave_cond=value4, max_cond=value5, min_jsc=value6**

n is the iteration count, **m** is the number of objective function evaluations performed per iteration, **value1** is the value of the objective function (this usually decreases monotonically), **value2** is the norm of the gradient (does not always decrease monotonically), and **value3** is the cumulative cpu time (in seconds) spent up to the current iteration. The minimum possible value of the objective function is zero but this is attained only for a perfect mesh. **ave_cond**, **max_cond**, and **min_jsc** are the average and maximum condition number, and the minimum scaled jacobian. **ave_cond** generally decreases monotonically because it is directly related to **value1**.

Edge Length

Applies to: Surfaces

Summary: This smoother tries to make all edge lengths equal

Syntax:

```
Surface <range> Smooth Scheme Edge Length
```

Discussion:

Edge Length smoothing in Cubit is provided by MESQUITE, a mesh optimization toolkit by Argonne National Laboratory and Sandia National Laboratories. (See Brewer, et al. 2003 for more details on the MESQUITE toolkit.) This smooth scheme may be useful for lengthening the shortest edge length in paved meshes.

Interior node positions are adjusted in an optimization loop where the optimal element has an ideal shape (square) and has an area equal to the average element area of the input mesh.

NOTE: This smoother should be avoided when the mesh contains high aspect-ratio elements that the user wants to keep.

Because this smoother essentially tries to make all the edge lengths equal, it is designed to work well on meshes whose elements have aspect ratios close to 1. The farther from 1 the aspect ratio is, the less applicable this smoother will be.

Equipotential

Applies to: Volume Meshes

Summary: Attempts to equalize the volume of elements attached to each node

Syntax:

```
Volume <range> Smooth Scheme Equipotential
[Free]
```

Discussion:

This smoother is a variation of the Equipotential (Jones, 74) algorithm that has been extended to manage non-regular grids (Tipton, 90). This method tends to equalize element volumes as it adjusts nodal locations. The advantage of the equipotential method is its tendency to "pull in" badly shaped meshes. This capability is not without cost: the equipotential method may take longer to converge or may be divergent. To impose an equipotential smooth on a volume, each element must be smoothed in every iteration--a typically expensive computation. While a Laplacian method can complete smoothing operations with only local nodal calculations, the equipotential method requires complete domain information to operate.

The **free** option allows the nodes on the boundary (surfaces and curves) to be moved during the smooth operation.

Laplacian

Applies to: Curve, Surface, and Volume meshes

Summary: Tries to make equal edge lengths

Syntax:

```
{Surface|Volume} <range> Smooth Scheme
Laplacian [Free] [Global]
```

Discussion:

The length-weighted Laplacian smoothing approach calculates an average element edge length around the mesh node being smoothed to weight the magnitude of the allowed node movement (Jones, 74). Therefore this smoother is highly sensitive to element edge lengths and tends to average these lengths to form better shaped elements. However, similar to the mapping transformations, the length-weighted Laplacian formulation has difficulty with highly concave regions.

Currently, the stopping criterion for curve smoothing is 0.005, i.e., nodes are no longer moved when smoothing moves the node less than 0.005 * the minimum edge length. The maximum number of smoothing iterations is the maximum of 100 and the number of nodes in the curve mesh. Neither of these parameters can currently be set by the user.

The **free** option allows the nodes on the boundary (surfaces and curves) to be moved during the smooth operation.

Using the **global** keyword when smoothing a group of surfaces will allow smoothing of mesh on shared curves to improve the quality of elements on both surfaces sharing that curve.

Mean Ratio

Applies to: Triangular or Quadrilateral Surface Meshes, Tetrahedral or Hexahedral Volume Meshes. Does not apply to Mixed Element Meshes.

Summary: Moves interior mesh nodes to optimize the average mean ratio metric value of the mesh.

Syntax:

```
Surface <surface_id_range> Smooth Scheme Mean
Ratio [cpu <double=10>]

Volume <volume_id_range> Smooth Scheme Mean
Ratio [cpu <double=10>]
```

Discussion:

CUBIT includes a mean ratio smoother provided by MESQUITE, a mesh optimization toolkit by Argonne National Laboratory and Sandia National Laboratories. (See Brewer, et al. 2003 for more details on the MESQUITE toolkit.) This smoother is similar in purpose to the Condition Number smoother. However, the Mean Ratio smoother uses a second order optimization method, and therefore it will often reach a near-optimal mesh more quickly than the Condition Number smoother. The Mean Ratio smoother requires the initial mesh to be untangled, but the smoother is guaranteed to not tangle the mesh. If the user attempts to call the Mean Ratio smoother on a tangled mesh, an untangler will first attempt to untangle the mesh before calling the Mean Ratio smoother.

The Mean Ratio smoother's optimization process terminates when one of the following three criteria is met:

1. The mesh is "close" to an optimal mesh configuration.
2. The maximum allotted time has been exceeded.
3. The user interrupts the smoothing process.

The user has control over the second and the third criteria only. For criterion 2, the default is for the smoother to terminate after ten minutes even if a near-optimal mesh has not been reached. The user can change this time bound by specifying the optional "cpu" argument in the command listed above. This argument takes a single, positive number that represents the time (in minutes) that will be used as a time bound. If the user wishes to terminate the process early, criteria three allows the user to "interrupt" (for example, on some platforms, by pressing CTRL-C) the process. If the process is terminated early, the mesh will not revert to the original node positions; CUBIT will instead keep the partially optimized mesh.

Mesh Smoothing

- Centroid Area Pull
- Equipotential
- Laplacian
- Smart Laplacian
- Condition Number
- Mean Ratio
- Winslow
- Untangle
- Edge Length

Related Topics

- Smoothing mesh-based geometry
- Smoothing free meshes

After generating the mesh, it is sometimes necessary to modify that mesh, either by changing the positions of the nodes or by removing the mesh altogether. CUBIT contains a variety of mesh smoothing algorithms for this purpose. Node positions can also be fixed, either by specific node or by geometry entity, to restrict the application of smoothing to non-fixed nodes.

Mesh smoothing in CUBIT operates in a similar fashion to mesh generation, i.e. it is a two-step process whereby a smooth scheme is chosen and set, then a smooth command performs the actual smoothing. Like meshing algorithms, there is a variety of smoothing algorithms available, some of which apply to multiple geometry entity types and some which only apply to one specific type (these algorithms are described below.) To smooth the mesh on a geometry entity, the user must perform the following steps:

1. Set the smooth scheme for the object using the following command:

```
{Curve|Surface|Volume} <range> Smooth
Scheme <scheme>
```

where **<scheme>** is any acceptable smooth scheme described in this section. Also set any scheme-specific information, using the smooth scheme setting commands described below.

2. Smooth the object, using the command:

```
Smooth Curve <range>

Smooth Surface <range> [Global]

Smooth {Body|Volume|Group} <range>
```

Groups of entities may be smoothed, by smoothing a group or a body.

If a Body is specified, the volumes in that Body are smoothed. If a Group is specified, only the volume meshes within these groups are smoothed - no smoothing of the surface meshes is performed.

Global Smoothing

When smoothing a set of surfaces, the keyword **global** can be added to the smooth command such as

Smooth Surface <range> [Global]

If the smoothing algorithm for two neighboring surfaces are both allowed to move boundary nodes, then appending the "global" keyword will often result in a higher quality mesh near the curve(s) shared by those two

surfaces.

Focused Smoothing on Groups of Mesh Entities

Meshed entities such as hexes or tris can be smoothed individually or in groups by specifying the entities in a list.

```
Smooth {Hex|Tet} <range>[Scheme
{Equipotential|Laplacian|Random}]

Smooth {Face|Tri} <range>[Scheme
{Laplacian|Centroid|Winslow}] [Target Surface
<id>]

Smooth Edge <id_range> [Scheme Laplacian]
[Target Surface <id>]
```

The **Smooth Edge** command allows the user to smooth individual edges owned by a curve. Specifying a target curve allows the user to move the edges on a meshed curve to a different curve. The target curve or surface does not necessarily need to be the owning curve or surface of the nodes. For example, if given two curves (A and B) and curve A was meshed, the target smoothing could be used to move all of the edges of curve A onto curve B. The smooth scheme option for the edge smoothing is currently limited only to the laplacian scheme.

The **Smooth Face|Tri** command is used to smooth individual faces or triangles. The target option is similar to the curve target option above. Faces or Tris can be smoothed to a surface that is not necessarily the owning surface; in fact, the faces or tris do not even have to be attached to any surface. This makes this option especially helpful for smoothing free meshes. Specifying a smooth scheme allows for relaxation based surface smoothers (i.e. centroid area pull, laplacian, winslow) to be utilized during targeted smoothing. It is not currently enabled for optimization based smoothing schemes.

Smooth Tolerance

Smoothing algorithms move nodes in an attempt to improve the quality of the mesh elements. Most of these algorithms are iterative, and the algorithm terminates when some criterion is met. Specifically, for the Laplacian and Equipotential style smoothers, smoothing is terminated either by satisfying a smoothing tolerance or by performing the maximum number of smoothing iterations. For these smoothers, the smooth tolerance may be set by the user:

```
[Set] Smooth Tolerance <tol>
```

The value **<tol>** tells the smoother to stop when node movement is less than **tol * local_minimum_edge_length**.

The default value for tol is 0.05. The maximum number of iterations may be set by the user. For volumes, the smooth tolerance and iterations may also be set by

(Note: The above command affects all smoother that respect tolerance.)

```
Volume Smooth Tolerance <tol>

Volume Smooth Iterations <iters>
```

(Note: The above two commands only affect the volume smoothers.)

Boundary Mesh Smoothing

Where used in the smooth schemes below, the Free keyword permits the nodes lying on the bounding entities to "float" along those entities; without

this keyword, boundary nodes remain fixed.

- Laplacian
- Centroid Area Pull
- Winslow
- Equipotential

Nodal positions may be fixed so that no smoothing scheme, either implicit or explicit, will move them, with the following command:

```
{Curve|Surface|Volume} <range> Node Position
{Fixed|Free}

Node <range> Position {Fixed|Free}
```

The following command does not fix nodal positions, but does fix the connectivity of the mesh, preventing certain volume schemes from changing the bounding mesh:

```
{Curve|Surface|Volume} Mesh {Fixed|Free}
```

The additional following scheme is available for research purposes and can be used only after issuing a **'set developer on'** command.

- Randomize

Smart Laplacian

Applies to: Surface and Volume meshes

Summary: Tries to make equal edge lengths while ensuring no degradation in element shape

Syntax:

```
{Surface|Volume} <range> Smooth Scheme Smart
Laplacian
```

Discussion:

The Smart Laplacian smoothing approach is a variation on the standard Laplacian algorithm. The algorithm iteratively loops over the mesh and updates nodes based on the location of their neighbors. First, a patch of elements is formed around a given node. The quality of this patch is assessed to determine the quality of the worst shaped element. Then a new candidate node position is calculated as the average of the neighboring nodes. The quality of the patch is assessed again using the candidate node position. If there has been no degradation in the quality of the elements in the patch, the candidate node position is accepted; otherwise, the candidate node position is rejected and the node is returned to its previous position.

The Smart Laplacian smoother is intended to provide a reliable smoother that is nearly as fast as the Length-Weighted Laplacian smoother. Due to the dual goals of this smoother, making equal edge length and improving element shape, it will not always be able to make progress. However, it is often useful as a quick alternative to the more time-consuming optimization methods like Mean Ratio or Condition Number. When this smoother fails to make significant progress, the optimization methods can be tried.

The Smart Laplacian Smoother uses the Mean Ratio quality measure to assess element shape. This smoother is ensuring no degradation in the minimum Mean Ratio. The Mean Ratio smoother is optimizing the same metric, but it is attempting to improve the average Mean Ratio quality.

Untangle

Applies to: Triangular or Quadrilateral Surface Meshes Tetrahedral or Hexahedral Volume Meshes. Does not apply to Mixed Element Meshes.

Summary: Removes as many negative Jacobians from the mesh as possible by minimizing a certain objective function.

Syntax:

```
Surface <surface_id_range> Smooth Scheme
Untangle [beta <double=0.02>] [cpu
<double=10>]

Volume <volume_id_range> Smooth Scheme
Untangle [beta <double=0.02>] [cpu
<double=10>]
```

Related Commands:

```
Condition Number
```

Discussion:

The Untangle 'smoother' is designed to eliminate negative Jacobians from a given mesh by moving nodes to appropriate locations. If a mesh node is not involved in causing a negative Jacobian it will not be moved. If a mesh has no negative Jacobians, the Untangler will not move any of the nodes. This smoother is not magic: if an untangled mesh does not exist for the given mesh topology, the untangler will not untangle the mesh. Instead, it will do the best it can and exit gracefully. An untangled mesh produced by this smoother will often have poor shape quality; in that case it is recommended that untangling be followed by condition number smoothing. The untangle smoother is automatically called by the condition number smoother.

There is no "fixed/free" option with this command; boundary nodes are always held fixed. As a result, users should be aware that the volume untangler cannot succeed if the volume contains a surface mesh which contains a negative Jacobian. In that case, one must first remove the surface mesh negative Jacobians by invoking the surface Untangler and then invoke the volume Untangler.

The command above only sets the smoothing scheme; to actually smooth the mesh one must subsequently issue the command "smooth surface <surface_id_range>" or "smooth volume <volume_id_range>".

Stopping Criteria: Untangling will proceed until the objective function has been minimized or the optional user input "cpu" has been satisfied. The latter stopping criterion tells the code how many minutes to spend trying to untangle the mesh. The default value is 10 minutes. Optimization may also be halted by using "control-C" on your keyboard.

Beta Parameter: An optional user input parameter "beta" plays a role in determining the optimal mesh. Optimization proceeds until the minimum scaled Jacobian of the mesh is (roughly) greater than beta. To remove negative Jacobians one would need beta=0 (however, as a safety margin, we choose beta=0.02 as the default). To further improve the scaled Jacobian of the mesh, input a larger value of "beta". If a mesh with all scaled Jacobians greater than "beta" does not exist, optimization will continue until the cpu time stopping criterion has been met. Therefore, it is best not to use "beta" values too large (say, greater than 0.2) without also decreasing the cpu time limit.

To view a detailed report of the smoothing in progress issue the command "set debug 91 on" prior to smoothing the surfaces or volumes. You will get a synopsis of whether or not untangling is needed and

whether the stopping criteria are satisfied. In addition the following printout information is given for each iteration of the conjugate gradient numerical optimization:

```
Iteration=n, Evals=m, Fcn=value1,
dfmax=value2, time=value3 min_jsc=value4
```

n is the iteration count, **m** is the number of objective function evaluations performed per iteration, **value1** is the value of the objective function (this usually decreases monotonically), **value2** is the norm of the gradient (does not always decrease monotonically), and **value3** is the cumulative cpu time (in seconds) spent up to the current iteration. The minimum possible value of the objective function is zero; this value is attained only when the minimum scaled Jacobian of the mesh exceeds "beta". The **minimum scaled jacobian** is also reported.

Winslow

Applies to: Surface meshes

Summary: Elliptic smoothing technique for structured and unstructured surface meshes

Syntax:

```
Surface <range> Smooth Scheme Winslow [Free]
```

Discussion:

Winslow elliptic smoothing (Knupp, 98) is based on solving Laplaces equation with the independent and dependent variables interchanged. The method is widely used in conjunction with the mapping and submapping methods to give smooth meshes with positive Jacobians, even on non-convex two-dimensional regions. The method has been extended in CUBIT to work on unstructured meshes.

The **free** option allows the nodes on the boundary (curves) to be moved during the smooth operation.

Align Mesh

At times it is desirable to have identical meshes on two different surfaces or curves. The align mesh command will attempt to assign correspondence between nodes on surfaces or curves and move the nodes on one surface or curve to match the configuration on the other. The command syntax is:

```
Align Mesh Surface <id> [CloseTo] Surface <id>
[Tolerance <tol>]

Align Mesh Curve <id> [CloseTo] Curve <id>
[Tolerance <tol>]
```

These two commands align the mesh on the first entity with that of the second entity. This means that nodes on the first entity will be moved to the closest location possible to their corresponding nodes on the second entity. This is done without regard to mesh quality, so it is possible to invert elements with this command.

```
Align Mesh Node <id> [CloseTo] Node <id>
[Tolerance <tol>]
```

This command aligns the first node with the second node, within the limits of the geometric entities that own the nodes. This is also done without respect for element quality.

And example of this is given as follows:

brick x 10
volume 1 copy move 11
surface all except 10 6 vis off
transparent
graphics perspective off
at 5.552503 3.832384 0.134127
from 34.651051 3.640138 -0.193121
up 0.006514 0.999945 -0.008172 mesh surface all
surface 6 smooth scheme randomize free
smooth surface 6
node 432 move 0 0 -0.2
align mesh node 944 node 432
node 432 move 0 0 0.4
align mesh curve 23 closeto curve 12
align mesh surf 10 closeto surf 6

Collapsing Mesh

CUBIT currently offers several options for modifying an existing finite element mesh. Triangle and tetrahedral meshes sometimes contain low quality elements that can be removed by the following operations.

Collapsing Mesh Using Quality Metrics

The following first two commands collapse triangles, while the third collapses tetrahedra. Collapses are performed only if the collapse operation does not cause worse quality in the surrounding, surviving mesh; otherwise the collapse is not performed. The collapse operation removes the element by consolidating two neighbor nodes of the specified mesh entity into one, to either node location or their midpoint. The specified metric is used to determine quality, the default being Scaled Jacobian. The **Interior** option of the Collapse Tet command limits the collapse to the interior of the volume mesh, so that the triangle surface mesh is not modified.

```
Collapse Edge <ids> [SCALED JACOBIAN|Aspect
Ratio|Shape|Shape and Size]
```

```
Collapse Tri <ids> [SCALED JACOBIAN|Aspect
Ratio|Shape|Shape and Size]
```

```
Collapse Tet <ids> [Interior] [Altitude|Aspect
Ratio|Aspect Ratio
Gam|Distortion|Inradius|Jacobian|Normalized
Inradius|Node Distance|SCALED
JACOBIAN|Shape|Shape and Size|Timestep]
```

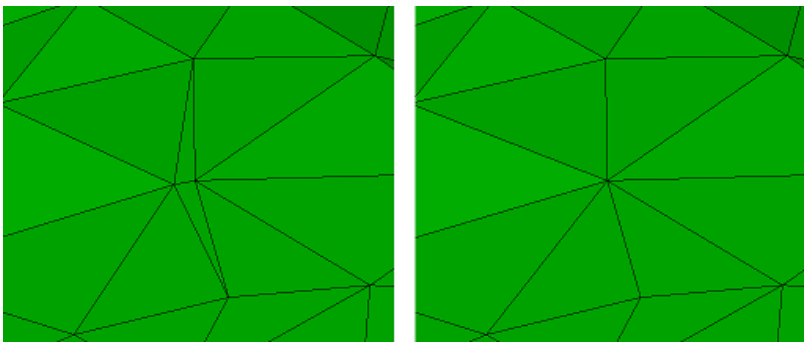


Figure 1. Before and After Triangle/Edge Collapse

Collapsing Mesh Manually

The following command collapses triangles containing the given edge, with no regard for quality.

```
Meshedit Collapse edge <id> [keep node <id>]
[compress_ids]
```

This command only works on triangle surface meshes. If volumetric elements, or quads, are attached to the edge, the command fails. The **keep node** option control which node to collapse to. The option **compress_ids** compresses holes in the mesh id space caused by the collapse.

Combining nodes to collapse tetrahedra can be done with the following command. The first node specified is merged into (deleted) the second node. If the first node cannot be moved due to geometry constraints (it's owned by a vertex), the operation fails. This command is behind a developer flag so **set dev on** must be issued before use.

```
Merge Node <id> <id>
```

Collapsing and Swapping Mesh Using Quality Metrics

Mesh quality can also be improved by swapping edges. The **improve tri** command uses both edge collapse and edge swap operations to improve triangle mesh quality. The operation that produces better quality, based on the specified metric, is used. Also how close the mesh approximates the geometry is considered. The command will fail if the triangles are in 3D mesh elements.

```
Improve Tri <ids> [SCALED JACOBIAN | Aspect
Ratio | Shape | Normalized Inradius]
```

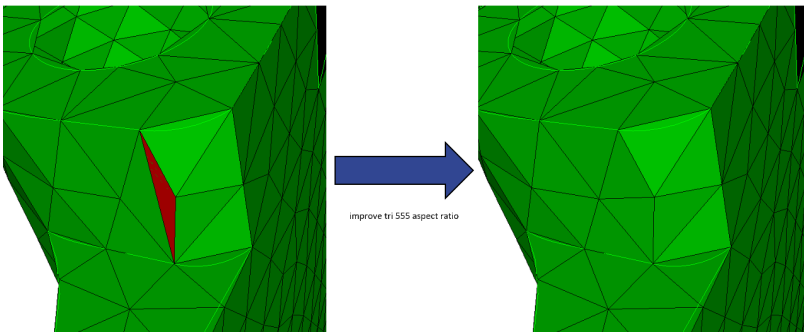


Figure 2. Before and After Improve Tri Command

Removing Overconstrianed Tets

Cubit offers commands to remove overconstrained tetrahedra - tets containing two triangles on the same surface and all four corner nodes on surfaces, curves, or vertices. These tets are not amenable to smoothing, leaving removal as the only viable solution to improve quality.

```
Remove Overconstrained {Tet <ids> | Tet Volume
<ids>} [preview]
```

The tets are deleted and the two back sides become surface triangles. If the tets have mid-edge nodes, the back mid-edge nodes are snapped to the surface. If this removal operation generates worse quality (aspect ratio) than in the tet removed, the operation is not performed. Figure 3 shows before and after image of removing overconstrained tets. If volumes are specified, the overconstrained tets are found automatically. The **preview** option draws the tets that will be removed.

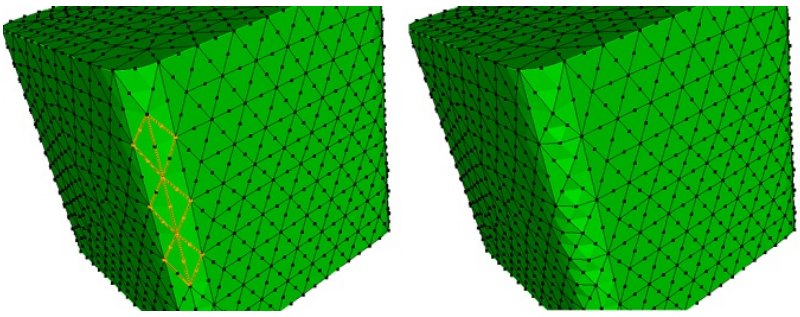


Figure 3. Before and After Removing Overconstrained Tets

Creating and Merging Mesh Elements

The following forms of the create and merge commands operate on meshed entities only. They allow low-level editing of meshes to make minor corrections to a mostly correct mesh. They are not designed for major modifications to existing meshes. Because Cubit's display routines were not designed with these type of operations in mind, these commands may cause the current display of the affected entities to take an unexpected form. An appropriate drawing command can be used to return the display to the desired view.

The delete commands for deleting individual elements are still under development, but they may be used after setting a developer flag.

Creating Mesh Elements

The create command uses existing mesh nodes to create new mesh entities.

Creating Hex and Tet Elements

```
Create {Hex|Tet} Node <range> [Owner Volume
<id>]
```

Using the nodes specified, this form of the command creates a new hex or tet that will be owned by the specified volume. For a hex, 8 nodes are required. The order in which the nodes are specified is very important. They should describe two opposing faces of the hex; the normal of the first face should point into the hex and the normal of the second face should point out of the hex. For example, to create the hex shown in Figure 1 below, the following command would be entered:

```
create hex node 1,2,3,4,5,6,7,8 owner volume 1
```

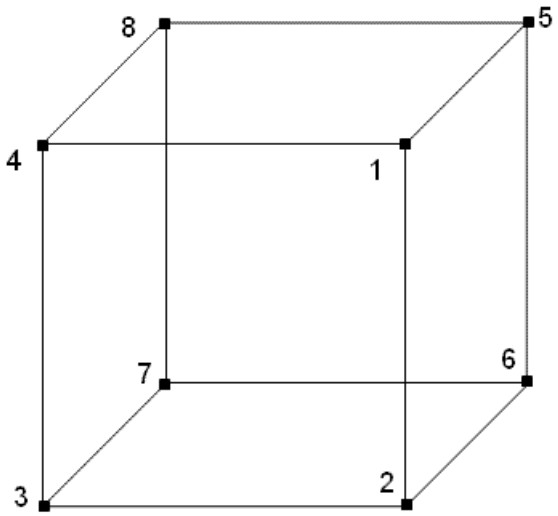


Figure 1. Node Numbering for the Create Hex command

To create a tet, 4 nodes are specified. The base is specified as a tri with the normal point toward the fourth node using the right hand rule. To create the tet shown in Figure 2, the following command would be entered:

```
create tet node 1,2,3,4 owner volume 1
```

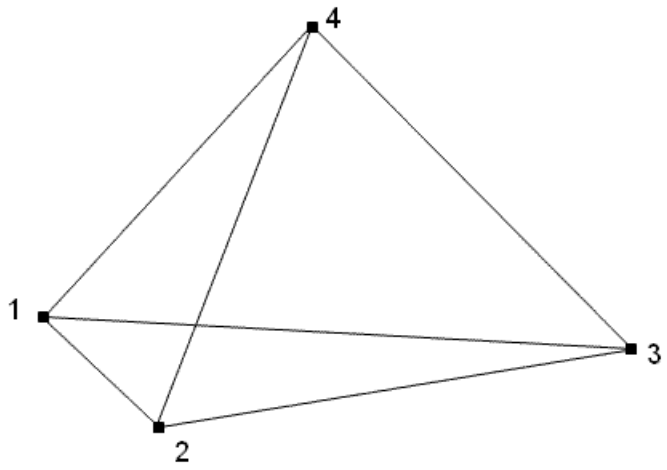


Figure 2. Node ordering for Create Tet Command

Creating Wedge Elements

```
Create Wedge Node <range> [Owner Volume <id>]
```

To create a wedge, 6 nodes are specified. The base is specified as a tri with the normal pointing inward using the right hand rule. To create the wedge shown in Figure 3, the following command would be entered:

```
create wedge node 1,2,3,4,5,6 owner volume 1
```

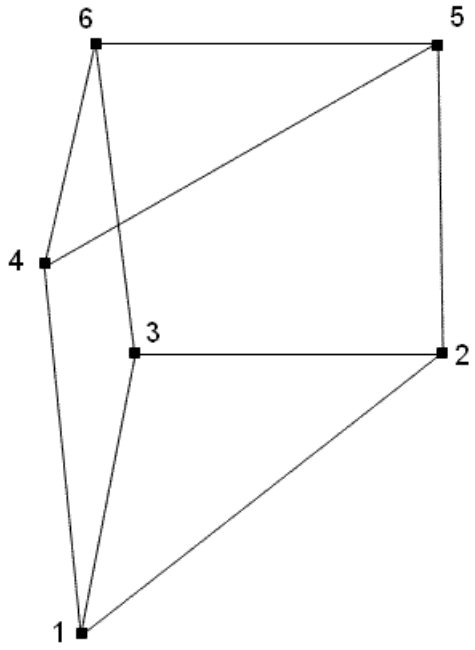


Figure 3. Node ordering for Create Wedge Command

Creating Face and Tri Elements

```
Create {Face|Tri} Node <range> [Owner
{Volume|Surface} <id>]
```

The next form of the command creates a face or tri that will be owned by the specified volume or surface. Four nodes are specified for a face, three nodes for a tri. The nodes should be specified in the order needed

to produce a face or tri with the normal in the desired direction using the right hand rule.

Creating Edge Elements

```
Create Edge Node <range> [Owner
{Volume|Surface|Curve} <id>]
```

This form of the command creates an edge that will be owned by the specified volume, surface, or curve. Two nodes must be specified; order is unimportant.

Creating Nodes

```
Create Node Location <x> <y> <z> Owner
{Volume|Surface|Curve|Vertex} <id>
```

The last form of the command creates a node at the specified location that will be owned by the specified volume, surface, curve, or vertex. The location is specified by three absolute values that represent the position of the node in 3D space.

Merging Nodes

The merge node command is used to join two mesh entities one node at a time. It should be used with care because merging nodes of different meshed entities may have unpredictable results. The syntax is:

```
Merge Node <id1> <id2>
```

The merge node command replaces the node specified as id1 with the node id2. The command is equivalent to deleting node id1 and creating node id2 in the same location. The resultant merged node takes on the characteristics of the replaced node such as position and owner. This may include some or all of the higher level mesh entities related to the merged node.

Caution should be taken when using the merge node command because other commands involving the related meshed entities may not work properly following the merge.

Mesh Cleanup

- Tetrahedral Mesh Cleanup
- Hexahedral Mesh Cleanup

Once a mesh has been created or imported, Cubit has tools to both manually and automatically improve the quality of a mesh. Mesh Cleanup is the name for the automatic tools which automatically find bad elements and fixing them by both recomputing node locations (i.e. smoothing) AND redefining the local element connectivity. To automatically cleanup a mesh, use the following command:

```
Cleanup {Volume|Block} <id_range> [angle
<value=150>]
```

This command will cleanup either a tet or a hex mesh as described below.

Cleaning Up a Tetrahedral Mesh

An alternative to the remesh command for tetrahedral meshes is the cleanup command. For this command the existing mesh is validated and "optimized" by the tetmesher, instead of being deleted and replaced with a different mesh.

To cleanup a tetrahedral volume mesh use the following command:

```
Cleanup {Volume|Block} <id_range>
```

A second variation of the Cleanup command allows remeshing of tetrahedra that are either part of a free mesh (not owned by a volume) or are a subset of the tetrahedra in the volume. The command is:

```
Cleanup Tet <id_range> [Free]
```

For example, the command

```
cleanup tet all free
```

will gather all tetrahedra in a free mesh or single volume, generate a triangle boundary surface, and "optimize" the mesh, ignoring any volume or blocks. Without the optional **free** keyword, the tets will be processed volume by volume or block by block retaining the boundary between adjacent volumes or blocks.

Also, the command

```
cleanup tet 200 to 300
```

will gather the tetrahedra in the range [200, 300], generate a triangle boundary surface, and "optimize" the mesh. If the tetrahedra in the range are disjointed, i.e., multiple, independent sets, this operation may fail. It is best to specify a contiguous set of elements.

Note: Cubit will issue an error if the tetrahedra are owned by more than one volume or mesh container.

Cleaning Up a Hexahedral Mesh

The command to cleanup a hex mesh is:

```
Cleanup Volume <id_range> [angle <value=150>]
```

Hexahedral mesh cleanup is newer to Cubit and currently only a single type of bad element is found and fixed. The hex mesh quality

configuration that is currently implemented is when a column of hex elements is on the boundary of a volume, the hexes in the column each have 2 adjacent quad faces on the boundary, and the dihedral angle between those 2 faces is greater than the specified **angle** tolerance. This situation is illustrated in Figure 1 where the red column of hexahedra has a good angle on the source surface, which flattens out to 180 degree angle on the target creating inverted hex elements. The **angle** parameter determines how large the angle can get before being cleaned up.

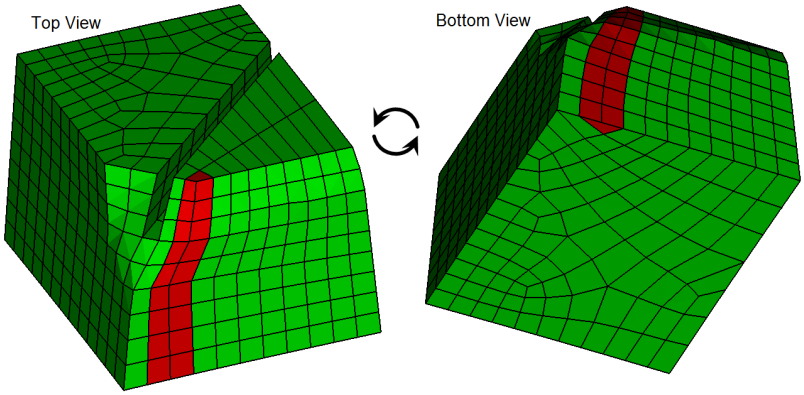


Figure 1. Example of hexahedral mesh with case handled by hex mesh cleanup.

Figure 2 illustrates the result of hex mesh cleanup. Internally, Cubit finds the column of hexahedra with the bad elements, as well as an adjacent column of hexahedra, and then automatically performs some hex column operations followed by smoothing to improve the quality of the elements locally.

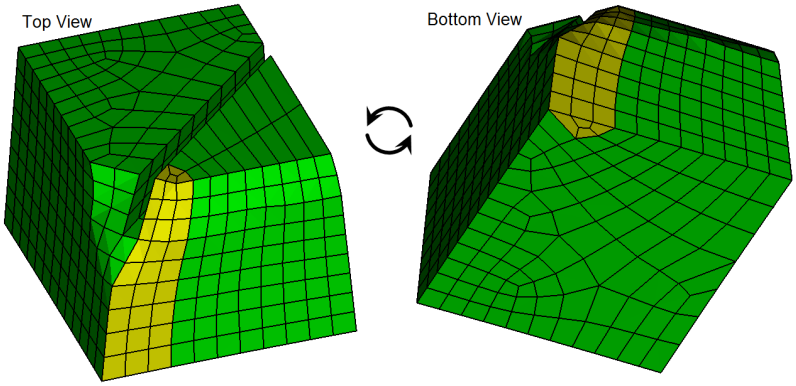


Figure 2. The mesh from Figure 1 after hex mesh cleanup.

Remeshing

Mesh generation is frequently an iterative process of meshing, deleting the mesh, and remeshing. The remesh command is a convenient tool to bypass the mesh deletion process when used to remesh a volume. You may also use the remesh command to replace a localized set of deformed triangles or tetrahedra after analysis. Thus, remeshing can become part of an optimization loop.

Use the following command to remesh hexahedra:

```
Remesh Volume <range>
```

Use the following command to remesh tetrahedra:

```
Remesh {Volume|Block|Tet} <range> [FIXED|free]
```

or to remesh a range of tets or tris based upon quality criteria:

```
Remesh Tet <id_range> | [quality <tet_metric>
[less than|greater than] <value> ...] [inflate
<value>][size <value>][FIXED|free][preview]
```

```
Remesh Tri <id_range> | [quality <tri_metric>
[less than|greater than] <value> ...] [inflate
<value>][preview]
```

Remeshing a Swept Volume Mesh

The remesh command can be useful when using the sweep scheme. When a sweep scheme is applied to the volume, it will delete the target surface mesh on a volume with one of the sweeping schemes and then remesh the volume. It is useful when changing between sweep smooth options as in the following example below.

```
volume 1 scheme sweep
mesh volume 1
```

At this stage, the user may discover that poor quality elements may have been generated. The user could then do the following:

```
volume 1 sweep smooth winslow
remesh volume 1
```

At this point, the volume is remeshed using the sweep smooth winslow option.

Remeshing Tetrahedra

When used for tetrahedra, the **Remesh** command generates a new tetrahedral mesh after deleting the existing mesh described by the list of tetrahedra, volumes, or blocks. When remeshing a list of tetrahedra, the smallest set of tets possible is replaced, which often means a partial remeshing of volumes, surfaces and/or curves. This set will always include the input list of tetrahedra but may include more.

Each tetrahedron may only be in one volume or block, but the list of tetrahedra may span volumes or blocks. Each block is treated individually if multiple blocks are specified.

The default **FIXED** option will ensure that any triangle or edge in the tetrahedron list to be remeshed that lie on geometric surfaces or curves will not be affected by the remesh operation. In contrast, the **free** option allows edges and triangles on curves and surfaces to be removed and remeshed. Use the **FIXED** option when it is important to maintain the boundary mesh configuration fixed, otherwise the free option will remesh the portions of curves and surfaces in the remesh region.

The **Remesh** command can be used to selectively remove and remesh a small portion of tetrahedron in the mesh that have been identified as poor quality. This can be an effective tool for improving mesh quality on a deformed mesh following an analysis without the need to regenerate the full mesh.

The **quality** option will identify those tetrahedra from the full model and apply the remeshing opertaion only to those tetrahedra. Any of the standard quality metrics for tetrahedra may be used as the **<tet_metric>**. These include: **Aspect Ratio Bet, Aspect Ratio Gam, Element Volume, Condition No., Jacobian, Scaled Jacobian, Shape, Relative Size, Shape And Size, Distortion, Allmetrics, Algebraic** and **Traditional.** The metric specification is used in conjunction with a **less than** or **greater than** specification and a threshold value. For example, the syntax below would remesh all tetrahedra in the mesh who's scaled jacobian metric was less than 0.2.

> **remesh tet quality Scaled Jacobian less than 0.2 inflate 1 free**

The **inflate** option can be used to expand the set of tets selected by the **quality** metric criteria. The **<value>** input following the **inflate** option is the number of tet layers surrounding the poor quality tets that will be included in the remesh region. Usually a value of 1 is sufficient to allow the tet mesher to generate better quality elements, however 2 or greater will remesh a larger portion of the mesh. A value of 0 is generally not recommended as it usually does not provide enough space for the tet mesher to improve element quality. The **inflate** option can also be used independently from the remesh command. See the **Inflate** command described below.

This command also allows for multiple quality criteria. For example, the following command would use both aspect ratio and scaled jacobian as criteria for remeshing. Any number of quality criteria may be included in the command syntax:

> **remesh tet quality Scaled Jacobian less than 0.2 quality Aspect Ratio Bet greater than 4 inflate 1 free**

The **preview** option will display the tetrahedra selected by the **quality** criteria and **inflate** options without actually performing the remeshing operation.

Sizing functions may be used with tet remeshing. See Mesh Adaptivity and Sizing Functions and Exodus II-based Field Function for more information.

The **size** parameter may be used to control the size of the tets created in the remesh.

Inflating a set of Tets

In cases where a set of tets are to be remeshed, it is useful to be able to expand the set to include additional surrounding tets. This is to allow the mesher more freedom to place good quality elements, but also to ensure a valid shape in which the mesher has to work. The Inflate command starts with a given set of tetrahedra and will expand the set based on the number of user defined layers as well as manifold criteria. The result will be added to the curent group, or a new group can be created. The following describes the syntax and arguments to this command:

```
Inflate {group <id>|tet <ids>} {manifold|layer
<value>}[{add|create <"name">}] [draw]
```

group<id>|tet<range>: input to this command can be with a group name, group id, or a range of tets. The group must contain at least 1 tet. The tets need not be contiguous.

manifold: This option will add tets to the set where the boundary or skin of the tets meet at a single edge or node. This ensures that a complete valid manifold definition of the boundary of the set of tetrahedra can be defined. This is important for the tetrahedral mesh generator which requires a manifold boundary definition. both **layer** and **manifold** can be used in the same command.

layer <value>: This option will add the number of layers of tets indicated by **value** to the set. A layer is defined by all tets connected by at least a node to the skin of the existing set. This option alone does not guarantee a valid manifold definition. Use both the **layer** and **manifold** in the same command options to ensure a manifold definition.

add|create<"name">: The **add** option will add tets in the inflated region to the input group. An input group must be specified for this to be a valid option. The **create** option will create a new group and add all tets (including the input), to a new group specified by **<"name">**. If neither **add** nor **create** are specified, a new default group named "**inflated_tets**" will be created. If a group of that name already exists, it will be added to.

draw: The **draw** option will display both the input set of tets and the inflated tets in the graphics window. The input tets will be displayed in green and the inflated tets will be displayed in red.

Examples:

Generate a simple tet mesh. For tets with ids 1 to 10, define a 1 layer buffer and ensure it maintains a manifold boundary. The result will be placed in a new group called "**inflated_tets**" and displayed in the graphics window.

```
brick x 10
vol 1 scheme tetmesh
mesh vol 1
inflate tet 1 to 10 layer 1 draw
```

Create a group called "bad_tets"containing all tets in volume 1 with quality metric (scaled Jacobian) less than 0.2. Expand that group by one layer and remesh it.

```
group 'bad_tets' equals qual vol 1 scaled high 0.2
inflate bad_tets layer 1 add
remesh tet in bad_tets
```

Remeshing Triangles

Remeshing triangles works in many of the same ways as remeshing tets, generating a new triangle mesh after deleting the existing mesh described by the list of triangles. When remeshing a list of triangles, the smallest set possible is replaced, which often means a partial remeshing of surfaces only. This set will always include the input list of triangles but may include more to ensure the set is non-manifold. Some important differences between remeshing triangles vs tets are:

- Triangle remeshing does not support the **free** option. So if the triangles include mesh edges that lie on geometric curves, these edges will not be affected by the remesh operation.
- Triangle remeshing uses the **trimesh** scheme to do the remeshing, whereas tet remeshing with the **free** option uses scheme **triadvance**.
- A **size** parameter allows for control of the size of the triangles generated in the remesh.

Edge Swapping

The edge swap command allows a user to target a specific edge between two triangles (similar functionality for quads and tets has not been included) and change the connectivity of the triangles. Multiple edges can be swapped simultaneously. The input order of the edges is the order in which the swaps will be performed.

Typically, the edge swap command is used to specifically repair local mesh connectivity.

```
Swap Edge <id_list>
```

The following images show the before and after views of a model where the highlighted edge is swapped. The edge in each image is the same edge.

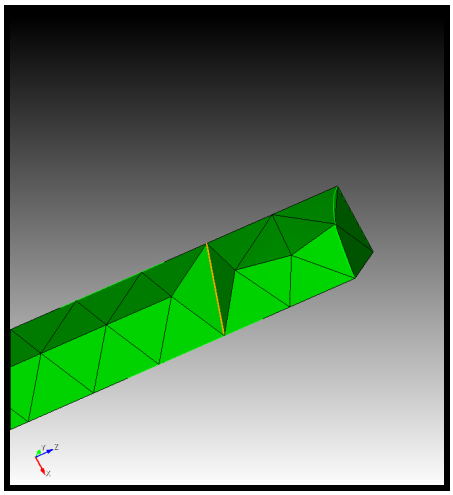


Image 1 - Before edge swapping

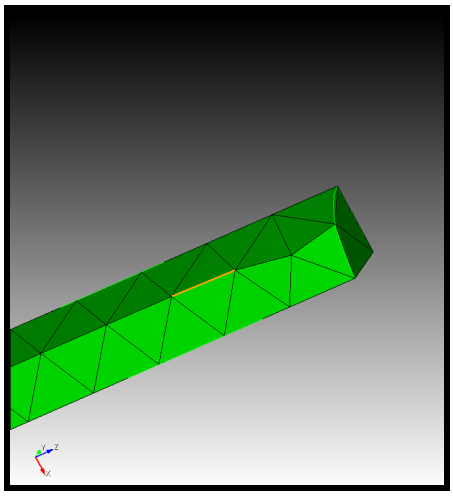


Image 2 - After edge swapping

Matching Tetrahedral Meshes

The intended use of this function is for importing two exodus or genesis files that have non-conforming mesh where they touch and modifying the meshes locally to make them conforming. The result is a single mesh that is stitched together at the locally modified region. This functionality is currently only available for tetrahedral meshes.

Tetrahedral mesh matching will work on free mesh only. The interface where the two meshes will be matched need not be planar. A single target sideset and one or more source sidesets should be provided. The source sideset should be completely enclosed in the target sideset so that the boundaries of the two sidesets do not intersect. The two meshes need not touch exactly at the sidesets but the closer the meshes are to touching the better the results will be. Small gaps or overlaps will generally be allowed. Both of the meshes involved in the matching should be contained in defined blocks prior to issuing the command.

The syntax for the command is:

```
Meshmatch tet sideset <id_list> onto sideset
<id>
```

The one or more sidesets specified *before* the '**onto**' keyword are the source sideset(s). The sideset *after* the '**onto**' keyword is the target sideset.

Mesh Coarsening

Hexahedral Coarsening

CUBIT provides a limited number of options for coarsening hexahedral meshes. The options currently available for hex coarsening rely on the hex sheet extraction process described in Mesh Refinement page. Removing a sheet from a hexahedral mesh essentially means that a complete layer of hexes will be removed and the adjacent layers expanded to take its place.

Extracting a Single Hex Sheet

The following command can be used to extract a single hex sheet.

```
Extract sheet { Edge <id> | Node <id_1> <id_2>
}
```

The edge or node pair are used to define the sheet that will be extracted. Figure 3 below shows an example of extracting a hex sheet. In this example the hex sheet is specified by the node pair highlighted in the images. Note that the entire layer of hexes between the highlighted nodes has been removed and the neighboring layers have been expanded to take its place.

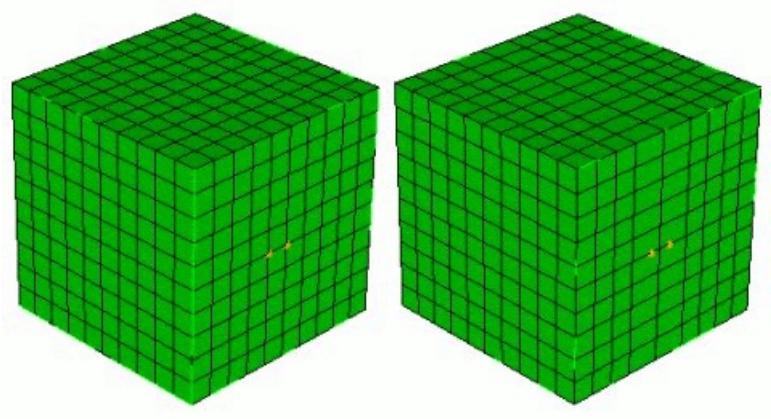


Figure 3. Example of Hex Sheet Extraction

Note: Also see the Mesh Refinement section for a description of hex sheet drawing.

Extracting multiple sheets along a curve

Another option for extracting hex sheets can be done by specifying a curve at which to perform the sheet extraction operations. In this case, multiple layers of hexes can be removed by specifying a curve perpendicular to the hex layers. The command for coarsening perpendicular to a curve is as follows:

```
Coarsen Mesh Curve <id> Factor <value>
[NO_SMOOTH|smooth]

Coarsen Mesh Curve <id> Remove
{<num_edges>|edge <id_ranges>}
[NO_SMOOTH|smooth]
```

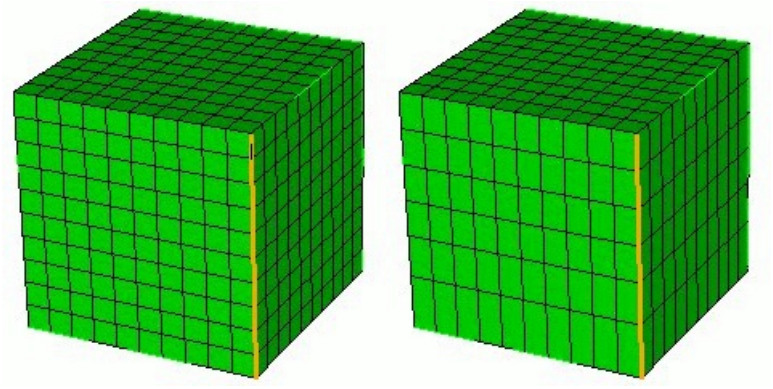


Figure 4. Coarsening a mesh by extracting sheets perpendicular to a curve

The first option uses the **Factor** argument. The factor argument controls how much larger the edges will be on the curve. For example, Figure 4 shows the coarsen mesh curve command used with a factor of 2. In this case, the command attempts to make the mesh edges approximately twice the length relative to their original length along the curve.

The second option uses the **Remove** argument. With this option, a specified number of layers may be removed from the mesh. This may be accomplished by indicating an exact number, or by providing a list of edge IDs that correspond to the layers that will be removed.

The **NO_SMOOTH|smooth** option allows the user to improve the element quality after the sheet extraction process by smoothing the remaining nodes. The default for both of these commands is to not smooth. Smoothing may also be accomplished after sheet extraction by using the smooth volume command.

Uniform hex coarsening

By applying the coarsen mesh curve command multiple times to curves that are orthogonal in the model, the effect of uniform coarsening of the mesh may be achieved.

Mesh Refinement

- Global Mesh Refinement
- Refining at a Geometric or Mesh Feature
- Hexahedral Refinement Using Sheet Insertion
- Local Refinement of Tets, Triangles, and Edges
- Parallel Refinement
- Uniform Mesh Refinement

CUBIT provides several methods for *conformally* refining an existing mesh. Conformal mesh refinement does not leave hanging nodes in the mesh after refinement operations, rather conformal mesh refinement provides transition elements to the existing mesh. Both local and global mesh refinement operations are provided.

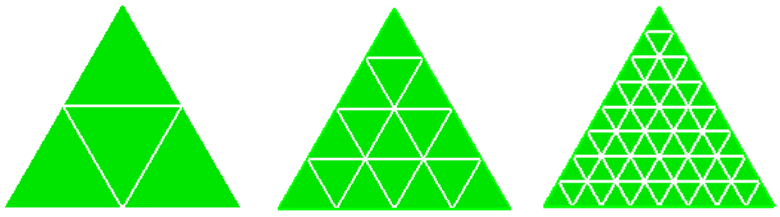
Global Mesh Refinement

The Refine Surface and Refine Volume commands provide capability for globally refining an entire surface or volume mesh. Global refinement will only be used if the entire body is included in the command. Otherwise, the command will be interpreted as local refinement (see below.) This distinction can be important because the global refinement algorithm divides each element into fewer sub-elements than local refinement. The command syntax is:

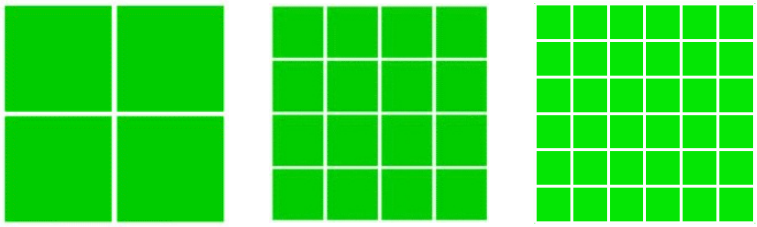
```
Refine Volume <range>numsplit<int>

Refine Surface <range>numsplit<int>
```

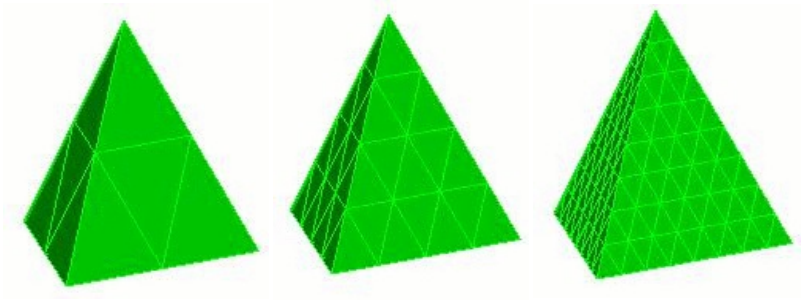
The numsplit option specifies how many times to subdivide an element. A value of 1 will split every triangle and quadrilateral into four pieces, and every tetrahedron and hexahedron into eight pieces. Examples of global refinement on each element are shown below.



original mesh NumSplit = 1 NumSplit = 2



original mesh NumSplit = 1 NumSplit = 2



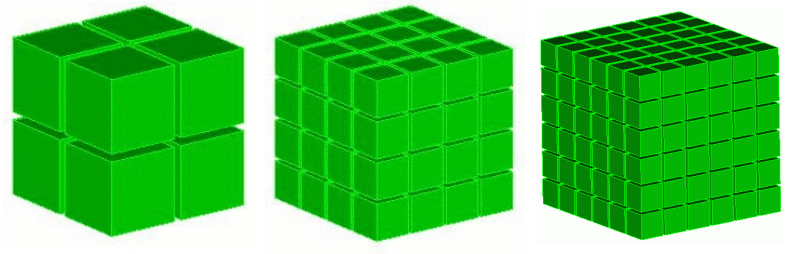


Figure 1. Example of uniform refinement for each of the mesh entities

Refining at a Geometric or Mesh Feature

CUBIT also provides methods for local refinement around geometric or mesh features. Individual elements or groups of elements can be refined in this manner using the following syntax.

```
Refine {Node|Edge|Tri|Face|Tet|Hex} <range>
[NumSplit<int = 1>|Size <double> [Bias
<double>]]
[Depth <int>|Radius <double>]
[Sizing_Function]
[Smooth]

Refine {Vertex|Curve|Surface} <range>
[NumSplit<int = 1>|Size <double> [Bias
<double>]]
[Depth <int>|Radius <double>]
[Sizing_Function]
[Smooth]
```

To use these commands, first select mesh or geometric entities at which you would like to perform refinement. Refinement will be applied to all mesh entities associated with or within proximity of the entities. The all keyword may be used to uniformly refine all elements in the model

The following is a description of refinement options.

NumSplit

Defines the number of times the refinement operation will be applied to the elements in the refinement region. For uniform or global refinement, where all elements in the model are to be refined, A NumSplit value of 1

will split each triangle and quadrilateral into four elements, and each tetrahedron and hexahedraon into eight elements. A numsplit of 2 would result in 9 and 27 elements respectively. For uniform refinement, the total number of elements obeys the following:

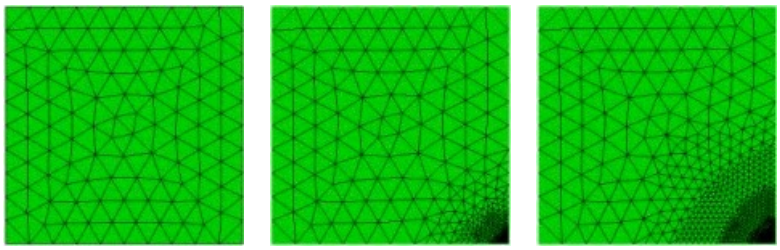
NE = NI * (NumSplit+1)^Dim

where **NE** is the final number of elements, **NI** is the initial number of elements and **Dim** is 2 or 3 for 2D and 3D elements respectively.

In cases where only a portion of the elements are selected for refinement, the elements at the boundary between the refined and non-refined elements will be split to accommodate a transition in element size. The transition pattern will vary depending on the local features and surrounding elements. For non-uniform refinement of hexahedron, for a numsplit of 1, each element in the uniform refinement zone will be subdivided into 27 elements rather than 8. This affords greater flexibility in transitioning between the refined and unrefined elements.

Size, Bias

The Size and Bias options are useful when a specific element size is desired at a known location. This might be used for locally refining around a vertex or curve. The Bias argument can be used with the Size option to define the rate at which the element sizes will change to meet the existing element sizes on the model. Figure 2 shows an example of using the Size and Bias options around a vertex. Valid input values for Bias are greater than 1.0 and represent the maximum change in element size from one element to the next. Since refinement is a discrete operation, the Size and Bias options can only approximate the desired input values. This may cause apparent discontinuities in the element sizes. Using the default smooth option can lessen this effect. It should also be noted that the Size option is exclusive of the NumSplit option. Either NumSplit or Size can be specified, but not both.

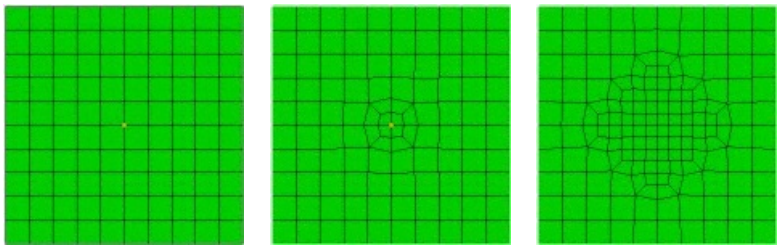


original mesh Bias=2.0 Bias=1.5

Figure 2. Example of using the Size and Bias options at a Vertex.

Depth

The Depth option permits the user to specify how many elements away from the specified entity will also be refined. Default Depth is 1. Figure 3 shows an example of using the depth option when refining at a node.



original mesh Depth=0 Depth=1

Figure 3. Example of using the Depth option at a node to control how far from the node to propagate the refinement.

Radius

Instead of specifying the number of elements to describe how far to propagate the refinement, a real Radius may be entered. The effects of the Radius are similar to that shown in Figure 3, except that the elements whose centroid fall within the specified Radius will be refined. Transition elements are inserted outside of this region to transition to the existing elements.

Sizing Function

Refinement may also be controlled by a sizing function. CUBIT uses sizing functions to control the local density of a mesh. Various options for setting up a sizing function are provided, including importing scalar field data from an exodus file. In order to use this option, a sizing function must first be specified on the surface or volume on which the refinement will be applied. See Adaptive Meshing for a description of how to define a sizing function.

Smooth

The default mode for refinement operations is to NOT perform smoothing after splitting the elements. In many cases, it may be necessary to perform smoothing on the model to improve quality. The smooth option provides this capability.

Controlling Regularity of Triangle Refinement

The default behavior of triangle refinement is to attempt to maximize element quality using the basic one->four template. This can sometimes result in an irregular pattern, where one or more edges are swapped. To enforce regularity of the triangle refinement pattern, regardless of quality, the folowing setting may be used.

```
Set Triangle Refine Regular {on|OFF}
```

Hexahedral Refinement Using Sheet Insertion

Several tools for refining a hexahedral mesh using sheet insertion and deletion are available in CUBIT.

- Refining at a Geometric Feature
- Refining along a Path
- Refining a Hex Sheet
- Directional Refinement
- Hex Sheet Drawing

Refining at a Geometric Feature

The following commands offer additional controls on refinement with respect to one or more geometric features of the model.

An existing hexahedral mesh can be refined at a geometric feature using the following command:

```
Refine Mesh Volume <id> Feature {Surface |
Curve | Vertex | Node} <id_range> Interval
<integer>
```

This command refines the mesh around a given feature by adding sheets of hexes. These sheets can be generalized as planes for surfaces, cylinders for curves, and spheres for vertices. The **interval** keyword specifies the number of intervals away from the feature to insert the new sheet of hexes. For this command a single sheet of hexes is inserted into the hexahedral mesh.

Figure 4 shows an example of this command where the feature at which refinement is to be performed is a curve. In this case the interval chosen was, 2. This indicated that the elements 2 intervals away from the curve would be refined.

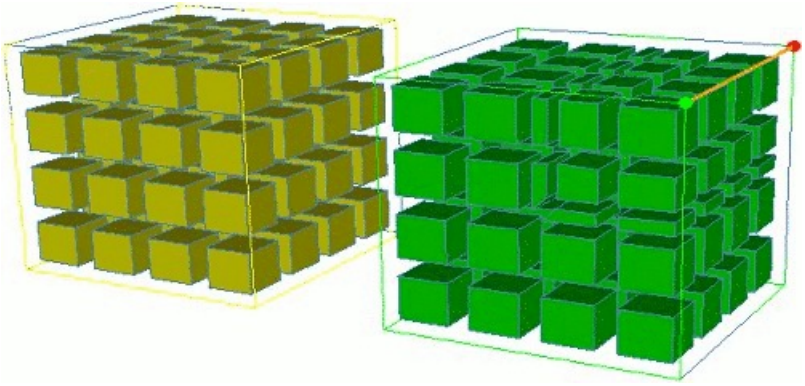


Figure 4. Example of Refinement at a curve

Refining along a path

Hexahedral meshes can be refined from a specific node and along a propagated path using the following command

```
Refine Mesh Start Node <id> Direction Edge
<id> End Node <id> [Smooth]
```

Figure 5 shows a swept mesh and its cross section. The cross section view on the left shows a path that has been propagated through the mesh between the start node and end node. This path is then projected along a chain of edges in the direction given by the direction edge as shown in Figure 5. The start node and end node must be on the same sweep layer. This refinement procedure also requires the volume's meshing scheme to be set to sweep. If the smooth keyword is given the mesh will be smoothed after the refinement step is complete.

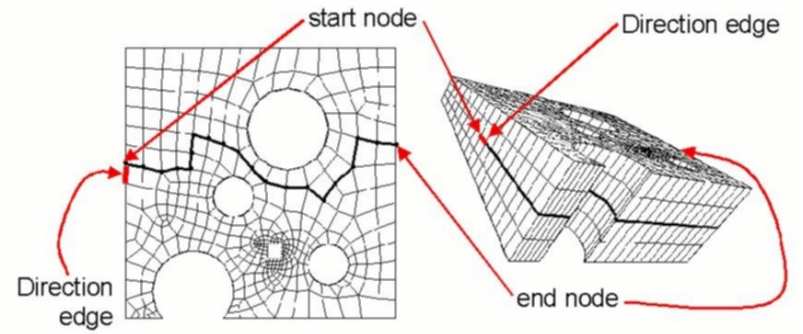


Figure 5. Refining a Mesh Along a Path

Refining a Hex Sheet

The following command can be used to refine the elements in one or more hex sheets:

```
Refine Mesh Sheet [Intersect] { Node <id_1>
<id_2> | Edge <id_range> } { Factor <double> |
Greater_than <size> } [Smooth] [in volume
<id_range> [depth <num_layers]]
```

The **node** and **edge** keywords are used to define the hex sheet(s) to be refined. If the node option is chosen, only one node pair can be entered (see Figure 6). If the edge option is chosen, one or more edges can be entered (see Figure 7).

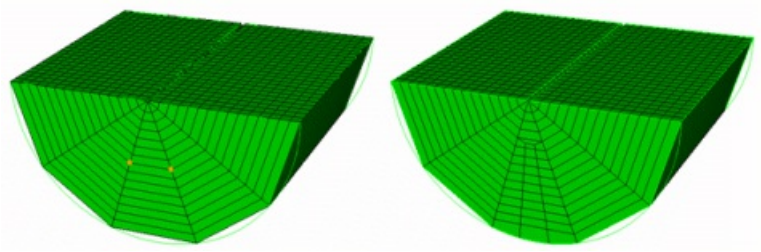


Figure 6. Refine mesh sheet node 796 782 greater_than 6

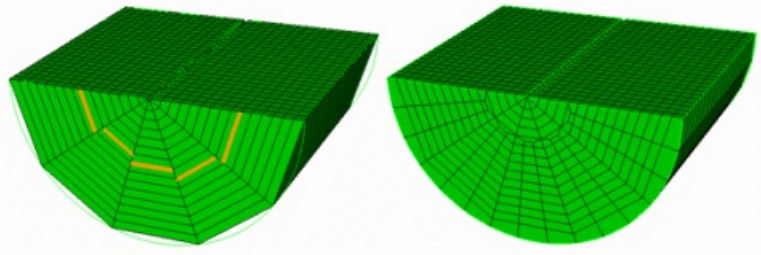


Figure 7. Refine mesh sheet edge 1584 1564 1533 1502 1471 greater_than 6

The **factor** and **greater_than** keywords are used to specify the refinement criteria for the selected hex sheet(s). If the factor keyword is used, the length of the smallest edge in the hex sheet is determined and any edge in the hex sheet with a length greater than the smallest length multiplied by the factor is refined. If the greater_than keyword is used, any edge in the hex sheet with a length greater than the specified size is refined.

The **intersect** keyword is optional. It is used to more easily define multiple hex sheets to be refined. If the intersect keyword is entered, the node and edge keywords are used to define a chord rather than a sheet (a chord is the two-dimensional equivalent of the three-dimensional sheet). The chord will be limited to the surface(s) associated with the nodes or edge entered, and all sheets intersecting the chord will be selected for refinement (see Figure 8). When the node keyword is used with the intersect option, the nodes must define an edge on the surface of the mesh.

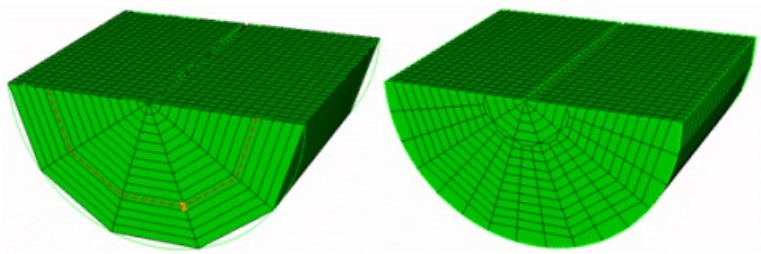


Figure 8. Refine mesh sheet intersect edge 1499 greater_than 6

The **smooth** keyword is also optional. When the smooth keyword is entered, the elements that have been refined are smoothed in an attempt to improve element quality. Figure 9 shows the same command as Figure 8 with the addition of the smooth keyword. Smoothing may or may not be beneficial, depending on the situation.

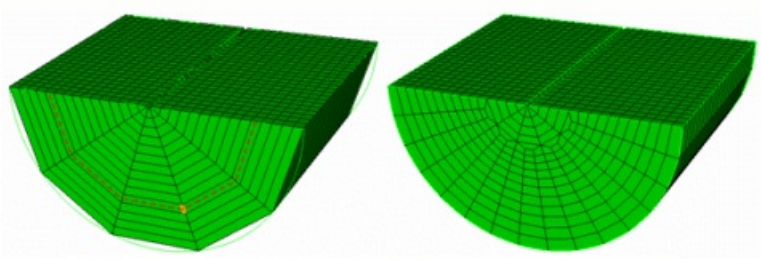


Figure 9. Refine mesh sheet intersect edge 1499 greater_than 6 smooth

Directional Refinement

Mesh sheet refinement can also be used to refine a mesh in a particular direction. This can help control anisotropy. The following command can be used as a short cut for specifying what sheets should be used in refinement.

```
Refine Volumes <id_range> using {Plane
<options> | Surface <id_range> | Curve
<id_range> } [ Depth <num_layers> ] [ Smooth ]
```

The volumes specified indicate which hexes can be refined. A transition layer will be made out of hexes surrounding the indicated volumes. If the **depth** option is used, additional layers of hexes around the specified volumes will be included in the refinement region. Behind the **using** option, if the **plane** option is employed, all the edges in the volume which are parallel to the plane (to a small tolerance) are used to specify the sheets to refine. If the **surface** or **curve** option is employed instead, all the edges in the surfaces or curves will be used.

For example, Figure 10 and 11 shows directional refinement using the plane option. The command used to convert the mesh in Figure 10 to Figure 11 is:

```
refine vol 2 using plane xplane depth 1
```

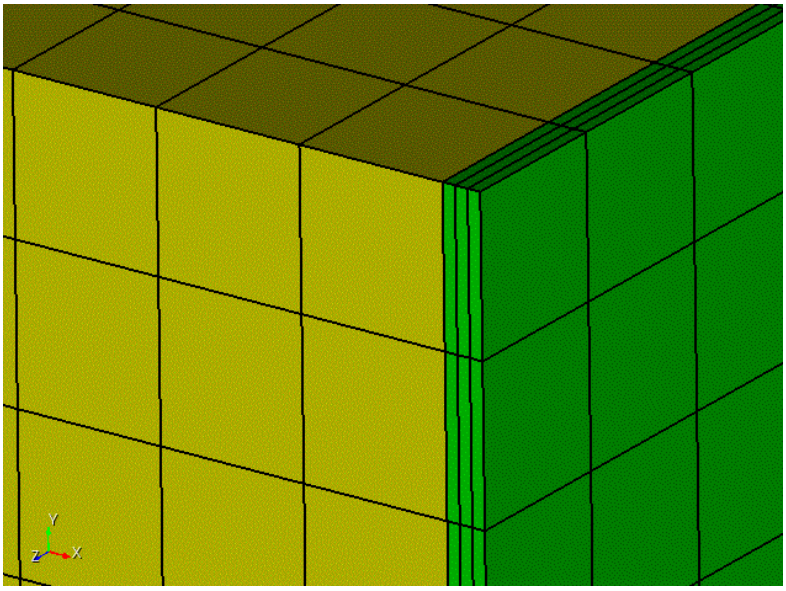


Figure 10. Starting mesh

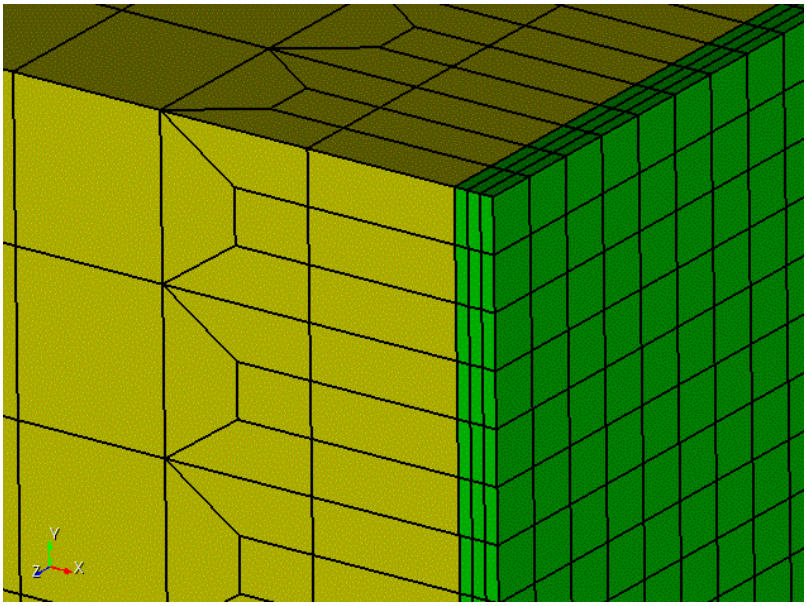


Figure 11. Directional result of refinement resulting from using the plane option on the refinement command.

Directional refinement can be used iteratively to reduce or create anisotropy of any level. This is done by applying the direction refinement command iteratively. A second iteration of directional refinement can be applied by issuing the same command again. To improve element quality, however, it is often recommended to perform refinement parallel to the plane before subsequent iterations. For example, taking the mesh in Figure 11 as input, the following commands will generate the mesh in Figure 12.

```
refine mesh sheet edge ( at 4.5 5 5 ordinal 1
) factor 0
```

```
refine vol 2 using plane xplane depth 1
```

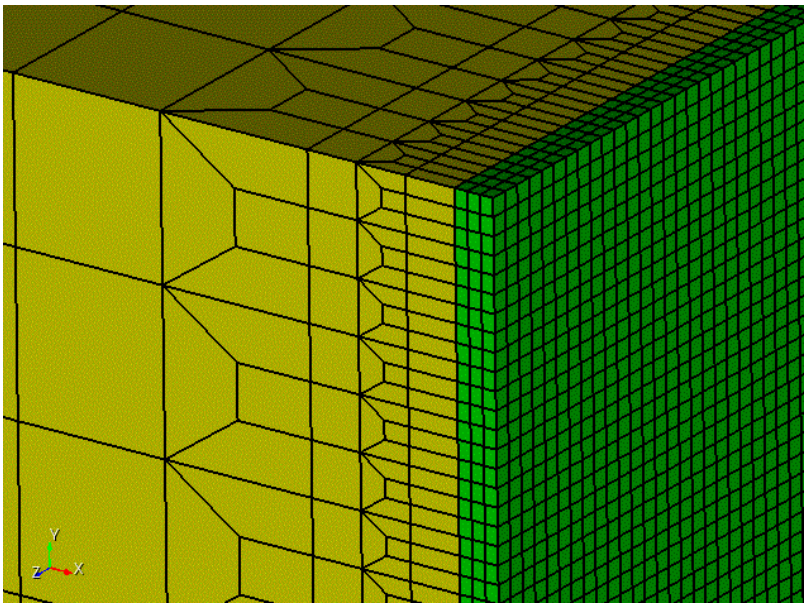


Figure 12. A 2nd iteration of direction refinement is applied.

Hex Sheet Drawing

Since refinement of hex meshes generally occurs by inserting hex sheets, tools have been provided to draw a specified sheet or group of sheets.

This command draws a sheet of hexes that is defined by the edge or node pair.

```
Draw Sheet {Edge <id> |Node <id_1> <id_2>}
[Mesh [List]] [Color <color_name>] [Gradient]
```

The following command draws the three sheets that intersect to define the given hex. These sheets are drawn green, yellow, and red. To draw a specific sheet, list its color in the command.

```
Draw Sheet Hex <id> [Green][Yellow][Red][Mesh
[List]] [Gradient]
```

The 'gradient' keyword for both commands draws the sheet in gradient shading according to the distance between opposite hex faces that are parallel to the sheet.

The 'mesh' keyword will draw the hexes in the hex sheet. If the 'list' keyword is also given, the ids of the hexes in the sheet will be listed.

Local Refinement of Tets, Triangles, and Edges

Local refinement of tets, triangles, and edges is available by refining individual entities or by refining to guarantee a user-specified number of tests through the thickness:

- Single Entity Refinement
- 'N' Tets Through the Thickness Refinement

Single Entity Refinement

Local refinement of tets, triangles, and edges is available. When refining triangles a node is inserted at the enter of the triangle and three new triangles are connected to this node. The original triangle is deleted. The command to refine triangles is:

```
Refine Local Tri <tri_id_list>
```

When refining an edge, a node splits the original edge between two triangles and four new triangles are created and connected to the new node. The command to refine an edge is:

```
Refine Local Edge <edge_id>
```

When refining a tet edge, the tet edge is split by a node and then all tets attached to the original edge are split into two through a triangle that goes through the new node. All other adjacent nodes and edges are unmodified by the operation. Note that on the interior of the mesh tet edges are not represented explicitly so the command takes two nodes as input to define the edge. The command to refine a tet edge is:

```
Refine Tet_edge Node <node1_id> <node2_id>
```

'N' Tets Through the Thickness Refinement

Cubit provides a capability to guarantee a user-specified number of tets through the thickness. This functionality is intended to work on an existing tet mesh using mesh refinement. The user specifies the geometry or mesh defining the thin region and also the number of desired tets through the thickness and the refinement algorithm will run until it meets this criteria. The number of tets through the thickness in this context is interpreted as the number of mesh edges through the thickness and the algorithm will continue to do refinement until there are no mesh edge paths through the thin region that contain fewer mesh edges than the number specified by the user. The command for doing

this is:

```
Refine min_through_thickness <val> source
{surface|node|tri|nodeset|sideset|block}
<id_range> target
{surface|node|tri|nodeset|sideset|block}
<id_range> [anisotropic] [single_iteration]
[dont_fill_in_gaps]
```

The various options are described below.

anisotropic: When this option is specified in the command the algorithm will only attempt to refine the edges that go roughly normal to the source and target entities. This will give an anisotropic result. The meshes on the source and target will generally not be affected when this option is used. When this option is not specified the refinement algorithm will be isotropic in nature and will propagate much more. However, it will tend to have better transitioning from the refined region to the non-refined regions.

dont_fill_in_gaps: When this option is specified in the command the algorithm will NOT try to grow the regions that will be refined. When the regions are grown it helps to avoid leaving small pockets of mesh that are not refined (splotchiness). This has effect only on isotropic refinement (when the "anisotropic" option is NOT used).

single_iteration: When this option is specified in the command the algorithm will only run for one iteration even if the min_through_thickness criteria is not met.

A quality command for querying the minimum number of tets through the thickness is found here.

Below is an example using the following commands:

refine min_through_thickness 4 source surf 1 target surf 2 anisotropic

refine min_through_thickness 4 source surf 7 target surf 13 3 14 anisotropic

Surfaces 1 and 2 are the two surfaces on opposite sides of the thin region in the green volume and surfaces 7 13 3 14 are the surfaces on opposite sides of the thin region in the yellow volume.

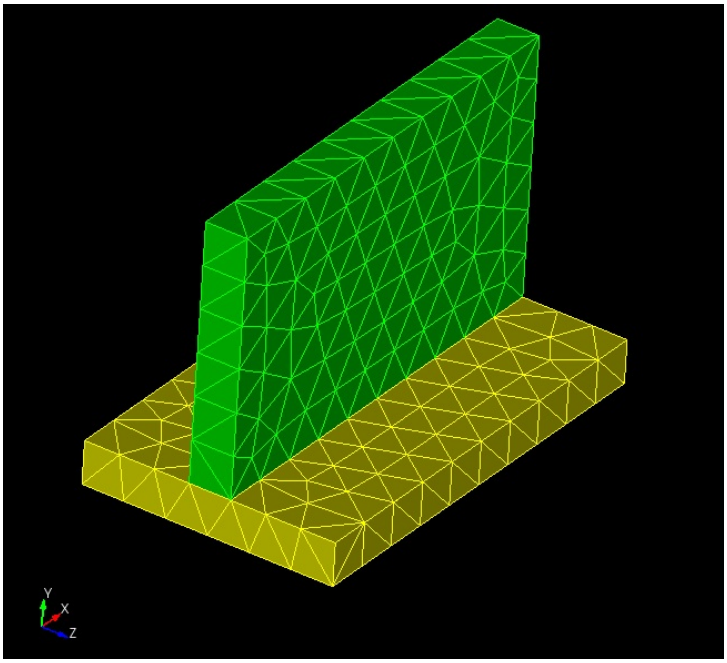


Figure 13. Before N through the thickness

refinement.

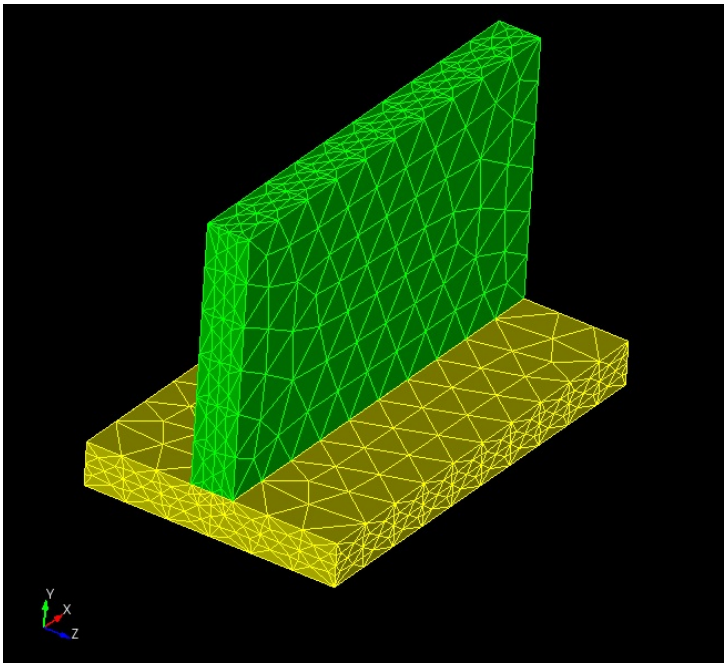


Figure 14. After N through the thickness refinement.

Parallel Refinement

Global mesh refinement can be used to increase global mesh density with a single command. If an extremely large mesh is desired, one approach is to generate a coarse mesh with the desired relative mesh gradations, and then perform global mesh refinement to scale the number of elements up across the model. Depending on the amount of refinement requested, this can exceed the memory limits of Cubit running on a single processor. Global parallel mesh refinement allows refinement to go beyond the memory limits of a single processor. The resulting mesh size is only limited by the number of processors you have available to perform the refinement. The command syntax is:

```
Refine Parallel [Fileroot <'root filename'>]
[Overwrite] [No_geom] [No_execute] [Processors
<int>] [Numsplit <int>] [Version <'Sierra
version'>]
```

This command causes Cubit to write two files to disk. First, Cubit writes an Exodus file named <root filename>.in.e which contains the mesh elements in the current Cubit session. Second, Cubit writes an OpenNURBS 3dm file http://www.opennurbs.org named .3dm, which contains a definition of the geometry from the current Cubit Session. The **Fileroot** argument specifies the full path and root of the files that will be written. Additional blocks are written to the Exodus file to correspond to the geometry entities in the 3dm file. The **Overwrite** argument specifies if existing files on disk with the same names should be overwritten or not.

When the mesh is refined in STK_Adapt, the new nodes created during refinement will be projected to the geometry definitions from the OpenNURBS file. If the **No_geom** argument is specified, only the Exodus file is written, and new nodes will be placed by evaluating the shape functions of the elements being evaluated.

The exported Exodus and OpenNURBS files are prepared specifically for input into the Sierra STK_Adapt program. By default, Cubit spawns STK_Adapt in the background after exporting the files. If the **No_execute** argument is specified, the Cubit command exports the files, but does not spawn STK_Adapt. The user can then move those files to a large parallel machine to perform the STK_Adapt refinement.

If **No_execute** is not specified, then Cubit will spawn Sierra STK_Adapt in the background to perform the refinement. The **Processors** argument specifies the number of processors to use for the STK_Adapt run. The **Numsplit** argument specifies how many times the global refinement should be performed. If **Numsplit** = 1, then each element edge is split into 2 sub-edges. If **Numsplit** = 2, then each element is split into 4 sub-edges, etc. The optional **Version** argument allows the user to specify which version of STK_Adapt should be run. Possible values for **Version** include "head", "4.22.0", etc.

Refine parallel command creates groups to visualize the association between mesh entities (edge, tri, and quads) and geometric entities (curves & surfaces). There are three types of groups that exist for each mesh entity type edge, tri, and quad. First group contains unique 1-1 map between mesh entity and geometric entity. Note that issuing "Debug 212" command before calling the refine parallel command, will create separate group for each geometric entity containing unique mesh entities. Next group contains mesh entities that point to multiple geometric entities. And the final group contains mesh entities not associated with any geometric entity.

After the Refine Parallel command finishes, the mesh in Cubit does not change, normally because the resulting mesh would be too big to store in Cubit on a single processor. Instead, the refined mesh is written to disk in a series of Exodus files, one per processor, using the **Fileroot** argument as the root of the Exodus file names. For example, if **Fileroot** is *"somemesh"* and **Processors** is 8, STK_Adapt will write out eight Exodus files named *somemesh.e.8.0*, *somemesh.e.8.1*, â€¦, *somemesh.e.8.7*. These files can be kept distributed for an analysis run, or united using the Sierra EPU command. In this example, Cubit would have written out a file called *somemesh.in.e*, which contains all of the sideset, nodeset, and block definitions defined in the Cubit session. All of these sidesets, nodesets and blocks are transferred to the refined exodus files (*somemesh.e.8.0*, etc.) for use in the subsequent analysis. The *somemesh.e.** files will also contain several other blocks which correspond to geometric entities defined in *somemesh.3dm* to enable the mesh to be refined to the CAD geometry, and should be ignored by downstream applications.

Sierra STK_Adapt must be in the PATH on the computer Cubit is running on. If Sierra STK_Adapt cannot be found, Cubit returns an error and no refinement is performed. Information on how to download and build Sierra STK_Adapt can be found at http://trilinos.sandia.gov/packages/stk/ .

Uniform Mesh Refinement

The Uniform Mesh Refinement (UMR) tool refines a mesh stored in the Exodus format uniformly, splitting every element in the mesh into a number of sub-elements, and writes the fine mesh to a new Exodus file. The resulting elements have roughly half the edge length of the original mesh. The algorithm uses an efficient streaming pipeline with low memory requirements. On recent CPUs with an SSD (solid state drive), it will write a mesh up to 4 billion elements or nodes in only a few minutes, and millions of elements or nodes in seconds or less.

In the alpha release of UMR, only blocks and sidesets of tetrahedra (4 node) and triangles (3 node) are supported. No projection of new nodes to geometry or smoothing is performed. Each TET element results in 8 new TET elements, each TRI results in 4 new TRI elements. Resulting blocks and sidesets in the output mesh will have the same IDs as the input mesh.

In its current form, UMR is run as an executable available on the LAN. It has the following options as displayed with the '-h' option:

Usage: ./build/extra/extra [OPTIONS] [INPUT] [OUTPUT]

Positionals	
INPUT FILE</>	Input mesh file (coarse mesh).
OUTPUT FILE</>	Output mesh file (refined mesh).

Options:	
-h,--help	Print this help message and exit
-g,--geometry FILE	Snap boundary to geometry given in STEP format
-a,--associate	Output the geometry association to a .exo file
-b,--boundary	Output the boundary mesh as OBJ file
-m,--metrics	Print element quality in input mesh and exit
-j,--jobs N:UINT in [0 - 30]	Number of concurrent jobs. 0 means maximum number of logical cores. (default = 0).
-v,--verbose N:INT in [0-3]	Write increasingly more output (default =1)
-q,--quiet	Suppress output (same as --verbose=0)
-d,--debug N:INT in [0 - 5]	Write more debug output to log files (default = 0).
-V,--version	Prints version information

Project home page: https://cee-gitlab.sandia.gov/meshing/extra

Mesh Scaling

Cubit supports the scaling of hexahedra and tetrahedra meshes. Mesh Scaling allows a series of meshes to be built [typically] for solution convergence studies or other purposes. Each mesh has progressively larger or smaller elements. For example, if the input mesh has 10,000 hexahedra, scaling with a multiplier of 2.0 will result in a mesh of about 20,000 hexahedra, with approximately the same element orientation and size gradations as the original, as seen in Figure 1. Additional meshes can be built by scaling the original mesh with multipliers of 4, 6, 8, etc. Scaling by a value less than 1.0 will produce a mesh with fewer elements. Scaling by a negative value is not allowed. Convergence studies can be performed with much less computational cost than if traditional global refinement is used, because the element increase at each step of the series can be smaller.

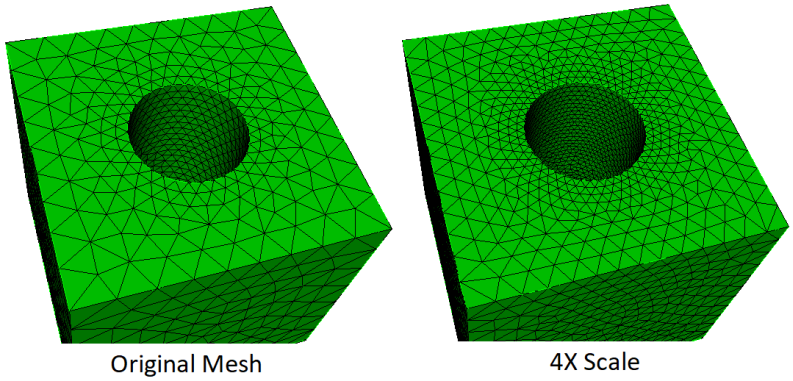


Figure 1.

Command Syntax

```
scale mesh [volume <ids>]
[multiplier <value, default=2.0>]
[minimum <value, default=1>]
[{SWEPT_BLOCKS | legacy | maintain_structure}]
[feature_angle <value>]
[force_structured in {[volume <ids>] [surface
<ids>]}]
[thin_gap_intervals <value, default=2>]
[fix_all_gaps]
[max_aspect_ratio <value> in volume <ids>]
[max_feature_length <value, default=30>]
[smooth_volume {ON|off}]
```

Command Options

scale mesh [volume <ids>]

Specifying a list of volumes is optional. By default, all volumes will be scaled. For Hex Mesh Scaling, the specified volumes, together with any volumes merged with them, will be scaled. Tet Mesh Scaling only scales the specified volumes, preserving the shared mesh boundary with any non-participating volumes. This approach allows individual assembly components to be scaled, without scaling the entire model.

[multiplier <value, default=2.0>]

The target number of output elements is the number of input elements times the **multiplier** parameter. The default value is 2.0. For example: the mesh in Figure 2a has 3025 hex elements. After scaling by 2.0, the

mesh in Figure 2b has 6804 hexes. Note the locations of new nodes are projected to lie on the associated CAD geometry, if any.

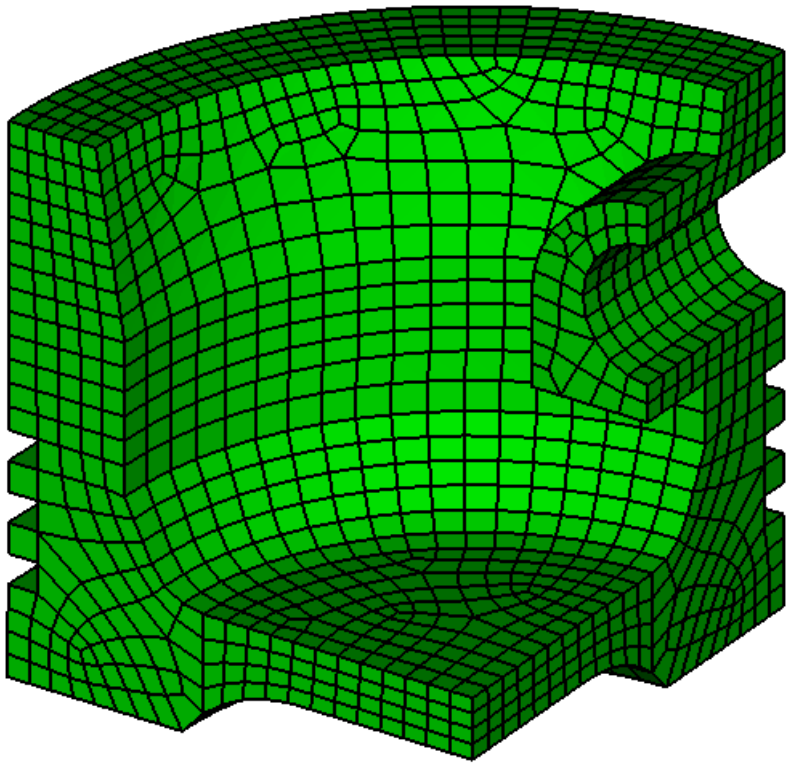


Figure 2a. Input mesh of 3025 hex elements.

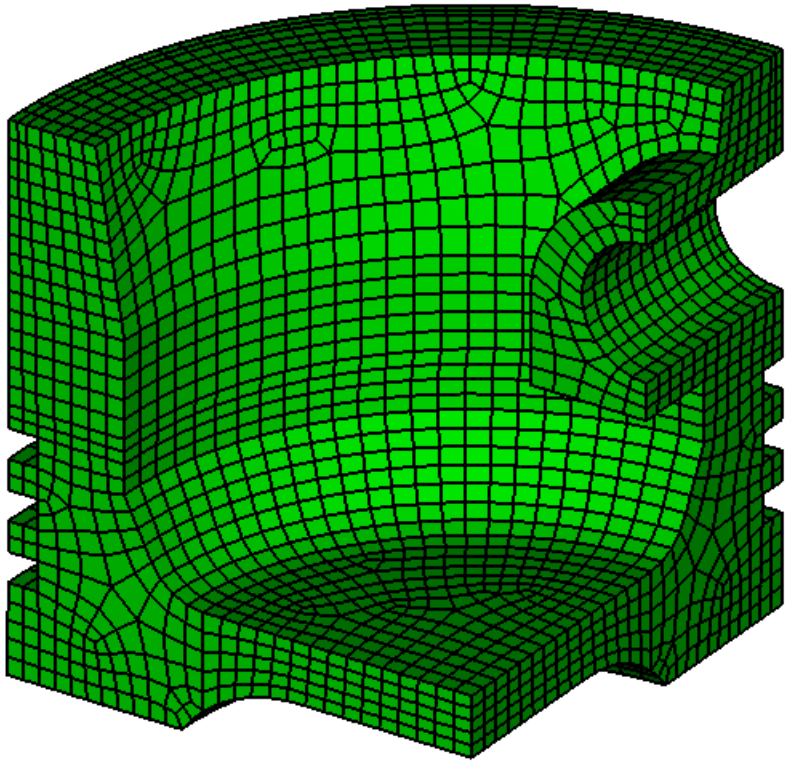


Figure 2b. Output mesh of 6804 hexes, after scaling with a multiplier of 2.0.

Tet Mesh Scaling

Tetmesh scaling woks by using the original tet mesh as a background sizing mesh, with a size on each node to achieve the desired scale factor. The only command parameter relevant to tet mesh scaling is the **multiplier** parameter. All others are used for Hex Mesh Scaling.

Hex Mesh Scaling

Hex Mesh Scaling is more flexible than template-based global refinement methods, because it does not require that every element is refined, or refined in the same way. Instead, Hex Mesh Scaling decomposes the entire mesh into larger "blocks" of hexes, and then refines the blocks. In this way, Mesh Scaling supports increasing the element count by small multiplicative factors, e.g. 1.5, that are impossible with template based refinement. However, like template refinement, it can ensure that every location of the mesh is refined. These features can be useful for solution verification.

A traditional template-based refinement replaces each hexahedron with a 2x2x2 structured grid of hexahedra, increasing the element count by a factor of 8X. In contrast, Hex Mesh Scaling refines "blocks" of elements (not to be confused with Exodus element blocks). The block decomposition subdivides the entire mesh into structured (mapped) and swept blocks. A block may contain many elements, but is not allowed to cross geometric boundaries, boundary conditions, and loading constraints. For example, a block cannot have a curve or nodeset in its interior, nor hexes from multiple Exodus blocks. Blocks may be structured or logical sweeps. A structured block is restricted to be a grid of MxNxO hexes, so, its extent is limited by any surface nodes that do not have exactly four edges, etc. Hex Mesh Scaling remeshes the entire model conforming to the block decomposition, using the original mesh as a sizing function, multiplied by the scale factor.

Hex Scale Mesh Command Options

[minimum <value, default=1>]

The **minimum** parameter provides further control over the level of refinement. It is the minimum number of intervals added to each block-curve. Specifying **minimum 1**, which is the default, will guarantee that at least one interval is added to every element block in all 3 directions, which guarantees every part of the domain is scaled by at least a little bit. This can be "turned off" by specifying **minimum 0**.

Refinement and Coarsening Levels

The multiplier determines the target number of output hexes. There is no guarantee it will be achieved exactly. For **minimum 0** and small multipliers, there is no guarantee that every block will be refined in all 3 directions. This is because the target number of elements may be reached first. This may lead to unevenly distributed refinement, with jumps in adjacent element sizes.

An uneven distribution may also result if adjacent blocks have significantly different MxNxO intervals; this is common with the **legacy** option. For example, for a 1x10x12 block adjacent to a 6x10x12 block, Mesh Scaling could output 2x11x13 and 7x11x13 blocks. The M value of the first block has doubled, 2/1, while the M value of the second block has only increased by 7/6. Thus, the user may observe a jump in the lengths of adjacent edges.

To coarsen a mesh, specify a multiplier less than one. For example, a multiplier of 0.9 will attempt to decrease the element count by 10%. Each block side will have its intervals *decreased* by the minimum value. A block must have at least one interval, so how far the mesh can be coarsened is limited by the distance between mesh irregularities, geometry, boundary condition and loading constraints.

Solution Verification

Mesh Scaling is useful for solution verification, as it can easily generate a series of similar meshes of increasing mesh density. For best results,

generate each mesh in the series by scaling the original mesh, rather than scaling the previous mesh of the series. It is suggested that each mesh uses a **multiplier** at least 2X larger, and a **minimum** at least one more, than the prior mesh. Small multipliers alone are unlikely to produce sufficient changes for solution verification. The **minimum** is especially useful for ensuring changes in regions that are initially coarse. A good set of (**multiplier, minimum**) parameters follows:

```
(2X, 1) (4X, 2) (8X, 3) (16X, 4), (prior *2,
prior +1), etc.
```

A large **minimum** can cause quality problems and generate too many elements. For example, the **minimum** in the parameter series **(2X, 1) (3X, 2) (4X, 3) (5X, 4) (6X, 5) (7X, 6) (8X, 7) etc.** would likely be too aggressive. It would produce many more elements than the specified multiplier, potentially causing poor element quality or even mesh scaling failure. For a slowly increasing set of multipliers, a less aggressive minimum series is recommended, such as

```
(2X, 1) (3X, 1) (4X, 2) (5X, 2) (6X, 2) (7X, 2)
(8X, 3) etc..
```

[{SWEPT_BLOCKS | legacy | maintain_structure}]

There are three major block decomposition variations to choose from. To understand their differences, one must first understand the two types of blocks: "structured" and "swept" blocks. A structured block is a MxNxO structured grid. A swept block is a single-source to single-target sweep of some subset of a single volume. That is, it is composed of a single surface of quad elements, projected some number of layers to form hexes.

The block decomposition options are **swept_blocks** (default), **maintain_structure** and **legacy**. For **legacy**, only structured blocks are used. For **swept_blocks** and **maintain_structure**, the decomposition constructs large swept blocks wherever logical sweeps can be identified, and structured blocks otherwise. The main difference is that **swept_blocks** remeshes the source surfaces of swept blocks from scratch. In contrast, **maintain_structure** partition each swept block into structured sub-blocks, and remeshes by selectively refining those sub-blocks. Thus **swept_blocks** may change the number and relative location of irregular nodes, whereas **maintain_structure** keeps them the same.

Typically, **swept_blocks** and **maintain_structure** provide smoother, more evenly distributed refinements compared to **legacy**. This is because with swept blocks, there are typically significantly fewer blocks in the decomposition. Having fewer blocks increases the likelihood that each block will receive at least some refinement before the multiplier is reached.

However, **legacy** and **maintain_structure** provide element orientations and structure closer to the original mesh than **swept_blocks**. This is because structured blocks maintain the irregular nodes.

Often **maintain_structure** provides both element orientations closer to the original mesh and a smoother, more evenly distributed refinement. Its structured blocks preserve orientations and structure. Its swept blocks provide the freedom to distribute changes, and smooth the mesh, across its structured sub-blocks.

[force_structured in {[volume <ids>] [surface <ids>]}]

In some cases, the user may want to use swept blocks in only some parts of the model. The original mesh may have small regions with carefully constructed meshes. Using **swept_blocks** can destroy these constructions, replacing them with pave-and-sweep meshes. These constructions can be preserved by specifying **force_structured** for the

surfaces and volumes containing them.

For example, see Figures 3, 4 and 5. In Figure 3 surface 108 was meshed with great care to ensure a structured mesh around the holes, while surface 34 was meshed with paving. Since surface 108 has irregular nodes, it appears to Mesh Scaling as the source surface of a swept block. Figure 4 illustrates the resulting mesh from the command **"scale mesh multi 2"**. Notice the structured meshes around the holes have been replaced with a standard paved mesh. Figure 5 illustrates preserving the structured holes of surface 108 while allowing surface 34 to be repaved. The command was **"scale mesh multi 2 force_structured in surface 108"**. A different mesh would result from the command **scale mesh multi 2 force_structured in volume 1** , because this would also preserve the irregular nodes in surface 34, resulting in more blocks and a less smooth mesh.

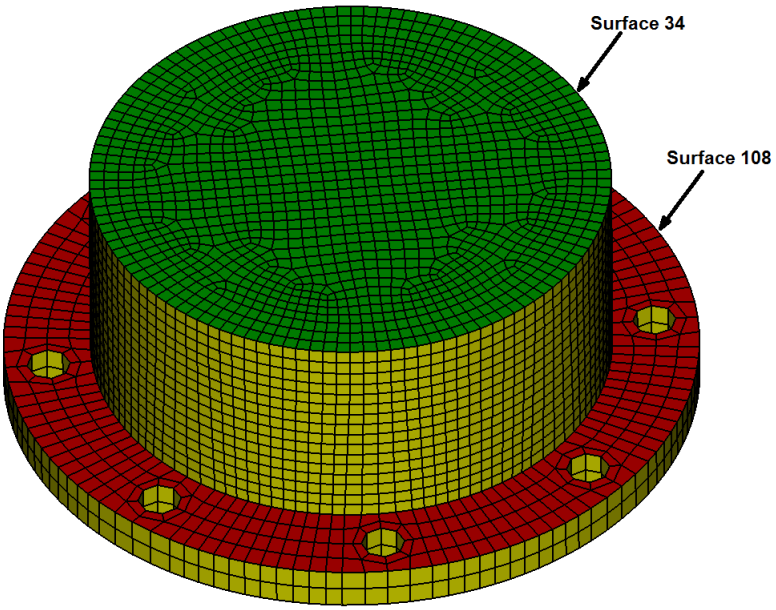


Figure 3. Input mesh. Surface 108's mesh has desired structure and irregular nodes. Surface 34 contains a paved mesh.

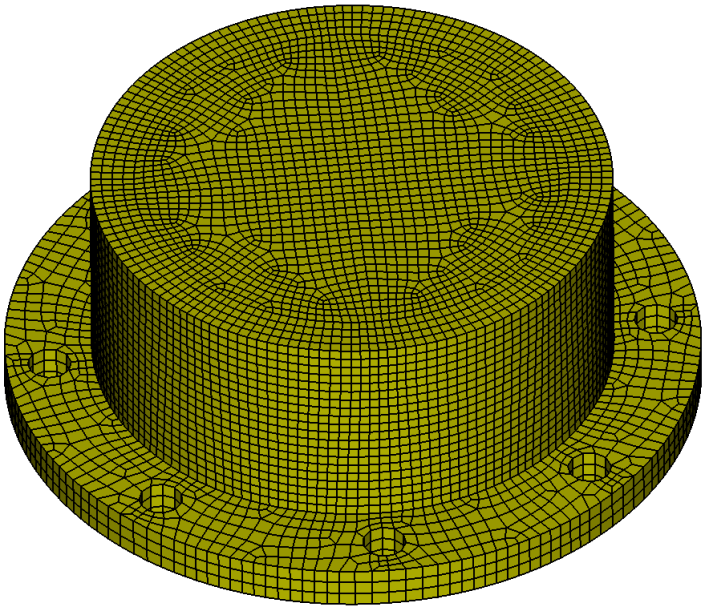


Figure 4. Output mesh from the command "Scale Mesh Multi 2". The mesh on surface 108 is replaced with a paved mesh.

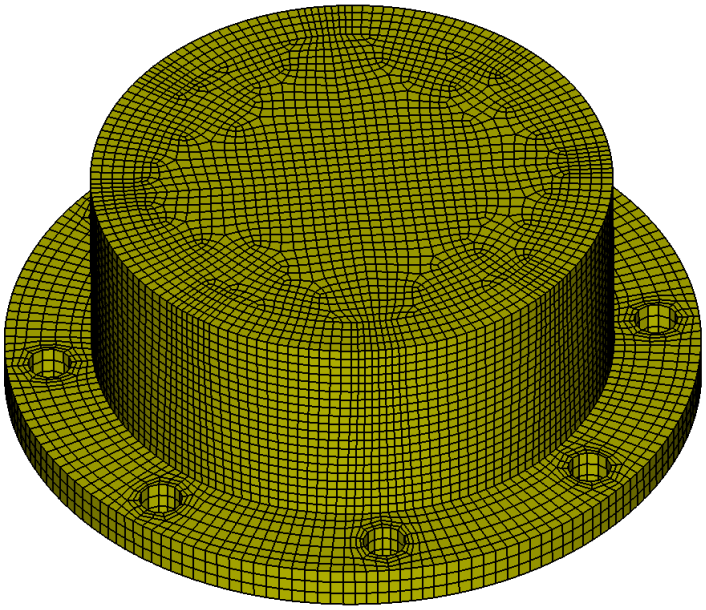


Figure 5. Output mesh from the command "Scale Mesh Multi 2 force_structured in Surface 108". The desired features are maintained, while swept blocks are used in unimportant regions.

[thin_gap_intervals <value, default=2>] [fix_all_gaps] [max_feature_length]

For the **maintain_structure** option, the **thin_gap_intervals** parameter determines how thin gaps are defined. If two disjoint curves of a surface come close together, the space between them is considered a "thin gap" if the number of intervals across that space is at most **thin_gap_intervals**. Mesh scaling gives high priority to adding intervals within thin gaps. By default, the position of *some* nodes within thin gaps are fixed to help reduce skew. The **fix_all_gaps** option fixes *all* nodes in thin gaps. Features, e.g. curves and gaps, longer than **max_feature_length** intervals will be split into multiple features. This results in fixing additional nodes along the feature to help reduce skew.

In general, **maintain_structure** should result in a smoother scaled mesh than the other algorithms, however, sometimes it can introduce some skew in the scaled elements. If skew is introduced by scaling using **maintain_structure**, try either increasing the **thin_gap_intervals** parameters, specifying **fix_all_gaps**, decreasing the **max_feature_length** parameter, or all three.

[feature_angle <value>] [max_aspect_ratio <value> in volume <ids>]

The **feature_angle** and **max_aspect_ratio** options affect the formation of swept blocks. These are alpha, experimental commands.

[smooth_volume {ON|off}]

If **smooth_volume** is on, then the volume mesh is smoothed as a post-process if it has poor quality elements, and smaller minimum quality than the original mesh. By default, smooth_volume is on.

Block Repositioning

A capability to reposition blocks is provided. This capability will retain all the current connectivity of the nodes involved. Unlike the Nodeset Move command, this command works for blocks containing free mesh (mesh not owned by geometry.)

```
Block <id_range> Move <delta_x><delta_y>
<delta_z>
```

Node and Nodeset Repositioning

A capability to reposition nodesets and individual nodes is provided. This capability will retain all the current connectivity of the nodes involved, but it cannot guarantee that the new locations of the moved nodes do not form intersections with previously existing mesh or geometry. This capability is provided to allow the user maximum control over the mesh model being constructed, and by giving this control the user can possible create mesh that is self-intersecting. The user should be careful that the nodes being relocated will not form such intersections.

The user can reposition nodes appearing in the same nodeset using the **NodeSet Move** command. Moves can be specified using either a relative displacement or an absolute position. The command to reposition nodes in a nodeset is:

```
Nodeset <nodeset_list> Move <delta_x>
<delta_y> <delta_z>

Nodeset <nodeset_list> Move To <x_pos> <y_pos>
<z_pos>
```

The first form of the command specifies a relative movement of the nodes by the specified distances and the second form of the command specifies absolute movement to the specified position.

Individual nodes can be repositioned using the **Node Move** command. Moves are specified as relative displacements. The command syntax is:

```
Node <range> Move <delta_x> <delta_y>
<delta_z>

Node <range> Move {[X <val>] [Y <val>] [Z
<val>]}

Node <range> Move Normal to Surface <id>
distance <val>

Node <range> Move Closest Surface <id>
distance <val>
```

Nodes can also be repositioned using a location or direction specification. See Location, Direction, and Axis Specification for details on the location and direction specification. The command syntax is:

```
Node <range> Move Location <options>

Node <range> Move Direction <options>
```

If **node constraint** is set to **on** then nodes will snap to geometry that they are associated with after the move. This only happens with the **node move** command and not with **nodeset move**. This behavior can be turned off with

```
set node constraint off
```

See also Transforming Mesh Coordinates.

Mesh Pillowing

Mesh pillowing is a mesh refinement technique that inserts a layer or 'pillow' of elements around the boundary of an enclosed mesh. It can be used to improve mesh quality while preserving the outer boundary of the selected element set. Mesh Pillowing can be used to quickly perform a number of meshing tasks, such as inserting a uniform boundary layer a specified distance from an outer boundary, or inserting a ring of elements around a hole.

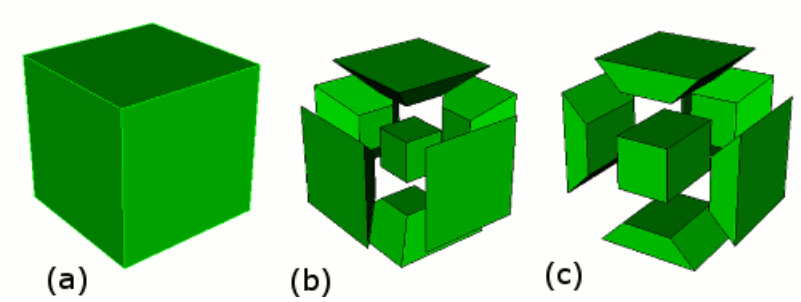


Figure 1: A single hex before (a) and after (b) a pillow operation. The far right (c) depicts a pillow operation with the front surface designated as a 'through' surface.

During a typical pillow operation, the user selects a set of elements, called a 'shrink set', to define what elements will be operated on. All elements on the outer boundary of the shrink set are then shrunk towards the center of the set. New elements are then created to fill the gap between the original boundary and the shrunk boundary. The newly created elements form the pillow around the selected shrink set. Figure 1a and 1b show an example of a pillow operation performed on a single hex. Geometry surfaces, or mesh element faces can be specified as **through** surfaces for the pillowing operation. This means that the pillow will extend through the selected surfaces, and no new elements will be created along them. Figure 1c shows the effect of pillowing a single hex with one surface selected as a through surface.

In some cases a shrink set may not be valid due to the geometry of a specific region. As the exterior nodes of the shrink set move towards the middle they must be able to maintain appropriate geometric associations. Nodes on vertices must move along curves, nodes on curves must move along surfaces. If there are multiple curves or surfaces along which an exterior node might travel, then the ownership is ambiguous and the pillowing will fail.

Using the optional **distance** keyword with a specified value allows manual control of the distance that each boundary element is shrunk towards the center of the shrink set. If no distance value is specified, an appropriate value is calculated for each element. If a distance value is specified, all newly created nodes will have their position fixed by default. This allows the user to smooth the mesh without altering the node positions of the newly created hexes. If the optional **unfix_nodes** keyword is used, this default behavior is changed, and any smooth operations will alter the newly created node locations. By default, a smooth operation is automatically performed following any pillow operation unless the optional **no_smooth** keyword is used.

Similar analogous commands are available for creating a pillow around a set of two dimensional faces.

Syntax:

```
Pillow Hex <ids> [ Through { [Surface <ids>]
[Face <ids>][Tri <ids>] } ] [ Distance <value>
```

```
] [ Unfix_nodes ] [ No_smooth ]

Pillow Face <ids> [ Through Curve <ids> ] [
Distance <value> ] [ Unfix_nodes ] [No_smooth]
```

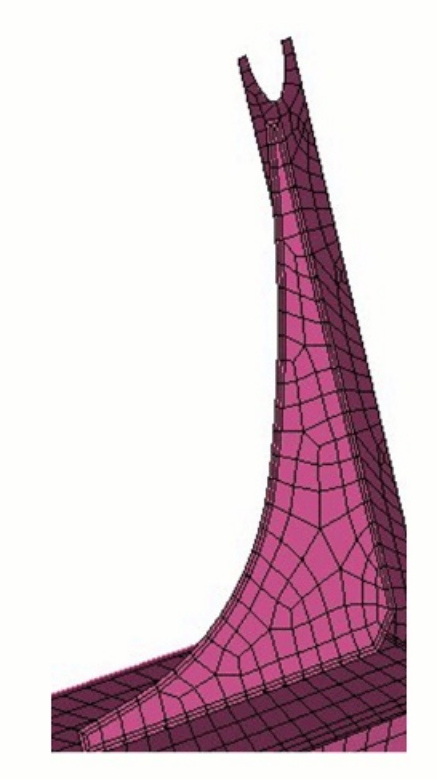


Figure 2: Example model using pillow operations to create ordered nodes a specified distance around the boundary of a mesh.

Mesh Column Operations

Column operations allow users direct control over the mesh connectivity while maintaining full-geometric associativity. Often, hex meshing schemes such as sweeping and mapping result in mesh topology forced into unnatural shapes, such as a square shaped source surface mesh getting swept into a circular target surface. Often forcing meshes into shapes like this results in poor element quality because of non-optimal element angles. The Column commands allow users to directly modify mesh topology to make minor tweaks to a mesh improving element quality. Column operations are almost always followed by smoothing to enable element quality improvement. Cubit provides tools to perform insertion, deletion, swapping, grouping, and drawing of hex columns.

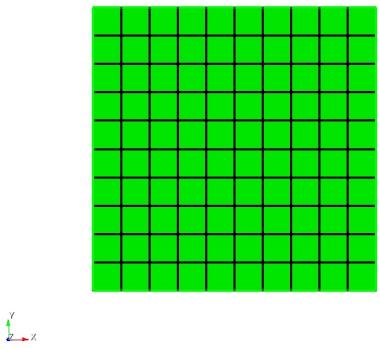
- Column Insertion
- Column Deletion
- Column Swapping
- Column Groups
- Drawing Columns

Column Insertion

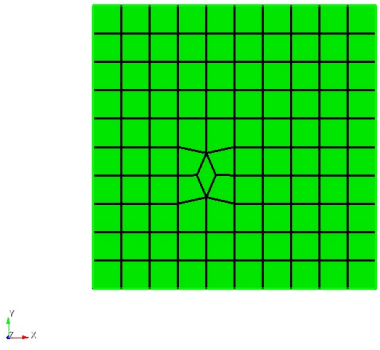
A single column can be inserted into the mesh by using the following command:

```
column open node <center node id> <orientation
node ids>
```

For example, given the following meshed brick:



we issue the command, **column open node 89 88 90 ,** to get this result:



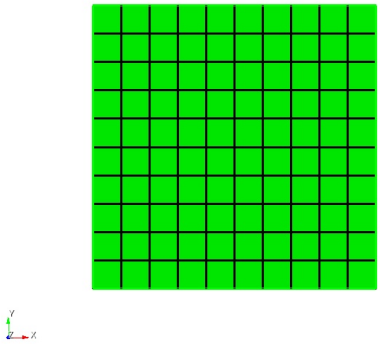
Column Deletion

Columns can be removed with neighboring columns being joined together using **collapse** commands. Collapse commands are of two types: interior and boundary.

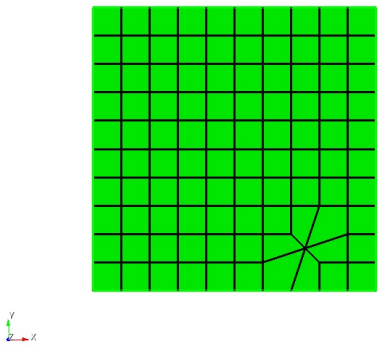
For interior node collapse, the two nodes which are opposite on a face are combined together. The column associated with the face is removed. Use the following command:

```
column collapse node <opposite node ids>
```

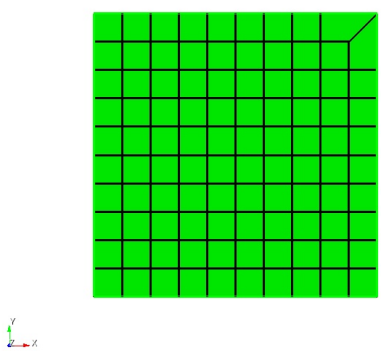
For example, given the following meshed brick:



we issue the command, **column collapse node 51 59**, to get this result:



The column collapse command can be used with boundary nodes. For example, we issue the command, **column collapse boundary node 13 2 11**, to get this result:

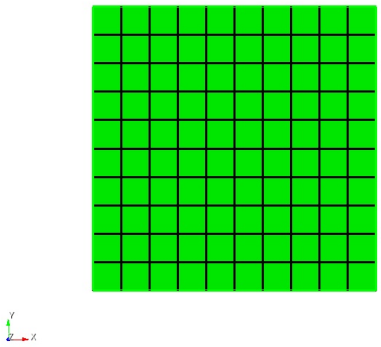


Column Swapping

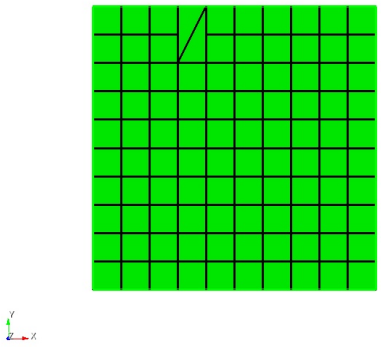
Faces between two hex columns can be swapped using the following command:

```
column swap node <old edge node ids> <new edge node ids>
```

For example, given the following meshed brick:



we issue the command, **column swap node 103 94 102 18**, to get this result:



Column Groups

A group consisting of hexes that comprise a column can be created using the following command:

```
column { face <id> | edge <id1> <id2> | hex <id1> <id2> } group
```

Drawing Columns

Columns can be drawn using the following command:

```
draw column { face <id> | edge <id1> <id2> | hex <id1> <id2>}
```

Removing Intersecting Mesh

In addition to finding mesh intersection , Cubit® can also remove mesh intersection by slightly repositioning the nodes of intersecting elements. Nodes are moved along surface normals just enough so that the intersection is zero or undetectable. The command syntax is:

```
Remove Mesh Intersection {Block|Body|Volume}
<id_list> [with {Block|Body|Volume} <id_list>]
[low <value=0.0001>] [high <value>] [detail]
```

The **low** and **high** options set how much cumulative intersection should be detected. A low value of 0.1 would ignore elements that do not intersect more than 10% of their volume. Similarly, a high value of 0.5 would discard elements that intersect more than 50% of their volume. Both low and high can be used simultaneously. The default for the **low** value is 0.0001

The **detail** option provides information, printing the ten nodes moved the largest distances, providing users an idea of the greatest changes in the mesh.

```
Largest node movements:
Node 165 moved a distance 0.003889
Node 170 moved a distance 0.003886
Node 1091 moved a distance 0.003861
Node 163 moved a distance 0.003832
Node 1100 moved a distance 0.003794
Node 1090 moved a distance 0.003739
Node 1101 moved a distance 0.003728
Node 117 moved a distance 0.003701
Node 164 moved a distance 0.003672
Node 121 moved a distance 0.003483
```

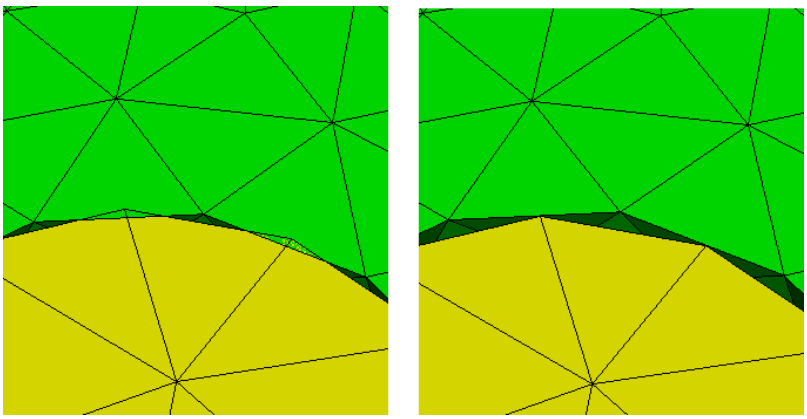


Figure 1. "Before and after mesh intersection removal"

Note:

- Elements that intersect more than 50% of their volume will not be modified, as element quality could degrade substantially. Users will be notified of these elements through a printed warning.
- The shape of higher-order mesh elements is considered when computing intersection.

Importing a Mesh

- Importing 2D Exodus II Files
- Importing Exodus II Files
- Importing Patran Files
- Importing I-DEAS Files
- Importing Abaqus Files
- Importing Nastran Files
- Importing Fluent Files

ExodusII finite element data files can be imported into CUBIT. Several options for importing the mesh are available, (including mesh transformations):

- Importing a free mesh without geometry.
- Importing a free mesh and associating the mesh with ACIS-based geometry currently residing in CUBIT.
- Importing a 2D mesh and constructing ACIS-based Geometry
- Importing a mesh and constructing Mesh-Based Geometry from dihedral angles and boundary conditions.
- Importing a preview mesh.

Importing Fluent Files

The command to import a mesh from a fluent format file is:

```
Import Fluent [Mesh Geometry]
'<input_filename>' [Feature Angle <angle>]
[nobcs]
```

Including the keyword **Mesh Geometry** will instruct CUBIT to create mesh-based geometry. This will provide the user with the ability to remesh geometric entities. If the user does not import with the Mesh Geometry flag, he will have to tell CUBIT to draw the mesh after the import is done in order to view it.

The **Feature Angle** is used when building the surface topology to determine when to split a surface into two surfaces. If the angle between two neighboring element normals is less than Feature Angle, then the two elements will be placed on separate surfaces. If the keyword Feature Angle is not supplied, the default 135 degrees is used. For a description of importing mesh geometry see Importing Exodus II Files.

The keyword **nobcs** can be included if boundary conditions are not to be imported.

It should be noted that CUBIT sometimes cannot successfully generate mesh-based geometry for complex models. If this occurs, import the mesh without the Mesh Geometry flag, and draw the mesh to view it.

Importing 2D Exodus Files

CUBIT has a limited capability to create ACIS Geometry from 2D ExodusII finite element mesh files. (For a more general capability, see the Import Mesh Geometry command, which will create Mesh-Based Geometry).

To import a 2D Exodus II file and create ACIS geometry, the following command can be used:

```
Import Free Mesh '<filename>' {Time <t> | Step
<step#> | Last}
```

CUBIT can create ACIS geometry from 2D Exodus II data files (4, 8, or 9 node QUAD or SHELL element types) that do not have enclosed voids (holes surrounded by mesh) and which were originally generated with CUBIT and exported to ExodusII with the Nodeset Associativity option set to on. The Nodeset Associativity command records the topology of the geometry into special nodesets which allow CUBIT to reconstruct a new solid model from the mesh even after it has been deformed. The new solid model of the deformed geometry can be remeshed with standard techniques or meshed with a sizing function that can also be imported into CUBIT from the same ExodusII file. CUBIT's implementation of the paving and triadvance algorithms can generate a mesh following a sizing function to capture a gradient of any variable (element or nodal) present in the ExodusII file.

In order for this feature to be effective, the following commands must be issued when the mesh is exported and later imported:

```
nodeset associativity on

set associativity complete on
```

The first command ensures that the geometry will be correctly recovered from the mesh, while the second ensures that boundary condition and material IDs will be recovered.

Importing Abaqus Files

The command to import a mesh from an Abaqus format file is:

```
Import Abaqus [Mesh Geometry]
'<input_filename>' [Feature Angle <angle>]
[Nobcs]
```

Including the keyword **Mesh Geometry** will instruct CUBIT to create mesh-based geometry. This will provide the user with the ability to remesh geometric entities. If the user does not import with the Mesh Geometry flag, he will have to tell CUBIT to draw the mesh after the import is done in order to view it.

The **Feature Angle** is used when building the surface topology to determine when to split a surface into two surfaces. If the angle between two neighboring element normals is less than Feature Angle, then the two elements will be placed on separate surfaces. If the keyword Feature Angle is not supplied, the default 135 degrees is used. For a description of importing mesh geometry see Importing Exodus II Files.

The keyword **nobcs** can be included if boundary conditions are not to be imported.

The Abaqus importer can import the following Abaqus file formats: flat file, part-independent, and part-dependent.

It should be noted that CUBIT sometimes cannot successfully generate mesh-based geometry for complex models. If this occurs, import the mesh without the Mesh Geometry flag, and draw the mesh to view it.

To list Abaqus cards supported by Cubit:

```
List Abaqus Import Cards
```

This command will list out all supported Abaqus cards that CUBIT can interpret.

Table 1. Supported Element Types

	1st Order	2nd Order
Triangle	S3 STRI3 CAX3 CPE3	STRI6 CAX6 CPE6
Quadrilateral	S4 CAX4 CPE4	S8 CAX8 CPE8
Tetrahedron	C3D4	C3D10
Hexahedron	C3D8 SC8R	C3D20
Wedge	C3D6	C3D15

Line Element	B21 B31 T2D2 T3D2 SPRINGA SPRING1 SPRING2 CONN3D2	B22 B32 T2D3 T3D3
Sphere	MASS	

See http://www.simulia.com/ for more information on the ABAQUS file format.

Importing Exodus II Files

- Importing a Free Mesh without Geometry
- Importing a Free Mesh onto Existing Geometry
- Creating Mesh-based Geometry on Import
- Importing a Lite Mesh

The commands to import meshes from an Exodus II format file are:

```
Import Mesh '<exodusII_filename>' No_Geom
[Block <block_ids>] [block_name <names>]
[{genesis_collision_fail|COMBINE_GENESIS_IDS|combine_genesis_names|unique_genesis_ids}]
[block_offset <value>] [sideset_offset
<value>] [nodeset_offset <value>]
[{node_id_collision_fail|UNIQUE_NODE_IDS}]
[{element_id_collision_fail|UNIQUE_ELEMENT_IDS}]
[node_offset <value>] [element_offset
<value>]] [nodal_var{<string>|all_nodal_vars}]
[element_var<string>|all_element_vars}]
[group_name '<free_elements>'] [[Time
<time>|Step <step>|Last] [Scale <value>]]

Import Mesh '<exodusII_filename>'
[Block <block_ids>] [block_name <names>]
[{genesis_collision_fail|COMBINE_GENESIS_IDS|combine_genesis_names|unique_genesis_ids}]
[block_offset <value>] [sideset_offset
<value>] [nodeset_offset <value>]
[{node_id_collision_fail|UNIQUE_NODE_IDS}]
[{element_id_collision_fail|UNIQUE_ELEMENT_IDS}]
[node_offset <value>] [element_offset
<value>]]
[{Group|Body|Volume|Surface|Curve|Vertex}
<id_range> | Preview]

Import Mesh Geometry '<exodusII_filename>'
[Block <id_range>|ALL] [block_name <names>]
[{genesis_collision_fail|COMBINE_GENESIS_IDS|combine_genesis_names|unique_genesis_ids}]
[block_offset <value>] [sideset_offset
<value>] [nodeset_offset <value>]
[{node_id_collision_fail|UNIQUE_NODE_IDS}]
[{element_id_collision_fail|UNIQUE_ELEMENT_IDS}]
[node_offset <value>] [element_offset
<value>]] [nodal_var{<string>|all_nodal_vars}]
[element_var<string>|all_element_vars}] [Use
[NODESET|no_nodeset] [SIDESET|no_sideset]
[Feature_Angle <angle>] [Surface_Feature_Angle
<angle>]
[LINEAR|Gradient|Quadratic|Spline|Acis]
[Deformed {Time <time>|Step <step>|Last}
[Scale <value>] ] [MERGE|No_Merge]
[Merge_nodes <tolerance>]
[{midnode_correction|NO_MIDNODE_CORRECTION}]

Import Mesh '<exodusII_filename>' Lite
[{genesis_collision_fail|COMBINE_GENESIS_IDS|combine_genesis_names|unique_genesis_ids}]
[block_offset <value>] [sideset_offset
<value>] [nodeset_offset <value>]
[{node_id_collision_fail|UNIQUE_NODE_IDS}]
[{element_id_collision_fail|UNIQUE_ELEMENT_IDS}]
[node_offset <value>] [element_offset
<value>]]
```

Related Commands:

```
Import Mesh Geometry (options)

Import Free Mesh (2D)

Delete Mesh Preview
```

```
Export [ Genesis | Mesh ] '<filename>'

List Import Mesh NodeSet Associativity

List [Export Mesh] NodeSet Associativity

[Set] Import Mesh NodeSet Associativity
[ON|off]

[Set] [Export Mesh] NodeSet Associativity
[on|OFF]

Transforming Mesh Coordinates

Set Import Mesh [Vertex] [Curve] [Surface]
Tolerance <distance>

Set Import Mesh NodeSet Order [On|Off]

List Import Mesh NodeSet Order
```

Common Options

Specifying a Portion of the Mesh to be Imported

The **Block** option in the **Import Mesh** command indicates that only the specified element block should be imported from the Exodus II file. The blocks are specified by ID. The **block_name** option can be used to import only the named blocks. Multiple blocks can be imported by making a comma separated list of names. The **block** and **block_name** options can be used together to specify some blocks by ID and others by name. If the **block** or **block_name** options are not specified, then the entire mesh file is read. These options are not yet supported for lite imports, which currently imports the entire mesh.

Combine Genesis IDs Option

The **combine_genesis_ids** option, is used to combine blocks where the IDs in the session and the file being imported are identical. This can occur when importing into an active session where Cubit IDs have already been assigned. The default behavior is to combine genesis entities based on IDs. If two entities have differnet IDs, but the same names, they will not be combined, and the import will fail.

Combine Genesis Names and Nodes IDs Options

The **combine_genesis_names** option, is used to combine blocks where the names in the session and the file being imported are identical. This can occur when importing into an active session where names have already been assigned. If two entities have different names, but identical IDs, they will not be combined, and the import will fail.

The **combine_node_ids** option is only available in the 'lite' form of the 'import mesh' command. This option combines nodes, where the ids in the session and the file being imported are identical, regardless of location (coincidence).

Unique Genesis IDs Option

The **unique_genesis_ids** option is used to renumber genesis entities from the genesis file in the case that ID overlap exists when importing into Cubit. The incoming genesis entities are kept unique and are not combined with genesis entities already in the session. In case of colliding IDs, a report displayed in the command window showing the original and new IDs. This renumbering can occur when importing into an active session where Cubit IDs have already been assigned. If an entity being imported has the same name as one already in the session, the entity being imported will be renamed with a number suffix '_N'.

Genesis Collision Fail Option

The **genesis_collision_fail** option allows the user to prevent the genesis file import if any incoming genesis entity IDs or names are already used by genesis entities in the session. This can occur when importing into an active session where Cubit IDs have already been assigned.

Block_Offset, Sideset_Offset, Nodeset_Offset Options

The **block_offset**, **sideset_offset**, and **nodeset_offset** options may be used to modify the IDs of genesis entities being imported. The provided value will be added to the ID in the file.

Unique Node and Element IDs Option

The **unique_node_ids**, and **unique_element_ids** options are used to automatically renumber nodes and elements from the genesis file in the case that ID overlap exists when importing into Cubit. If there are no overlaps, the IDs in the file will be preserved. This can occur when importing into an active session where Cubit IDs have already been assigned.

Element and Node ID Collision Fail Option

The **node_id_collision_fail**, and **element_id_collision_fail** options allows one to prevent the genesis file import if any incoming node or element IDs are already used by mesh entities in the session. This can occur when importing into an active session where Cubit IDs have already been assigned.

Node_Offset and Element_Offset Options

The **node_offset**, and **element_offset** options may be used to modify the IDs of nodes and elements being imported. The provided value will be added to the ID in the file.

Node/Element Variables

The **nodal_var** and **element_var** options allow nodal and element variable information respectively to be imported from the Exodus file. To import a specific varible, a name can be specified. Using the ***_all** option imports all variable information. By default, no variable information is imported.

Importing a Free Mesh Without Geometry

The command to import a free mesh from an Exodus II format file without mesh-based geometry is:

```
Import Mesh '<exodusII_filename>' No_Geom
[group_name '<free_elements>'] [[Time
<time>|Step <step>|Last] [Scale <value>]]
```

When a free Exodus II mesh is imported into Cubit, it contains no geometric or topological information. Previously, the user could either associate that mesh with existing geometry, or build mesh-based geometry to fit the mesh. A third option, as of Cubit 11.1, allows the user to retain the disassociated mesh as a free mesh inside Cubit.

A free mesh may be modified as described in the Free Mesh section of the documentation. This includes limited access to smoothing, renumbering, transformations, refinement, mesh quality, and other mesh centric operations.

When an Exodus II File is imported as a free mesh, Cubit will automatically create a group called "free_elements" to contain the free mesh elements. The 'group_name' option can be used to give the group a different name.

Deformation information can be read in via the **Time/Step/Last** and **Scale** parameters.

Note: The **Import Mesh <filename> No_Geom** command is not to be confused with the **Import Free Mesh** command which applies only to 2D Exodus II Files.The term "Free Mesh" in both places of the documentation refers to the same thing - a mesh without geometry. However, in the case of all other import mesh commands, the imported free mesh ends up associated with geometry. The **Import Mesh <filename> No_Geom** is the only way to import a free mesh that remains disassociated from geometry.

Importing a Mesh Onto Existing Geometry

The command to import a free mesh from an Exodus II format file and associate it with existing geometry is:

```
Import Mesh '<exodusII_filename>'
[{Group|Body|Volume|Surface|Curve|Vertex}
<id_range> | Preview]
```

The user can import a mesh from an Exodus II file and associate the mesh with matching geometry. The resulting mesh may then be manipulated normally. For example, the mesh may be smoothed or portions of it deleted and remeshed. The user can save their work by exporting the geometry and mesh, and then restore the geometry and mesh later. In some cases, saving and restoring can be faster or more reliable than replaying journal files.

Saving and importing a mesh may be useful for teams working on creating a conforming mesh of a large assembly so that they can pass information to one another. For example, a team member can export the mesh on the surfaces between two parts, and another team member import the mesh for use on an adjoining part of the assembly.

As of cubit version 7.0, any higher order elements, block definitions, nodesets, and sidesets are retained on import.

Importing a Mesh with Nodeset Associativity

Meshes can be imported into Cubit that contain nodeset associativity data used for defining finite element boundary conditions. If an exported Cubit mesh is going to be imported back onto the same geometry, then before exporting the user should issue the following command:

```
set export mesh nodeset associativity on
```

This causes extra nodeset data to be written, which associates every node to a geometric entity, resulting in an import which is more reliable. When importing, if the user does not want to use the nodeset associativity data that exists in a file, then before importing the following command should be used:

```
set import mesh nodeset associativity off
```

The user may wish to turn geometry associativity off if, for example, the geometry is no longer identical as a result of curves being composited, or Cubit names changed due to a ACIS version changes.

Importing a Mesh onto Modified Geometry

Although there are some exceptions, Cubit requires that the mesh be imported onto the same geometry from which it was exported.

Since merge information is not stored with the ACIS representation, care should be taken that the geometry is merged the same way on export and import of the mesh. If not, importing the mesh one block at a time in successive commands may increase the chance of a successful import, at the cost of more memory and time.

Between exporting and importing a mesh, the geometry may be modified

slightly by compositing entities. Mesh import will, however not be successful if entities are partitioned or a body is webcut. In some cases mesh import may be successful on modified geometry if the new vertices match up exactly with nodes of the mesh, and the new curves match up exactly with edge chains of the mesh. Unless this criteria is met, associating the mesh with the geometry will be unsuccessful.

Mesh Import Tolerance

To change the tolerance with which imported mesh must line up with geometry issue the command:

```
Set Import Mesh [Vertex] [Curve] [Surface]
Tolerance <distance>
```

Specifying a Portion of the Mesh to be Imported

The **Block** option in the **Import Mesh** command indicates that only the specified element block should be imported from the Exodus II file. In the same manner, the **Volume** and other geometry options provide a way to import the nodes and element on the indicated geometry. If neither a **block** nor a **geometry entity** is specified, then the entire mesh file is read.

If a **block** is specified without specifying a **geometry entity**, associativity or proximity is used to determine which volume the block elements should be associated with. If a **block** and a **volume** are specified, the block elements are associated with the specified volume, provided they actually match. If a **volume** is specified without a **block**, associativity data is used to find a block corresponding to the given volume.

Nodeset Ordering

If the Import mesh NodeSet Order flag is on, the nodesets will be read in a manner which allows them to be associated with existing geometry. This means the nodesets are assumed to be in ascending order. If the flag is set to false, the goemetry nodesets in imported mesh files are assumed to be in random order. This value is on by default, and should not need to be changed by the user.

Creating Mesh-Based Geometry on Import

Cubit's mesh generation tools require an underlying geometry representation. In most cases, the ACIS solid modeling engine, compiled with Cubit, is used to represent the geometry. However, in some cases, an ACIS representation is not available, and a previously developed finite element mesh is the only available representation of the model. In order to utilize Cubit's mesh generation tools, the **import mesh geometry** command provides an option for creating geometry directly from the finite element mesh.

The **import mesh geometry** command will create a new volume for every block defined in the Exodus II file. It will also create curves, surfaces and vertices at appropriate locations on the model based on dihedral angles (also called feature angles) and assigned nodesets and/or sidesets. The mesh used to construct the geometry will be owned by the new geometric entities. This means that the mesh can be deleted, remeshed, or smoothed using any of Cubit's meshing tools by simply using the new geometry definition. Cubit will assign appropriate intervals to the new curves as well as determine an acceptable meshing scheme for surfaces and volumes.

The command to import a finite element mesh from an ExodusII format file and generate geometry from the mesh is:

```
Import Mesh Geometry '<exodusII_filename>'
[Block <id_range>|ALL] [Use
```

```
[NODESET|no_nodeset] [SIDESET|no_sideset]
[Feature_Angle <angle>] [Surface_Feature_Angle
<angle>]
[LINEAR|Gradient|Quadratic|Spline|Acis]
[Deformed {Time <time>|Step <step>|Last}
[Scale <value>] ] [MERGE|No_Merge]
[Merge_nodes <tolerance>]
```

File Name

Type the name of file to import in single quotation marks. The file must reside in the current directory. For information on changing the current directory, see Cubit environment commands. To list all the files in the current directory, type **ls** at the command prompt.

Blocks

Use this option to select the specific blocks to be imported from the Exodus II file. If no blocks are entered, then all blocks will be read and imported from the file. Standard ID parsing can also be used in this argument to select a range of blocks. For example "**1 to 5**" or "**1, 5 to 10 except 6**".

Each unique block selected to be imported will define a new body in the geometric model. Figure 1 shows a simple example of the geometry generated from the 3D finite element mesh.

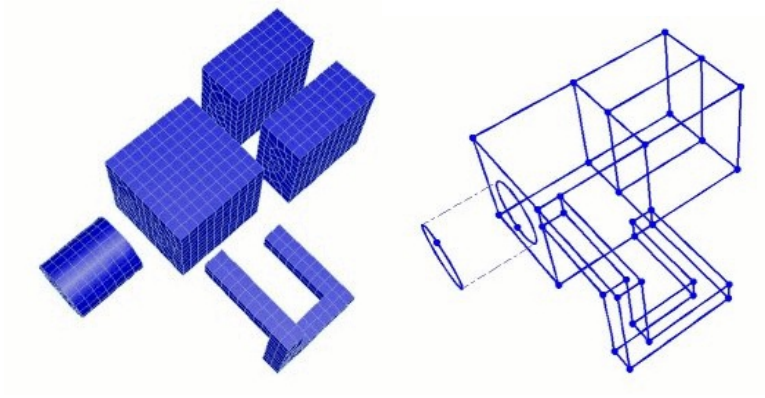


Figure 1. Example of mesh based geometry (right) created from a finite element mesh (left)

Blocks may be composed of 1D, 2D or 3D elements. For blocks composed of 2D elements (i.e. QUAD4, SHELL etc.), a sheet body will be created. One dimensional elements (i.e.. BEAM, TRUSS, etc.) will define curves. Where a block may be composed of more than one disconnected sets of elements, one body will be created for each continuous region of elements assigned to the same block. Where possible, the ID of the new body will be the same as the block ID. Since IDs must be unique, if a body ID is already in use, the next available ID will automatically assigned by the program.

Nodesets/Sidesets

Use the **nodeset** and **sideset** options to use any nodeset and sideset information in the Exodus II file in constructing geometry. Recall that nodesets and sidesets are generic boundary condition data assigned to nodes, edges or faces of the finite elements. It is useful to group mesh entities belonging to unique boundary conditions into geometric entities. This permits the user to remesh a particular region of the model without having to reassign boundary conditions.

If the **nodeset** and **sideset** arguments are given, geometric entities will be generated for each unique set of nodes, edges or element faces assigned to a nodeset or sideset. The default is to use any nodeset and sideset information available in the file. Figure 2 shows an example of how nodeset and sideset information might be used to generate

geometry.

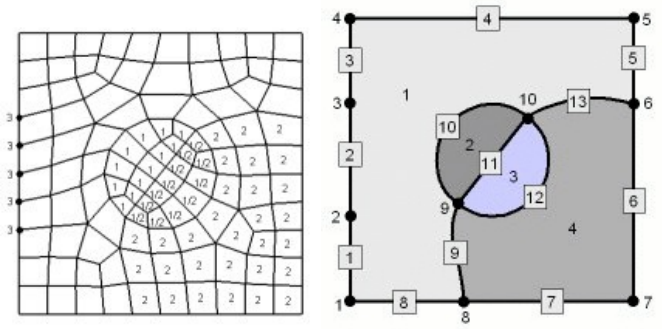


Figure 2. Example of geometry created from mesh entities assigned to nodesets (3) and sidesets (1 and 2).

Upon import, nodesets and sidesets are automatically created with the appropriate geometric entities assigned to them. The IDs of the new geometric entities, if generated from boundary condition data, will be the same as the nodeset and sideset IDs. Where doing so would conflict with existing geometric IDs, the program will automatically select the next available ID.

Feature Angle

Use this option to specify the angle at which surfaces will be split by a curve or where curves will be split by a vertex. 180 degrees will generate a surface for every element face, while 0 degrees will define a single, unbroken surface from the shell of the mesh. The default angle is 135 degrees.

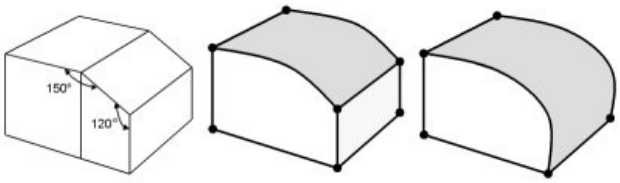


Figure 3. Example use of Feature Angle

Figure 3 shows an example of the use of different feature angles. On the left is a simple two-element hex mesh. Specifying a feature angle greater than 120 degrees would create the geometry in the center image. Using a feature angle less than 120 degrees and greater than 90 degrees would define the geometry on the right.

It is possible to independently control the feature angle for surfaces and curves. If Surface_Feature_Angle is specified, it controls the angle at which surfaces will be split by a curve, while Feature_Angle controls the angle at which curves will be split by a vertex. If Surface_Feature_Angle is not specified, Feature_Angle will control the angle for both surfaces and curves.

Smooth Curves and Surfaces

This argument allows the option of using a higher-order approximation of the surface when remeshing/refining the resulting geometry. Default is to use the original mesh faces themselves as the curve and surface geometry representation. If the finite element model to be imported is to represent geometry with curved surfaces, it may be useful to select this option. If selected, it will use a 4th order B-Spline approximation to the surface [Walton,96]. Figure 4 shows the effect of the smooth curve and surface option.

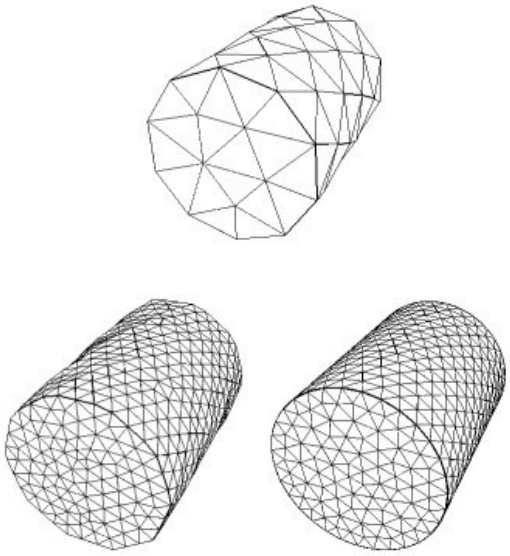


Figure 4. Effect of Smooth Curve and Surface Option for remeshing of mesh-based geometry

In this figure the top image is the original finite element mesh imported into Cubit. In this example both models have been remeshed with the same element size. The difference is that the figure on the right uses the smooth curve and surface option. While this option can improve the surface representation, it should be noted that memory requirements and meshing times can sometimes be affected.

If importing the Exodus II file using the command line, other options for surface representations are also available.

```
[LINEAR|Gradient|Quadratic|Spline|Acis]
```

The method used from the GUI is either **Linear** or **Spline**. The **Gradient** and **Quadratic** methods are still somewhat experimental and may not be as general purpose as the **Spline** representation. The **Acis** option will attempt to create ACIS geometry from the mesh. This option is an alpha feature and can only be used if developer commands have been turned on. For more detail see: Acis Geometry From Mesh

Apply Deformations

The **deformed** option permits the user to import time-dependant deformation information from the Exodus file. For this option, any vector data in the Exodus II file is assumed to be deformation information. If selected, deformations will be applied to the nodes upon import. Enter a specific time step value, integer step, or the last time available in the file. If time-dependant data is available in the Exodus II file, selecting the down arrow in the edit field will display the available time steps in the file. Default time is the last time step.

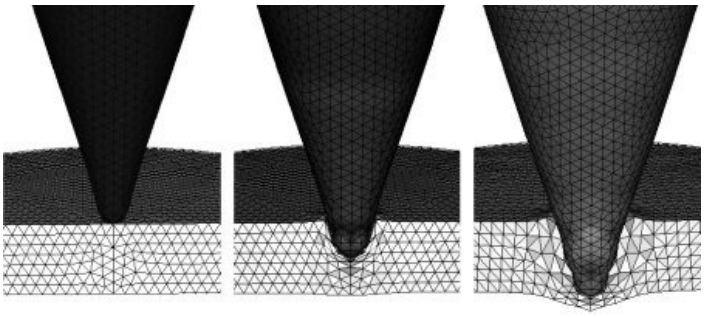


Figure 5. Example of remeshing of a deformed finite element mesh

Figure 5 shows an example of using Mesh-Based Geometry for a large deformation analysis. In this case, the analysis [Attaway et. al.,98] began and continued until mesh quality became unacceptable. At that point, the

mesh was imported into Cubit and geometry re-created from the computed deformations. The finite element mesh could then be removed, remeshed or improved and written back to an Exodus II file. After remapping [Wellman,99] the appropriate analysis variables back to the mesh, the analysis could then be restarted. This process was repeated multiple times until the desired results were achieved.

Note: Care should be taken when using large deformations, as inverted elements (negative Jacobians) may produce unpredictable results with the resulting geometric representation.

Also available is an optional **scale** factor. This applies the indicated scale to all deformations. Default is 1.0.

Merge

This option allows the user to either merge or not merge the resulting volumes. The default option is to merge adjacent volumes. This results in non-manifold topology, where neighboring volumes share common surfaces. Using the no_merge option, adjacent volumes will generate distinct/separate surfaces.

Merge Nodes

The **merge_nodes** option will allow the user to specify a different tolerance for merging nodes on import. The default value is 1e-6.

Note: Care should be taken when setting import merge tolerances. Setting a tolerance too low will not merge adjacent nodes. Setting the tolerance too high can produce undesirable results, and severely tangle the mesh.

Midnode Correction

When importing a mesh in Cubit, the software performs quality checks on the elements. By default, if any elements fall below the specified quality threshold, Cubit notifies the user. This quality assessment applies to imported mesh-based geometry that uses high-order tetrahedral elements (tet10).

For TET10 elements, the **node constraint smart** command evaluates element quality using a designated quality metric, typically the default **normalized inradius**. If the quality of an element falls below a defined threshold (default: **0.15**) and straightening the edge of the TET10 would improve the quality, Cubit automatically performs midnode correction. This correction can also be enabled during the import process.

By default, the midnode correction option is set to **no_midnode_correction**, meaning that the mesh will be imported without modifying midnode locations. However, users have the flexibility to choose the **midnode_correction** option when importing mesh-based geometry. Enabling this option allows Cubit to straighten high-order edges if the mesh quality metric falls below the designated threshold and if it results in improved quality.

Importing a Lite Mesh

The command to import a lite mesh from an Exodus II format file is:

```
Import Mesh '<exodusII_filename>' lite
```

When an Exodus II mesh is imported into Cubit using the lite option, it contains no geometric or topological information.

The lite mesh import may be an option for users wanting to quickly view the mesh without gaining all the abilities to modify the mesh.

More information on how a lite mesh may be viewed or modified is described in the Lite Mesh section of the documentation.

Importing I-DEAS Files

The command to import a mesh from an I-DEAS format file is:

```
Import Ideas [Mesh Geometry]
'<input_filename>' [Feature Angle <angle>]
[Nobcs]
```

Including the keyword **Mesh Geometry** will instruct CUBIT to create mesh-based geometry. This will provide the user with the ability to remesh geometric entities. If the user does not import with the Mesh Geometry flag, he will have to tell CUBIT to draw the mesh after the import is done in order to view it.

The **Feature Angle** is used when building the surface topology to determine when to split a surface into two surfaces. If the angle between two neighboring element normals is less than Feature Angle, then the two elements will be placed on separate surfaces. If the keyword Feature Angle is not supplied, the default 135 degrees is used. For a description of importing mesh geometry see Importing Exodus II Files.

The keyword **nobcs** can be included if boundary conditions are not to be imported.

It should be noted that CUBIT sometimes cannot successfully generate mesh-based geometry for complex models. If this occurs, import the mesh without the Mesh Geometry flag, and draw the mesh to view it.

To see more information on the I-DEAS file format, visit their website at www.siemens.com.

Importing Nastran Files

The command to import a mesh from an Nastran format file is:

```
Import Nastran [Mesh Geometry]
'<input_filename>' [Feature Angle <angle>]
[Nobcs]
```

Including the keyword **Mesh Geometry** will instruct CUBIT to create mesh-based geometry. This will provide the user with the ability to remesh geometric entities. If the user does not import with the Mesh Geometry flag, he will have to tell CUBIT to draw the mesh after the import is done in order to view it.

The **Feature Angle** is used when building the surface topology to determine when to split a surface into two surfaces. If the angle between two neighboring element normals is less than Feature Angle, then the two elements will be placed on separate surfaces. If the keyword Feature Angle is not supplied, the default 135 degrees is used. For a description of importing mesh geometry see Importing Exodus II Files.

The keyword **nobcs** can be included if boundary conditions are not to be imported.

It should be noted that CUBIT sometimes cannot successfully generate mesh-based geometry for complex models. If this occurs, import the mesh without the Mesh Geometry flag, and draw the mesh to view it.

See http://en.wikipedia.org/wiki/Nastran for more information on the NASTRAN file format.

Importing Patran Files

The command to import a mesh from an Patran format file is:

```
Import Patran '<neutral_filename>'

Import Patran Mesh Geometry
'<neutral_filename>' [Use [Feature_Angle
<angle>] [Linear|Gradient|Quadratic|Spline] ]
```

See Importing Exodus II Files for a description of the import options.

For more information on the Patran file format, see their website at www.mscsoftware.com.

Geometry Adaptive Sizing for TriMesh and TetMesh Schemes

The **TriMesh** and **TetMesh** schemes in Cubit are based upon third party libraries known as MeshGems that are developed and distributed by Distene. They are robust and fast triangle and tet meshing algorithms that have built in capabilities for adaptively controlling the mesh size based upon feature sizes. In most cases the sizing controls provided as part of the **scheme** command are sufficient to control mesh sizes. As such, the sizing functions described in this section cannot be used with the the MeshGems triangle and tet meshing algorithms. If a sizing function is assigned to a volume or surface, and the **TriMesh** or **TetMesh** scheme is selected, rather than using the MeshGems algorithm for meshing the surfaces, it will automatically revert to using the **TriAdvance** scheme. Any settings defined with the **TriMesh** or **TetMesh** scheme will be ignored and the sizing function will be used to determine local mesh sizes.

When using the **TriMesh** and **TetMesh** schemes, recommended practice is to mesh all surfaces and volumes simultaneously. This provides the greatest flexibility to the algorithms to determine feature sizes and their effect on neighboring surfaces and volumes. The default settings for **TriMesh** and **TetMesh** schemes will automatically provide geometry adaptive mesh sizing. These default settings can however be adjusted by using the settings on the **scheme** command. The scheme settings are described in the **TetMesh** and **TriMesh** sections of the documentation.

Mesh Adaptivity and Sizing Functions

CUBIT provides several options for controlling the density of a mesh by adapting to various geometric, analysis, or user-defined properties. Interval sizes are defined automatically, explicitly, or through *sizing functions*. The sizing functions can be based on the physical features of the model, a previous analysis solution, or a user-specified bias. Adaptivity can apply to meshing either curves or surfaces.

Adaptive Curve Meshing

CUBIT provides several ways to adaptively mesh curves. Three curve meshing schemes are provided for this purpose. They include the following schemes:

- Curvature
- FeatureSize

The first two schemes use characteristics of the geometric model to define element sizes. The third scheme uses a field function typically defined from a previous analysis solution. FeatureSize is an alpha feature and should be used with caution.

Adaptive Surface Meshing

Adaptive surface meshing in CUBIT produces a function following mesh which sizes elements based on the value of the driving function at the spatial location at which the element is to be placed. Adaptive surface meshing is performed using the paving, triadvance or tridelaunay algorithms in combination with an appropriate sizing function. The types of sizing functions that can be used are

- Bias Sizing
- Constant Sizing
- Curvature Sizing
- Linear Sizing
- Interval Sizing
- Inverse Sizing
- Super Sizing
- Exodus-based field function
- Geometry Adaptive (Skeleton Sizing)
- Geometry Adaptive for TriMesh and TetMesh Schemes

The Super sizing function is an alpha feature and should be used with caution.

The procedure for adaptively meshing a surface is to designate paving, triadvance or tridelaunay as the mesh scheme for that surface, assign sizing function types, and mesh the surface.

The command syntax of these commands is:

```
Surface < id > Scheme
{Pave|TriAdvance|TriDelaunay}

then

Import Sizing Function '<exodusII_filename>'
Block <block_id> Variable '<variable_name>'
Time <time> [Deformed]

Surface <id> Sizing Function [Type] Exodus
```

```
[Min <min_value> Max <max_value>]
```

or

```
Surface <id> Sizing Function [Type]
{Constant|Curvature|Interval|Inverse|Linear|Super|None}]
[Neighbor [<max_neighbors>]]
```

(See note below regarding 'Neighbor' parameter)

or

```
Surface <id> Sizing Function [Type] Bias Start
Curve <id_range> {Finish Curve <id_range>|
Factor <val>}
```

then

```
Mesh Surface <id>
```

Adaptive Volume Meshing

Adaptive volume meshing in CUBIT produces a function following mesh that sizes elements based on the value of the driving function at the spatial location at which the element is to be placed. Adaptive volume meshing is performed using the tetmesh scheme in combination with an appropriate sizing function. The types of sizing functions that can be used are constant, geometry adaptive and geometry adaptive (skeleton sizing). Other sizing functions will be added in future versions of Cubit.

The procedure for adaptively meshing a volume is to designate **tetmesh** as the mesh scheme for that volume, assign sizing function types, and mesh the volume.

The command syntax of these commands is:

```
Volume <id> scheme tetmesh
Volume <id> Sizing Function [Type]
{Constant|None}
Mesh Surface <id>
```

The following sections describe details of the various volume sizing methods.

- Constant Sizing
- Geometry Adaptive (Skeleton Sizing)
- Geometry Adaptive for TriMesh and TetMesh Schemes

Note regarding 'Neighbor' parameter:

The **maximum neighbors** is the number of points used by the sizing function to compute the size at the requested point. If the number of neighbors is zero, all of the points on the boundary are used in the size calculation. If the number of neighbors is some other number, only that number of closest points are used in the calculation.

Bias Sizing Function

Syntax:

```
Surface <id> Sizing Function Type Bias Start
Curve <id_range>
{Finish Curve <id_range>| Factor <val>}
```

Synopsis:

The **Bias** sizing function for surfaces is similar to biasing curves. Indeed, setting a bias sizing function for a surface will bias the boundary curves, as well as control paving to follow the bias inside the surface. You first specify the size of a couple of bounding curves (the start curves), then specify the bias sizing function for the surface.

Discussion:

Recall that for biasing curves, you specify the start and end vertex. For the bias sizing function, you specify the start curves, from which to bias away. The sizes of these curves should already be set before setting the surface sizing function since their average size is taken to be the starting size (almost). If the start curve sizes change, then you should set the surface sizing function again.

You can either supply a geometric factor, or the set of finish curves whose sizes you want to match at that distance. A geometric factor. It automatically sizes and biases or dualbiases the non-start curves, including any finish curves. These curves need not be perpendicular to the starting curves. The interval count and scheme are soft-set, so they won't be changed if they are already hard-set. If the size of the start curves or finish curves are changed, then the sizing function command should be re-issued.

The sizing function value at a point is defined in terms of the straight-line distance from the point to the closest starting curve. So, it works best if all the starting curves have the same size, and the surface is relatively flat. But, starting curves need not be parallel to one another. Similarly, the non-start curves need not have any particular orientation wrt the start curves.

The bias sizing function was designed to easily set the sizes of a sequence of adjoining surfaces: assign a size to the curve you want to bias away from, then set the bias sizing function of the first surface, with its finish curves being the start curve of the second surface, etc. See the last example below.

Examples:

Here are some example journal files and resulting pictures:

```
# bias_sz_fn_demo.jou
brick x 100 y 10 z 10
color vol 1 red
surface 1 scheme pave
surface all except 1 visibility off
# label curve interval
# graph text 2
display

# mesh 1
curve 4 size 2
surface 1 sizing function type bias start curve 4 factor 1.3
mesh surface 1
# see figure 1
```

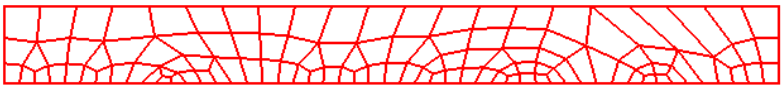


Figure 1. Surface with bias sizing function factor > 1.

```
# mesh 2
delete mesh
surface 1 sizing function type bias start curve 4 factor
{1/1.1}
mesh surface 1
# see figure 2
```

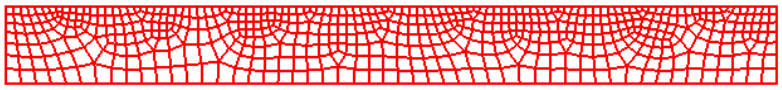


Figure 2. Surface with bias sizing function factor < 1

```
# mesh 3
reset
cyl rad 6 z 1
cyl rad 4 z 1
sub 2 from 1
section body 1 yplane
section body 1 xplane
surf all except 19 vis off
color vol 1 red
display

# finish curve mesh
surf 19 scheme qtri base scheme pave
surface 19 size 0.7
curve 26 size 0.07
surface 19 sizing function type bias start curve 26 finish
curve 25
mesh surface 19
pause
# see figure 3
```

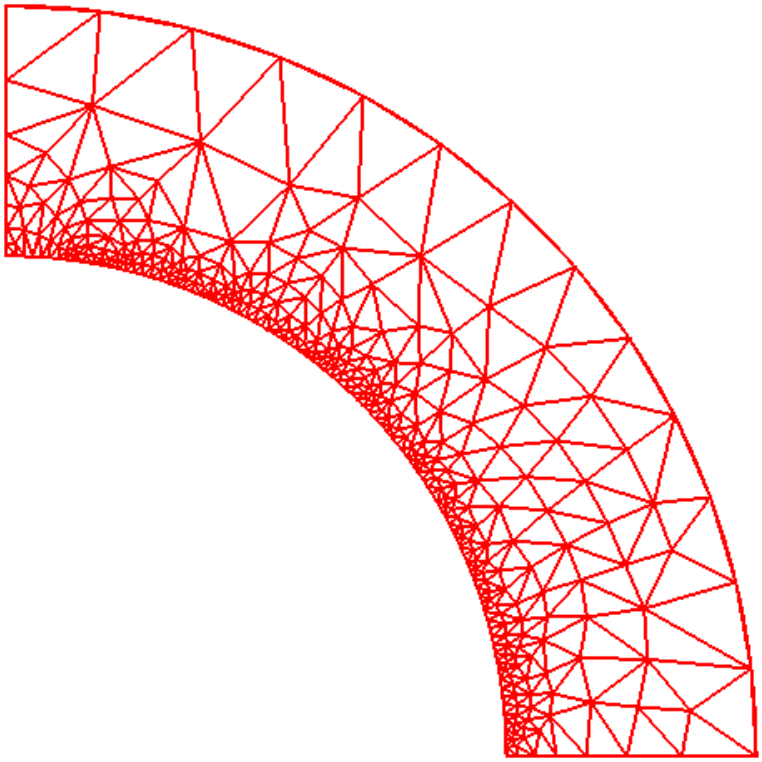


Figure 3. Surface with bias sizing function start and finish curve. Scheme qtri, base scheme pave.

```
# dual bias mesh
delete mesh
curve 25 26 size 0.02
curve 25 26 scheme equal
surface 19 sizing function type bias start curve 26 25 factor
1.3
mesh surface 19
zoom curve 12
pause
# see figure 4
```

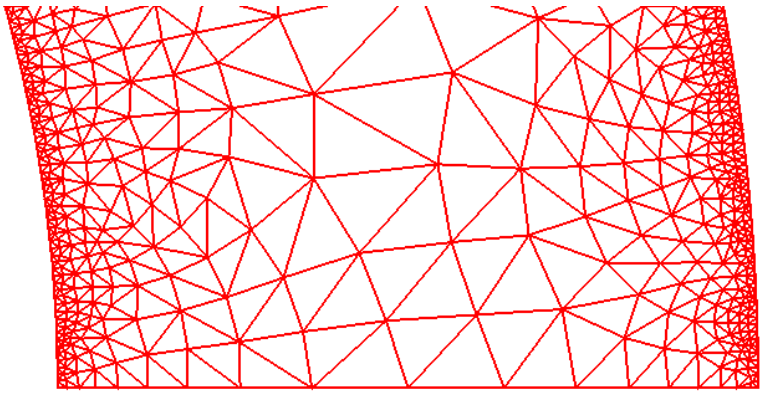


Figure 4. Close up of surface with dual bias sizing function start and finish curve. Scheme qtri, base scheme pave.

```
# funny face
reset
prism sides 5 z 1 radius 1
cylinder radius 0.1 z 1
body 2 move -0.4 0 0
subtract 2 from 1
cylinder radius 0.1 z 1
body 3 move 0.2 0 0
```

```
subtract 3 from 1
prism sides 6 radius 0.2 z 1
body 4 move 0 -0.4 0
subtract 4 from 1
surface all except 34 visibility off
color vol 1 red
display
surface 34 scheme pave
curve 23 19 size 0.01
surface 34 sizing function type bias start curve 19 23 factor
1.3
mesh surface 34
# see figure 5
```

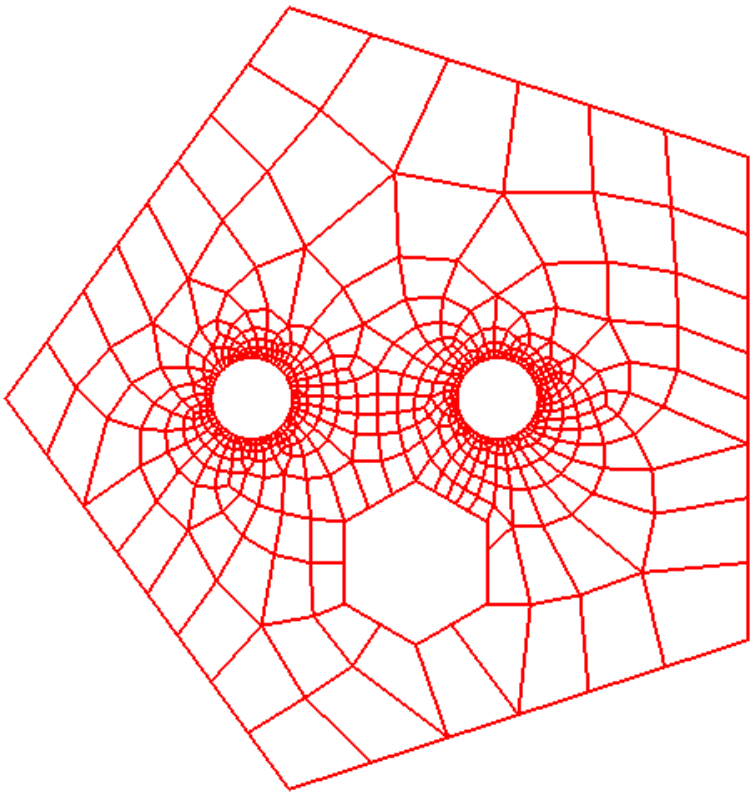


Figure 5. Bias away from two round holes.

```
# bias surface chain
reset
cylinder radius 1 z 1
cylinder radius 0.2 z 1
cylinder radius 0.4 z 1
cylinder radius 0.8 z 1
imprint body all
delete body 2 3 4
section body 1 xplane
section body 1 yplane
surface all except 42 43 44 45 vis off
color volume 1 red
surface all scheme pave
curve 55 interval 36
surface 43 sizing function type bias start curve 55 factor
1.3
surface 44 sizing function type bias start curve 57 factor
1.3
# curve 57 had its size determined by a prior bias sizing
function
surface 45 sizing function type bias start curve 58 factor
1.3
surface 42 sizing function type bias start curve 55 factor
```

```
1.3
mesh surface 42 43 44 45
display
highlight curve in surface 42 43 44 45
# see figure 6
```

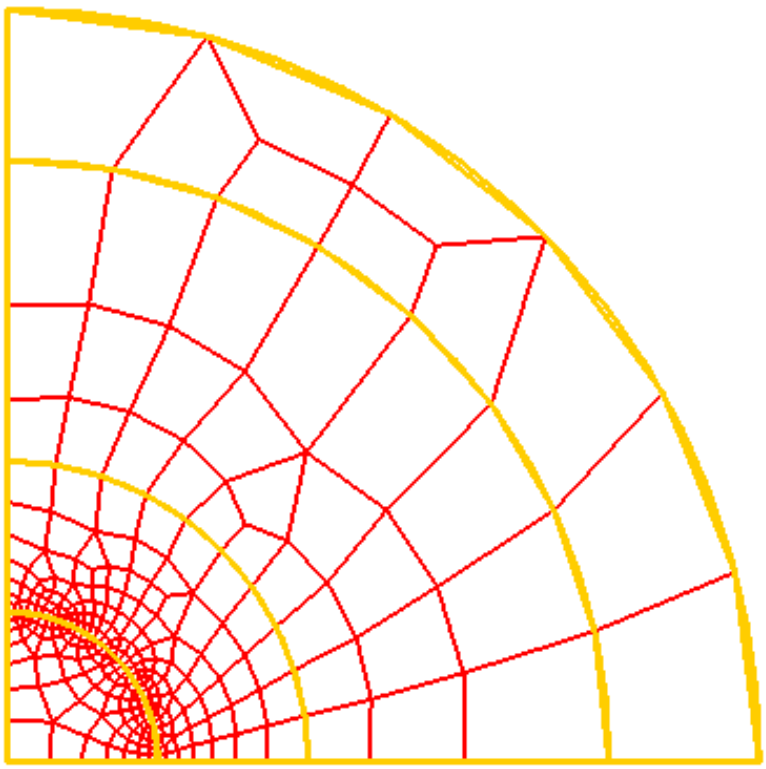


Figure 6. A chain of biased surfaces. Only one curve's intervals were explicitly set.

Constant Sizing Function

Syntax:

```
Surface <id> Sizing Function [Type] Constant

Volume <id> Sizing Function [Type] Constant
```

Synopsis:

The **Constant** sizing function specifies that a constant element size be used over the interior of the surface or volume. The value used as the constant size is the interval size that has been set for the entity. For example, the following commands will cause the mesh size to be smaller on the interior than on the surface's bounding curves.

```
reset
brick x 10
surface 1 scheme pave
curve in surface 1 interval 5
surface 1 size 0.5
surface 1 sizing function constant
mesh surface 1
```

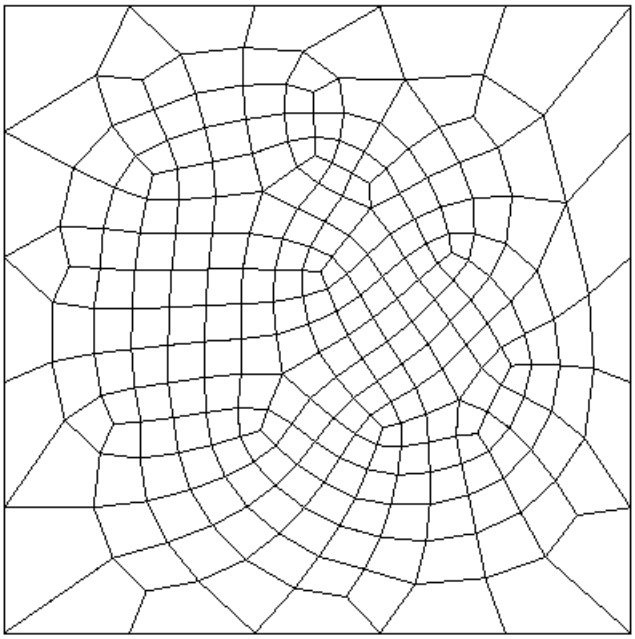


Figure 1. Constant Sizing Function

Curvature Sizing Function

The **Curvature** sizing function determines element size based on the curvature evaluation of a surface at the current location. Two surface curvature values (taken perpendicular to each other) are compared at the location of interest, and the largest is used as the sizing function for the mesh. Figure 1 shows a solid with a highly deformed surface which displays rapid change of surface curvature at several locations.

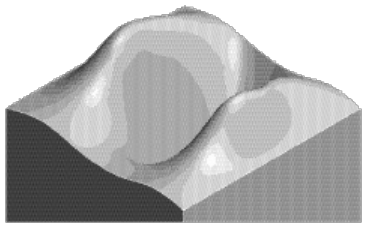


Figure 1. NURB solid with high surface curvature change

Figure 2 depicts a normal paved mesh of this surface using a common size on all bounding curves and no sizing function in the interior. The total number of quadrilateral shell elements for this case is 1988. Figure 3 shows a mesh which was generated with the curvature sizing function option. The mesh is graded denser in the regions of quickly changing curvature, such as at the tops of the hills and at the bottom of the valley. Due to the intense interrogation of the underlying geometric modeler which the curvature method relies on, this option can be very computationally expensive.

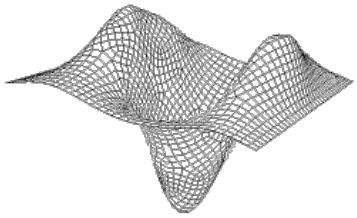


Figure 2. NURB mesh with no interior sizing function

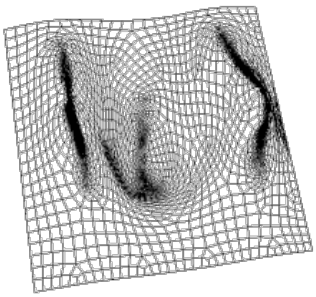


Figure 3. NURB mesh with curvature sizing function

Exodus II-based Field Function

The ability to specify the size of elements based on a general field function is also available in CUBIT. With this capability, the desired element size can be determined using a field variable read from a time-dependent variable in an Exodus II file. Both quadrilateral and triangle elements are supported for surfaces, and both tetrahedral and hexahedral elements are supported for volumes.

A field function is a time-dependent variable in an Exodus II file. Either node-based or element-based variables may be used. Currently, field functions are imported from element and node-based Exodus II data. The mesh block containing the corresponding elements must be imported along with the field function data.

Exodus variable-based adaptive meshing is accomplished in CUBIT in several steps:

1. Surface mesh scheme set to Pave or TriMesh, and/or volume mesh scheme set to Tetmesh.
2. An Exodus mesh and variable for that mesh is read into CUBIT.
3. The surface or volume sizing function type is designated and the Exodus variable is mapped to give normalized and localized size measures.
4. The geometry is meshed

Importing a field function, and normalizing that function are done in two separate steps to allow renormalization. The following command is used to read in a mesh for the field function:

```
Import Sizing Function '<exodusII_filename>'
Block <block_id> Variable `<variable_name>'
[Time <time_val> | Step <step> | Last]
[Deformed]
```

The **block_id** is the element block to be read, which can be a single block id or the word **all**. The **variable_name** is an Exodus time-dependent variable name (either element-based or nodal-based) which values are used to drive the mesh size. The timestep for the time-dependent variable can be specified as a **time** with a value, as a **step** with an index or **Last** to use the last timestep in the file. The **Deformed** keyword indicates whether to read the deformed field function mesh, which should align with the geometry being meshed and needs to be accounted for in the field function data. (For information on creating deformed geometry from EXODUSII data, see Importing 2D EXODUSII Files and Importing EXODUSII Files) .

Note that when a field function is read in, the mesh is stored in the background, and therefore the geometry is not considered meshed. Also note that if deformation is not being modeled, the geometry should be in the same state as it was when that mesh was written (see Importing a Mesh for more details on importing meshes).

Once the field function has been read in, it can be normalized before being used to generate a mesh. The normalization parameters (**Min_size** and **Max_size**) are specified in the same command that is used to specify the sizing function type for the surface or volume. The syntax of these commands are:

```
Surface <id> Sizing Function Type Exodus
[Min_size <min_val> Max_size <max_val> Log_Map
Inverse_Map Scale_Mesh_Multiplier <value>]

Volume <id> Sizing Function Type Exodus
[Min_size <min_val> Max_size <max_val> Log_Map
```

```
Inverse_Map Scale_Mesh_Multiplier <value>]
```

If normalization parameters are specified, the field function is normalized so that its range falls between the minimum and maximum values input. If an element-based variable is used for the sizing function each node is assigned a value that is the average of variables on all connected elements. Nodal variables are used directly.

The **Log_Map** option maps the range so that it is logarithmic, base 10.

Inverse_Map flips the mapping so that smaller and larger values in the mapping generate larger and smaller elements respectively. See figure 1 below.

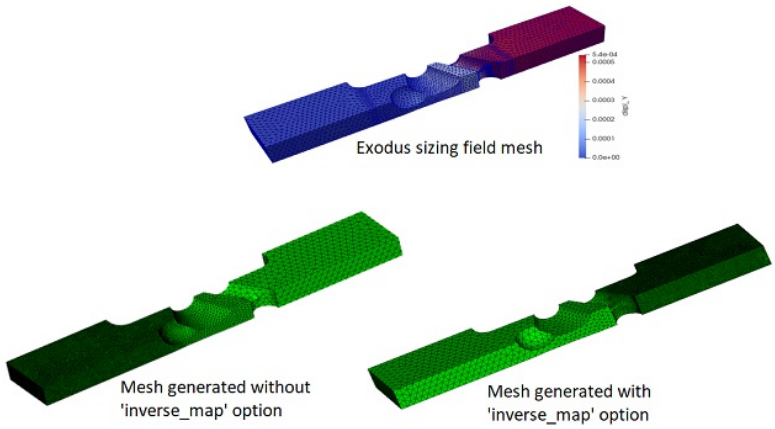


Figure 1. Effect of 'inverse_map' option

Scale_Mesh_Multiplier scales the size that the field data ultimately yields.

After the sizing function normalization, the geometry may be meshed using the normal meshing command.

For example, the left image in Figure 2 depicts a plastic strain metric which was generated by PRONTO-3D [Taylor, 89] a transient solid dynamics solver, and recorded into an ExodusII data file. When the file is read back into CUBIT, the paving algorithm is driven by the function values at the original node locations, resulting in an adaptively generated mesh [Attaway, 93]. The right image in Figure 1 depicts the resulting mesh from this plastic strain objective function.

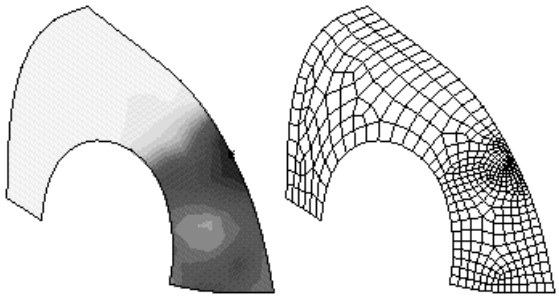


Figure 2. Plastic strain metric and the adaptively generated mesh

Surface/Curve Meshing with Exodus II - based Field Functions

While adaptively meshing a surface using a field function, the curves will be meshed using the Exodus II information. To override this, curves may have their meshing scheme set to equal or some other desired scheme. While adaptively meshing a volume using a field function, the surfaces and curves will be meshed using the Exodus II information. To override

this for a surface, one can set the sizing function to "none" for that surface.

Geometry Adaptive Sizing Function (Skeleton Sizing)

The **Geometry Adaptive Sizing Function**, also referred to as the **Skeleton Sizing Function** (Quadros 2005; Quadros 2004; Quadros 2004(2)), automatically generates a mesh sizing function based upon geometric properties of the model. This sizing scheme attempts to create a sizing function that allows unstructured meshing schemes to generate a mesh with the following properties:

- The sizes of the mesh elements vary smoothly throughout the mesh
- The mesh elements resolve the geometry to a sufficient degree
- The mesh elements do not over-resolve the geometry.

The geometry adaptive sizing function can be used to create sizing information for surfaces, solids, and assemblies.

This sizing function uses geometric properties to influence mesh size. The scheme calculates or estimates:

- 3D-proximity (thickness though the volume)
- 2D-proximity (thickness across a surface)
- 1D-proximity (curve length)
- Surface curvature
- Curve curvature.

These properties are then used to calculate a sizing function throughout the geometric entity (or entities). Regions of relatively high complexity will have a fine mesh size, while regions of relatively low complexity will have a coarse mesh size. For example, generally, a high-curvature region on a surface will have a finer mesh size than a low-curvature region on that surface

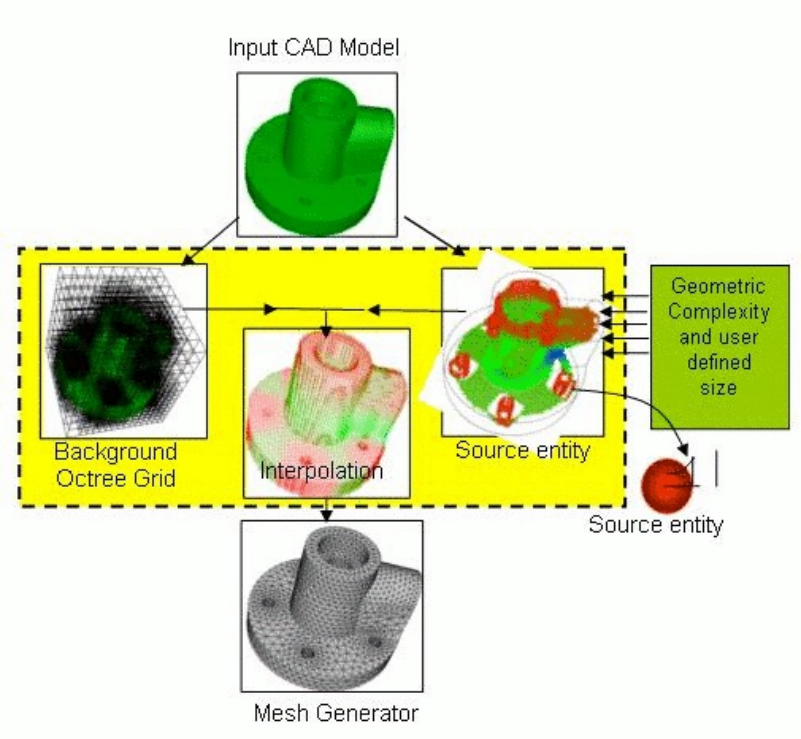


Figure 1: Overview of Computational Framework

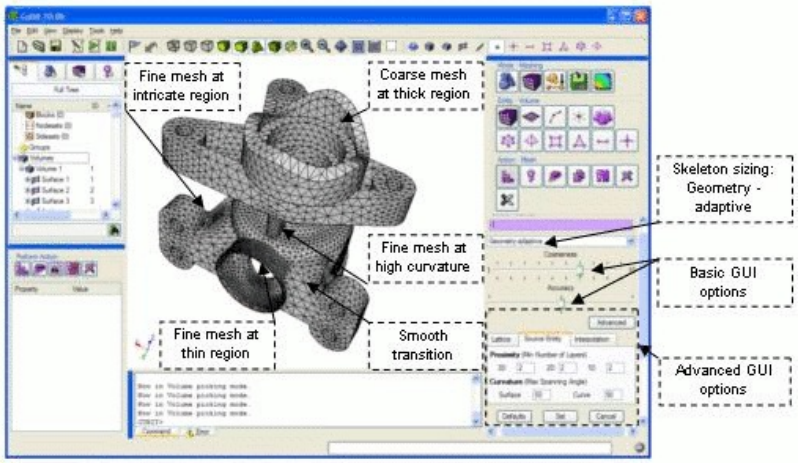


Figure 2: Skeleton Sizing Function example in the GUI

Skeleton Sizing Behaviors

Skeleton sizing can be specified on single or multiple surface(s)/volume(s) at a time from the GUI (Meshing Control Panel) or the command-line. The following describes how specifying sizing on entities can change skeleton sizing's behavior:

Single surfaces/volumes – If skeleton sizing is applied to surfaces/volumes one at a time, each entity's sizing is not influenced by the others. On the command-line, issue a separate command for each entity. In the GUI, specify only one surface or volume before selecting "Apply Size".

Multiple surfaces – If skeleton sizing is applied on multiple surfaces together, then geometric features of a particular surface may affect its neighboring surfaces.

Multiple volumes (assembly sizing) – Skeleton sizing can be applied to assembly models so that geometric features of a volume may influence its neighbors. Volumes should first be imprinted and merged before they are specified together for skeleton sizing.

Command Line Syntax

Skeleton sizing on surfaces:

```
Surface <surface_id_range> Sizing Function
Skeleton
{[scale <1 to 10 = 7>] [time_accuracy_level <1
to 3 = 2>]
[min_depth <3 to 8 = 5>] [max_depth <4 to 9 =
7>] [facet_extract_ang <1 to 30 = 10>]
[min_num_layers_2d < 1 to N = 1>]
[min_num_layers_1d < 1 to N = 1>]
[max_span_ang_surf <5.0 to 75.0 = 45.0
degrees>]
[max_span_ang_curve <5.0 to 75.0 = 45.0
degrees>]
[min_size <float>] [max_size <float>]
[max_gradient <float=1.5>]}
```

Skeleton sizing on volumes:

```
Volume <range> Sizing Function Skeleton
{[scale <1 to 10 = 7>] [time_accuracy_level <1
to 3 = 2>]
[min_depth <3 to 8 = 5>] [max_depth <4 to 9 =
7>] [facet_extract_ang <1 to 30 = 10>]
[min_num_layers_3d < 1 to N = 1>]
```

```
[min_num_layers_2d < 1 to N = 1>]
[min_num_layers_1d < 1 to N = 1>]
[max_span_ang_surf <5.0 to 75.0 = 45.0 degrees>]
[max_span_ang_curve <5.0 to 75.0 = 45.0
degrees>]
[min_size <float>] [max_size <float>]
[max_gradient <float=1.5>]}
```

The options are explained below:

Basic Arguments

- **max_size** (default=auto): The value for max_size is calculated automatically by default. Users can specify any positive real number based on the dimensions of the model to control the max size of the elements. If the skeleton sizing function creates large elements, than this argument can be used to control the maximum element size.
- **min_size**(default=auto): The value for min_size is calculated automatically by default. Users can specify any positive real number based on dimension of the model to specify the minimum size of the elements.
- **max_gradient** (1.0 to 3.0, default 1.5): The transition in element size is controlled using this parameter. Larger values of max_gradient result in fewer elements, but also lead to more abrupt transitions in size and possibly poorer quality elements.

Scaling and Accuracy Arguments:

- **scale** (1 to 10, default 7): The overall size of the elements is controlled by this argument. A coarser mesh can be generated by increasing the value of scale up to 10.0. To get a finer mesh, decrease the value of the scale (minimum value = 1).
- **time_accuracy_level** (1 to 3, default 2): This controls the computational time and accuracy level by adjusting various internal parameters of the skeleton sizing function. Users should try levels in increasing order. Level 1 takes the shortest time to compute the skeleton sizing function and Level 3 takes the longest time to compute the skeleton sizing function. However, Level 1 is less accurate than Level 2 and Level 3.

Advanced Arguments

Lattice Arguments:

The skeleton sizing function is generated and stored on a background octree grid whose cells are subdivided based on the graphics facets of the model. The level of subdivision of the background grid affects how well the sizing function captures the geometric complexity of features. Reasonable defaults have been selected for the following two refinement (subdivision) parameters, but these may be overridden for use with simple (decrease parameters) or more complex (increase parameters) models.

- **min_depth** (default auto): min_depth controls the maximum cell dimension of the background octree grid. The higher the value of min_depth, the smaller the dimension of the maximum-sized cell. Computational time increases with increasing min_depth. By default the min_depth is calculated based on the geometric complexity of the input model and mesh size specified on sub-entities.
- **max_depth** (default auto): max_depth controls the minimum cell dimension. If the object contains very fine features then increasing the value of max_depth is suggested. The maximum depth has been limited to 9. By default the max_depth is calculated based on the geometric complexity of the input model and mesh size

specified on sub-entities.
- **facet_extract_ang** (default 10 degree): facet_extract_ang is used to control the faceted representation of NURBS model. This option gives control of the accuracy of a faceted approximation of the model used to compute the adaptive sizing. For models with high curvature regions, decreasing the tolerance will give a better approximation of the geometry and avoid the creation of random dense meshes. Note that increasing this angle too much can generate invalid facets over curved regions, while decreasing the angle too much can cause signficant slowdowns in sizing calculations.

Source Entity Arguments

- **min_num_layers_3d** (Any value greater than 1, default 1): This parameter ensures that a minimum specified number of layers exist across the thickness of the volume. This parameter could be useful in generating meshes for mold flow simulation.
- **min_num_layers_2d** (Any value greater than 1, default 1): This parameter ensures that a minimum specified number of layers exist across the thickness of a surface.
- **min_num_layers_1d** (Any positive integer value, default 1): This ensures that any curve contains a minimum specified number of intervals.
- **max_span_ang_curve** (Range 5.0 to 75.0, default 45.0): Maximum spanning angle is a parameter that controls the mesh size at curved regions of curves. It is defined as the angle subtended by the normals at the end nodes of the mesh edge in the curved region of a curve. When a finer mesh is needed at curved regions of curves, then max_span_ang_curve should be decreased.
- **max_span_ang_surf** (Range 5.0 to 75.0, default 45.0 deg): Maximum spanning angle is a parameter that controls the mesh size at curved regions of surfaces. It is the angle subtended by the normals at the end nodes of the mesh edge in a curved region of a surface. When a finer mesh is needed at curved regions of surfaces, then max_span_ang_surf should be decreased.

Note: These arguments override the basic arguments. For example, time accuracy level 1 internally sets min_depth = 4 and max_depth = 6, and when min_depth is set to 4 and max_depth is set to 7 in the advanced options (recommended for models with fine features), then advanced options override the basic options. In the command-line, to override the depths set by a time_accuracy_level, specify min_depth and max_depth after it.

Adding User Specified Sizing Sources

Skeleton sizing function gives an option to manually add sizing sources on geometric entiies such as vertices, curves, and surfaces. These sizing sources control the size and scope (region of influence via num_layers) at geometric entities. The below command gives the syntax for adding sizing sources. Please note that the below command for adding sizing sources should be called after issuing the above given skeleton sizing command. First, the skeleton sizing command automatically generates sizing sources based on the geometric factors such as proximity, surface curvature, curve length, etc. Issuing the below command creates sizing sources in addition to the automatically generated sizing sources. Finally, when the meshing command is called, the mesh sizing function is calculated using all the sizing sources.

```
Volume <vol_id_range> Sizing Function Skeleton
add size_source {vertex|curve|surface}
<id_range> size <double> num_layers <int>
```

Skeleton with Other Sizing Controls

Skeleton sizing function produces a smooth sizing function when called with other sizing controls available in Cubit. Skeleton sizing function behaves as SOFT firmness level. Skeleton sizing function always respects interval count specified on the curves. Skeleton sizing function respects interval size on curves and surfaces only if it is specified after calling the skeleton sizing function.

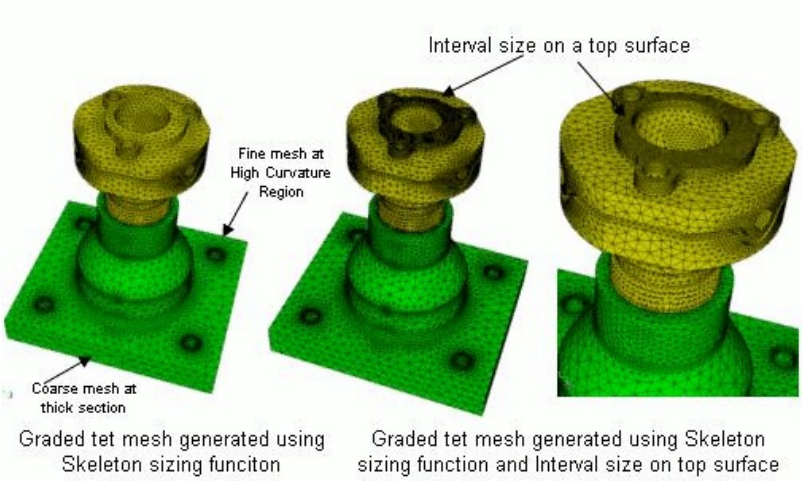


Figure 3: Skeleton sizing function with other sizing controls

Limitations

- **Currently, the skeleton sizing function is primarily intended for use with ACIS models. Skeleton sizing may be used on facet-based models (STL, facet, and MBG format) models, but results are not guaranteed. Sizing function generation with other geometry engines in Cubit is not guaranteed or supported in Cubit 10.1.**
- **The skeleton sizing function has mainly been tested with trimesh and tetmesh schemes. In general, structured or semi-structured meshing schemes do not have enough flexibility to utilize the skeleton sizing function. It is recommended that the skeleton sizing be used only with unstructured meshing schemes. However, if using skeleton sizing in conjunction with the pave scheme for surfaces, decreasing the max_gradient and scale arguments is suggested.**
- **For sizing function generation of assemblies in Cubit 10.1, at least time_accuracy_level 2 is generally recommended. This helps ensure that the geometric complexity of small features is captured. For example, "volume all sizing function skeleton time_accuracy_level 2"**

Interval Sizing Function

The **Interval** sizing function is similar to the Linear function, but bases edge length at a location on the squared lengths of edges bounding the surface weighted by their inverse distance from the current location. An example is shown below.

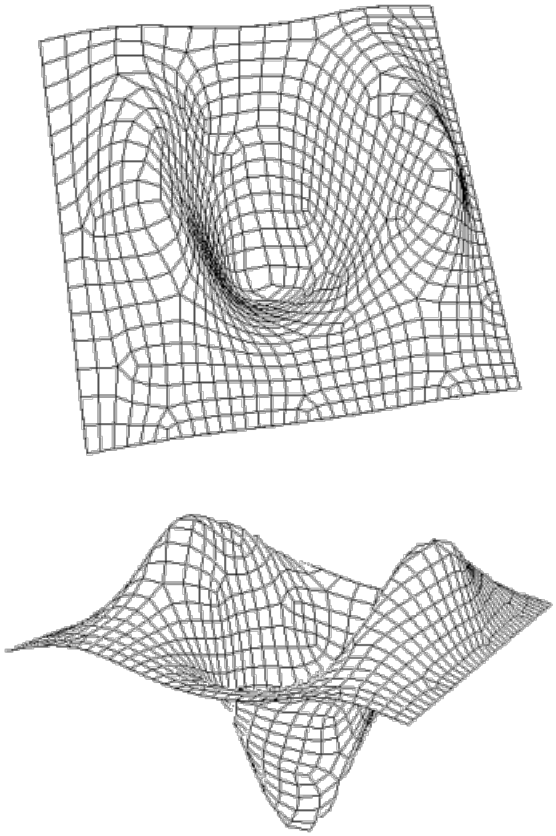


Figure 1. NURB mesh with interval sizing function, 34 by 16 density

Inverse Sizing Function

The **Inverse** sizing function is also similar to the Linear function, but this method bases edge length at a location on the inverse lengths of edges bounding the surface weighted by their inverse distance from the current location (see Figure 1). The difference between the three linear sizing functions (Linear, Interval, Inverse) is sometimes subtle, but is driven by the geometry being meshed since the influence of these functions is strongly controlled by the number, positioning, and mesh density of the bounding curves relative to the interior surface area.

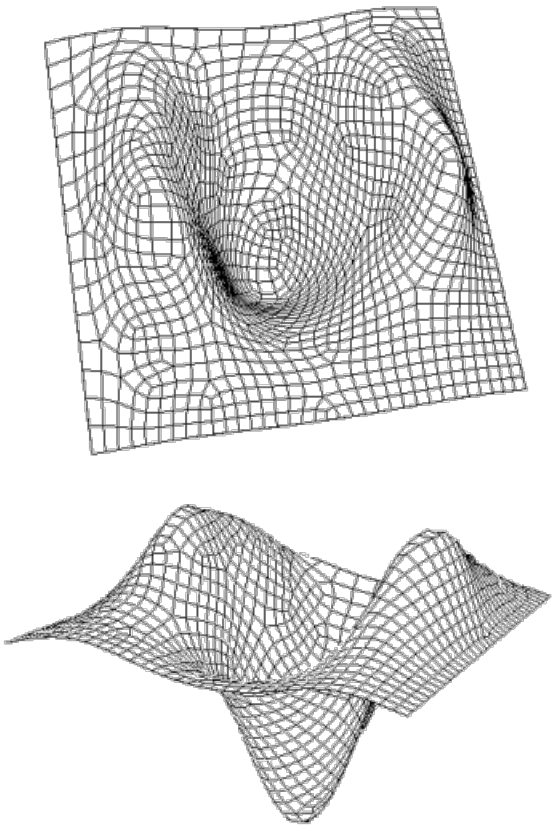


Figure 1. NURB mesh with inverse sizing function, 34 by 16 density

Linear Sizing Function

The **Linear** class of sizing functions determines element size based on a weighted average of edge lengths for mesh edges bounding the surface being meshed. There are several variants of this class of sizing function. The Linear function bases edge length at a location on the lengths of edges bounding the surface weighted by their inverse distance from the current location. The result of this weighting is a more gradual change in mesh density during a transition between dense and coarse mesh. Figure 1 shows the same NURB surface mesh but with intervals of 34 on two curves and intervals of 16 on the remaining two bounding curves and no sizing function. It can be observed that the mesh progresses more rapidly inward from the coarser meshed curves, which locates the transition region much closer to the finer meshed curves. To combat this, the Linear function weights the sizing of new elements such that these transitions occur slower. Figure 2 displays two views of the same NURB geometry with the same bounding curve mesh density using the linear sizing function.

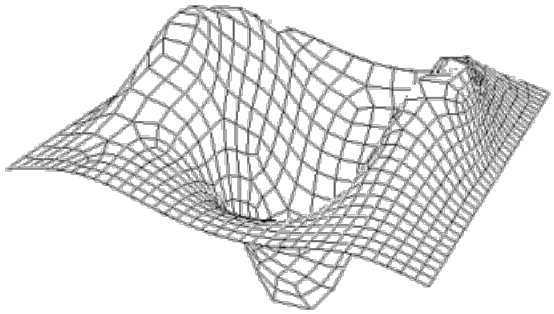


Figure 1. NURB mesh with no sizing function, 34 by 16 density

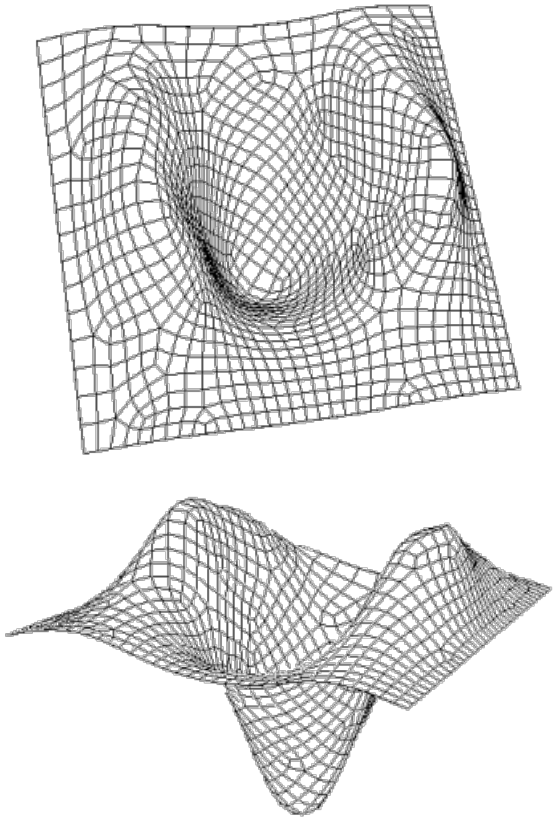


Figure 2. NURB mesh with linear sizing function, 34 by 16 density

Hyperbolic Tan Sizing Function

A new mesh sizing capability developed to improve mesh quality and control. This function can be specified on curves to ensure that the mesh transitions smoothly and accurately, enhancing the overall quality of the simulation. The command syntax for specifying this mesh sizing function is as follows:

```
Curve <curve_id_range> Sizing Function Tanh
[start_vertex <id>]
[start_size <value>]
[end_size <value>]
[stretch_para <value>]
[interval <int>]
[Type {"TANH_HALF"|"TANH_FULL"}]
```

- `curve <curve_id_range>`: Specifies the range of curve IDs to which the sizing function will be applied.
- `start_vertex <id>` (Optional): Specifies the starting vertex ID.
- `start_size <value>` (Optional): Specifies the starting size of the mesh.
- `end_size <value>` (Optional): Specifies the ending size of the mesh.
- `stretch_para <value>` (Optional): Specifies the stretch parameter for the sizing function.
- `interval <int>` (Optional): Specifies the number of intervals for the mesh.
- `Type {"TANH_HALF"|"TANH_FULL"}` (Optional): Specifies the type of hyperbolic tangent function to be used. TANH_HALF is the default type.

In the below Figure 1, the TANH_HALF function shows that the mesh size increases linearly at the start_vertex (Para t = 0.0) and mesh size gradation decreases as the parameter, t, goes towards 1.0. Figure 2 shows the mesh generated by placing nodes on the curve using TANH_HALF sizing function.

In the below Figure 1, TANH_FULL shows that the mesh size increases gradually at start_vertex, which might be desirable in capturing boundary layers. Then mesh size increases linearly and finally the mesh size gradation flattens resulting in uniform mesh size in the far field.

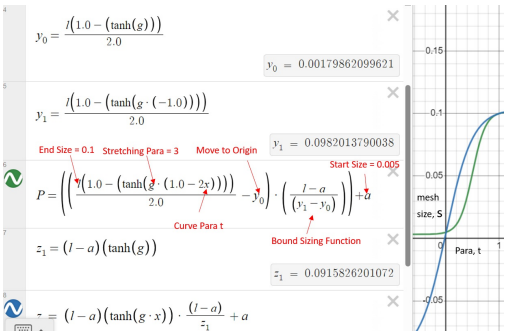


Figure 1: Hyperbolic Tanh half and full symmetry function

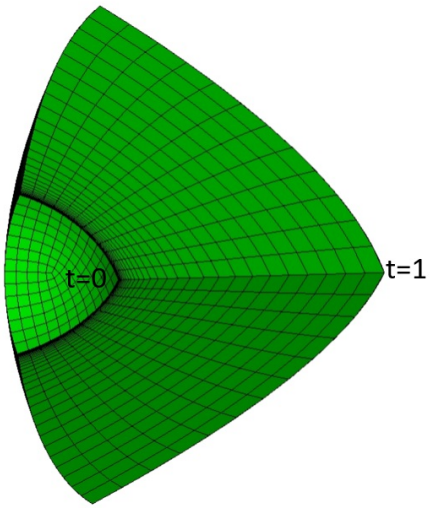


Figure 2: Hyperbolic Tanh half sizing function along the radial curves to create boundary layers

The below Cubit® commands demonstrate the different options of the Hyperbolic Tan Sizing Function. A simple geometry, such as a brick, is used for the demo.

Option 1: Sizing function type TANH_HALF and TANH_FULL

The mesh generated using the TANH_HALF type shows that the mesh

edges grow faster at the start vertex. Therefore, TANH_HALF results in fewer intervals along the curve compared to TANH_FULL

```
brick x 10 y 35 z 50
curve 3 sizing func tanh start_vertex 4 start_size 0.1 end_size 4 stretch_para 1 type "TANH_HALF"
mesh curve 3
brick x 10 y 35 z 50
curve 3 sizing func tanh start_vertex 4 start_size 0.1 end_size 4 stretch_para 1 type "TANH_FULL"
mesh curve 3
```

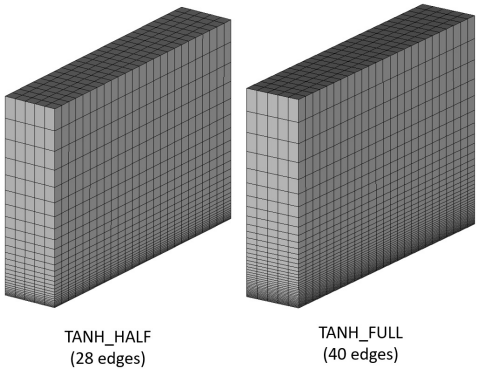


Figure 3: TANH_HALF produces less intervals than TANH_FULL

Option 2: Stretching parameter stretch_para

This example demonstrates the basic application of the Hyperbolic Tan Sizing Function with a single stretch parameter and type "TANH_HALF".

This example illustrates the flexibility of the Hyperbolic Tan Sizing Function in handling multiple stretch parameters. As the stretch_para increases, the mesh size grows faster and will result in fewer mesh edges on the curve.

```
brick x 10 y 35 z 50
curve 3 sizing func tanh start_vertex 4 start_size 0.1 end_size 4 stretch_para 1 type "TANH_HALF"
mesh curve 3
brick x 10 y 35 z 50
curve 3 sizing func tanh start_vertex 4 start_size 0.1 end_size 4 stretch_para 5 type "TANH_HALF"
mesh curve 3
```

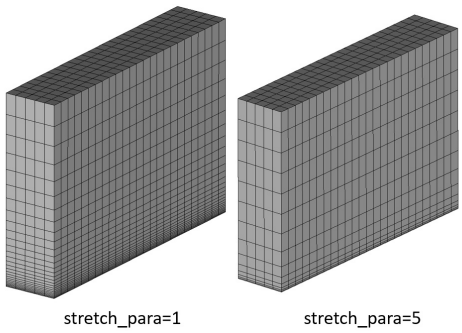


Figure 4: Mesh edge grown rate is controlled by stretch_para

Option 3: start_size, end_size, and interval

The Hyperbolic Tanh Sizing Function allows specification of start_size and end_size; however, when interval is also specified the priority will be given to interval than respecting the start and end sizes. When interval is set to 0, only the start and end size will be respected and tanh sizing function is free to choose any number of interval.

```
brick x 10 y 35 z 50
curve 3 sizing func tanh start_vertex 4 start_size 0.1 end_size 4 stretch_para 5 interval 0 type "TANH_HALF"
mesh curve 3
brick x 10 y 35 z 50
curve 3 sizing func tanh start_vertex 4 start_size 0.1 end_size 4 stretch_para 5 interval 30 type "TANH_HALF"
mesh curve 3
```

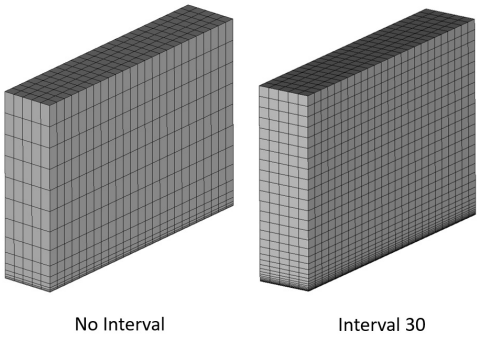


Figure 4: Interval overrides the start_size and end_size

Python Inputs for UGRID:

UGRID is a Python script developed at Sandia for automething mesh generation for aero simulations. The `ugrid_inputs.py` Python script specifies the input arguments for the Hyperbolic Tan Sizing Function. UGRID then calls Cubit's tanh command using the arguments specified in `ugrid_inputs.py`. Below are examples of `ugrid_inputs.py` demonstrating sizing control in both radial and axial directions:

Example 1: Sizing control using TANH_HALF

```
# ugrid_inputs.py
curve_id_range = "3"
start_vertex = 4
start_size = 0.1
end_size = 4
stretch_para = 1
tanh_type = "TANH_HALF"
# Command to be executed by UGRID
command = f"Curve {curve_id_range} Sizing Function Tanh start_vertex {start_vertex} start_size {start_size} end_size {end_size} stretch_para {stretch_para} type {tanh_type}"
```

Example 2: Specified interval overrides the default Tanh sizing function

```
# ugrid_inputs.py
curve_id_range = "3"
start_vertex = 4
start_size = 0.1
end_size = 4
stretch_para = 5
interval = "30"
tanh_type = "TANH_HALF"
# Command to be executed by UGRID
command = f"Curve {curve_id_range} Sizing Function Tanh start_vertex {start_vertex} start_size {start_size} end_size {end_size} stretch_para {stretch_para} interval {interval
```

Free Meshes

A free mesh is a mesh that is not associated with any underlying geometric entities. A free mesh contains only mesh elements (hexahedra, triangles, edges, nodes, etc), and not volumes, surfaces, etc. Since there is no underlying geometry, operations on free meshes are limited. The following operations can be performed on free meshes in some capacity:

- Creating a free mesh
- Creating mesh-based geometry to fit a free mesh
- Mesh merging
- Mesh transformations
- Mesh smoothing
- Mesh quality operations
- Mesh refinement
- Cleaning up a free mesh
- Assigning boundary conditions
- Skinning a free mesh
- Mesh deletion
- Bottom-up element creation
- Exporting a free mesh

Creating a free mesh

A free mesh can be created in three ways.

1. Importing a mesh into Cubit using the **Import Mesh [No_Geom]** command. This option is discussed in detail in Importing Exodus II Files.
2. Disassociating an existing mesh from its geometry
3. Creating a mesh with the geometry-tolerant mesh scheme

Disassociating a mesh from its geometry

The command to disassociate a mesh from existing geometry is:

```
Disassociate Mesh [From]
{Volume|Surface|Curve|Vertex} <id_range>
```

For example:

```
brick x 10
mesh volume all
disassociate mesh from volume 1
delete volume 1
```

When a mesh is disassociated from its geometry, a group called 'disassociate elements' is created to contain the free mesh.

Creating Mesh-Based Geometry to fit a Free Mesh

It is possible to create underlying mesh-based geometry to own a free mesh. It is similar in functionality to the Import Mesh Geometry command, but it does not require the extra import/export step. For example, a user would be able to read in a free mesh, fix any mesh problems, and then create the mesh-based geometry without having to

write the mesh to a file first. The command syntax is:

```
Create Mesh Geometry {Hex|Tet|Face|Tri|Block}
<range> [no_nodeset] [no_sideset] [exclude
block <range>] [Feature_Angle <angle=135>]
[Acis] [Keep]
```

The command also applies to any subset of the mesh. For example, you can create mesh geometry for a group of hexes or element blocks.

If the **keep** option is specified, the mesh will be duplicated so you will have two copies of the mesh: The original mesh and the new mesh that is owned by the new MBG geometry. If the keep option is not specified, the existing mesh will be reused, and duplicate elements will not be created. Elements will now be owned by the new MBG geometry. The command will check for mesh ownership and will issue a warning. Use the keep option if the mesh is already owned. The keep option is not specified by default.

By default, genesis entities will be used as criteria for building the new MBG geometry. The **no_nodeset** and **no_sideset** options can be specified to prevent this. Any genesis entities defined on the free mesh are transferred to the new MBG geometry. When the **keep** option is used copies of genesis entities are made on the new MBG geometry.

The **exclude block** option excludes specified blocks from feature angle detection. Any volumes created from these excluded blocks will have only one surface.

The **Acis** option will attempt to create ACIS geometry from the mesh. This option is an alpha feature and can only be used if developer commands have been turned on. For more detail see: Acis Geometry From Mesh

Merging a free mesh

To merge two free meshes, the equivalence command may be used. The command syntax is:

```
Equivalence Node <range> [Tolerance <value>]
[Preview]
```

All nodes in the given range that are within the specified tolerance will be merged. The merged and unchanged (unmerged) nodes are put into groups. The maximum distance between merged nodes, and the minimum distance between unmerged nodes, are listed for the user because if these values are close together it can indicate a problem with the Tolerance. Nodes within tolerance and part of the same element will not be merged. This prevents collapsing elements into degenerate forms. With the Preview option, the nodes aren't actually merged, but the ones that would have merged are drawn and grouped. For example:

```
br x 10
volume 1 copy move x 10
mesh volume all
disassociate mesh from volume 1 2
delete volume 1 2
equivalence node all tolerance 0.05
## merges all nodes that are within 0.05 of each
other
```

Free Mesh Transformation Operations

Mesh transformations for free meshes are achieved through the use of the group transformation commands, given in Basic Group Operations. All members of a free mesh are automatically assigned to a group. These groups can then be modified using group operations. The following command sequence illustrates how transformations might be applied to a free mesh.

```
brick x 10
mesh volume 1
disassociate mesh from volume 1
delete volume 1
group disassociated_elements move x 10
group disassociated_elements rotate 15 about x
group disassociated_elements scale 0.25
group disassociated_elements reflect 1 1 0
group 'node_group' add node 1 to 121
group node_group move z 5
##The moved nodes do not also move the attached
geometry, as one might expect.
```

If a group is composed of mesh entities, these commands will only operate on the nodes in the group. All nodes of the group will be moved, scaled, rotated, or reflected as specified. If there are no nodes in the group, Cubit will return an error. Including all nodes in the group will transform the whole model. Including only a subset of nodes will transform those nodes and their enclosed elements, but it will not transform the whole mesh.

Disassociated mesh elements cannot be copied using the Group copy commands. To create a copy they must be exported and reimported. Alternatively, they can be associated with mesh-based geometry, and then copied using the typical copy commands.

Extruding Mesh Elements

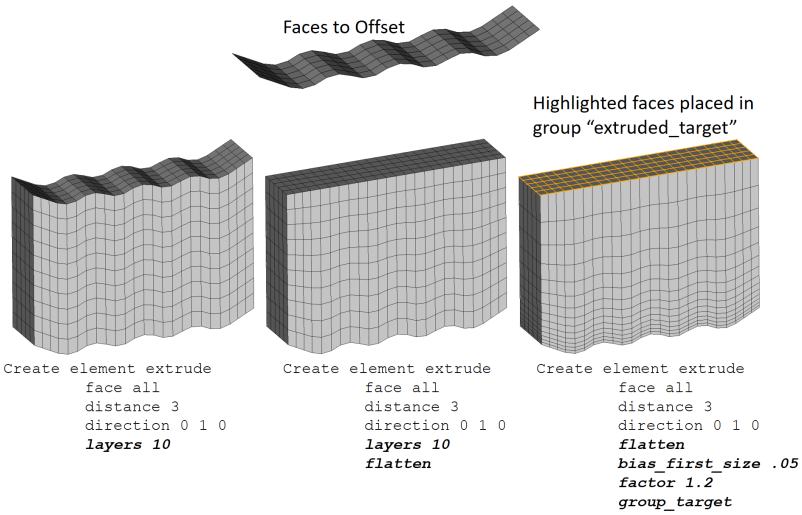
Mesh elements can be extruded to create new elements from existing nodes, edges, faces or triangles. There are two forms of the extrude command as follows:

```
 Create Element Extrude {Node|Edge|Face|Tri}
<element_list> Direction <options> [Distance
<value>] {Layers <num_layers |
{bias_first_size <value> factor <value>}}
[Twist <angle> Axis <axis_options>] [flatten]
[group_target]
```

```
 Create Element Extrude {Node|Edge|Face|Tri}
<element_list> Along Curve <curve_list>
[Layers <num_layers]
```

In the first form of the command, a direction and **distance** are specified to define the extrusion. To define node spacing along the extrusion, the command takes either the **layers** or **bias_first_size** and **factor** parameters, but not both. Specifying a value for the **layers** option determines how many evenly sized elements will be created in the given distance. The **bias_first_size** and **factor** parameters define a bias that will be used along the extrusion distance. When specifying entities to be extruded which are either non-planar, or not orthogonal to the extrusion direction, the **flatten** parameter results in the extrusion terminating on a single plane orthogonal to the extrusion direction. The optional **group_target** parameter places all of the faces, tris, edges, or nodes at the end of the extrusion into a group named "extruded_target" which can be conveniently used to define a subsequent extrusion with a different set of extrusion parameters. **Twist** can also be specified and requires an angle of twist and a twist axis.

The figure below illustrates the new **flatten**, **bias_first_size**, **factor**, and **group_target** keywords. At the top is a set of non-planar faces to offset. On the bottom-left, each node on all of the faces are extruded the same amout resulting in a target of the extrusion with identical curvatuve to the input faces. In the bottom-middle, the **flatten** keyword is used, resulting in the extrusion terminating on a single plane. In the bottom-right, the **bias_first_size**, **factor**, and **group_target** keywords are used, resulting in a biased extrusion, and the creation of a group named "extruded_target", which contains the faces on the end of the extrusion.



In the second form, a curve is specified, along with the input entities will be extruded. Extruding along a curve supports the **layers** parameter, but does not currently support the **distance**, **bias_first_size**, **factor**, **flatten**, **twist** or **group_target** parameters.

```
#Extrude a face in a given direction:
create node location 0 0 0
create node location 1 0 0
create node location 1 1 0
create node location 0 1 0
create face node 1 to 4
create element extrude face 1 direction 0 0 1
distance 3 layers 3
create element extrude face 1 direction 0 0 1
distance 3 layers 3 twist 90 axis direction 0 0 1
origin 0 0 0

#Sweep face along curve
create node location 0 0 0
create node location 1 0 0
create node location 1 1 0
create node location 0 1 0
create face node 1 to 4
create vertex location position 0 0 0
create vertex location position 0 .2 1
create vertex location position 0 1 2
create vertex location position 0 3 2
create vertex location position 0 4 1
create vertex location position 0 5 0
create curve spline vertex 1 2 3 4 5
create element extrude face 1 layers 5 along curve 1
```

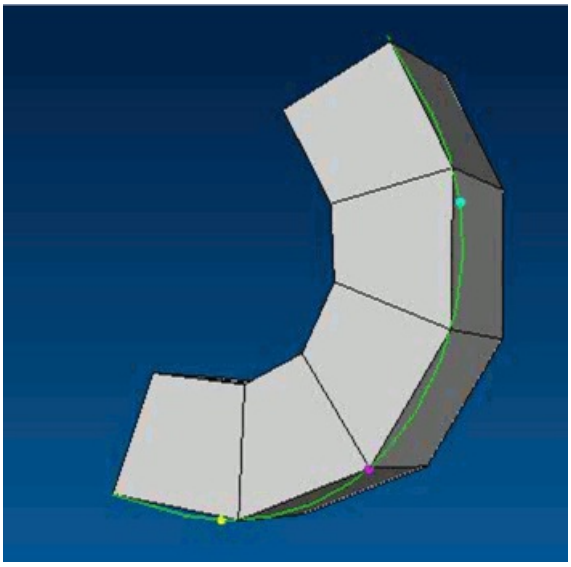


Figure 1. Extruding mesh elements along a spline

Offsetting Mesh Elements

Faces and triangle elements can be used to create hexahedral and wedge elements from an offset command. The default offest direction is normal to the selected face. The **Oppposite_normal** option will use the reverse direction. The **layers** parameter determines how many elements will be created in the given direction.

```
Create Element Offset {Face|Tri}
<element_list> [Normal_to|Opposite_normal]
{Distance <value>] [Layers <num_layers>]
```

```
#Create wedge and hex elements from face and tri
elements via offset
create node location 0 0 0
create node location 1 0 0
create node location 1 1 0
create node location 0 1 0
create node location 2 0 1
create node location 2 1 1
create node location 1 2 0
create face node 1 to 4
create face node 3 2 5 6
create tri node 7 4 3
create tri node 7 3 6
create element offset face all tri all distance 3
layers 3 opposite_normal
```

Revolving Mesh Elements

Elements can be created by revolving an existing element around a given axis. The **Attempt_fix** parameter will try to fix any poorly formed hex elements by collapsing them into wedge elements. **Angle** determines the amount of rotation around the axis. The **Layers** option determines how many elements will be created in the given rotation. The **quadratic** option will result in the creation of quadratic elements. For example, it can result in HEX20 or TET10 elements being created. To further specify the element type, for the created elements, one may put them into a block.

```
Create Element Revolve {Edge|Face|Tri}
<element_list> Axis <axis_options> Angle
<angle> [Layers <num_layers>] [Attempt_fix]
[quadratic]
```

```
#Revolve 2 faces around the Y-axis and collapse
```

```
inner hexes to wedges
create node location 0 0 0
create node location 1 0 0
create node location 1 1 0
create node location 0 1 0
create node location 2 0 0
create node location 2 1 0
create face node 1 2 3 4
create face node 2 5 6 3
create element revolve face 1 2 axis direction 0 1 0
angle 180 layers 4 attempt_fix
block 1 hex all
block 2 wedge all
block 1 element type hex8
block 2 element type wedge6
```

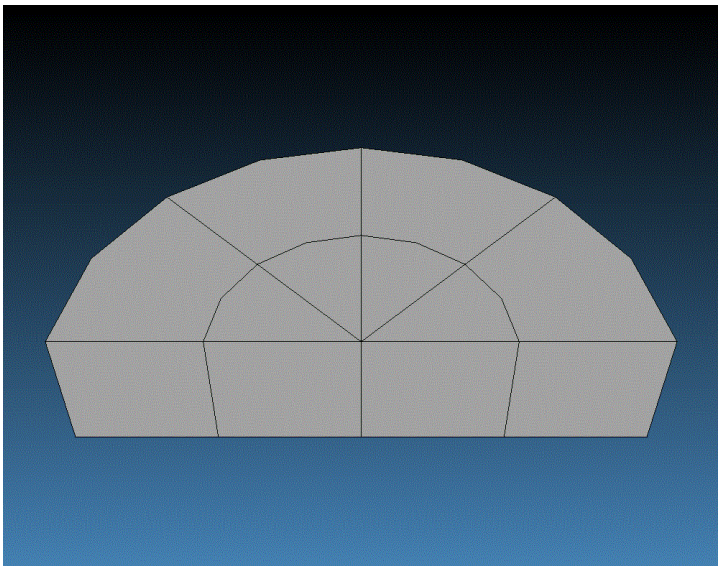


Figure 2. Revolving free mesh elements to create hex and wedge elements

Smoothing a free mesh

Interior nodes can be smoothed using commands such as *smooth hex all*, or *smooth tet all in block 100*. These commands will smooth only the interior node on the elements used in the command. The nodes on the boundary will remain unchanged. To smooth nodes on a boundary, the target smoothing option can be used. Targeted smoothing allows the user to smooth a group of mesh elements to a surface or curve that is not their owner. Targeted smoothing is discussed under Mesh Smoothing. The following sequence of commands illustrate the capability of smoothing a free mesh to a target surface.

```
sphere rad 25
webcut vol 1 plane xplane offset 18
delete vol 2
webcut volume 1 plane yplane offset 8
webcut volume 1 plane yplane offset -8
delete vol 1 3
surf 16 copy
delete vol 4
surf 18 scheme pave
surf 18 size 2
mesh surf 18
disassociate mesh surf 18 ##Mesh and geometry
overlap
```

```
refine face 1 radius 3
set developer on ## Smoothing free mesh is a
developer command
smooth face all scheme laplacian
##Smoothed mesh is away from surface
smooth face all scheme laplacian target surface 18
##Smoothed mesh is aligned with surface
```

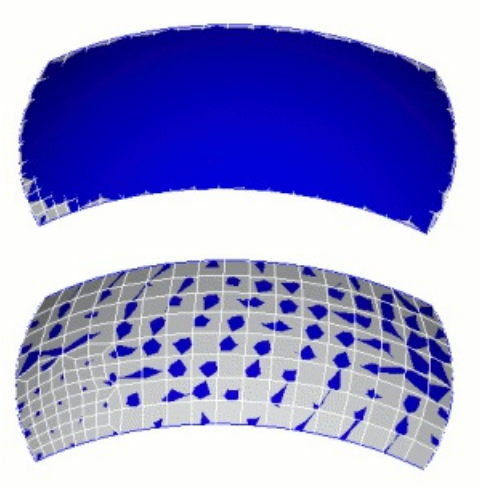


Figure 3. Smoothing without a target (above) and smoothing to a target surface (below).

Mesh quality on a free mesh

The mesh quality checks for a free mesh are the same as for other geometry-based meshes. The difference is in how you specify elements in the command. Instead of specifying volumes or surfaces you would specify groups of hexes, faces, tris, or tets. Examples are given below:

```
quality hex all
quality face all scaled jacobian
quality tet 1 to 100 draw mesh
```

Mesh refinement on a free mesh

Refinement for a free mesh is limited to refinement of mesh elements. Refinement may be accomplished by specifying groups of mesh elements which to refine using the regular refinement options. For boundary elements, the refinement scheme will use averaging methods to determine node placement, in the absence of a boundary geometry to define node placement.

Cleaning up a free mesh

A free tet mesh may be cleaned up using the Cleanup Tet command. For example

```
cleanup tet all
#cleans up all tets
cleanup tet 1 to 1000
#cleans up all tets in the range [1,1000]
```

It is best to specify contiguous sets of elements for this command.

Assigning boundary conditions

Assigning boundary conditions on free meshes can be accomplished by explicitly specifying mesh elements, by creating a sideset or block from the skin of a group of elements, or by creating groups based on feature angle using the seed method. Once the group is created it is easy to assign it to a nodeset or sideset.

Cubit will respect block, nodeset, and sideset data that is associated with an imported free mesh, or disassociated mesh. The following command sequence illustrates how the group seed operation could be used for assigning boundary conditions on free meshes.

```
##Creating blocks, nodesets and sidesets on free
meshes
cylinder radius 3 z 12
volume 1 size 0.5
mesh volume 1
disassociate mesh from volume 1
delete volume 1
group 'mygroup1' add seed face 752 feature_angle 45
##Groups all faces on the cylindrical surface
group 'mygroup2' add seed face 752 feature_angle 45
divergence
##Groups only faces within 45 degrees of seed face
sideset 1 group mygroup1
sideset 2 group mygroup2
block 1 hex all
draw sideset 1
draw sideset 2
draw block 1
```

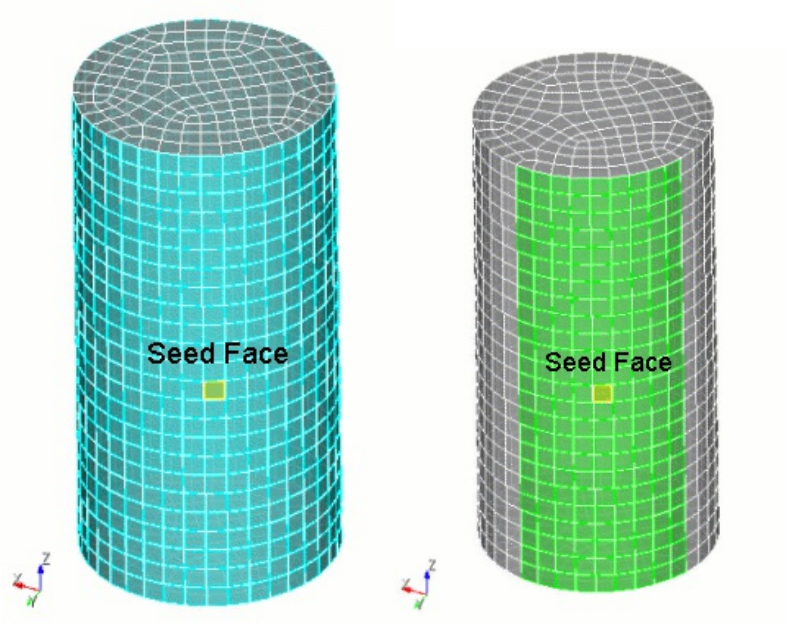


Figure 4. Grouping faces on free meshes using the seed method. The feature angle method is used on the left with a feature angle of 45 degrees. On the right is the result if using the divergence method.

Even though boundary conditions can be defined directly only on geometry entities, these geometry-based BCs will be maintained on the free mesh following the disassociate command. The following command line sequence illustrates this capability.

```
##Respecting blocks, nodesets and sidesets in mesh
elements after disassociation
```

```
brick x 10
mesh vol 1
sideset 1 surface 1
nodeset 1 curve 1
block 1 volume 1
disassociate mesh from volume 1
draw sideset 1
draw nodeset 1
draw block 1
```

Skinning a free mesh

The skin command takes a list of mesh elements and returns the triangles and faces on the boundary of that group. The group of elements returned from the command can be assigned to either a group, sideset, or block. Free meshes can be skinned by specifying either a list of hexahedra, a list of tetrahedra, or a list of blocks.

Deleting free mesh elements

Typically meshes are deleted by specifying owning geometry. For free meshes, the meshes cannot be deleted in this fashion. Instead, the mesh may be deleted using the **Delete mesh** command. The syntax is:

```
Delete Mesh
```

This command will delete all mesh entities in the entire model. To specify groups of elements for deletion, you can use the individual deletion commands. The command to delete a group of free mesh elements is:

```
Delete {Node|Hex|Tet|Face|Tri} <id_range>
[No_propagate]
```

When deleting elements, the default behavior will be that the child mesh entities will be deleted when they become orphaned. For example, when a hex is deleted, if its faces, edges and vertices are no longer used by adjacent hex elements, then they will also be deleted. The **no_propagate** option will leave any child mesh entities regardless if they become orphaned.

Bottom-up element creation

Bottom-up mesh element creation methods are available for free meshes. The difference between element creation methods for free meshes versus associated meshes is that the free meshes commands do not have a command option to associate the elements with an owning body. Otherwise the commands are identical to mesh element creation commands for associated meshes. The command syntax for free meshes is:

```
Create Node <x> <y> <z>

Create {Hex|Tet|Tri|Face|Edge} Node <id_range>
```

Exporting free meshes

Free meshes can be exported as ExodusII files. All elements belonging to any block are exported. Any elements not belonging to a block will not be

exported (i.e. Cubit will not assign default blocks).

Mesh Deletion

Meshing a complex model often involves iteration between setting mesh parameters, meshing, and checking mesh quality. This often requires removing mesh, for only an entity or for an entity and all its lower order geometry, or sometimes for the entire model.

The command to remove all existing mesh entities from the model is:

```
Delete Mesh
```

The command for deleting mesh on a specific entity is:

```
Delete Mesh {geom_list} [Propagate]
```

These commands automatically cause deletion of mesh on higher dimensional entities owning the target geometry.

If the Propagate keyword is used, mesh on lower order entities is deleted as well, but only if that mesh is not used by another higher order entity. For example, if two surfaces (surfaces 1 and 2) sharing a single curve are meshed, and the command "delete mesh surface 1 propagate" is entered, the mesh on surface 1 is deleted, as well as the mesh on all the curves bounding surface 1 except the curve shared by surface 2. In some cases, the capability to delete individual mesh faces on a surface is needed. Deleting a mesh face involves closing a face by merging two mesh nodes indicated in the input. The syntax for this command is:

```
Delete Face <face_id> Node <node_id> [Node
<diagonal_node_id>]
```

This command is provided primarily for developers' use, but also provides the user fine control over surface meshes. At the present time, this command works only with faces appearing on geometric surfaces and should be used before any hex meshing is performed on any volume containing the face to be deleted.

Automatic Mesh Deletion

Cubit will automatically delete the mesh from a geometry that is about to be modified by a geometry modification command. To change this behavior, so that Cubit will issue an error instead of automatically deleting the mesh, use the following command.

```
Set Mesh Autodelete [ON|Off]
```

Mesh Validity

After a mesh is generated, it is checked to ensure that the mesh has valid connectivity. If an invalid mesh is formed, then CUBIT automatically deletes it. This default behavior can be changed with the following command:

```
Set Keep Invalid Mesh [on|off]
```

The current behavior can be viewed with the following command:

```
List Keep Invalid Mesh
```

The Jacobian quality metric is also computed automatically to check quality after a mesh is generated. If the quality is poor, a warning is printed to the terminal.

Skinning a Mesh

The Skin command takes a range of hexahedra, tetrahedra, blocks, or volumes and generates a collection of triangles or quadrilaterals on the exterior of the volumetric elements. This is the skin mesh.

```
Skin {Block|Volume} <range> [Individual]
[Nomake]

Skin
{Element|Hex|Tet|Wedge|Pyramid|Face|Tri|Block|Volume}
<range> [Nomake]

Skin
{Element|Hex|Tet|Wedge|Pyramid|Face|Tri|Block|Volume}
<range> [Make {Block|Sideset [<id>] |Group
[<name>|<id>]}

Skin
{Element|Hex|Tet|Wedge|Pyramid|Face|Tri|Block|Volume}
<range> {Add|Replace} {Block|Sideset [<id>]
|Group [<name>|<id>]}
```

The **Individual** keyword tells Cubit to skin Blocks or Volumes, one by one independently of each other, even if they share merged surfaces.

The **Nomake** keyword tells Cubit to not create any kind of grouping of the mesh faces resulting from the skinning operation.

If the **Make** option and its arguments are present, then the specified object (block, sideset or group) receives the skin mesh. The command fails if an object with the optional identifier already exists. If the object identifier is omitted, the identifier is set to the next object of that type. The skin mesh is stored in the next available sideset if the Make option is missing.

Another command form has two options, **Add** and **Replace**. Each option has a required, associated identifier. If the identifier is missing or invalid, the command fails. The Add option appends the skin mesh to the object. The Replace option removes any existing mesh from the object before adding the skin mesh.

The skin mesh will respect the merged volumes. If two adjacent volumes are merged, the skin mesh will not include the merged surface. If the volumes are not merged, each volume will generate a separate skin surface. If volumes are not merged, they are treated separately. The skin command will also respect any number of interior voids. All surface elements will be oriented forward with respect to the originating volumes.

The primary use for the skin command is to generate surface meshes of quads or tris for sidesets and remeshing.

For **Face** and **Tri** elements (2d elements), the skin is a set of edges (1d elements.) The skin for 3d elements is a set of 2d elements.

Lite Meshes

Cubit has the ability to represent mesh using either a lightweight or heavy representation. The lightweight representation option is new for Cubit 15.3, and can be referred to as lite. The heavy representation is useful for supporting all the various mesh manipulation operations available in Cubit. While still under development, the lite representation option is intended to be a quick way to display larger meshes while supporting a smaller subset of mesh manipulation operations.

The following are supported operations with lite mesh:

- Creating a lite mesh
- Graphics
- Information
- Modifications to lite mesh
- Exporting a lite mesh
- Equivalencing of nodes

The following items are not yet supported:

- Information and listing of individual hexes, tets, wedges, pyramids, faces, tris and edges
- Modifying the content of blocks, sidesets and nodesets
- Modifying the element type of a block
- Renumbering blocks, sidesets and nodesets
- Topology checks on blocks
- Cleanup on blocks
- Skinning on blocks
- Quality calculations on elements
- Moving blocks or nodesets
- Deleting individual blocks, nodesets and sidesets
- Creating rebar elements
- Smoothing elements
- Refining the mesh
- Merging nodes

Creating a lite mesh

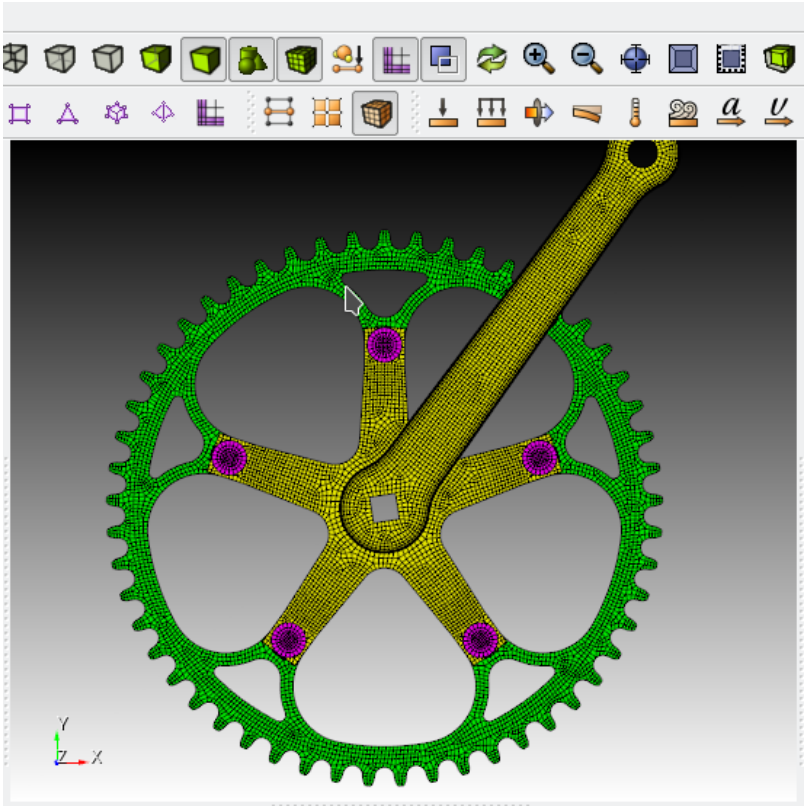
To use the lite mesh representation, one may import a mesh file using the 'lite' option. There is not currently another way to create lite mesh other than importing from a file. The command to import a lite mesh is:

```
Import mesh "<filename>" lite
```

Additional options for lite import can be found under the Import Mesh Lite command.

Graphics

Generally, the graphical features for lite meshes is supported at the same level as for heavy meshes, including the ability to draw, pick, select, highlight, zoom among other operations. The coloring of the mesh is based on blocks, and may be adjusted by the user. Toggling visibility of all sidesets and nodesets can be done by clicking the Display Boundary Conditions toolbar button or with the **`bc visibility {on|off}`** command. Toggling visibilty of all blocks can be done by clicking the Display Mesh toolbar button or with the **`mesh visibility {on|off}`** command.



The draw, zoom and select commands work on blocks, sidesets and nodesets. Also, selecting those genesis entities in the graphics window will result in them being highlighted in both the graphics window and in the tree. Selecting of nodes and elements has not yet been implemented for meshes imported in lite mode.

Information

There are several ways to view information about the lite mesh. The tree and the property page can show information about the blocks, sidesets and nodesets. Also, the **`list`** command can print information about individual blocks, sidesets, and nodesets. The **`list element`** command will print out the ID space used by elements. The **`list node`** command will show the ID space used by nodes. Listing of individual elements and nodes is not yet supported.

Modification to lite mesh

Some modifications to genesis entities are supported. Blocks, sidesets and nodesets may have names assigned to them. Blocks may have their attributes modified, and materials may be assigned to blocks. Not supported is the ability to modify the contents of blocks, nodesets and sidesets.

Exporting a lite mesh

Exporting a lightweight mesh to an Exodus file is supported. This includes writing out blocks, nodesets, sidesets, element ids, node ids, etc... Not all Exodus data is read in, and if there is some data not recognized by Cubit, it will not be exported. Field data is an example of

Exodus data not recognized by Cubit, nor exported. Importing multiple Exodus files and exporting a single file is supported. Importing a single Exodus file and exporting a portion of it is supported. Distribution factors are preserved when reading/writing in lite mode.

Equivalencing of nodes

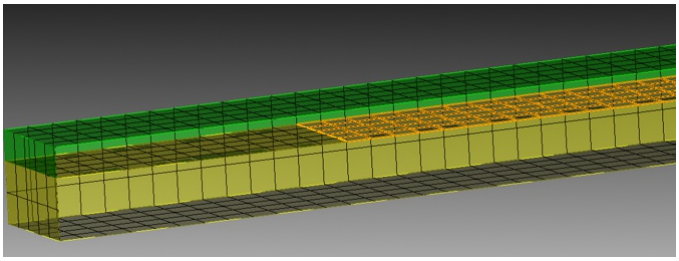
Equivalencing of nodes in a lightweight mesh is supported using this command.

```
Equivalence Node <range> [Tolerance <value>]
[Preview]
```

All nodes given and within tolerance of each other will be merged. More information on equivalencing nodes can be found here

Creating Cohesive Elements

To model adhesives, interfaces in composite materials or crack propagation, it is possible to create cohesive elements.



A geometry with highlighted surface indicating location for cohesive elements

The above example has a shared surface between the green and yellow volumes which is already cracked, using the unmerge command in Cubit. The highlighted merged surface can then used for creating cohesive elements. The following command will do so.

```
Create Cohesive Elements Block <value> Surface
<ids>
```

The **Block** keyword indicates the block id for the newly created block.

The **Surface** keyword indicates which surfaces to unmerge and insert cohesive elements..

Finite Element Model

- Global Element IDs
- Exodus Boundary Conditions
- Non-Exodus Boundary Conditions
- Exporting the Finite Element Model

This chapter describes the techniques used to complete the definition of the finite element model. The definitions of the basic items in an Exodus database are briefly presented, followed by a description of the commands a user would typically enter to produce a customized finite element problem description, and how to export the finite element model.

Global Element IDs

Cubit Mesh Entity ID Spaces

All mesh entities have an ID associated with them which is unique within the corresponding mesh entity ID space. For example, a hex will have an id which can be used in commands such as "list hex 17". However, this ID is only unique amoungst the hexahedra in the Cubit session. There could also be a tet, quad, tri, edge, node, etc. with ID 17.

Global Element IDs

Whenever a hex, tet, quad, tri, etc. gets put into an element block, it is assigned another ID which is called the *Global Element ID*. The *Global Element ID* is unique amoungst all element which have been put into any block. Starting in Cubit 14.0, it is exported to the Exodus file format so that downstream analysis applications can map elements back to the corresponding Cubit hex, quad, etc. In Cubit 14.0, Global Element are not supported by the other exporters, but will be in future releases.

Interacting with GlobalElement IDs

After a hex, quad, etc. is placed into an element block, you can see which ID was assigned to it with the **list** command. For example:

```
reset
bri x 10
mesh vol all
block 1 hex all
list hex 1
```

The resulting output will contain the following:

```
CUBIT> list hex 1
Hex 1
Global Element ID = 1
```

In this simple example, the Hex ID is the same as the Global Element ID, but this will not always be true.

These Global Element IDs are exported as the global id in the Exodus file. If during an analysis run, a particular element needs to be identified back in the Cubit session, they can be found with any of the following commands:

> **List element <id_range>**
>
> **Draw element <id_range>**
>
> **Highlight element <id_range>**
>
> **List element <id_range>**

Users can control the assigned *Global Element ID* with the renumber command.

Export Mesh and Its Geometry Association

Cubit offers the option to export a complete finite element mesh, along with its association to an ACIS geometry model. This is useful if a 3rd party application is going to be used to modify the mesh after exporting from Cubit, and you want the geometry available to project to during the modification operations. The command is:

```
Export m2g '<fileroot>' [{opennurbs|ACIS}]
[overwrite]
```

The **fileroot** argument to this command is not a complete filename, rather, it is a full path and filename without the file extension. The **export m2g** command will write out:

- **fileroot.XYZ:** A geometry file containing the geometric model, where XYZ is the 3 letter file extension of the requested geometry output.
- **fileroot.exo:** The full mesh in the exodus format.
- **fileroot.m2g:** An ascii file defining how the mesh in fileroot.exo is associated to fileroot.sat.

The **export m2g** command can export either an ACIS (*.sat) file or an OpenNurbs (*.3dm) file depending on the supported geometry types of the downstream 3rd party application in use. The default is to export an ACIS file.

An example usage of the **export m2g** command is the Sierra mesh_scale command. Sierra mesh_scale is a batch program which performs the same mesh scaling as can be performed in Cubit with the **scale mesh** command. When generating new nodes on the boundary of the mesh, Sierra mesh_scale projects the locations of the new nodes to the ACIS geometry model by leveraging the information exported by the **export m2g** command.

Exporting Sierra Files

Sierra input decks can be exported from Cubit. This capability was added in response to a need to translate Abaqus input decks to Sierra input decks by importing the Abaqus deck into CUBIT and then immediately exporting the Sierra deck. Therefore, it is assumed that most of the input deck information has been created outside of CUBIT and that the user will not interact with it in CUBIT .

The Sierra input deck writer is simply another export format and as a result it can be used for any currently defined mesh and input deck info defined in Cubit.

The Sierra input deck exporter relies on some of the mesh-specific information that is generated when exporting the Genesis mesh. Therefore, you should **export the Genesis mesh before exporting the** Sierra **input deck.**

Defining PARAMS for NASTRAN

List Nastran Exporter Params

Set Nastran Exporter Params Add '<param_string>'

Set Nastran Exporter Params Remove '<param_string>'

Set Nastran Exporter Params Clear

Nastran uses “PARAMS” to define additional instructions and settings in its Bulk Data file. Any string can be defined as a Nastran Exporter Param, and it will be exported to the Nastran file as “PARAM, <string>”.

Exporting ABAQUS

Mesh can be exported from CUBIT in the ABAQUS format. The command to export to ABAQUS is:

```
Export Abaqus <'filename'> [Block <id_list>]
[Sideset <id_list>] [Nodeset <id_list>] [BCSet
<id_list>] [Group <id_list>] {[Instance Block
<id_list> [Source_csys <id>] Target_csys
<id_list>] | [Instance_per_block]} [partial]
[Overwrite] [Everything]
```

The ABAQUS file written by CUBIT will contain a single part, or mulitple parts. Multiple parts can be defined by using the **instance** option. Additionally, a single block/part can be instanced multiple times by giving multiple target coordinate systems. A bolt mesh used several times is an example where one might want multiple instances. To instance a block, a source coordinate system and a target coordinate system (where the mesh will be translated and rotated to) need to be defined. If no source coordinate system is given in the command, the default (global) coordinate system is used. The default (global) coordinate system can be referenced specifcally using '0' for the id. The instance keyword can be used as many times as needed. For example, **block 1** can be instanced 3 times using 2 defined coordinate systems and the global coordinate system using:

```
create coordinate frame origin location 2 0 0
tag 'R'

create coordinate frame origin location -2 0 0
tag 'R'

Export Abaqus "myfile.inp" Instance Block 1
Target_csys 0 1 2
```

To enable automatic instancing, with the global coordinate system, based on defined blocks, the **Instance_per_block** option can be used.

Materials are supported with ABAQUS export as well. See documentation on materials for more information.

```
create material "Steel-200" property_group
"CUBIT-ABAQUS"

modify material "Steel-200" scalar_properties
"DENSITY" 7.8240

block 1 add material "Steel-200"
```

Additionally, Groups may be used to define additional node and element sets. For example:

```
group 'set-material-statistic-200' add tet in
block 1 node in block 1
```

Note: By default, the Abaqus exporter writes 6 decimal places. The command "**set Abaqus precision <n>**" can be used to change the number of decimal places written.

Exporting an Exodus II File

After defining the element blocks, nodesets and sidesets for a model, the model can be written to the Exodus II file using the command:

```
Export [Genesis|Mesh] '<filename>' [dimension
{2|3}] [Block <id_list>] [no_ids]
[Qualityfile] [XML '<filename>'] [Overwrite]
```

Note: The ordering of options in this command is important. Misordering the options can cause the command to ignore some options.

The **Genesis** or **Mesh** arguments are optional and both indicate that an Exodus II format will be written. The filename can be any valid filename. Where a full path is not specified, the file will be written in the current working directory.

The **dimension** argument is also optional. Most element types have an inherent dimensionality associated with them. For example, a truss or beam element is inherently 2D while a hex or tetra element is 3D. Without this argument, only the x-y location of the nodal coordinates of 2D elements are written to the Exodus II file. Using the argument *dimension 3* in this example permits the full 3D coordinates to be written.

The optional **Block** argument may also be added to the **Export** command. Without this argument all blocks defined in the current model will be exported to the Exodus II file. This argument permits the user to specify only a subset of all the blocks in the model. The **<id_list>** may be any valid set of integers corresponding to the block ids in the current model.

no_ids prevents the writing of node and element id maps in the Exodus file. This can be useful if users do not want Cubit ids in downstream workflows.

The **Qualityfile** option exports, in addition to the mesh, a text file containing a printout of the element quality using the 'Allmetrics' option. The name of this file is the base name of the mesh file (file extension removed) and "_quality.txt" added.

The **XML** optional argument may also be added to the **Export** command. When this argument is included and assembly data exists in the model, an XML file is written which describes the relationship between block IDs in the Exodus II file and parts in the assembly. See the Parts, Assemblies and Metadata section for details.

Element and Node ID Maps

Element ID map and node ID map are always written to the Exodus II file. The IDs written to the node ID map are the node IDs used to refer to nodes at the Cubit command line. The IDs written to the element ID map are the Global Element IDs which are assigned to the hex, tet, quad, etc. when they are added to an element block. The node and element ID maps can be used when a particular element or node is refered to in a downstream application and the corresponding node or element in Cubit must be found. Some analysis and post-processing applications consider these maps to be optional, while others ignore the maps even if they are present. See the Exodus manual for more information on element and node ID maps.

Exporting a Parallel Mesh for pCAMAL

```
Export Parallel "<filename>" [Block <id_list>]
```

```
[Overwrite] [Processor <number>]
```

The **Export Parallel** command is used to output an ExodusII file with the boundary mesh or shell for sweepable volumes that were meshed with set parallel meshing enabled. The options are the same as those for the "export genesis" command except for the addition of the processor option.

The processor option allows the user to specify the number of processors that will be used to mesh the volume with the pCAMAL option. This same option exists in the pCAMAL application and is more often used there since the number of available processors is known then rather than when the output file is created in Cubit.

If the processor option is given, Cubit attempts to balance the number of sweepable volumes to run on n processors by converting many-to-one sweeps to one-to-one sweeps, subdividing the sweep volume along its sweep direction, or partitioning the source surface of a one-to-one sweep if the number of source quads is much larger than the number of layers.

Converting an Exodus II file to ASCII

The Exodus II file format is binary. It is frequently necessary to view the contents of the Exodus II file as plain text. A publicly available tool known as **ncdump** can be used to view the contents of an Exodus II file. **ncdump** is part of the **netCDF** library and is currently available from Unidata at the following URL:

http://www.unidata.ucar.edu/

On a UNIX platform, typical use of the **ncdump** utility is:

```
ncdump filename.e > filename.txt
```

In this format, the **ncdump** utility will take the Exodus II file, **filename.e**, and dump the contents to an ASCII file **filename.txt**

Another option for converting between binary and ASCII formats of Exodus II files is a utility known as **exotxt**. Exotxt is part of the SEACAS tool suite. Contact the Sandia CUBIT development team for a copy of this utility.

Controlling Exodus II Output Precision

By default, exodus files are written with double precision numbers. It may be useful to change this for large meshes to decrease output file size. This can be done using the following command:

```
Set Exodus Single Precision [On|Off]
```

This command toggles the Exodus output file between single precision (floats) and double precision.

64 bit Ids

The **Set Exodus 64bit** command enables 64 bit ids in the Exodus files. This may be useful when the file includes a large number of entities such that a 32 bit id can no longer represent distinct entities.

```
Set Exodus 64bit [on|OFF]
```

Large Exodus Format

The **Set Large Exodus** command enables the large exodus file setting to create a model that can store individual datasets larger than 2 gigabytes. This modifies the internal storage used by ExodusII and also puts the underlying netcdf file into the "64-bit offset" mode.

```
Set Large Exodus [ON|Off]
```

Exodus NetCDF4/HDF5 Format

The **Set Exodus NetCDF** command enables the exodus NetCDF4/HDF5 file setting to create a model that can store even larger files with unlimited dimensions. This modifies the internal storage used by ExodusII to an HDF5 based file. This setting overrides the **Set Large Exodus** setting.

```
Set Exodus NetCDF4 [On|OFF]
```

Exporting Geometry Association with the Exodus Mesh

Optionally, you can also export the associated ACIS geometry and the correspondence between the mesh and the geometry by using the **export m2g** command.

Exporting the Finite Element Model

For information on exporting an Exodus File, see Exporting Exodus II Files. Custom translators are available to translate between the Exodus II format and a limited number of other analysis code formats. Contact the cubit development team for a current list of supported translator formats. For information on the GDF format, see Exporting GDF Files. The general syntax for the various exporters is as follows. The specific exporter commands are listed below.

```
Export {Abaqus [Explicit]* [Partial]* | CGNS |
Nastran | Ideas | Patran | LSDyna | Fluent}
<'filename'> [Block <id_list>] [Sideset
<id_list>] [Nodeset <id_list>] [BCSet
<id_list>] [dimension {2|3}***] [Overwrite]
[Everything] [NX]**

Export {Sierra | VRML} <'filename'>
[Overwrite]
```

* **Explicit** and **Partial** keywords only available with Abaqus Exporter

** **NX** keyword only available with I-DEAS Exporter

***The **dimension** argument is also optional. Most element types have an inherent dimensionality associated with them. For example, a truss or beam element is inherently 2D while a hex or tetra element is 3D. Without this argument, only the x-y location of the nodal coordinates of 2D elements are written to the Exodus II file. Using the argument dimension 3, in this example, permits the full 3D coordinates to be written.

The Abaqus Exporter has a few additional keywords available. See the last paragraph below for an explanation of those keywords:

```
Export Abaqus <'filename'> [Block <id_list>]
[Sideset <id_list>] [Nodeset <id_list>] [BCSet
<id_list>] [dimension {2|3}] [nodefile
<'filename'>] [elementfile <'filename'>] [flat]
[overwrite] [everything]
```

If no blocks are exported, Cubit will export all nodes and elements in the model. If one or more blocks are entered in the command, only those blocks will be exported. Similarly, if no BCSets are entered in the command, Cubit will export all boundary conditions as a single BCSet. If one or more BCSets are entered into the command, only those BCSets will be exported. Use the overwrite flag to overwrite an existing file.

By default, Cubit will reassign node and element IDs based on which block they are in. If the everything keyword is present, Cubit will export all nodes and elements in the model, whether they are in a block or not.

The I-DEAS Universal file can be read into Siemen's NX application if the file is generated using the **NX** keyword. This is because extra information must be written to an I-DEAS Universal file in order for NX to be able to read it.

There are a few keywords specifically for the Abaqus exporter. **Flat** can be used if the user desires Cubit to write out the model as a "flat file." Abaqus refers to files a "flat files" when they do not use the *PART/*INSTANCE structure. All nodes and elements will be defined at the global level. The keywords **elementfile** and **nodefile** can be used to instruct Cubit to export the nodes and/or elements to a separate file.

If the **Explicit** keyword is used with Abaqus, Cubit will write an Abaqus Explicit deck. The one Explicit-only feature that Cubit supports is **Fixed Mass Scaling**.

If the **Partial** keyword is used with Abaqus, Cubit will write a partial Abaqus deck. Cubit will output the mesh as defined by the Abaqus keywords **PART, NODE, ELEMENT, NSET, ELSET, and SURFACE**. Everything else is ignored. Use the Abaqus keyword **INCLUDE** to include this file in a master Abaqus deck for analysis.

Specific Exporter Commands:

```
Export Abaqus [explicit] '<filename>' [Block
<id_list>] [Sideset <id_list>]    [Nodeset
<id_list>] [BCSet <id_list>] [group <id_list>]
 [instance block <id_list> [source_csys
<id_list>] target_csys <id_list>
```

```
[preview]] [dimension {2|3}] [overwrite]
[everything] [partial]

Set Abaqus Precision <n=6>
```

Note: This command can be used to control the number of decimal places written to the Abaqus file.

```
Export CGNS '<filename>' [Block <id_list>]
[Sideset <id_list>] [Nodeset <id_list>] [BCSet
<id_list>] [dimension {2|3}] [overwrite]
[everything]

Export Nastran '<filename>' [Block <id_list>]
[Sideset <id_list>]  [Nodeset <id_list>] [BCSet
<id_list>] [dimension {2|3}] [overwrite]
[everything]

Export Ideas '<filename>' [NX] [Block
<id_list>] [Sideset <id_list>]    [Nodeset
<id_list>] [BCSet <id_list>] [dimension {2|3}]
[overwrite] [everything]

Export Patran '<filename>' [Block <id_list>]
[Sideset <id_list>]    [Nodeset <id_list>]
[BCSet <id_list>] [overwrite] [everything]
[dimension {2|3}]

Export Lsdyna '<filename>' [Block <id_list>]
[Sideset <id_list>]    [Nodeset <id_list>]
[BCSet <id_list>] [overwrite]

Export Fluent '<filename>' [Block <id_list>]
[Sideset <id_list>]    [Nodeset <id_list>]
[BCSet <id_list>] [dimension {2|3}] [overwrite]
[everything]
```

Note: The following command is for exporting mesh geometry, (**.msh format**.)

```
Export Fluent '<filename>' [Surface
<id_list>|Volume <id_list>] [Overwrite]

Export Sierra <'filename'> [Overwrite]

Export VRML <'filename'> [Overwrite]
```

Additional Information on building Cubit models for CFD

Exporting ABAQUS

Defining PARAMS for NASTRAN

Supported element types

Cubit Element Type	ExodusII	CGNS	Abaqus	Nastran	I-DEAS UNV	Patran	LS-DYNA	Fluent
Sphere	SPHERE						ELEMENT_SPH	
Spring	SPRING		SPRINGA/SPRING1/SPRING2	CBUSH1D**				
Bar	BAR		B21**	CROD**	121**	Bar2		
Bar2	BAR2		B21**	CROD**	121**	Bar2		
Bar3	BAR3		B22**	CROD**	121**			
Beam	BEAM		B31	CROD**	21**	Bar2	ELEMENT_BEAM	
Beam2	BEAM2		B31	CROD**	21**	Bar2	ELEMENT_BEAM	
Beam3	BEAM3		B32	CROD**	24**			
Truss	TRUSS		T3D2/T3D2T*,**	CROD**	121**	Bar2	ELEMENT_BEAM	
Truss2	TRUSS2		T3D2/T3D2T*,**	CROD**	121**	Bar2	ELEMENT_BEAM	
Truss3	TRUSS3		T3D2/T3D2T*,**	CROD**	121**			
Quad	QUAD	QUAD_4	CPE4R/CPE4RT*	CQUAD4	54	Quad4	ELEMENT_SHELL	3
Quad4	QUAD4	QUAD_4	CPE4R/CPE4RT*	CQUAD4	54	Quad4	ELEMENT_SHELL	3
Quad5	QUAD5					Quad5		
Quad8	QUAD8		CPE8R/CPE8RT*	CQUAD8	55	Quad8		
Quad9	QUAD9		S9R5	CQUAD	55	Quad9		
Shell	SHELL		S4R/S4RT*	CQUAD4	94***	Quad4	ELEMENT_SHELL	
Shell4	SHELL4		S4R/S4RT*	CQUAD4	94***	Quad4	ELEMENT_SHELL	
Shell8	SHELL8		S8R/S8RT*	CQUAD8	95***	Quad8		
Shell9	SHELL9		S9R5		95***	Quad9		
Tri	TRI	TRI_3	CPS3/CPS3T*	CTRIA3	51	Tri3	ELEMENT_SHELL	1
Tri3	TRI3	TRI_3	CPS3/CPS3T*	CTRIA3	51	Tri3	ELEMENT_SHELL	1
Tri6	TRI6		CPS6/CPS6T*	CTRIA6	52	Tri6		
Tri7	TRI7				52	Tri7		

Trishell	TRISHELL		STRI3	CTRIA3	91	Tri3	ELEMENT_SHELL	
Trishell3	TRISHELL3		STRI3	CTRIA3	91	Tri3	ELEMENT_SHELL	
Trishell6	TRISHELL6		STRI65	CTRI6	92	Tri6		
Trishell7	TRISHELL7				92	Tri7		
Hex	HEX	HEXA_8	C3D8R/C3D8RT*	CHEXA	115	Hex8	ELEMENT_SOLID	4
Hex8	HEX8	HEXA_8	C3D8R/C3D8RT*	CHEXA	115	Hex8	ELEMENT_SOLID	4
Hex9	HEX9							
Hex20	HEX20		C3D20R/C3D20RT*	CHEXA	116	Hex20		
Hex27	HEX27			CHEXA	116			
Tetra	TETRA	TETRA_4	C3D4/C3D4T*	CTETRA	111	Tet4	ELEMENT_SOLID	2
Tetra4	TETRA4	TETRA_4	C3D4/C3D4T*	CTETRA	111	Tet4	ELEMENT_SOLID	2
Tetra8	TETRA8							
Tetra10	TETRA10		C3D10/C3D10MT*	CTETRA	118	Tet10	ELEMENT_SOLID	
Tetra14	TETRA14							
Wedge	WEDGE		C3D6/C3D6T*	CPENTA	112		ELEMENT_SOLID	6
Hexshell	HEXSHELL							
Pyramid	PYRAMID			CPYRAM	115†			
Pyramid5	PYRAMID5			CPYRAM	115†			
Pyramid8	PYRAMID8			CPYRAM	116†			
Pyramid13	PYRAMID13			CPYRAM	116†			
Pyramid18	PYRAMID18			CPYRAM	116†			
Superelement	SUPERELEMENT_TOPOLOGY_XXX							

*Thermal element

**Check to make sure the element's properties are correct after exporting

***Also exports lofting factor for shell elements (IDEAS)

† The element type will be HEX but the number of nodes will be the number of nodes in the pyramid.

Supported boundary conditions types

Cubit Element Type	ExodusII	CGNS	Abaqus	Nastran	I-DEAS UNV	Patran	LS-DYNA	Fluent
BC Set			*STEP	SUBCASE	2428			
Displacement				SPC	791	08		
Temperature			*BOUNDARY	TEMP	791	10		
Force			*CLOAD	FORCE/MOMENT	790	07		
Pressure			*DSLOAD	PLOAD4	790	6		
Convection			*SFILM ***	CONV	790	17		
Heat Flux			*DSFLUX	QHBDY	790	16		
Contact			*CONTACT		2471			
Materials			*MATERIAL	MAT1_, MAT4_	1716	03		
CFD Boundary Conditions								
Interior								2
Wall								3
Inlet Pressure								4
Inlet Massflow								20
Inlet Velocity								10
Outlet Pressure								5
Far-field Pressure								9
Symmetry								7

*** Does not allow separate temperatures for top and bottom of shell elements. Values will be averaged.

Exporting Fluent Grid Files

Geometry can be exported from Cubit to the Fluent **.msh** format. This format can be used to exchange grid information between **.msh** compatible programs including Fluent, GAMBIT, and TGrid. The command used to export the mesh geometry is:

```
Export Fluent '<filename>' [Surface
<id_list>|Volume <id_list>] [Overwrite]
```

The filename should be enclosed in either single or double quotes. By convention, the file extension .msh is applied to grid files. The extension should be included in the filename section. Other file extensions such as .cas may be used, but they cannot be guaranteed to be compatible with either GAMBIT or TGrid.

In order to guarantee that the grid file will be compatible with Fluent, all bodies must be merged (See Geometry Merging). Several types of Fluent boundary condition zones are now implemented in Cubit. They are:

- axis
- exhaust fan
- fan
- inlet vent
- intake fan
- interface
- interior
- mass flow inlet
- outflow
- outlet vent
- periodic
- periodic shadow
- porous jump
- pressure far field
- pressure inlet
- pressure outlet
- radiator
- symmetry
- velocity inlet
- wall

Boundary condition zones created in two different ways. The first way involves user-defined mesh groups consisting only of quads (3D), triangles (3D), or element edges (2D) (See Geometry Groups). The second way involves sidesets. Specifying a boundary condition consists of selecting a user-defined mesh group or a sideset, or a surface. Selecting a surface automatically assigns the boundary condition to the sideset associated with that surface. The boundary condition type is specified and is either given a name or an id (See Using CFD Boundary Conditions). Groups or sidesets of mixed type (e.g. hexes and faces) will not be exported. All surfaces not set to one of the first seven boundary condition types are automatically set to type ‘wall’. The various parameters for each of the boundary condition types must be set within either Fluent or GAMBIT.

Cell zones are automatically created for 3D meshes containing blocks. Blocks must contain entire and continuous volumes in order to create a valid grid. In 2D models, the cell zones are created from sidesets containing only quads or tris. In order to create a valid grid, these sidesets must contain whole, continuous surfaces. All cell zones are by default set to type ‘fluid.’

If no entities are specified, the entire model is exported. In order to export selected entities, the types ‘volume’ and ‘surface’ can be specified. In 2D cases, use ‘surface’ while in the 3D case use ‘volume.’

Transforming Mesh Coordinates

A mesh can be scaled and transformed to a new location as it is written to or read from an Exodus file. To transform a mesh during import or export use the following command:

```
Transform Mesh {Input|Output}
[Scale <xyz_factor>]
[Scale <x_factor> <y_factor> <z_factor>]]
[Scale {X|Y|Z} <factor>]
[Translate <dx> [<dy> [<dz>]]]
[Translate {X|Y|Z} <distance>]
[Rotate <degrees> about {X|Y|Z}]
[Reset]
```

This command may be repeated any number of times using any number of options. Transform commands are cumulative, added to the effect of previous transforms. If more than one transformation is entered in the same command, transformations are applied in the order they appear in the command.

To clear a transformation matrix, use the **Reset** option:

```
Transform Mesh {Input|Output} Reset
```

Mesh input and output transformations are also cleared when you reset the entire model using the **Reset** command.

Transforming a mesh during output **does not** change the position of the mesh within CUBIT. It only changes the nodal positions written to the Exodus file. Nodal positions may be changed within CUBIT by transforming the body that contains the mesh. See Geometry Transforms for information on how to apply transformations to a Body.

Transforming a mesh during input **does** change the position of the mesh with CUBIT. The file being read is not modified.

Transformations applied during mesh input are independent of transformations applied during mesh output.

The following example generates a simple mesh, writes the mesh with its coordinates scaled by a factor of 2, and then re-imports that mesh, restoring the scaling to what it originally was in CUBIT.

```
brick x 10
volume 1 interval 4
mesh vol 1
transform mesh output scale 2
export mesh 'temp.exo'
delete mesh
transform mesh input scale .5
import mesh 'temp.exo'
```

See Geometry Transforms for information on how to apply transformations to a Body.

See Nodeset and Nodeset Repositioning

See Importing a Mesh

See Mesh Based Geometry

Coordinate Frames

CUBIT allows the user to define coordinate systems (frames) that are written to a mesh file. These coordinate frames are generally used as reference coordinate systems during analysis. In CUBIT, the user may define multiple exodus coordinate frames. When created, a coordinate frame is assigned an id. Coordinate frames can be created using x-y-z coordinates, nodes or vertices with the following commands:

```
Create Coordinate Frame
<xval> <yval> <zval>//origin
<xval> <yval> <zval> //z-axis
<xval> <yval> <zval> //xz-plane
[tag { 'R' | 'C' | 'S' } ]

Create Coordinate Frame Node
<node_origin_id>
<node_zaxis_id>
<node_xzplane_id>
[tag { 'R' | 'C' | 'S' } ]

Create Coordinate Frame Vertex
<vertex_origin_id>
<vertex_zaxis_id>
<vertex_xzplane_id>
[tag { 'R' | 'C' | 'S' } ]
```

Using the 'tag' option specifies the type of coordinate frame, i.e., rectangular (R), cylindrical (C) or spherical (S). The default coordinate frame type is rectangular. Exodus coordinate frames may also be listed and deleted using the commands below:

```
List Coordinate Frame [ids] [ <frame_id>]

Delete Coordinate Frame [ids] [ <frame_id>|
all]
```

Any coordinate frames that exist at the time the exodus file is exported will be written out in the exodus file.

These coordinate frames can also be referenced when exporting an Abaqus file.

Exporting GDF Files

The Geometric Data Format (GDF) is an export format used by WAMIT. The file format consists of quadrilateral and triangular elements.

```
Export GDF '<filename>' [entity_list] [Block
<id_list>] [ulen <value=1.0>] [gravity
<value=9.80665>] [isx <value=0>] [isy
<value=0>] [Overwrite]
```

All elements to be exported should be part of a block. All blocks are exported unless otherwise specified in the command. The standard GDF parameters can be specified on export.

Element Block Specification

- Creating Blocks
- Assigning a Name or Description to an Element Block
- Defining the Element Type
- Node Constraints for High Order Elements
- Default Element Blocks
- Duplicate Block Elements
- Assigning Attributes
- Displaying Blocks
- Deleting Blocks
- Renumbering Element Blocks
- Automatically Assigning Mesh Edges to a Block (Rebar)
- Creating Spider Blocks
- Creating Beam Blocks
- Creating Spring Blocks
- Creating Sphere Blocks
- 2d Elements
- Mixed Element Output
- Adding Materials to a Block
- SUPERELEMENT_TOPOLOGY_XXX Support

Element blocks are the method CUBIT uses to group related sets of elements into a single entity. Each element in an element block must have the same basic and specific element type.

The preferred method for defining blocks is to use geometric entities such as volumes, surfaces or curves. Blocks can also be defined using mesh entities. If a block is defined at a geometric entity, each of the elements *owned* by the geometry are automatically assigned to the block. Deleting or remeshing the geometry automatically changes the set of elements grouped into the block. If mesh entities are used to specify a block, deleting the mesh will also delete the elements from the block.

Some important notes regarding Element Blocks are as follows:

- Multiple volumes, surfaces, and curves can be contained in a single element block
- A volume, surface, or curve can only be in one element block
- Element Block id's are arbitrary and user-defined. They do not need to be in any contiguous sequence of integers.
- Element Blocks can be assigned a single floating point number, referred to as the block Attribute; this number is used to represent the length or thickness of Bar and Shell elements, respectively. The attribute defaults to 1.0 if not specified.

Creating Element Blocks

Element blocks are defined with the following Block commands.

```
Block <block_id> [ADD|Remove] {Vertex | Node}
<range>

Block <block_id> [ADD|Remove] {Curve | Edge}
<range>

Block <block_id> [ADD|Remove] {Surface | Face
| Tri} <range>

Block <block_id> [ADD|Remove] {Volume | Hex |
Tet | Pyramid | Wedge} <range>

Block <block_id> [ADD|Remove] Group <range>
```

These commands define blocks based on a list of geometric or mesh entities. A block can only hold entities of the same dimensionality. For example, a block defined to hold vertices and nodes cannot also hold hexes. The above commands reflect this restriction. This restriction also applies when adding entities using groups. When creating a block using a group containing entities of different dimensionality the behavior is undefined.

Adding geometric entities to a block effectivily adds all mesh entities of the same dimensionality contained in the geometric entity to the block. For example, adding a volume to a block adds all hexes, tets, pyramids and wedges contained in the volume to the block. Removing geometry entities works in the same manner. Thus the following commands:

```
Block 1 add volume 1

Block 1 remove hex 1
```

Creates block 1 containing all of the hexes, tets, pyramids and wedges in volume 1 except for hex 1.

When a mesh entity, or a meshed geometric entity is put into a block, it is assigned a *Global Element ID* which is exported to the exodus file for tracking during analysis.

Assigning a Name or Description to an Element Block

The following commands can be used to assign a name or description to an element block. Assigning a name to a block can be more intuitive than using traditional integer IDs, and the name and description are preserved in DART metadata-enabled applications (like SIMBA). This command is also available for nodesets and sidesets.

```
Block<ids> Name "<new_name>"

Block<ids> Description "<description>"
```

Defining the Element Type

Each block must have a specific element type associated with it. To assign an element type to a block, use the following command:

```
Block <block_id_range> Element Type <type>
```

Available element types are defined by the Exodus II file format specification (Schoof, 95). CUBIT supports the following element types:

Nodes: SPHERE SPRING

Curves: BAR BAR2 BAR3 BEAM BEAM2 BEAM3 TRUSS TRUSS2 TRUSS3 SPRING

Surfaces: QUAD QUAD4 QUAD5 QUAD8 QUAD9 SHELL SHELL4 SHELL8 SHELL9 HEXSHELL TRI TRI3 TRI6 TRI7 TRISHELL TRISHELL3 TRISHELL6 TRISHELL7

Volumes: HEX HEX8 HEX9 HEX20 HEX27 TETRA TETRA4 TETRA8 TETRA10 TETRA14 TETRA15 PYRAMID PYRAMID5 PYRAMID13 PYRAMID18 WEDGE WEDGE6 WEDGE15 WEDGE16 WEDGE20 WEDGE21

If the element type is not assigned for an element block, it will be assigned a default type depending on which type of geometry entity is contained in the block. The default values used for element type are:

Volume: 8-node hexahedral elements (HEX8) will be generated for hex meshes. TETRA4 will be generated for tet meshes.

> **Surface**: 4-node shell elements (SHELL4) will be generated for quad meshes and TRISHELL3 for tri meshes.
>
> **Curve**: 2-node bar elements (BAR2) will be generated.
>
> **Node**: 1-node elements (SPHERE) will be generated.

Node Constraints for High Order Elements

Higher order nodes are moved to curved geometry by default. To change this, use the following command:

```
set Node Constraint {on|off|SMART [tet quality
{distortion|NORMALIZED INRADIUS}][threshold
<value=0.15>]]}
```

On means higher order mid-nodes snap to curved geometry. **Off** means the mid-nodes retain their positions. **"smart"** means higher order mid-nodes will only snap to geometry if they do not cause quality problems after being moved. Nodes that cannot be moved without causing quality problems are placed at the average location of the element nodes: for edges, this means on the line containing the edge; for 2d elements, this usually means on the plane containing the element.

When the **smart** option is used, the **tet quality** and **threshold** options can also be used. **Tet quality** indicates the quality metric that will be used for determining whether mid-nodes will be projected or straightened. This option is currently only valid for high order tets (TETRA10) and tris (TRI6). **Normalized Inradius** or **Distortion** metrics may be selected as criteria for projections. The **threshold** value indicates the quality value at which mid-nodes will not be projected. For example, if **Normalized Inradius** falls below the threshold value, the element edge will be straightened. Those with metrics above the threshold will be projected.

This setting also has an effect when moving nodes with the **node <range> move** command. If nodes are associated with geometry, they will snap to that geometry after that are moved when **node constraint** is set to **on**.

Default Element Blocks

When exporting an ExodusII file, if the user has not specified any Element Blocks, by default element blocks will be written for any meshed volumes. This default behavior can be changed, to write surface, volume, or no meshes by default. This option can be set using the command

```
Set Default Block
[ON|off|Volume|Surface|Curve]
```

Default behavior, **ON**, is for the blocks to automatically be written based on their owning geometry. When the **OFF** setting is used, only the mesh contained in blocks created by the user will be exported. Mesh not in an element block at export time, will not be exported. The export will still succeed and no error will be thrown. If **Volume** is specified, only elements contained in volumes will have default blocks specified. Similarly, the **Surface** or **Curve** argument indicates that only surfaces or curves containing elements will use default blocks, respectively.

When default blocks are used, the IDs for the resulting blocks will be the ID of the owning geometry.

Duplicate Block Elements

By default, any given element cannot be included in more than one block. However, when using the following command, an element may be included in more than one block. Please note, since material properties

are assigned to blocks, using this command to allow duplicate block elements may result in an element being assigned to multiple materials.

```
Set Duplicate Block Elements {on|OFF}
```

Cubit stores only a single Global Element ID (GID) for each element. If an element is placed into more than one block, when the model is exported to Exodus, new additional GIDs will be assigned to the element for each additional block that an element is in. These additional GIDs are exported to the exodus file, but Cubit currently only stores and tracks the first GID assigned.

Assigning Attributes to Blocks

It may be necessary to associate attributes with a specific element block. Attributes are generally integer or floating point values that represent some physical property in the region occupied by the block, such as material properties or shell thickness. To assign the number of attributes for an element block, use the following command:

```
Block <id_range> Attribute Count <0-20>
```

CUBIT will store up to 20 attributes per block. Specify the maximum number of attributes to be stored on the block with this command. Once this command has been executed, individual attributes may be set using the following command:

```
Block <id_range> Attribute Index <index>
<value>
```

The index is an integer from 1 to the maximum count specified in the Block Attribute Count command. The value may be any valid floating point number.

To assign a value to all attributes of an element block, use the command:

```
Block <block_id_range> Attribute <value>
```

Displaying Element Blocks

Blocks can be viewed individually with CUBIT by employing the following command:

```
Draw Block <block_id_range> [Color
<color_spec>] [add] [thickness [offset [scale
<val>] | include_normal]]
```

For blocks that are of type SHELL and TRISHELL or one of its variants including the [thickness] keyword and parameters will result in the blocks being color-coded by shell thickness with a corresponding color bar. Blocks can be drawn with their specified thickness, so they visually have a thickness. This thickness can also be scaled in the draw command. Arrows defining the shell normal direction will be displayed as well as a legend showing the thickness values.

Block colors can also be changed using the following command:

```
Color Block <block_id_range> {color|Default}
```

Deleting Element Blocks

All Nodesets, Sidesets and Blocks may be deleted from the model using the following command:

```
Reset Genesis
```

To remove only Blocks, the following may be used:

```
Reset Block
```

To remove a specific block, use:

```
Delete Block <block_id_range>
```

Renumbering Element Blocks

The block renumber command gives the user the ability to renumber blocks to fit the user's needs. The command is:

```
Block <id_range> renumber start_id <id>
[uniqueids]
```

The **id_range** must include existing entities or the command will fail.

The **start_id** plus the number of entities must specify a new id space that does not overlap with the existing block ids. In other words, if the current block numbers are 100, 105, 106, and 109, a **start_id** of 102 would suggest new block numbers of 102, 103, 104, and 105. This would cause an id space conflict and the command will fail.

If the user specifies the **uniqueids** option, then the new entity id space must not conflict with the existing id space of all blocks, nodesets, and sidesets.

Example:

Assume:

block ids: 100, 105, 106, 109

block all renumber start 20
block 20 renumber start 24

After commands:

block ids: 21, 22, 23, 24

To renumber the elements within a block, see the renumber command

Automatically Assigning Mesh Edges to a Block (Rebar)

After a mesh has been defined within a volume, it may be useful to use the existing mesh edges as the basis for an element block. Such an element block might be composed of bars or truss type elements that might propagate through a solid medium such as rebar placed in reinforced concrete. Although the **Block <id> Edge <range>** command could be used for this task, it would prove extremely tedious defining the individual edges to add to the block. To make this process easier, the following command can be used:

```
Rebar Start <x> <y> <z> Direction <x> <y> <z>
[Length <value>] Block <id> [Element Type
{bar|bar2|bar3|BEAM|beam2|beam3|truss|truss2|truss3}]
```

The **Rebar** command allows the user to specify a starting location for a set of edges and an initial direction. The program will find the closest existing node in the mesh to **Start <x> <y> <z>** and begin propagating through the mesh in the specified **Direction <x> <y> <z>**, adding edges to the block as it propagates through the mesh. The edge that is attached to the last node and is within a fixed 30 degrees of the specified direction is added to the block. The Propagation of the edges continues until either

the optional **Length** value is reached or an edge does not meet the **Direction** criteria. Also required with this command is a **block** ID. An **Element Type** can also be specified.

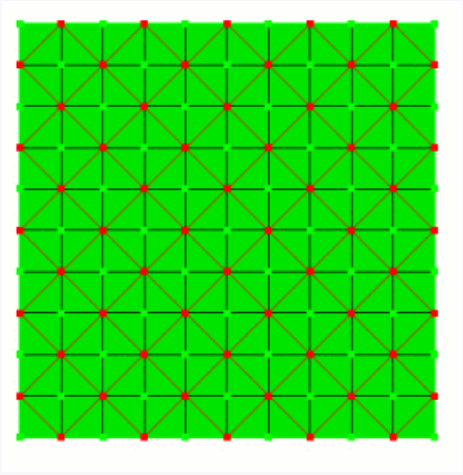
Similarly, you can use the following command which will use the 30 degree cone described above to gather edges from a surface into a single block using the Cartesian x, y, and/or z vectors.

```
Rebar Surface <range> [x] [y] [z] Block <id>
[Element Type
{bar|bar2|bar3|BEAM|beam2|beam3|truss|truss2|truss3}]
[Propagate]
```

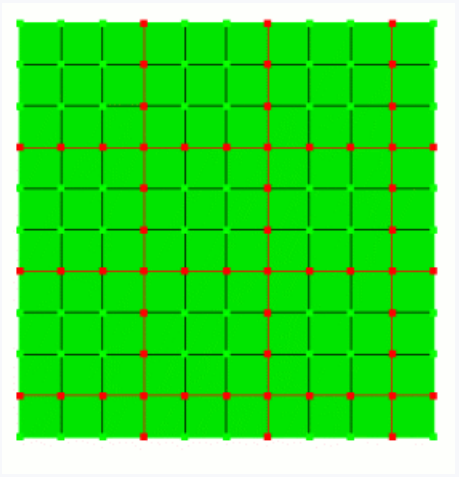
Diagonal and Orthogonal Rebar Blocks

Another method for generating rebar blocks include the Diagonal/Orthogonal option. This command can only be used on surfaces that have been meshed with the mapping scheme. This command will create a block of edges from the mapped mesh by starting in one corner and gathering edges orthogonally, or creating new edges diagonally based on the option specified, using the parametric coordinate system dictated by the mapping scheme on the surface. The spacing option dictates how many edges are skipped over before starting the next set of rebar edges.

```
Rebar Surface <range> {Diagonal|Orthogonal}
[Spacing <int>] [Block <id> [Element Type
{bar|bar2|bar3|BEAM|beam2|beam3|truss}]
```



CUBIT> rebar surf 1 diagonal spacing 2 block 2

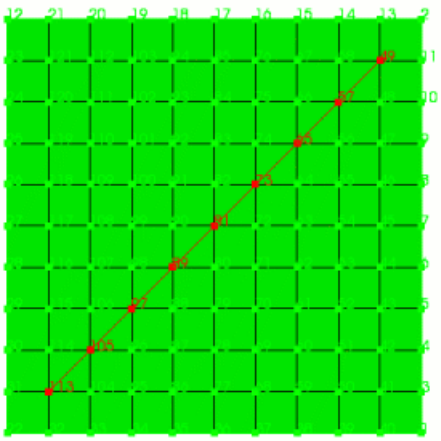


CUBIT> rebar surf 1 orthogonal spacing 3 block 3

Specifying a set of nodes

A final rebar option allows the user to create or group rebar edges into a specified block using nodes. Edges are created, or gathered, using the ordered list of nodes specified in the command.

```
Rebar Node <range> [Target Block <id>]
[Element Type
{bar|bar2|bar3|BEAM|beam2|beam3|truss}]
```



CUBIT> rebar node 113 105 97 89 81 73 65 57 49 target block 1

A related command for creating curve geometry directly from mesh edges is the Create Curve from Mesh command. See Curve creation for more details.

Creating Spider Blocks

The block creation tool also allows the user to create a special block of bar elements that can be used as part of the boundary specification. This command creates bar type elements directly without creating any underlying geometry.

The command for creating this type of block is:

```
Block <id> Joint [Vertex <id> | Node <id> ] Spider
{Surface|Curve|Vertex|Face|Tri|Node} <range>
[preview] [Element Type
{BAR|bar2|bar3|beam|beam2|beam3|truss|truss2|truss3}]
```

The **joint node** is the starting location of the bar elements and the **spider** location is the terminating location of the bar elements. You can specify the joint node as either a node or a vertex. Optionally, if no joint node is specified, a joint node will automatically be created at the centroid of the nodes on the specified terminating location. You can specify the terminating location as either a node, vertex, geometric surface or the face of a mesh entity.

Some analysis codes refer to these bar elements as tied contacts or rigid bar elements. They can be used to tie models together or to enforce specific kinds of boundary conditions. For example, in the figure below a block of beam elements is used to tie a node at the center of the circle to every node on the edge of the circle. This arrangement can be used to enforce circularity but still allow for displacement of the entire circle. This may occur if there are additional structures above the cylinder that are being excluded from the current finite element model. The beam elements were created by a series of commands of the form

```
block 10 joint node 1 spider node 2 element type beam
```

The **preview** option can be included to draw the location of the beam blocks on the screen without actually executing the command.

If geometry (surfaces, curves, or vertices) is specified to define the spider, the spider will be 'tied' to that geometry, meaning:

- if the geometry is meshed, the edges will be created
- if the geometry is deleted, the mesh will be deleted
- if the geometry is translated, the edges will be translated.

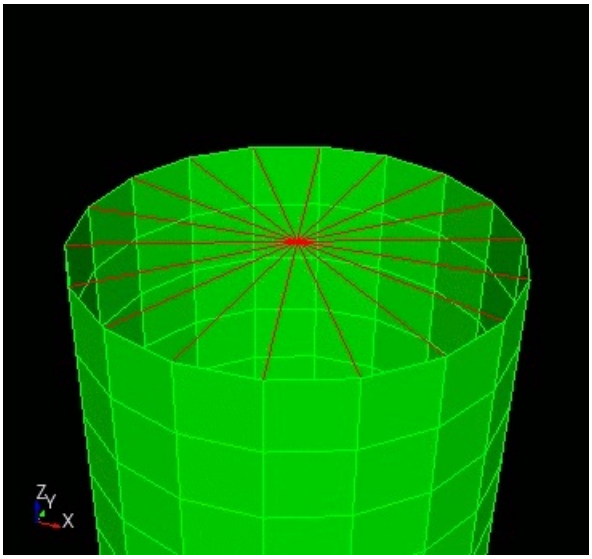


Figure 1. Beam elements created with the Spider command

Creating Beam Blocks

Properties for blocks that are beam types (beam, beam2, beam3) have additional commands to define a cross-sectional area. The following command can be used to change the type of cross-sectional area of a beam block:

```
Block <id> beam_type
{CIRCLE|box|rectangle|pipe|ibeam|general}
```

The dimensions are set by listing them after the keyword **beam_dimensions**:

```
Block <id> beam_dimensions <values>
```

The order in which the values need to be specified are described in the chart below.

If the solver used is to integrate over the section during the simulation, turn **section_integration** on using the following command:

Block <id> section_integration {ON|off}

The beam normal vector is a vector normal to the plane of motion and tangent to the first bending axis. This vector can be set using the following command:

```
Block <id> beam_normal <x><y><z>
```

Section Profile	Order to Specify Dimensions
Circle	Radius
Pipe	Outer radius, wall thickness
Rectangle	Width, height
Box	Total width, total height, thickness (right), thickness (top), thickness (left), thickness (bottom)

I-Beam	Distance to bending axis (from bottom), total height, bottom width, top width, thickness (bottom), thickness (top), thickness (web)
General	Area, Ixx, Ixy, Iyy, Polar moment of inertia (J)

Creating Spring Blocks

Spring blocks that will be exported to Abaqus can contain additional properties related to Abaqus springs. Users can specify the spring type, stiffness, and DOFs associated with Abaqus springs. The spring type mapping to Abaqus elements is in the following table.

CUBIT Block Spring Type	Abaqus Element Type
Node_to_node	SPRINGA
Node_to_node	SPRING1
Node_to_ground_fixed	SPRING2

The spring type is set using the **spring_type** keyword. In order to use this command, the block must already have an element type of "SPRING." If a DOF is associated with a spring, the **spring_dof_1** keyword is used to specify the DOF on the first node and **spring_dof_2** is used to specify the DOF on the second node (SPRING2 only).

```
Block <id> [spring_type {NODE_TO_NODE |
node_to_node_fixed_axis | node_to_ground}]
[stiffness <k>] [spring_dof_1 <n>]
[spring_dof_2 <n>]
```

Creating Sphere Blocks

Sphere elements are created in CUBIT by inserting either nodes or vertices into a block.

```
Block <id> {node|vertex} <id_range>
```

The command above causes CUBIT to internally create a sphere element and associate it to the inserted node, or to the node associated to the inserted vertex.

> **Example:**
>
> **brick x 10**
> **vol all size 5**
> **mesh vol all**
> **create vertex 0 0 10**
> **#{sph_vtx_id=Id("vertex")}**
> **mesh vertex {sph_vtx_id}**
> **#{sph_nd=Id("node")}**
> **block 1 volume 1**
> **block 2 vertex {sph_vtx_id}**
> **block 3 joint node {sph_nd} spider surf 1**
> **locate sphere all**

The example commands above will generate the model illustrated in the figure below.

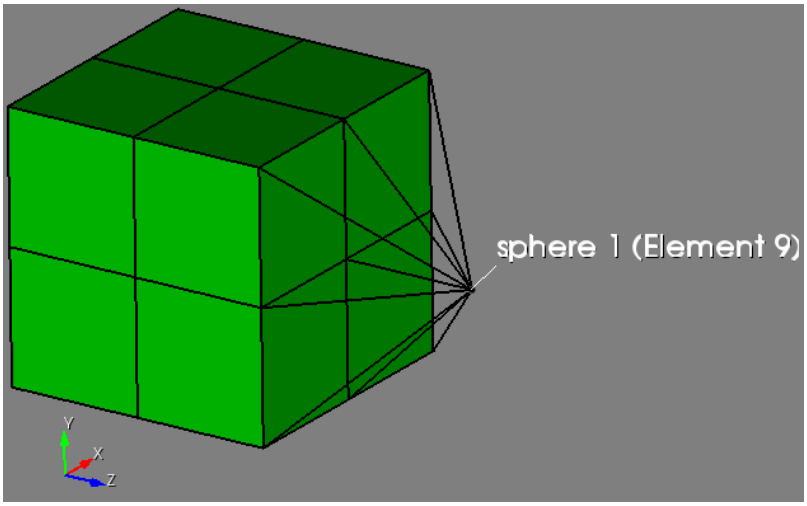


Figure 2. A sphere element created and connected to a solid mesh with 2d elements.

You can interact with sphere elements in Cubit with the commands below:

```
locate sphere <id_range>

draw sphere <id_range>

highlight sphere <id_range>

list sphere ids

list sphere <id_range>
```

2D Elements

CUBIT is a 3d mesh generator by default. Element types, by default, are respectively TRISHELL and SHELL for triangle and quad elements. If a 2d mesh is desired, blocks types must be explicitly set to TRI or QUAD.

> **Example:**
>
> **create brick x 10**
> **surface 1 scheme trimesh**
> **mesh surface 1**
> **block 1 surface 1**
> **block 1 element type tri**
> **export mesh "mymesh.exo"**

Sideset 1 will be based on the TRI and QUAD elements in **blocks 1 and 2**, with the side numbering referring to the edges of the triangles and quads.

Mixed Element Output

The **Set Block Mixed Output** command controls the behavior of blocks containing different element types when exporting in a file format that doesn't support blocks with mixed element types. If DEGENERATE, all elements will be exported in one block, but tets and pyramids will be written as degenerate hexes, and triangles will be written as degenerate quads. If OFFSET (set by default), then new element blocks will be created separating the types. Hex and Quad blocks retain the block id, whereas tets, triangle, pyramids and wedges get put into other blocks. The ids of the other blocks are based on the block id plus the offset for that type. Those values are set using the offset commands.

```
Set Block Mixed Element Output { OFFSET |
Degenerate }
```

```
Set Block Triangle Offset <value>

Set Block Tetrahedron Offset <value>

Set Block Pyramid Offset <value>
```

Adding Materials to a Block

```
Block <id> Material <id|'name'>
```

If a material is assigned to an element block, the material properties will be associated with the block's elements when the mesh is exported. If no material is assigned to a block, a default material will be used during export.

SUPERELEMENT_TOPOLOGY_XXX Support

Cubit has basic support for Superelements. Importing mesh with Superelements is supported using either the lite, no_geom or geometry options. When using the geometry option to construct Mesh-Based geometry, superelements do not have geometry created, but continue to exist as free elements. Superelements are visually represented as a point cloud since there is no connecting topology. Cubit does not have the ability to create these elements. Importing and exporting these elements is supported.

Exodus II File Specification

Exodus II Manual

The full Exodus II manual is available from the web.

Element Block Definition Examples

Multiple Element Blocks

Multiple element blocks are often used when generating a finite element mesh. For example, if the finite element model consists of a block which has a thin shell encasing the volume mesh, the following block commands would be used:

> **Block 100 Volume 1**
> **Block 100 Element Type Hex8**
> **Block 200 Surface 1 To 6**
> **Block 200 Element Type Shell4**
> **Block 200 Attribute 0.01**
> **Mesh Volume 1**
> **Export Genesis `block.g'**

This sequence of commands defines two element blocks (100 and 200). Element block 100 is composed of 8-node hexahedral elements and element block 200 is composed of 4-node shell elements on the surface of the block. The "thickness" of the shell elements is 0.01. The finite element code which reads the Genesis file (block.g) would refer to these blocks using the element block IDs 100 and 200. Note that the second line and the fourth line of the example are not required since both commands represent the default element type for the respective element blocks.

Surface Mesh Only

If a mesh containing only the surface of the block is desired, the first two lines of the example would be omitted and the Mesh Volume 1 line would be changed to, for example

> **Mesh Surface 1 To 6**.

Two-dimensional Mesh

CUBIT also provides the capability of writing two-dimensional Genesis databases similar to FASTQ. The user must first assign the appropriate surfaces in the model to an element block. Then a Quad* type element may be specified for the element block. For example

> **Block 1 Surface 1 To 4**
> **Block 1 Element Type Quad4**

In this case, it is important for users to note that a two-dimensional Genesis database will result. In writing a two-dimensional Genesis database, CUBIT ignores all z-coordinate data. Therefore, the user must ensure that the Element Block is assigned to a planar surface lying in a plane parallel to the x-y plane. Currently, the Quad* element types are the only supported two-dimensional elements. Two-dimensional shell elements will be added in the near future if required.

Exodus II Model Title

CUBIT will automatically generate a default title for the Genesis database. The default title has the form:

```
cubit(genesis_filename): date: time
```

The title can be changed using the command:

```
Title '<title_string>'
```

Defining Materials and Media Types

Materials can be defined in CUBIT and assigned to element blocks. If an element block is exported without a material assigned to it, a default material (with properties for common steel) will be exported for it.

```
Create Material [id] [Name <'name'>]
[Elastic_modulus <value>] [Poisson_ratio
<value>] [Shear_modulus <value>] [Density
<value>] [Specific_heat <value>] [Conductivity
<value>] [User constants <value ...>] [DepVar
<value>]

Modify Material <id_list|'name'|all> [Name
<'name'>] [Elastic_modulus <value>]
[Poisson_ratio <value>] [Shear_modulus
<value>] [Density <value>] [Specific_heat
<value>] [Conductivity <value>] [User
constants <value ...>] [DepVar <value>]

Create Media [id] [Name <'name'>]
[Fluid|Porous|Solid]

Modify Media <id_list|'name'|all> [Name
<'name'>] [Fluid|Porous|Solid]
```

Materials can be created with any number of the following material properties:

- Elastic modulus
- Poisson Ratio
- Density
- Specific Heat
- Conductivity
- Shear Modulus (must satisfy E = 2G(1+v))
- User Constants
- DepVar (Only written to Abaqus file)

Media types include:

- Fluid
- Porous
- Solid

Any properties that are not initialized by the user will have a default value of 0.

Materials and media types can be listed and deleted using the following commands:

```
List Material <id_list|'name'|all>

Delete material <id_list|'name'|all>

List Media <id_list|'name'|all>

Delete Media <id_list|'name'|all>
```

Materials and media can be added to an existing block using the following command:

```
Block <id> Material <id|'name'>

Block <id> Media <id|'name'>
```

Custom Material Commands

The Cubit SDK allows custom material properties to be defined using the MaterialInterface. The following versions of the material commands allow users to create materials with custom properties that have already been defined using the SDK.

```
Create {material|media} 'material_name'
property_group 'group_name' [id <requested_id>]
[description 'string']

Modify {material|media} 'material_name'
[property_group 'group_name'] [id
<requested_id>] [description 'string'] [rename
'new_name']
[scalar_properties ('property_name'
<property_value>)...]
[vector_property 'property_name' <val1>
<val2>...]
[matrix_property 'property_name' <val1>
<val2>...]
[clear properties 'property_name1'
'property_name2'...]
```

A scalar property has a single value associated with it. The scalar properties defined by Cubit are:

- "MODULUS" (The modulus of elasticity)
- "SHEAR_MODULUS"
- "POISSON" (Poisson's ratio)
- "DENSITY"
- "SPECIFIC_HEAT"
- "CONDUCTIVITY" (Thermal conductivity, not electrical)
- "THERMAL_EXPANSION"
- "YIELD_STRENGTH"
- "ULTIMATE_STRENGTH"
- "ULTIMATE_STRAIN"
- "DEPVAR" (Property required for Abaqus export)
- "CFD_MEDIA_TYPE" (0 = fluid; 1 = porous; 2 = solid)

Vector properties are given in the command as a list of values. The vector properties defined in Cubit are:

- "USER_CONSTANTS" (User-defined material constants)

Matrix properties are also given as a list of values. The number of columns in the matrix is defined by the specific property, and Cubit automatically divides the given values into rows and columns based on the column count. For example, if a matrix property has 2 columns, the value list "2 33.1 3 18.9" is interpreted as the matrix:

2	33.1
3	18.9

Cubit defines the following matrix properties:

- "ELASTIC_MODULUS_VS_TEMPERATURE" (2 columns)
- "POISSONS_RATIO_VS_TEMPERATURE" (2 columns)
- "DENSITY_VS_TEMPERATURE" (2 columns)
- "YIELD_STRESS_VS_STRAIN_VS_TEMPERATURE" (3 columns)
- "SPECIFIC_HEAT_VS_TEMPERATURE" (2 columns)
- "CONDUCTIVITY_VS_TEMPERATURE" (2 columns)

A property group is a collection of material properties. Its main purpose is to help define what properties a material should have, even if a value is not given for the property. Cubit defines the following property groups:

- "CUBIT-FEA" (generic FEA material)
- "CUBIT-ABAQUS" (material specific to Abaqus export)
- "CUBIT-CFD" (generic CFD media)

Table 1. Property groups defined in CUBIT

Property	"CUBIT-FEA"	"CUBIT-ABAQUS"	"CUBIT-CFD"
"CFD_MEDIA_TYPE"			x
"MODULUS"	x	x	
"SHEAR_MODULUS"	x	x	
"POISSON"	x	x	
"DENSITY"	x	x	
"SPECIFIC_HEAT"	x	x	
"CONDUCTIVITY"	x	x	
"THERMAL_EXPANSION"	x	x	
"YIELD_STRENGTH"	x	x	
"ULTIMATE_STRENGTH"	x	x	
"ULTIMATE_STRAIN"	x	x	
"DEPVAR"		x	
"USER_CONSTANTS"	x	x	
"ELASTIC_MODULUS_VS_TEMPERATURE"		x	
"POISSONS_RATIO_VS_TEMPERATURE"		x	
"DENSITY_VS_TEMPERATURE"		x	
"YIELD_STRESS_VS_STRAIN_VS_TEMPERATURE"		x	
"SPECIFIC_HEAT_VS_TEMPERATURE"		x	
"CONDUCTIVITY_VS_TEMPERATURE"		x	

Exodus Boundary Conditions

Sandia's finite element analysis codes have been written to transfer mesh definition data in the ExodusII file format (citation Schoof, 95). The ExodusII database exported during a CUBIT session is sometimes referred to as a Genesis database file; this term is used to refer to a subset of an Exodus file containing the problem definition only, i.e., no analysis results are included in the database.

The ExodusII database contains mechanisms for grouping elements into Element Blocks, which are used to define material types of elements. ExodusII also allows the definition of groups of nodes and element sides in Nodesets and Sidesets, respectively; these are useful for defining boundary and initial conditions. Using Element Blocks, Nodesets and Sidesets allows the grouping of elements, nodes and sides for use in defining boundary conditions, without storing analysis code-specific boundary condition types. This allows CUBIT to generate meshes for many different types of finite element codes.

Element Blocks

Element Blocks (also referred to as simply, Blocks) are a logical grouping of elements all having the same basic geometry and number of nodes. All elements within an Element Block are required to have the same element type. Access to an Element Block is accomplished through a user-specified integer Block ID. Typically, Element Blocks can also be assigned material properties to associate material properties with a group of elements.

Nodesets

Nodesets are a logical grouping of nodes accessed through a user-specified Nodeset ID. Nodesets provide a means to reference a group of nodes with a single ID. They are typically used to specify load or boundary conditions on portions of the CUBIT model or to identify a group of nodes for a special output request in the finite element analysis code.

Sidesets

Sidesets are another mechanism by which constraints may be applied to the model. Sidesets represent a grouping of element sides and are also referenced using an integer Sideset ID. They are typically used in situations where a constraint must be associated with element sides to satisfactorily represent the physics (for example, a contact surface or a pressure.

Element Types

The basic elements used to discretize geometry were described in the mesh generation chapter. Within each basic element type, several specific element types are available. These specific element types vary by the number of nodes used to define the element, and result in different orders of accuracy of the element. The element types available for each basic element type defined in CUBIT are summarized in the following table.

Table 1. Element Types Defined in CUBIT

Basic Element Type	Specific Element Type	Notes
Node	SPHERE	

Node	SPHERE	
Edge	BAR, BAR2, BAR3, BEAM, BEAM2, BEAM3, TRUSS, TRUSS2, TRUSS3, SPRING	By default, Bars have 2 DOF's per node; Beams, trusses and springs have 3
Triangle	TRI, TRI3, TRI6, TRI7, TRISHELL, TRISHELL3, TRISHELL6, TRISHELL7	Tri element nodal coordinates are always 3D.
Quadrilateral	QUAD, QUAD4, QUAD8, QUAD9; SHELL, SHELL4, SHELL8, SHELL9, HEXSHELL	By default, quad element nodal coordinates are 2D, i.e., only x and y coordinates. Shell element nodal coordinates are 3D.
Tetrahedron	TETRA, TETRA4, TETRA8, TETRA10, TETRA14, TETRA15	TETRA8 contains vertex nodes and mid-face nodes.
Hexahedron	HEX, HEX8, HEX9, HEX20, HEX27	
Pyramid	PYRAMID, PYRAMID5, PYRAMID8, PYRAMID13, PYRAMID18	A PYRAMID8 is output as a degenerate HEX.
Wedge	WEDGE, WEDGE6, WEDGE15, WEDGE16, WEDGE20, WEDGE21	
Superelement	SUPERELEMENT_TOPOLOGY_XXX	An element composed of an variable number of nodes.

For a description of the node and side numbering conventions for each specific element type, see the Appendix. Element types can be set for individual Element Blocks, either before or after meshing has been performed.

Nodeset and Sideset Specification

- Creating Nodesets and Sidesets
- Assigning Names and Descriptions to Nodesets and Sidesets
- Grouping Faces on a Surface into a Sideset
- Deleting Nodesets and Sidesets
- Renumbering Nodesets and Sidesets
- Transfer Nodesets and Sidesets
- Displaying Nodesets and Sidesets
- Nodeset Associativity Data
- Equation-Controlled Distribution Factors
- Nodesets/Sidesets/Blocks Behavior with Geometric Entity Copy

Boundary conditions such as constraints and loads are applied to the finite element model using *nodesets or sidesets*, also known as Genesis entities. Rather than attempting to maintain specific boundary condition information, such as load, temperature, constraint, etc., Genesis entities are the generic vehicle for the user to set up boundary conditions on the model. Nodes, elements and element faces are instead grouped together and assigned unique IDs. Node, element and face IDs assigned to Genesis entities can then be written to the Exodus II mesh file. Once imported to the intended analysis application, the nodeset and sideset IDs can be appropriately interpreted as specific physical boundary conditions.

The preferred method for creating Genesis entities is to assign vertices, curves, surfaces or volumes to a specific nodeset or sideset ID. Any mesh entity *owned* by the geometric entity in a nodeset or sideset is automatically assigned to the same nodeset or sideset. This allows greatest flexibility in generating and updating the finite element mesh. For example, if a surface belongs to a specific sideset, remeshing the surface will automatically delete any old faces from the sideset and add the faces of the new mesh.

In some cases, the geometric model does not provide enough resolution to define the desired boundary conditions. In this case, the model may be partitioned using CUBIT's virtual geometry features. Where this may not be feasible, mesh entities can also be added directly to the desired nodeset or sideset. Where individual mesh entities have been added to nodesets or sidesets, deleting the mesh will also remove these elements from the Genesis entity. If the geometry is remeshed, the new mesh entities must also be added once again to the nodesets or sidesets.

Nodesets can be created from groups of nodes categorized by their owning volumes, surfaces, curves or vertex. Individual nodes may also be added to a nodeset. Nodes can belong to more than one nodeset.

Sidesets can be created from groups of element sides or faces categorized by their owning surfaces or curves or by their individual face IDs. Element sides and faces can also belong to more than one sideset.

Creating Nodesets and Sidesets

Nodesets and Sidesets are created in CUBIT by assigning the appropriate geometry or mesh entities in the model to a nodeset or sideset ID. The following commands can be used:

```
Nodeset <nodeset_id> [ADD|Remove] {Curve |
Surface | Volume | Vertex | Node} <range>

Sideset <sideset_id> [ADD|Remove] Group
<id_range>
```

```
Sideset <sideset_id> [ADD|Remove]
{Curve|Surface|Edge|Face|Tri} <id_range>

Sideset <sideset_id> [Add] Edge <id_range>
[wrt {{Tri|Face} <id_range> | all } ]

Sideset <sideset_id> [Add] Face <id_range>
[wrt {Hex <id_range> | all} ]

Sideset <sideset_id> [Add] Tri <id_range> [wrt
{Tet <id_range> | all} ]

Sideset <sideset_id> [Add] Surface <id_range>
[wrt {{Volume|Surface} <id_range> | all} ]
[FORWARD|Reverse|Both]

Sideset <sideset_id> [Add] Curve <id_range>
[wrt {Surface <id_range> | all} ]
```

Like element blocks, Nodesets and Sidesets are given arbitrary, user-defined ID numbers. If there are no user-defined Nodesets or Sidesets, none are written to the Exodus II file.

With Sidesets, direction is often important. For surfaces, the direction may be specified using the **Forward**, **Reverse**, or **Both** options. The **Forward** option will write a sideset in relation to hexes in the surface's forward volume, which is the volume that the surface's normal points away from. The **Reverse** option will write a sideset in relation to hexes in the surface's reverse volume, which is the volume that the surface's normal points into. The **Both** option will allow sidesets to be written in relation to the hexes that lie in volumes on both sides of the surface. The default is **Forward**. The user can additionally specify the volume from which the hexes should be taken in relation to by using the **wrt Volume** option.

Direction is equally important for curves in Sidesets. The **wrt Surface** option allows the user to indicate which surface's faces will be included in the Sideset. The **wrt All** option will include all faces attached to the curve. The default is wrt All.

Useful hint:

When creating nodesets and sidesets it is often userful to use the Extended Command Line Entity Specification. Here is an example that creates a nodeset which includes all the nodes on the exterior of the geometry:

```
# Create the geometry
Create brick x 10
Create cylinder height 10 radius 2
Move volume 2 z 10
# Merge the geometry
Merge volume all
# Mesh the geometry
Mesh volume all
# Create a nodeset that includes only those
nodes
# located on the exterior of the geometry
Nodeset 1 add surface in volume all with not
is_merged
```

The following commands remove nodes from the nodeset that belong to a surface. Continuing from the previous example:

```
# Remove surface 2 from the nodeset
Nodeset 1 remove surface 2
# Remove nodes from the nodeset
# that belong to the curves that bound surface
2
Nodeset 1 remove node in curve in surface 2
```

Nodes can also be added or removed based upon their coordinates. Here is an example that removes all the nodes with a z coordinate equal to 15. Continuing from the previous example:

```
# Remove the nodes with a z coordinate equal
to 15
Nodeset 1 remove node in surface all with
z_coord = 15
```

Assigning Names and Descriptions to Nodesets and Sidesets

Nodesets and sidesets can be assigned names and descriptions. Using names and descriptions is often more intuitive than using traditional integer IDs. When exporting a mesh as a DART artifact, names and descriptions are included in the metadata, making them available to DART metadata-enabled applications such as SIMBA. To give a name or description to nodeset or sideset, use one of the following commands:

```
{Nodeset|Sideset} <ids> Name "<new_name>"

{Nodeset|Sideset} <ids> Description "
<description>"
```

This command can also be used to define names and descriptions for Element Blocks.

Grouping Faces on a Surface into a Sideset

A sideset can be created from a subset of the faces on a given surface by using one of the following commands:

```
SideSet <sideset_id> Surface <id_range> Patch
Maximum <x> <y> <z> Minimum <x> <y> <z>

SideSet <sideset_id> Surface <id_range> Patch
Center <x> <y> <z> Radius <value> [Filter]
[Partition]

SideSet <sideset_id> Surface <id_range> Patch
Center <x> <y> <z> Outer_radius <value>
Inner_radius <value> [Filter] [Partition]

SideSet <sideset_id> Surface <id_range> Patch
Cylinder <axis_specification> Radius <rad>
[Filter] [Partition]

SideSet <sideset_id> Surface <id_range> Patch
Cylinder <axis_specification> Outer_radius
<rad> Inner_radius <rad> [Filter] [Partition]
```

These commands place only the faces meeting the specified criteria into the sideset.

- Using the **maximum** and **minimum** options will include all faces on the surface whose centroid falls within the axis-aligned box defined by the maximum and minimum points.
- Using the **center** and **radius** options will include all faces on the surface whose centroid falls within the sphere defined by center and radius.
- Using the **center**, **outer_radius**, and **inner_radius** options will include all faces on the surface whose centroid falls within the sphere defined by center and outer_radius, but excluding those faces whose centroid falls within the sphere defined by center and inner_radius. In other words, a face will be included if the distance between the face and the center point is between inner_radius and

outer_radius.
- Using the **cylinder** option will include all faces whose centroid falls within a cylinder of infinite length with the given axis and radius. The axis is specified as described in Specifying an Axis. Using the optional inner_radius will exclude those faces whose centroid is closer to the axis than the specified inner_radius.

Normally, these commands place the individual elements into the sideset. If the mesh on the surface is deleted, the elements will be removed from the sideset. If the surface is then remeshed, new elements will NOT automatically be added to the sideset. This is usually the intended behavior.

If the **filter** option is included, only a single connected set of elements is added to the sideset. If the shape of the surface is such that multiple disconnected sets of elements fall within the specified spherical or cylindrical region, the filter option will limit the faces added to the sideset to the one set closest to center.

Using the **partition** option changes this behavior. The partition option causes the surface to be split, based on the faces included in the patch. The newly created patch surface will be added to the sideset instead of the individual elements. If the mesh is deleted and a new mesh is generated, the new mesh on the patch surface will automatically be included in the sideset, just as occurs with other geometric entities assigned to sidesets.

Note that the sideset patch commands work with both triangular and quadrilateral faces.

Grouping elements in voids and enclosures

The **sideset start enclosure** command creates sidesets of monotonically increasing ID numbers containing the elements comprising the watertight skin of the input elements. When there's a 'void' in the middle of the elements, a region devoid of elements, though still enclosed by elements, this enclosed region will also have a sideset defined on the skin of the enclosed region.

```
Sideset Start <id> Enclosure {Volume|Hex|Tet}
<range>
```

The start id is the id of the sideset at which to start. The ID numbers will increase monotonically unless there is a conflicting ID number. The command will add as many sidesets as there are fully continuous regions or tris or faces in the input group. This function can be particularly helpful for calculations for radiation enclosures.

Deleting Nodesets and Sidesets

All Nodesets, Sidesets and Blocks may be deleted from the model using the following command:

```
Reset Genesis
```

To remove only nodesets or sidesets, the following may be used:

```
Reset Nodeset

Reset Sideset
```

To remove a specific nodeset or sideset, use:

```
Delete Nodeset <nodeset_id_range>

Delete Sideset <sideset_id_range>
```

Renumbering Nodesets and Sidesets

The nodeset and sideset renumber commands give the user the ability to renumber these entities to fit the user's needs. The command is:

```
{Nodeset|Sideset} <id_range> renumber start_id
<id> [uniqueids]
```

Example:

Assume:

sideset ids: 1, 2, 4, 6, 10

sideset all renumber start 30

After renumbering:

sideset ids: 30, 31, 32, 33, 34

The **id_range** must specify existing nodesets or sidesets, respectively, or the command will fail.

The new ids to be assigned cannot contain the id of an existing nodeset (when renumbering nodesets), or an existing sideset (when renumbering sidesets).

For example, given sidesets with ids 100, 105, 106, and 109, the command

```
sideset all renumber start_id 102
```

would attempt to renumber the sideset ids to 102, 103, 104, and 105. Since sideset 105 already exists, the command will fail.

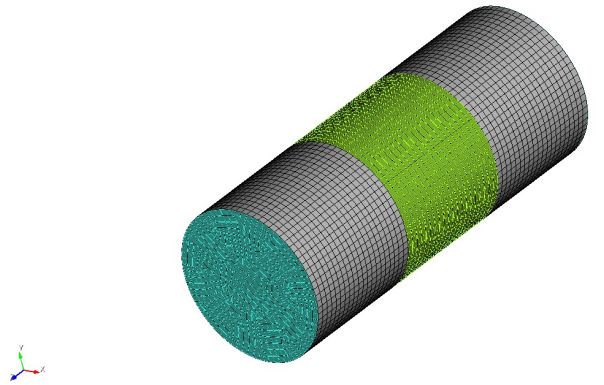
When the **uniqueids** option is specified, the new ids to be assigned cannot contain the id of an existing nodeset OR an existing sideset OR an existing block. For example, given sidesets with ids 100, 105, 106, and 109, and given blocks with ids 201, 202, and 203, the command

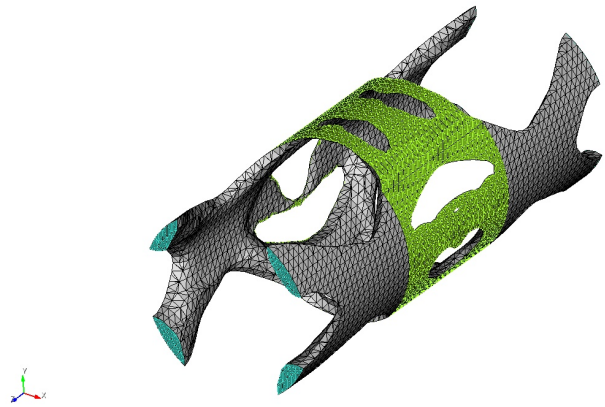
```
sideset all renumber start_id 200 uniqueids
```

would attempt to renumber the sideset ids to 200, 201, 202, and 203. While this does not conflict with existing sideset ids, it does conflict with the existing block ids and so the command will fail.

Transfer Nodesets and Sidesets

The capability shown in the below figure automates the transfer of sidesets and nodesets from the source mesh (e.g. design domain model) to the target mesh (e.g. topology optimized shape). Transfer of sidesets and nodesets was a user-intensive, error-prone, bottleneck as users were required to go through the painful step of manually selecting elements to define equivalent sidesets on the target mesh.





The commands to automatically transfer nodesets and sidesets are given below:

```
transfer nodeset <ids> onto {tri <ids> |face <ids>
|tet <ids> |hex <ids> } [tolerance <value> ]

transfer sideset <ids> onto {tri <ids> |face <ids>
|tet <ids> |hex <ids> } [tolerance <value> ] [log]
```

tri <ids> and face <ids> : if the target mesh is a surface mesh, use tri or face option.

tet <ids> and hex <ids>: this option can be used and the boundary skin mesh of the volumetric mesh will be extracted automatically to associate skin elements and nodes to the sidesets and nodesets, respectively.

tolerance <value>: The tolerance option must be used only if the default smart local tolerance doesn't give the desired results.

log : The log option outputs the area of sidesets in source and target meshes and the ratio of the areas. The below table shows an example output. For example, the area_ratio can be used to adjust the pressure boundary conditions to maintain same forces.

Log output of transfer sideset command

sideset_id	old_area ID	new_area	area_ratio ID
1	56.4843	4.8099	0.09
2	94.2209	68.4337	0.73

The below Python script Illustrates transfer of sidesets and nodesets from a source mesh to a target mesh using the existing and the new commands

```
#!python
import cubit
cubit.init(['cubit','-nojournal'])
```

```
# STEP 1: import the source mesh as mesh based geometry
(MBG); delete mesh and blocks
cubit.cmd('import mesh geometry "design_domain_mesh.exo"
feature_angle 135.00 merge ')
cubit.cmd('del mesh')
cubit.cmd('del block all')
# STEP 2: import the target mesh as a free mesh
cubit.cmd('import mesh "optimized_mesh.exo" no_geom')
# STEP 3: transfer sideset and nodeset from the source mesh to
the target mesh
cubit.cmd('transfer sideset all onto tet all')
cubit.cmd('transfer nodeset all onto tet all')
# STEP 4: export optimized mesh with sidesets
cubit.cmd('export mesh \'optimized_mesh_with_bc.exo\' block all
sideset all nodeset all')
```

Displaying Nodesets and Sidesets

Nodesets and Sidesets can be viewed individually through CUBIT by employing the following commands:

```
Draw NodeSet <nodeset_id_range> [Color
<color_spec>] [add]

Draw SideSet <sideset_id_range> [Color
<color_spec>] [add]
```

Nodeset and Sideset colors can also be changed using the following commands:

```
Color NodeSet <nodeset_id_range>
{color|Default}

Color SideSet <sideset_id_range>
{color|Default}
```

Nodeset Associativity Data

Nodesets can be used to store geometry associativity data in the Exodus II file. This data can be used to associate the corresponding mesh to an existing geometry in a subsequent CUBIT session. This functionality can be used either to associate a previously-generated mesh with a geometry (See Importing an Exodus II File), or to associate a field function with a geometry for adaptive surface meshing (See Adaptive Meshing).

The commands to control and list whether associativity data is written or read from an Exodus II files are the following:

```
List Import Mesh NodeSet Associativity

List [Export Mesh] NodeSet Associativity

List [Export Mesh] NodeSet Associativity
Complete

set Import Mesh NodeSet Associativity [ON|off]

[set] [Export Mesh] NodeSet Associativity
[on|OFF]

[set] [Export Mesh] NodeSet Associativity
Complete [On|OFF]
```

Associativity data is stored in the Exodus II file in two locations. First, a nodeset is written for each piece of geometry (vertices, curves, etc) containing the nodes owned for that geometry. Then, the name of each geometry entity is associated with the corresponding nodeset by writing a property name and designating the corresponding nodeset as having that property. Nodeset numbers used for associativity nodesets are

determined by adding a fixed base number (depending on the order of the geometric entity) to the geometric entity id number. The base numbers for various orders of geometric entities are shown in the following table. For example, nodes owned by curve number 26 would be stored in associativity nodeset 40026.

Table 1. Nodeset ID base numbers for geometric entities

Geometric Entity	Base Nodeset ID
Vertex	50000
Curve	40000
Surface	30000
Volume	20000

Instead of storing just the nodes owned by a particular entity, nodes for lower order entities are also stored. For example, the associativity nodeset for a surface would contain all nodes owned by that surface as well as the nodes on the bounding curves and vertices.

Equation-Controlled Distribution Factors

By default, distribution factors on nodesets or sidesets are written with a constant value of "1" at each node. It is also possible to vary the distribution factor for each node in a nodeset or sideset, using an equation to control the value of the distribution factor at each node. To do so, an equation must first be defined using the command:

```
Create Equation "<expression>" name "<name>"
```

where **expression** is any mathematical expression which evaluates to a single number, and **name** is the name by which this equation will be known. The expression is written using aprepro syntax, with a few differences from the use of APREPRO in its usual context.

1. The expression as a whole is not wrapped in curly braces "{" and"}".
2. The expression may include any of the following pre-defined variables:

x - The x-coordinate of the current node
y - The y-coordinate of the current node
z - The z-coordinate of the current node
n - The CUBIT ID of the current node. This is the ID of the node in CUBIT, which may not be the same as the node's ID in the Exodus II file.

For example, to define an equation which varies from -10 to 10 based on the sine of the node's x_coordinate:

```
Create Equation "10*sin(x)" Name "my_equation"
```

Once an equation has been defined, it can be applied to a nodeset or sideset:

```
Nodeset <id> Distribution Equation "<equation_name>"

Sideset <id> Distribution Equation "<equation_name>"
```

For example, to apply the equation created earlier to nodeset 10:

```
Nodeset 10 Distribution Equation "my_equation"
```

When nodeset 10 is written to an Exodus II file, "my_equation" will be evaluated once for each node in the nodeset, with the values of x, y, z, and n set to appropriate values for the node. The result is used as the distribution factor for that node.

Here is a complete example that writes out the distribution factors 0.0,

0.5, and 1.0 for the 3 nodes on the curve:

```
# Create a straight line from (0,0,0) to (1,0,0)
create vertex 0 0 0
create vertex 1 0 0
create curve vertex 1 2
# Mesh with 3 nodes
curve 1 interval 2
mesh curve 1
# Create a block and a nodeset
block 1 curve 1
nodeset 1 curve 1
# Define an equation and apply it to the nodeset
create equation "x" name "simple_eq"
nodeset 1 distribution equation "simple_eq"
# Write the mesh
export mesh "temp.g" overwrite
```

Here is another complete example that varies the distribution factors for sideset 20 from zero to 1, depending on the node's x-coordinate. The sideset is applied to sides of HEX20 elements, so each element side has 8 different distribution factors.

```
# Mesh a cube
brick x 10
mesh volume 1
# Create a block of 20-noded hexes
block 1 volume 1
block 1 element type hex20
# Apply a sideset to be used for a variable pressure
sideset 20 surface 1
# Define an equation and apply it to the sideset
create equation "(x+5)/10" name "zero_to_one"
sideset 20 distribution equation "zero_to_one"
# Write the mesh
export mesh "temp.g" overwrite
```

Note that distribution equations only affect Exodus II output. Equations are currently ignored for other mesh file types.

See APREPRO in the appendix.

Nodesets/Sidesets/Blocks Behavior with Geometric Entity Copy

The below commands can be used to set the behavior of bc (boundary contitions of nodesets/sidesets/blocks) propagation when a geometric entity is copied. The default **OFF** option specifies that any bc containing the copied geometry and/or mesh of the copied geometry will not be contained in a bc. The **use_original** option will add the new, copied geometry and/or mesh into the existing bc. The **on** option will create new bcs, containing the copied geometry and/or mesh, mirroring the original bcs.

set copy_nodeset_on_geometry_copy [on | OFF| use_original]

set copy_sideset_on_geometry_copy [on | OFF| use_original]

set copy_block_on_geometry_copy [on | OFF| use_original]

CUBIT Initial Conditions

In CUBIT, initial conditions can be applied to nodesets. CUBIT supports the following types of initial conditions: displacement, velocity, acceleration, temperature, and generic field. For now, initial conditions are only supported by CUBIT's Abaqus exporter. The commands to create an initial condition are:

```
Create initialcondition [id] type temperature
[name <'name'>]   [{add|on} nodeset
<entity_list>] [value <val>]

Create initialcondition [id] type displacement
[name <'name'>]   [{add|on} nodeset
<entity_list>]   [dof {1|2|3|4|5|6} {value
<value>|off}]

Create initialcondition [id] type velocity
[name <'name'>]   [{add|on} nodeset
<entity_list>]   [dof {1|2|3|4|5|6} {value
<value>|off}]

Create initialcondition [id] type acceleration
[name <'name'>]   [{add|on} nodeset
<entity_list>]   [dof {1|2|3|4|5|6} {value
<value>|off}]

Create initialcondition [id] type field [name
<'name'>]   [{add|on} nodeset <entity_list>]
[variable <n> value <val>
```

For most of the initial conditions, only two pieces of data are required: a list of nodesets this IC is applied to, and an initial value. Optionally, a name can be specified for the initial condition. To modify an initial condition, replace the word “create” with the word “modify.” If modifying an IC, the IC’s ID must be passed in so CUBIT knows which IC you are modifying. Example:

Modify initialcondition 3 value 1.23

Use this command to list the information about a set of initial conditions:

List initialcondition <id_list>

Use this command to delete a set of initial conditions:

Delete initialcondition <id_list>

Using Constraints

Constraints couple the motion of a set of nodes to the motion of a reference node. Rigid bodies and kinematic constraints do exactly this for blocks and sidesets, respectively. A distributing constraint allows users to average the constrained motion of a sideset by using weight factors to control force transmission (to be specified outside of CUBIT). A tie constraint can be used to tie the elements of one sideset to the elements of another. Currently, only the Abaqus Exporter supports this type of constraint.

Note that as of CUBIT 13.0, constraints are supported by the Abaqus Importer/Exporter only. Contact the CUBIT support team if support in additional file formats is needed.

To create a constraint, use one of the following commands:

```
Create Constraint {Kinematic|Distributing}
[name '<name>'] [vertex|node] <id> sideset <id>

Create Constraint Rigidbody [name '<name>']
[vertex|node] <id> block <id>

Create Constraint Tie [name '<name>'] primary
sideset <id> secondary sideset <id>
```

A constraint's name, dependent object, and independent object can be changed using the following commands:

```
Modify Constraint <id|name> [name '<name>']
[vertex|node] <id> sideset <id>

Modify Constraint <id|name> [name '<name>']
[vertex|node] <id> block <id>

Modify Constraint <id|name> [primary sideset
<id>] [secondary sideset <id>]
```

Constraints can be listed or deleted using the following commands:

```
List Constraint <id_range>

Delete Constraint <id_range>
```

Cubit Boundary Conditions

- Sets
- Restraints
- Loads
- Contacts
- CFD Boundary Conditions
- Miscellaneous commands
- CUBIT Initial Conditions

In CUBIT, boundary conditions are applied to sidesets or nodesets. Sidesets and nodesets can contain geometry or mesh. This means that models can be remeshed without worrying about losing boundary condition data if the boundary condition is applied to a geometry-based sideset/nodeset.

The sideset/nodeset used by a boundary condition will be visible to the user, and the user can modify the sideset/nodeset separately from the boundary condition. Sidesets/nodesets can be assigned to (or removed from) a boundary condition at any time.

Boundary conditions are broken into four groups: Restraints, loads, contact, and cfd. Each restraint that is created will belong to a restraint set, each load will belong to a load set, and each contact definition will belong to a contact set. A boundary condition set consists of any number of restraints, contact pairs, and loads. CFD boundary conditions do not belong to boundary condition sets.

Table 1: Overview of boundary condition entities available in Cubit

Entity	Description and scope
Acceleration	Creates an acceleration boundary condition (acts on a body, volume, surface, curve, or vertex)
Velocity	Creates a velocity boundary condition (acts on a body, volume, surface, curve, or vertex)
Boundary Condition Set	Creates a BC set (contains restraint, load and contact sets)
Contact Region	Creates a contact region between two surfaces or two curves
Contact Pair	Creates a contact pair between two previously defined contact regions
Displacement	Creates a displacement boundary condition (acts on a body, volume, surface, curve or vertex)
Temperature	Create a temperature boundary condition (acts on a surface, curve or vertex)
Force	Creates a force boundary condition (acts on a surface, curve or vertex)
Pressure	Creates a pressure boundary condition (acts on a surface or curve)
Heat flux	Creates a heat flux boundary condition (acts on a surface or curve)
Inlet Velocity	Creates an inlet velocity boundary condition (acts on a surface)
Inlet Pressure	Creates an inlet pressure boundary condition (acts on a surface)
Inlet Massflow	Creates an inlet massflow boundary condition (acts on a surface)
Outlet Pressure	Creates an outlet pressure boundary condition (acts on a surface)
Farfield Pressure	Creates a farfield pressure boundary condition (acts on a surface)

Symmetry	Creates a symmetry boundary condition (acts on a surface)

CFD Boundary Conditions

CUBIT can export models to the Fluent mesh format and CNGS unstructured mesh format for CFD analysis. CUBIT also supports defining the below CFD boundary conditions. Only the region on which the BC acts can be defined in CUBIT. The data associated with each boundary condition (pressure, velocity, mass values, etc.) is not defined within CUBIT and must be assigned using a CFD model editor, such as Fluent.

```
create cfd_bc [id]
{axis|exhaustfan|fan|inletvent|intakefan|interface|
interior|massflowinlet|outflow|outletvent|periodic|periodicshadow|
porousjump|pressurefarfield|pressureinlet|pressureoutlet|radiator|
supersonicinflow|supersonicoutflow|symmetry|velocityinlet|wall}
[name <'name'>] [{add|on} {sideset|surface}
<entity_list>
```

The following shows the commands for modifying, deleting, and lising CFD boundary conditions.

```
modify cfd_bc [id] [name <'name'>] [{add|remove}
{sideset|surface} <entity_list>

delete cfd_bc {<ids>|'<string>'}

list cfd_bc {<ids>}
```

Using Contact Surfaces

- Contact Region
- Contact Pair
- Auto-Contact Tool

The Contact Region

To define contact between two entities, Cubit requires each entity to be defined as a separate **contact region**. Each region can be made up of multiple 1D or 2D entities.

```
Create Contact Region [id] [Name <'name'>]
[{Add|On}
{Sideset|Surface|Curve|Face|Tri|Edge}
<entity_list>]

Modify Contact Region {id_list|'name'|All}
[Name <'name'>] [{Add|Remove}
{Sideset|Surface|Curve|Face|Tri|Edge}
<entity_list>]
```

The Contact Pair

```
create contact pair [id] [name <'name'>]
[primary contact region <id|'name'>]
 [secondary contact region <id|'name'>]
[friction <value>] [tolerance <value>]  [tied
{on|OFF}] [General <on|OFF> [Exterior
<on|OFF>]]

modify contact pair {id_list|'name'|all} [name
<'name'>] [primary contact region <id|'name'>]
[secondary contact region <id|'name'>]
[friction <value>] [tolerance <value>] [tied
{on|OFF}] [General <on|OFF> [Exterior
<on|OFF>]]
```

A contact pair is composed of two contact regions. One region will be the 'primary' surface, and the other will be the 'secondary.' 2D contact regions can not be mixed with 1D contact regions. The friction coefficient can also be included. The **tolerance** keyword is currently unused. Use the **tied** keyword to specify that the contact is to define tied contact between the two contact regions, essentially "gluing" the parts together. **Currently, this option is only available when using the Abaqus Exporter**.

The **General** keyword can be used to specify general (i.e. global) contact without specifying surfaces/curves to use as contact pairs. Currently, this keyword is only valid when exporting to Abaqus. If the **Exterior** keyword is used with the **General** keyword, then Abaqus will consider all exterior surfaces when determining contact regions. If the Exterior keyword is omitted, then the user must provide a primary contact region and/or a secondary contact region.

Auto-Contact Tool

With the auto-contact tool, Cubit can search for contact pairs and

With the auto-contact tool, Cubit can search for contact pairs and

automatically set up all of the necessary contact regions and contact pairs.

```
Create Contact Autoselect
[{Volume|Surface|Curve} <ids>] [Primary Volume
<id>] [Maxgap <value>] [Curve_Contact]
```

The optional geometry list can be used to limit Cubit's search to only a subset of entities. If this list is omitted, all bodies in the model will be searched. The optional **primary volume** keyword can be used to tell Cubit which volume should be used as the primary contact region. If this keyword is omitted, the user will not have control over which volume is the primary region. The **maxgap** keyword can be used to control how Cubit searches for contact regions. This value is used as the maximum amount of gap that can exist between two surfaces and be identified as a contact region. If this keyword is omitted, the geometry tolerance is used. The **curve_contact** keyword can be used to indicate the model requires curve contact as opposed to surface contact.

Using Loads

- Force
- Pressure
- Heat Flux
- Convection

Forces

```
Create Force [id] [Name <'name'>] [ {Add|On}
{Nodeset|Surface|Curve|Vertex|Face|Tri|Edge|Node}
<entity_list>] [Force Value <val>] [Moment
Value <val>] [Direction { direction_options}]

Create Force [id] [Name <'name'>] [ {Add|On}
{Nodeset|Surface|Curve|Vertex|Face|Tri|Edge|Node}
<entity_list>] [ Vector <val> <val> <val>
<val> <val> <val>]

Modify Force {id_list|'name'|all} [Name
<'name'>] [ {Add|Remove}
{Nodeset|Surface|Curve|Vertex|Face|Tri|Edge|Node}
<entity_list>] [Force Value <val>] [Moment
Value <val>] [Direction { direction_options}]

Modify Force {id_list|'name'|all} [Name
<'name'>] [ {Add|Remove}
{Nodeset|Surface|Curve|Vertex|Face|Tri|Edge|Node}
<entity_list>] [ Vector <val> <val> <val>
<val> <val> <val>]
```

A CUBIT user has the ability to create forces on 0D, 1D, and 2D entities. A force can be created using the direction syntax (see Specifying Direction). If the vector keyword is used, the first three values are the force components, and the last three values are the moment components.

The use of the **force** and **moment** keywords specify the type of load. If both a force and a moment are to be applied, first create one of them, then modify it to add the other. Note that every instance of a **force** or **moment** keyword must have an accompanying **value** keyword.

Regarding **force** and **moment** keywords, the following detail may be helpful:

A user may create a force and moment at the same time, but can only specify a direction once. If the force and moment have the same unit vector, it will be successful. If a users wants to create a force in the direction 1,2,3 and a moment in the direction 1,0,0, the user will have to create one, then add the other by modifying it.

Using Pressure

```
Create Pressure [id] [Name <'name'>] [{Add|On}
{Sideset|Surface|Curve|Face|Tri|Edge}
<entity_list>] [Magnitude <value>]
[TOP|Bottom] [PRESSURE|Totalforce]

Modify Pressure {id_list|'name'|all} [Name
<'name'>] [{Add|Remove}
{Sideset|Surface|Curve|Face|Tri|Edge}
<entity_list>] [Magnitude <value>]
[TOP|Bottom] [PRESSURE|Totalforce]
```

Cubit users can create pressure boundary conditions on 1D and 2D

CUBIT users can create pressure boundary conditions on 1D and 2D

entities. Positive surface pressures acting on solid elements are defined as pointing into the face of the elements. Pressures are always normal to the face. For shells and independent surfaces, a 'left-hand-rule' is employed. Point your left thumb at the surface in question. If the direction your fingers curl matches the direction of ascending vertex numbering, the direction of the pressure vectors will match the direction of your thumb.

Value

The **value** variable is the magnitude of the pressure boundary condition. If the user leaves this value blank, CUBIT will assign the pressure magnitude to zero (possibly a trivial case) and issue a warning. Typing a negative value will not flip the direction of the pressure arrows on the display; instead, the pressure magnitude will be negative.

Pressure and Total Force

The pressure and totalforce keywords are used to modify the pressure boundary condition. The pressure keyword is the default. All pressures applied with this keyword present (or with both of these keywords absent from the command string) are pure pressures. If the user enters the totalforce keyword, the pressure magnitude is divided by the area of the surface the pressure is acting on (or the length of the curve, for a curve pressure). In effect, the user is entering a force that is treated and exported as a pressure.

Top and Bottom

The **top** keyword (default) indicates the pressure will occur on the top of a shell element. Specifying **bottom** will cause the pressure to be applied to the bottom of the element.

Using Heat Flux

```
Create Heatflux [id] [Name <'name'>] [{Add|On}
{Sideset|Surface|Curve|Face|Tri|Edge}
<entity_list>] [Value <value>]

Create Heatflux [id] [Name <'name'>] [{Add|On}
{Sideset|Surface|Face|Tri} <entity_list>] [Top
<value> Bottom <value>]

Modify Heatflux {id_list|'name'|All} [Name
<'name'>] [{Add|Remove}
{Sideset|Surface|Curve|Face|Tri|Edge}
<entity_list>] [Value <value>]

Modify Heatflux {id_list|'name'|All} [Name
<'name'>] [{Add|Remove}
{Sideset|Surface|Face|Tri} <entity_list>] [Top
<value> Bottom <value>]
```

A CUBIT user may apply heat flux boundary conditions to 1D and 2D entities, including thin-shell elements.

Top and Bottom Values

Heat fluxes can be applied to thin-shell elements as well. The same rules apply to thin-shell heat fluxes as to thin-shell temperatures: thin-shell heat fluxes can only be applied to surfaces and heat flux boundary conditions cannot contain regular and thin-shell heat flux values (see journal below). However, thin-shell heat flux commands do not contain gradient or middle keyword options. Only top and bottom keywords are supported.

Using Convection

```
Create Convection [id] [Name <'name'>]
[{Add|On}
{Sideset|Surface|Curve|Face|Tri|Edge}
<entity_list>] [Surrounding {<value>| Top
<value> Bottom <value>} Coefficient {<value>|
Top <value> Bottom <value>}]

Modify Convection [id] [Name <'name'>]
[{Add|On}
{Sideset|Surface|Curve|Face|Tri|Edge}
<entity_list>] [Surrounding {<value>| Top
<value> Bottom <value>} Coefficient {<value>|
Top <value> Bottom <value>}]
```

A Cubit user can apply convection boundary conditions to 1D and 2D entities. Convection is a transport of thermal energy that is proportional to the difference between the surface temperature and the temperature of the surroundings.

Surrounding

The surrounding keyword specifies the temperature surrounding the entity with the convection boundary condition.

Coefficient

The coefficient keyword is a convection coefficient, in units of energy per length times time times temperature (i.e., [energy]/([length]*[time]*[temperature])).

Miscellaneous Boundary Condition Commands

- Delete
- List
- Draw
- Highlight

Delete

The **BC delete** keyword combination is used to delete boundary conditions. The current list of all entities that can be deleted using this command were shown in Table 1. Cubit currently has no 'undo' command to 'undelete' a boundary condition deletion.

```
Delete {bc_type} [<id-range>|All]

Delete Boundary Conditions
```

Every set (and boundary condition within them) can be deleted at once by typing **delete boundary conditions**. This command will delete all boundary conditions from your model.

List

The **List** keyword combination is used to list boundary conditions. The current list of all entities that can be listed using this command was shown in Table 1. Cubit's parser can evaluate boundary conditions given the entities they act on. For example, "List pressure in surface 1" will list all pressures that act on Surface 1.

```
List {bc_type} [<id-range>]

List Boundary Conditions
```

Every set (and boundary condition within them) may be listed at once by typing **list boundary conditions**. CUBIT will list the number of sets and individual boundary conditions in your model. This command will list the total number of each type of set and boundary condition, including boundary conditions that are not a part of a BC set.

Draw

```
Draw {bc_type} {<id-range>|all}[Add]
```

The **draw** keyphrase allows a CUBIT user to draw any type of boundary condition. This command will clear the graphics window of every part of the model except for the selected boundary condition. Using the **add** keyword will permit multiple boundary conditions to be drawn at the same time. Any combination of boundary conditions and entities that were valid for delete and list are also valid for draw.

Highlight

```
Highlight {bc_type} {<id-range>|All}
```

The **highlight** keyphrase allows a CUBIT user to highlight any boundary condition. Highlighting a boundary condition will turn it bright orange and the vectors defining it will thicken. The highlight command is similar to the draw command.

Using Restraints

- Displacement
- Acceleration
- Velocity
- Temperature

Displacements/Accelerations/Velocities

A CUBIT user has the ability to create displacement boundary conditions on most geometric entities found within Cubit.

```
Create Displacement [id] [Name <'name'>]
[{Add|On}
{Nodeset|Volume|Surface|Curve|Vertex|Hex|Tet|Face|Tri|Edge|Node}
<entity_list>] [DOF {All|{[1][2][3][4][5][6]}}
Fix <value>]
[SmallestCombine|Average|LargestCombine|OVERWRITE]

Modify Displacement {id_list|'name'|all} [name
<'name'>] [{Add|Remove}
{Nodeset|Volume|Surface|Curve|Vertex|Hex|Tet|Face|Tri|Edge|Node}
<entity_list>] [DOF {All|{[1][2][3][4][5][6]}}
{Fix <value>|Free}]
[SmallestCombine|Average|LargestCombine|OVERWRITE]
```

```
Create Acceleration [id] [Name <'name'>]
[{Add|On}
{Nodeset|Volume|Surface|Curve|Vertex|Hex|Tet|Face|Tri|Edge|Node}
<entity_list>] [DOF {All|{[1][2][3][4][5][6]}}
Fix <value>]
[SmallestCombine|Average|LargestCombine|OVERWRITE]

Modify Acceleration {id_list|'name'|all} [name
<'name'>] [{Add|Remove}
{Nodeset|Volume|Surface|Curve|Vertex|Hex|Tet|Face|Tri|Edge|Node}
<entity_list>] [DOF {All|{[1][2][3][4][5][6]}}
{Fix <value>|Free}]
[SmallestCombine|Average|LargestCombine|OVERWRITE]

Create Velocity [id] [Name <'name'>] [{Add|On}
{Nodeset|Volume|Surface|Curve|Vertex|Hex|Tet|Face|Tri|Edge|Node}
<entity_list>] [DOF {All|{[1][2][3][4][5][6]}}
Fix <value>]
[SmallestCombine|Average|LargestCombine|OVERWRITE]

Modify Velocity {id_list|'name'|all} [name
<'name'>] [{Add|Remove}
{Nodeset|Volume|Surface|Curve|Vertex|Hex|Tet|Face|Tri|Edge|Node}
<entity_list>] [DOF {All|{[1][2][3][4][5][6]}}
{Fix <value>|Free}]
[SmallestCombine|Average|LargestCombine|OVERWRITE]
```

A number of required and optional keywords make the BC create displacement command one of the more complicated of the boundary condition commands. These keywords will be examined individually in detail.

Degrees of Freedom

The **dof** keyword is the heart of this command. It specifies how to constrain the entity in question. The keyword is an abbreviation for ‘degree of freedom’. Typing the optional keyword all tells CUBIT that the entered command will encompass all six degrees of freedom. The degrees of freedom (1 - 6) are defined below in Table 2.

Table 2: CUBIT definitions of the six degrees of freedom.

DOF	Physical analog
1	x-translation
2	y-translation
3	z-translation
4	x-rotation
5	y-rotation
6	z-rotation

CUBIT will allow displacement commands to be applied upon between one and all six of the degrees of freedom. The degrees of freedom do not need to be entered in order. The command strings ' 1 2 3 4 5 6 ' '2 6 1 4 3 5' and 'all' will end with the same result.

Fixed or Free

The **fix** and **free** keywords tell CUBIT whether an entity's displacement defined by the dof keyword is to be enforced with a finite value or not. If the displacement is fixed, the entity will be constrained in the pre-specified degrees of freedom. A decimal number entered after the fix keyword will be the value of the enforced degree(s) of freedom. CUBIT allows the user to leave this value blank if the enforced displacement is to be zero, for convenience. However, entering '0' is still permitted. If a user wishes to remove a displacement from an entity, he or she should just delete it rather than trying to set all of the degrees of freedom to free.

Displacement Combinations

The **SmallestCombine**, **Average** and **LargestCombine** keywords deal with displacement combinations. These keywords only apply when a user is modifying an existing displacement boundary condition.

The **SmallestCombine** keyword will compare the existing displacement values with the current (residing on the command line) displacement values. The keyword will modify the displacement to the match the displacements dictated by the boundary condition that has the smallest absolute value. If the boundary condition with the smallest absolute value is the existing value, the displacement boundary condition will be unchanged. If the current boundary condition has a smaller absolute value than the existing displacement, the displacement boundary condition will be changed to incorporate the new values.

The **Average** keyword will average the existing displacement values with the current (residing on the command line) displacement values. Note that these averages are not continually updated (i.e., they are not weighted). If a user created a displacement boundary condition and constrained a degree of freedom to 10.0 and then constrained the same degree of freedom to 20.0 with the Average keyword, the new displacement value would be 15.0. But if a user constrained the same degree of freedom to 30.0, while using the Average keyword, the new displacement value would be 22.5 ([15+30]/2), not 20.0 ([10+20+30]/3).

The **LargestCombine** keyword will compare the existing displacement values with the current (residing on the command line) displacement values. The keyword will modify the displacement to the match the displacements dictated by the boundary condition that has the largest absolute value. If the boundary condition with the largest absolute value is the existing value, the displacement boundary condition will be unchanged. If the current boundary condition has a larger absolute value than the existing displacement, the displacement boundary condition will be changed to incorporate the new values.

When none of these keywords are specified, CUBIT will combine displacements in its default mode, Overwrite. The Overwrite keyword overwrites the entity's previous displacement boundary condition(s) with the displacement values specified in the command.

Temperature

CUBIT can create temperature boundary conditions on most geometric and mesh entities. The temperature boundary condition can also be applied to thin-shell elements.

```
Create Temperature [id] [Name <'name'>]
[{Add|On}
{Nodeset|Volume|Surface|Curve|Vertex|Hex|Tet|Face|Tri|Edge|Node}
<entity_list>] [Value <val>]

Create Temperature [id] [Name <'name'>]
[{Add|On}
{Nodeset|Volume|Surface|Curve|Vertex|Hex|Tet|Face|Tri|Edge|Node}
<entity_list>] [{ Top <val> Bottom <val> |
[Middle <val>] [Gradient <val>] } ]

Modify Temperature {id_list|'name'|all} [name
<'name'>] [{Add|Remove}
{Nodeset|Volume|Surface|Curve|Vertex|Hex|Tet|Face|Tri|Edge|Node}
<entity_list>] [Value <val>]

Modify Temperature {id_list|'name'|all} [name
<'name'>] [{Add|Remove}
{Nodeset|Volume|Surface|Curve|Vertex|Hex|Tet|Face|Tri|Edge|Node}
<entity_list>] [{ Top <val> Bottom <val> |
[Middle <val>] [Gradient <val>] } ]
```

The **value** keyword defines the amplitude (temperature). The other command options are discussed below

Top, Gradient, Middle, Bottom

The above keywords are only used for thin-shell elements (i.e., 2D entities). The valid combinations are limited to: top and bottom, middle and gradient, only gradient or only middle. It should be noted that temperature boundary conditions cannot contain regular and thin-shell temperature values.

Boundary Condition Sets

```
Create bcset [id] [name <'name'>] [After bcset
<id>] [{Add|Remove} {bc_type}  <id-range |
<with name 'name'> >] [analysistype
{STATIC|heat|dynamic|modal}]
[modal_max_frequency <value>]

Modify bcset {id_list|'name'|all} [name
<'name'>] [After bcset <id>] [{Add|Remove}
{bc_type}  <id-range | <with name 'name'> >]
[analysistype {STATIC|heat|dynamic|modal}]
```

*** ABAQUS Parameters ***

```
Modify bcset {id_list|'name'|all}
[max_step_increments <value>]
    [nonlinear_geometry <on|OFF>][perturbation
<on|OFF>][stabilize <on|OFF>]  [steadystate
<on|OFF>][modal_max_frequency <value>]

Modify bcset {id_list|'name'|all}
[initial_step_size <value>]     [step_period
<value>][min_step_size <value>]  [max_step_size
<value>][min_step_temperature_change <value>]

Modify bcset {id_list|'name'|all} [mass_scaling
<on|OFF>]   [mass_scaling_dt <value>]
[mass_scaling_factor <value>]
    [mass_scaling_type
<'uniform'|'BELOW_MIN'|'set_equal_dt'>]

Modify bcset {id_list|'name'|all} [restart
<on|OFF>][restart_overlay <on|OFF>]
 [{restart_frequency|restart_num_intervals}
<value>]

Modify bcset {id_list|'name'|all} [output_field
<on|OFF>]     [output_field_frequency <value>]
[output_history <on|OFF>]
    [output_history_frequency <value>]

Modify bcset {id_list|'name'|all} [el_file
<on|OFF>][el_file_frequency <value>]
 [node_file <on|OFF>][node_file_frequency
<value>]

Modify bcset {id_list|'name'|all} [el_print
<on|OFF>][el_print_frequency <value>]
 [node_print <on|OFF>][node_print_frequency
<value>]
```

*** NASTRAN Parameters ***

```
Modify bcset {id_list|'name'|all}
 {displacement_output <on|OFF>
{PLOT|print|punch|punchprint} {group <ALL|none|
<id>> }}

Modify bcset {id_list|'name'|all} {oload
<on|OFF> {PLOT|print|punch|punchprint} {group
<ALL|none|<id>> }}

Modify bcset {id_list|'name'|all} {mpcforces
<on|OFF> {PLOT|print|punch|punchprint}  {group
<ALL|none|<id>> }}

Modify bcset {id list|'name'|all}  {spcforces
```

```
<on|OFF> {PLOT|print|punch|punchprint}  {group
<ALL|none|<id>> }}

Modify bcset {id_list|'name'|all}  {stress
<on|OFF> {PLOT|print|punch|punchprint} {group
<ALL|none|<id>> }
    {CENTER|cubic|sgage|corner}
{VONMISES|maxs}}

Modify bcset {id_list|'name'|all}
 {element_strain_energy <on|OFF>
{PLOT|print|punch|punchprint}  {group
<ALL|none|<id>> } {AVERAGE|amplitude|peak}}
```

CUBIT can create BC sets, which is a group of previously defined loads, restraints and contact pairs. A BCSet is used to define a load case (analysis step) when writing out 3rd party analysis decks. A BCSet can be a static analysis set, a thermal analysis set, a modal analysis set, or a dynamic analysis set by specifying the **analysistype**. The **After** keyword can be used to define the order that the BCSets will be written when the model is exported.

Several solver-specific parameters can be set for a BCSet. For **ABAQUS**, parameters associated with *STEP, *STATIC, *DYNAMIC, *FREQUENCY, *HEAT TRANSFER, *MASS SCALING, *RESTART, *OUTPUT, *EL FILE, *NODE FILE, *EL PRINT, and *NODE PRINT can be modified. For **Nastran**, output requests can be defined for Displacement, Reaction Loads, MPC Forces, SPC Forces, Stress, and Element Strain Energy.

Boundary Layer Meshing

Boundary layer meshing is best accessed via the GUI.

To create a boundary layer:

1. On the Command Panel, click on **Mesh** and then **Boundary Layer**.
2. Click on the **Create** action button.
3. Select **System Assigned ID** or manually enter an ID.
4. On the **Settings** tab, enter the appropriate size for the first row in the boundary layer.
5. Enter a value for the **Growth Factor**.
6. Specify the **Number of Layer**.
7. Optionally select **Internal Continuity**.

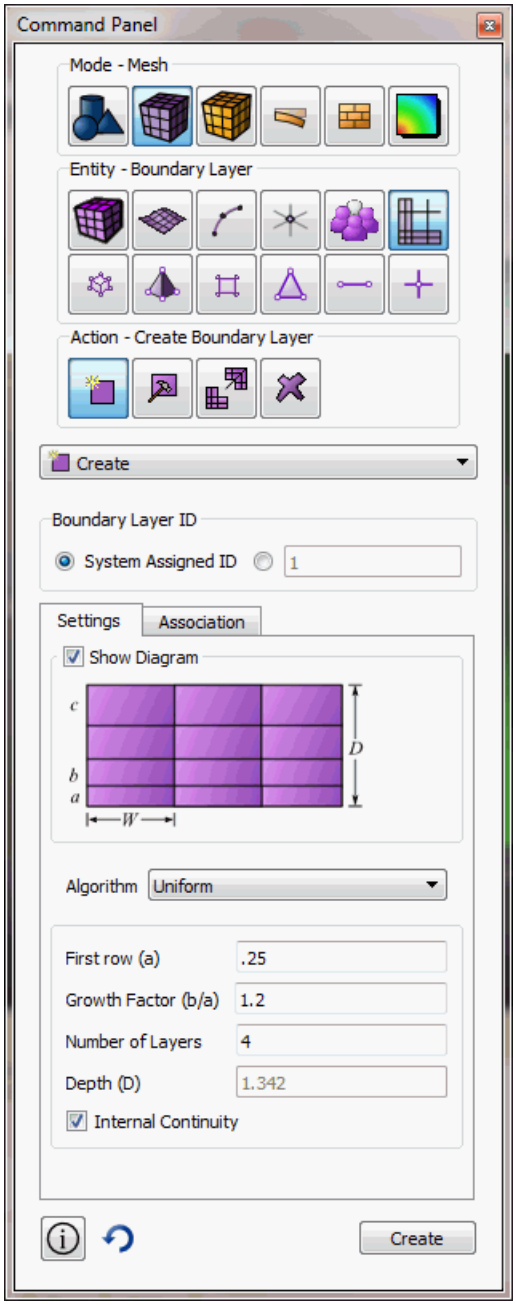


Figure 1 - Settings Panel

First row(a) -- the height of the first layer in the boundary layer

Growth Factor(b/a) -- the factor by which each layer grows

Number of Layers -- the number of layers that make up a boundary layer

Internal Continuity -- continuity flag for boundary layers. If on, all intersections are a side type.

8. On the **Association** tab, select **Curve** for 2D boundary layers or **Surface** for 3D boundary layers.

   For 2D boundary layers:

   1. In the **Curve ID(s)** field, enter the curve ID(s) where the boundary layer(s) begins.
   2. In the **Surface ID** field, enter the surface ID that contains the boundary layer(s).
   3. Click **Add** to add the curves to the boundary layer(s).
   4. Click **Create** to create the boundary layer(s).

   For 3D boundary layers:

   1. In the **Surface ID(s)** field, enter the surface ID(s) where the boundary layer(s) begins.
   2. In the **Volume ID** field, enter the volume ID that contains the boundary layer(s).
   3. Click **Add** to add the curves to the boundary layer(s).
   4. Click **Create** to create the boundary layer(s).

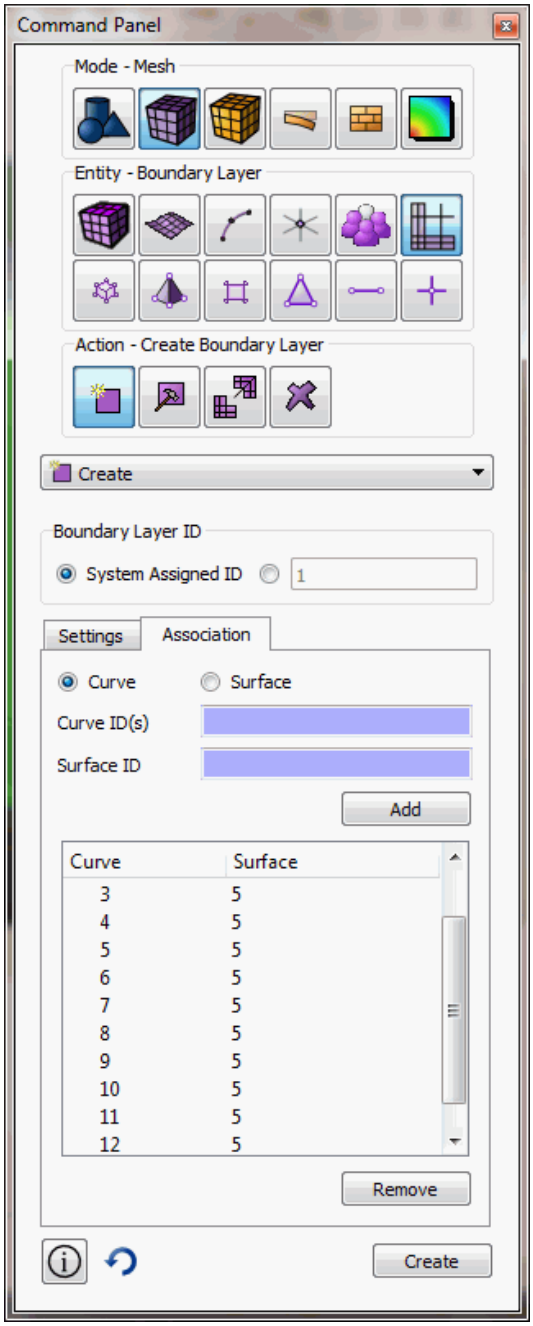


Figure 2 - Association Panel

For 2d boundary layers, a curve/surface pair is given to create a boundary layer starting from a curve and growing out on the given surface.

For 3d boundary layers, a surface/volume pair is given to create a boundary layer starting from a surface and growing out on the given volume.

Intersection Types

In some cases, the user may want to adjust the intersection types. This could be because the automatic intersection type is not desired, or because it is not workable due to ambiguity.

The four intersection types are:

- end - suitable for angles between 0 and 135 degrees.
- side - suitable for angles between 135 and 225
- corner - suitable for angles between 225 and 315
- reversal - suitable for angles 315 to 360

These intersection types may be set on a vertex/surface basis and on a curve/volume basis.

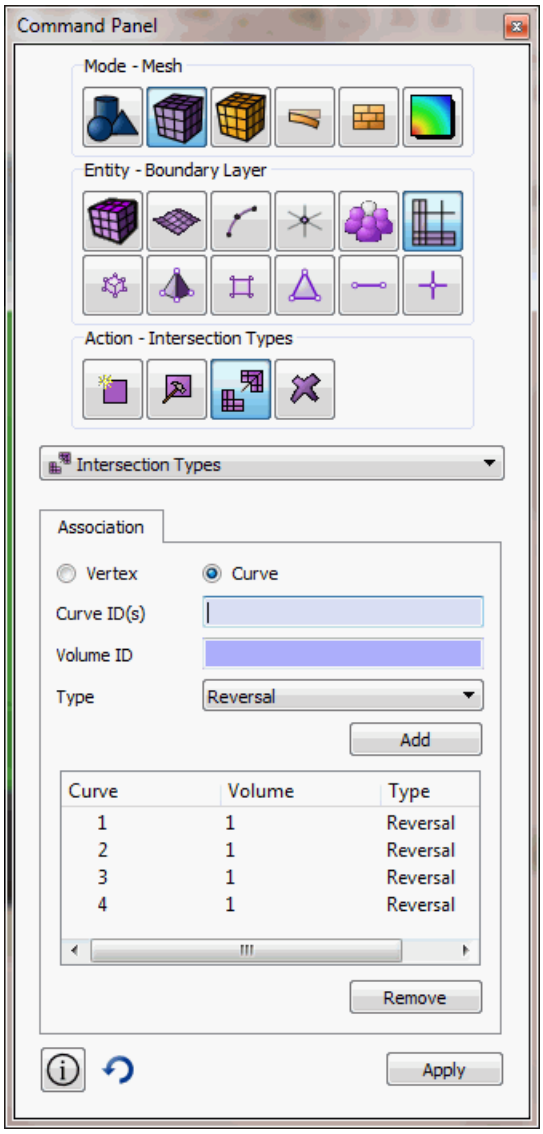


Figure 3 - Intersections Type Panel

Current Limitations

Not all combinations of intersection types and topology are supported for 3d cases. An end, corner, or reversal may not span multiple curves in a single volume. A possible workaround is to composite the curves to make a single curve.

Not all meshing schemes may be used in combination with boundary layers. In cases where it is not supported, the boundary layer will be ignored in mesh generation. It is supported with the following schemes:

- trimesh
- pave
- map
- submap
- sweep
- tetmesh.

Underlying Trelis Commands

```
Create Boundary_layer <id>

Delete Boundary_layer <id>
```

```
Modify Boundary_layer <id> add Curve
<id_range> Surface <id>

Modify Boundary_layer <id> remove Curve
<id_range> Surface <id>

Modify Boundary_layer <id> add Surface
<id_range> Volume <id>

Modify Boundary_layer <id> remove Surface
<id_range> Volume <id>

"*** Only three of the four parameters
should be specified ***

"***           (Height, Growth, Layer, or
Depth)          ***

Modify Boundary_layer <id> uniform Height
<double> Growth <double> Layers <double>
Depth <double>

Modify Boundary_layer <id> continuity
{yes | no}

set boundary_layer intersection volume
<id> curve <ids> type {end, side, corner,
reversal, default}

set boundary_layer intersection surface
<id> vertex <ids> type {end, side,
corner, reversal, default}
```

Sample Journal Files

Example 1

```
reset

create surface rectangle  width 10 height 3

create surface circle radius 5 zplane

surf 2 move z -10

create volume loft surface 1 2

delete surf 1 2

create boundary_layer 1

modify boundary_layer 1 uniform height 0.1 growth 1.2 layers 4

modify boundary_layer 1 add surface 4 volume 3 surface 5 volume
3 surface 6 volume 3 surface 7 volume 3

set boundary_layer intersection volume 3 curve 10 type side

set boundary_layer intersection volume 3 curve 12 type side

set boundary_layer intersection volume 3 curve 14 type side

set boundary_layer intersection volume 3 curve 16 type side

mesh vol 3
```

Example 2

```
reset

create surface rectangle width 2

cylinder radius 0.1 z 0.1

cylinder radius 0.02 z 0.1

section volume 2 xplane reverse

section volume 3 xplane

volume 3 move x 0.5

create volume loft surface 12 8

unite volume 2 3 4

volume 2 copy

volume 5 scale 0.3 0.3 1

volume 5 rotate -10 about z

volume 5 move x 0.55 y -0.1

volume 2 5 move x -0.25

imprint all

delete vol 2 5

surf all scheme trimesh

group "profile" add curve in surface 30 29

create boundary_layer 1

modify boundary_layer 1 uniform height 0.002 growth 1.2 layers 6

modify boundary_layer 1 add curve in group with name "profile"
surface 28

curve in group with name "profile" size 0.02

surface 28 size 0.3

surface 28 sizing function linear neighbor 2

mesh surface 28
```

Step-By-Step Tutorials

The purpose of this chapter is to demonstrate the capabilities of CUBIT for finite element mesh generation as well as provide a brief tutorial on the use of the software package. This chapter is designed to demonstrate step-by-step instructions for generating a simple mesh on a perforated brick.

The following activity demonstrates the basics of using CUBIT to generate and mesh a geometry. By following these steps, you will become familiar with the basics of the command-line and GUI interfaces without stopping for detailed explanations. All the commands introduced in this tutorial are documented in subsequent chapters on this manual.

Here are a few tips for the examples in the tutorial:

- Focus on the instructions preceded with "Step" numbers. These walk you through a series of explicit activities that describe how to complete the task.
- Refer to the screen shots and other pictures that show what you should see on your own monitor as you progress through the tutorial.
- In this tutorial, command line options will look like this:

```
cubit> <Your commands go here>
```

If you do not have the Graphical User Interface (GUI) version of CUBIT, follow the steps in the right column below, otherwise, proceed through the steps on the left:

GUI	CL	
Overview	Overview	
Step 1	Step 1	Beginning Execution
Step 2	Step 2	Creating the Brick
Step 3	Step 3	Creating the Cylinder
Step 4	Step 4	Adjusting the Graphics Display
Step 5	Step 5	Forming the Hole
Step 6	Step 6	Setting Interval Sizes
Step 7	Step 7	Surface Meshing
Step 8	Step 8	Volume Meshing
Step 9	Step 9	Inspecting the Model
Step 10	Step 10	Defining Boundary Conditions
Step 11	Step 11	Exporting the Mesh

Additional Tutorials

ITEM Tutorial - A tutorial on the new ITEM wizard.

Power Tools GUI Tutorial - A tutorial on geometry decomposition and cleanup using the Power Tools on the new CUBIT GUI.

Decomposition Tutorial - A series of webcutting hints and suggestions for creating sweepable volumes on various models.

Geometry Cleanup Process Flow - A flowchart on geometry cleanup and defeaturing.

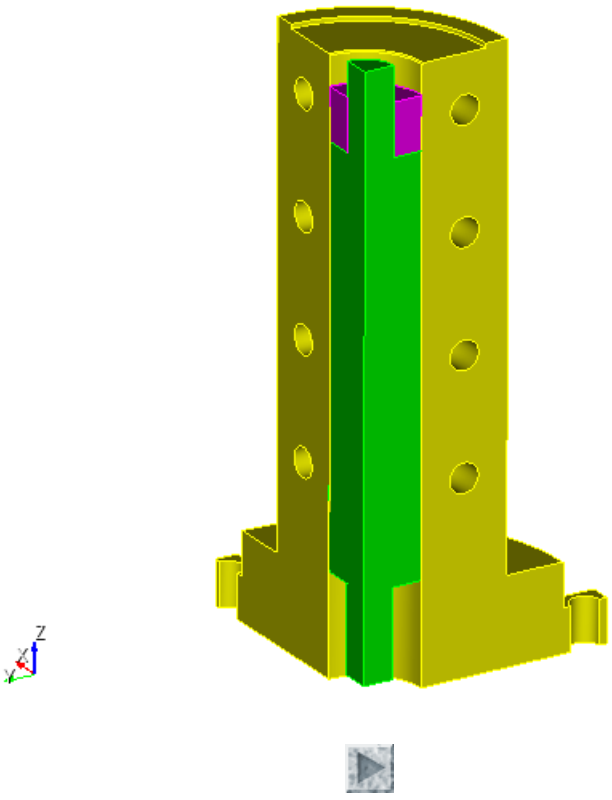
ITEM Tutorial

Overview

This tutorial will demonstrate the use of the Immersive Topology Environment for Meshing (ITEM) to create a finite element mesh. ITEM is a wizard-like environment that guides a user through a typical mesh generation process from import to export. Each page in the ITEM workflow is linked to other pages, and one can easily move around in the environment by clicking on links on each page. Most of the pages contain diagnostic tools that search the model for specific geometry or mesh-related issues. Clicking on a entity in the ITEM output window will then generate specific command suggestions to resolve the problem. The following topics are included in this tutorial:

- Importing a geometry
- Creating the finite element model
- Removing small features
- Using merge tolerance to find and fix small misalignments
- Decomposing a model
- Generating a mesh
- Validating the mesh
- Creating boundary conditions
- Exporting the mesh

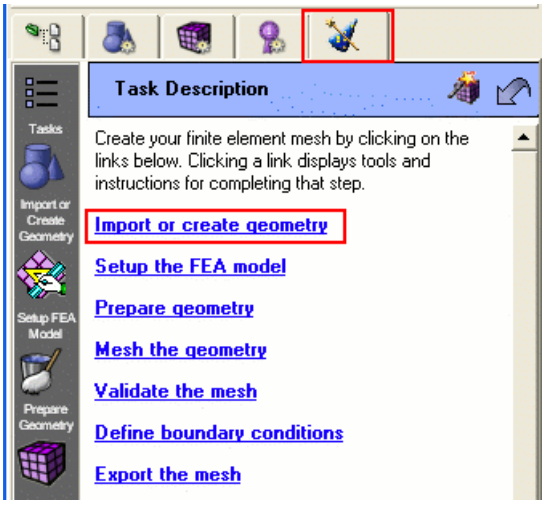
The model that will be used is shown below.



ITEM Tutorial Step 1

Step 1: Import Geometry

- Click on **ITEM tab** on the power tools panel on the left hand side of the screen.
- Click on **Import or create geometry**



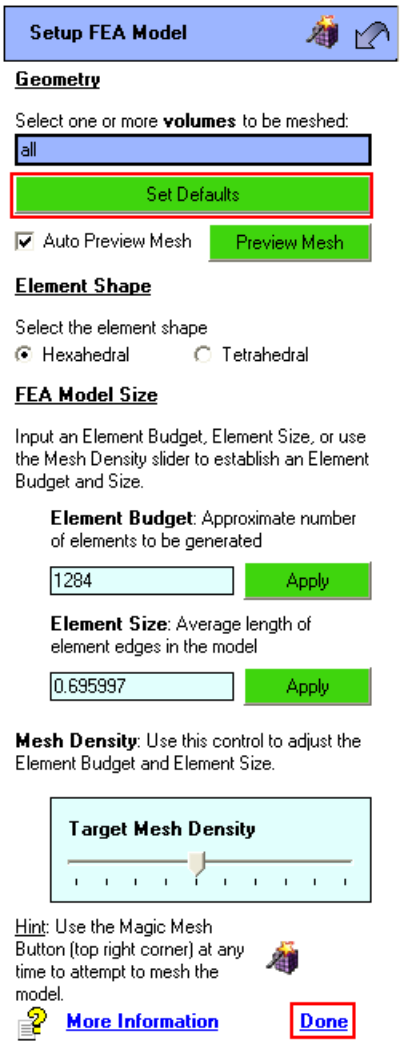
- Click on **Import a CAD model**
- Click on **Acis**
- Browse for the "item_tutorial.sat" file and import. The file may be in the Cubit directory/folder under 'tutorials.'
- Leave the default options selected on the import dialog box.
- Click **Done**
- Click **Done**

In most cases, clicking the Done button also acts like a "Back" button. Clicking Done will return the user to the previous page while preserving any changes made on that page.

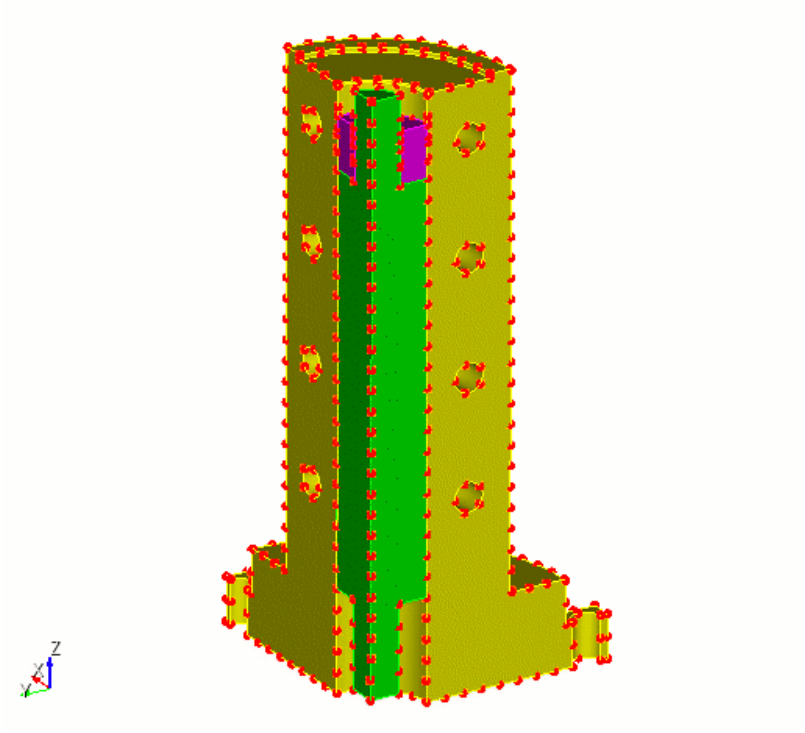
ITEM Tutorial Step 2

Step 2: Setup The FEA Model

- Click on **Setup the FEA model**
- Click on **Set Defaults** (this determines a default mesh size and populates the fields accordingly-it also previews the mesh if the **Auto Preview Mesh** checkbox is checked)



- Examine the preview mesh

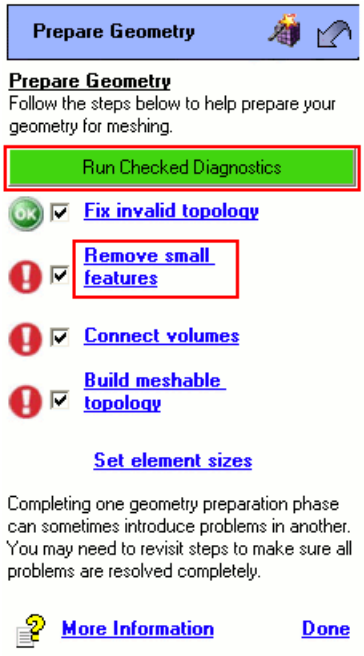


- Click **Done** to return to the previous page.
- Click **Done** again to return to the main ITEM task page

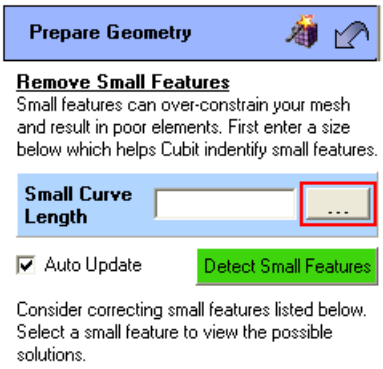
ITEM Tutorial Step 3

Step 3: Remove Small Features

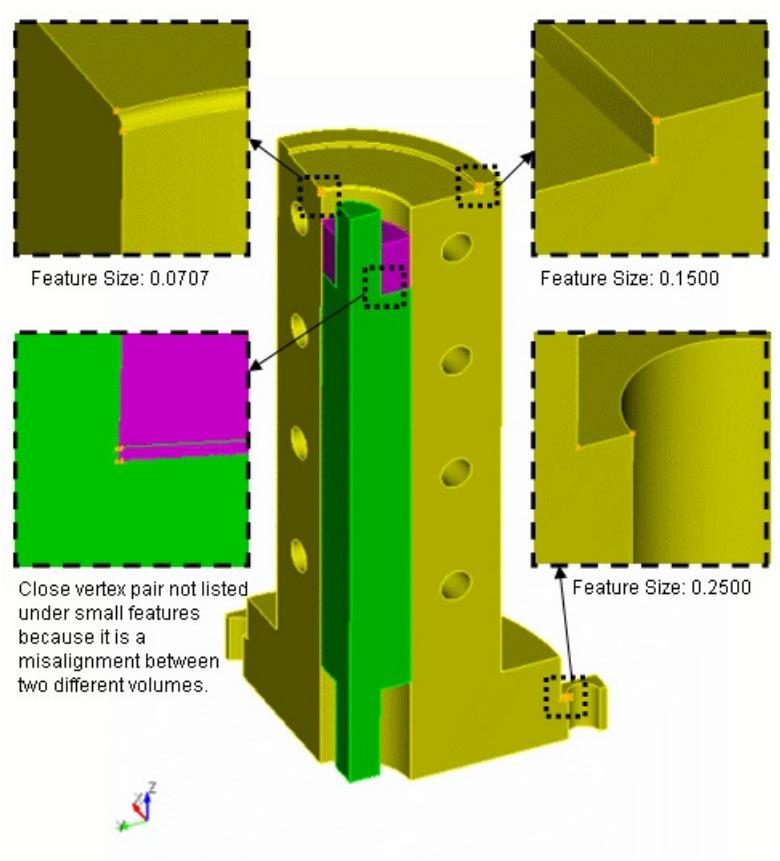
- Click **Prepare Geometry**
- Click **Run Checked Diagnostics**: The red exclamation point indicates a problem area that may need to be addressed



- Click on **Remove small features**
- Click on the button with three dots (...) next to the small curve length input box. This will open the small feature size panel. The small feature size panel is a tool to help the user find an appropriate small feature size. The smallest feature size is the smallest detail in the model that needs to be resolved in the mesh. It is also used for several calculations and diagnostic tools in the geometry ITEM panels. Any feature that is smaller than the smallest feature size will be flagged as a small curve or surface, and will need to be removed.

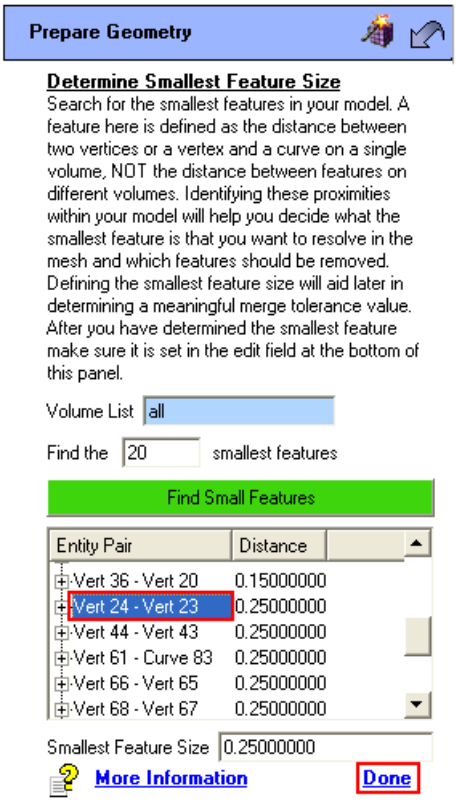


- Click on **Find small features**
- Change the Number to **20**
- Click on **Find Small features** again. One important thing to note is that the smallest features will only include vertex-vertex and vertex-curve pairs on the same volume. The small gaps and misalignments between volumes will be addressed during the imprinting and merging step.

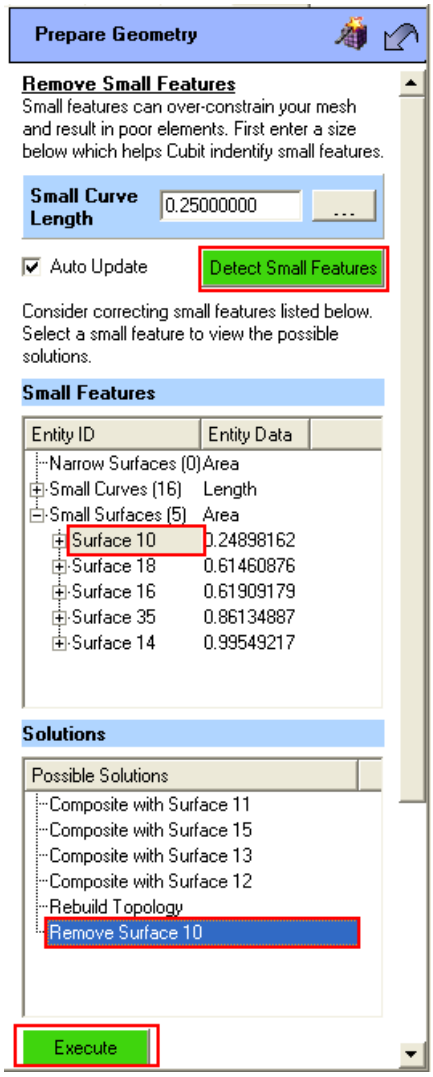


In addition to setting the small feature size, the smallest feature size panel of itself is a useful tool for visualizing and grouping small features. Sometimes it is useful just to have a list of the smallest features and a means of quickly visualizing and grouping them.

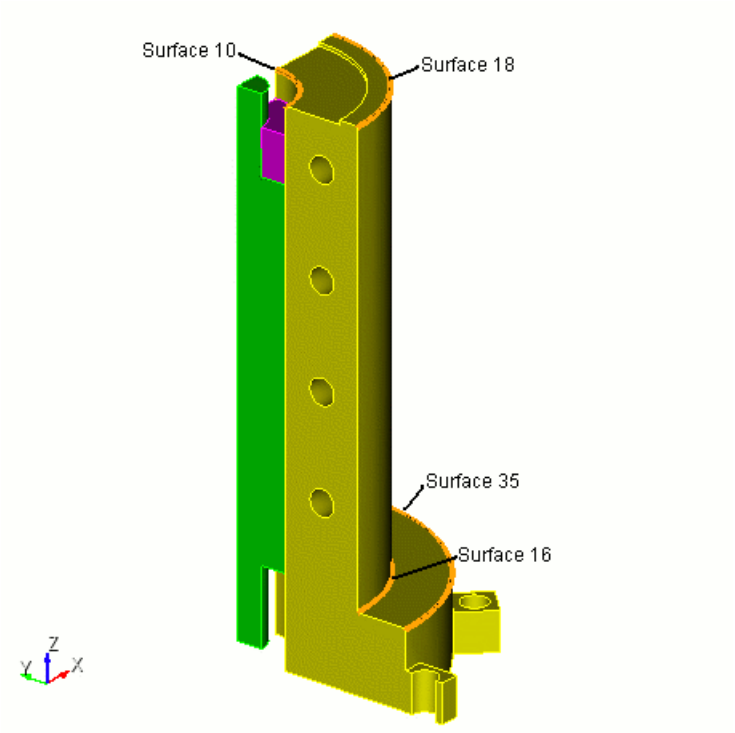
- Scan through the vertex-vertex pairs on the list. Right-click on any of the entities on the list and select **Zoom to Pair** or **Fly-in** to zoom in on the small feature.
- Locate pair **Vertex 75 and 74**
- Clicking on the pair should populated the small feature size input box and the bottom of the window.
- Click Done



- You should now be on the **Remove Small Features** page. Click **Detect small features**. There should be 4 small surfaces on the list.
- Click on the plus sign to open the **Small Surfaces** group. The first four surfaces are small filleted surfaces that are easily removed.
- Click on **Surface 10**
- Click on **Remove Surface** from the Solutions window
- Click **Execute**

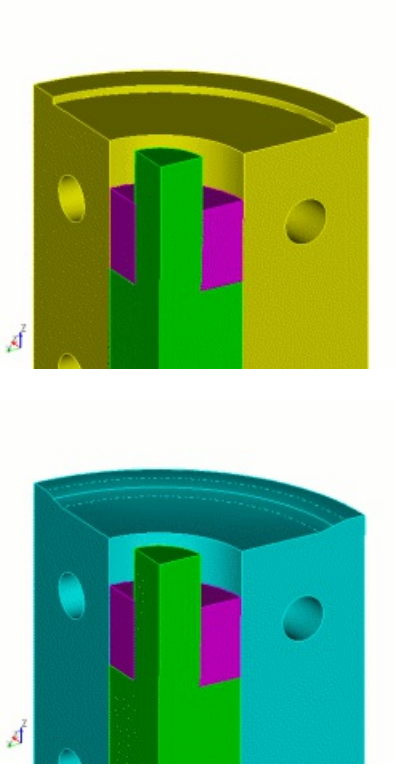


- Repeat for **Surface 18**, **Surface 16**, **Surface 35**



- On the 4th small surface execute the **Rebuild Topology** solution. The rebuild topology option will remove curves or surfaces from a model and reconstruct the geometry using real geometry

operations. In this case, it replaces the sharp lip surface with a gradual surface, as shown in the following image.



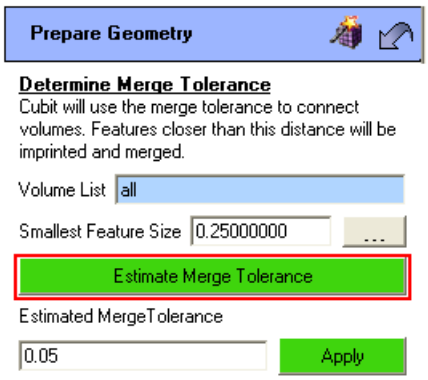
- The rest of the small curves in the model are at the small feature size limit of 0.25. To remove these from the list, either change small feature size to a smaller value like 0.24, Or click **Mark ALL as Okay**
- Click **Done** to return to the main Prepare Geometry page

ITEM Tutorial Step 4

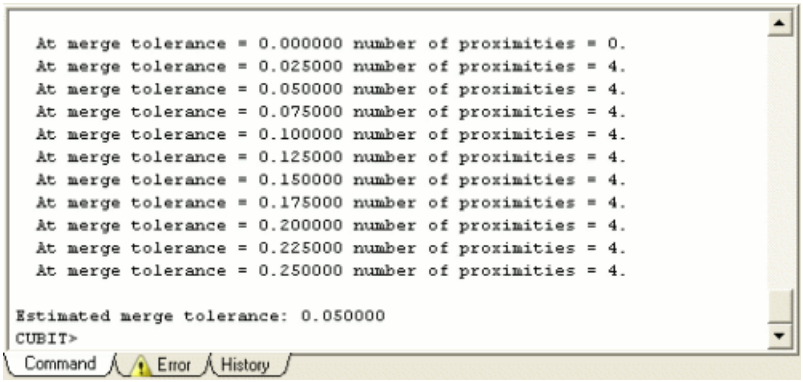
Step 4: Connect Volumes

The next step in the mesh generation process is to merge all shared curves and surfaces. This is necessary so that adjacent volumes can shared boundary meshes. For most geometries, this step presents no major complications. But in many cases, misalignments, tolerance problems, or other cleanup operations can prevent proper merging. The ITEM panel is designed to guide users through imprint/merge problems.

- Click **Connect Volumes**
- Click **Imprint and merge**
- Click on the button with three small dots **(...)** next to the Merge Tolerance field
- Click **Estimate Merge Tolerance**. The merge tolerance panel is used to help the user find an appropriate merge tolerance. In addition to determining a proper tolerance for merging, the merge tolerance can also be used as a diagnostic tool to find small misalignments, as will be demonstrated below. Many of these can be resolved prior to imprinting and merging.

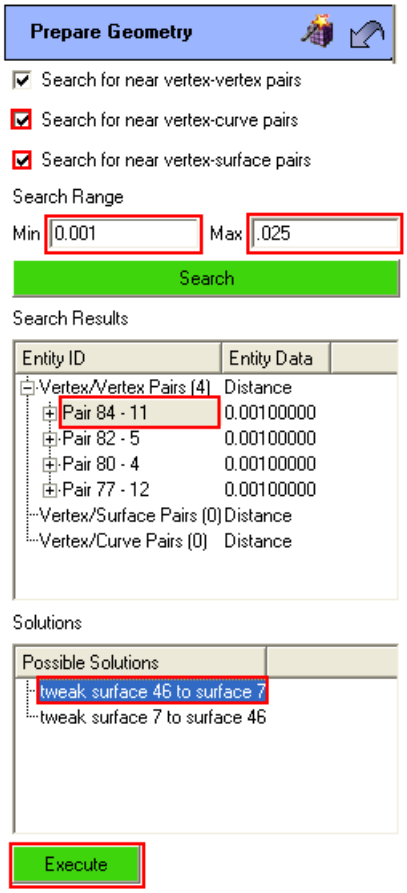


- Check vertex-curve and vertex-surface pairs
- Look in the ouput of the command line workspace. A proximity is nearly coincident entities that would be merged at the given tolerance. From the given list, you can tell that there are 4 entities that would be merged at a merge tolerance of 0.025 which would not be merged if the merge tolerance were 0. This means that those entities are less than 0.025 apart.



- Change the search parameter to very small number like Min=0.001 and search again.
- Four **Vertex-Vertex Pairs** appear
- Open **Vertex/Vertex** pairs group
- Right click on first pair and choose **Label Pair** to view
- Click on the first solution (**tweak surface 46 to surface 7**) and

click **Execute**



- Click **Done**
- Click **Imprint/Merge button**
- Click **Detect Potential Problems**

.

No problems should appear on the list, signifying that imprinting and merging has most likely been successful. There are several diagnostic tools on this page that help to determine if imprint/merging has been successful. These include:

- **Overlapping Surfaces**- Surfaces that overlap, but are not merged.
- **Non-manifold curves**- Two curves that are merged but that don't have any merged surfaces.
- **Non-manifold vertices**- Two vertices that are merged, but do not share any merged curves.
- **Floating volumes**- Volumes that are not connected to any other volumes (meaning they are not merged)

All of these diagnostics could be run at any time but the results are most meaningful after an imprint/merge operation.

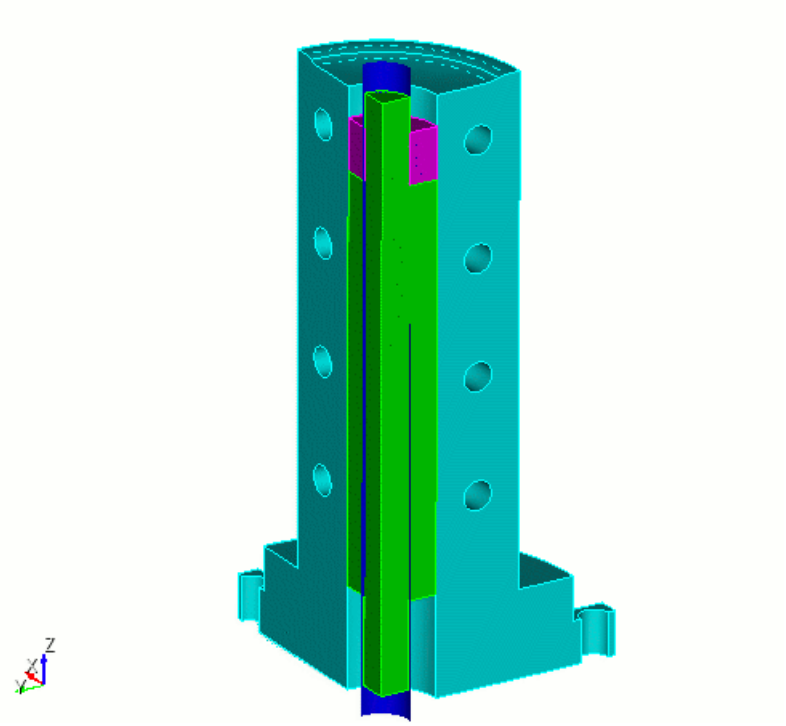
- Click **Done**
- Click **Done**

ITEM Tutorial Step 5

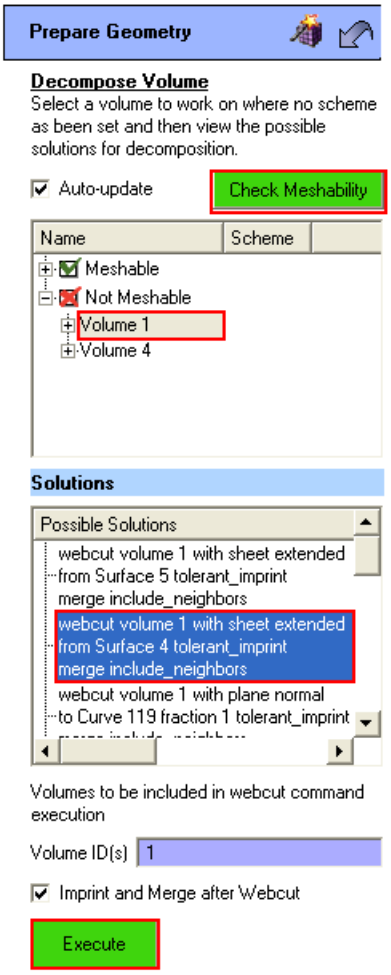
Step 5: Build a Meshable Topology

The next step in the mesh generation process can be one of the most challenging. Building a meshable topology involves decomposing an assembly into meshable parts. For sweeping, this means decomposing it into volumes composed of many-to-one and one-to-one sweepable parts. Each decomposed volume is further constrained because it needs to be able to share boundary meshes on merged surfaces. Since the number of possible decomposition strategies are numerous, it is not yet possible to automatically decompose most models. Instead, the ITEM framework seeks to provide possible decomposition options to the user, which they can be easily executed (and if necessary, quickly undone).

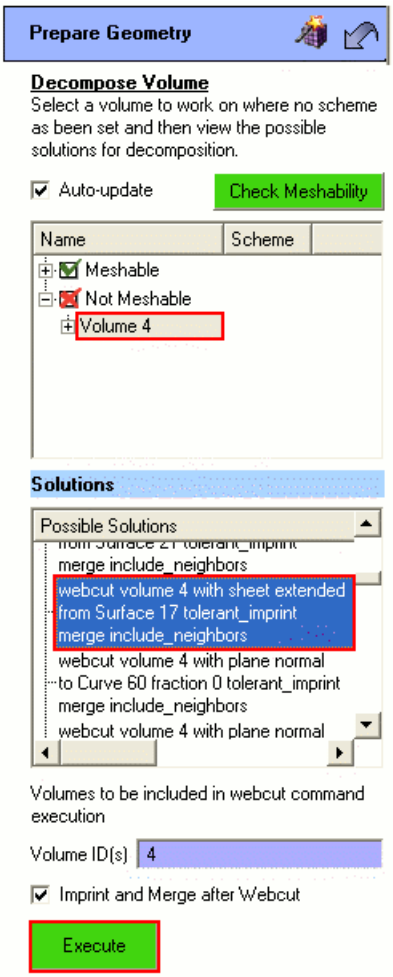
- Click **Build meshable topology**
- Click **Decompose volume**
- Click **Check Meshability**
- Examine **Volume 3**. There is 1 source and 1 target surface and it is sweepable.
- Examine **Volume 1**. There are two source surfaces and two target surfaces. It needs to be webcut cylindrically through middle

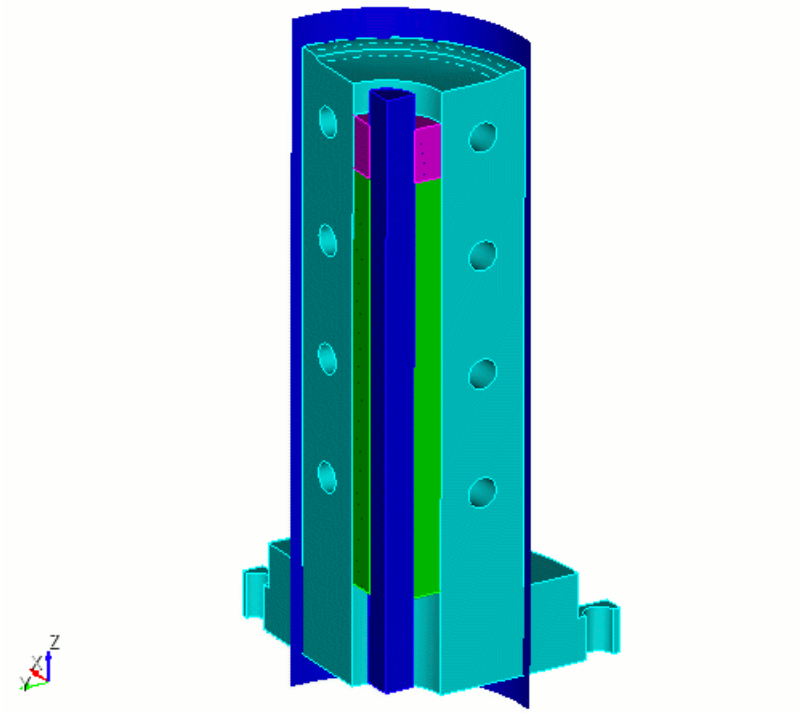


- Click on **Volume 1**
- Choose cylindrical webcut and execute making sure the imprint and merge after webcut is checked



- Examine **Volume 2**. The tall portion should be swept around in the direction of the holes. The bottom portion should be swept from top to bottom. It is many to one sweepable if webcut cylindrically around inner cylinder.
- Choose cylindrical webcut that cuts off thicker part at bottom of **Volume 4.**





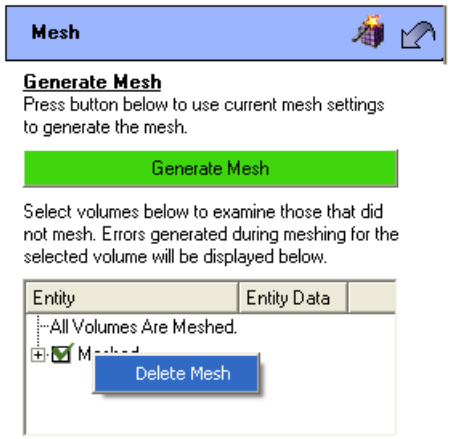
- Click **Done**.
- Click **Done** after running checked diagnostics again to verify.
- Click **Done**.

ITEM Tutorial Step 6

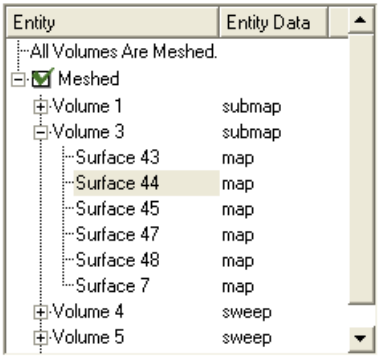
Step 6: Meshing the Geometry

The actual mesh generation process is usually quite iterative. Rare is the case where meshing succeeds perfectly on the first try, even when all volumes are "meshable". Even if it does succeed, it is usually constrained by areas of poor quality elements. ITEM was designed to help users navigate the iterative mesh generation process. When meshing fails, the mesh generation panel helps to explain common error messages and suggest possible strategies for getting a model to mesh.

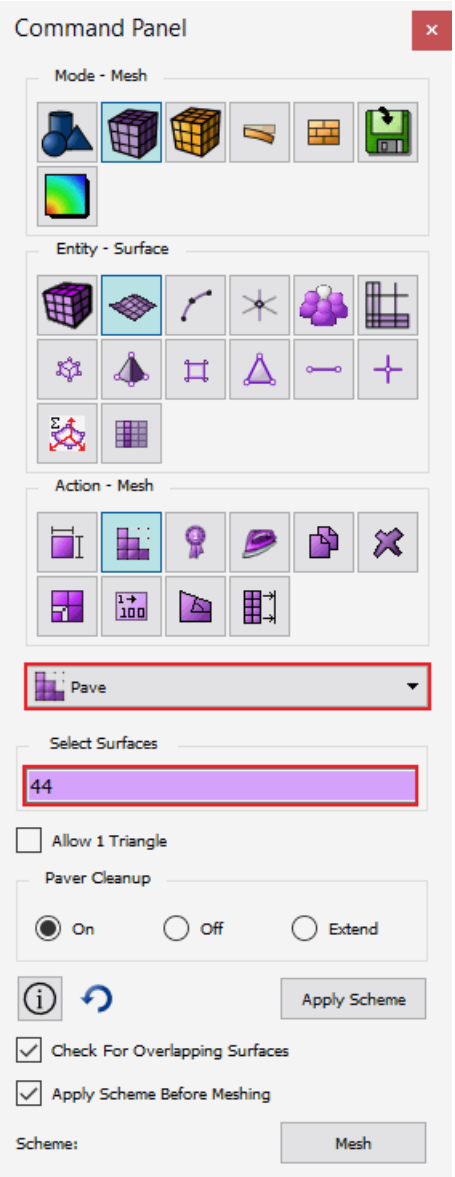
- Click **Mesh the geometry**
- Click **Generate Mesh**. Meshing succeeds in this case, but on closer inspection, it appears as though it has forced a mapped mesh on two surfaces that would be better as paved surfaces.
- Right click on **Meshed Volumes** and select **Delete Mesh**



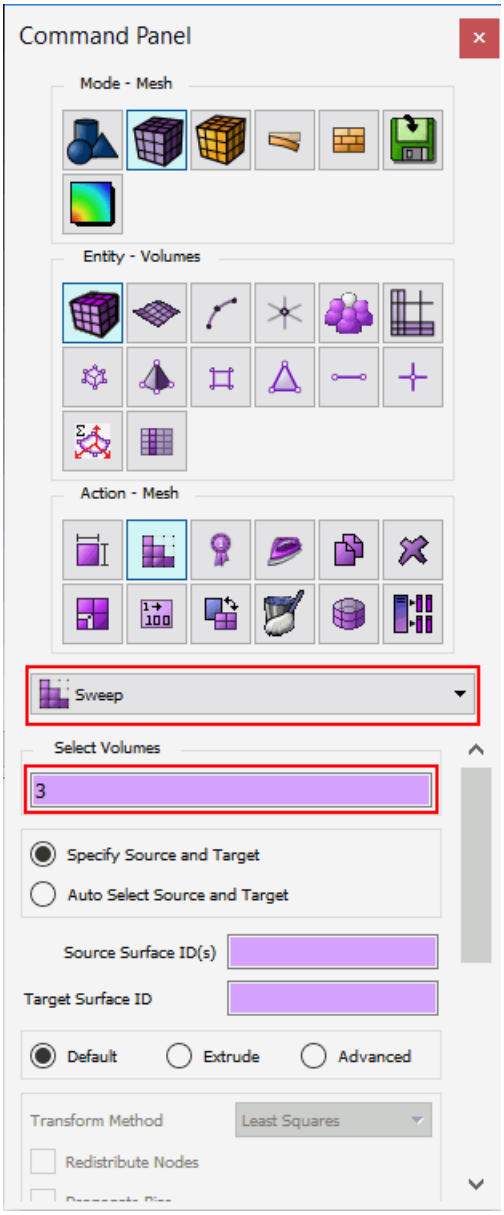
- Click on **Volume 3-Surface 44** - Notice the scheme is set to map



- Click on the **Meshing-Surface** command panel and set the scheme to pave for **Surface 44**

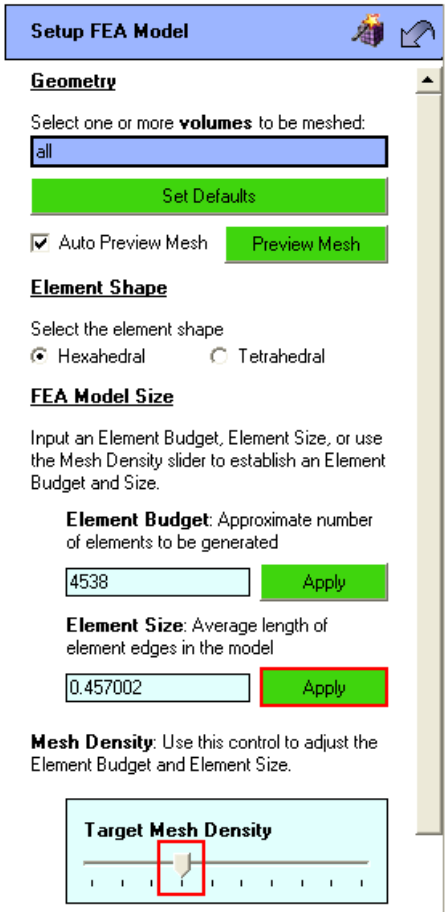


- Repeat for **Surface 20**
- Go to command panel for **Meshing-Volume** and select **Volume 3**. Change the scheme to **Sweep**.

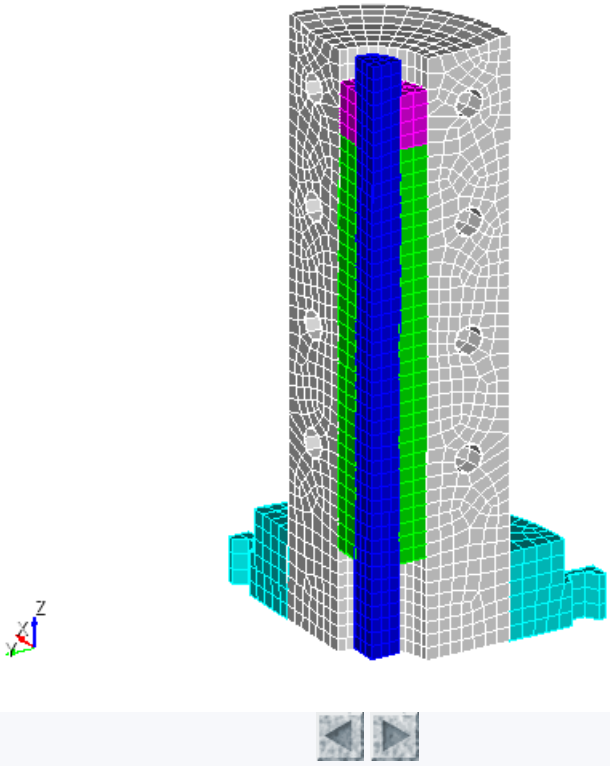


The mesh density didn't adequately capture the mesh features. To decrease the mesh size, return to the setup panel.

- Back on the ITEM Panel, click on **Setup the FEA model** button on the left panel.
- Change the mesh size by moving the target mesh density slider one position to the left.
- Click **Apply** next to the **Element Size**



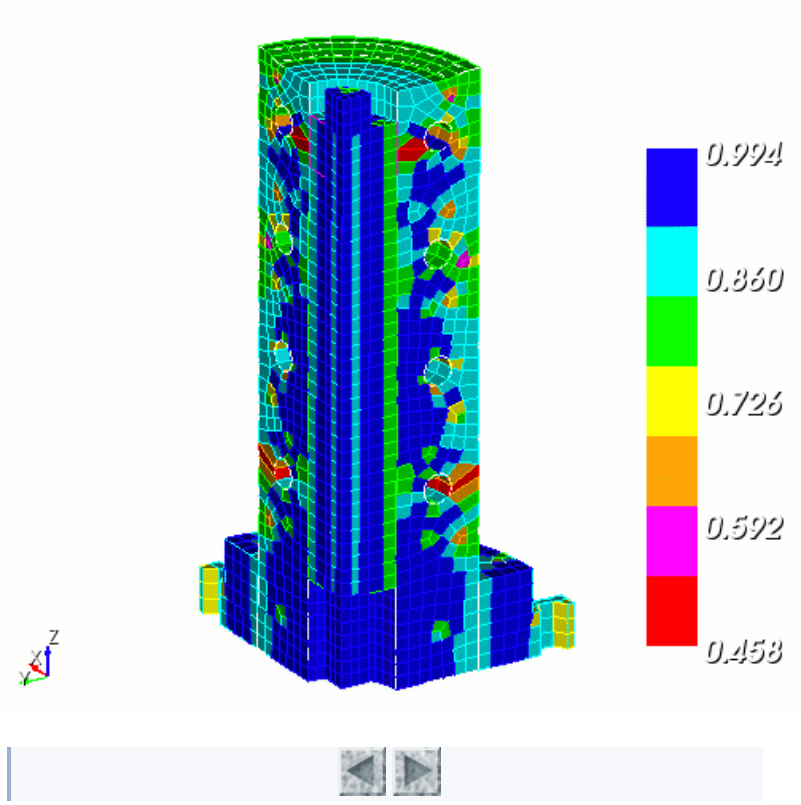
- Click on **Done**
- Click on **Mesh the geometry**
- Click on **Generate Mesh**



ITEM Tutorial Step 7

Step 7: Validate Mesh

- Click on **Validate Mesh**
- Click on **Check mesh quality**
- Click on **Analyze Quality** - No bad elements found

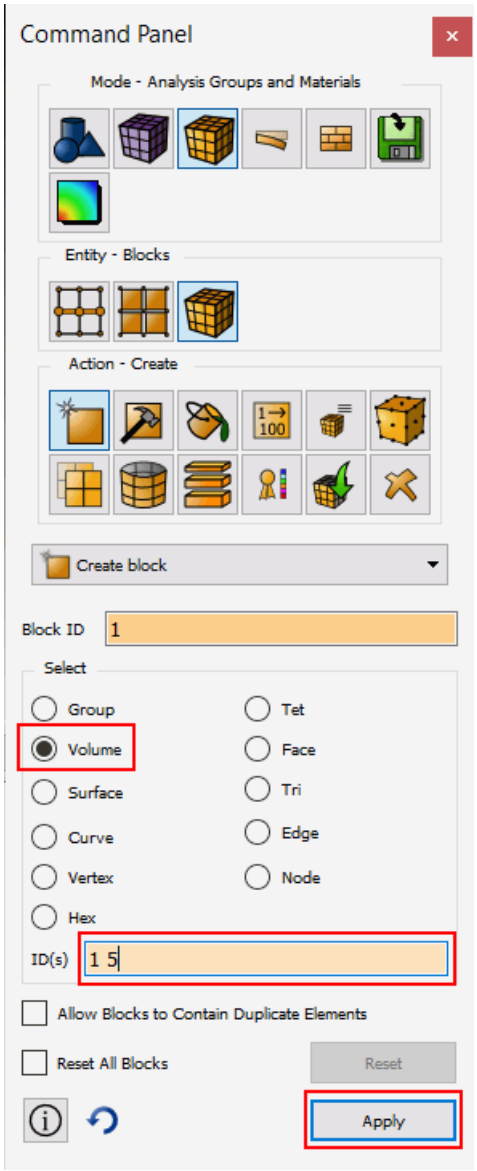


ITEM Tutorial Step 8

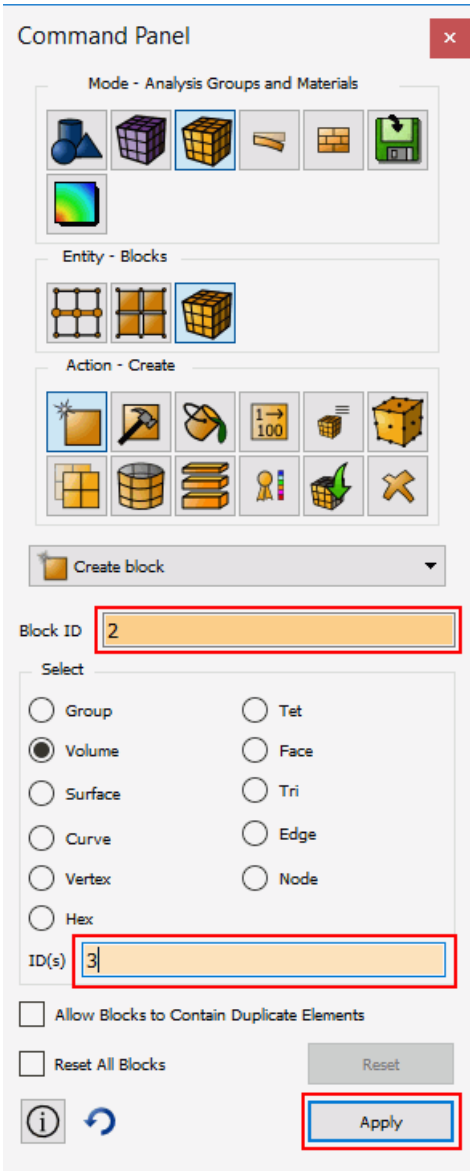
Step 8: Define Boundary Conditions

Exodus boundary conditions are specified as generic blocks, nodesets, and sidesets. Clicking on a boundary condition type on the ITEM panel will open the corresponding command panel.

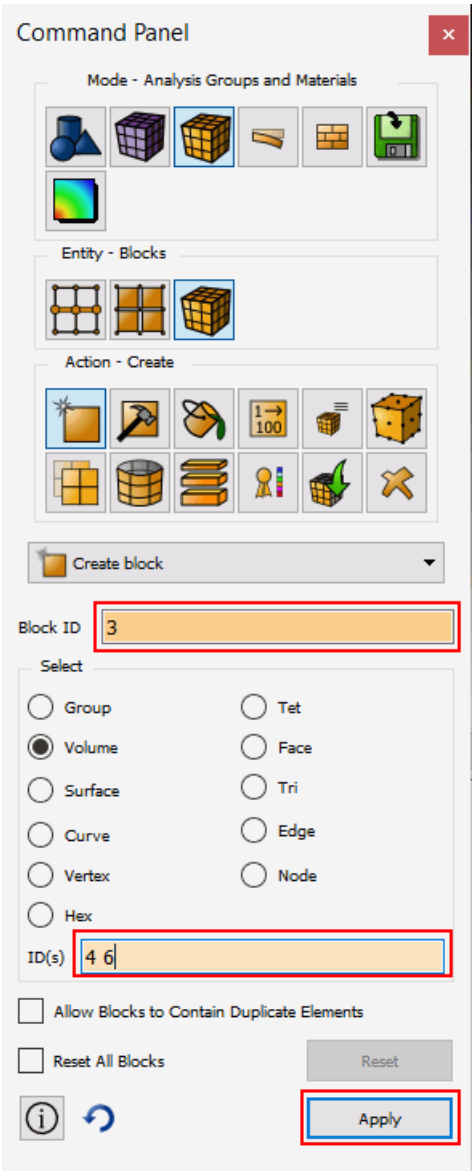
- Click on **Define boundary conditions** on the ITEM panel
- Click on **Define Block**. This opens the Material Properties->Block Panel and automatically assigns a default block id of 1.
- Set selection type to **Volume**
- Select **Volumes 1 5**
- Click **Apply**



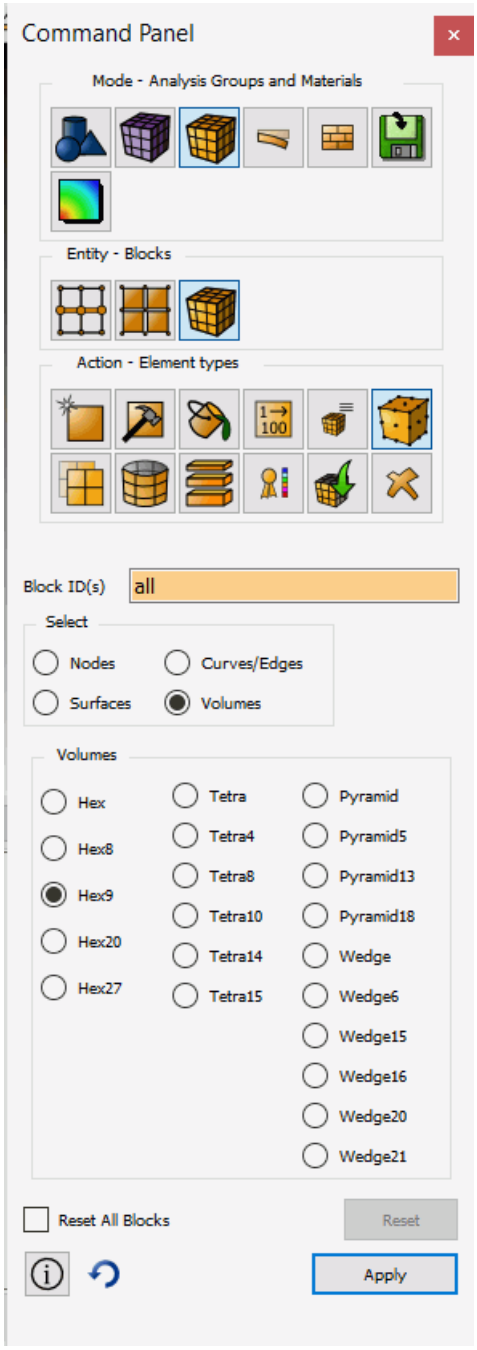
- Click on **Define Block** on the ITEM Panel. It should automatically increment the Block ID for you.
- Select **Volume 3**
- Click **Apply**



- Click **Create Block** on the ITEM Panel to increment block ID
- Select **Volumes 4 6**
- Click **Apply**



- Change to the **Element Type** panel by selecting the element type button
- Change the **Element Type** to **Hex9**
- Click **Apply**



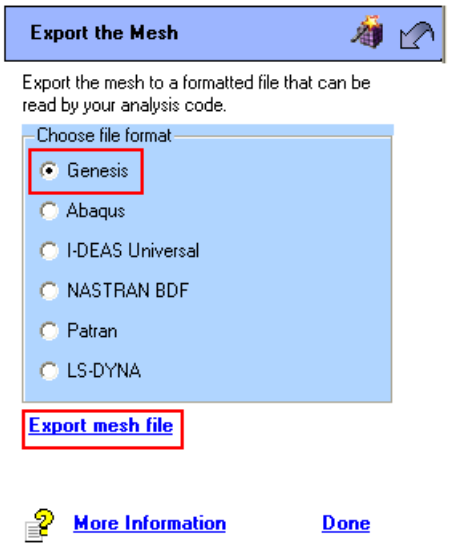
- On the ITEM panel click **Done**

ITEM Tutorial Step 9

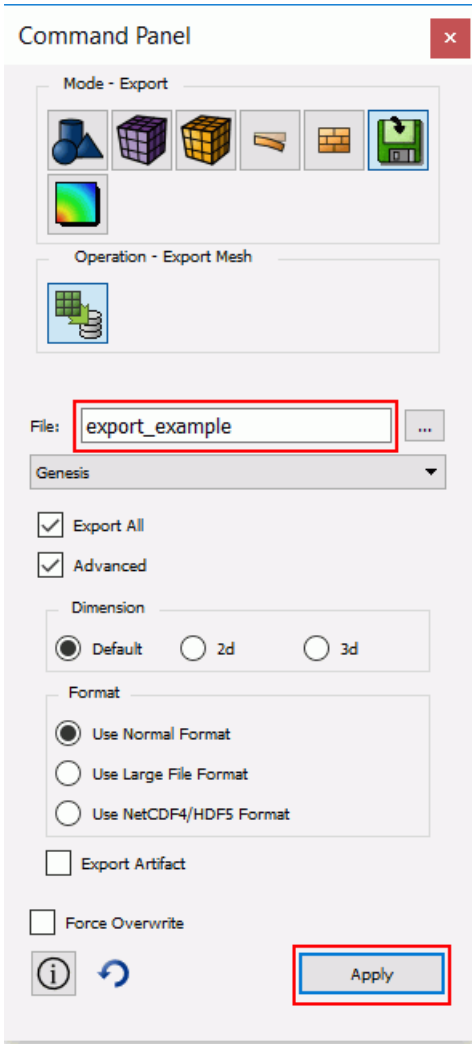
Step 9: Export the Exodus Model

Cubit primarily supports the Exodus format for mesh export. But there are also limited export abilities for other formats as well. For a list of export capabilities see Exporting the Finite Element Model

- Click on the **Export the Mesh** link from the main ITEM task page
- Set the **export type** to **Genesis**. This opens the export mesh dialog box on the command panels.



- Assign a filename
- Click **Apply** (all blocks will exported by default).



Congratulations on completing the ITEM tutorial. Click on the arrow to return to the main tutorial page.

Power Tools GUI Tutorial

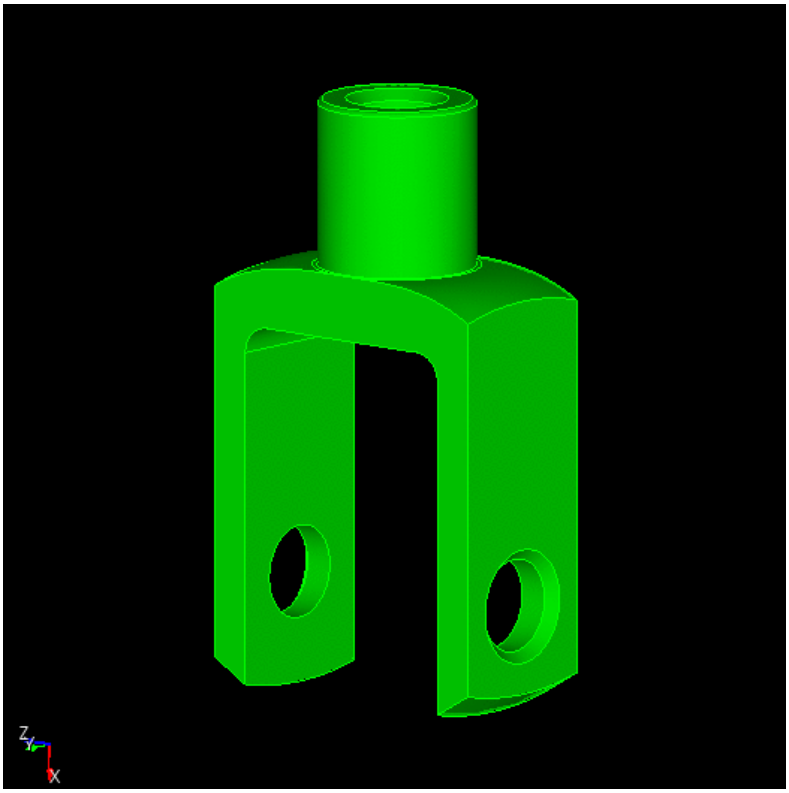
Overview

This tutorial demonstrates using the Power Tools on the CUBIT GUI for geometry decomposition and cleanup. The following features will be covered:

- Importing Geometry
- Analyzing Geometry
- Geometry Power Tools
- Webcutting
- Imprint/Merge
- Mesh Power Tools
- Meshing

Each of these steps is described in detail in the following sections. For this tutorial you will need to have a basic understanding of the CUBIT GUI functionality, including how to select entities, maneuver in the graphics window, operate the Control Panel, and use toolbars. If you have not already done so, we recommend completing the Basic Tutorial first. The following image shows the geometry that will be used for this tutorial.

NOTE: Many of the steps in this tutorial include operations on specific entities which are identified by ID. When the solid modeling kernal is updated in Cubit the ID space may change. As such, you may not be able to rely on the IDs specified in this tutorial. Please look at the associated graphics to determine which entity/ID is being referred to.

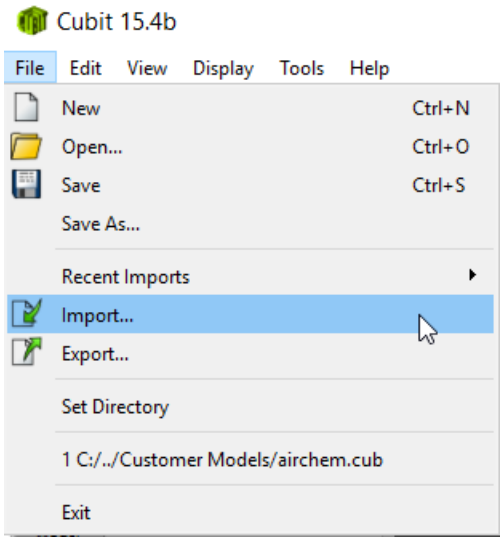


Power Tools GUI Tutorial Step 1

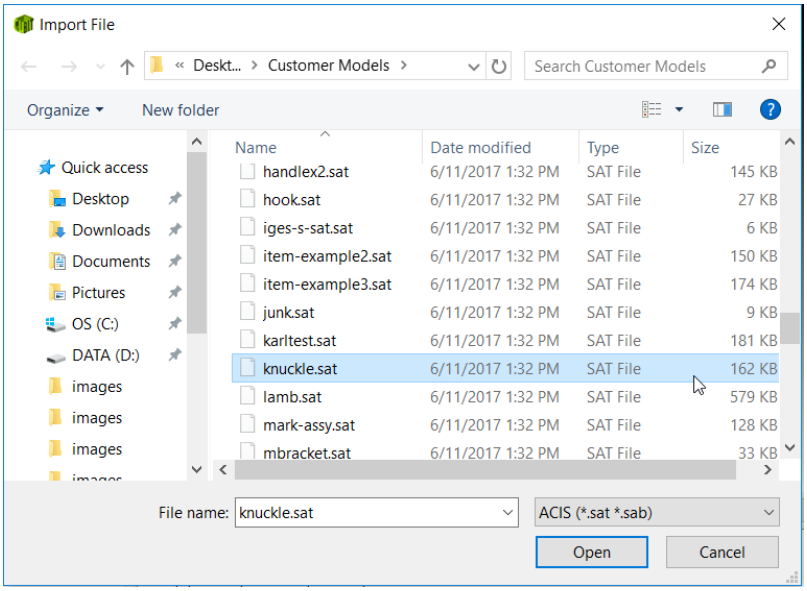
Step 1: Import the Geometry

Begin by opening a new session of CUBIT. To complete this tutorial, you will need to download the ACIS file that contains the geometry definition.

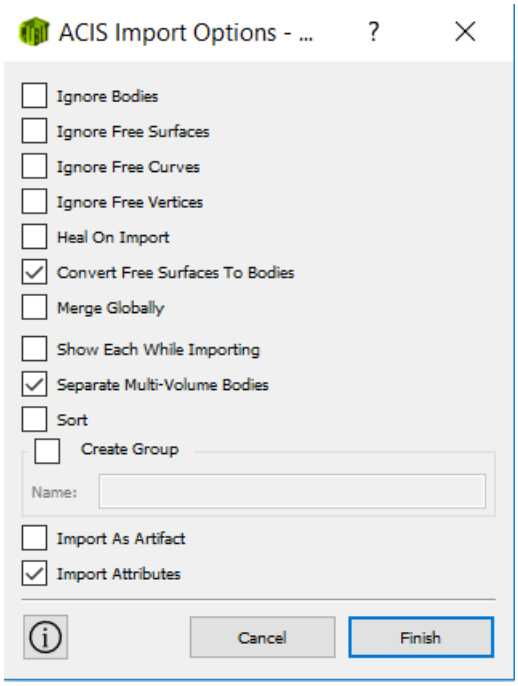
- Download geometry file knuckle.sat (**Note**: This link will not work from within Cubit. You will need to access this documentation from the cubit web site, or locate the file on your computer. It is included in the distribution of CUBIT under components\cubit\help\step_by_step_tutorials\power_tools)
- Select the **Import** option from **File** menu



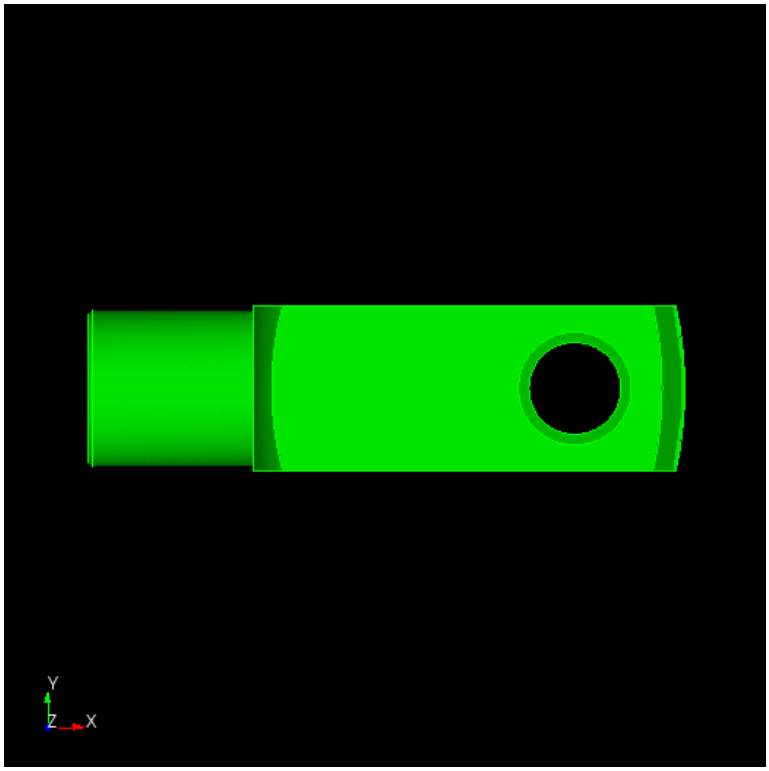
- The following dialog box will appear. Open the file by clicking on the name and selecting **Open**. If you do not see the file, make sure that you are in the right directory, and that the file type is set to ACIS.



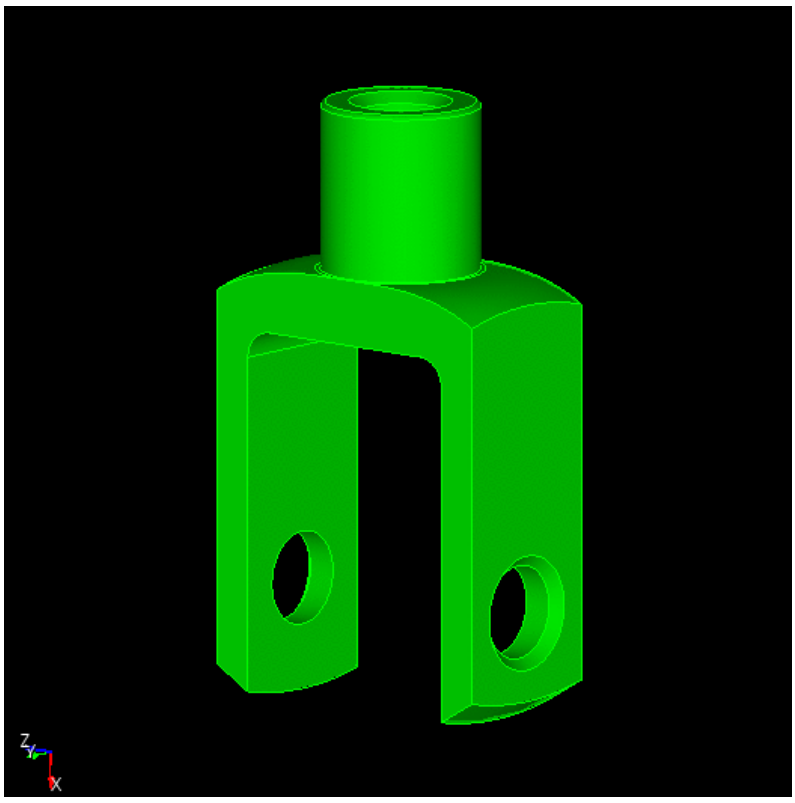
- Leave all of the import settings on their default settings and select **Finish**



Your graphics window should now appear as follows:



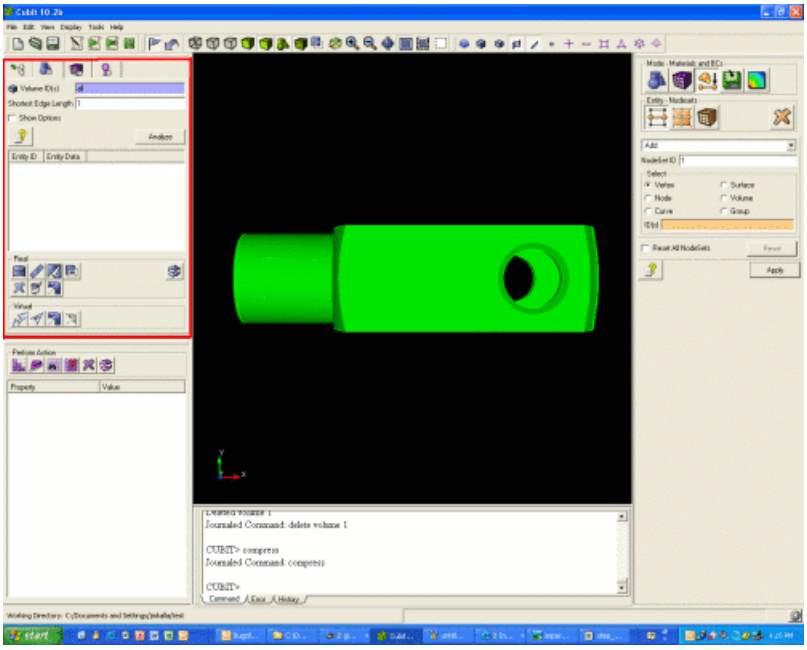
- Use the mouse to rotate the image in the graphics window to get a better perspective. For help with using the mouse in the graphics window, see Mouse Based Zoom, Pan and Rotate .


Z
X

Power Tools GUI Tutorial Step 2

Step 2: Analyze the Geometry

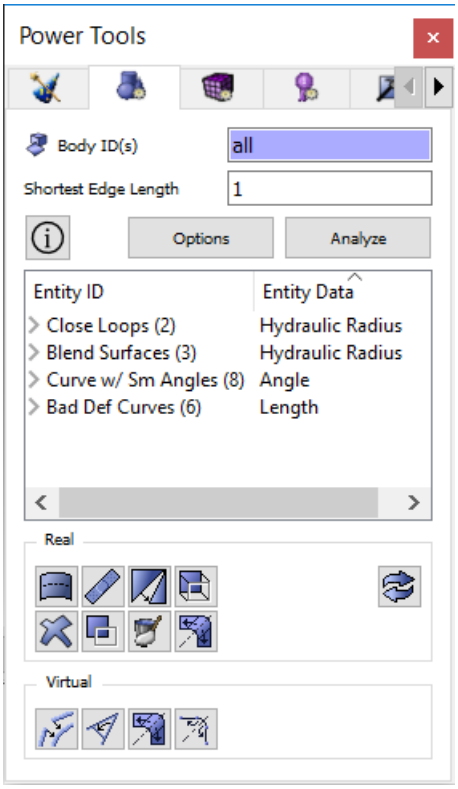
The Geometry Power Tools are located in the Entity Tree Window under the blue geometry tab. This menu provides access to many of the geometry analysis and clean-up tools in CUBIT.



Many geometries that are imported from other solid modeling software contain inconsistencies or small gaps that can cause meshing to fail. These problems are the result of differences in tolerances, file transfer loss, or inherent limitations in the parent system. In other instances, the geometry has no inconsistencies, but may be unsuitable for meshing because of topology such as small angles, overlap, or features smaller than the desired meshing size. The geometry analysis tool will analyze the volumes and return a list of suspected problems. To see a list of analysis options, click the "Show Options" box below the Analyze button.

Many of these problems can be fixed using the tools on the Power Tools menu. These include Split Surface, Heal, Tweak, Remove, Merge, Composite, Collapse Angle, Collapse Curve, and Collapse Surface. Many of these tools will be demonstrated in this tutorial.

- Open the **geometry repair** tab in the Entity Tree window
- Type all in the **Volumes to Analyze** field
- Set the **Shortest Edge Length** to 1
- Press **Analyze**



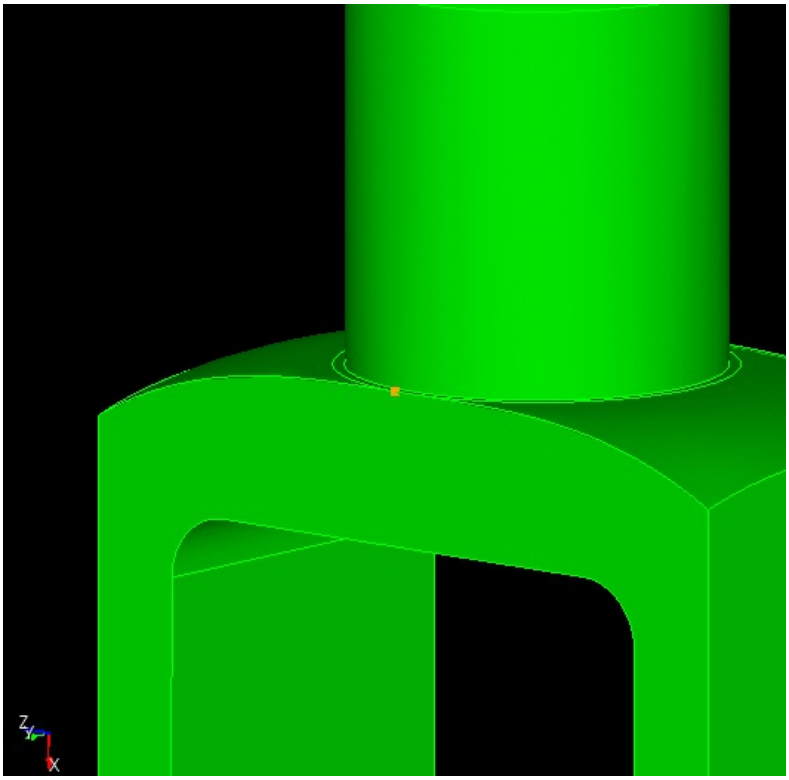
After the Analyze Button is pushed, display area will appear as shown above. There are four suspected problems with this geometry: Curves with Small Angles, Blend Surfaces, Close Loops, and Badly Defined Geometry. The numbers in parentheses indicate the number of occurrences of this problem in the model. Clicking on the + sign by each label will list the CUBIT entities by ID with this problem. Clicking on the + sign by each entity will cause that entity's children or parents to be listed (depending on the entity and the type of geometry test). See documentation on Geometry Repair for more information about the display window. Clicking on the name of an entity will highlight that entity in the graphics window.

- Select **Vertex 45** under Curves with Small Angles

Observe that this vertex is highlighted in the graphics window.

- Right click and select **Zoom To** from the list of options

The graphics window should look like this:



- Right Click on Vertex 45 and select **Reset Zoom** from the list of options

The image should now be reset to the previous graphics state.

You can experiment with some of the other options in the top half of the right click menu. They are:

- **Fly-in** - Animated zoom feature
- **Locate** - Labels entity
- **Draw** - Draw this entity by itself
- **Draw with Neighbors** - Draw this entity with all adjacent curves and surfaces
- **Clear Highlights** - Clear all highlighted entities
- **Reset Graphics** - Refresh graphics screen

The graphics window may also be reset by pressing the reset graphics button  on the menu.

Power Tools GUI Tutorial Step 3

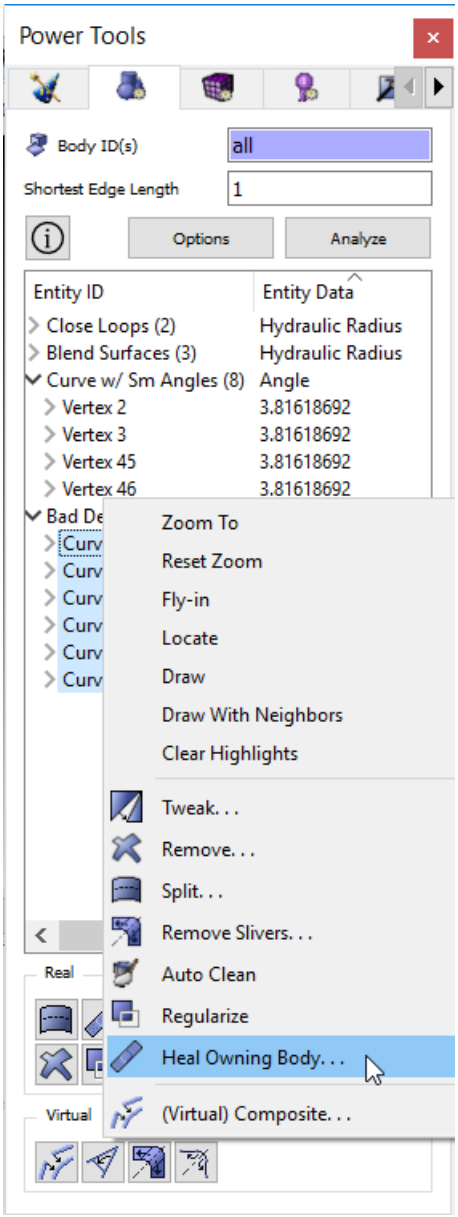
Step 3: Healing the Geometry

The first step to improving any geometry is to look for badly defined geometry and to fix it using the Autoheal tool in CUBIT. The Geometry Analysis tool may detect these inconsistencies, but only if such a function exists in the parent software. It is always a good idea to run the Autoheal on imported geometry. In this example, the Power Tools has located some badly defined curves. This step will show you how to use the geometry repair tool to fix these curves.

- **Highlight** all of the badly defined curves by holding down the Shift key while selecting
- Right click and select **Heal Owning Body** from the list of options

OR

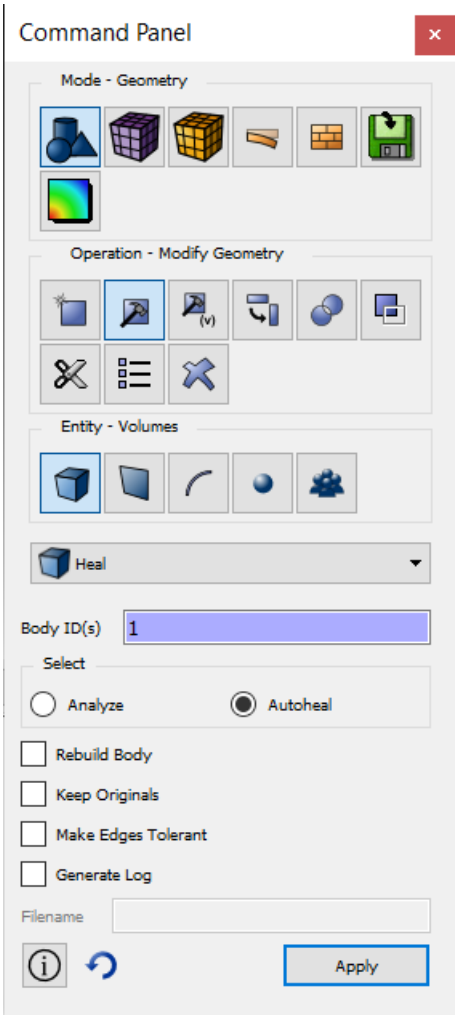
- Click the  button



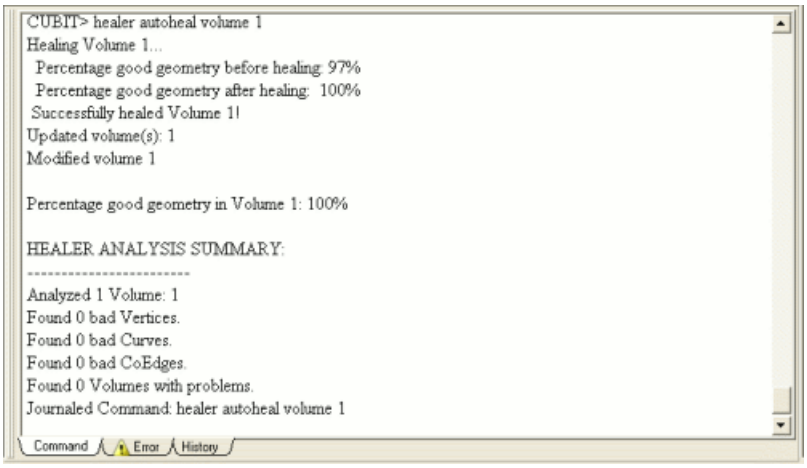
The Geometry Repair Tool does not execute any geometry clean-up

commands directly, but directs you to the place on the Control Panel where this function can be executed. The following menu will appear on the Control Panel. Notice that the id of the owning body has already been pasted into the input window.

- Select the **Autoheal** button
- Press **Apply**



The output window on the CUBIT GUI should appear with the following message. You may have to scroll to see the whole thing. The percentage before and after healing are 97% to 100%. Healing has been successful.



Run the geometry analysis test again to guarantee that all bad geometry has been removed.

- Press the **Analyze** Button in the Geometry Repair window

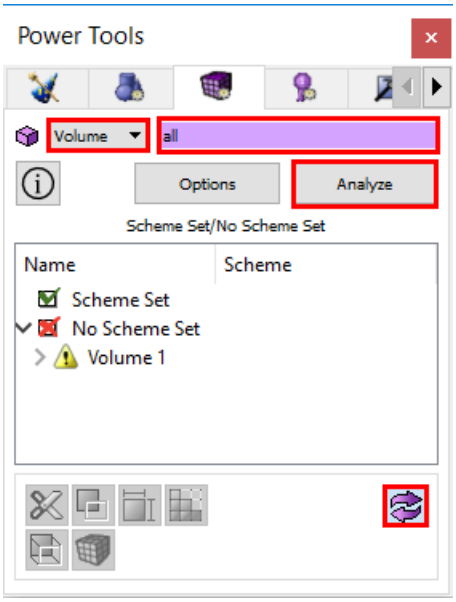


Power Tools GUI Tutorial Step 4

Step 4: Mesh Power Tools

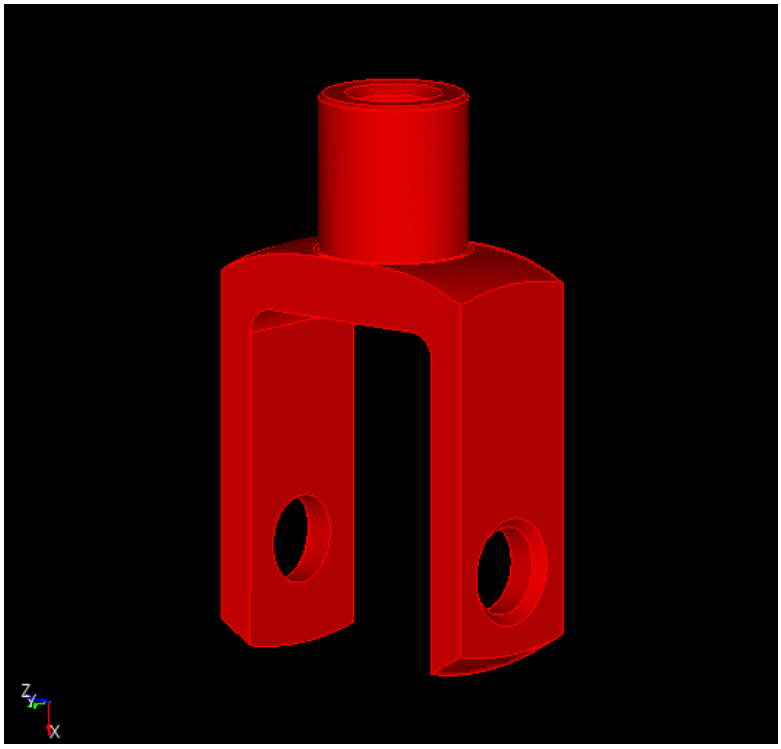
The Mesh Power Tool provides an easy and graphical way to determine if volumes are meshable. This tool will employ the AutoScheme feature in CUBIT to select and assign schemes to meshable volumes. If a volume is not currently meshable, it will be flagged and highlighted. Use the Mesh Power Tool to determine if the volume is currently meshable.

- Click on the purple Mesh Tools tab in the Power Tools window.
- Select **Volume** as the entity type in the pull-down menu (It may already be selected)
- Enter **all** in the input window
- Press **Analyze**



Volume 1 will appear under the "No Scheme Set" heading.

The graphics window should look like this with Volume 1 highlighted in red. Using this graphics feature, all volumes that are meshable will be highlighted in green, and all volumes that are not currently meshable will be highlighted in red.



- Toggle the **Graphics Button** (located in the bottom right corner of the tool) off so that Volume 1 is shown in green again

Power Tools GUI Tutorial Step 5

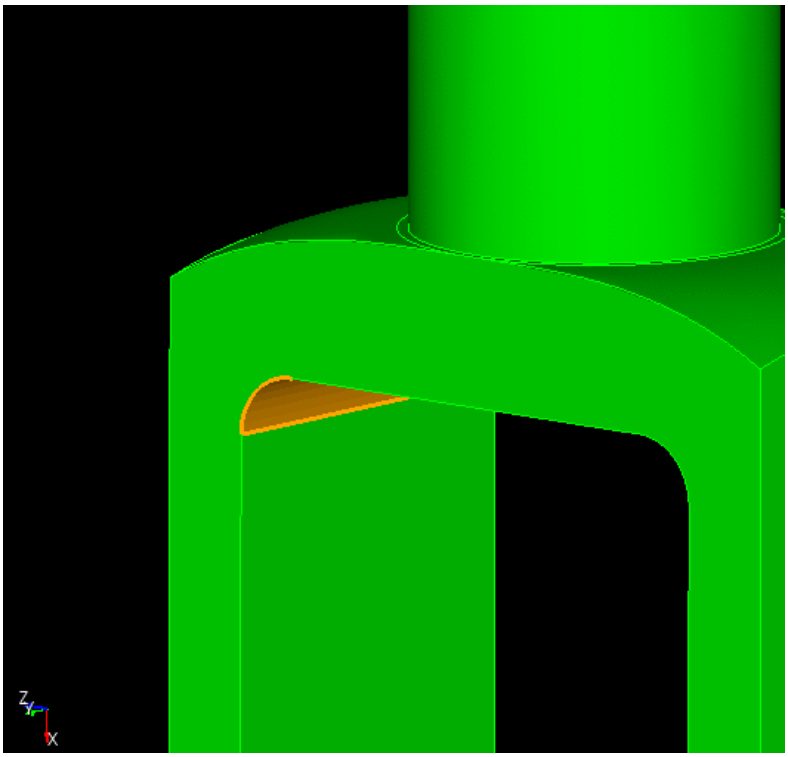
Step 5: Splitting Filleted Surfaces

The previous step determined that the volume was not currently meshable, and that further decomposition was required. This decomposition can be performed using the tools in the Geometry Repair power tools. A good place to start is with blend surfaces.

A blend surface is a transitional surface that connects two orthogonal planes, also known as a fillet. Blend surfaces can be problematic in meshing because there is no clear transition between the two orthogonal surfaces, making sweeping or mapping algorithms difficult. The Split Surface function divides these blend surfaces (or any surface) into two distinct surfaces.

- Select **Surface 22** from the list of blend surfaces
- Right click and select **Zoom To** from the list of options

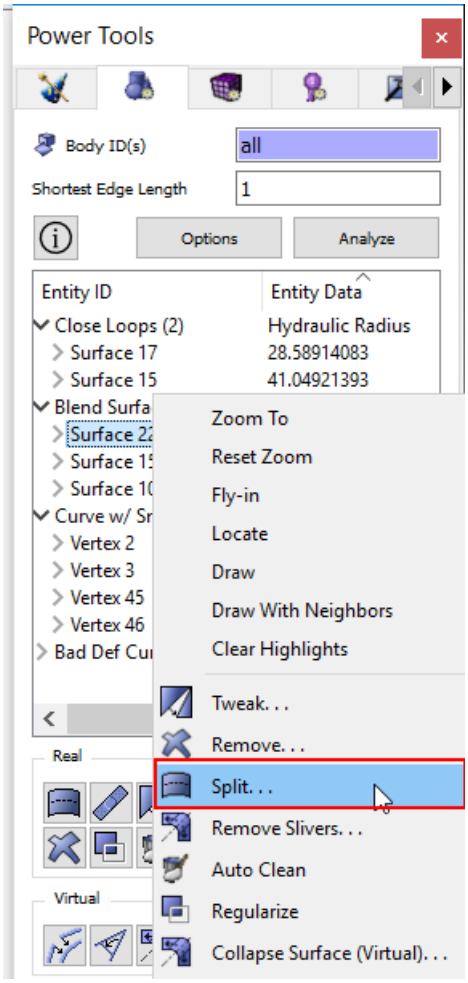
The graphics window should look like this:



- Right click with Surface 22 highlighted and select the **Split** button

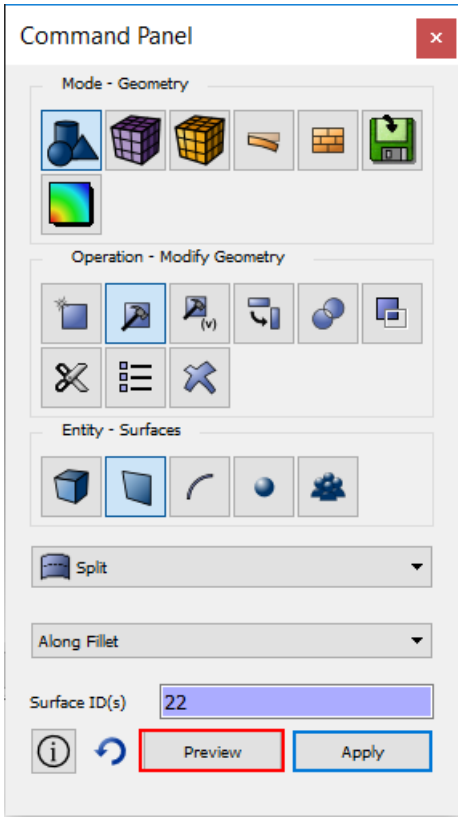
OR

- Click the  button on the tool panel

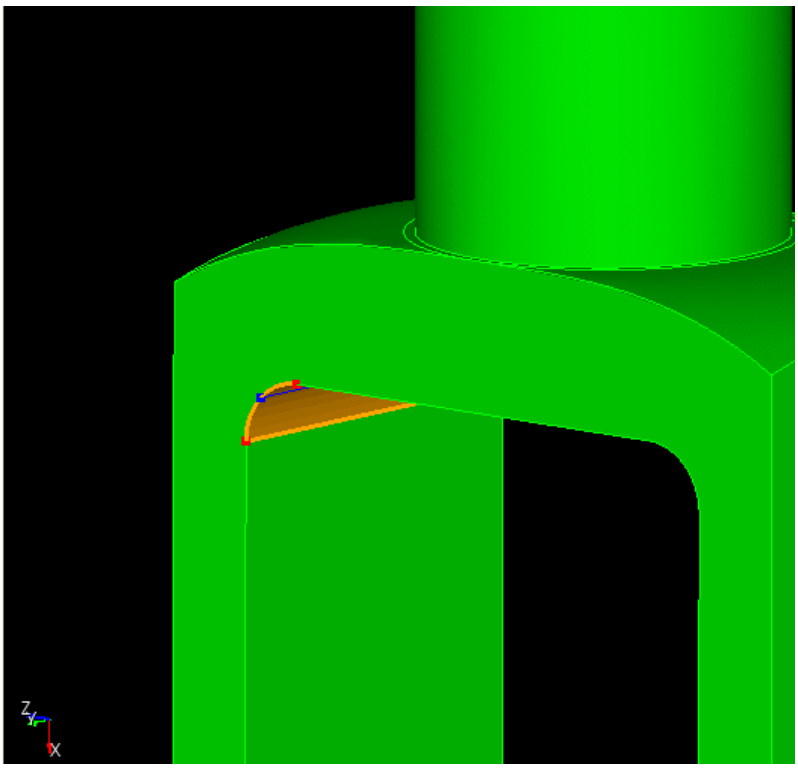


The Geometry-Surface-Modify-Split Menu will appear on the Control Panel. Make sure the Surface id is input in the window.

- Press the **Preview** Button



The blue line shows where the surface will be split.



- Press the **Apply** Button

The surface should now appear split.

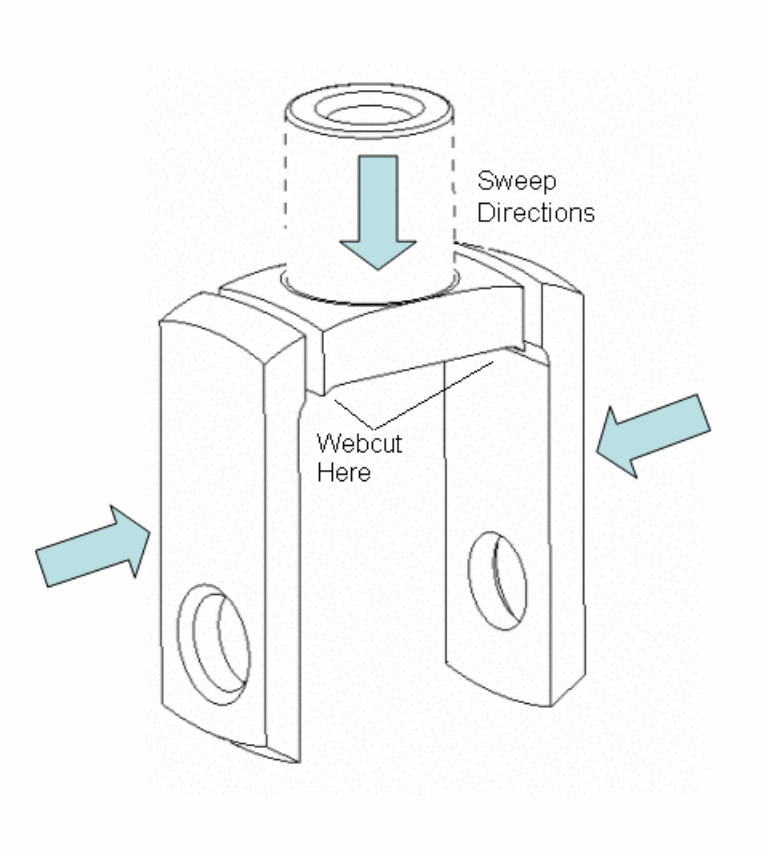
- Repeat these steps with the opposite blend surface (ID 10)

Power Tools GUI Tutorial Step 6

Step 6: Web Cutting

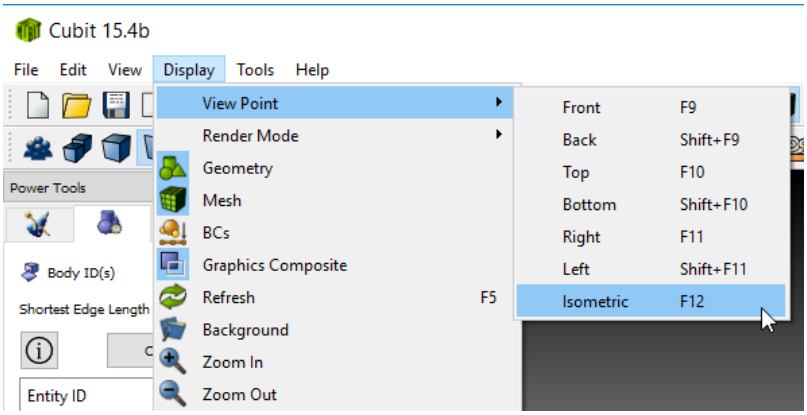
Since the model has several through holes, sweeping is not possible from a single source and target. However, it is possible to divide the model into three sweepable regions. The figure below shows where to divide the model to get it into sweepable regions. These regions coincide with the holes in the model.

Web cutting is this process of dividing volumes into sweepable regions by cutting with a plane. For this exercise, you will use the curves that were just created with the split surface command to cut the volume.



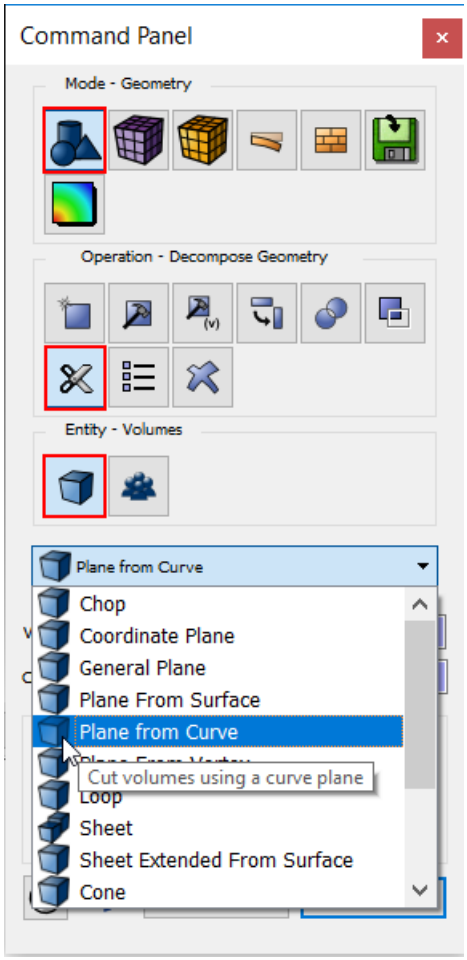
In order to visualize the process more clearly, switch to the isometric view.

- Change the view to isometric in the **Display** menu under **View Point**

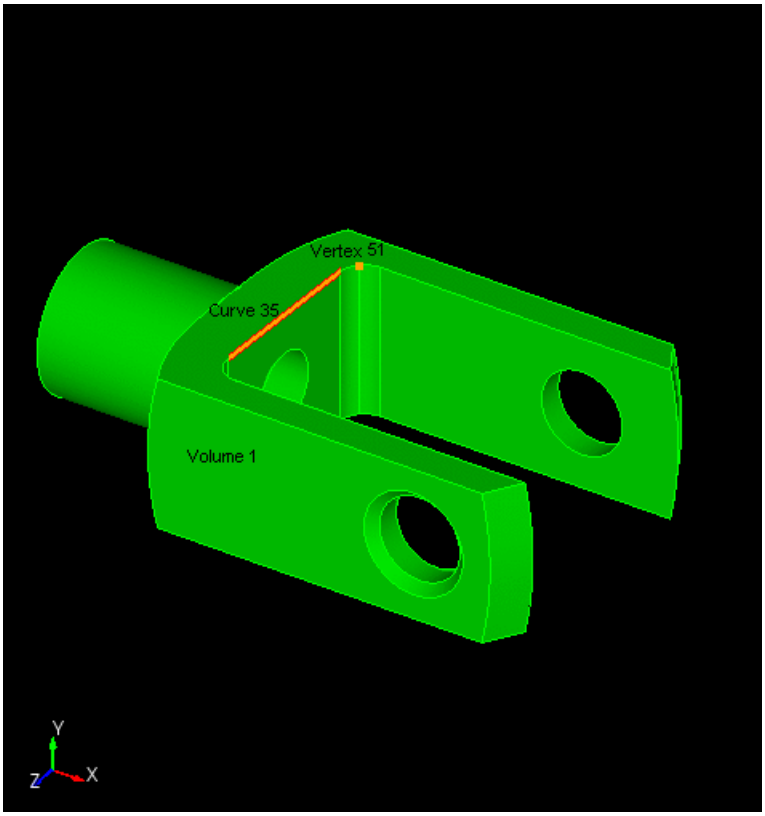


The web cutting menu is located under Geometry-Webcut-Volume on the Control Panel.

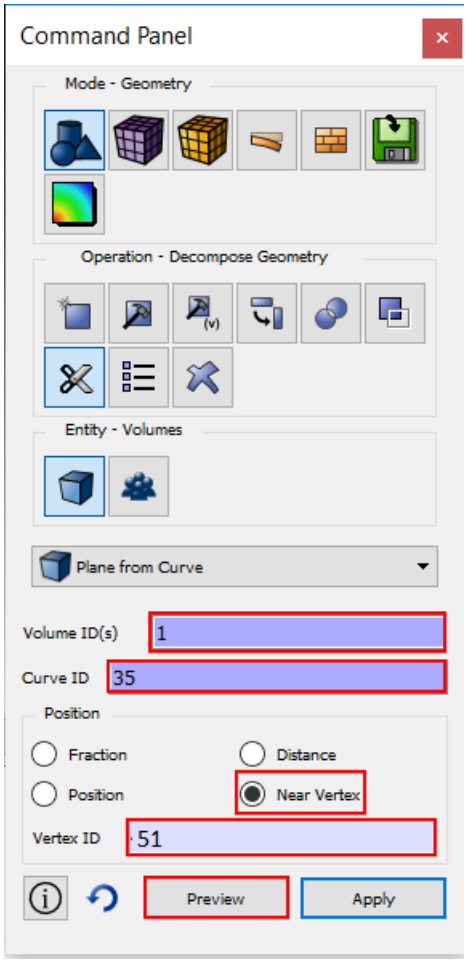
- Click on **Geometry**, then **Webcut**, then **Volume** on the Control Panel
- Select **Plane from Curve** from the list of options



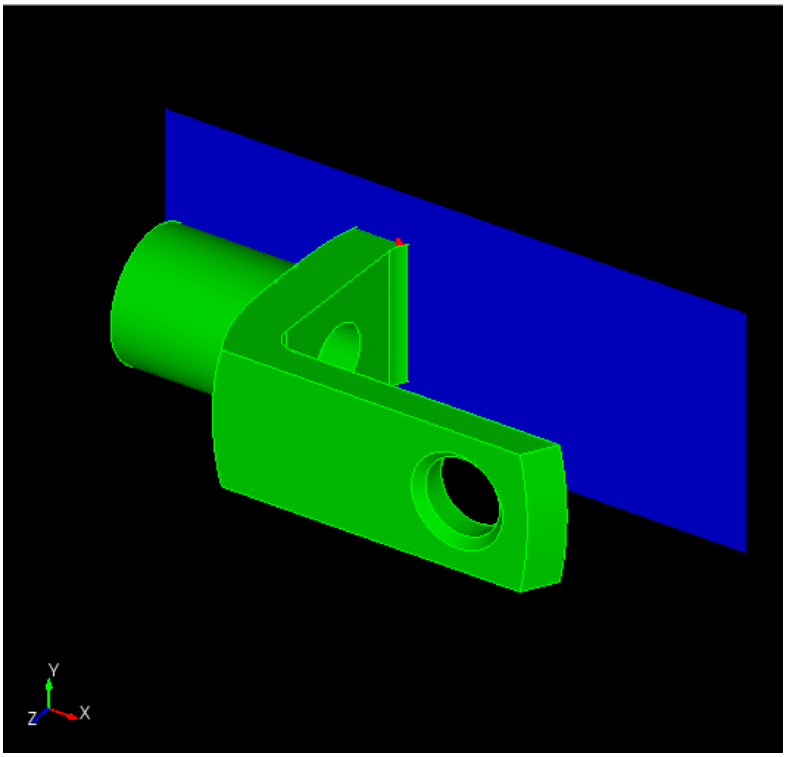
The following image shows the entity ids that will be used to webcut the volume. Select entities with the mouse by clicking on them.



- Enter **Volume 1** by typing it or selecting from the graphics window
- Enter **Curve 35** by typing it or selecting from the graphics window
- Change the **Type** to **Near Vertex**
- Enter **Vertex 51** by typing it or selecting from the graphics window
- Press **Preview**



A blue preview plane should appear in the following position. Check to make sure that your model looks the same.



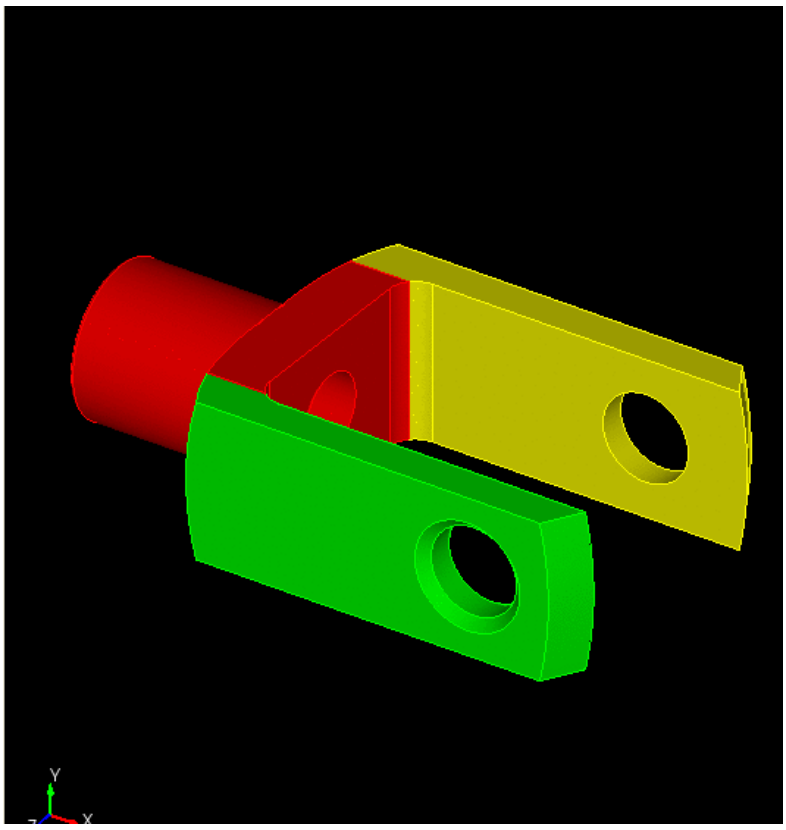
- Press **Apply**

The volume has now been split into two volumes. Volume 2 is shown in yellow.

Repeat these steps with the other side of the part. The Volume and Curve ids will remain the same.

- Enter **Vertex 49** in the input window or select from the graphics window
- Press **Preview**, then **Apply**

The final webcut volume should look like this:

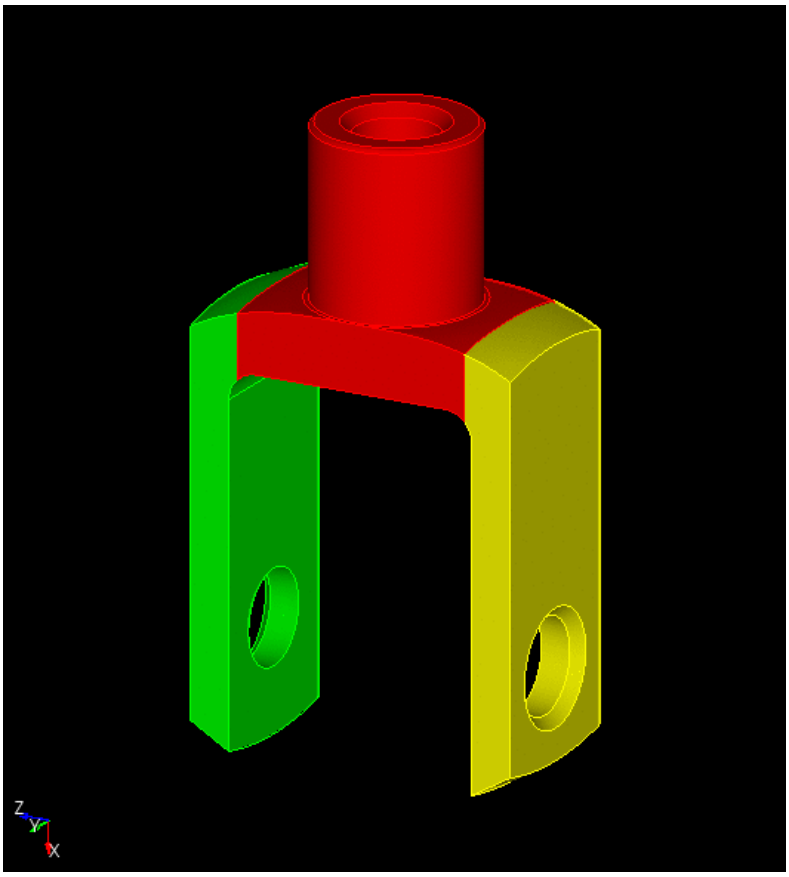

Y
X

Power Tools GUI Tutorial Step 7

Step 7: Removing Small Surfaces

Some surfaces are too small for analysis and should be removed from the model. In this example, Surface 15 and Surface 17 may fall into that category, assuming that the distance between curves on these surfaces is smaller than the desired final mesh size. You can remove these surfaces by extending adjacent surfaces until they intersect.

- **Rotate** the model to the following orientation



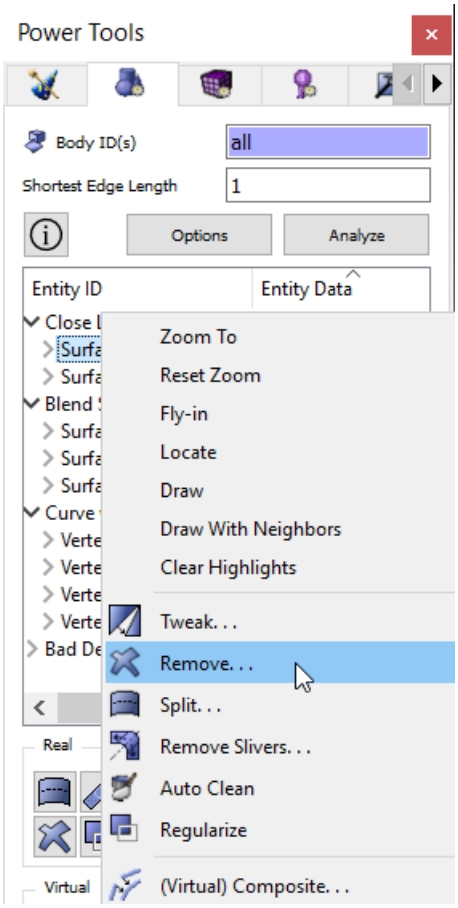
- Press **Analyze** on the Geometry Power Tools menu

You will notice that a new category has appeared labeled Overlapping Surfaces. This is because there are two new surfaces created for each of the webcuts that overlap a surface on the original body. This can be removed using the Imprint/Merge function which will be explained in Step 9.

- **Zoom** to **Surface 17** in the graphics display
- **Right Click** on **Surface 17** in the Geometry Repair window and select **Remove**

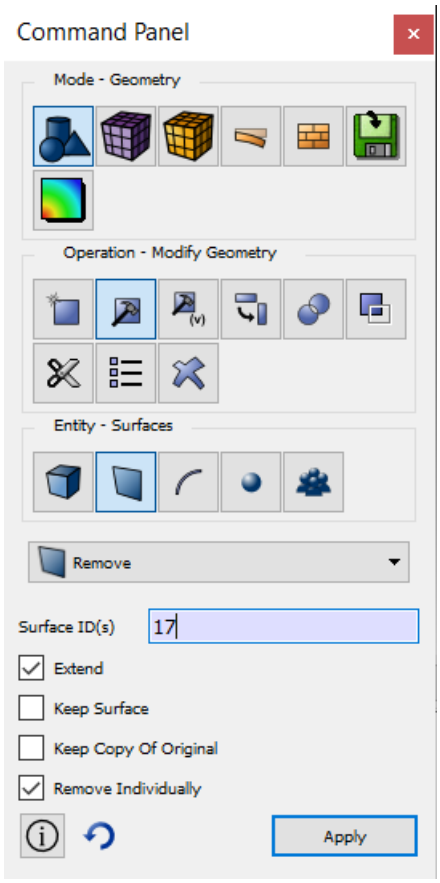
OR

- Press the **Remove Button**  on the tool bar

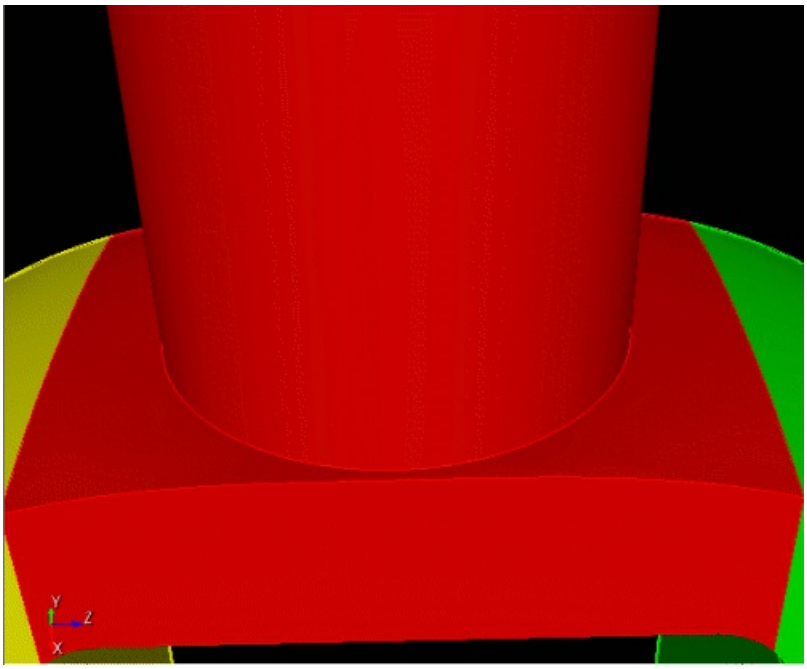


The Control Panel will appear under the Geometry-Surface-Modify-Remove heading. The Surface id should appear in the input window.

- Make sure that **Surface 17** appears in the window and the **Extend** button is checked
- Press **Apply**

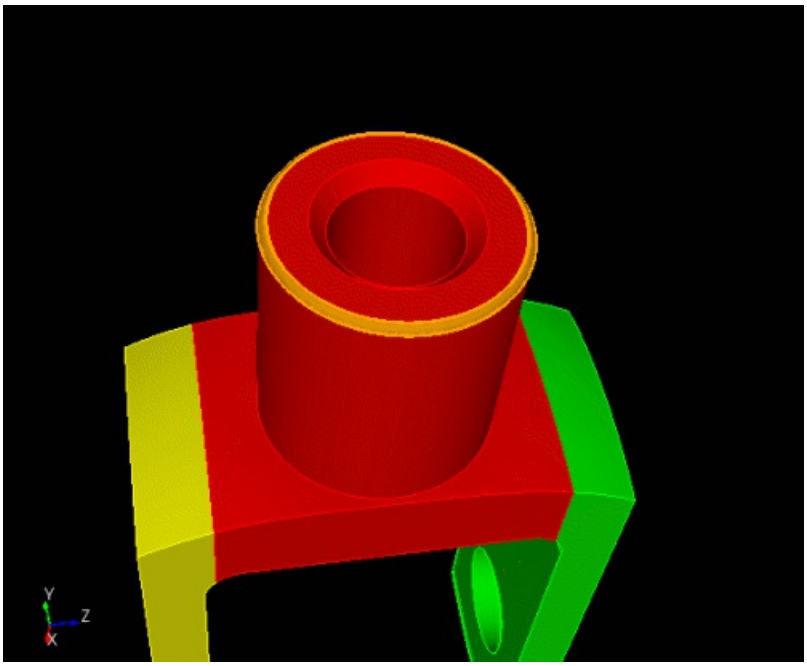


The small surface no longer appears.



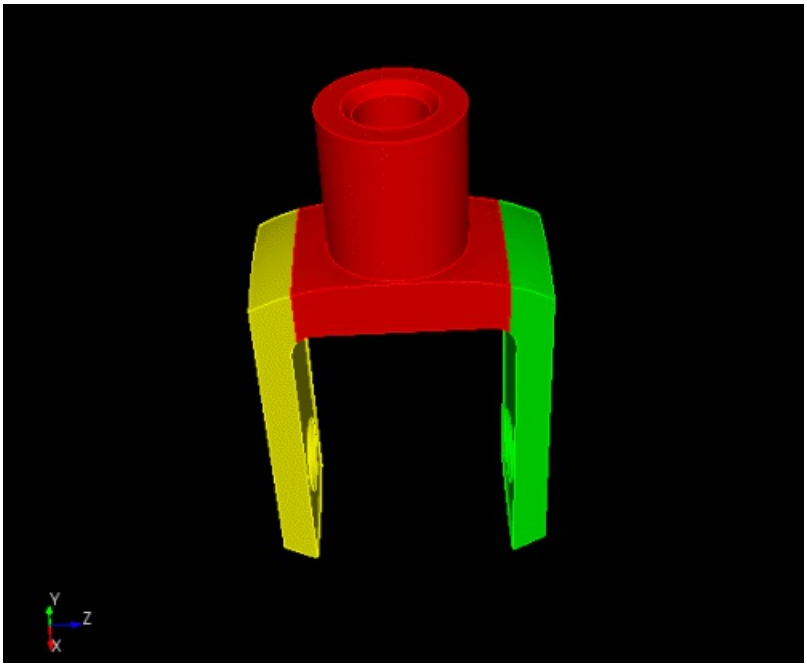
- Highlight **Surface 15** and select the **Remove** option

Surface 15 is shown highlighted in the following image.



- The Geometry-Surface-Modify-Remove option appears on the Control Panel. Make sure that **Surface 15** appears in the input window.
- Press **Apply**

Reset the Zoom to show the entire model.


Y
Z
X

Power Tools GUI Tutorial Step 8

Step 8: Tweaking Surfaces

Tweaking is the process of deleting, moving, or offsetting, surfaces and extending or trimming adjacent surfaces to fill in the gaps. Tweaking is useful for eliminating gaps between components, simplifying geometry or changing the dimensions of an entity. Tweaking will be used in this example to decrease the radius of the upper cylinder.

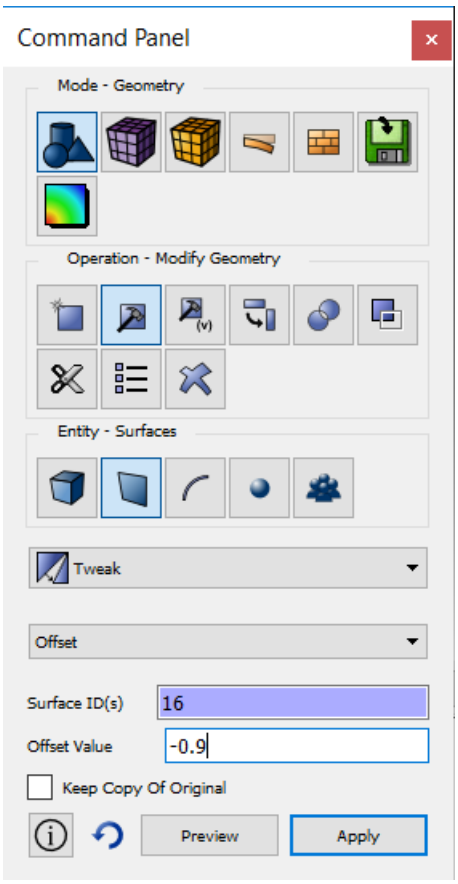
Begin by reanalyzing the geometry.

- Press **Analyze** on the Power Tools menu

There should be 1 entry under the "Close Loops" category for Surface 38. A close loop (pronounced KLOS) is a surface which has two loops that are within some small distance of each other at their closest points. The parameter for distance is the square of the shortest edge length parameter.

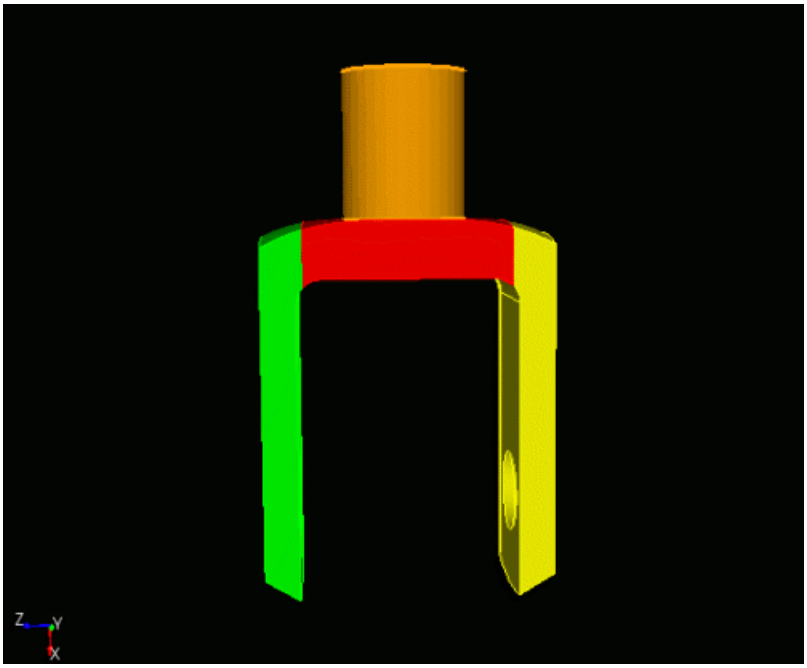
- Press the **Tweak Button**  (since you are not tweaking Surface 41 directly, the surface does not need to be highlighted when you press the tweak button)

The **Geometry-Surface-Modify-Tweak** will open on the Control Panel as shown below.



- Enter **Surface 16** by typing it in at the input line or selecting from the graphics window
- Select the **Offset** option from the pull-down menu

Surface 16 is shown highlighted below.

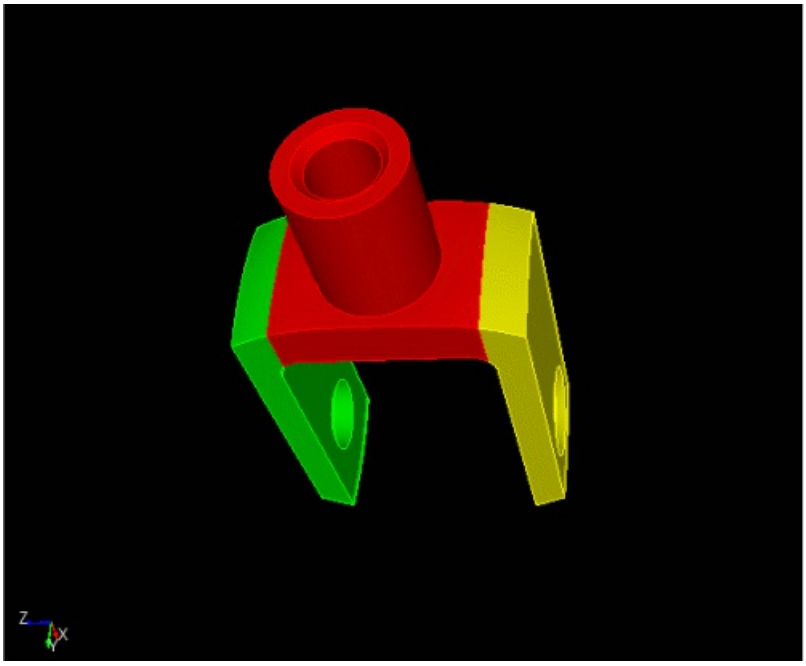


- Enter an Offset **Value** of **-0.9**.

The offset value is a percentage of the current size. Entering -0.9 will decrease the radius by 10 percent.

- Press **Apply**

The graphics window should now look like this. Notice that the radius of the cylinder has shrunk inward, increasing the gap between the edges on Surface 41.

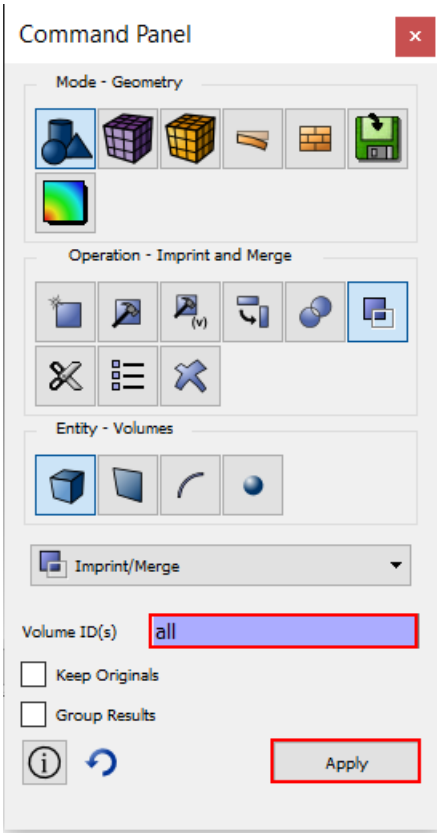


Power Tools GUI Tutorial Step 9

Step 9: Imprint/Merge

Imprinting is the process of projecting curves from one surface onto an overlapping surface. Merging is the process of taking two overlapping surfaces and merging them into one surface shared by two volumes, creating non-manifold geometry. Both imprinting and merging are necessary to make adjacent volumes have identical meshes at their intersection. Imprinting and merging is almost always necessary after webcutting.

- To open the imprint/merge menu, select the **Geometry** icon, then **Imprint and Merge**, then **Volumes** on the Control Panel
- Enter **all** in the input window.
- Press **Apply**



You will not notice any visible changes in the graphics window after imprint/merge operations, but results of the operations will be printed in the output window. Confirm that both surfaces have been merged by reading the output in the graphics window (You may have to scroll to see all of the results)

You can return to the Power Tools menu to see that the Close Loops and Overlapping Surfaces are gone.

- Press **Analyze** in the Power Tools menu

The display window will now read "Nothing Found" to indicate that are no geometry tests that fail.

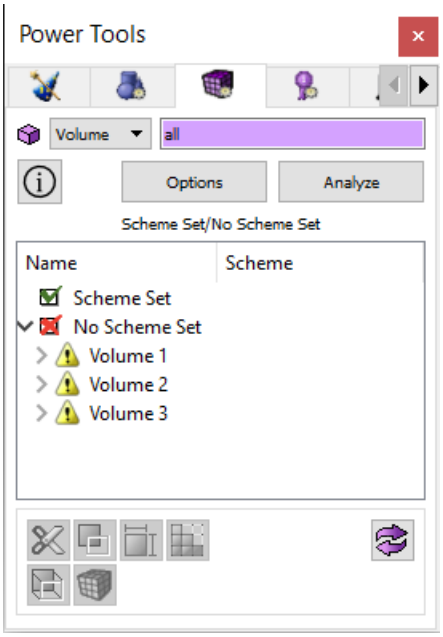


Power Tools GUI Tutorial Step 10

Step 10: Compositing Surfaces

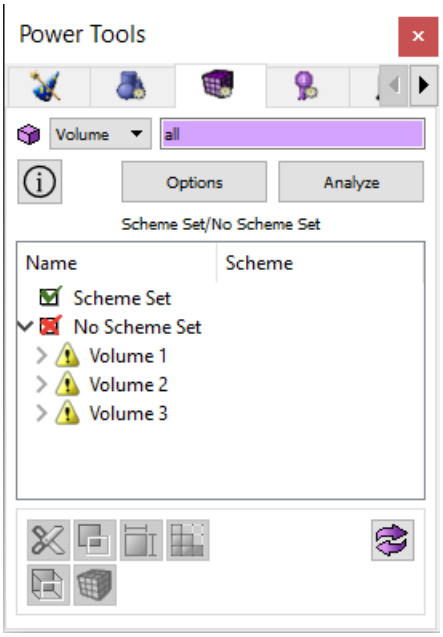
Composite surfaces are adjacent surfaces that have been merged into one surface. Composite surfaces are created using Virtual Geometry, which is a built-in geometry kernel that sits on top of the existing geometry, and does not change the underlying geometry definition. Virtual geometry has the added advantage of being reversible. It can be removed after meshing. The general purpose for using composite surfaces is to deconstrain the mesh. For example, compositing two surfaces will remove the requirement that nodes be placed on the curve between the surfaces. Composite surfaces will be used in this example to facilitate the sweeping algorithm.

- Open the **Mesh Tools** tab
- Enter '**all**' in the input field
- Press **Analyze**
- Toggle the **Reset Graphics** button to show entities in green and red (for meshable and non-meshable volumes)

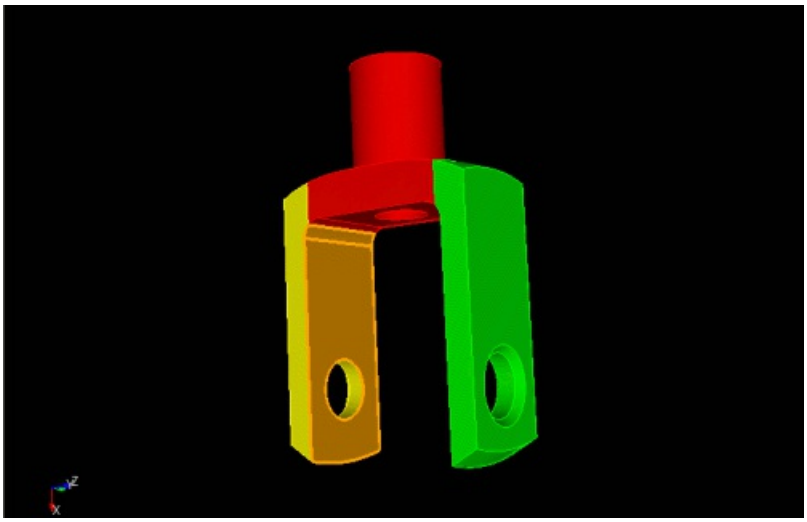


No volumes are listed as automatically meshable. In the graphics window, red indicates that the volume scheme has not been set. Green indicates that the scheme has been set.

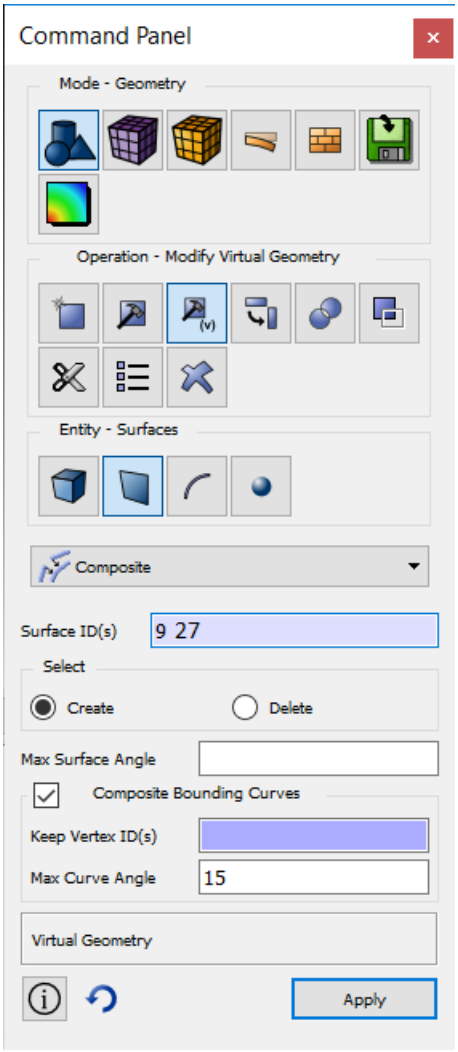
- Toggle the **Reset Graphics** button so it returns to the normal colors
- Open the **Geometry Tools** tab
- Press the **Composite Button**  on the toolbar



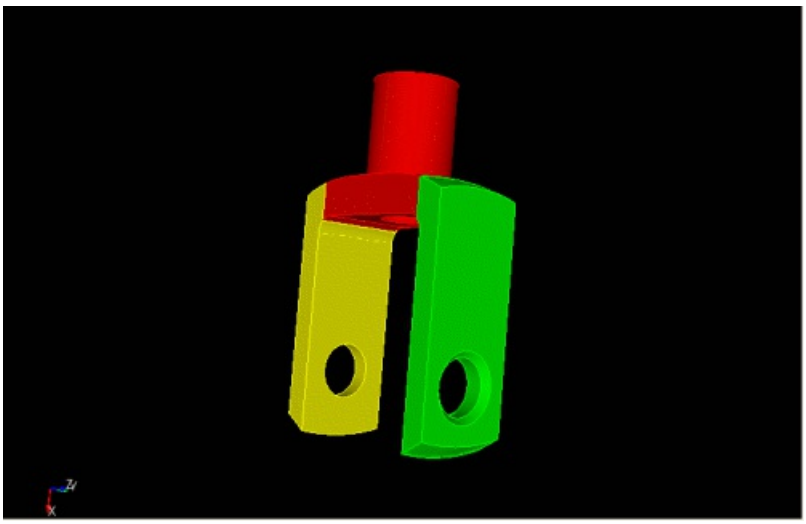
The Geometry-Surface-Modify-Composite menu will open on the Control Panel.



- Select **Surfaces 9** and **27** (shown in the image above) by entering them in at the input line, using CTRL-Click (Windows) in the graphics window, or Command Key-Click (Macintosh) in the graphics window
- Make sure the **Create** button is checked
- Press **Apply**

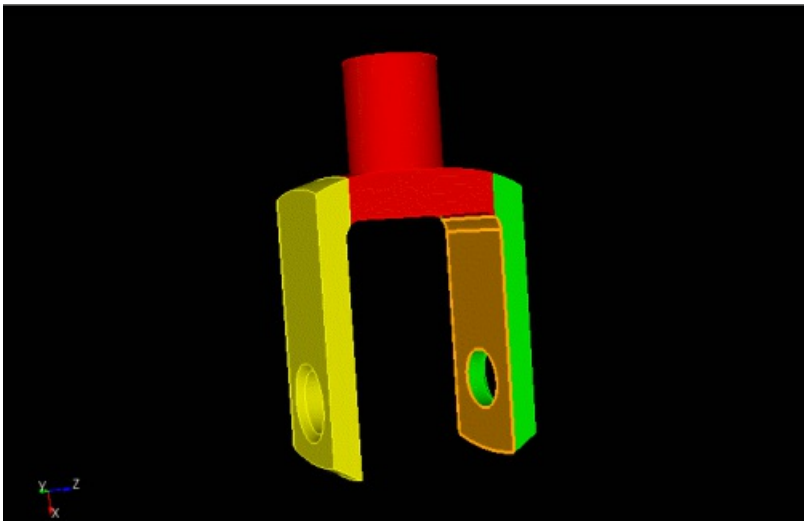


The two surfaces should appear merged.

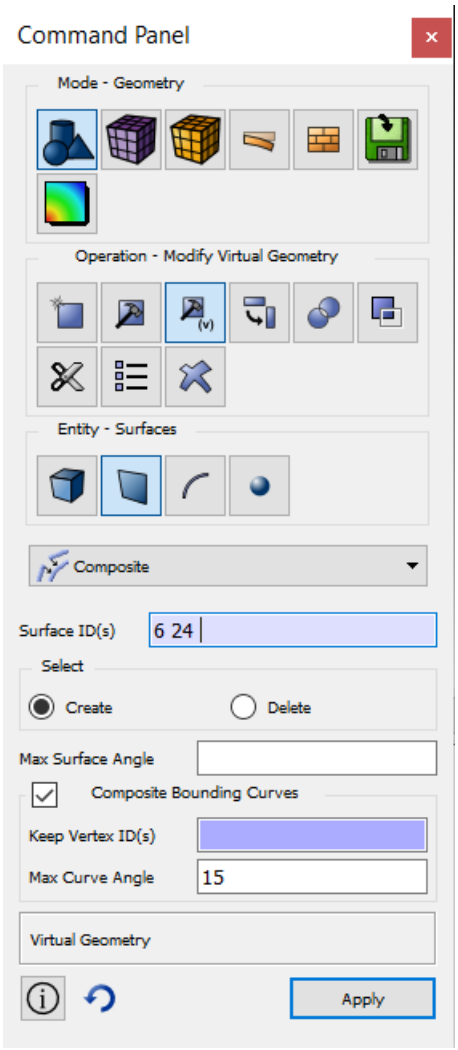


Repeat these steps with the opposite side.

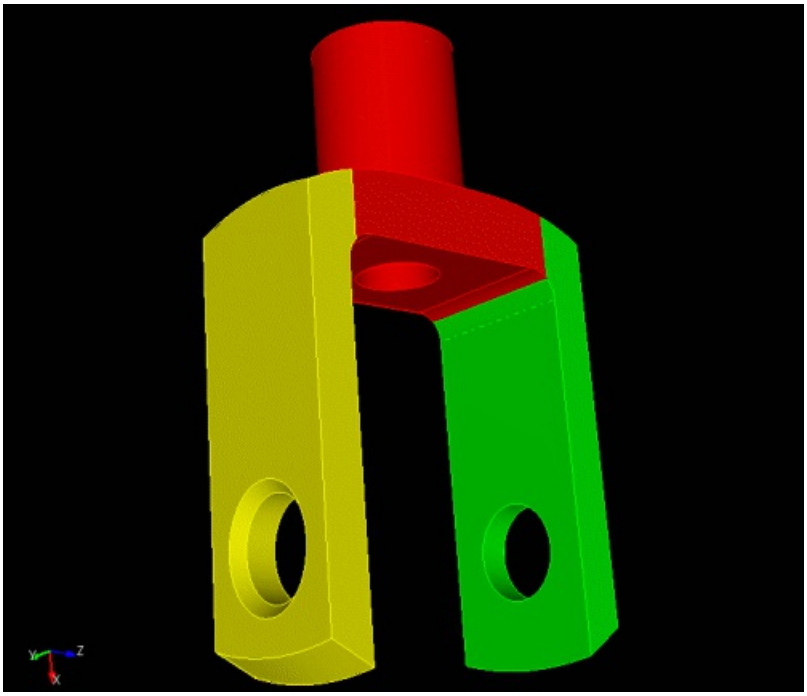
- Rotate the view window so **Surface 6** and **24** are visible



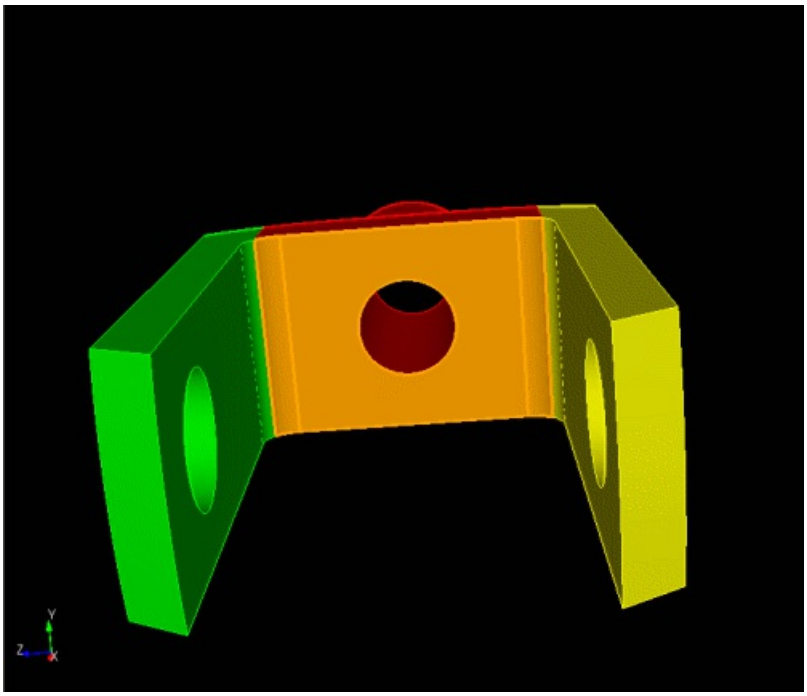
- Select **Surface 6** and **Surface 24** by using CTRL-Click (Windows), Command Key-Click (Macintosh), or entering the ids the input window
- Press Apply



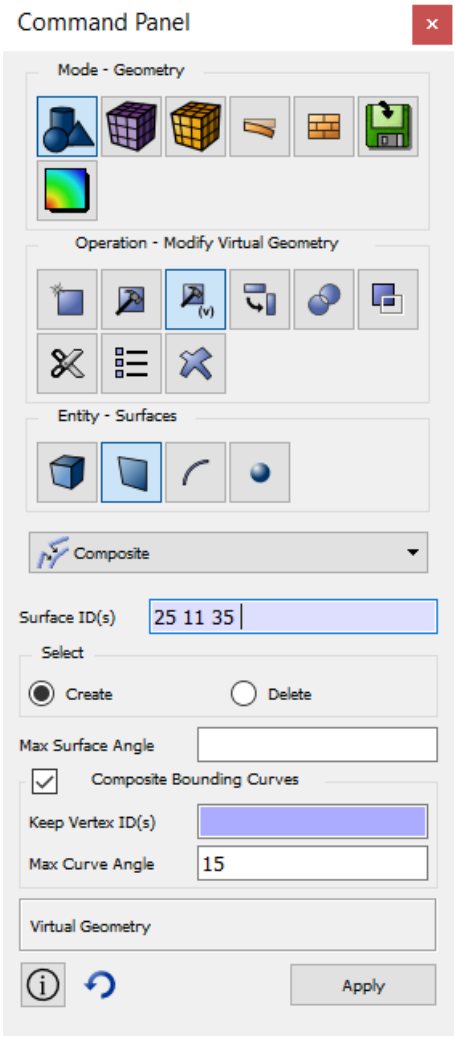
Check to see that the surfaces have been composited and that your graphics window looks like the following image.



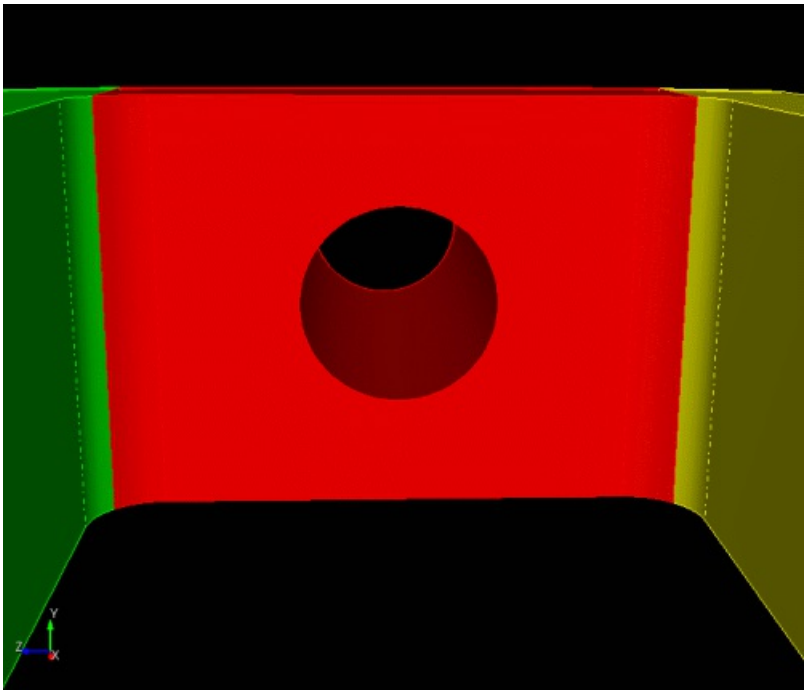
Finally, surfaces 11, 25, and 35 (shown below) need to be composited.



Use the command panel to choose surfaces for the composite command.



Press the **apply** button and check the results in the graphics window.

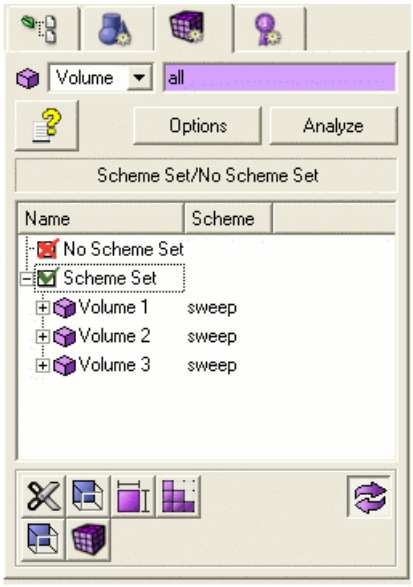


Power Tools GUI Tutorial Step 11

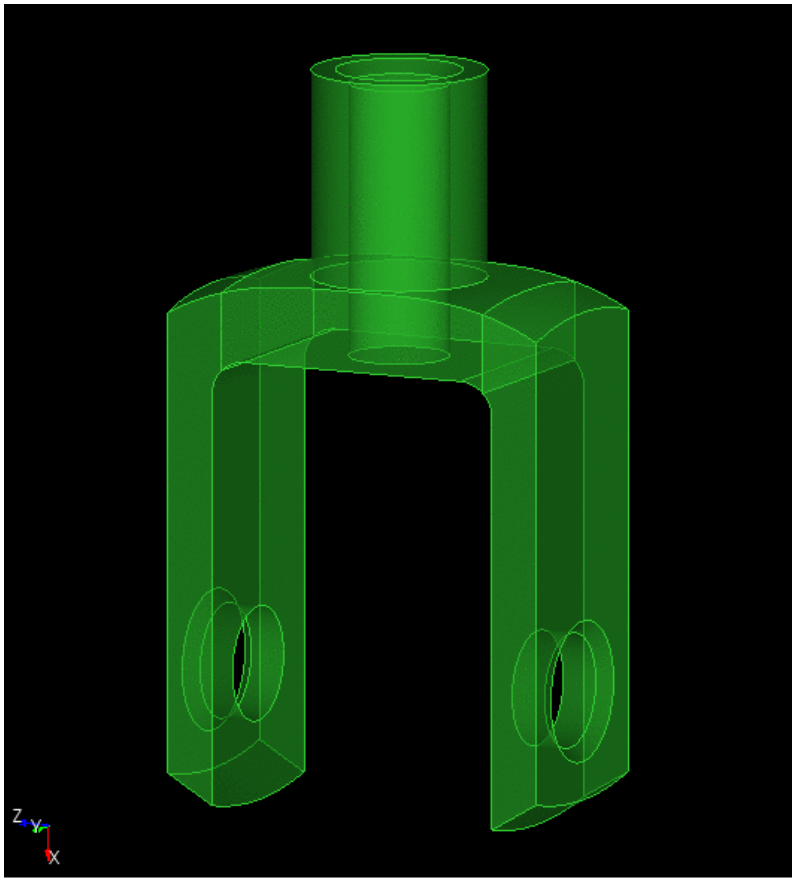
Step 11: Meshing the Model

Use the Mesh Power Tools to apply schemes to the remaining volumes.

- Press the **Mesh Tools** tab in the Power Tools window
- Press **Analyze**



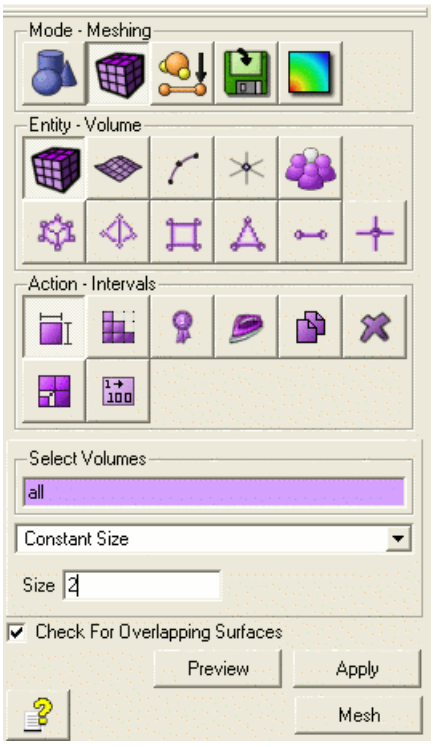
All of the schemes have now been set with a sweeping algorithm. The model is ready to be meshed. All volumes should appear green in the graphics window.



- Toggle the **Reset Graphics** button to return volumes to their original colors

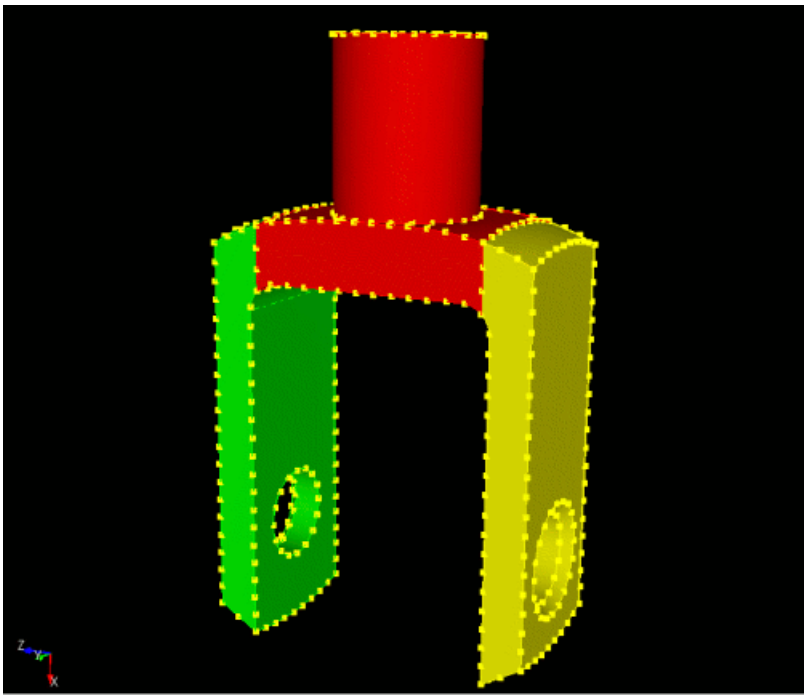
Select **Volume** as the entity, and **Intervals** as the Action.

- Enter **all** in the "Select Volumes" input window
- Select **Constant Size** from the list of sizing options
- Enter **2** for the size
- Press **Apply Size**
- Press **Preview**



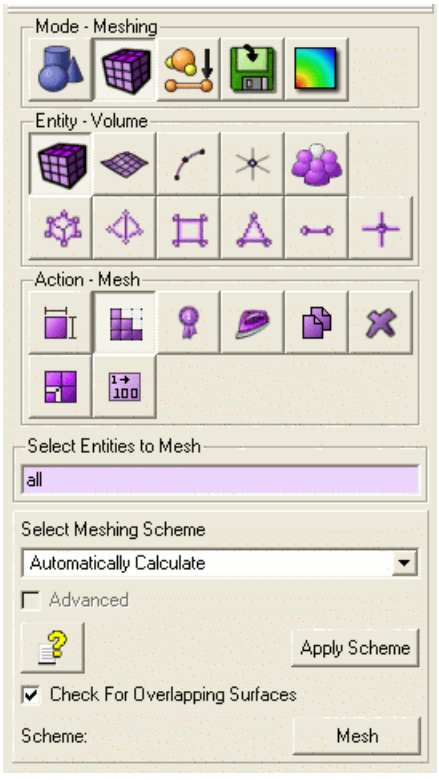
The graphics window should appear as follows, with the mesh size

increments highlighted on all of the curves in the model.

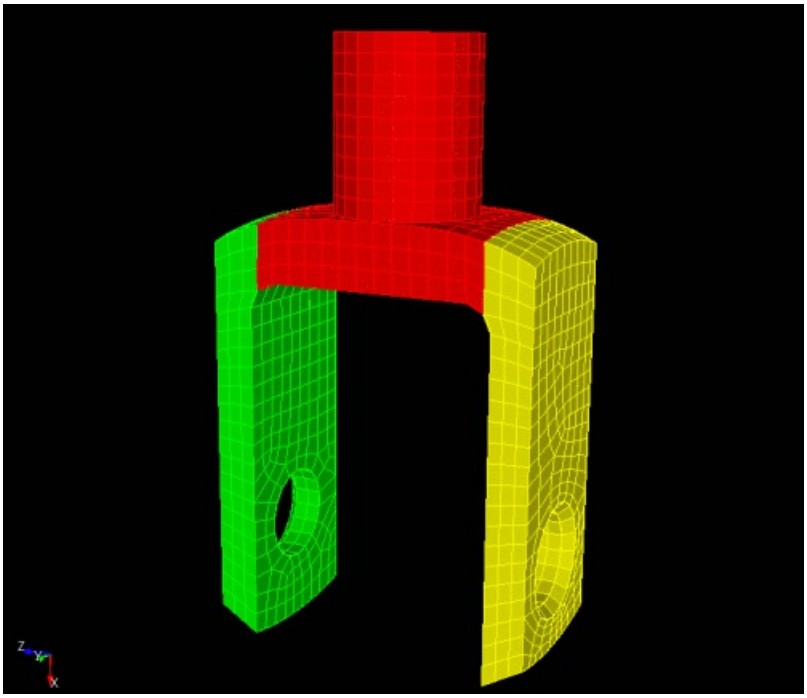


- Go to Mode - Meshing, Entity - Volume, Action - Mesh, and press the **Mesh** Button

There is no need to press the Apply Scheme button since the scheme have already been set in the Meshing Tools.



The final mesh should look like this:



Congratulations! You have just completed the Power Tools Tutorial. Click on the arrow to return to the Tutorial home.

Decomposition Tutorial

Creating Sweepable Volumes Through Webcutting

Most volumes require some measure of decomposition before they can be meshed with a hexahedral meshing scheme. The most common hexahedral meshing tool is the sweeping algorithm. Sweeping is the process of creating a hexahedral mesh by extruding a quadrilateral surface mesh from a source surface onto a topologically similar target surface by way of a linking surface. The surface mesh can be meshed with any surface meshing scheme (i.e. structured or unstructured mesh), but the most common surface meshing scheme for the sweeping algorithm is the pave scheme. In fact, the sweeping algorithm is sometimes called the "pave-sweep" algorithm. Most volumes aren't automatically sweepable, which is why geometry decomposition is so important to the meshing process. Decomposition usually involves a series of webcutting, boolean, and virtual geometry operations that break up a larger model into sweepable regions. Studies have shown that this step in the meshing process is the most time consuming for the analyst. The goals of this tutorial are for the user to learn to:

1. Recognize sweepable volumes
2. Recognize how to decompose a model into sweepable parts
3. Gain proficiency with webcutting and other decomposition techniques
4. Avoid common pitfalls with decomposition and sweeping

Why use sweeping?

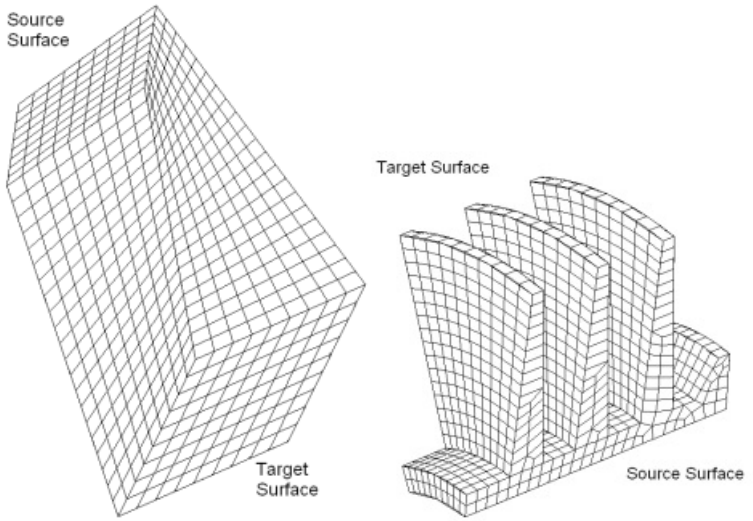
Of all the hexahedral meshing schemes in the Cubit toolkit, sweeping is considered the most reliable at producing high quality elements. Although decomposing a model into sweepable volumes can be time-consuming, and sometimes falls into the realm of trying to fit a square peg into a round hole, the pave-sweep algorithm has a high rate of success, and it sometimes the only way to get a hexahedral mesh on a model.

What makes a volume sweepable?

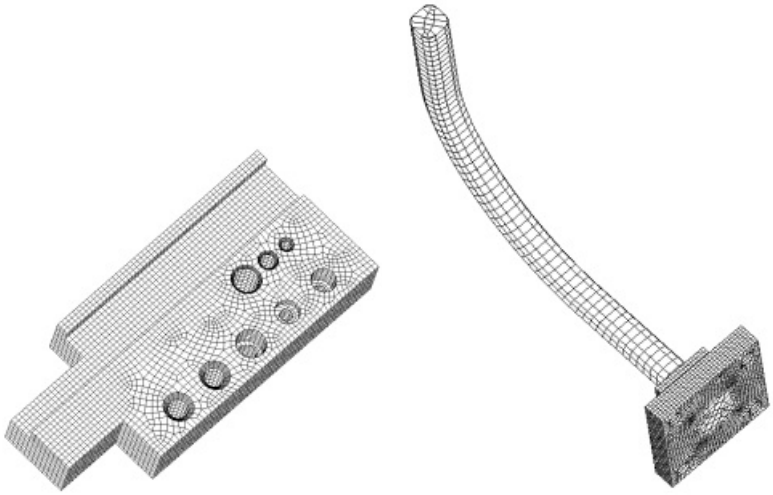
Recognizing sweepable topologies can be an art form. Sweepable volumes can be comprised of many different topologies. We typically classify sweeping problems into three groups, based on the number of source/target surfaces.

Basic Sweep Groups

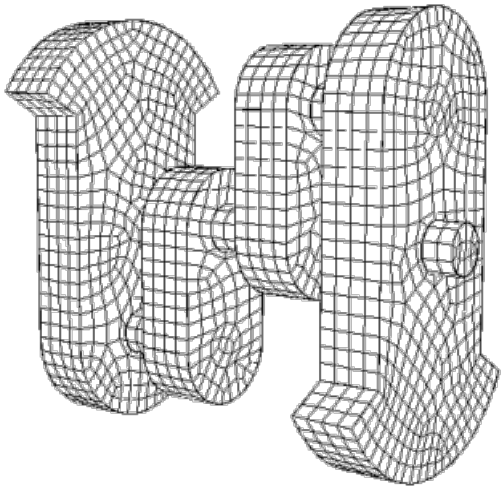
One-to-one: A volume with a one source surface and one target surface.



Many-to-one: A volume with multiple source surfaces and one target surface



Multisweep (or Many-to-Many): A volume with multiple target surfaces



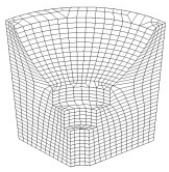
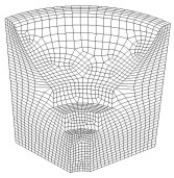
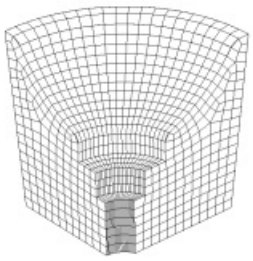
Points to consider when determining whether a volume is sweepable

- Swept surface meshes can be extruded through a volume which is rotated or translated. However, if the translation/rotation is severe then the quality of the resulting mesh may be poor.
- A volume with multiple target surfaces and a single source surface

can sometimes be inverted and handled as a many-to-one sweepable volume. Otherwise, it is treated as a multisweep problem.
- Imprinting introduces new topology onto surfaces. Sweepable volumes may not be sweepable after imprinting and merging adjacent surfaces
- Multisweep is still under development, and has limitations, so if you are having difficulty with the multisweep algorithm, it is usually a good idea to decompose it into many-to-one or one-to-one sweepable regions.
- Cubit won't always automatically recognize your volume as a sweepable volume, even if it is. Sometimes, you have to give it a list of source/target surfaces explicitly.

Basic Sweep Paths

In addition to the different topologies, sweepable volumes can be classified by the sweep direction. These include: top-to-bottom, inside-to-outside, and around (rotational). Be sure to consider all the possibilities for sweep directions when you begin decomposing a model. And keep in mind that sweep paths must be compatible with adjacent volumes. To be compatible, overlapping surfaces must have the same scheme (i.e. both must be a linking surface or a paved surface). The volume below is meshed three different times with the three different sweep directions. Notice the difference in element sizes and orientations between the meshes. See if you can pick out the different source and target surfaces in each example. As an exercise, try to mesh this model with each of the different sweep paths.

		
Top-to-Bottom	Inside-to-Outside	Around (Rotational)
Many-to-one	Many-to-one	One-to-one (this is the default sweep direction for this model)

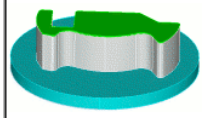
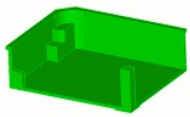
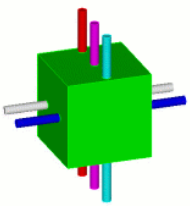
What are some good strategies for decomposing my model?

Recognizing when a volume is sweepable is a difficult task of itself, but being able to come up with viable webcutting, compositing, and boolean strategies to make a volume sweepable is even more difficult, and can only be achieved through practice. Here are some general principles to follow when decomposing a model.

1. Select your sweep path
2. Use as few webcuts as possible
3. Set your own source and target surfaces if Cubit does not pick them automatically
4. If one of your volumes does not mesh, or has an undesirable mesh, try changing the order in which you mesh volumes. This will hardset the intervals on the volumes.
5. The Reset Volume command will remove all schemes and interval settings from volumes.
6. If changing the mesh order or resetting the volumes does not work and you continue to get "Matching Intervals Failed" errors, set explicit intervals on some or all curves.
7. Make additional webcuts if necessary.
8. Check for sliver surfaces or curves that may have been introduced

during decomposition and remove these through tweaking collapsing, or compositing.
9. Change surface vertex types on mapped or submapped surfaces if you need to force a certain configuration
10. Use partitioning to introduce virtual geometry constraints without affecting the underlying geometry
11. Composite surfaces to remove constraints without affecting the underlying geometry
12. Save your work often. For a complex model, the meshing process can be very iterative. You may need to start over many times until you find an acceptable solution.

The following is a compilation of several different decomposition problems of varying difficulty. If you accessed this help from the Cubit program (as opposed to the web documentation), you will need to browse for the geometry files from within your Cubit installation directory. They should be located in the "/components/cubit/help/step_by_step_tutorials/decomposition" directory of the Cubit installation folder.

Example	Image	File
Beginner		
Sweeping through multiple adjacent volumes		example01.sat
Interlocking rings		example02.sat
Webcutting using the "sweep" option		example03.sat
Using the loft command		example04.sat
Multiple sweep directions		example05.sat
Advanced		

Employing symmetry and controlling skew		example06.sat
Using virtual geometry		example07.sat
Sweeping volumes with narrow angles and surfaces		example08.sat

Example 1. Sweeping multiple adjacent volumes

The following model has several interior volumes which share surfaces. This example may at first seem complex, but it actually requires very little decomposition. The key to this example is that each of the interior volumes is already sweepable, oriented along the same sweep axis, and none of the linking surfaces have additional topology introduced through imprint/merge. In fact, there is only one required webcut to make this model automatically sweepable.

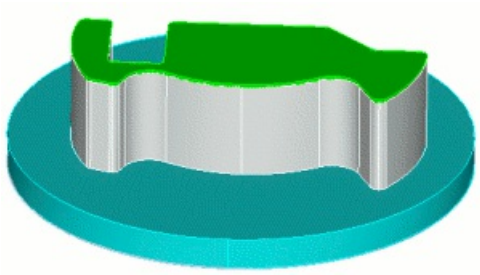


Figure 1. Exterior view

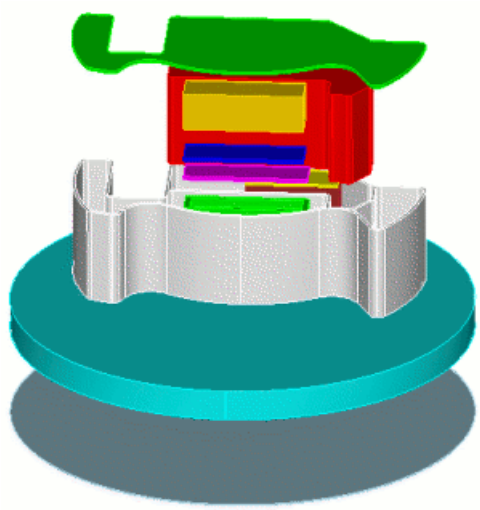
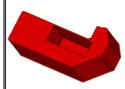
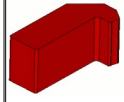
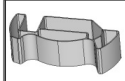
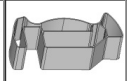
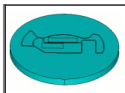
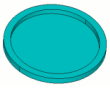


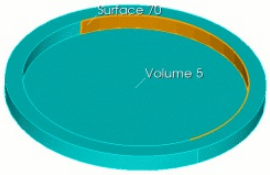
Figure 2. Interior view

We examine several of the volumes below.

Source Surface(s)	Target Surface(s)	Sweep type
		**Many-to-one Sweepable** Imprinting and merging adjacent volumes creates additional partitions on the source surface, but the target surface does not contain imprints.
		**Many-to-one Sweepable** Multiple source surfaces due to interior void
		**One-to-One Sweepable** Source and target surfaces are single surfaces, and there are no imprints on the linking surfaces
		**Many-to-one Sweepable** Interior void causes multiple source surfaces.

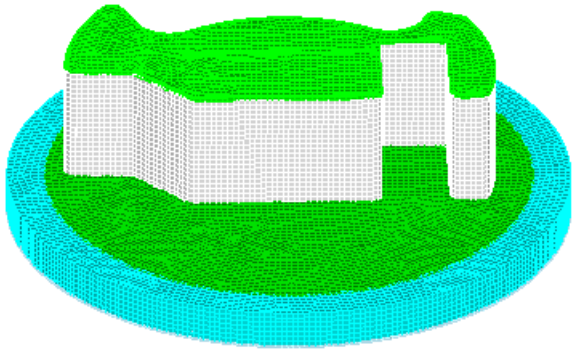
		**Multisweep** Imprinting causes multiple source surfaces and interior void causes multiple target surfaces. This volume requires decomposition

Suggested webcut

Webcut	Command
	**CUBIT> webcut volume 5 with sheet extended from surface 70** **CUBIT> imprint all** **CUBIT> merge all** **CUBIT> volume all size 0.15** **CUBIT> volume all scheme auto**

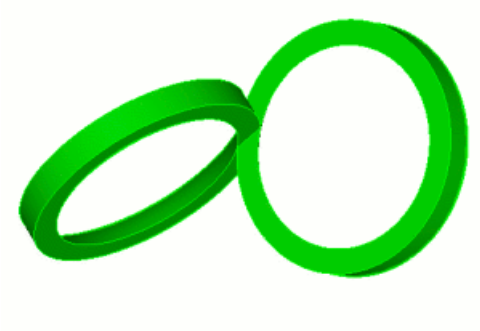
Final mesh

The final mesh is created at a size of 0.15 for all volumes.

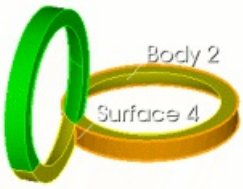


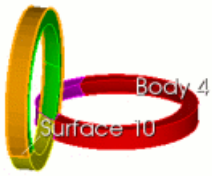
Example 2. Interlocking rings

The following example is composed of two rings of constant cross-section that can be swept along their axes. The problem here is that the rings overlap, forming a tetrahedral shape which cannot be swept. The key to solving this problem is separating out the region of overlap, explicitly setting the source and target surfaces, and using the tetprimitive scheme on the tetrahedral region.



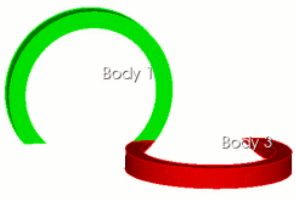
Suggested webcuts

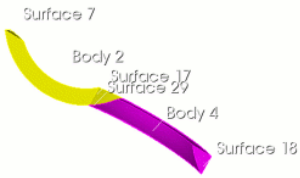
Webcut	Command
	`CUBIT> webcut body 1 plane surface 5`
	`CUBIT> webcut body 2 sheet extended from surface 4`
	`CUBIT> webcut body 3 plane surface 12`

	`CUBIT> webcut body 4 sheet extended from surface 10`
	`CUBIT> imprint all`
	`CUBIT> merge all`

There are five volumes that result from the webcutting. Two of them are automatically sweepable. Two of them must have their schemes set explicitly, and one of them is meshed using the tetprimitive scheme.

Webcut	Command

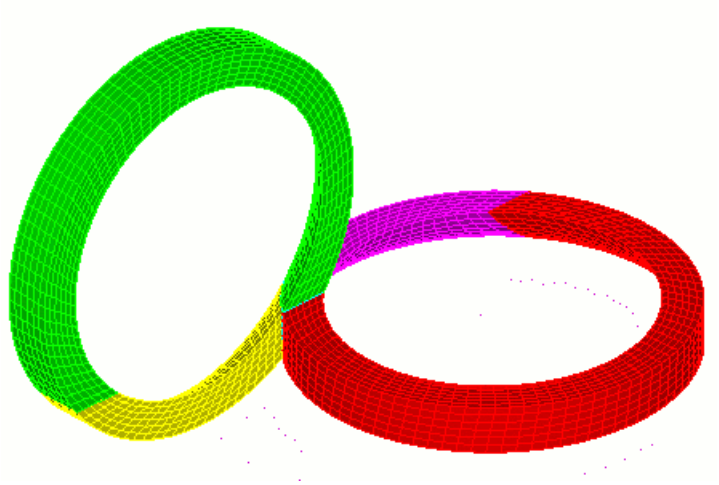
	One-to-one Sweepable Source and target are set automatically using autoscheme `CUBIT> volume 1 3 scheme auto`

	One-to-one Sweepable
	Must have source and target set explicitly
	`CUBIT> volume 2 scheme sweep source 17 target 7`
	`CUBIT> volume 4 scheme sweep source 29 target 18`

	Non-sweepable
	Use the tetprimitive scheme
	`CUBIT> curve in volume 5 interval 6`
	`CUBIT> volume 5 scheme tetprimitive`
	`CUBIT> volume all size 0.5`
	`CUBIT> mesh volume all`

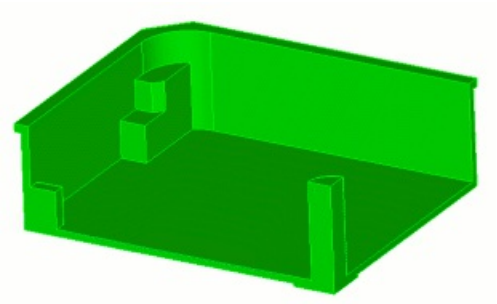
Final mesh

The final mesh is created at a size of 0.5 for all volumes.

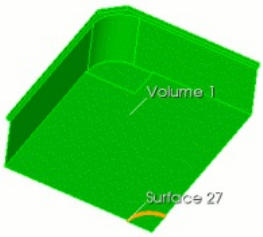
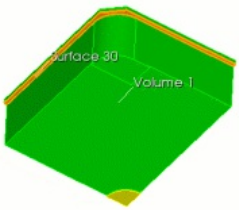
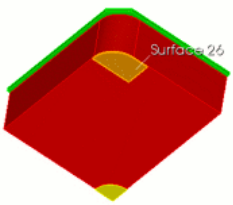


Example 3. Webcutting using the sweep option

This example introduces additional webcutting options. This example would be a simple many-to-one sweep except for the overhanging lip and the protrusions on the bottom surface. To a beginner user, it may at first seem reasonable to use the bottom surface as a webcutting plane. However, this will not create a many-to-one sweepable volume. Instead, you need to use the protruding surfaces as cutting planes, and extend them through the entire volume.

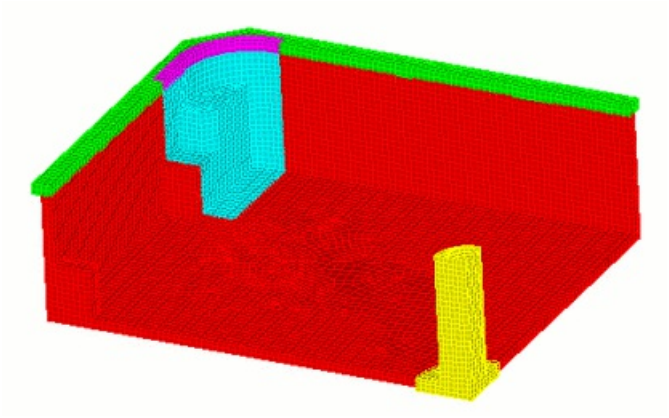


Suggested webcuts

Webcut	Command
	CUBIT> webcut volume 1 with sheet extended from surface 27
	CUBIT> webcut volume 1 with plane surface 30
	CUBIT> webcut vol all sweep surf 26 vector -1 0 0 through_all Now Volume 3 (red) has only 1 target surface. CUBIT> imprint all CUBIT> merge all CUBIT> volume all size 0.05 CUBIT> mesh volume all

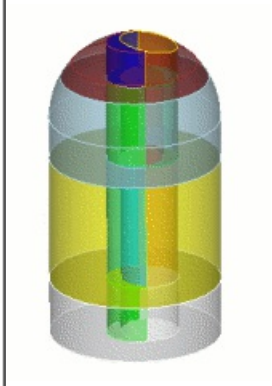
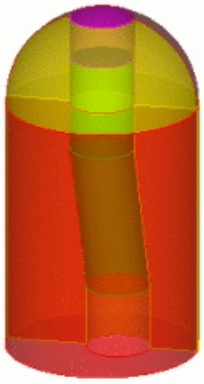
Final mesh

The final mesh is created at a size of 0.05 for all volumes.

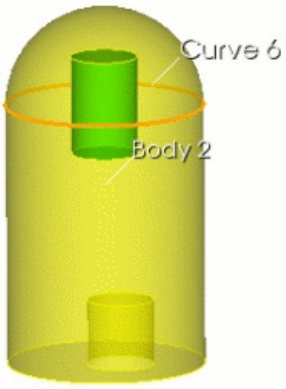
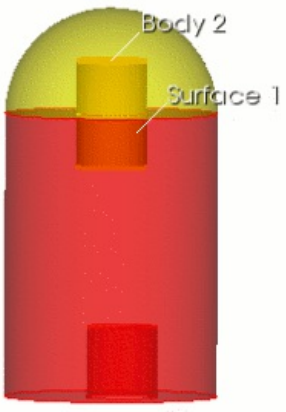


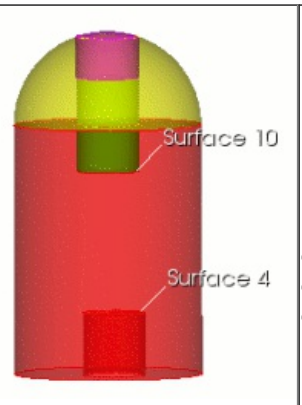
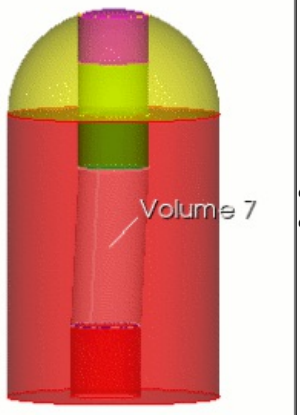
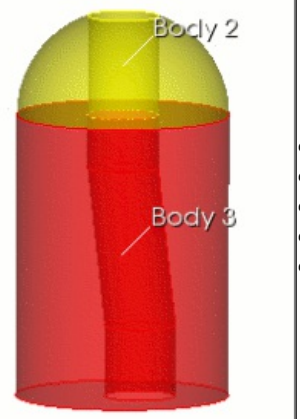
Example 4. Using the Loft command

In the next example, the loft command significantly decreases the number of required webcuts. This model also demonstrates using 2 separate sweep paths (top-to-bottom and rotational) on adjacent volumes.

		
Original Volume	Webcuts created from sweeping surfaces (not recommended)	Webcuts using loft command (recommended)

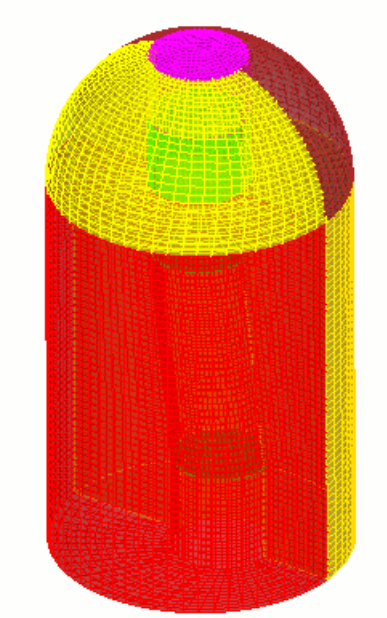
Suggested webcuts

Webcut	Command
	`CUBIT> webcut body 2 loop curve 6`
	`CUBIT> webcut body 2 sheet extended from surface 1`

	CUBIT> create surface from surface 10 CUBIT> create surface from surface 4 CUBIT> create body loft surface 19 20
	CUBIT> webcut body 3 tool body 7 CUBIT> delete body 5 6 7
	CUBIT> webcut body 2 3 plane yplane CUBIT> imprint all CUBIT> merge all CUBIT> volume all size 0.15 CUBIT> mesh volume all

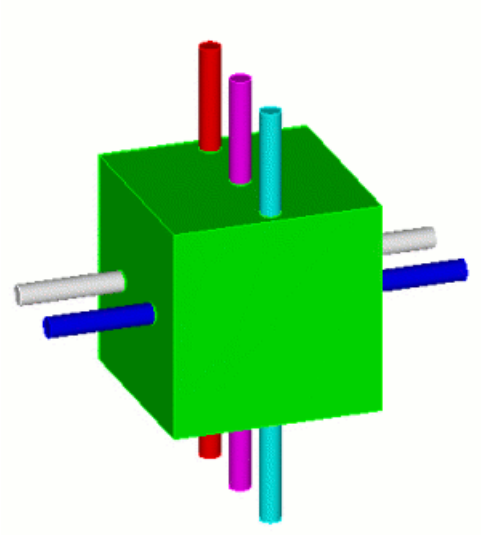
Final mesh

The final webcut model consists of a central shaft which can be swept top to bottom, and a surrounding casing which can be swept around. This is possible because the shared surface is a linking surface for both types of sweeps. The final mesh is created with a size of 0.15

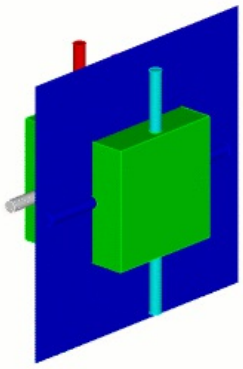


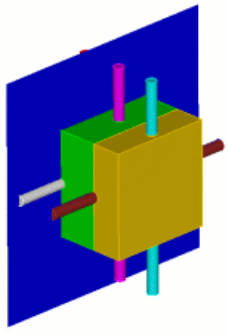
Example 5. Multiple sweep directions

The next example gives another example of using different sweep directions on the same model. The following model shows a brick which is perforated by several cylindrical shafts. The shafts do not intersect each other.

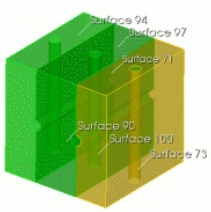


Suggested webcuts

Webcut	Command
	`CUBIT> webcut volume all with plane yplane offset 20`

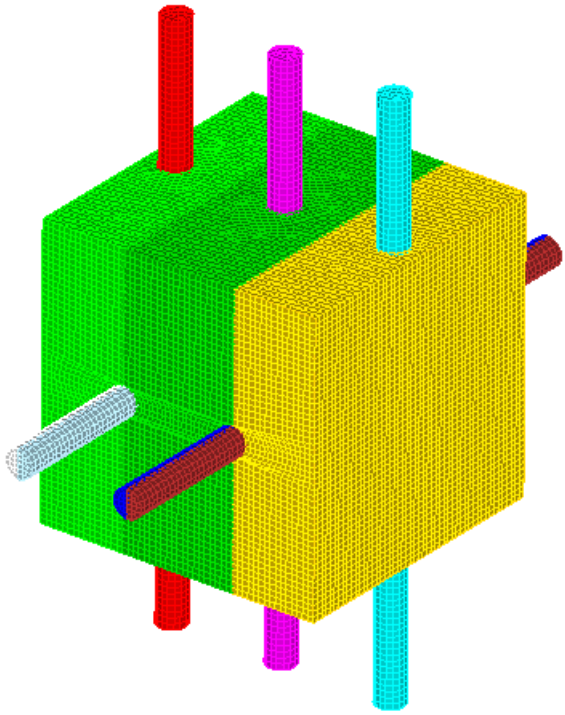
	`CUBIT> webcut volume all with plane yplane offset -20`
	`CUBIT>imprint all`
	`CUBIT>merge all`

All of the volumes in this model are now one-to-one sweepable. However, the source and target surfaces for the main block portions must be set explicitly

	CUBIT>volume 8 scheme Sweep source surface 94 target surface 90 rotate off CUBIT>volume 10 scheme sweep source surface 71 target surface 73 rotate off CUBIT>volume 12 scheme Sweep source surface 97 target surface 100 rotate off CUBIT>volume all size 2 CUBIT>mesh volume all

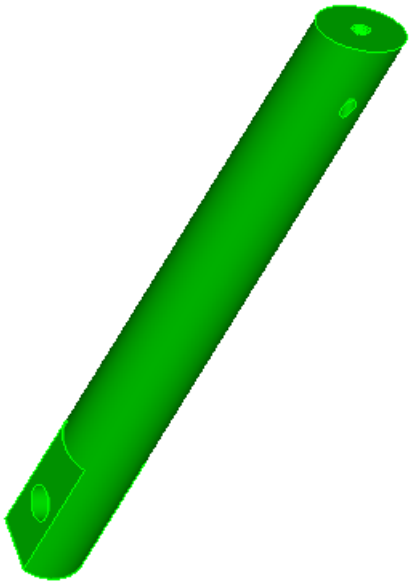
Final mesh

In this model it is possible to have different sweep directions since the surfaces which overlap are both linking surfaces. The final mesh is created with a mesh size of 2 and is shown below.

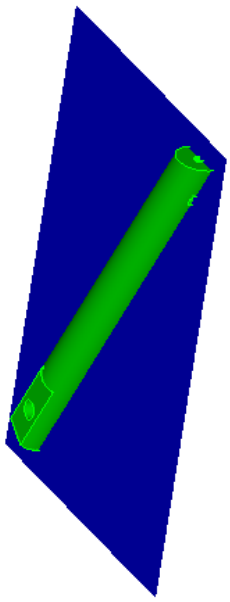
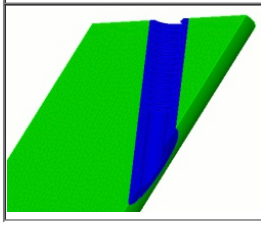


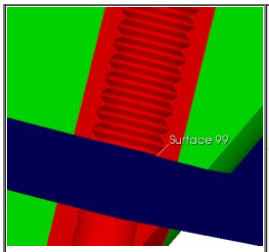
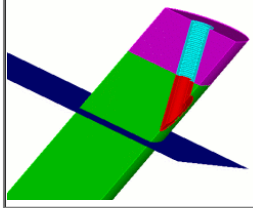
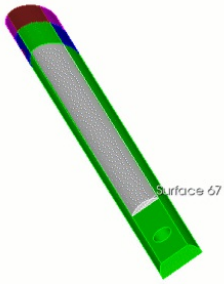
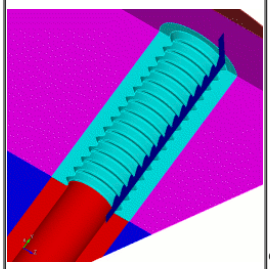
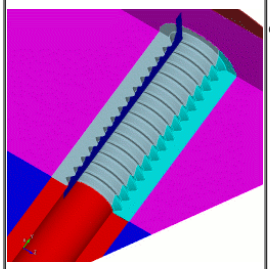
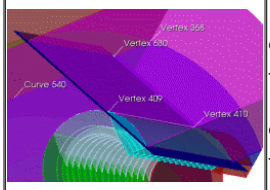
Example 6. Employing Symmetry

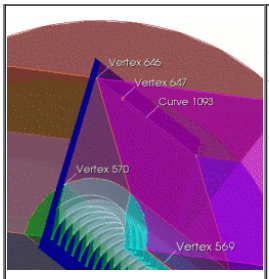
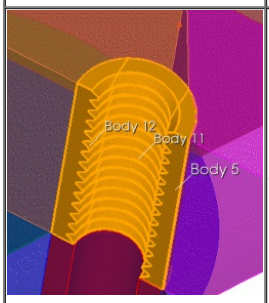
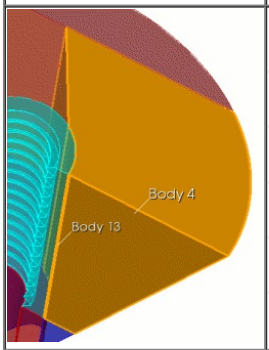
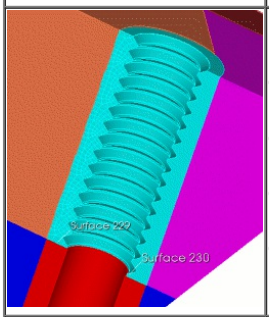
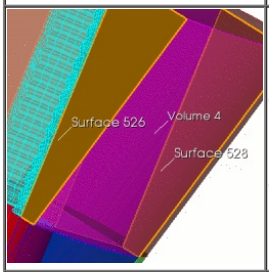
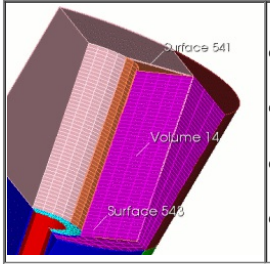
One technique for creating a symmetric mesh on a symmetric model is to mesh only half of the volume, then copy the mesh onto the other half. The following example employs this technique. This model at first appears quite simple, but it actually requires a good deal of webcutting to get a reasonable mesh that is not highly skewed.

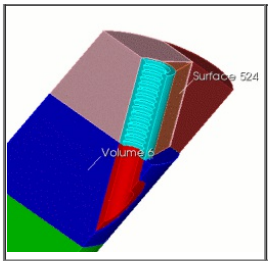
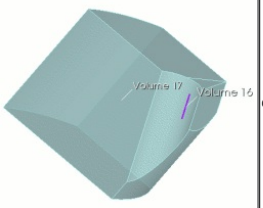
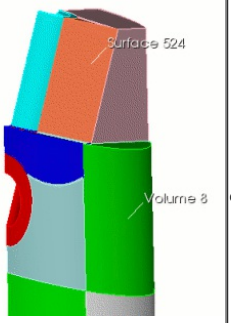
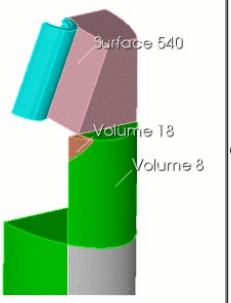
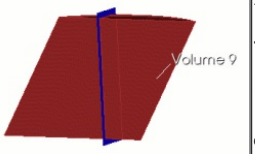
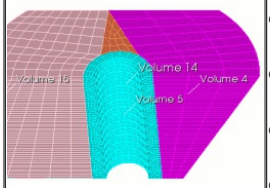


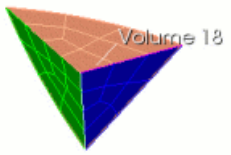
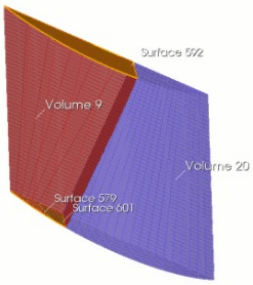
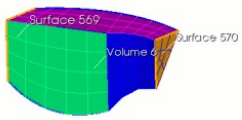
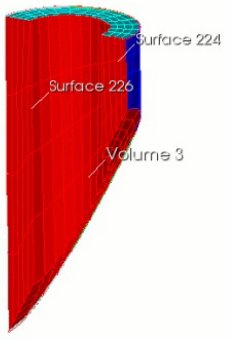
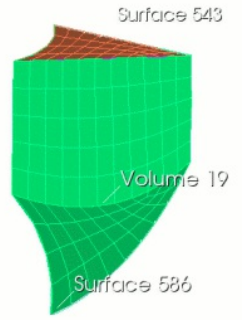
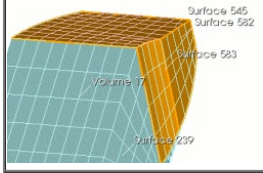
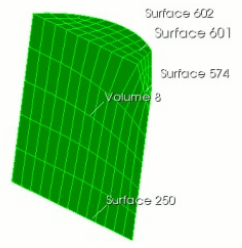
Suggested webcuts

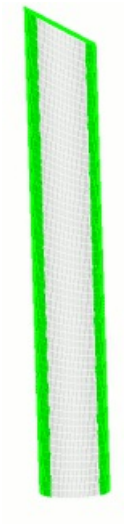
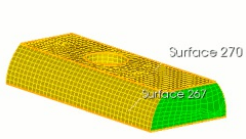
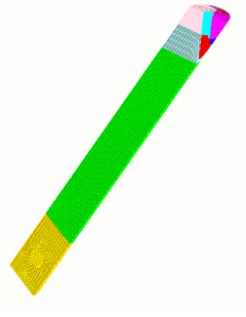
Webcut	Command
	CUBIT> webcut body 1 with plane xplane offset 0 CUBIT> delete body 2
	CUBIT> webcut body 1 with cylinder radius 2.75 axis y

	CUBIT> webcut body 1 3 with plane yplane offset 0
	CUBIT> webcut body 1 with plane yplane offset -15
	CUBIT> webcut body 1 6 4 with plane surface 64
	CUBIT> webcut body 1 with plane surface 67
 	CUBIT> webcut body 5 with plane zplane offset 1.5 CUBIT> webcut body 11 with plane zplane offset -1.5
	CUBIT> create vertex on curve 540 distance 2 from vertex 368 CUBIT> webcut body 4 with plane vertex 409 vertex 410 vertex 630

	CUBIT> create vertex on curve 1093 distance 3 from vertex 646 CUBIT> webcut body 14 with plane vertex 570 vertex 569 vertex 647 This wedge shape webcut is a method of controlling skew in the final mesh.
	CUBIT> unite body 5 11 12
	CUBIT> unite body 4 13 CUBIT> delete vertex all CUBIT> imprint all CUBIT> merge all CUBIT> vol all size .5
	CUBIT> surf 229 size .25 CUBIT> mesh surf 229 CUBIT> volume 5 scheme sweep source 229 target 230 CUBIT> mesh volume 5
	CUBIT> volume 4 scheme sweep source surface 526 target 528 CUBIT> mesh volume 4
	CUBIT> volume 14 scheme sweep source 543 target 541 CUBIT> mesh volume 14 CUBIT> delete mesh CUBIT> unmerge all

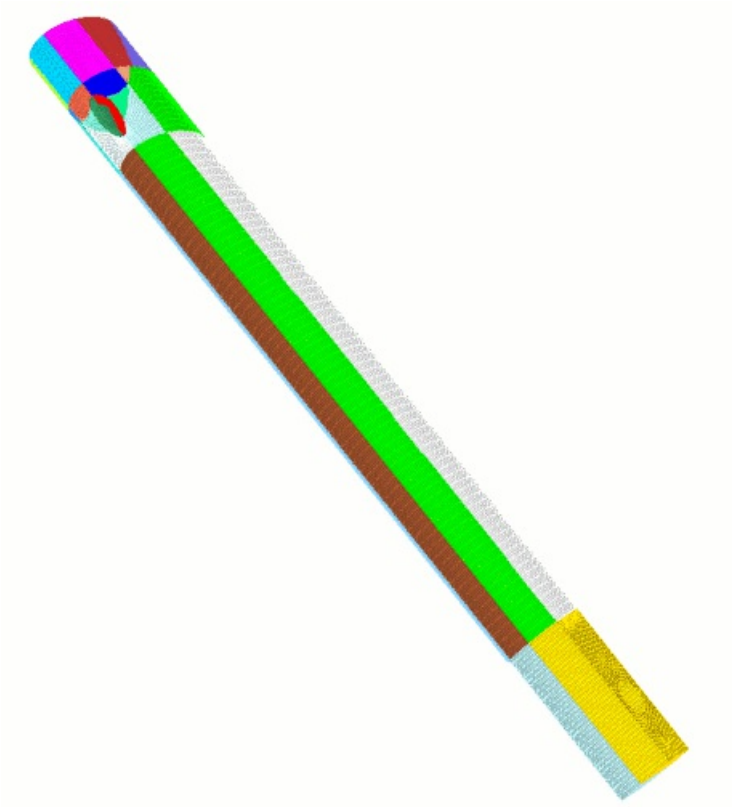
	CUBIT> webcut body 6 with plane surface 524
	CUBIT> unite body 16 17
	CUBIT> webcut body 8 with plane surface 524
	CUBIT> webcut body 18 with plane surface 540
	CUBIT> webcut volume 9 with plane zplane offset -3 CUBIT> rotate 5 about x This is another effort to prevent skew in the final mesh CUBIT> imprint all CUBIT> merge all
	CUBIT> mesh volume 5 (swept around) CUBIT> mesh volume 4 (mapped) CUBIT> mesh volume 14 (swept top to bottom) CUBIT> volume 15 scheme map CUBIT> curve all in volume 15 size 0.5 CUBIT> mesh volume 15

	**CUBIT> volume 18 scheme tetprimitive** **CUBIT> volume 18 interval 3** **CUBIT> mesh volume 18**
	**CUBIT> volume 9 scheme sweep source surface 579 601 target surface 592 rotate off** **CUBIT> mesh volume 9** **CUBIT> mesh volume 20**
	**CUBIT> volume 6 scheme sweep source 569 target 570** **CUBIT> mesh volume 6**
	**CUBIT> volume 3 scheme sweep source 224 target 226** **CUBIT> surf 224 226 scheme map** **CUBIT> mesh volume 3**
	**CUBIT> volume 19 scheme sweep source 543 target 586** **CUBIT> mesh volume 19**
	**CUBIT> volume 17 scheme sweep source 545 583 582 target 239** **CUBIT> mesh volume 17**
	**CUBIT> volume 8 scheme sweep source 574 597 601 target 241** **CUBIT> mesh volume 8**

	CUBIT> volume 7 1 size 2 CUBIT> volume 7 1 scheme auto
	CUBIT> volume 10 scheme sweep source 270 target 267 CUBIT> mesh volume 7 1 CUBIT> mesh volume 10
	CUBIT> unmerge all CUBIT> body all copy reflect x CUBIT> merge all

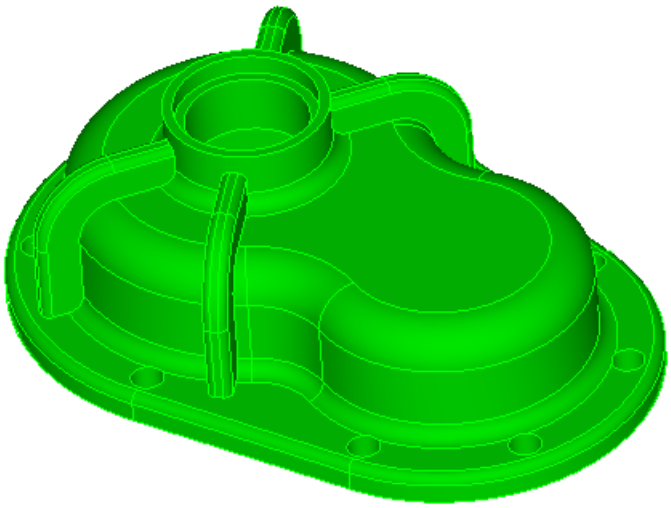
Final mesh

The entire mesh is copied and reflected around the x axis during the last step. The advantage of symmetry in this example is that it cuts the decomposition in half, and it also ensures a perfectly symmetrical mesh.



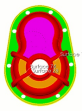
Example 7. Using virtual geometry in geometry decomposition

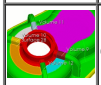
Virtual geometry is used to change the properties of mesh without changing the underlying geometry. The next example uses virtual geometry to remove unwanted sliver curves, and to create a sweepable volume. The composite curve function is used to combine sliver curves that are created from webcutting a slightly curved surface. Then the partition surface command is used to create additional partitions on a surface to ensure sweepability.

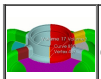


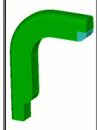
Suggested webcuts

Webcut	Command
	`CUBIT> webcut volume 1 sweep surface 2 vector 0 0 -1 through_all`
	`CUBIT> webcut volume 3 sweep surface 108 vector 0 0 -1 through_all`

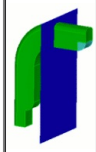
	`CUBIT> webcut volume 3 sweep surface 13 vector 0 0 -1 through_all`
	`CUBIT> webcut volume 3 sweep surface 28 vector 0 0 -1 through_all`
	`CUBIT> webcut volume 3 sweep surface 74 vector 0 0 -1 through_all`

	`CUBIT> webcut volume 3 with sheet extended from surface 197`
	`CUBIT> webcut volume 8 with sheet extended from surface 224`
	`CUBIT> webcut volume 11 10 12 9 with plane surface 28`
	`CUBIT> webcut volume 3 with plane normal to curve 116 fraction 0.5`

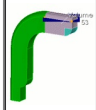
	CUBIT> webcut volume 3 17 with plane normal to curve 835 close_to vertex 487
	CUBIT> webcut volume 18 19 with sheet extended from surface 376
	CUBIT> webcut volume 3 17 with sheet extended from surface 378
	CUBIT> webcut volume 8 with sheet extended from surface 73 CUBIT> webcut volume 8 with sheet extended from surface 72 CUBIT> webcut volume 8 with sheet extended from surface 133 CUBIT> webcut volume 8 with sheet extended from surface 71
	CUBIT> webcut volume 8 with plane vertex 709 vertex 713 vertex 702
	CUBIT> unite volume 36 45 CUBIT> unite volume 37 43 CUBIT> unite volume 35 44 CUBIT> unite volume 39 42
	CUBIT> webcut volume 29 with plane vertex 81 vertex 93 vertex 154
	CUBIT> unite volume 33 36 50 11 CUBIT> unite volume 10 49 37 31 CUBIT> unite volume 12 52 35 34 CUBIT> unite volume 9 51 39 32 CUBIT> unite volume 9 22 CUBIT> unite volume 12 23 CUBIT> unite volume 20 33 CUBIT> unite volume 21 10



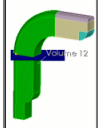
```
CUBIT> webcut volume 12 with plane vertex 86 vertex 71 vertex 76
CUBIT> webcut volume 53 with plane vertex 738 vertex 87 vertex 741
```



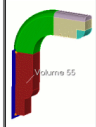
```
CUBIT> webcut volume 12 with plane vertex 72 vertex 85 vertex 74
CUBIT> webcut volume 55 with plane vertex 754 vertex 205 vertex 208
CUBIT> webcut volume 12 sweep surface 731 along curve 1073 through_all
CUBIT> unite volume 53 57 56
CUBIT> unite volume 54 12 55
```



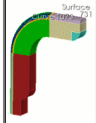
```
CUBIT> webcut volume 9 with plane vertex 99 vertex 101 vertex 103
CUBIT> webcut volume 58 with plane vertex 769 vertex 98 vertex 772
CUBIT> webcut volume 9 with plane vertex 106 vertex 104 vertex 100
CUBIT> webcut volume 60 with plane vertex 781 vertex 201 vertex 198
```



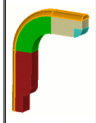
```
CUBIT> webcut volume 9 sweep surface 764 along curve 1078 through_all
CUBIT> unite volume 58 62 60
CUBIT> unite volume 59 9 61
```



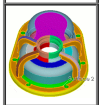
```
CUBIT> webcut volume 20 with plane vertex 140 vertex 138 vertex 135
CUBIT> webcut volume 63 with plane vertex 139 vertex 137 vertex 134
CUBIT> webcut volume 20 with plane vertex 141 vertex 800 vertex 796
CUBIT> webcut volume 64 with plane vertex 803 vertex 220 vertex 223
CUBIT> webcut volume 63 sweep surface 803 along curve 1238 through_all
CUBIT> unite volume 20 67 66
CUBIT> unite volume 65 63 64
```



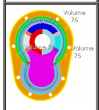
```
CUBIT> webcut volume 21 with plane vertex 165 vertex 163 vertex 160
CUBIT> webcut volume 68 with plane vertex 164 vertex 162 vertex 159
CUBIT> webcut volume 21 with plane vertex 825 vertex 169 vertex 822
CUBIT> webcut volume 69 with plane vertex 830 vertex 216 vertex 213
CUBIT> webcut volume 68 sweep surface 836 along curve 1069 through_all
CUBIT> unite volume 21 72 69
CUBIT> unite volume 70 68 71
```



These are the steps to webcut each of the stiffeners into the configuration shown. It is repeated for each of the stiffeners. This is also the step which creates the sliver curves which must be composited out later.



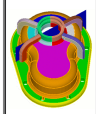
```
CUBIT> webcut volume 70 65 59 54 with plane surface 2
```



```
CUBIT> unite volume 1 76 75 73 74
```



```
CUBIT> unite volume 28 47 46 41 48 38 8 30 29 40
```



```
CUBIT> webcut volume 28 with plane surface 870

CUBIT> webcut volume 28 77 with plane surface 871

CUBIT> webcut volume 28 77 with plane surface 878

CUBIT> webcut volume 28 77 with plane surface 879
```



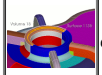
```
CUBIT> webcut volume 1 81 2 82 with plane normal to curve 1849 fraction 0.5
```



```
CUBIT>webcut volume 19 18 with plane normal to curve 843 fraction 0.75
```



```
CUBIT> create curve vertex 1122 vertex 471 on surface 1134
```



```
CUBIT> webcut volume 19 sweep curve 2073 along curve 2042 through_all

CUBIT> webcut volume 18 with sheet extended from surface 1146

CUBIT> webcut volume 18 with sheet extended from surface 1135
```



```
CUBIT> unite volume 91 92

CUBIT> delete curve 2073
```



```
CUBIT> unite volume 89 18

CUBIT> unite volume 88 19

CUBIT> imprint all

CUBIT> merge all
```



Composite small curves formed from webcuts

```
CUBIT> composite create curve 1456 1468
CUBIT> composite create curve 1459 1467
CUBIT> composite create curve 1499 1511
CUBIT> composite create curve 1502 1510
CUBIT> composite create curve 1371 1379
CUBIT> composite create curve 1370 1381
CUBIT> composite create curve 1423 1413
CUBIT> composite create curve 1422 1414
CUBIT> volume all scheme auto
```



Create the partitioned curves shown using existing vertices

```
CUBIT> partition create surface 1067 vertex 311 175
CUBIT> partition create surface 1067 vertex 174 312
CUBIT> partition create surface 1063 vertex 123 294
CUBIT> partition create surface 1251 vertex 170 226
CUBIT> partition create surface 1082 vertex 195 115
CUBIT> partition create surface 1082 vertex 242 116
CUBIT> partition create surface 1077 vertex 117 309
CUBIT> partition create surface 1255 vertex 118 310
```

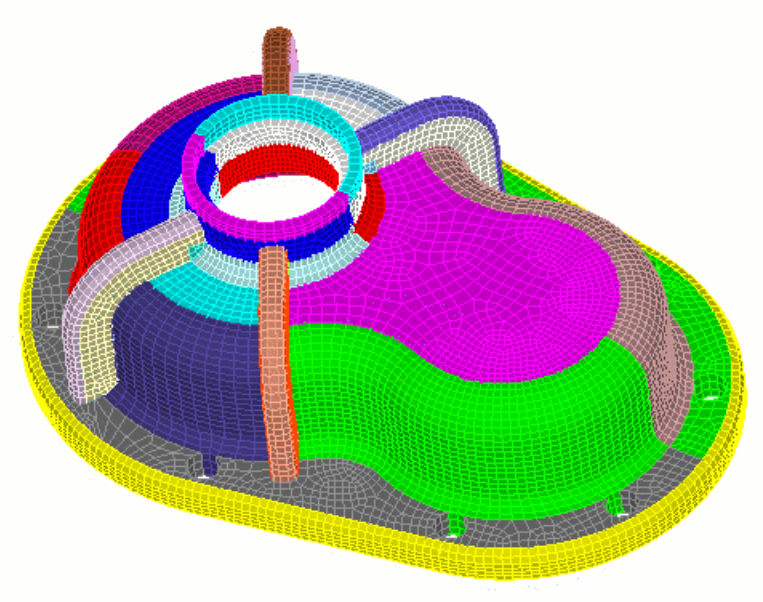


Meshing order is significant in this case. Since meshing a volume will hard set the interval counts on curves and surfaces, you will need to make sure that all of the interval counts are the same on adjacent volumes. Usually the meshing algorithm can handle this interval matching, but sometimes it helps to mesh volumes in a certain order. In this case, the meshing order also significantly changes the quality in the resulting mesh.

```
CUBIT> reset volume all
CUBIT> volume all scheme auto
CUBIT> volume 81 scheme sweep source surface 979 target surface 1061 rotate off
CUBIT> volume 81 sweep smooth auto
CUBIT> volume 85 scheme sweep source surface 1061 target surface 889 rotate off
CUBIT> volume 85 sweep smooth auto
CUBIT> volume all size 0.1
CUBIT> curve 2125 2122 interval 12
CUBIT> mesh vol 5 6 7 13 14 15 16 (COLORED GREEN)
CUBIT> mesh Volume 85 81 77 83 78 82 87 28 80 79 (COLORED RED)
CUBIT> mesh vol 88 89 91 90 17 3 (COLORED YELLOW)
CUBIT> mesh volume with not is_meshed (COLORED WHITE)
```

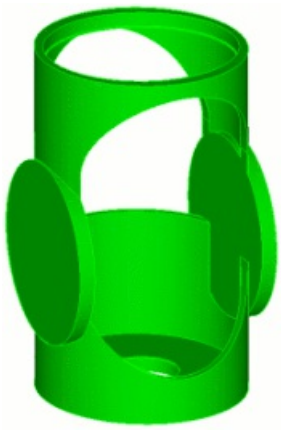
Final mesh

The final mesh is shown below.

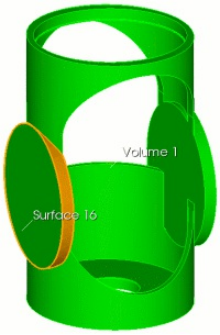
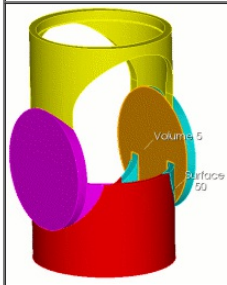
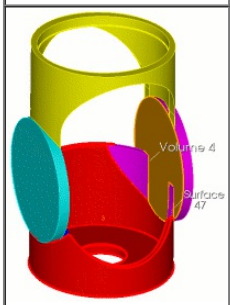


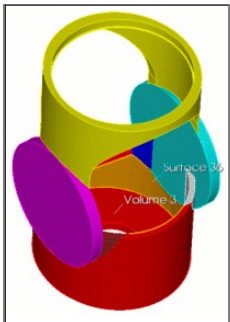
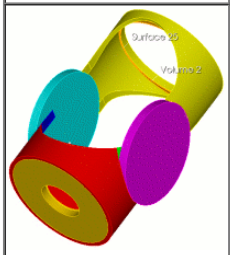
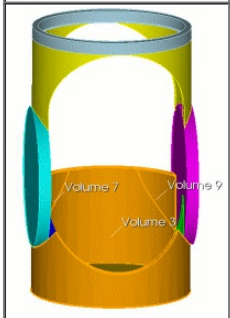
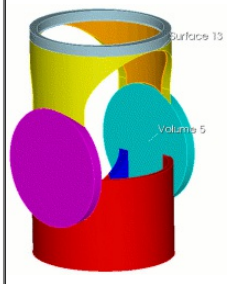
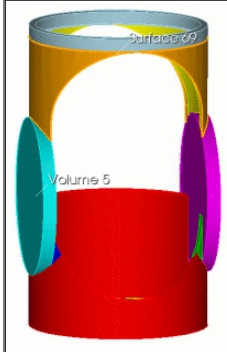
Example 8. Sweeping volumes with narrow angles and surfaces

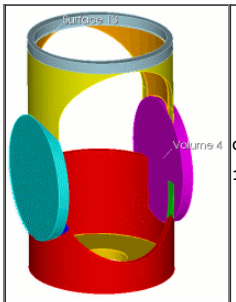
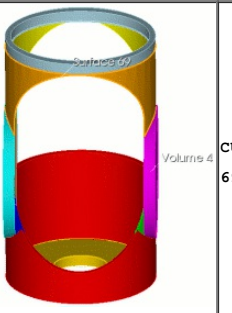
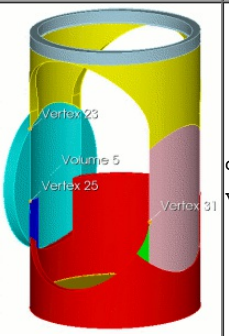
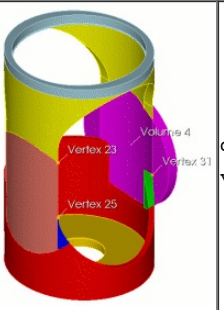
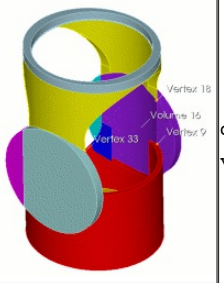
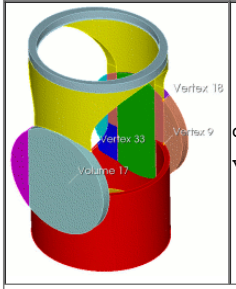
Narrow angles are a challenge for sweeping algorithms. In the next example, a well-placed webcut shaves off the end of the small angle to create an additional surface for the sweeping algorithm.

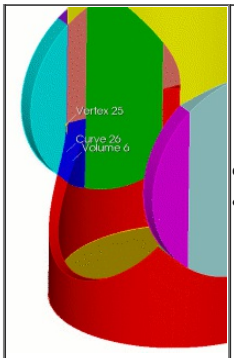
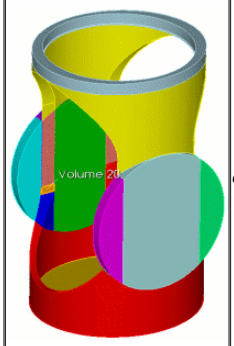
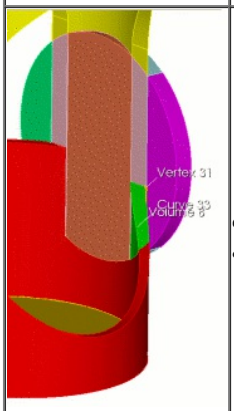
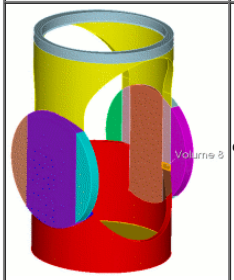
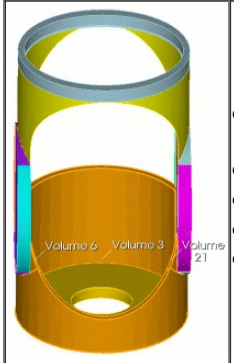


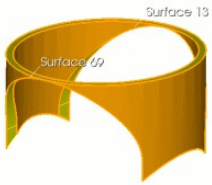
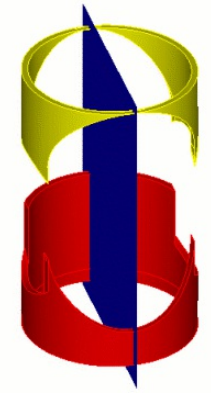
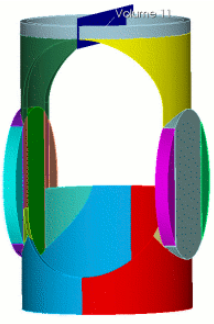
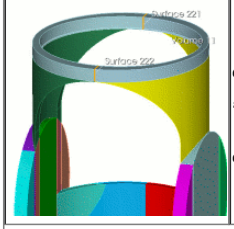
Suggested webcuts

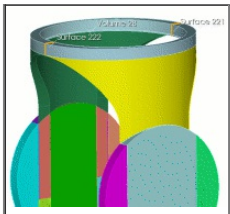
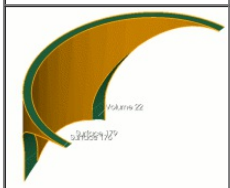
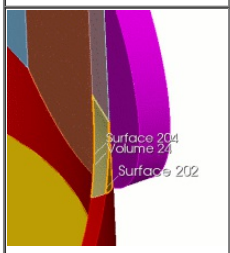
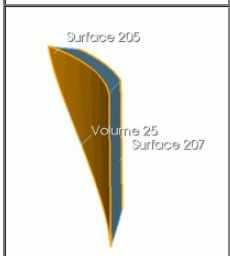
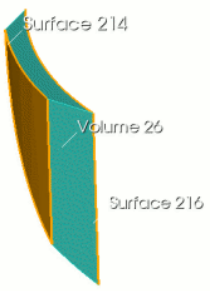
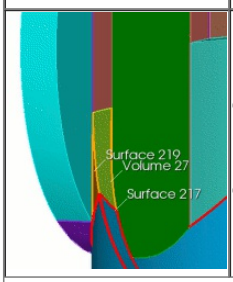
Webcut	Command
	`CUBIT> webcut volume 1 with sheet extended from surface 16`
	`CUBIT> webcut volume 5 with plane surface 50`
	`CUBIT> webcut volume 4 with plane surface 47`

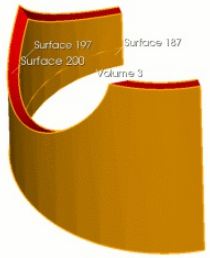
	**CUBIT> webcut volume 3 with sheet extended from surface 36**
	**CUBIT> webcut volume 2 with plane surface 25**
	**CUBIT> unite volume 3 9 7**
	**CUBIT> webcut volume 5 with sheet extended from surface 13**
	**CUBIT> webcut volume 5 with sheet extended from surface 69**

	**CUBIT> webcut volume 4 with sheet extended from surface 13**
	**CUBIT> webcut volume 4 with sheet extended from surface 69**
	**CUBIT> webcut volume 5 with plane vertex 23 vertex 25 vertex 31**
	**CUBIT> webcut volume 4 with plane vertex 23 vertex 25 vertex 31**
	**CUBIT> webcut volume 16 with plane vertex 18 vertex 9 vertex 33**
	**CUBIT> webcut volume 17 with plane vertex 18 vertex 9 vertex 33**

	CUBIT> webcut volume 6 with plane normal to curve 26 distance 0.6 from vertex 25
	CUBIT> delete volume 20
	CUBIT> webcut volume 8 with plane normal to curve 33 distance 0.6 from vertex 31
	CUBIT> delete volume 8
	CUBIT> unite volume 3 21 6 CUBIT> imprint volume all CUBIT> merge volume all CUBIT> volume all size 0.3 CUBIT> volume all scheme auto

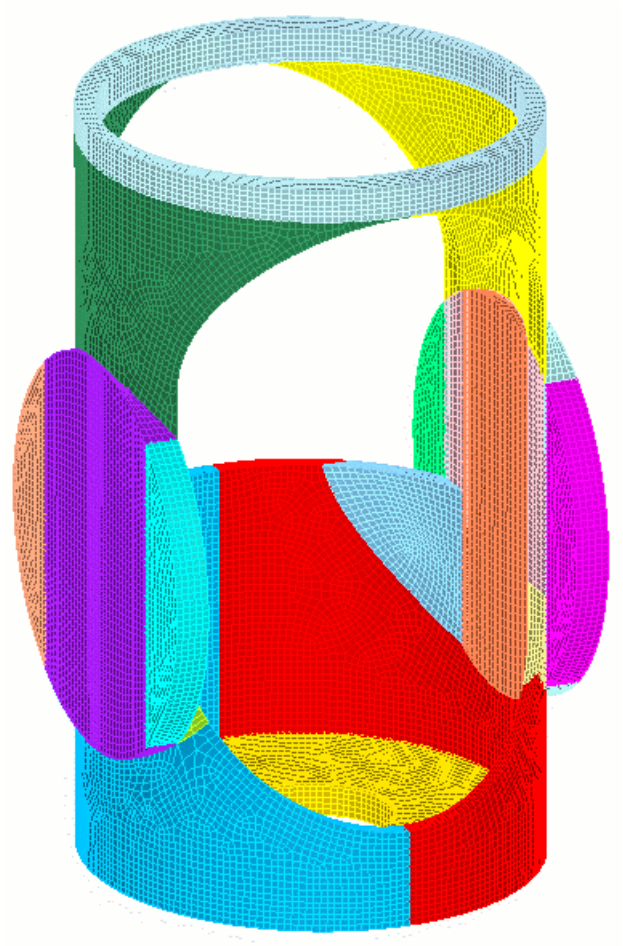
	**CUBIT> volume 2 scheme sweep source 13 target 69** **CUBIT> volume 2 sweep smooth auto** **CUBIT> unmerge volume all**
	**CUBIT> webcut volume 2 3 with plane zplane**
	**CUBIT> webcut volume 3 with sheet extended from surface 154**
	**CUBIT> webcut volume 23 with sheet extended from surface 153**
	**CUBIT> webcut volume 11 with plane zplane noimprint nomerge** **CUBIT> imprint volume all** **CUBIT> merge volume all**
	**CUBIT> volume 11 scheme sweep source surface 221 target surface 222 rotate off** **CUBIT> volume 11 sweep smooth auto**

	**CUBIT> volume 28 scheme sweep source surface 222 target surface 221 rotate off** **CUBIT> volume 28 sweep smooth auto**
	**CUBIT> volume 22 scheme sweep source surface 176 target surface 179 rotate off** **CUBIT> volume 22 sweep smooth auto**
	**CUBIT> volume 2 scheme sweep source surface 173 target surface 170 rotate off** **CUBIT> volume 2 sweep smooth auto**
	**CUBIT> volume 24 scheme sweep source surface 204 target surface 202 rotate off** **CUBIT> volume 24 sweep smooth auto**
	**CUBIT> volume 25 scheme sweep source surface 205 target surface 207 rotate off** **CUBIT> volume 25 sweep smooth auto**
	**CUBIT> volume 26 scheme sweep source surface 214 target surface 216 rotate off** **CUBIT> volume 26 sweep smooth auto**
	**CUBIT> volume 27 scheme sweep source surface 217 target surface 219 rotate off** **CUBIT> volume 27 sweep smooth auto**

	CUBIT> volume 3 scheme sweep source surface 197 187 target surface 200 rotate off CUBIT> volume 3 sweep smooth auto
	CUBIT> volume 23 scheme sweep source surface 212 193 target surface 210 rotate off CUBIT> volume 23 sweep smooth auto CUBIT> volume all scheme auto CUBIT> volume all size 0.2 CUBIT> mesh volume all

Final mesh

The final mesh is shown below.



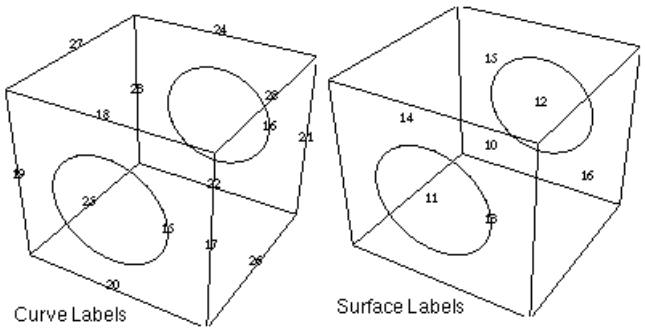
GUI Basic Tutorial

Overview

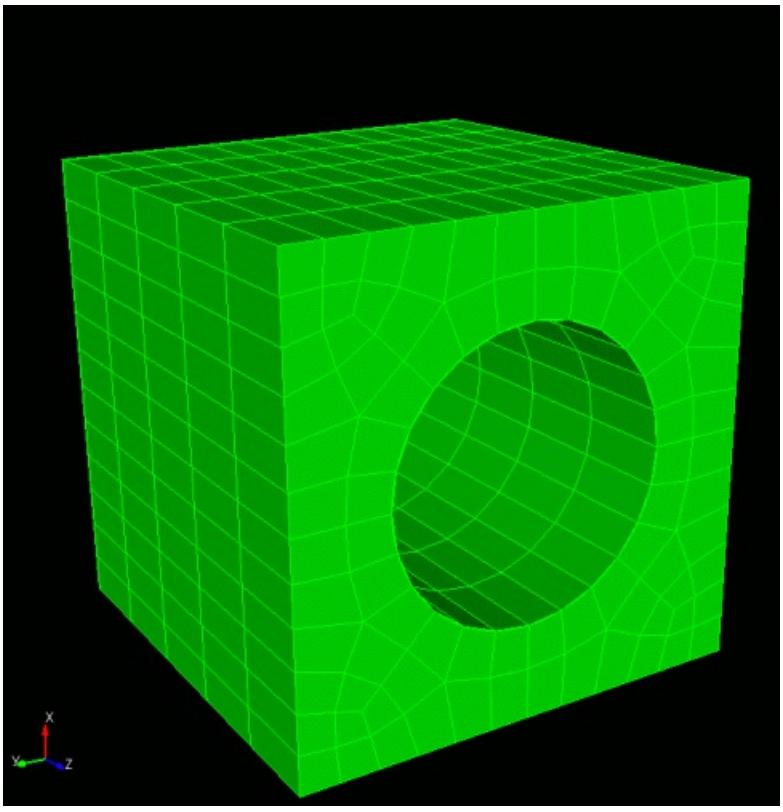
This tutorial demonstrates the use of CUBIT to create and mesh a brick with a through-hole. The primary steps in performing this task are:

- Creating the geometry
- Setting the interval sizes and meshing schemes
- Meshing the geometry
- Specifying the boundary conditions
- Exporting the mesh

The geometry for this tutorial is a brick with a cylindrical hole in the center, shown in the figure below. This figure also shows the curve and surface identification (ID) numbers, which are referenced in the command lines options shown with each step. The final meshed body is shown in the next figure.



Geometry for Brick with Cylindrical Hole



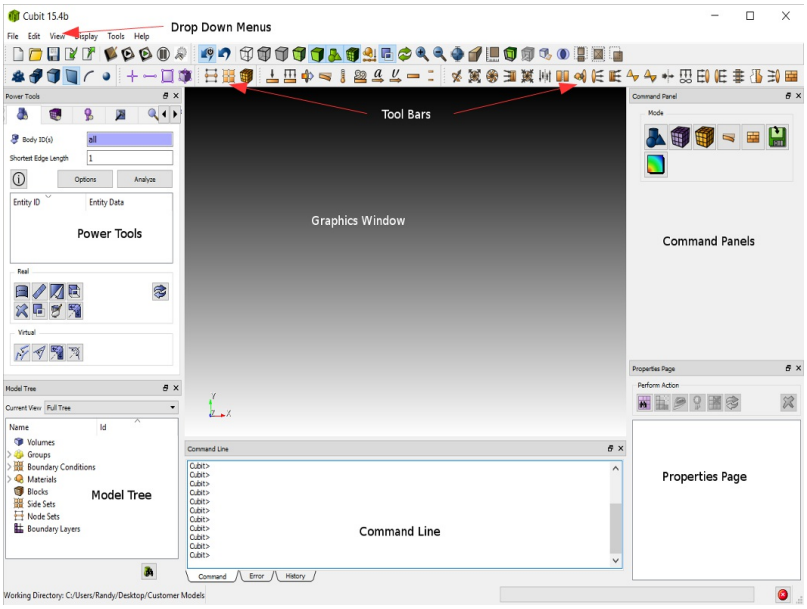
Generated Mesh for Brick with Cylindrical Hole



GUI Basic Tutorial Step 1

Step 1: Beginning Execution

Type "**cubit**" from a UNIX prompt or select cubit from the start menu if you are running on a PC with Windows. The CUBIT Application Window will appear as illustrated below:



CUBIT Application Window

The use of each window in the CUBIT program is described briefly below

Graphics Window	The current model will be displayed here. Zooming, panning, and rotating are also performed in this window.
Drop Down Menus	Functions such as file management, edit controls, display options, user preferences, journal file management, window manipulation, and help are available in the pull-down menus.
Toolbars	This is a large selection of selectable icons that duplicate the functions found in the pull-down menus. Additionally, picking types, and mouse selection controls are found here.
Power Tools	The Power Tools contains the ITEM workflow, geometry repair power tools, meshing power tools, mesh/geometry comparison tool, defeaturing tool, assembly tool, and mesh quality power tools.
Command Line Workspace	The command line workspace contains both the cubit command, error, history, and script windows. The command window is used to enter cubit commands and view the output. The error window is used to view cubit errors. The history window is used to view recent commands. The optional script window is used for Python programming.
Command Panels	Most Cubit commands are available in the command panels. The panel is organized topologically, by mode.
Properties Page	This is a list of properties of the selected geometry, mesh, boundary condition, or assembly. Some of the properties can also be edited from this window.
Model Tree	The model tree shows all geometric entities and their relationships, boundary conditions, boundary layers, and groups.

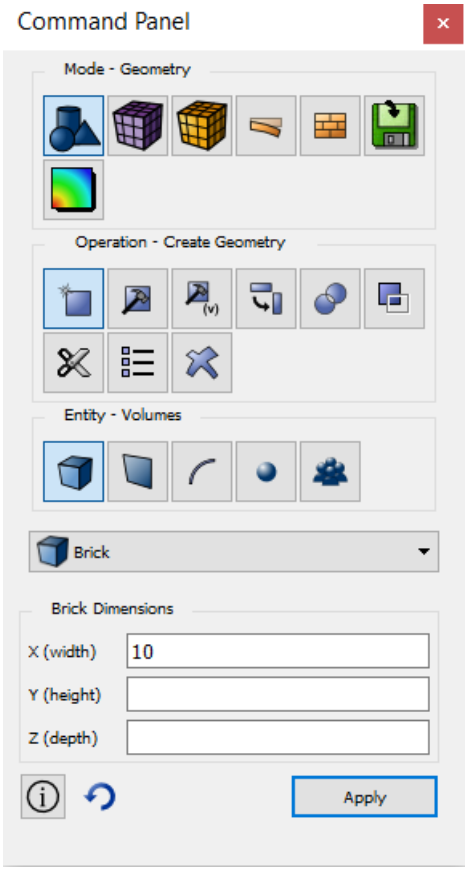


GUI Basic Tutorial Step 2

Step 2: Creating the Brick

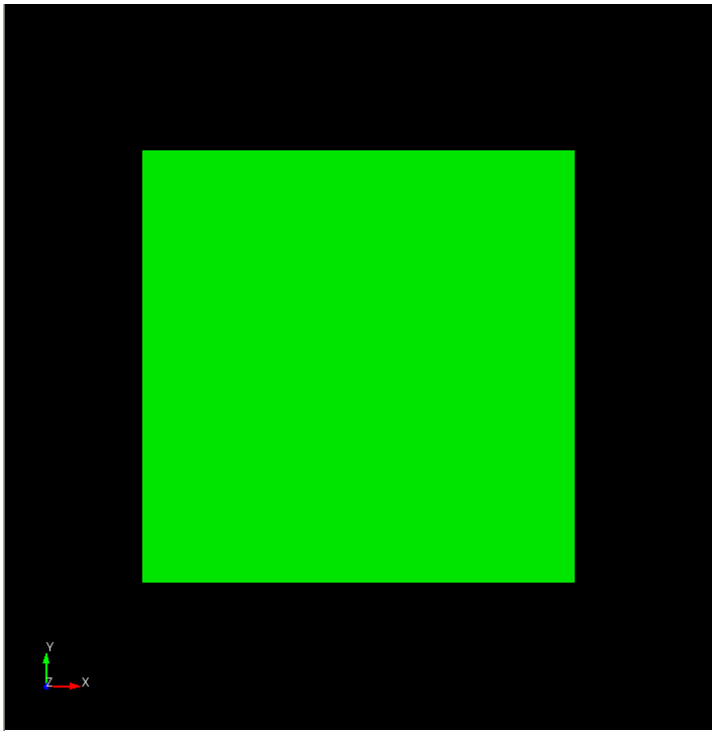
Now you may begin generating the geometry to be meshed. You will create a brick of width 10, depth 10 and height 10. The width and depth correspond to the x and y dimensions of the object being created. The "width" or x-dimension is screen-horizontal and the "depth" or y-dimension is screen-vertical. The height or z-dimension is out of the screen.

- On the Command Panel, select **Geometry**, then **Create Geometry**, then **Volume**. Brick is the default type.



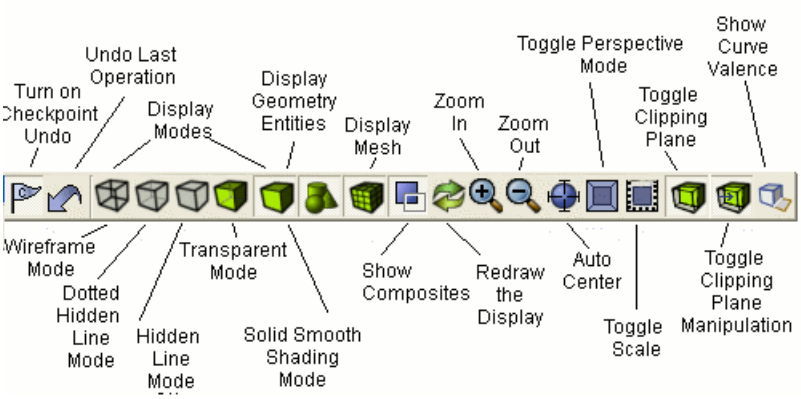
- Enter values for X, Y, and Z. Note, X (width) has a default value of 10. Select **Apply** to create the brick.

The brick should appear in your Graphics window as shown below.



Display of Brick

If you would like to change the rendering mode of your model, you may click on one of the view buttons in the Display Tools tool bar.

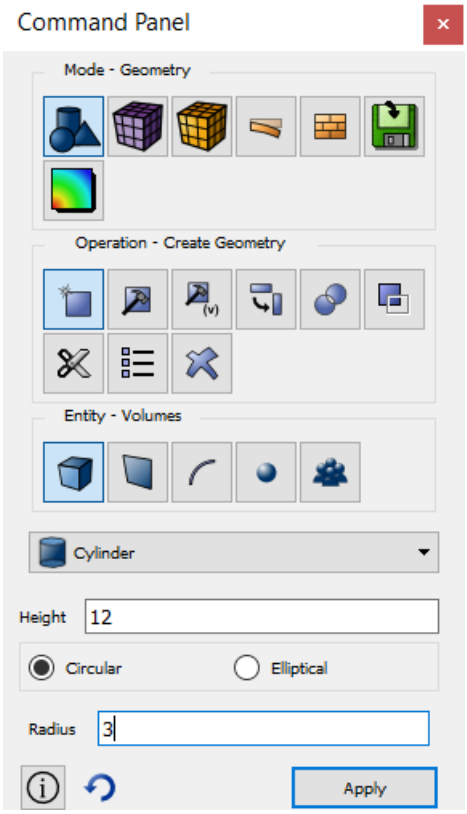


GUI Basic Tutorial Step 3

Step 3: Creating the Cylinder

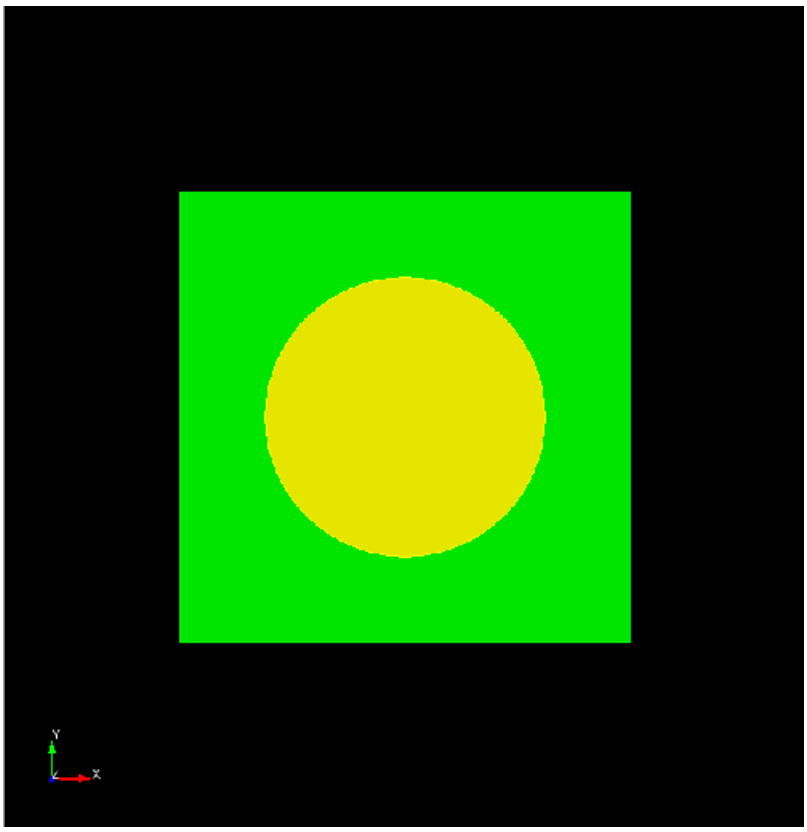
Now you must form the cylinder which will be used to cut a hole in the brick.

- Select **Cylinder** from the **Volume-type** combo box.



- Enter **12** for the height and **3** for the radius. Then select **Apply**.

The brick and the cylinder should appear in your display window as shown below:



Brick and Cylinder

GUI Basic Tutorial Step 4

Step 4: Adjusting the Graphics Display

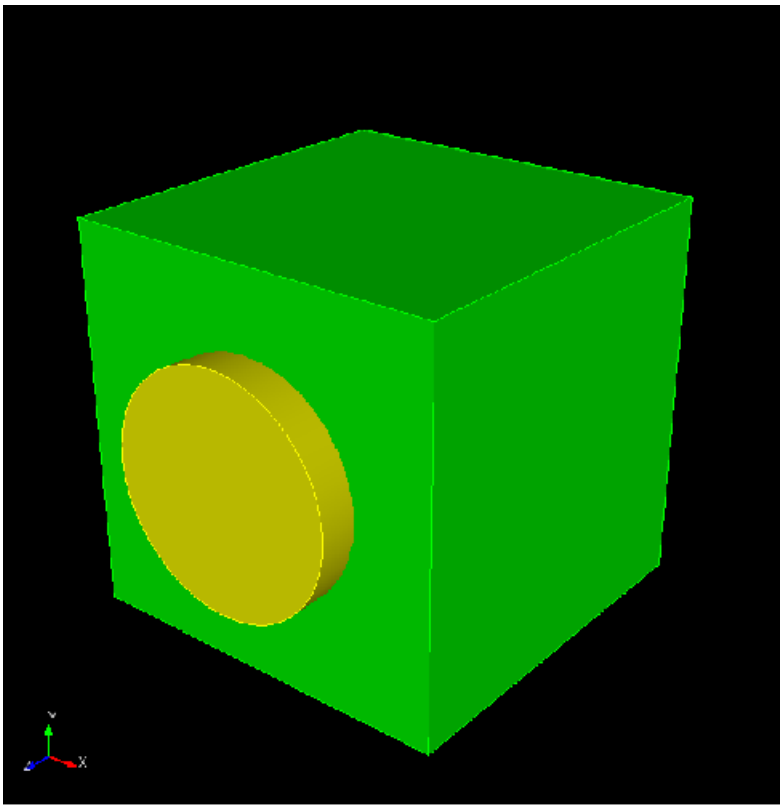
The geometry is drawn in the graphics window in perspective mode, by default from a viewing direction of the +z axis. This view can now be adjusted to verify the proper orientation of the geometry just created.

The following button clicks apply for 3-button mice (these are the default GUI settings):

- **left** will pick when the mouse is over an entity. Left click will also pan when held down.
- **middle** will rotate
- **right** will show a context menu when an entity is selected. Right click will zoom when no entity is selected.

Mouse button behavior can be customized from the Tools-Options menu for use with non 3-button mice.

Use the mouse buttons to make the display look like the figure below.



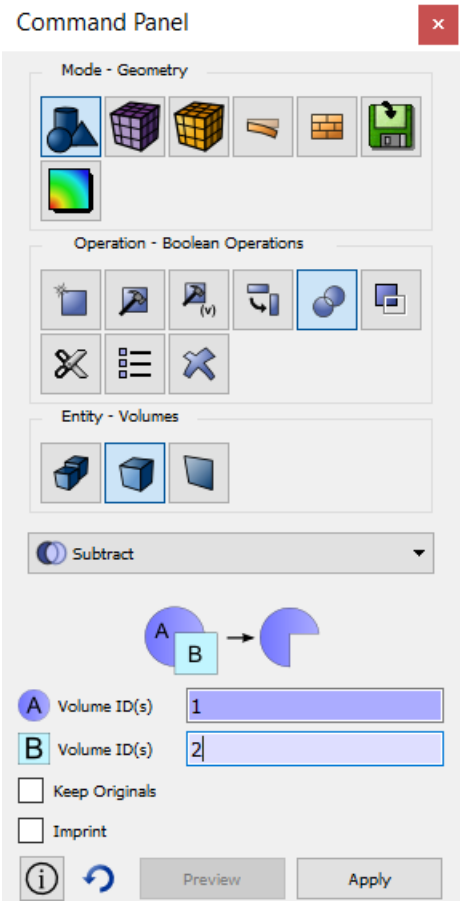
View from Different Perspective

GUI Basic Tutorial Step 5

Step 5: Forming the Hole

Now the cylinder can be subtracted from the brick to form the hole in the block.

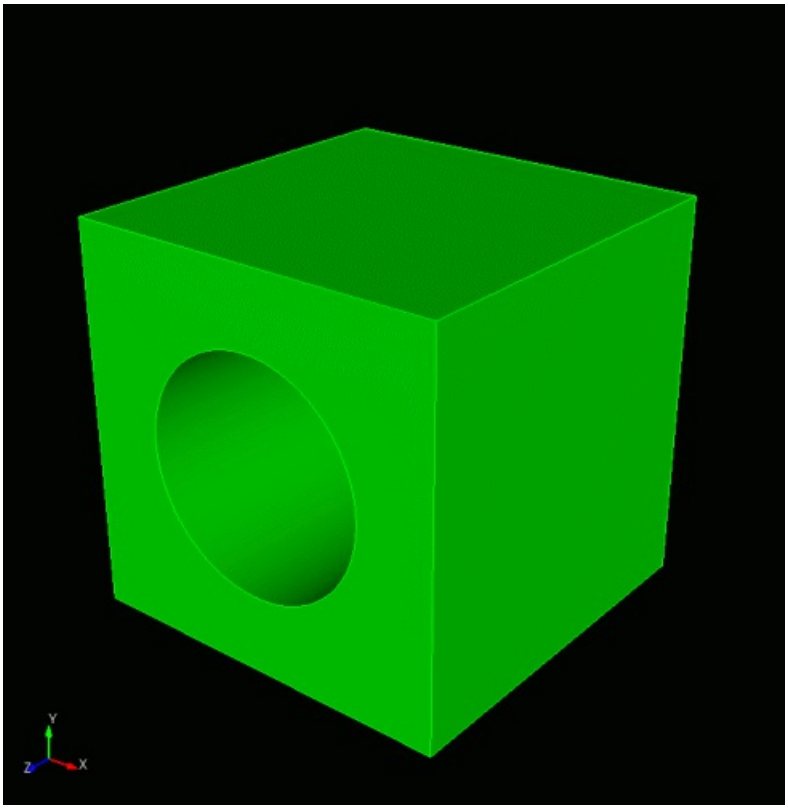
- Select the **Boolean** operation button. Select **Volume** as the entity type. Then select **Subtract** from the Boolean combo box.



- Enter **2** for **Subtract Volume ID(s)** and **1** for **From Volume ID(s)**.
- Select the **Apply** button

You can also select the brick or cylinder interactively. Place the cursor in the **Subtract Volume ID(s)** field and click. This field is known as a **Pick Widget**. Clicking in a pick widget automatically sets the graphics pick mode for the entity type expected by the pick widget. Move the cursor to the graphics window and, using the left mouse button, select an entity. The id of the selected entity will be echoed into the pick widget field. Holding the control key while selecting entities in the graphics window will select multiple entities.

Notice that both original volumes are deleted in the Boolean operation and replaced with a new volume, with an id of 1. The result of this operation is a single volume, a brick with a hole through it, as shown below.



Brick after Subtracting Cylinder

We have now completed creating the geometry, and are ready to generate a mesh.

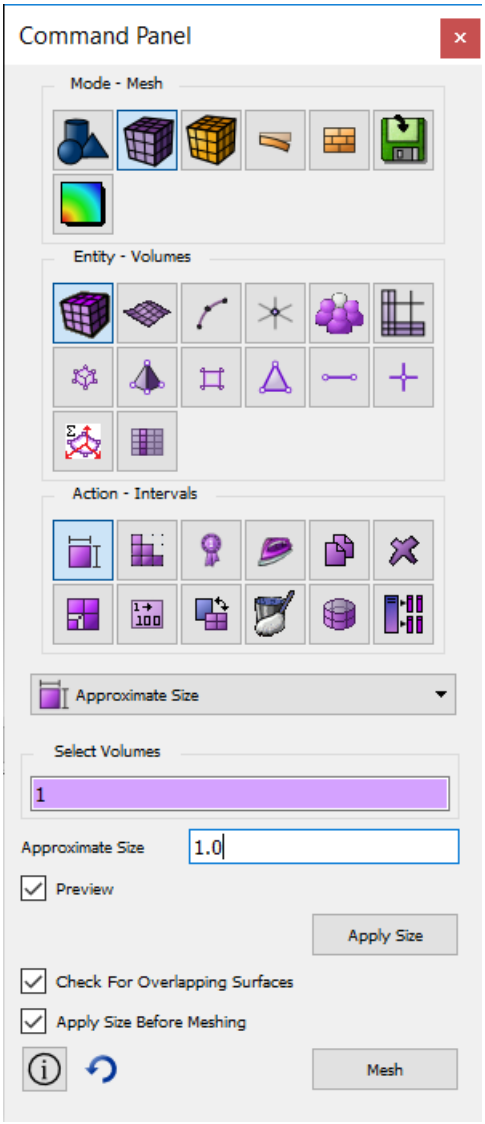
GUI Basic Tutorial Step 6

Step 6: Setting Interval Sizes

The volume shown in Step 5 will be meshed by sweeping a surface mesh from one side of the brick to the other. Before generating any mesh, the user must specify the size of the elements to be generated. In this example, one element size will be specified for the volume as a whole and a smaller size will be specified for around the hole. A direct interval setting will be specified for the sweep direction.

To set the interval size for the overall volume, do the following:

- Change the mode to **Meshing**, then select **Volume** followed by **Intervals**.



- Place the cursor into the **Select Volumes** field. Since this is a pick widget, click anywhere on the volume in the graphics window. Alternatively, type 1 in the field. Set the **Interval Size** to **1.0** and select **Apply Size**

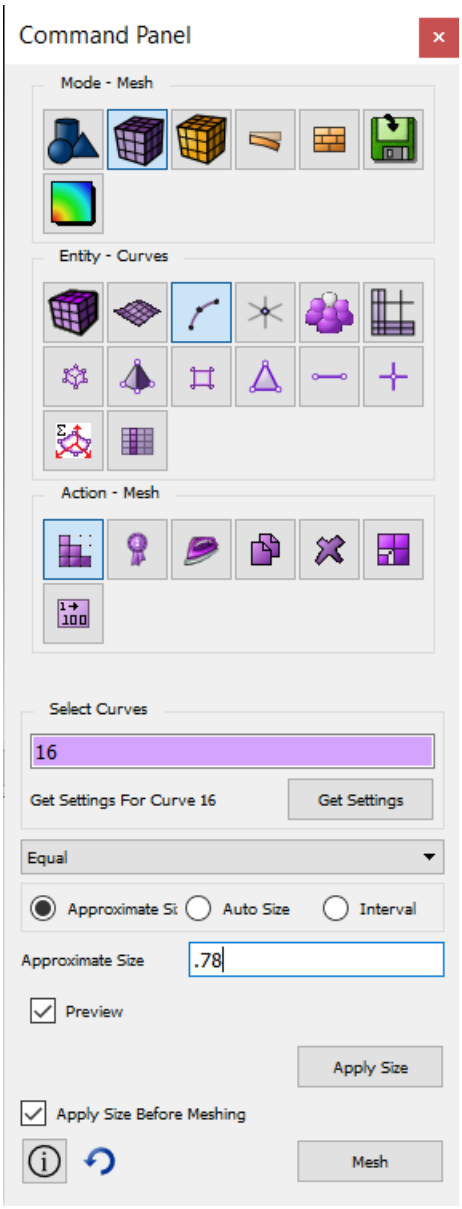
Since the brick is 10 units in length on a side, this specifies that each straight curve is to receive approximately 10 mesh elements.

In order to better resolve the hole in the middle of the top surface, we set

a smaller size for the curve bounding this hole.

- Change the object of the command panel to curve by selecting **Curve** from the **Entity** buttons and **Mesh** from the **Action** Buttons.

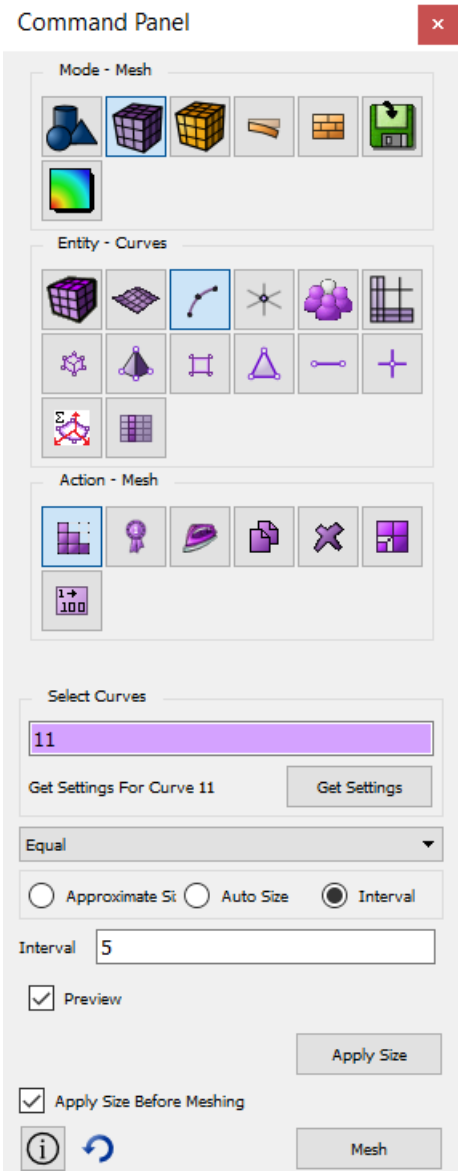
Note: There is not a separate interval action panel for curves. The interval and mesh actions for curves are grouped together in one panel.



- Place the cursor into the **Select Curves** pick widget field. Select the near end of the cylinder in the graphics window. Once you have selected the curve, the id of that curve, **16** should appear in the Selected Curves field. Select **Size**
- Enter **0.78** for the size and select **Apply Size**.

Finally, we would like to generate exactly 5 element layers in the sweep direction. This is accomplished by setting the intervals on one of the curves in the sweep direction.

- Place the cursor back into the **Selected Curves** field and enter **11**.
- Select the **Interval** radio button
- Enter an interval count of **5** and select **Apply**.

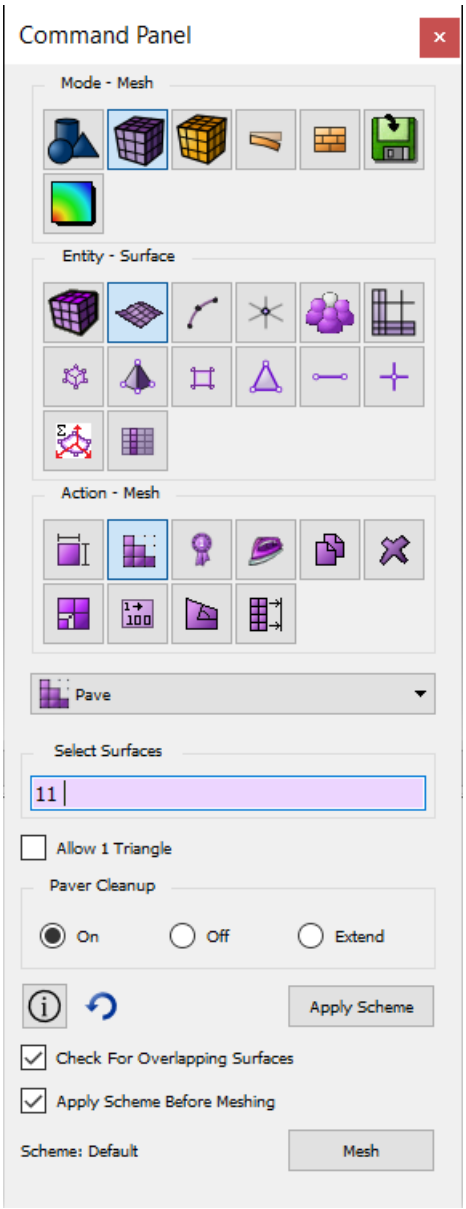

Command Panel
Mode - Mesh
Entity - Curves
Action - Mesh
Select Curves
11
Get Settings For Curve 11
Get Settings
Equal
Approximate Si
Auto Size
Interval
Interval
5
Preview
Apply Size
Apply Size Before Meshing
Mesh

GUI Basic Tutorial Step 7

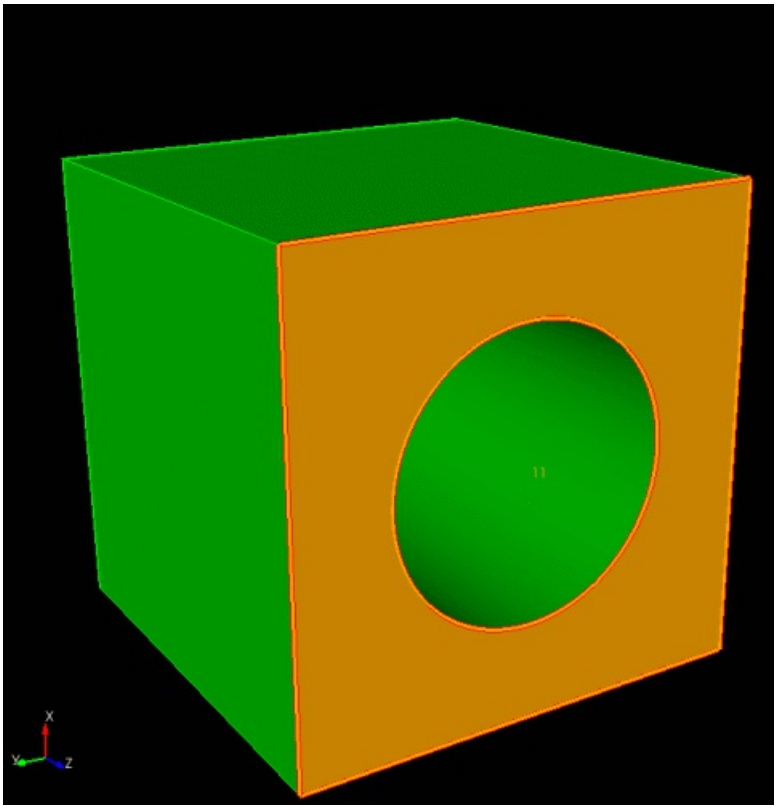
Step 7: Surface Meshing

Now all necessary intervals have been set, the meshing can proceed. Begin by meshing the front surface (with the hole) using the paving algorithm. This is done in two steps. First, set the scheme for that surface to Pave, then issue the command to Mesh.

- Select **Surface** then **Mesh** buttons in the Control Panel.
- Select **Pave** in the **Available Mesh Schemes** combo box.

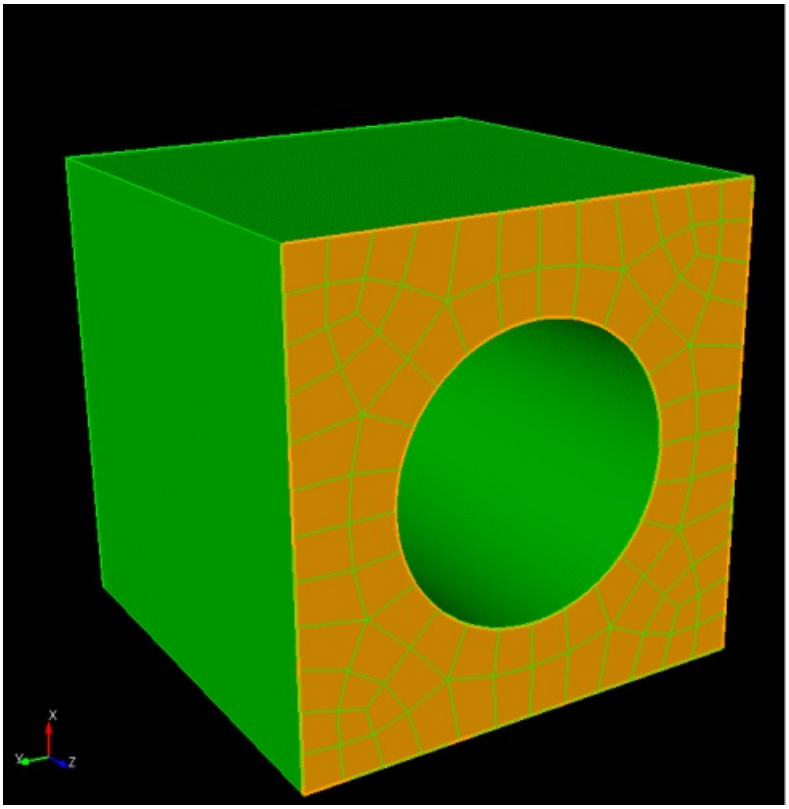


Place the cursor into the **Surface ID(s)** field. Select the front surface of the object by selecting anywhere within the region indicated. The id of **Surface 11** will be echoed in the field.



- Select the **Apply** button to set the scheme.
- Select the **Mesh** button to mesh the surface.

A mesh should be generated on surface 11 using the paving algorithm. The result is shown below.



Surface Meshed with Paving

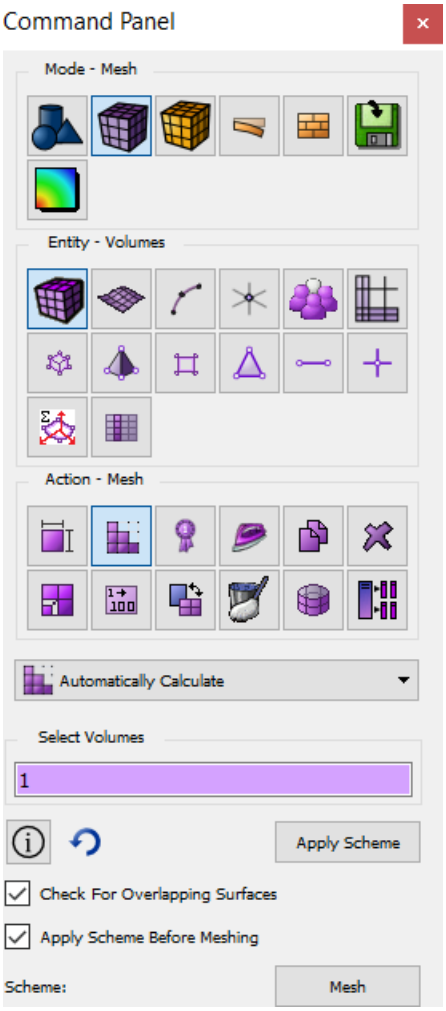
GUI Basic Tutorial Step 8

Step 8: Volume Meshing

The volume mesh can now be generated. Again, the first step is to specify the type of meshing scheme that should be used and the second step is to issue the order to mesh. In certain cases, the scheme can be determined by CUBIT automatically. For sweepable volumes, the automatic scheme detection algorithm also identifies the source and target surfaces of the sweep automatically.

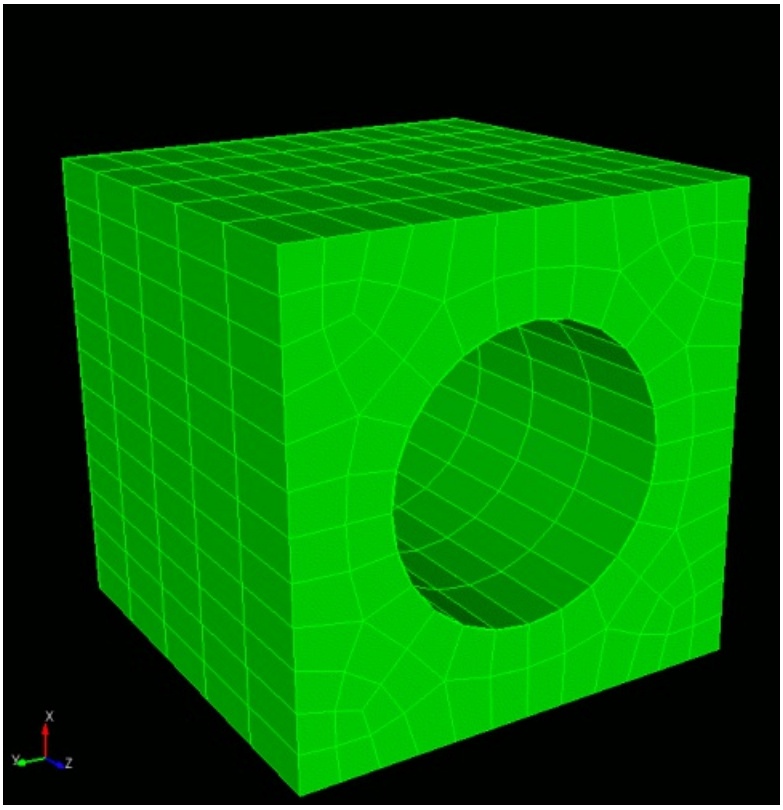
To instruct the code to automatically determine the meshing scheme, and in this case the source and target surfaces, do the following:

- Select **Volume** then **Mesh** on the control panel.



- Place the cursor into the **Volume ID(s)** field then select the volume in the Graphics Window. The id of **Volume 1** should appear in the field. Choose the **Automatically Calculate** scheme using the combo box provided.
- Select **Apply Scheme** to set the scheme. Then select **Mesh** to mesh the volume.

The final meshed body will appear in the Graphics Window, as shown below:

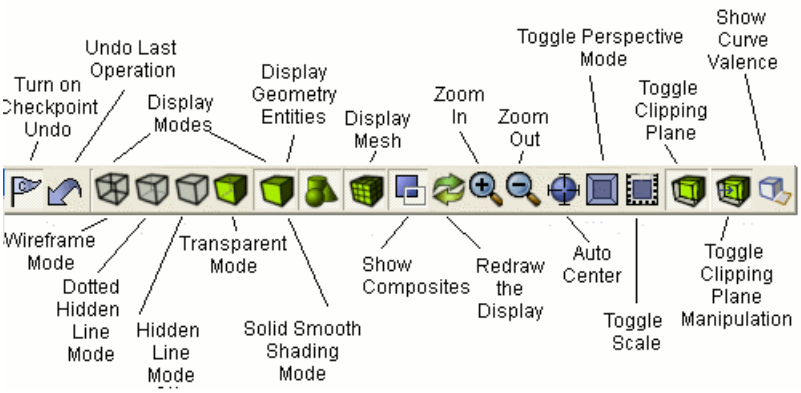


Smooth Shade View of Volume Mesh

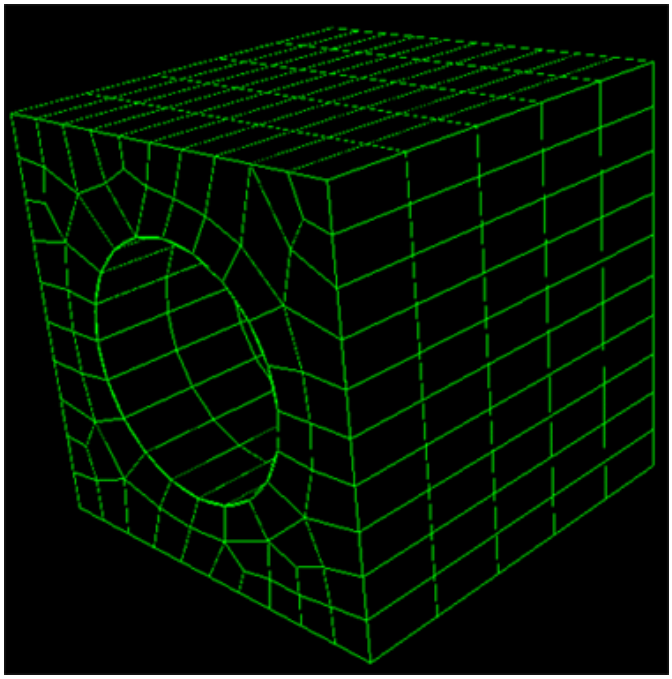
GUI Basic Tutorial Step 9

Step 9: Inspecting the Model

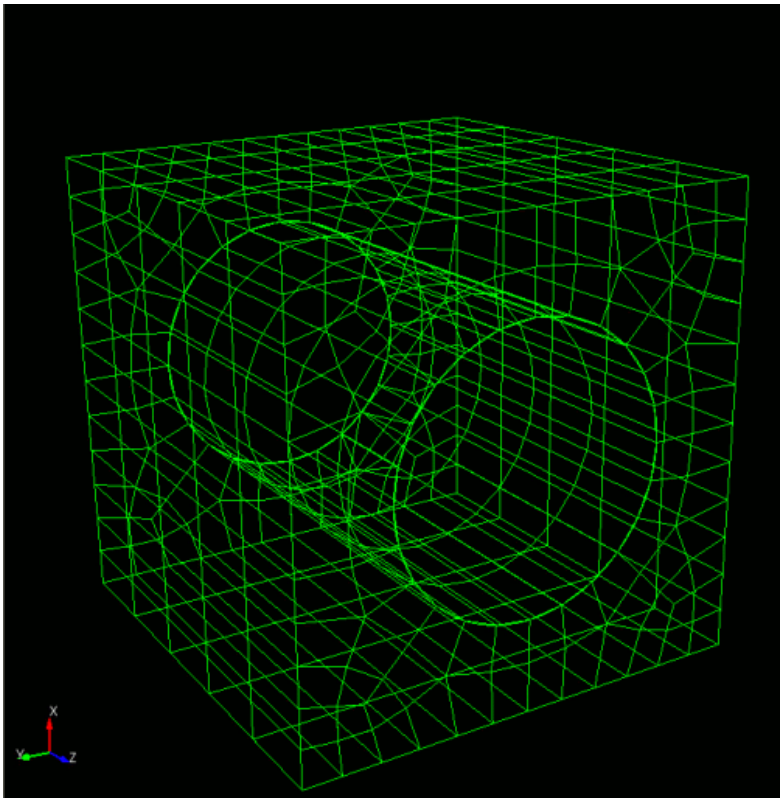
The type, quality, and speed of rendering the image can be controlled in CUBIT by selecting one of the buttons in the **Display** icon group. These icons appear by default in the icon bar above the graphics window. They can be used to change the display mode to wire frame, hidden line, true hidden line, transparent or smooth shade.



For example, the following two figures result from selecting the **Hidden Line** and **Wire Frame Mode** buttons respectively.



Hidden Line View of Mesh



Wire Frame View of Mesh

Although CUBIT automatically computes limited quality metrics after generating a mesh and warns the user about certain cases of bad quality, it is still a good idea to inspect a broader set of quality measures. To do this, use the **Command Window** to enter the command:

```
CUBIT> quality volume 1 Allmetrics
```

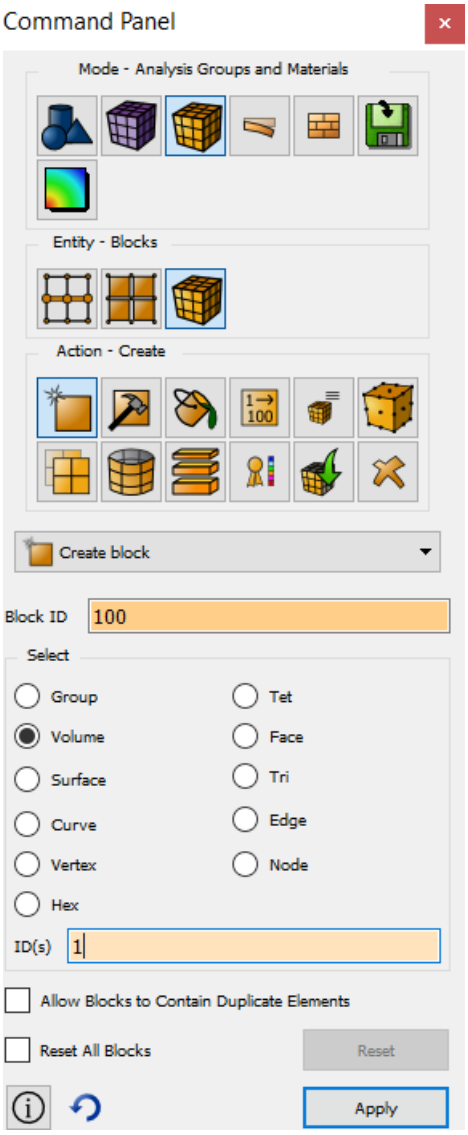
The results of the quality are displayed in the Command Window. For an explanation of each quality metric along with acceptable ranges, see Mesh Quality Assessment. For the purposes of this tutorial, you can assume the quality metrics shown are in an acceptable range.

GUI Basic Tutorial Step 10

Step 10: Defining Boundary Conditions

Let us assume that we need to define one material type for the entire mesh, and a single node-based boundary condition on all surfaces. This is accomplished by identifying an Element Block and a Nodeset, respectively; the id numbers assigned to these entities are assigned by the user, usually by some convention meaningful to the analysis to be done. The element block and nodeset are identified from the Materials and Properties button on the control panel.

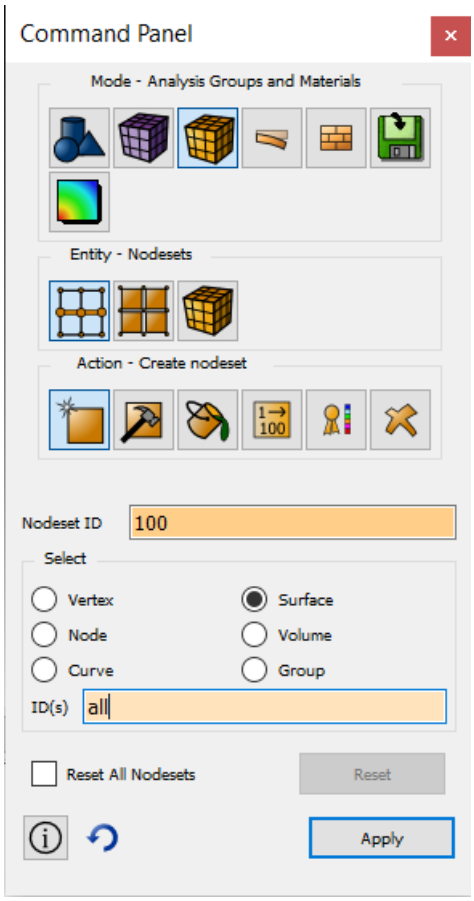
- Select the **Analysis Groups and Materials** button and then **Blocks** in the Control Panel window
- Select the **Create** button
- Select **Create block** in the pull down menu
- Enter 100 into the Block ID field
- Select the **Volume** radio button
- Enter the id of **Volume 1** by selecting it in the graphics window, or just manually entering in **ID(s)** field
- Press **Apply**



Create a nodeset by following the steps below

- Open the **Nodeset** window on the Control Panel

- Select the **Create** button
- Enter a Nodeset id of **100**
- Select the **Surface** radio button and type **all** in the ID(s) field
- Press **Apply**

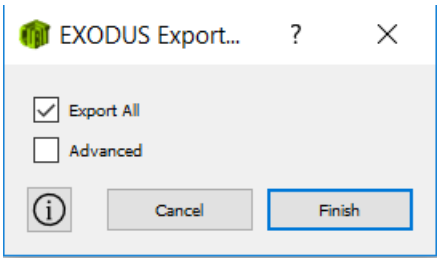


GUI Basic Tutorial Step 11

Step 11: Exporting the Mesh

Finally, the mesh needs to be written to an Exodus II file. This is easily done:

- From the **File** menu, select **Export**.
- Set the file export type to **Genesis Files** from file type combo box.
- Enter a file name in the dialog, such as **brick_with_hole.g**, and select **Save**. Since this is a standard file management dialog, the user may browse or use any other file management functionality supported by the platform.
- Select the **Export All** check box



- Select **Finish** to export the mesh.



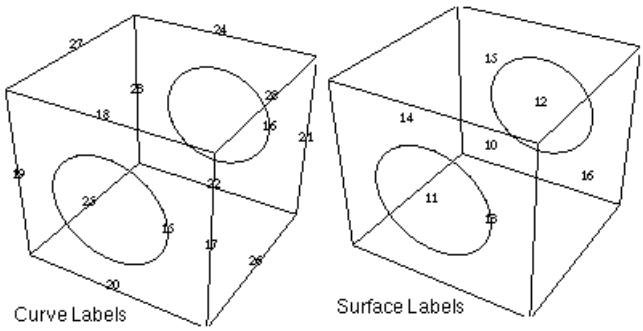
Command Line Basic Tutorial

Overview

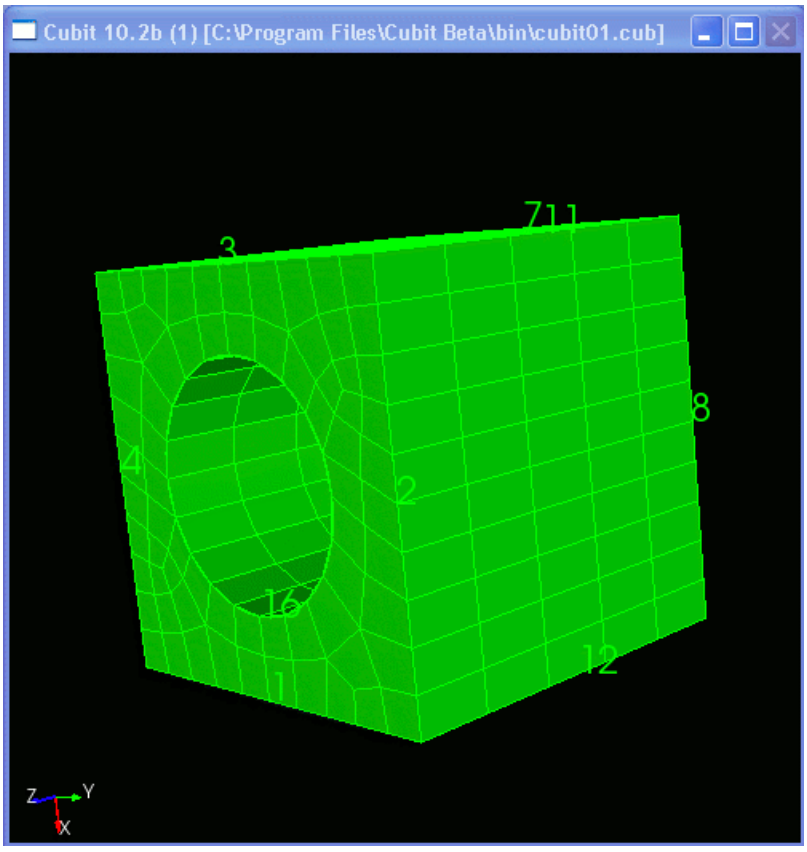
This tutorial demonstrates the use of CUBIT to create and mesh a brick with a through-hole. The primary steps in performing this task are:

- Creating the geometry
- Setting the interval sizes and meshing schemes
- Meshing the geometry
- Specifying the boundary conditions
- Exporting the mesh

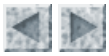
Each of these steps is described in detail in the following sections. The geometry in this tutorial is a brick with a cylindrical hole in the center, shown in the figure below. This figure also shows the curve and surface identification (ID) numbers, which are referenced in the command lines shown with each step. The final meshed body is shown in the next figure.



Geometry for Cube with Cylindrical Hole



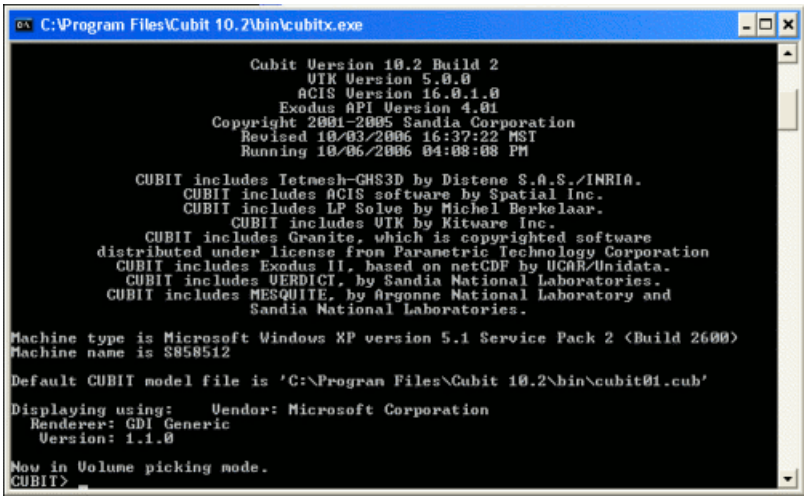
Generated Mesh for Cube with Cylindrical Hole



CL Basic Tutorial Step 1

Step 1: Beginning Execution

Type "cubit" from a UNIX prompt to begin execution of CUBIT. A CUBIT console window will appear which tells the user which CUBIT version is being run and the most recent revision date. An example of the UNIX output window is shown below. This window echoes the commands and relays information about the success or failure of attempted actions.



Some things to notice are:

- At the top of the CUBIT window you will be told where the commands entered in this CUBIT session will be journaled. For example: "Commands will be journaled to `cubit01.jou' for this example.
- In addition to the CUBIT version, the code also reports the versions of ACIS and VTK that have been compiled into CUBIT.
- The command line prompt appears after the banner screen, and appears as "**CUBIT>**".
- Commands are entered at that prompt, followed by the "Enter" key.
- Upon startup, a graphics window should also appear, with an axis triad in the lower left hand corner (this window will not appear if CUBIT is started with the -nographics option.)



CL Basic Tutorial Step 2

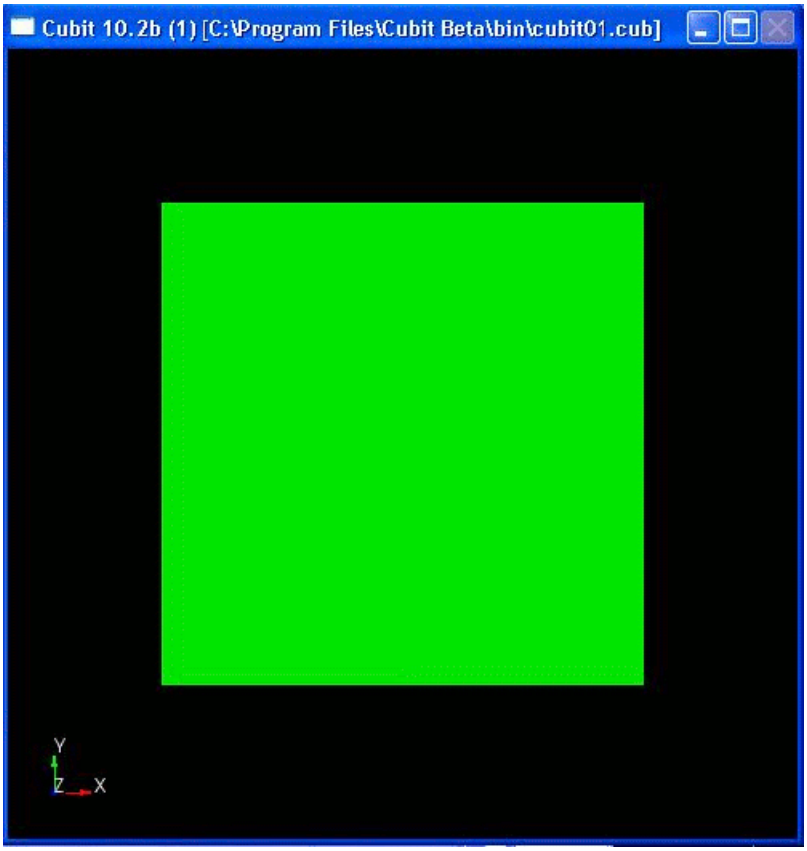
Step 2: Beginning Execution

Now you may begin generating the geometry to be meshed. You will create a brick of width 10, depth 10 and height 10. The width and depth correspond to the x and y dimensions of the object being created. The "width" or x-dimension is screen-horizontal and the "depth" or y-dimension is screen-vertical. The height or z-dimension is out of the screen. The command to create this object is:

```
cubit> create brick width 10 depth 10 height
10 (OR)

cubit> create brick x 10
```

The cube should appear in your display window as shown below:



Display of Brick



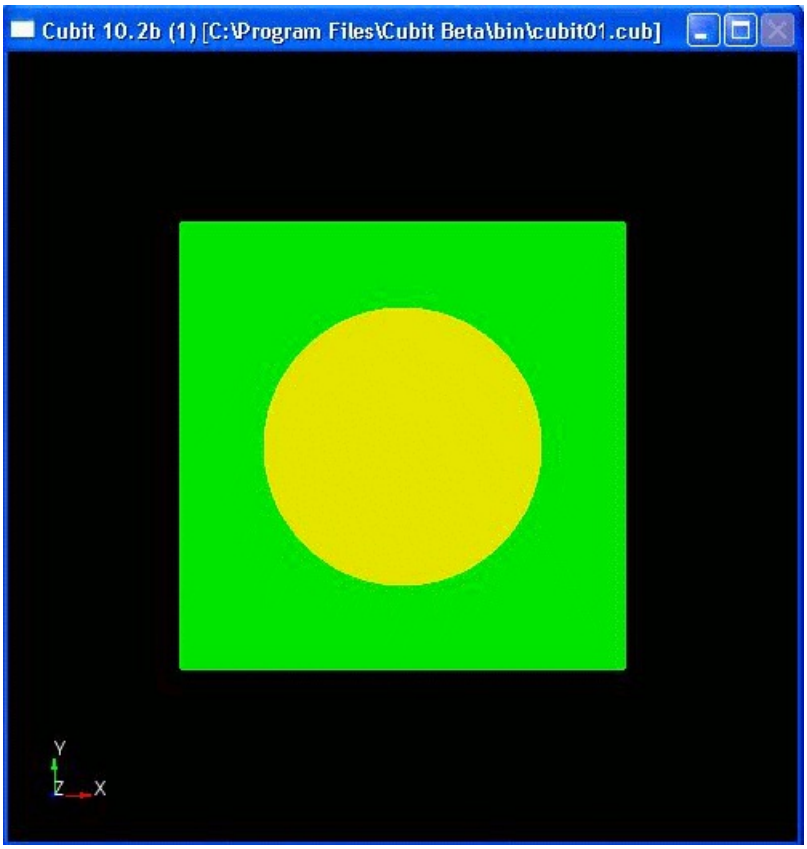
CL Basic Tutorial Step 3

Step 3: Creating the Cylinder

Now you must form the cylinder which will be used to cut the hole from the brick. This is accomplished with the command:

```
cubit> create cylinder height 12 radius 3
```

At this point you will see both a brick and a cylinder appear in the CUBIT display window, as shown below:



Brick and Cylinder



CL Basic Tutorial Step 4

Step 4: Adjusting the Graphics Display

The geometry is drawn in the graphics display in perspective mode by default from a viewing direction of the +z axis. This view can now be adjusted to verify the proper orientation of the geometry just created. The orientation of the geometry can be adjusted using the command line or interactively with the mouse.

Command Line

You can adjust the orientation of the object from the command line. For example, the from command can be used as follows

```
cubit>from 20 15 25

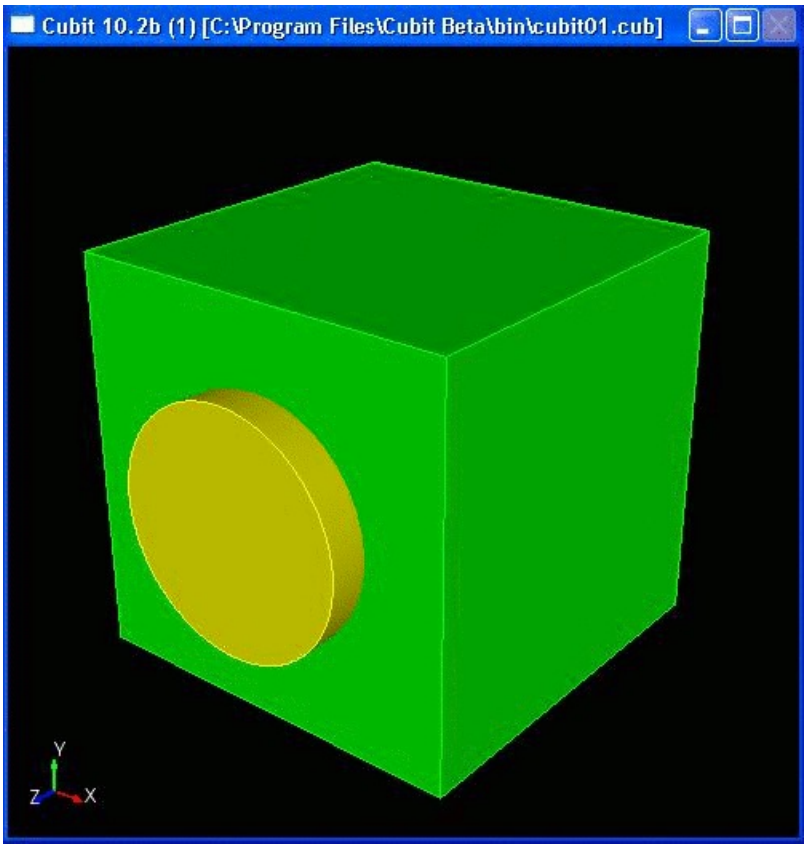
cubit>display
```

Mouse

To interactively change the orientation, activate your graphics window by placing your cursor in the window or by clicking at the top of it (this will vary depending upon your window settings in your operating system).

- Use the **left mouse button** to interactively rotate the view
- Use the **middle mouse button** to zoom in or out
- Use the **right mouse button** to pan the view.

Use the mouse buttons to make the display look the figure below:



View from a Different Perspective



CL Basic Tutorial Step 5

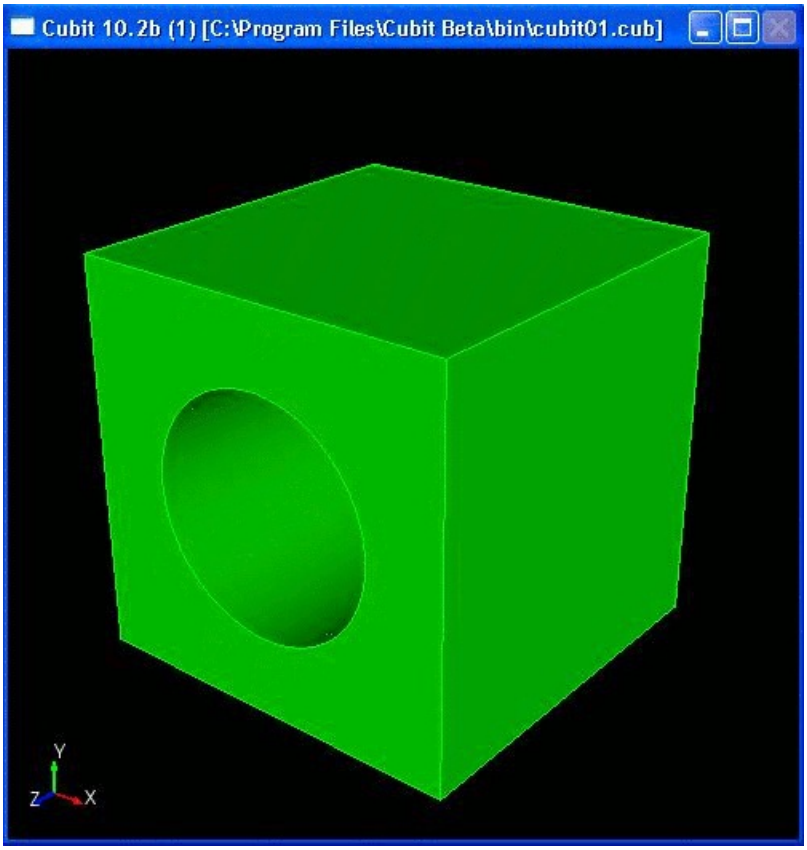
Step 5: Forming the Hole

Now, the cylinder can be subtracted from the brick to form the hole in the block. Issue the following command:

```
cubit> subtract 2 from 1
```

Note that both original volumes are deleted in the Boolean operation and replaced with a new volume (with an id of 1) which is the result of the Boolean operation Subtract .

The result of this operation is a single body, a brick with a hole through as shown below:



Brick after Subtracting the Cylinder

We have now completed creating the geometry, and are ready to generate a mesh.



CL Basic Tutorial Step 6

Step 6: Setting Interval Sizes

The volume shown in Step 5 will be meshed by sweeping a surface mesh from one side of the brick to the other. Before generating any mesh, the user must specify the size of the elements to be generated. In this example, one element size will be specified for the volume as a whole and a smaller size will be specified for around the hole. A direct interval setting will be specified for the sweep direction.

To set the interval size for the overall volume, enter the command

```
cubit> volume 1 size 1.0
```

Since the brick is 10 units in length on a side, this specifies that each straight curve is to receive approximately 10 mesh elements.

In order to better resolve the hole in the middle of the top surface, we set a smaller size for the curve bounding this hole. To find the id number of the curve bounding the hole, the user can either pick the curve (See Selecting Entities with the Mouse) or turn curve labels on and regenerate the view. To do the latter, use the command

```
cubit> label curve on

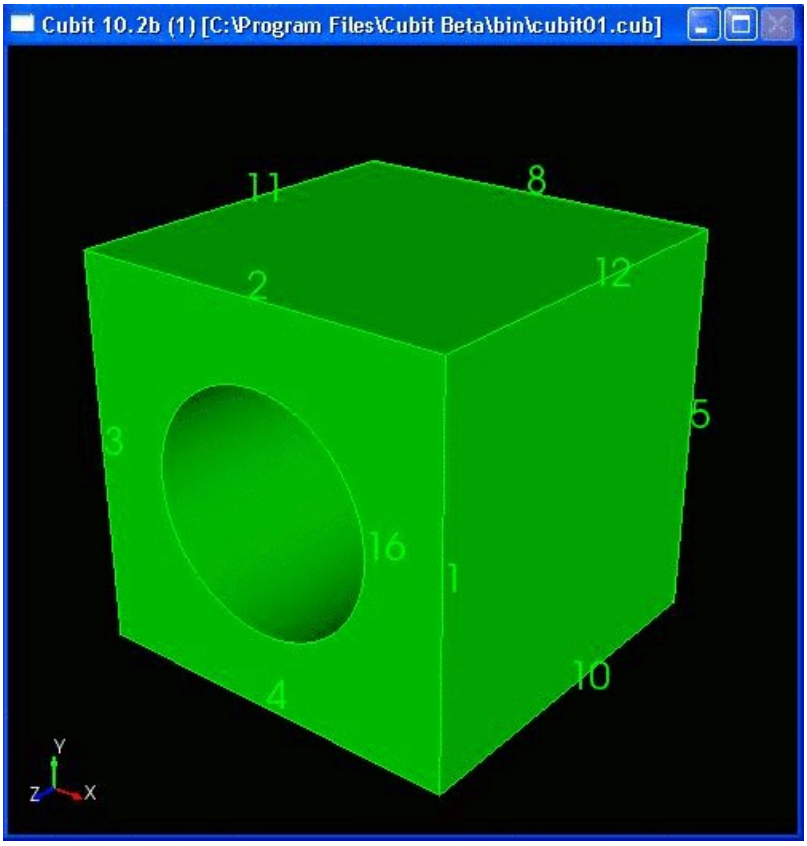
cubit> display
```

The default size of the labels can sometimes be too small to read. To change the text size, use the graphics text size command:

```
cubit> graphics text size 2

cubit> display
```

The result is shown in the figure below. Then the interval size can be set for the appropriate curve:



Geometry with Curve Labeling Turned on

```
cubit> curve 16 interval size 0.78
```

Finally, we would like to generate exactly 5 element layers in the sweep direction. This is accomplished by setting the intervals on curve 11:

```
cubit> curve 11 interval 5
```

CL Basic Tutorial Step 7

Step 7: Surface Meshing

Now that all the necessary intervals have been set, the meshing can proceed. Begin by meshing the front surface (with the hole) using the paving algorithm. This is done in two steps. First, set the scheme for that surface to Pave; then, issue the command to Mesh. Since the surface to be paved is number 11, issue the command:

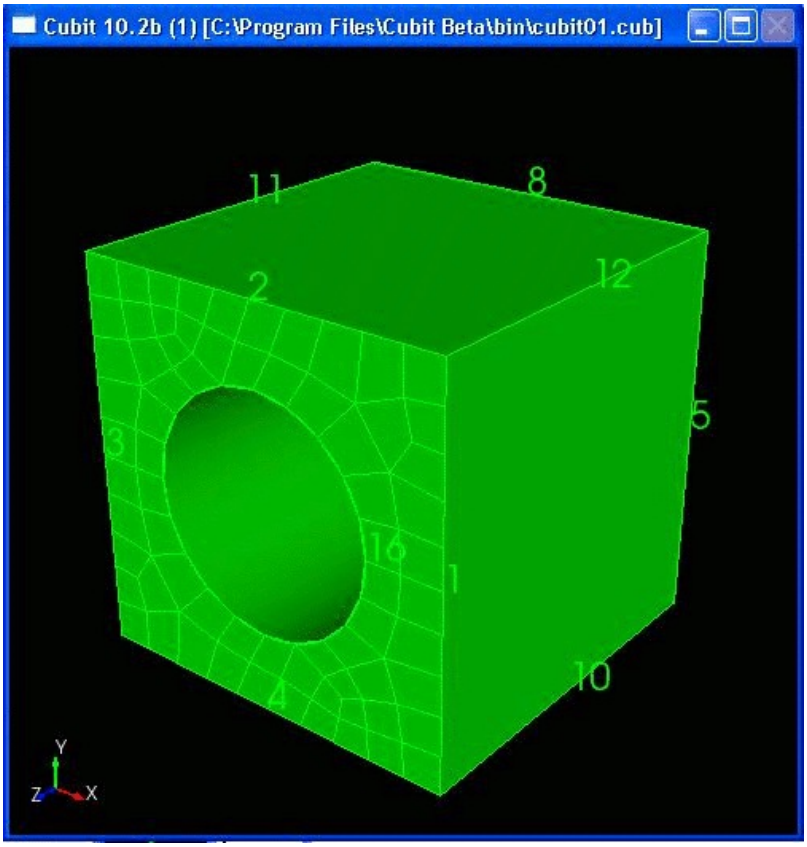
```
cubit> surface 11 scheme pave
```

With the meshing scheme specified, we proceed to mesh the surface:

```
cubit> mesh surface 11

cubit>display
```

The results are shown below:



Surface Meshed with Paving

CL Basic Tutorial Step 8

Step 8: Surface Meshing

The volume mesh can now be generated. Again, the first step is to specify the type of meshing scheme to be used and the second step is to issue the order to mesh. In certain cases, the scheme can be determined by CUBIT automatically. For sweepable volumes, the automatic scheme detection algorithm also identifies the source and target surfaces of the sweep automatically.

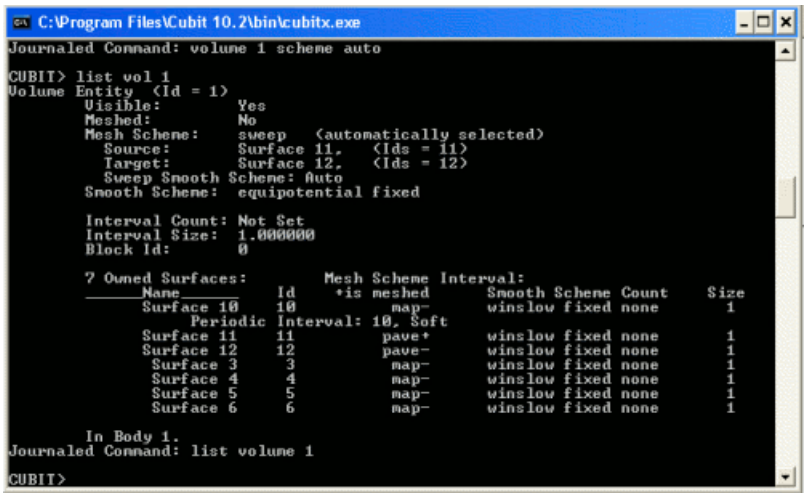
To instruct the code to automatically determine the meshing scheme and in this case the source and target surfaces, enter the command:

```
cubit> volume 1 scheme auto
```

To view the results of auto scheme selection, certain data about the volume can be listed:

```
cubit> list volume 1
```

The results of this command are shown below; note that the scheme, and in this case the source and target surfaces, are reported toward the top of the list output.

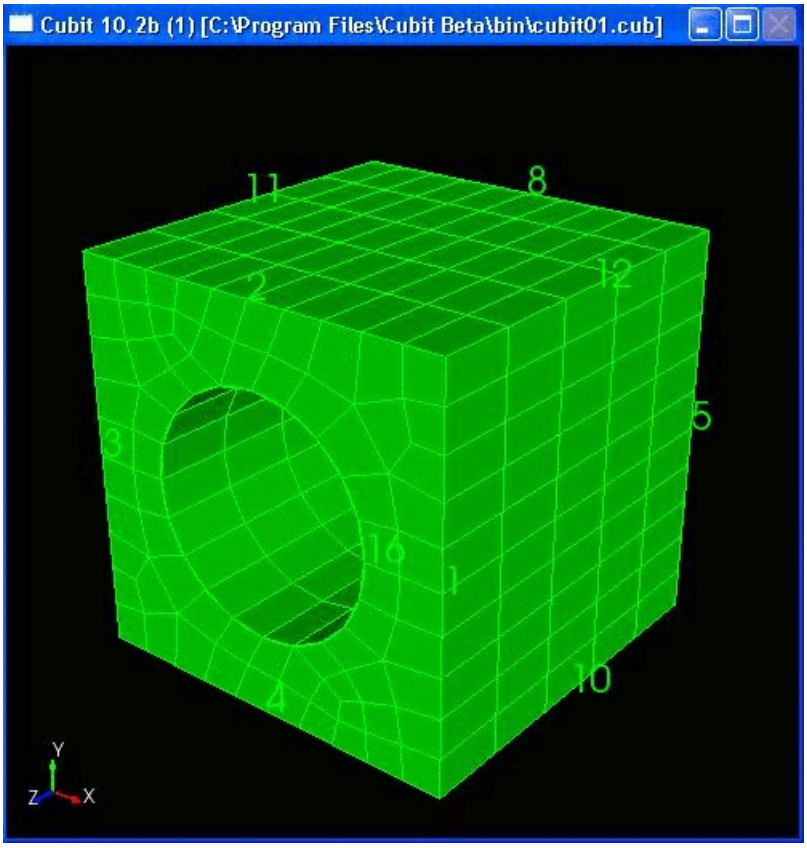


Output from Listing Volume 1

With the scheme set, the mesh command may be given:

```
cubit> mesh volume 1
```

The final meshed body will appear in the display window, as shown below:



View of Volume Mesh

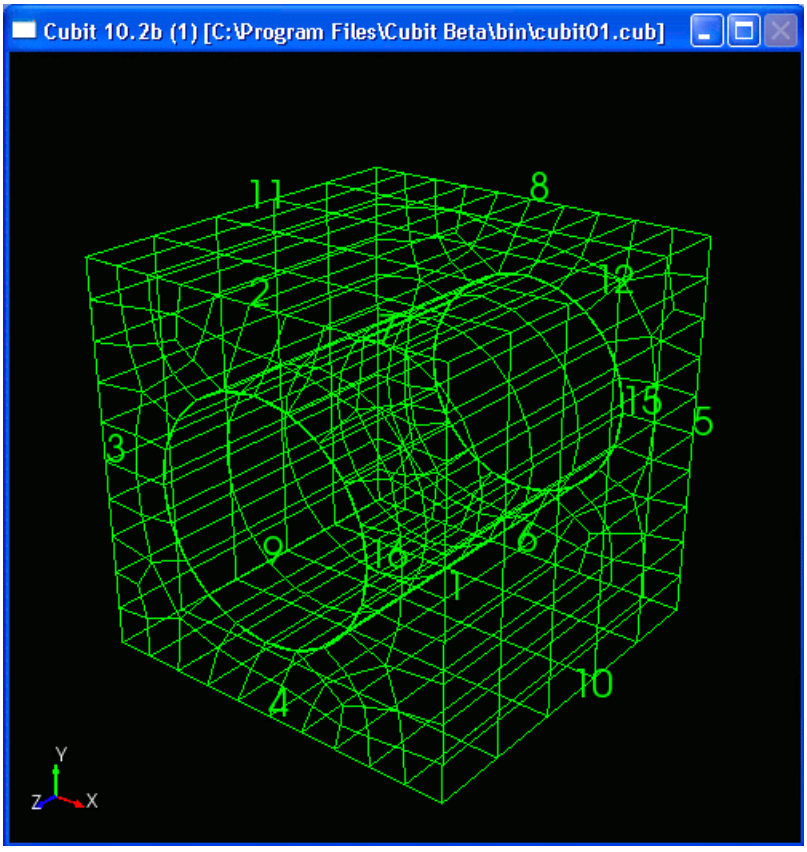
CL Basic Tutorial Step 9

Step 9: Inspecting the Model

The type, quality, and speed of rendering the image can be controlled in CUBIT by using several graphics mode commands, such as Wire Frame, Hidden Line, Transparent and Smooth Shade. For example:

```
cubit> graphics mode wireframe
```

The wire frame display is illustrated below:

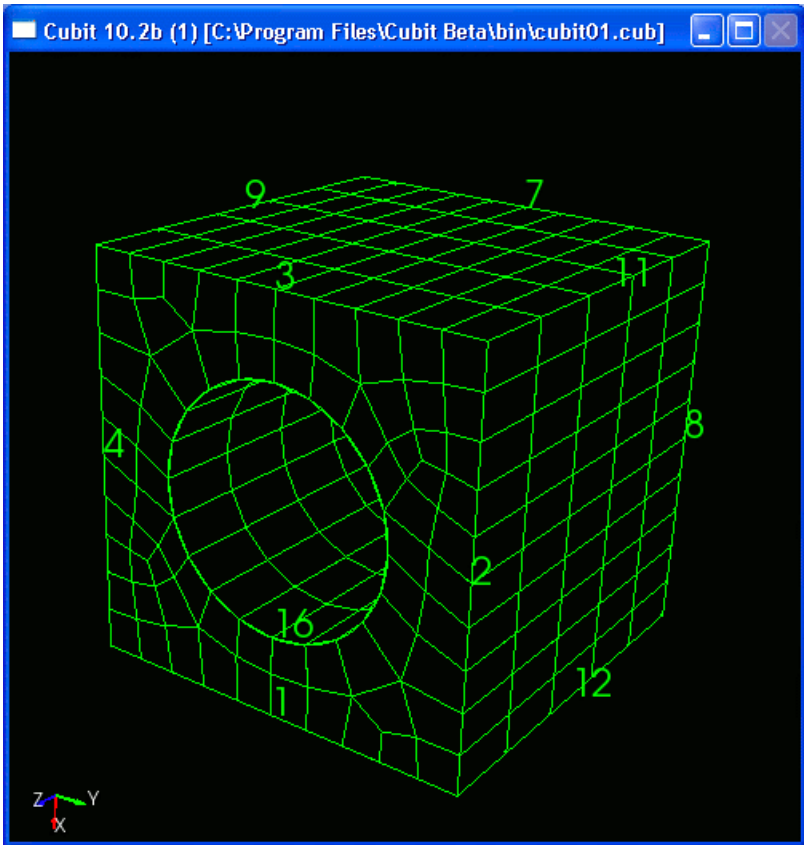


Wire Frame View of Mesh

Next, try:

```
cubit> graphics mode hiddenline
```

The hidden line display is illustrated below:

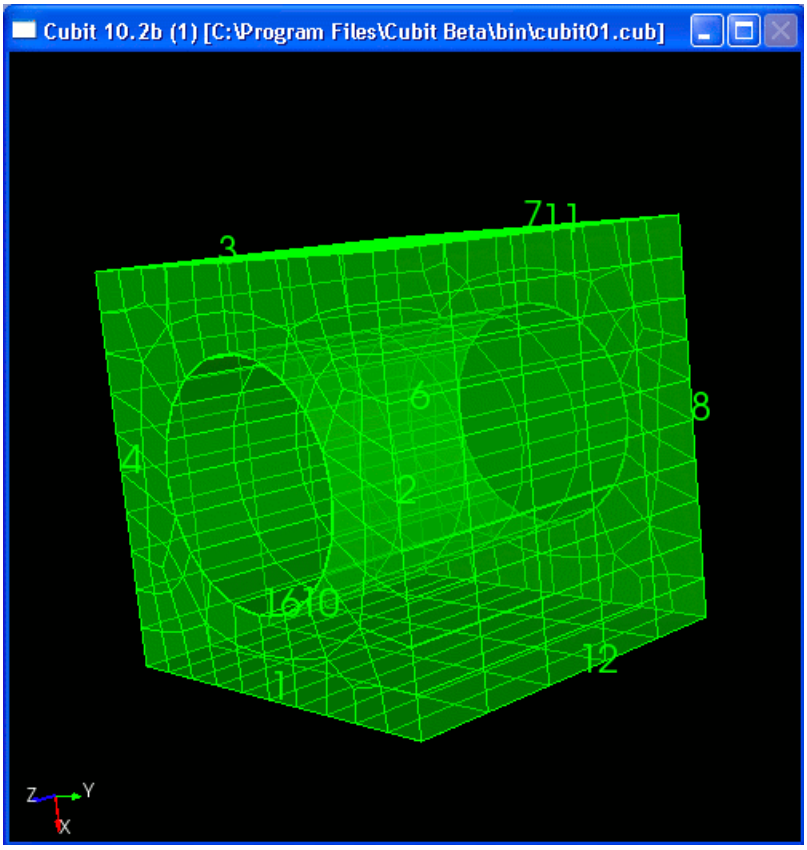


Hidden Line View of Mesh

Next, try:

```
cubit> graphics mode transparent
```

The transparent display is shown below.

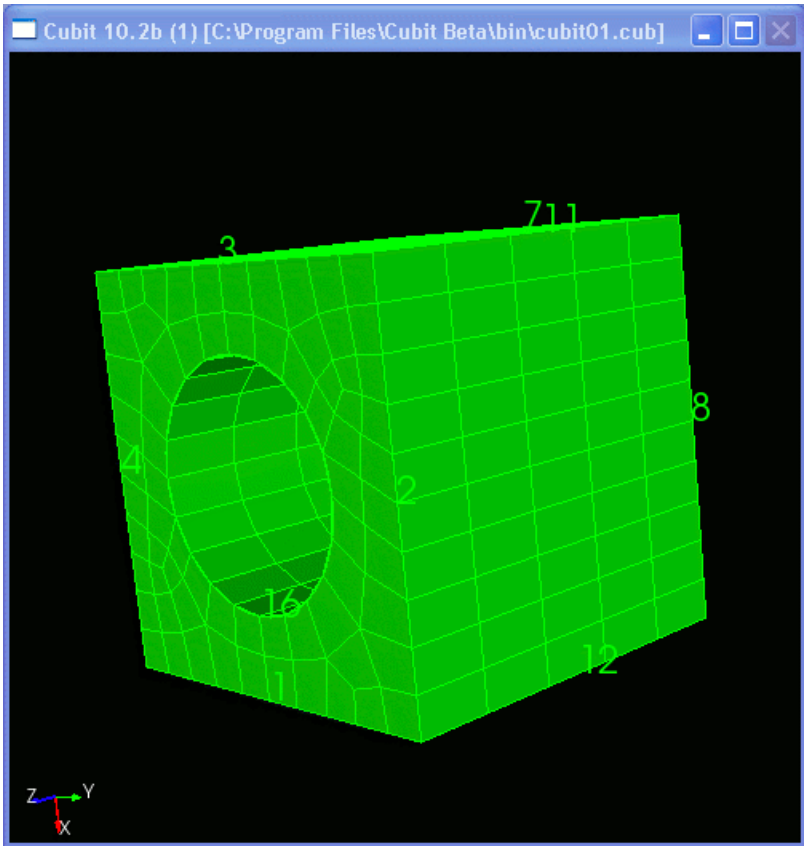


Transparent View of Mesh

Next, try:

```
cubit> graphics mode smoothshade
```

The smooth shade display is shown below. For detailed information on the viewing mode options, See Graphics Modes.

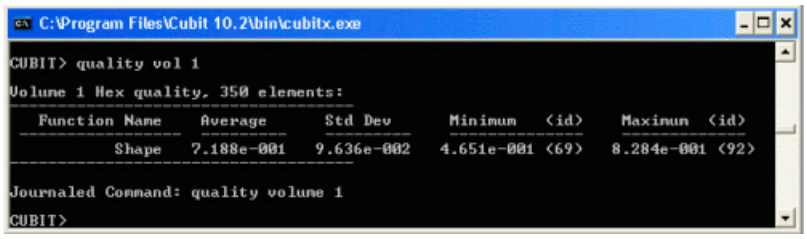


Smooth Shade View of Mesh

Although CUBIT automatically computes limited quality metrics after generating a mesh and warns the user about certain cases of bad quality, it is still a good idea to inspect a broader set of quality measures. To do this, enter the command:

```
cubit> quality volume 1
```

The results of the quality output are shown below. For an explanation of quality metrics along with acceptable ranges, see Mesh Quality Assessment. For the purposes of this tutorial, you can assume the quality metrics shown below are in an acceptable range.



Quality Table from Volume 1's Hex Mesh

CL Basic Tutorial Step 10

Step 10: Defining Boundary Conditions

Let us assume that we need to define one material type for the entire mesh, and a single node-based boundary condition on all surfaces. This is accomplished by identifying an Element Block and a Nodeset, respectively; the id numbers assigned to these entities are assigned by the user, usually by some convention meaningful to the analysis to be done. The element block and nodeset are identified using the commands:

```
cubit> block 100 volume 1

cubit> nodeset 100 surface all in volume 1
```



CL Basic Tutorial Step 11

Step 11: Exporting the Mesh

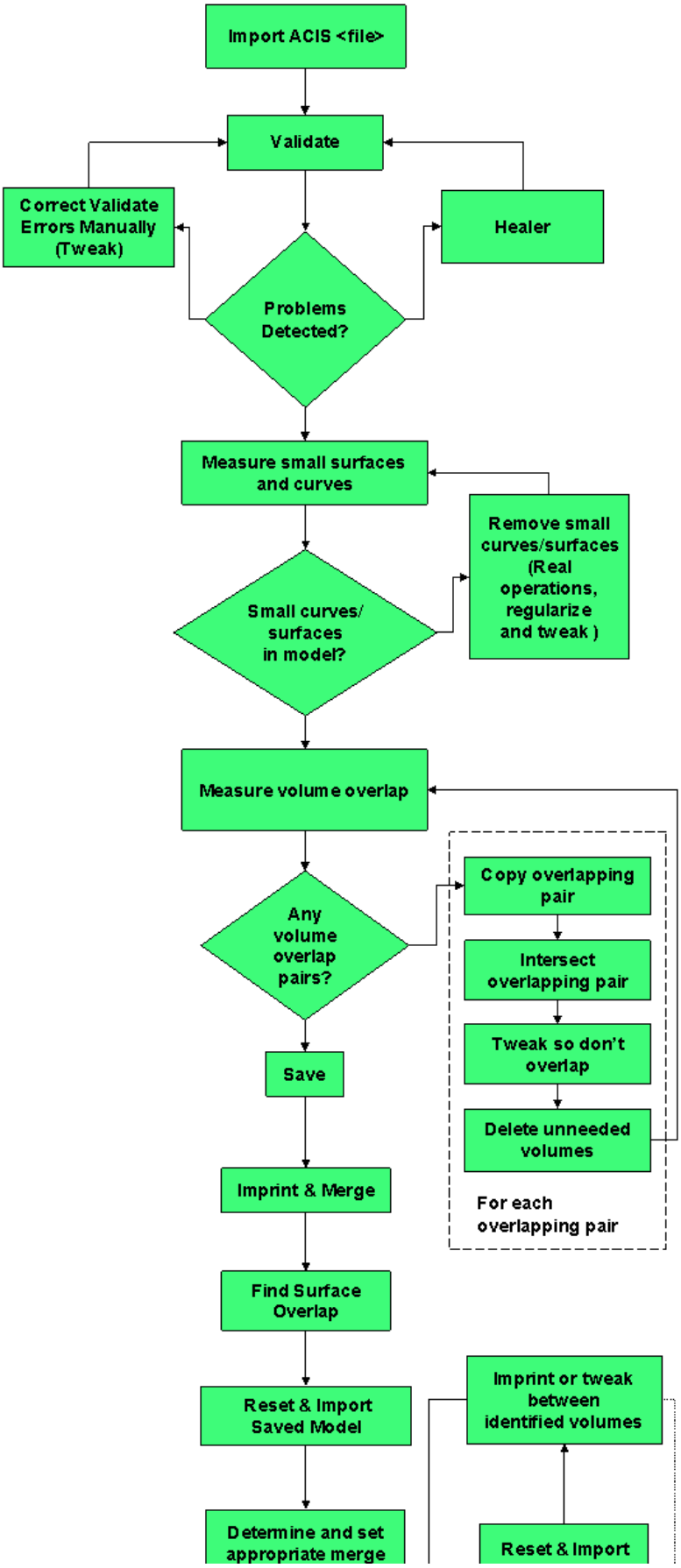
Finally, the mesh needs to be written to an ExodusII file. This is easily done:

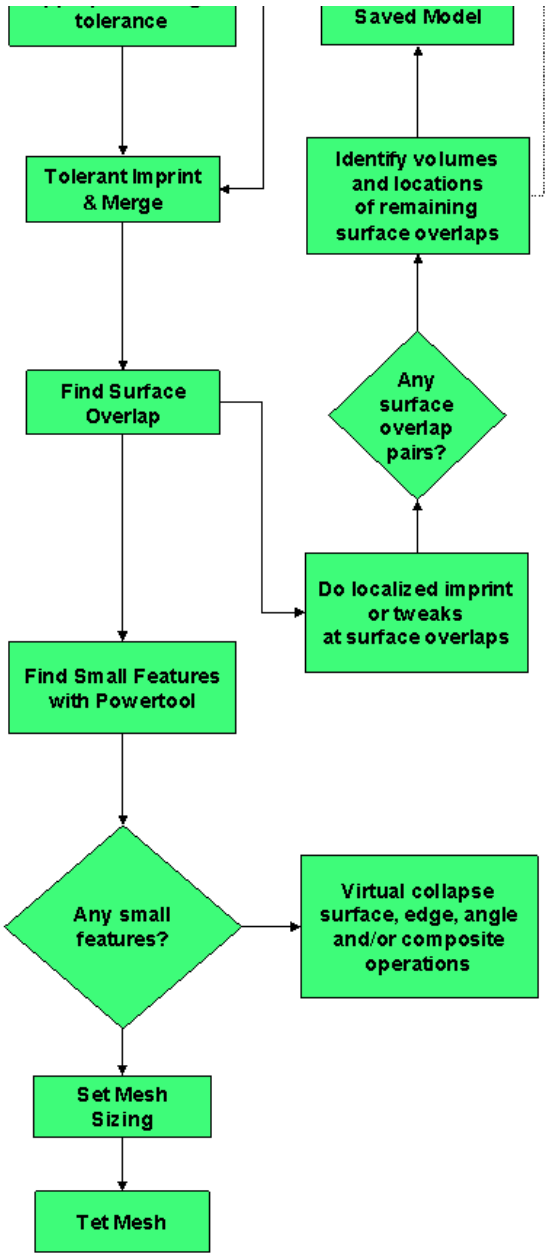
```
cubit> export genesis `brick_with_hole.g'
```

The filename and extension are arbitrary and, like the block and nodeset numbers, are usually named according to a convention meaningful to the analysis.



Geometry Cleanup Process Flow




tolerance
Saved Model
Tolerant Imprint & Merge
Identify volumes and locations of remaining surface overlaps
Find Surface Overlap
Any surface overlap pairs?
Do localized imprint or tweaks at surface overlaps
Find Small Features with Powertool
Any small features?
Virtual collapse surface, edge, angle and/or composite operations
Set Mesh Sizing
Tet Mesh

Immersive Topology Environment for Meshing (ITEM)

The Cubit Geometry and Meshing Toolkit team at Sandia has taken on the ambitious task of reducing the time for simulation by specifically addressing the bottlenecks in the mesh generation process. It is not unusual for the meshing process to take upwards of three-quarters of the entire simulation time. With its many tools developed for a wide range of application areas, it takes time to gain enough proficiency in Cubit to quickly generate a mesh from a complex geometry. As a result, the Immersive Topology Environment for Meshing (ITEM) was developed. ITEM is a user-interactive meshing tool that guides the user through a typical mesh generation process.

With the ultimate goal of reducing the time to generate a mesh for simulation, ITEM has been developed within the Cubit Geometry and Meshing Toolkit to take advantage of its extensive tool suite. Built on top of these tools it attempts to improve the user experience by accomplishing three main tasks:

1. Guiding the user through the workflow
2. Providing the user with smart options
3. Automating geometry and meshing tasks

Guiding the user through the workflow.

In software of any complexity where usage may be occasional or infrequent, the overhead of learning the new tool to a point of proficiency may be daunting. Given a solid model that may have been designed for manufacturing purposes, the analysts may be faced with generating a mesh. They may not be working with Cubit on a daily basis, but would like to take advantage of the powerful tools provided by the software.

To address this, ITEM provides a wizard-like environment that steps the user through the geometry and meshing process. For someone unfamiliar with the software, it provides an interactive, step-by-step set of tools for accomplishing the major tasks in the process. For those more familiar with the tools, it serves as a reminder of the major tasks, but is flexible enough to accommodate a more iterative approach, allowing them to jump between major tasks easily. Currently restricting the workflow to models requiring three-dimensional, solid elements, ITEM uses the following steps:

1. **Define the Geometric Model:** Import a CAD model or create geometry within the Cubit environment.
2. **Set up the model:** Define basic information such as element shape, volumes to be meshed and element sizes or budgets.
3. **Clean up the geometry:** Detect common issues and simplify geometric features on the CAD model.
4. **Meshing:** Perform operations to make the model meshable, such as imprint/merge, scheme selection, decomposition and performing the meshing.
5. **Validate the Mesh:** Check element quality and perform mesh improvement operations
6. **Apply boundary conditions regions:** Define regions where boundary conditions may be applied using nodeset, sideset and block definitions.
7. **Export the mesh:** Define a target analysis code format and export the mesh.

Providing the user with smart options.

Solid models used for analysis may have a huge variety of different

characteristics that may prevent them from being easily meshed.

Questions such as, What are the problems associated with my model? What are the current roadblocks to generating a mesh on this model? and What should I do to resolve the problems, are constantly being asked by the analysts. Without an extensive knowledge of the tools and algorithms, it may be difficult to answer these questions effectively.

ITEM addresses this issue by providing smart options to the user. Based on the current state of the model, it will automatically run diagnostics and determine potential solutions that the user may consider. For example, where unwanted small features may exist in the model, ITEM will direct the user to these features and provide a range of geometric solutions to the problem. Scrolling through the solutions provides a preview of the expected result. The user can then select the solution that seems most appropriate and execute the solution to change or simplify the geometry. This diagnostic-solution approach is the basis for the ITEM design and is the common mode of user interaction while in this environment. This contrasts with the more traditional hunt-and-guess approach of providing the user with an array of buttons and icons that they may choose from and guessing what may result. ITEM, on the other hand, serves in effect, as an expert providing guidance to the user as they proceed through the geometry and meshing process.

Automating geometry and meshing tasks.

With all of the advanced research and development that has gone into the meshing and geometry problem, a push-button solution for any arbitrary solid model may seem like the ideal objective of any meshing tool. Although for many cases, this would be the best solution, for others it may not even be desirable. A push-button solution assumes a certain amount of trust in the geometric reasoning the software chooses to provide. This may be more trust than an occasional user who is tasked with a high consequence simulation may be willing to give. Even if the user is willing to accept full automation, in many cases, the geometric complexity of the model may be beyond the capability of current algorithms to adequately resolve.

On the other hand, once the user is familiar with the characteristics of the solutions that the software provides, they may not be concerned with examining and intervening on every detail of the model creation process. Instead, in the interest of increasing efficiency, they may want the fastest solution possible. Providing the option for the user to automate as much of the geometry and meshing process as possible is another important aspect of ITEM.

For various characteristic geometric problems that are encountered in a solid model, ITEM can determine from the potential geometric solutions, which of them may be most applicable and apply that solution without any user intervention. For many configurations of geometry, a completely automated solution may be available. For others, only a portion of the process may be able to be automated. Where an adequate solution cannot be determined automatically, the smart options described above are available to help guide the user. As new advances in geometric reasoning and advanced meshing algorithms are developed, ITEM will incorporate these into the solutions for automation.

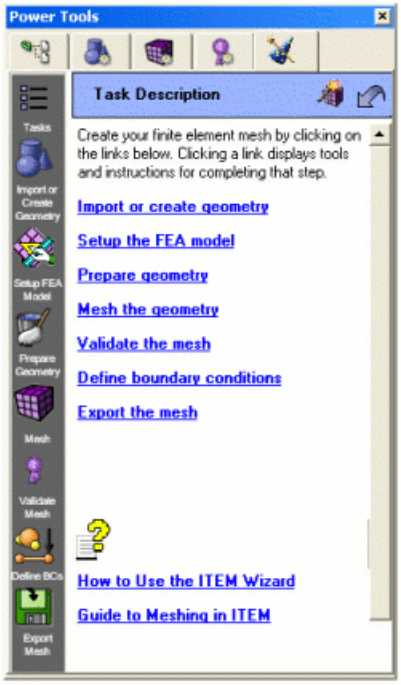
It should be clear that ITEM is not intended to be a fully automated system for meshing solid models. Instead it is intended to be a flexible environment that will guide the user through the model generation process by offering solution alternatives and providing automation should the user choose. The remainder of this document is organized according to the basic workflow used in ITEM. The objective is to describe the general problems that may be encountered in developing an analysis model and how ITEM and Cubit may be used to address the problems. In developing this environment, many new innovative tools were invented and developed to help support this new approach to mesh and model generation.

How to Use the ITEM Wizard

The ITEM Workflow

The Immersive Topology Environment for Meshing (ITEM) is a wizard-like environment that guides the user through the mesh generation process from geometry definition to export. ITEM was designed to provide a step-by-step set of tools to help new users generate a mesh with very little previous knowledge of the CUBIT program. But ITEM is also flexible enough to accomodate advanced users who want to use a more iterative approach, or who just want to use ITEM for a specific tool or panel.

The main ITEM task page is shown below. To access this page, click on the "wizard hat" icon from the Power Tools window.



Main ITEM Task Panel

The main item tasks are shown both in the text window, and also along the sidebar. The icons in the sidebar are available from any of the ITEM panels. It is acceptable to jump to different tasks during the process, although beginning users may just want to follow the steps in order. To get to the main task page, click on the Task icon on the sidebar during any step in the process.

Many meshing tasks require an iterative approach to the mesh generation process. For your convenience, if you do click on one of the task buttons from a different panel, it will take you to the last visited panel in that section. For example, if you are on the mesh generation page, and you click on the prepare geometry section, it will take you to the last page you visited in the prepare geometry section.

There are two help links at the bottom of the main task page. The first link will open this document which describes the general ITEM process and how to use the panels. This page is only accessible from the main task page. The second link opens the main ITEM documentation which describes each process in the ITEM mesh generation process in detail. This document can be accessed from any of the ITEM panels.

To proceed through the ITEM panels you must either click on a task or

click on the "Done" button at the bottom of each page. There is no "Back" button on the ITEM interface. But in most cases, clicking the "Done" button works like a "Back" button.

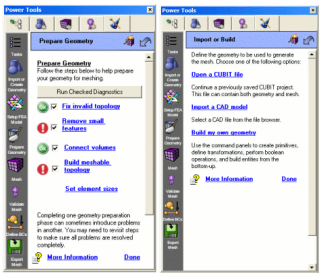
Using an ITEM Panel

The item panels are designed to be self-explanatory, with plenty of documentation on each page, and access to more help if needed. However, it does help to be generally familiar with the main types of panels.

Task panels that link to other ITEM panels

Some ITEM panels provide a list of tasks that link to other ITEM panels. Sometimes the tasks are designed to be completed in sequential or iterative fashion. In that case, you will be returned to the task page after selecting done on each sub-panel where you can select the next task. The Prepare Geometry panel is an example of this case. Each of the tasks with a warning flag should be completed. As you return to this panel, you may need to run the diagnostics again, and possibly even revisit previous task pages.

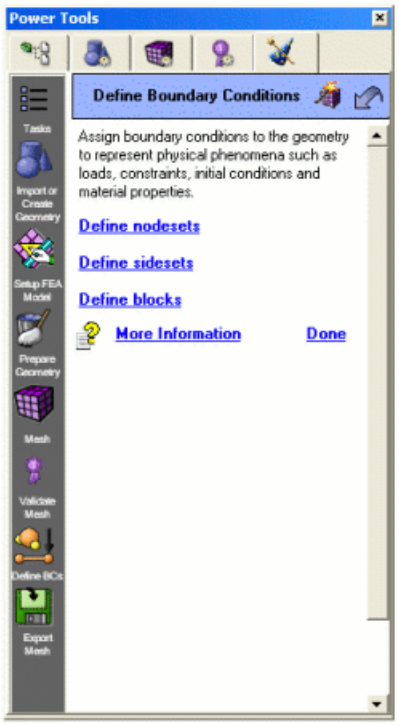
In other cases, the list of tasks is a presents a list of choices, from which you will only select one option. The Import Geometry Page shown below is such an example. It gives a list of different geometry import/creation options and you just select one of the alternatives.



Prepare Geometry ITEM Panel **Import Geometry ITEM Panel**

Task Panels that Link to Control Panels

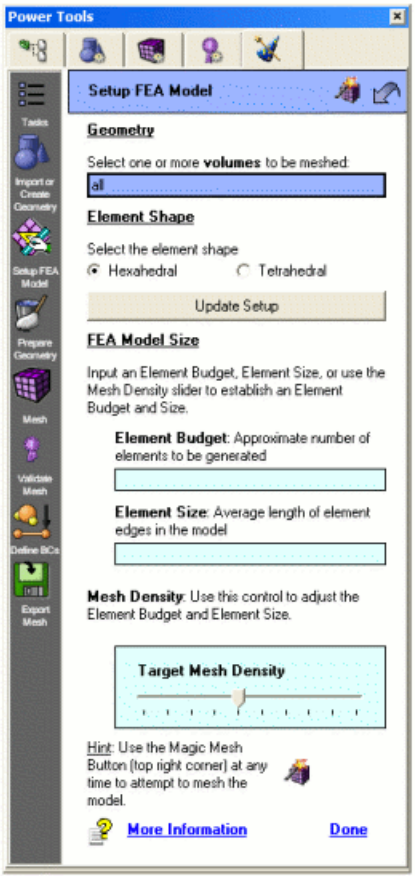
A few of the ITEM task panels will provide links to existing control panel topics. Clicking on a link from one of these panels will NOT open a new panel, but will open the corresponding control panel. The Define Boundary Conditions page is an example of this type of panel.



Define Boundary Conditions Panel

Set-up Panels

A set-up panel is used to provide input or set-up options for your model. The most prominent set-up panel is the Set-up FEA Model page which is used to define mesh budget, element type, and element size. Another set-up page is the Define Metrics page under the Validate Mesh task. This panel is used to define quality metrics for your model. These panels provide useful information for the diagnostics used in other panels.



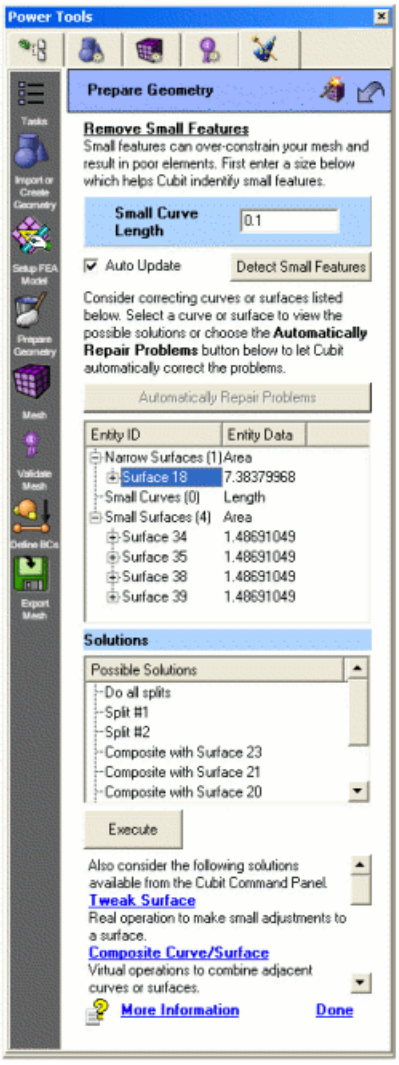
Setup FEA Model Panel

Diagnostic Panels

The most useful type of ITEM panel is the diagnostic panel. These panels each focus on a specific diagnostic such as invalid topology, small features, blend surfaces, overlapping surfaces, or meshability. Most of theses panels contain some or all of the following features.

- **Diagnostic Button** - Clicking on this button will run a series of tests on the model.
- **Output Window** - Displays the results of the diagnostics and lists entities with problems. Includes a right-click menu with visualization and other options.
- **Automatically Repair Button** - Tries to solve the problems automatically.
- **Solution Window** - Presents a list of specific solutions based on the entity you select in the output window. This window also contains several right-click context menu items for each solution, including a "More Information" button which will open the documentation to information about that specific task. Another useful feature of the solution window is that in most cases clicking on one of the solutions will preview that option in the graphics window.
- **Execute Button** - Executes the solution selected.
- **Additional Options** - Sometimes you won't see your desired solution in the list. Additional solutions with brief descriptions are provided at the bottom of the panel. Clicking on these links will open the corresponding control panel.
- **More Information Link** - Opens a page describing the diagnostics and solutions used for this panel.

The Small Features Panel shows an example diagnostic panel in ITEM.



Undo Button

The Undo button allows you to reverse the most recent command. To enable the Undo button, click on the "Enable Undo" option from the Edit menu. The undo button works by saving information about your model after each step. For large or complex models, this can be time consuming, so you may need to disable the undo feature. Additionally, not all commands are enabled for undo. Many of the graphics and meshing commands, and various default settings are not included. Within ITEM, many commands are bundled into a single button click. Clicking undo will attempt to reverse all of the executed commands. See the command line window for the results of the undo command.

Magic Mesh Button

This button, shown at the top of each ITEM panel, provides the user with the opportunity to use Cubit's internal automation algorithms to generate a mesh. In addition to simply issuing a mesh command, it will attempt to execute the following steps.

- **Geometry Cleanup:** Check for small or ill-defined geometry and automatically resolve it
- **Auto-scheme:** Automatically set meshing schemes and select sources and targets for hex meshing
- **Decomposition:** If hex meshing, attempt to decompose the volume to admit a sweep or mapped mesh
- **Force Sweeps:** For almost-sweepable geometry, modify the linking surfaces to force a sweep
- **Imprint/Merge:** For assemblies, imprint adjacent volumes and merge common surfaces
- **Overlap check:** Check for any remaining overlapping volumes and attempt to resolve merge problems
- **Mesh sizing:** For tetrahedral meshing, automatically define a sizing function based on geometry characteristics
- **Interval Matching:** For hex meshing, coordinate the assignment of curve intervals.
- **Sweep grouping:** Determine an appropriate order to mesh volumes to reduce dependencies
- **Mesh:** Perform the mesh operation volume(s)
- **Mesh Quality:** Check mesh quality and locally optimize if necessary

If for any reason, Cubit is unable to complete these steps without further user intervention, the process will stop and the user will directed to continue with the ITEM workflow. For simple geometries, executing the magic mesh button at this phase of the workflow may be all that is necessary to completely define a good quality mesh. For other more complex geometry, considerable user intervention may be required.

The magic mesh button may be executed at any time during the ITEM workflow by selecting the button at the top right corner of the ITEM panel. Once the user has visited the various panels of the ITEM interface to provide user intervention, the automatic execution of the appropriate operations will not longer be attempted.

Getting Help

There are several ways to get help from within the ITEM interface. Most of these have already been discussed, but they are listed here again for reference:

- **How to Use ITEM** - This document which is available only from the main task page
- **Guide to Meshing in ITEM** - A document which describes the

ITEM workflow, and how to use the diagnostics on each page. This is accessible from each page using the More Information links.

- **Individual help topics for specific solutions** - Opens the documentation to help for each specific solution topic. This is accessible from the right-click menu when a command is selected in the solutions window.
- **Documentation included on panels** - Many of the panels contain brief descriptions and explanations to describe the features and tools on that panel.

Setting up the Finite Element Model

Once the geometry to be meshed has been imported or created, the first step to defining the mesh is to set up the model. Basic parameters that are needed through the rest of the ITEM workflow are defined at this stage. Subsequent diagnostics and workflow may change based on how the model is initially set up.

Element Shape

Either a hexahedral or tetrahedral element shape may be selected. The meshing algorithm used to mesh the volumes will change based on this setting. Specific element characteristics such as the order of the element (i.e. TET10, HEX20) may be specified at a later time. The steps that will be displayed in the workflow will change based on the element type that is selected.

FEA Model Size

The number of elements or average size of the elements is an important aspect of defining your analysis model. Geometric features that are considerably smaller than the average element size, in most cases should be ignored since the mesh resolution will not be able to adequately capture them. Defining the element size at this point in the workflow permits subsequent diagnostic tests and operations to have a relative measure of what is “small”. More detailed sizing attributes such as biasing and geometry-adaptive sizing may be defined later in the ITEM workflow.

One of three different mechanisms may be used to define the size, element budget, element size and mesh density. Each of these values is dependent on the other. As a result, changing one value will automatically change the other.

- *Element Budget:* This value is an approximate number of elements that should be generated in the entire model. The element budget for hexahedra, Nhex, is related to the element size, esize, by the following relationship:

  $$e_{size} = \sqrt[3]{\frac{V_{model}}{N_{hex}}}$$

  Where Vmodel is the geometric volume of the solid model. The element budget for tetrahedra vs. hexahedra is approximately 1:7. That is, for an equivalent edge length, a tetrahedral mesh will contain roughly seven times as many elements as a hexahedral mesh.

- *Element Size:* Element budget and mesh density are indirect methods for setting the element size, esize. This value can also be set explicitly. It represents the approximate average edge length of elements in the model. This size will determine the relative definition of small for subsequent diagnostic tests and will be used to set the mesh size the meshing algorithms will use.

- *Mesh Density:* The mesh density is represented by an integer between 1 and 10, where 1 is the finest resolution and 10 is the coarsest. It is a heuristic measure of how fine of a mesh will be generated and permits the user to indirectly set an element size without explicitly defining a real value. In most cases, the mesh density, md is related to the element size, esize by the following heuristic relationship:

$$e_{size} = \sqrt[3]{V_{max}}\,(0.03 + 0.00045 m_d^{3.1})$$

Where Vmax is the of the geometric volume of the largest volume in the solid model. Changing the target mesh density will display a preview of the approximate nodal spacing on the curves of the model in the graphics window.

Defining the Geometric Model

Various methods may be used to define a geometric model. In most cases, a solid model is created in a commercial CAD tool such as Pro/Engineer or Solidworks. It can also be generated natively within Cubit using geometry commands. One of the most time consuming tasks in developing an analysis model is in dealing with geometric anomalies. Carefully considering how the model is constructed and what format the model will be defined in can eliminate many potential problems downstream in the model creation workflow. The following describes the various solutions for defining geometry within Cubit along with their pros and cons:

- Geometry Formats
- Creating Your Own Geometry
- Scripting
- CUB Files

Geometry Formats

Cubit can use one of three different commercial geometry representations, ACIS (.sat, .sab), Pro/E (.g) or Catia (.cat). It may also use a facetted format (MBG) that is developed in-house at Sandia. When a model of any of these formats is imported, Cubit uses the appropriate third party geometry kernel to directly manage and evaluate the geometry. Since the geometry is considered “native” when any of these formats is used, no translation step is required.

Since commercial solid modelers do not necessarily agree on formats and representations, using a translation process to convert a non-native format to a native format, can introduce errors in the geometry. While this in itself may not be a show-stopper, it can frequently add hours to an otherwise simple process while the user is forced to clean up dirty geometry. Neutral formats such as STEP and IGES are common in the CAE industry. They can often be an ideal solution for representing the analysis solid model. In Cubit, when importing a neutral format, it is automatically translated to the ACIS format. The user should be careful however in selecting these formats as commercial solid modeling engines frequently interpret standard specifications for these formats in different ways sometimes resulting in unusual results. Wherever possible a native format should be used.

Native geometry kernels provide the most accurate way for transferring data between solid-model based applications. Since these geometry kernels must be licensed and incorporated into the Cubit distribution separately, one drawback is the additional licensing and cost for maintaining these kernels. Cubit is currently able to provide licenses for ACIS and Pro/E kernels for government and academic use. Additional licensing arrangements may be required for Catia or for any commercial use.

Creating your own geometry

Cubit offers a wide variety of tools for creating geometry natively. The advantage to this is the ability to control the geometry creation process without the need for another CAD tool. Although Cubit is not designed to be a CAD tool it does provide many tools for both bottom-up and primitive creation.

Bottom-up creation refers to the process of building geometry from its basic components starting with vertices, curves, surfaces and then volumes. This process can be somewhat tedious, but is often useful for generating auxiliary geometry once a CAD model has been imported.

Primitive creation refers to the various operations for generating geometric primitives such as bricks, spheres, cylinders and cones. Once defined, operations for repositioning the objects and performing Boolean

defined, operations for repositioning the objects and performing Boolean operations between them may be used. Relatively complex models may be generated using this approach.

Scripting

One advantage to generating your own geometry within Cubit is the ability to parameterize the construction of the model. Cubit utilizes a rich command language that can be stored as a script or journal file. Parameters representing dimensions of objects may be defined in the script and conveniently adjusted to update the geometry representation. For more ambitious users, Cubit also has the ability to interpret python scripts, allowing a high degree of customization that can employ the full capability of the python scripting language.

It should be noted that when using Cubit, commands are automatically echoed to an external temporary journal file on disk and to the history window. Observing these commands is a good way to become familiar with Cubit's internal command language. Copying and pasting selected commands to a text editor is an ideal method for building a parameterized journal file. Journal files may be built up and played back to reproduce the entire process of building an analysis model.

CUB Files

A CUB file is Cubit's database file. You may want to think of it as a snap-shot of the current state of the model. While journal files record the process for creating the model, a CUB file stores only the end state. It can include both geometry in its native format and any mesh information as well as attributes and boundary condition information. Restoring a CUB file will write over any existing data you currently have defined.

Generating a Mesh in ITEM

The mesh generation panel in ITEM is different from the other panels in Cubit. Meshing errors can arise from a number of different problems. Many of these problems are caused from improper geometry preparation/cleanup. Other problems can be caused from improper interval settings, or meshing schemes. Instead of suggesting specific operations as it does on other panels, the meshing panel in ITEM will suggest several possible solutions based on the error message output. Each of these solutions may require significant user input, and may require you to revisit previous ITEM panels or Control panels. To open the appropriate Control panel, you can right click on the solution and select "Show Command Panel". For convenience, these general solutions are described here, including which ITEM panels and which Control panels they refer to. References to help topics are also included.



Figure 1. ITEM Mesh Panel

ITEM Meshing Suggestions

1. ***The volume is not decomposed enough. It may need to be webcut.***

   Diagnostic: This solution message appears when auto scheme selection fails. There are many reasons that auto scheme selection may have failed. Check to make sure that your volume is broken up into meshable parts. For sweepable volumes, this means that each volume should only have one target surface.

   Action: Right-clicking on this solution and selecting the "Show command panel" option will open the webcutting commands on the control panel. Alternatively, you can also return to the ITEM decomposition panel for more webcutting suggestions.

Help Topics:
Geometry Decomposition explains diagnostics and solutions on the ITEM decomposition panel
Decomposition Tutorial has several webcutting tips and examples.
Web Cutting Documentation contains all of the syntax for webcutting commands in Cubit.

2. ***Meshing schemes may need to be manually set.***

   Diagnostic: This solution message appears when auto scheme selection fails, interval matching fails, or interval assignments fail. Setting the schemes manually may help resolve some of these issues. It may also help to set source and target surfaces explicitly for swept meshes.

   Action: The volume schemes can be set explicitly from the Volume-Mesh control panel. The "Set Source and Target" panel in ITEM can be used to aid in setting explicit source and target surfaces for swept meshes.

   Help Topics:
   Recognizing Nearly Sweepable Regions explains how ITEM might be used to recognize nearly sweepable regions.
   Meshing the Geometry has some suggestions for getting difficult geometry to mesh.
   Decomposition Tutorial has several examples where the meshing schemes have to be set manually.
   Meshing Schemes gives an overview of all of the meshing schemes in Cubit.

3. ***The mesh size or number of intervals on a volume may need to be changed.***

   Diagnostic: This solution message appears for many reasons: auto scheme selection fails, interval matching fails, interval assignments fail, inconsistent edge-face ratios, odd number of intervals on a paver loop, or connectivity problems. Setting explicit intervals may be necessary

   Action: The volume mesh intervals can be set explicitly from the Volume-Interval control panel. The "Set Element Sizes" panel in ITEM can be used to aid in setting explicit sizes and sizing functions for meshes.

   Help Topics:
   Interval Assignment has links to different interval assignment methods in Cubit
   Bias, Dualbias describes how to create a biased mesh and Controlling Mesh Quality describes how to propagate a curve bias.
   Decomposition Tutorial has several examples where the meshing intervals are set manually.
   Mesh Adaptivity and Sizing Functions describes how to use sizing functions in Cubit.

4. ***Compositing surfaces or curves to remove unnecessary details may resolve the problem.***

   Diagnostic: This solution message appears when auto-scheme selection fails. A model may contain small curves or surfaces that need to be composited with adjacent surfaces. Or it may just contain more detail than is needed for analysis. Compositing surfaces and curves does not affect the underlying geometry.

   Action: The Remove Small Features or Force Sweep Topology panels in ITEM may suggest several possible candidates for compositing. The Surface-Modify-Composite or the Curve-Modify-Composite panels can be used to composite surfaces or curves respectively. These panels are also used to delete virtual geometry from curves or surfaces.

   Help Topics:

Removing Small and Narrow Features describes using ITEM to remove small and narrow features in your model.
Forced Sweepability describes using ITEM to force sweepability using virtual geometry.
Composite Curves explains how to composite curves in Cubit.
Composite Surfaces explains how to composite surfaces in Cubit.
Decomposition Tutorial Example 7 has an example of using composite curves to improve meshability.
Power Tools Tutorial has another example of using composite geometry.

5. ***Collapsing surfaces, curves, or angles to remove unnecessary details may resolve the problem.***

   Diagnostic: This solution message appears when auto-scheme selection fails. Collapsing a surface involves splitting a surface, and compositing it with adjacent surfaces.

   Action: The Remove Small Features panel in ITEM may suggest several possible candidates for collapse. The Surface-Modify-Collapse, Curve-Modify-Collapse, or Vertex-Modify-Collapse Angle panels can also be used to collapse surfaces, curves, or angles respectively.

   Help Topics:
   Removing Small and Narrow Features describes using ITEM to remove small and narrow features in your model.
   Collapse Angle explains how to collapse angles in Cubit.
   Collapse Curves explains how to collapse curves in Cubit.
   Collapse Surfaces explains how to collapse surfaces in Cubit.
6. ***Removing unnecessary surfaces or curves to simplify geometry may improve the chances that a volume will mesh.***

   Diagnostic: This solution message appears when auto-scheme selection fails. Removing unnecessary surfaces may improve meshability.

   Action: The Remove Small Features panel in ITEM may suggest several possible candidates for removal. The Surface-Modify-Tweak panel, Surface-Modify-Remove panel, Curve-Modify-Tweak or the Volume-Modify-Remove Slivers panels are also used to remove unnecessary features in a model.

   Help Topics:
   Removing Small and Narrow Features describes using ITEM to remove small and narrow features in your model.
   Removing Geometric Features describes the syntax for removing unneeded surfaces and vertices, including sliver surfaces.
   Tweaking Geometry contains the syntax for tweaking surfaces, curves, and vertices.
   Power Tools Tutorial has an example of using the tweak surface command to simplify a model.
7. ***Smoothing the mesh may improve the mesh quality.***

   Diagnostic: This solution message appears when mesh generation creates poor quality elements, particularly if it creates inverted or "negative Jacobian" elements. In some cases, smoothing a mesh may get rid of these bad elements.

   Action: Depending on the geometry type, the smoothing panel can be accessed from the Control panel under Volume-Smooth or Surface-Smooth panels. It is also helpful to use the Validate Mesh page in ITEM for assessing quality metrics.

   Help Topics:
   Mesh Smoothing describes the different smoothing schemes in Cubit and how to use them.
   Mesh Validation describes how to use quality metrics in ITEM and

gives suggestions on smoothing schemes to try.
Mesh Quality Assessment describes the different quality metrics in Cubit and how to use them.

8. ***Deleting the mesh on an entity in order to further decompose or modify it may be necessary.***

   Diagnostic: This solution message appears when mesh generation creates a poor quality mesh, due to negative Jacobians, inconsistent edge-face ratios, connectivity problems, or any other invalid mesh configuration. Mesh generation can be a very iterative process. It is sometimes necessary to delete a mesh and try different schemes, sizes, or even just change the meshing order. Sometimes you must further decompose or modify your geometry to get it to mesh.

   Action: To delete a mesh, you can select it in the graphics window and choose Delete Mesh from the right-click context menu. You can also delete a mesh from any of the Mesh-Entity-Delete panels on the Control Panel.

   Help Topics:
   Mesh Deletion describes command line syntax for deleting a mesh.

9. ***Changing vertex types may make the surface or volume meshable.***

   Diagnostic: This solution message appears when mesh generation fails to assign valid vertex types on mapped or submapped surfaces.

   Action: To change the vertex type on a surface, select the Surface-Mesh-Submap-Advanced or Surface-Mesh-Map-Advanced panels. From here you can assign and view vertex types.

   Help Topics:
   Surface Vertex Types describes how to change the vertex types on a geometry.

Validating the Mesh in ITEM

Advancements in the mesh generation algorithms have significantly reduced the amount of quality problems seen in the initially generated mesh. Further, ITEM generally relies on the most robust meshing algorithms available in CUBIT, specifically sweeping for hexahedral mesh generation (Scott,05) and the MeshGems (George,91) meshing software (See http://www.distene.com). However, some problems can still exist, and therefore ITEM has integrated quality diagnostics and solution options.

Diagnostics: After the mesh has been generated, the user may choose to perform element quality checks. ITEM utilizes the Verdict (Stimpson,07) library where a large number of mesh quality metrics have been defined and available as a modular library. If no user preference is specified, ITEM uses the Scaled Jacobian distortion metric to determine element quality. This check will warn users of any elements that are below a default or user-specified threshold, allowing various visualization options for displaying element quality.

Solutions: If the current element quality is unacceptable, ITEM will present several possible mesh improvement solutions. The most promising solutions are provided through ITEM's interface to two smoothers: mean ratio optimization and Laplacian smoothing. These are provided as part of the Mesquite (Brewer,03) mesh quality improvement tool built within CUBIT. The user has the option of performing these improvements on the entire mesh, subsets of the mesh defined by the element quality groups, or on individual elements. The Laplacian smoothing scheme allows the users to smooth just the interior nodes or to simultaneously smooth both the interior and boundary nodes in an attempt to improve surface element quality.

Recognizing Nearly Sweepable Regions

The purpose of geometry operations such as decomposition is to transform an unmeshable region into one or more meshable regions. However, even the operations suggested by the decomposition tool can degenerate into guesswork if they are not performed with a specific purpose in mind. Without a geometric goal to work toward, it can be difficult to recognize whether a particular operation will be useful.

Incorporated within the proposed ITEM environment are algorithms that are able to detect geometry that is nearly sweepable, but which are not fully sweepable due to some geometric feature or due to incompatible constraints between adjacent sections of geometry. By presenting potential sweeping configurations to the user, ITEM provides suggested goals to work towards, enabling the user to make informed decisions while preparing geometry for meshing.

Unlike the decomposition solutions presented in the previous section, the purpose of recognizing nearly sweepable regions is to show potential alternative source-target pairs for sweeping even when the autoscheme tool does not recognize the topology as strictly sweepable. When combined with the decomposition solutions and the forced sweepability capability described later, it provides the user with an additional powerful strategy for building a hexahedral mesh topology.

Diagnostics: In recognizing nearly sweepable regions, the diagnostic tool employed is once again the autoscheme tool described in [White, 00]. Volumes that do not meet the criteria defined for mapping or sweeping are presented to the user. The user may then select from these volume for which potential source-target pairs are computed.

Solutions: The current algorithm for determining possible sweep configurations is an extension of the autoscheme algorithm described in [White, 00]. Instead of rejecting a configuration which does not meet the required sweeping constraints, the sweep suggestion algorithm ignores certain sweeping roadblocks until it has identified a nearly feasible sweeping configuration. The suggestions are presented graphically, as seen in Figure 1. In most cases, the source-target pairs presented by the sweep suggestion algorithm are not yet feasible for sweeping given the current topology. The user may use this information for further decomposition or to apply solutions identified by the forced sweepability capability described next. The sweep suggest algorithm also provides the user with alternative feasible sweep direction solutions as shown in Figure 1. This is particularly useful when dealing with interconnected volumes where sweep directions are dependent on neighboring volumes.

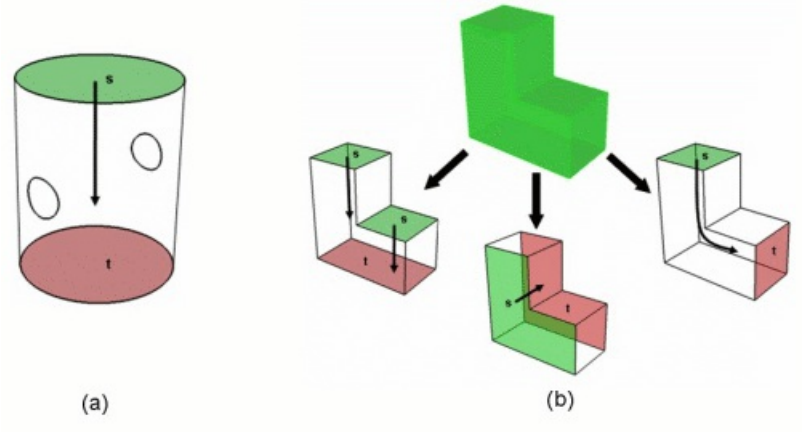


Figure 1. (a) ITEM displays the source and target of a geometry

that is nearly sweepable. The region is not currently sweepable due to circular imprints on the side of the cylinder. (b) Alternative feasible sweep directions are also computed.

Blend Surfaces

Blend surfaces are common in solid model meshing problems. A blend surface, also known as a fillet or chamfer, is problematic for sweeping algorithms which have trouble assigning vertex types on blend surfaces. While blend surfaces present a challenge for meshing applications, there are many tools within ITEM to help guide the user through possible solutions.

Diagnostic: Blend surfaces are detected by looping over the curves on a surface and examining the angles, surface normals, and curvature of curves and adjacent surfaces.

Solutions: The current solution to blend surfaces is to remove the surface and attempt to extend adjacent surfaces to fill in the gap. An example of blend surfaces that have been removed is shown below. This is useful for models which can be simplified without losing important topology.

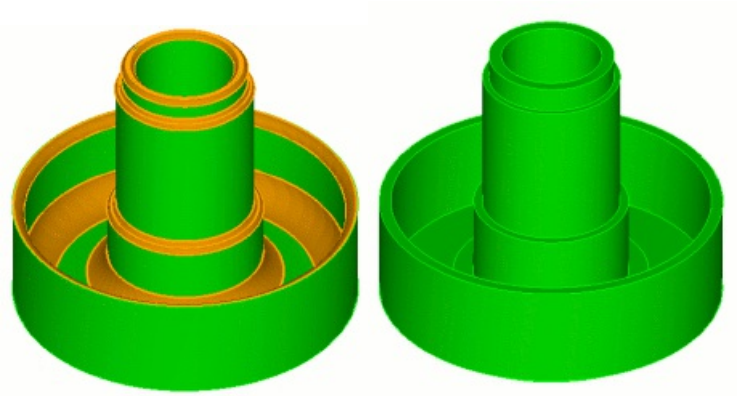


Figure 1. A volume which has been simplified by removing blend surfaces.

Clean Up the Geometry

Meshing packages have the challenge of dealing with a host of geometry problems. Many of these problems can be generalized as file translation issues. Typically, the geometry used in a meshing package has not been created there but in one of many CAD packages. Exporting these files out of CAD and into a neutral file format (IGES, STEP, SAT) accepted by the meshing software can introduce misrepresentations in the geometry. If the CAD and meshing packages do not support the same file formats, a second translation may be necessary, possibly introducing even more problems.

Another complication caused by file translation is that of tolerances. Some CAD packages see two points as coincident if they are within 1e-3 units, while others use 1e-6. If the meshing software's tolerance is finer than the CAD package's, this disparity in tolerance can cause subsequent geometry modification operations in the meshing package to inadvertently create sliver features, which tend to be difficult and tedious to deal with. This tolerance problem also causes misalignment issues between adjacent volumes of assemblies, hindering the sharing of coincident geometry in order to produce a conformal mesh.

Modeling errors caused by the user in the CAD package is another problem that the meshing package has to correct. In the CAD package, the user may not create the geometry correctly, causing some parts to overlap, or introduce small gaps between parts that should touch. Many times these problems are detected in the meshing package at a point when it is not feasible to simply go back into the CAD system and fix the problem, so the meshing package must be capable of correcting it.

Several approaches for addressing the geometry cleanup problem have been proposed in the literature, but they typically provide operations that are automatically applied to the geometry once one or more topology problems have been identified. While very effective in many cases, they generally lack the ability for the user to have control over the resolution of these CAD issues while still maintaining the option for automation. The ITEM environment provides tools to both diagnose these common issues and to provide a list of solutions from which the user may select that will correct the problems.

For the purposes of mesh generation, features in a solid model that should be carefully considered and addressed prior to meshing generally fit in one of four categories:

- Bad geometry representation As a result of translation errors between CAD representations, errors or differences in the way the geometry is interpreted may occur. Depending on the severity of the problem, sometimes a mesh can be generated even with a less-than perfect geometric representation, however, in most cases, these should be resolved before meshing.
- Small details in the model In some cases there exist small details in the geometry that, if meshed, would result in very small elements and a potentially huge element budget. Small curves and surfaces can sometimes result from details in the design solid model that may not be necessary for analysis or may even be a result of careless construction of the CAD model. In either case, it is important to remove or modify these features before meshing.
- Compatible topology for meshing scheme Several meshing algorithms, such as the structured, mapping and sweeping techniques require a specific configuration of vertices, curves and surfaces in order to operate. Operations to decompose the geometry into a meshable topology are often needed. Other unstructured techniques like paving, and tetrahedral meshing do not require decomposition.
- Conformal topology for assemblies Assemblies of parts are often required to have a conformal mesh across their interface, (i.e.

required to have a conformal mesh across their interface. (i.e. Shared nodes at a common boundary). The operations imprint and merge are often required to connect parts together so that when meshed, the representation will be a single continuous mesh.

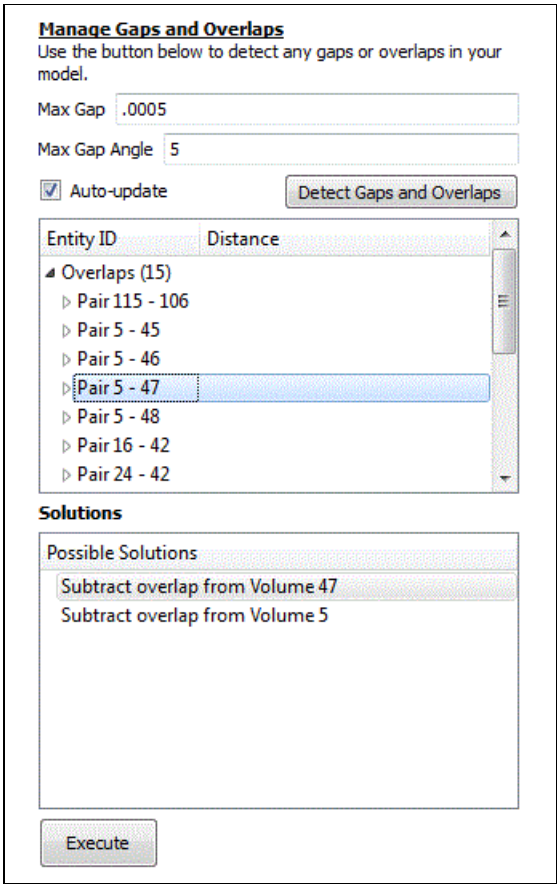
Being able to recognize when a problem exists and what operations to apply to resolve issues in each of the four categories described above, is indeed an art-form and requires significant experience to become proficient. ITEM will not take the place of an experienced user, but it is intended to offer the user help along the way by detecting potential problems and suggesting solutions they might consider.

Resolving Problems with Conformal Assemblies

Where more than a single geometric volume is to be modeled, a variety of common problems may arise that must be resolved prior to mesh generation. These are typically a result of misaligned volumes defined in the CAD package or problems arising from the imprint and merge operations in the meshing package. ITEM addresses some of the same problems by allowing the option for user interaction as well as full automation using the CAD geometry representation. The proposed environment utilizes two main diagnostics to detect potential problems: the misalignment check, and the overlapping surfaces check. Associated with both of these are solutions that are specific to the entity and from which the user may preview and select to resolve the problem.

Resolving Misaligned Volumes with Manage Gaps/Overlaps Tool

The Manage Gaps/Overlaps Tool within the geometry cleanup area of ITEM allows the user to quickly search an assembly for gaps and overlaps between assembly components. The search criteria for gaps is a tolerance specified by the user and defines the maximum gap between components to look for. A gap angle can also be specified which specifies how "parallel" two entities must be to be considered in the gap check. The overlap check simply asks Cubit to see if any of the volumes are overlapping and doesn't require a tolerance from the user. The results are displayed in a list of pairs of volumes. The user can right-click on these pairs and tell Cubit to draw the pair. A useful graphical depiction of the gap or overlap will be displayed. When the user clicks on a pair in the list a set of solutions for fixing the gap or overlap will also be displayed below in a separate list. The user can select a solution and click the "Execute" button to execute it. The gap solutions are either a surface "tweak" operation and the overlap solution can be either a tweak operation or a Boolean operation to remove the overlap. This tool provides a powerful way to quickly work through the assembly and fix gaps and overlaps.



Resolving Misaligned Volumes with Near Coincident Vertex Checks

The near coincident vertex check or misalignment check is used to diagnose possible misalignments between adjacent volumes. This diagnostic is performed prior to the imprint operation in order to reduce the sliver surfaces and other anomalies which can occur as a result of imprinting misaligned volumes. With this diagnostic, the distance between pairs of vertices on different volumes are measured and flagged when they are just beyond the merge tolerance. The merge tolerance, **T**, is the maximum distance at which the geometry kernel will consider the vertices the same entity. A secondary tolerance, $\mathbf{T_s}$, is defined where $\mathbf{T_s} > \mathbf{T}$ which is used for determining which pairs of vertices may also be considered for merging. Pairs of vertices whose distance, **d** is $\mathbf{T} < \mathbf{d} > \mathbf{T_s}$ are presented to the user, indicating areas in the model that may need to be realigned. The misalignment check should also detect small distances between vertices and curves on adjacent volumes.

When pairs of vertices are found that are slightly out of tolerance, the current solution is to move one of the surfaces containing one vertex of the pair to another surface containing the other vertex in the pair. Moving or extending a surface is known as tweaking.

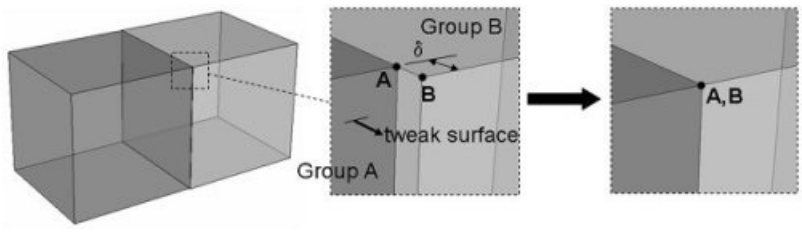


Figure 1. Example of a solution generated to correct misaligned volumes using the tweak operator

The result of this procedure will be a list of possible solutions that will be

presented to the users. They can then graphically preview the solutions and select the one that is most appropriate to correct the problem.

Correcting Merge Problems

The merge operation is usually performed immediately following imprinting and is also subject to occasional tolerance problems. In spite of correcting misalignments in the volume, the geometry kernel may still miss merging surfaces that may occupy the same space on adjacent volumes. If volumes in an assembly are not correctly merged, the subsequent meshes generated on the volumes will not be conformal. As a result, it is vital that all merging issues be resolved prior to meshing. The ITEM environment provides a diagnostic and several solutions for addressing these issues.

An overlapping surface check is performed to diagnose the failed sharing of topology between adjacent volumes. In contrast to the misalignment check, the check for overlapping surfaces is performed after the imprinting and merging operations. The overlapping surface check will measure the distance between surfaces on neighboring volumes to ensure that they are greater than the merge tolerance apart. Pairs of surfaces that failed to merge and that are closer than the merge tolerance are flagged and displayed to the user as potential problems.

A test for nonmanifold curves and vertices is also performed after imprinting and merging to find geometry that was not merged correctly. The test for nonmanifold curves is looking for curves that are merged, but do not share merged surfaces. Similarly, the test for nonmanifold vertices is looking for merged vertices that do not share any merged curves. Another test for floating volumes is performed to identify volumes that are not attached to any other entities.

If imprinting and merging has been performed and a subsequent overlapping surface check finds overlapping surface pairs, the user may be offered three different options for correcting the problem: force merge, tolerant imprint of vertex locations and tolerant imprint of curves.

If the topology for both surfaces in the pair is identical, the force merge operation can generally be utilized. The merge operation will remove one of the surface definitions in order to share a common surface between two adjacent volumes. Normally this is done only after topology and geometry have been determined to be identical, however the force merge will bypass the geometry criteria and perform the merge. Figure 2 shows a simple example where the bounding vertices are identical but the surface definitions are slightly different so that the merge operation fails. Force merge in this case would be an ideal choice.

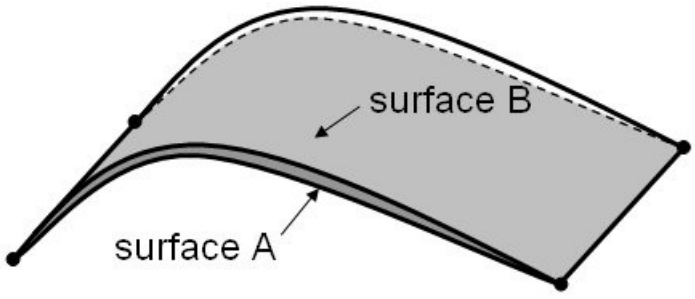


Figure 2. Example where the merge operation will fail, but force merge will be successful

The force merge operation is presented as a solution where a pair of overlapping surfaces are detected and if any of the following criteria are satisfied:

- All curves of both surfaces are merged
- All vertices between the two surfaces are merged and all the curves

are coincident to within 1% of their length or 0.005, whichever is larger
- All the curves of both surfaces are either merged or overlapping and a vertex of any curve of one surface that will imprint onto any other curve of the other surface cannot be identified
- At least one curve of one surface may be imprinted onto the other and if both surfaces have an equal number of curves and vertices, and the overlapping area between the 2 surfaces is more than 99% of the area of each surface. This situation generally prevents generating sliver surfaces
- At least one vertex of surface B may be imprinted onto surface A, and if both surfaces have equal number of curves and vertices, and the vertex(s) of surface B to imprint onto surface A lies too close to any vertices of surface A
- All the curves of both surfaces are either merged or overlapping and no vertices of any curve of surface A will imprint onto any other curve of surface B

Individual vertices may need to be imprinted in order to accomplish a successful merge. The solution of imprinting a position x,y,z onto surface A or B is presented to the user if the following criteria is met

- Curves between the two surfaces overlap within tolerance, and a vertex of curve A lies within tolerance to curve B and outside tolerance to any vertex of curve B. Tolerance is 0.5% of the length of the smaller of the 2 curves or the merge tolerance (0.0005), whichever is greater.

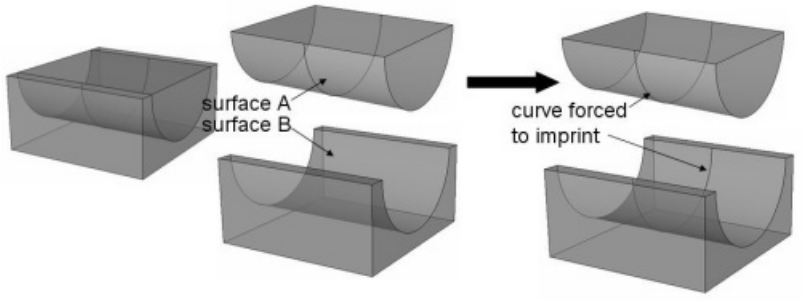


Figure 3. Curve on surface A was not imprinted on surface B due to tolerance mismatch. Solution is defined to detect and imprint the curve

In some cases one or more curves may not have been correctly imprinted onto an overlapping surface which may be preventing merging. This may again be the result of a tolerance mismatch in the CAD translation. If this situation is detected a tolerant imprint operation may be performed which will attempt to imprint the curve onto the adjacent volume. Figure 3 shows an example where a curve on surface A is forced to imprint onto surface B using tolerant imprint, because it did not imprint during normal imprinting. The solution of a curve of surface A to be imprinted onto surface B may be presented to the user if all 3 of the following conditions are satisfied:

- There are no positions to imprint onto the owning volume of either surface
- Curve of surface A is not overlapping another curve of surface B
- Curve of surface A passes tests to ensure that it is really ON surface B

Contact Surfaces

A contact surface is two surfaces which overlap, but are not merged. In a physical sense, this could represent two surfaces which come in contact with each other, as opposed to two surfaces which merely form a partition for meshing purposes. It is easy using the ITEM interface to identify and select contact surfaces in your model. Simply select surfaces in the graphics window and press the "Add" button on the ITEM interface. The contact surfaces will be shown in the window.

To remove a contact surface from the list, right click on the surface and select "Not a Contact Surface" from the context menu to remove that specific surface, or "Remove all contact surfaces" to remove all contact surfaces. Several other visualization tools are also available from the context menu including Zoom, Fly-in, Draw, List, Locate, etc.

Geometry Decomposition

Automatic decomposition has been researched and tools have been developed which have met with some limited success [Lu,99 , Staten,05]. Automatic decomposition requires complex feature detection and sub-division algorithms. The decomposition problem is at least on the same order of difficulty as the auto-hex meshing problem. Fully automatic methods for quality hexahedral meshing have been under research and development for many years [Blacker,93 , Folwell,98 , Price,95]. However, a method that can reliably generate hexahedral meshes for arbitrary volumes, without user intervention and that will build meshes of an equivalent quality to mapping and sweeping techniques, has yet to be realized. Although fully automatic techniques continue to progress [Staten,06], the objective of the proposed environment is to reduce the amount of user intervention required while utilizing the tried and true mapping and sweeping techniques as its underlying meshing engine.

Instead of trying to solve the all-hex meshing problem automatically, the ITEM approach to this problem is to maintain user interaction. The ITEM algorithms determine possible decompositions and suggest these to the user. The user can then make the decision as to whether a particular cut is actually useful. This process helps guide new users by demonstrating the types of decompositions that may be useful. It also aids experienced users by reducing the amount of time required to set up decomposition commands.

Diagnostics: The current diagnostic for determining whether a volume is mappable or sweepable is based upon the autoscheme tool described in [White,00]. Given a volume, the autoscheme tool will determine if the topology will admit a mapping, sub-mapping or sweeping meshing scheme. For volumes where a scheme cannot be adequately determined, a set of decomposition solutions are generated and presented to the user.

Solutions: The current algorithm for determining possible cut locations is based on the algorithm outlined in [Lu,99] and is described here for clarity:

- Find all curves that form a dihedral angle less than an input value (currently 135)
- Build a graph of these curves to determine connectivity
- Find all curves that form closed loops
- For each closed loop:
  - Find the surfaces that bound the closed loop
  - Save the surface
  - Remove the curves in the closed loop from the processing list
- For each remaining curve:
  - Find the open loops that terminates at a boundary
  - For each open loop:
    - Find the surfaces that bound the open loop
    - Save the surfaces
- For each saved surface:
  - Create an extension of the surface
  - Present the extended surface to the user as a possible decomposition location.

This relatively simple algorithm detects many cases that are useful in decomposing a volume. Future work will include determining symmetry, sweep, and cylindrical core decompositions. These additional decomposition options should increase the likelihood of properly decomposing a volume for hexahedral meshing.

Figure 1 shows an example scenario for using this tool. The simple model at the top is analyzed using the above algorithm. This results in

several different solutions being offered to the user, three of which are illustrated here. As each of the options is selected, the extended cutting surface is displayed providing rapid feedback to the user as to the utility of the given option. Note that all solutions may not result in a volume that is closer to being successfully hex-meshed. Instead the system relies on some user understanding of the topology required for sweeping.

Each time a decomposition solution is selected and performed, additional volumes may be added, which will in turn be analyzed by the autoscheme diagnostic tool. This interactive process continues until the volume is successfully decomposed into a set of volumes which are recognized as either mappable or sweepable.

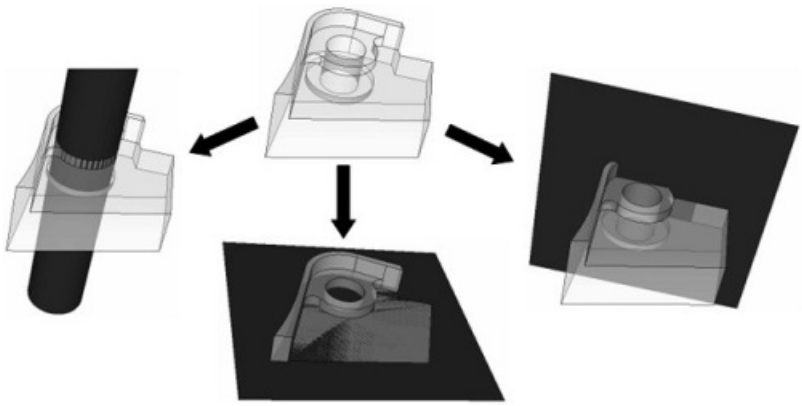


Figure 1. ITEM decomposition tool shows 3 of the several solutions generated that can be selected to decompose the model for hex meshing

Forced Sweepability

In some cases, decomposition alone is not sufficient to provide the necessary topology for sweeping. The forced sweepability capability attempts to force a model to have sweepable topology given a set of source and target surfaces. The source-target pairs may have been identified manually by the user, or defined as one the solutions from the sweep suggestion algorithm described above. All of the surfaces between source and target surfaces are referred to as linking surfaces. Linking surfaces must be mappable or submappable in order for the sweeping algorithm to be successful. There are various topology configurations that will prevent linking surfaces from being mappable or submappable.

Diagnostics: The first check that is made is for small curves. Small curves will not necessarily introduce topology that is not mappable or submappable but will often enforce unneeded mesh resolution and will often degrade mesh quality as the mesh size has to transition from small to large. Next, the interior angles of each surface are checked to see if they deviate far from 90 multiples. As the deviation from 90 multiples increases the mapping and submapping algorithms have a harder time classifying corners in the surface. If either of these checks identify potential problems they are flagged and potential solutions are generated.

Solutions: If linking surface problems are identified ITEM will analyze the surface and generate potential solutions for resolving the problem. Compositing the problem linking surface with one of its neighbors is a current solution that is provided. ITEM will look at the neighboring surfaces to decide which combination will be best. When remedying bad interior angles the new interior angles that would result after the composite are calculated in order to choose the composite that would produce the best interior angles. Another criterion that is considered is the dihedral angle between the composite candidates. Dihedral angles close to 180 are desirable. The suggested solutions are prioritized based on these criteria before being presented to the user. Figure 1 shows an example of a model before and after running the forced sweepability solutions. The top and bottom of the cylinder were chosen as the source and target surfaces respectively.

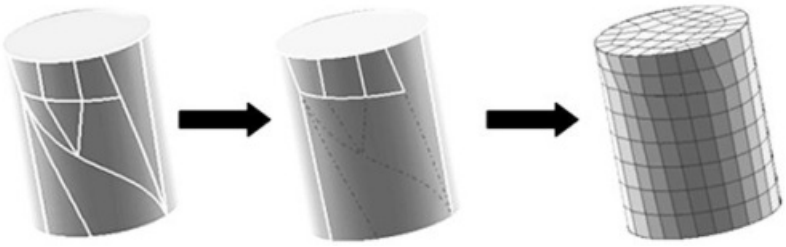


Figure 1. Non-submappable linking surface topology is composited out to force a sweepable volume topology

Bad geometry representation

As a result of translation errors between CAD representations, errors or differences in the way the geometry is interpreted may occur. Depending on the severity of the problem, sometimes a mesh can be generated even with a less-than perfect geometric representation, however, in most cases, these should be resolved before meshing.

Detecting Invalid Geometry

In most cases, bad or invalid topology or geometry definition comes from problems which arise in the CAD translation process. CUBIT's main geometry kernel, ACIS is used to represent the model if it has been imported using an IGES or STEP format. Translation to and from these neutral formats is frequently the cause of bad geometry. ITEM will use the geometry validation procedures built into the ACIS kernel to detect if there is any bad geometry and will list the entities that may be causing a problem.

Since the validation procedures are specific to ACIS, models that may have been imported from another native format such as Pro/E will not provide this diagnostic. Although this may seem like a severe limitation, importing native formats rarely have bad geometry, since no translation process is necessary.

It is good practice to always check your model for bad geometry before proceeding to other geometry or meshing operations. In some cases, if a webcut or meshing operation fails, the cause is an invalid geometric definition that has not been adequately healed. Resolving bad geometry problems up front, in most cases is essential to obtaining a mesh. On the other hand, if the location of the bad geometry in the model is such that it will not effect subsequent Boolean or decomposition operations, there may be a chance that completely resolving bad geometry is not necessary. Simply ignoring bad geometry that cannot be easily repaired with automatic procedures may be a reasonable solution, provided the user is aware of the potential limitations.

Resolving Invalid Geometry

To resolve invalid geometry, ITEM uses the heal procedure built into the ACIS geometry kernel. In almost all cases, this is a fully automatic procedure. Simply selecting the automatic repair button will make the appropriate adjustments to the geometry. This can be done one volume at a time by healing the owning volume, or by healing the full model all at once. If healing was successful, No problems detected should be displayed.

If auto repair does not successfully repair the geometry, you may want to try additional options available in Cubit for healing. See the Cubit documentation for a complete description of additional healing options.

Determining an Appropriate Merge Tolerance

Determining the appropriate merge tolerance for a model can be essential for creating conformal meshes on some models. The merge tolerance is a value that identifies at which distance different entities can be considered the same entity. Many entities will fail to merge because of widespread geometry tolerance or alignment problems that are either too difficult, time-consuming or even impossible to resolve. Specifying a merge tolerance that is larger than these small discrepancies allows the user to account for geometry that is misaligned. But specifying a merge tolerance that is too large can combine features the user wishes to keep, and possibly corrupt the model. The ideal merge tolerance should be smaller than the smallest feature, but larger than the biggest gap or misalignment that cannot be resolved. Since it is not always a simple task to determine either of these features, the ITEM workflow provides a diagnostic tool designed to guide the user to find the small misalignments that may lead to merge problems. It then presents possible solutions to fix these problems, or the ability to change the merge tolerance to ignore them.

Opening the Merge Tolerance Panel

To open the merge tolerance tool from the ITEM Wizard, click on Prepare Geometry->Connect Volumes->Imprint and Merge. Then click on the button with three dots next to the **Merge Tolerance** input field.

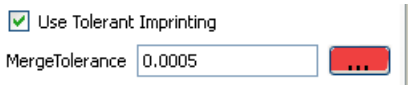


Figure 1. How to open the merge tolerance panel

The merge tolerance panel is shown in the following image.

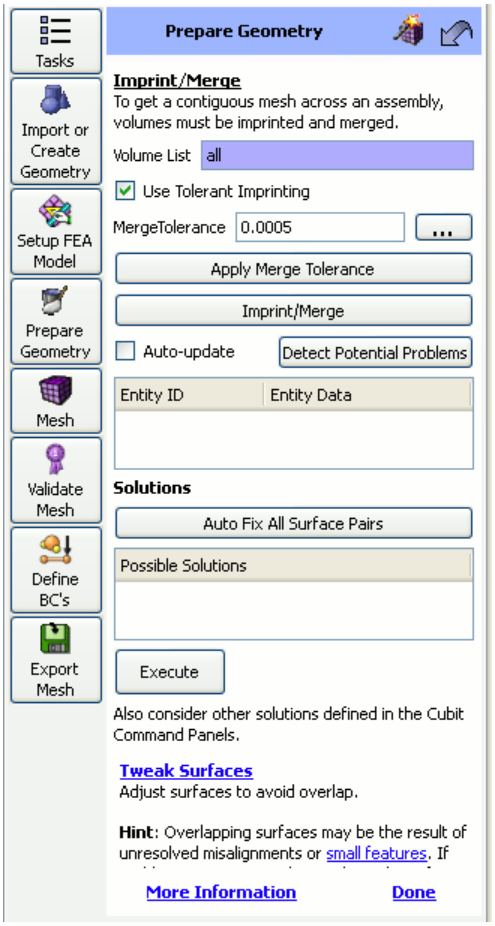


Figure 2. The Merge Tolerance Diagnostic Panel

Estimating Merge Tolerance with Small Feature Size

Since the merge tolerance must be smaller than the smallest feature in the mesh, the best place to start is by finding the smallest feature and using that value to create an estimate for the merge tolerance. To find the smallest feature, click on the small button with three dots next to the input box for Small Features.

Note: The small feature checks will not find misalignments between different volumes- it will only list vertex-vertex pairs and vertex-curve pairs on the same volume. The small feature size is used on the merge tolerance panel to find an initial estimate for the merge tolerance.

After determining the smallest feature size, click on the *Estimate Merge Tolerance* button to come up with a rough estimate for the merge tolerance. It is important to note that this is only an estimate. After an initial estimate is made, it can be fine tuned using the Fine Tune Merge Tolerance tool.

Fine Tuning the Merge Tolerance

In the fine tune merge tolerance area, the user may search for vertex-vertex, vertex-curve, and vertex-surface pairs that are within user-specified ranges. This includes checks between entities on different volumes. This allows the user to determine if the merge tolerance he/she has determined will capture all of the merges he/she intends. The user can check/uncheck which pairs to search for and what range to look in. The results from the search will show up in the window below and the user can select the results, right click on it, and choose *Draw with Volumes* to zoom into that pair of features. For vertex-vertex pairs there may be tweak solutions presented to the user in the list box below for

fixing the problems.

Setting the Merge Tolerance

The Apply button next to Estimated Merge Tolerance edit field is used to take the estimated merge tolerance and use it to set the merge tolerance in CUBIT by issuing the Merge Tolerance <val> command.

Building a Sweepable Topology

The hex meshing problem presents a number of additional challenges to the user that tetrahedral meshing does not. Where a good quality tetrahedral mesh can generally be created once small features and imprint/merge problems have been addressed, the hexahedral meshing problem poses additional topology constraints which must be met.

Although progress has been made in automating the hex meshing process, the most robust meshing algorithms still rely on geometric primitives. Mapping [Cook, 82] and sub-mapping [Whiteley, 96] algorithms rely on parametric cubes and sweeping[Knupp, 98; Scott, 05] relies on extrusions. Most real world geometries do not automatically fit into one of these categories so the topology must be changed to match the criteria for one of these meshing schemes. ITEM addresses the hex meshing topology problem through four primary diagnostic and solution mechanisms.

1. Detecting blend surfaces
2. Detecting and suggesting decomposition operations
3. Recognizing nearly sweepable topologies and suggesting source-target pairs
4. Detecting and compositing surfaces to force a sweep topology

Small details in the model

The small feature removal area of ITEM focuses on identifying and removing small features in the model that will either inhibit meshing or force excessive mesh resolution near the small feature. Small features may result from translating models from one format to another or may be intentional design features. Regardless of the origin small features must often be removed in order to generate a high quality mesh.

ITEM will recognize small features that fall in four classifications:

1. small curves
2. small surfaces
3. narrow surfaces
4. surfaces with narrow regions

These operations may involve either real, virtual or a combination of both types of operations to remove these features. A virtual operation is one in which does not modify the CAD model, but rather modifies an overlay topology on the original CAD model. Real operations, on the other hand directly modify the CAD model. Where real operations are provided by the solid modeling kernel upon which CUBIT is built, virtual operations are provided by CUBIT's CGM (Tautges, 00) module and are implemented independently of the solid modeling kernel. The following describes the diagnostics for finding each of the four classifications of small features and the methods for removing them.

Small Curves

Diagnostic: Small curves are found by simply comparing each curve length in the model to a user-specified characteristic small curve size. A default epsilon (ε) is automatically calculated as 10 percent of the user specified mesh size, but can be overridden by the user.

Solutions: ITEM provides three different solutions for eliminating small curves from the model. The first solution uses a virtual operation to composite surfaces. Two surfaces near the small curve can often be composited together to eliminate the small curve as shown in Figure 1(a).

The second solution for eliminating small curves is the collapse curve operation. This operation combines partitioning and compositing of surfaces near the small curve to generate a topology that is similar to pinching the two ends of the curve together into a single point. The partitioning can be done either as a real or virtual operation. Figure 1(b) illustrates the collapse curve operation.

The third solution for eliminating small curves is the remove topology operation. This operation can be thought of as cutting out an area around the small curve and then reconstructing the surfaces and curves in the cut-out region so that the small curves no longer exist. (Clark, 07) provides a detailed description of the remove topology operation. This operation has more impact on the actual geometry of the model because it redefines surfaces and curves in the vicinity of a small curve. The reconstruction of curves and surfaces is done using real operations followed by composites to remove extra topology introduced during the operation. Figure 1(c) shows the results using the remove topology operation.

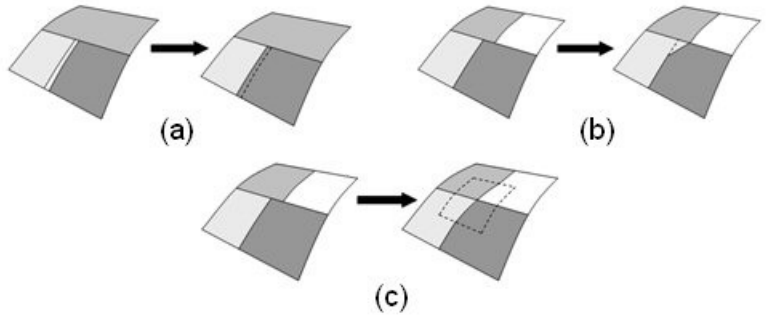


Figure 1. Three operators used for removing small curves (a) composite; (b) collapse curve; (c) remove topology

Small and Narrow Surfaces

ITEM also addresses the problem of small and narrow surfaces. Both are dealt with in a similar manner and are described here.

Diagnostic: Small surfaces are found by comparing the surface area with a characteristic *small area*. The characteristic small area is defined simply as the characteristic small curve length squared or ε^2.

Narrow surfaces are distinguished from *surfaces with narrow regions* by the characteristic that the latter can be split such that the narrow region is separated from the rest of the surface. Narrow surfaces are themselves a narrow region and no further splits can be done to separate the narrow region. Figure 2 shows examples of each. ITEM provides the option to split off the narrow regions, subdividing the surface so the narrow surfaces can be dealt with independently.

Narrow regions/surfaces are also recognized using the characteristic value of ε. The distance, d_i from the endpoints of each curve in the surface to the other curves in the surface are computed and compared to ε. When $d_i<\varepsilon$ other points on the curve are sampled to identify the beginning and end of the narrow region. If the narrow region encompasses the entire surface, the surface is classified as a narrow surface. If the region contains only a portion of the surface, it is classified as a surface with a narrow region.

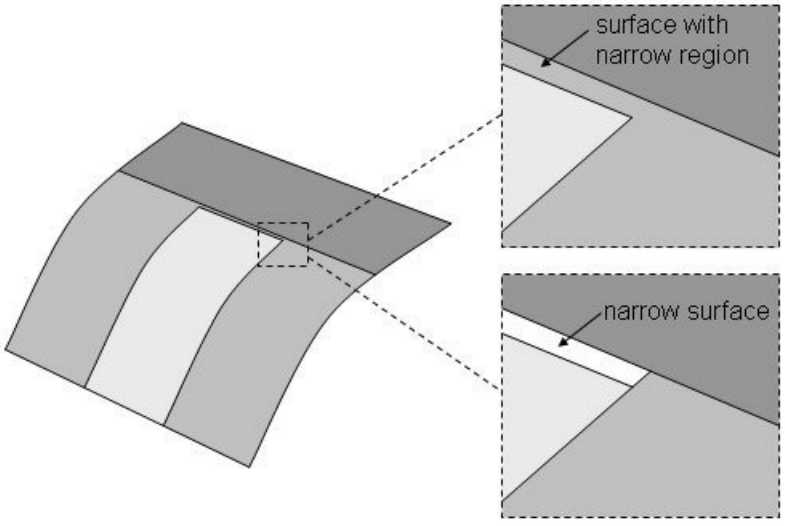


Figure 2. Two cases illustrating the difference between surfaces with narrow regions and narrow surfaces

Solutions: ITEM provides four different solutions for eliminating small and narrow surfaces from the model. The first solution uses the regularize operation. Regularize is a real operation provided by the solid modeling kernel that removes unnecessary/redundant topology in the

model. In many cases a small/narrow surface's definition may be the same as a surface next to it and therefore the curve between them is not necessary and can be regularized out. An example of regularizing a small/narrow surface out is shown in Figure 3.

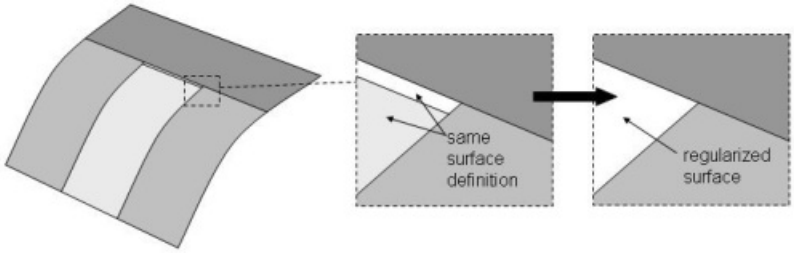


Figure 3. When the small surface's underlying geometric definition is the same as a neighbor the curve between them can be regularized out.

The second solution for removing small/narrow surfaces uses the remove operation. Remove is also a real operation provided by the solid modeling kernel. However, it differs from regularize in that it doesn't require the neighboring surface(s) to have the same geometric definition. Instead the remove operation removes the specified surface from the model and then attempts to extend and intersect adjacent surfaces to close the volume. An example of using the remove solution is shown in Figure 4.

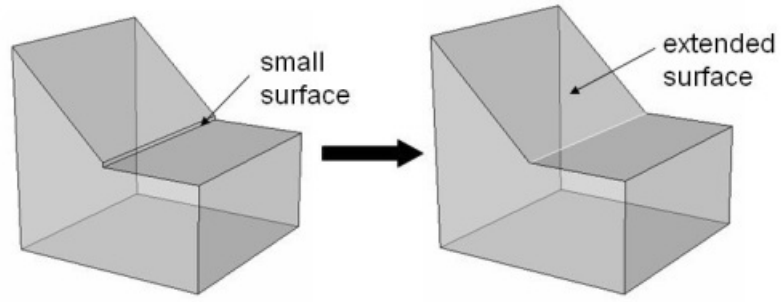


Figure 4. The remove operation extends an adjacent surface to remove a small surface

The third solution for removing small/narrow surfaces uses the virtual composite operation to composite the small surface with one of its neighbors. This is very similar to the use of composites for removing small curves. An example is shown in Figure 5.

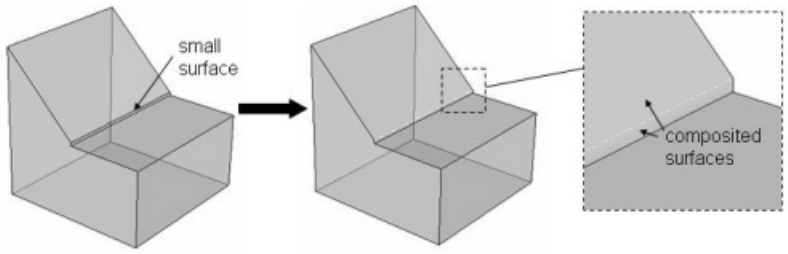


Figure 5. Composite solution for removing a narrow surface

The final solution for removing small/narrow surfaces uses the remove topology operation (Clark, 07). The remove topology operation behaves the same as when used for removing small curves in that it cuts out the area of the model around the small/narrow surface and replaces it with a simplified topology. In the case of a small surface where all of the curves on the surface are smaller than the characteristic small curve length, the small surface is replaced by a single vertex. In the case of a narrow surface where the surface is longer than the characteristic small curve length in one of its directions, the surface is replaced with a curve. The remove topology operation can be thought of as a local dimensional

reduction to simplify the topology. The remove topology operation can also be used to remove networks of small/narrow surfaces in a similar fashion. Examples of using the remove topology solution to remove small/narrow surfaces are shown in Figures 6 and Figure 7.

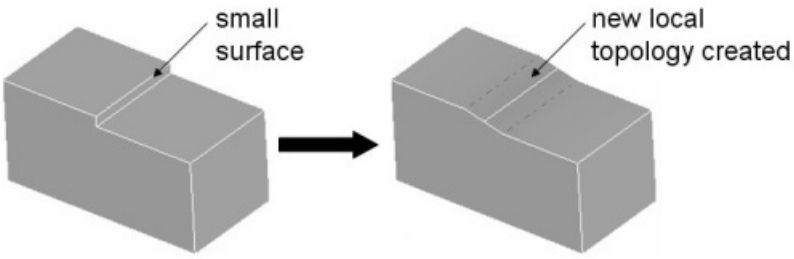


Figure 6. Remove topology solution for removing a narrow surface

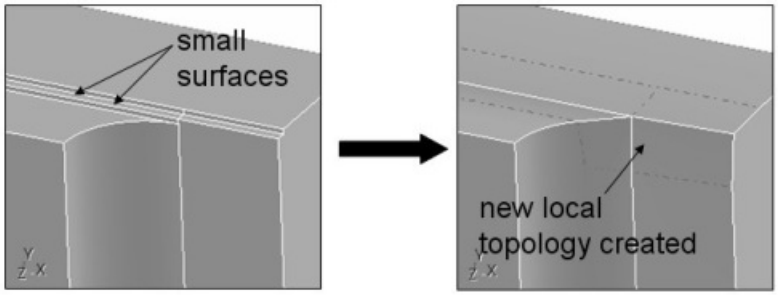


Figure 7.Remove topology solution for removing a network of narrow surfaces

Determining the Small Feature Size

The smallest feature size is a value that represents the size of the smallest detail in the volume that the user wants to include in the final mesh. Any details that are smaller than this size should be removed from the model before completing the other steps of the meshing process. Small details can result from a variety of different reasons. Sometimes the model contains excessive detail that the user does not need. Other times, small features such as extra curves are created during import to account for a mismatching topology. Still other times, the small features are the result of webcutting or other decomposition methods. Ideally there should be a minimum threshold at which the user decides to keep all features above the given size, and remove the rest. The smallest feature size is used for other diagnostic tools, so selecting an appropriate feature size is important for other steps in the mesh generation process.

After the **Find Small Features** button is pressed, Cubit lists the 10 closests vertex-vertex and vertex-curve pairs. The pairs are listed in the display window from smallest to largest. To see more pairs, change the search parameter in the input box. To visualize each pair, the user can right click on a feature and select the *Draw Pair with Volumes* option. After determining the smallest feature size the user can enter it in the edit field at the bottom of the panel and it will be used in later calculations. The user can also right click on one of the pairs in the list and choose *Use as smallest feature* to populate the edit field at the bottom of the panel.

Why doesn’t the list include small gaps between volumes?

The smallest feature check is only searching over vertex-vertex and vertex-curve pairs in the same volume. Small gaps and misalignments are not included in this list. The purpose of the small feature diagnostic panel is to search for features that need to be removed prior to meshing. A feature is an entity such as a small curve or sliver surface that exists on a single volume which must be resolved by the mesh. A gap or misalignment is two entities that should be coincident, but are not, due to translation or other problems. Gaps and misalignments may not hinder mesh generation on a given volume, but they do prevent proper imprinting and merging.

The imprint/merge, merge tolerance, and overlapping volume panels contain diagnostics that check for misalignment problems. The purpose of those diagnostics is to enable imprinting and merging of a volume with small misalignments.

Note: The smallest feature size is used as a metric on the merge tolerance page, but it is only used to get an initial estimate for the merge tolerance. Small feature size and merge tolerance represent different metrics, and should not be confused.

In Figure 1, the small feature size diagnostic finds small features with lengths of 0.707, 0.15 and 0.25. The user may decide that the smallest feature he or she wishes to keep is the one at the 0.25 size. If he sets the small feature size to 0.25, the other features will be flagged as small curves and surfaces on the Small Features page. They can then be removed using tweaking and other geometry clean-up commands. If he sets the small feature size to 0.707, none of the features will be flagged as small features.

In addition to the features shown, this model contains two vertices that

are slightly misaligned due to geometry translation problems. The nearly coincident vertices are not listed on the small features list because the vertices lie on different volumes. To find these near coincident vertices, the user would use the merge tolerance panel.

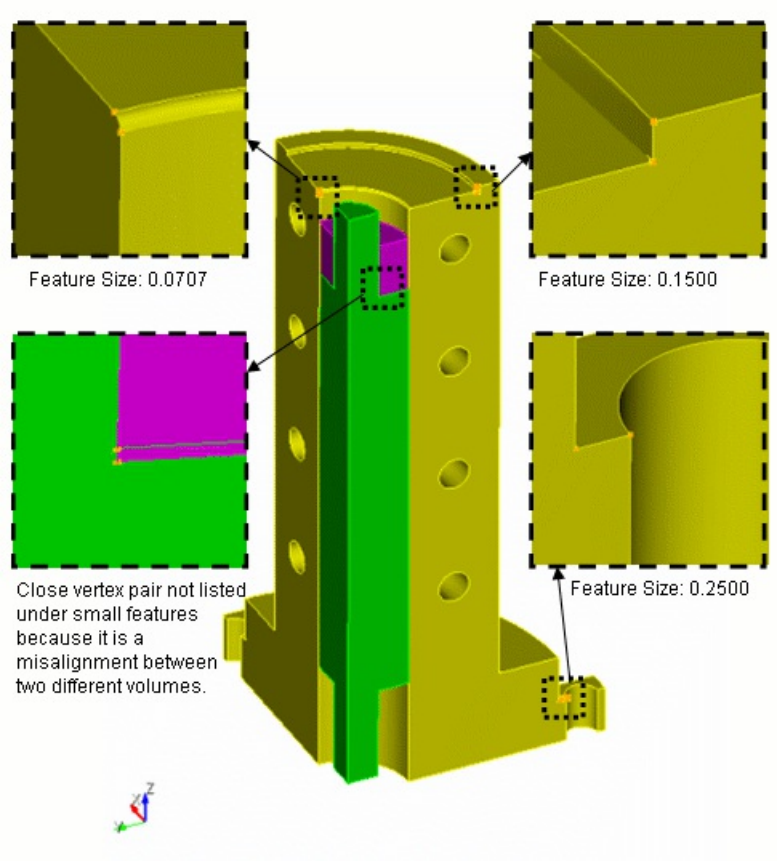


Figure 1. Small Features and Overlap on a Model

Appendix

- Alpha Commands
- Available Colors
- Element Numbering
- FullHex vs. NodeHex Representation
- APREPRO
- Python Cubit Enhancement Scripts
- Cubit Python Interface
- Navigation XML Files
- FASTQ
- Periodic Space-filling Models (Tile)
- Generating Meshes for Adaptive Topological Optimization (ATO)
- References

Alpha Commands

CUBIT has several functions that are currently in development and are considered "Alpha" features. These features can be can be accessed or hidden within Cubit by typing the following command:

> **Set Developer Commands {On|OFF}**

The commands that are currently developer commands are:

- Automatic Detail Suppression
- Deleting Mesh Elements
- Feature Size
- Optimize Jacobian
- Mesh Cutting
- Mesh Grafting
- Randomize Smoothing
- Refine Mesh Boundary
- Super Sizing Function
- Triangle Mesh Coarsening
- Transition
- Importing MBG Files
- Exporting MBG Files
- Remove Tiny Edge Length
- Exporting SGM Files

Creating ACIS Geometry From Mesh 

Note: These features are under development. The command to enable or disable features under development is:

Set Developer Commands {On|OFF}

Using the **Acis** options (in red) in the commands below will produce ACIS geometry instead of mesh-based geometry. ACIS geometry is generally more desirable than mesh-based geometry because it can be modified easily.

Importing a Mesh

```
Import Mesh Geometry '<exodusII_filename>'
[Block <id_range>|ALL] [Unique Genesis IDs]
[Start_id <id>] [Use [NODESET|no_nodeset]
[SIDESET|no_sideset] [Feature_Angle <angle>]
[LINEAR|Gradient|Quadratic|Spline|Acis]
[Deformed {Time <time>|Step <step>|Last}
[Scale <value>] ] [MERGE|No_Merge]
[Export_facets <1|2|3>] [Merge_nodes
<tolerance>]
```

This command tries to associate the mesh to the ACIS geometry that is created. If the association fails, the mesh ends up as free mesh. For more information on this command see: Importing Exodus II Files

Existing Mesh - Create Mesh Geometry

```
 Create Mesh Geometry {Hex|Tet|Face|Tri|Block}
<range> [Feature_Angle <angle=135>] [Acis]
[Keep]
```

This command tries to associate the mesh to the ACIS geometry that is created. If the association fails, the mesh ends up as free mesh. For more information on this command see: Free Meshes

Existing Mesh - Create Geometry

```
  Create Geometry {Hex|Tet|Face|Tri} <id_range>
```

These two commands do not use blocks, sidesets, nodesets or a user-specified dihedral angle to create vertices, curves, and surfaces. These commands use a proprietary third-party routine to create geometry. They also do not associate the mesh to the ACIS geometry that is created.

Deleting Mesh Elements 

Element deletion for owned geometry is no longer available unless the developer flag is turned on. Element deletion is still available without the developer flag for free meshes. The command to enable or disable features under development is:

```
Set Developer Commands {On|OFF}
```

The following forms of the delete commands operate on meshed entities only. They allow low-level editing of meshes to make minor corrections to a mostly correct mesh. They are not designed for major modifications to existing meshes. Because Cubit's display routines were not designed with these type of operations in mind, these commands may cause the current display of the affected entities to take an unexpected form. An appropriate drawing command can used to return the display to the desired view.

When deleting elements, the default behavior will be that the child mesh entities will be deleted when they become orphaned. For example, when a hex is deleted, if its faces, edges and vertices are no longer used by adjacent hex elements, then they will also be deleted. The **no_propagate** option will leave any child mesh entities regardless if they become orphaned.

The delete command removes one or more mesh entities from an existing mesh. Additional mesh entities may be deleted as well depending on the particular form of the command. Exactly which entities are removed is explained in the following descriptions.

```
Delete {Hex|Tet} <range> [No_Propagate]
```

Deletes the specified hexes or tets. All associated tris, faces, edges, and nodes are also deleted unless the *no_propagate* option is given.

```
Delete Wedge <range>
```

Deletes the specified wedges. No other mesh entities are affected.

```
Delete {Face|Tri} <range> [No_Propagate]
```

Deletes the specified faces or tris. For faces, all hexes that contain the face are also deleted. For tris, all tets that contain the tri are also deleted. All associated edges and nodes are also deleted unless the *no_propagate* option is given.

```
Delete Edge <range> [No_Propagate]
```

Deletes the specified edges. Any associated tris, faces, hexes, and tets are also deleted. Any associated nodes are also deleted unless the *no_propagate* option is given.

```
Delete Node <range>
```

Deletes the specified nodes. Any associated edges, tris, faces, hexes, and tets are also deleted.

FeatureSize

Note: This feature is under development. The command to enable or disable features under development is:

```
Set Developer Commands {On|OFF}
```

Applies to: Curves

Summary: Meshes a curve based on its proximity to nearby geometry and size of nearby geometric features. This is an alpha feature and should be used with caution.

Syntax:

```
Curve <range> Scheme Featuresize
```

Related Commands:

```
Curve <range> Density <density_factor>
```

Discussion:

The user may also automatically bias the mesh from small elements near complicated geometry to large elements near expanses of simple geometry. Meshing a curve with scheme featuresize places nodes roughly proportional to the distance from the node to a piece of geometry that is foreign to the curve. Foreign means that the geometric entity doesn't contain the curve, or any of its vertices (i.e. the entity's intersection with the curve is empty). It is known that featuresize is a continuous function that varies slowly. Featuresize meshing is very automatic and integrated with interval matching. Featuresize meshing works well with paving, and in some cases with structured surface-meshing schemes (map, submap) as well.

If desired, the user may specify the exact or goal number of intervals with a size or interval command, and then the featuresize function will be used to space the nodes.

The featuresize function may also be scaled by the user to produce a finer or coarser mesh using the **density** command as follows:

```
Curve <range> Density <density_factor>
```

The default scaling factor or **density** is 1. Higher densities also reduce the transition rate of the node spacing. A density of 2 usually gives a good quality mesh. A density below about 0.5 could produce rapid transitions and poor mesh quality. The following shows an example of different density values when using the featuresize scheme.



Importing Abaqus Files

Note: This feature is under development. The command to enable or disable features under development is:

```
Set developer commands {on|OFF}
```

The command to import a mesh from an Abaqus format file is:

```
Import Abaqus [Mesh Geometry]
'<input_filename>' [Feature Angle <angle>]
```

For a description of importing mesh geometry see Importing Exodus II Files.

Importing Meshed Based Geometry Files (MBG) 

Note: This feature is under development. The command to enable or disable features under development is:

```
Set Developer Commands {On|OFF}
```

CUBIT provides the capability to import a model composed of mesh based geometry. The command to import meshed based geometry is:

```
Import mbg "<filename>"
```

MBG is created in Cubit when one meshes a volume or imputs the mesh from a previously meshed volume with the **import mesh geometry** command. Optionaly there one may create geometry with the "set dev on" option.

In order to create, import and export MBG one needs to set the geometry engine to facet with the following command "set geom eng facet".

The following commands create a brick and export and import it as a MBG file: set geometry engine facet set dev on brick x 10 export mbg "brick.mbg" overwrite reset import mbg "brick.mbg"

set geometry engine facet
set dev on
brick x 10
export mbg "brick.mbg" overwrite
reset
import mbg "brick.mbg"

Exporting Meshed Based Geometry Files (MBG) 

Note: This feature is under development. The command to enable or disable features under development is:

```
Set Developer Commands {On|OFF}
```

CUBIT provides the capability to import a model composed of mesh based geometry. The command to import meshed based geometry is:

Export mbg "<filename>"

MBG is created in Cubit when one meshes a volume or imputs the mesh from a previously meshed volume with the import mesh geometry command. Optionaly one may create geometry with the "set dev on" option.

In order to create, import and export MBG one needs to set the geometry engine to facet with the following command "set geom eng facet".

The following commands create a brick and export and import it as a MBG file:

set geometry engine facet
set dev on
brick x 10
export mbg "brick.mbg" overwrite
reset
import mbg "brick.mbg"

Mesh Cutting 

Note: This feature is under development. The command to enable or disable features under development is:

Set Developer Commands {On|OFF}

The term "mesh cutting" refers to modifying an existing mesh by moving nodes to a cutting entity and modifying the connectivity of the mesh so that the original mesh fits a new geometry. The behavior of mesh cutting is intended to be similar to web cutting in that the process results in a decomposition of the original geometry. The difference is that the decomposition is performed on meshed geometry and results in the creation of virtual geometry partitions. The underlying acis body remains unchanged. The user has the option to determine what is partitioned during mesh cutting: the volume, the surfaces only, or nothing.

The current scope of mesh cutting is limited to cutting hex meshed volumes with planes and extended surfaces. These cutting entities are also limited in that mesh cutting will not work if they pass through a vertex at the end of more than two curves. Mesh cutting does not work on tet meshes or surface meshes.

The steps of mesh cutting include:

- **Create a starting mesh**. This mesh is typically simpler than the desired final mesh and can be created with sweeping, mapping, or some other available meshing algorithm. Currently, the starting mesh must be a single volume: mesh cutting does not handle merged volumes or assemblies.
- **Create a cutting entity** that can be used to capture the new detail in the mesh. Currently, mesh cutting works with planes or sheets extended from surfaces. It is important to note that if an extended surface is used, mesh cutting will not capture any geometric features (curves or vertices) of the surface.
- **Issue the command** to cut the mesh. The meshcut commands are similar in syntax and behavior to the webcut commands.

The following entities with the associated commands are available for mesh cutting:

Coordinate Plane

A coordinate plane can be used to cut the model, and can optionally be offset a positive or negative distance from its position at the origin.

```
Meshcut Volume <range> Plane
{xplane|yplane|zplane} [offset <dist>]
```

The planar surface to be used for mesh cutting can also be previewed using the Draw Plane command.

Planar Surface

An existing planar surface can also be used to cut the model.

```
Meshcut Volume <range> Plane Surface
<surface_id>
```

The planar surface to be used for mesh cutting can also be previewed using the Draw Plane command.

Plane from 3 points

Any arbitrary planar surface can be used by specifying three nodes that define the plane.

```
Meshcut Volume <range> Plane Node <3_node_ids>
```

Extended Surface

An extended surface or "sheet" can also be used for mesh cutting. In this case, the sheet is not restricted to be planar and will be extended in all directions possible. When cutting with an extended surface mesh cutting will ignore all curves and vertices of the surface. Also, the resolution of the mesh will determine how well curved surfaces are captured with meshcutting. A surface with high curvature will not be captured accurately with a coarse mesh. Note that some spline surfaces are limited in extent and may not give an expected result from mesh cutting.

```
Meshcut Volume <range> Sheet [Extended From]
Surface <surface_id>
```

Note: When cutting with surfaces extended from composite surfaces the default underlying surface approximation may result in a poor final mesh for mesh cutting. This problem can be fixed using the following command:

```
Composite closest_pt surface <id> gme
```

See the discussion on composite geometry for a more detailed description of this command.

Meshcut Options

The following options can be used with all the meshcut commands:

[PARTITION VOLUME|partition surface|no_partition]: By default, mesh cutting will create virtual partitions of the volume being cut to match the cutting entity. This option allows mesh cutting to also create only the surface partitions or create no partitions for the volume or surfaces.

[no_refine]: This option tells mesh cutting not to refine the mesh around the cutting entity.

[no_smooth]: This option tells mesh cutting not to perform the final smoothing step after the cut has been made.

Meshcutting Scope

The following is a list of the current scope and limitations of meshcutting.

- Meshcutting only works on hex meshes.
- Meshcutting only works for single volumes. It currently does not handle assembly meshes.
- Currently, only planes and extended surfaces can be used as the cutting entity.
- Curves and vertices on the cutting entity will not be captured in the mesh.
- Meshcutting will not work if the cutting entity passes through a meshed vertex that is at the end of more than two curves.
- The resolution of the mesh determines how well a non-planar cutting entity will be captured in the resulting mesh. Small features and high curvature will not be captured by a coarse mesh.
- Spline surfaces are limited in extent and may not give expected results if used as an extended cutting surface.

Meshcutting Example

The figures below show an example of mesh cutting. Figure 1 shows the body that will be meshed. This body is a brick with intersecting through-

holes. The steps to create a mesh for this body are listed below.

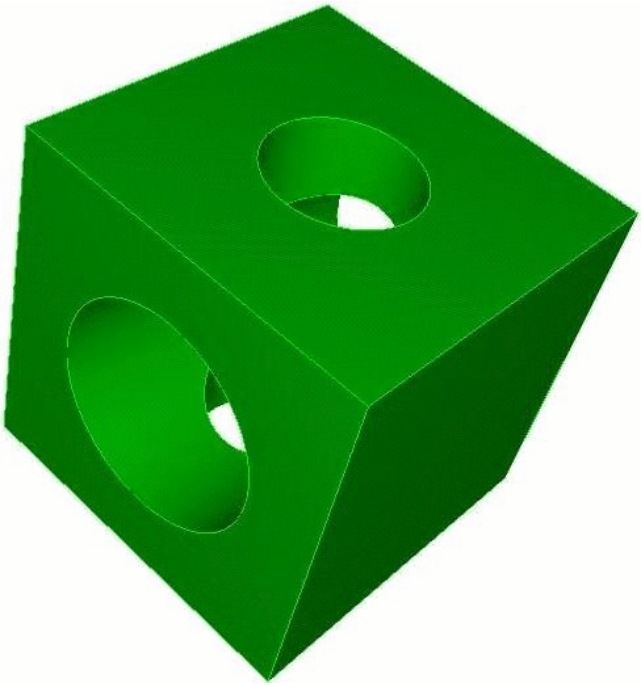


Figure 1: The original, unmeshed body

Step 1: Create a starting mesh. Figure 2 below shows the starting mesh for this problem. The commands for this mesh are:

```
cubit> reset
cubit> set dev on
cubit> create brick x 10
cubit> create cylinder radius 3 z 15
cubit> subtract 2 from 1
cubit> volume 1 scheme sweep
cubit> volume 1 size .75
cubit> webcut volume 1 with plane xplane offset 0
cubit> merge all
cubit> mesh volume 1 3
```

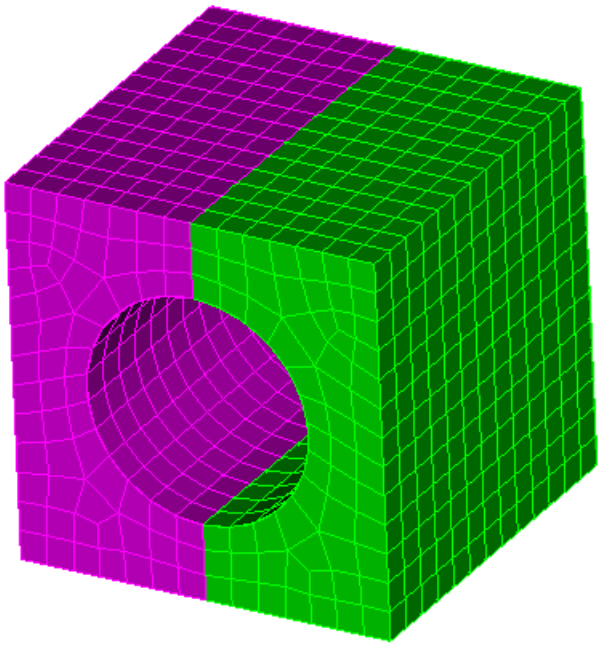


Figure 2: The starting mesh

Step 2: Create a cutting entity. Figure 3 shows the volume that will be used to cut the mesh. The commands are:

```
cubit> create cylinder radius 2 z 15
cubit> rotate volume 4 about x angle 90
```

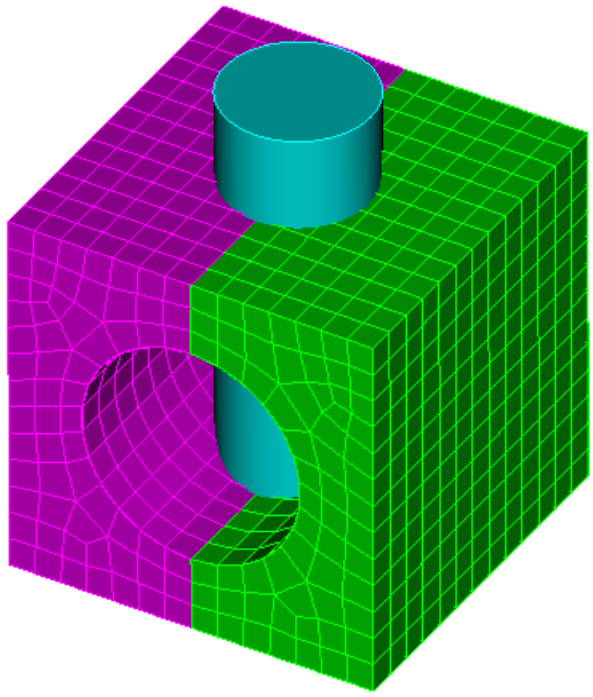


Figure 3: The starting mesh and cutting entity

Step 3: Cut the mesh. Figure 4 shows the new mesh after the original mesh has been cut. Also recommend smoothing the mesh:

```
cubit> meshcut vol 1 3 sheet surface 27
cubit> surf in vol 1 3 smooth sch condition number
beta 2 cpu 0.5
cubit> smooth surface in volume 1 3
cubit> vol 1 3 smooth scheme condition number beta 2
cpu 0.5
cubit> smooth volume 1 3
cubit> draw volume 1 3
```

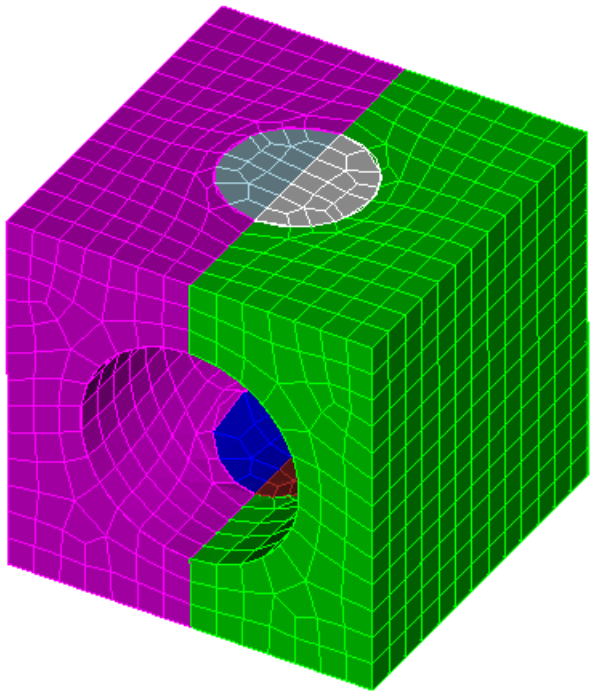


Figure 4: The mesh after meshcutting

Step 4: Final step. Figure 5 shows the quality of the final mesh. The command is:

```
cubit> quality volume 1 3 scaled jacobian global
draw mesh
```

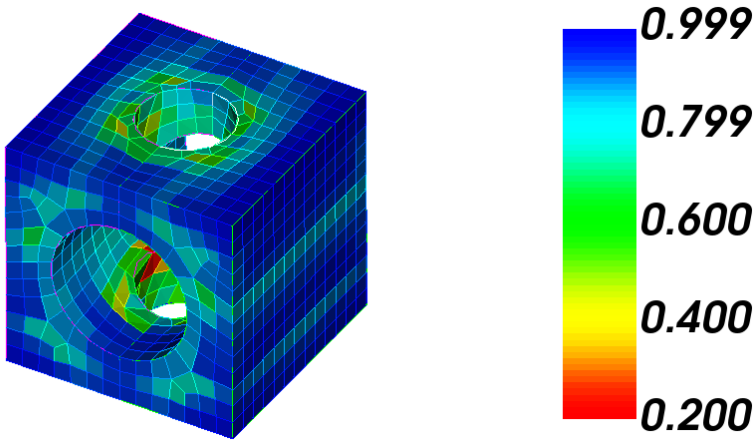


Figure 5: Quality of final mesh

Mesh Grafting

Note: This feature is under development. The command to enable or disable features under development is:

```
Set Developer Commands {On|OFF}
```

Grafting is used to merge a meshed surface with a dissimilar unmeshed surface. In the process, the location of the nodes on the meshed surface will be adjusted to fit to the bounding curves of the unmeshed surface and the connectivity of the original mesh may be changed to improve the final quality of the mesh. This allows an unmeshed volume to be attached--or grafted--onto a meshed volume. Grafting is particularly useful for models that have intersecting sweep directions (see example below).

The command syntax for grafting is:

```
Graft {Surface <range> | Volume <id>} onto
Volume <id> [no_refine] [no_smooth]
```

The Graft command will check that the second volume is meshed. It then searches for surfaces on the second volume that overlap with the other volume or range of surfaces that is specified. If overlapping surfaces are found the mesh will then be adjusted on the second volume and after any needed imprinting is done the overlapping surfaces will be merged together.

Grafting Options

[no_refine]: This option tells grafting not to modify the connectivity of the original mesh. The mesh is still adjusted to fit the boundary of the branch surface but no new elements are added.

[no_smooth]: This option tells grafting not to perform the final smoothing of the modified surface or volume mesh.

Grafting Scope

The following is a list describing the current scope and limitations of grafting:

- Grafting only works on volumes meshed with hex elements.
- The unmeshed branch surface cannot have any point outside the boundary of the meshed trunk surface.
- Grafting may have difficulty with branch surfaces that are very thin with respect to the element size of the meshed surface or that have sharp angles.
- If grafting fails some of the nodes of the original mesh may have been moved. Check the mesh quality and re-smooth if needed.

Grafting Example

This example shows the four basic steps of grafting:

1. Partition the geometry (optional).
2. Mesh the trunk volume.
3. Graft the branch volume onto the trunk volume.
4. Mesh the branch volume.

Step 1: Partition the geometry

Figure 1 shows the model that will be meshed. The arrows in the figure

show the two intersecting sweep directions. Figure 2 shows the model decomposed for grafting.

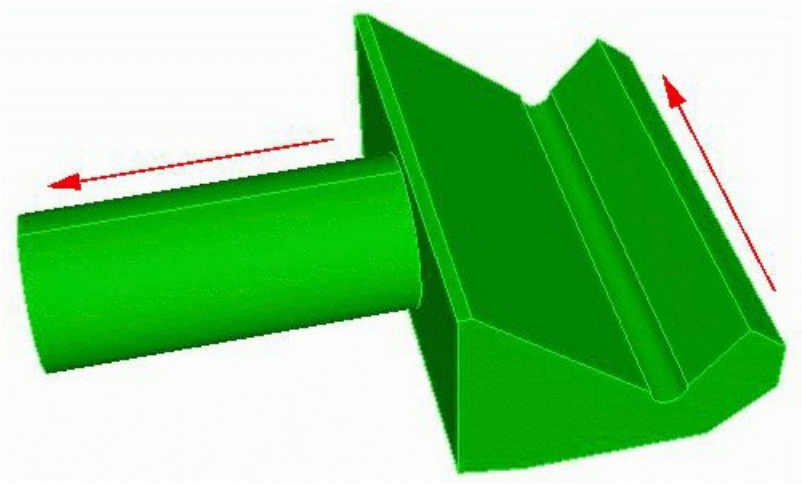


Figure 1. A model with two intersecting sweep directions.

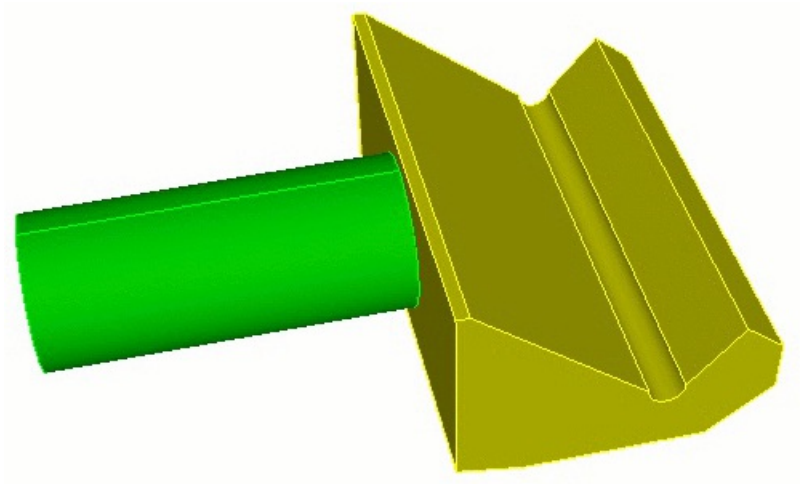


Figure 2. The model decomposed for grafting

Step 2: Mesh the trunk volume.

Figure 3 shows the mesh of the trunk volume. At this point the mesh on the trunk surface adjacent to the branch surface is a structured mesh that does not align with the boundary of the branch surface. The trunk and branch surfaces are two separate surfaces.

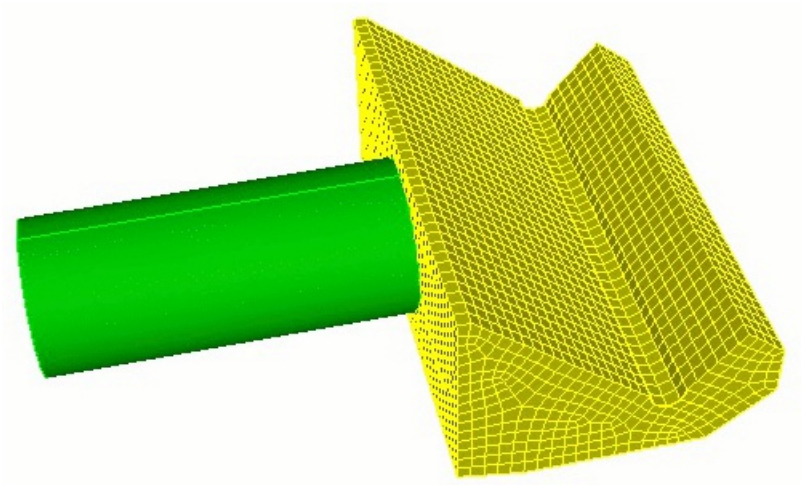


Figure 3. Meshed trunk volume.

Step 3: Graft the branch onto the trunk

Figure 4 shows the trunk surface after it has been modified to fit the branch surface. At this point the two surfaces have been merged together.

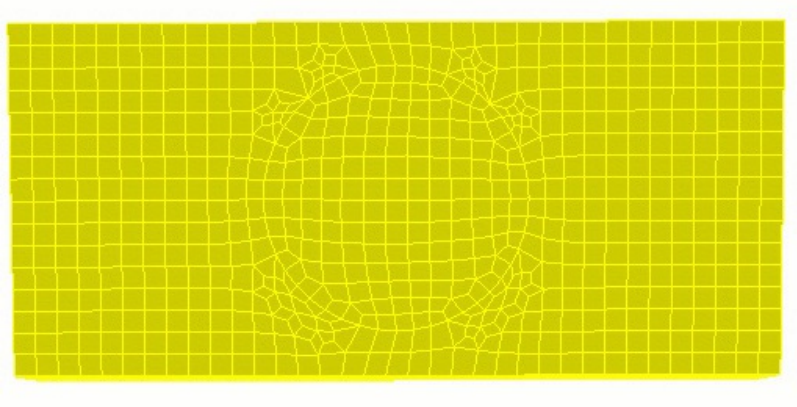


Figure 4. Trunk surface after grafting.

Step 4: Mesh the branch volume.

The final mesh is shown in Figure 5.

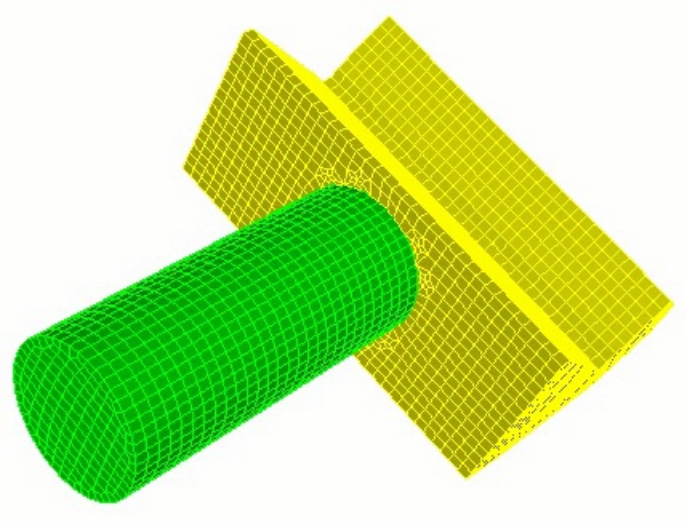


Figure 5. Final mesh

Optimize Jacobian 

Note: This feature is under development. The command to enable or disable features under development is:

```
Set Developer Commands {On|OFF}
```

Applies to: Volume meshes

Summary: Produces locally-uniform hex meshes by optimizing element Jacobians

Syntax:

```
Volume <range> Smooth Scheme Optimize Jacobian
[param]
```

Discussion:

The Optimize Jacobian method minimizes the sum of the squares of the Jacobians (i.e., volumes) attached to the smooth node. Meshes smoothed by this means tend to have locally-uniform hex volumes.

The parameter <**param**> has a default value of 1, meaning that the method will attempt to make local volumes equal. The parameter, which should always be between 1 and 2 (with 1.05 recommended), can be used to sacrifice local volume equality in favor of moving towards meshes with all-positive Jacobians.

Randomize 

Note: This feature is under development. The command to enable or disable features under development is:

```
Set Developer Commands {On|OFF}
```

Applies to: Curve, Surface and Volume meshes

Summary: Randomizes the placement of nodes on a geometry entity

Syntax:

```
{Surface|Volume} <range> Smooth Scheme
Randomize [percent]
```

Discussion:

This scheme will create non-smooth meshes. If a percent argument is given, this sets the amount by which nodes will be moved as a percentage of the local edge length. The default value for percent is 0.40. This smooth scheme is primarily a research scheme to help test other smooth schemes.

Refine Mesh Boundary

Note: This feature is under development. The command to enable or disable features under development is:

Set Developer Commands {On|OFF}

Boundary effects to be modeled in the analysis code frequently require a refined mesh near a specific surface. CUBIT provides this capability with the Refine Mesh Boundary command. This command is similar to the Refine Mesh Volume Feature command except that it can insert multiple sheets of hexes near the specified surface.

```
Refine Mesh Boundary Surface <range> Volume
<id> {Bias <double>} {First_delta <double> |
Thickness <double>} [Layer <num_layers=1>]
[SMOOTH|No_smooth]
```

With this command **num_layers** of hexes can be inserted at the first interval from the specified surface. A **bias factor** indicating the change in element size must be specified. You must also indicate a **first_delta** or **thickness** which represents the distance to the first inserted layer. The mesh in Figure 5 with bias 1.0 and first_delta of 5. The default smooth option provides the capability to smooth the mesh following the refinement procedure.

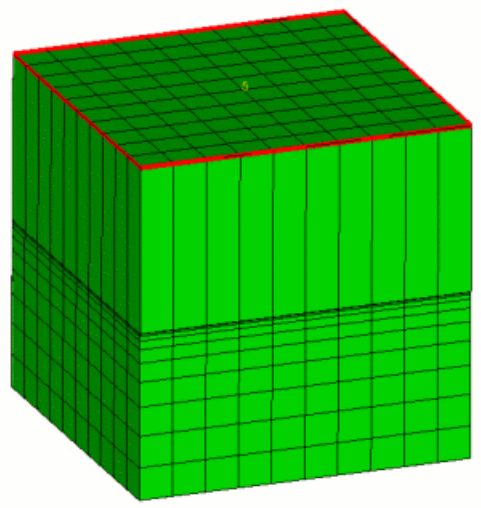


Figure 5. Example of Boundary Surface Refinement

Remove Tiny Edge Length

Note: This feature is under development. The command to enable or disable features under development is:

```
Set Developer Commands {On|OFF}
```

Applies to: Trimesh Surface Scheme

Summary: Tolerance specified to prevent small edges in a triangle mesh

Syntax:

```
  [Set] Trimesher Remove Tiny Edge Length
{<value>|[off]}
```

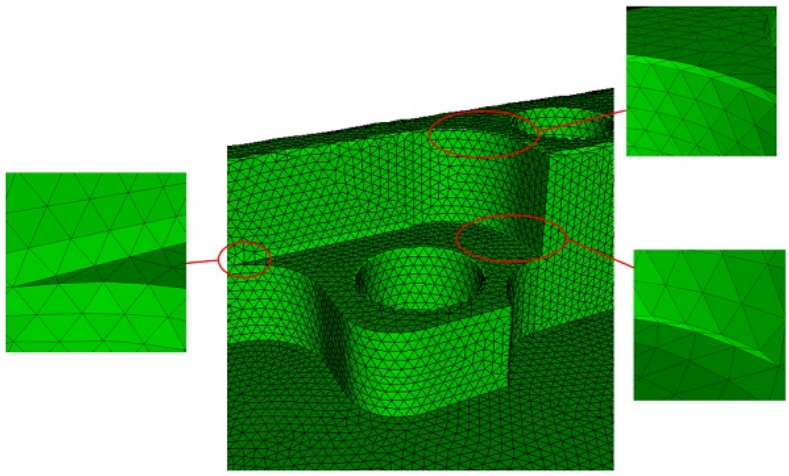
Discussion:

Setting the tiny edge length forces the MeshGems trimesher to generate triangles with edges greater than the specified value. It is actually a post processing step that collapses triangles with edges less than the specified value. This setting is necessary because the MeshGems triangle mesher sometimes inserts triangles with small edges along high curvature features, even though a larger size has been specified and geometry approximation has been turned off. Using this setting is the only way to guarantee that no edges smaller than the specified value will be created.

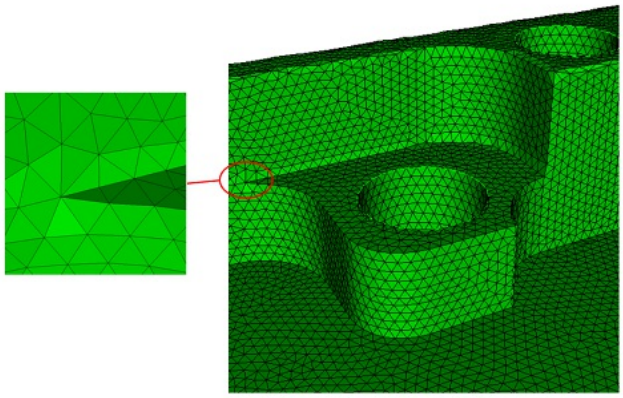
The **off** option resets the 'tiny edge length' value so it is not used.

The user should not use 'tiny edge length' values approaching the mesh size because an invalid mesh can result.

The images below show meshing a surface with and without setting a 'tiny edge length' value. In this example all surfaces have been composited into a single surface. Compositing small surfaces with larger neighbors in conjunction with using 'tiny edge length' has the effect of washing-over small features.



Without using 'remove tiny edge length' triangles with short edges remain.



With appropriate 'remove tiny edge length' value triangles with short edges are collapsed.

Super Sizing Function

Note: This feature is under development. The command to enable or disable features under development is:

```
Set Developer Commands {On|OFF}
```

The **Super** sizing function computes both the Curvature and the Linear function and takes the smaller value of the two. This is an alpha feature and should be used with caution. The following is an example of Super element sizing.

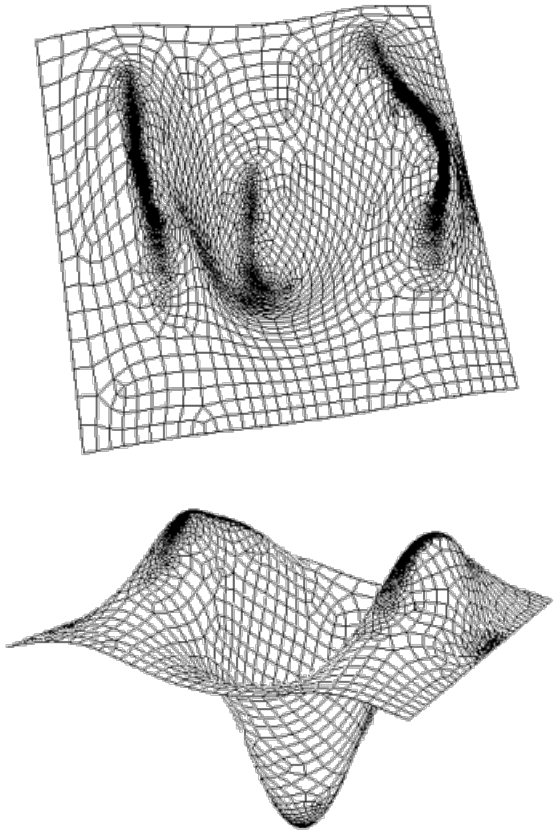


Figure 1. NURB mesh with super sizing function, 34 by 16 density

Test Sizing Function

Note: This feature is under development. The command to enable or disable features under development is:

```
Set Developer Commands {On|OFF}
```

The **Test** sizing function is a hardwired numerical function used to demonstrate the transitional effect of sizing function-based and adaptive paving. The function is a periodic function which is repeated in 50x50 unit intervals on a 2D surface in the first quadrant (x > 0, y > 0, z = 0). This is an alpha feature and should be used with caution. An example of a surface meshed with this sizing function is shown in Figure 1.

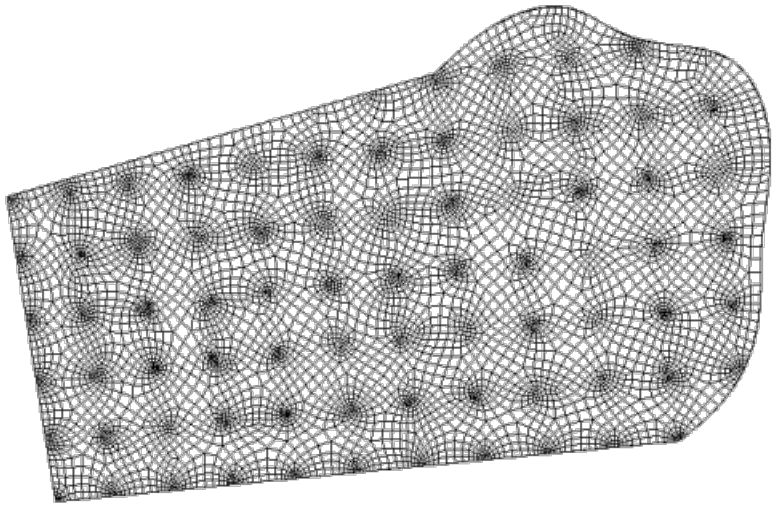


Figure 1. Test sizing function for spline geometry

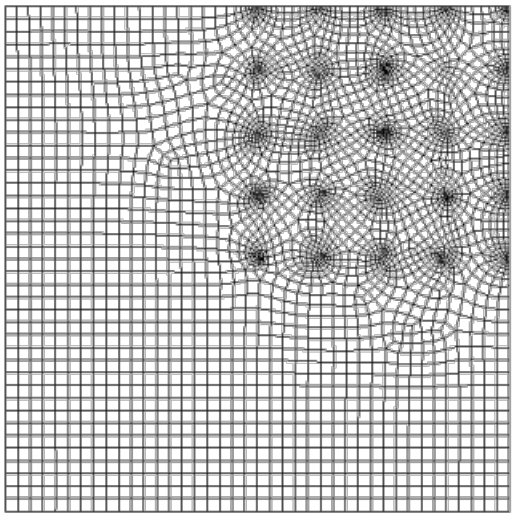


Figure 2. Test sizing function for square geometry

Transition

Note: This feature is under development. The command to enable or disable features under development is:

```
Set Developer Commands {On|OFF}
```

Applies to: Surfaces

Summary: Produces a specified transition mesh for specific situations

Syntax:

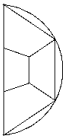
```
Surface <range> Scheme Transition
{Triangle|Half_circle|Three_to_one|Two_to_one|Convex_corner|Four_to_two}
[Source Curve <id>] [Source Vertex <id>]
```

Discussion:

The *transition* scheme supplies a set of transition primitives which serve to transition a mesh from one density to another across a given surface. The six transition sub-types are demonstrated here.



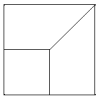
Scheme Transition **Triangle** creates four quads in a triangle that has sides of three, two, and one intervals.



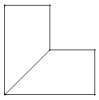
Scheme Transition **Half_circle** creates three intervals on the flat and three on the curved part of the half-circle, then creates four quads in the surface.



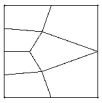
Scheme Transition **Three_to_one** creates four quads on a rectangular surface that has intervals of three, one, one, and one on its sides.



Scheme Transition **Two_to_one** creates three quads on a rectangular surface that has intervals of two, two, one and one on its sides :



Scheme Transition **Convex_corner** takes a six-sided block with a convex corner and connects that inner corner to the opposite one, creating two quads on the surface.



Scheme Transition **Four_to_two** creates seven quads on a rectangular surface that has intervals of four, two, two, and two on its sides.

The user also has the option of specifying a source curve and/or a source vertex. The rules for these specifications are as follows

- If both a curve and vertex are specified, the vertex must be on the curve.
- The Convex_corner sub-type does not allow a source curve.
- The Four_to_two sub-type does not allow a source vertex.
- The source curve will be the curve that will be given the fewest intervals.
- The source vertex will specify which corner will be used for the scheme, in cases where this makes sense (primarily in the Triangle, and Two_to_one cases).
- If none of the optional information is given, the program will assign the source curve to be the shortest one on the face, in keeping with the most probable

Triangle Mesh Coarsening

Note: This feature is under development. The command to enable or disable features under development is:

```
Set Developer Commands {On|OFF}
```

CUBIT provides the capability for coarsening triangle surface meshes. Triangle coarsening uses a technique known as edge collapsing to coarsen a mesh. With this technique, triangle edges are selectively eliminated from the mesh until the specified criteria have been met. The following commands will coarsen an existing triangle surface mesh:

```
Coarsen {Node|Edge|Tri} <range> {Factor|Size
<double> [Bias <double>]} [Depth <int>|Radius
<double>] [Sizing_Function] [no_smooth]

Coarsen {Vertex|Curve|Surface} <range>
{Factor|Size<double> [Bias<double>]}
[Depth<int>|Radius<double>] [Sizing_Function]
[no_smooth]
```

Important: These commands are currently implemented only for *triangle* shaped elements.

To use these commands, first select mesh or geometric entities at which you would like to perform coarsening. Coarsening operations will be applied to all mesh entities associated with or within proximity of the entities. The **all** keyword may be used to uniformly coarsen all triangles in the model.

Following is a description of each of the coarsen options:

Factor

Defines the approximate size relative to the existing edge lengths for which the coarsening will be applied. For example, a factor of 2 will attempt to make every edge length within the specified region approximately twice the size. A factor of 3 will make everything three times the size. Valid input values for factor must be greater than 1. Figure 1 shows an example where a coarsening factor of 2 was applied

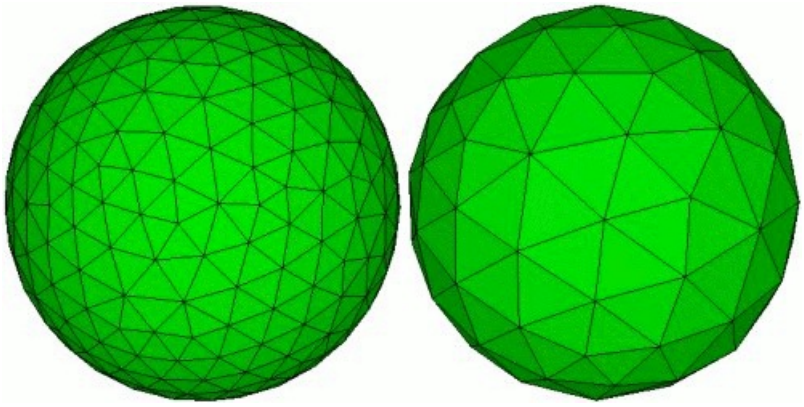


Figure 1. Example of coarsening all triangles with a factor of 2.

Size, Bias

The Size and Bias options are useful when a specific element size is desired at a known location. This might be used for locally coarsening around a vertex or curve. The Bias argument can be used with the Size option to define the rate at which the element sizes will change to meet

the existing element sizes on the model. Valid input values for Bias are greater than 1.0 and represent the maximum change in element size from one element to the next. Since coarsening is a discrete operation, the Size and Bias options can only approximate the desired input values. This may cause apparent discontinuities in the element sizes. Using the default smooth option can lessen this effect. It should also be noted that the Size option is exclusive of the Factor option. Either Factor or Size can be specified, but not both.

Depth

The Depth option permits the user to specify how many elements away from the specified entity will also be coarsened. Default Depth is 1.

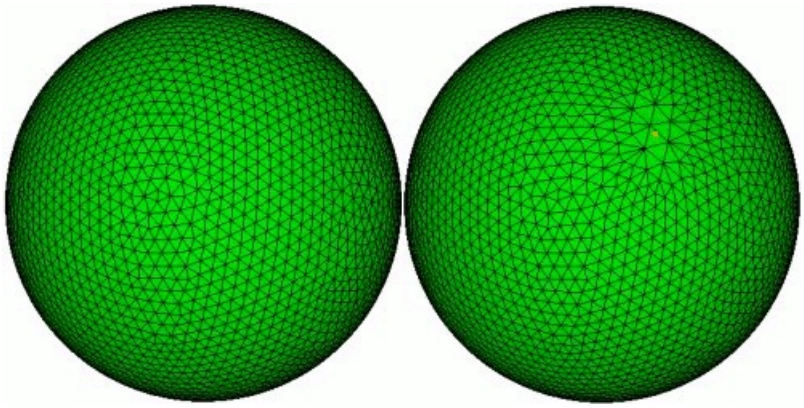


Figure 2. Coarsening performed at a node with factor = 3 and depth = 3

Radius

Instead of specifying the number of elements to describe how far to propagate the coarsening, a real Radius may be entered.

Sizing Function

Coarsening may also be controlled by a sizing function. CUBIT uses sizing functions to control the local density of a mesh. Various options for setting up a sizing function are provided, including importing scalar field data from an exodus file. In order to use this option, a sizing function must first be specified on the surface on which the coarsening will be applied. See Adaptive Meshing for a description of how to define a sizing function.

No_Smooth

The default mode for coarsening operations is to perform smoothing after coarsening the elements. This will generally provide better quality elements. In some cases it may be necessary to retain the original node locations after coarsening. The no_smooth option provides this capability.

Higher Order Element Metrics

Note: This feature is under development.

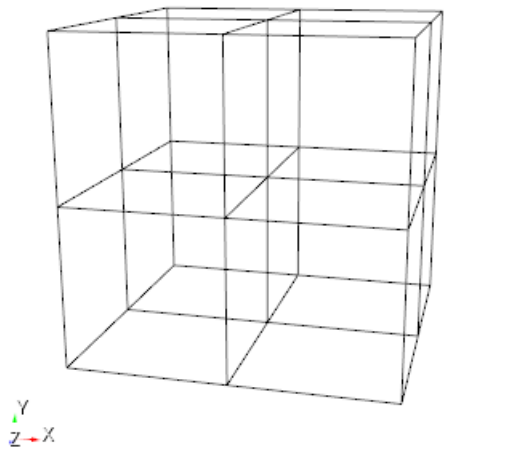
Cubit contains an **Node Distance** and **Altitude** quality metric for higher order elements. Currently, HEX27, WEDGE21 and TET15 are supported. The metrics are designed to help identify small distances which limit the analysis time step.

These metrics may be accessed by using the commands:

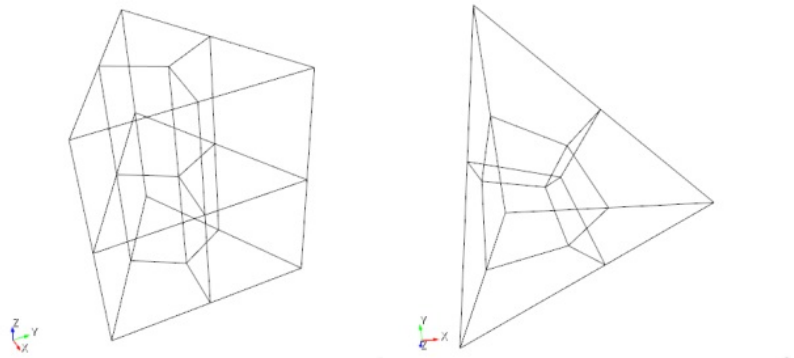
```
Quality <element list> Node Distance

Quality <element list> Altitude
```

The **Node Distance** quality metric for a single element is the minimum distance between nodes within that element. Not every node is compared against every other node within an element, rather, a structured approach is used. For a single HEX27 element, the element can be subdivided using the 27 nodes to form 8 logical hexes with 8 nodes each. The edge lengths of the logical hexes are the distances considered for the HEX27 element. See the image below for a representation of the edges used to compute node distance in a single HEX27 element.

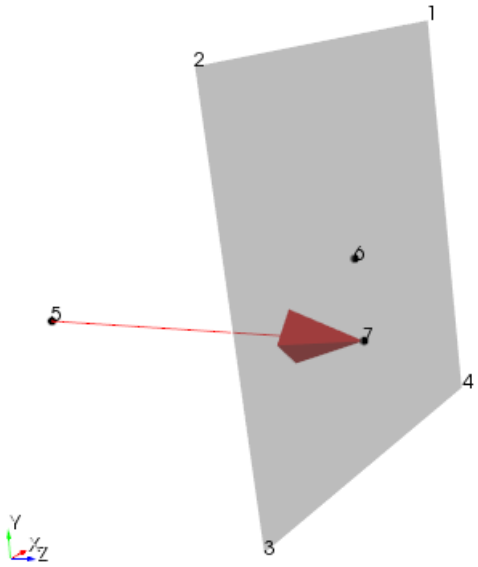


Additionally, the representation of edges used for a single WEDGE21 and TET15 element are given in the following images.



The **Altitude** quality metric for a single element is the minimum distance between the body center node and the projection of the body center node to the faces. Specifically, for a HEX27, the body center node is projected

to each of the 6 faces of the element, and the minimum distance between the body center node and the 6 projections is the altitude metric for that element. The face projection is based on the linear approximation of the higher order face. The image below shows an example of the distance calculated when a body center node is projected to a face.



APREPRO

Within CUBIT there is support for a programming language called *APREPRO* (An Algebraic Preprocessor for Parameterizing Finite Element Analyses). In addition to the standard APREPRO functionality, CUBIT extends the language with its own functions to aid in the meshing process. Included here is a summary of the CUBIT-specific APREPRO functionality. For a description of the APREPRO language and its usage, please refer to the APREPRO user's manual (PDF).

- Using APREPRO in CUBIT
- APREPRO Functions
- APREPRO Journaling

Note: APREPRO variables can be created and modified from the GUI. Enable/disable the editor from the View/Aprepro editor menu option. The editor is a docking window and can be placed anywhere in the GUI.

Using APREPRO in CUBIT

To use APREPRO within CUBIT, simply enclose APREPRO statements within curly braces '**{}**' as part of the CUBIT command. Any APREPRO statements included in a command will be evaluated before the command is executed. For example, if the APREPRO variable 'my_x' is given the value of 3, the command

```
brick x {my_x}
```

will become

```
brick x 3
```

before the command is executed by CUBIT. Note that this means APREPRO will NOT give CUBIT parametric modeling abilities. In the above example, if the value of 'my_x' is later changed to 5, the size of the brick already created will not automatically change to five.

APREPRO expressions can also exist on separate lines. When doing this, it is recommended to add the CUBIT comment character '**#**' before the APREPRO statement. This will tell CUBIT to treat the evaluated expression as a comment, which will prevent errors from being issued in many cases.

Consider the following example:

```
#{my_x = 3}

#{my_y = my_x + 2}

#{if(my_y < my_x)}

brick x {my_x} y {my_y}

#{else}

create cylinder radius {my_x} height {my_y}

#{endif}
```

In the first two lines, only APREPRO statements are being executed (values are assigned to the variables 'my_x' and 'my_y'). After being evaluated by APREPRO, these two lines will be sent to CUBIT as

#3

#5

If the comment character was omitted instead CUBIT would issue several errors about incorrect command syntax. However, because these lines start with the comment character, they are ignored by CUBIT. Also note that the character '$' may be used in place of '#' for comments.

Loops

Repeated processing of a group of lines can be controlled with the **{loop(control)}**, **{endloop}** commands, as noted in section 6.2.5 of the APREPRO documentation.

A loop may also be terminated before running the specified number of times using a **#{break}** statement. As soon as a **#{break}** statement is encountered, the loop is exited and the rest of the statements in the loop will not execute. Additional iterations of the loop will not be executed either.

For example, the following commands will create 3 bricks:

```
#{_x = 1}
#{loop(10)}

  brick x 1

  #{if(_x == 2)}

    #{break}

  #{endif}

  #{_x++}

  pause

  brick x 1

#{endloop}
```

When a **#{break}** statement executes, anything in the loop following the **#{break}** statement will be skipped, including the **#{endif}**. For this reason, a **#{break}** statement not only exits the loops, but also terminates the most recent **#{if}** statement exactly as **#{endif}** would do. **#{break}** statements should not be used outside of **#{if}** statements.

It is also possible to terminate a loop using the **#{abortloop}** statement. **#{abortloop}** will terminate all loops (including nested loops) without executing the contents of the loop(s). This can be useful when a typo is made while manually entering a loop at the command line. Instead of ending the loop normally and waiting for the loop to execute with numerous errors, the loop will end immediately without any execution or errors. Please note, however, that the **#{abortloop}** statement is only valid within a loop block; otherwise, it will generate errors.

When creating a loop, APREPRO will record all lines that are given to the command line until the corresponding **#{endloop}** is reached. During this process, no commands will be passed to CUBIT. Once the terminating **#{endloop}** is reached, APREPRO will expand the loop, repeating the recorded lines the number of times specified by the loop counter, and send the expanded list of commands to CUBIT. If the terminating **#{endloop}** is accidentally omitted, CUBIT may appear to be unresponsive to commands because APREPRO is still recording lines for the loop. In situations like these, the **#{abortloop}** statement may be used to terminate any unfinished loops and restore the command line to a working state.

Also note that it is not recommended to use the '**pause**' command within a loop, as it can lead to situations in which the user must repeatedly enter the command '**resume**' to execute the entire loop. In situations like these **#{abortloop}** will NOT terminate the loop because it has already been expanded by APREPRO and CUBIT is simply executing a list of commands.

Deleting APREPRO Variables

There are two ways to delete an APREPRO variable in CUBIT. The first is to use the APREPRO '**delete**' function. The delete function takes the name of the variable to be deleted as its argument, as shown in the following example:

```
#{my_var = 2}

...

#{delete('my_var')}
```

The second way to delete an APREPRO variable is by using the '**reset aprepro**' command:

```
#{my_var = 2}
```

```
#{some_var = 3}

...

reset aprepro
```

This will delete all APREPRO variables and reset APREPRO to its initial state.

Other Examples

The following example shows the use of some of the string functions.

```
#{t1 = "ATAN2"}{t2="(0,-1)"}

#{t3 = tolower(t1 // t2)}
```

... The variable t3 is equal to the string atan2(0,-1)

```
#{execute(t3)}
```

...t3 = 3.141592654

The result is the same as executing {atan2(0,-1)} This is admittedly a very contrived example; however, it does illustrate the workings of several of the functions. In the first example, an expression is constructed by concatenating two strings together and converting the resulting string to lowercase. This string is then executed.

The following example uses the **rescan** function to illustrate a basic macro capability in APREPRO. The example creates vertices in CUBIT equally spaced about the circumference of a 180 degree arc of radius 10. Note that the macro is 5 lines long (2 of the lines start with #, with the exception of the looping constructs - the actual journal file for this would not continue lines but would put each one on one long line).

```
#{num = 0} {rad = 10} {nintv = 10} {nloop =
nintv + 1}

#{line = 'Create Vertex {polarX(rad,(++num-
1)*180/nintv)} {polarY(rad,(num-
1)*180/nintv)}'}

#{loop(nloop)}

{rescan(line)}

#{endloop}
```

Note the loop construct to automatically repeat the **rescan** line. To modify this example to calculate the coordinates of 101 points rather than eleven, the only change necessary would be to set {nintv=100}.

APREPRO Functions

CUBIT adds a number of APREPRO functions to aid in the meshing process. A description of each function is available in the categories below.

- Geometry Query Functions
- Mesh Query Functions
- Group, Block, and Assembly Functions
- Id Functions
- Miscellaneous Functions
- Pre-defined Variables

Table 1. Geometry Functions

Syntax	Description
BBox_XMin("type", id)	Returns the xmin value of the bounding box of the specified geometric entity. “type” can be “volume”, “surface”, “curve”, “vertex”, or “group”. If “volume”, “surface”, “curve”, or “vertex” it will calculate the bounding box for the entity with the given id. If “group” it will calculate the combined bounding box for the group. A group can have any of the geometry types (vol, surf, curve, vert) in it and can be of mixed types.
BBox_XMax("type", id)	Returns the xmax value of the bounding box of the specified geometric entity. “type” can be “volume”, “surface”, “curve”, “vertex”, or “group”. If “volume”, “surface”, “curve”, or “vertex” it will calculate the bounding box for the entity with the given id. If “group” it will calculate the combined bounding box for the group. A group can have any of the geometry types (vol, surf, curve, vert) in it and can be of mixed types.
BBox_YMin("type", id)	Returns the ymin value of the bounding box of the specified geometric entity. “type” can be “volume”, “surface”, “curve”, “vertex”, or “group”. If “volume”, “surface”, “curve”, or “vertex” it will calculate the bounding box for the entity with the given id. If “group” it will calculate the combined bounding box for the group. A group can have any of the geometry types (vol, surf, curve, vert) in it and can be of mixed types.
BBox_YMax("type", id)	Returns the ymax value of the bounding box of the specified geometric entity. “type” can be “volume”, “surface”, “curve”, “vertex”, or “group”. If “volume”, “surface”, “curve”, or “vertex” it will calculate the bounding box for the entity with the given id. If “group” it will calculate the combined bounding box for the group. A group can have any of the geometry types (vol, surf, curve, vert) in it and can be of mixed types.

BBox_ZMin("type", id)	Returns the zmin value of the bounding box of the specified geometric entity. “type” can be “volume”, “surface”, “curve”, “vertex”, or “group”. If “volume”, “surface”, “curve”, or “vertex” it will calculate the bounding box for the entity with the given id. If “group” it will calculate the combined bounding box for the group. A group can have any of the geometry types (vol, surf, curve, vert) in it and can be of mixed types.
BBox_ZMax("type", id)	Returns the zmax value of the bounding box of the specified geometric entity. “type” can be “volume”, “surface”, “curve”, “vertex”, or “group”. If “volume”, “surface”, “curve”, or “vertex” it will calculate the bounding box for the entity with the given id. If “group” it will calculate the combined bounding box for the group. A group can have any of the geometry types (vol, surf, curve, vert) in it and can be of mixed types.
GeomCentroid_X("type", id)	Returns the x coordinate of the centroid of the specified geometric entity. "type" can be "volume" or "group". If “volume” it calculates the centroid for the volume with the given id (single volume). If “group” it must be a group of volumes and it will calculate the combined centroid for the whole group with the given id.
GeomCentroid_Y("type", id)	Returns the y coordinate of the centroid of the specified geometric entity. "type" can be "volume" or "group". If “volume” it calculates the centroid for the volume with the given id (single volume). If “group” it must be a group of volumes and it will calculate the combined centroid for the whole group with the given id.
GeomCentroid_Z("type", id)	Returns the z coordinate of the centroid of the specified geometric entity. "type" can be "volume" or "group". If “volume” it calculates the centroid for the volume with the given id (single volume). If “group” it must be a group of volumes and it will calculate the combined centroid for the whole group with the given id.
Length(id)	Returns the length of the curve with the given id.
Length(x, y, z, ord)	Returns the length of the curve identified by the given center point coordinates and ordinal value.
NumCurves()	Returns the number of curves in the model.
NumSurfaces()	Returns the number of surfaces in the model.
NumVertices()	Returns the number of vertices in the model.
NumVolumes()	Returns the number of volumes in the model.
Radius(id)	Returns the radius of the curve at its midpoint.
Radius(x, y, z, ord)	Returns the radius of the curve identified by the given center point coordinates and ordinal value.

SurfaceArea(id)	Returns the surface area of the surface with the given id.
SurfaceArea(x, y, z, ord)	Returns the surface area of the surface identified by the given center point coordinates and ordinal value.
Type("entity name")	Returns the type of the specified entity name
Volume(id)	Gets the geometric volume of the volume with the given id.
Volume(x, y, z, ord)	Gets the geometric volume of the volume identified by the given center point coordinates and ordinal value.
Vx(id), Vy(id), Vz(id)	Gets the x, y or z coordinate of vertex with the given id.
Vx(x, y, z, ord) Vy(x, y, z, ord) Vz(x, y, z, ord)	Gets the x, y or z coordinate of vertex identified by the given center point coordinates and ordinal value.
VertexAt(x, y, z, ordinal)	Returns the id of the vertex with the idless reference, x,y,z,ordinal.
CurveAt(x, y, z, ordinal)	Returns the id of the curve with the idless reference, x,y,z,ordinal.
SurfaceAt(x, y, z, ordinal)	Returns the id of the surface with the idless reference, x,y,z,ordinal.
VolumeAt(x, y, z, ordinal)	Returns the id of the volume with the idless reference, x,y,z,ordinal.

Table 2. Mesh Functions

Syntax	Description
EdgeLength(id)	Returns the length of the edge with the given id.
EdgeLength(x, y, z, ord)	Returns the length of the edge identified by the given center point coordinates and ordinal value.
FaceArea(id)	Returns the area of the face with the given id.
FaceArea(x, y, z, ord)	Returns the area of the face identified by the given center point coordinates and ordinal value.
HexVolume(id)	Returns the volume of the hex with the given id.
HexVolume(x, y, z, ord)	Returns the volume of the hex identified by the given center point coordinates and ordinal value.
IntNum(id)	Returns the number of intervals on a curve with the given id.
IntNum(x, y, z, ord)	Returns the number of intervals on a curve identified by the given center point coordinates and ordinal value.
IntSize(id)	Returns the interval size on a curve with the given id.
IntSize(x, y, z, ord)	Returns the interval size on a curve identified by the given center point coordinates and ordinal value.

MeshCentroid_X("type", id)	Returns the x coordinate of the centroid of the specified mesh entity. “type” can be “volume”, “block”, or “group”. If “volume” it calculates the centroid of the 3D elements in the volume with the given id. If “block” it calculates the centroid of the elements in the block with the given id. If “group” it must be a group of volumes and it calculates the centroid of the group with the given id.
MeshCentroid_Y("type", id)	Returns the y coordinate of the centroid of the specified mesh entity. “type” can be “volume”, “block”, or “group”. If “volume” it calculates the centroid of the 3D elements in the volume with the given id. If “block” it calculates the centroid of the elements in the block with the given id. If “group” it must be a group of volumes and it calculates the centroid of the group with the given id.
MeshCentroid_Z("type", id)	Returns the z coordinate of the centroid of the specified mesh entity. “type” can be “volume”, “block”, or “group”. If “volume” it calculates the centroid of the 3D elements in the volume with the given id. If “block” it calculates the centroid of the elements in the block with the given id. If “group” it must be a group of volumes and it calculates the centroid of the group with the given id.
MeshLength(id)	Gets the length of the meshed curve with the given id.
MeshLength(x, y, z, ord)	Gets the length of the meshed curve identified by the given center point coordinates and ordinal value.
MeshSurfaceArea(id)	Returns the total area of all triangle or quadrilateral elements on the surface with the given id. This will vary from the geometric surface area since the mesh approximates the boundary with linear mesh edges.
MeshSurfaceArea(x, y, z, ord)	Returns the total area of all triangle or quadrilateral elements on the surface identified by the given center point coordinates and ordinal value. This will vary from the geometric surface area since the mesh approximates the boundary with linear mesh edges.
MeshVolume(id)	Returns the total volume of all mesh elements in the volume with the given id. This will vary from the actual geometric volume since the mesh approximates curved boundaries with linear mesh edges.
MeshVolume(x, y, z, ord)	Returns the total volume of all mesh elements in the volume identified by the given center point coordinates and ordinal value. This will vary from the actual geometric volume since the mesh approximates curved boundaries with linear mesh edges.

MinSurfaceMeshQuality(id, "metric")	Returns the worst value of the specified element quality metric of all elements on the given surface. Acceptable metrics include: shape aspect ratio condition no distortion element area jacobian maximum angle minimum angle relative size scaled jacobian shape and size shear and size shear skew stretch taper warpage
MinSurfaceMeshQuality(x, y, z, ord, "metric")	Returns the worst value of the specified element quality metric of all elements on the surface identified by the given center point coordinates and ordinal value. Acceptable metrics include: shape aspect ratio condition no distortion element area jacobian maximum angle minimum angle relative size scaled jacobian shape and size shear and size shear skew stretch taper warpage

MinVolumeMeshQuality(id, "metric")	Returns the worst value of the specified element quality metric of all elements in the volume with the given id. Acceptable metrics include: shape aspect ration bet aspect ratio gam aspect ratio condition no diagonal ratio dimension distortion element volume jacobian mass increase ratio mean ratio inradius node distance normalized inradius relative size scaled jacobian shape and size shear and size shear skew stretch taper timestep
MinVolumeMeshQuality(x, y, z, ord, "metric")	Returns the worst value of the specified element quality metric of all elements in the volume identified by the given center point coordinates and ordinal value. Acceptable metrics include: shape aspect ration bet aspect ratio gam aspect ratio condition no diagonal ratio dimension distortion element volume jacobian mass increase ratio mean ratio inradius node distance normalized inradius relative size scaled jacobian shape and size shear and size shear skew stretch taper timestep
NumEdgesOnCurve(id)	Returns the number of edges on the curve with the given id.
NumEdgesOnCurve(x, y, z, ord)	Returns the number of edges on the curve identified by the given center point coordinates and ordinal value.

NumElemsOnSurface(id)	Returns the number of elements on the surface with the given id.
NumElemsOnSurface(x, y, z, ord)	Returns the number of elements on the surface identified by the given center point coordinates and ordinal value.
NumElemsInVolume(id)	Returns the number of elements in the volume with the given id.
NumElemsInVolume(x, y, z, ord)	Returns the number of elements in the volume identified by the given center point coordinates and ordinal value.
Nx(id), Ny(id), Nz(id)	Gets the x, y or z coordinate of node with the given id.
Nx(x, y, z, ord) Ny(x, y, z, ord) Nz(x, y, z, ord)	Gets the x, y or z coordinate of node identified by the given center point coordinates and ordinal value.
TetVolume(id)	Returns the volume of the tet with the given id.
TetVolume(x, y, z, ord)	Returns the volume of the tet identified by the given center point coordinates and ordinal value.
TriArea(id)	Returns the area of the tri with the given id.
TriArea(x, y, z, ord)	Returns the area of the tri identified by the given center point coordinates and ordinal value. .

Table 3. Group, Block, and Assemblyl Metadata Functions

Syntax	Description
BlockAttributeName(id, index)	Returns the name for the specified attribute index in the block within the given id
BlockAttributeValue(id, index)	Returns the value for the specified attribute index in the block within the given id
NumBlocks()	Returns the number of blocks in the model.
NumSidesets()	Returns the number of sidesets in the model.
NumNodesets()	Returns the number of nodesets in the model.
NumInGrp("groupname")	Returns the number of entities in the given group.
NumTypeInGroup("group_name", "entity_type")	Returns the number of "entity_type" in group "group_name".
NumVolsInPart("part_name")	Returns the number of volumes assigned to the part with the specified name.

PartInVol(id)	Returns the name and instance number of the part that the volume has been assigned to.

Table 4. ID Functions

Syntax	Description
CurveAt(x, y, z, ordinal)	Returns the id of the curve with the idless reference, x,y,z,ordinal.
EdgeAt(x, y, z, ordinal)	Returns the id of the edge with the idless reference, x,y,z,ordinal.
FaceAt(x, y, z, ordinal)	Returns the id of the quad face with the idless reference, x,y,z,ordinal.
GroupMemberId("group_name", "entity_type", index)	Returns the ID of "entity_type" in group "group_name" at the specified index. If the group contains multiple entity types the index will only be relevant for the entity type specified and will behave as if the group only contained that entity type.
HexAt(x, y, z, ordinal)	Returns the id of the hexahedra element with the idless reference, x,y,z,ordinal.
Id("type")	Returns the ID of the entity most recently created with the specified type. Acceptable types include: "body", "volume", "surface", "curve", "vertex", "group", "node", "edge", "quad", "face", "tri", "hex", "tet", or "pyramid".
NodeAt(x, y, z)	Returns the id of the node closest to the xyz location.
NodeAt(x, y, z, ordinal)	Returns the id of the Node with the idless reference, x,y,z,ordinal.
PyramidAt(x, y, z, ordinal)	Returns the id of the pyramid element with the idless reference, x,y,z,ordinal.
SurfaceAt(x, y, z, ordinal)	Returns the id of the surface with the idless reference, x,y,z,ordinal.
TetAt(x, y, z, ordinal)	Returns the id of the tetraheral element with the idless reference, x,y,z,ordinal.
TriAt(x, y, z, ordinal)	Returns the id of the triangle with the idless reference, x,y,z,ordinal.
VertexAt(x, y, z, ordinal)	Returns the id of the vertex with the idless reference, x,y,z,ordinal.
VolumeAt(x, y, z, ordinal)	Returns the id of the volume with the idless reference, x,y,z,ordinal.
WedgeAt(x, y, z, ordinal)	Returns the id of the wedge element with the idless reference, x,y,z,ordinal.

Table 5. Miscellaneous Functions

Syntax	Description

FileExists("file name")	Checks if the given file exists. Returns non-zero if true.
GeometryEngineVersion("engine name")	Get the version for the specified geometry engine.
get_error_count()	Returns the current error count in CUBIT
get_warning_count()	Returns the current warning count in CUBIT
HasFeature("feature name")	Checks if the specified feature is available. Returns non-zero if the feature is enabled.
Print(msg)	Prints msg
PrintError(svar)	Outputs the string **svar** to stderr.
Quote(svar)	Returns the string **svar**, enclosed in single quotes.
SessionId()	Returns the unique id for the current CUBIT session.
GetMachineType()	Returns the string name of the platform Cubit is running on ("Linux", "Darwin", "Microsoft Windows", etc.).
set_error_count(val)	Sets the error count in CUBIT to val
set_warning_count(val)	Sets the warning count in CUBIT to val
TimerStart()	Starts the CPU timer
TimerStop()	Stops the CPU timer

Table 6. Pre-defined Variables

The following APREPRO variables are predefined in CUBIT.

Variable	**Description**
CUBIT	Variable to indicate that CUBIT is defined
CUBIT_VERSION	Current version of CUBIT (not to be confused with **VERSION**, which stores the current version of APREPRO)

APREPRO Journaling

When using APREPRO, statements can be echoed to a journal file. To do so, use the following command:

```
[set] Journal [Graphics|Names|Aprepro|Errors]
[on|off]
```

Simply typing "journal aprepro" without an argument will display the current aprepro journaling setting.

For example,

```
bri x {2*5.0}
```

is journaled as

```
brick x {2*5.0}
```

if aprepro journaling is ON, or

```
brick x 10
```

if aprepro journaling is off. The default is **ON**.

APREPRO Comments

Comments are also journaled. This is useful for documenting aprepro definitions and descriptions.

Comments on the same line as a command get split into two separate lines in the journal file.

Significant Figures

When journal aprepro is **ON**, numbers are journaled exactly as they are entered. The maximum number of significant digits is determined by the command input.

When journal aprepro is **off**, numeric results of aprepro statements are journaled according to the maximum number of significant digits hard-coded into CUBIT, using the value of **DBL_DIG**.

Loops and Journaling

Loops are not journaled as loops, per se. For example, the APREPRO expression:

```
{loop(3)}
    bri x {x}
{endloop}
```

is journaled as:

```
bri x {x}
bri x {x}
bri x {x}
```

Multi-line Strings

Multi-line strings are currently not journaled (both definitions and when they are expanded). For example,

```
#{line = 'bri x 10
mesh vol 1'}
```

{line}

will be journaled as

bri x 10
mesh vol 1

Note that **bri x 10\n mesh vol 1** was not journaled as **{line}**

Python Interface

The following Python functions and objects provide capability to query and modify Cubit models.

Functions

CubitInterface - Cubit model query and modify functions.

Classes

Entity - The base class of all the geometry and mesh types.
GeomEntity - The base class for specifically the Geometry types (Body, Surface, etc.).
Body - Defines a body object that mostly parallels Cubit's Body class.
Volume - Defines a volume object that mostly parallels Cubit's RefVolume class
Surface - Defines a surface object that mostly parallels Cubit's RefFace class.
Curve - Defines a curve object that mostly parallels Cubit's RefEdge class.
Vertex - Defines a vertex object that mostly parallels Cubit's RefVertex class.
CFD BC Interface -Defines the interface to CFD Boundary Condition entities
Direction Interface -Defines the interface to a Direction object
Location Interface -Defines the interface to a Location object
CubitFailureException - An exception class to alert the caller when the underlying Cubit function fails.
InvalidEntityException - An exception class to alert the caller that an invalid entity was attempted to be used.
InvalidInputException - An exception class to alert the caller of a function that invalid inputs were entered.
MeshError - Mesh error interface
AssemblyItem - AssemblyItem interface

Importing Cubit into Python

Python users are able to import Cubit into Python and make calls into Cubit via CubitInterface and the other Python classes described in this section. Below is a simple Python script. The key parts are ensuring the Cubit libraries are on the path and ensuring the cubit.init() call is made first.

```
import sys

# add Cubit libraries to your path
sys.path.append('/opt/cubit/bin')

import cubit

#start cubit - this step is key
#cubit.init does not require any arguments.
#If you do want to provide arguments, you must
#provide 2 or more, where the first must
#be "cubit", and user args start as the 2nd argument.
#If only one argument is used, it will be ignored.
cubit.init(['cubit','-nojournal'])

height = 1.2
blockHexRadius = 0.1732628

#hexagon
baseBlock = cubit.prism(height, 6, blockHexRadius, blockHexRadius)

#etc . . .
```

Python Cubit Enhancement Scripts

The Python-Cubit enhancement code base is intended to be used as an extension to already existing Cubit functionality. It provides the user with a number of functionalities that are either currently outside the realm of the python functions which cubit supplies internally (such as vector math), or that are comprised of commonly used combinations of already existing python functionalities.

Python Cubit Enhancement Scripts

Cubit Python API 17.05

cubit Namespace Reference

Classes | Functions | Variables

Classes

class **AssemblyItem**
Class to implement assembly tree interface. More...

class **Body**
Defines a body object that mostly parallels Cubit's **Body** class. More...

class BoltHoleInfo

class **CFD_BC_Entity**
Class to implement cfd bc data retrieval. More...

class CIObserve

class **CubitFailureException**
An exception class to alert the caller when the underlying Cubit function fails. More...

class CubitMessageHandler
CubitMessageHandler provides a way to override how messages are displayed to the user. More...

class CubitProgressHandler

class CubitStringMessageHandler

class **Curve**
Defines a curve object that mostly parallels Cubit's RefEdge class. More...

class **Dir**
Defines a direction object. More...

class **Entity**
The base class of all the geometry and mesh types. More...

class ExternalExitHandler

class **GeomEntity**
The base class for specifically the Geometry types (**Body**, **Surface**, etc.) More...

class **InvalidEntityException**
An exception class to alert the caller that an invalid entity was attempted to be used. More...

class **InvalidInputException**
An exception class to alert the caller of a function that invalid inputs were entered. More...

class **Loc**
Defines a location object. More...

class **MeshErrorFeedback**
Class to implement mesh command feedback processing. More...

class MeshImport

class **Surface**
Defines a surface object that mostly parallels Cubit's RefFace class. More...

class **Vertex**
Defines a vertex object that mostly parallels Cubit's RefVertex class. More...

class **Volume**
Defines a volume object that mostly parallels Cubit's RefVolume class. More...

class VolumeGap

Functions

def **body** (id_in)
Retrieve a body by its ID. More...

def **brick** (width, depth=-1, height=-1)
Create a brick of specified width, depth, and height. More...

def **copy_body** (init_body)
Create a duplicate of the specified body. More...

def **create_arc_curve** (v0, v1, intermediate_point)
Create an arc curve using end vertices and an intermediate point. More...

def **create_curve** (v0, v1)
Create a curve between two vertices. More...

def **create_spline** (points, surface_id)
Create a spline curve through a sequence of 3D points on a surface. More...

def **create_surface** (curves)
Create a sheet body from boundary curves. More...

def **create_vertex** (x, y, z)
Create a vertex at specified coordinates. More...

def **curve** (id_in)
Retrieve a curve object by its ID. More...

def **cylinder** (height, x_radius, y_radius, top_radius)
Create a cylinder or truncated cone of specified dimensions. More...

def **get_all_cfd_bcs** ()

def **get_assembly_children** (assembly_id)

def **get_assembly_items** ()

def **get_mesh_error_count** ()

def **get_mesh_errors** ()

def **get_top_level_assembly_items** ()

def **get_volume_CAD_material** (volume_id)

def **get_volumes_for_node** (node_name, node_instance)

def **move** (entity, vector, preview=False)
Translates an **Entity** by a specified vector. More...

def **prism** (height, sides, major, minor)
Create a prism of the specified dimensions. More...

def **pyramid** (height, sides, major, minor, top=0.0)
Create a pyramid of specified dimensions. More...

def **reflect** (entity, axis, preview=False)
Reflects an **Entity** about a specified axis (e.g., plane normal). More...

def **reset** ()
Reset Cubit state. More...

def **scale** (entity, factor, preview=False)
Scales an **Entity** uniformly by a specified factor. More...

def **set_element_variable** (element_ids, variable_name, variables)
Sets scalar variables on specified mesh elements for Exodus export. More...

def **set_nodal_variable** (node_ids, variable_name, variables)
Sets scalar variables on specified mesh nodes. More...

def **sphere** (radius, x_cut=0, y_cut=0, z_cut=0, inner_radius=0)
Create all or part of a sphere. More...

def **subtract** (tool_in, from_in, imprint_in=False, keep_old_in=False)
Performs a boolean subtract operation: removes tool bodies from target bodies. More...

def **surface** (id_in)
Retrieve a surface object by its ID. More...

def **sweep_curve** (curves, along_curves, draft_angle, draft_type, rigid)
Sweep one or more curves along a path to create sheet bodies. More...

def **torus** (center_radius, swept_radius)
Create a torus of specified dimensions. More...

def **tweak_curve_offset** (curves, distances, keep_old=False, preview=False)
2D equivalent of tweak_surface_offset: offsets specified curves on a sheet body. More...

def **tweak_curve_remove** (curves, keep_old=False, preview=False)
Removes specified curves and extends adjacent surfaces on a sheet body. More...

def **tweak_surface_offset** (surfaces, distances)
Offsets specified surfaces by given distances. More...

def **tweak_surface_remove** (surfaces, extend_ajoining=True, keep_old=False, preview=False)
Removes specified surfaces from a body, optionally extending adjacent surfaces. More...

def **tweak_vertex_fillet** (verts, radius, keep_old=False, preview=False)
2D sheet vertex chamfer: creates chamfers at specified vertices by offsetting adjacent curves. More...

def **unite** (body_in, keep_old_in=False)
Performs a boolean unite operation: merges specified bodies into one. More...

def **vertex** (id_in)
Retrieve a vertex object by its ID. More...

def **volume** (id_in)
Retrieve a volume by its ID. More...

CubitInterface Control

def **init** (argv)
Initialize the Cubit engine with optional startup arguments. More...

def **destroy** ()
Shut down Cubit and close the active journal file. More...

def **set_cubit_message_handler** (hdlr)
Redirect Cubit output to a custom message handler. More...

def **get_cubit_message_handler** ()
Retrieve the current Cubit message handler. More...

def **set_exit_handler** (hdlr)
Set a custom exit callback for Cubit. More...

def **set_progress_handler** (progress)
Register a progress-bar callback handler with Cubit. More...

def **replace_progress_handler** (progress)
Replace the current progress-bar callback handler and return the old one. More...

def **set_cubit_interrupt** (interrupt)
Enable or disable interruptible operations in Cubit. More...

System Control and Data

def **set_playback_paused_on_error** (pause)
Configure whether playback pauses on error. More...

def **is_playback_paused_on_error** ()
Query whether playback is paused on error. More...

def **pause_playback** ()
Pause journal playback immediately. More...

def **stop_playback** ()

Stop journal playback entirely. More...

def **resume_playback** ()
Resume a paused journal playback. More...

def **is_playback_paused** ()
Check if journal playback is currently paused. More...

def **developer_commands_are_enabled** ()
Check if developer commands are enabled. More...

def **get_interface** (interface_name)
Retrieve a Cubit interface by name. More...

def **release_interface** (instance)
Release a previously retrieved Cubit interface. More...

def **add_filename_to_recent_file_list** (filename)
Add a filename to Cubit's recent-file list in the GUI File menu. More...

def **get_version** ()
Retrieve the current Cubit version string. More...

def **get_revision_date** ()
Retrieve the Cubit revision date. More...

def **get_build_number** ()
Retrieve the Cubit build number. More...

def **get_acis_version** ()
Retrieve the ACIS kernel version string. More...

def **get_acis_version_as_int** ()
Retrieve the ACIS kernel version as an integer. More...

def **get_sgm_version** ()
Retrieve the SGM (Solid Geometry Manager) version. More...

def **get_exodus_version** ()
Retrieve the Exodus mesh library version. More...

def **get_meshgems_version** ()
Retrieve the MeshGems library version. More...

def **get_graphics_version** ()
Retrieve the VTK (Visualization Toolkit) version used by Cubit. More...

def **get_python_version** ()
Retrieve the Python interpreter version used by Cubit. More...

def **get_default_geometry_engine** ()
Get the name of the default modeler engine. More...

def **is_catia_engine_available** ()
Determine whether the CATIA geometry engine is available. More...

def **is_acis_engine_available** ()
Determine whether the ACIS geometry engine is available. More...

def **is_opencascade_engine_available** ()
Determine whether the OpenCASCADE geometry engine is available. More...

def **is_special_build** (build_type)
Check if a special build type is available. More...

def **print_cmd_options** ()
Display Cubit's supported startup options. More...

def **is_modified** ()
Check if the model has been modified since import or last save. More...

def **set_modified** ()
Reset the model's modified status to "unmodified". More...

def **is_undo_save_needed** ()
Check if the model requires an undo checkpoint save. More...

def **set_undo_saved** ()
Clear the undo-needed flag for the model. More...

def **is_performing_undo** ()
Query whether an undo operation is currently being performed. More...

def **was_last_cmd_undoable** ()
Report whether the last executed command was undoable. More...

def **get_undo_enabled** ()
Query whether undo is currently enabled. More...

def **number_undo_commands** ()
Query the number of undoable commands in the stack. More...

def **is_command_echoed** ()
Check whether Cubit echoes commands to the console. More...

def **get_command_from_history** (command_number)
Retrieve a specific command from Cubit's history buffer. More...

def **get_command_history_count** ()
Get the number of commands in Cubit's history buffer. More...

def **get_next_command_from_history** ()
Get the next command from the history buffer. More...

def **get_previous_command_from_history** ()
Get the previous command from the history buffer. More...

def **write_to_journal** (words)
Append a custom entry to Cubit's journal and recording streams. More...

def **journal_commands** (state)
Enable or disable journaling of Cubit commands. More...

def **is_command_journaled** ()
Query whether Cubit is journaling commands. More...

def **get_current_journal_file** ()
Get the filename of the current journal file. More...

def **is_working_dir_set** ()
Check if the "-workingdir" option was provided at Cubit startup. More...

def **app_util** ()
Return Cubit's AppUtil interface for global services. More...

def **cgm_iface** ()
Return the CGM-based geometry interface for Cubit. More...

def **cgm** ()
Return the raw CGMApp kernel instance used by Cubit. More...

Command Utilities

def **cmd** (input_string)
Execute a raw Cubit command string immediately (modifies model state). More...

def **silent_cmd** (input_string)
Execute a Cubit command without echoing or verbose output. More...

def **print_info** (message)
Print a message through Cubit's messaging system. More...

def **parse_cubit_list** (type, entity_list_string)
Parse a Cubit-style entity selection expression into a list of IDs. More...

def **parse_locations** (location_str)
Parse a Cubit location specification into concrete 3D coordinates. More...

def **string_from_id_list** (ids)

Convert a list of integers into a compact Cubit-style ID string. More...

def **get_id_string** (entity_ids, sort=True)
Convert a list of integers into a compact Cubit-style ID range string. More...

def **print_raw_help** (input_line, order_dependent, consecutive_dependent)
Display context-sensitive help while typing commands. More...

def **get_error_count** ()
Retrieve the total number of errors in the current Cubit session. More...

def **get_cubit_digits_setting** ()
Retrieve the current Cubit "digits" setting. More...

def **is_part_of_list** (target_id, id_list)
Check if an ID is present in a list of IDs. More...

def **get_last_id** (entity_type)
Get the ID of the last created entity of the given type. More...

def **entity_exists** (entity_type, id)
Check whether an entity of the specified type and ID exists. More...

def **get_idless_signature** (entity_type, entity_id)
Get the idless signature of a geometric or mesh entity. More...

def **get_idless_signatures** (entity_type, entity_id_list)
Get the idless signatures of a range of geometric or mesh entities. More...

def **get_aprepro_vars** ()
Retrieve the current Aprepro variable names. More...

def **get_aprepro_value_as_string** (variable_name)
Get the string value of an Aprepro variable. More...

def **get_aprepro_numeric_value** (variable_name)
Get the numeric value of a specified Aprepro variable. More...

Graphics Viewing

def **get_view_distance** ()
Get the camera's distance between its position and focus point. More...

def **get_view_at** ()
Get the camera's current "at" (target) point. More...

def **get_view_from** ()
Get the camera's current 'from' (position) point. More...

def **get_view_up** ()
Get the camera's 'up' direction vector. More...

def **reset_camera** ()
Reset the camera view and clear auxiliary graphics windows. More...

def **flush_graphics** ()
Force immediate rendering of pending graphics operations. More...

def **is_perspective_on** ()
Check if perspective projection is enabled. More...

def **is_occlusion_on** ()
Check if occlusion is enabled. More...

def **is_scale_visibility_on** ()
Check if the scale annotation is visible in the graphics window. More...

def **is_mesh_visibility_on** ()
Check if mesh graphics are visible in the graphics window. More...

def **is_geometry_visibility_on** ()

Check if geometry graphics are visible in the graphics window. More...

def **get_rendering_mode** ()
Get the current graphics rendering mode. More...

def **set_rendering_mode** (mode)
Set the current graphics rendering mode (equivalent to "Graphics Mode <option>"). More...

def **set_label_type** (entity_type, label_flag)
Sets label display type for a given entity type. More...

def **get_label_type** (entity_type)
Gets the current label display type for a given entity type. More...

Graphics Entity Selections

def **clear_drawing_set** (set_name)
Clear all geometry in a named drawing set (e.g., mesh preview). More...

def **unselect_entity** (entity_type, entity_id)
Unselect an entity that is currently selected. More...

def **get_rubberband_shape** ()
Get the current rubberband selection shape. More...

def **is_select_partial_on** ()
Check if partial selection is enabled. More...

def **clear_highlight** ()
Clear all entity highlights. More...

def **clear_preview** ()
Clear preview graphics without affecting other display settings. More...

def **highlight** (entity_type, entity_id)
Highlight the given entity in the graphics window without selecting it. More...

def **get_selected_ids** ()
Retrieve all currently selected entity IDs in pick order. More...

def **get_selected_id** (index)
Get the selected entity ID by index. More...

def **get_selected_type** (index)
Get the selected entity type by index. More...

def **get_pick_type** ()
Get the current pick mode for entity selection. More...

def **set_pick_type** (pick_type, silent=False)
Set the current pick mode for entity selection. More...

def **set_filter_types** (num_types, filter_types)
Set multiple pick filter types for graphics selections. More...

def **add_filter_type** (filter_type)
Add an entity type to the graphics pick filter. More...

def **remove_filter_type** (filter_type)
Remove an entity type from the graphics pick filter. More...

def **is_type_filtered** (filter_type)
Check if a specific entity type is currently excluded from picking. More...

def **get_pick_filters** ()
Retrieve the list of entity types currently allowed for picking. More...

def **clear_picked_list** ()
Clear the list of currently picked entities. More...

def **step_next_possible_selection** ()
Advance to the next entity in the current selection list. More...

def **step_previous_possible_selection** ()
Move back to the previous entity in the current selection list. More...

def **print_current_selections** ()
Print all selected entities and their types. More...

def **print_currently_selected_entity** ()
Print details of the current selected entity. More...

def **current_selection_count** ()
Get the number of entities in the current selection list. More...

Meshing Schemes Support

def **get_default_element_type** ()
Retrieve the current default element type for meshing. More...

def **is_volume_meshable** (volume_id)
Check if a volume is meshable under the current meshing scheme. More...

def **is_surface_meshable** (surface_id)
Check if a surface is meshable under the current meshing scheme. More...

def **get_vertex_type** (surface_id, vertex_id)
Retrieve the vertex type for a given vertex on a surface. More...

def **get_submap_corner_types** (surface_id)
Get a list of vertex IDs and their corner-type codes for a surface submap. More...

def **get_curve_bias_type** (curve_id)
Retrieve the bias scheme type applied to a curve. More...

def **get_curve_bias_geometric_factor** (curve_id)
Get the primary geometric progression factor used for biasing a curve. More...

def **get_curve_bias_geometric_factor2** (curve_id)
Get the secondary geometric progression factor for a dual-bias curve. More...

def **get_curve_bias_first_interval_length** (curve_id)
Query the length of the first interval on a biased curve. More...

def **get_curve_bias_first_interval_fraction** (curve_id)
Retrieve the fraction of curve length used as the first interval size. More...

def **get_curve_bias_fine_size** (curve_id)
Get the "fine size" parameter for a biased curve, if set. More...

def **get_curve_bias_coarse_size** (curve_id)
Get the "coarse size" parameter for a biased curve, if set. More...

def **get_curve_bias_first_last_ratio1** (curve_id)
Retrieve the ratio of first-to-last interval at the start of a biased curve. More...

def **get_curve_bias_first_last_ratio2** (curve_id)
Retrieve the ratio of first-to-last interval at the end of a biased curve. More...

def **get_curve_bias_last_first_ratio1** (curve_id)
Retrieve the ratio of last-to-first interval at the start of a biased curve. More...

def **get_curve_bias_last_first_ratio2** (curve_id)
Retrieve the ratio of last-to-first interval at the end of a biased curve. More...

def **get_curve_bias_from_start** (curve_id, value)
Determine if the bias is measured from the curve's start vertex. More...

def **get_curve_bias_from_start_set** (curve_id)
Check if the "bias from start" flag has been explicitly set on a curve. More...

def **get_curve_bias_start_vertex_id** (curve_id)
Retrieve the vertex ID designated as the start of bias on a curve. More...

def **get_curve_mesh_scheme_curvature** (curve_id)
Retrieve the curvature mesh scheme adaptation value for a curve. More...

def **get_curve_mesh_scheme_pinpoint_locations** (curve_id)
Retrieve the pinpoint mesh scheme locations for a curve. More...

def **get_mesh_scheme** (geometry_type, entity_id)
Retrieve the meshing scheme applied to a geometric entity. More...

def **get_smooth_scheme** (geometry_type, entity_id)
Retrieve the smoothing scheme for a specified geometry entity. More...

def **get_mesh_scheme_firmness** (geometry_type, entity_id)
Retrieve the meshing scheme firmness for a specified surface or volume. More...

def **get_tetmesh_proximity_flag** (volume_id)
Get the proximity-layer flag for tet meshing on a volume. More...

def **get_tetmesh_proximity_layers** (volume_id)
Get the number of proximity layers for tet meshing on a volume. More...

def **get_tetmesh_growth_factor** (volume_id)
Get the growth factor for tet meshing on a volume. More...

def **get_tetmesh_parallel** ()
Get the global parallel-tetmesher flag (HPC) setting. More...

def **get_tetmesh_num_anisotropic_layers** ()
Get the global number of anisotropic tet layers setting. More...

def **get_tetmesh_optimization_level** ()
Get the global tet meshing optimization level. More...

def **get_tetmesh_insert_mid_nodes** ()
Get the global flag indicating insertion of mid-edge (quadratic) nodes during tet meshing. More...

def **get_tetmesh_optimize_mid_nodes** ()
Get the global flag indicating optimization of mid-edge nodes during tet meshing. More...

def **get_tetmesh_optimize_overconstrained_tets** ()
Get the global flag indicating optimization of overconstrained tetrahedra. More...

def **get_tetmesh_optimize_overconstrained_edges** ()
Get the global flag indicating optimization of overconstrained edges. More...

def **get_tetmesh_minimize_slivers** ()
Get the global flag indicating minimization of sliver tetrahedra. More...

def **get_tetmesh_minimize_interior_points** ()
Get the global flag indicating minimization of interior points in tet meshing. More...

def **get_tetmesh_relax_surface_constraints** ()
Get the global flag indicating relaxation of surface mesh constraints in tet meshing. More...

def **get_mesh_geometry_approximation_angle** (geometry_type, entity_id)
Get the geometry approximation angle for TriMesh/TetMesh on a given entity. More...

def **get_trimesh_surface_gradation** ()
Retrieve the global surface mesh gradation for the TriMesh scheme. More...

def **get_trimesh_volume_gradation** ()

Retrieve the global volume mesh gradation for the TriMesh scheme. More...

def **get_trimesh_use_surface_proximity** ()
Retrieve the global surface proximity flag for the TriMesh scheme. More...

def **get_trimesh_surface_proximity_ratio** ()
Retrieve the global surface proximity ratio for the TriMesh scheme. More...

def **get_trimesh_target_min_size** (geometry_type, entity_id)
Retrieve the target minimum triangle size set on a specific geometry entity for the TriMesh scheme. More...

def **get_trimesh_geometry_sizing** ()
Retrieve the global geometry sizing flag for the TriMesh scheme. More...

def **get_trimesh_num_anisotropic_layers** ()
Retrieve the global number of anisotropic triangle layers for the TriMesh scheme. More...

def **get_trimesh_split_overconstrained_edges** ()
Retrieve the global flag for splitting overconstrained edges in the TriMesh scheme. More...

def **get_trimesh_tiny_edge_length** ()
Retrieve the global tiny edge length for the TriMesh scheme. More...

def **get_trimesh_ridge_angle** ()
Retrieve the global ridge angle for the TriMesh scheme. More...

def **get_node_constraint** ()
Query whether node constraint is enabled (move mid-nodes to geometry). More...

def **get_node_constraint_value** ()
Query the numeric value of the node constraint setting. More...

def **get_node_constraint_smart_threshold** ()
Query the current quality threshold for smart node constraint. More...

def **get_node_constraint_smart_metric** ()
Query the current quality metric for smart node constraint. More...

def **get_dbl_sculpt_default** (variable)
Retrieve the default value of a named sculpt parameter. More...

def **get_int_sculpt_default** (variable)
Retrieve default integer-valued sculpt parameter. More...

def **get_bool_sculpt_default** (variable)
Retrieve default boolean-valued sculpt parameter. More...

def **get_string_sculpt_default** (variable)
Retrieve default string-valued sculpt parameter. More...

Mesh Sizing Support

def **get_mesh_intervals** (geometry_type, entity_id)
Get the mesh interval count for a specified entity. More...

def **get_mesh_size** (geometry_type, entity_id)
Retrieve the effective target edge length for meshing an entity. More...

def **get_requested_mesh_size** (geometry_type, id)
Retrieve the mesh size explicitly set on a geometry entity. More...

def **has_valid_size** (geometry_type, entity_id)
Check whether a geometric entity has a valid size. More...

def **get_mesh_interval_firmness** (geometry_type, entity_id)
Retrieve the interval firmness for a specified geometry entity. More...

def **get_requested_mesh_interval_firmness** (geometry_type, entity_id)
Retrieve the explicitly requested interval firmness for a specified entity. More...

def **get_mesh_size_type** (geometry_type, entity_id)
Retrieve the mesh size setting type for a specified entity. More...

def **get_requested_mesh_size_type** (geometry_type, entity_id)
Retrieve the mesh size setting type explicitly requested on a specified entity. More...

def **auto_size_needs_to_be_calculated** ()
Check if automatic mesh sizing is outdated and needs recomputation. More...

def **get_default_auto_size** ()
Compute Cubit's heuristic default automatic mesh size for the current model. More...

def **get_requested_mesh_intervals** (geometry_type, entity_id)
Retrieve the mesh interval count explicitly set on a geometry entity. More...

def **is_interval_count_odd** (surface_id)
Check if any loop on a surface has an odd number of mesh intervals. More...

def **get_auto_size** (geometry_type, entity_id_list, auto_factor)
Predict the automatic mesh size for a set of entities. More...

def **get_element_budget** (element_type, entity_id_list, auto_factor)
Estimate the total element count for a set of volumes given size settings. More...

def **get_exodus_sizing_function_variable_name** ()
Retrieve the Exodus-based sizing function variable name. More...

def **get_exodus_sizing_function_file_name** ()
Retrieve the Exodus sizing function file name. More...

def **get_sizing_function_name** (entity_type, entity_id)
Retrieve the meshing sizing function type for a surface or volume. More...

def **exodus_sizing_function_file_exists** ()
Check if the Exodus sizing function file currently exists. More...

Mesh Query Support

def **is_meshed** (geometry_type, entity_id)
Determine whether a specified geometry entity has been meshed. More...

def **get_hex_count** ()
Retrieve the count of hexahedral elements in the current model. More...

def **get_pyramid_count** ()
Retrieve the count of pyramid elements in the current model. More...

def **get_tet_count** ()
Retrieve the count of tetrahedral elements in the current model. More...

def **get_quad_count** ()
Retrieve the count of quadrilateral elements in the current model. More...

def **get_tri_count** ()
Retrieve the count of triangular elements in the current model. More...

def **get_edge_count** ()
Retrieve the count of edges between surface elements in the current model. More...

def **get_sphere_count** ()
Retrieve the count of sphere elements in the current model. More...

def **get_wedge_count** ()
Retrieve the count of wedge-shaped elements in the current model. More...

def **get_node_count** ()
Retrieve the count of nodes in the current model. More...

def **get_element_count** ()
Retrieve the count of exportable elements in the current model. More...

def **get_volume_element_count** (volume_id)
Retrieve the count of 3D elements in a specified volume. More...

def **get_surface_element_count** (surface_id)
Retrieve the count of elements on a specified surface. More...

def **volume_contains_tets** (volume_id)
Determine whether a specified volume contains any tetrahedral elements. More...

def **get_hex_sheet** (node_id_1, node_id_2)
Retrieve the IDs of all hexahedral elements forming a hex sheet through two nodes. More...

def **get_mesh_edge_length** (edge_id)
Compute the length of a specified mesh edge. More...

def **get_meshed_volume_or_area** (geometry_type, entity_ids)
Sum mesh volumes or surface areas for CAD entities or mesh elements. More...

def **get_elem_quality_stats** (entity_type, id_list, metric_name, single_threshold, use_low_threshold, low_threshold, high_threshold, make_group)
Python-friendly version of get_quality_stats without reference parameters. More...

def **get_quality_stats_at_geometry** (geom_type, mesh_type, geom_id_list, expand_levels, metric_name, single_threshold, use_low_threshold, low_threshold, high_threshold, make_group)
Python-friendly version of get_quality_stats operating on geometry entities. More...

def **get_quality_value** (mesh_type, mesh_id, metric_name)
Retrieve a specific quality metric value for a single mesh element. More...

def **get_quality_values** (mesh_type, mesh_ids, metric_name)
Retrieve quality metric values for multiple mesh elements. More...

def **calculate_timestep_estimate** (entity_type, entity_ids)
Estimate the time step based on element sizes and material properties. More...

def **calculate_timestep_estimate_with_props** (entity_type, entity_id_list, density, youngs_modulus, poissons_ratio)
Estimate the stable time step using user-specified material properties. More...

def **get_target_timestep** ()
Return the current target time step threshold used in the density multiplier metric. More...

def **get_overconstrained_tets_in_volumes** (volumes)
Retrieve IDs of overconstrained tetrahedra within specified volumes. More...

def **best_edge_to_collapse_interior_node** (node_id)
Find the best edge to collapse to remove an interior node in a triangular mesh. More...

def **get_edges_to_swap** (curve_id)

Get edges on triangles at a knife-edge curve that are candidates for swapping. More...

def **get_n_largest_distances_between_meshes** (n, entity_type1, ids1, entity_type2, ids2)
Finds the N largest node-to-element distances between two meshes. More...

Geometry Topology Support

def **get_entities** (entity_type)
Retrieve IDs of all current entities of a specified type (geometry and mesh). More...

def **get_current_ids** (entity_type)
Retrieve IDs of all current geometry entities of a specified type. More...

def **get_body_count** ()
Get the current number of bodies in the model. More...

def **get_volume_count** ()
Get the current number of volumes in the model. More...

def **get_surface_count** ()
Get the current number of surfaces in the model. More...

def **get_curve_count** ()
Get the current number of curves in the model. More...

def **get_vertex_count** ()
Get the current number of vertices in the model. More...

def **get_curve_count_in_volumes** (target_volume_ids)
Get the current number of curves in the specified volumes. More...

def **get_relatives** (source_geometry_type, source_id, target_geom_type)
Get the relatives (parents or children) of a specified entity. More...

def **are_adjacent_surfaces** (surface_ids)
Return whether two or more surfaces share at least one manifold curve. More...

def **are_adjacent_curves** (curve_ids)
Return whether two or more curves share at least one manifold vertex. More...

def **get_adjacent_surfaces** (geometry_type, entity_id)
Get a list of surfaces adjacent to a specified surface (including the surface itself). More...

def **get_adjacent_volumes** (geometry_type, entity_id)
Get a list of adjacent volumes to a specified entity. More...

def **get_num_volume_shells** (volume_id)
Get the number of shells in a volume. More...

def **get_valence** (vertex_id)
Get the valence (number of incident edges) for a specific vertex. More...

def **get_list_of_free_ref_entities** (geometry_type)
Get all free (unattached) entities of a given geometry type. More...

def **get_owning_body** (geometry_type, entity_id)
Get the owning body for a specified entity. More...

def **get_owning_volume** (geometry_type, entity_id)
Get the owning volume for a specified entity. More...

def **get_owning_volume_by_name** (entity_name)
Get the owning volume for an entity by its name. More...

def **get_common_curve_id** (surface_1_id, surface_2_id)
Return a curve shared by two surfaces. More...

def **get_common_vertex_id** (curve_1_id, curve_2_id)
Return a vertex shared by two curves. More...

def **get_connected_surfaces** (surf_ids)
Find surfaces connected to a given set of surfaces. More...

def **gather_surfaces_by_orientation** (seed_surf_ids, all_surf_ids)
Gathers surfaces connected across shared edges, forming a surface enclosure. More...

Geometry Attribute Support

def **is_visible** (geometry_type, entity_id)
Query visibility for a specific geometry entity. More...

def **is_virtual** (geometry_type, entity_id)
Query whether a specified geometry entity is virtual. More...

def **contains_virtual** (geometry_type, entity_id)
Query whether any child entities of a specified geometry entity are virtual. More...

def **is_merged** (geometry_type, entity_id)
Check if a specified geometry entity has been merged into another. More...

def **get_merge_setting** (geometry_type, entity_id)
Get merge mode ("on", "off", or "auto") for a given entity. More...

def **get_source_surfaces** (volume_id)
Retrieve the list of sweep source surfaces for a specified volume. More...

def **get_target_surfaces** (volume_id)
Retrieve the list of sweep target surfaces for a specified volume. More...

def **get_entity_modeler_engine** (geometry_type, entity_id)
Get the modeler engine type for a specified entity. More...

def **get_entity_name** (entity_type, entity_id, no_default=False)

Get the name of a specified entity. More...

def **get_entity_names** (entity_type, entity_id, no_default=False, first_name_only=False)
Get all names associated with a specified entity. More...

def **set_entity_name** (entity_type, entity_id, new_name)
Set the name of a specified entity. More...

def **get_id_from_name** (name)
Retrieve the integer ID of an entity by its name. More...

def **get_all_ids_from_name** (geo_type, name)
Retrieve all IDs of entities of a specified geometry type whose names start with a given prefix. More...

def **get_entity_color** (entity_type, entity_id)
Get the color of a specified entity. More...

def **get_entity_color_index** (entity_type, entity_id)
Get the color index of a specified entity. More...

def **is_multi_volume** (body_id)
Check if a body contains multiple volumes. More...

def **is_sheet_body** (volume_id)
Determine if a volume is a sheet body (zero thickness). More...

Geometry Query Support

def **get_vertex_coordinates** (entityID)
Get the 3D coordinates of a vertex. More...

def **get_distance_between** (vertex_id_1, vertex_id_2)
Get the Euclidean distance between two vertices. More...

def **get_curve_type** (curve_id)
Get the curve type for a specified curve. More...

def **get_curve_length** (curve_id)
Get the 3D length of a specified curve. More...

def **get_arc_length** (curve_id)
Get the parametric (arc) length of a specified curve. More...

def **get_distance_from_curve_start** (x_coordinate, y_coordinate, z_coordinate, curve_id)
Get the distance along a curve from its start to the closest point on the curve. More...

def **get_curve_radius** (curve_id)
Compute the radius of a specified curve. More...

def **get_curve_center** (curve_id)
Retrieve the center point of a specified curve. More...

def **get_arc_center_radius** (curve_id)
Get the center point and radius of a specified arc curve. More...

def **evaluate_exterior_angle_at_curve** (curve_id, volume_id)
Return the exterior angle at a single curve with respect to a volume. More...

def **evaluate_exterior_angle** (curve_list, test_angle)
Find curves whose exterior angle between adjacent surfaces is less than a given threshold. More...

def **get_similar_curves** (curve_ids, tol=1e-3, use_percent_tol=True, on_similar_vols=True)
Find curves with lengths similar to a given curve. More...

def **get_surface_type** (surface_id)
Get the surface type for a specified surface. More...

def **get_surface_normal** (surface_id)
Get the unit normal vector at the center of a specified surface. More...

def **get_surface_normal_at_coord** (surface_id, coord)
Get the unit normal vector at a specified point on a surface. More...

def **get_surface_centroid** (surface_id)
Get the approximate centroid of a specified surface based on graphics faceting. More...

def **get_surface_sense** (surface_id)
Get the orientation ("sense") of a specified surface. More...

def **is_surface_planer** (surface_id)
Query whether a specified surface is planar. More...

def **is_surface_planar** (surface_id)

def **get_surface_area** (surface_id)
Get the area of a specified surface. More...

def **get_hydraulic_radius_surface_area** (surface_id)
Compute the hydraulic radius of a specified surface. More...

def **get_surface_principal_curvatures** (surface_id)
Get the principal curvatures of a surface at its midpoint. More...

def **evaluate_surface_angle_at_vertex** (surf_id, vert_id)
Return the interior angle at a vertex on a specified surface. More...

def **project_unit_square** (pts, surface_id, quad_id, node00_id, node10_id)
Map points in a unit square (u-v coordinates on a quad face) and project them onto a surface. More...

def **get_similar_surfaces** (surface_ids, tol=1e-3, use_percent_tol=True, on_similar_vols=True)
Find surfaces with similar area and curve count to given surfaces. More...

def **print_surface_summary_stats** ()
Print summary statistics for all surfaces in the model. More...

def **get_volume_area** (volume_id)
Get the total surface area of a specified volume. More...

def **get_volume_volume** (vol_id)
Get the enclosed volume of a specified volume. More...

def **get_hydraulic_radius_volume_area** (volume_id)
Compute the hydraulic radius of a specified volume. More...

def **get_similar_volumes** (volume_ids, tol=1e-3, use_percent_tol=True)
Find volumes with similar size and face count to given volumes. More...

def **get_bounding_box** (geometry_type, entity_id)
Get the axis-aligned bounding box for a specified entity. More...

def **get_total_bounding_box** (geometry_type, entity_list)
Get the combined bounding box for a list of entities. More...

def **get_tight_bounding_box** (geometry_type, entity_list)
Get the tight bounding box for a list of entities. More...

def **get_total_volume** (volume_list)
Get the total volume for a list of volumes. More...

def **print_volume_summary_stats** ()
Print summary statistics for all volumes in the model. More...

def **get_center_point** (entity_type, entity_id)
Get the 3D center or coordinates of a specified entity. More...

def **is_periodic** (geometry_type, entity_id)
Query whether a specified surface or curve is periodic. More...

def **get_geometry_type** (geom_type, ent_ids)
Return the analytic geometry type for a list of surfaces or curves. More...

def **get_distance_between_entities** (geom_type_1, entity_id_1, geom_type_2, entity_id_2)
Get the minimum distance between two geometry entities. More...

def **is_point_contained** (geometry_type, entity_id, xyz_point)
Determine if a point is inside, outside, on, or unknown relative to a given entity. More...

def **snap_locations_to_geometry** (locations, entity_type, entity_id, tol)
Snaps given XYZ locations to nearest points on specified entity. More...

def **measure_between_entities** (entity_type1, entity_id1, entity_type2, entity_id2)
Returns the shortest distance between two geometry entities and their closest points. More...

def **get_coordinate_systems_id_list** ()
Returns a list of coordinate system IDs. More...

Geometry-Mesh Entity Support

def **get_geometry_node_count** (entity_type, entity_id)
Get the number of mesh nodes on a geometric entity. More...

def **get_vertex_node** (vertex_id)
Get the node ID owned by a vertex. More...

def **get_curve_nodes** (curve_id)
Get the list of node IDs owned by a curve. More...

def **get_curve_edges** (curve_id)
Get the list of edge element IDs on a curve. More...

def **get_surface_nodes** (surface_id)
Get the list of node IDs owned by a surface. More...

def **get_surface_quads** (surface_id)
Get the list of quadrilateral (quad) element IDs on a surface. More...

def **get_surface_tris** (surface_id)
Get the list of triangle (tri) element IDs on a surface. More...

def **get_volume_nodes** (volume_id)
Get the list of node IDs owned by a volume. More...

def **get_volume_hexes** (volume_id)
Get the list of hexahedron (hex) IDs contained in a volume. More...

def **get_volume_tets** (volume_id)
Get the list of tetrahedron (tet) IDs contained in a volume. More...

def **get_volume_wedges** (volume_id)
Get the list of wedge IDs contained in a volume. More...

def **get_volume_pyramids** (volume_id)
Get the list of pyramid IDs contained in a volume. More...

def **get_surface_num_loops** (surface_id)
Get the number of loops on a surface. More...

def **get_surface_loop_nodes** (surface_id)
Get the ordered list of node IDs on the loops of a surface. More...

def **get_entity_sense** (source_type, source_id, sideset_id)
Get the sense of an entity in a sideset. More...

def **get_wrt_entity** (source_type, source_id, sideset_id)
Get the 'with-respect-to' entity of an item in a sideset. More...

def **get_geometric_owner** (mesh_entity_type, mesh_entity_list)
Get geometric owners for a set of mesh entities. More...

def **get_geometry_owner** (entity_type, entity_id)
Get the geometric owner of a mesh element. More...

def **get_all_geometric_owners** (mesh_entity_type, mesh_entity_list)
Get the list of geometric owners for a set of mesh entities and their child entities. More...

Mesh Element Queries

def **get_connectivity** (entity_type, entity_id)
Get the list of node IDs comprising a mesh element. More...

def **get_expanded_connectivity** (entity_type, entity_id)
Get the list of node IDs for a mesh element, including interior nodes. More...

def **get_sub_elements** (entity_type, entity_id, dimension)
Get the lower-dimensional entities of a higher-dimensional mesh element. More...

def **get_node_exists** (node_id)
Check whether a node exists in the model. More...

def **get_exodus_id** (entity_type, entity_id)
Get the Global Element ID for a mesh entity. More...

def **get_element_exists** (element_id)
Check whether a global mesh element ID exists in the model. More...

def **get_element_type** (element_id)
Get the specific mesh element type for a global element ID. More...

def **get_element_type_id** (element_id)
Map a Global Element ID back to its local mesh entity ID. More...

def **get_element_block** (element_id)
Get the block ID containing a given global element. More...

def **get_global_element_id** (element_type, id)
Alias for get_exodus_id: get the Global Element ID for a local mesh entity. More...

def **get_hex_global_element_id** (hex_id)
Get the Global Element ID for a specific hexahedral element. More...

def **get_tet_global_element_id** (tet_id)
Get the Global Element ID for a specific tetrahedral element. More...

def **get_wedge_global_element_id** (wedge_id)
Get the Global Element ID for a specific wedge element. More...

def **get_pyramid_global_element_id** (pyramid_id)
Get the Global Element ID for a specific pyramid element. More...

def **get_tri_global_element_id** (tri_id)
Get the Global Element ID for a specific triangular element. More...

def **get_quad_global_element_id** (quad_id)
Get the Global Element ID for a specific quadrilateral element. More...

def **get_edge_global_element_id** (edge_id)
Get the Global Element ID for a specific edge element. More...

def **get_sphere_global_element_id** (sphere_id)
Get the Global Element ID for a specific node (SPHERE element). More...

def **get_node_global_id** (node_id)
Get the global node ID assigned in the Exodus file for a mesh node. More...

def **get_closest_node** (x_coordinate, y_coordinate, z_coordinate)
Find the mesh node closest to a given point. More...

def **get_nodal_coordinates** (node_id)
Get the coordinates of a mesh node. More...

def **get_node_faces** (node_id)
Get the IDs of all quadrilateral elements (faces) adjacent to a node. More...

def **get_node_tris** (node_id)
Get the IDs of all triangular elements adjacent to a node. More...

def **get_node_edges** (node_id)
Get the IDs of all edge elements adjacent to a node. More...

def **get_node_position_fixed** (node_id)
Query whether a mesh node is fixed (constrained against smoothing). More...

def **get_mesh_element_type** (entity_type, entity_id)
Get the mesh element type applied to a geometric entity. More...

Geometry Defeaturing Support

def **get_small_curves** (target_volume_ids, mesh_size)
Find curves with edge length below a threshold within given volumes. More...

def **get_smallest_curves** (target_volume_ids, number_to_return)
Return IDs of the smallest curves in the specified volumes. More...

def **get_small_surfaces** (target_volume_ids, area_threshold)
Find surfaces with area below a given threshold. More...

def **get_small_surfaces_HR** (target_volume_ids, mesh_size)
Python-callable version: identify small hydraulic-radius surfaces. More...

def **is_narrow_surface** (surface_id, mesh_size)
Determine if any two non-adjacent edges on the surface form a narrow region. More...

def **get_narrow_surfaces** (target_volume_ids, mesh_size)
Find surfaces with narrow regions in specified volumes. More...

def **get_small_and_narrow_surfaces** (target_ids, small_area, small_curve_size)

Find surfaces that are either small in area or contain narrow regions. More...

def **get_closed_narrow_surfaces** (target_ids, narrow_size)
Find faces on closed surfaces whose two boundary edges remain within a distance threshold. More...

def **get_narrow_regions** (target_ids, narrow_size)
Find faces containing narrow regions within specified volumes. More...

def **get_surfs_with_narrow_regions** (target_ids, narrow_size)
Find surfaces with narrow regions based on edge-pair orientation and proximity. More...

def **is_close_loop_surface** (surface_id, mesh_size)
Check if a surface has multiple loops closer than a given threshold. More...

def **get_close_loops** (target_volume_ids, mesh_size)
Find faces with multiple loops closer than a given threshold. More...

def **get_close_loops_with_thickness** (target_volume_ids, mesh_size, genus)
Find faces with multiple loops closer than a threshold and return their minimum loop separations. More...

def **get_close_loop_thickness** (surface_id)
Compute the minimum separation distance between loops on a surface. More...

def **get_small_volumes** (target_volume_ids, mesh_size)
Find volumes whose size is below a threshold based on mesh size. More...

def **is_chamfer_surface** (surface_id, thickness_threshold)
Determine if a face's underlying surface is a chamfer. More...

def **get_chamfer_surfaces** (target_volume_ids, thickness_threshold)
Get the list of chamfer surfaces for a list of volumes. More...

def **get_chamfer_chains** (surface_id)
Returns the chamfer chains for a given surface. More...

def **get_chamfer_chain_collections** (volume_list, thickness_threshold)
Return collections of surfaces that form chamfer chains in the specified volumes. More...

def **is_cylinder_surface** (surface_id)
Determine if a given face's underlying surface is a circular cylinder. More...

def **is_blend_surface** (surface_id)
Determine if a face's underlying surface is a blend (fillet). More...

def **get_blend_surfaces** (target_volume_ids)
Find blend (fillet) surfaces within specified volumes. More...

def **get_small_radius_blend_surfaces** (target_volume_ids, max_radius)
Find blend surfaces with radius of curvature <= max_radius. More...

def **get_blend_chains** (surface_id)
Returns the blend chains for a given surface. More...

def **get_blend_chain_collections** (volume_list, radius_threshold)
Return collections of surfaces that form blend chains in the specified volumes. More...

def **is_cone_surface** (surface_id)
Determine whether a surface is a cone. More...

def **find_cone_surfaces** (surface_id)
Identify cone surface(s) starting from a candidate surface. More...

def **get_cone_surfaces** (target_volume_ids)
Identify cone surfaces in a set of volumes. More...

def **get_surface_cone_collections** (volume_list, radius_threshold=0.0)
Identify collections of surfaces that comprise cones in specified volumes. More...

def **is_cavity_surface** (surface_id)
Return whether the specified surface is part of a cavity. More...

def **get_cavity_surfaces** (surface_id)
Return the surfaces in the cavity adjacent to the specified surface. More...

def **get_surface_cavity_collections** (volume_list, area_threshold=-1, angle_tolerance=-1, combine_cavities=True)
Return collections of surfaces that form cavities in the specified volumes. More...

def **is_hole_surface** (surface_id, radius_threshold)
Return whether the specified surface is part of a hole. More...

def **get_hole_surfaces** (surface_id)
Return the surfaces in the hole adjacent to the specified surface. More...

def **get_surface_hole_collections** (volume_list, radius_threshold)
Return collections of surfaces that form holes in the specified volumes. More...

def **is_continuous_surface** (surface_id, angle_tol)
Return whether the surface has any adjacent continuous surfaces. More...

def **get_continuous_surfaces** (surface_id, angle_tol)
Return the list of adjacent continuous surfaces. More...

def **get_continuous_curves** (curve_id, angle_tol, require_two_valent=False)

Return the list of adjacent continuous curves. More...

def **get_continuous_curve_collections** (volume_list, angle_tolerance=-1)
Returns collections of continuous curves in the given volumes. More...

def **get_sharp_angle_vertices** (target_volume_ids, upper_bound, lower_bound)
Identify vertices at sharp curve angles in a set of volumes. More...

def **get_blunt_tangency_default_depth** (vert_id, angle, add_material)
Computes the default depth used to blunt a tangency at a vertex. More...

def **get_tangential_intersections** (target_volume_ids, upper_bound, lower_bound)
Find surfaces with tangential intersection angles outside specified bounds. More...

def **get_coincident_vertices** (target_volume_ids, high_tolerance)
Find vertex pairs within a specified tolerance across given volumes. More...

def **get_close_vertex_curve_pairs** (target_volume_ids, high_tolerance)
Find pairs of vertices and curves within a specified tolerance across given volumes. More...

def **get_overlapping_surfaces_in_bodies** (body_ids, filter_slivers=False)
Identify overlapping surfaces between different volumes in a set of bodies. More...

def **find_overlapping_curves** (curve_ids)
Identify overlapping curves in a specified list of curves. More...

def **get_gaps_between_volumes** (target_volume_ids, maximum_gap_tolerance, maximum_gap_angle, cache_overlaps=0)
Identify gaps between surfaces in a list of volumes. More...

def **get_overlapping_volumes** (target_volume_ids)
Identify overlapping volumes in a list of volumes. More...

def **get_overlapping_volumes_at_volume** (volume_id, compare_volumes)
Identify volumes in the model that overlap a single target volume. More...

def **get_overlapping_surfaces_at_surface** (surface_id, compare_volumes, cache_overlaps=0)
Identify surfaces in the model that overlap a single target surface. More...

def **get_overlap_max_gap** ()
Get the current maximum gap tolerance used for calculating surface overlaps. More...

def **set_overlap_max_gap** (maximum_gap)
Set the maximum gap tolerance for calculating surface overlaps. More...

def **get_overlap_min_gap** ()
Get the current minimum gap tolerance used for calculating surface overlaps. More...

def **set_overlap_min_gap** (min_gap)
Set the minimum gap tolerance for calculating surface overlaps. More...

def **get_overlap_max_angle** ()
Get the current maximum angle tolerance used for calculating surface overlaps. More...

def **set_overlap_max_angle** (maximum_angle)
Set the maximum angle tolerance for calculating surface overlaps. More...

def **get_nearby_entities** (gtype, ent_ids, compare_ents, distance)
Identify nearby entities of type curve, surface, or volume for a given list of the same type. More...

def **get_nearby_volumes_at_volume** (volume_id, compare_volumes, distances, progress=None)
Identify nearby volumes in the model for a list of target volumes. More...

def **get_unmerged_curves_on_shells** (shell_vols, thickness)
Identify unmerged curves between sheet (shell) volumes for use in shell construction workflows. More...

def **get_mergeable_vertices** (target_volume_ids)
Get the list of mergeable vertices from a list of volumes or bodies. More...

def **get_mergeable_curves** (target_volume_ids)
Get the list of mergeable curves from a list of volumes or bodies. More...

def **get_mergeable_surfaces** (target_volume_ids)
Get the list of mergeable surfaces from a list of volumes or bodies. More...

def **get_merge_tolerance** ()
Return the current merge tolerance value. More...

Geometry ITEM Solutions

def **get_solutions_for_close_loop** (surface_id, mesh_size)
Retrieve recommended remedies for a close-loop surface to resolve narrow regions. More...

def **get_solutions_for_near_coincident_vertices** (vertex_id_1, vertex_id_2)
Provide remedies for two nearly coincident vertices on different volumes to resolve gaps or misalignments. More...

def **get_solutions_for_bad_geometry** (geom_type, geom_id)
Provide remedy for bad geometry via ACIS healing (deprecated). More...

def **get_solutions_for_overlapping_volumes** (volume_id_1, volume_id_2, maximum_gap_tolerance, maximum_gap_angle)
Provide remedies for two overlapping volumes based on gap and angular tolerances. More...

def **get_solutions_for_overlapping_surfaces** (surface_id_1, surface_id_2)
Suggests remedies for two overlapping surfaces on different volumes. More...

def **get_volume_gap_solutions** (surface_id_1, surface_id_2)
Suggests remedies for a gap between two surfaces on different volumes. More...

def **get_solutions_for_near_coincident_vertex_and_curve** (vertex_id, curve_id)
Suggests remedies when a vertex lies nearly on a curve from a different volume. More...

def **get_solutions_for_near_coincident_vertex_and_surface** (vertex_id, surface_id)
Suggests remedies when a vertex lies nearly on a surface from a different volume. More...

def **get_solutions_for_imprint_merge** (surface_id1, surface_id2)
Suggests remedies for imprint/merge when two overlapping surfaces are on different volumes. More...

def **get_solutions_for_volumes** (vol_id, small_curve_size, mesh_size)
Recommend remediation operations for volumes with features below a size threshold. More...

def **get_solutions_for_classified_volume** (classification, vol_id)
Suggests geometry fixes or feature operations for a volume based on its classification. More...

def **get_solutions_for_bolt** (bolt_id, insert_id, threaded_vol_id)
Suggests operations for a volume classified as a bolt using known subcomponent IDs. More...

def **get_solutions_for_bolt_hole** (bearing_hole, threaded_holes)
Suggests repair or modification options for a set of concentric fastener pilot holes. More...

def **get_solutions_for_classified_surface** (classification, surf_id)
Suggests remedies or modifications for a classified surface. More...

def **get_solutions_for_thin_volume** (vol_id, near_vols, include_weights=False, include_type=False)
Generate candidate operations to reduce a thin volume into a sheet body. More...

def **get_solutions_for_sheet_volumes** (vol_ids, thickness)
Propose surface-extension and imprint/merge operations to connect sheet bodies. More...

def **get_solutions_for_sheet_volume_connection** (args, kwargs)
Propose surface-tweak, imprint, and merge operations to bridge two sheet-volume sets. More...

def **get_solutions_for_small_surfaces** (surface_id, small_curve_size, mesh_size)
Recommend remedial operations on a surface already flagged as small. More...

def **get_solutions_for_small_curves** (curve_id, small_curve_size, mesh_size)
Recommend remedial operations on a curve already flagged as small. More...

def **get_solutions_for_sharp_angle_vertex** (vertex_id, small_curve_size, mesh_size)
Retrieve recommended remedies for a sharp angle at a vertex. More...

def **get_solutions_for_surfaces_with_narrow_regions** (surface_id, small_curve_size, mesh_size)
Recommend operations to eliminate narrow regions on a surface identified as narrow. More...

def **get_solutions_for_cone_surface** (surface_id)
Suggests remedies for a surface classified as a cone. More...

def **get_solutions_for_decomposition** (volume_list, exterior_angle, do_imprint_merge, tolerant_imprint)
Provide possible decomposition solutions for volumes based on exterior angle criteria. More...

def **get_solutions_for_blends** (surface_id)
Provide possible blend removal solutions for a given surface (and its blend chain if applicable). More...

def **get_solutions_for_cavity_surface** (surface_id)
Provide possible cavity removal or adjustment solutions for a given surface (and its cavity if applicable). More...

def **get_mesh_error_solutions** (error_code)
Retrieve recommended solutions and context cues for a mesh error. More...

Geometry Idealization Support

def **get_reduce_bolt_core_default_dimensions** (vol_id)
Computes default core dimensions for a bolt volume in reduce bolt core operation. More...

def **get_bolt_diameters** (vol_ids)
Get the shank diameters of specified bolt volumes. More...

def **get_bolt_axis** (vol_id)
Get the normalized axis vector of a bolt volume. More...

def **get_bolt_shigley_radius** (bolt_id, angle, washer_id=0)
Get the equivalent Shigley frustum radius at the bolt interface. More...

def **get_bolt_washer** (bolt_id)
Get the washer volume ID associated with a given bolt. More...

def **get_bolt_coordinate_system** (geom_type, id)
Get local coordinate system for a bolt or bolt hole. More...

def **get_bolt_clamped_members** (vol_ids, nearby_vols, progress=None)
Identify clamped members associated with a bolt. More...

def **get_bolts_in_clamped_members** (geo_type, clamped_vols, candidate_bolts)
Identify bolts clamping the given volumes or blocks. More...

def **get_bolt_holes_info** (geo_type, clamped_members, radius_threshold, gap_threshold)
Identify concentric pilot holes across clamped members. More...

def **get_bolt_holes** (geo_type, clamped_members, radius_threshold, gap_threshold)
Identify upper and lower pilot hole surfaces from clamped members. More...

def **get_2D_sheet_volumes** (vol_id)
Get associated 2D sheet volumes from a reduced 3D thin volume. More...

def **get_3D_thin_volume** (vol_id)
Get the original 3D thin volume associated with a 2D sheet volume. More...

Blocks, Sidesets, and Nodesets

def **get_block_count** ()
Get the current number of element blocks in the model. More...

def **get_block_id** (entity_type, entity_id)
Get the element block ID associated with a geometric entity. More...

def **get_block_ids** (mesh_geometry_file_name)
Get list of block IDs from a mesh geometry file. More...

def **get_block_id_list** ()
Get a list of all active block IDs. More...

def **get_next_block_id** ()
Get the next available block ID. More...

def **get_block_volumes** (block_id)
Get the list of volume IDs contained in a block. More...

def **get_block_surfaces** (block_id)
Get the list of surface IDs contained in a block. More...

def **get_block_curves** (block_id)
Get the list of curve IDs contained in a block. More...

def **get_block_vertices** (block_id)
Get the list of vertex IDs contained in a block. More...

def **get_block_nodes** (block_id)
Get the list of node IDs contained in a block. More...

def **get_block_edges** (block_id)
Get the list of edge IDs contained in a block. More...

def **get_block_tris** (block_id)
Get the list of triangle (tri) IDs contained in a block. More...

def **get_block_faces** (block_id)
Get the list of face IDs contained in a block. More...

def **get_block_pyramids** (block_id)
Get the list of pyramid IDs contained in a block. More...

def **get_block_wedges** (block_id)
Get the list of wedge IDs contained in a block. More...

def **get_block_tets** (block_id)
Get the list of tetrahedron (tet) IDs contained in a block. More...

def **get_block_hexes** (block_id)
Get the list of hexahedron (hex) IDs contained in a block. More...

def **get_block_element_type** (block_id)
Get the element type associated with a block. More...

def **get_block_attribute_count** (block_id)
Get the number of attributes assigned to a block. More...

def **get_block_element_attribute_count** (block_id)
Get the number of attributes defined for elements in the specified block. More...

def **get_block_attribute_value** (block_id, attribute_index)
Get the value of a specific attribute for a block. More...

def **get_block_attribute_name** (block_id, attribute_index)
Get the name of a specific attribute for a block. More...

def **get_block_element_attribute_names** (block_id)
Get the list of attribute names associated with block elements. More...

def **get_valid_block_element_types** (block_id)
Get a list of valid element types for the specified block. More...

def **get_block_material** (block_id)
Get the ID of the material assigned to the specified block. More...

def **get_blocks_with_materials** ()
Get all block IDs and their associated material IDs. More...

def **get_sideset_count** ()
Get the current number of sidesets in the model. More...

def **get_sideset_id_list** ()

Get a list of all active sideset IDs. More...

def **get_next_sideset_id** ()

Get the next available sideset ID. More...

def **get_sideset_curves** (sideset_id)
Get the list of curve IDs associated with a sideset. More...

def **get_sideset_edges** (sideset_id)
Get the list of edge IDs contained in a sideset. More...

def **get_sideset_surfaces** (sideset_id)
Get the list of surface IDs contained in a sideset. More...

def **get_sideset_quads** (sideset_id)
Get the list of quadrilateral (quad) element IDs contained in a sideset. More...

def **get_sideset_tris** (sideset_id)
Get the list of triangle (tri) element IDs contained in a sideset. More...

def **get_sideset_area** (sideset_id)
Get the total area of a sideset. More...

def **get_sideset_element_type** (sideset_id)
Get the element type of a sideset. More...

def **get_nodeset_count** ()
Get the current number of nodesets in the model. More...

def **get_nodeset_id_list** ()
Get a list of all active nodeset IDs. More...

def **get_next_nodeset_id** ()
Get the next available nodeset ID. More...

def **get_nodeset_volumes** (nodeset_id)
Get the list of volume IDs associated with a nodeset. More...

def **get_nodeset_surfaces** (nodeset_id)
Get the list of surface IDs associated with a nodeset. More...

def **get_nodeset_curves** (nodeset_id)
Get the list of curve IDs associated with a nodeset. More...

def **get_nodeset_vertices** (nodeset_id)
Get the list of vertex IDs associated with a nodeset. More...

def **get_nodeset_nodes** (nodeset_id)
Get the list of node IDs explicitly assigned to a nodeset. More...

def **get_nodeset_nodes_inclusive** (nodeset_id)
Get the list of node IDs associated with a nodeset (inclusive). More...

def **get_nodeset_node_count** (nodeset_id)
Get the number of nodes in a nodeset. More...

def **get_exodus_element_count** (entity_id, entity_type)
Get the number of elements in a Exodus entity. More...

def **get_exodus_entity_name** (entity_type, entity_id)
Get the name of an Exodus entity. More...

def **get_exodus_entity_type** (entity_type, entity_id)
Get the Exodus type of an entity. More...

def **get_exodus_entity_description** (entity_type, entity_id)
Get the description of an Exodus entity. More...

def **get_all_exodus_times** (filename)
Get all available time steps from an Exodus file. More...

def **get_exodus_variable_names** (container_type, container_id)
Get the names of Exodus variables in a nodeset, sideset, or block. More...

def **get_all_exodus_variable_names** (filename, variable_type)
Get all variable names of a given type from an Exodus file. More...

def **get_exodus_variable_count** (container_type, container_id)
Get the number of Exodus variables in a nodeset, sideset, or block. More...

def **get_copy_nodeset_on_geometry_copy_setting** ()

Get the current nodeset behavior setting for geometry copy. More...

def **get_copy_sideset_on_geometry_copy_setting** ()
Get the current sideset behavior setting for geometry copy. More...

def **get_copy_block_on_geometry_copy_setting** ()
Get the current block behavior setting for geometry copy. More...

def **set_copy_nodeset_on_geometry_copy_setting** (val)
Set the behavior for nodeset propagation during geometry copy. More...

def **set_copy_sideset_on_geometry_copy_setting** (val)
Set the behavior for sideset propagation during geometry copy. More...

def **set_copy_block_on_geometry_copy_setting** (val)
Set the behavior for block propagation during geometry copy. More...

Group Support

def **get_group_groups** (group_id)
Retrieve direct child group IDs for a specified parent group, if any. More...

def **get_group_volumes** (group_id)
Return direct child volume IDs for a specified group, if any. More...

def **get_group_bodies** (group_id)
Return direct child body IDs for a specified group, if any. More...

def **get_group_surfaces** (group_id)
Return direct child surface IDs for a specified group, if any. More...

def **get_group_curves** (group_id)
Return direct child curve IDs for a specified group, if any. More...

def **get_group_vertices** (group_id)
Return direct child vertex IDs for a specified group, if any. More...

def **get_group_nodes** (group_id)
Return direct child node IDs for a specified group, if any. More...

def **get_group_edges** (group_id)
Return direct child edge IDs for a specified group, if any. More...

def **get_group_quads** (group_id)
Return direct child quad IDs for a specified group, if any. More...

def **get_group_tris** (group_id)
Return direct child triangle IDs for a specified group, if any. More...

def **get_group_tets** (group_id)
Return direct child tetrahedron IDs for a specified group, if any. More...

def **get_group_wedges** (group_id)
Return direct child wedge element IDs for a specified group, if any. More...

def **get_group_pyramids** (group_id)
Return direct child pyramid element IDs for a specified group, if any. More...

def **get_group_spheres** (group_id)
Return direct child sphere element IDs for a specified group, if any. More...

def **get_group_hexes** (group_id)
Return direct child hexahedral element IDs for a specified group, if any. More...

def **get_next_group_id** ()
Return the next available group ID from Cubit. More...

def **delete_all_groups** ()
Remove all groups from the current Cubit session. More...

def **delete_group** (group_id)
Delete a specific group by ID. More...

def **set_max_group_id** (maximum_group_id)
Reset Cubit's internal maximum group ID to a specified value. More...

def **create_new_group** ()
Create a new, empty group and return its ID. More...

def **remove_entity_from_group** (group_id, entity_id, entity_type)
Remove a specific entity from a given group. More...

def **add_entity_to_group** (group_id, entity_id, entity_type)
Add a specific entity to a given group. More...

def **add_entities_to_group** (group_id, entity_ids, entity_type)
Add a list of entities to a specified group. More...

def **group_names_ids** ()
Retrieve names and IDs of all defined groups (excluding the pick group). More...

def **get_mesh_group_parent_ids** (element_type, element_id)
Retrieve parent group IDs for a specific mesh element. More...

def **is_mesh_element_in_group** (element_type, element_id)
Check if a specific mesh element belongs to any group. More...

Metadata Support

def **get_assembly_classification_level** ()
Get classification level metadata. More...

def **get_assembly_classification_category** ()
Get classification category metadata. More...

def **get_assembly_weapons_category** ()
Get weapons category metadata. More...

def **get_assembly_metadata** (volume_id, data_type)
Get metadata for a specified volume. More...

def **is_assembly_metadata_attached** (volume_id)
Check if metadata is attached to a volume. More...

def **get_assembly_name** (assembly_id)
Get the name of an assembly node. More...

def **get_assembly_path** (assembly_id)
Get the full path of an assembly node, identifying its hierarchy. More...

def **get_assembly_type** (assembly_id)
Get the type of an assembly node ("part" or "assembly"). More...

def **get_parent_assembly_path** (assembly_id)
Get the path of an assembly node's parent. More...

def **get_assembly_level** (assembly_id)
Get the hierarchy level of an assembly node. More...

def **get_assembly_description** (assembly_id)
Get the description of an assembly node. More...

def **get_assembly_instance** (assembly_id)
Get the instance number of an assembly node. More...

def **get_parent_assembly_instance** (assembly_id)

Get the instance number of the parent of an assembly node. More...

def **get_assembly_file_format** (assembly_id)
Get the file format from which the assembly node was imported. More...

def **get_assembly_units** (assembly_id)
Get the units of measurement used in the assembly node. More...

def **get_assembly_material_description** (assembly_id)
Get the material description for a part. More...

def **get_assembly_material_specification** (assembly_id)
Get the material specification for a part. More...

Non-Exodus Boundary Condition Support

def **get_bc_id_list** (bc_type_enum)
Get active boundary condition (BC) IDs of a specified type. More...

def **get_bc_name** (bc_type_enum, bc_id)
Get the name of a specific boundary condition (BC). More...

def **get_nodeset_id_list_for_bc** (bc_type_enum, bc_id)
Get list of nodeset IDs associated with a boundary condition. More...

def **get_sideset_id_list_for_bc** (bc_type_enum, bc_id)
Get list of sideset IDs associated with a boundary condition. More...

def **is_on_thin_shell** (bc_type_enum, entity_id)
Check if a boundary condition is applied to a thin shell. More...

def **temperature_is_on_solid** (bc_type_enum, entity_id)
Check if a temperature BC is applied to a solid region. More...

def **convection_is_on_solid** (entity_id)
Check if a convection BC is on a solid. More...

def **convection_is_on_shell_area** (entity_id, shell_area_enum)
Check if a convection BC is on a shell top or bottom. More...

def **get_convection_coefficient** (entity_id, bc_type_enum)
Get convection coefficient from a convection BC. More...

def **get_bc_temperature** (bc_type_enum, entity_id, temp_type_enum)
Get the temperature value for a specified BC area. More...

def **temperature_is_on_shell_area** (bc_type_enum, bc_area_enum, entity_id)
Check if a temperature BC is on a shell area. More...

def **heatflux_is_on_shell_area** (bc_area_enum, entity_id)
Check if a heatflux BC is on a shell area. More...

def **get_heatflux_on_area** (bc_area_enum, entity_id)
Get the heatflux value on a specified area. More...

def **get_cfd_type** (entity_id)
Get the CFD boundary condition subtype. More...

def **get_contact_pair_friction_value** (entity_id)
Get the friction value for a contact pair. More...

def **get_contact_pair_tolerance_value** (entity_id)
Get the upper bound tolerance for a contact pair. More...

def **get_contact_pair_tol_lower_value** (entity_id)
Get the lower bound tolerance for a contact pair. More...

def **get_contact_pair_tied_state** (entity_id)
Get the tied state of a contact pair. More...

def **get_contact_pair_general_state** (entity_id)
Get the general contact state. More...

def **get_contact_pair_exterior_state** (entity_id)
Get the exterior state of a contact pair. More...

def **get_velocity_combine_type** (entity_id)
Get the velocity's combine type. More...

def **get_acceleration_combine_type** (entity_id)
Get the combine method for an acceleration BC. More...

def **get_displacement_combine_type** (entity_id)
Get the combine method for a displacement BC. More...

def **get_pressure_value** (entity_id)
Get the magnitude value of a pressure BC. More...

def **get_pressure_function** (entity_id)
Get the function expression associated with a pressure BC. More...

def **get_force_magnitude** (entity_id)
Get the force magnitude of a force BC. More...

def **get_moment_magnitude** (entity_id)
Get the moment magnitude of a force BC. More...

def **get_force_direction_vector** (entity_id)
Get the direction vector of a force BC. More...

def **get_force_moment_vector** (entity_id)
Get the moment vector of a force BC. More...

def **get_constraint_type** (constraint_id)
Get the type of a specified constraint. More...

def **get_constraint_reference_point** (constraint_id)
Get the reference point of a specified constraint. More...

def **get_constraint_dependent_entity_point** (constraint_id)
Get the dependent entity of a specified constraint. More...

def **get_material_property** (material_property_enum, entity_id)
Get the value of a material property. More...

def **get_media_property** (entity_id)

Get the media classification of a material. More...

def **get_media_name_list** ()
Get list of all CFD media names. More...

def **get_material_name_list** ()
Get list of all material names. More...

def **get_material_name** (material_id)
Get the name of a material or CFD media by ID. More...

Boundary Layer Support

def **get_next_boundary_layer_id** ()
Get the next available boundary layer ID. More...

def **is_boundary_layer_id_available** (boundary_layer_id)
Check if a boundary layer ID is currently available. More...

def **get_boundary_layer_algorithm** (boundary_layer_id)
Get the algorithm type used by the specified boundary layer. More...

def **get_boundary_layers_by_base** (base_type, base_id)
Get all boundary layers associated with a specified base entity. More...

def **get_boundary_layers_by_pair** (base_type, base_id, parent_id)
Get all boundary layers associated with a base-parent pair. More...

def **get_boundary_layer_continuity** (boundary_layer_id)
Get the continuity setting of a boundary layer. More...

def **get_boundary_layer_id_list** ()
Get a list of all defined boundary layer IDs. More...

Machine Learning Support

def **load_ML** (args, kwargs)
Load machine learning training data into memory. More...

def **unload_ML** (args, kwargs)
Unload machine learning training data from memory. More...

def **get_ML_operation_features** (ml_op_names, entity1_ids, entity2_ids, params, mesh_size, reduced_features=False)
Compute ML feature vectors for a list of Cubit operations. More...

def **get_ML_features** (ml_op_names, entity1_ids, entity2_ids, params, mesh_size, reduced_features=False)
Compute ML feature vectors for operations permitting multiple entities. More...

def **get_ML_predictions** (ml_op_names, entity1_ids, entity2_ids, params, mesh_size, reduced_features=False)
Compute ML predictions for a list of operations on single-entity pairs. More...

def **do_ML_predictions** (ml_op_names, entity1_ids, entity2_ids, params, mesh_size, reduced_features=False)
Compute ML predictions for a list of operations with multiple entities. More...

def **get_ML_classification** (geom_type, ent_id)
Get the top classification label for a single volume or surface. More...

def **get_ML_classifications** (geom_type, ent_ids)
Classify multiple volumes or surfaces in a single batch operation. More...

def **get_ML_classification_categories** (geom_type)
Retrieve all available classification categories for a geometry type. More...

def **ML_train** (geom_type)
Force retraining of the ML classification model for a specific geometry type. More...

def **get_ML_operation_feature_types** (ml_op_name, reduced_features=False)
Get the data-type descriptor for each feature of a specified ML operation. More...

def **get_ML_operation_feature_names** (ml_op_name, reduced_features=False)
Retrieve the list of feature names for a specified ML operation. More...

def **get_ML_operation_feature_size** (ml_op_name, reduced_features=False)
Retrieve the expected feature-vector length for a given ML operation. More...

def **get_ML_operation_label_size** (ml_op_name)
Retrieve the expected label vector length for a given ML operation. More...

def **get_ML_classification_models** ()
Retrieve the list of supported classification ML model names. More...

def **get_ML_regression_models** ()
Retrieve the list of supported regression ML model names. More...

def **get_ML_model_name** (model_ID)
Retrieve the name of an ML model based on its ID. More...

def **get_ML_model_ID** (model_name)
Retrieve the unique numeric ID for a given ML model or operation name. More...

def **get_ML_operation** (op_name, entity_id1, entity_id2, params, small_curve_size, mesh_size)
Generate the command, display label, and preview text for an ML operation. More...

def **get_ML_feature_importances** (op_name)
Retrieve feature importance scores for a given ML operation. More...

def **get_ML_feature_distance** (op_name, f1, f2)
Compute the weighted distance between two feature vectors. More...

def **set_ML_base_user_dir** (path, **print_info**=False)
Set the root directory for user-provided ML training data (classification only). More...

def **get_ML_initialize_surviving_entities** (op_name, ent1, ent2)
Capture entities for survival tracking before a CAD operation. More...

def **get_ML_surviving_entities** (op_name, begin_ents)
Identify which entities survive after a CAD operation. More...

Variables

BAR = _cubit3.BAR
BAR2 = _cubit3.BAR2
BAR3 = _cubit3.BAR3
BEAM = _cubit3.BEAM
BEAM2 = _cubit3.BEAM2
BEAM3 = _cubit3.BEAM3
CI_BCENTITY_CC = _cubit3.CI_BCENTITY_CC
CI_BCENTITY_CC_SHELL_BOTTOM = _cubit3.CI_BCENTITY_CC_SHELL_BOTTOM
CI_BCENTITY_CC_SHELL_TOP = _cubit3.CI_BCENTITY_CC_SHELL_TOP
CI_BCENTITY_FIRST = _cubit3.CI_BCENTITY_FIRST
CI_BCENTITY_LAST = _cubit3.CI_BCENTITY_LAST
CI_BCENTITY_SHELL_AREA_BOTTOM = _cubit3.CI_BCENTITY_SHELL_AREA_BOTTOM
CI_BCENTITY_SHELL_AREA_GRADIENT = _cubit3.CI_BCENTITY_SHELL_AREA_GRADIENT
CI_BCENTITY_SHELL_AREA_MIDDLE = _cubit3.CI_BCENTITY_SHELL_AREA_MIDDLE
CI_BCENTITY_SHELL_AREA_TOP = _cubit3.CI_BCENTITY_SHELL_AREA_TOP
CI_BCENTITY_SOLID_HEATFLUX = _cubit3.CI_BCENTITY_SOLID_HEATFLUX
CI_BCENTITY_SOLID_TEMP = _cubit3.CI_BCENTITY_SOLID_TEMP
CI_BCENTITY_TEMP = _cubit3.CI_BCENTITY_TEMP
CI_BCENTITY_UNKNOWN = _cubit3.CI_BCENTITY_UNKNOWN
CI_BCTYPE_ACCELERATION = _cubit3.CI_BCTYPE_ACCELERATION
CI_BCTYPE_AXIS = _cubit3.CI_BCTYPE_AXIS
CI_BCTYPE_CONSTRAINT = _cubit3.CI_BCTYPE_CONSTRAINT
CI_BCTYPE_CONTACT_PAIR = _cubit3.CI_BCTYPE_CONTACT_PAIR
CI_BCTYPE_CONTACT_REGION = _cubit3.CI_BCTYPE_CONTACT_REGION
CI_BCTYPE_CONVECTION = _cubit3.CI_BCTYPE_CONVECTION
CI_BCTYPE_DISPLACEMENT = _cubit3.CI_BCTYPE_DISPLACEMENT
CI_BCTYPE_EXHAUSTFAN = _cubit3.CI_BCTYPE_EXHAUSTFAN
CI_BCTYPE_FAN = _cubit3.CI_BCTYPE_FAN
CI_BCTYPE_FIRST = _cubit3.CI_BCTYPE_FIRST
CI_BCTYPE_FORCE = _cubit3.CI_BCTYPE_FORCE
CI_BCTYPE_GAP = _cubit3.CI_BCTYPE_GAP
CI_BCTYPE_HEATFLUX = _cubit3.CI_BCTYPE_HEATFLUX
CI_BCTYPE_INLETVENT = _cubit3.CI_BCTYPE_INLETVENT
CI_BCTYPE_INTAKEFAN = _cubit3.CI_BCTYPE_INTAKEFAN
CI_BCTYPE_INTERFACE = _cubit3.CI_BCTYPE_INTERFACE
CI_BCTYPE_INTERIOR = _cubit3.CI_BCTYPE_INTERIOR
CI_BCTYPE_LAST = _cubit3.CI_BCTYPE_LAST
CI_BCTYPE_LUMPMASS = _cubit3.CI_BCTYPE_LUMPMASS
CI_BCTYPE_MASSFLOWINLET = _cubit3.CI_BCTYPE_MASSFLOWINLET
CI_BCTYPE_MATERIAL = _cubit3.CI_BCTYPE_MATERIAL
CI_BCTYPE_MEDIA = _cubit3.CI_BCTYPE_MEDIA
CI_BCTYPE_MPC = _cubit3.CI_BCTYPE_MPC
CI_BCTYPE_OUTFLOW = _cubit3.CI_BCTYPE_OUTFLOW
CI_BCTYPE_OUTLETVENT = _cubit3.CI_BCTYPE_OUTLETVENT
CI_BCTYPE_PERIODIC = _cubit3.CI_BCTYPE_PERIODIC
CI_BCTYPE_PERIODICSHADOW = _cubit3.CI_BCTYPE_PERIODICSHADOW
CI_BCTYPE_POROUSJUMP = _cubit3.CI_BCTYPE_POROUSJUMP
CI_BCTYPE_PRESSURE = _cubit3.CI_BCTYPE_PRESSURE
CI_BCTYPE_PRESSUREFARFIELD = _cubit3.CI_BCTYPE_PRESSUREFARFIELD
CI_BCTYPE_PRESSUREINLET = _cubit3.CI_BCTYPE_PRESSUREINLET

CI_BCTYPE_PRESSUREOUTLET = _cubit3.CI_BCTYPE_PRESSUREOUTLET
CI_BCTYPE_RADIATOR = _cubit3.CI_BCTYPE_RADIATOR
CI_BCTYPE_RIGIDBODY = _cubit3.CI_BCTYPE_RIGIDBODY
CI_BCTYPE_SPRING = _cubit3.CI_BCTYPE_SPRING
CI_BCTYPE_SUPERSONICINFLOW = _cubit3.CI_BCTYPE_SUPERSONICINFLOW
CI_BCTYPE_SUPERSONICOUTFLOW = _cubit3.CI_BCTYPE_SUPERSONICOUTFLOW
CI_BCTYPE_SYMMETRY = _cubit3.CI_BCTYPE_SYMMETRY
CI_BCTYPE_TEMPERATURE = _cubit3.CI_BCTYPE_TEMPERATURE
CI_BCTYPE_UNKNOWN = _cubit3.CI_BCTYPE_UNKNOWN
CI_BCTYPE_VELOCITY = _cubit3.CI_BCTYPE_VELOCITY
CI_BCTYPE_VELOCITYINLET = _cubit3.CI_BCTYPE_VELOCITYINLET
CI_BCTYPE_WALL = _cubit3.CI_BCTYPE_WALL
CI_CONDUCTIVITY = _cubit3.CI_CONDUCTIVITY
CI_DENSITY = _cubit3.CI_DENSITY
CI_ERROR = cvar.CI_ERROR
CI_MATERIAL_FIRST = _cubit3.CI_MATERIAL_FIRST
CI_MATERIAL_LAST = _cubit3.CI_MATERIAL_LAST
CI_MATERIAL_UNKNOWNPROPERTY = _cubit3.CI_MATERIAL_UNKNOWNPROPERTY
CI_MODULUS = _cubit3.CI_MODULUS
CI_POISSON = _cubit3.CI_POISSON
CI_SHEAR_MODULUS = _cubit3.CI_SHEAR_MODULUS
CI_SPECIFIC_HEAT = _cubit3.CI_SPECIFIC_HEAT
CI_THERMAL_EXPANSION = _cubit3.CI_THERMAL_EXPANSION
CI_ULTIMATE_STRAIN = _cubit3.CI_ULTIMATE_STRAIN
CI_ULTIMATE_STRENGTH = _cubit3.CI_ULTIMATE_STRENGTH
CI_YIELD_STRENGTH = _cubit3.CI_YIELD_STRENGTH
CUBIT_TRI = _cubit3.CUBIT_TRI
CURVE = _cubit3.CURVE
cvar = _cubit3.cvar
HEX = _cubit3.HEX
HEX20 = _cubit3.HEX20
HEX21 = _cubit3.HEX21
HEX26 = _cubit3.HEX26
HEX27 = _cubit3.HEX27
HEX8 = _cubit3.HEX8
HEX9 = _cubit3.HEX9
HEXSHELL = _cubit3.HEXSHELL
INVALID_ELEMENT_TYPE = _cubit3.INVALID_ELEMENT_TYPE
PYRAMID = _cubit3.PYRAMID
PYRAMID13 = _cubit3.PYRAMID13
PYRAMID18 = _cubit3.PYRAMID18
PYRAMID5 = _cubit3.PYRAMID5
PYRAMID8 = _cubit3.PYRAMID8
QUAD = _cubit3.QUAD
QUAD4 = _cubit3.QUAD4
QUAD5 = _cubit3.QUAD5
QUAD8 = _cubit3.QUAD8
QUAD9 = _cubit3.QUAD9

SHARED_PTR_DISOWN = _cubit3.SHARED_PTR_DISOWN
SHEL = _cubit3.SHEL
SHELL4 = _cubit3.SHELL4
SHELL8 = _cubit3.SHELL8
SHELL9 = _cubit3.SHELL9
SPHERE_EXO = _cubit3.SPHERE_EXO
SPRING = _cubit3.SPRING
SUPERELEMENT_TOPOLOGY = _cubit3.SUPERELEMENT_TOPOLOGY
SURFACE = _cubit3.SURFACE
TETRA = _cubit3.TETRA
TETRA10 = _cubit3.TETRA10
TETRA14 = _cubit3.TETRA14
TETRA15 = _cubit3.TETRA15
TETRA4 = _cubit3.TETRA4
TETRA8 = _cubit3.TETRA8
TRI3 = _cubit3.TRI3
TRI4 = _cubit3.TRI4
TRI6 = _cubit3.TRI6
TRI7 = _cubit3.TRI7
TRISHELL = _cubit3.TRISHELL
TRISHELL3 = _cubit3.TRISHELL3
TRISHELL4 = _cubit3.TRISHELL4
TRISHELL6 = _cubit3.TRISHELL6
TRISHELL7 = _cubit3.TRISHELL7
TRUSS = _cubit3.TRUSS
TRUSS2 = _cubit3.TRUSS2
TRUSS3 = _cubit3.TRUSS3
VERTEX = _cubit3.VERTEX
VOLUME = _cubit3.VOLUME
WEDGE = _cubit3.WEDGE
WEDGE15 = _cubit3.WEDGE15

WEDGE16 = _cubit3.WEDGE16
WEDGE20 = _cubit3.WEDGE20
WEDGE21 = _cubit3.WEDGE21
WEDGE6 = _cubit3.WEDGE6

Function Documentation

◆ add_entities_to_group()

def cubit.add_entities_to_group (*group_id*, *entity_ids*, *entity_type*)

Add a list of entities to a specified group.

.. code-block:: python

```
# Initialize geometry

cubit.cmd
("brick x 10" )

# Create a new empty group and
retrieve its ID.

group_id = cubit.create_new_group
()

# Prepare a list of vertices 1
through 8.

verts_to_add = list(range(1,9))

# Add multiple vertices in one
call.

cubit.add_entities_to_group
(group_id, verts_to_add, "vertex" )

# Verify by querying group
vertices.

verts = cubit.get_group_vertices
(group_id)

print("Group
vertices:" , verts) #
prints [1, 2, 3, 4, 5, 6, 7, 8]
```

```
@n type of group_id:  int
```

Parameters

group_id ID of the group to which entities will be added.
type of entity_ids: std::vector< int,std::allocator< int > >

entity_ids Vector of entity IDs to add.
type of entity_type: string

entity_type Type of the entities (e.g., "vertex", "curve", "surface", "volume"). Only geometric entities are supported.
return type of : void

Returns
None.

◆ add_entity_to_group()

def cubit.add_entity_to_group (*group_id*, *entity_id*, *entity_type*)

Add a specific entity to a given group.

.. code-block:: python

```
# Initialize geometry.

cubit.cmd
("brick x 10" )

# Create a new empty group and
retrieve its ID.

group_id = cubit.create_new_group
()

# Add vertex with ID 1 to the new
group.

cubit.add_entity_to_group
(group_id, 1, "vertex" )

# Verify by querying group
vertices.

verts = cubit.get_group_vertices
(group_id)

print("Group
vertices:" , verts) #
prints "[1]"
```

```
@n type of group_id:  int
```

Parameters

group_id ID of the group to which the entity will be added.
type of entity_id: int

entity_id ID of the entity to add.
type of entity_type: string

entity_type Type of the entity (e.g., "vertex", "curve", "surface", "volume"). Only geometric entities are supported.
return type of : void

Returns

None.

◆ add_filename_to_recent_file_list()

def cubit.add_filename_to_recent_file_list (*filename*)

Add a filename to Cubit's recent-file list in the GUI File menu.

Registers the specified file path so it appears under "Recent Files" in Cubit's GUI.

```
.. code-block:: python
```

```
import cubit

filename = "C:/models/part.sat"

cubit.add_filename_to_recent_file_list
(filename)
```

```
@n type of filename:  string
```

Parameters

filename Reference to the file path string to add to the recent-file list.

◆ add_filter_type()

def cubit.add_filter_type (*filter_type*)

Add an entity type to the graphics pick filter.

Allows picking entities of the specified type in addition to any existing filters.

.. code-block:: python

```
                import cubit

                # Add "volume" to the pick
                filter

                cubit.add_filter_type
                ("volume" )
```

@n type of filter_type: string

Parameters

filter_type **Entity** type to add to the pickable set (e.g., "vertex", "curve", "surface", "volume", "node", "edge", "face").

◆ app_util()

def cubit.app_util ( )

Return Cubit's AppUtil interface for global services.

Provides access to application-level utilities–logging, configuration, file I/O, and more–required by higher-level modules like CubitML.

Python usage example:

.. code-block:: python

```
                import cubit, cubitml

                # Initialize Cubit (no GUI, no
                journal, offscreen driver)

                cubit.init
                (['' , '-nojournal' , '-driver' ,
    'offscreen' ])

                util = cubit.app_util
                ()

                cml = cubitml.CubitML(util)

                geom = cubit.cgm_iface
                ()

                ml = cubitml.MLUtil(cml, geom)

                ml.load_ML()
```

Typically used together with **cgm_iface()** to bootstrap ML workflows.

return type of : std:: shared_ptr< AppUtil >

Returns

Shared pointer to the AppUtil instance.

◆ are_adjacent_curves()

def cubit.are_adjacent_curves (*curve_ids*)

Return whether two or more curves share at least one manifold vertex.

Given a list of curve IDs, this function returns whether the curves are adjacent. Two curves are considered adjacent if they share at least one manifold vertex: a vertex that is connected to exactly two curves.

If the shared vertex is connected to more than two curves, the curves are **not considered adjacent** .

```
.. code-block:: python
```

```
cubit.cmd
("brick x 10" )


print(cubit.are_adjacent_curves
([1, 2]))

# Expected output:

# False


cubit.cmd
("surface 1 copy" )
# Creates sheet body with curves
13, 14, etc.


print(cubit.are_adjacent_curves
([13, 14]))

# Expected output:

# True
```

```
@n type of curve_ids:  std::vector< int,std::allocator< int > >
```

Parameters

curve_ids IDs of curves to query.
return type of : boolean

Returns

True if the curves are adjacent (share at least one manifold vertex - vertex connected to exactly two curves), false otherwise.

◆ are_adjacent_surfaces()

def cubit.are_adjacent_surfaces (*surface_ids*)

Return whether two or more surfaces share at least one manifold curve.

Given a list of surface IDs, this function returns whether the surfaces are adjacent. Two surfaces are considered adjacent if they share at least one manifold curve (a curve that is part of exactly two surfaces).

```
.. code-block:: python
```

```
cubit.cmd
("brick x 10" )


print(cubit.are_adjacent_surfaces
([1, 6]))

# Expected output:

# True


print(cubit.are_adjacent_surfaces
([1, 2]))

# Expected output:

# False
```

```
@n type of surface_ids:  std::vector< int,std::allocator< int > >
```

Parameters

surface_ids IDs of surfaces to query.
return type of : boolean

Returns

True if the surfaces are adjacent (share at least one manifold curve), false otherwise.

◆ auto_size_needs_to_be_calculated()

def cubit.auto_size_needs_to_be_calculated ( )

Check if automatic mesh sizing is outdated and needs recomputation.

Returns true if the model has changed since the last auto size calculation, indicating that automatic sizes (which can be expensive to compute) should be recalculated before generating a mesh.

.. code-block:: python

```
import cubit

# Recompute auto size on volume 1
if needed
```

if **cubit.auto_size_needs_to_be_calculated()** : **cubit.cmd** ("volume 1 size auto factor 3")

@n return type of : boolean

Returns

True if auto sizing is out of date and should be recalculated; false otherwise.

◆ best_edge_to_collapse_interior_node()

def cubit.best_edge_to_collapse_interior_node (*node_id*)

Find the best edge to collapse to remove an interior node in a triangular mesh.

For a given interior mesh node (no geometry association), this function identifies the best adjacent edge whose collapse would remove the node. If no suitable edge is found, returns 0.

.. code-block:: python

```
# 1) Assume vertex 50 is an
interior node in a surface mesh.

best_edge =
cubit.best_edge_to_collapse_interior_node
(50)
```

if best_edge > 0: print(f"Collapse interior node 50 along edge {best_edge}") else:

```
print("No valid edge to
collapse for node 50" )
```

type of node_id: int

Parameters

node_id ID of the interior mesh node to remove.

return type of : int

Returns

ID of the best edge to collapse; 0 if no valid edge found.

◆ body()

def cubit.body (*id_in*)

Retrieve a body by its ID.

Retrieves the body object corresponding to the provided ID.

.. code-block:: python

```
# Create a 10x10x10 brick (body
ID 1).

cubit.cmd
("brick x 10" )

# Select and retrieve body with
ID 1.

body = cubit.body
(1)

print("Body ID:" ,
body.id())
```

@n type of id_in: int

Parameters

id_in ID of the body to retrieve.

return type of : :py:class: **Body**

Returns

Body object corresponding to the specified ID.

◆ brick()

```
def cubit.brick ( width,
                  depth = -1,
                  height = -1
                )
```

Create a brick of specified width, depth, and height.

Creates a brick geometry with the given dimensions. If only width is provided (depth and height default to -1), a cube of side length `width` is generated.

.. code-block:: python

```python
import cubit

# Create a cube of side 10 (width
only).

cube = cubit.brick
(10.0)

print("Created cube Body
ID:" , cube.id())

print("Cube volume:"
, cube.volume())

# Expected output:

# Created cube Body ID: 1

# Cube volume: 1000


# Create a brick of width=2,
depth=3, height=4.

brick_block = cubit.brick
(2.0, 3.0, 4.0)

print("Created brick Body
ID:" , brick_block.id())

print("Brick
volume:" , brick_block.volume())

# Expected output:

# Created brick Body ID: 2

# Brick volume: 24
```

@n type of width: float, in

Parameters

width Width of the brick (or cube side length if only this argument is supplied).
type of depth: float, in, optional

depth Depth of the brick (default = -1, uses `width`).
type of height: float, in, optional

height Height of the brick (default = -1, uses `width`).
return type of : :py:class: **Body**

Returns

Body object representing the newly created brick geometry.

◆ calculate_timestep_estimate()

def cubit.calculate_timestep_estimate (*entity_type*, *entity_ids*)

Estimate the time step based on element sizes and material properties.

Calculates a time step estimate for specified mesh elements of a given type. Supported "tet", "hex", "volume", "block", "group". Elements must belong to a single block with an assigned material defining at least elastic_modulus, poisson_ratio, and density.

```
.. code-block:: python
```

```
                # Create geometry and define
                material properties.


                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("create material \"steel\"
                property_group \"CUBIT-ABAQUS\"" )


                cubit.cmd
                ("modify material \"steel\"
                scalar_properties \"MODULUS\" 2.068e+05
                \"SHEAR_MODULUS\" 8.016e+04 \"POISSON\"
      0.29
                \"DENSITY\" 7e-06 \"SPECIFIC_HEAT\" 500
                \"CONDUCTIVITY\" 0.045
      \"THERMAL_EXPANSION\" 1.2e-05
                \"YIELD_STRENGTH\" 374
      \"ULTIMATE_STRENGTH\"
                590" )


                # Assign material and
                mesh.


                cubit.cmd
                ("block 1 add volume
                1" )


                cubit.cmd
                ("block 1 material
                'steel'" )


                cubit.cmd
                ("mesh volume 1" )


                # Estimate time step.


                dt = cubit.calculate_timestep_estimate
                ('volume' , [1])


                print(f"Estimated time
                step: {dt}" )
```

```
@n type of entity_type:  string
```

Parameters

entity_type Mesh element type string ("tet", "hex", "volume", "block", "group").
type of entity_ids: std::vector< int,std::allocator< int > >

entity_ids List of IDs of elements to include in the estimate.
return type of : float

Returns

Smallest stable time step across specified elements.

◆ calculate_timestep_estimate_with_props()

def cubit.calculate_timestep_estimate_with_props (*entity_type*, *entity_id_list*, *density*, *youngs_modulus*, *poissons_ratio*)

Estimate the stable time step using user-specified material properties.

Calculates a time step estimate for mesh elements of the given type, using provided density, Young's modulus, and Poisson's ratio rather than block-assigned materials.

.. code-block:: python

```
# Create geometry and mesh volume
1

cubit.cmd
("brick x 10" )

cubit.cmd
("mesh volume 1" )

vol_list = [1]

# Estimate time step with
density=2.7, E=70, nu=0.35

dt =
cubit.calculate_timestep_estimate_with_props
('volume' ,
vol_list,

2.7, 70.0, 0.35)

print(f"Custom estimate:
{dt}" )
```

@n type of entity_type: string

Parameters

entity_type	Mesh element type ("tet", "hex", "volume", "block", "group"). type of entity_id_list: std::vector< int,std::allocator< int > >
entity_id_list	Vector of element IDs to include. type of density: float
density	Material density. type of youngs_modulus: float
youngs_modulus	Young's modulus. type of poissons_ratio: float
poissons_ratio	Poisson's ratio. return type of : float

Returns

Smallest stable time step estimate.

◆ cgm()

def cubit.cgm ( )

Return the raw CGMApp kernel instance used by Cubit.

Grants direct access to lower-level CGM application calls (version, kernel settings, etc.).

return type of : CGMApp

Returns

Pointer to the active CGMApp instance (or None if CGM is unavailable).

◆ cgm_iface()

def cubit.cgm_iface ( )

Return the CGM-based geometry interface for Cubit.

Yields a CubitGeometryInterface implemented on the CGM kernel, enabling CAD topology queries and edits needed for ML feature extraction.

Python usage example:

.. code-block:: python

```
import cubit, cubitml

cubit.init
(['' , '-nojournal' , '-driver' ,
'offscreen' ])

util = cubit.app_util
()

cml = cubitml.CubitML(util)

geom = cubit.cgm_iface
()

ml = cubitml.MLUtil(cml, geom)

ml.load_ML()
```

Use this interface for all geometry operations in the ML pipeline.

return type of : CubitGeometryInterface

Returns
Pointer to the CGM-backed CubitGeometryInterface.

◆ clear_drawing_set()

def cubit.clear_drawing_set (*set_name*)

Clear all geometry in a named drawing set (e.g., mesh preview).

Removes any preview graphics associated with the specified set name, leaving the set empty for fresh drawing operations.

.. code-block:: python

```
import cubit

# Preview a mesh

cubit.cmd
("mesh volume 5 scheme
sweep preview mesh_preview" )

# Now clear the preview
set

cubit.clear_drawing_set
("mesh_preview" )
```

@n type of set_name: string

Parameters
set_name Name of the drawing set to clear.

◆ clear_highlight()

def cubit.clear_highlight ( )

Clear all entity highlights.

Removes highlights and selections from every entity in the graphics window.

.. code-block:: python

```
import cubit

# First select nodes 10 and
20

cubit.cmd
("select node 10 20"
)

# Now clear all highlights and
selections

cubit.clear_highlight
()
```

◆ clear_picked_list()

def cubit.clear_picked_list ( )

Clear the list of currently picked entities.

Empties the internal pick list without altering the pick filter or highlights.

.. code-block:: python

```
import cubit

# Clear all picked
entities

cubit.clear_picked_list
()
```

◆ clear_preview()

def cubit.clear_preview ( )

Clear preview graphics without affecting other display settings.

Removes only the temporary preview without altering the main geometry or mesh visibility.

.. code-block:: python

```
import cubit

# Create a brick and preview
moving it along the X axis by 10 units

cubit.cmd
("brick x 10" )

# A blue wireframe preview of the
moved brick appears

cubit.cmd
("move volume 1 x 10
preview" )


# Now clear the blue wireframe
preview

cubit.clear_preview
()
```

◆ cmd()

def cubit.cmd (*input_string*)

Execute a raw Cubit command string immediately (modifies model state).

This is one of the two primary ways (silent_cmd) to change the CAD model within Cubit. Sends the exact command text to Cubit for execution with no pre-validation. See Cubit's command syntax in the online documentation for a full description.

.. code-block:: python

```
import cubit

# Create a 10x10x10 block

cubit.cmd
("brick x 10" )

# Then mesh volume 1

cubit.cmd
("mesh volume 1" )
```

```
@n type of input_string:  string
```

Parameters

input_string Null-terminated C string containing the complete Cubit command.
return type of : boolean

Returns

true if the command executed successfully; false otherwise.

◆ contains_virtual()

def cubit.contains_virtual (*geometry_type*, *entity_id*)

Query whether any child entities of a specified geometry entity are virtual.

.. code-block:: python

```
cubit.cmd
("brick x 10" )
# create brick (Volume ID =
1)


cubit.cmd
("composite create surf 1
6" ) # create composite
surface (Surface ID = 7)


# Check if volume 1 contains any
virtual children (surface 7 is virtual)


has_virt = cubit.contains_virtual
("volume" , 1)


print("Volume 1 contains
virtual children:" , has_virt)


cubit.cmd
("composite delete surf
7" ) # delete composite
surface


has_virt = cubit.contains_virtual
("volume" , 1)


print("Volume 1 contains
virtual children:" , has_virt)
```

```
@n type of geometry_type:  string
```

Parameters

geometry_type The type of geometry ("curve", "surface", "volume", etc.).
type of entity_id: int

entity_id The ID of the entity to query.
return type of : boolean

Returns

True if any child entities of the specified entity are virtual; false otherwise.

◆ convection_is_on_shell_area()

def cubit.convection_is_on_shell_area (*entity_id*, *shell_area_enum*)

Check if a convection BC is on a shell top or bottom.

type of entity_id: int

Parameters

entity_id ID of the BC.
type of shell_area_enum: int

shell_area_enum Use CI_BCEntityTypes: 7 for top, 8 for bottom.
return type of : boolean

Returns

True if BC is applied to specified shell area.

◆ convection_is_on_solid()

def cubit.convection_is_on_solid (*entity_id*)

Check if a convection BC is on a solid.

type of entity_id: int

Parameters

entity_id ID of the convection boundary condition.
return type of : boolean

Returns

True if the BC is applied to a solid region.

◆ copy_body()

def cubit.copy_body (*init_body*)

Create a duplicate of the specified body.

Returns a new **Body** object that is an exact copy of the input body, including geometry and mesh data.

```
.. code-block:: python

                import cubit

                # Create a unit cube body.

                original = cubit.brick
                (1.0)

                print("Original Body
                ID:" , original.id())


                # Duplicate the body.

                copy = cubit.copy_body
                (original)

                print("Copied Body
                ID:" , copy.id())

                # Verify the copy has the same
                volume.

                print("Copy volume:"
                , copy.volume())

                # Expected output:

                # Original Body ID: 1

                # Copied Body ID: 2

                # Copy volume: 1.0
```

```
@n type of init_body:  :py:class:`Body`, in
```

Parameters

init_body **Body** object to duplicate.
type of init_body: :py:class: **Body**, in

init_body **Body** object to duplicate.
return type of : :py:class: **Body**

Returns

New **Body** object that is a copy of the input. :raises: **CubitFailureException** Thrown if copying fails.

◆ create_arc_curve()

def cubit.create_arc_curve (*v0*,
v1,
intermediate_point
)

Create an arc curve using end vertices and an intermediate point.

Constructs a circular arc passing through the specified start and end vertices and the given intermediate point.

.. code-block:: python

```
import cubit

# Create vertices for arc
endpoints.

v0 = cubit.create_vertex
(1, 0, 0)

v1 = cubit.create_vertex
(-1, 0, 0)

# Intermediate point on the arc
(half-circle of radius 1).

mid_pt = [0.0, 1.0, 0.0]

# Create the arc curve.

arc = cubit.create_arc_curve
(v0, v1, mid_pt)

print("Arc curve
ID:" , arc.id())

print("Arc curve
length:" , arc.length())

# Expected output:

# Arc curve ID: 1

# Arc curve length:
3.14159
```

@n type of v0: :py:class:`Vertex`, in

Parameters

v0	The start vertex of the arc. type of v1: :py:class: **Vertex**, in
v1	The end vertex of the arc. type of intermediate_point: std::array< double,3 >, in
intermediate_point	Array of three coordinates representing a point on the arc. return type of : :py:class: **Curve**

Returns

Curve object representing the created arc.

create_curve()

def cubit.create_curve (*v0*,
v1

)

Create a curve between two vertices.

Constructs a straight-line curve connecting the two specified vertices.

.. code-block:: python

```
import cubit

# Create two vertices at (0,0,0)
and (1,1,0).

v0 = cubit.create_vertex
(0, 0, 0)

v1 = cubit.create_vertex
(1, 1, 0)

# Create a curve between v0 and
v1.

curve = cubit.create_curve
(v0, v1)

print("Curve ID:" ,
curve.id())

# Retrieve and print curve
length.

print("Curve
length:" , curve.length())

# Expected output:

# Curve ID: 1

# Curve length: 1.41421
```

@n type of v0: :py:class:`Vertex`, in

Parameters

v0 The start vertex of the curve.
type of v1: :py:class: **Vertex**, in

v1 The end vertex of the curve.
return type of : :py:class: **Curve**

Returns

Curve object representing the connection between the vertices.

◆ create_new_group()

def cubit.create_new_group ( )

Create a new, empty group and return its ID.

.. code-block:: python

```
# Initialize geometry.

cubit.cmd
("brick x 10" )

# Create a new group and retrieve
its ID.

new_id = cubit.create_new_group
()

# Add vertices in volume 1 to the
new group using shorthand.

cubit.cmd
(f"group {new_id} add
vertex in vol 1" )
```

@n return type of : int

Returns

ID of the newly created group.

◆ create_spline()

def cubit.create_spline (*points*,
surface_id
)

Create a spline curve through a sequence of 3D points on a surface.

Constructs a smooth spline curve passing through the provided points, projected onto the specified surface. Projection is mandatory: the `surface_id` must refer to an existing surface entity. At least two points are required; otherwise, a **CubitFailureException** is thrown. A non-existent `surface_id` produces an **InvalidEntityException** . Other errors during projection or spline creation will result in a **CubitFailureException** or runtime error.

.. code-block:: python

```
import cubit

# Create a unit cube using the
brick function (cube of side 1).

cube = cubit.brick
(1.0)

# Retrieve the front surface with
ID 1.

front_surface = cubit.surface
(1)


# Define spline points on the
plane z=0.5 (front face).

points = [[0.0, 0.0, 0.5], [0.5, 0.0, 0.5], [0.5, 0.5,
0.5]]

# Create spline projected onto
surface ID 1.

spline = cubit.create_spline
(points, front_surface.id())

print("Spline ID:" ,
spline.id())

print("Spline
length:" , spline.length())

# Expected output:

# Spline ID: 1

# Spline length:
1.0304240189190117
```

@n type of points: std::vector< std::array< double,3 >,std::allocator< std::array< double,3 > > >, in

Parameters

points Ordered list of 3D coordinates (each array length = 3); must contain at least two points.
type of surface_id: int, in

surface_id ID of the surface onto which points are projected; must be a valid surface ID.
return type of : :py:class: **Curve**

Returns

Curve object representing the projected spline. :raises: **CubitFailureException** Thrown for invalid input or projection failure. :raises: **InvalidEntityException** Thrown if `surface_id` does not correspond to an existing surface.

◆ create_surface()

def cubit.create_surface (*curves*)

Create a sheet body from boundary curves.

Constructs a sheet body bounded by the specified closed curves. The returned **Body** contains exactly one surface and one volume. To access the underlying **Surface** , retrieve it from the body's surface list.

.. code-block:: python

```
import cubit

# Create four vertices of a unit
square on the XY-plane.

v0 = cubit.create_vertex
(0, 0, 0)

v1 = cubit.create_vertex
(1, 0, 0)

v2 = cubit.create_vertex
(1, 1, 0)

v3 = cubit.create_vertex
(0, 1, 0)

# Create boundary curves of the
square.

c0 = cubit.create_curve
(v0, v1)

c1 = cubit.create_curve
(v1, v2)

c2 = cubit.create_curve
(v2, v3)

c3 = cubit.create_curve
(v3, v0)

boundary = [c0, c1, c2, c3]


# Create a sheet body from the
boundary curves.

sheet_body = cubit.create_surface
(boundary)

print("Sheet Body
ID:" , sheet_body.id())

# Access the underlying surface
object.

surface = sheet_body.surfaces()[0]

print("Surface
area:" , surface.area())

# Expected output:

# Sheet Body ID: 1

# Surface area: 1.0
```

@n type of curves: std::vector< CubitInterface::Curve,std::allocator< CubitInterface::Curve > >, in

Parameters

curves List of **Curve** objects forming a closed boundary loop.
return type of : :py:class: **Body**

Returns

Body object representing the created sheet (contains one surface and one volume). :raises:
CubitFailureException Thrown if curves do not form a valid loop or creation fails.

◆ create_vertex()

def cubit.create_vertex (*x*,
y,
z
)

Create a vertex at specified coordinates.

Creates a 0-dimensional geometric entity (vertex) at the given (x, y, z) location. If no coordinates are specified, the vertex is placed at the origin (0,0,0).

.. code-block:: python

```
import cubit

# Create a vertex at the default
origin (0,0,0).

origin = cubit.create_vertex
()

coords = origin.coordinates()

print("Origin Vertex
ID:" , origin.id())

print(f"Coordinates:
({coords[0]}, {coords[1]},
{coords[2]})" )


# Create a vertex at (1.0, 2.0,
3.0).

v = cubit.create_vertex
(1.0, 2.0, 3.0)

coords = v.coordinates()

print("Vertex at (1,2,3)
ID:" , v.id())

print(f"Coordinates:
({coords[0]}, {coords[1]},
{coords[2]})" )
```

@n type of x: float, in

Parameters

x X-coordinate of the vertex (default = 0).
type of y: float, in

y Y-coordinate of the vertex (default = 0).
type of z: float, in

z Z-coordinate of the vertex (default = 0).
return type of : :py:class: **Vertex**

Returns

Vertex object located at the specified coordinates.

◆ current_selection_count()

def cubit.current_selection_count ( )

Get the number of entities in the current selection list.

Returns how many entities have been picked and are available for navigation.

.. code-block:: python

```
import cubit

count = cubit.current_selection_count
()

print("Selections:"
, count)
```

@n return type of : int

Returns

Number of entities currently selected.

◆ curve()

def cubit.curve (*id_in*)

Retrieve a curve object by its ID.

Retrieves the curve object corresponding to the provided ID.

```
.. code-block:: python
```

```
                import cubit


                # Create a 10x10x10 brick (curve
                IDs include edges).


                cubit.cmd
                ("brick x 10" )


                curve = cubit.curve
                (1)


                print("Curve ID:" ,
                curve.id())
```

```
@n type of id_in:  int
```

Parameters

id_in ID of the curve to retrieve.
return type of : :py:class: `Curve`

Returns

Curve object corresponding to the specified ID.

◆ cylinder()

def cubit.cylinder (*height*,
x_radius,
y_radius,
top_radius
)

Create a cylinder or truncated cone of specified dimensions.

Creates a cylindrical or conical body based on provided bottom and top radii. A zero top radius yields a cone; a top radius equal to the x and y radii yields a straight cylinder.

```
.. code-block:: python
```

```
                import cubit


                # Create a cone: height=5,
                x_radius=2, y_radius=2, top_radius=0.


                my_cone = cubit.cylinder
                (5, 2, 2, 0)


                print("Cone ID:" ,
                my_cone.id())


                print("Cone volume:"
                , my_cone.volume())


                # Create a straight cylinder:
                height=5, x_radius=2, y_radius=2,
    top_radius=2.


                my_cylinder = cubit.cylinder
                (5, 2, 2, 2)


                print("Cylinder ID:"
                , my_cylinder.id())


                print("Cylinder
                volume:" , my_cylinder.volume())
```

```
@n type of height:  float, in
```

Parameters

height Height of the cylinder.
type of x_radius: float, in

x_radius Radius in the x direction (base ellipse x-axis).
type of y_radius: float, in

y_radius Radius in the y direction (base ellipse y-axis).
type of top_radius: float, in

top_radius Radius at the top: 0 for cone, equal to x and y radii for straight cylinder.
return type of : :py:class: `Body`

Returns

Body object representing the newly created cylinder or cone.

◆ delete_all_groups()

def cubit.delete_all_groups ( )

Remove all groups from the current Cubit session.

.. code-block:: python

```
# Create some groups.

cubit.cmd
("brick x 10" )

cubit.cmd
("group \"g1\" add vertex
all" )

cubit.cmd
("group \"g2\" add volume
all" )

# Delete all groups.

cubit.delete_all_groups
()

# Verify that no groups remain by
querying entities of type "group".

groups = cubit.get_entities
("group" )

print("Remaining
groups:" , len(groups)) #
prints "0"
```

```
@n return type of :  void
```

Returns
None.

◆ delete_group()

def cubit.delete_group (*group_id*)

Delete a specific group by ID.

.. code-block:: python

```
# Create and name two
groups.


cubit.cmd
("brick x 10" )


cubit.cmd
("group \"g1\" add vertex
all" )


cubit.cmd
("group \"g2\" add volume
all" )


# Retrieve ID of "g1" and delete
it.


g1_id = cubit.get_id_from_name
("g1" )


cubit.delete_group
(g1_id)


# Verify deletion: remaining
groups should not include g1.


groups = cubit.get_entities
("group" )


print("Remaining
groups:" , groups) #
prints list containing only g2's ID
```

```
@n type of group_id:  int
```

Parameters

group_id ID of the group to delete.

return type of : void

Returns

None.

◆ destroy()

def cubit.destroy ( )

Shut down Cubit and close the active journal file.

Flushes any pending commands, closes the journal, and releases Cubit resources. After calling **destroy()** , no further Cubit calls should be made.

```
.. code-block:: python
```

```
import cubit


status = cubit.destroy
()
```

return type of : int

Returns

0 on success; non-zero error code on failure.

◆ developer_commands_are_enabled()

def cubit.developer_commands_are_enabled ( )

Check if developer commands are enabled.

Returns true if Cubit is running in developer mode, allowing access to internal or experimental commands.

```
.. code-block:: python
```

```
import cubit


# Enable developer
commands


cubit.cmd
("set dev on" )



# Check developer mode
status
```

if **cubit.developer_commands_are_enabled()** : print("Developer commands are enabled.")

```
  @n return type of :  boolean
```

Returns

True if developer mode is active; false otherwise.

◆ do_ML_predictions()

```
def cubit.do_ML_predictions (  ml_op_names,
                               entity1_ids,
                               entity2_ids,
                               params,
                               mesh_size,
                               reduced_features = False
                            )
```

Compute ML predictions for a list of operations with multiple entities.

This is a batch variant of **get_ML_predictions()** that accepts lists of entity IDs per operation, enabling more flexible inputs. Features are computed and predictions are generated via a scikit-learn ensemble of decision trees.

type of ml_op_names: std::vector< std::string,std::allocator< std::string > >

Parameters

ml_op_names	Vector of ML operation or model names. type of entity1_ids: std::vector< std::vector< size_t,std::allocator< size_t > >,std::allocator< std::vector< size_t,std::allocator< size_t > > > >
entity1_ids	Vector of vectors of primary entity IDs for each operation. type of entity2_ids: std::vector< std::vector< size_t,std::allocator< size_t > >,std::allocator< std::vector< size_t,std::allocator< size_t > > > >
entity2_ids	Vector of vectors of secondary entity IDs for each operation. type of params: std::vector< std::vector< double,std::allocator< double > >,std::allocator< std::vector< double,std::allocator< double > > > >
params	Vector of parameter lists for each operation. type of mesh_size: float
mesh_size	Target mesh size used in feature calculation. type of reduced_features: boolean, optional
reduced_features	If true (where supported), use the reduced feature set for faster inference. return type of : std::vector< std::vector< double,std::allocator< double > >,std::allocator< std::vector< double,std::allocator< double > > > >

Returns

Outer vector matches ml_op_names length; each inner vector contains the predicted values for that operation.

.. code-block:: python

```
                ml_op_names = ['volume_no_op' ,
    'classify_surface' ]

                entity1_ids = [[5, 10], [20, 25]]

                entity2_ids = [[0, 0], [0, 0]]

                params = [[-1, -1, -1], [-1, -1, -1]]

                mesh_size = 1.5

                predictions = cubit.do_ML_predictions
                (

                ml_op_names, entity1_ids, entity2_ids,
    params,
                mesh_size)
```

◆ entity_exists()

def cubit.entity_exists (*entity_type,*
id
)

Check whether an entity of the specified type and ID exists.

This function returns `true` if an entity of the given type and ID exists in the model; otherwise it returns `false` .

Supported entity types include: "vertex", "curve", "surface", "volume", "body", etc.

`.. code-block:: python`

```
# Check whether surface with ID
12 exists

exists = cubit.entity_exists
("surface" , 12)

print(f"Does surface 12
exist? {'Yes' if exists else 'No'}" )
```

`@n type of entity_type:  string`

Parameters

entity_type Type of the entity being queried (e.g. "surface", "curve", "volume").
type of id: int

id ID of the entity.
return type of : boolean

Returns

`true` if the entity exists, `false` otherwise.

◆ evaluate_exterior_angle()

def cubit.evaluate_exterior_angle (*curve_list*, *test_angle*)

Find curves whose exterior angle between adjacent surfaces is less than a given threshold.

This function implements the Cubit "draw curve with exterior_angle < test_angle" test. For each curve in curve_list, it computes the exterior angle (the angle on the outside of the volume) between the two faces sharing that curve and returns those curves whose exterior angle is below test_angle.

Example setup:

.. code-block:: python

```
# Create two 10x10x10
bricks.

cubit.cmd
("brick x 10" )
# volume 1

cubit.cmd
("brick x 10" )
# volume 2

# Move and subtract to form
L-shaped volume.

cubit.cmd
("vol 2 move x 5 y
5" )

cubit.cmd
("subtract vol 2 from vol
1" )

# Get all curve IDs.

all_curves = cubit.get_current_ids
("curve" )

# Find curves with exterior angle
< 100 degrees (returns [27]).

result = cubit.evaluate_exterior_angle
(all_curves, 100.0)

print(result) # prints
(27,)
```

@n type of curve_list: std::vector< int,std::allocator< int > >

Parameters

curve_list Vector of curve IDs to test.
type of test_angle: float

test_angle Exterior angle threshold in degrees; curves with exterior angle < test_angle are returned.
return type of : std::vector< int,std:: allocator< int > >

Returns

A list (vector<int> or Python tuple) of curve IDs satisfying the test.

◆ evaluate_exterior_angle_at_curve()

def cubit.evaluate_exterior_angle_at_curve (*curve_id*, *volume_id*)

Return the exterior angle at a single curve with respect to a volume.

Computes the exterior angle (in degrees) on the outside of the specified volume between the two faces sharing the given curve.

.. code-block:: python

```
                # Create two 10x10x10 bricks and
                subtract to form L-shape.

                cubit.cmd
                ("brick x 10" )

                cubit.cmd
                ("brick x 10" )

                cubit.cmd
                ("vol 2 move x 5 y
                5" )

                cubit.cmd
                ("subtract vol 2 from vol
                1" )

                # Query the exterior angle at
                curve 27 on volume 1.

                angle =
    cubit.evaluate_exterior_angle_at_curve
                (27, 1)

                print("Exterior angle at
                curve 27 on volume 1:" , angle) #
    prints 90
```

@n type of curve_id: int

Parameters

curve_id ID of the curve.
type of volume_id: int

volume_id ID of the volume.
return type of : float

Returns

Exterior angle in degrees between the two faces sharing the curve on the specified volume.

◆ evaluate_surface_angle_at_vertex()

def cubit.evaluate_surface_angle_at_vertex (*surf_id*, *vert_id*)

Return the interior angle at a vertex on a specified surface.

Computes the angle in degrees between the two edges meeting at vert_id on surf_id, measured inside the surface. For a planar cube face, each corner vertex has an interior angle of 90 degrees.

.. code-block:: python

```
                # Create a 10x10x10 brick
                centered at the origin; surface 1 is a
    square
                face.

                cubit.cmd
                ("brick x 10" )

                # Vertex 1 on surface 1 is a
                corner with interior angle of 90
    degrees.

                angle =
    cubit.evaluate_surface_angle_at_vertex
                (1, 1)

                print("Interior angle at
                vertex 1 on surface 1:" , angle) #
    prints 90
```

@n type of surf_id: int

Parameters

surf_id ID of the surface.
type of vert_id: int

vert_id ID of the vertex on that surface.
return type of : float

Returns

Interior angle in degrees at the specified vertex on the given surface.

◆ exodus_sizing_function_file_exists()

def cubit.exodus_sizing_function_file_exists ( )

Check if the Exodus sizing function file currently exists.

Returns true if an Exodus II file has been imported as a sizing function and still resides on disk; returns false otherwise.

```
.. code-block:: python
```

```
                import cubit


                # After importing a field
                function:


                # cubit.cmd("Import Sizing
                Function 'adapt.exo' Block 3 Variable
      'strain'
                Last")


                exists =
      cubit.exodus_sizing_function_file_exists
                ()


                print("Sizing file
                exists?" , exists)
```

```
 @n return type of :  boolean
```

Returns

True if the imported Exodus II sizing file still exists on disk; false otherwise.

find_cone_surfaces()

def cubit.find_cone_surfaces (*surface_id*)

Identify cone surface(s) starting from a candidate surface.

Given a surface ID, this function determines if the surface is part of a cone. If the surface is a cone with an adjacent surface also part of the same cone, both surface IDs are returned. Otherwise, only the input surface ID is returned.

```
.. code-block:: python
```

```
                # Create a cylinder volume with a
                cone on top.


                cubit.cmd
                ('cylinder rad 1 height
                5' ) # Create
                cylinder


                cubit.cmd
                ('create frustum height 1
                radius 1 top 0' ) # Create
                cone (frustum with top radius 0)


                cubit.cmd
                ('vol 2 mov z 3' )
                # Move cone on top of
                cylinder


                cubit.cmd
                ('unite vol all' )
                # Unite volumes into one


                cubit.cmd
                ('split period vol
                1' ) # Split cylinder and
                cone surfaces



                cone_surfaces1 =
      cubit.find_cone_surfaces
                (4) # from frustum


                cone_surfaces2 =
      cubit.find_cone_surfaces
                (1) # from cylinder



                print("Cone surfaces
                starting from 4:" , [int(sid) for sid
      in cone_surfaces1])


                print("Cone surfaces
                starting from 1:" , [int(sid) for sid
      in cone_surfaces2])


                # Expected output:


                # Cone surfaces starting from 4:
                [4]


                # Cone surfaces starting from 1:
                [1]
```

```
 @n type of surface_id:  int
```

Parameters

surface_id **Surface** ID to test as a candidate cone.
return type of : std::vector< int,std:: allocator< int > >

Returns

List of surface IDs:

- If the input surface is a cone with a cone neighbor, both surface IDs are returned.
- If not, the list contains only the input surface ID.

find_overlapping_curves()

def cubit.find_overlapping_curves (*curve_ids*)

Identify overlapping curves in a specified list of curves.

The result is a vector of vectors. Each vector contains the IDs of curves that overlap with one another. Only overlaps between curves of different volumes or touching surfaces are reported.

```
.. code-block:: python
```

```
                # Create two volumes placed
                exactly next to each other so that surfaces
    are
                touching.


                cubit.cmd
                ('brick x 10' )
                # Create volume 1


                cubit.cmd
                ('brick x 10' )
                # Create volume 2


                cubit.cmd
                ('vol 2 mov x 10' )
                # Move volume 2 exactly next to
                volume 1



                curve_ids = cubit.get_entities
                ('curve' )
                # Get all curve IDs


                my_overlaps = cubit.find_overlapping_curves
                (curve_ids)



                print("Overlapping
                curves:" )
```

for overlap in my_overlaps: print("Curves:", tuple(int(cid) for cid in overlap))

```
                # Expected output:


                # Curves: (10, 21)


                # Curves: (1, 15)


                # Curves: (5, 19)


                # Curves: (12, 23)
```

```
 @n type of curve_ids:  std::vector< int,std::allocator< int > >
```

Parameters

curve_ids List of curve IDs to search for curve overlaps.
return type of : std::vector< std::vector< int,std::allocator< int > >,std::allocator< std::vector< int,std:: allocator< int > > > >

Returns

A list of curve overlap results. Each entry is a vector of curve IDs that overlap.

◆ flush_graphics()

def cubit.flush_graphics ( )

Force immediate rendering of pending graphics operations.

Ensures that any queued draw calls (for example, draw location or mesh preview) are displayed before subsequent commands execute. Requires an active graphics window.

```
.. code-block:: python
```

```
                import cubit


                cubit.cmd
                ("draw location 1 2
                3" )


                cubit.flush_graphics
                ()
```

◆ gather_surfaces_by_orientation()

def cubit.gather_surfaces_by_orientation (*seed_surf_ids*, *all_surf_ids*)

Gathers surfaces connected across shared edges, forming a surface enclosure.

Starting from a list of seed surfaces, recursively finds and returns all surfaces that are connected to them across shared edges. The resulting set typically forms a closed or connected surface group (enclosure).

If `all_surf_ids` is provided, the search is limited to those surfaces. If empty, all model surfaces are considered.

This function is useful for finding surface enclosures, identifying connected outer boundaries, or selecting surface groups in models with voids or non-manifold geometry.

`.. code-block:: python`

```
# Geometry setup

cubit.cmd
("brick x 10" )

cubit.cmd
("brick x 5" )

cubit.cmd
("subtract vol 2 from vol
1" )


seed_surf_ids = [1] # Starting
from one outer surface

all_surf_ids = [] # Search all
surfaces


gathered_surfs = cubit.gather_surfaces_by_orientation
(seed_surf_ids, all_surf_ids)

print("Gathered
surfaces:" , gathered_surfs)

# Expected behavior:

# Surfaces on outer boundary of
volume 1 will be gathered.

# Interior surfaces of the void
will not be included.
```

`@n type of seed_surf_ids:  std::vector< int,std::allocator< int > >`

Parameters

seed_surf_ids List of starting surface IDs to grow from.
type of all_surf_ids: std::vector< int,std::allocator< int > >

all_surf_ids Optional list of candidate surface IDs to search within (empty means all surfaces).

`@n return type of : std::vector< int,std:: allocator< int > >`

Returns

List of surface IDs connected across shared edges.

◆ get_2D_sheet_volumes()

def cubit.get_2D_sheet_volumes (*vol_id*)

Get associated 2D sheet volumes from a reduced 3D thin volume.

Returns the IDs of sheet bodies created from a 3D thin volume using reduce thin commands. These sheet volumes preserve thickness and loft attributes for shell FEA purposes.

```
.. code-block:: python
```

```
                thin_vol = 1001

                sheet_vols = cubit.get_2D_sheet_volumes
                (thin_vol)
```

```
@n type of vol_id:  int
```

Parameters

vol_id **Volume** ID representing the 3D thin volume.
return type of : std::vector< int,std:: allocator< int > >

Returns

IDs of 2D sheet volumes generated from the 3D volume.

◆ get_3D_thin_volume()

def cubit.get_3D_thin_volume (*vol_id*)

Get the original 3D thin volume associated with a 2D sheet volume.

Used to retrieve the parent 3D geometry from a 2D sheet volume generated by a reduce thin operation. Supports traceability in shell FEA workflows.

```
.. code-block:: python
```

```
                sheet_vol = 2002

                parent_thin_vol =
    cubit.get_3D_thin_volume
                (sheet_vol)
```

```
@n type of vol_id:  int
```

Parameters

vol_id **Volume** ID representing a 2D sheet volume.
return type of : int

Returns

ID of the associated 3D thin volume.

◆ get_acceleration_combine_type()

def cubit.get_acceleration_combine_type (*entity_id*)

Get the combine method for an acceleration BC.

Valid options include "Overwrite", "Average", "SmallestCombine", or "LargestCombine".
type of entity_id: int

Parameters

entity_id ID of the acceleration BC.
return type of : string

Returns

Combine method as a string.

◆ get_acis_version()

def cubit.get_acis_version ( )

Retrieve the ACIS kernel version string.

Returns the ACIS geometric modeling kernel version used by Cubit.

```
.. code-block:: python
```

```
                import cubit

                acis_version = cubit.get_acis_version
                ()

                print("ACIS
                version:" , acis_version)
```

```
@n return type of :  string
```

Returns

A string containing the ACIS version number.

◆ get_acis_version_as_int()

def cubit.get_acis_version_as_int ( )

Retrieve the ACIS kernel version as an integer.

Returns the ACIS version encoded as an integer (e.g., 202107 for version 2021.07).

```
.. code-block:: python
```

```
                # Python example

                import cubit

                acis_version_int =
    cubit.get_acis_version_as_int
                ()

                print("ACIS version
                (int):" , acis_version_int)
```

```
@n return type of :  int
```

Returns
An integer representing the ACIS version.

◆ get_adjacent_surfaces()

def cubit.get_adjacent_surfaces (*geometry_type*, *entity_id*)

Get a list of surfaces adjacent to a specified surface (including the surface itself).

For a given surface, this returns all surfaces that either own the specified entity or share a boundary with it. The returned list includes the queried surface as well.

```
.. code-block:: python
```

```
                # Create a 10x10x10 brick
                centered at the origin; surfaces 1-6
    are
                generated.

                cubit.cmd
                ("brick x 10" )


                # Query adjacent surfaces to
                surface 1.

                adj_surfs = cubit.get_adjacent_surfaces
                ("surface" , 1)

                print(adj_surfs) # prints (1, 6,
                5, 4, 3)
```

```
@n type of geometry_type:  string
```

Parameters

geometry_type	The type of the entity ("surface" for this function). type of entity_id: int
entity_id	ID of the surface to query. return type of : std::vector< int,std:: allocator< int > >

Returns
A list (vector or Python tuple) of surface IDs adjacent to-and including-the queried surface.

◆ get_adjacent_volumes()

def cubit.get_adjacent_volumes (*geometry_type*, *entity_id*)

Get a list of adjacent volumes to a specified entity.

For a specified entity, find all volumes that own the entity and volumes that touch the volume that owns this entity.

.. code-block:: python

```
                volume_id_list =
      cubit.get_adjacent_volumes
                ("curve" , 22)
```

Parameters

geom_type Specifies the geometry type of the entity type of entity_id: int

entity_id Specifies the id of the entity return type of : std::vector< int,std:: allocator< int > >

Returns

A list (python tuple) of volume ids

◆ get_all_cfd_bcs()

def cubit.get_all_cfd_bcs ( )

◆ get_all_exodus_times()

def cubit.get_all_exodus_times (*filename*)

Get all available time steps from an Exodus file.

Opens an Exodus II file and returns a vector of all stored time values (time steps). These time values are used when importing deformed meshes (via the `Time <time>` or `Step <step>` options) to select which deformation state to import. The last time in the list corresponds to the default import time.

.. code-block:: python

```
                # Query all available time steps
                in an Exodus file

                times = cubit.get_all_exodus_times
                ("deformed_mesh.e" )

                # Print the times

                print("Available time
                steps:" )
```

for t in times: print(" Time:", t)

```
                # Expected output
                (example):

                # Available time steps:

                # Time: 0.0

                # Time: 1.0

                # Time: 2.0

                # Time: 3.0
```

```
 @n type of filename:  string
```

Parameters

filename Fully qualified Exodus II file name. return type of : std::vector< double,std:: allocator< double > >

Returns

Vector (Python tuple) of available time steps stored in the file.

◆ get_all_exodus_variable_names()

def cubit.get_all_exodus_variable_names (*filename*, *variable_type*)

Get all variable names of a given type from an Exodus file.

Opens an Exodus II file and returns a vector of all variable names of the requested type. These variable names are used when importing nodal or element variable data into Cubit via the `nodal_var` and `element_var` import options.

Valid variable types:

- "g' `= global variables` -'n' `= nodal variables` -'e`` = element variables

`.. code-block:: python`

```
# Query all nodal variable names
in an Exodus file

nodal_vars =
cubit.get_all_exodus_variable_names
("deformed_mesh.e" ,
"n" )

# Print the variable names

print("Available nodal
variables:" )
```

for var_name in nodal_vars: print(" ", var_name)

```
# Expected output
(example):

# Available nodal
variables:

# displacement_x

# displacement_y

# displacement_z

# temperature
```

`@n type of filename:  string`

Parameters

filename	Fully qualified Exodus II file name. type of variable_type: string
variable_type	Variable type: 'g' (global), 'n' (nodal), or 'e' (element). return type of : std::vector< std::string,std::allocator< std:: string > >

Returns

Vector (Python tuple) of variable names of the requested type stored in the file.

◆ get_all_geometric_owners()

def cubit.get_all_geometric_owners (*mesh_entity_type*, *mesh_entity_list*)

Get the list of geometric owners for a set of mesh entities and their child entities.

Returns the geometric owners of the specified mesh entities and any child entities (e.g., edges or nodes of a quad).

Title Supported mesh entity types

- quad
- face
- tri
- hex
- tet
- edge
- node

.. code-block:: python

```
# Get owners of quads 1-8 and
their child edges/nodes


owners = cubit.get_all_geometric_owners
("quad" ,
"1 to 8" )


print("Owners:" ,
owners) # Output: ['surface 1',
'edge 1', 'node 1', 'node 2', ...]
```

@n type of mesh_entity_type: string

Parameters

mesh_entity_type Mesh entity type: quad, face, tri, hex, tet, edge, or node.
type of mesh_entity_list: string

mesh_entity_list Space-delimited mesh entity IDs (e.g., "all", "1 to 8", "1 2 3").
return type of : std::vector< std::string,std::allocator< std:: string > >

Returns

Vector of owner strings for each specified entity and its children.

◆ get_all_ids_from_name()

def cubit.get_all_ids_from_name (*geo_type*, *name*)

Retrieve all IDs of entities of a specified geometry type whose names start with a given prefix.

.. code-block:: python

```
# Create and name two
volumes.


cubit.cmd
("volume 1 name
'member_2A'" )


cubit.cmd
("volume 3 name
'member_2B'" )


# Query all volume IDs with names
starting with "member_2".


ids = cubit.get_all_ids_from_name
("volume" ,
"member_2" )


print("Found IDs:" ,
ids) # prints "[1, 3]"
```

@n type of geo_type: string

Parameters

geo_type Geometry entity type ("vertex", "curve", "surface", or "volume").
type of name: string

name Prefix of the entity names to search for.
return type of : std::vector< int,std:: allocator< int > >

Returns

Vector of integer IDs of entities matching the name prefix.

◆ get_aprepro_numeric_value()

def cubit.get_aprepro_numeric_value (*variable_name*)

Get the numeric value of a specified Aprepro variable.

Aprepro is a lightweight macro language in Cubit used for variable substitution and simple scripting.

```
.. code-block:: python
```

```
                    # Define an Aprepro variable with
                    a numeric value.

                    cubit.cmd
                    ("#{test_var=3.14}"
                    )

                    val = cubit.get_aprepro_numeric_value
                    ("test_var" )

                    print("test_var =" ,
                    val) # prints 3.14
```

```
 @n type of variable_name:  string
```

Parameters

variable_name Name of the Aprepro variable to query.
return type of : float

Returns

The variable's value as a double, or CUBIT_DBL_MAX on failure.

◆ get_aprepro_value_as_string()

def cubit.get_aprepro_value_as_string (*variable_name*)

Get the string value of an Aprepro variable.

Aprepro is a lightweight macro language in Cubit used for variable substitution and simple scripting.

```
.. code-block:: python
```

```
                    # Define an Aprepro string
                    variable.

                    cubit.cmd
                    ("#{mesh_scheme='pave'}" )

                    val = cubit.get_aprepro_value_as_string
                    ("mesh_scheme" )

                    print("mesh_scheme
                    =" , val) # prints
                    "pave"
```

```
 @n type of variable_name:  string
```

Parameters

variable_name Name of the Aprepro variable to query.
return type of : string

Returns

The string value of the specified Aprepro variable.

◆ get_aprepro_vars()

def cubit.get_aprepro_vars ( )

Retrieve the current Aprepro variable names.

Aprepro is a lightweight macro language in Cubit used for variable substitution and simple scripting.

```
.. code-block:: python
```

```
                # Define two Aprepro
                variables.


                cubit.cmd
                ("#{first_var=1}" )


                cubit.cmd
                ("#{second_var=2}" )


                # Query the list of Aprepro
                variables.


                vars = cubit.get_aprepro_vars
                ()


                print(vars) # prints
                ('first_var', 'second_var')
```

```
@n return type of : std::vector< std::string,std::allocator< std:: string > >
```

Returns

A list (vector<int> or Python tuple) of the current Aprepro variable names.

◆ get_arc_center_radius()

def cubit.get_arc_center_radius (*curve_id*)

Get the center point and radius of a specified arc curve.

If the curve is a circular or elliptical arc (e.g., an end circle of a cylinder), returns its center (X,Y,Z) and constant radius. For any other curve type, returns {0,0,0,0} and emits a warning.

```
.. code-block:: python
```

```
                    # 1) Create a cylinder of radius
                    3 and height 2

                    cubit.cmd
                    ("cylinder rad 3 height
                    2" )


                    # 2) Query one of its end-circle
                    curves (ID 2)

                    curve_id = 2

                    arc_data = cubit.get_arc_center_radius
                    (curve_id)

                    print(f"Center =
                    ({arc_data[0]:.0f}, {arc_data[1]:.0f},
                    {arc_data[2]:.0f})" )

                    print(f"Radius =
                    {arc_data[3]:.0f}" )

                    # Expected output:

                    # Center = (0, 0, 1)

                    # Radius = 3
```

```
@n type of curve_id:  int
```

Parameters

curve_id ID of the RefEdge to query.
return type of : std:: array< double,4 >

Returns

Array of four doubles: [0] = center.x, [1] = center.y, [2] = center.z, [3] = radius; returns {0,0,0,0} on unsupported curve types or failure.

◆ get_arc_length()

def cubit.get_arc_length (*curve_id*)

Get the parametric (arc) length of a specified curve.

Returns the length measured along the curve's parameterization. For most curves in Cubit, this matches the physical 3D length, but for certain spline or NURBS representations, arc length may be computed by integrating the parametric form.

```
.. code-block:: python
```

```
                    # Create a cylinder of radius 1
                    (curves 1 and 2 are circular edges).


                    cubit.cmd
                    ("create cylinder radius 1
                    height 2" )


                    # Curve 1 is a circle of radius
                    1: arc length = 2*PI.


                    arc_length = cubit.get_arc_length
                    (1)


                    print("Curve 1 arc
                    length:" , arc_length) #
                    prints approximately 6.28319
```

```
@n type of curve_id:  int
```

Parameters

curve_id ID of the curve to query.

return type of : float

Returns

The parametric (arc) length of the curve in model units.

◆ get_assembly_children()

def cubit.get_assembly_children (*assembly_id*)

◆ get_assembly_classification_category()

def cubit.get_assembly_classification_category ( )

Get classification category metadata.

return type of : string

Returns

Classification category as a string.

◆ get_assembly_classification_level()

def cubit.get_assembly_classification_level ( )

Get classification level metadata.

return type of : string

Returns

Classification level as a string.

◆ get_assembly_description()

def cubit.get_assembly_description (*assembly_id*)

Get the description of an assembly node.

type of assembly_id: int

Parameters
assembly_id ID of the assembly node.
return type of : string

Returns
Description string.

◆ get_assembly_file_format()

def cubit.get_assembly_file_format (*assembly_id*)

Get the file format from which the assembly node was imported.

type of assembly_id: int

Parameters
assembly_id ID of the assembly node.
return type of : string

Returns
File format string, e.g., "ACIS" or "STEP".

◆ get_assembly_instance()

def cubit.get_assembly_instance (*assembly_id*)

Get the instance number of an assembly node.

Distinguishes nodes with the same name.
type of assembly_id: int

Parameters
assembly_id ID of the assembly node.
return type of : int

Returns
Instance number.

◆ get_assembly_items()

def cubit.get_assembly_items ( )

◆ get_assembly_level()

def cubit.get_assembly_level (*assembly_id*)

Get the hierarchy level of an assembly node.

Level 0 is the root.
type of assembly_id: int

Parameters
assembly_id ID of the assembly node.
return type of : int

Returns
Integer hierarchy level.

◆ get_assembly_material_description()

def cubit.get_assembly_material_description (*assembly_id*)

Get the material description for a part.

type of assembly_id: int

Parameters
assembly_id ID of the part node.
return type of : string

Returns
Material description string.

◆ get_assembly_material_specification()

def cubit.get_assembly_material_specification (*assembly_id*)

Get the material specification for a part.

type of assembly_id: int

Parameters
assembly_id ID of the part node.
return type of : string

Returns
Material specification string.

◆ get_assembly_metadata()

def cubit.get_assembly_metadata (*volume_id*, *data_type*)

Get metadata for a specified volume.

Returns metadata associated with a volume's part, such as part number, description, material info, or file reference.
type of volume_id: int

Parameters
volume_id ID of the volume.
type of data_type: int
data_type Integer representing the metadata type: 1 = Part Number, 2 = Description, 3 = Material Description, 4 = Material Specification, 5 = Assembly Path, 6 = Original File.
return type of : string

Returns
Corresponding metadata string.

◆ get_assembly_name()

def cubit.get_assembly_name (*assembly_id*)

Get the name of an assembly node.

type of assembly_id: int

Parameters
assembly_id ID of the assembly node.
return type of : string

Returns
Name string of the node.

◆ get_assembly_path()

def cubit.get_assembly_path (*assembly_id*)

Get the full path of an assembly node, identifying its hierarchy.

type of assembly_id: int

Parameters
assembly_id ID of the assembly node.
return type of : string

Returns
Full path string, e.g., "/root/subassembly/part".

◆ get_assembly_type()

def cubit.get_assembly_type (*assembly_id*)

Get the type of an assembly node ("part" or "assembly").

type of assembly_id: int

Parameters
assembly_id ID of the assembly node.
return type of : string

Returns
Type string.

◆ get_assembly_units()

def cubit.get_assembly_units (*assembly_id*)

Get the units of measurement used in the assembly node.

type of assembly_id: int

Parameters

assembly_id ID of the assembly node.
return type of : string

Returns

Units string, e.g., "mm", "inch".

◆ get_assembly_weapons_category()

def cubit.get_assembly_weapons_category ( )

Get weapons category metadata.

return type of : string

Returns

Weapons category as a string.

◆ get_auto_size()

def cubit.get_auto_size (*geometry_type*,
entity_id_list,
auto_factor
)

Predict the automatic mesh size for a set of entities.

Calculates the mesh size (target edge length) that would be applied if the command 'size auto factor n' were issued on the given entities. This does not modify the model or set any sizes-it only returns the value.

```
.. code-block:: python
```

```
import cubit

# Predict auto mesh size with
factor 3 on surfaces 4 and 5

size = cubit.get_auto_size
("surface" , [4, 5],
3.0)

print("Predicted auto mesh
size:" , size)
```

```
@n type of geometry_type:  string
```

Parameters

geometry_type **Entity** type: "curve", "surface", or "volume".
type of entity_id_list: std::vector< int,std::allocator< int > >

entity_id_list Vector of IDs for the specified entities.
type of auto_factor: float

auto_factor The factor n used in `size auto factor n`.
return type of : float

Returns

The mesh size that would result from auto-sizing.

◆ get_bc_id_list()

def cubit.get_bc_id_list (*bc_type_enum*)

Get active boundary condition (BC) IDs of a specified type.

Returns a list of active BC IDs matching the given type enum.

type of bc_type_enum: int

Parameters

bc_type_enum Boundary condition type enum (CI_BCTypes). FEA types: 1-9, CFD types: 10-30.
return type of : std::vector< int,std:: allocator< int > >

Returns

List of active BC IDs.

◆ get_bc_name()

def cubit.get_bc_name (*bc_type_enum*, *bc_id*)

Get the name of a specific boundary condition (BC).

Retrieves the name associated with a given BC ID and type.

type of bc_type_enum: int

Parameters

bc_type_enum BC type enum (CI_BCTypes).
type of bc_id: int

bc_id ID of the boundary condition.
return type of : string

Returns

Name string of the specified BC.

◆ get_bc_temperature()

def cubit.get_bc_temperature (*bc_type_enum*, *entity_id*, *temp_type_enum*)

Get the temperature value for a specified BC area.

type of bc_type_enum: int

Parameters

bc_type_enum BC type (temperature = 4, convection = 7).
type of entity_id: int

entity_id ID of the BC.
type of temp_type_enum: int

temp_type_enum CI_BCEntityTypes for area: 2 (solid convection), 3 (solid temp), 7 (top), 8 (bottom), 9 (gradient), 10 (middle).
return type of : float

Returns

Temperature value.

◆ get_blend_chain_collections()

def cubit.get_blend_chain_collections (*volume_list*, *radius_threshold*)

Return collections of surfaces that form blend chains in the specified volumes.

A blend chain is a group of contiguous surfaces that together form a smooth transition feature, such as a fillet or round. Blend chains are filtered by their computed radius.

If radius_threshold is provided, only blend chains with radius less than this value are returned.

.. code-block:: python

```
cubit.cmd
("brick x 10" )

cubit.cmd
("modify curve 2 12 8 11
blend radius 1" ) #
Surfaces 7, 8, 9, 10 are created


volume_ids = [1]

radius_thresh = 2.0


blend_collections =
cubit.get_blend_chain_collections
(volume_ids, radius_thresh)
```

for surfaces, radius in blend_collections: print("Blend chain surfaces:", tuple(surfaces), "Radius:", radius)

```
# Expected output:

# Blend chain surfaces: (7, 8, 9,
10), Radius: 1.0
```

@n type of volume_list: std::vector< int,std::allocator< int > >

Parameters

volume_list List of volume IDs to query.
type of radius_threshold: float

radius_threshold Return only blend chains whose computed radius is less than this value.
return type of : std::vector< std::pair< std::vector< int,std::allocator< int > >,double >,std::allocator< std::pair< std::vector< int,std::allocator< int > >,double > > >

Returns

: A list of pairs each pair contains (1) a list of surface IDs defining a blend chain, and (2) the computed radius of the blend chain.

◆ get_blend_chains()

def cubit.get_blend_chains (*surface_id*)

Returns the blend chains for a given surface.

Given a surface ID, this function returns all blend chains associated with that surface. If the surface is part of one or more blend chains, each chain will be returned as a list of surface IDs.

.. code-block:: python

```
cubit.cmd
("brick x 10" )


cubit.cmd
("modify curve 12 2 8 11
blend radius 1" ) # Blend
the four curves on surface 5


surface_id = 10


blend_chains = cubit.get_blend_chains
(surface_id)
```

for chain in blend_chains: print("Blend chain:", tuple(chain))

```
# Expected output:


# Blend chain: (7, 8, 9,
10)
```

@n type of surface_id: int

Parameters

surface_id **Surface** to retrieve the blend chains from.
return type of : std::vector< std::vector< int,std::allocator< int > >,std::allocator< std::vector< int,std:: allocator< int > > > >

Returns

A list of lists of surface IDs in each blend chain.

Note

If using Python, lists will be returned as Python tuples.

◆ get_blend_surfaces()

def cubit.get_blend_surfaces (*target_volume_ids*)

Find blend (fillet) surfaces within specified volumes.

Iterates over all faces of volumes in target_volume_ids and returns those for which is_blend_surface returns true.

.. code-block:: python

```
# Create a 2x2x2 brick and fillet
edge 1 with radius 0.5.


cubit.cmd
("brick x 2 y 2 z 2"
)


cubit.cmd
("modify curve 1 blend
radius 0.5" )


# Identify blend surfaces in
volume 1.


blends = cubit.get_blend_surfaces
([1])


print(blends) # expect [7]
```

@n type of target_volume_ids: std::vector< int,std::allocator< int > >

Parameters

target_volume_ids Vector of volume IDs to examine.
return type of : std::vector< int,std:: allocator< int > >

Returns

Vector of IDs of surfaces that are blends (fillets).

◆ get_block_attribute_count()

def cubit.get_block_attribute_count (*block_id*)

Get the number of attributes assigned to a block.

Each block can store up to 20 attributes, which represent physical properties like material parameters.

```
.. code-block:: python
```

```
                cubit.cmd
                ("brick x 10" )

                cubit.cmd
                ("block 1 vol 1" )

                cubit.cmd
                ("block 1 attribute count
                3" )


                attr_count =
      cubit.get_block_attribute_count
                (1)

                print("Attribute
                count:" , attr_count)

                # Expected output: Attribute
                count: 3
```

```
 @n type of block_id:  int
```

Parameters

block_id The ID of the block to query.
return type of : int

Returns

The number of attributes currently assigned to the block.

◆ get_block_attribute_name()

def cubit.get_block_attribute_name (*block_id*, *attribute_index*)

Get the name of a specific attribute for a block.

Returns the name associated with the attribute at the specified index for a given block. Attribute names are typically assigned to describe the purpose or meaning of each attribute, such as "thickness" or "thermal_conductivity".

```
.. code-block:: python
```

```
                cubit.cmd
                ("brick x 10" )

                cubit.cmd
                ("block 1 vol 1" )

                cubit.cmd
                ("block 1 attribute count
                2" )

                cubit.cmd
                ("block 1 attribute index 1
                10.0" )

                cubit.cmd
                ("block 1 attribute name 1
                'thickness'" )


                attr_name =
      cubit.get_block_attribute_name
                (1, 1)

                print("Attribute name at
                index 1:" , attr_name)

                # Expected output: Attribute name
                at index 1: thickness
```

```
@n type of block_id:  int
```

Parameters

block_id ID of the block.
type of attribute_index: int

attribute_index Index of the attribute (1-based).
return type of : string

Returns

Name of the attribute at the specified index.

◆ get_block_attribute_value()

def cubit.get_block_attribute_value (*block_id*, *attribute_index*)

Get the value of a specific attribute for a block.

Returns the floating-point value of the attribute at the specified index for a given block. Attributes are user-defined values assigned to represent physical or material properties in a block, such as shell thickness or temperature.

.. code-block:: python

```
cubit.cmd
("brick x 10" )

cubit.cmd
("block 1 vol 1" )

cubit.cmd
("block 1 attribute count
2" )

cubit.cmd
("block 1 attribute index 2
5.5" )


attr_value =
cubit.get_block_attribute_value
(1, 2)

print("Attribute value at
index 2:" , attr_value)

# Expected output: Attribute
value at index 2: 5.5
```

@n type of block_id: int

Parameters

block_id ID of the block.
type of attribute_index: int

attribute_index Index of the attribute (1-based).
return type of : float

Returns

Value of the attribute at the specified index.

◆ get_block_count()

def cubit.get_block_count ( )

Get the current number of element blocks in the model.

In Cubit, an element block groups related mesh elements of the same type into a single entity. Blocks can be defined by geometric entities (volumes, surfaces, curves) or by directly specifying mesh entities. Once defined, all elements owned by those entities become part of the block.

.. code-block:: python

```
# Create a 10x10x10 brick and
mesh it.

cubit.cmd
("brick x 10" )

cubit.cmd
("mesh volume 1" )


# Define block 20 to contain
volume 1.

cubit.cmd
("Block 20 add volume
1" )


# get_block_count() returns
1.

count = cubit.get_block_count
()

print("Block count:"
, count) # prints 1
```

@n return type of : int

Returns

The number of element blocks currently defined in the model.

◆ get_block_curves()

def cubit.get_block_curves (*block_id*)

Get the list of curve IDs contained in a block.

This function returns the IDs of all curves that are part of the specified block.

```
.. code-block:: python
```

```
                # Create a brick and assign
                curves 1 to 12 to block 30


                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("block 30 curve 1 to
                12" )



                # Query the curves in block
                30


                curve_ids = cubit.get_block_curves
                (30)


                print(f"Block 30 contains
                {len(curve_ids)} curve(s)." )
```

for curve_id in curve_ids: print(f" Curve ID: {curve_id}")

```
  @n type of block_id:  int
```

Parameters

block_id ID of the block to examine.
return type of : std::vector< int,std:: allocator< int > >

Returns

List of curve IDs contained in the block.

◆ get_block_edges()

def cubit.get_block_edges (*block_id*)

Get the list of edge IDs contained in a block.

This function returns the IDs of all edges that are part of the specified block.

```
.. code-block:: python
```

```
                # Create a brick


                cubit.cmd
                ("brick x 10" )



                # Assign edges from curves in
                volume 1 to block 20


                cubit.cmd
                ("block 20 edge in curve in
                vol 1" )



                # Query the edges in block
                20


                edge_ids = cubit.get_block_edges
                (20)


                print(f"Block 20 contains
                {len(edge_ids)} edge(s)." )
```

for edge_id in edge_ids: print(f" Edge ID: {edge_id}")

```
  @n type of block_id:  int
```

Parameters

block_id ID of the block to examine.
return type of : std::vector< int,std:: allocator< int > >

Returns

List of edge IDs contained in the block.

◆ get_block_element_attribute_count()

def cubit.get_block_element_attribute_count (*block_id*)

Get the number of attributes defined for elements in the specified block.

This returns the number of attribute values associated with each element in a block. Attributes can represent properties like shell thickness, material coefficients, etc.

.. code-block:: python

```
cubit.cmd
("brick x 10" )

cubit.cmd
("block 1 vol 1" )

cubit.cmd
("block 1 attribute count
2" )


count =
cubit.get_block_element_attribute_count
(1)

print("Element attribute
count:" , count)

# Expected output: Element
attribute count: 2
```

@n type of block_id: int

Parameters

block_id The ID of the block.
return type of : int

Returns

Number of element-level attributes defined in the block.

◆ get_block_element_attribute_names()

def cubit.get_block_element_attribute_names (*block_id*)

Get the list of attribute names associated with block elements.

Returns the names of all attributes associated with the elements of a given block. These names describe the meaning of each attribute (e.g., "thickness", "material_id").

```
.. code-block:: python
```

```
                cubit.cmd
                ("brick x 10" )

                cubit.cmd
                ("block 1 vol 1" )

                cubit.cmd
                ("block 1 attribute count
                2" )

                cubit.cmd
                ("block 1 attribute index 1
                10.0" )

                cubit.cmd
                ("block 1 attribute index 2
                25.0" )

                cubit.cmd
                ("block 1 attribute name 1
                'thickness'" )

                cubit.cmd
                ("block 1 attribute name 2
                'density'" )


                names =
    cubit.get_block_element_attribute_names
                (1)
```

for name in names: print(name)

```
                # Expected output:

                # thickness

                # density
```

```
 @n type of block_id:  int
```

Parameters

block_id ID of the block.
return type of : std::vector<
std::string,std::allocator< std:: string > >

Returns

Vector of attribute names associated with the block elements.

◆ get_block_element_type()

def cubit.get_block_element_type (*block_id*)

Get the element type associated with a block.

Returns the finite element type (e.g., HEX8, HEX20, TET10) used in the specified block.

.. code-block:: python

```
cubit.cmd
("brick x 10" )


cubit.cmd
("block 1 vol 1" )


cubit.cmd
("block 1 element type
hex20" )


cubit.cmd
("mesh vol 1" )



elem_type =
cubit.get_block_element_type
(1)


print("Element
type:" , elem_type)


# Expected output: Element type:
HEX20
```

@n type of block_id: int

Parameters

block_id ID of the block to query.
return type of : string

Returns

A string indicating the element type used in the block.

◆ get_block_faces()

def cubit.get_block_faces (*block_id*)

Get the list of face IDs contained in a block.

This function returns the IDs of all face elements that are part of the specified block.

.. code-block:: python

```
# Create a brick and mesh volume
1


cubit.cmd
("brick x 10" )


cubit.cmd
("mesh vol 1" )



# Assign faces from surface 1 to
block 40


cubit.cmd
("block 40 face in surface
1" )



# Query the faces in block
40


face_ids = cubit.get_block_faces
(40)


print(f"Block 40 contains
{len(face_ids)} face(s)." )
```

for face_id in face_ids: print(f" Face ID: {face_id}")

@n type of block_id: int

Parameters

block_id ID of the block to examine.
return type of : std::vector< int,std:: allocator< int > >

Returns

List of face IDs contained in the block.

◆ get_block_hexes()

def cubit.get_block_hexes (*block_id*)

Get the list of hexahedron (hex) IDs contained in a block.

This function returns the IDs of all hexahedron (hex) elements that are part of the specified block.

```
.. code-block:: python
```

```
                    # Create a brick and mesh volume
                    1


                    cubit.cmd
                    ("brick x 10" )


                    cubit.cmd
                    ("mesh vol 1" )


                    # Assign hexes in volume 1 to
                    block 80


                    cubit.cmd
                    ("block 80 hex in vol
                    1" )


                    # Query the hexes in block
                    80


                    hex_ids = cubit.get_block_hexes
                    (80)


                    print(f"Block 80 contains
                    {len(hex_ids)} hex(s)." )
```

for hex_id in hex_ids: print(f" Hex ID: {hex_id}")

```
  @n type of block_id:  int
```

Parameters

block_id ID of the block to examine.
return type of : std::vector< int,std:: allocator< int > >

Returns

List of hexahedron (hex) IDs contained in the block.

◆ get_block_id()

def cubit.get_block_id (*entity_type*,
entity_id
)

Get the element block ID associated with a geometric entity.

Returns the ID of the element block that is associated with the specified curve, surface, or volume. A geometric entity may be assigned to only one block. If no block is assigned, this function returns 0.

```
.. code-block:: python
```

```
                # Create geometry and assign it
                to a block

                cubit.cmd
                ("brick x 10 y 10 z
                10" )

                cubit.cmd
                ("block 5 add volume
                1" )


                # Query the block ID associated
                with volume 1

                block_id = cubit.get_block_id
                ("volume" , 1)


                # Print the result

                print("Block ID for volume
                1:" , block_id)


                # Expected output
                (example):

                # Block ID for volume 1: 5
```

```
@n type of entity_type:  string
```

Parameters

entity_type Type of the entity: "curve", "surface", or "volume".
type of entity_id: int

entity_id ID of the entity.
return type of : int

Returns

Block ID associated with the entity, or zero (0) if none.

◆ get_block_id_list()

def cubit.get_block_id_list ( )

Get a list of all active block IDs.

Returns a vector of all currently active element block IDs in the model. A geometric entity may be assigned to only one block.

```
.. code-block:: python
```

```
                # Create geometry

                cubit.cmd
                ("brick x 10" )

                cubit.cmd
                ("brick x 10" )

                cubit.cmd
                ("vol 2 mov x 10" )


                # Create a blocks and assign
                volumes

                cubit.cmd
                ("block 10 vol 1" )

                cubit.cmd
                ("block 20 vol 2" )


                # Query all active block
                IDs

                block_ids = cubit.get_block_id_list
                ()


                # Print the block IDs

                print("Active block
                IDs:" )
```

for block_id in block_ids: print(" Block ID:", block_id)

```
                # Expected output
                (example):

                # Active block IDs:

                # Block ID: 10

                # Block ID: 20
```

```
@n return type of : std::vector< int,std:: allocator< int > >
```

Returns

Vector (Python tuple) of all active block IDs.

◆ get_block_ids()

def cubit.get_block_ids (*mesh_geometry_file_name*)

Get list of block IDs from a mesh geometry file.

Opens a mesh geometry file (Exodus II format) and returns a vector of all element block IDs defined in the file. This allows users to inspect available blocks before importing the mesh, and to selectively import blocks by ID.

.. code-block:: python

```
# Query all block IDs in a mesh
geometry file

block_ids = cubit.get_block_ids
("mesh_geometry.e" )


# Print the block IDs

print("Available block
IDs:" )
```

for block_id in block_ids: print(" Block ID:", block_id)

```
# Expected output
(example):

# Available block IDs:

# Block ID: 1

# Block ID: 5

# Block ID: 10
```

@n type of mesh_geometry_file_name: string

Parameters

mesh_geometry_file_name Fully qualified name of a mesh geometry file (Exodus II format).
return type of : std::vector< int,std:: allocator< int > >

Returns

Vector (Python tuple) of block IDs defined in the mesh geometry file.

◆ get_block_material()

def cubit.get_block_material (*block_id*)

Get the ID of the material assigned to the specified block.

Returns the material ID associated with the block. This ID corresponds to a defined material (as created with material commands). A value of 0 indicates that no material is currently assigned.

.. code-block:: python

```
cubit.cmd
("brick x 10" )

cubit.cmd
("block 1 vol 1" )

cubit.cmd
("material 1 name
'Steel'" )

cubit.cmd
("block 1 material
1" )


mat_id = cubit.get_block_material
(1)

print("Material ID:"
, mat_id)
```

@n type of block_id: int

Parameters

block_id ID of the block.
return type of : int

Returns

Material ID assigned to the block, or 0 if none.

◆ get_block_nodes()

def cubit.get_block_nodes (*block_id*)

Get the list of node IDs contained in a block.

This function returns the IDs of all nodes that are part of the specified block.

```
.. code-block:: python
```

```
                # Create a brick and mesh
                it


                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("mesh vol 1" )


                # Assign nodes from surface 1 to
                block 1


                cubit.cmd
                ("block 1 node in surface
                1" )


                # Query the nodes in block
                1


                node_ids = cubit.get_block_nodes
                (1)


                print(f"Block 1 contains
                {len(node_ids)} node(s)." )
```

for node_id in node_ids: print(f" Node ID: {node_id}")

```
  @n type of block_id:  int
```

Parameters

block_id ID of the block to examine.
return type of : std::vector< int,std:: allocator< int > >

Returns

List of node IDs contained in the block.

◆ get_block_pyramids()

def cubit.get_block_pyramids (*block_id*)

Get the list of pyramid IDs contained in a block.

This function returns the IDs of all pyramid elements that are part of the specified block.

Note
In Cubit, only one type of element can be assigned to a block. To separate element types, use multiple blocks. For example, after meshing with tets and pyramids, assign tets and pyramids to separate blocks as shown below.

```
.. code-block:: python
```

```
                    # Create two bricks and mesh them
                    with different schemes

                    cubit.cmd
                    ("brick x 10" )

                    cubit.cmd
                    ("vol 1 copy mov x
                    10" )

                    cubit.cmd
                    ("merge all" )


                    cubit.cmd
                    ("vol 1 scheme map"
                    )

                    cubit.cmd
                    ("mesh vol 1" )


                    cubit.cmd
                    ("vol 2 scheme
                    tetmesh" )

                    cubit.cmd
                    ("mesh vol 2" )


                    # Assign tets and pyramids in
                    volume 2 to separate blocks

                    cubit.cmd
                    ("block 2 tet in vol
                    2" )

                    cubit.cmd
                    ("block 3 pyramid in vol
                    2" )


                    # Query the pyramids in block
                    3

                    pyramid_ids = cubit.get_block_pyramids
                    (3)

                    print(f"Block 3 contains
                    {len(pyramid_ids)} pyramid(s)." )
```

for pyr_id in pyramid_ids: print(f" Pyramid ID: {pyr_id}")

```
  @n type of block_id:  int
```

Parameters

block_id ID of the block to examine.
return type of : std::vector< int,std:: allocator< int > >

Returns

List of pyramid IDs contained in the block.

◆ get_block_surfaces()

def cubit.get_block_surfaces (*block_id*)

Get the list of surface IDs contained in a block.

This function returns the IDs of all surfaces that are part of the specified block.

```
.. code-block:: python
```

```
                # Create a brick and assign
                surfaces 1 to 6 to block 20


                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("block 20 surface 1 to
                6" )



                # Query the surfaces in block
                20


                surface_ids = cubit.get_block_surfaces
                (20)


                print(f"Block 20 contains
                {len(surface_ids)} surface(s)." )
```

for surf_id in surface_ids: print(f" Surface ID: {surf_id}")

```
  @n type of block_id:  int
```

Parameters

block_id ID of the block to examine.
return type of : std::vector< int,std:: allocator< int > >

Returns

List of surface IDs contained in the block.

◆ get_block_tets()

def cubit.get_block_tets (*block_id*)

Get the list of tetrahedron (tet) IDs contained in a block.

This function returns the IDs of all tetrahedron (tet) elements that are part of the specified block.

```
.. code-block:: python
```

```
                # Create a brick and tetmesh
                volume 1


                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("vol 1 scheme
                tetmesh" )


                cubit.cmd
                ("mesh vol 1" )


                # Assign tets in volume 1 to
                block 60


                cubit.cmd
                ("block 60 tet in vol
                1" )


                # Query the tets in block
                60


                tet_ids = cubit.get_block_tets
                (60)


                print(f"Block 60 contains
                {len(tet_ids)} tet(s)." )
```

for tet_id in tet_ids: print(f" Tet ID: {tet_id}")

```
  @n type of block_id:  int
```

Parameters

block_id ID of the block to examine.
return type of : std::vector< int,std:: allocator< int > >

Returns

List of tetrahedron (tet) IDs contained in the block.

◆ get_block_tris()

def cubit.get_block_tris (*block_id*)

Get the list of triangle (tri) IDs contained in a block.

This function returns the IDs of all triangle elements that are part of the specified block.

```
.. code-block:: python
```

```
                # Create a brick and tetmesh
                volume 1


                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("vol 1 scheme
                tetmesh" )


                cubit.cmd
                ("mesh vol 1" )


                # Assign triangles from surface 1
                to block 30


                cubit.cmd
                ("block 30 tri in surface
                1" )


                # Query the triangles in block
                30


                tri_ids = cubit.get_block_tris
                (30)


                print(f"Block 30 contains
                {len(tri_ids)} triangle(s)." )
```

for tri_id in tri_ids: print(f" Tri ID: {tri_id}")

```
  @n type of block_id:  int
```

Parameters

block_id ID of the block to examine.
return type of : std::vector< int,std:: allocator< int > >

Returns

List of triangle (tri) IDs contained in the block.

◆ get_block_vertices()

def cubit.get_block_vertices (*block_id*)

Get the list of vertex IDs contained in a block.

This function returns the IDs of all vertices that are part of the specified block.

```
.. code-block:: python
```

```
                # Create a brick and assign
                vertices 1 to 8 to block 40


                cubit.cmd
                ("brick x 1" )


                cubit.cmd
                ("block 40 vertex 1 to
                8" )


                # Query the vertices in block
                40


                vertex_ids = cubit.get_block_vertices
                (40)


                print(f"Block 40 contains
                {len(vertex_ids)} vertex(s)." )
```

for vertex_id in vertex_ids: print(f" Vertex ID: {vertex_id}")

```
  @n type of block_id:  int
```

Parameters

block_id ID of the block to examine.
return type of : std::vector< int,std:: allocator< int > >

Returns

List of vertex IDs contained in the block.

◆ get_block_volumes()

def cubit.get_block_volumes (*block_id*)

Get the list of volume IDs contained in a block.

This function returns the IDs of all volumes that are part of the specified block.

```
.. code-block:: python
```

```
                # Create a brick and assign it to
                block 1


                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("block 1 vol 1" )


                # Query the volumes in block
                1


                volume_ids = cubit.get_block_volumes
                (1)


                print(f"Block 1 contains
                {len(volume_ids)} volume(s)." )
```

for vol_id in volume_ids: print(f" Volume ID: {vol_id}")

```
  @n type of block_id:  int
```

Parameters

block_id ID of the block to examine.
return type of : std::vector< int,std:: allocator< int > >

Returns

List of volume IDs contained in the block.

◆ get_block_wedges()

def cubit.get_block_wedges (*block_id*)

Get the list of wedge IDs contained in a block.

This function returns the IDs of all wedge elements that are part of the specified block.

```
.. code-block:: python
```

```
                # Create a brick and mesh surface
                1 with triangles

                cubit.cmd
                ("brick x 10" )

                cubit.cmd
                ("surface 1 scheme
                trimesh" )

                cubit.cmd
                ("mesh surface 1" )


                # Sweep the surface mesh to
                create wedges

                cubit.cmd
                ("vol 1 scheme
                sweep" )

                cubit.cmd
                ("mesh vol 1" )


                # Assign wedges in volume 1 to
                block 50

                cubit.cmd
                ("block 50 wedge in vol
                1" )


                # Query the wedges in block
                50

                wedge_ids = cubit.get_block_wedges
                (50)

                print(f"Block 50 contains
                {len(wedge_ids)} wedge(s)." )
```

for wedge_id in wedge_ids: print(f" Wedge ID: {wedge_id}")

```
  @n type of block_id:  int
```

Parameters

block_id ID of the block to examine.
return type of : std::vector< int,std:: allocator< int > >

Returns

List of wedge IDs contained in the block.

◆ get_blocks_with_materials()

def cubit.get_blocks_with_materials ( )

Get all block IDs and their associated material IDs.

Returns a list of all existing block IDs and the corresponding material ID assigned to each. This function helps identify which blocks are assigned materials. Blocks without an assigned material will have a material ID of 0 in the returned pair.

```
.. code-block:: python
```

```
                cubit.cmd
                ("brick x 10" )

                cubit.cmd
                ("block 1 vol 1" )

                cubit.cmd
                ("material 1 name
                'Steel'" )

                cubit.cmd
                ("block 1 material
                1" )


                block_mats = cubit.get_blocks_with_materials
                ()
```

for pair in block_mats: print("Block", pair[0], "has material", pair[1])

```
  @n return type of : std::vector< std::vector< int,std::allocator< int > >,std::allocator< std::vector< int,std:: allocator< int > > > >
```

Returns

List of [block_id, material_id] pairs. Material ID is 0 if unassigned.

◆ get_blunt_tangency_default_depth()

def cubit.get_blunt_tangency_default_depth (*vert_id*, *angle*, *add_material*)

Computes the default depth used to blunt a tangency at a vertex.

Used with the `blunt tangency` command to eliminate small angles caused by fillets. This function estimates the default `depth` parameter based on the specified angle and whether material is added or removed.

.. code-block:: python

```
cubit.cmd
("brick x 10" )

cubit.cmd
("cylinder rad 5 height
10" )

cubit.cmd
("vol 2 mov z 10" )

cubit.cmd
("unite vol 1 2" )


depth =
cubit.get_blunt_tangency_default_depth
(12, 10.0, True )

print("Default
depth:" , depth)
```

@n type of vert_id: int

Parameters

vert_id ID of the vertex to modify.
type of angle: float

angle Desired target angle after blunting.
type of add_material: boolean

add_material If true, material is added; if false, material is removed.

@n return type of : float

Returns

Suggested depth value for the operation.

◆ get_body_count()

def cubit.get_body_count ( )

Get the current number of bodies in the model.

.. code-block:: python

```
# Create two bricks as separate
bodies.

cubit.cmd
("brick x 10" )
# body 1

cubit.cmd
("brick x 5" )
# body 2

body_count = cubit.get_body_count
() # returns 2

print("Body count:"
, body_count)
```

@n return type of : int

Returns

The number of bodies currently defined in the model.

◆ get_bolt_axis()

def cubit.get_bolt_axis (*vol_id*)

Get the normalized axis vector of a bolt volume.

Returns the primary axis direction of the specified bolt geometry, normalized to unit length. If the volume is invalid or the axis cannot be determined, the returned vector will be [0, 0, 0].

```
.. code-block:: python
```

```
                axis = cubit.get_bolt_axis
                (2)

                print("Bolt axis:" ,
                axis)
```

type of vol_id: int

Parameters

vol_id **Volume** ID representing a bolt geometry.
return type of : std::vector< double,std:: allocator< double > >

Returns

Normalized axis vector of the bolt, or [0, 0, 0] if invalid.

◆ get_bolt_clamped_members()

def cubit.get_bolt_clamped_members (*vol_ids*,
nearby_vols,
progress = `None`
)

Identify clamped members associated with a bolt.

Returns the ordered list of volume IDs that represent the bolt's clamped components:

- Bearing surface member
- Sandwiched members (if any)
- Threaded member

If `nearby_vols` is empty, Cubit will automatically determine nearby volumes (less efficient).

```
.. code-block:: python
```

```
                clamped = cubit.get_bolt_clamped_members
                ([12], [])
```

for group in clamped: print(group)

```
@n type of vol_ids:  std::vector< int,std::allocator< int > >
```

Parameters

vol_ids List of bolt volume IDs.
type of nearby_vols: std::vector< std::vector< int,std::allocator< int > >,std::allocator< std::vector< int,std::allocator< int > > > >

nearby_vols Optional list of nearby volume IDs for each bolt. Leave empty to compute automatically.
type of progress: :py:class: `CubitProgressHandler`, optional

progress Optional CubitProgressHandler for monitoring progress.
return type of : std::vector< std::vector< int,std::allocator< int > >,std::allocator< std::vector< int,std:: allocator< int > > > >

Returns

: Nested list one entry per bolt, each a list of volume IDs representing clamped members.

◆ get_bolt_coordinate_system()

def cubit.get_bolt_coordinate_system (*geom_type*, *id*)

Get local coordinate system for a bolt or bolt hole.

Returns the coordinate system as three 3D points defining the origin, the Z-direction point, and a point in the XZ plane. This is useful for aligning or analyzing bolt and hole geometry in local space.

For hole-based queries, only one surface from the hole is needed; it must uniquely identify the hole.

.. code-block:: python

```
# For a bolt volume

bolt_coord =
cubit.get_bolt_coordinate_system
("bolt" , 12)

origin = bolt_coord[0]

z_axis_point = bolt_coord[1]

xz_plane_point = bolt_coord[2]


# For a hole surface

hole_coord =
cubit.get_bolt_coordinate_system
("hole" , 35)

h_origin = hole_coord[0]

h_z_axis_point = hole_coord[1]

h_xz_plane_point = hole_coord[2]
```

@n type of geom_type: string

Parameters

geom_type "bolt" for bolt volume or "hole" for hole surface.
type of id: int

id **Volume** ID for bolts or surface ID from the hole geometry.
return type of : std::vector< std::vector< double,std::allocator< double > >,std::allocator< std::vector< double,std:: allocator< double > > > >

Returns

: Three 3D points as vectors origin, Z-direction point, and XZ-plane point.

◆ get_bolt_diameters()

def cubit.get_bolt_diameters (*vol_ids*)

Get the shank diameters of specified bolt volumes.

Returns the estimated shank diameter for each volume in the input list. Volumes that are invalid or for which the diameter cannot be determined will return a value of 0.

.. code-block:: python

```
diameters = cubit.get_bolt_diameters
([2, 3])


print("Bolt diameter for
vol 2:" , diameters[0])


print("Bolt diameter for
vol 3:" , diameters[1])
```

type of vol_ids: std::vector< int,std::allocator< int > >

Parameters

vol_ids **Volume** IDs representing bolt geometries.
return type of : std::vector< double,std:: allocator< double > >

Returns

Vector of shank diameters (in same order as vol_ids).

◆ get_bolt_holes()

def cubit.get_bolt_holes (*geo_type*,
clamped_members,
radius_threshold,
gap_threshold
)

Identify upper and lower pilot hole surfaces from clamped members.

Analyzes the provided volumes or blocks to extract the top and bottom surfaces of pilot holes (e.g., for bolts) based on radius and proximity criteria. The output can be passed directly to reduce bolt commands.

.. code-block:: python

```
clamped_vols = [301, 302, 303]

upper_surfs, lower_surfs = cubit.get_bolt_holes
("volume" ,
clamped_vols, 6.0, 0.25)
```

@n type of geo_type: string

Parameters

geo_type	"volume" or "block", specifying clamped member type. type of clamped_members: std::vector< int,std::allocator< int > >
clamped_members	IDs of volumes or blocks with candidate pilot holes. type of radius_threshold: float
radius_threshold	Maximum radius for a hole to be considered. type of gap_threshold: float
gap_threshold	Maximum separation for concentric hole matching.

@n return type of : std::vector< std::vector< int,std::allocator< int > >,std::allocator< std::vector< int,std:: allocator< int > > > >

Returns

A vector of two surface ID lists:

- First list: upper hole surfaces.
- Second list: lower hole surfaces (may include more than one per hole).

◆ get_bolt_holes_info()

def cubit.get_bolt_holes_info (*geo_type*,
clamped_members,
radius_threshold,
gap_threshold
)

Identify concentric pilot holes across clamped members.

Analyzes a list of clamped volumes or blocks to identify sets of concentric holes that qualify as bolt pilot holes, based on a maximum radius and proximity criteria. Returns detailed information including hole surfaces, radii, axes, and associated volumes.

.. code-block:: python

```
clamped_vols = [301, 302, 303]

pilot_holes = cubit.get_bolt_holes_info
("volume" ,
clamped_vols, 6.0, 0.25)
```

for hole in pilot_holes: print("Bearing volume:", hole.bearingVolume, "radius:", hole.bearingRadius)

@n type of geo_type: string

Parameters

geo_type	"volume" or "block", specifying how the clamped members are defined. type of clamped_members: std::vector< int,std::allocator< int > >
clamped_members	IDs of volumes or blocks potentially containing pilot holes. type of radius_threshold: float
radius_threshold	Maximum hole radius to be considered a pilot hole. type of gap_threshold: float
gap_threshold	Maximum gap between concentric holes to be treated as aligned.

@n return type of : std::vector< BoltHoleInfo,std:: allocator< BoltHoleInfo > >

Returns

A list of BoltHoleInfo structs containing:

- bearing hole surface IDs and volume,
- bearing hole axis and radius,
- thread hole surface IDs, volumes, axes, and radii,
- bounding box of the full hole set and total volume.

◆ get_bolt_shigley_radius()

def cubit.get_bolt_shigley_radius (*bolt_id*, *angle*, *washer_id* = 0)

Get the equivalent Shigley frustum radius at the bolt interface.

Calculates the radius of a Shigley-style frustum representing the stress cone between upper and lower volumes surrounding a bolt. This is typically used to evaluate stress distribution. A standard angle of 30 degrees is common.

Optionally accounts for a washer in the bolt-washer clamp mechanism by specifying its entity ID (set to 0 if no washer is used).

.. code-block:: c++

```
# Without washer

radius1 = cubit.get_bolt_shigley_radius
(2, 0, 30.0)

print("Shigley radius (no
washer):" , radius1)

# With washer (ID 5)

radius2 = cubit.get_bolt_shigley_radius
(2, 5, 30.0)

print("Shigley radius (with
washer):" , radius2)
```

type of bolt_id: int

Parameters

bolt_id ID of the bolt volume entity.
type of washer_id: int, optional

washer_id Optional ID of the washer entity in the bolt-washer mechanism (use 0 if none).
type of angle: float

angle Cone half-angle in degrees (typical value is 30).
return type of : float

Returns

Radius at the bolt interface, or x1 if it cannot be computed.

◆ get_bolt_washer()

def cubit.get_bolt_washer (*bolt_id*)

Get the washer volume ID associated with a given bolt.

Returns the volume ID of the washer corresponding to the specified bolt ID. If no washer is associated with the bolt, the function returns 0.

.. code-block:: python

```
# Assume bolt 1 has been
created

washer_id = cubit.get_bolt_washer
(1)

print("Washer volume
ID:" , washer_id)
```

@n type of bolt_id: int

Parameters

bolt_id The ID of the bolt whose washer volume ID is requested.
return type of : int

Returns

The volume ID of the washer associated with the bolt, or 0 if none exists.

◆ get_bolts_in_clamped_members()

def cubit.get_bolts_in_clamped_members (*geo_type*, *clamped_vols*, *candidate_bolts*)

Identify bolts clamping the given volumes or blocks.

Performs ML-based classification on provided candidate volumes to identify which are bolts, then checks which of those bolts clamp the specified clamped members.

```
.. code-block:: python
```

```
                clamped_ids = [201, 202]

                candidate_bolts = [100, 105, 110]

                bolts =
    cubit.get_bolts_in_clamped_members
                ("volume" ,
                clamped_ids, candidate_bolts)

                print(bolts)
```

```
 @n type of geo_type:  string
```

Parameters

geo_type "volume" or "block", indicating the type of clamped entities.
type of clamped_vols: std::vector< int,std::allocator< int > >

clamped_vols List of volume or block IDs that are clamped by one or more bolts.
type of candidate_bolts: std::vector< int,std::allocator< int > >

candidate_bolts Volumes to consider as potential bolts. ML classification is run on them to identify actual bolts.
return type of : std::vector< int,std:: allocator< int > >

Returns

List of volume IDs classified as bolts that clamp the given entities.

◆ get_bool_sculpt_default()

def cubit.get_bool_sculpt_default (*variable*)

Retrieve default boolean-valued sculpt parameter.

Matches the input variable name substring to known sculpt parameters and returns its default.

Title Available boolean parameters

- OVERWRITE
- EXECUTE
- VOID
- SEPARATE_VOID_BLOCKS
- HTET

```
                    * PATH_ABSOLUTE, PATH_COMMAND

                    * CLEAN, SHOW, IMPORT, COMBINE
```

- BBOX_EXPAND
- GEN_INPUT_FILE
- PERIODIC

```
                    * PILLOW_SURFACES,
    PILLOW_CURVES,
                    PILLOW_BOUNDARIES,
    PILLOW_SMOOTH
```

- DEFEATURE_BBOX

```
                    * ADAPT_EXPORT,
    ADAPT_NON_MANIFOLD,
                    USE_DEFAULT_ADAPT_THRESHOLD

                    * QUALITY, COMPARE_VOLUME,
    COMPUTE_SS_STATS

                    * WRITE_GEOM, WRITE_MBG
```

- UNIQUE_GENESIS_IDS

type of variable: string

Parameters

variable Name of the sculpt parameter substring.
return type of : boolean

Returns

Default boolean value for the specified parameter.

◆ get_boundary_layer_algorithm()

def cubit.get_boundary_layer_algorithm (*boundary_layer_id*)

Get the algorithm type used by the specified boundary layer.

Returns the name of the algorithm (e.g., "Structured", "Unstructured").
type of boundary_layer_id: int

Parameters

boundary_layer_id ID of the boundary layer.
return type of : string

Returns

Algorithm name as a string.

◆ get_boundary_layer_continuity()

def cubit.get_boundary_layer_continuity (*boundary_layer_id*)

Get the continuity setting of a boundary layer.

Indicates whether the boundary layer is continuous across adjacent entities.
type of boundary_layer_id: int

Parameters

boundary_layer_id ID of the boundary layer.
return type of : boolean

Returns

True if the boundary layer is continuous; otherwise false.

◆ get_boundary_layer_id_list()

def cubit.get_boundary_layer_id_list ( )

Get a list of all defined boundary layer IDs.

return type of : std::vector< int,std:: allocator< int > >

Returns
List of boundary layer IDs currently defined in the model.

◆ get_boundary_layers_by_base()

def cubit.get_boundary_layers_by_base (*base_type*, *base_id*)

Get all boundary layers associated with a specified base entity.

Retrieves boundary layer IDs that use the given base entity.
type of base_type: string

Parameters
base_type Type of base entity (e.g., "surface", "volume").
type of base_id: int

base_id ID of the base entity.
return type of : std::vector< int,std:: allocator< int > >

Returns
Vector of boundary layer IDs using the specified base.

◆ get_boundary_layers_by_pair()

def cubit.get_boundary_layers_by_pair (*base_type*, *base_id*, *parent_id*)

Get all boundary layers associated with a base-parent pair.

Returns boundary layers defined using both the base and parent entities.
type of base_type: string

Parameters
base_type Type of base entity (e.g., "surface", "volume").
type of base_id: int
base_id ID of the base entity.
type of parent_id: int
parent_id ID of the parent entity.
return type of : std::vector< int,std:: allocator< int > >

Returns
Vector of boundary layer IDs that match the base-parent pair.

◆ get_bounding_box()

def cubit.get_bounding_box (*geometry_type*, *entity_id*)

Get the axis-aligned bounding box for a specified entity.

.. code-block:: python

```
bbox = cubit.get_bounding_box
("surface" , 22)

print(bbox) # Displays 10 values:
minx, maxx, rangex, miny, maxy, rangey,
minz, maxz,
rangez, diagonal
```

```
@n type of geometry_type:  string
```

Parameters

geometry_type Specifies the geometry type ("body", "volume", "surface", "curve", or "vertex").
type of entity_id: int

entity_id ID of the entity.
return type of : std:: array< double,10 >

Returns

A 10-element array containing the bounding box parameters.

Title: Array contents

- [0] min x coordinate
- [1] max x coordinate
- [2] x-range (max x ? min x)
- [3] min y coordinate
- [4] max y coordinate
- [5] y-range (max y ? min y)
- [6] min z coordinate
- [7] max z coordinate
- [8] z-range (max z ? min z)
- [9] diagonal length of the box

◆ get_build_number()

def cubit.get_build_number ( )

Retrieve the Cubit build number.

Returns the build identifier for the current Cubit binary.

```
.. code-block:: python
```

```
import cubit

build_number = cubit.get_build_number
()

print("Build
number:" , build_number)
```

```
@n return type of :  string
```

Returns

A string containing the current Cubit build number.

◆ get_cavity_surfaces()

def cubit.get_cavity_surfaces (*surface_id*)

Return the surfaces in the cavity adjacent to the specified surface.

The function returns a list of surface IDs that belong to the same cavity as the given surface. The input surface_id must already be part of a cavity. The result includes surface_id itself.

```
.. code-block:: python
```

```
                cubit.cmd
                ("brick x 10" )

                cubit.cmd
                ("brick x 5" )

                cubit.cmd
                ("vol 5 move x 5" )

                cubit.cmd
                ("subtract volume 2 from
                volume 1" )


                surf_id = 17

                cavity_surfs =
    cubit.get_cavity_surfaces
                (surf_id)


                print("Cavity
                surfaces:" , tuple(cavity_surfs))

                # Expected output:

                # Cavity surfaces: (17, 15, 16,
                13, 14)
```

```
@n type of surface_id:  int
```

Parameters

surface_id **Surface** ID that is part of a cavity.
return type of : std::vector< int,std:: allocator< int > >

Returns

A list of surface IDs belonging to the cavity (includes surface_id itself).

◆ get_center_point()

def cubit.get_center_point (*entity_type*, *entity_id*)

Get the 3D center or coordinates of a specified entity.

Returns the centroid for geometry entities (body, volume, surface), midpoint for curves, coordinates for vertices, or center for mesh entities (node, edge, face, tri, quad, hex, tet, wedge, pyramid, sphere).

```
.. code-block:: python
```

```
                cubit.cmd
                ("brick x 10" )

                center = cubit.get_center_point
                ("surface" , 1)

                print("Surface 1
                center:" , center) # (0.0,
                0.0, 5.0)

                # For a vertex:

                vcoord = cubit.get_center_point
                ("vertex" , 12)

                print("Vertex 12
                coords:" , vcoord)
```

```
@n type of entity_type:  string
```

Parameters

entity_type **Entity** type: "body", "volume", "surface", "curve", "vertex", "node", "edge", "face", "tri", "quad", "hex", "tet", "wedge", "pyramid", or "sphere".
type of entity_id: int

entity_id ID of the entity to query.
return type of : std:: array< double,3 >

Returns

A 3-element array [x, y, z] representing the entity's center or vertex coordinates.

◆ get_cfd_type()

def cubit.get_cfd_type (*entity_id*)

Get the CFD boundary condition subtype.

type of entity_id: int

Parameters

entity_id ID of the CFD boundary condition.
return type of : int

Returns

CFD type as defined in CI_BCTypes.

◆ get_chamfer_chain_collections()

def cubit.get_chamfer_chain_collections (*volume_list*, *thickness_threshold*)

Return collections of surfaces that form chamfer chains in the specified volumes.

A chamfer chain is a group of contiguous planar surfaces that together form a chamfer feature. Chamfer chains are filtered by their computed thickness (distance between chamfer edges).

If thickness_threshold is provided, only chamfer chains with thickness less than this value are returned.

.. code-block:: python

```
cubit.cmd
("brick x 10" )


cubit.cmd
("modify curve 2 12 8 11
chamfer radius 1" ) #
Surfaces 7, 8, 9, 10 are created


volume_ids = [1]


thickness_thresh = 2.0


chamfer_collections =
cubit.get_chamfer_chain_collections
(volume_ids, thickness_thresh)
```

for surfaces, thickness in chamfer_collections: print("Chamfer chain surfaces:", tuple(surfaces), "Thickness:", thickness)

```
# Expected output:


# Chamfer chain surfaces: (7, 8,
9, 10), Thickness: 1.0
```

@n type of volume_list: std::vector< int,std::allocator< int > >

Parameters

volume_list List of volume IDs to query.
type of thickness_threshold: float

thickness_threshold Return only chamfer chains whose computed thickness is less than this value.
return type of : std::vector< std::pair< std::vector< int,std::allocator< int > >,double >,std::allocator< std::pair< std::vector< int,std:: allocator< int > >,double > > >

Returns

: A list of pairs each pair contains (1) a list of surface IDs defining a chamfer chain, and (2) the computed thickness of the chamfer chain.

◆ get_chamfer_chains()

def cubit.get_chamfer_chains (*surface_id*)

Returns the chamfer chains for a given surface.

Given a surface ID, this function returns all chamfer chains associated with that surface. If the surface is part of one or more chamfer chains, each chain will be returned as a list of surface IDs.

```
.. code-block:: python
```

```
                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("modify curve 4 1 2 3
                chamfer radius 1" ) #
                Chamfer the four curves on surface 5


                surface_id = 10


                chamfer_chains =
    cubit.get_chamfer_chains
                (surface_id)
```

for chain in chamfer_chains: print("Chamfer chain:", tuple(chain))

```
                # Expected output:


                # Chamfer chain: (7, 8, 9,
                10)
```

```
 @n type of surface_id:  int
```

Parameters

surface_id **Surface** to retrieve the chamfer chains from.
return type of : std::vector< std::vector< int,std::allocator< int > >,std::allocator< std::vector< int,std:: allocator< int > > > >

Returns

A list of lists of surface IDs in each chamfer chain.

Note

If using Python, lists will be returned as Python tuples.

◆ get_chamfer_surfaces()

def cubit.get_chamfer_surfaces (*target_volume_ids*, *thickness_threshold*)

Get the list of chamfer surfaces for a list of volumes.

type of target_volume_ids: std::vector< int,std::allocator< int > >

Parameters

target_volume_ids List of volume ids to examine.
type of thickness_threshold: float

thickness_threshold max thickness criteria for chamfer
return type of : std::vector< std::vector< double,std::allocator< double > >,std::allocator< std::vector< double,std:: allocator< double > > > >

Returns

List (python tuple) of chamfer surface ids (as doubles) and their thicknesses

◆ get_close_loop_thickness()

def cubit.get_close_loop_thickness (*surface_id*)

Compute the minimum separation distance between loops on a surface.

For a surface with two or more boundary loops, returns the smallest distance between any two loops. If the surface has fewer than two loops, returns 0.

.. code-block:: python

```
                # Create two concentric cylinders
                and subtract inner from outer.


                cubit.cmd
                ("cylinder x 3 height
                10" )


                cubit.cmd
                ("cylinder x 2 height
                10" )


                cubit.cmd
                ("subtract volume 2 from
                volume 1" )


                # Surface 9 has two loops; get
                their minimum separation.


                thickness =
      cubit.get_close_loop_thickness
                (9)


                print("Surface 9
                thickness:" , thickness) #
                expect 1.0
```

@n type of surface_id: int

Parameters

surface_id ID of the surface to measure.
return type of : float

Returns

Minimum loop-to-loop distance (thickness). Returns 0 if fewer than two loops.

◆ get_close_loops()

def cubit.get_close_loops (*target_volume_ids*, *mesh_size*)

Find faces with multiple loops closer than a given threshold.

Iterates over all faces of volumes in target_volume_ids. A face is included if it has two or more boundary loops and the minimum distance between any two loops is <= mesh_size.

This differs from get_narrow_regions and get_surfs_with_narrow_regions by only checking loop-to-loop proximity on faces with multiple loops, rather than edge splits or oriented edge-pair tests.

.. code-block:: python

```
# Create two concentric cylinders
and subtract inner from outer.


cubit.cmd
("cylinder x 3 height
10" )


cubit.cmd
("cylinder x 2 height
10" )


cubit.cmd
("subtract volume 2 from
volume 1" )


# Surface 9 has two loops
separated by distance 1.


mesh_size = 1.0


close_loop_faces = cubit.get_close_loops
([1], mesh_size)


print(close_loop_faces) # expect
[9]
```

@n type of target_volume_ids: std::vector< int,std::allocator< int > >

Parameters

target_volume_ids Vector of volume IDs to examine.
type of mesh_size: float

mesh_size Distance threshold for loop separation.
return type of : std::vector< int,std:: allocator< int > >

Returns

Vector of surface IDs whose loops lie within mesh_size of each other.

◆ get_close_loops_with_thickness()

def cubit.get_close_loops_with_thickness (*target_volume_ids*, *mesh_size*, *genus*)

Find faces with multiple loops closer than a threshold and return their minimum loop separations.

Iterates over all faces of volumes in target_volume_ids. A face is included if it has (genus + 1) loops (where genus = num_loops - 1) and the minimum distance between any two loops is <= mesh_size. Returns a list of [surface_id, min_distance] pairs for each qualifying face.

This function differs from get_close_loops by filtering faces by genus and also returning the minimum loop-to-loop distance for each face.

.. code-block:: python

```
# Create two concentric cylinders
and subtract inner from outer.


cubit.cmd
("cylinder x 3 height
10" )


cubit.cmd
("cylinder x 2 height
10" )


cubit.cmd
("subtract volume 2 from
volume 1" )


# Now surface 9 has two loops
(genus = 1) separated by distance 1.


mesh_size = 1.0


genus = 1 # Only surfaces with
exactly two loops.


results = cubit.get_close_loops_with_thickness
([1], mesh_size, genus)
```

for sid, dist in results: print("Surface", sid, "has min loop distance", dist) # expect: **Surface** 9 has min loop distance 1.0

```
@n type of target_volume_ids:  std::vector< int,std::allocator< int > >
```

Parameters

target_volume_ids Vector of volume IDs to examine.
type of mesh_size: float

mesh_size Distance threshold for loop separation.
type of genus: int

genus Number of loops minus one. To return faces of any genus, use genus < 0.
return type of : std::vector< std::vector< double,std::allocator< double > >,std::allocator< std::vector< double,std:: allocator< double > > > >

Returns

Vector of [surface_id, min_distance] for each face whose loops lie within mesh_size and match the specified genus.

◆ get_close_vertex_curve_pairs()

def cubit.get_close_vertex_curve_pairs (*target_volume_ids*, *high_tolerance*)

Find pairs of vertices and curves within a specified tolerance across given volumes.

Unlike get_coincident_vertices, which identifies vertices that coincide with other vertices, this function finds vertices that lie within high_tolerance of a curve. Iterates over all vertices and curves of volumes in target_volume_ids and returns pairs whose shortest distance is <= high_tolerance.

```
.. code-block:: python

                # Create two 1x1x1 bricks and
                offset the second so its vertices are close to
    curves
                on the first.

                cubit.cmd
                ("brick x 1" )

                cubit.cmd
                ("brick x 1" )

                cubit.cmd
                ("volume 2 move x 1 y
                0.5" )

                tol = 1e-6 # Tolerance for close
                detection.

                pairs = cubit.get_close_vertex_curve_pairs
                ([1, 2], tol)

                # Expected output: (15, 5, 12, 1,
                5, 19, 2, 15)
```

for i in range(0, len(pairs), 2): print("Vertex", pairs[i], "is close to Curve", pairs[i+1])

```
  @n type of target_volume_ids:  std::vector< int,std::allocator< int > >
```

Parameters

target_volume_ids Vector of volume IDs to examine.
type of high_tolerance: float

high_tolerance Distance threshold below which a vertex and a curve are considered close.
return type of : std::vector< int,std:: allocator< int > >

Returns

Flat vector of [vertex_id, curve_id, ...] representing close vertex-curve pairs.

◆ get_closed_narrow_surfaces()

def cubit.get_closed_narrow_surfaces (*target_ids*, *narrow_size*)

Find faces on closed surfaces whose two boundary edges remain within a distance threshold.

A surface is "closed in U" (or "closed in V") if traversing its U (or V) parameter from minimum to maximum returns to the same point (no open edge). For each face of volumes in target_ids, the following criteria are applied:

- The underlying surface is closed in U or V (no open boundary in that parametric direction).
- The face has exactly two loops and exactly two edges (inner and outer boundaries).
- Sampling five points uniformly along one edge, each point's shortest distance to the other edge must be <= narrow_size. Faces meeting all criteria are returned.

This function differs from get_narrow_regions and get_surfs_with_narrow_regions by only considering closed two-loop faces and sampling along their boundary loops, rather than testing all faces or arbitrary edge pairs.

.. code-block:: python

```
cubit.cmd
("cylinder radius 3 height
10" )


cubit.cmd
("modify curve 1 blend
radius 0.5" )


cubit.cmd
("cylinder radius 2 height
10" )


cubit.cmd
("subtract volume 2 from
volume 1" )


narrow_thresh = 1.0


closed_narrow =
cubit.get_closed_narrow_surfaces
([1], narrow_thresh)


print(closed_narrow) # expect two
IDs
```

@n type of target_ids: std::vector< int,std::allocator< int > >

Parameters

target_ids Vector of volume IDs whose faces are checked.
type of narrow_size: float

narrow_size Distance threshold for edge proximity.
return type of : std::vector< int,std:: allocator< int > >

Returns

Vector of IDs of faces whose underlying surface is closed in U or V, has two loops, and whose two edges remain within narrow_size.

◆ get_closest_node()

def cubit.get_closest_node (*x_coordinate*, *y_coordinate*, *z_coordinate*)

Find the mesh node closest to a given point.

This function searches the current mesh and returns the ID of the node whose coordinates are nearest to the specified (x, y, z) point. If the mesh contains no nodes, it returns 0.

```
.. code-block:: python
```

```
                    # Create a brick and mesh its
                    volume

                    cubit.cmd
                    ("brick x 10" )

                    cubit.cmd
                    ("mesh vol 1" )


                    # Find the node closest to (5, 5,
                    5)

                    closest = cubit.get_closest_node
                    (5.0, 5.0, 5.0)

                    print("Closest node to
                    (5,5,5):" , closest)
```

```
@n type of x_coordinate:  float
```

Parameters

x_coordinate X coordinate of the query point.
type of y_coordinate: float

y_coordinate Y coordinate of the query point.
type of z_coordinate: float

z_coordinate Z coordinate of the query point.
return type of : int

Returns

ID of the closest node, or 0 if no nodes are present.

◆ get_coincident_vertices()

def cubit.get_coincident_vertices (*target_volume_ids*, *high_tolerance*)

Find vertex pairs within a specified tolerance across given volumes.

Iterates over all vertices of volumes in target_volume_ids and identifies pairs whose distance is <= high_tolerance. Returns a flat list of vertex ID pairs: [v1_a, v1_b, v2_a, v2_b, . . .].

.. code-block:: python

```
# Create two 1x1x1 bricks and
move the second so they share a face.


cubit.cmd
("brick x 1" )


cubit.cmd
("brick x 1" )


cubit.cmd
("volume 2 move x 1"
)


tol = 1e-6 # Tolerance for
coincident detection.


pairs = cubit.get_coincident_vertices
([1, 2], tol)


# Expected output: (6, 15, 5, 16,
2, 11, 1, 12)
```

for i in range(0, len(pairs), 2): print("Coincident pair:", pairs[i], "and", pairs[i+1])

```
@n type of target_volume_ids:  std::vector< int,std::allocator< int > >
```

Parameters

target_volume_ids Vector of volume IDs to examine.
type of high_tolerance: float

high_tolerance Distance threshold below which two vertices are considered coincident.
return type of : std::vector< int,std:: allocator< int > >

Returns

Flat vector of vertex ID pairs [v1, v2, v3, v4, . . .] representing coincident vertices.

◆ get_command_from_history()

def cubit.get_command_from_history (*command_number*)

Retrieve a specific command from Cubit's history buffer.

.. code-block:: python

```
import cubit


count = cubit.get_command_history_count
()
```

for i in range(count): print(cubit.get_command_from_history(i))

```
@n type of command_number:  int
```

Parameters

command_number Zero-based index into the history buffer.
return type of : string

Returns

The command string at the given index.

◆ get_command_history_count()

def cubit.get_command_history_count ( )

Get the number of commands in Cubit's history buffer.

.. code-block:: python

```
import cubit

count = cubit.get_command_history_count
()
```

```
@n return type of :  int
```

Returns
The total number of commands stored in history.

◆ get_common_curve_id()

def cubit.get_common_curve_id (*surface_1_id*, *surface_2_id*)

Return a curve shared by two surfaces.

Finds the curve that bounds both input surfaces. Assumes both surface IDs are valid and that only topologically adjacent surfaces can share a curve. Returns -1 if no shared curve exists.

```
.. code-block:: python
```

```
cubit.cmd
("brick x 10" )
# surfaces 1-6

cid = cubit.get_common_curve_id
(1, 6) # Expected: 1

print(cid)
```

```
@n type of surface_1_id:  int
```

Parameters
surface_1_id ID of the first surface.
type of surface_2_id: int
surface_2_id ID of the second surface.
return type of : int

Returns
ID of the shared curve, or -1 if none exists.

◆ get_common_vertex_id()

def cubit.get_common_vertex_id (*curve_1_id*, *curve_2_id*)

Return a vertex shared by two curves.

Finds the vertex that bounds both input curves. Assumes both curves exist and that only adjacent curves share a vertex. Returns 0 if no shared vertex exists.

```
.. code-block:: python
```

```
cubit.cmd("brick x 10")                    // curves 1-12
vid = cubit.get_common_vertex_id(1, 2)     // expected: 2

print(vid)
```

```
@n type of curve_1_id:  int
```

Parameters
curve_1_id ID of the first curve.
type of curve_2_id: int
curve_2_id ID of the second curve.
return type of : int

Returns
ID of the shared vertex, or 0 if none exists.

◆ get_cone_surfaces()

def cubit.get_cone_surfaces (*target_volume_ids*)

Identify cone surfaces in a set of volumes.

This function returns the surfaces that represent cones defined by a conic surface and a hard point in the specified volumes.

The function is callable from both C++ and Python.

.. code-block:: python

```
cubit.cmd
('brick x 10' )


cubit.cmd
('cylinder rad 1 height
5' )


cubit.cmd
('create frustum height 1
radius 1 top 0' )


cubit.cmd
('vol 2 mov z 3' )


cubit.cmd
('unite vol all' )


cone_surfaces = cubit.get_cone_surfaces
([1])


print("Cone
surfaces:" , [int(sid) for sid in cone_surfaces])


# Expected output:


# Cone surfaces: [4]
```

@n type of target_volume_ids: std::vector< int,std::allocator< int > >

Parameters

target_volume_ids List of volume IDs to analyze.
return type of : std::vector< int,std:: allocator< int > >

Returns

List of surface IDs that are cones defined by a conic surface and a hard point.

◆ get_connected_surfaces()

def cubit.get_connected_surfaces (*surf_ids*)

Find surfaces connected to a given set of surfaces.

Given a list of surface IDs, this function groups them into "patches" of connected surfaces (connection means sharing a common curve). Merged surfaces are always excluded.

Title Behavior based on the number of patches found:

- If the input surfaces form a single patch, return all surfaces connected to that patch (excluding merged surfaces).
- If the input surfaces form exactly two separate patches, return all surfaces that lie between those two patches (excluding merged surfaces), plus the original patches.
- If the input surfaces form more than two patches, return an empty list.

```
.. code-block:: python
```

```
                # Create a 10x10x10 brick;
                surfaces 1-6 are generated.


                cubit.cmd
                ("brick x 10" )



                # Surfaces 1 and 2 are opposite
                faces on volume 1, forming two separate
    patches.


                connected =
    cubit.get_connected_surfaces
                ([1, 2])


                # connected will be (1, 2, 3, 4,
                5, 6)


                print("Connected
                surfaces:" , connected)
```

Parameters

surface_ids IDs of the surfaces to start from.
return type of : std::vector< int,std:: allocator< int > >

Returns

A list (vector or Python tuple) of surface IDs that satisfy the connectivity rules above.

◆ get_connectivity()

def cubit.get_connectivity (*entity_type*, *entity_id*)

Get the list of node IDs comprising a mesh element.

This function returns the connectivity of the specified mesh element by listing the IDs of its corner nodes. The ordering of nodes follows the Exodus convention; see the Exodus documentation for the element type-specific node ordering.

.. code-block:: python

```
# Create geometry and mesh
it

cubit.cmd
("brick x 10" )

cubit.cmd
("mesh vol 1" )


# Query connectivity of hex
element 221

node_id_list = cubit.get_connectivity
("hex" , 221)

print("Hex 221 is connected
to nodes:" , node_id_list)
```

@n type of entity_type: string

Parameters

entity_type Mesh element type (e.g., "hex", "tet", "tri", "quad").
type of entity_id: int

entity_id ID of the mesh element.
return type of : std::vector< int,std:: allocator< int > >

Returns

List of node IDs defining the element's connectivity.

◆ get_constraint_dependent_entity_point()

def cubit.get_constraint_dependent_entity_point (*constraint_id*)

Get the dependent entity of a specified constraint.

type of constraint_id: int

Parameters

constraint_id ID of the constraint.
return type of : string

Returns

Dependent entity as a string.

◆ get_constraint_reference_point()

def cubit.get_constraint_reference_point (*constraint_id*)

Get the reference point of a specified constraint.

type of constraint_id: int

Parameters

constraint_id ID of the constraint.
return type of : string

Returns

Reference point as a string.

◆ get_constraint_type()

def cubit.get_constraint_type (*constraint_id*)

Get the type of a specified constraint.

type of constraint_id: int

Parameters
constraint_id ID of the constraint.
return type of : string

Returns
: Constraint type "Kinematic", "Distributing", or "Rigidbody".

◆ get_contact_pair_exterior_state()

def cubit.get_contact_pair_exterior_state (*entity_id*)

Get the exterior state of a contact pair.

type of entity_id: int

Parameters
entity_id ID of the contact pair.
return type of : boolean

Returns
True if contact is considered exterior.

◆ get_contact_pair_friction_value()

def cubit.get_contact_pair_friction_value (*entity_id*)

Get the friction value for a contact pair.

type of entity_id: int

Parameters
entity_id ID of the contact pair.
return type of : float

Returns
Friction value.

◆ get_contact_pair_general_state()

def cubit.get_contact_pair_general_state (*entity_id*)

Get the general contact state.

type of entity_id: int

Parameters
entity_id ID of the contact pair.
return type of : boolean

Returns
True if general contact is active.

◆ get_contact_pair_tied_state()

def cubit.get_contact_pair_tied_state (*entity_id*)

Get the tied state of a contact pair.

type of entity_id: int

Parameters
entity_id ID of the contact pair.
return type of : boolean

Returns
True if tied state is active.

◆ get_contact_pair_tol_lower_value()

def cubit.get_contact_pair_tol_lower_value (*entity_id*)

Get the lower bound tolerance for a contact pair.

type of entity_id: int

Parameters

entity_id ID of the contact pair.
return type of : float

Returns

Lower tolerance value.

◆ get_contact_pair_tolerance_value()

def cubit.get_contact_pair_tolerance_value (*entity_id*)

Get the upper bound tolerance for a contact pair.

type of entity_id: int

Parameters

entity_id ID of the contact pair.
return type of : float

Returns

Upper tolerance value.

◆ get_continuous_curve_collections()

def cubit.get_continuous_curve_collections (*volume_list*, *angle_tolerance* = -1)

Returns collections of continuous curves in the given volumes.

Continuous curves are defined as curves connected at 2-valent vertices, where the tangents at the common vertex form an angle of 180 degrees +/- the specified angle tolerance.

.. code-block:: python

```
                volume_ids = [1, 2]

                angle_tol_deg = 5.0

                collections =
    cubit.get_continuous_curve_collections
                (volume_ids, angle_tol_deg)
```

for curve_ids, total_length in collections: print("Curve IDs:", curve_ids, "Length:", total_length)

@n type of volume_list: std::vector< int,std::allocator< int > >

Parameters

volume_list List of volume IDs to query.
type of angle_tolerance: float, optional

angle_tolerance Tangent angle tolerance in degrees. Continuous curves must have tangents at 180 +/- this tolerance. If angle_tolerance < 0 or not specified, defaults to 10 degrees.
return type of : std::vector< std::pair< std::vector< int,std::allocator< int > >,double >,std::allocator< std::pair< std::vector< int,std:: allocator< int > >,double > > >

Returns

: Ordered list of pairs (1) vector of contiguous curve IDs forming a continuous curve, (2) total length of the curves. The list is ordered from smallest to largest total length.

◆ get_continuous_curves()

def cubit.get_continuous_curves (*curve_id*,
angle_tol,
require_two_valent = `False`
)

Return the list of adjacent continuous curves.

Two curves are considered continuous if the angle between them at a shared vertex is 180 degrees +/- angle_tol.

The returned list includes the input curve_id and any connected continuous curves.

If require_two_valent is true, continuity does not extend across vertices connected to more than two curves.

`.. code-block:: python`

```
                cubit.cmd
                ("cylinder rad 1 height
                5" )


                cubit.cmd
                ("vol 1 split
                periodic" ) # Splits the
                cylinder surface into surfaces 4 and 5
      with continuous
                curves


                continuous_curves =
      cubit.get_continuous_curves
                (4, 0.1)


                print("Continuous
                curves:" , tuple(continuous_curves))


                # Expected output:


                # (4, 8)
```

`@n type of curve_id:  int`

Parameters

curve_id	**Curve** ID to query. type of angle_tol: float
angle_tol	Angle tolerance for continuity (degrees). type of require_two_valent: boolean, optional
require_two_valent	If true, continuity does not extend across vertices with more than two adjacent curves. return type of : std::vector< int,std::allocator< int > >

Returns

A list of curve IDs in the continuous set (including curve_id).

◆ get_continuous_surfaces()

def cubit.get_continuous_surfaces (*surface_id, angle_tol*)

Return the list of adjacent continuous surfaces.

Two surfaces are considered continuous if the exterior angle between them at a shared curve is 180 degrees +/- angle_tol.

The returned list includes the input surface_id and any connected continuous surfaces.

.. code-block:: python

```
cubit.cmd
("cylinder rad 1 height
5" )


cubit.cmd
("vol 1 split
periodic" ) # Splits
cylinder surface into surfaces 4 and 5


continuous_surfs =
cubit.get_continuous_surfaces
(4, 0.1)


print("Continuous
surfaces:" , tuple(continuous_surfs))


# Expected output:

# Continuous surfaces: (4,
5)
```

@n type of surface_id: int

Parameters

surface_id **Surface** ID to query.
type of angle_tol: float

angle_tol Angle tolerance for continuity (degrees).
return type of : std::vector< int,std:: allocator< int > >

Returns

A list of surface IDs in the continuous set (including surface_id).

◆ get_convection_coefficient()

def cubit.get_convection_coefficient (*entity_id, bc_type_enum*)

Get convection coefficient from a convection BC.

type of entity_id: int

Parameters

entity_id ID of the convection BC.
type of bc_type_enum: int

bc_type_enum CI_BCEntityTypes: 1 (normal), 5 (shell top), 6 (shell bottom).
return type of : float

Returns

Convection coefficient value.

◆ get_coordinate_systems_id_list()

def cubit.get_coordinate_systems_id_list ( )

Returns a list of coordinate system IDs.

Retrieves the IDs of all coordinate systems currently defined in the model.

In Python, the returned list is a tuple of IDs.

.. code-block:: python

```
coord_ids =
cubit.get_coordinate_systems_id_list
()


print("Coordinate system
IDs:" , coord_ids)
```

@n return type of : std::vector< int,std:: allocator< int > >

Returns

List of coordinate system IDs.

get_copy_block_on_geometry_copy_setting()

def cubit.get_copy_block_on_geometry_copy_setting ( )

Get the current block behavior setting for geometry copy.

When a geometric entity is copied, this setting controls how blocks are propagated:

- "OFF": Copied geometry/mesh is not included in any block.
- "use_original": Copied geometry/mesh is added to the original block.
- "on": New blocks are created for the copied geometry/mesh, mirroring the originals.

```
.. code-block:: python
```

```
# Create a brick and assign a
block


cubit.cmd
("brick x 10" )


cubit.cmd
("block 10 volume 1"
)


# Set block copy OFF and verify
setting
```

◆

get_copy_nodeset_on_geometry_copy_setting()

```
cubit.set_copy_block_on_geometry_copy_setting
("OFF" )

setting =
cubit.get_copy_block_on_geometry_copy_setting
()

print("After setting
OFF:" , setting)


# Copy volume; block should NOT
be copied

cubit.cmd
("volume 1 copy" )


cubit.cmd
("move volume 2 x
10" )


# List blocks to verify
behavior


cubit.cmd
("list block" )


cubit.cmd
("list block 10" )


# Set block copy ON and verify
setting


cubit.set_copy_block_on_geometry_copy_setting
("on" )

setting =
cubit.get_copy_block_on_geometry_copy_setting
()

print("After setting
ON:" , setting)


# Copy volume; block should now
be copied as new block 11


cubit.cmd
("volume 1 copy" )


cubit.cmd
("move volume 3 x
20" )


# List blocks to verify
behavior


cubit.cmd
("list block" )


cubit.cmd
("list block 11" )


# Final check of the
setting


setting =
cubit.get_copy_block_on_geometry_copy_setting
()

print("Final
setting:" , setting)
```

def cubit.get_copy_nodeset_on_geometry_copy_setting ( )

Get the current nodeset behavior setting for geometry copy.

When a geometric entity is copied, this setting controls how nodesets are propagated:

- "OFF": Copied geometry/mesh is not included in any nodeset.
- "use_original": Copied geometry/mesh is added to the original nodeset.
- "on": New nodesets are created for the copied geometry/mesh, mirroring the originals.

```
.. code-block:: python
```

```
@n return type of :  string
```

Returns

Current copy block on geometry copy setting ("on", "OFF", or "use_original").

```
# Create a brick and assign a
nodeset

cubit.cmd
("brick x 10" )

cubit.cmd
("nodeset 10 vertex 1
2" )


# Set nodeset copy OFF and verify
setting


cubit.set_copy_nodeset_on_geometry_copy_setting
("OFF" )

setting =
cubit.get_copy_nodeset_on_geometry_copy_setting
()

print("After setting
OFF:" , setting)

# Copy volume; nodeset should NOT
be copied

cubit.cmd
("volume 1 copy" )

cubit.cmd
("move volume 2 x
10" )

# List nodesets to verify
behavior

cubit.cmd
("list nodeset" )

cubit.cmd
("list nodeset 10" )


# Set nodeset copy ON and verify
setting


cubit.set_copy_nodeset_on_geometry_copy_setting
("on" )

setting =
cubit.get_copy_nodeset_on_geometry_copy_setting
()

print("After setting
ON:" , setting)


# Copy volume; nodeset should now
be copied as new nodeset 11

cubit.cmd
("volume 1 copy" )

cubit.cmd
("move volume 3 x
20" )


# List nodesets to verify
behavior

cubit.cmd
("list nodeset" )

cubit.cmd
("list nodeset 11" )


# Final check of the
setting

setting =
cubit.get_copy_nodeset_on_geometry_copy_setting
()

print("Final
setting:" , setting)
```

◆

get_copy_sideset_on_geometry_copy_setting()

def cubit.get_copy_sideset_on_geometry_copy_setting ( )

Get the current sideset behavior setting for geometry copy.

When a geometric entity is copied, this setting controls how sidesets are propagated:

- "OFF": Copied geometry/mesh is not included in any sideset.
- "use_original": Copied geometry/mesh is added to the original sideset.
- "on": New sidesets are created for the copied geometry/mesh, mirroring the originals.

```
.. code-block:: python
```

```
@n return type of :  string
```

Returns

Current copy nodeset on geometry copy setting ("on", "OFF", or "use_original").

```
# Create a brick and assign a
sideset


cubit.cmd
("brick x 10" )


cubit.cmd
("sideset 10 surface
1" )


# Set sideset copy OFF and verify
setting


cubit.set_copy_sideset_on_geometry_copy_setting
("OFF" )


setting =
cubit.get_copy_sideset_on_geometry_copy_setting
()


print("After setting
OFF:" , setting)


# Copy volume; sideset should NOT
be copied


cubit.cmd
("volume 1 copy" )


cubit.cmd
("move volume 2 x
20" )


# List sidesets to verify
behavior


cubit.cmd
("list sideset" )


cubit.cmd
("list sideset 10" )
```

◆ get_cubit_digits_setting()

def cubit.get_cubit_digits_setting ( )

Retrieve the current Cubit "digits" setting.

Returns the number of digits after the decimal point that Cubit uses when printing numeric output. This corresponds to the value set by the Cubit command `set digits N`.

.. code-block:: python

```
import cubit

# Set sideset copy ON and verify
setting
# Set digits to 10 via Python
command interface


cubit.set_copy_sideset_on_geometry_copy_setting
("ON" )


cubit.cmd
("set digits 10" )


digits = cubit.get_cubit_digits_setting
()


setting =
cubit.get_copy_sideset_on_geometry_copy_setting
()


print("Digits
setting:", digits) #
print("After setting
prints 10
ON:", setting)
```

@n return type of : float

Returns
: The number of decimal digits Cubit will print, or -1 if no setting exists.

```
# Copy volume; sideset should now
be copied as new sideset 11


cubit.cmd
("volume 1 copy" )


cubit.cmd
("move volume 3 x
20" )


# List sidesets to verify
behavior


cubit.cmd
("list sideset" )


cubit.cmd
("list sideset 11" )


# Final check of the
setting


setting =
cubit.get_copy_sideset_on_geometry_copy_setting
()


print("Final
setting:", setting)
```

◆ get_cubit_message_handler()

def cubit.get_cubit_message_handler ( )

Retrieve the current Cubit message handler.

Returns the handler instance currently receiving Cubit messages.

.. code-block:: python

```
import cubit


handler =
cubit.get_cubit_message_handler
()


handler.print_message("Test
info message" )


handler.print_error("Test
error message" )
```

@n return type of : :py:class: `CubitMessageHandler`

Returns
: Current copy sideset on geometry copy setting ("on", "OFF", or "use_original").

Returns
: Shared pointer to the active CubitMessageHandler.

◆ get_current_ids()

def cubit.get_current_ids (*entity_type*)

Retrieve IDs of all current geometry entities of a specified type.

Returns the IDs of every geometry entity matching the given type. Supported types are "body", "volume", "surface", "curve", and "vertex".

Notes To include mesh entities (e.g. "node", "element"), use **get_entities()** .

```
.. code-block:: python
```

```
cubit.cmd
("brick x 10" )
# body 1


cubit.cmd
("brick x 5" )
# body 2


body_ids = cubit.get_current_ids
("body" )
# returns (1, 2)
```

```
@n type of entity_type:  string
```

Parameters

entity_type Geometry entity type: "body", "volume", "surface", "curve", or "vertex".
return type of : std::vector< int,std:: allocator< int > >

Returns

A vector<int> (or Python tuple) of IDs of all current geometry entities of the specified type.

◆ get_current_journal_file()

def cubit.get_current_journal_file ( )

Get the filename of the current journal file.

Returns the path to the active journal file. With journaling on by default, this shows which file Cubit is writing to upon startup.

```
.. code-block:: python
```

```
import cubit


print("Journal
file:" , cubit.get_current_journal_file
())
```

```
@n return type of :  string
```

Returns

The full path to the active journal file, or an empty string if journaling is off.

◆ get_curve_bias_coarse_size()

def cubit.get_curve_bias_coarse_size (*curve_id*)

Get the "coarse size" parameter for a biased curve, if set.

Returns the user-specified "coarse" interval size for the curve's bias scheme. If no coarse size is set, returns 0.

```
.. code-block:: python
```

```
                import cubit


                # Create geometry and set
                fine/coarse sizes.


                cubit.cmd
                ("create vertex 0 0
                0" )


                cubit.cmd
                ("create vertex 1 2
                0" )


                cubit.cmd
                ("create curve vertex 1
                2" ) # Curve ID = 1


                cubit.cmd
                ("curve 1 scheme bias fine
                size 0.1" )


                cubit.cmd
                ("curve 1 scheme bias
                coarse size 0.5" )



                # Query the coarse size.


                coarse_size =
      cubit.get_curve_bias_coarse_size
                (1)


                print("Curve 1 coarse
                size:" , coarse_size) #
                Should print 0.5
```

```
 @n type of curve_id:  int
```

Parameters

curve_id ID of the curve whose coarse size is retrieved.
return type of : float

Returns

Coarse interval size (double > 0), or 0 if not defined.

◆ get_curve_bias_fine_size()

def cubit.get_curve_bias_fine_size (*curve_id*)

Get the "fine size" parameter for a biased curve, if set.

Returns the user-specified "fine" interval size for the curve's bias scheme. If no fine size is set, returns 0.

```
.. code-block:: python
```

```
                import cubit


                # Create geometry and set fine
                size.


                cubit.cmd
                ("create vertex 0 0
                0" )


                cubit.cmd
                ("create vertex 0 3
                0" )


                cubit.cmd
                ("create curve vertex 1
                2" ) # Curve ID = 1


                cubit.cmd
                ("curve 1 scheme bias fine
                size 0.1" )



                # Query the fine size.


                fine_size =
        cubit.get_curve_bias_fine_size
                (1)


                print("Curve 1 fine
                size:" , fine_size) #
                Should print 0.1
```

```
 @n type of curve_id:  int
```

Parameters

curve_id ID of the curve whose fine size is retrieved.
return type of : float

Returns

Fine interval size (double > 0), or 0 if not defined.

◆ get_curve_bias_first_interval_fraction()

def cubit.get_curve_bias_first_interval_fraction (*curve_id*)

Retrieve the fraction of curve length used as the first interval size.

Returns the first interval length expressed as a fraction of total curve length. (e.g., if fraction = 0.25, the first interval = 0.25 * curve_length).

```
.. code-block:: python
```

```
                import cubit

                # Create geometry and set bias
                fraction.

                cubit.cmd
                ("create vertex 0 0
                0" )

                cubit.cmd
                ("create vertex 0 2
                0" )

                cubit.cmd
                ("create curve vertex 1
                2" ) # Curve ID = 1

                cubit.cmd
                ("curve 1 scheme bias
                fraction 0.25" )

                # Query the first interval
                fraction.

                frac =
    cubit.get_curve_bias_first_interval_fraction
                (1)

                print("Curve 1 first
                interval fraction:" , frac) # Should
    print 0.25
```

```
@n type of curve_id:  int
```

Parameters

curve_id ID of the curve whose first interval fraction is retrieved.
return type of : float

Returns

Fraction (0 < fraction <= 1) of total curve length, or 0 if not set.

◆ get_curve_bias_first_interval_length()

def cubit.get_curve_bias_first_interval_length (*curve_id*)

Query the length of the first interval on a biased curve.

Returns the absolute length of the first (smallest) mesh interval on the curve. This corresponds to the "first_delta" if that form was used, or the automatically computed value.

type of curve_id: int

Parameters

curve_id ID of the curve whose first interval length is retrieved.
return type of : float

Returns

Length of the first mesh interval (double > 0), or 0 if not set.

```
.. code-block:: python
```

```
                import cubit

                # Create geometry and set
                first_delta.

                cubit.cmd
                ("create vertex 0 0
                0" )

                cubit.cmd
                ("create vertex 1 1
                0" )

                cubit.cmd
                ("create curve vertex 1
                2" ) # Curve ID = 1

                cubit.cmd
                ("curve 1 scheme bias
                first_delta 0.2" )

                # Query the first interval
                length.

                first_len =
    cubit.get_curve_bias_first_interval_length
                (1)

                print("Curve 1 first
                interval length:" , first_len) # Should
    print 0.2
```

◆ get_curve_bias_first_last_ratio1()

def cubit.get_curve_bias_first_last_ratio1 (*curve_id*)

Retrieve the ratio of first-to-last interval at the start of a biased curve.

Returns the ratio of the length of the first interval to the last interval at the curve's start vertex. Useful for understanding how aggressively the mesh is biased at the beginning.

```
.. code-block:: python
```

```
                    import cubit


                    # Create geometry and set bias
                    factor.


                    cubit.cmd
                    ("create vertex 0 0
                    0" )


                    cubit.cmd
                    ("create vertex 0 5
                    0" )


                    cubit.cmd
                    ("create curve vertex 1
                    2" ) # Curve ID = 1


                    cubit.cmd
                    ("curve 1 scheme bias
                    factor 2.0" )


                    # Query the start first/last
                    ratio.


                    ratio1 =
          cubit.get_curve_bias_first_last_ratio1
                    (1)


                    print("Curve 1 start
                    first/last ratio:" , ratio1)
```

```
@n type of curve_id:  int
```

Parameters

curve_id ID of the curve whose start ratio is retrieved.
return type of : float

Returns

Ratio (double > 0) of first interval length to last interval length at start.

◆ get_curve_bias_first_last_ratio2()

def cubit.get_curve_bias_first_last_ratio2 (*curve_id*)

Retrieve the ratio of first-to-last interval at the end of a biased curve.

Returns the ratio of the length of the first interval to the last interval at the curve's end vertex. Useful for understanding how aggressively the mesh is biased near the end.

```
.. code-block:: python
```

```
                    import cubit


                    # Create geometry and set bias
                    factor.


                    cubit.cmd
                    ("create vertex 0 0
                    0" )


                    cubit.cmd
                    ("create vertex 0 5
                    0" )


                    cubit.cmd
                    ("create curve vertex 1
                    2" ) # Curve ID = 1


                    cubit.cmd
                    ("curve 1 scheme bias
                    factor 2.0" )


                    # Query the end first/last
                    ratio.


                    ratio2 =
          cubit.get_curve_bias_first_last_ratio2
                    (1)


                    print("Curve 1 end
                    first/last ratio:" , ratio2)
```

```
 @n type of curve_id:  int
```

Parameters

curve_id ID of the curve whose end ratio is retrieved.
return type of : float

Returns

Ratio (double > 0) of first interval length to last interval length at end.

◆ get_curve_bias_from_start()

def cubit.get_curve_bias_from_start (*curve_id*, *value*)

Determine if the bias is measured from the curve's start vertex.

By default, any "bias" scheme is applied from the start vertex. If you call "curve <id> reverse bias", this flips the progression so it is measured from the end. This function sets the output boolean to true if the bias progression begins at the start vertex, or false if it was reversed.

```
.. code-block:: python
```

```
                import cubit

                # 1) Create curve and set bias
                (starts from start by default).

                cubit.cmd
                ("create vertex 0 0
                0" )

                cubit.cmd
                ("create vertex 0 2
                0" )

                cubit.cmd
                ("create curve vertex 1
                2" ) # Curve ID = 1

                cubit.cmd
                ("curve 1 scheme bias
                factor 1.2" )

                # 2) Query from_start (should be
                True).

                from_start =
      cubit.get_curve_bias_from_start
                (1)

                print("Bias from
                start?" , from_start) #
                True

                # 3) Reverse bias, then query
                again (should be False).

                cubit.cmd
                ("curve 1 reverse
                bias" )

                from_start =
      cubit.get_curve_bias_from_start
                (1)

                print("Bias from
                start?" , from_start) #
                False
```

```
@n type of curve_id:  int
```

Parameters

curve_id ID of the curve to query.
type of value: boolean, out

value Set to true if bias is from the start vertex; false if reversed or not set.
return type of : boolean

Returns

True if the curve exists (and output is written); false if curve_id is invalid.

◆ get_curve_bias_from_start_set()

def cubit.get_curve_bias_from_start_set (*curve_id*)

Check if the "bias from start" flag has been explicitly set on a curve.

Returns true if the bias-from-start setting (or "reverse bias") has been defined for this curve. If the curve is unmeshed but you have never called "reverse bias", this may still report true (default).

```
.. code-block:: python
```

```
                import cubit

                # 1) Create curve and set bias
                (default from-start).

                cubit.cmd
                ("create vertex 0 0
                0" )

                cubit.cmd
                ("create vertex 1 1
                0" )

                cubit.cmd
                ("create curve vertex 1
                2" ) # Curve ID = 1

                cubit.cmd
                ("curve 1 scheme bias
                factor 1.3" )

                # 2) Query if from-start is
                explicitly set (True by default).

                is_set =
      cubit.get_curve_bias_from_start_set
                (1)

                print("Bias-from-start
                explicitly set?" , is_set) # True

                # 3) Reverse bias, then query
                again (still True).

                cubit.cmd
                ("curve 1 reverse
                bias" )

                is_set =
      cubit.get_curve_bias_from_start_set
                (1)

                print("Bias-from-start
                explicitly set?" , is_set) # True
```

```
 @n type of curve_id:  int
```

Parameters

curve_id ID of the curve to query.
return type of : boolean

Returns

True if bias-from-start (or reverse) has been defined; false if not or curve invalid.

◆ get_curve_bias_geometric_factor()

def cubit.get_curve_bias_geometric_factor (*curve_id*)

Get the primary geometric progression factor used for biasing a curve.

Retrieves the ratio between successive edge lengths at the start of the curve. This "factor" defines the geometric progression for a biased mesh from the first vertex.

type of curve_id: int

Parameters

curve_id ID of the curve whose first bias factor is retrieved.
return type of : float

Returns

The first geometric bias factor (ratio > 0).

```
.. code-block:: python
```

```
                import cubit

                # Create geometry and set bias
                factor.

                cubit.cmd
                ("create vertex 0 0
                0" )

                cubit.cmd
                ("create vertex 1 0
                0" )

                cubit.cmd
                ("create curve vertex 1
                2" ) # Curve ID = 1

                cubit.cmd
                ("curve 1 scheme bias
                factor 2.0" )

                # Query the first bias
                factor.

                factor1 =
      cubit.get_curve_bias_geometric_factor
                (1)

                print("Curve 1 first bias
                factor:" , factor1) #
                Should print 2.0
```

◆ get_curve_bias_geometric_factor2()

def cubit.get_curve_bias_geometric_factor2 (*curve_id*)

Get the secondary geometric progression factor for a dual-bias curve.

Retrieves the same "factor" used from the end vertices toward the middle when using dualbias. For a dualbias, there is only one factor value (applied symmetrically from both ends).

type of curve_id: int

Parameters

curve_id ID of the curve whose dualbias factor is retrieved.
return type of : float

Returns

The geometric factor (ratio > 0) for a dualbias curve.

```
.. code-block:: python
```

```
                import cubit

                # Create geometry and set
                dualbias factor.

                cubit.cmd
                ("create vertex 0 0
                0" )

                cubit.cmd
                ("create vertex 0 1
                0" )

                cubit.cmd
                ("create curve vertex 1
                2" ) # Curve ID = 1

                cubit.cmd
                ("curve 1 scheme dualbias
                factor 1.5" )

                # Query the dualbias
                factor.

                factor2 =
      cubit.get_curve_bias_geometric_factor2
                (1)

                print("Curve 1 dualbias
                factor:" , factor2) #
                Should print 1.5
```

◆ get_curve_bias_last_first_ratio1()

def cubit.get_curve_bias_last_first_ratio1 (*curve_id*)

Retrieve the ratio of last-to-first interval at the start of a biased curve.

Returns the inverse of the first/last ratio at the curve's start vertex (i.e., last interval length / first interval length).

```
.. code-block:: python
```

```
                    import cubit


                    # Create geometry and set bias
                    factor.


                    cubit.cmd
                    ("create vertex 0 0
                    0" )


                    cubit.cmd
                    ("create vertex 0 5
                    0" )


                    cubit.cmd
                    ("create curve vertex 1
                    2" ) # Curve ID = 1


                    cubit.cmd
                    ("curve 1 scheme bias
                    factor 2.0" )


                    # Query the start last/first
                    ratio.


                    inv_ratio1 =
          cubit.get_curve_bias_last_first_ratio1
                    (1)


                    print("Curve 1 start
                    last/first ratio:" , inv_ratio1)
```

```
@n type of curve_id:  int
```

Parameters

curve_id ID of the curve whose start inverse ratio is retrieved.
return type of : float

Returns

Ratio (double > 0) of last interval length to first interval length at start.

◆ get_curve_bias_last_first_ratio2()

def cubit.get_curve_bias_last_first_ratio2 (*curve_id*)

Retrieve the ratio of last-to-first interval at the end of a biased curve.

Returns the inverse of the first/last ratio at the curve's end vertex (i.e., last interval length / first interval length)

```
.. code-block:: python
```

```
                    import cubit


                    # Create geometry and set bias
                    factor.


                    cubit.cmd
                    ("create vertex 0 0
                    0" )


                    cubit.cmd
                    ("create vertex 0 5
                    0" )


                    cubit.cmd
                    ("create curve vertex 1
                    2" ) # Curve ID = 1


                    cubit.cmd
                    ("curve 1 scheme bias
                    factor 2.0" )


                    # Query the end last/first
                    ratio.


                    inv_ratio2 =
          cubit.get_curve_bias_last_first_ratio2
                    (1)


                    print("Curve 1 end
                    last/first ratio:" , inv_ratio2)
```

```
@n type of curve_id:  int
```

Parameters

curve_id ID of the curve whose end inverse ratio is retrieved.
return type of : float

Returns

Ratio (double > 0) of last interval length to first interval length at end.

◆ get_curve_bias_start_vertex_id()

def cubit.get_curve_bias_start_vertex_id (*curve_id*)

Retrieve the vertex ID designated as the start of bias on a curve.

Returns the ID of the vertex used as the bias "start" point when using curve-based vertex sizes. If no custom start vertex is defined, returns -1.

```
.. code-block:: python
```

```
                import cubit


                # 1) Create curve, set vertex
                sizes, then bias from vertex 1.


                cubit.cmd
                ("create vertex 0 0
                0" )


                cubit.cmd
                ("create vertex 0 4
                0" )


                cubit.cmd
                ("create curve vertex 1
                2" ) # Curve ID = 1


                cubit.cmd
                ("vertex 1 size 0.5"
                )


                cubit.cmd
                ("vertex 2 size 1.0"
                )


                cubit.cmd
                ("curve 1 scheme bias
                factor 1.8 start vertex 1" )


                # 2) Query the start vertex for
                bias.


                start_vertex =
    cubit.get_curve_bias_start_vertex_id
                (1)
```

if start_vertex >= 0: print("Start vertex for bias:", start_vertex) # Should print 1

```
  @n type of curve_id:  int
```

Parameters

curve_id ID of the curve whose start-vertex is queried.
return type of : int

Returns

Vertex ID for the bias start, or -1 if not defined or invalid.

◆ get_curve_bias_type()

def cubit.get_curve_bias_type (*curve_id*)

Retrieve the bias scheme type applied to a curve.

Returns a string describing the bias scheme for the specified curve. Possible values include "Bias", "Dualbias", "Multi_bias", or "None".

type of curve_id: int

Parameters

curve_id ID of the curve whose bias type is queried.
return type of : string

Returns

String indicating the curve's bias type.

```
.. code-block:: python
```

```
import cubit


# 1) Create two vertices and a
curve (Curve ID = 1).


cubit.cmd
("create vertex 0 0
0" ) # Vertex ID =
1


cubit.cmd
("create vertex 0 1
0" ) # Vertex ID =
2


cubit.cmd
("create curve vertex 1
2" ) # Curve ID = 1



# 2) Apply a bias scheme (factor
= 1.5).


cubit.cmd
("curve 1 scheme bias
factor 1.5" )



# 3) Query the bias type.


bias_type = cubit.get_curve_bias_type
(1)


print("Curve 1 bias
type:" , bias_type) #
Should print "Bias"
```

◆ get_curve_center()

def cubit.get_curve_center (*curve_id*)

Retrieve the center point of a specified curve.

Title Center computation

- For circular and elliptical arcs, returns the exact geometric center.
- For straight-line curves, returns the midpoint between endpoints.
- For all other curve types or if the center cannot be determined, returns (0, 0, 0).

.. code-block:: python

```
cubit.cmd
("create vertex 0 0
0" ); cubit.cmd
("create vertex 1 0
0" )


cubit.cmd
("create curve vertex 1
2" )


c1 = cubit.get_curve_center
(1); print(c1) # Output: (0.5,
0.0, 0.0)



cubit.cmd
("cylinder rad 3 height
1" )


c2 = cubit.get_curve_center
(2); print(c2) # Output: (0.0,
0.0, 0.0)
```

@n type of curve_id: int

Parameters

curve_id ID of the curve to query.
return type of : std:: array< double,3 >

Returns

Three-element array {x, y, z} representing the curve's center or midpoint; returns {0, 0, 0} if undefined.

◆ get_curve_count()

def cubit.get_curve_count ( )

Get the current number of curves in the model.

.. code-block:: python

```
# Create a 10x10x10 brick; this
generates twelve curves (edges).


cubit.cmd
("brick x 10" )


curve_count = cubit.get_curve_count
() # returns 12


print("Curve count:"
, curve_count)
```

@n return type of : int

Returns

The number of curves currently defined in the model.

◆ get_curve_count_in_volumes()

def cubit.get_curve_count_in_volumes (*target_volume_ids*)

Get the current number of curves in the specified volumes.

.. code-block:: python

```
# Create two bricks: volume 1
(10x10x10) and volume 2 (5x5x5).


cubit.cmd
("brick x 10" )
# volume 1, 12 curves


cubit.cmd
("brick x 5" )
# volume 2, 12 curves



# Count curves in volumes 1 and
2.


volumes = [1, 2]


curve_count = cubit.get_curve_count_in_volumes
(volumes) # returns 24


print("Curve count in
volumes 1 and 2:" , curve_count)
```

@n type of target_volume_ids: std::vector< int,std::allocator< int > >

Parameters

target_volume_ids List of volume IDs to query.
return type of : int

Returns

The total number of curves contained in the specified volumes.

◆ get_curve_edges()

def cubit.get_curve_edges (*curve_id*)

Get the list of edge element IDs on a curve.

This function returns the IDs of all edge elements that are on the specified curve.

.. code-block:: python

```
# Create a brick and mesh volume
1


cubit.cmd
("brick x 10" )


cubit.cmd
("mesh vol 1" )



# Query the edges on curve
1


edge_ids = cubit.get_curve_edges
(1)


print(f"Curve 1 contains
{len(edge_ids)} edge(s)." )
```

for edge_id in edge_ids: print(f" Edge ID: {edge_id}")

@n type of curve_id: int

Parameters

curve_id ID of the curve to examine.
return type of : std::vector< int,std:: allocator< int > >

Returns

List of edge element IDs on the curve.

◆ get_curve_length()

def cubit.get_curve_length (*curve_id*)

Get the 3D length of a specified curve.

This returns the physical length of the curve in model units. For closed curves (e.g., circles), it equals the circumference; for lines, the straight-line distance.

```
.. code-block:: python
```

```
                    # Create a cylinder of radius 1
                    (curves 1 and 2 are circular edges).


                    cubit.cmd
                    ("create cylinder radius 1
                    height 2" )


                    # Curve 1 is a circle of radius
                    1: length = 2*PI.


                    length = cubit.get_curve_length
                    (1)


                    print("Curve 1
                    length:" , length) #
                    prints approximately 6.28319
```

```
 @n type of curve_id:  int
```

Parameters

curve_id ID of the curve to query.
return type of : float

Returns

The total 3D length of the curve in model units.

◆ get_curve_mesh_scheme_curvature()

def cubit.get_curve_mesh_scheme_curvature (*curve_id*)

Retrieve the curvature mesh scheme adaptation value for a curve.

Returns the double parameter that controls how node spacing adapts to local curvature. A value of zero produces nearly equal intervals; positive values concentrate nodes in high-curvature regions.

```
.. code-block:: python
```

```
                import cubit


                # 1) Create three vertices and
                build a spline curve (Curve ID = 1).


                cubit.cmd
                ("create vertex 0 0
                0" )


                cubit.cmd
                ("create vertex 1 0
                0" )


                cubit.cmd
                ("create vertex 1 1
                0" )


                cubit.cmd
                ("create curve vertex 1 2 3
                spline delete" ) # Curve
                ID = 1



                # 2) Set the curvature mesh
                scheme value to 0.5 on curve 1 and mesh
    it.


                cubit.cmd
                ("curve 1 scheme curvature
                0.5" )


                cubit.cmd
                ("mesh curve 1" )



                # 3) Query the curvature scheme
                value.


                value =
    cubit.get_curve_mesh_scheme_curvature
                (1)


                print("Curve 1 curvature
                scheme value:" , value) #
                Should print 0.5
```

```
@n type of curve_id:  int
```

Parameters

curve_id ID of the curve whose curvature scheme value is retrieved.
return type of : float

Returns

Curvature adaptation value (double). Zero = uniform intervals.

◆ get_curve_mesh_scheme_pinpoint_locations()

def cubit.get_curve_mesh_scheme_pinpoint_locations (*curve_id*)

Retrieve the pinpoint mesh scheme locations for a curve.

Returns a list of absolute positions along the curve (measured from the start vertex) where nodes have been placed by the pinpoint scheme.

```
.. code-block:: python
```

```
                import cubit


                # 1) Create two vertices and a
                linear curve between them (Curve ID =
    1).


                cubit.cmd
                ("create vertex 0 0
                0" ) # Vertex ID =
                1


                cubit.cmd
                ("create vertex 2 0
                0" ) # Vertex ID =
                2


                cubit.cmd
                ("create curve vertex 1
                2" ) # Curve ID = 1


                # 2) Apply a pinpoint scheme at
                positions 0.5 and 1.5.


                cubit.cmd
                ("curve 1 scheme pinpoint
                location 0.5 1.5" )


                # 3) Query the pinpoint
                locations.


                locations =
    cubit.get_curve_mesh_scheme_pinpoint_locations
                (1)


                print("Curve 1 pinpoint
                locations:" , locations) #
                Should print: [0.5, 1.5]
```

```
 @n type of curve_id:  int
```

Parameters

curve_id ID of the curve whose pinpoint locations are retrieved.
return type of : std::vector< double,std:: allocator< double > >

Returns

std::vector<double> of positions along the curve where nodes are placed.

◆ get_curve_nodes()

def cubit.get_curve_nodes (*curve_id*)

Get the list of node IDs owned by a curve.

This function returns the IDs of nodes owned by the specified curve. Nodes on the bounding vertices of the curve are excluded.

```
.. code-block:: python
```

```
                # Create a brick and mesh volume
                1


                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("mesh vol 1" )


                # Query the nodes on curve
                1


                curve_nodes = cubit.get_curve_nodes
                (1)


                print(f"Curve 1 owns
                {len(curve_nodes)} node(s)." )
```

for node_id in curve_nodes: print(f" Node ID: {node_id}")

```
  @n type of curve_id:  int
```

Parameters

curve_id ID of the curve.
return type of : std::vector< int,std:: allocator< int > >

Returns

List of node IDs owned by the curve (excluding nodes on bounding vertices).

◆ get_curve_radius()

def cubit.get_curve_radius (*curve_id*)

Compute the radius of a specified curve.

Title Computation

- For circular arcs, returns the constant radius.
- For ellipses, parabolas, splines, helices, and hyperbolas, returns the radius of curvature at the midpoint (curvature evaluated and inverted; below numerical tolerance treated as infinite and returned as 0.0).
- For straight lines or invalid/unsupported curve IDs, returns 0.0.

.. code-block:: python

```
# 1) Create a straight
line


cubit.cmd
("create vertex 0 0
0" )


cubit.cmd
("create vertex 1 0
0" )


cubit.cmd
("create curve vertex 1
2" ) # curve 1


r1 = cubit.get_curve_radius
(1)


print("Curve 1
radius:" , r1) # Expected:
0.0



# 2) Create a cylinder of radius
2 and height 5


cubit.cmd
("cylinder rad 2 height
5" ) # curve 2 (end
circle)


r2 = cubit.get_curve_radius
(2)


print("Curve 2
radius:" , r2) # Expected:
2.0
```

@n type of curve_id: int

Parameters

curve_id ID of the curve to query.
return type of : float

Returns

Constant radius for circular arcs; radius of curvature at midpoint for other curved types; 0.0 for straight lines or invalid/unsupported IDs.

◆ get_curve_type()

def cubit.get_curve_type (*curve_id*)

Get the curve type for a specified curve.

Returns a descriptive string indicating the curve's geometry.

Title Available curve types

- Arc curve
- Ellipse curve
- Parabola curve
- Segmented/facet curve
- Spline curve
- Straight curve
- Point curve
- Helical curve
- Undefined curve type

```
.. code-block:: python
```

```
                # Assume curve ID 12 corresponds
                to a spline.

                curve_type = cubit.get_curve_type
                (12)

                print("Curve 12 is
                a:" , curve_type) #
                Output: Curve 12 is a: Spline curve
```

```
 @n type of curve_id:  int
```

Parameters

curve_id ID of the curve to query.
return type of : string

Returns

Descriptive string of the curve's geometry type.

◆ get_dbl_sculpt_default()

def cubit.get_dbl_sculpt_default (*variable*)

Retrieve the default value of a named sculpt parameter.

Matches the given substring against known double-valued sculpt parameters and returns its default.

Title Available parameters

- AUTO_SIZE
- OPT_THRESHOLD
- CURVE_OPT_THRESHOLD
- GQ_THRESHOLD
- DEFAULT_GEO_SMOOTH_MAX_DEVIATION
- PCOL_THRESHOLD
- DEG_THRESHOLD
- HTET_THRESHOLD
- ADAPT_THRESHOLD
- CAPTURE_ANGLE
- EXPAND_PERCENT
- THICKEN_VOID
- PILLOW_THRESHOLD
- REMOVE_BAD
- MIN_ELEM_THICKNESS

```
                * XTRANSLATE, YTRANSLATE,
    ZTRANSLATE

                * XSCALE, YSCALE, ZSCALE
```

type of variable: string

Parameters

variable Substring of the sculpt parameter name to look up.
return type of : float

Returns

The default double value for the matched parameter.

◆ get_default_auto_size()

def cubit.get_default_auto_size ( )

Compute Cubit's heuristic default automatic mesh size for the current model.

Returns the target edge length determined by Cubit's auto-sizing heuristics (model dimensions, curve lengths, etc.). This is equivalent to running:

.. code-block:: c++

```
cubit.cmd
("vol all size auto factor
5" )
```

and then querying:

.. code-block:: c++

```
get_mesh_size("volume" , 1)
```

.. code-block:: python

```
import cubit

size = cubit.get_default_auto_size
()

print("Default auto mesh
size:" , size)
```

```
@n return type of :  float
```

Returns
: Target edge length computed by Cubit's auto-sizing algorithm.

◆ get_default_element_type()

def cubit.get_default_element_type ( )

Retrieve the current default element type for meshing.

.. code-block:: python

```
default_type =
cubit.get_default_element_type
()

print("Default element
type:" , default_type)
```

```
@n return type of :  string
```

Returns
: A string indicating the default mesh type:

- "tri" indicates a tri/tet mesh default
- "hex" indicates a quad/hex mesh default
- "none" indicates no default has been assigned

◆ get_default_geometry_engine()

def cubit.get_default_geometry_engine ( )

Get the name of the default modeler engine.

The default engine is the geometry kernel used for creating new geometry. ACIS is the default but can be changed to facet.

```
.. code-block:: python
```

```
cubit.cmd
("set geom engine
facet" ) # set engine to
facet

engine =
cubit.get_default_geometry_engine
()

print("Default
engine:" , engine) #
prints facet
```

```
@n return type of :  string
```

Returns
: The name of the default modeler engine ACIS, CATIA, OCC, or facet.

◆ get_displacement_combine_type()

def cubit.get_displacement_combine_type (*entity_id*)

Get the combine method for a displacement BC.

Valid options include "Overwrite", "Average", "SmallestCombine", or "LargestCombine".
type of entity_id: int

Parameters

entity_id ID of the displacement BC.
return type of : string

Returns

Combine method as a string.

◆ get_distance_between()

def cubit.get_distance_between (*vertex_id_1*, *vertex_id_2*)

Get the Euclidean distance between two vertices.

.. code-block:: python

```
                # Create two standalone vertices
                at (0,0,0) and (1,1,1).


                cubit.cmd
                ("create vertex 0 0
                0" ) # vertex 1


                cubit.cmd
                ("create vertex 1 1
                1" ) # vertex 2


                dist = cubit.get_distance_between
                (1, 2)


                print("Distance between
                vertices 1 and 2:" , dist) # prints
    1.73205
```

```
@n type of vertex_id_1:  int
```

Parameters

vertex_id_1 ID of the first vertex.
type of vertex_id_2: int

vertex_id_2 ID of the second vertex.
return type of : float

Returns

The straight-line distance between the two vertices.

◆ get_distance_between_entities()

def cubit.get_distance_between_entities (*geom_type_1*, *entity_id_1*, *geom_type_2*, *entity_id_2*)

Get the minimum distance between two geometry entities.

Computes the shortest straight-line distance between any point on the first entity and any point on the second entity. Supported entity types are "vertex", "curve", "surface", and "volume".

.. code-block:: python

```
                # Create a 10x10x10 brick
                centered at the origin; volume 1
    occupies -5 to +5 in
                X.

                cubit.cmd
                ("brick x 10" )


                # Create a vertex at (10, 0, 0);
                this is vertex 9.

                cubit.cmd
                ("create vertex 10 0
                0" )


                # Distance from vertex 9 to
                volume 1 is |10 - 5| = 5.

                dist =
    cubit.get_distance_between_entities
                ("vertex" , 9,
                "volume" , 1)

                print("Distance between
                vertex 9 and volume 1:" , dist) #
    prints 5
```

@n type of geom_type_1: string

Parameters

geom_type_1 Type of the first entity ("vertex", "curve", "surface", or "volume").
type of entity_id_1: int

entity_id_1 ID of the first entity.
type of geom_type_2: string

geom_type_2 Type of the second entity ("vertex", "curve", "surface", or "volume").
type of entity_id_2: int

entity_id_2 ID of the second entity.
return type of : float

Returns

The minimum straight-line distance between the two specified entities.

◆ get_distance_from_curve_start()

def cubit.get_distance_from_curve_start (*x_coordinate*, *y_coordinate*, *z_coordinate*, *curve_id*)

Get the distance along a curve from its start to the closest point on the curve.

If the given xyz is not exactly on the curve, the closest point on the curve is used.

```
.. code-block:: python
```

```
                # Create a cylinder of radius 1
                and height 1; its circular edge is
    curve 1.


                cubit.cmd
                ("cylinder rad 1 height
                1" )



                # Measure distance from the start
                of curve 1 to the closest point on
    curve 1 from
                (-1,0,0).


                dist =
    cubit.get_distance_from_curve_start
                (-1.0, 0.0, 0.0, 1)


                print("Distance along curve
                1:" , dist) # prints
                approximately 3.14159
```

```
@n type of x_coordinate:  float
```

Parameters

x_coordinate X coordinate of the query point.
type of y_coordinate: float

y_coordinate Y coordinate of the query point.
type of z_coordinate: float

z_coordinate Z coordinate of the query point.
type of curve_id: int

curve_id ID of the curve to measure along.
return type of : float

Returns

The arc distance from the curve's start to the closest point on the curve to the specified xyz.

◆ get_edge_count()

def cubit.get_edge_count ( )

Retrieve the count of edges between surface elements in the current model.

For a volume mesh, only edges on the surface-between surface elements-are counted.

```
.. code-block:: python
```

```
                cubit.cmd
                ("brick x 10" )
                # create brick (Volume ID =
                1)


                cubit.cmd
                ("mesh vol 1" )
                # mesh volume 1 (hex or
                tet)


                edge_count = cubit.get_edge_count
                ()


                print("Edge count:"
                , edge_count)
```

```
@n return type of :  int
```

Returns

Number of edges between surface elements in the current model's mesh.

◆ get_edge_global_element_id()

def cubit.get_edge_global_element_id (*edge_id*)

Get the Global Element ID for a specific edge element.

Cubit assigns a Global Element ID to each mesh element when it is placed into a block. This function returns the Global Element ID corresponding to the given local edge ID within its type-specific ID space.

```
.. code-block:: python
```

```
                    # Create geometry and mesh a
                    curve with equal spacing

                    cubit.cmd
                    ("brick x 10" )

                    cubit.cmd
                    ("curve 1 scheme
                    equal" )

                    cubit.cmd
                    ("mesh curve 1" )


                    # Assign all edge elements on
                    curve 1 to block 1

                    cubit.cmd
                    ("block 1 edge in curve
                    1" )


                    # Query the Global Element ID of
                    local edge 3

                    gid = cubit.get_edge_global_element_id
                    (3)

                    print("Global Element ID of
                    edge 3:" , gid) # prints
                    3
```

```
 @n type of edge_id:  int
```

Parameters

edge_id Local edge element ID within its type-specific ID space.
return type of : int

Returns

Global Element ID for the specified edge, or 0 if not placed into a block.

◆ get_edges_to_swap()

def cubit.get_edges_to_swap (*curve_id*)

Get edges on triangles at a knife-edge curve that are candidates for swapping.

Given a curve that defines a knife-edge between two triangle-meshed surfaces, this function returns the IDs of mesh edges on the triangles at the curve that are good candidates for edge swapping.

Swapping these edges can improve the local dihedral angles between adjacent triangles, resulting in larger available volumes for successful tet meshing.

This function is commonly used as a postprocessing step after tri or tet meshing to identify edges that may be swapped to improve mesh quality. The Cubit command: `Swap Edge <ids>` can then be used to perform the actual swapping.

`.. code-block:: python`

```
                # Identify candidate edges to
                swap at curve 12

                edge_ids = cubit.get_edges_to_swap
                (12)

                print(f"Curve 12 has
                {len(edge_ids)} candidate edge(s) to
    swap." )
```

if edge_ids: swap_cmd = "Swap Edge" for edge_id in edge_ids: swap_cmd += f" {edge_id}"

```
                print(f" Edge ID:
                {edge_id}" )


                # Execute the Swap Edge
                command

                cubit.cmd
                (swap_cmd)
```

`@n type of curve_id:  int`

Parameters

curve_id ID of the curve defining the knife-edge.
return type of : std::vector< int,std:: allocator< int > >

Returns

List of edge IDs that are good candidates for swapping.

◆ get_elem_quality_stats()

def cubit.get_elem_quality_stats (*entity_type*, *id_list*, *metric_name*, *single_threshold*, *use_low_threshold*, *low_threshold*, *high_threshold*, *make_group*)

Python-friendly version of get_quality_stats without reference parameters.

Computes quality statistics and returns all results–including failing element IDs–in a single vector.

`.. code-block:: python`

```
# Python example

id_list = [223, 226, 256]

single_threshold = 0.2

use_low_threshold = False

low_threshold = 0.0

high_threshold = 0.0

make_group = True


quality_data =
cubit.get_elem_quality_stats
(

"hex" , id_list,
"scaled jacobian" ,

single_threshold, use_low_threshold,

low_threshold, high_threshold,

make_group)


min_value = quality_data[0]

max_value = quality_data[1]

mean_value = quality_data[2]

std_value = quality_data[3]

min_element_id = int(quality_data[4])

max_element_id = int(quality_data[5])

element_type = int(quality_data[6]) #
0=edge,1=tri,2=quad,3=tet,4=hex

bad_group_id = int(quality_data[7])

num_failures = int(quality_data[8])

mesh_list = [int(x) for x in
quality_data[9:9+num_failures]]
```

@n type of entity_type: string

Parameters

entity_type	Geometry type ("edge", "tri", "quad", "tet", "hex"). type of id_list: std::vector< int,std::allocator< int > >
id_list	List of element IDs to evaluate. type of metric_name: string
metric_name	Quality metric name (e.g., "scaled jacobian"). type of single_threshold: float
single_threshold	Threshold when using a single bound. type of use_low_threshold: boolean
use_low_threshold	If true, apply [low_threshold, high_threshold]; otherwise use single threshold. type of low_threshold: float
low_threshold	Lower bound for quality range. type of high_threshold: float
high_threshold	Upper bound for quality range. type of make_group: boolean
make_group	If true, create a group for failing elements and return its ID. return type of : std::vector< double,std::allocator< double > >

◆ get_element_block()

Returns

Vector<double> containing: [0] min_value [1] max_value [2] mean_value [3] std_value [4] min_element_id [5] max_element_id [6] element_type (0=edge,1=tri,2=quad,3=tet,4=hex) [7] bad_group_id [8] mesh_list size (n) [9..] mesh_list element IDs

def cubit.get_element_block (*element_id*)

Get the block ID containing a given global element.

In Cubit, elements receive a Global Element ID when placed into a block. This function returns the ID of the block that contains the specified global element. Returns 0 if the element is not assigned to any block. You can also use the interactive command `list element <global_id>` to see the block.

```
.. code-block:: python
```

```
                # Create geometry and mesh volume
                1


                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("mesh vol 1" )


                # Assign hex elements to block 1
                and faces to block 2


                cubit.cmd
                ("block 1 hex all" )
                # global IDs start at 1


                cubit.cmd
                ("block 2 face in surface
                1" ) # global IDs follow
                block 1


                # Query block for hex 1 (should
                be 1)


                block1 = cubit.get_element_block
                (1)


                print(f"Element 1 is in
                block {block1}" ) # prints
                1


                # Query block for face 1 (global
                ID 1001 ? block 2)


                block2 = cubit.get_element_block
                (1001)


                print(f"Element 1001 is in
                block {block2}" ) # prints
                2
```

```
 @n type of element_id:  int
```

Parameters

element_id Global Element ID to query.
return type of : int

Returns

ID of the block containing the element, or 0 if none.

◆ get_element_budget()

def cubit.get_element_budget (*element_type*, *entity_id_list*, *auto_factor*)

Estimate the total element count for a set of volumes given size settings.

Calculates an approximate "element budget" (total number of elements) for the specified volumes, based on the current mesh size factor and element type. For hexahedral meshes, the target edge length (esize) is related to the model volume (Vmodel) and the hex count (Nhex) by:

.. code-block:: c++

```
esize = (Vmodel / Nhex)^(1/3)
```

Solving for Nhex gives:

.. code-block:: c++

```
Nhex = Vmodel / (esize^3)
```

For tetrahedral meshes, the element count is roughly seven times that of a hex mesh with the same edge length (i.e., Ntet ~ 7 * Nhex)

```
.. code-block:: python
```

```
import cubit

# Estimate hex element budget on
volumes 1 and 2 with auto factor = 3

vols = [1, 2]

hex_budget = cubit.get_element_budget
("hex" , vols, 3)

print("Hex element
budget:" , hex_budget)


# Estimate tet element budget on
volumes 1 and 2 with auto factor = 3

tet_budget = cubit.get_element_budget
("tet" , vols, 3)

print("Tet element
budget:" , tet_budget)
```

```
@n type of element_type:  string
```

Parameters

element_type Element type: "hex" or "tet". Determines whether to estimate a hexahedral or tetrahedral budget.
type of entity_id_list: std::vector< int,std::allocator< int > >

entity_id_list Vector of volume IDs over which to compute the total model volume.
type of auto_factor: int

auto_factor The "auto size" factor n (as in `size auto factor n`). This factor scales the base size; a larger factor produces larger elements and fewer total elements.
return type of : int

Returns

Approximate number of elements that will be generated (integer).

◆ get_element_count()

def cubit.get_element_count ( )

Retrieve the count of exportable elements in the current model.

Exportable elements are those that have been assigned to a block. If no blocks are defined, all elements (nodes, edges, quad, hex, tet, tri, wedge, sphere, etc.) are considered exportable.

```
.. code-block:: python
```

```
cubit.cmd
("brick x 10" )
# create first brick (Volume ID =
1)


cubit.cmd
("brick x 10" )
# create second brick (Volume ID
= 2)


cubit.cmd
("vol 1 mov x 10" )
# move volume 1 to abut volume
2


cubit.cmd
("merge all" )
# merge all volumes into one
model


cubit.cmd
("mesh vol all" )
# mesh every volume (hex
meshes)


cubit.cmd
("block 1 vol 1" )
# assign volume 1 elements to
block 1


# Only hex elements from volume 1
are now deemed exportable


elem_count = cubit.get_element_count
() # count exportable
elements


print("Element
count:" , elem_count)
```

```
@n return type of :  int
```

Returns

Number of exportable elements in the current model's mesh.

◆ get_element_exists()

def cubit.get_element_exists (*element_id*)

Check whether a global mesh element ID exists in the model.

In Cubit, elements receive a Global Element ID when they are placed into a block. This function returns `true` if the specified global element ID has been established (i.e., the element was assigned to a block); otherwise it returns `false` .

```
.. code-block:: python
```

```
                # Create geometry and mesh volume
                1


                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("mesh vol 1" )



                # Before block assignment, no
                global IDs exist


                exists_before =
    cubit.get_element_exists
                (10)


                print(f"Element 10 exists
                before block? {'Yes' if exists_before
    else
                'No'}" ) # No



                # Assign all hex elements to
                block 1 (establishes global IDs)


                cubit.cmd
                ("block 1 hex all" )



                # After block assignment, global
                IDs exist


                exists_after = cubit.get_element_exists
                (10)


                print(f"Element 10 exists
                after block? {'Yes' if exists_after
    else 'No'}"
                ) # Yes
```

```
@n type of element_id:  int
```

Parameters

element_id Global element ID to check (as established by block assignment).
return type of : boolean

Returns

`true` if the element has a Global Element ID, `false` otherwise.

◆ get_element_type()

def cubit.get_element_type (*element_id*)

Get the specific mesh element type for a global element ID.

CUBIT supports a variety of element types, each with different node counts and accuracy orders. This function returns one of the specific type strings defined in CUBIT; for example, "HEX20", "TETRA4", "TRISHELL7", etc. Supported element types are detailed in the CUBIT documentation or the Exodus manual.

```
.. code-block:: python
```

```
                # Create a brick and assign hex20
                elements to block 1


                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("block 1 vol 1" )


                cubit.cmd
                ("block 1 element type
                hex20" )


                cubit.cmd
                ("mesh vol 1" )


                cubit.cmd
                ("export mesh
                \"model.exo\"" )


                # Query the type of the first
                element in block 1 (global ID 1)


                elem_type = cubit.get_element_type
                (1)
```

print("Element 1 type:", elem_type) // should print HEX20

```
  @n type of element_id:  int
```

Parameters

element_id Global element ID (as assigned by Exodus/Genesis).
return type of : string

Returns

A string naming the specific element type (for example, HEX, HEX20, TETRA4, TRISHELL7, etc.).

◆ get_element_type_id()

def cubit.get_element_type_id (*element_id*)

Map a Global Element ID back to its local mesh entity ID.

Cubit assigns each element a Global Element ID when it is placed into a block. This function returns the corresponding local mesh entity ID within that element's type-specific ID space (hex, tet, tri, etc.). Together with get_element_type, it lets you determine both the element's type and its local ID.

```
.. code-block:: python
```

```
                # Create geometry and mesh volume
                1

                cubit.cmd
                ("brick x 10" )

                cubit.cmd
                ("mesh vol 1" )

                # Assign hex elements to block 1
                (global IDs start at 1)

                cubit.cmd
                ("block 1 hex all" )

                # Assign face elements on surface
                1 to block 2 (global IDs start after
    hexes)

                cubit.cmd
                ("block 2 face in surface
                1" )


                # Query element 1001 (first face
                in block 2)

                elem_type = cubit.get_element_type
                (1001)

                local_id = cubit.get_element_type_id
                (1001)

                print(f"Element 1001: type
                = {elem_type}, local ID = {local_id}" )

                # Expected output: Element 1001:
                type = face, local ID = 1
```

```
 @n type of element_id:  int
```

Parameters

element_id Global Element ID (as assigned in the Exodus file).
return type of : int

Returns

Local mesh entity ID within its type-specific ID space.

◆ get_entities()

def cubit.get_entities (*entity_type*)

Retrieve IDs of all current entities of a specified type (geometry and mesh).

Returns the IDs of every entity matching the given type, including both geometry entities (body, volume, surface, curve, vertex) and mesh entities (node, element).

Notes To restrict the results to geometry entities only, use **get_current_ids()** .

```
.. code-block:: python
```

```
                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("create cylinder radius 3
                height 12" )


                volume_ids = cubit.get_entities
                ("volume" )
                # returns (1, 2, ...)


                node_ids = cubit.get_entities
                ("node" )
                # returns (101, 102, ...)
```

```
 @n type of entity_type:  string
```

Parameters

entity_type Type of entity to query: geometry types ("body", "volume", "surface", "curve", "vertex") or mesh types ("node", "element").
return type of : std::vector< int,std:: allocator< int > >

Returns

A vector<int> (or Python tuple) of IDs of all entities of the specified type.

◆ get_entity_color()

def cubit.get_entity_color (*entity_type*,
entity_id
)

Get the color of a specified entity.

Returns the RGBA color as four doubles in [0, 1] for the given entity.

```
.. code-block:: python
```

```
                # Assume curve 33 has been
                assigned a color earlier.


                color = cubit.get_entity_color
                ("curve" , 33)


                print(f"R: {color[0]} G:
                {color[1]} B: {color[2]} A: {color[3]}"
    )
```

```
 @n type of entity_type:  string
```

Parameters

entity_type Type of the entity ("body", "volume", "surface", "curve", or "vertex").
type of entity_id: int

entity_id ID of the entity to query.
return type of : std:: array< double,4 >

Returns

A 4-element array [R, G, B, A], each component in [0, 1].

◆ get_entity_color_index()

def cubit.get_entity_color_index (*entity_type*, *entity_id*)

Get the color index of a specified entity.

Cubit uses predefined color indices to represent common colors.

Title Available color indices

- -1 default color
- 0 black (0,0,0)
- 1 grey (127,127,127)
- 2 orange (255,165,0)
- 3 red (255,0,0)
- 4 green (0,255,0)
- 5 yellow (255,255,0)
- 6 magenta (255,0,255)
- 7 cyan (0,255,255)
- 8 blue (0,0,255)
- 9 white (255,255,255)
- 10 brown (165,42,42)
- 11 gold (255,215,0)
- 12 lightblue (173,216,230)
- 13 lightgreen (0,204,0)
- 14 salmon (250,128,114)
- 15 coral (255,127,80)
- 16 pink (255,192,203)

```
.. code-block:: python
```

```
                # Set the color of curve 33 to
                red.

                cubit.cmd
                ("color curve 33
                red" )

                idx = cubit.get_entity_color_index
                ("curve" , 33)

                print("Color index:"
                , idx) # prints 3
```

```
 @n type of entity_type:  string
```

Parameters

entity_type Type of the entity ("body", "volume", "surface", "curve", or "vertex").
type of entity_id: int

entity_id ID of the entity to query.
return type of : int

Returns

: The color index -1 (default) or 0-16 for predefined colors.

◆ get_entity_modeler_engine()

def cubit.get_entity_modeler_engine (*geometry_type*, *entity_id*)

Get the modeler engine type for a specified entity.

Returns the engines associated with this entity. Valid return strings are: acis, facet, virtual.

```
.. code-block:: python
```

```
                engine_list =
      cubit.get_entity_modeler_engine
                ("surface" , 47)

                print(engine_list) # e.g.,
                ('acis', 'facet')
```

Parameters

geom_type Specifies the geometry type ("body", "volume", "surface", "curve", or "vertex").
type of entity_id: int

entity_id ID of the entity.
return type of : std::vector< std::string,std::allocator< std:: string > >

Returns

: A list of modeler engine names "acis", "facet", or "virtual".

◆ get_entity_name()

def cubit.get_entity_name (*entity_type*,
entity_id,
no_default = `False`
)

Get the name of a specified entity.

Returns either a user-defined name (stored in Cubit) or a default name generated at run-time. If no_default is true and the entity has only a default name, an empty string is returned.

`.. code-block:: python`

```
# Create and name a
vertex.


cubit.cmd
("vertex 22 name
'point_load'" )


# Query the name of vertex
22.


name = cubit.get_entity_name
("vertex" , 22,
True )


print("Entity name:"
, name) # prints
"point_load"
```

`@n type of entity_type:  string`

Parameters

entity_type Type of the entity ("body", "volume", "surface", "curve", or "vertex").
type of entity_id: int

entity_id ID of the entity.
type of no_default: boolean, optional

no_default If true, return an empty string when only a default name exists.
return type of : string

Returns

The entity's name, or an empty string if no_default is true and the name is default.

◆ get_entity_names()

def cubit.get_entity_names (*entity_type*,
entity_id,
no_default = `False`,
first_name_only = `False`
)

Get all names associated with a specified entity.

Retrieves every name attribute set on the entity. By default, returns all names, but if first_name_only is true, only the first name is returned. Setting no_default to true excludes default names.

`.. code-block:: python`

```
# Create and assign two names to
vertex 22.


cubit.cmd
("vertex 22 name
'point_load'" )


cubit.cmd
("vertex 22 name
'PSI'" )



# Retrieve all names.


names = cubit.get_entity_names
("vertex" , 22,
True , False )


print(names) # prints
('point_load', 'PSI')



# Retrieve only the first
name.


first_only = cubit.get_entity_names
("vertex" , 22,
True , True )


print(first_only[0]) # prints
"point_load"
```

`@n type of entity_type:  string`

Parameters

entity_type	Type of the entity ("body", "volume", "surface", "curve", or "vertex"). type of entity_id: int
entity_id	ID of the entity. type of no_default: boolean, optional
no_default	If true, excludes default names from the returned list. type of first_name_only: boolean, optional
first_name_only	If true, returns only the first name attribute. return type of : std::vector< std::string,std::allocator< std:: string > >

Returns

A vector of all name strings (or a single-element vector if first_name_only is true).

◆ get_entity_sense()

def cubit.get_entity_sense (*source_type*,
source_id,
sideset_id
)

Get the sense of an entity in a sideset.

This function returns the sense of the specified entity (face, quad, or tri) within the specified sideset. The sense indicates the orientation of the entity relative to the sideset.

```
.. code-block:: python
```

```
                # Query the sense of face 332 in
                sideset 2

                sense = cubit.get_entity_sense
                ("face" , 332, 2)

                print(f"Sense of face 332
                in sideset 2: {sense}" )
```

```
@n type of source_type:  string
```

Parameters

source_type Item type: "face", "quad", or "tri".
type of source_id: int

source_id ID of the entity.
type of sideset_id: int

sideset_id ID of the sideset.
return type of : string

Returns

Sense of the specified entity in the sideset.

◆ get_error_count()

def cubit.get_error_count ( )

Retrieve the total number of errors in the current Cubit session.

Use this to check whether any errors have occurred since Cubit was started (or since the last manual reset).

Errors can be cleared by issuing the Cubit command `reset error` .

```
.. code-block:: python
```

```
                import cubit

                # Execute some commands...

                errors = cubit.get_error_count
                ()

                print("Number of
                errors:" , errors)


                # Reset errors

                cubit.cmd
                ("reset error" )

                print("Number of errors
                after reset:" , cubit.get_error_count
                ())
```

```
@n return type of :  int
```

Returns

Integer count of errors recorded in this Cubit session.

◆ get_exodus_element_count()

def cubit.get_exodus_element_count (*entity_id*, *entity_type*)

Get the number of elements in a Exodus entity.

.. code-block:: python

```
                element_count =
      cubit.get_exodus_element_count
                (2, "sideset" )
```

```
@n type of entity_id:  int
```

Parameters

entity_id The id of the entity
type of entity_type: string

entity_type The type of the entity
return type of : int

Returns

Number of Elements

◆ get_exodus_entity_description()

def cubit.get_exodus_entity_description (*entity_type*, *entity_id*)

Get the description of an Exodus entity.

Returns the description string associated with a block, sideset, or nodeset entity. If no description is assigned or the entity does not exist, an empty string is returned.

```
.. code-block:: python
```

```
                # Issue Cubit commands to create
                a brick and assign a sideset with a
      description


                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("sideset 1 surface
                1" )


                cubit.cmd
                ("sideset 1 description
                \"Pressure boundary for inlet\"" )


                # Query the description of
                sideset 1


                entity_description =
      cubit.get_exodus_entity_description
                ("sideset" , 1)


                # Print the result


                print("Entity
                description:" , entity_description)


                # Expected output
                (example):


                # Entity description: Pressure
                boundary for inlet
```

```
 @n type of entity_type:  string
```

Parameters

entity_type Type of the entity: "block", "sideset", or "nodeset".
type of entity_id: int

entity_id ID of the entity.
return type of : string

Returns

Description of the entity, or an empty string if none is assigned.

◆ get_exodus_entity_name()

def cubit.get_exodus_entity_name (*entity_type*, *entity_id*)

Get the name of an Exodus entity.

Retrieves the user-defined name of an Exodus entity of the given type and ID.

```
.. code-block:: python
```

```
                # Issue Cubit commands to create
                a brick and assign a sideset with a
    name


                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("sideset 1 surface
                1" )


                cubit.cmd
                ("sideset name
                \"pressure_load\"" )


                # Query the name of sideset
                1


                entity_name =
    cubit.get_exodus_entity_name
                ("sideset" , 1)


                # Print the result


                print("Entity name:"
                , entity_name)


                # Expected output:


                # Entity name:
                pressure_load
```

```
@n type of entity_type:  string
```

Parameters

entity_type Type of the entity: "block", "sideset", or "nodeset".
type of entity_id: int

entity_id ID of the entity.
return type of : string

Returns

Name of the entity, or an empty string if none is assigned.

◆ get_exodus_entity_type()

def cubit.get_exodus_entity_type (*entity_type*, *entity_id*)

Get the Exodus type of an entity.

Returns the Exodus type string associated with the mesh group of a block, sideset, or nodeset. If the entity does not exist, an empty string is returned.

.. code-block:: python

```
                # Issue Cubit commands to create
                a brick and assign a sideset

                cubit.cmd
                ("brick x 10" )

                cubit.cmd
                ("sideset 1 surface
                1" )

                # Query the Exodus type of
                sideset 1

                entity_type =
      cubit.get_exodus_entity_type
                ("sideset" , 1)

                # Print the result

                print("Entity type:"
                , entity_type)

                # Expected output
                (example):

                # Entity type: lite
```

@n type of entity_type: string

Parameters

entity_type Type of the entity: "block", "sideset", or "nodeset".
type of entity_id: int

entity_id ID of the entity.
return type of : string

Returns

Exodus type string of the entity (e.g., "lite"), or an empty string if the entity does not exist.

◆ get_exodus_id()

def cubit.get_exodus_id (*entity_type*,
entity_id
)

Get the Global Element ID for a mesh entity.

In Cubit, each mesh element type (hex, tet, quad, tri, etc.) has a local ID unique only among its type. When elements are placed into a block, they receive a Global Element ID that is unique across all blocks and element types. Global Element IDs are exported to Exodus files for downstream applications to map back to the original Cubit elements.

Notes In an interactive session you can also use `list <type> <local_id>` (e.g. `list hex 100`) to display the Global Element ID.

`.. code-block:: python`

```
# Reset session and create a
brick

cubit.cmd
("reset" )

cubit.cmd
("brick x 10" )


# Mesh all volumes

cubit.cmd
("mesh vol all" )


# Place all hex elements into
block 1

cubit.cmd
("block 1 hex all" )

# Place all face elements on
surface 1 into block 2

cubit.cmd
("block 2 face in surface
1" )


# Query the Global Element ID of
local hex 1

hex_gid = cubit.get_exodus_id
("hex" , 1)

print("Global Element ID of
hex 1: " , hex_gid) #
prints 1


# Query the Global Element ID of
local face 1

face_gid = cubit.get_exodus_id
("face" , 1)

print("Global Element ID of
face 1:" , face_gid) #
prints 1001 (block 2 follows block 1)
```

`@n type of entity_type:  string`

Parameters

entity_type Mesh element type ("hex", "tet", "quad", "tri", "face", etc.).
type of entity_id: int

entity_id Local mesh entity ID within its type.
return type of : int

Returns

Global Element ID (unique across all blocks and types), or 0 if the mesh has not been exported.

◆ get_exodus_sizing_function_file_name()

def cubit.get_exodus_sizing_function_file_name ( )

Retrieve the Exodus sizing function file name.

Returns the path or file name of the Exodus II file from which the sizing function was imported. If no file has been imported, returns an empty string.

.. code-block:: python

```
                import cubit

                # After reading an Exodus field
                function:

                # cubit.cmd("Import Sizing
                Function 'mesh.sfb' Block all Variable
      'curvature' Time
                0")

                file_name =
      cubit.get_exodus_sizing_function_file_name
                ()

                print("Exodus sizing
                file:" , file_name)
```

@n return type of : string

Returns
The file name of the imported Exodus II sizing function, or an empty string.

◆ get_exodus_sizing_function_variable_name()

def cubit.get_exodus_sizing_function_variable_name ( )

Retrieve the Exodus-based sizing function variable name.

Returns the name of the field variable currently used as the sizing function. This is the Exodus variable (node- or element-based) driving adaptive meshing.

.. code-block:: python

```
                import cubit

                # After reading an Exodus field
                function:

                # cubit.cmd("Import Sizing
                Function 'strain.exo' Block 10 Variable
                'plastic_strain' Last")

                var_name =
      cubit.get_exodus_sizing_function_variable_name
                ()

                print("Current Exodus
                sizing variable:" , var_name)
```

@n return type of : string

Returns
The name of the Exodus variable used for sizing.

◆ get_exodus_variable_count()

def cubit.get_exodus_variable_count (*container_type*, *container_id*)

Get the number of Exodus variables in a nodeset, sideset, or block.

Parameters
entity_type : nodeset, sideset, or block
type of container_id: int
container_id The block id
return type of : int

Returns
Number of Exodus variables

◆ get_exodus_variable_names()

def cubit.get_exodus_variable_names (*container_type*, *container_id*)

Get the names of Exodus variables in a nodeset, sideset, or block.

Parameters

entity_type : nodeset, sideset, or block
type of container_id: int

container_id The block id
return type of : std::vector<
std::string,std::allocator< std:: string > >

Returns

Names of Exodus variables

◆ get_exodus_version()

def cubit.get_exodus_version ( )

Retrieve the Exodus mesh library version.

Returns the version string of the Exodus library used for mesh I/O.

```
.. code-block:: python
```

```
import cubit

exodus_version =
cubit.get_exodus_version
()

print("Exodus
version:" , exodus_version)
```

```
@n return type of :  string
```

Returns

A string containing the Exodus library version.

◆ get_expanded_connectivity()

def cubit.get_expanded_connectivity (*entity_type*, *entity_id*)

Get the list of node IDs for a mesh element, including interior nodes.

This function returns the IDs of all nodes associated with the specified mesh element, including both corner (boundary) and interior nodes. The ordering follows the Exodus convention; see the Exodus documentation for element-specific node ordering.

.. code-block:: python

```
                # Create a brick and assign
                volume 1 to block 1

                cubit.cmd
                ("brick x 10" )

                cubit.cmd
                ("block 1 vol 1" )

                # Specify a 20-node hexahedral
                element type for block 1

                cubit.cmd
                ("block 1 element type
                hex20" )

                # Mesh volume 1, producing hex20
                elements

                cubit.cmd
                ("mesh vol 1" )


                # Query expanded connectivity of
                the first hex20 element (ID 1)

                node_id_list =
    cubit.get_expanded_connectivity
                ("hex20" , 1)

                print(f"Hex20 element 1 has
                {len(node_id_list)} nodes (expected
    20):" ,
                node_id_list)
```

@n type of entity_type: string

Parameters

entity_type Mesh element type (e.g., "hex", "tet", "tri", "quad", "hex20").
type of entity_id: int

entity_id ID of the mesh element.
return type of : std::vector< int,std:: allocator< int > >

Returns

List of node IDs associated with the element, including interior nodes.

◆ get_force_direction_vector()

def cubit.get_force_direction_vector (*entity_id*)

Get the direction vector of a force BC.

type of entity_id: int

Parameters

entity_id ID of the force BC.
return type of : std:: array< double,3 >

Returns

Direction vector as [x, y, z].

◆ get_force_magnitude()

def cubit.get_force_magnitude (*entity_id*)

Get the force magnitude of a force BC.

type of entity_id: int

Parameters
entity_id ID of the force BC.
return type of : float

Returns
Force magnitude.

◆ get_force_moment_vector()

def cubit.get_force_moment_vector (*entity_id*)

Get the moment vector of a force BC.

type of entity_id: int

Parameters
entity_id ID of the force BC.
return type of : std:: array< double,3 >

Returns
Moment vector as [x, y, z].

◆ get_gaps_between_volumes()

def cubit.get_gaps_between_volumes (*target_volume_ids*, *maximum_gap_tolerance*, *maximum_gap_angle*, *cache_overlaps* = `0`)

Identify gaps between surfaces in a list of volumes.

This is a Python-friendly version of `get_volume_gaps` .

For each pair of volumes that have surface gaps, a `VolumeGap` object is returned. Each `VolumeGap` contains:

- the pair of volume IDs,
- a list of surface ID pairs (`surfPairs`),
- corresponding distances (`gaps`),
- corresponding overlap areas (`overlapAreas`).

Caching can be used to avoid redundant distance calculations when this function is called together with `get_overlapping_surfaces_in_volumes` . Both functions require the same underlying distance computations between surfaces in the specified volumes. If caching is enabled, the results of these distance calculations will be saved and reused when the second function is called. If you are calling only this function by itself, caching provides no performance benefit.

`.. code-block:: python`

```
cubit.cmd
('brick x 10' )
# Create volume 1


cubit.cmd
('brick x 10' )
# Create volume 2


cubit.cmd
('vol 2 mov x 10.01'
) # Move volume 2 slightly away
to create a small gap


target_volumes = [1, 2]


gaps = cubit.get_gaps_between_volumes
(target_volumes, 0.02, 5.0, 0)


print("Volume gaps:"
)
```

for gap in gaps: print(f"Volumes: ({gap.volume1}, {gap.volume2})") for surf_pair, dist, area in zip(gap.surfPairs, gap.gaps, gap.overlapAreas): print(f" Surfaces: ({surf_pair[0]}, {surf_pair[1]}), " f {dist:.6f}, Overlap area: {area}")

```
# Expected output:


# Volumes: (1, 2)


# Surfaces: (6, 10), Distance:
0.010000, Overlap area: 100.0
```

`@n type of target_volume_ids:  std::vector< int,std::allocator< int > >`

Parameters

target_volume_ids	List of volume IDs to examine. type of maximum_gap_tolerance: float
maximum_gap_tolerance	Maximum distance between surfaces to be considered a gap. type of maximum_gap_angle: float
maximum_gap_angle	Maximum angle between surfaces to be considered a gap. type of cache_overlaps: int, optional
cache_overlaps	Caching mode: 0 = no caching, 1 = clear cache first, 2 = use and add to existing cache. return type of : std::vector< VolumeGap,std::allocator< VolumeGap > >

Returns

A list of `VolumeGap` objects, one per volume pair with detected gaps. Each `VolumeGap` contains:

- `volume1`, `volume2`: the volume IDs
- `surfPairs`: list of surface ID pairs
- `gaps`: list of distances between surfaces
- `overlapAreas`: list of overlap areas between surfaces.

◆ get_geometric_owner()

def cubit.get_geometric_owner (*mesh_entity_type*, *mesh_entity_list*)

Get geometric owners for a set of mesh entities.

Returns the geometric owners of the specified mesh entities (e.g. "surface 3", "curve 5").

Title Supported mesh entity types

- quad
- face
- tri
- hex
- tet
- edge
- node

```
.. code-block:: python
```

```
                # Get owners of quads 1-8

                owners = cubit.get_geometric_owner
                ("quad" ,
                "1 to 8" )

                print("Geometric owners of
                quads 1 to 8:" , owners)

                # Example output: ['surface 1',
                'curve 2', 'curve 3', 'surface 4', ...]
```

```
 @n type of mesh_entity_type:  string
```

Parameters

mesh_entity_type Mesh entity type: "quad", "face", "tri", "hex", "tet", "edge", or "node".
type of mesh_entity_list: string

mesh_entity_list Space-delimited mesh entity IDs (e.g. "all", "1 to 8", "1 2 3").
return type of : std::vector< std::string,std::allocator< std:: string > >

Returns

Vector of owner strings in the form "surface x", "curve y", etc.

◆ get_geometry_node_count()

def cubit.get_geometry_node_count (*entity_type*, *entity_id*)

Get the number of mesh nodes on a geometric entity.

This function returns the count of mesh nodes associated with the specified geometric entity (surface, curve, etc.).

```
.. code-block:: python
```

```
                # Create and mesh a brick

                cubit.cmd
                ("brick x 10" )

                cubit.cmd
                ("mesh vol 1" )

                # Query node count on surface
                1

                node_count =
      cubit.get_geometry_node_count
                ("surface" , 1)

                print(f"Surface 1 has
                {node_count} node(s)." )
```

```
 @n type of entity_type:  string
```

Parameters

entity_type Type of the geometric entity ("surface", "curve", "volume", etc.).
type of entity_id: int

entity_id ID of the geometric entity.
return type of : int

Returns

Number of mesh nodes on the entity.

◆ get_geometry_owner()

def cubit.get_geometry_owner (*entity_type*,
entity_id
)

Get the geometric owner of a mesh element.

This function returns the geometric entity that owns the specified mesh element. For example, it may return "surface 3", "volume 1", etc., indicating where the element lives.

```
.. code-block:: python
```

```
                    # Create geometry and mesh
                    it


                    cubit.cmd
                    ("brick x 10" )


                    cubit.cmd
                    ("mesh vol 1" )



                    # Query the geometric owner of
                    hex element 221


                    geom_owner = cubit.get_geometry_owner
                    ("hex" , 221)


                    print(f"Geometric owner of
                    hex 221: {geom_owner}" )
```

```
 @n type of entity_type:  string
```

Parameters

entity_type Mesh element type (e.g., "hex", "tet", "tri", "quad", etc.).
type of entity_id: int

entity_id ID of the mesh element.
return type of : string

Returns

Geometric owner of the element (e.g., "surface 3").

◆ get_geometry_type()

def cubit.get_geometry_type (*geom_type*,
ent_ids
)

Return the analytic geometry type for a list of surfaces or curves.

This function behaves similar to get_surface_type and get_curve_type but accepts multiple IDs. Note also the difference in return types

```
.. code-block:: python
```

```
                    # Create a cylinder (curves 1 and
                    2 are circular edges).


                    cubit.cmd
                    ("create cylinder radius 3
                    height 10" )


                    curve_ids = [1, 2]


                    curve_types = cubit.get_geometry_type
                    ("curve" ,
                    curve_ids)


                    print(curve_types) # e.g.,
                    ['circle', 'circle']
```

```
 @n type of geom_type:  string
```

Parameters

geom_type Specifies the entity type: "curve" or "surface".
type of ent_ids: std::vector< int,std::allocator< int > >

ent_ids List of IDs for curves or surfaces to query.
return type of : std::vector< std::string,std::allocator< std:: string > >

Returns

A vector of geometry-type strings in the same order as ent_ids. Valid curve types: "arc", "ellipse", "parabola", "segmented", "spline", "straight", "point", "helix", "hyperbola", "undefined", "unknown". Valid surface types: "cone", "plane", "sphere", "spline", "torus", "best_fit", "facet", "cylinder", "undefined", "unknown".

◆ get_global_element_id()

def cubit.get_global_element_id (*element_type*, *id*)

Alias for get_exodus_id: get the Global Element ID for a local mesh entity.

This function is equivalent to get_exodus_id and returns the Global Element ID assigned to the specified local mesh entity within its type-specific ID space.

.. code-block:: python

```
# Create geometry, mesh it, and
assign hex elements to block 1

cubit.cmd
("brick x 10" )

cubit.cmd
("mesh vol 1" )

cubit.cmd
("block 1 hex all" )


# Query the Global Element ID of
local hex 22

gid = cubit.get_global_element_id
("hex" , 22)

print("Global Element ID of
hex 22:" , gid)
```

@n type of element_type: string

Parameters

element_type Type of the mesh entity ("hex", "tet", "wedge", "pyramid", "tri", "quad", "face", "edge", or "sphere").
type of id: int

id Local mesh entity ID within its type.
return type of : int

Returns

Global Element ID corresponding to the specified local entity.

◆ get_graphics_version()

def cubit.get_graphics_version ( )

Retrieve the VTK (Visualization Toolkit) version used by Cubit.

Returns the version string of the VTK graphics library integrated into Cubit.

.. code-block:: python

```
import cubit

vtk_version =
cubit.get_graphics_version
()

print("VTK version:"
, vtk_version)
```

@n return type of : string

Returns

A string containing the VTK version number.

◆ get_group_bodies()

def cubit.get_group_bodies (*group_id*)

Return direct child body IDs for a specified group, if any.

.. code-block:: python

```
# Create geometry and copy the
initial volume.


cubit.cmd
("brick x 10" )


cubit.cmd
("vol 1 copy mov x
10" )


# Define group containing all
bodies.


cubit.cmd
("group \"body_group\" add
body all" )


# Query group ID by name.


body_group_id = cubit.get_id_from_name
("body_group" )


# Get direct child body
IDs.


bodies = cubit.get_group_bodies
(body_group_id)


# Print tuple-like output.


print(tuple(bodies)) # prints
"(1,2)"
```

```
@n type of group_id:  int
```

Parameters

group_id ID of the group to examine.
return type of : std::vector< int,std:: allocator< int > >

Returns

Vector of body IDs (empty if none).

◆ get_group_curves()

def cubit.get_group_curves (*group_id*)

Return direct child curve IDs for a specified group, if any.

.. code-block:: python

```
# Create a brick and group all
curves.


cubit.cmd
("brick x 10" )


cubit.cmd
("group \"curve_group\" add
curve all" )


# Query group ID by name.


curve_group_id = cubit.get_id_from_name
("curve_group" )


# Get direct child curve
IDs.


curves = cubit.get_group_curves
(curve_group_id)


# Print tuple-like output.


print(tuple(curves)) # prints
"(1,2,3,4,5,6,7,8,9,10,11,12)"
```

```
@n type of group_id:  int
```

Parameters

group_id ID of the group to examine.
return type of : std::vector< int,std:: allocator< int > >

Returns

Vector of curve IDs (empty if none).

◆ get_group_edges()

def cubit.get_group_edges (*group_id*)

Return direct child edge IDs for a specified group, if any.

.. code-block:: python

```
# Create geometry and mesh
it.


cubit.cmd
("brick x 10" )


cubit.cmd
("mesh volume all" )


# Group all mesh edges on surface
1.


cubit.cmd
("group \"edge_group\" add
edge in surf 1" )


# Query group ID by name.


edge_group_id = cubit.get_id_from_name
("edge_group" )


# Get direct child edge
IDs.


edges = cubit.get_group_edges
(edge_group_id)


# Print tuple-like output.


print(tuple(edges)) # prints
something like "(1,2,3,...,n)"
```

```
@n type of group_id:  int
```

Parameters

group_id ID of the group to examine.
return type of : std::vector< int,std:: allocator< int > >

Returns

Vector of edge IDs (empty if none).

◆ get_group_groups()

def cubit.get_group_groups (*group_id*)

Retrieve direct child group IDs for a specified parent group, if any.

.. code-block:: python

```
                # Create geometry and define
                groups.

                cubit.cmd
                ("brick x 10" )

                cubit.cmd
                ("group \"vert_group\" add
                vert in vol 1" )

                cubit.cmd
                ("group 'curve_group' add
                curve in vol 1" )

                cubit.cmd
                ("group \"parent_group\"
                add vert_group curve_group" )

                # Query group IDs.

                vert_group_id = cubit.get_id_from_name
                ("vert_group" )

                curve_group_id = cubit.get_id_from_name
                ("curve_group" )

                parent_group_id =
      cubit.get_id_from_name
                ("parent_group" )

                # Get direct child group
                IDs.

                child_groups = cubit.get_group_groups
                (parent_group_id)

                # Print tuple-like output.

                print(tuple(child_groups)) #
                prints "(2,3)"
```

```
@n type of group_id:  int
```

Parameters

group_id ID of the parent group.
return type of : std::vector< int,std:: allocator< int > >

Returns

Vector of child group IDs (empty if none).

◆ get_group_hexes()

def cubit.get_group_hexes (*group_id*)

Return direct child hexahedral element IDs for a specified group, if any.

Hexahedral elements are generated on volumes using the default map scheme and meshed via: mesh volume all

```
.. code-block:: python
```

```
                    # Create and mesh geometry with
                    hex elements.

                    cubit.cmd
                    ("brick x 10" )

                    cubit.cmd
                    ("mesh volume all" )

                    # Group all hex elements in
                    volume 1.

                    cubit.cmd
                    ("group \"hex_group\" add
                    hex in vol 1" )

                    # Query group ID by name.

                    hex_group_id = cubit.get_id_from_name
                    ("hex_group" )

                    # Get direct child hex
                    IDs.

                    hexes = cubit.get_group_hexes
                    (hex_group_id)

                    # Print tuple-like output.

                    print(tuple(hexes)) # prints
                    "(1,2,3,...,n)"
```

```
 @n type of group_id:  int
```

Parameters

group_id ID of the group to examine.
return type of : std::vector< int,std:: allocator< int > >

Returns

Vector of hex IDs (empty if none).

◆ get_group_nodes()

def cubit.get_group_nodes (*group_id*)

Return direct child node IDs for a specified group, if any.

.. code-block:: python

```
# Create geometry, mesh it, and
group all nodes.


cubit.cmd
("brick x 10" )


cubit.cmd
("mesh volume all" )


cubit.cmd
("group \"node_group\" add
node in vol 1" )


# Query group ID by name.


node_group_id = cubit.get_id_from_name
("node_group" )


# Get direct child node
IDs.


nodes = cubit.get_group_nodes
(node_group_id)


# Print tuple-like output.


print(tuple(nodes)) # prints
something like "(1,2,3,...,n)"
```

```
@n type of group_id:  int
```

Parameters

group_id ID of the group to examine.
return type of : std::vector< int,std:: allocator< int > >

Returns

Vector of node IDs (empty if none).

◆ get_group_pyramids()

def cubit.get_group_pyramids (*group_id*)

Return direct child pyramid element IDs for a specified group, if any.

.. code-block:: python

```
                # Create geometry, duplicate
                volume 1 to create vol 2;

                # mesh volume 1 with default hex
                scheme and vol 2 with tet elements,

                # auto-generating pyramids at
                their interface.

                cubit.cmd
                ("brick x 10" )

                cubit.cmd
                ("vol 1 copy mov x
                10" )

                cubit.cmd
                ("mesh volume 1" )

                cubit.cmd
                ("vol 2 scheme
                tetmesh" )

                cubit.cmd
                ("mesh volume 2" )

                # Group all pyramid elements in
                vol 2.

                cubit.cmd
                ("group \"pyramid_group\"
                add pyramid in vol 2" )

                # Query group ID by name.

                pyramid_group_id =
        cubit.get_id_from_name
                ("pyramid_group" )

                # Get direct child pyramid
                IDs.

                pyramids = cubit.get_group_pyramids
                (pyramid_group_id)

                # Print tuple-like output.

                print(tuple(pyramids)) # prints
                something like "(1,2,3,...,n)"
```

```
@n type of group_id:  int
```

Parameters

group_id ID of the group to examine.
return type of : std::vector< int,std:: allocator< int > >

Returns

Vector of pyramid IDs (empty if none).

◆ get_group_quads()

def cubit.get_group_quads (*group_id*)

Return direct child quad IDs for a specified group, if any.

.. code-block:: python

```
# Create and mesh
geometry.


cubit.cmd
("brick x 10" )


cubit.cmd
("mesh volume all" )


# Group all quadrilateral faces
on surface 1.


cubit.cmd
("group \"quad_group\" add
face in surf 1" )


# Query group ID by name.


quad_group_id = cubit.get_id_from_name
("quad_group" )


# Get direct child quad
IDs.


quads = cubit.get_group_quads
(quad_group_id)


# Print tuple-like output.


print(tuple(quads)) # prints
something like "(1,2,3,...,n)"
```

```
@n type of group_id:  int
```

Parameters

group_id ID of the group to examine.
return type of : std::vector< int,std:: allocator< int > >

Returns

Vector of quad IDs (empty if none).

◆ get_group_spheres()

def cubit.get_group_spheres (*group_id*)

Return direct child sphere element IDs for a specified group, if any.

Sphere elements are generated by inserting nodes or vertices into a block using: Block <id> {node|vertex} <id_range>

Once created, spheres can be located and grouped for querying.

```
.. code-block:: python
```

```
                # Create volumes, set mesh size,
                and mesh all volumes.


                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("vol all size 5" )


                cubit.cmd
                ("mesh volume all" )


                # Create a vertex to seed sphere
                and mesh to generate node.


                cubit.cmd
                ("create vertex 0 0
                10" )


                sph_vtx_id = cubit.get_id_from_name
                ("vertex" )


                cubit.cmd
                (f"mesh vertex
                {sph_vtx_id}" )


                sph_nd = cubit.get_id_from_name
                ("node" )


                # Generate sphere elements via
                block and spider commands on surface 1.


                cubit.cmd
                ("block 1 volume 1"
                )


                cubit.cmd
                (f"block 2 vertex
                {sph_vtx_id}" )


                cubit.cmd
                (f"block 3 joint node
                {sph_nd} spider surf 1" )


                cubit.cmd
                ("locate sphere all"
                )


                cubit.cmd
                ("group \"sphere_group\"
                add sphere all" )


                # Query and print sphere group
                elements.


                sphere_group_id =
      cubit.get_id_from_name
                ("sphere_group" )


                spheres = cubit.get_group_spheres
                (sphere_group_id)


                print(tuple(spheres)) # prints
                something like "(1,2,3,...,n)"
```

```
 @n type of group_id:  int
```

Parameters

group_id ID of the group to examine.
return type of : std::vector< int,std:: allocator< int > >

Returns

Vector of sphere IDs (empty if none).

◆ get_group_surfaces()

def cubit.get_group_surfaces (*group_id*)

Return direct child surface IDs for a specified group, if any.

.. code-block:: python

```
# Create geometry (brick produces
six surfaces).


cubit.cmd
("brick x 10" )


# Define group containing all
surfaces.


cubit.cmd
("group \"surf_group\" add
surface all" )


# Query group ID by name.


surf_group_id = cubit.get_id_from_name
("surf_group" )


# Get direct child surface
IDs.


surfaces = cubit.get_group_surfaces
(surf_group_id)


# Print tuple-like output.


print(tuple(surfaces)) # prints
"(1,2,3,4,5,6)"
```

```
@n type of group_id:  int
```

Parameters

group_id ID of the group to examine.
return type of : std::vector< int,std:: allocator< int > >

Returns

Vector of surface IDs (empty if none).

◆ get_group_tets()

def cubit.get_group_tets (*group_id*)

Return direct child tetrahedron IDs for a specified group, if any.

.. code-block:: python

```
# Create geometry, set meshing
scheme, and mesh.


cubit.cmd
("brick x 10" )


cubit.cmd
("vol 1 scheme
tetmesh" )


cubit.cmd
("mesh volume all" )


# Group all tetrahedral elements
in volume 1.


cubit.cmd
("group \"tet_group\" add
tet in vol 1" )


# Query group ID by name.


tet_group_id = cubit.get_id_from_name
("tet_group" )


# Get direct child tetrahedron
IDs.


tets = cubit.get_group_tets
(tet_group_id)


# Print tuple-like output.


print(tuple(tets)) # prints
something like "(1,2,3,...,n)"
```

```
@n type of group_id:  int
```

Parameters

group_id ID of the group to examine.
return type of : std::vector< int,std:: allocator< int > >

Returns

Vector of tetrahedron IDs (empty if none).

◆ get_group_tris()

def cubit.get_group_tris (*group_id*)

Return direct child triangle IDs for a specified group, if any.

.. code-block:: python

```
# Create and mesh
geometry.


cubit.cmd
("brick x 10" )


cubit.cmd
("mesh volume all" )


# Group all triangular faces on
surface 1.


cubit.cmd
("group \"tri_group\" add
tri in surf 1" )


# Query group ID by name.


tri_group_id = cubit.get_id_from_name
("tri_group" )


# Get direct child triangle
IDs.


tris = cubit.get_group_tris
(tri_group_id)


# Print tuple-like output.


print(tuple(tris)) # prints
something like "(1,2,3,...,n)"
```

```
@n type of group_id:  int
```

Parameters

group_id ID of the group to examine.
return type of : std::vector< int,std:: allocator< int > >

Returns

Vector of triangle IDs (empty if none).

◆ get_group_vertices()

def cubit.get_group_vertices (*group_id*)

Return direct child vertex IDs for a specified group, if any.

.. code-block:: python

```
# Create geometry (brick creates
vertices).


cubit.cmd
("brick x 10" )


# Define group containing all
vertices.


cubit.cmd
("group \"vert_group\" add
vertex all" )


# Query group ID by name.


vert_group_id = cubit.get_id_from_name
("vert_group" )


# Get direct child vertex
IDs.


vertices = cubit.get_group_vertices
(vert_group_id)


# Print tuple-like output.


print(tuple(vertices)) # prints
"(1,2,3,4,5,6,7,8)"
```

```
@n type of group_id:  int
```

Parameters

group_id ID of the group to examine.
return type of : std::vector< int,std:: allocator< int > >

Returns

Vector of vertex IDs (empty if none).

◆ get_group_volumes()

def cubit.get_group_volumes (*group_id*)

Return direct child volume IDs for a specified group, if any.

.. code-block:: python

```
# Create geometry and copy
volume.

cubit.cmd
("brick x 10" )

cubit.cmd
("vol 1 copy mov x
10" )

# Define group containing all
volumes.

cubit.cmd
("group \"vol_group\" add
vol all" )

# Query group ID by name.

vol_group_id = cubit.get_id_from_name
("vol_group" )

# Get direct child volume
IDs.

volumes = cubit.get_group_volumes
(vol_group_id)

# Print tuple-like output.

print(tuple(volumes)) # prints
"(1,2)"
```

```
@n type of group_id:  int
```

Parameters

group_id ID of the group to examine.
return type of : std::vector< int,std:: allocator< int > >

Returns

Vector of volume IDs (empty if none).

◆ get_group_wedges()

def cubit.get_group_wedges (*group_id*)

Return direct child wedge element IDs for a specified group, if any.

.. code-block:: python

```
                # Create geometry, set surface
                meshing scheme to tris and sweep to
    create
                wedges.

                cubit.cmd
                ("brick x 10" )

                cubit.cmd
                ("surf 1 scheme
                trimesh" )

                cubit.cmd
                ("mesh surf 1" )

                cubit.cmd
                ("mesh volume all" )

                # Group all wedge elements in
                volume 1.

                cubit.cmd
                ("group \"wedge_group\" add
                wedge in vol 1" )

                # Query group ID by name.

                wedge_group_id = cubit.get_id_from_name
                ("wedge_group" )

                # Get direct child wedge
                IDs.

                wedges = cubit.get_group_wedges
                (wedge_group_id)

                # Print tuple-like output.

                print(tuple(wedges)) # prints
                something like "(1,2,3,...,n)"
```

```
@n type of group_id:  int
```

Parameters

group_id ID of the group to examine.
return type of : std::vector< int,std:: allocator< int > >

Returns

Vector of wedge IDs (empty if none).

◆ get_heatflux_on_area()

def cubit.get_heatflux_on_area (*bc_area_enum*, *entity_id*)

Get the heatflux value on a specified area.

type of bc_area_enum: int

Parameters

bc_area_enum CI_BCEntityTypes: 4 (solid), 7 (shell top), 8 (shell bottom).
type of entity_id: int

entity_id ID of the heatflux BC.
return type of : float

Returns

Heatflux value.

◆ get_hex_count()

def cubit.get_hex_count ( )

Retrieve the count of hexahedral elements in the current model.

.. code-block:: python

```
cubit.cmd
("brick x 10" )
# create brick (Volume ID =
1)


cubit.cmd
("mesh vol 1" )
# mesh as hexes


hex_count = cubit.get_hex_count
()


print("Hex count:" ,
hex_count)
```

```
@n return type of :  int
```

Returns

Number of hexahedral elements in the current model's mesh.

◆ get_hex_global_element_id()

def cubit.get_hex_global_element_id (*hex_id*)

Get the Global Element ID for a specific hexahedral element.

This function returns the Global Element ID assigned to the given local hex ID when the element was placed into a block.

```
.. code-block:: python
```

```
# Create geometry, mesh it, and
assign all hexes to block 1


cubit.cmd
("brick x 10" )


cubit.cmd
("mesh vol 1" )


cubit.cmd
("block 1 hex all" )


# Query the Global Element ID of
local hex 22


gid = cubit.get_hex_global_element_id
(22)


print("Global Element ID of
hex 22:" , gid)
```

```
@n type of hex_id:  int
```

Parameters

hex_id Local hexahedral element ID.
return type of : int

Returns

Global Element ID for the specified hex.

◆ get_hex_sheet()

def cubit.get_hex_sheet (*node_id_1*, *node_id_2*)

Retrieve the IDs of all hexahedral elements forming a hex sheet through two nodes.

A hex sheet is a layer of contiguous hexes. The two node IDs define an edge perpendicular to that layer.

```
.. code-block:: python
```

```
                    cubit.cmd
                    ("brick x 10" )
                    # create brick (Volume ID =
                    1)


                    cubit.cmd
                    ("mesh vol 1" )
                    # mesh volume 1 with hexes


                    # Assume nodes 5 and 6 form an
                    edge perpendicular to a hex layer


                    hex_ids = cubit.get_hex_sheet
                    (5, 6)


                    print("Hex sheet
                    IDs:" , hex_ids)
```

```
 @n type of node_id_1:  int
```

Parameters

node_id_1 First node ID defining the edge perpendicular to the hex layer.
type of node_id_2: int

node_id_2 Second node ID defining the edge perpendicular to the hex layer.
return type of : std::vector< int,std:: allocator< int > >

Returns

A list (vector<int> or Python tuple) of hex IDs forming the contiguous sheet.

◆ get_hole_surfaces()

def cubit.get_hole_surfaces (*surface_id*)

Return the surfaces in the hole adjacent to the specified surface.

The function returns a list of surface IDs that belong to the same hole as the given surface. The input surface_id must already be part of a hole. The result includes surface_id itself.

```
.. code-block:: python
```

```
                cubit.cmd
                ("brick x 10" )

                cubit.cmd
                ("cylinder rad 1 height
                10" )

                cubit.cmd
                ("vol 2 move z 5" )

                cubit.cmd
                ("subtract volume 2 from
                volume 1" )


                surf_id = 10

                hole_surfs = cubit.get_hole_surfaces
                (surf_id)


                print("Hole
                surfaces:" , tuple(hole_surfs))


                # Expected output:

                # Hole surfaces: (10, 11)
```

```
@n type of surface_id:  int
```

Parameters

surface_id **Surface** ID that is part of a hole.
return type of : std::vector< int,std:: allocator< int > >

Returns

A list of surface IDs belonging to the hole (includes surface_id itself).

◆ get_hydraulic_radius_surface_area()

def cubit.get_hydraulic_radius_surface_area (*surface_id*)

Compute the hydraulic radius of a specified surface.

Title Computation

- hydraulic_radius = 4 * (surface_area / total_edge_length), where surface_area is the face area and total_edge_length is the sum of all edge lengths bounding the face
- If total_edge_length is zero, returns 0.0

```
.. code-block:: python
```

```
                cubit.cmd
                ("brick x 10" )

                hr1 =
    cubit.get_hydraulic_radius_surface_area
                (1)

                print("Cube face hydraulic
                radius:" , hr1) # Output:
                10.0


                cubit.cmd
                ("brick x 10 y 1 z
                10" )

                hr2 =
    cubit.get_hydraulic_radius_surface_area
                (1)

                print("Skinny face
                hydraulic radius:" , hr2) # Output:
    1.81818
```

```
@n type of surface_id:  int
```

Parameters

surface_id ID of the surface to query.
return type of : float

Returns

Hydraulic radius of the specified surface (4 * area / total_edge_length), or 0.0 if no edges.

◆ get_hydraulic_radius_volume_area()

def cubit.get_hydraulic_radius_volume_area (*volume_id*)

Compute the hydraulic radius of a specified volume.

Title Computation

- hydraulic_radius = 6 * (enclosed_volume / total_surface_area)
- enclosed_volume: measured volume of the body
- total_surface_area: sum of areas of all faces
- Returns 0.0 if total_surface_area is zero

.. code-block:: python

```
                # Cube: volume = 1000, total face
                area = 600

                cubit.cmd
                ("brick x 10" )

                hr_cube =
    cubit.get_hydraulic_radius_volume_area
                (1)

                print("Cube hydraulic
                radius:" , hr_cube) #
                Output: 10.0


                # Skinny brick: volume = 100,
                total face area = 240

                cubit.cmd
                ("brick x 10 y 1 z
                10" )

                hr_skinny =
    cubit.get_hydraulic_radius_volume_area
                (1)

                print("Skinny brick
                hydraulic radius:" , hr_skinny) #
    Output: 2.5
```

@n type of volume_id: int

Parameters

volume_id ID of the volume to query.
return type of : float

Returns

Hydraulic radius = 6 * (enclosed_volume / total_surface_area); 0.0 if total_surface_area is zero.

◆ get_id_from_name()

def cubit.get_id_from_name (*name*)

Retrieve the integer ID of an entity by its name.

.. code-block:: python

```
                # Create and name a
                vertex.

                cubit.cmd
                ("vertex 22 name
                'point_load'" )

                # Query the ID using the
                name.

                entity_id = cubit.get_id_from_name
                ("point_load" )

                print("Entity ID:" ,
                entity_id) # prints "22"
```

@n type of name: string

Parameters

name Name of the entity whose ID is requested.
return type of : int

Returns

Integer ID of the named entity.

◆ get_id_string()

def cubit.get_id_string (*entity_ids*,
sort = `True`
)

Convert a list of integers into a compact Cubit-style ID range string.

Collapses consecutive IDs into "start to end" ranges and separates entries with commas. Unlike **string_from_id_list()** , this variant does not insert any newline characters.

`.. code-block:: python`

```
import cubit

ids = [3, 1, 2, 5, 6, 8]

# Default: sort then
collapse

s = cubit.get_id_string
(ids)

# s == "1 to 3, 5 to 6, 8"


# Without sorting:

s = cubit.get_id_string
(ids, False )

# s == "3, 1 to 2, 5 to 6,
8"
```

`@n type of entity_ids:  std::vector< int,std::allocator< int > >`

Parameters

entity_ids Vector of integer IDs to format.
type of sort: boolean, optional

sort If true (default), sort IDs ascending before collapsing ranges; if false, preserve the original order.
return type of : string

Returns

A single-line string representing the ID ranges (no line breaks).

◆ get_idless_signature()

def cubit.get_idless_signature (*entity_type*,

entity_id
)

Get the idless signature of a geometric or mesh entity.

This function returns the *idless signature* of the specified entity. The idless signature is a position-based and ordinal-based reference to the entity, independent of its current ID.

Idless signatures are used to create **version-independent journal files** . Because entity IDs can change between different Cubit versions or after model operations (e.g., webcuts), using the idless signature ensures that journal files will still refer to the correct entity by position and ordinal rather than ID.

Example format of an idless signature: `"volume at 3.42 5.66 6.32 ordinal 2"` or `"curve at (1 1 0 ordinal 2)"`

Common usage scenario:

- Enable `journal idless on`
- Journal commands will record using idless references (rather than entity IDs)
- Use `get_idless_signature` to retrieve the current idless reference for an entity programmatically

`.. code-block:: python`

```
# Create a brick

cubit.cmd
("brick x 10" )


# Retrieve the idless signature
of surface 1

signature = cubit.get_idless_signature
("surface" , 1)

print(f"Idless signature of
surface 1: {signature}" )
```

`@n type of entity_type:  string`

Parameters

entity_type Type of the entity (e.g. "curve", "surface", "vertex", "volume", "face", etc.).
type of entity_id: int

entity_id ID of the requested entity.
return type of : string

Returns

Idless signature of the entity (e.g. "volume at x y z ordinal n").

Notes: For full context, see the Cubit command: `journal idless {on | off | reverse}` .

◆ get_idless_signatures()

def cubit.get_idless_signatures (*entity_type, entity_id_list*)

Get the idless signatures of a range of geometric or mesh entities.

This function returns the *idless signatures* of the specified entities as a single string. The idless signature is a position-based and ordinal-based reference to each entity, independent of its current ID.

Idless signatures are used to create **version-independent journal files** . Since entity IDs can change between different Cubit versions or after model operations (e.g., webcuts), using idless signatures ensures that journal files remain valid by referring to entities by position and ordinal instead of ID.

The return string will contain one idless signature per entity, space-separated. Title Example:

- "curve at (1 1 0 ordinal 2) curve at (0 0 1 ordinal 1) ..."

```
.. code-block:: python
```

```
                # Create a brick and mesh
                it

                cubit.cmd
                ("brick x 10" )

                cubit.cmd
                ("mesh vol 1" )


                # Prepare a list of surface
                IDs

                surface_ids = [1, 2, 3]


                # Get idless signatures of
                surfaces 1, 2, 3

                signatures =
      cubit.get_idless_signatures
                ("surface" ,
                surface_ids)

                print(f"Idless signatures
                of surfaces 1-3: {signatures}" )
```

```
 @n type of entity_type:  string
```

Parameters

entity_type Type of the entities (e.g. "curve", "surface", "vertex", "volume", "face", etc.).
type of entity_id_list: std::vector< int,std::allocator< int > >

entity_id_list List of entity IDs.
return type of : string

Returns

A single string containing the idless signatures of the entities, space-separated.

Notes: For full context, see the Cubit command: `journal idless {on | off | reverse}`.

◆ get_int_sculpt_default()

def cubit.get_int_sculpt_default (*variable*)

Retrieve default integer-valued sculpt parameter.

Matches the input variable name substring to known sculpt parameters and returns its default value.

Title Available integer parameters

- NUM_PROC
- DEBUG
- NUM_LAPLACE
- MAX_OPT_SMOOTH
- MAX_PCOL
- MAX_GQ_ITERS
- DEG_ITERS
- STAIR
- CSMOOTH
- SMOOTH

```
                    * VOID_MAT, VOID
```

- SIDESETS

```
                    * HTET_METHOD, HTET
```

- TRI_MESH
- PILLOW_LAYERS

```
                    * ADAPT_TYPE, ADAPT_LEVELS,
    ADAPT_LOAD_BALANCE,
                    ADAPT_MEMORY_STATS
```

- ALIGN

```
                    * CAPTURE_SIDE, CAPTURE

                    * DEFEATURE_MIN_VOL_CELLS,
    DEFEATURE_ITERS,
                    DEFEATURE

                    * MICRO_EXPAND, MICRO_SHAVE
```

- QUIET

```
                    * WRITE_GEOM, WRITE_MBG
```

- MESHING_SCHEME
- PRINT_ONLY
- THREADS
- BBOX_EXPAND
- WEAR_METHOD
- MIN_NUM_LAYERS
- INPUT_MESH_MATERIAL

type of variable: string

Parameters

variable Name of the sculpt parameter substring.
return type of : int

Returns

Default integer value for the specified parameter.

◆ get_interface()

def cubit.get_interface (*interface_name*)

Retrieve a Cubit interface by name.

Returns a pointer to the requested CubitBaseInterface instance, or nullptr if no matching interface is registered.

```
.. code-block:: python

                import cubit

                iface = cubit.get_interface
                ("MeshInterface" )
```

if iface is not None:

use iface ...

cubit.release_interface(iface)

```
  @n type of interface_name:  string
```

Parameters

interface_name Name of the interface to retrieve.
return type of : CubitBaseInterface

Returns

Pointer to the CubitBaseInterface, or nullptr if not found.

◆ get_label_type()

def cubit.get_label_type (*entity_type*)

Gets the current label display type for a given entity type.

Queries the label display type currently associated with the specified entity type.

Valid `entity_type` values:

```
"RefVolume" ,
"RefFace" ,
"RefEdge" ,
"RefVertex" ,


"CubitHex" ,
"CubitTet" ,
"CubitFace" ,
"CubitTri" ,


"CubitEdge" ,
"CubitNode" ,
"CubitSphere"
```

Returned value corresponds to `SVUtil::LabelType` :
0=CUBIT_LABEL_NONE, 1=CUBIT_LABEL_ID,
2=CUBIT_LABEL_ELEMENT_ID, 3=CUBIT_LABEL_NAME,
4=CUBIT_LABEL_INTERVAL, 5=CUBIT_LABEL_SIZE,
6=CUBIT_LABEL_MERGE, 7=CUBIT_LABEL_IS_MERGED,
8=CUBIT_LABEL_FIRMNESS, 9=CUBIT_LABEL_SCHEME,
10=CUBIT_LABEL_NAME_ID,
11=CUBIT_LABEL_NAME_ONLY,
12=CUBIT_LABEL_SPHERE_ID

`.. code-block:: python`

```
label_type = cubit.get_label_type
("RefFace" )


print("Label type for
RefFace:" , label_type)
```

`@n type of entity_type:  string`

Parameters

entity_type **Entity** type to query.
return type of : int

Returns

Label display type (see list above).

◆ get_last_id()

def cubit.get_last_id (*entity_type*)

Get the ID of the last created entity of the given type.

This function returns the ID of the most recently created entity of the specified type.

Supported entity types include: "vertex", "curve", "surface", "volume", "body", etc.

`.. code-block:: python`

```
# Create a brick


cubit.cmd
("brick x 10" )


# Query the last created surface
ID


last_id = cubit.get_last_id
("surface" )


print(f"Last created
surface ID: {last_id}" )
```

`@n type of entity_type:  string`

Parameters

entity_type Type of the entity being queried (e.g. "surface", "curve", "volume").
return type of : int

Returns

Integer ID of the last created entity of the specified type.

◆ get_list_of_free_ref_entities()

def cubit.get_list_of_free_ref_entities (*geometry_type*)

Get all free (unattached) entities of a given geometry type.

.. code-block:: python

```
                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("create vertex 0 0
                0" )


                free_vertices =
      cubit.get_list_of_free_ref_entities
                ("vertex" )


                print(free_vertices) # prints
                (9,)
```

```
 @n type of geometry_type:  string
```

Parameters

geometry_type Specifies the geometry type ("vertex", "curve", "surface", etc.).
return type of : std::vector< int,std:: allocator< int > >

Returns

A list (vector or Python tuple) of IDs of detached entities of the specified type.

◆ get_material_name()

def cubit.get_material_name (*material_id*)

Get the name of a material or CFD media by ID.

type of material_id: int

Parameters

material_id ID of the material or media.
return type of : string

Returns

Material or media name.

◆ get_material_name_list()

def cubit.get_material_name_list ( )

Get list of all material names.

return type of : std::vector< std::string,std::allocator< std:: string > >

Returns

List of material names.

◆ get_material_property()

def cubit.get_material_property (*material_property_enum*, *entity_id*)

Get the value of a material property.

type of material_property_enum: int

Parameters

material_property_enum Property type (e.g., 0 = Elastic Modulus, 1 = Shear Modulus, etc.).
type of entity_id: int

entity_id ID of the material.
return type of : float

Returns

Value of the specified material property.

◆ get_media_name_list()

def cubit.get_media_name_list ( )

Get list of all CFD media names.

return type of : std::vector< std::string,std::allocator< std:: string > >

Returns
List of media names.

◆ get_media_property()

def cubit.get_media_property (*entity_id*)

Get the media classification of a material.

type of entity_id: int

Parameters
entity_id ID of the media.
return type of : int

Returns
: Media type 0 = FLUID, 1 = POROUS, 2 = SOLID.

◆ get_merge_setting()

def cubit.get_merge_setting (*geometry_type*, *entity_id*)

Get merge mode ("on", "off", or "auto") for a given entity.

By disabling merge on an entity, adjacent geometry remains separate-preserving material boundaries, distinct mesh regions, and contact interfaces.

.. code-block:: python

```
cubit.cmd
("brick x 10" )
# create first brick (vol 1,
surfaces 1-6)


cubit.cmd
("brick x 10" )
# create second brick (vol 2,
surfaces 7-12)


cubit.cmd
("vol 2 move x 10" )
# move second brick so surfaces 6
and 10 touch


cubit.cmd
("surface 6 merge
off" ) # prevent merging
on surface 6


cubit.cmd
("merge all" )
# attempt to merge; volumes
remain separate


mode = cubit.get_merge_setting
("surface" , 6)
# returns "off"


print("Merge
setting:" , mode) # prints
"off"
```

Parameters
geom_type Geometry type ("body", "volume", "surface", "curve", or "vertex").
type of entity_id: int
entity_id ID of the entity.
return type of : string

Returns
: The merge setting "on", "off", or "auto".

◆ get_merge_tolerance()

def cubit.get_merge_tolerance ( )

Return the current merge tolerance value.

The merge tolerance is an absolute distance value used to determine geometric correspondence when merging entities. Vertices, curves, and surfaces are compared using spatial checks within this tolerance. Entities closer than the merge tolerance are considered equivalent and may be merged, even if their parameterizations differ.

The default merge tolerance in Cubit is 5.0e-4. The lower limit is 1.0e-6. There is no defined upper limit.

```
.. code-block:: python
```

```
                # Set the merge tolerance using a
                Cubit command:


                cubit.cmd
                ("merge tolerance
                0.005" )


                # Get the current merge
                tolerance:


                tol = cubit.get_merge_tolerance
                ()


                print("Current merge
                tolerance:" , tol)


                # Expected output
                (example):


                # Current merge tolerance:
                0.005
```

```
 @n return type of :  float
```

Returns
The current merge tolerance value.

◆ get_mergeable_curves()

def cubit.get_mergeable_curves (*target_volume_ids*)

Get the list of mergeable curves from a list of volumes or bodies.

Given a list of volume IDs, this function returns a list of lists of potentially mergeable curves. Each inner list contains the curve IDs of one set of curves that can be merged. Each set may contain more than two curves.

Curves are considered mergeable if they are within the current merge tolerance. If not explicitly set, the default merge tolerance is 1e-6.

In Python, the lists will be returned as Python tuples.

.. code-block:: python

```
cubit.cmd
('brick x 10' )
# Volume 1


cubit.cmd
('brick x 10' )
# Volume 2


cubit.cmd
('vol 2 mov x 10' )
# Move volume 2 exactly adjacent
to volume 1


target_volumes = [1, 2]


mergeable_curves = cubit.get_mergeable_curves
(target_volumes)


print("Mergeable
curves:" )
```

for group in mergeable_curves: print(group)

```
# Expected output:

# (1, 15)

# (5, 19)

# (10, 21)

# (12, 23)
```

@n type of target_volume_ids: std::vector< int,std::allocator< int > >

Parameters

target_volume_ids List of volume IDs to examine.
return type of : std::vector< std::vector< int,std::allocator< int > >,std::allocator< std::vector< int,std:: allocator< int > > > >

Returns

List of lists of mergeable curve IDs (potentially more than a pair). In Python, each inner list is returned as a tuple.

◆ get_mergeable_surfaces()

def cubit.get_mergeable_surfaces (*target_volume_ids*)

Get the list of mergeable surfaces from a list of volumes or bodies.

Given a list of volume IDs, this function returns a list of lists of potentially mergeable surfaces. Each inner list contains the surface IDs of one set of surfaces that can be merged. Each set may contain more than two surfaces.

Surfaces are considered mergeable if they are within the current merge tolerance. If not explicitly set, the default merge tolerance is 1e-6.

In Python, the lists will be returned as Python tuples.

.. code-block:: python

```
cubit.cmd
('brick x 10' )
# Volume 1


cubit.cmd
('brick x 10' )
# Volume 2


cubit.cmd
('vol 2 mov x 10' )
# Move volume 2 exactly adjacent
to volume 1


target_volumes = [1, 2]


mergeable_surfaces = cubit.get_mergeable_surfaces
(target_volumes)


print("Mergeable
surfaces:" )
```

for group in mergeable_surfaces: print(group)

```
# Expected output:


# (6, 10)
```

@n type of target_volume_ids: std::vector< int,std::allocator< int > >

Parameters

target_volume_ids List of volume IDs to examine.
return type of : std::vector< std::vector< int,std::allocator< int > >,std::allocator< std::vector< int,std:: allocator< int > > > >

Returns

List of lists of mergeable surface IDs (potentially more than a pair). In Python, each inner list is returned as a tuple.

◆ get_mergeable_vertices()

def cubit.get_mergeable_vertices (*target_volume_ids*)

Get the list of mergeable vertices from a list of volumes or bodies.

Given a list of volume IDs, this function returns a list of lists of potentially mergeable vertices. Each inner list contains the vertex IDs of one set of vertices that can be merged. Each set may contain more than two vertices.

Vertices are considered mergeable if they are within the current merge tolerance. If not explicitly set, the default merge tolerance is 1e-6.

In Python, the lists will be returned as Python tuples.

```
.. code-block:: python
```

```
                cubit.cmd
                ('brick x 10' )
                # Volume 1


                cubit.cmd
                ('brick x 10' )
                # Volume 2


                cubit.cmd
                ('vol 2 mov x 10' )
                # Move volume 2 exactly adjacent
                to volume 1


                target_volumes = [1, 2]


                mergeable_vertices = cubit.get_mergeable_vertices
                (target_volumes)


                print("Mergeable
                vertices:" )
```

for group in mergeable_vertices: print(group)

```
                # Expected output:

                # (1, 12)

                # (2, 11)

                # (5, 16)

                # (6, 15)
```

```
@n type of target_volume_ids:  std::vector< int,std::allocator< int > >
```

Parameters

target_volume_ids List of volume IDs to examine.
return type of : std::vector< std::vector< int,std::allocator< int > >,std::allocator< std::vector< int,std:: allocator< int > > > >

Returns

List of lists of mergeable vertex IDs (potentially more than a pair). In Python, each inner list is returned as a tuple.

◆ get_mesh_edge_length()

def cubit.get_mesh_edge_length (*edge_id*)

Compute the length of a specified mesh edge.

Returns the geometric length, in model units, of the mesh edge with the given ID.

```
.. code-block:: python
```

```
                import cubit

                # Get the length of edge
                10

                length = cubit.get_mesh_edge_length
                (10)

                print("Edge 10
                length:" , length)
```

```
@n type of edge_id:  int
```

Parameters

edge_id ID of the mesh edge whose length is requested.
return type of : float

Returns

The length of the specified mesh edge.

◆ get_mesh_element_type()

def cubit.get_mesh_element_type (*entity_type*, *entity_id*)

Get the mesh element type applied to a geometric entity.

Returns the type of elements used to mesh the specified geometry. Possible return values: "hex", "tet", "pyramid", "wedge", "face" (quad), "tri", "edge", or "node". Returns an empty string if the entity has not been meshed.

.. code-block:: python

```
                # Create a block and mesh volume
                1 with tets

                cubit.cmd
                ("brick x 10" )

                cubit.cmd
                ("vol all scheme
                tetmesh" )

                cubit.cmd
                ("mesh vol 1" )


                # Query the mesh type on surface
                1

                mesh_type = cubit.get_mesh_element_type
                ("surface" , 1)

                print(f"Mesh element type
                on surface 1: {mesh_type}" ) # prints
    "tri"
```

@n type of entity_type: string

Parameters

entity_type Type of the geometry ("surface", "curve", "volume", etc.).
type of entity_id: int

entity_id ID of the geometry entity.
return type of : string

Returns

Name of the mesh element type on that entity, or empty if none.

◆ get_mesh_error_count()

def cubit.get_mesh_error_count ( )

◆ get_mesh_error_solutions()

def cubit.get_mesh_error_solutions (*error_code*)

Retrieve recommended solutions and context cues for a mesh error.

Given a mesh error code, returns a triplet of strings: 1) Solution text describing how to correct the error. 2) Help context cue for detailed guidance. 3) Command-panel cue suggesting the UI panel or command to use.

```
.. code-block:: python
```

```
                    import cubit

                    # Lookup mesh error solutions for
                    code 102

                    cues = cubit.get_mesh_error_solutions
                    (102)

                    print("Solution: " ,
                    cues[0])

                    print("Help context:
                    " , cues[1])

                    print("Command panel:
                    " , cues[2])
```

```
 @n type of error_code:  int
```

Parameters

error_code Integer mesh error code for which to retrieve guidance.
return type of : std::vector< std::string,std::allocator< std:: string > >

Returns

Vector of three strings: [0] solution text, [1] help context cue, [2] command-panel cue.

◆ get_mesh_errors()

def cubit.get_mesh_errors ( )

◆ get_mesh_geometry_approximation_angle()

def cubit.get_mesh_geometry_approximation_angle (*geometry_type, entity_id*)

Get the geometry approximation angle for TriMesh/TetMesh on a given entity.

Returns the maximum deviation angle (in degrees) used to approximate curved CAD surfaces when meshing. A smaller angle yields more finely triangulated surfaces. The value may be explicitly set on a curve, surface, or volume, or computed from adjacent geometry if not user-set.

```
.. code-block:: python
```

```
                # 1) Set geometry approximation
                angle to 10 degrees on volume 5.

                cubit.cmd
                ("volume 5 scheme tetmesh
                geom approximation angle 10" )

                # 2) Query the geometry
                approximation angle for volume 5.

                angle =
      cubit.get_mesh_geometry_approximation_angle
                ("volume" , 5)

                print(f"Volume 5 geometry
                approximation angle: {angle} degrees" )
                # Expected: 10
```

```
@n type of geometry_type:  string
```

Parameters

geometry_type Geometry type ("curve", "surface", or "volume").
type of entity_id: int

entity_id ID of the entity whose approximation angle is queried.
return type of : float

Returns

Geometry approximation angle in degrees (double); default is 15 if not set.

◆ get_mesh_group_parent_ids()

def cubit.get_mesh_group_parent_ids (*element_type, element_id*)

Retrieve parent group IDs for a specific mesh element.

Returns IDs of all groups containing the given mesh element (excluding pick group).

```
.. code-block:: python
```

```
                # Fetch groups that contain
                triangle element 332.

                parent_id_list =
      cubit.get_mesh_group_parent_ids
                ("tri" , 332)

                print("Parent groups for
                tri 332:" , parent_id_list)
```

```
@n type of element_type:  string
```

Parameters

element_type Mesh element type (e.g., "node", "edge", "face"/"tri"/"quad", "volume"/"tet"/"hex").
type of element_id: int

element_id ID of the mesh element.
return type of : std::vector< int,std:: allocator< int > >

Returns

Vector of group IDs containing this element.

◆ get_mesh_interval_firmness()

def cubit.get_mesh_interval_firmness (*geometry_type*, *entity_id*)

Retrieve the interval firmness for a specified geometry entity.

Interval firmness determines whether a curve's interval count or size can be modified by other commands. Title Possible return values:

- "HARD" - interval is locked (user-set directly via hard command).
- "SOFT" - interval is a goal (may be adjusted by matching or other commands).
- "LIMP" - no interval was explicitly set (default state).

```
.. code-block:: python
```

```
                # 1) Create two vertices and a
                linear curve between them (Curve ID =
    1).


                cubit.cmd
                ("create vertex 0 0
                0" ) # Vertex ID =
                1


                cubit.cmd
                ("create vertex 1 0
                0" ) # Vertex ID =
                2


                cubit.cmd
                ("create curve vertex 1
                2" ) # Curve ID = 1


                # 2) By default, interval
                firmness is LIMP.


                firmness =
    cubit.get_mesh_interval_firmness
                ("curve" , 1)


                print("Curve 1 interval
                firmness:" , firmness) #
                Expected: "LIMP"


                # 3) Set a "soft" interval count
                (upgrades firmness to SOFT).


                cubit.cmd
                ("curve 1 interval count
                10" )


                firmness =
    cubit.get_mesh_interval_firmness
                ("curve" , 1)


                print("Curve 1 interval
                firmness:" , firmness) #
                Expected: "SOFT"


                # 4) Lock the interval with a
                "hard" command.


                cubit.cmd
                ("curve 1 interval
                hard" )


                firmness =
    cubit.get_mesh_interval_firmness
                ("curve" , 1)


                print("Curve 1 interval
                firmness:" , firmness) #
                Expected: "HARD"
```

```
 @n type of geometry_type:  string
```

Parameters

geometry_type Geometry type of the entity ("curve", "surface", or "volume").

Note

interval firmness applies to curves; "surface" and "volume" queries return combined firmness of contained curves.
type of entity_id: int

Parameters

entity_id ID of the entity whose interval firmness is queried.
return type of : string

Returns

: The current interval firmness "HARD", "SOFT", or "LIMP".

◆ get_mesh_intervals()

def cubit.get_mesh_intervals (*geometry_type*, *entity_id*)

Get the mesh interval count for a specified entity.

Title Retrieves the number of mesh intervals (curve subdivisions) defined on a geometry entity.

- For curves, this is the number of mesh edges along that curve.
- For surfaces or volumes, it is the default interval count that their boundary curves inherit if no curve-level setting exists.
- Curve-level interval settings always override surface- or volume-level settings.

.. code-block:: python

```
import cubit

curve_int = cubit.get_mesh_intervals
("curve" , 5)

surf_int = cubit.get_mesh_intervals
("surface" , 12)
```

@n type of geometry_type: string

Parameters

geometry_type **Entity** type: "curve", "surface", or "volume".
type of entity_id: int

entity_id ID of the geometry entity.
return type of : int

Returns

Mesh interval count on the curve, or inherited count for surface or volume (overridden by any curve-level setting).

◆ get_mesh_scheme()

def cubit.get_mesh_scheme (*geometry_type*, *entity_id*)

Retrieve the meshing scheme applied to a geometric entity.

Returns the name of the scheme currently set on the specified entity. This can be called at any time after a scheme has been assigned (e.g., via `cubit.cmd("vol 1 scheme sweep")`) to confirm which scheme is in use.

.. code-block:: python

```
cubit.cmd
('surface 12 scheme
pave' )

scheme = cubit.get_mesh_scheme
("surface" , 12)

print("Surface 12 uses
scheme:" , scheme)
```

@n type of geometry_type: string

Parameters

geometry_type Type of geometry ("curve", "surface", "volume").
type of entity_id: int

entity_id ID of the entity whose scheme is queried.
return type of : string

Returns

The meshing scheme name currently applied (e.g., "sweep", "bias", "tetmesh").

◆ get_mesh_scheme_firmness()

def cubit.get_mesh_scheme_firmness (*geometry_type*, *entity_id*)

Retrieve the meshing scheme firmness for a specified surface or volume.

Scheme firmness controls whether a user-assigned scheme can be overridden by automatic selection. Valid firmness values are "Default" (automatic selection allowed), "Soft" (automatic selection preferred), and "Hard" (scheme locked and not changed by automatic selection). Only "surface" and "volume" entities support scheme firmness.

.. code-block:: python

```
                # Assume surface 12 already
                exists and has a meshing scheme
      assigned.

                # 1) Lock the surface's scheme by
                setting firmness to "Hard":

                cubit.cmd
                ("surface 12 scheme
                hard" )

                # 2) Query the firmness for
                surface 12:

                firmness =
      cubit.get_mesh_scheme_firmness
                ("surface" , 12)

                print("Surface 12
                firmness:" , firmness) #
                Expected: "Hard"
```

@n type of geometry_type: string

Parameters

geometry_type Geometry type ("surface" or "volume").
type of entity_id: int

entity_id ID of the entity whose scheme firmness is queried.
return type of : string

Returns

The current scheme firmness ("Default", "Soft", or "Hard").

◆ get_mesh_size()

def cubit.get_mesh_size (*geometry_type*, *entity_id*)

Retrieve the effective target edge length for meshing an entity.

Returns the mesh size (in model units) used when generating 2D or 3D elements.

- If no size is set on the entity, it inherits the size from its parent.
- If no size is set anywhere in the hierarchy, the default automatic size is used.

.. code-block:: python

```
                import cubit

                size = cubit.get_mesh_size
                ("curve" , 7)

                print("Mesh size on curve
                7:" , size)
```

@n type of geometry_type: string

Parameters

geometry_type **Entity** type: "curve", "surface", or "volume".
type of entity_id: int

entity_id ID of the entity to query.
return type of : float

Returns

Effective mesh edge length (inherited or default auto size).

◆ get_mesh_size_type()

def cubit.get_mesh_size_type (*geometry_type*, *entity_id*)

Retrieve the mesh size setting type for a specified entity.

Unlike **get_mesh_interval_firmness()** , which returns the firmness of an interval count (number of edges), this function returns the status of a target edge length ("mesh size") setting. The returned value may reflect direct user input, automatic calculation from connected geometry, or lack of any setting.

Title Possible return values are:

- "USER_SET" - The mesh size (edge length) was explicitly set by the user on this entity.
- "CALCULATED" - The mesh size was inferred or computed based on adjacent geometry or default rules.
- "NOT_SET" - No mesh size has been set, either by user or by calculation.

```
.. code-block:: python
```

```
                # 1) Create two vertices and a
                linear curve between them (Curve ID =
    1).


                cubit.cmd
                ("create vertex 0 0
                0" ) # Vertex ID =
                1


                cubit.cmd
                ("create vertex 1 0
                0" ) # Vertex ID =
                2


                cubit.cmd
                ("create curve vertex 1
                2" ) # Curve ID = 1


                # 2) Initially, no mesh size is
                set, so status is NOT_SET.


                size_type = cubit.get_mesh_size_type
                ("curve" , 1)


                print("Curve 1 mesh size
                type:" , size_type) #
                Expected: "NOT_SET"


                # 3) Explicitly set a mesh size
                (target edge length) of 0.1 on curve 1
    and mesh
                it.


                cubit.cmd
                ("curve 1 size 0.1"
                )


                cubit.cmd
                ("mesh curve 1" )


                size_type = cubit.get_mesh_size_type
                ("curve" , 1)


                print("Curve 1 mesh size
                type:" , size_type) #
                Expected: "USER_SET"


                # 4) If a parent surface or
                automatic process assigns a size,
    querying the curve
                may yield "CALCULATED".
```

```
 @n type of geometry_type:  string
```

Parameters

geometry_type	Geometry type ("curve", "surface", or "volume"). type of entity_id: int
entity_id	ID of the entity whose mesh size type is queried. return type of : string

Returns

: The entity's mesh size status "USER_SET", "CALCULATED", or "NOT_SET".

◆ get_meshed_volume_or_area()

def cubit.get_meshed_volume_or_area (*geometry_type*, *entity_ids*)

Sum mesh volumes or surface areas for CAD entities or mesh elements.

- If geometry_type is "volume" or "surface", sums the volumes or areas of all mesh elements attached to those entities.
- If geometry_type is "hex", "tet", "tri", or "quad", sums volumes (hex,tet) or areas (tri,quad) of the specified mesh element IDs.

```
.. code-block:: python
```

```
                import cubit

                vol = cubit.get_meshed_volume_or_area
                ("volume" , [1])

                area = cubit.get_meshed_volume_or_area
                ("tri" , [10, 11])
```

```
 @n type of geometry_type:  string
```

Parameters

geometry_type "volume","surface","hex","tet","tri", or "quad".
type of entity_ids: std::vector< int,std::allocator< int > >

entity_ids Vector of IDs for the specified type.
return type of : float

Returns

Total mesh volume (3D) or area (2D).

◆ get_meshgems_version()

def cubit.get_meshgems_version ( )

Retrieve the MeshGems library version.

Returns the version string of the MeshGems library used by Cubit.

```
.. code-block:: python
```

```
                import cubit

                meshgems_version =
      cubit.get_meshgems_version
                ()

                print("MeshGems
                version:" , meshgems_version)
```

```
 @n return type of :  string
```

Returns

A string containing the MeshGems library version.

◆ get_ML_classification()

def cubit.get_ML_classification (*geom_type*, *ent_id*)

Get the top classification label for a single volume or surface.

This function performs the full classification workflow, equivalent to calling **get_ML_operation_features()** followed by **get_ML_predictions()** . It uses the appropriate model ("volume_no_op" for volumes or "classify_surface" for surfaces) and returns the category with the highest predicted probability.

type of geom_type: string

Parameters

geom_type Geometry type indicator: "volume" or "surface".
type of ent_id: int

ent_id ID of the volume or surface entity to classify.
return type of : string

Returns

String label of the highest-confidence category.

```
   .. code-block:: python
```

```
                classification =
      cubit.get_ML_classification
                ("surface" , 20)

                print("Predicted
                classification:" , classification)
```

◆ get_ML_classification_categories()

def cubit.get_ML_classification_categories (*geom_type*)

Retrieve all available classification categories for a geometry type.

This function queries the ML system for every label used to classify volumes or surfaces. It can also accept the specific ML model type, such as "volume_no_op" or "surface_classification".

type of geom_type: string

Parameters

geom_type Geometry type ("volume" or "surface"). Also supports the ML model type, e.g., "volume_no_op" or "surface_classification".
return type of : std::vector< std::string,std::allocator< std:: string > >

Returns

Vector of category names.

```
.. code-block:: python

        categories =
    cubit.get_ML_classification_categories
        ("surface" )

        print("Classification
        categories for surface:" , categories)
```

◆ get_ML_classification_models()

def cubit.get_ML_classification_models ( )

Retrieve the list of supported classification ML model names.

Queries the ML subsystem for all available classification models used for labeling or categorization.

return type of : std::vector< std::string,std::allocator< std:: string > >

Returns

Vector of classification model names; empty if none are registered.

```
.. code-block:: python

        models =
    cubit.get_ML_classification_models
        ()

        print("Available
        classification models:" , models)
```

◆ get_ML_classifications()

def cubit.get_ML_classifications (*geom_type*,

ent_ids

)

Classify multiple volumes or surfaces in a single batch operation.

Perform classification on a collection of entities at once to improve efficiency over repeated individual calls.

type of geom_type: string

Parameters

geom_type Geometry type to classify ("volume" or "surface").
type of ent_ids: std::vector< size_t,std::allocator< size_t > >

ent_ids Vector of entity IDs (volumes or surfaces) to classify.
return type of : std::vector< std::string,std::allocator< std:: string > >

Returns

Vector of predicted category labels, in the same order as ent_ids.

◆ get_ML_feature_distance()

def cubit.get_ML_feature_distance (*op_name*,
f1,
f2
)

Compute the weighted distance between two feature vectors.

Applies operation-specific importance weights derived from an ensemble of decision trees to each feature, then calculates the Euclidean distance between the weighted vectors. The distance indicates the geometric or topological similarity between the two entities, where a distance of zero signifies an exact match.

type of op_name: string

Parameters

op_name ML operation name (see **get_ML_operation_features()**).
type of f1: std::vector< double,std::allocator< double > >

f1 First feature vector; must match f2 in length.
type of f2: std::vector< double,std::allocator< double > >

f2 Second feature vector; must match f1 in length.
return type of : float

Returns

Weighted distance between the two feature vectors. A distance of zero indicates an exact match between the entities.

.. code-block:: python

```
                # Retrieve features for two
                entities

                op_names = ['surface_no_op' ]

                entity1_ids = [5, 10] # Two
                entities

                entity2_ids = [0, 0] # None for
                surface_no_op

                params = [[-1, -1, -1]]

                mesh_size = 1.5

                features =
    cubit.get_ML_operation_features
                (

                op_names, entity1_ids, entity2_ids,
    params, mesh_size)


                # Compute the weighted distance
                between the features of the two
    entities

                distance =
    cubit.get_ML_feature_distance
                ("surface_no_op" ,
                features[0], features[1])

                print("Weighted
                distance:" , distance)
```

◆ get_ML_feature_importances()

def cubit.get_ML_feature_importances (*op_name*)

Retrieve feature importance scores for a given ML operation.

Queries the trained model to obtain the relative importance of each feature used in the specified operation.

type of op_name: string

Parameters

op_name ML operation identifier (e.g., "remove_cone"). return type of : std::vector< double,std:: allocator< double > >

Returns

Vector of importance values for each feature, in the same order as **get_ML_operation_features()**.

```
.. code-block:: python
```

```
                feature_names =
    cubit.get_ML_operation_feature_names
                ("remove_cone" )


                importances =
    cubit.get_ML_feature_importances
                ("remove_cone" )


                print("Feature importances
                for remove_cone:" )
```

for i in range(len(feature_names)): print(f" {feature_names[i]}\t{importances[i]}")

◆ get_ML_features()

def cubit.get_ML_features (*ml_op_names*,
entity1_ids,
entity2_ids,
params,
mesh_size,
reduced_features = `False`
)

Compute ML feature vectors for operations permitting multiple entities.

This is a batch variant of **get_ML_operation_features()** that accepts lists of IDs per operation, enabling feature extraction across multiple entities at once.

type of ml_op_names: std::vector< std::string,std::allocator< std::string > >

Parameters

ml_op_names Vector of ML operation/model names.
type of entity1_ids: std::vector< std::vector< size_t,std::allocator< size_t > >,std::allocator< std::vector< size_t,std::allocator< size_t > > > >

entity1_ids Vector of vectors of primary entity IDs per operation.
type of entity2_ids: std::vector< std::vector< size_t,std::allocator< size_t > >,std::allocator< std::vector< size_t,std::allocator< size_t > > > >

entity2_ids Vector of vectors of secondary entity IDs per operation.
type of params: std::vector< std::vector< double,std::allocator< double > >,std::allocator< std::vector< double,std::allocator< double > > > >

params Vector of parameter lists for each operation.
type of mesh_size: float

mesh_size Target mesh size for feature calculations.
type of reduced_features: boolean, optional

reduced_features If true (where supported), compute a reduced feature set for faster performance.
return type of : std::vector< std::vector< double,std::allocator< double > >,std::allocator< std::vector< double,std::allocator< double > > > >

Returns

Outer vector size == ml_op_names.size(); each inner vector contains the feature values for that operation.

.. code-block:: python

```
ml_op_names = ['surface_no_op' , 'surface_no_op' ]

entity1_ids = [[20, 25], [30, 35]]

entity2_ids = [[0, 0], [0, 0]]

params = [[-1, -1, -1], [-1, -1, -1]]

mesh_size = 1.5924

features = cubit.get_ML_features
(

ml_op_names, entity1_ids, entity2_ids, params,
mesh_size)
```

◆ get_ML_initialize_surviving_entities()

def cubit.get_ML_initialize_surviving_entities (*op_name*, *ent1*, *ent2*)

Capture entities for survival tracking before a CAD operation.

This function sets a baseline for CAD operation models that measure mesh quality before and after the operation. It is used with **get_ML_surviving_entities()** to identify entities that replace or correspond to the originals after the operation.

type of op_name: string

Parameters

op_name ML operation name (e.g., "remove_cone").
type of ent1: std::vector< int,std::allocator< int > >

ent1 List of primary entity IDs involved in the operation.
type of ent2: std::vector< int,std::allocator< int > >

ent2 List of secondary entity IDs involved in the operation.
return type of : std::vector< int,std:: allocator< int > >

Returns

Vector of entity IDs for survival tracking.

.. code-block:: python

```
ent1 = [5]

ent2 = [0]

survivors =
cubit.get_ML_initialize_surviving_entities
("remove_cone" ,
ent1, ent2)

print("Surviving entity
ID:" , survivors[0])
```

◆ get_ML_model_ID()

def cubit.get_ML_model_ID (*model_name*)

Retrieve the unique numeric ID for a given ML model or operation name.

Looks up the internal mapping from a model or operation name to its assigned ID.

type of model_name: string

Parameters

model_name Name of the ML model or operation to query.
return type of : int

Returns

Positive integer ID on success; 0 if not found or on error.

.. code-block:: python

```
model_id = cubit.get_ML_model_ID
("surface_no_op" )

print("Model ID for
surface_no_op:" , model_id)
```

◆ get_ML_model_name()

def cubit.get_ML_model_name (*model_ID*)

Retrieve the name of an ML model based on its ID.

This function returns the name of the ML model corresponding to the provided model ID. The model name is used to identify a specific machine learning model that can be used for various operations such as classification or prediction.

type of model_ID: int

Parameters

model_ID The unique identifier for the ML model.
return type of : string

Returns

The name of the ML model corresponding to the provided ID.

```
.. code-block:: python

                model_ID = 1


                model_name = cubit.get_ML_model_name
                (model_ID)


                print("Model name for
                ID" , model_ID, ":" , model_name)
```

◆ get_ML_operation()

def cubit.get_ML_operation (*op_name*,
entity_id1,
entity_id2,
params,
small_curve_size,
mesh_size
)

Generate the command, display label, and preview text for an ML operation.

Constructs three strings for the specified operation and entities:

1. the Cubit command to execute the operation,
2. a human-readable display label,
3. a preview description.

type of op_name: string

Parameters

op_name	Name of the ML operation (see get_ML_operation_features). type of entity_id1: int
entity_id1	ID of the primary entity involved in the operation. type of entity_id2: int
entity_id2	ID of the secondary entity involved in the operation. type of params: std::vector< double,std::allocator< double > >
params	Optional vector of parameter values for the operation. type of small_curve_size: float
small_curve_size	Threshold below which a curve is considered "small." Used for tailoring command generation. type of mesh_size: float
mesh_size	Target mesh size parameter for the operation. return type of : std::vector< std::string,std::allocator< std:: string > >

Returns

A vector of three strings:

- [0] = execution command,
- [1] = display label,
- [2] = preview text.

.. code-block:: python

```
operation = cubit.get_ML_operation
(

"surface_no_op" , 5,
0, [-1, -1, -1], 0.1, 1.5)

print("Command:" ,
operation[0])

print("Label:" ,
operation[1])

print("Preview:" ,
operation[2])
```

◆ get_ML_operation_feature_names()

def cubit.get_ML_operation_feature_names (*ml_op_name*, *reduced_features* = `False`)

Retrieve the list of feature names for a specified ML operation.

This function returns a vector of strings naming each feature in the operation's input vector.

type of ml_op_name: string

Parameters

ml_op_name Name of the ML operation or model (as defined by **get_ML_operation_features()**).
type of reduced_features: boolean, optional

reduced_features If true (currently only for "volume_no_op"), return the reduced set of feature names (9) instead of the full set (46) for faster inference.

`@n return type of : std::vector< std::string,std::allocator< std:: string > >`

Returns

Vector of feature name strings, in the same order as the corresponding feature values.

`.. code-block:: python`

```
# print names of all features
with their type and feature value for
surface_no_op


feature_names = cubit.get_ML_operation_feature_names
("surface_no_op" )


feature_types = cubit.get_ML_operation_feature_types
("surface_no_op" )


features = cubit.get_ML_operation_features
(['surface_no_op' ],
[[20]], [[0]], [[-1, -1, -1]], 1.5)


# Print the feature
details


print("Feature details for
surface_no_op:" )
```

for i in range(len(feature_names)): print(f"{i}\t{feature_names[i]}\t{feature_types[i]}\t{features[0][i]}")

◆ get_ML_operation_feature_size()

def cubit.get_ML_operation_feature_size (*ml_op_name*, *reduced_features* = `False`)

Retrieve the expected feature-vector length for a given ML operation.

For the specified operation (as described by **get_ML_operation_features()**), this function returns the number of features that the model expects as input.

type of ml_op_name: string

Parameters

ml_op_name Name of the ML operation or model.
type of reduced_features: boolean, optional

reduced_features If true (currently only supported for "volume_no_op"), returns the reduced feature count (9) instead of the full feature set (46) for more efficient predictions.
return type of : int

Returns

Number of features expected by the operation.

`.. code-block:: python`

```
feature_size =
cubit.get_ML_operation_feature_size
("surface_no_op" )


print("Expected feature
size for surface_no_op:" ,
feature_size)
```

◆ get_ML_operation_feature_types()

def
cubit.get_ML_operation_feature_types (*ml_op_name*,
reduced_features = `False`
)

Get the data-type descriptor for each feature of a specified ML operation.

This function returns a list of strings indicating the type of data expected for each feature in the operation's input vector. The types can be:

- "boolean" (binary flags: 0 or 1)
- "categorical" (integer codes representing discrete categories, e.g., surface type)
- "continuous" (numeric values spanning a range, e.g., curve count or surface area)

type of ml_op_name: string

Parameters

ml_op_name Name of the ML model or operation (see **get_ML_operation_features()**).
type of reduced_features: boolean, optional

reduced_features If true (currently only for "volume_no_op"), returns types for the reduced feature set (9) instead of the full set (46).
return type of : std::vector< std::string,std::allocator< std:: string > >

Returns

Vector of data-type strings matching the order of the feature values.

.. code-block:: python

```
feature_types =
cubit.get_ML_operation_feature_types
("surface_no_op" )


print("Feature types for
surface_no_op:" , feature_types)
```

◆ get_ML_operation_features()

```
def cubit.get_ML_operation_features (  ml_op_names,
                                       entity1_ids,
                                       entity2_ids,
                                       params,
                                       mesh_size,
                                       reduced_features = False
                                    )
```

Compute ML feature vectors for a list of Cubit operations.

This function computes machine learning feature vectors for a list of Cubit operations based on specified parameters, including entity IDs and target mesh size. It returns the computed feature vectors for each operation.

.. code-block:: python

```
ml_op_names = ['surface_no_op' , 'surface_no_op' ]

entity1_ids = [20, 25] # surface
IDs

entity2_ids = [0, 0] # none for
surface_no_op

params = [[-1, -1, -1], [-1, -1, -1]]

mesh_size = 1.5924

features = cubit.get_ML_operation_features
(

ml_op_names, entity1_ids, entity2_ids, params,
mesh_size)
```

Supported operations:

.. code-block:: c++

```
ID | Type | #Labels | Name                    | Entity1 | Entity2  | Params
-------------------------------------------------------------------------
1 |  R   |   3     | vertex_no_op            | vertex  | none     |
2 |  R   |   3     | curve_no_op             | curve   | none     |
3 |  R   |   3     | surface_no_op           | surface | none     |
4 |  C   |   1     | volume_no_op            | volume  | none     |
5 |  R   |   3     | remove_surface          | surface | none     |
6 |  R   |   3     | tweak_replace_surface   | surface | surface  |
7 |  R   |   3     | composite_surfaces      | surface | surface  |
8 |  R   |   3     | collapse_curve          | curve   | vertex   |
9 |  R   |   3     | remove_topology_curve   | curve   | curve    |
10 |  R   |   3     | virtual_collapse_curve  | curve   | vertex   |
11 |  R   |   3     | remove_topology_surface | surface | surface  |
```

```
12 | R | 3 | blunt_tangency | vertex | none |
remove_mat, angle, depth
```

13 | R | 3 | remove_cone | surface | none |

```
14 | R | 3 | collapse_angle | vertex | none |
real_split, angle, comp_vertex
```

15 | R | 3 | remove_blend | surface | none | 16 | R | 3 | remove_cavity | surface | none |

```
17 | R | 3 | copy_surface | volume | surface | nsurfs,
id0, loft0, thick0, ...

18 | R | 3 | midsurface | volume | surf0 | surf1, loft,
thick
```

19 | C | 1 | classify_surface | surface | none |

type of ml_op_names: std::vector< std::string,std::allocator< std::string > >

Parameters

ml_op_names	Vector of ML operation/model names. type of entity1_ids: std::vector< size_t,std::allocator< size_t > >
entity1_ids	Vector of primary entity IDs for each operation. type of entity2_ids: std::vector< size_t,std::allocator< size_t > >
entity2_ids	Vector of secondary entity IDs for each operation. type of params: std::vector< std::vector< double,std::allocator< double > >,std::allocator< std::vector< double,std::allocator< double > > > >
params	Vector of parameter lists for each operation. type of mesh_size: float
mesh_size	Target mesh size for feature calculations. type of reduced_features: boolean, optional
reduced_features	If true (where supported), compute a reduced feature set for faster performance. return type of : std::vector< std::vector< double,std::allocator< double > >,std::allocator< std::vector< double,std:: allocator< double > > > >

Returns

Outer vector size == ml_op_names.size(); each inner vector contains the feature values for that operation.

◆ get_ML_operation_label_size()

def cubit.get_ML_operation_label_size (*ml_op_name*)

Retrieve the expected label vector length for a given ML operation.

Returns the number of entries in the output label vector for the specified operation or model.

type of ml_op_name: string

Parameters

ml_op_name Name of the ML operation or model.
return type of : int

Returns

Number of labels in the operation's output vector.

```
.. code-block:: python

                label_size =
    cubit.get_ML_operation_label_size
                ("surface_no_op" )

                print("Label size for
                surface_no_op:" , label_size)
```

◆ get_ML_predictions()

def cubit.get_ML_predictions (*ml_op_names*,
entity1_ids,
entity2_ids,
params,
mesh_size,
reduced_features = `False`
)

Compute ML predictions for a list of operations on single-entity pairs.

This function loads the ML training data (if not already loaded), computes features for each specified operation and its associated entities, and then runs predictions using a scikit-learn ensemble of decision trees.

type of ml_op_names: std::vector< std::string,std::allocator< std::string > >

Parameters

ml_op_names	Vector of ML operation or model names.
Classification	"volume_no_op", "classify_surface" (returns confidence per category).
Regression	various operation IDs (returns one or more quality metrics). type of entity1_ids: std::vector< size_t,std::allocator< size_t > >
entity1_ids	Vector of primary entity IDs for each operation. type of entity2_ids: std::vector< size_t,std::allocator< size_t > >
entity2_ids	Vector of secondary entity IDs for each operation. type of params: std::vector< std::vector< double,std::allocator< double > >,std::allocator< std::vector< double,std::allocator< double > > > >
params	Vector of parameter lists for each operation. type of mesh_size: float
mesh_size	Target mesh size used in feature calculation. type of reduced_features: boolean, optional
reduced_features	If true (where supported), use the reduced feature set for faster inference. return type of : std::vector< std::vector< double,std::allocator< double > >,std::allocator< std::vector< double,std::allocator< double > > > >

Returns

Outer vector matches ml_op_names length; each inner vector contains the predicted values for that operation:

- Classification: confidences (0-1) per category.
- Regression: one or more metrics as defined per model.

.. code-block:: python

```
                ml_op_names = ['volume_no_op' ,
    'classify_surface' ]

                entity1_ids = [5, 10] # volume
                IDs

                entity2_ids = [0, 0] # none for
                classification

                params = [[-1, -1, -1], [-1, -1, -1]]

                mesh_size = 1.5

                predictions = cubit.get_ML_predictions
                (

                ml_op_names, entity1_ids, entity2_ids,
    params,
                mesh_size)
```

◆ get_ML_regression_models()

def cubit.get_ML_regression_models ( )

Retrieve the list of supported regression ML model names.

Queries the ML subsystem for all available regression models used for prediction or analysis.

return type of : std::vector< std::string,std::allocator< std:: string > >

Returns

Vector of regression model names; empty if no models are registered.

.. code-block:: python

```
models = cubit.get_ML_regression_models
()

print("Available regression
models:" , models)
```

◆ get_ML_surviving_entities()

def cubit.get_ML_surviving_entities (*op_name*, *begin_ents*)

Identify which entities survive after a CAD operation.

Uses the snapshot from **get_ML_initialize_surviving_entities()** to find entities that remain or correspond to the initial set after the operation is completed.

type of op_name: string

Parameters

op_name Name of the ML operation.
type of begin_ents: std::vector< int,std::allocator< int > >

begin_ents **Entity** ID list from **get_ML_initialize_surviving_entities()**.
return type of : std::pair< std::string,std::vector< int,std:: allocator< int > > >

Returns

A pair:

- first = entity type ("vertex", "curve", etc.),
- second = list of surviving entity IDs.

.. code-block:: python

```
# Step 1: Initialize the baseline entity IDs

ent1 = [5] # Primary entity (surface 5)

ent2 = [0] # No secondary entities for the operation

params = [[-1, -1, -1]]

mesh_size = 1.5

begin_ents = cubit.get_ML_initialize_surviving_entities ("remove_surface" , ent1, ent2)

# Step 2: Perform the operation (remove surface 5)

cubit.cmd ("remove surface 5" )

# Step 3: Retrieve the surviving entities

result = cubit.get_ML_surviving_entities ("remove_surface" , begin_ents)

# Print the surviving entity type and ID

print("Surviving entity type:" , result[0])
```

for id in result[1]: print("Surviving entity ID:", id)

◆ get_moment_magnitude()

def cubit.get_moment_magnitude (*entity_id*)

Get the moment magnitude of a force BC.

type of entity_id: int

Parameters

entity_id ID of the force BC.
return type of : float

Returns

Moment magnitude.

◆

get_n_largest_distances_between_meshes()

def cubit.get_n_largest_distances_between_meshes (*n*,
entity_type1,
ids1,
entity_type2,
ids2
)

Finds the N largest node-to-element distances between two meshes.

For each node on the entities in `ids1` , computes its distance to nearby elements on `ids2` .

```
  * Only triangle and quad elements are supported. For a distance to be included:

      * The node and element must be sufficiently facing each other (dot product of normals < -0.65).
      * The distance must be smaller than the element's bounding box size.



  * The function returns the N largest such distances found. For each, the following are returned:

      * Distance between the node and element.
      * Node ID.
      * Element ID.



  * The result is a flat vector of 3 * N doubles, ordered by decreasing distance.

  * This function is useful for:

      * Checking mesh alignment between surfaces.
      * Identifying large mesh gaps.
      * Identifying overlapping regions where contact conditions are expected.



  * Overlapping meshes in such cases can cause problems in FEA analyses if not corrected.




.. code-block:: python
```

```
                  # Surface IDs for first mesh
                  (source nodes)


                  ids1 = [1, 2]


                  # Surface IDs for second mesh
                  (target elements)


                  ids2 = [3, 4]



                  distances = cubit.get_n_largest_distances_between_meshes
                  (5, "surface" ,
                  ids1, "surface" ,
                  ids2)
```

for i in range(0, len(distances), 3): print("Distance:", distances[i], "Node ID:", int(distances[i+1]), "Element ID:", int(distances[i+2]))

```
  @n type of n:  int
```

Parameters

n	Number of largest distances to return. type of entity_type1: string
entity_type1	**Entity** type for first mesh (e.g., "surface"). type of ids1: std::vector< int,std::allocator< int > >
ids1	IDs of entities to use for nodes. type of entity_type2: string
entity_type2	**Entity** type for second mesh (e.g., "surface"). type of ids2: std::vector< int,std::allocator< int > >
ids2	IDs of entities to use for elements.

```
  @n return type of : std::vector< double,std:: allocator< double > >
```

Returns

Flat vector of (distance, node ID, element ID) triplets, size = 3 * N.

◆ get_narrow_regions()

def cubit.get_narrow_regions (*target_ids*, *narrow_size*)

Find faces containing narrow regions within specified volumes.

Collects all unique faces of volumes in target_ids. For each face:

- If the face has a single boundary loop, looks for "split" locations between non-adjacent edges where edges come within narrow_size without a short vertex chain between them.
- If the face has multiple loops, computes the minimum distance between loops; if any two loops come within narrow_size, the face is included. Faces meeting either criterion are returned.

This function differs from get_closed_narrow_surfaces by applying to all faces (not just closed two-loop faces), and from get_surfs_with_narrow_regions by using split-point and loop-to-loop proximity tests rather than purely edge-pair orientation checks.

.. code-block:: python

```
cubit.cmd
("cylinder radius 3 height
10" )


cubit.cmd
("modify curve 1 blend
radius 0.5" )


cubit.cmd
("cylinder radius 2 height
10" )


cubit.cmd
("subtract volume 2 from
volume 1" )


narrow_thresh = 1.0


narrow_faces = cubit.get_narrow_regions
([1], narrow_thresh)


print(narrow_faces)
```

@n type of target_ids: std::vector< int,std::allocator< int > >

Parameters

target_ids Vector of volume IDs to examine.
type of narrow_size: float

narrow_size Distance threshold for detecting narrow regions.
return type of : std::vector< int,std:: allocator< int > >

Returns

Vector of surface IDs that contain narrow regions.

◆ get_narrow_surfaces()

def cubit.get_narrow_surfaces (*target_volume_ids*, *mesh_size*)

Find surfaces with narrow regions in specified volumes.

For each face of volumes in target_volume_ids, checks if any two non-adjacent edges form a narrow region (distance <= mesh_size and orientation difference > 15deg). Returns all face IDs that meet this criterion.

.. code-block:: python

```
volume_ids = [1, 2]


mesh_size = 0.1 # Distance
threshold.


narrow_surfaces = cubit.get_narrow_surfaces
(volume_ids, mesh_size)
```

for sid in narrow_surfaces: print("Narrow surface ID:", sid)

@n type of target_volume_ids: std::vector< int,std::allocator< int > >

Parameters

target_volume_ids Vector of volume IDs to examine.
type of mesh_size: float

mesh_size Distance threshold for detecting narrow regions.
return type of : std::vector< int,std:: allocator< int > >

Returns

Vector of IDs of surfaces containing a narrow region.

◆ get_nearby_entities()

def cubit.get_nearby_entities (*gtype*,
ent_ids,
compare_ents,
distance
)

Identify nearby entities of type curve, surface, or volume for a given list of the same type.

Returns a list of entities of type `gtype` that are within the specified `distance` of the entities in `ent_ids` . If `compare_ents` is empty, all entities of type `gtype` in the model will be used for comparison.

If `distance` is set to -1, a default tolerance will be computed internally.

`.. code-block:: python`

```
                cubit.cmd
                ('brick x 10' )
                # Create volume 1


                cubit.cmd
                ('brick x 10' )
                # Create volume 2


                cubit.cmd
                ('vol 2 mov x 10.5'
                ) # Move volume 2 slightly away
                from volume 1


                gtype = "volume"


                ent_ids = [1]


                compare_ents = [] # Empty list =
                check against all volumes


                distance = 1.0


                nearby_ents = cubit.get_nearby_entities
                (gtype, ent_ids, compare_ents,
    distance)



                print("Nearby volumes to
                volume 1:" )
```

for id in nearby_ents: print(id)

```
                # Expected output:


                # 2
```

`@n type of gtype:  string`

Parameters

gtype	**Entity** type to query: "curve", "surface", or "volume". type of ent_ids: std::vector< int,std::allocator< int > >
ent_ids	Entities of type `gtype` to check. type of compare_ents: std::vector< int,std::allocator< int > >
compare_ents	Entities of type `gtype` to check against. If empty, checks against all entities of type `gtype` in the model. type of distance: float
distance	Maximum distance between entities. If -1, a default tolerance will be computed. return type of : std::vector< int,std:: allocator< int > >

Returns

List of entity IDs of type `gtype` that are within `distance` of the entities in `ent_ids`.

◆ get_nearby_volumes_at_volume()

def cubit.get_nearby_volumes_at_volume (*volume_id*, *compare_volumes*, *distances*, *progress* = None)

Identify nearby volumes in the model for a list of target volumes.

For each volume in `volume_id` , returns a list of volumes from `compare_volumes` (or all volumes in the model if `compare_volumes` is empty) that are within the specified distance.

The `distances` vector specifies the maximum distance to use for each corresponding volume in `volume_id` . If a value is set to -1, a default tolerance will be computed for that volume.

The result is a vector of vectors: one list of nearby volumes per input volume in `volume_id` .

`.. code-block:: python`

```
                cubit.cmd
                ('brick x 10' )
                # Create volume 1


                cubit.cmd
                ('brick x 10' )
                # Create volume 2


                cubit.cmd
                ('vol 2 mov x 10.5'
                ) # Move volume 2 slightly away
                from volume 1


                volume_ids = [1]


                compare_volumes = [] # Empty list
                = check against all volumes


                distances = [1.0] # Per-volume
                distance


                nearby_volumes =
      cubit.get_nearby_volumes_at_volume
                (volume_ids, compare_volumes, distances)


                print("Nearby
                volumes:" )
```

for i, near_list in enumerate(nearby_volumes): print(f"Volume {volume_ids[i]} nearby volumes:", " ".join(str(v) for v in near_list))

```
                # Expected output:


                # Volume 1 nearby volumes:
                2
```

`@n type of volume_id: std::vector< int,std::allocator< int > >`

Parameters

volume_id	List of volume IDs to check. type of compare_volumes: std::vector< int,std::allocator< int > >
compare_volumes	List of volumes to check against. If empty, checks against all volumes in the model. type of distances: std::vector< double,std::allocator< double > >
distances	Maximum distance to use per volume in `volume_id`. If a value is -1, a default tolerance will be computed. type of progress: :py:class: `CubitProgressHandler`, optional
progress	Optional progress handler. return type of : std::vector< std::vector< int,std::allocator< int > >,std::allocator< std::vector< int,std:: allocator< int > > > >

Returns

A vector of vectors. For each volume in `volume_id`, returns a list of nearby volume IDs from `compare_volumes`.

◆ get_next_block_id()

def cubit.get_next_block_id ( )

Get the next available block ID.

.. code-block:: python

```
cubit.cmd
("brick x 10" )

cubit.cmd
("brick x 10" )

cubit.cmd
("vol 2 mov x 10" )


# Create blocks and assign
volumes

cubit.cmd
("block 10 vol 1" )

cubit.cmd
("block 20 vol 2" )


next_id = cubit.get_next_block_id
()

print("Next available block
ID:" , next_id)


# Expected output:

# Next available block ID:
21
```

```
@n return type of :  int
```

Returns
Next available block ID.

◆ get_next_boundary_layer_id()

def cubit.get_next_boundary_layer_id ( )

Get the next available boundary layer ID.

This ID can be used to define a new boundary layer entity.
return type of : int

Returns
Next available unique boundary layer ID.

◆ get_next_command_from_history()

def cubit.get_next_command_from_history ( )

Get the next command from the history buffer.

Advances an internal history pointer and returns the next command.

```
.. code-block:: python
```

```
import cubit

next_cmd =
cubit.get_next_command_from_history
()

print(next_cmd)
```

```
@n return type of :  string
```

Returns
The next command in the history sequence.

◆ get_next_group_id()

def cubit.get_next_group_id ( )

Return the next available group ID from Cubit.

.. code-block:: python

```
# Initialize geometry.

cubit.cmd
("brick x 10" )

# Retrieve the next free group ID
(should be 2).

next_id = cubit.get_next_group_id
()

print("Next group
ID:" , next_id) # prints
"Next group ID: 2"

# Create a new group; it will use
the next available ID.

cubit.cmd
("group \"vert_group\" add
vert in vol 1" )

# Verify assignment by fetching
the ID from the group name.

vert_group_id = cubit.get_id_from_name
("vert_group" )

print("vert_group
ID:" , vert_group_id) #
prints "vert_group ID: 2"
```

```
@n return type of :  int
```

Returns
Next unused integer ID for a new group.

◆ get_next_nodeset_id()

def cubit.get_next_nodeset_id ( )

Get the next available nodeset ID.

.. code-block:: python

```
cubit.cmd
("brick x 10" )

cubit.cmd
("nodeset 30 vertex 1
2" )

cubit.cmd
("nodeset 31 vertex 3
4" )

next_id = cubit.get_next_nodeset_id
()

print("Next available
nodeset ID:" , next_id)

# Expected output:

# Next available nodeset ID:
32
```

```
@n return type of :  int
```

Returns
Next available nodeset ID.

◆ get_next_sideset_id()

def cubit.get_next_sideset_id ( )

Get the next available sideset ID.

.. code-block:: python

```
cubit.cmd
("brick x 10" )


cubit.cmd
("sideset 20 surface
1" )


cubit.cmd
("sideset 21 surface
2" )


next_id = cubit.get_next_sideset_id
()


print("Next available
sideset ID:" , next_id)


# Expected output:


# Next available sideset ID:
22
```

@n return type of : int

Returns
Next available sideset ID.

◆ get_nodal_coordinates()

def cubit.get_nodal_coordinates (*node_id*)

Get the coordinates of a mesh node.

This function returns the (x, y, z) coordinates of the specified node ID. In C++, the coordinates are returned as a std::array<double,3>. In Python, the coordinates are returned as a tuple of three floats.

.. code-block:: python

```
# Create a brick and mesh its
volume


cubit.cmd
("brick x 10" )


cubit.cmd
("mesh vol 1" )


# Get coordinates of node
1


coords = cubit.get_nodal_coordinates
(1)


print(f"Node 1 at
({coords[0]}, {coords[1]},
{coords[2]})" )
```

@n type of node_id: int

Parameters
node_id ID of the node.
return type of : std:: array< double,3 >

Returns
: Array (or tuple) of three doubles the x, y, and z coordinates of the node.

◆ get_node_constraint()

def cubit.get_node_constraint ( )

Query whether node constraint is enabled (move mid-nodes to geometry).

Node constraints control how higher-order mid-nodes snap to curved geometry:

- off : mid-nodes retain their original positions.
- on : mid-nodes are projected to the geometry.
- smart : mid-nodes only snap if doing so does not degrade element quality.

```
.. code-block:: python
```

```
                # Enable node constraint
                (mid-nodes snap).


                cubit.cmd
                ("set Node Constraint
                on" )


                enabled = cubit.get_node_constraint
                () # returns True


                print("Node constraint
                enabled:" , enabled)



                # Disable node constraint.


                cubit.cmd
                ("set Node Constraint
                off" )


                enabled = cubit.get_node_constraint
                () # returns False


                print("Node constraint
                enabled:" , enabled)
```

```
@n return type of :  boolean
```

Returns
: True if node constraint is "on" or "smart"; otherwise false.

◆ get_node_constraint_smart_metric()

def cubit.get_node_constraint_smart_metric ( )

Query the current quality metric for smart node constraint.

When node constraint is in "smart" mode for tets or tris, this returns either "distortion" or "normalized inradius", indicating which metric controls snapping.

```
.. code-block:: python
```

```
                # Set smart mode with normalized
                inradius metric.


                cubit.cmd
                ("set Node Constraint SMART
                tet quality normalized inradius
      threshold 0.2" )


                metric =
      cubit.get_node_constraint_smart_metric
                () # returns "normalized
                inradius"


                print("Smart
                metric:" , metric)
```

```
@n return type of :  string
```

Returns
: The quality metric name for smart mode ("distortion" or "normalized inradius").

◆ get_node_constraint_smart_threshold()

def cubit.get_node_constraint_smart_threshold ( )

Query the current quality threshold for smart node constraint.

When node constraint is in "smart" mode, mid-nodes are only projected if element quality remains above this threshold.

```
.. code-block:: python
```

```
                # Set smart mode with threshold
                0.25.

                cubit.cmd
                ("set Node Constraint SMART
                tet quality normalized inradius
      threshold 0.25"
                )

                thr =
      cubit.get_node_constraint_smart_threshold
                () # returns 0.25

                print("Smart
                threshold:" , thr)
```

```
@n return type of :  float
```

Returns
The quality threshold used for smart mode (double).

◆ get_node_constraint_value()

def cubit.get_node_constraint_value ( )

Query the numeric value of the node constraint setting.

Returns 0 for off, 1 for on, or 2 for smart. This corresponds to the "set Node Constraint" options.

```
.. code-block:: python
```

```
                # Set to on.

                cubit.cmd
                ("set Node Constraint
                on" )

                mode = cubit.get_node_constraint_value
                () # returns 1

                print("Node constraint
                mode:" , mode)
```

```
@n return type of :  int
```

Returns
0 if off, 1 if on, or 2 if smart.

◆ get_node_count()

def cubit.get_node_count ( )

Retrieve the count of nodes in the current model.

.. code-block:: python

```
                cubit.cmd
                ("brick x 10" )
                # create brick (Volume ID =
                1)

                cubit.cmd
                ("mesh vol 1" )
                # mesh volume 1

                node_count = cubit.get_node_count
                ()

                print("Node count:"
                , node_count)
```

```
@n return type of :  int
```

Returns
Number of nodes in the current model's mesh.

◆ get_node_edges()

def cubit.get_node_edges (*node_id*)

Get the IDs of all edge elements adjacent to a node.

Edge elements are created on meshed curves and throughout meshed surfaces, but not within the interior volume mesh. Nodes not on any meshed curve or surface will have no adjacent edges.

```
.. code-block:: python
```

```
                    # Create a brick and mesh a curve
                    with equal spacing

                    cubit.cmd
                    ("brick x 10" )

                    cubit.cmd
                    ("curve 1 scheme
                    equal" )

                    cubit.cmd
                    ("mesh curve 1" )


                    # Query edge elements adjacent to
                    node 1

                    edges = cubit.get_node_edges
                    (1)

                    print(f"Node 1 is adjacent
                    to {len(edges)} edge(s):" , edges)
```

```
@n type of node_id:  int
```

Parameters

node_id ID of the node.
return type of : std::vector< int,std:: allocator< int > >

Returns

List of edge element IDs adjacent to the node (empty if none).

◆ get_node_exists()

def cubit.get_node_exists (*node_id*)

Check whether a node exists in the model.

This function returns `true` if the specified node ID exists in the current model; otherwise it returns `false` .

```
.. code-block:: python
```

```
                    # Create geometry and mesh volume
                    1

                    cubit.cmd
                    ("brick x 10" )

                    cubit.cmd
                    ("mesh vol 1" )


                    # Check if node 1 exists

                    exists = cubit.get_node_exists
                    (1)

                    print(f"Node 1 exists?
                    {'Yes' if exists else 'No'}" )
```

```
@n type of node_id:  int
```

Parameters

node_id ID of the node to check.
return type of : boolean

Returns

`true` if the node exists, `false` otherwise.

◆ get_node_faces()

def cubit.get_node_faces (*node_id*)

Get the IDs of all quadrilateral elements (faces) adjacent to a node.

In Cubit, "faces" are represented as quad elements on surfaces. Nodes in the interior of volumes will have no adjacent faces returned.

```
.. code-block:: python
```

```
                # Create and mesh a brick
                volume

                cubit.cmd
                ("brick x 10" )

                cubit.cmd
                ("mesh vol 1" )


                # Query quad faces adjacent to an
                interior node

                quads = cubit.get_node_faces
                (1)

                print(f"Node 1 is adjacent
                to {len(quads)} quad(s)." ) # likely 0


                # Mesh the surface and query
                again

                cubit.cmd
                ("surface 1 mesh" )

                quads = cubit.get_node_faces
                (1)

                print(f"After surface mesh,
                Node 1 is adjacent to {len(quads)}
    quad(s):" ,
                quads)
```

```
@n type of node_id:  int
```

Parameters

node_id ID of the node.
return type of : std::vector< int,std:: allocator< int > >

Returns

List of quad element IDs (faces) adjacent to the node (empty if none).

◆ get_node_global_id()

def cubit.get_node_global_id (*node_id*)

Get the global node ID assigned in the Exodus file for a mesh node.

In Cubit, each node has a local ID unique within the session. Upon exporting the mesh to an Exodus file, nodes are renumbered into a global ID space from 1 to N. This function returns the Exodus global node ID for the given local node ID, or 0 if the mesh has not been exported.

```
.. code-block:: python
```

```
                # Create and mesh a brick
                volume

                cubit.cmd
                ("brick x 10" )

                cubit.cmd
                ("mesh vol 1" )

                # Before export, no global node
                IDs exist

                gid = cubit.get_node_global_id
                (1)

                print(f"Global node ID of
                node 1 before export: {gid}" ) # prints
    0

                # Export mesh to assign global
                node IDs

                cubit.cmd
                ("export mesh
                \"model.exo\"" )

                # After export, global and local
                node IDs match

                gid = cubit.get_node_global_id
                (1)

                print(f"Global node ID of
                node 1 after export: {gid}" ) # prints
    1
```

```
@n type of node_id:  int
```

Parameters

node_id Local Cubit node ID.
return type of : int

Returns

Global node ID assigned in the Exodus file, or 0 if not yet exported.

◆ get_node_position_fixed()

def cubit.get_node_position_fixed (*node_id*)

Query whether a mesh node is fixed (constrained against smoothing).

A fixed node will not be moved by mesh smoothing or optimization operations.

.. code-block:: python

```
                # Create a brick and set up
                tetmeshing on volume 1

                cubit.cmd
                ("brick x 10" )

                cubit.cmd
                ("vol 1 scheme
                tetmesh" )

                cubit.cmd
                ("mesh vol 1" )


                # Fix all nodes on surface
                1

                cubit.cmd
                ("node in surface 1
                position fixed" )


                # Attempt to smooth all
                surfaces

                cubit.cmd
                ("smooth surface
                all" )


                # Get the center point of surface
                1

                center = cubit.get_center_point
                ("surface" , 1)


                # Find the node closest to that
                center and check its fixed state

                node_id = cubit.get_closest_node
                (*center)

                is_fixed =
      cubit.get_node_position_fixed
                (node_id)

                print(f"Node {node_id}
                fixed? {is_fixed}") #
                Expected: True
```

@n type of node_id: int

Parameters

node_id ID of the node to query.
return type of : boolean

Returns

`true` if the node is fixed (constrained), `false` if it is not.

◆ get_node_tris()

def cubit.get_node_tris (*node_id*)

Get the IDs of all triangular elements adjacent to a node.

In Cubit, triangular "faces" are represented as tris on surfaces. Nodes in the interior of volumes will have no adjacent tris returned.

```
.. code-block:: python
```

```
                # Create a brick and mesh its
                surfaces with triangles


                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("surface 1 scheme
                trimesh" )


                cubit.cmd
                ("mesh surface 1" )



                # Query triangular faces (tris)
                adjacent to node 1


                tris = cubit.get_node_tris
                (1)


                print(f"Node 1 is adjacent
                to {len(tris)} tri(s):" , tris)
```

```
@n type of node_id:  int
```

Parameters

node_id ID of the node.
return type of : std::vector< int,std:: allocator< int > >

Returns

List of triangular element IDs (tris) adjacent to the node (empty if none).

◆ get_nodeset_count()

def cubit.get_nodeset_count ( )

Get the current number of nodesets in the model.

In Cubit, a nodeset groups mesh nodes for applying boundary conditions or loads. Nodesets can be defined by assigning vertices, curves, surfaces, volumes, or individual nodes to a given ID.

```
.. code-block:: python
```

```
                # Create and mesh a brick; nodes
                are generated on volume 1.


                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("mesh volume 1" )


                # Define nodeset 201 to include
                nodes on surface 1.


                cubit.cmd
                ("Nodeset 201 add surface
                1" )


                # get_nodeset_count() returns
                1.


                count = cubit.get_nodeset_count
                ()


                print("Nodeset
                count:" , count) # prints
                1
```

```
@n return type of :  int
```

Returns

The number of nodesets currently defined in the model.

◆ get_nodeset_curves()

def cubit.get_nodeset_curves (*nodeset_id*)

Get the list of curve IDs associated with a nodeset.

This function returns the IDs of all curves that are associated with the specified nodeset.

```
.. code-block:: python
```

```
                # Create a brick and assign
                curves in volume 1 to nodeset 30


                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("nodeset 30 curve in vol
                1" )



                # Query the curves in nodeset
                30


                curve_ids = cubit.get_nodeset_curves
                (30)


                print(f"Nodeset 30 contains
                {len(curve_ids)} curve(s)." )
```

for curve_id in curve_ids: print(f" Curve ID: {curve_id}")

```
  @n type of nodeset_id:  int
```

Parameters

nodeset_id ID of the nodeset to examine.
return type of : std::vector< int,std:: allocator< int > >

Returns

List of curve IDs contained in the nodeset.

◆ get_nodeset_id_list()

def cubit.get_nodeset_id_list ( )

Get a list of all active nodeset IDs.

Returns a vector of all currently active nodeset IDs in the model.

.. code-block:: python

```
# Create geometry

cubit.cmd
("brick x 10" )

# Create nodesets on
vertices

cubit.cmd
("nodeset 30 vertex 1
2" )

cubit.cmd
("nodeset 31 vertex 3
4" )

# Query all active nodeset
IDs

nodeset_ids = cubit.get_nodeset_id_list
()

# Print the nodeset IDs

print("Active nodeset
IDs:" )
```

for nodeset_id in nodeset_ids: print(" Nodeset ID:", nodeset_id)

```
# Expected output
(example):

# Active nodeset IDs:

# Nodeset ID: 30

# Nodeset ID: 31
```

return type of : std::vector< int,std:: allocator< int > >

Returns
Vector (Python tuple) of all active nodeset IDs.

◆ get_nodeset_id_list_for_bc()

def cubit.get_nodeset_id_list_for_bc (*bc_type_enum*, *bc_id*)

Get list of nodeset IDs associated with a boundary condition.

Returns nodesets to which the BC is applied.

type of bc_type_enum: int

Parameters

bc_type_enum	BC type enum (CI_BCTypes). type of bc_id: int
bc_id	ID of the boundary condition. return type of : std::vector< int,std:: allocator< int > >

Returns
List of associated nodeset IDs.

◆ get_nodeset_node_count()

def cubit.get_nodeset_node_count (*nodeset_id*)

Get the number of nodes in a nodeset.

type of nodeset_id: int

Parameters

nodeset_id The nodeset id
return type of : int

Returns

Number of nodes in the nodeset

◆ get_nodeset_nodes()

def cubit.get_nodeset_nodes (*nodeset_id*)

Get the list of node IDs explicitly assigned to a nodeset.

This function returns the IDs of nodes that were specifically assigned to the nodeset.

If the nodeset was created on geometry (e.g. a surface or volume), this function will **not** return the nodes on that geometry unless they were explicitly assigned. To include nodes on geometry, use 'get_nodeset_nodes_inclusive' instead.

```
.. code-block:: python
```

```
                # Create a brick and mesh volume
                1

                cubit.cmd
                ("brick x 10" )

                cubit.cmd
                ("mesh vol 1" )

                # Assign nodes from surface 1 to
                nodeset 50

                cubit.cmd
                ("nodeset 50 node in
                surface 1" )

                # Query the nodes in nodeset
                50

                node_ids = cubit.get_nodeset_nodes
                (50)

                print(f"Nodeset 50 contains
                {len(node_ids)} node(s)." )
```

for node_id in node_ids: print(f" Node ID: {node_id}")

```
@n type of nodeset_id:  int
```

Parameters

nodeset_id ID of the nodeset to examine.
return type of : std::vector< int,std:: allocator< int > >

Returns

List of node IDs explicitly assigned to the nodeset.

◆ get_nodeset_nodes_inclusive()

def cubit.get_nodeset_nodes_inclusive (*nodeset_id*)

Get the list of node IDs associated with a nodeset (inclusive).

This function returns the IDs of all nodes associated with the specified nodeset. It includes:

- Nodes explicitly assigned to the nodeset.
- Nodes associated with any geometry used to define the nodeset (e.g. surfaces or volumes).

Use this function when the nodeset was created using geometry and you want **all associated nodes** . For only explicitly assigned nodes, see 'get_nodeset_nodes'.

```
.. code-block:: python
```

```
                # Create a brick and mesh volume
                1


                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("mesh vol 1" )



                # Create nodeset 50: assign nodes
                from surface 1


                cubit.cmd
                ("nodeset 50 node in
                surface 1" )


                # Add surface 6 to nodeset 50
                (nodes on surface 6 will also be
    included)

                cubit.cmd
                ("nodeset 50 add surface
                6" )


                # Query the nodes in nodeset 50
                (inclusive)


                node_ids =
    cubit.get_nodeset_nodes_inclusive
                (50)


                print(f"Nodeset 50
                (inclusive) contains {len(node_ids)}
    node(s)." )
```

for node_id in node_ids: print(f" Node ID: {node_id}")

```
  @n type of nodeset_id:  int
```

Parameters

nodeset_id ID of the nodeset to examine.
return type of : std::vector< int,std:: allocator< int > >

Returns

List of node IDs contained in the nodeset, including nodes on associated geometry.

◆ get_nodeset_surfaces()

def cubit.get_nodeset_surfaces (*nodeset_id*)

Get the list of surface IDs associated with a nodeset.

This function returns the IDs of all surfaces that are associated with the specified nodeset.

```
.. code-block:: python
```

```
                # Create a brick and assign
                surfaces 1 to 6 to nodeset 20


                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("nodeset 20 surface 1 to
                6" )



                # Query the surfaces in nodeset
                20


                surface_ids =
      cubit.get_nodeset_surfaces
                (20)


                print(f"Nodeset 20 contains
                {len(surface_ids)} surface(s)." )
```

for surf_id in surface_ids: print(f" Surface ID: {surf_id}")

```
  @n type of nodeset_id:  int
```

Parameters

nodeset_id ID of the nodeset to examine.
return type of : std::vector< int,std:: allocator< int > >

Returns

List of surface IDs contained in the nodeset.

◆ get_nodeset_vertices()

def cubit.get_nodeset_vertices (*nodeset_id*)

Get the list of vertex IDs associated with a nodeset.

This function returns the IDs of all vertices that are associated with the specified nodeset.

```
.. code-block:: python
```

```
                # Create a brick and assign
                vertices in volume 1 to nodeset 40


                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("nodeset 40 vertex in vol
                1" )



                # Query the vertices in nodeset
                40


                vertex_ids = cubit.get_nodeset_vertices
                (40)


                print(f"Nodeset 40 contains
                {len(vertex_ids)} vertex(s)." )
```

for vertex_id in vertex_ids: print(f" Vertex ID: {vertex_id}")

```
  @n type of nodeset_id:  int
```

Parameters

nodeset_id ID of the nodeset to examine.
return type of : std::vector< int,std:: allocator< int > >

Returns

List of vertex IDs contained in the nodeset.

◆ get_nodeset_volumes()

def cubit.get_nodeset_volumes (*nodeset_id*)

Get the list of volume IDs associated with a nodeset.

This function returns the IDs of all volumes that are associated with the specified nodeset.

```
.. code-block:: python
```

```
                # Create a brick and assign
                volume 1 to nodeset 1


                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("nodeset 1 vol 1" )



                # Query the volumes in nodeset
                1


                volume_ids = cubit.get_nodeset_volumes
                (1)


                print(f"Nodeset 1 contains
                {len(volume_ids)} volume(s)." )
```

for vol_id in volume_ids: print(f" Volume ID: {vol_id}")

```
  @n type of nodeset_id:  int
```

Parameters

nodeset_id ID of the nodeset to examine.
return type of : std::vector< int,std:: allocator< int > >

Returns

List of volume IDs contained in the nodeset.

◆ get_num_volume_shells()

def cubit.get_num_volume_shells (*volume_id*)

Get the number of shells in a volume.

A "shell" is a closed set of faces bounding a region within the volume. For a simple solid with no internal cavities, this returns 1. If the volume contains voids or nested regions, each closed boundary counts as a separate shell.

```
.. code-block:: python
```

```
                # Create two bricks: volume 1
                (10x10x10) and volume 2 (5x5x5).


                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("brick x 5" )


                # Subtract the smaller brick from
                the larger one, creating a cavity
      inside volume
                1.


                cubit.cmd
                ("subtract vol 2 from vol
                1" )


                # Now volume 1 has 2 shells: the
                outer boundary and the inner cavity.


                shells = cubit.get_num_volume_shells
                (1) # returns 2


                print("Volume 1
                shells:" , shells)
```

```
 @n type of volume_id:  int
```

Parameters

volume_id ID of the volume to query.
return type of : int

Returns

Number of shells (closed face loops) in the specified volume.

◆ get_overconstrained_tets_in_volumes()

def cubit.get_overconstrained_tets_in_volumes (*volumes*)

Retrieve IDs of overconstrained tetrahedra within specified volumes.

Overconstrained tetrahedra are those that have two triangular faces on the same surface and all four corner nodes lying on surfaces, curves, or vertices. Such tets cannot be smoothed and are typically removed to improve mesh quality.

type of volumes: std::vector< int,std::allocator< int > >

Parameters

volumes Vector of volume IDs to search for overconstrained tets.
return type of : std::vector< int,std:: allocator< int > >

Returns

std::vector<int> containing the IDs of all overconstrained tets found.

.. code-block:: python

```
import cubit

# 1) Create and mesh a spherical
volume (Volume ID = 1).

cubit.cmd
("create sphere rad
1" ) # Create a sphere of
radius 1 (volume ID = 1)

cubit.cmd
("volume 1 scheme
tetmesh" ) # Apply TetMesh
scheme to volume 1

cubit.cmd
("mesh volume 1" )
# Generate tetrahedral
mesh

# 2) Identify overconstrained
tets in volume 1.

vols = [1]

over_tets =
cubit.get_overconstrained_tets_in_volumes
(vols)
```

if over_tets:

3) Convert the integer list to a space-separated string.

ids_str = cubit.string_from_id_list(over_tets)

```
# 4) Remove all overconstrained
tets.

cubit.cmd
(f"remove overconstrained
tet {ids_str}" )

print(f"Removed
overconstrained tets: {ids_str}" )
```

else:

```
print("No overconstrained
tets found in volume 1." )
```

◆ get_overlap_max_angle()

def cubit.get_overlap_max_angle ( )

Get the current maximum angle tolerance used for calculating surface overlaps.

Returns the maximum angle (in degrees) allowed between normals of adjacent surfaces when determining overlaps. This setting controls how much angular deviation is tolerated before surfaces are considered non-overlapping.

.. code-block:: python

```
# Retrieve the default max angle
setting.

max_angle = cubit.get_overlap_max_angle
()

print("Current overlap max
angle:" , max_angle)
```

@n return type of : float

Returns

The current maximum angle tolerance (in degrees) used for surface overlap calculations.

◆ get_overlap_max_gap()

def cubit.get_overlap_max_gap ( )

Get the current maximum gap tolerance used for calculating surface overlaps.

Returns the maximum allowable gap between adjacent surfaces when determining overlaps. This setting controls how large a gap is tolerated before surfaces are considered non-overlapping.

.. code-block:: python

```
# Retrieve the default max gap
setting.

max_gap = cubit.get_overlap_max_gap
()

print("Current overlap max
gap:" , max_gap)
```

@n return type of : float

Returns

The current maximum gap tolerance (in model units) used for surface overlap calculations.

◆ get_overlap_min_gap()

def cubit.get_overlap_min_gap ( )

Get the current minimum gap tolerance used for calculating surface overlaps.

Returns the minimum allowable gap between adjacent surfaces when determining overlaps. This setting can be used to ignore negligible gaps that should be treated as overlapping.

.. code-block:: python

```
# Retrieve the default min gap
setting.

min_gap = cubit.get_overlap_min_gap
()

print("Current overlap min
gap:" , min_gap)
```

@n return type of : float

Returns

The current minimum gap tolerance (in model units) used for surface overlap calculations.

◆ get_overlapping_surfaces_at_surface()

def cubit.get_overlapping_surfaces_at_surface (*surface_id*, *compare_volumes*, *cache_overlaps* = 0)

Identify surfaces in the model that overlap a single target surface.

Returns a list of surfaces that overlap the specified `surface_id` . If `compare_volumes` is empty, all volumes in the model will be used for comparison.

Caching can be used to avoid redundant distance calculations when this function is called together with `get_overlapping_surfaces_in_volumes` . Both functions require the same underlying distance computations between surfaces in the specified volumes. If caching is enabled, the results of these distance calculations will be saved and reused when the second function is called. If you are calling only this function by itself, caching provides no performance benefit.

`.. code-block:: python`

```
cubit.cmd
('brick x 10' )
# Create volume 1


cubit.cmd
('brick x 10' )
# Create volume 2


cubit.cmd
('vol 2 mov x 10' )
# Move volume 2 exactly next to
volume 1


surface_id = 6 # Example surface
on volume 1


compare_volumes = [] # Empty list
= check against all volumes


overlapping_surfaces =
cubit.get_overlapping_surfaces_at_surface
(surface_id, compare_volumes, 0)


print(f"Surfaces
overlapping surface {surface_id}:" )
```

for s_id in overlapping_surfaces: print(s_id)

```
# Expected output:


# 10
```

`@n type of surface_id:  int`

Parameters

surface_id	**Surface** ID to check. type of compare_volumes: std::vector< int,std::allocator< int > >
compare_volumes	List of volumes to check against. If empty, checks against all volumes in the model. type of cache_overlaps: int, optional
cache_overlaps	Caching mode: 0 = no caching, 1 = clear cache first, 2 = use and add to existing cache. return type of : std::vector< int,std::allocator< int > >

Returns

List of surface IDs that overlap the specified `surface_id`.

◆ get_overlapping_surfaces_in_bodies()

def cubit.get_overlapping_surfaces_in_bodies (*body_ids*, *filter_slivers* = `False`)

Identify overlapping surfaces between different volumes in a set of bodies.

Overlaps are only reported between surfaces from *different volumes* . The result is a vector of vectors. The first surface ID in each vector overlaps with all subsequent surfaces in that vector.

`.. code-block:: python`

```
cubit.cmd
('brick x 10' )
# Create volume 1


cubit.cmd
('brick x 10' )
# Create volume 2


cubit.cmd
('vol 2 mov x 10' )
# Move volume 2 to overlap with
volume 1


body_ids = [1, 2]

overlaps =
cubit.get_overlapping_surfaces_in_bodies
(body_ids)


print("Overlapping
surfaces:" )
```

for surfaces in overlaps: print("Surfaces:", tuple(int(sid) for sid in surfaces))

```
# Expected output:

# Surfaces: (6, 10)

# Surfaces: (10, 6)
```

type of body_ids: std::vector< int,std::allocator< int > >

Parameters

body_ids List of body IDs to search for surface overlaps.
type of filter_slivers: boolean, optional

filter_slivers Optional flag to filter out overlaps where surfaces share a merged curve.
return type of : std::vector< std::vector< int,std::allocator< int > >,std::allocator< std::vector< int,std:: allocator< int > > > >

Returns

A list of surface overlap results. Each entry is a vector of surface IDs where the first ID overlaps with all subsequent IDs in that vector. Overlaps are only reported between surfaces from different volumes.

◆ get_overlapping_volumes()

def cubit.get_overlapping_volumes (*target_volume_ids*)

Identify overlapping volumes in a list of volumes.

For each pair of overlapping volumes, two volume IDs are returned in the output list. The first volume ID overlaps with the second, the third overlaps with the fourth, and so on. The list should always contain an even number of volume IDs (modulus 2 = 0).

```
.. code-block:: python
```

```
                cubit.cmd
                ('brick x 10' )
                # Create volume 1


                cubit.cmd
                ('brick x 10' )
                # Create volume 2


                cubit.cmd
                ('vol 2 mov x 9.5' )
                # Move volume 2 to overlap with
                volume 1



                target_volumes = [1, 2]


                overlapping_volumes = cubit.get_overlapping_volumes
                (target_volumes)



                print("Overlapping
                volumes:" )
```

for i in range(0, len(overlapping_volumes), 2): print(f"Volumes: ({overlapping_volumes[i]}, {overlapping_volumes[i+1]})")

```
                # Expected output:


                # Volumes: (1, 2)
```

```
@n type of target_volume_ids:  std::vector< int,std::allocator< int > >
```

Parameters

target_volume_ids List of volume IDs to examine.
return type of : std::vector< int,std:: allocator< int > >

Returns

A list of overlapping volume IDs. The first ID overlaps with the second, the third with the fourth, and so on.

◆ get_overlapping_volumes_at_volume()

def cubit.get_overlapping_volumes_at_volume (*volume_id*, *compare_volumes*)

Identify volumes in the model that overlap a single target volume.

Returns a list of volumes that overlap the specified `volume_id` . If `compare_volumes` is empty, all volumes in the model will be used for comparison.

`.. code-block:: python`

```
                cubit.cmd
                ('brick x 10' )
                # Create volume 1


                cubit.cmd
                ('brick x 10' )
                # Create volume 2


                cubit.cmd
                ('vol 2 mov x 9.5' )
                # Move volume 2 to overlap with
                volume 1


                volume_id = 1


                compare_volumes = [] # Empty list
                = check against all volumes


                overlapping_volumes =
      cubit.get_overlapping_volumes_at_volume
                (volume_id, compare_volumes)


                print(f"Volumes overlapping
                volume {volume_id}:" )
```

for v_id in overlapping_volumes: print(v_id)

```
                # Expected output:


                # 2
```

`@n type of volume_id:  int`

Parameters

volume_id	**Volume** ID to check. type of compare_volumes: std::vector< int,std::allocator< int > >
compare_volumes	List of volumes to check against. If empty, checks against all volumes in the model. return type of : std::vector< int,std::allocator< int > >

Returns

List of volume IDs that overlap the specified `volume_id`.

◆ get_owning_body()

def cubit.get_owning_body (*geometry_type*, *entity_id*)

Get the owning body for a specified entity.

Returns the ID of the body that contains (owns) the given entity.

```
.. code-block:: python
```

```
                    # Create a 10x10x10 brick (body
                    1).

                    cubit.cmd
                    ("brick x 10" )

                    # Query the owning body for curve
                    12.

                    body_id = cubit.get_owning_body
                    ("curve" , 12)
                    # returns 1

                    print("Owning body of curve
                    12:" , body_id)
```

```
 @n type of geometry_type:  string
```

Parameters

geometry_type Specifies the entity type ("body", "volume", "surface", "curve", or "vertex").
type of entity_id: int

entity_id ID of the entity whose owning body is requested.
return type of : int

Returns

ID of the body that owns the specified entity (or 0 if none).

◆ get_owning_volume()

def cubit.get_owning_volume (*geometry_type*, *entity_id*)

Get the owning volume for a specified entity.

Returns the ID of the volume that contains (owns) the given entity.

```
.. code-block:: python
```

```
                    # Create a 10x10x10 brick (volume
                    1).

                    cubit.cmd
                    ("brick x 10" )

                    # Query the owning volume for
                    curve 12.

                    volume_id = cubit.get_owning_volume
                    ("curve" , 12)
                    # returns 1

                    print("Owning volume of
                    curve 12:" , volume_id)
```

```
 @n type of geometry_type:  string
```

Parameters

geometry_type Specifies the entity type ("body", "volume", "surface", "curve", or "vertex").
type of entity_id: int

entity_id ID of the entity whose owning volume is requested.
return type of : int

Returns

ID of the volume that owns the specified entity (or 0 if none).

◆ get_owning_volume_by_name()

def cubit.get_owning_volume_by_name (*entity_name*)

Get the owning volume for an entity by its name.

.. code-block:: python

```
                # Create a brick and name one
                surface.


                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("surface 1 name
                'TipSurface'" )


                volume_id =
      cubit.get_owning_volume_by_name
                ("TipSurface" )


                print("Owning
                volume:" , volume_id) #
                prints 1
```

```
@n type of entity_name:  string
```

Parameters

entity_name The name of the entity as defined in Cubit.
return type of : int

Returns

ID of the volume that owns the named entity, or 0 if the name is not found.

◆ get_parent_assembly_instance()

def cubit.get_parent_assembly_instance (*assembly_id*)

Get the instance number of the parent of an assembly node.

type of assembly_id: int

Parameters

assembly_id ID of the assembly node.
return type of : int

Returns

Parent instance number.

◆ get_parent_assembly_path()

def cubit.get_parent_assembly_path (*assembly_id*)

Get the path of an assembly node's parent.

type of assembly_id: int

Parameters

assembly_id ID of the assembly node.
return type of : string

Returns

Parent path string.

◆ get_pick_filters()

def cubit.get_pick_filters ( )

Retrieve the list of entity types currently allowed for picking.

Returns all types in the active pick filter. Only entities of these types can be selected in the graphics window.

.. code-block:: python

```
import cubit

filters = cubit.get_pick_filters
()
```

for t in filters: print("Pick filter:", t)

```
@n return type of : std::vector< std::string,std::allocator< std:: string > >
```

Returns
Vector of strings naming the current pick filter types.

◆ get_pick_type()

def cubit.get_pick_type ( )

Get the current pick mode for entity selection.

Returns the pick type that the graphics system is using for selection. This corresponds to the GUI icons and can be one of: "vertex", "curve", "surface", "volume", "node", "edge", "face", etc.

.. code-block:: python

```
import cubit

mode = cubit.get_pick_type
()

print("Current pick
mode:" , mode)

# Retrieve selected entities of
that type

ids = cubit.get_selected_ids
()
```

for id in ids: print(f"Selected {mode}: {id}")

```
@n return type of :  string
```

Returns
Null-terminated string naming the current pick type.

◆ get_pressure_function()

def cubit.get_pressure_function (*entity_id*)

Get the function expression associated with a pressure BC.

type of entity_id: int

Parameters
entity_id ID of the pressure BC.
return type of : string

Returns
Pressure function as a string.

◆ get_pressure_value()

def cubit.get_pressure_value (*entity_id*)

Get the magnitude value of a pressure BC.

type of entity_id: int

Parameters

entity_id ID of the pressure BC.
return type of : float

Returns

Pressure magnitude.

◆ get_previous_command_from_history()

def cubit.get_previous_command_from_history ( )

Get the previous command from the history buffer.

Moves the internal history pointer backwards and returns that command.

```
.. code-block:: python
```

```
                    import cubit

                    prev_cmd =
        cubit.get_previous_command_from_history
                    ()

                    print(prev_cmd)
```

```
 @n return type of :  string
```

Returns

The previous command in the history sequence.

◆ get_pyramid_count()

def cubit.get_pyramid_count ( )

Retrieve the count of pyramid elements in the current model.

.. code-block:: python

```
cubit.cmd
("brick x 10" )
# Volume ID = 1


cubit.cmd
("brick x 10" )
# Volume ID = 2


cubit.cmd
("vol 1 mov x 10" )
# move volume 1 to abut volume
2


cubit.cmd
("merge all" )
# merge volumes


cubit.cmd
("vol 1 scheme map"
) # hex mesh for volume 1


cubit.cmd
("mesh vol 1" )


cubit.cmd
("vol 2 scheme
tetmesh" ) # tet mesh for
volume 2


cubit.cmd
("mesh vol 2" )


# pyramids are generated at the
interface to join hex and tet elements


pyramid_count = cubit.get_pyramid_count
()


print("Pyramid
count:" , pyramid_count)
```

```
@n return type of :  int
```

Returns

Number of pyramid elements in the current model's mesh.

◆ get_pyramid_global_element_id()

def cubit.get_pyramid_global_element_id (*pyramid_id*)

Get the Global Element ID for a specific pyramid element.

This function returns the Global Element ID assigned to the given local pyramid ID when the element is placed into a block.

```
.. code-block:: python
```

```
                # Create two volumes and generate
                pyramids at their interface

                cubit.cmd
                ("brick x 10" )

                cubit.cmd
                ("vol 1 copy mov x
                10" )

                cubit.cmd
                ("merge all" )

                # Mesh volume 1 with mapped
                hexes

                cubit.cmd
                ("vol 1 scheme map"
                )

                cubit.cmd
                ("mesh vol 1" )

                # Mesh the other side with
                tetmeshing to create pyramids

                cubit.cmd
                ("vol 2 scheme
                tetmesh" )

                cubit.cmd
                ("mesh vol 2" )


                # Assign all pyramid elements to
                block 1

                cubit.cmd
                ("block 1 pyramid
                all" )


                # Query the Global Element ID of
                local pyramid 22

                gid =
      cubit.get_pyramid_global_element_id
                (22)

                print("Global Element ID of
                pyramid 22:" , gid)
```

```
 @n type of pyramid_id:  int
```

Parameters

pyramid_id Local pyramid element ID within its type.
return type of : int

Returns

Global Element ID for the specified pyramid, or 0 if not placed into a block.

◆ get_python_version()

def cubit.get_python_version ( )

Retrieve the Python interpreter version used by Cubit.

Returns the version string of the embedded Python interpreter in Cubit.

```
.. code-block:: python
```

```
                import cubit

                python_version =
      cubit.get_python_version
                ()

                print("Python
                version:" , python_version)
```

```
 @n return type of :  string
```

Returns

A string containing the Python version number.

◆ get_quad_count()

def cubit.get_quad_count ( )

Retrieve the count of quadrilateral elements in the current model.

For a hexahedral volume mesh, this returns the number of quad faces on the surface.

.. code-block:: python

```
cubit.cmd
("brick x 10" )
# create brick (Volume ID =
1)


cubit.cmd
("vol 1 scheme map"
) # set hex scheme


cubit.cmd
("mesh vol 1" )
# mesh as hexes (surface faces
are quads)


quad_count = cubit.get_quad_count
()


print("Quad count:"
, quad_count)
```

@n return type of : int

Returns

Number of quadrilateral elements (surface faces of a hex mesh).

◆ get_quad_global_element_id()

def cubit.get_quad_global_element_id (*quad_id*)

Get the Global Element ID for a specific quadrilateral element.

Cubit assigns a Global Element ID to each element when it is placed into a block. This function returns the Global Element ID corresponding to the given local quad ID within its type-specific ID space.

.. code-block:: python

```
# Create a brick and mesh its
volume


cubit.cmd
("brick x 10" )


cubit.cmd
("mesh vol 1" )


cubit.cmd
("block 1 quad in surface
1" ) # assign all quads on
surface 1 to block 1



# Query the Global Element ID of
local quad 22


gid = cubit.get_quad_global_element_id
(22)


print("Global Element ID of
quad 22:" , gid)
```

@n type of quad_id: int

Parameters

quad_id Local quadrilateral element ID within its type.
return type of : int

Returns

Global Element ID for the specified quad, or 0 if not placed into a block.

◆ get_quality_stats_at_geometry()

def cubit.get_quality_stats_at_geometry (*geom_type*, *mesh_type*, *geom_id_list*, *expand_levels*, *metric_name*, *single_threshold*, *use_low_threshold*, *low_threshold*, *high_threshold*, *make_group*)

Python-friendly version of get_quality_stats operating on geometry entities.

Computes quality statistics over all mesh elements attached to the specified geometry entities (curves, surfaces, or volumes) up to a given adjacency level, using the specified metric and threshold criteria. Results are packed into a single std::vector<double> as follows:

- [0] min_value
- [1] max_value
- [2] mean_value
- [3] std_value
- [4] min_element_id
- [5] max_element_id
- [6] element_type (0=edge, 1=tri, 2=quad, 3=tet, 4=hex)
- [7] bad_group_id
- [8] number of failing elements (n)
- [9]...[9+n-1] IDs of failing elements

Unlike **get_elem_quality_stats()** , which operates on a flat list of mesh-element IDs, this function first gathers all mesh elements of mesh_type attached to the given geometry_type entities (expanding connectivity up to expand_levels) and then calls **get_elem_quality_stats()** internally.

```
.. code-block:: python
```

```
                # 1) Create two vertices and a
                linear curve (Curve ID = 1), then mesh
      that
                curve.

                cubit.cmd
                ("create vertex 0 0
                0" )

                cubit.cmd
                ("create vertex 1 0
                0" )

                cubit.cmd
                ("create curve vertex 1
                2" ) # Curve ID = 1

                cubit.cmd
                ("curve 1 size 0.1"
                )

                cubit.cmd
                ("mesh curve 1" )


                # 2) Compute quality stats for
                all "edge" elements attached to that
      curve.

                # expand_levels = 0 means only
                directly connected edges.

                curves = [1]

                quality_data =
      cubit.get_quality_stats_at_geometry
                (

                "curve" ,
                # geom_type

                "edge" ,
                # mesh_type

                curves, # geom_id_list

                0, # expand_levels

                "scaled jacobian" ,
                # metric_name

                0.2, # single_threshold

                False , # use_low_threshold

                0.0, # low_threshold

                0.0 # high_threshold

                True# make_arrays
                )
                min_value = quality_data[0]
                max_value = quality_data[1]
                mean_value = quality_data[2]
                std_value = quality_data[3]
                skew_val = cubit.get_quality_value
                (
                "hex" , # mesh_type
                223, # mesh_id
                "skew"# metric_name
                )
                num_failures = int(quality_data[8])
                mesh_list = [int(x) for x in
      quality_data[9:9+num_failures]]
```

◆ get_quality_value()

def cubit.get_quality_value(*mesh_type*,
mesh_id,
metric_name
)

Retrieve a specific quality metric value for a single mesh element.

Returns the requested quality metric for the given mesh entity.

.. code-block:: python

```
                min_element_id = int(quality_data[4])
                max_element_id = int(quality_data[5])
                element_type = int(quality_data[6]) #
      0=edge
                bad_group_id = int(quality_data[7])
                geom_id_list = [int(x) for x in
      quality_data[10+num_failures:10+num_failures+num_geom]]
```

@n type of mesh_type: string

Parameters

mesh_type Type of mesh element ("hex", "tet", "tri", "quad").

@n type of geom_type: string

Parameters

mesh_id type of mesh_id: int
Identifier of the mesh element.

geom_type type of metric_name: string
Geometry type ("curve", "surface", or "volume").

metric_name Name of the quality metric (e.g. "skew", "taper", "scaled jacobian").

mesh_type type of mesh_type: string
Mesh element type to evaluate ("edge", "tri", "quad", "tet", "hex").

return type of : float

type of geom_id_list: std::vector< int,std::allocator< int > >

Returns

The value of the specified quality metric.

geom_id_list Vector of geometry IDs whose attached mesh elements are considered.

type of expand_levels: int

expand_levels Number of adjacency levels to include (0 = only directly attached elements).

type of metric_name: string

◆ get_quality_values()

def cubit.get_quality_values (*mesh_type, mesh_ids, metric_name*)

Retrieve quality metric values for multiple mesh elements.

Returns a vector of metric values for the specified list of element IDs. This differs from `get_quality_value()`, which only returns the quality of a single element's metric.

```
.. code-block:: python
```

```
                mesh_ids = [223, 224, 225]
                skew_vals = cubit.get_quality_values
                (
                "hex", # mesh_type
                mesh_ids, # list of element IDs
                "skew"# metric_name

                )

for eid, val in zip(mesh_ids, skew_vals): print(f"Element {eid} skew: {val}")
```

```
  @n type of mesh_type:  string
```

Parameters

mesh_type Type of mesh element ("hex", "tet", "tri", "quad").
type of mesh_ids: std::vector< int,std::allocator< int > >

mesh_ids Vector of mesh element IDs to query.
type of metric_name: string

metric_name Name of the quality metric (e.g., "skew", "taper", "scaled jacobian").
return type of : std::vector< double,std:: allocator< double > >

Returns

std::vector<double> containing metric values corresponding to each mesh_id.

◆ get_reduce_bolt_core_default_dimensions()

def cubit.get_reduce_bolt_core_default_dimensions (*vol_id*)

Computes default core dimensions for a bolt volume in reduce bolt core operation.

Used with `reduce volume <id> bolt core` to estimate the default values for c1, c2, and c3, which define the extent of the core geometry surrounding the bolt.

```
.. code-block:: python
```

```
                dims =
        cubit.get_reduce_bolt_core_default_dimensions
                (2)

                print("c1:" ,
                dims[0], "c2:" ,
                dims[1], "c3:" ,
                dims[2])
```

```
 @n type of vol_id:  int
```

Parameters

vol_id **Volume** ID representing bolt geometry.
return type of : std::vector< double,std:: allocator< double > >

Returns

Vector containing [c1, c2, c3] dimensions for the bolt core.

◆ get_relatives()

def cubit.get_relatives (*source_geometry_type*,
source_id,
target_geom_type
)

Get the relatives (parents or children) of a specified entity.

Use this to fetch either ancestor (parent) or descendant (child) entities of one type for a given source entity. For example, to list all curves bounding surface 12, call with source_geometry_type="surface", source_id=12, target_geom_type="curve".

.. code-block:: python

```
# Create a brick and extract
curves for surface 1.


cubit.cmd
("brick x 10" )


curve_list = cubit.get_relatives
("surface" , 1,
"curve" )


print(curve_list) # e.g., (1, 2,
3, 4)
```

@n type of source_geometry_type: string

Parameters

source_geometry_type The type of the source entity ("body", "volume", "surface", "curve", or "vertex").
type of source_id: int

source_id ID of the source entity.
type of target_geom_type: string

target_geom_type The target entity type to retrieve ("body", "volume", "surface", "curve", or "vertex").
return type of : std::vector< int,std:: allocator< int > >

Returns

A list (vector or Python tuple) of IDs matching target_geom_type related to source_id.

◆ get_rendering_mode()

def cubit.get_rendering_mode ( )

Get the current graphics rendering mode.

Returns an integer code for the active rendering style:

- 0 = wireframe only
- 1 = wireframe with hidden-line removal
- 2 = true hidden-line shading
- 3 = smooth shaded surfaces
- 4 = transparent surfaces
- 5 = geometry facets only

.. code-block:: python

```
import cubit


# Set rendering mode to
hidden-line removal and verify


cubit.cmd
("Graphics Mode
HiddenLine" )


mode = cubit.get_rendering_mode
()


print("Rendering
mode:" , mode) # should
print 1
```

@n return type of : int

Returns

Integer code for the current rendering mode.

◆ get_requested_mesh_interval_firmness()

def cubit.get_requested_mesh_interval_firmness (*geometry_type*, *entity_id*)

Retrieve the explicitly requested interval firmness for a specified entity.

Unlike **get_mesh_interval_firmness()** , which returns the effective firmness after considering influences from connected topology, this function returns only the firmness setting directly assigned by the user on the entity itself (no inheritance or propagation).

Title Possible return values are:

- "HARD" - interval was explicitly locked by using a hard interval command on this entity.
- "SOFT" - interval was explicitly set (or upgraded) to soft on this entity.
- "LIMP" - no interval firmness was ever set directly on this entity.

```
.. code-block:: python
```

```
                # Assume surface 12 already
                exists.

                # 1) Explicitly set surface 12's
                interval firmness to SOFT:

                cubit.cmd
                ("surface 12 interval
                soft" )

                # 2) Query only the requested
                interval firmness for surface 12:

                firmness =
      cubit.get_requested_mesh_interval_firmness
                ("surface" , 12)

                print("Surface 12 requested
                interval firmness:" , firmness) #
      Expected: "SOFT"

                # 3) Query again (still "SOFT",
                unaffected by connected geometry).

                firmness =
      cubit.get_requested_mesh_interval_firmness
                ("surface" , 12)

                print("Surface 12 requested
                interval firmness:" , firmness) #
      Expected: "SOFT"
```

```
 @n type of geometry_type:  string
```

Parameters

geometry_type Geometry type of the entity ("curve", "surface", or "volume").
type of entity_id: int

entity_id ID of the entity whose requested interval firmness is queried.
return type of : string

Returns

: The requested interval firmness on that entity "HARD", "SOFT", or "LIMP". "LIMP" indicates no firmness was explicitly set here.

◆ get_requested_mesh_intervals()

def cubit.get_requested_mesh_intervals (*geometry_type*, *entity_id*)

Retrieve the mesh interval count explicitly set on a geometry entity.

Returns the number of subdivisions assigned directly to the specified geometry entity.

- If no interval is set on the entity, it inherits the size from its parent.
- If no interval is set anywhere in the hierarchy, the default automatic size is used.

.. code-block:: python

```
import cubit

size = cubit.get_requested_mesh_intervals ("curve" , 7)

print("Mesh size on curve 7:" , size)
```

@n type of geometry_type: string

Parameters

geometry_type **Entity** type: "curve", "surface", or "volume".
type of entity_id: int

entity_id ID of the geometry entity.
return type of : int

Returns

Interval count set on that entity.

◆ get_requested_mesh_size()

def cubit.get_requested_mesh_size (*geometry_type*, *id*)

Retrieve the mesh size explicitly set on a geometry entity.

Returns the target edge length that was directly assigned to the entity, not inherited from parent entities. If no explicit size was set, returns -1.

.. code-block:: python

```
import cubit

# Get the requested mesh size on surface 4

requested_size = cubit.get_requested_mesh_size ("surface" , 4)

print("Requested mesh size:" , requested_size)
```

@n type of geometry_type: string

Parameters

geometry_type **Entity** type: "curve", "surface", or "volume".
type of id: int

id ID of the geometry entity.
return type of : float

Returns

Mesh size explicitly set on the entity, or -1 if none.

◆ get_requested_mesh_size_type()

def cubit.get_requested_mesh_size_type (*geometry_type*, *entity_id*)

Retrieve the mesh size setting type explicitly requested on a specified entity.

Unlike **get_mesh_size_type()** , which may return "CALCULATED" if the size was inherited or computed, this function returns only the status of a mesh size setting directly applied by the user on this entity.

Title Possible return values are:

- "USER_SET" - The mesh size (target edge length) was explicitly set on this entity.
- "CALCULATED" - The mesh size was inferred or computed on this entity (but not directly by the user).
- "NOT_SET" - No mesh size was explicitly set on this entity.

.. code-block:: python

```
                # 1) Create two vertices and a
                linear curve between them (Curve ID =
    1).

                cubit.cmd
                ("create vertex 0 0
                0" ) # Vertex ID =
                1

                cubit.cmd
                ("create vertex 1 0
                0" ) # Vertex ID =
                2

                cubit.cmd
                ("create curve vertex 1
                2" ) # Curve ID = 1


                # 2) By default, no mesh size is
                set, so this returns "NOT_SET".

                requested_type =
    cubit.get_requested_mesh_size_type
                ("curve" , 1)

                print("Curve 1 requested
                mesh size type:" , requested_type) #
    Expected: "NOT_SET"


                # 3) Explicitly set a mesh size
                (target edge length) of 0.1 on curve 1.

                cubit.cmd
                ("curve 1 size 0.1"
                )


                # 4) Now
                get_requested_mesh_size_type returns
    "USER_SET".

                requested_type =
    cubit.get_requested_mesh_size_type
                ("curve" , 1)

                print("Curve 1 requested
                mesh size type:" , requested_type) #
    Expected: "USER_SET"
```

@n type of geometry_type: string

Parameters

geometry_type Geometry type ("curve", "surface", or "volume").
type of entity_id: int

entity_id ID of the entity whose requested mesh size status is queried.
return type of : string

Returns

: The requested mesh size status on that entity "USER_SET", "CALCULATED", or "NOT_SET".

◆ get_revision_date()

def cubit.get_revision_date ( )

Retrieve the Cubit revision date.

Returns the date of the last code revision applied to the Cubit engine.

```
.. code-block:: python
```

```
import cubit

rev_date = cubit.get_revision_date
()

print("Revision
date:" , rev_date)
```

```
@n return type of :  string
```

Returns
A string containing Cubit's last revision date.

◆ get_rubberband_shape()

def cubit.get_rubberband_shape ( )

Get the current rubberband selection shape.

Returns an integer code for the active selection shape in the graphics window:

- 0 = box
- 1 = polygon
- 2 = circle

```
.. code-block:: python
```

```
import cubit

shape = cubit.get_rubberband_shape
()

print("Rubberband
shape:" , shape)
```

```
@n return type of :  int
```

Returns
Integer code indicating the current rubberband selection shape.

◆ get_selected_id()

def cubit.get_selected_id (*index*)

Get the selected entity ID by index.

Use together with **get_selected_ids()** and **get_selected_type()** to inspect selections.

```
.. code-block:: python
```

```
import cubit

# Select surfaces 3 and 4

cubit.cmd
("select surface 3
4" )

# Retrieve all selected IDs by
index

ids = cubit.get_selected_ids
()
```

for i in range(len(ids)): id = cubit.get_selected_id(i)

```
type = cubit.get_selected_type
(i)

print("Selection" ,
i, ":" , type, id)
```

```
@n type of index:  int
```

Parameters
index Zero-based index into the current selection list.
return type of : int

Returns
Entity ID at the given index.

◆ get_selected_ids()

def cubit.get_selected_ids ( )

Retrieve all currently selected entity IDs in pick order.

Returns a vector of IDs for entities selected via the graphics interface or programmatic pick commands. The order reflects the sequence in which entities were picked.

```
.. code-block:: python
```

```
                    import cubit


                    # Select surfaces 3 and 4, then
                    iterate through the selection


                    cubit.cmd
                    ("select surface 3
                    4" )


                    ids = cubit.get_selected_ids
                    ()
```

for i, id in enumerate(ids): t = cubit.get_selected_type(i)

```
                    print("Selected" ,
                    t, "with ID" , id)
```

```
 @n return type of : std::vector< int,std:: allocator< int > >
```

Returns

Vector of selected entity IDs; empty if no entities are selected.

◆ get_selected_type()

def cubit.get_selected_type (*index*)

Get the selected entity type by index.

Use together with **get_selected_ids()** and **get_selected_id()** to inspect selections.

```
.. code-block:: python
```

```
                    import cubit


                    # Select surfaces 3 and 4


                    cubit.cmd
                    ("select surface 3
                    4" )


                    # If there is at least one
                    selection, print its type and ID


                    ids = cubit.get_selected_ids
                    ()
```

if ids: id = cubit.get_selected_id(0)

```
                    t = cubit.get_selected_type
                    (0)


                    print(f"First selection is
                    a {t} with ID {id}" )
```

```
 @n type of index:  int
```

Parameters

index Zero-based index into the current selection list.
return type of : string

Returns

Entity type string ("vertex", "curve", "surface", "volume") at that index.

◆ get_sgm_version()

def cubit.get_sgm_version ( )

Retrieve the SGM (Solid Geometry Manager) version.

Returns the version string of the SGM engine integrated with Cubit.

.. code-block:: python

```
import cubit

sgm_version = cubit.get_sgm_version
()

print("SGM version:"
, sgm_version)
```

@n return type of : string

Returns
A string containing the SGM version number.

◆ get_sharp_angle_vertices()

def cubit.get_sharp_angle_vertices (*target_volume_ids*, *upper_bound*, *lower_bound*)

Identify vertices at sharp curve angles in a set of volumes.

This function computes the interior angles at curve intersections (sharp corners) on the surfaces of each of the specified volumes. Vertices are identified where the interior angle exceeds the specified *upper_bound* or falls below the *lower_bound* threshold.

.. code-block:: python

```
cubit.cmd
('brick x 10' )

cubit.cmd
('cylinder rad 5 height
10' )

cubit.cmd
('vol 2 mov z 10' )

cubit.cmd
('unite vol all' )


sharp_vertex_data = cubit.get_sharp_angle_vertices
([1], 350, 10)


vertex_ids = sharp_vertex_data[0]

angles = sharp_vertex_data[1]


print("Sharp
vertices:" )
```

for vid, angle in zip(vertex_ids, angles): print(f"Vertex {int(vid)} angle {angle:.1f}")

```
# Expected output:

# Vertex 11 angle 0.0

# Vertex 12 angle 0.0

# Vertex 13 angle 0.0

# Vertex 14 angle 0.0
```

@n type of target_volume_ids: std::vector< int,std::allocator< int > >

Parameters

target_volume_ids	List of volume IDs to analyze. type of upper_bound: float
upper_bound	Upper threshold angle, in degrees. type of lower_bound: float
lower_bound	Lower threshold angle, in degrees. return type of : std::vector< std::vector< double,std::allocator< double > >,std::allocator< std::vector< double,std:: allocator< double > > > >

Returns
: A 2D array [vertex IDs[], corresponding angles[]]. The first array contains the unique vertex IDs (as integers). The second array contains the corresponding angle at each vertex (as floating-point values).

◆ get_sideset_area()

def cubit.get_sideset_area (*sideset_id*)

Get the total area of a sideset.

A sideset can contain triangle (tri) or quadrilateral (face) elements. This function returns the sum of the areas of those elements, if they exist. If no such elements are present in the sideset, the function returns zero.

```
.. code-block:: python
```

```
                # Create a brick, mesh the
                volume, and assign faces on surface 1
      to sideset
                1


                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("mesh vol 1" )


                cubit.cmd
                ("sideset 1 face in surface
                1" )


                # Query the area of sideset
                1


                area = cubit.get_sideset_area
                (1)


                print(f"Sideset 1 total
                area: {area}" )
```

```
 @n type of sideset_id:  int
```

Parameters

sideset_id ID of the sideset to examine.
return type of : float

Returns

Sum of the areas of all tris or faces in the sideset.

◆ get_sideset_count()

def cubit.get_sideset_count ( )

Get the current number of sidesets in the model.

In Cubit, a sideset groups element faces (or edges) for applying boundary conditions. Sidesets can be defined by assigning surfaces, curves, faces, or mesh entities to a given ID.

```
.. code-block:: python
```

```
                # Create and mesh a brick;
                surfaces 1-6 become mesh faces.


                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("mesh volume 1" )


                # Define sideset 101 to include
                faces on surface 1.


                cubit.cmd
                ("Sideset 101 add surface
                1" )


                # get_sideset_count() returns
                1.


                count = cubit.get_sideset_count
                ()


                print("Sideset
                count:" , count) # prints
                1
```

```
 @n return type of :  int
```

Returns

The number of sidesets currently defined in the model.

◆ get_sideset_curves()

def cubit.get_sideset_curves (*sideset_id*)

Get the list of curve IDs associated with a sideset.

This function returns the IDs of all curves that are associated with the specified sideset.

```
.. code-block:: python
```

```
                # Create a brick and assign
                curves in volume 1 to sideset 10


                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("sideset 10 curve in vol
                1" )



                # Query the curves in sideset
                10


                curve_ids = cubit.get_sideset_curves
                (10)


                print(f"Sideset 10 contains
                {len(curve_ids)} curve(s)." )
```

for curve_id in curve_ids: print(f" Curve ID: {curve_id}")

```
  @n type of sideset_id:  int
```

Parameters

sideset_id ID of the sideset to examine.
return type of : std::vector< int,std:: allocator< int > >

Returns

List of curve IDs contained in the sideset.

◆ get_sideset_edges()

def cubit.get_sideset_edges (*sideset_id*)

Get the list of edge IDs contained in a sideset.

A sideset can contain edge elements. This function returns the IDs of those edge elements, if they exist. An empty list will be returned if there are no edges in the sideset.

```
.. code-block:: python
```

```
                # Create a brick, mesh volume 1,
                and assign edges in volume 1 to sideset
    20


                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("mesh vol 1" )


                cubit.cmd
                ("sideset 20 edge in curve
                in vol 1" )



                # Query the edges in sideset
                20


                edge_ids = cubit.get_sideset_edges
                (20)


                print(f"Sideset 20 contains
                {len(edge_ids)} edge(s)." )
```

for edge_id in edge_ids: print(f" Edge ID: {edge_id}")

```
  @n type of sideset_id:  int
```

Parameters

sideset_id ID of the sideset to examine.
return type of : std::vector< int,std:: allocator< int > >

Returns

List of edge IDs contained in the sideset.

◆ get_sideset_element_type()

def cubit.get_sideset_element_type (*sideset_id*)

Get the element type of a sideset.

type of sideset_id: int

Parameters

sideset_id The id of the sideset to be queried
return type of : string

Returns

Element type

◆ get_sideset_id_list()

def cubit.get_sideset_id_list ( )

Get a list of all active sideset IDs.

Returns a vector of all currently active sideset IDs in the model.

```
.. code-block:: python
```

```
                # Create geometry

                cubit.cmd
                ("brick x 10" )

                # Create sidesets on
                surfaces

                cubit.cmd
                ("sideset 20 surface
                1" )

                cubit.cmd
                ("sideset 21 surface
                2" )

                # Query all active sideset
                IDs

                sideset_ids = cubit.get_sideset_id_list
                ()

                # Print the sideset IDs

                print("Active sideset
                IDs:" )
```

for sideset_id in sideset_ids: print(" Sideset ID:", sideset_id)

```
                # Expected output
                (example):

                # Active sideset IDs:

                # Sideset ID: 20

                # Sideset ID: 21
```

```
@n return type of : std::vector< int,std:: allocator< int > >
```

Returns

Vector (Python tuple) of all active sideset IDs.

◆ get_sideset_id_list_for_bc()

def cubit.get_sideset_id_list_for_bc (*bc_type_enum*, *bc_id*)

Get list of sideset IDs associated with a boundary condition.

Returns sidesets to which the BC is applied.

type of bc_type_enum: int

Parameters

bc_type_enum BC type enum (CI_BCTypes).
type of bc_id: int

bc_id ID of the boundary condition.
return type of : std::vector< int,std::
allocator< int > >

Returns

List of associated sideset IDs.

◆ get_sideset_quads()

def cubit.get_sideset_quads (*sideset_id*)

Get the list of quadrilateral (quad) element IDs contained in a sideset.

A sideset can contain quadrilateral elements (faces). This function returns the IDs of those quad elements, if they exist. An empty list will be returned if there are no quads in the sideset.

```
.. code-block:: python

                # Create a brick, mesh the
                volume, and assign faces on surface 1
    to sideset
                4


                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("mesh vol all" )


                cubit.cmd
                ("sideset 4 face in surface
                1" )


                # Query the quads in sideset
                4


                quad_ids = cubit.get_sideset_quads
                (4)


                print(f"Sideset 4 contains
                {len(quad_ids)} quad(s)." )
```

for quad_id in quad_ids: print(f" Quad ID: {quad_id}")

```
  @n type of sideset_id:  int
```

Parameters

sideset_id ID of the sideset to examine.
return type of : std::vector< int,std:: allocator< int > >

Returns

List of quadrilateral (quad) element IDs contained in the sideset.

◆ get_sideset_surfaces()

def cubit.get_sideset_surfaces (*sideset_id*)

Get the list of surface IDs contained in a sideset.

A sideset can contain surfaces. This function returns the IDs of those surfaces, if they exist. An empty list will be returned if there are no surfaces in the sideset.

```
.. code-block:: python
```

```
                # Create a brick and assign
                surface 3 to sideset 3


                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("sideset 3 surface
                3" )



                # Query the surfaces in sideset
                3


                surface_ids =
      cubit.get_sideset_surfaces
                (3)


                print(f"Sideset 3 contains
                {len(surface_ids)} surface(s)." )
```

for surf_id in surface_ids: print(f" Surface ID: {surf_id}")

```
  @n type of sideset_id:  int
```

Parameters

sideset_id ID of the sideset to examine.

return type of : std::vector< int,std:: allocator< int > >

Returns

List of surface IDs contained in the sideset.

◆ get_sideset_tris()

def cubit.get_sideset_tris (*sideset_id*)

Get the list of triangle (tri) element IDs contained in a sideset.

A sideset can contain triangle (tri) elements. This function returns the IDs of those tri elements, if they exist. An empty list will be returned if there are no tris in the sideset.

.. code-block:: python

```
                # Create a brick, tetmesh volume
                1, and assign tris on surface 6 to
      sideset 6


                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("vol 1 scheme
                tetmesh" )


                cubit.cmd
                ("mesh vol 1" )


                cubit.cmd
                ("sideset 6 tri in surface
                6" )



                # Query the tris in sideset
                6


                tri_ids = cubit.get_sideset_tris
                (6)


                print(f"Sideset 6 contains
                {len(tri_ids)} tri(s)." )
```

for tri_id in tri_ids: print(f" Tri ID: {tri_id}")

```
  @n type of sideset_id:  int
```

Parameters

sideset_id ID of the sideset to examine.
return type of : std::vector< int,std:: allocator< int > >

Returns

List of triangle (tri) element IDs contained in the sideset.

◆ get_similar_curves()

def cubit.get_similar_curves (*curve_ids*,
tol = `1e-3`,
use_percent_tol = `True`,
on_similar_vols = `True`
)

Find curves with lengths similar to a given curve.

This function compares the length of the curve in curve_ids against all other curves in the model. Curves whose lengths differ by no more than tol (interpreted as a fraction if use_percent_tol is true, or as an absolute length if false) are considered similar. If on_similar_vols is true, only curves on volumes with the same geometry as the volume owning curve_ids are compared.

`.. code-block:: python`

```
                # Create a cylinder of radius 1
                and height 2; its circular edges are curves
      1 (back)
                and 2 (front)


                cubit.cmd
                ("create cylinder radius 1
                height 2" )


                # Find curves similar to curve
                1.


                matches = cubit.get_similar_curves
                ([1])


                print(matches) # prints (1,
                2)
```

`@n type of curve_ids:  std::vector< int,std::allocator< int > >`

Parameters

curve_ids List of curve IDs to compare against.
type of tol: float, optional

tol Tolerance for length comparison (fraction if use_percent_tol is true, otherwise absolute).
type of use_percent_tol: boolean, optional

use_percent_tol If true, interpret tol as a fraction of the curve length; otherwise interpret as an absolute length.
type of on_similar_vols: boolean, optional

on_similar_vols If true, restrict comparison to curves on volumes similar to the volume owning curve_ids.
return type of : std::vector< int,std:: allocator< int > >

Returns

A list of curve IDs whose lengths match within tolerance.

◆ get_similar_surfaces()

def cubit.get_similar_surfaces (*surface_ids*,
tol = `1e-3`,
use_percent_tol = `True`,
on_similar_vols = `True`
)

Find surfaces with similar area and curve count to given surfaces.

This function compares each surface in surface_ids against all other surfaces in the model. Surfaces whose areas differ by no more than tol (interpreted as a fraction if use_percent_tol is true, or as an absolute area if false) and that have the same number of bounding curves are considered similar. If on_similar_vols is true, only surfaces on volumes with the same geometry as the volumes owning surface_ids are compared.

`.. code-block:: python`

```
                # Create a cylinder of radius 1
                and height 2; its circular end faces are
    surfaces 2 and
                3.


                cubit.cmd
                ("create cylinder radius 1
                height 2" )



                # Find surfaces similar to
                surface 2.


                matches = cubit.get_similar_surfaces
                ([2])


                print(matches) # prints (2,
                3)
```

`@n type of surface_ids:  std::vector< int,std::allocator< int > >`

Parameters

surface_ids List of surface IDs to compare against.
type of tol: float, optional

tol Tolerance for area comparison (fraction if use_percent_tol is true, otherwise absolute).
type of use_percent_tol: boolean, optional

use_percent_tol If true, interpret tol as a fraction of the surface area; otherwise interpret as an absolute area.
type of on_similar_vols: boolean, optional

on_similar_vols If true, restrict comparison to surfaces on volumes similar to the volumes owning surface_ids.
return type of : std::vector< int,std:: allocator< int > >

Returns

A list of surface IDs whose areas and curve counts match within tolerance.

◆ get_similar_volumes()

def cubit.get_similar_volumes (*volume_ids*,
tol = `1e-3`,
use_percent_tol = `True`
)

Find volumes with similar size and face count to given volumes.

This function compares each volume in volume_ids against all other volumes in the model. Volumes whose volumes differ by no more than tol (interpreted as a fraction if use_percent_tol is true, or as an absolute volume if false) and that have the same number of faces are considered similar.

`.. code-block:: python`

```
# Create two identical bricks of
size 10 (volumes 1 and 2).

cubit.cmd
("brick x 10" )

cubit.cmd
("brick x 10" )

cubit.cmd
("vol 2 move x 20" )
# separate the second
brick


# Find volumes similar to volume
1.

matches = cubit.get_similar_volumes
([1])

print(matches) # prints (1,
2)
```

`@n type of volume_ids:  std::vector< int,std::allocator< int > >`

Parameters

volume_ids List of volume IDs to compare against.
type of tol: float, optional

tol Tolerance for volume comparison (fraction if use_percent_tol is true, otherwise absolute).
type of use_percent_tol: boolean, optional

use_percent_tol If true, interpret tol as a fraction of the volume; otherwise interpret as an absolute volume.
return type of : std::vector< int,std:: allocator< int > >

Returns

A list (vector or Python tuple) of volume IDs whose volumes and face counts match within tolerance.

◆ get_sizing_function_name()

def cubit.get_sizing_function_name (*entity_type*,
entity_id
)

Retrieve the meshing sizing function type for a surface or volume.

Returns the sizing function type assigned to the specified entity. Possible return values: "constant", "curvature", "interval", "inverse", "linear", "super", "test", "exodus", or "none".

`.. code-block:: python`

```
import cubit

# After setting a volume sizing
function:

# cubit.cmd("Volume 2 Sizing
Function Type Curvature");

fn = cubit.get_sizing_function_name
("volume" , 2)

print("Volume 2 sizing
function:" , fn) # prints
"curvature"
```

`@n type of entity_type:  string`

Parameters

entity_type Either "surface" or "volume".
type of entity_id: int

entity_id ID of the surface or volume.
return type of : string

Returns

The sizing function name assigned to that entity.

◆ get_small_and_narrow_surfaces()

def cubit.get_small_and_narrow_surfaces (*target_ids*, *small_area*, *small_curve_size*)

Find surfaces that are either small in area or contain narrow regions.

For each face of volumes in target_ids, measures its area and checks if area <= small_area. Also checks for narrow regions by evaluating pairs of non-adjacent edges: if their distance <= small_curve_size and orientation difference > 15deg, the face is narrow. Returns all face IDs meeting either criterion.

.. code-block:: python

```
volume_ids = [1, 2]

area_thresh = 0.25 # e.g.,
mesh_size * mesh_size.

narrow_thresh = 0.1 #
Edge-distance threshold.

results = cubit.get_small_and_narrow_surfaces
(volume_ids, area_thresh, narrow_thresh)
```

for sid in results: print("Surface ID:", sid)

```
@n type of target_ids:  std::vector< int,std::allocator< int > >
```

Parameters

target_ids Vector of volume IDs to examine.
type of small_area: float

small_area Area threshold; faces with area <= small_area qualify as "small."
type of small_curve_size: float

small_curve_size Distance threshold; faces with any two non-adjacent edges within small_curve_size and orientation difference > 15deg qualify as "narrow."
return type of : std::vector< int,std:: allocator< int > >

Returns

Vector of IDs of faces that are small or narrow.

◆ get_small_curves()

def cubit.get_small_curves (*target_volume_ids*, *mesh_size*)

Find curves with edge length below a threshold within given volumes.

For each curve on faces of volumes in target_volume_ids, measures its edge length. Curves with length <= mesh_size are returned.

.. code-block:: python

```
volume_ids = [1, 2]

mesh_size = 0.5 # Edge-length
threshold.

small_curves = cubit.get_small_curves
(volume_ids, mesh_size)
```

for cid in small_curves: print("Small curve ID:", cid)

```
@n type of target_volume_ids:  std::vector< int,std::allocator< int > >
```

Parameters

target_volume_ids Vector of volume IDs to examine.
type of mesh_size: float

mesh_size Edge-length threshold for "small" curves.
return type of : std::vector< int,std:: allocator< int > >

Returns

Vector of IDs of curves with length <= mesh_size.

◆ get_small_radius_blend_surfaces()

def cubit.get_small_radius_blend_surfaces (*target_volume_ids*, *max_radius*)

Find blend surfaces with radius of curvature <= max_radius.

Iterates over all faces of volumes in target_volume_ids and returns those blend surfaces whose radius of curvature is <= max_radius. If max_radius = 0, all blend surfaces are returned.

.. code-block:: python

```
                # Create a 10x10x10 brick and
                apply two blends: one with radius 0.5 and one with
                radius 2.


                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("modify curve 1 blend
                radius 0.5" ) # blend
                surface 7


                cubit.cmd
                ("modify curve 2 blend
                radius 2.0" ) # blend
                surface 8


                # Get only the small-radius
                blends (<= 1.0).


                max_radius = 1.0


                small_blends =
      cubit.get_small_radius_blend_surfaces
                ([1], max_radius)


                print(small_blends) # expect
                [7]
```

@n type of target_volume_ids: std::vector< int,std::allocator< int > >

Parameters

target_volume_ids Vector of volume IDs to examine.
type of max_radius: float

max_radius Maximum radius of curvature; if 0, return all blend surfaces.
return type of : std::vector< int,std:: allocator< int > >

Returns

Vector of blend surface IDs with radius <= max_radius.

◆ get_small_surfaces()

def cubit.get_small_surfaces (*target_volume_ids*, *area_threshold*)

Find surfaces with area below a given threshold.

Gathers all unique faces from the specified volumes, measures each face's area, and returns those with area <= area_threshold.

.. code-block:: python

```
volume_ids = [1, 2]

area_threshold = 0.25 # e.g.,
mesh_size * mesh_size.

small_surfaces = cubit.get_small_surfaces
(volume_ids, area_threshold)
```

for sid in small_surfaces: print("Small surface ID:", sid)

```
@n type of target_volume_ids:  std::vector< int,std::allocator< int > >
```

Parameters

target_volume_ids Vector of volume IDs to examine.
type of area_threshold: float

area_threshold Area threshold; faces with area <= area_threshold are returned.
return type of : std::vector< int,std:: allocator< int > >

Returns

Vector of IDs of surfaces with area <= area_threshold.

◆ get_small_surfaces_HR()

def cubit.get_small_surfaces_HR (*target_volume_ids*, *mesh_size*)

Python-callable version: identify small hydraulic-radius surfaces.

Computes hydraulic radius = 4*(area/perimeter) for each face in target_volume_ids. Returns IDs of faces with hydraulic radius <= mesh_size.

.. code-block:: python

```
volume_ids = [1, 2]

mesh_size = 0.5 # Hydraulic
radius threshold.

small_surfaces = cubit.get_small_surfaces_HR
(volume_ids, mesh_size)
```

for sid in small_surfaces: print("Small surface ID:", sid)

```
@n type of target_volume_ids:  std::vector< int,std::allocator< int > >
```

Parameters

target_volume_ids Vector of volume IDs to examine.
type of mesh_size: float

mesh_size Hydraulic radius threshold (faces with 4* (area/perimeter) <= mesh_size are "small").
return type of : std::vector< int,std:: allocator< int > >

Returns

Vector of IDs of small hydraulic-radius surfaces.

◆ get_small_volumes()

def cubit.get_small_volumes (*target_volume_ids*, *mesh_size*)

Find volumes whose size is below a threshold based on mesh size.

Volumes with actual volume < 10 * mesh_size^3 are considered "small".

```
.. code-block:: python
```

```
                # Create two small and one larger
                brick.


                cubit.cmd
                ("brick x 0.5 y 0.5 z
                0.5" ) # Vol 1


                cubit.cmd
                ("brick x 1 y 1 z 1"
                ) # Vol 2


                cubit.cmd
                ("brick x 3 y 3 z 3"
                ) # Vol 3


                mesh_size = 1.0 # 10 * 1^3 =
                10.


                small_vols = cubit.get_small_volumes
                ([1, 2, 3], mesh_size)


                print(small_vols) # expect [1,
                2]
```

```
 @n type of target_volume_ids:  std::vector< int,std::allocator< int > >
```

Parameters

target_volume_ids Vector of volume IDs to examine.
type of mesh_size: float

mesh_size Threshold defining "small": volumes with volume < 10 * mesh_size^3.
return type of : std::vector< int,std:: allocator< int > >

Returns

Vector of IDs of volumes that meet the "small" criterion.

◆ get_smallest_curves()

def cubit.get_smallest_curves (*target_volume_ids*, *number_to_return*)

Return IDs of the smallest curves in the specified volumes.

Measures the length of each curve on faces of volumes in target_volume_ids and returns the number_to_return curves with the shortest lengths.

```
.. code-block:: python
```

```
                volume_ids = [1, 2]


                number_to_return = 5 # Get five
                shortest curves.


                smallest_curves = cubit.get_smallest_curves
                (volume_ids, number_to_return)
```

for cid in smallest_curves: print("Curve ID:", cid)

```
  @n type of target_volume_ids:  std::vector< int,std::allocator< int > >
```

Parameters

target_volume_ids Vector of volume IDs to examine.
type of number_to_return: int

number_to_return Number of smallest curves to return.
return type of : std::vector< int,std:: allocator< int > >

Returns

Vector of IDs of the smallest curves.

◆ get_smooth_scheme()

def cubit.get_smooth_scheme (*geometry_type*, *entity_id*)

Retrieve the smoothing scheme for a specified geometry entity.

This returns the name of the smoothing scheme applied to the given entity ("curve", "surface", or "volume") and ID. If none is set, returns an empty string.

.. code-block:: python

```
cubit.cmd
("surface 10 smooth scheme
mean ratio" )

scheme = cubit.get_smooth_scheme
("surface" , 10)

print("Smooth
scheme:" , scheme)
```

@n type of geometry_type: string

Parameters

geometry_type The geometry type: "curve", "surface", or "volume".
type of entity_id: int

entity_id The ID of the entity.
return type of : string

Returns

The smoothing scheme name, or an empty string if none is set.

◆ get_solutions_for_bad_geometry()

def cubit.get_solutions_for_bad_geometry (*geom_type*, *geom_id*)

Provide remedy for bad geometry via ACIS healing (deprecated).

Bad geometry often results from imperfect CAD translations. This function invokes ACIS's built-in healing operation.

Note

ACIS healing support is deprecated and will be removed in a future release. Alternative solutions for bad geometry are pending.

type of geom_type: string

Parameters

geom_type Type of entity: "curve", "surface", "volume", or "body".
type of geom_id: int

geom_id ID of the geometry entity to query.
return type of : std::vector< std::vector< std::string,std::allocator< std::string > >,std::allocator< std::vector< std::string,std::allocator< std:: string > > > >

Returns

A vector of three string vectors: [0] Display string, [1] Command string, [2] Preview string.

◆ get_solutions_for_blends()

def cubit.get_solutions_for_blends (*surface_id*)

Provide possible blend removal solutions for a given surface (and its blend chain if applicable).

Given a surface ID, this function returns possible solutions for removing or adjusting blends on that surface. If the surface is part of a blend chain, the solutions will include operations for the entire blend chain as well.

```
.. code-block:: python
```

```
                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("modify curve 12 2 8 11
                blend radius 1" ) # Blend
                the four curves on surface 5


                # Get blend removal solutions for
                surface 10:

                surface_id = 10


                solutions =
      cubit.get_solutions_for_blends
                (surface_id)
```

for i in range(len(solutions[0])): print("Option:", solutions[0][i])

```
                print(" Command:" ,
                solutions[1][i])

                print(" Preview:" ,
                solutions[2][i])

                print(" Shorthand:"
                , solutions[3][i])


                # Expected output:

                # Option: Remove Blend Chain at
                Surface 10

                # Command: remove surface 10
                blend_chain

                # Preview: remove surface 10
                blend_chain preview

                # Shorthand: 15 10
```

```
 @n type of surface_id:  int
```

Parameters

surface_id The surface being queried. If part of a blend chain, solutions will include the full chain. return type of : std::vector< std::vector< std::string,std::allocator< std::string > >,std::allocator< std::vector< std::string,std::allocator< std:: string > > > >

Returns

A vector of four string vectors: [0] Display strings for each recommended blend solution, [1] Cubit command strings to apply each blend solution, [2] Cubit preview command strings, [3] Shorthand descriptions (operation ID and involved surface IDs).

◆ get_solutions_for_bolt()

def cubit.get_solutions_for_bolt (*bolt_id*, *insert_id*, *threaded_vol_id*)

Suggests operations for a volume classified as a bolt using known subcomponent IDs.

This function provides modification options for a bolt volume using known insert and threaded subcomponent volumes. It is more efficient than get_solutions_for_classified_volume when those IDs are already available.

Set insert_id or threaded_vol_id to 0 if the corresponding volume is not present or not known. To classify bolt volumes in the model, use **get_ML_classification()** .

```
.. code-block:: python
```

```
                cubit.cmd
                ("cylinder radius 1 height
                10" ) # volume 1


                cubit.cmd
                ("cylinder radius 2 height
                2" ) # volume 2


                cubit.cmd
                ("volume 2 move z 5"
                )


                cubit.cmd
                ("unite volume all"
                ) # volume 1 becomes a
                bolt


                solutions =
      cubit.get_solutions_for_bolt
                (1, 0, 0)
```

for i in range(3): print("Option:", solutions[0][i])

```
                print(" Command:" ,
                solutions[1][i])


                print(" Preview:" ,
                solutions[2][i])


                print(" ML Op:" ,
                solutions[3][i])


                # Expected output (first three
                results only):


                # Option: Delete Bolt Volume
                1


                # Command: delete volume 1


                # Preview: highlight volume
                1


                # ML Op: 40 1 0 0


                # Option: Reduce Bolt 1
                Simplify


                # Command: reduce volume 1


                # Preview: highlight volume
                1


                # ML Op: 42 1 0 0 1 1 0


                # Option: Reduce Bolt 1 Tight
                Fit


                # Command: reduce volume 1 bolt
                fit_volume tight_fit remove_key merge


                # Preview: highlight volume
                1


                # ML Op: 42 1 0 0 1 1 1
```

```
 @n type of bolt_id:  int
```

Parameters

bolt_id	ID of the volume classified as a bolt. type of insert_id: int
insert_id	Optional ID of the insert volume at the bolt interface. Use 0 if absent. type of threaded_vol_id: int
threaded_vol_id	Optional ID of the threaded (lower) volume. Use 0 if absent or unknown. return type of : std::vector< std::vector< std::string,std::allocator< std::string > >,std::allocator< std::vector< std::string,std::allocator< std:: string > > > >

Returns

A vector of four aligned string vectors: [0] Display strings for each recommended operation, [1] Cubit commands to execute the operations, [2] Preview commands for visualization, [3] Operation strings formatted for machine learning tools. In Python, the result is a tuple of lists.

◆ get_solutions_for_bolt_hole()

def cubit.get_solutions_for_bolt_hole (*bearing_hole*, *threaded_holes*)

Suggests repair or modification options for a set of concentric fastener pilot holes.

Given a bearing hole surface ID and a list of concentric threaded hole surface IDs, this function identifies potential repair or optimization operations.

.. code-block:: python

```
                cubit.cmd
                ("brick x 10 y 10 z
                2" )

                cubit.cmd
                ("brick x 10 y 10 z
                2" )

                cubit.cmd
                ("volume 2 move z 2"
                )

                cubit.cmd
                ("cylinder radius 1 height
                10" )

                cubit.cmd
                ("subtract volume 3 from
                volume 1 2" )

                threaded_holes = [19]

                solutions =
    cubit.get_solutions_for_bolt_hole
                (16, threaded_holes)
```

for i in range(3): print("Option:", solutions[0][i])

```
print(" Command:" ,
solutions[1][i])


print(" Preview:" ,
solutions[2][i])


print(" Shorthand:"
, solutions[3][i])


# Expected output (first three
results only):


# Option: Delete Pilot Hole at
Surfaces 16 19


# Command: remove surface 16 19
include hole


# Preview: highlight surface 16
19


# Shorthand: 57 16 19


# Option: Reduce Hole at Surfaces
16 19 Spider Wagon Wheel


# Command: reduce upper surface
16 lower surface 19 hole spider wagon
    wheel mesh


# Preview: highlight surface 16
19


# Shorthand: 58 16 19 3 1
0


# Option: Reduce Hole at Surfaces
16 19 Spider J2G


# Command: reduce upper surface
16 lower surface 19 hole spider j2g
    mesh


# Preview: highlight surface 16
19


# Shorthand: 58 16 19 3 2
0
```

```
@n type of bearing_hole:  int
```

Parameters

bearing_hole	ID of the surface on the bearing hole (one per hole). type of threaded_holes: std::vector< int,std::allocator< int > >
threaded_holes	List of surface IDs from concentric threaded holes (one per hole). return type of : std::vector< std::vector< std::string,std::allocator< std::string > >,std::allocator< std::vector< std::string,std::allocator< std:: string > > > >

Returns

A vector of four aligned string vectors: [0] Display strings for each proposed operation, [1] Cubit commands to execute them, [2] Preview commands for visualization, [3] Shorthand strings identifying operation and surface IDs. In Python, the result is a tuple of lists.

◆ get_solutions_for_cavity_surface()

def cubit.get_solutions_for_cavity_surface (*surface_id*)

Provide possible cavity removal or adjustment solutions for a given surface (and its cavity if applicable).

Given a surface ID, this function returns possible solutions for removing or adjusting cavity geometry on that surface. If the surface is part of a cavity, the solutions will include operations for the entire cavity as well.

.. code-block:: python

```
cubit.cmd
("brick x 10" )


cubit.cmd
("brick x 5" )


cubit.cmd
("vol 5 move x 5" )


cubit.cmd
("subtract volume 2 from
volume 1" )


# Get cavity removal solutions
for surface 17:


surface_id = 17


solutions =
cubit.get_solutions_for_cavity_surface
(surface_id)
```

for i in range(len(solutions[0])): print("Option:", solutions[0][i])

```
print(" Command:" ,
solutions[1][i])


print(" Preview:" ,
solutions[2][i])


# Expected output:

# Option: Remove Cavity at
Surface 17


# Command: remove surface 17
cavity


# Preview: remove surface 17
cavity preview
```

```
@n type of surface_id:  int
```

Parameters

surface_id The surface being queried. If part of a cavity, solutions will include the full cavity.
return type of : std::vector< std::vector< std::string,std::allocator< std::string > >,std::allocator< std::vector< std::string,std::allocator< std:: string > > > >

Returns

A vector of three string vectors: [0] Display strings for each recommended cavity solution, [1] Cubit command strings to apply each cavity solution, [2] Cubit preview command strings.

◆ get_solutions_for_classified_surface()

def cubit.get_solutions_for_classified_surface (*classification, surf_id*)

Suggests remedies or modifications for a classified surface.

.. code-block:: python

```
#Subtract a smaller cylinder from
a larger one.


cubit.cmd
("cylinder rad 10 height
3" ) # volume 1


cubit.cmd
("cylinder rad 8 height
3" ) # volume 2


cubit.cmd
("subtract vol 2 from vol
1" ) # creates annulus
surface 9


cubit.cmd
("classify surface 9
\"slot\"" ) # machine
learning operation


solutions =
cubit.get_solutions_for_classified_surface
("slot" , 9)
```

for disp, cmd, prev, shrt in zip(solutions[0], solutions[1], solutions[2], solutions[3]): print("Option:", disp)

```
print(" Command:" ,
cmd)


print(" Preview:" ,
prev)


print(" Shorthand:"
, shrt)
```

```
@n type of classification:  string
```

Parameters

classification Type assigned to the surface (e.g., "slot", "hole", "fillet").
type of surf_id: int

surf_id ID of the classified surface.
return type of : std::vector< std::vector< std::string,std::allocator< std::string > >,std::allocator< std::vector< std::string,std::allocator< std:: string > > > >

Returns

A vector of four string vectors: [0] Display strings describing each remedy, [1] Cubit command strings to execute each remedy, [2] Preview commands for visualization, [3] Shorthand operation descriptors.

◆ get_solutions_for_classified_volume()

def cubit.get_solutions_for_classified_volume (*classification, vol_id*)

Suggests geometry fixes or feature operations for a volume based on its classification.

Given a classification type and a volume ID, this function generates recommended modifications.

To obtain classifications from Cubit's ML system, use **get_ML_classification()** .

```
.. code-block:: python
```

```
cubit.cmd
("cylinder radius 1 height
10" ) # volume 1


cubit.cmd
("cylinder radius 2 height
2" ) # volume 2


cubit.cmd
("volume 2 move z 5"
)


cubit.cmd
("unite volume all"
) # volume 1 becomes a
bolt


solutions =
cubit.get_solutions_for_classified_volume
("bolt" , 1)
```

for i in range(3): print("Option:", solutions[0][i])

```
                print(" Command:" ,
                solutions[1][i])


                print(" Preview:" ,
                solutions[2][i])


                print(" ML Op:" ,
                solutions[3][i])


                # Expected output (first three
                results only):


                # Option: Delete Bolt Volume
                1


                # Command: delete volume 1


                # Preview: highlight volume
                1


                # ML Op: 40 1 0


                # Option: Reduce Bolt 1
                Simplify


                # Command: reduce volume 1


                # Preview: highlight volume
                1


                # ML Op: 42 1 0


                # Option: Reduce Bolt 1 Tight
                Fit


                # Command: reduce volume 1 bolt
                fit_volume tight_fit remove_key merge


                # Preview: highlight volume
                1


                # ML Op: 42 1 0 1 1 0
```

```
@n type of classification:  string
```

Parameters

classification Classification type: one of "bolt", "nut", "washer", "spring", "ball", "race", "pin", "gear", "insert", or "other".
type of vol_id: int

vol_id ID of the volume to process.
return type of : std::vector< std::vector< std::string,std::allocator< std::string > >,std::allocator< std::vector< std::string,std::allocator< std:: string > > > >

Returns

A vector of four aligned string vectors: [0] Display strings for each suggested operation, [1] Cubit command strings to apply the operations, [2] Preview command strings for visualization, [3] Machine-learning-formatted operation strings.

◆ get_solutions_for_close_loop()

def cubit.get_solutions_for_close_loop (*surface_id*, *mesh_size*)

Retrieve recommended remedies for a close-loop surface to resolve narrow regions.

.. code-block:: python

```
cubit.cmd
("cylinder radius 3 height
10" )

cubit.cmd
("modify curve 2 blend
radius 0.5" )

mesh_size = 1.0

solutions =
cubit.get_solutions_for_close_loop
(4, mesh_size)
```

for disp, cmd, prev, shrt in zip(solutions[0], solutions[1], solutions[2], solutions[3]): print("Option:", disp)

```
print(" Command:" ,
cmd)

print(" Preview:" ,
prev)

print(" Shorthand:"
, shrt)

# Expected output:

# Option: Remove Surface 4

# Command: remove surface 4
extend

# Preview: remove surface 4
extend preview

# Shorthand: 5 4

# Option: Composite Surfaces 4
and 3

# Command: composite create
surface 4 3 keep angle 180.0

# Preview: highlight surface 4
3

# Shorthand: 7 4 3

# Option: Composite Surfaces 4
and 1

# Command: composite create
surface 4 1 keep angle 180.0

# Preview: highlight surface 4
1

# Shorthand: 7 4 1
```

```
@n type of surface_id:  int
```

Parameters

surface_id ID of the close-loop surface being queried.
type of mesh_size: float

mesh_size Distance threshold that defines a narrow loop region.
return type of : std::vector< std::vector< std::string,std::allocator< std::string > >,std::allocator< std::vector< std::string,std::allocator< std:: string > > > >

Returns

A vector of four string vectors: [0] Display strings for each remedy, [1] Cubit command strings to apply each remedy, [2] Cubit preview command strings, [3] Shorthand descriptions (operation ID and entity IDs) for reconstructing the operation.

◆ get_solutions_for_cone_surface()

def cubit.get_solutions_for_cone_surface (*surface_id*)

Suggests remedies for a surface classified as a cone.

Returns recommended operations for a conical surface, including human-readable descriptions, Cubit command strings to apply them, preview commands for visualization, and shorthand strings for serialization or machine learning workflows.

.. code-block:: python

```
                cubit.cmd
                ("create frustum height 2
                radius 2 top 0" )


                cubit.cmd
                ("cylinder rad 2 height
                10" )


                cubit.cmd
                ("vol 2 mov z -6" )


                cubit.cmd
                ("unite vol all" )


                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("subtract vol 1 from vol
                3" ) # cone surface
                becomes surface 13


                solutions =
      cubit.get_solutions_for_cone_surface
                (13)
```

for disp, cmd, prev, shrt in zip(solutions[0], solutions[1], solutions[2], solutions[3]): print("Option:", disp)

```
                print(" Command:" ,
                cmd)


                print(" Preview:" ,
                prev)


                print(" Shorthand:"
                , shrt)



                # Output:


                # Option: Remove Cone Surface
                13


                # Command: tweak surface 13
                cone


                # Preview: highlight surface
                13


                # Shorthand: 13 13


                # Option: Composite Surfaces 13
                and 12


                # Command: composite create
                surface 13 12 keep angle 180.0


                # Preview: highlight surface 13
                12


                # Shorthand: 7 13 12
```

```
@n type of surface_id:  int
```

Parameters

surface_id ID of the conical surface.
return type of : std::vector< std::vector< std::string,std::allocator< std::string > >,std::allocator< std::vector< std::string,std::allocator< std:: string > > > >

Returns

A vector of four string vectors: [0] Display strings for each recommended operation, [1] Cubit command strings, [2] Preview command strings, [3] Shorthand operation descriptors.

Note

In Python, returned vectors will be tuples.

◆ get_solutions_for_decomposition()

def cubit.get_solutions_for_decomposition (*volume_list*, *exterior_angle*, *do_imprint_merge*, *tolerant_imprint*)

Provide possible decomposition solutions for volumes based on exterior angle criteria.

Given a list of volumes and an exterior angle threshold, this function suggests possible decompositions. Optionally, imprinting and merging can be performed to support the decomposition process.

.. code-block:: python

```
cubit.cmd
("brick x 10 y 10 z
2" ) # Volume 1


cubit.cmd
("brick x 10 y 2 z
10" ) # Volume 2


cubit.cmd
("volume 2 move y
-5" )


cubit.cmd
("unite volume all"
)



volume_list = [1]


exterior_angle = 135.0


do_imprint_merge = False


tolerant_imprint = False



solutions =
cubit.get_solutions_for_decomposition
    (volume_list, exterior_angle,
do_imprint_merge,
    tolerant_imprint)
```

for i in range(len(solutions[0])): print("Option:", solutions[0][i])

```
print(" Command:" ,
solutions[1][i])


print(" Preview:" ,
solutions[2][i])



# Expected output
(partial):


# Option: Webcut Volume 1 With
Sheet Extended From Surface 17


# Command: webcut volume 1 with
sheet extended from surface 17 imprint merge
include_neighbors


# Preview: webcut volume 1 with
sheet extended from surface 17 imprint merge
include_neighbors preview


# Option: Webcut Volume 1 With
Sheet Extended From Surface 16


# Command: webcut volume 1 with
sheet extended from surface 16 imprint merge
include_neighbors


# Preview: webcut volume 1 with
sheet extended from surface 16 imprint merge
include_neighbors preview


# ... additional options
...
```

@n type of volume_list: std::vector< int,std::allocator< int > >

Parameters

volume_list	List of volume IDs to query. type of exterior_angle: float
exterior_angle	Threshold value for the exterior angle (in degrees) used to determine decomposition faces. type of do_imprint_merge: boolean
do_imprint_merge	If true, perform imprint and merge operations to support decomposition. type of tolerant_imprint: boolean
tolerant_imprint	If true, perform tolerant imprinting. return type of : std::vector< std::vector< std::string,std::allocator< std::string > >,std::allocator< std::vector< std::string,std::allocator< std:: string > > > >

Returns

A vector of four string vectors: [0] Display strings for each recommended decomposition option, [1] Cubit command strings to apply each decomposition option, [2] Cubit preview command strings, [3] Shorthand descriptions (operation ID and involved

volume or surface IDs). This may be empty depending on the solution.

◆ get_solutions_for_imprint_merge()

def cubit.get_solutions_for_imprint_merge (*surface_id1*, *surface_id2*)

Suggests remedies for imprint/merge when two overlapping surfaces are on different volumes.

Given two overlapping surface IDs from different volumes, this function provides suggested operations to resolve the overlap.

To detect overlapping surfaces beforehand, use get_overlapping_surfaces_at_surface (for a single surface) or get_overlapping_surfaces_in_volumes (for sets of volumes).

.. code-block:: python

```
                cubit.cmd
                ("brick x 10" )

                cubit.cmd
                ("cylinder radius 3 height
                10" )

                cubit.cmd
                ("volume 2 move z
                10" )

                solutions =
      cubit.get_solutions_for_imprint_merge
                (1, 8)
```

for i in range(len(solutions[0])): print("Option:", solutions[0][i])

```
                print(" Command:" ,
                solutions[1][i])

                print(" Preview:" ,
                solutions[2][i])

                # Expected output (3
                remedies):

                # Option: Imprint with curves
                13

                # Command: imprint tolerant
                surface 1 with curve 13 merge

                # Preview: draw surface 1 8
                overlap;highlight curve 13

                # Option: Auto Fix

                # Command: merge surface 1 8
                auto_imprint

                # Preview:

                # Option: Tolerant Imprint Owning
                Volumes

                # Command: imprint tolerant
                volume 1 2 merge

                # Preview:
```

```
@n type of surface_id1:  int
```

Parameters

surface_id1 ID of the first overlapping surface (must be on a different volume than surface_id2).
type of surface_id2: int

surface_id2 ID of the second overlapping surface (must be on a different volume than surface_id1).
return type of : std::vector< std::vector< std::string,std::allocator< std::string > >,std::allocator< std::vector< std::string,std::allocator< std:: string > > > >

Returns

A vector of four aligned string vectors: [0] Display strings for each recommended remedy, [1] Cubit command strings to apply each remedy, [2] Cubit preview command strings for each remedy, [3] Shorthand descriptions (operation ID and surface/volume IDs; may be empty).

◆ get_solutions_for_near_coincident_vertex_and_curve()

def cubit.get_solutions_for_near_coincident_vertex_and_curve (*vertex_id*, *curve_id*)

Suggests remedies when a vertex lies nearly on a curve from a different volume.

Given a vertex and a nearby curve from different volumes, this function identifies operations to resolve the near-coincidence without introducing geometry issues.

To detect such vertex-curve pairs, use get_close_vertex_curve_pairs.

.. code-block:: python

```
                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("volume 2 move x 10 y 5 z
                0.01" )


                solutions =
      cubit.get_solutions_for_near_coincident_vertex_and_curve
                (12, 1)
```

for i in range(3): print("Option:", solutions[0][i])

```
                print(" Command:" ,
                solutions[1][i])


                print(" Preview:" ,
                solutions[2][i])


                print(" Shorthand:"
                , solutions[3][i])


                # Expected output (first three
                results only):


                # Option: tweak surfaces in
                vertex 12 to curve 1


                # Command: tweak surface in
                vertex 12 move vertex 12 location 5 0 5


                # Preview: tweak surface in
                vertex 12 move vertex 12 location 5 0 5
      preview


                # Shorthand: 47 12 1 5 0 5


                # Option: tweak surfaces in curve
                1 to vertex 12


                # Command: tweak surface in curve
                1 move curve 1 location vertex 12


                # Preview: tweak surface in curve
                1 move curve 1 location vertex 12 preview


                # Shorthand: 48 1 12 0 0 0


                # Option: move volume 2 from
                vertex 12 to curve 1


                # Command: move vertex 12
                location 5 0 5 include_merged


                # Preview: move vertex 12
                location 5 0 5 include_merged preview


                # Shorthand: 52 12 1 5 0 5
```

```
 @n type of vertex_id:  int
```

Parameters

vertex_id ID of the vertex that is nearly coincident with the curve (must be on a different volume).
type of curve_id: int

curve_id ID of the curve that lies near the vertex (must be on a different volume).
return type of : std::vector< std::vector< std::string,std::allocator< std::string > >,std::allocator< std::vector< std::string,std::allocator< std:: string > > > >

Returns

A vector of four aligned string vectors: [0] Display strings for each recommended remedy, [1] Cubit command strings to apply each remedy, [2] Cubit preview command strings for each remedy, [3] Shorthand descriptions (operation ID and vertex/curve IDs).

◆

get_solutions_for_near_coincident_vertex_and_surface()

def cubit.get_solutions_for_near_coincident_vertex_and_surface (*vertex_id*, *surface_id*)

Suggests remedies when a vertex lies nearly on a surface from a different volume.

Given a vertex and a nearby surface from different volumes, this function identifies potential fixes such as tweaks or volume adjustments.

To identify near-coincident vertex-surface pairs, use get_close_vertex_surface_pairs.

.. code-block:: python

```
                cubit.cmd
                ("brick x 10" )

                cubit.cmd
                ("brick x 10" )

                cubit.cmd
                ("volume 2 move x 10.01 y 5
                z 5" )

                solutions =
    cubit.get_solutions_for_near_coincident_vertex_and_surface
                (15, 6)
```

for i in range(3): print("Option:", solutions[0][i])

```
                print(" Command:" ,
                solutions[1][i])

                print(" Preview:" ,
                solutions[2][i])

                print(" Shorthand:"
                , solutions[3][i])

                # Expected output (first three
                results only):

                # Option: tweak surfaces in
                vertex 15 to surface 6

                # Command: tweak surface in
                vertex 15 move vertex 15 location 5 0 0

                # Preview: tweak surface in
                vertex 15 move vertex 15 location 5 0 0
    preview

                # Shorthand: 49 15 6 5 0 0

                # Option: tweak surface 6 to
                vertex 15

                # Command: tweak surface 6 move
                surface 6 location vertex 15

                # Preview: tweak surface 6 move
                surface 6 location vertex 15 preview

                # Shorthand: 50 6 15 0 0 0

                # Option: move volume 2 from
                vertex 15 to surface 6

                # Command: move vertex 15
                location 5 0 0 include_merged

                # Preview: move vertex 15
                location 5 0 0 include_merged preview

                # Shorthand: 54 15 6 5 0 0
```

```
@n type of vertex_id:  int
```

Parameters

vertex_id ID of the vertex that is nearly coincident with the surface (must be on a different volume).
type of surface_id: int

surface_id ID of the surface that lies near the vertex (must be on a different volume).
return type of : std::vector< std::vector< std::string,std::allocator< std::string > >,std::allocator< std::vector< std::string,std::allocator< std:: string > > > >

Returns

A vector of four aligned string vectors: [0] Display strings for each recommended remedy, [1] Cubit command strings to apply each remedy, [2] Cubit preview command strings for each remedy, [3] Shorthand descriptions (operation ID and vertex/surface IDs).

◆ get_solutions_for_near_coincident_vertices()

def cubit.get_solutions_for_near_coincident_vertices (*vertex_id_1*, *vertex_id_2*)

Provide remedies for two nearly coincident vertices on different volumes to resolve gaps or misalignments.

Vertices must belong to different volumes.

```
.. code-block:: python
```

```
                cubit.cmd
                ("brick x 10" )

                cubit.cmd
                ("brick x 10" )

                cubit.cmd
                ("volume 2 move x 10 y
                0.01" )

                solutions =
    cubit.get_solutions_for_near_coincident_vertices
                (2, 11)
```

for disp, cmd, prev, shrt in zip(solutions[0][:3], solutions[1][:3], solutions[2][:3], solutions[3][:3]): print("Option:", disp)

```
                print(" Command:" ,
                cmd)

                print(" Preview:" ,
                prev)

                print(" Shorthand:"
                , shrt)

                print("and more" )

                # Expected output:

                # Option: tweak surfaces in
                vertex 2 to vertex 11

                # Command: tweak surface in
                vertex 2 move vertex 2 location vertex
    11

                # Preview: tweak surface in
                vertex 2 move vertex 2 location vertex
    11
                preview

                # Shorthand: 46 2 11 0 0 0

                # Option: move volume 1 from
                vertex 2 to vertex 11

                # Command: move vertex 2 location
                vertex 11 include_merged

                # Preview: move vertex 2 location
                vertex 11 include_merged preview

                # Shorthand: 51 2 11 0 0 0

                # Option: tweak surfaces in
                vertex 11 to vertex 2

                # Command: tweak surface in
                vertex 11 move vertex 11 location
    vertex 2

                # Preview: tweak surface in
                vertex 11 move vertex 11 location
    vertex 2
                preview

                # Shorthand: 46 11 2 0 0 0

                # and more
```

```
@n type of vertex_id_1:  int
```

Parameters

vertex_id_1 ID of the first vertex (on volume 1).
type of vertex_id_2: int

vertex_id_2 ID of the second vertex (on volume 2).
return type of : std::vector< std::vector< std::string,std::allocator< std::string > >,std::allocator< std::vector< std::string,std::allocator< std:: string > > > >

Returns

A vector of four string vectors: [0] Display strings for each recommended remedy, [1] Cubit command strings to apply each remedy, [2] Cubit preview command strings, [3]

Shorthand descriptions (operation ID and vertex IDs).

get_solutions_for_overlapping_surfaces()

def cubit.get_solutions_for_overlapping_surfaces (*surface_id_1*, *surface_id_2*)

Suggests remedies for two overlapping surfaces on different volumes.

Given two overlapping surface IDs from different volumes that share the same spatial region, this function provides imprint and merge options for resolving the overlap.

To identify overlapping surfaces beforehand, use get_overlapping_surfaces_at_surface or get_overlapping_surfaces_in_volumes.

.. code-block:: python

```
cubit.cmd
("brick x 10" )

cubit.cmd
("brick x 10" )

cubit.cmd
("volume 2 move x
10" )

solutions =
cubit.get_solutions_for_overlapping_surfaces
(6, 10)
```

for i in range(len(solutions[0])): print("Option:", solutions[0][i])

```
print(" Command:" ,
solutions[1][i])


print(" Preview:" ,
solutions[2][i])


print(" Shorthand:"
, solutions[3][i])


# Expected output:


# Option: Imprint Volumes 1
2


# Command: imprint volume 1
2


# Preview: highlight volume 1
2


# Shorthand: 45 1 2


# Option: Imprint & Merge
Volumes 1 2


# Command: imprint volume 1
2;merge volume 1 2


# Preview: highlight volume 1
2


# Shorthand: 44 1 2
```

```
@n type of surface_id_1:  int
```

Parameters

surface_id_1 ID of the first overlapping surface (must be on a different volume than surface_id_2).
type of surface_id_2: int

surface_id_2 ID of the second overlapping surface (must be on a different volume than surface_id_1).
return type of : std::vector< std::vector< std::string,std::allocator< std::string > >,std::allocator< std::vector< std::string,std::allocator< std:: string > > > >

Returns

A vector of four aligned string vectors: [0] Display strings for each recommended remedy, [1] Cubit command strings to apply each remedy, [2] Cubit preview command strings for each remedy, [3] Shorthand descriptions (operation ID and volume IDs).

get_solutions_for_overlapping_volumes()

def cubit.get_solutions_for_overlapping_volumes (*volume_id_1*, *volume_id_2*, *maximum_gap_tolerance*, *maximum_gap_angle*)

Provide remedies for two overlapping volumes based on gap and angular tolerances.

If two volume IDs represent overlapping solids and the overlap exceeds a given gap or angle tolerance, this function suggests operations to resolve the interference.

```
.. code-block:: python
```

```
cubit.cmd
("brick x 10" )


cubit.cmd
("cylinder radius 3 height
10" )


cubit.cmd
("volume 2 move x
9.9" )


solutions =
cubit.get_solutions_for_overlapping_volumes
(1, 2, 0.2, 5.0)
```

for i in range(len(solutions[0])): print("Option:", solutions[0][i])

```
print(" Command:" ,
solutions[1][i])


print(" Preview:" ,
solutions[2][i])


print(" Shorthand:"
, solutions[3][i])


# Expected output:


# Option: Remove Overlap from
Smaller Volume (2)


# Command: remove overlap volume
1 2 modify smaller


# Preview: highlight volume
2


# Shorthand: 41 2 1


# Option: Remove Overlap from
Larger Volume (1)


# Command: remove overlap volume
1 2 modify larger


# Preview: highlight volume
1


# Shorthand: 41 1 2


# Option: Replace Surface 1 with
Surface 8


# Command: tweak surface 1
replace with surface 8


# Preview: tweak surface 1
replace with surface 8 preview


# Shorthand: 6 1 8


# Option: Replace Surface 8 with
Surface 1


# Command: tweak surface 8
replace with surface 1


# Preview: tweak surface 8
replace with surface 1 preview


# Shorthand: 6 8 1
```

```
@n type of volume_id_1:  int
```

Parameters

volume_id_1	ID of the first volume. type of volume_id_2: int
volume_id_2	ID of the second volume. type of maximum_gap_tolerance: float
maximum_gap_tolerance	Maximum allowed gap before volumes are considered overlapping. type of maximum_gap_angle: float
maximum_gap_angle	Maximum allowed angular deviation at overlap regions. return type of : std::vector< std::vector< std::string,std::allocator< std::string > >,std::allocator< std::vector< std::string,std::allocator< std:: string > > > >

Returns

A vector of four string vectors: [0] Display strings for each recommended remedy, [1] Cubit command strings to apply each remedy, [2] Cubit preview command strings, [3] Shorthand descriptions (operation ID and volume or surface IDs).

◆ get_solutions_for_sharp_angle_vertex()

def cubit.get_solutions_for_sharp_angle_vertex (*vertex_id*, *small_curve_size*, *mesh_size*)

Retrieve recommended remedies for a sharp angle at a vertex.

.. code-block:: python

```
                cubit.cmd
                ("brick x 10" )

                cubit.cmd
                ("cylinder rad 5 height
                10" )

                cubit.cmd
                ("vol 2 mov z 10" )

                cubit.cmd
                ("unite vol 1 2" )


                small_curve_size = 0.25

                mesh_size = 1.0

                solutions =
      cubit.get_solutions_for_sharp_angle_vertex
                (12, small_curve_size, mesh_size)
```

for disp, cmd, prev, shrt in zip(solutions[0], solutions[1], solutions[2], solutions[3]): print("Option:", disp)

```
print(" Command:" ,
cmd)


print(" Preview:" ,
prev)


print(" Shorthand:"
, shrt)



# Expected output (first 3
options shown, more options are
returned):


# Option: Blunt Tangency at
Vertex 12 Remove Material using Angle
15


# Command: blunt tangency vertex
12 remove_material angle 15


# Preview: blunt tangency vertex
12 remove_material angle 15 preview


# Shorthand: 12 12 0 1 15
-1


# Option: Blunt Tangency at
Vertex 12 Add Material using Angle 15


# Command: blunt tangency vertex
12 angle 15


# Preview: blunt tangency vertex
12 angle 15 preview


# Shorthand: 12 12 0 0 15
-1


# Option: Blunt Tangency at
Vertex 12 Remove Material using Angle
30


# Command: blunt tangency vertex
12 remove_material angle 30


# Preview: blunt tangency vertex
12 remove_material angle 30 preview


# Shorthand: 12 12 0 1 30
-1


# ...


# (Additional options omitted for
brevity.)
```

◆

get_solutions_for_sheet_volume_connection()

```
def cubit.get_solutions_for_sheet_volume_connection ( args, kwargs )
```

Propose surface-tweak, imprint, and merge operations to bridge two sheet-volume sets.

Unlike get_solutions_for_sheet_volume_connection (single group), this function connects two distinct sheet-volume sets by first merging within each set (if needed) and then suggesting tweaks, imprint, and merge steps between them. Sheets remain separated unless a tweak operation creates material assignments.

```
.. code-block:: python
```

Parameters

vertex_id ID of the vertex with the sharp angle being queried.
type of small_curve_size: float

small_curve_size Threshold that defines what is considered a "small" curve.
type of mesh_size: float

mesh_size Element size used for meshing, used to determine tweak operations.
type of args: std::vector< std::vector< std::string,std::allocator< std:: string > > >

Returns

A vector of four string vectors: [0] Display strings for each remedy, [1] Cubit command strings to apply each remedy, [2] Cubit preview command strings, [3] Shorthand descriptions (operation ID and entity IDs) for reconstructing the operation.

```
import cubit

cubit.cmd
("reset" )

cubit.cmd
("brick x 10 y 1 z
10" )

cubit.cmd
("brick x 1 z 10 y
10" )

cubit.cmd
("vol 2 mov x 5.5" )

cubit.cmd
("reduce vol 1 thin copy
surface 5 loft factor 0 thickness 1" )

cubit.cmd
("reduce vol 2 thin
midsurface surface 10 12 loft factor 0.5
thickness
1" )

v1 = [3]; v2 = [4]

sols =
cubit.get_solutions_for_sheet_volume_connection
(v1, v2, 1.0, 1.0, "" , 0)

print("Option:" ,
sols[0][0])

print(" Command:" ,
sols[1][0])

print(" Preview:" ,
sols[2][0])

print(" Shorthand:"
, sols[3][0])

# Expected output:

# Option: Tweak Curve 25 to Sheet
4 (Surface 14)

# Command: tweak curve 25 target
surface 14;imprint volume 4 with curve
25;merge volume
4 3

# Preview: tweak curve 25 target
surface 14 preview

# Shorthand: 38 25 14 0.5 0
0
```

@n type of voll_sheets: std::vector< int,std::allocator< int > >

Parameters

vol1_sheets IDs of the first sheet-volume set (common parent).
type of vol2_sheets: std::vector< int,std::allocator< int > >

vol2_sheets IDs of the second sheet-volume set (common parent).
type of thickness1: float

thickness1 Thickness of sheets in vol1_sheets.
type of thickness2: float

thickness2 Thickness of sheets in vol2_sheets.
type of close_type: string, optional

close_type Optional entity type to focus near (e.g., "curve"). Default = "".
type of close_id: int, optional

close_id Optional ID of close_type entity. Default = 0.
return type of : std::vector< std::vector< std::string,std::allocator< std::string > >,std::allocator< std::vector< std::string,std::allocator< std:: string > > > >

Returns

A vector of four string vectors: [0] Display labels of each option, [1] Commands to execute each option, [2] Preview commands, [3] Shorthand strings for ML training.

◆ get_solutions_for_sheet_volumes()

def cubit.get_solutions_for_sheet_volumes (*vol_ids*, *thickness*)

Propose surface-extension and imprint/merge operations to connect sheet bodies.

Analyze sheet-body volumes and suggest surface-tweak commands that extend surfaces to fill gaps between them, then propose imprint and merge operations. The sheets remain separate bodies to accommodate unique material assignments.

```
.. code-block:: python

                cubit.cmd
                ("brick x 10 y 1 z
                10" )

                cubit.cmd
                ("brick x 1 z 10 y
                10" )

                cubit.cmd
                ("vol 2 mov x 5.5" )

                cubit.cmd
                ("reduce vol 1 thin copy
                surface 5 loft factor 0 thickness 1" )
                # -> sheet 3

                cubit.cmd
                ("reduce vol 2 thin
                midsurface surface 10 12 loft factor 0.5
    thickness
                1" ) # -> sheet
                4


                vol_ids = [3, 4]

                thickness = [1.0, 1.0]

                solutions =
    cubit.get_solutions_for_sheet_volumes
                (vol_ids, thickness)
```

for disp, cmd, prev, shrt in zip(*solutions): print("Option:", disp)

```
                print(" Command:" ,
                cmd)

                print(" Preview:" ,
                prev)

                print(" Shorthand:"
                , shrt)

                # Expected output (only one
                option shown):

                # Option: Tweak Curve 25 to Sheet
                4 (Surface 14)

                # Command: tweak curve 25 target
                surface 14;imprint volume 4 with curve
    25;merge volume
                4 3

                # Preview: tweak curve 25 target
                surface 14 preview

                # Shorthand: 38 25 14 0.5 0
                0
```

```
 @n type of vol_ids:  std::vector< int,std::allocator< int > >
```

Parameters

vol_ids IDs of the sheet bodies to connect.
type of thickness: std::vector< double,std::allocator< double > >

thickness Corresponding thickness values for each sheet body.
return type of : std::vector< std::vector< std::string,std::allocator< std::string > >,std::allocator< std::vector< std::string,std::allocator< std:: string > > > >

Returns

A vector of four string vectors: [0] Display descriptions of each connection option, [1] Cubit commands to execute the extension, imprint and merge, [2] Cubit preview commands, [3] Shorthand strings (for ML training).

◆ get_solutions_for_small_curves()

def cubit.get_solutions_for_small_curves (*curve_id*, *small_curve_size*, *mesh_size*)

Recommend remedial operations on a curve already flagged as

small.

Operates on a curve marked small by small_curve_size and mesh_size, and suggests surface removals, surface-replacement tweaks, curve collapses to vertices, or topology rebuild operations.

```
.. code-block:: python
```

```
                import cubit

                cubit.cmd
                ("reset" )

                cubit.cmd
                ("brick x 10" )

                cubit.cmd
                ("brick x 0.1" )

                cubit.cmd
                ("vol 2 mov x 5 y 5 z
                5" )

                cubit.cmd
                ("subtract vol 2 from vol
                1" )

                curve_id = 28

                small_curve_size = 0.25

                mesh_size = 1.0

                solutions =
    cubit.get_solutions_for_small_curves
                (

                curve_id, small_curve_size, mesh_size)
```

for disp, cmd, prev, shrt in zip(*solutions)[:3]: print("Option:", disp)

```
                print(" Command:" ,
                cmd)

                print(" Preview:" ,
                prev)

                print(" Shorthand:"
                , shrt)

                # Expected output (first three of
                many):

                # Option: Remove Surface
                13

                # Command: remove surface 13
                extend

                # Preview: remove surface 13
                extend preview

                # Shorthand: 5 13

                # Option: Remove Surface
                16

                # Command: remove surface 16
                extend

                # Preview: remove surface 16
                extend preview

                # Shorthand: 5 16

                # Option: Replace Surface 18 with
                Surface 14

                # Command: tweak surface 18
                replace with surface 14

                # Preview: tweak surface 18
                replace with surface 14 preview

                # Shorthand: 6 18 14

                # ...and more options
```

```
@n type of curve_id:  int
```

Parameters

curve_id	ID of the small curve. type of small_curve_size: float
small_curve_size	Curve-length threshold defining "small" features. type of mesh_size: float
mesh_size	Target mesh element size for context. return type of : std::vector< std::vector< std::string,std::allocator< std::string > >,std::allocator< std::vector< std::string,std::allocator< std:: string > > > >

Returns

A vector of four string vectors: [0] Display labels, [1] Cubit commands, [2] Preview commands, [3] Shorthand strings for

ML training.

◆ get_solutions_for_small_surfaces()

def cubit.get_solutions_for_small_surfaces (*surface_id*, *small_curve_size*, *mesh_size*)

Recommend remedial operations on a surface already flagged as small.

Operates on a surface marked small by small_curve_size and mesh_size, and suggests composite merges, topology rebuilds, surface removals, or surface-replacement tweaks.

.. code-block:: python

```
import cubit

cubit.cmd
("reset" )

cubit.cmd
("brick x 10" )

cubit.cmd
("brick x 0.1" )

cubit.cmd
("vol 2 mov x 5 y 5 z
5" )

cubit.cmd
("subtract vol 2 from vol
1" )

surface_id = 13

solutions =
cubit.get_solutions_for_small_surfaces
(

surface_id, 0.25, 1.0)
```

for disp, cmd, prev, shrt in zip(*solutions)[:3]: print("Option:", disp)

```
print(" Command:" ,
cmd)

print(" Preview:" ,
prev)

print(" Shorthand:"
, shrt)

# ...and more options
```

@n type of surface_id: int

Parameters

surface_id	ID of the surface to remediate. type of small_curve_size: float
small_curve_size	Threshold under which curves are deemed small. type of mesh_size: float
mesh_size	Target mesh element size for context. return type of : std::vector< std::vector< std::string,std::allocator< std::string > >,std::allocator< std::vector< std::string,std::allocator< std:: string > > > >

Returns

Vector of four string vectors: [0] Display labels, [1] Execution commands, [2] Preview commands, [3] Shorthand strings for ML training.

◆ get_solutions_for_surfaces_with_narrow_regions()

def cubit.get_solutions_for_surfaces_with_narrow_regions (*surface_id*, *small_curve_size*, *mesh_size*)

Recommend operations to eliminate narrow regions on a surface identified as narrow.

Analyzes a surface with narrow regions and suggests operations.

type of surface_id: int

Parameters

surface_id	ID of the surface to analyze. type of small_curve_size: float
small_curve_size	Length threshold defining narrow regions. type of mesh_size: float
mesh_size	Target mesh element size for context. return type of : std::vector< std::vector< std::string,std::allocator< std::string > >,std::allocator< std::vector< std::string,std::allocator< std:: string > > > >

Returns

A vector of three string vectors: [0] Display descriptions for each solution, [1] Cubit command strings to execute solutions, [2] Cubit preview command strings. (In Python, returned as a tuple of lists.)

◆ get_solutions_for_thin_volume()

def cubit.get_solutions_for_thin_volume (*vol_id*, *near_vols*, *include_weights* = `False`, *include_type* = `False`)

Generate candidate operations to reduce a thin volume into a sheet body.

Suggest copying or midsurfacing operations (with optional weights/types) to reduce a volume to a shell.

.. code-block:: python

```
cubit.cmd
("brick x 10 y 1 z
10" )

cubit.cmd
("brick x 1 y 10 z
10" )

cubit.cmd
("move volume 2 by x 4.5 y
4.5" )

cubit.cmd
("unite volume all"
)

vol_id = 1

near_vols = []

solutions =
cubit.get_solutions_for_thin_volume
(vol_id, near_vols)
```

for disp, cmd, prev, shrt in zip(*solutions[:3]): print("Option:", disp)

```
                print(" Command:" ,
                cmd)


                print(" Preview:" ,
                prev)


                print(" Shorthand:"
                , shrt)


                print("...and more options
                available" )


                # Expected output for first three
                options:


                # Option: Copy Surfaces 13
                15


                # Command: reduce volume 1 thin
                copy surface 13 15 loft factor 0 0
      thickness 1 1


                # Preview: reduce volume 1 thin
                copy surface 13 15 loft factor 0 0
      thickness 1 1
                preview


                # Shorthand: 17 1 13 2 13 0 1 15
                0 1


                # Option: Copy Surfaces 6
                17


                # Command: reduce volume 1 thin
                copy surface 6 17 loft factor 0 0
      thickness 1 1


                # Preview: reduce volume 1 thin
                copy surface 6 17 loft factor 0 0
      thickness 1 1
                preview


                # Shorthand: 17 1 6 2 6 0 1 17 0
                1


                # Option: Copy Surface 15


                # Command: reduce volume 1 thin
                copy surface 15 loft factor 0 thickness
      1


                # Preview: reduce volume 1 thin
                copy surface 15 loft factor 0 thickness
      1
                preview


                # Shorthand: 17 1 15 1 15 0
                1


                # ...and more options
```

```
@n type of vol_id:  int
```

Parameters

vol_id	ID of the thin volume to convert to a sheet. type of near_vols: std::vector< int,std::allocator< int > >
near_vols	List of neighboring volume IDs to limit influence; empty uses all. type of include_weights: boolean, optional
include_weights	If true, prepend each operation string with heuristic weight (1=best ... 0=worst). type of include_type: boolean, optional
include_type	If true, include connection type code (long_long=4, continuous=3, midsurface=2, copy=1). return type of : std::vector< std::vector< std::string,std::allocator< std::string > >,std::allocator< std::vector< std::string,std::allocator< std:: string > > > >

Returns

A vector of four string vectors: [0] Display labels for each candidate operation, [1] Cubit commands to perform each operation, [2] Cubit preview commands, [3] Shorthand strings encoding type, vol, surfaces, factors, etc. (In Python this returns as a tuple of lists.)

◆ get_solutions_for_volumes()

def cubit.get_solutions_for_volumes (*vol_id*, *small_curve_size*, *mesh_size*)

Recommend remediation operations for volumes with features below a size threshold.

Analyze a given volume to find curves smaller than a user-specified threshold and suggest operations to eliminate those features before meshing.

```
.. code-block:: python
```

```
                cubit.cmd
                ("brick x 10" )

                cubit.cmd
                ("brick x 0.1" )

                vol = 2

                small_curve_size = 0.25

                mesh_size = 1.0

                solutions =
      cubit.get_solutions_for_volumes
                (vol, small_curve_size, mesh_size)
```

for disp, cmd, prev, shrt in zip(*solutions): print("Option:", disp)

```
                print(" Command:" ,
                cmd)

                print(" Preview:" ,
                prev)

                print(" Shorthand:"
                , shrt)

                # Expected output:

                # Option: Delete Volume 2

                # Command: delete volume 2

                # Preview: highlight volume
                2

                # Shorthand: 40 2 0
```

```
 @n type of vol_id:  int
```

Parameters

vol_id	ID of the volume to inspect. type of small_curve_size: float
small_curve_size	Maximum curve length considered "small". type of mesh_size: float
mesh_size	Target mesh element size for the volume. return type of : std::vector< std::vector< std::string,std::allocator< std::string > >,std::allocator< std::vector< std::string,std::allocator< std:: string > > > >

Returns

A vector of four string vectors: [0] Display labels for each remedy option, [1] Cubit commands to execute each remedy, [2] Cubit preview commands, [3] Shorthand operation strings for ML training.

◆ get_source_surfaces()

def cubit.get_source_surfaces (*volume_id*)

Retrieve the list of sweep source surfaces for a specified volume.

When a volume uses the Sweep scheme, source surfaces are the starting faces from which the hexahedral mesh is extruded. This function returns all surface IDs designated as sources. If no explicit source was set, returns an empty list.

```
.. code-block:: python
```

```
                # 1) Create a brick (Volume ID =
                1).


                cubit.cmd
                ("brick x 10" )
                # Volume ID = 1



                # 2) Specify surface 2 as the
                source and surface 5 as the target for
      sweeping.


                cubit.cmd
                ("vol 1 scheme sweep source
                surf 2 target surf 5" )


                # 3) Query the source surfaces
                for volume 1.


                sources = cubit.get_source_surfaces
                (1)


                # Expected output: (2,)


                print("Source
                surfaces:" , sources)
```

```
 @n type of volume_id:  int
```

Parameters

volume_id ID of the volume to query.
return type of : std::vector< int,std:: allocator< int > >

Returns

A list (vector<int> or Python tuple) of surface IDs designated as sweep sources.

◆ get_sphere_count()

def cubit.get_sphere_count ( )

Retrieve the count of sphere elements in the current model.

Sphere elements are nodes whose element type has been set to "sphere".

```
.. code-block:: python
```

```
                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("mesh vol all" )


                cubit.cmd
                ("block 1 node all"
                )


                cubit.cmd
                ("block 1 element type
                sphere" )


                sphere_count = cubit.get_sphere_count
                ()


                print("Sphere
                count:" , sphere_count)
```

```
 @n return type of :  int
```

Returns

Number of sphere elements (nodes flagged as "sphere") in the current model.

◆ get_sphere_global_element_id()

def cubit.get_sphere_global_element_id (*sphere_id*)

Get the Global Element ID for a specific node (SPHERE element).

Cubit assigns a Global Element ID to each mesh entity when it is placed into a block. Nodes are represented as SPHERE elements in the mesh. This function returns the Global Element ID corresponding to the given local sphere (node) ID within its type-specific ID space.

```
.. code-block:: python
```

```
                # Create and mesh a brick
                volume

                cubit.cmd
                ("brick x 10" )

                cubit.cmd
                ("mesh vol 1" )

                # Assign all vertices (nodes) in
                volume 1 to block 1

                cubit.cmd
                ("block 1 vertex in vol
                1" )

                # Query the Global Element ID of
                local node 1

                gid =
    cubit.get_sphere_global_element_id
                (1)

                print("Global ID of node
                1:" , gid) # prints
                1
```

```
@n type of sphere_id:  int
```

Parameters

sphere_id Local node (SPHERE) ID within its type-specific ID space.
return type of : int

Returns

Global Element ID for the specified node, or 0 if not placed into a block.

◆ get_string_sculpt_default()

def cubit.get_string_sculpt_default (*variable*)

Retrieve default string-valued sculpt parameter.

Matches the input variable name substring to known sculpt parameters and returns its default.

Title Available string parameters

- ROOT_FILE
- QUALITY_FILE
- VOLFRAC_FILE

type of variable: string

Parameters

variable Name of the sculpt parameter substring.
return type of : string

Returns

Default string value for the specified parameter.

◆ get_sub_elements()

def cubit.get_sub_elements (*entity_type*, *entity_id*, *dimension*)

Get the lower-dimensional entities of a higher-dimensional mesh element.

This function returns the IDs of all sub-entities of the specified dimension for a given mesh element. For example, to get all faces (dimension 2) of a hexahedron, or all edges (dimension 1) of a triangle.

Notes Faces and edges are only created on surfaces and curves, respectively. Interior faces or edges within volumes are not generated. In the example below, only three faces appear because they lie on the exterior surfaces of volume 1.

```
.. code-block:: python
```

```
                # Create and mesh a brick
                volume

                cubit.cmd
                ("brick x 10" )

                cubit.cmd
                ("mesh volume 1" )


                # Query the exterior faces
                (dimension 2) of hex element 1

                faces = cubit.get_sub_elements
                ("hex" , 1, 2)

                print("Hex 1 has
                faces:" , faces) # e.g.
                (391, 201, 91)
```

```
@n type of entity_type:  string
```

Parameters

entity_type Mesh element type of the higher-dimensional entity (e.g., "hex", "tet", "tri", "quad").
type of entity_id: int

entity_id ID of the mesh element.
type of dimension: int

dimension Dimension of sub-entities to retrieve: 0 = vertices, 1 = edges, 2 = faces, 3 = volumes.
return type of : std::vector< int,std:: allocator< int > >

Returns

List of IDs of the requested sub-entities (Python tuple in Python API).

◆ get_submap_corner_types()

def cubit.get_submap_corner_types (*surface_id*)

Get a list of vertex IDs and their corner-type codes for a surface submap.

Vertex types classify each corner based on how many mesh elements meet there:

- -1 unknown : not classified or not applicable
- 1 end : interior angle ~ 90deg (one element)
- 2 side : interior angle ~ 180deg (two elements)
- 3 corner : interior angle ~ 270deg (three elements)
- 4 reverse : interior angle ~ 360deg (four elements)
- 5 triangle corner : corner of a triangular mesh element
- 6 non-triangle : corner of a non-triangular mesh element

type of surface_id: int

Parameters

surface_id ID of the surface to query.
return type of : std::vector< std::pair< int,int >,std::allocator< std:: pair< int,int > > >

Returns

Vector of <vertex_id, corner_type> pairs for each corner.

◆ get_surface_area()

def cubit.get_surface_area (*surface_id*)

Get the area of a specified surface.

.. code-block:: python

```
                # Create a 10x10x10 brick
                centered at the origin; surfaces 1-6
    are
                generated.

                cubit.cmd
                ("brick x 10" )

                area = cubit.get_surface_area
                (1) # top face area = 100

                print("Surface 1
                area:" , area) # prints
                100
```

```
@n type of surface_id:  int
```

Parameters

surface_id ID of the surface to query.
return type of : float

Returns

The area of the specified surface.

◆ get_surface_cavity_collections()

def cubit.get_surface_cavity_collections (*volume_list*, *area_threshold* = `-1`, *angle_tolerance* = `-1`, *combine_cavities* = `True`)

Return collections of surfaces that form cavities in the specified volumes.

A cavity is a collection of contiguous surfaces, bounded by curves where the exterior angle is greater than or equal to (180 - angle_tolerance). Cavities are filtered by their total surface area.

If combine_cavities is true, small adjacent cavities are merged after initial cavity detection.

If area_threshold < 0.0, all cavities are returned. If angle_tolerance < 0.0, a default of 0.01 degrees is used.

`.. code-block:: python`

```
                cubit.cmd
                ("brick x 10" )

                cubit.cmd
                ("brick x 5" )

                cubit.cmd
                ("vol 5 move x 5" )

                cubit.cmd
                ("subtract volume 2 from
                volume 1" )


                volume_ids = [1]

                area_thresh = -1.0 # return all
                cavities

                angle_tol_deg = 0.1

                combine = False


                cavity_collections =
      cubit.get_surface_cavity_collections
                (volume_ids, area_thresh, angle_tol_deg,
      combine)
```

for surfaces, area in cavity_collections: print("Cavity surfaces:", tuple(surfaces), "Area:", area)

```
                # Expected output:

                # Cavity surfaces: (17, 15, 16,
                13, 14), Area: <value>
```

`@n type of volume_list:  std::vector< int,std::allocator< int > >`

Parameters

volume_list List of volume IDs to query.
type of area_threshold: float, optional

area_threshold Return cavities whose total surface area is less than this value. If less than 0.0, return all cavities.
type of angle_tolerance: float, optional

angle_tolerance Bounding curves will have an exterior angle greater than or equal to (180 - angle_tolerance). If less than 0.0, default is 0.01 degrees.
type of combine_cavities: boolean, optional

combine_cavities If true, small adjacent cavities are combined after initial cavity detection.
return type of : std::vector< std::pair< std::vector< int,std::allocator< int > >,double >,std::allocator< std::pair< std::vector< int,std::allocator< int > >,double > > >

Returns

: Ordered list of pairs each pair contains (1) a list of contiguous surface IDs defining a cavity, and (2) the total surface area of that cavity (ordered from smallest to largest area).

◆ get_surface_centroid()

def cubit.get_surface_centroid (*surface_id*)

Get the approximate centroid of a specified surface based on graphics faceting.

The centroid is computed from the faceted representation of the surface, so it is an approximation.

```
.. code-block:: python
```

```
                    cubit.cmd
                    ("brick x 10" )
                    # create a 10x10x10 brick
                    centered at origin


                    centroid = cubit.get_surface_centroid
                    (1)


                    print(centroid) # expected: (0.0,
                    0.0, 5.0)
```

```
@n type of surface_id:  int
```

Parameters

surface_id ID of the surface to query.
return type of : std:: array< double,3 >

Returns

A 3-element array [x, y, z] representing the approximate surface centroid.

◆ get_surface_cone_collections()

def cubit.get_surface_cone_collections (*volume_list*, *radius_threshold* = `0.0`)

Identify collections of surfaces that comprise cones in specified volumes.

Cones may be represented by a single surface or by two adjacent surfaces symmetrically split. Results can be filtered by a maximum cone radius.

`.. code-block:: python`

```
                # Create a cylinder volume with a
                cone on top.


                cubit.cmd
                ('cylinder rad 1 height
                5' ) # Create
                cylinder


                cubit.cmd
                ('create frustum height 1
                radius 1 top 0' ) # Create
                cone (frustum with top radius 0)


                cubit.cmd
                ('vol 2 mov z 3' )
                # Move cone on top of
                cylinder


                cubit.cmd
                ('unite vol all' )
                # Unite volumes into one


                cubit.cmd
                ('split period vol
                1' ) # Split cylinder and
                cone surfaces


                cone_collections =
        cubit.get_surface_cone_collections
                ([1], 0.0)


                print("Cone
                collections:" )
```

for surfaces, radius in cone_collections: print(f"Radius: {radius}, Surfaces:", [int(sid) for sid in surfaces])

```
                # Expected output:


                # Radius: <value>,
                Surfaces: [surface ids of cone faces]
```

`@n type of volume_list:  std::vector< int,std::allocator< int > >`

Parameters

volume_list List of volume IDs to query.
type of radius_threshold: float, optional

radius_threshold Optional maximum radius. Return cones with computed radius less than or equal to *radius_threshold*. If 0, all cone collections are returned.
return type of : std::vector< std::pair< std::vector< int,std::allocator< int > >,double >,std::allocator< std::pair< std::vector< int,std::allocator< int > >,double > > >

Returns

A list of (surface ID list, radius) pairs. Each pair corresponds to one cone collection. Surfaces are grouped per individual cone.

◆ get_surface_count()

def cubit.get_surface_count ( )

Get the current number of surfaces in the model.

.. code-block:: python

```
                # Create a 10x10x10 brick; this
                generates six surfaces.


                cubit.cmd
                ("brick x 10" )


                surf_count = cubit.get_surface_count
                () # returns 6


                print("Surface
                count:" , surf_count)
```

`@n return type of :  int`

Returns

The number of surfaces currently defined in the model.

◆ get_surface_element_count()

def cubit.get_surface_element_count (*surface_id*)

Retrieve the count of elements on a specified surface.

Returns the total number of quadrilateral and triangular elements present on the given surface, regardless of block assignments.

```
.. code-block:: python
```

```
                cubit.cmd
                ("brick x 10" )
                # create brick (Volume ID =
                1)


                cubit.cmd
                ("mesh vol 1" )
                # mesh volume 1 (surface faces
                become quads or tris)


                surf_count =
      cubit.get_surface_element_count
                (1)


                print("Surface 1
                elements:" , surf_count)
```

```
@n type of surface_id:  int
```

Parameters

surface_id ID of the surface to query.
return type of : int

Returns

Number of quadrilateral and triangular elements on that surface.

◆ get_surface_hole_collections()

def cubit.get_surface_hole_collections (*volume_list*, *radius_threshold*)

Return collections of surfaces that form holes in the specified volumes.

A hole is a collection of contiguous surfaces that form a cylindrical or approximately cylindrical feature. Holes are filtered by their computed radius.

If radius_threshold is provided, only holes with radius less than or equal to this value are returned.

.. code-block:: python

```
cubit.cmd
("brick x 10" )


cubit.cmd
("cylinder rad 1 height
10" )


cubit.cmd
("vol 2 move z 5" )


cubit.cmd
("subtract volume 2 from
volume 1" )


volume_ids = [1]


radius_thresh = 2.0


hole_collections =
cubit.get_surface_hole_collections
(volume_ids, radius_thresh)
```

for surfaces, radius in hole_collections: print("Hole surfaces:", tuple(surfaces), "Radius:", radius)

```
# Expected output:


# Hole surfaces: (10, 11),
Radius: 1.0
```

@n type of volume_list: std::vector< int,std::allocator< int > >

Parameters

volume_list List of volume IDs to query.
type of radius_threshold: float

radius_threshold Return holes with computed radius less than or equal to this value.
return type of : std::vector< std::pair< std::vector< int,std::allocator< int > >,double >,std::allocator< std::pair< std::vector< int,std::allocator< int > >,double > > >

Returns

: A list of pairs each pair contains (1) a list of contiguous surface IDs defining a hole, and (2) the computed radius of the hole.

◆ get_surface_loop_nodes()

def cubit.get_surface_loop_nodes (*surface_id*)

Get the ordered list of node IDs on the loops of a surface.

This function returns the ordered list of node IDs for each loop on the specified surface. The first loop in the list is the external loop. Each loop is returned as a separate list of node IDs.

Loops are ordered as follows:

- First list: external loop (outer boundary)
- Subsequent lists: internal loops (holes, if present)

```
.. code-block:: python
```

```
                # Create a brick and subtract a
                cylinder to create an internal loop


                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("cylinder rad 3 height
                10" )


                cubit.cmd
                ("subtract vol 2 from vol
                1" )


                # Mesh the curves to ensure nodes
                exist on loops


                cubit.cmd
                ("mesh curve in vol
                1" )


                # Query the nodes on the loops of
                surface 1


                loop_nodes =
    cubit.get_surface_loop_nodes
                (1)


                print(f"Surface 1 has
                {len(loop_nodes)} loop(s)." )
```

for i, loop in enumerate(loop_nodes): print(f" Loop {i+1} has {len(loop)} node(s):", " ".join(str(node_id) for node_id in loop))

```
  @n type of surface_id:  int
```

Parameters

surface_id ID of the surface.
return type of : std::vector< std::vector< int,std::allocator< int > >,std::allocator< std::vector< int,std:: allocator< int > > > >

Returns

List of lists of node IDs, one list per loop (first list is the external loop).

◆ get_surface_nodes()

def cubit.get_surface_nodes (*surface_id*)

Get the list of node IDs owned by a surface.

This function returns the IDs of nodes owned by the specified surface. Nodes on the bounding curves and vertices of the surface are excluded.

.. code-block:: python

```
# Create a brick and mesh volume
1


cubit.cmd
("brick x 10" )


cubit.cmd
("mesh vol 1" )



# Query the nodes on surface
5


surface_nodes = cubit.get_surface_nodes
(5)


print(f"Surface 5 owns
{len(surface_nodes)} node(s)." )
```

for node_id in surface_nodes: print(f" Node ID: {node_id}")

```
@n type of surface_id:  int
```

Parameters

surface_id ID of the surface.
return type of : std::vector< int,std:: allocator< int > >

Returns

List of node IDs owned by the surface (excluding nodes on bounding curves and vertices).

◆ get_surface_normal()

def cubit.get_surface_normal (*surface_id*)

Get the unit normal vector at the center of a specified surface.

.. code-block:: python

```
# Create a 10x10x10 brick;
surfaces 1-6 are generated.


cubit.cmd
("brick x 10" )



# Measure the normal for surface
1 (the top face).


normal = cubit.get_surface_normal
(1)


print("Normal:" ,
normal) # Expected output: (0.0,
0.0, 1.0)
```

```
@n type of surface_id:  int
```

Parameters

surface_id ID of the surface to query.
return type of : std:: array< double,3 >

Returns

A 3-element array [nx, ny, nz] representing the unit normal at the surface center.

◆ get_surface_normal_at_coord()

def cubit.get_surface_normal_at_coord (*surface_id*, *coord*)

Get the unit normal vector at a specified point on a surface.

.. code-block:: python

```
                # Create a 10x10x10 brick
                centered at the origin; surfaces 1-6
    are
                generated.

                cubit.cmd
                ("brick x 10" )


                # Pick a point at the center of
                surface 1: (0, 0, 5).

                coord = (0.0, 0.0, 5.0)

                normal =
    cubit.get_surface_normal_at_coord
                (1, coord)

                print("Normal:" ,
                normal) # Expected output: (0.0,
                0.0, 1.0)
```

```
@n type of surface_id:  int
```

Parameters

surface_id ID of the surface to query.
type of coord: std::array< double,3 >

coord A 3-element array {x, y, z} representing a point on the surface.
return type of : std:: array< double,3 >

Returns

A 3-element array [nx, ny, nz] representing the unit normal at coord.

◆ get_surface_num_loops()

def cubit.get_surface_num_loops (*surface_id*)

Get the number of loops on a surface.

This function returns the number of loops on the specified surface. A loop is a closed boundary on the surface. Surfaces with holes or internal boundaries will have multiple loops.

```
.. code-block:: python
```

```
                # Create a brick and subtract a
                cylinder to create an internal loop

                cubit.cmd
                ("brick x 10" )

                cubit.cmd
                ("cylinder rad 3 height
                12" )

                cubit.cmd
                ("subtract vol 2 from vol
                1" )


                # Query the number of loops on
                surface 1

                num_loops = cubit.get_surface_num_loops
                (1)

                print(f"Surface 1 has
                {num_loops} loop(s)." )
```

```
@n type of surface_id:  int
```

Parameters

surface_id ID of the surface.
return type of : int

Returns

Number of loops on the surface.

◆ get_surface_principal_curvatures()

def cubit.get_surface_principal_curvatures (*surface_id*)

Get the principal curvatures of a surface at its midpoint.

Principal curvatures quantify how a surface bends in two orthogonal directions at a point. At the surface midpoint, this function returns two scalar values: k1 and k2. For a planar surface, both values are 0. For a sphere of radius R, both values equal 1/R.

```
.. code-block:: python
```

```
                # Create a sphere of radius 2;
                surface 1 is the spherical face.

                cubit.cmd
                ("sphere rad 2" )

                curvatures =
    cubit.get_surface_principal_curvatures
                (1)

                print("Principal
                curvatures:" , curvatures) # prints
    (0.5, 0.5)


                # Create a planar brick face;
                surface 1 is planar.

                cubit.cmd
                ("brick x 10" )

                curvatures =
    cubit.get_surface_principal_curvatures
                (1)

                print("Principal
                curvatures:" , curvatures) # prints
    (0.0, 0.0)
```

```
 @n type of surface_id:  int
```

Parameters

surface_id ID of the surface to query.
return type of : std::vector< double,std::
allocator< double > >

Returns

: A vector of two doubles the first and second principal curvatures at the surface midpoint.

◆ get_surface_quads()

def cubit.get_surface_quads (*surface_id*)

Get the list of quadrilateral (quad) element IDs on a surface.

This function returns the IDs of all quadrilateral (quad) elements on the specified surface.

```
.. code-block:: python
```

```
                # Create a brick and mesh volume
                1

                cubit.cmd
                ("brick x 10" )

                cubit.cmd
                ("mesh vol 1" )


                # Query the quads on surface
                1

                quad_ids = cubit.get_surface_quads
                (1)

                print(f"Surface 1 contains
                {len(quad_ids)} quad(s)." )
```

for quad_id in quad_ids: print(f" Quad ID: {quad_id}")

```
  @n type of surface_id:  int
```

Parameters

surface_id ID of the surface.
return type of : std::vector< int,std:: allocator< int > >

Returns

List of quadrilateral (quad) element IDs on the surface.

◆ get_surface_sense()

def cubit.get_surface_sense (*surface_id*)

Get the orientation ("sense") of a specified surface.

.. code-block:: python

```
cubit.cmd
("brick x 10" )
# create a 10x10x10 brick


sense = cubit.get_surface_sense
(1) # e.g., "Forward"


print("Surface 1
sense:" , sense)
```

```
@n type of surface_id:  int
```

Parameters

surface_id ID of the surface to query.
return type of : string

Returns

: The surface sense "Reversed", "Forward", or "Both".

◆ get_surface_tris()

def cubit.get_surface_tris (*surface_id*)

Get the list of triangle (tri) element IDs on a surface.

This function returns the IDs of all triangle (tri) elements on the specified surface.

```
.. code-block:: python
```

```
# Create a brick, apply tetmesh
scheme, and mesh volume 1


cubit.cmd
("brick x 10" )


cubit.cmd
("vol 1 scheme
tetmesh" )


cubit.cmd
("mesh vol 1" )



# Query the tris on surface
1


tri_ids = cubit.get_surface_tris
(1)


print(f"Surface 1 contains
{len(tri_ids)} tri(s)." )
```

for tri_id in tri_ids: print(f" Tri ID: {tri_id}")

```
@n type of surface_id:  int
```

Parameters

surface_id ID of the surface.
return type of : std::vector< int,std:: allocator< int > >

Returns

List of triangle (tri) element IDs on the surface.

◆ get_surface_type()

def cubit.get_surface_type (*surface_id*)

Get the surface type for a specified surface.

Title Available surface types

- cone surface
- plane surface
- sphere surface
- spline surface
- torus surface
- best fit surface
- facet surface
- undefined surface type

```
.. code-block:: python
```

```
                # Assume surface ID 5 corresponds
                to a plane.

                surf_type = cubit.get_surface_type
                (5)

                print("Surface 5 is
                a:" , surf_type) # Output:
                Surface 5 is a: plane surface
```

```
 @n type of surface_id:  int
```

Parameters

surface_id ID of the surface to query.
return type of : string

Returns

Descriptive string indicating the surface's geometry type.

◆ get_surfs_with_narrow_regions()

def cubit.get_surfs_with_narrow_regions (*target_ids*, *narrow_size*)

Find surfaces with narrow regions based on edge-pair orientation and proximity.

For each face of volumes in target_ids, iterates over every pair of non-adjacent edges. For each pair:

- Computes the shortest distance between edges.
- If distance <= narrow_size, computes local edge tangents and surface normals to form cross-product vectors.
- If the dot product of those cross-product vectors is < -cos(15deg), the face is included. Returns IDs of faces where any such qualifying edge pair exists.

This function differs from get_closed_narrow_surfaces by checking all faces (not only closed two-loop faces), and from get_narrow_regions by requiring an oriented edge-pair test rather than loop-split or loop-proximity tests.

```
.. code-block:: python
```

```
                cubit.cmd
                ("cylinder radius 3 height
                10" )

                cubit.cmd
                ("modify curve 1 blend
                radius 0.5" )

                cubit.cmd
                ("cylinder radius 2 height
                10" )

                cubit.cmd
                ("subtract volume 2 from
                volume 1" )

                narrow_thresh = 1.0

                oriented_narrow =
      cubit.get_surfs_with_narrow_regions
                ([1], narrow_thresh)

                print(oriented_narrow)
```

```
 @n type of target_ids:  std::vector< int,std::allocator< int > >
```

Parameters

target_ids Vector of volume IDs to examine.
type of narrow_size: float

narrow_size Distance threshold for detecting narrow edge pairs with orientation checks.
return type of : std::vector< int,std:: allocator< int > >

Returns

Vector of surface IDs containing at least one narrow edge-pair region.

◆ get_tangential_intersections()

def cubit.get_tangential_intersections (*target_volume_ids*, *upper_bound*, *lower_bound*)

Find surfaces with tangential intersection angles outside specified bounds.

Iterates over all faces of volumes in target_volume_ids. For each face, computes any tangential intersection angles between adjacent faces. A surface is included if any angle is < lower_bound or > upper_bound.

type of target_volume_ids: std::vector< int,std::allocator< int > >

Parameters

target_volume_ids Vector of volume IDs to examine.
type of upper_bound: float

upper_bound Upper threshold for acceptable tangential angle.
type of lower_bound: float

lower_bound Lower threshold for acceptable tangential angle.
return type of : std::vector< int,std:: allocator< int > >

Returns

Vector of surface IDs having any tangential intersection angle < lower_bound or > upper_bound.

◆ get_target_surfaces()

def cubit.get_target_surfaces (*volume_id*)

Retrieve the list of sweep target surfaces for a specified volume.

When a volume uses the Sweep scheme, target surfaces are the ending faces to which the hexahedral mesh is extruded. This function returns all surface IDs designated as targets. If no explicit target was set, returns an empty list.

.. code-block:: python

```
                # 1) Create a brick (Volume ID =
                1).


                cubit.cmd
                ("brick x 10" )
                # Volume ID = 1


                # 2) Specify surface 2 as the
                source and surface 5 as the target for
      sweeping.


                cubit.cmd
                ("vol 1 scheme sweep source
                surf 2 target surf 5" )


                # 3) Query the target surfaces
                for volume 1.


                targets = cubit.get_target_surfaces
                (1)


                # Expected output: (5,)


                print("Target
                surfaces:" , targets)
```

@n type of volume_id: int

Parameters

volume_id ID of the volume to query.
return type of : std::vector< int,std:: allocator< int > >

Returns

A list (vector<int> or Python tuple) of surface IDs designated as sweep targets.

◆ get_target_timestep()

def cubit.get_target_timestep ( )

Return the current target time step threshold used in the density multiplier metric.

This threshold influences adaptive time stepping by multiplying element-based estimates.

.. code-block:: python

```
                # Set the target time step via
                Cubit command and query it.


                cubit.cmd
                ("set target timestep
                0.005" )


                target = cubit.get_target_timestep
                ()


                print("Target time
                step:" , target) # prints
                0.005
```

@n return type of : float

Returns

The currently configured target time step threshold.

◆ get_tet_count()

def cubit.get_tet_count ( )

Retrieve the count of tetrahedral elements in the current model.

.. code-block:: python

```
cubit.cmd
("brick x 10" )
# create brick (Volume ID =
1)


cubit.cmd
("vol 1 scheme
tetmesh" )


cubit.cmd
("mesh vol 1" )


tet_count = cubit.get_tet_count
()


print("Tet count:" ,
tet_count)
```

```
@n return type of :  int
```

Returns

Number of tetrahedral elements in the current model.

◆ get_tet_global_element_id()

def cubit.get_tet_global_element_id (*tet_id*)

Get the Global Element ID for a specific tetrahedral element.

This function returns the Global Element ID assigned to the given local tet ID when the element was placed into a block.

```
.. code-block:: python
```

```
# Create a brick, set up tetmesh,
and mesh volume 1


cubit.cmd
("brick x 10" )


cubit.cmd
("vol 1 scheme
tetmesh" )


cubit.cmd
("mesh vol 1" )


# Assign all tets in volume 1 to
block 1 (establishes global IDs)


cubit.cmd
("block 1 tet all" )



# Query the Global Element ID of
local tet 22


gid = cubit.get_tet_global_element_id
(22)


print("Global Element ID of
tet 22:" , gid)
```

```
@n type of tet_id:  int
```

Parameters

tet_id Local tetrahedral element ID.
return type of : int

Returns

Global Element ID for the specified tetrahedron, or 0 if the tet is not in a block.

◆ get_tetmesh_growth_factor()

def cubit.get_tetmesh_growth_factor (*volume_id*)

Get the growth factor for tet meshing on a volume.

The growth factor controls how quickly tetrahedral sizes can change when transitioning from small to larger elements within the volume. Valid values range from 1.0 (uniform sizing) to 10.0 (rapid growth).

```
.. code-block:: python
```

```
                # 1) Set growth factor to 1.3 on
                volume 5.


                cubit.cmd
                ("volume 5 scheme tetmesh
                growth_factor 1.3" )


                # 2) Query the growth
                factor.


                gf = cubit.get_tetmesh_growth_factor
                (5)


                print(f"Volume 5 tetmesh
                growth factor: {gf}" ) #
                Expected: 1.3
```

```
@n type of volume_id:  int
```

Parameters

volume_id ID of the volume to query.
return type of : float

Returns

Growth factor (double) currently set on the volume.

◆ get_tetmesh_insert_mid_nodes()

def cubit.get_tetmesh_insert_mid_nodes ( )

Get the global flag indicating insertion of mid-edge (quadratic) nodes during tet meshing.

Returns true if the tet mesher is configured to create mid-edge nodes while generating tets (improves quadratic element quality); false if midnodes are inserted post-meshing.

```
.. code-block:: python
```

```
                # 1) Enable automatic insertion
                of mid-edge nodes.


                cubit.cmd
                ("set tetmesher add
                mid_edge_nodes on" )


                # 2) Query the mid-edge node
                insertion flag.


                mid_node_on =
        cubit.get_tetmesh_insert_mid_nodes
                ()


                print(f"Insert mid-edge
                nodes during tet meshing?
        {mid_node_on}" )
```

```
@n return type of :  boolean
```

Returns

True if mid-edge nodes are inserted during meshing; false otherwise.

◆ get_tetmesh_minimize_interior_points()

def cubit.get_tetmesh_minimize_interior_points ( )

Get the global flag indicating minimization of interior points in tet meshing.

Returns true if the tet mesher will attempt to reduce the number of interior nodes inserted; false if interior point insertion is unrestricted. Disabling interior point insertion may produce a mesh with fewer nodes but can fail in complex geometries.

.. code-block:: python

```
# 1) Disable interior point
insertion.

cubit.cmd
("set tetmesher interior
points off" )

# 2) Query the interior point
minimization flag.

min_interior =
cubit.get_tetmesh_minimize_interior_points
()

print(f"Minimize interior
points? {min_interior}" )
```

@n return type of : boolean

Returns
True if minimizing interior points is ON; false otherwise.

◆ get_tetmesh_minimize_slivers()

def cubit.get_tetmesh_minimize_slivers ( )

Get the global flag indicating minimization of sliver tetrahedra.

Returns true if the tet mesher will apply additional processing to identify and remove sliver-shaped tetrahedra (minimize slivers); false if sliver removal is not explicitly requested.

.. code-block:: python

```
# 1) Enable sliver
minimization.

cubit.cmd
("set tetmesher optimize
sliver on" )

# 2) Query the sliver
minimization flag.

min_sliver =
cubit.get_tetmesh_minimize_slivers
()

print(f"Minimize sliver
tetrahedra? {min_sliver}" )
```

@n return type of : boolean

Returns
True if sliver minimization is ON; false otherwise.

◆ get_tetmesh_num_anisotropic_layers()

def cubit.get_tetmesh_num_anisotropic_layers ( )

Get the global number of anisotropic tet layers setting.

Returns how many anisotropic layers the tet mesher will attempt to place in thin regions. If this global setting is zero, anisotropic layering is disabled.

```
.. code-block:: python
```

```
                # 1) Enable 2 anisotropic layers
                globally.

                cubit.cmd
                ("set tetmesher anisotropic
                layers 2" )


                # 2) Query the number of
                anisotropic layers.

                num_aniso =
      cubit.get_tetmesh_num_anisotropic_layers
                ()

                print(f"Global anisotropic
                tet layers: {num_aniso}" ) # Expected:
      2
```

```
 @n return type of :  int
```

Returns
Number of anisotropic layers (integer); 0 if disabled.

◆ get_tetmesh_optimization_level()

def cubit.get_tetmesh_optimization_level ( )

Get the global tet meshing optimization level.

Returns the integer optimization level in use for the tet mesher, ranging from 0 (none) to 6 (extreme). Higher levels yield better element quality at the expense of runtime.

```
.. code-block:: python
```

```
                # 1) Set tet mesher optimization
                level to 4 (strong).

                cubit.cmd
                ("set tetmesher optimize
                level 4" )


                # 2) Query the optimization
                level.

                opt_level =
      cubit.get_tetmesh_optimization_level
                ()

                print(f"Tet mesher
                optimization level: {opt_level}" ) #
      Expected: 4
```

```
 @n return type of :  int
```

Returns
Optimization level (int in [0..6]).

◆ get_tetmesh_optimize_mid_nodes()

def cubit.get_tetmesh_optimize_mid_nodes ( )

Get the global flag indicating optimization of mid-edge nodes during tet meshing.

Returns true if the tet mesher is configured to optimize (adjust) mid-edge node positions during meshing to improve element quality; false if no midnode optimization is performed.

```
.. code-block:: python
```

```
                # 1) Enable mid-edge node
                optimization.


                cubit.cmd
                ("set tetmesher optimize
                surface mid_edge_nodes on" )



                # 2) Query the mid-edge node
                optimization flag.


                opt_mid_on =
    cubit.get_tetmesh_optimize_mid_nodes
                ()


                print(f"Optimize mid-edge
                nodes during tet meshing? {opt_mid_on}"
    )
```

```
 @n return type of :  boolean
```

Returns

True if mid-edge node optimization is ON; false otherwise.

◆ get_tetmesh_optimize_overconstrained_edges()

def cubit.get_tetmesh_optimize_overconstrained_edges ( )

Get the global flag indicating optimization of overconstrained edges.

Returns true if the tet mesher will split edges that connect two surface nodes but do not lie on the surface ("overconstrained edges"); false if not.

```
.. code-block:: python
```

```
                # 1) Enable optimization of
                overconstrained edges.


                cubit.cmd
                ("set tetmesher optimize
                overconstrained edges on" )



                # 2) Query the overconstrained
                edges optimization flag.


                opt_over_edge =
    cubit.get_tetmesh_optimize_overconstrained_edges
                ()


                print(f"Optimize
                overconstrained edges? {opt_over_edge}"
    )
```

```
 @n return type of :  boolean
```

Returns

True if overconstrained edges optimization is ON; false otherwise.

◆ get_tetmesh_optimize_overconstrained_tets()

def cubit.get_tetmesh_optimize_overconstrained_tets ( )

Get the global flag indicating optimization of overconstrained tetrahedra.

Returns true if the tet mesher will optimize any tetrahedra that have more than one triangular face on the same geometric surface (removes "overconstrained" tets); false if not.

```
.. code-block:: python
```

```
                # 1) Enable optimization of
                overconstrained tetrahedra.


                cubit.cmd
                ("set tetmesher optimize
                overconstrained tetrahedra on" )



                # 2) Query the overconstrained
                tetrahedra optimization flag.


                opt_over_tet =
    cubit.get_tetmesh_optimize_overconstrained_tets
                ()


                print(f"Optimize
                overconstrained tets? {opt_over_tet}" )
```

```
@n return type of :  boolean
```

Returns

True if overconstrained tetrahedra optimization is ON; false otherwise.

◆ get_tetmesh_parallel()

def cubit.get_tetmesh_parallel ( )

Get the global parallel-tetmesher flag (HPC) setting.

Returns true if the MeshGems-Tetra HPC (parallel) tet mesher is enabled globally; false if using the older serial tet mesher. This setting applies to all subsequent tet meshes until changed.

```
.. code-block:: python
```

```
                # 1) Enable parallel tet mesher
                globally.


                cubit.cmd
                ("set tetmesher hpc on
                threads 4" )



                # 2) Query the parallel
                flag.


                parallel_on =
    cubit.get_tetmesh_parallel
                ()


                print(f"Parallel tet mesher
                enabled? {parallel_on}" )
```

```
@n return type of :  boolean
```

Returns

True if parallel (HPC) tet mesher is ON; false if OFF.

◆ get_tetmesh_proximity_flag()

def cubit.get_tetmesh_proximity_flag (*volume_id*)

Get the proximity-layer flag for tet meshing on a volume.

Returns whether proximity layers are enabled for the specified volume. When enabled, the tet mesher will insert a minimum number of tetrahedral layers in thin regions to capture physical features.

```
.. code-block:: python
```

```
                # 1) Enable proximity layers
                (e.g., 3 layers) on volume 5.


                cubit.cmd
                ("volume 5 scheme tetmesh
                proximity layers 3" )



                # 2) Query the proximity
                flag.


                prox_on =
    cubit.get_tetmesh_proximity_flag
                (5)


                print(f"Proximity layers
                enabled for volume 5? {prox_on}" )
```

```
@n type of volume_id:  int
```

Parameters

volume_id ID of the volume to query.
return type of : boolean

Returns

True if proximity layers are ON; false if OFF.

◆ get_tetmesh_proximity_layers()

def cubit.get_tetmesh_proximity_layers (*volume_id*)

Get the number of proximity layers for tet meshing on a volume.

Returns the integer number of tetrahedral layers configured for thin regions. Only valid if proximity is enabled; otherwise, returns zero.

```
.. code-block:: python
```

```
                # 1) Enable 4 proximity layers on
                volume 5.


                cubit.cmd
                ("volume 5 scheme tetmesh
                proximity layers 4" )



                # 2) Query the number of
                proximity layers.


                num_layers =
    cubit.get_tetmesh_proximity_layers
                (5)


                print(f"Volume 5 proximity
                layers: {num_layers}" ) #
                Expected: 4
```

```
@n type of volume_id:  int
```

Parameters

volume_id ID of the volume to query.
return type of : int

Returns

Number of proximity layers (integer); 0 if proximity is OFF.

◆ get_tetmesh_relax_surface_constraints()

def cubit.get_tetmesh_relax_surface_constraints ( )

Get the global flag indicating relaxation of surface mesh constraints in tet meshing.

Returns true if the tet mesher is configured to relax surface mesh conformity constraints, allowing insertion of tetrahedra that may deviate slightly from the exact surface mesh; false otherwise.

```
.. code-block:: python
```

```
                # 1) Enable relaxing of surface
                constraints.


                cubit.cmd
                ("set tetmesher boundary
                recovery on" )


                # 2) Query the relax surface
                constraints flag.

                relax_surf =
      cubit.get_tetmesh_relax_surface_constraints
                ()

                print(f"Relax surface
                constraints? {relax_surf}" )
```

```
 @n return type of :  boolean
```

Returns
: True if relaxing surface constraints is ON; false otherwise.

◆ get_tight_bounding_box()

def cubit.get_tight_bounding_box (*geometry_type*, *entity_list*)

Get the tight bounding box for a list of entities.

Computes an oriented (tight) bounding box for the specified entities. The box is aligned with the principal axes of the geometry, not the global axes. This provides a minimal-volume bounding box that follows the geometry's orientation.

Title Array contents

- [0-2] center coordinates (x, y, z)
- [3-5] U-axis (normalized) vector (x, y, z)
- [6-8] V-axis (normalized) vector (x, y, z)
- [9-11] W-axis (normalized) vector (x, y, z)
- [12-14] lengths along U, V, W axes

```
.. code-block:: python
```

```
                cubit.cmd
                ("brick x 10" );
                cubit.cmd
                ("rotate volume 1 angle 45
                about z" )


                bbox = cubit.get_tight_bounding_box
                ("volume" , [1])


                print("Center:" ,
                bbox[0:3], "U-axis:"
                , bbox[3:6], "V-axis:" , bbox[6:9],


                "W-axis:" ,
                bbox[9:12], "Lengths:" , bbox[12:15])


                # Expected output: Center: (0.0,
                0.0, 0.0) ... Lengths: (5.0, 5.0, 5.0)
```

```
 @n type of geometry_type:  string
```

Parameters

geometry_type Geometry type ("body", "volume", "surface", "curve", or "vertex").
type of entity_list: std::vector< int,std::allocator< int > >

entity_list List of entity IDs.
return type of : std:: array< double,15 >

Returns
: 15-element array representing the tight bounding box.

◆ get_top_level_assembly_items()

def cubit.get_top_level_assembly_items ( )

◆ get_total_bounding_box()

def cubit.get_total_bounding_box (*geometry_type*, *entity_list*)

Get the combined bounding box for a list of entities.

.. code-block:: python

```
                # Prepare a list of surface
                IDs.

                entity_list = [22, 23, 24]

                # Query the total bounding box
                for those surfaces.

                bbox = cubit.get_total_bounding_box
                ("surface" ,
                entity_list)

                print(bbox) # Prints 10 values:
                x-min, x-max, x-range, y-min, y-max, y-
    range, z-min,
                z-max, z-range, diagonal
```

```
@n type of geometry_type:  string
```

Parameters

geometry_type Specifies the geometry type ("body", "volume", "surface", "curve", or "vertex"). type of entity_list: std::vector< int,std::allocator< int > >

entity_list List of IDs associated with geometry_type. return type of : std:: array< double,10 >

Returns

A 10-element array containing the combined bounding box parameters.

Title Array contents

- [0] x-min
- [1] x-max
- [2] x-range (x-max ? x-min)
- [3] y-min
- [4] y-max
- [5] y-range (y-max ? y-min)
- [6] z-min
- [7] z-max
- [8] z-range (z-max ? z-min)
- [9] diagonal length of the bounding box

◆ get_total_volume()

def cubit.get_total_volume (*volume_list*)

Get the total volume for a list of volumes.

.. code-block:: python

```
                # Assume volumes 1, 2, and 3
                exist.

                volume_list = [1, 2, 3]

                total = cubit.get_total_volume
                (volume_list)

                print("Total
                volume:" , total)
```

```
@n type of volume_list:  std::vector< int,std::allocator< int > >
```

Parameters

volume_list List of volume IDs to include. return type of : float

Returns

The sum of the volumes for all specified IDs.

◆ get_tri_count()

def cubit.get_tri_count ( )

Retrieve the count of triangular elements in the current model.

For a tetrahedral volume mesh, returns the number of surface triangles.

```
.. code-block:: python
```

```
cubit.cmd
("brick x 10" )
# create brick (Volume ID =
1)


cubit.cmd
("vol 1 scheme
tetmesh" )


cubit.cmd
("mesh vol 1" )
# mesh as tets (surface faces are
tris)


tri_count = cubit.get_tri_count
()


print("Tri count:" ,
tri_count)
```

```
@n return type of :  int
```

Returns

Number of triangular elements (surface faces of a tet mesh) in the current model's mesh.

◆ get_tri_global_element_id()

def cubit.get_tri_global_element_id (*tri_id*)

Get the Global Element ID for a specific triangular element.

Cubit assigns a Global Element ID to each element when it is placed into a block. This function returns the Global Element ID corresponding to the given local tri ID within its type-specific ID space.

```
.. code-block:: python
```

```
                    # Create a brick and mesh its
                    surfaces with triangles

                    cubit.cmd
                    ("brick x 10" )

                    cubit.cmd
                    ("surface 1 scheme
                    trimesh" )

                    cubit.cmd
                    ("mesh surface 1" )


                    # Assign all tri elements on
                    surface 1 to block 1

                    cubit.cmd
                    ("block 1 tri all" )


                    # Query the Global Element ID of
                    local tri 22

                    gid = cubit.get_tri_global_element_id
                    (22)

                    print("Global Element ID of
                    tri 22:" , gid)
```

```
 @n type of tri_id:  int
```

Parameters

tri_id Local triangular element ID within its type.
return type of : int

Returns

Global Element ID for the specified triangle, or 0 if not placed into a block.

◆ get_trimesh_geometry_sizing()

def cubit.get_trimesh_geometry_sizing ( )

Retrieve the global geometry sizing flag for the TriMesh scheme.

If ON, TriMesh will incorporate the geometry approximation angle when computing element sizes. If OFF, geometry approximation is ignored.

.. code-block:: python

```
                # 1) Enable geometry-based sizing
                globally.

                cubit.cmd
                ("set trimesher geometry
                sizing on" )


                # 2) Query the geometry sizing
                flag.

                geom_size_on =
      cubit.get_trimesh_geometry_sizing
                ()

                print("TriMesh geometry
                sizing enabled?" , geom_size_on) #
      Expected: True
```

@n return type of : boolean

Returns
True if TriMesh geometry sizing is ON; false otherwise.

◆ get_trimesh_num_anisotropic_layers()

def cubit.get_trimesh_num_anisotropic_layers ( )

Retrieve the global number of anisotropic triangle layers for the TriMesh scheme.

When ON, TriMesh will attempt to place specified layers of triangles in thin surface regions. Returns the number of layers; zero if anisotropic layers are disabled.

.. code-block:: python

```
                # 1) Enable TriMesh anisotropic
                layers with 3 layers globally.

                cubit.cmd
                ("set trimesher anisotropic
                layers on 3" )


                # 2) Query the number of
                anisotropic layers.

                layers =
      cubit.get_trimesh_num_anisotropic_layers
                ()

                print("Global TriMesh
                anisotropic layers:" , layers) #
      Expected: 3
```

@n return type of : int

Returns
Number of anisotropic layers (int); 0 if disabled.

◆ get_trimesh_ridge_angle()

def cubit.get_trimesh_ridge_angle ( )

Retrieve the global ridge angle for the TriMesh scheme.

The ridge angle determines which angles in a discrete surface are preserved as ridges when meshing. Triangles on discrete surfaces with dihedral angles above this threshold remain as ridges.

```
.. code-block:: python
```

```
                # 1) Set TriMesh ridge angle to
                30 degrees.

                cubit.cmd
                ("set trimesher ridge angle
                30" )


                # 2) Query the current ridge
                angle.

                ridge = cubit.get_trimesh_ridge_angle
                ()

                print("TriMesh ridge
                angle:" , ridge, "degrees" ) #
      Expected: 30
```

```
 @n return type of :  float
```

Returns

Ridge angle in degrees (double).

◆ get_trimesh_split_overconstrained_edges()

def cubit.get_trimesh_split_overconstrained_edges ( )

Retrieve the global flag for splitting overconstrained edges in the TriMesh scheme.

If ON, TriMesh will split edges owned by a surface but whose nodes lie on curves, ensuring two elements through thickness in those regions.

```
.. code-block:: python
```

```
                # 1) Enable splitting
                overconstrained edges globally.

                cubit.cmd
                ("set trimesher split
                overconstrained edges on" )


                # 2) Query the split
                overconstrained edges flag.

                split_on =
      cubit.get_trimesh_split_overconstrained_edges
                ()

                print("TriMesh split
                overconstrained edges?" , split_on) #
      Expected: True
```

```
 @n return type of :  boolean
```

Returns

True if splitting overconstrained edges is ON; false otherwise.

◆ get_trimesh_surface_gradation()

def cubit.get_trimesh_surface_gradation ( )

Retrieve the global surface mesh gradation for the TriMesh scheme.

The surface gradation controls how quickly triangle sizes change on surfaces when using MeshGems. A value > 1.0 limits the size ratio of adjacent triangles; larger values yield faster growth.

```
.. code-block:: python
```

```
                # 1) Set global TriMesh surface
                gradation to 1.2.


                cubit.cmd
                ("set trimesher surface
                gradation 1.2" )



                # 2) Query the current surface
                gradation.


                grad =
    cubit.get_trimesh_surface_gradation
                ()


                print("Global TriMesh
                surface gradation:" , grad) # Expected:
    1.2
```

```
 @n return type of :  float
```

Returns
Global surface gradation value (double).

◆ get_trimesh_surface_proximity_ratio()

def cubit.get_trimesh_surface_proximity_ratio ( )

Retrieve the global surface proximity ratio for the TriMesh scheme.

The proximity ratio is used when surface proximity is ON to scale the feature-based size in thin regions.

```
.. code-block:: python
```

```
                # 1) Enable surface proximity
                with a ratio of 2.0 globally.


                cubit.cmd
                ("set trimesher surface
                proximity on ratio 2.0" )



                # 2) Query the surface proximity
                ratio.


                ratio =
    cubit.get_trimesh_surface_proximity_ratio
                ()


                print("TriMesh surface
                proximity ratio:" , ratio) # Expected:
    2.0
```

```
 @n return type of :  float
```

Returns
Global surface proximity ratio (double).

◆ get_trimesh_target_min_size()

def cubit.get_trimesh_target_min_size (*geometry_type*, *entity_id*)

Retrieve the target minimum triangle size set on a specific geometry entity for the TriMesh scheme.

Returns the local "minimum size" constraint for the given entity (curve, surface, or volume) when using TriMesh. If no minimum was set on that entity, returns 0.

.. code-block:: python

```
                # 1) Create a surface (Surface ID
                = 12) and set its local TriMesh minimum
    size to
                0.05.


                cubit.cmd
                ("surface 12 scheme trimesh
                minimum size 0.05" )



                # 2) Query the target minimum
                size for surface 12.


                min_size =
    cubit.get_trimesh_target_min_size
                ("surface" , 12)


                print("Surface 12 TriMesh
                minimum size:" , min_size) # Expected:
    0.05
```

Parameters

geom_type Geometry type of the entity ("curve", "surface", or "volume").
type of entity_id: int

entity_id ID of the entity whose TriMesh minimum size is queried.
return type of : float

Returns

Target minimum triangle size (double) set on that entity; 0 if not set.

◆ get_trimesh_tiny_edge_length()

def cubit.get_trimesh_tiny_edge_length ( )

Retrieve the global tiny edge length for the TriMesh scheme.

The tiny edge length setting defines the minimum edge length below which elements are considered "tiny". TriMesh may use this value during cleanup or facet generation.

.. code-block:: python

```
                # 1) Set TriMesh tiny edge length
                to 0.01.


                cubit.cmd
                ("set trimesher tiny edge
                length 0.01" )



                # 2) Query the current tiny edge
                length.


                tiny =
    cubit.get_trimesh_tiny_edge_length
                ()


                print("TriMesh tiny edge
                length:" , tiny) #
                Expected: 0.01
```

@n return type of : float

Returns

Global tiny edge length value (double).

◆ get_trimesh_use_surface_proximity()

def cubit.get_trimesh_use_surface_proximity ( )

Retrieve the global surface proximity flag for the TriMesh scheme.

When enabled, surface proximity adds refinement in thin regions of surfaces based on a proximity ratio.

.. code-block:: python

```
                # 1) Enable surface proximity
                globally with default ratio of 1.0.


                cubit.cmd
                ("set trimesher surface
                proximity on" )



                # 2) Query whether surface
                proximity is on.


                prox =
      cubit.get_trimesh_use_surface_proximity
                ()


                print("TriMesh surface
                proximity enabled?" , prox) # Expected:
      True
```

@n return type of : boolean

Returns
True if global surface proximity is ON; false if OFF.

◆ get_trimesh_volume_gradation()

def cubit.get_trimesh_volume_gradation ( )

Retrieve the global volume mesh gradation for the TriMesh scheme.

The volume gradation controls how triangle sizes on adjacent surfaces respond to nearby features. This affects how small surface triangles propagate to neighboring regions.

.. code-block:: python

```
                # 1) Set global TriMesh volume
                gradation to 1.5.


                cubit.cmd
                ("set trimesher volume
                gradation 1.5" )



                # 2) Query the current volume
                gradation.


                vgrad =
      cubit.get_trimesh_volume_gradation
                ()


                print("Global TriMesh
                volume gradation:" , vgrad) # Expected:
      1.5
```

@n return type of : float

Returns
Global volume gradation value (double).

◆ get_undo_enabled()

def cubit.get_undo_enabled ( )

Query whether undo is currently enabled.

.. code-block:: python

```
# Enable undo.


cubit.cmd
("set undo on" )


enabled = cubit.get_undo_enabled
() # returns True


print("Undo
enabled:" , enabled)


# Disable undo.


cubit.cmd
("set undo off" )


enabled = cubit.get_undo_enabled
() # returns False


print("Undo
enabled:" , enabled)
```

```
@n return type of :  boolean
```

Returns
True if undo is enabled; otherwise false.

◆ get_unmerged_curves_on_shells()

def cubit.get_unmerged_curves_on_shells (*shell_vols*, *thickness*)

Identify unmerged curves between sheet (shell) volumes for use in shell construction workflows.

This function is normally used in the construction of shell representations from thin volumes. It identifies unmerged curve IDs on the specified shell_vols that are within the corresponding thickness distance of faces on other shell volumes in the list. These curves typically represent locations where the sheet bodies would need to be extended or trimmed to fully match the geometry of the original thin volumes.

Only sheet (shell) volumes are processed; solid volumes are ignored.

The thickness list specifies a proximity distance (normally the thickness of the original thin volumes) to use for detecting nearby curves. The list must match the length of shell_vols.

.. code-block:: python

```
cubit.cmd
('brick x 10 y 10 z
1' ) # Thin volume
1


cubit.cmd
('brick x 10 y 1 z
10' ) # Thin volume
2


cubit.cmd
('vol 2 mov x y 4.5 z
5.5' ) # Position thin
volume 2 to create L shape


cubit.cmd
('surface 2 copy' )
# Generates sheet body 3


cubit.cmd
('surface 11 copy' )
# Generates sheet body 4


shell_vols = [3, 4] # The sheet
body volume IDs


thickness = [1.0, 1.0] #
Thickness of original thin volumes


unmerged_curves =
cubit.get_unmerged_curves_on_shells
(shell_vols, thickness)


print("Unmerged curves
between sheet bodies:" )
```

for c_id in unmerged_curves: print(c_id)

```
# Expected output:


# 28


# 30


# Example use: Extend curve 30 to
fill the gap between the two sheet bodies:


cubit.cmd
('tweak curve 30 target
surface 2' )
```

@n type of shell_vols: std::vector< int,std::allocator< int > >

Parameters

shell_vols List of volume IDs to check. Only sheet (shell) volumes will be processed.
type of thickness: std::vector< double,std::allocator< double > >

thickness List of thickness values (proximity distance) corresponding to shell_vols. Length must match shell_vols.
return type of : std::vector< int,std:: allocator< int > >

Returns

List of unmerged curve IDs on the specified shell volumes that are in proximity to faces on other shell volumes.

◆ get_valence()

def cubit.get_valence (*vertex_id*)

Get the valence (number of incident edges) for a specific vertex.

The valence of a vertex is the number of curves (edges) that meet at that vertex.

```
.. code-block:: python
```

```
                # Create a 10x10x10 brick
                centered at the origin; vertex 1 is a
    corner with three
                incident edges.


                cubit.cmd
                ("brick x 10" )


                val = cubit.get_valence
                (1)


                print("Valence of vertex
                1:" , val) # prints
                3
```

```
@n type of vertex_id:  int
```

Parameters

vertex_id ID of the vertex to query.
return type of : int

Returns

The number of edges (curves) meeting at the specified vertex.

◆ get_valid_block_element_types()

def cubit.get_valid_block_element_types (*block_id*)

Get a list of valid element types for the specified block.

Returns the list of supported element types (e.g., "hex8", "hex20", "tet10") that can be assigned to the given block, based on its associated geometry or mesh configuration.

```
.. code-block:: python
```

```
                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("block 1 vol 1" )


                valid_types =
    cubit.get_valid_block_element_types
                (1)
```

for t in valid_types: print(t)

```
  @n type of block_id:  int
```

Parameters

block_id ID of the block.
return type of : std::vector<
std::string,std::allocator< std:: string > >

Returns

Vector of supported element type strings.

◆ get_velocity_combine_type()

def cubit.get_velocity_combine_type (*entity_id*)

Get the velocity's combine type.

Returns how multiple velocity boundary conditions are combined. Possible values include "Overwrite", "Average", "SmallestCombine", or "LargestCombine".

type of entity_id: int

Parameters

entity_id ID of the velocity boundary condition.
return type of : string

Returns

Combine type string for the given velocity BC.

◆ get_version()

def cubit.get_version ( )

Retrieve the current Cubit version string.

Returns the version identifier of the active Cubit engine, typically in "major.minor.patch" format.

```
.. code-block:: python
```

```
                import cubit


                version = cubit.get_version
                ()


                print("Cubit
                version:" , version)
```

```
@n return type of :  string
```

Returns
A string containing the current Cubit version.

◆ get_vertex_coordinates()

def cubit.get_vertex_coordinates (*entityID*)

Get the 3D coordinates of a vertex.

Returns the exact 3D coordinates for a vertex.

```
.. code-block:: python
```

```
                v_id = 12 # vertex 12


                vcoord = cubit.get_vertex_coordinates
                (v_id)


                print(f"Vertex {v_id}
                coords: {vcoord}" )
```

Parameters
entity_id ID of the vertex to query.
return type of : std:: array< double,3 >

Returns
A 3-element array [x, y, z] representing the vertex coordinates.

◆ get_vertex_count()

def cubit.get_vertex_count ( )

Get the current number of vertices in the model.

.. code-block:: python

```
                # Create a 10x10x10 brick; this
                generates eight corner vertices.


                cubit.cmd
                ("brick x 10" )


                vert_count = cubit.get_vertex_count
                () # returns 8


                print("Vertex
                count:" , vert_count)
```

```
@n return type of :  int
```

Returns
The number of vertices currently defined in the model.

◆ get_vertex_node()

def cubit.get_vertex_node (*vertex_id*)

Get the node ID owned by a vertex.

This function returns the ID of the node associated with the specified vertex. If the vertex does not have an associated node, the function returns -1.

```
.. code-block:: python
```

```
# Create a brick and mesh volume
1


cubit.cmd
("brick x 10" )


cubit.cmd
("mesh vol 1" )


# Query the node owned by vertex
1


node_id = cubit.get_vertex_node
(1)


print(f"Vertex 1 owns node
ID: {node_id}" )
```

```
@n type of vertex_id:  int
```

Parameters

vertex_id ID of the vertex.
return type of : int

Returns

ID of the node owned by the vertex. Returns -1 if no node exists.

◆ get_vertex_type()

def cubit.get_vertex_type (*surface_id*,
vertex_id
)

Retrieve the vertex type for a given vertex on a surface.

Vertex types classify each corner based on how many mesh elements meet there:

- "end" : interior angle ~ 90deg (one element)
- "side" : interior angle ~ 180deg (two elements)
- "corner" : interior angle ~ 270deg (three elements)
- "reverse" : interior angle ~ 360deg (four elements)
- "unknown" : not classified or not applicable

```
.. code-block:: python
```

```
# Create a brick; surface 1 has
four "end" vertices.


cubit.cmd
("brick x 10" )


# Query the type of vertex 1 on
surface 1.


vtype = cubit.get_vertex_type
(1, 1)


print("Vertex 1 on surface
1 type:" , vtype)
```

```
@n type of surface_id:  int
```

Parameters

surface_id ID of the surface containing the vertex.
type of vertex_id: int

vertex_id ID of the vertex to query.
return type of : string

Returns

: The vertex type "end", "side", "corner", "reverse", or "unknown".

◆ get_view_at()

def cubit.get_view_at ( )

Get the camera's current "at" (target) point.

Returns the xyz coordinates of the point that the camera is looking at. Requires an active graphics window; if graphics are disabled, this call may fail.

```
.. code-block:: python
```

```
                    import cubit


                    # Display current view (outputs
                    from/at/up vectors)


                    cubit.cmd
                    ("journal view" )


                    # Get the "at" coordinates


                    at = cubit.get_view_at
                    ()


                    print("Camera at:" ,
                    at) # e.g., [0.0, 0.0,
                    0.0]


                    # Set a new "at" point via Cubit
                    command


                    cubit.cmd
                    ("at 5 5 5" )


                    at = cubit.get_view_at
                    ()


                    print("Camera at after
                    setting:" , at)
```

```
 @n return type of : std:: array< double,3 >
```

Returns
: Array of three doubles {x, y, z} representing the camera's target point.

◆ get_view_distance()

def cubit.get_view_distance ( )

Get the camera's distance between its position and focus point.

Returns the straight-line distance from the camera "from" point to its "at" point. Requires an active graphics window; if graphics are disabled, this call may fail.

```
.. code-block:: python
```

```
                    import cubit


                    # Display current view parameters
                    (prints from/at/up vectors)


                    cubit.cmd
                    ("journal view" )


                    # Explicitly set the camera
                    'from' point to (10, 5, 15)


                    cubit.cmd
                    ("from 10 5 15" )


                    # Now get the updated camera
                    distance


                    dist = cubit.get_view_distance
                    ()


                    print("Camera distance
                    after setting 'from':" , dist)
```

```
 @n return type of :  float
```

Returns
: Distance (double) between the camera position ("from") and its focus point ("at").

◆ get_view_from()

def cubit.get_view_from ( )

Get the camera's current 'from' (position) point.

Returns the xyz coordinates of the camera's location in space. Requires an active graphics window; if graphics are disabled, this call may fail.

```
.. code-block:: python
```

```
                    import cubit


                    # Display current view (outputs
                    from/at/up vectors)


                    cubit.cmd
                    ("journal view" )


                    # Get the "from"
                    coordinates


                    frm = cubit.get_view_from
                    ()


                    print("Camera from:"
                    , frm) # e.g., [10.0, 5.0,
                    15.0]


                    # Move the camera to a new "from"
                    point


                    cubit.cmd
                    ("from 20 10 30" )


                    frm = cubit.get_view_from
                    ()


                    print("Camera from after
                    setting:" , frm)
```

```
 @n return type of : std:: array< double,3 >
```

Returns

Array of three doubles {x, y, z} representing the camera's position.

◆ get_view_up()

def cubit.get_view_up ( )

Get the camera's 'up' direction vector.

Returns the normalized xyz vector that defines the upward orientation of the camera. Requires an active graphics window; if graphics are disabled, this call may fail.

```
.. code-block:: python
```

```
                    import cubit


                    # Display current view (outputs
                    from/at/up vectors)


                    cubit.cmd
                    ("journal view" )


                    # Get the "up" coordinates


                    up = cubit.get_view_up
                    ()


                    print("Camera up
                    vector:" , up) # e.g.,
                    [0.0, 1.0, 0.0]


                    # Change the camera up direction
                    (example command)


                    cubit.cmd
                    ("up 0 1 0" )


                    up = cubit.get_view_up
                    ()


                    print("Camera up vector
                    after setting:" , up)
```

```
 @n return type of : std:: array< double,3 >
```

Returns

Array of three doubles {x, y, z} representing the camera's up direction vector.

◆ get_volume_area()

def cubit.get_volume_area (*volume_id*)

Get the total surface area of a specified volume.

This returns the sum of all face areas bounding the volume.

```
.. code-block:: python
```

```
                # Create a 10x10x10 brick
                centered at the origin; volume 1 is
      generated.


                cubit.cmd
                ("brick x 10" )


                area = cubit.get_volume_area
                (1) # 6 faces x (10x10) =
                600


                print("Volume 1 surface
                area:" , area) # prints
                600
```

```
 @n type of volume_id:  int
```

Parameters

volume_id ID of the volume to query.

return type of : float

Returns

Total surface area of the specified volume.

◆ get_volume_CAD_material()

def cubit.get_volume_CAD_material (*volume_id*)

◆ get_volume_count()

def cubit.get_volume_count ( )

Get the current number of volumes in the model.

```
.. code-block:: python
```

```
                # Create a 10x10x10 brick; this
                generates one volume (ID 1).


                cubit.cmd
                ("brick x 10" )


                vol_count = cubit.get_volume_count
                () # returns 1


                print("Volume
                count:" , vol_count)
```

```
 @n return type of :  int
```

Returns

The number of volumes currently defined in the model.

◆ get_volume_element_count()

def cubit.get_volume_element_count (*volume_id*)

Retrieve the count of 3D elements in a specified volume.

Returns the total number of hexahedra, tetrahedra, pyramids, and wedges within the given volume (regardless of block assignments).

```
.. code-block:: python
```

```
                cubit.cmd
                ("brick x 10" )
                # create brick (Volume ID =
                1)


                cubit.cmd
                ("vol 1 scheme
                tetmesh" )


                cubit.cmd
                ("mesh vol 1" )
                # mesh as tets


                count3D =
      cubit.get_volume_element_count
                (1)


                print("3D elements in
                volume 1:" , count3D)
```

```
@n type of volume_id:  int
```

Parameters

volume_id ID of the volume to query.
return type of : int

Returns

Number of hexes, tets, pyramids, and wedges in the specified volume.

◆ get_volume_gap_solutions()

def cubit.get_volume_gap_solutions (*surface_id_1*, *surface_id_2*)

Suggests remedies for a gap between two surfaces on different volumes.

Given two surface IDs from different volumes that are close but separated by a gap, this function provides options to close or reconcile the gap.

To identify surface gaps, use get_gaps_between_volumes or get_volume_gaps.

.. code-block:: python

```
                cubit.cmd
                ("brick x 10" )

                cubit.cmd
                ("brick x 10" )

                cubit.cmd
                ("volume 2 move x
                10.1" ) # small gap
                between face 6 and face 10

                solutions =
      cubit.get_volume_gap_solutions
                (6, 10)
```

for i in range(len(solutions[0])): print("Option:", solutions[0][i])

```
                print(" Command:" ,
                solutions[1][i])

                print(" Preview:" ,
                solutions[2][i])

                print(" Shorthand:"
                , solutions[3][i])

                # Expected output (first two
                results):

                # Option: Replace Surface 6 with
                Surface 10

                # Command: tweak surface 6
                replace with surface 10

                # Preview: tweak surface 6
                replace with surface 10 preview

                # Shorthand: 6 6 10

                # Option: Replace Surface 10 with
                Surface 6

                # Command: tweak surface 10
                replace with surface 6

                # Preview: tweak surface 10
                replace with surface 6 preview

                # Shorthand: 6 10 6
```

@n type of surface_id_1: int

Parameters

surface_id_1 ID of the first surface forming a gap (must be on a different volume than surface_id_2). type of surface_id_2: int

surface_id_2 ID of the second surface forming a gap (must be on a different volume than surface_id_1). return type of : std::vector< std::vector< std::string,std::allocator< std::string > >,std::allocator< std::vector< std::string,std::allocator< std:: string > > > >

Returns

A vector of four aligned string vectors: [0] Display strings for each recommended remedy, [1] Cubit command strings to apply each remedy, [2] Cubit preview command strings for each remedy, [3] Shorthand descriptions (operation ID and surface/volume IDs).

◆ get_volume_hexes()

def cubit.get_volume_hexes (*volume_id*)

Get the list of hexahedron (hex) IDs contained in a volume.

This function returns the IDs of all hexahedron (hex) elements that are part of the specified volume.

```
.. code-block:: python
```

```
                    # Create a brick and mesh volume
                    1


                    cubit.cmd
                    ("brick x 10" )


                    cubit.cmd
                    ("mesh vol 1" )


                    # Query the hexes in volume
                    1


                    hex_ids = cubit.get_volume_hexes
                    (1)


                    print(f"Volume 1 contains
                    {len(hex_ids)} hex(s)." )
```

for hex_id in hex_ids: print(f" Hex ID: {hex_id}")

```
  @n type of volume_id:  int
```

Parameters

volume_id ID of the volume to examine.
return type of : std::vector< int,std:: allocator< int > >

Returns

List of hexahedron (hex) IDs contained in the volume.

◆ get_volume_nodes()

def cubit.get_volume_nodes (*volume_id*)

Get the list of node IDs owned by a volume.

This function returns the IDs of nodes owned by the specified volume. Nodes on bounding surfaces, curves, and vertices are excluded. Only interior (volume-owned) nodes are returned.

```
.. code-block:: python
```

```
                    # Create a brick and mesh volume
                    1


                    cubit.cmd
                    ("brick x 10" )


                    cubit.cmd
                    ("mesh vol 1" )


                    # Query the nodes owned by volume
                    1


                    volume_nodes = cubit.get_volume_nodes
                    (1)


                    print(f"Volume 1 owns
                    {len(volume_nodes)} node(s)." )
```

for node_id in volume_nodes: print(f" Node ID: {node_id}")

```
  @n type of volume_id:  int
```

Parameters

volume_id ID of the volume.
return type of : std::vector< int,std:: allocator< int > >

Returns

List of node IDs owned by the volume (excluding nodes on bounding surfaces, curves, and vertices).

◆ get_volume_pyramids()

def cubit.get_volume_pyramids (*volume_id*)

Get the list of pyramid IDs contained in a volume.

This function returns the IDs of all pyramid elements that are part of the specified volume.

```
.. code-block:: python
```

```
                # Create two bricks and mesh them
                with different schemes


                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("vol 1 copy mov x
                10" )


                cubit.cmd
                ("merge all" )



                cubit.cmd
                ("vol 1 scheme map"
                )


                cubit.cmd
                ("mesh vol 1" )



                cubit.cmd
                ("vol 2 scheme
                tetmesh" )


                cubit.cmd
                ("mesh vol 2" )



                # Query the pyramids in volume
                2


                pyramid_ids = cubit.get_volume_pyramids
                (2)


                print(f"Volume 2 contains
                {len(pyramid_ids)} pyramid(s)." )
```

for pyr_id in pyramid_ids: print(f" Pyramid ID: {pyr_id}")

```
  @n type of volume_id:  int
```

Parameters

volume_id ID of the volume to examine.
return type of : std::vector< int,std:: allocator< int > >

Returns

List of pyramid IDs contained in the volume.

◆ get_volume_tets()

def cubit.get_volume_tets (*volume_id*)

Get the list of tetrahedron (tet) IDs contained in a volume.

This function returns the IDs of all tetrahedron (tet) elements that are part of the specified volume.

```
.. code-block:: python
```

```
                # Create a brick and tetmesh
                volume 1


                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("vol 1 scheme
                tetmesh" )


                cubit.cmd
                ("mesh vol 1" )


                # Query the tets in volume
                1


                tet_ids = cubit.get_volume_tets
                (1)


                print(f"Volume 1 contains
                {len(tet_ids)} tet(s)." )
```

for tet_id in tet_ids: print(f" Tet ID: {tet_id}")

```
  @n type of volume_id:  int
```

Parameters

volume_id ID of the volume to examine.
return type of : std::vector< int,std:: allocator< int > >

Returns

List of tetrahedron (tet) IDs contained in the volume.

◆ get_volume_volume()

def cubit.get_volume_volume (*vol_id*)

Get the enclosed volume of a specified volume.

.. code-block:: python

```
                # Create a 10x10x10 brick
                centered at the origin; volume 1 is
      generated.


                cubit.cmd
                ("brick x 10" )


                vol = cubit.get_volume_volume
                (1) # returns 1000


                print("Volume 1
                volume:" , vol) # prints
                1000
```

```
 @n type of vol_id:  int
```

Parameters

vol_id ID of the volume to query.
return type of : float

Returns

The enclosed volume of the specified volume.

◆ get_volume_wedges()

def cubit.get_volume_wedges (*volume_id*)

Get the list of wedge IDs contained in a volume.

This function returns the IDs of all wedge elements that are part of the specified volume.

```
.. code-block:: python
```

```
                # Create a brick and mesh surface
                1 with triangles


                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("surface 1 scheme
                trimesh" )


                cubit.cmd
                ("mesh surf 1" )


                # Sweep the surface mesh to
                create wedges in volume 1


                cubit.cmd
                ("vol 1 scheme
                sweep" )


                cubit.cmd
                ("mesh vol 1" )


                # Query the wedges in volume
                1


                wedge_ids = cubit.get_volume_wedges
                (1)


                print(f"Volume 1 contains
                {len(wedge_ids)} wedge(s)." )
```

for wedge_id in wedge_ids: print(f" Wedge ID: {wedge_id}")

```
  @n type of volume_id:  int
```

Parameters

volume_id ID of the volume to examine.
return type of : std::vector< int,std:: allocator< int > >

Returns

List of wedge IDs contained in the volume.

◆ get_volumes_for_node()

def cubit.get_volumes_for_node (*node_name*, *node_instance*)

◆ get_wedge_count()

def cubit.get_wedge_count ( )

Retrieve the count of wedge-shaped elements in the current model.

.. code-block:: python

```
cubit.cmd
("brick x 10" )
# create brick (Volume ID =
1)


cubit.cmd
("surface 1 scheme
trimesh" ) # set
triangular scheme on surface 1


cubit.cmd
("mesh surface 1" )
# mesh surface 1 with
triangles


cubit.cmd
("mesh vol 1" )
# sweep triangles, generating
wedges


wedge_count = cubit.get_wedge_count
()


print("Wedge count:"
, wedge_count)
```

```
@n return type of :  int
```

Returns

Number of wedge-shaped elements in the current model (e.g., generated by sweeping triangles).

◆ get_wedge_global_element_id()

def cubit.get_wedge_global_element_id (*wedge_id*)

Get the Global Element ID for a specific wedge element.

This function returns the Global Element ID assigned to the given local wedge ID when the element is placed into a block.

```
.. code-block:: python
```

```
                # Create a brick and mesh its top
                surface with triangles

                cubit.cmd
                ("brick x 10" )

                cubit.cmd
                ("surface 1 scheme
                trimesh" )

                cubit.cmd
                ("mesh surface 1" )


                # Sweep the triangular surface to
                generate wedges in volume 1

                cubit.cmd
                ("vol 1 scheme
                sweep" )

                cubit.cmd
                ("mesh vol 1" )


                # Assign all wedges in volume 1
                to block 1 (establishes global IDs)

                cubit.cmd
                ("block 1 wedge all"
                )


                # Query the Global Element ID of
                local wedge 22

                gid = cubit.get_wedge_global_element_id
                (22)

                print("Global Element ID of
                wedge 22:" , gid)
```

```
 @n type of wedge_id:  int
```

Parameters

wedge_id Local wedge element ID within its type.
return type of : int

Returns

Global Element ID for the specified wedge, or 0 if not placed into a block.

◆ get_wrt_entity()

def cubit.get_wrt_entity (*source_type*,
source_id,
sideset_id
)

Get the 'with-respect-to' entity of an item in a sideset.

This function returns the "with-respect-to" (WRT) entity associated with the specified entity (face, quad, or tri) in the specified sideset. The WRT entity provides context for how the sideset references or is oriented with respect to the entity.

```
.. code-block:: python
```

```
# Query the with-respect-to
entity of face 332 in sideset 2

wrt_entity = cubit.get_wrt_entity
("face" , 332, 2)

print(f"WRT entity for face
332 in sideset 2: {wrt_entity}" )
```

```
@n type of source_type:  string
```

Parameters

source_type Item type: "face", "quad", or "tri".
type of source_id: int

source_id ID of the entity.
type of sideset_id: int

sideset_id ID of the sideset.
return type of : string

Returns

'With-respect-to' entity of the specified entity in the sideset.

◆ group_names_ids()

def cubit.group_names_ids ( )

Retrieve names and IDs of all defined groups (excluding the pick group).

Returns a vector of (name, ID) pairs for each active group in the current session.

```
.. code-block:: python
```

```
# Create and name two
groups.

cubit.cmd
("brick x 10" )

cubit.cmd
("group \"g1\" add vertex
in vol 1" )

cubit.cmd
("group \"g2\" add volume
in vol 1" )

# Fetch group names and
IDs.

groups = cubit.group_names_ids
()
```

for name, gid in groups: print(f"Group '{name}' has ID {gid}")

```
@n return type of : std::vector< std::pair< std::string,int >,std::allocator< std::pair< std:: string,int > > >
```

Returns

Vector of (group name, group ID) pairs.

◆ has_valid_size()

def cubit.has_valid_size (*geometry_type*, *entity_id*)

Check whether a geometric entity has a valid size.

Title This function returns whether the specified geometric entity currently has a valid size.

- All entities have a size if **auto sizing** is enabled.
- If **auto sizing** is off, an entity only has a size if it has been explicitly set (e.g. via a `size` command).

Title Return value:

- `1` if the entity has a valid size
- `0` if the entity does not have a valid size

`.. code-block:: python`

```
                # Create a brick

                cubit.cmd
                ("brick x 10" )


                # Check whether surface 1 has a
                valid size

                has_size = cubit.has_valid_size
                ("surface" , 1)

                print(f"Surface 1 has valid
                size? {'Yes' if has_size else 'No'}" )
```

`@n type of geometry_type:  string`

Parameters

geometry_type Type of the geometric entity (e.g. "surface", "curve", "volume", etc.).
type of entity_id: int

entity_id ID of the geometric entity.
return type of : int

Returns

`1` if the entity has a valid size, `0` if it does not.

◆ heatflux_is_on_shell_area()

def cubit.heatflux_is_on_shell_area (*bc_area_enum*, *entity_id*)

Check if a heatflux BC is on a shell area.

type of bc_area_enum: int

Parameters

bc_area_enum CI_BCEntityTypes: 7 (top), 8 (bottom).
type of entity_id: int

entity_id ID of the heatflux BC.
return type of : boolean

Returns

True if the BC is applied to the specified shell area.

◆ highlight()

def cubit.highlight (*entity_type*,
entity_id
)

Highlight the given entity in the graphics window without selecting it.

Adds a visual highlight (for example, an orange colored overlay) to the specified entity. Unlike selection, highlighting does not add the entity to the pick list or change its selection state. Selecting an entity (for example, via "select volume 5") also produces a highlight, but calling this function only affects appearance.

```
.. code-block:: python
```

```
import cubit

# Select volume 5 (adds to pick
list and highlights it)

cubit.cmd
("select volume 5" )

# Highlight volume 6 without
selecting it

cubit.highlight
("volume" , 6)

# Retrieve the list of selected
IDs (only volume 5 is selected)

ids = cubit.get_selected_ids
()

print("Selected
IDs:" , ids)
```

```
@n type of entity_type:  string
```

Parameters

entity_type **Entity** type to highlight ("vertex", "curve", "surface", or "volume").
type of entity_id: int

entity_id ID of the entity to highlight.

◆ init()

def cubit.init (*argv*)

Initialize the Cubit engine with optional startup arguments.

Must be called before any other Cubit API functions. Passing a single empty string in the argument list will launch Cubit with default settings.

```
.. code-block:: python
```

```
import cubit

cubit.init
([""])
```

type of argv: std::vector< std::string,std::allocator< std::string > >

Parameters

argv List of startup directives for Cubit (e.g., command-line flags). Use {""} for defaults.

◆ is_acis_engine_available()

def cubit.is_acis_engine_available ( )

Determine whether the ACIS geometry engine is available.

Notes ACIS is the default engine in standard Cubit distributions and should always be available.

```
.. code-block:: python

                has_acis =
      cubit.is_acis_engine_available
                ()


                print("ACIS engine
                available:" , has_acis)
```

```
@n return type of :  boolean
```

Returns
True if ACIS support is present; otherwise false.

◆ is_assembly_metadata_attached()

def cubit.is_assembly_metadata_attached (*volume_id*)

Check if metadata is attached to a volume.

Used to validate part association or identify unassociated geometry.
type of volume_id: int

Parameters
volume_id ID of the volume.
return type of : boolean

Returns
True if metadata exists, false otherwise.

◆ is_blend_surface()

def cubit.is_blend_surface (*surface_id*)

Determine if a face's underlying surface is a blend (fillet).

Checks whether the surface with ID surface_id represents a blend created by a fillet operation.

```
.. code-block:: python

                # Create a 2x2x2 brick and fillet
                edge 1 with radius 0.5.


                cubit.cmd
                ("brick x 2 y 2 z 2"
                )


                cubit.cmd
                ("modify curve 1 blend
                radius 0.5" )


                # Surface 7 should now be a
                blend.


                is_blend = cubit.is_blend_surface
                (7)


                print("Surface 7 is
                blend:" , is_blend)
```

```
@n type of surface_id:  int
```

Parameters
surface_id ID of the surface to check.
return type of : boolean

Returns
True if the surface is a blend (fillet); otherwise false.

◆ is_boundary_layer_id_available()

def cubit.is_boundary_layer_id_available (*boundary_layer_id*)

Check if a boundary layer ID is currently available.

Useful for verifying uniqueness before assignment.
type of boundary_layer_id: int

Parameters
boundary_layer_id Boundary layer ID to check.
return type of : boolean

Returns
True if ID is available, false if already in use.

◆ is_catia_engine_available()

def cubit.is_catia_engine_available ( )

Determine whether the CATIA geometry engine is available.

Notes CATIA is not included in standard Cubit distributions. Only special builds with CATIA support will return true.

```
.. code-block:: python
```

```
                has_catia =
        cubit.is_catia_engine_available
                ()


                print("CATIA engine
                available:" , has_catia)
```

```
 @n return type of :  boolean
```

Returns

True if CATIA support is compiled in; otherwise false.

◆ is_cavity_surface()

def cubit.is_cavity_surface (*surface_id*)

Return whether the specified surface is part of a cavity.

A cavity surface is a surface that bounds a closed void within the model geometry.

If the surface is part of a cavity (such as the inner walls of a subtracted volume), this function returns true; otherwise, false.

```
.. code-block:: python
```

```
                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("brick x 5" )


                cubit.cmd
                ("vol 5 move x 5" )


                cubit.cmd
                ("subtract volume 2 from
                volume 1" )


                surf_id = 17


                is_cavity = cubit.is_cavity_surface
                (surf_id)


                print(f"Surface {surf_id}
                is {'a cavity surface' if is_cavity
        else 'not a cavity
                surface'}" )


                # Expected output
                (example):


                # Surface 17 is a cavity
                surface.
```

```
 @n type of surface_id:  int
```

Parameters

surface_id ID of the surface to query.

return type of : boolean

Returns

True if the surface is part of a cavity, false otherwise.

◆ is_chamfer_surface()

def cubit.is_chamfer_surface (*surface_id*,

thickness_threshold

)

Determine if a face's underlying surface is a chamfer.

Checks whether the surface with ID surface_id represents a chamfer by comparing its maximum thickness to thickness_threshold.

.. code-block:: python

```
# Create a 2x2x2 brick and
chamfer edge 1 with radius 0.5.

cubit.cmd
("brick x 2 y 2 z 2"
)

cubit.cmd
("modify curve 1 chamfer
radius 0.5" )

# Surface 7 should now be a
chamfer.

is_chamfer = cubit.is_chamfer_surface
(7, 1.0)

print("Surface 7 is
chamfer:" , is_chamfer)
```

@n type of surface_id: int

Parameters

surface_id ID of the surface to check.
type of thickness_threshold: float

thickness_threshold Max thickness to qualify as chamfer.
return type of : boolean

Returns

True if the surface thickness <= thickness_threshold, indicating a chamfer.

◆ is_close_loop_surface()

def cubit.is_close_loop_surface (*surface_id*,
mesh_size
)

Check if a surface has multiple loops closer than a given threshold.

Returns true if the face has two or more boundary loops and any two edges from different loops lie within mesh_size of each other. Otherwise returns false.

.. code-block:: python

```
# Create two concentric cylinders
and subtract inner from outer.

cubit.cmd
("cylinder x 3 height
10" )

cubit.cmd
("cylinder x 2 height
10" )

cubit.cmd
("subtract volume 2 from
volume 1" )

# Surface 9 (annulus-like face)
has two loops separated by distance 1.

has_close = cubit.is_close_loop_surface
(9, 1.0)

print("Surface 9 has close
loops:" , has_close) #
True
```

@n type of surface_id: int

Parameters

surface_id ID of the surface to check.
type of mesh_size: float

mesh_size Distance threshold for loop separation.
return type of : boolean

Returns

True if the surface has two or more loops and at least one pair of edges between loops within mesh_size.

◆ is_command_echoed()

def cubit.is_command_echoed ( )

Check whether Cubit echoes commands to the console.

Returns true if each command is printed as it is executed.

```
.. code-block:: python
```

```
import cubit

print(cubit.is_command_echoed
()) # True or False
```

```
@n return type of :  boolean
```

Returns
: True if command echoing is enabled; false otherwise.

◆ is_command_journaled()

def cubit.is_command_journaled ( )

Query whether Cubit is journaling commands.

Returns true if command journaling is currently enabled. Journaling is on by default, but may be turned off explicitly using a command or API call.

```
.. code-block:: python
```

```
import cubit

# Disable journaling

cubit.journal_commands
(False )


# Check journaling state
```

if not **cubit.is_command_journaled()** : print("Journaling is disabled.")

```
# Re-enable journaling

cubit.journal_commands
(True )
```

```
@n return type of :  boolean
```

Returns
: True if command journaling is enabled; false otherwise.

◆ is_cone_surface()

def cubit.is_cone_surface (*surface_id*)

Determine whether a surface is a cone.

Returns whether the specified surface is a conical surface.

```
.. code-block:: python
```

```
# Create a cylinder volume with a
cone on top.


cubit.cmd
('cylinder rad 1 height
5' ) # Create
cylinder


cubit.cmd
('create frustum height 1
radius 1 top 0' ) # Create
cone (frustum with top radius 0)


cubit.cmd
('vol 2 mov z 3' )
# Move cone on top of
cylinder


cubit.cmd
('unite vol all' )
# Unite volumes into one


cubit.cmd
('split period vol
1' ) # Split cylinder and
cone surfaces



result1 = cubit.is_cone_surface
(4) # from frustum


result2 = cubit.is_cone_surface
(1) # from cylinder



print("Surface 4 is
cone:" , result1)


print("Surface 1 is
cone:" , result2)



# Expected output:


# Surface 4 is cone: True


# Surface 1 is cone: False
```

```
@n type of surface_id:  int
```

Parameters

surface_id ID of the surface to test.
return type of : boolean

Returns

True if the surface is a cone, false otherwise.

◆ is_continuous_surface()

def cubit.is_continuous_surface (*surface_id*, *angle_tol*)

Return whether the surface has any adjacent continuous surfaces.

Two surfaces are considered continuous if the exterior angle between them at a shared curve is 180 degrees +/- angle_tol.

.. code-block:: python

```
                    cubit.cmd
                    ("cylinder rad 1 height
                    5" )

                    cubit.cmd
                    ("vol 1 split
                    periodic" ) # Splits
                    cylinder surface into surfaces 4 and 5


                    result = cubit.is_continuous_surface
                    (4, 0.1)


                    print("Surface 4 has
                    adjacent continuous surfaces:" ,
      result)


                    # Expected output:

                    # True
```

@n type of surface_id: int

Parameters

surface_id ID of surface to query.
type of angle_tol: float

angle_tol Angle tolerance for continuity (degrees).
return type of : boolean

Returns

True if the surface has at least one adjacent continuous surface, false otherwise.

◆ is_cylinder_surface()

def cubit.is_cylinder_surface (*surface_id*)

Determine if a given face's underlying surface is a circular cylinder.

Checks whether the surface associated with the specified face ID is a true cylindrical surface.

.. code-block:: python

```
                    # Create a cylinder of radius 2
                    and height 5.

                    cubit.cmd
                    ("cylinder radius 2 height
                    5" )

                    # Surface 1 (the lateral face)
                    should be a cylinder.

                    is_cyl = cubit.is_cylinder_surface
                    (1)

                    print("Surface 1 is
                    cylinder:" , is_cyl) #
                    True
```

@n type of surface_id: int

Parameters

surface_id ID of the surface to check.
return type of : boolean

Returns

True if the surface is a cylindrical surface; otherwise false.

◆ is_geometry_visibility_on()

def cubit.is_geometry_visibility_on ( )

Check if geometry graphics are visible in the graphics window.

Geometry display can be toggled on or off to show or hide CAD entities. This is useful when you want to view only the mesh without underlying geometry.

.. code-block:: python

```
import cubit

# Enable geometry visibility and
verify

cubit.cmd
("geometry vis off"
)

geomVis =
cubit.is_geometry_visibility_on
()

print("Geometry
visible?" , geomVis)
```

@n return type of : boolean

Returns

true if geometry graphics are currently visible; false otherwise.

◆ is_hole_surface()

def cubit.is_hole_surface (*surface_id*,
radius_threshold
)

Return whether the specified surface is part of a hole.

A hole surface is one that bounds a small feature, such as a drilled or machined hole, whose maximum radius does not exceed the given threshold.

If radius_threshold < 0, the default is used: 3 x mesh_size.

```
.. code-block:: python
```

```
                cubit.cmd
                ("brick x 10" )


                cubit.cmd
                ("cylinder rad 1 height
                10" )


                cubit.cmd
                ("vol 2 move z 5" )


                cubit.cmd
                ("subtract volume 2 from
                volume 1" )


                surf_id = 10


                radius_thresh = 2.0


                is_hole = cubit.is_hole_surface
                (surf_id, radius_thresh)


                print(f"Surface {surf_id}
                is {'a hole surface' if is_hole else
      'not a hole
                surface'}" )


                # Expected output
                (example):


                # Surface 10 is a hole
                surface.
```

```
@n type of surface_id:  int
```

Parameters

surface_id ID of the surface to query.
type of radius_threshold: float

radius_threshold Maximum radius criterion for hole detection. If < 0, defaults to 3 x mesh_size.
return type of : boolean

Returns

True if the surface is part of a hole, false otherwise.

◆ is_interval_count_odd()

def cubit.is_interval_count_odd (*surface_id*)

Check if any loop on a surface has an odd number of mesh intervals.

The pave meshing scheme requires each loop on a surface to have an even number of intervals so that quads can be generated. Use this function to detect if a surface contains any loop with an odd interval count, which would prevent pave meshing.

```
.. code-block:: python
```

```
                # Assume surface 5 has been
                meshed with interval counts.

                has_odd = cubit.is_interval_count_odd
                (5)
```

if has_odd: print("Surface 5 contains an odd loop; cannot apply pave mesh.") else:

```
                print("Surface 5 can be
                paved with all quads." )
```

```
@n type of surface_id:  int
```

Parameters

surface_id ID of the surface to check.
return type of : boolean

Returns

True if any loop on the surface has an odd number of intervals; otherwise false.

◆ is_merged()

def cubit.is_merged (*geometry_type*, *entity_id*)

Check if a specified geometry entity has been merged into another.

This function returns true if the given entity-identified by its geometry type ("curve", "surface", or "volume") and integer ID-has been merged into a different entity; otherwise, it returns false.

After performing merge operations (for example, merge vol all), shared or redundant entities may no longer exist as standalone objects. Use **is_merged()** to verify whether a particular entity was absorbed by another during such operations.

```
.. code-block:: python
```

```
                    # 1) Create two unit cubes
                    (Volume IDs = 1 and 2).


                    cubit.cmd
                    ("brick x 10" );
                    # Volume ID = 1


                    cubit.cmd
                    ("brick x 10" );
                    # Volume ID = 2


                    # 2) Move volume 2 ten units in
                    the x-direction so they share a face.


                    cubit.cmd
                    ("vol 2 mov x 10" );


                    # 3) Merge all volumes into a
                    single volume.


                    cubit.cmd
                    ("merge all" );


                    # 4) Check if surface 6 (the
                    original shared face) has been merged.
```

if (**cubit.is_merged** ("surface", 6)) { std::cout << "Surface 6 has been merged." << std::endl; } else { std::cout << "Surface 6 is not merged." << std::endl; }

```
  @n type of geometry_type:  string
```

Parameters

geometry_type The type of geometry to check. Valid values are "curve", "surface", or "volume".
type of entity_id: int

entity_id The integer ID of the entity to query.
return type of : boolean

Returns

true if the specified entity was merged into another; false otherwise.

◆ is_mesh_element_in_group()

def cubit.is_mesh_element_in_group (*element_type*, *element_id*)

Check if a specific mesh element belongs to any group.

.. code-block:: python

```
# Determine if tetrahedron with
ID 445 is in a group.
```

if **cubit.is_mesh_element_in_group** ("tet", 445): print("Tet 445 is in a group") else:

```
print("Tet 445 is not in
any group" )
```

```
@n type of element_type:  string
```

Parameters

element_type Mesh element type (e.g., "node", "edge", "tri", "quad", "tet", "hex").
type of element_id: int

element_id ID of the mesh element to test.
return type of : boolean

Returns

True if the element is contained in at least one group, false otherwise.

◆ is_mesh_visibility_on()

def cubit.is_mesh_visibility_on ( )

Check if mesh graphics are visible in the graphics window.

Mesh display can be toggled on or off to show or hide mesh elements. Hiding mesh can improve performance when displaying the geometry.

```
.. code-block:: python
```

```
import cubit

# Enable mesh visibility and
verify

cubit.cmd
("mesh vis off" )

meshVis = cubit.is_mesh_visibility_on
()

print("Mesh
visible?" , meshVis)
```

```
@n return type of :  boolean
```

Returns

true if mesh graphics are currently visible; false otherwise.

◆ is_meshed()

def cubit.is_meshed (*geometry_type*,
entity_id
)

Determine whether a specified geometry entity has been meshed.

Returns true if the given entity (curve, surface, or volume) already has a valid mesh; otherwise false.

```
.. code-block:: python
```

```
                    # 1) Mesh surface 137.

                    cubit.cmd
                    ("mesh surface 137"
                    )


                    # 2) Check if surface 137 is now
                    meshed.
```

if **cubit.is_meshed** ("surface", 137): print("Surface 137 is meshed.") else:

```
                    print("Surface 137 is not
                    meshed." )
```

```
 @n type of geometry_type:  string
```

Parameters

geometry_type Geometry type ("curve", "surface", or "volume").
type of entity_id: int

entity_id ID of the entity to check.
return type of : boolean

Returns

True if the specified entity is already meshed; false otherwise.

◆ is_modified()

def cubit.is_modified ( )

Check if the model has been modified since import or last save.

Returns true if any CAD operation has altered the current model state (e.g., `remove surface 10` after an import), false if the model is still in its original, unmodified state.

```
.. code-block:: python
```

```
                    import cubit

                    # Import and modify

                    cubit.cmd
                    ('import "part.sat"'
                    )

                    cubit.cmd
                    ("remove surface 10"
                    )

                    modified = cubit.is_modified
                    ()

                    print("Model
                    modified?" , modified)
```

```
 @n return type of :  boolean
```

Returns

True if the model has been changed by any operation; false otherwise.

◆ is_multi_volume()

def cubit.is_multi_volume (*body_id*)

Check if a body contains multiple volumes.

Cubit bodies typically consist of a single volume; multi-volume bodies are rare and usually arise from importing ACIS models. To split a multi-volume body into separate volumes, use: **cubit.cmd** ('split body <id>')

```
.. code-block:: python
```

```
# Assume body 10 may contain
multiple volumes.

multi = cubit.is_multi_volume
(10)
```

if multi: print("Body 10 has multiple volumes. Splitting...")

```
cubit.cmd
("split body 10" )
```

```
@n type of body_id:  int
```

Parameters

body_id ID of the body to query.
return type of : boolean

Returns

True if the body contains more than one volume; otherwise false.

◆ is_narrow_surface()

def cubit.is_narrow_surface (*surface_id*, *mesh_size*)

Determine if any two non-adjacent edges on the surface form a narrow region.

Iterates over all pairs of edges without a shared vertex. For each pair, if their shortest distance <= mesh_size and their local edge directions (relative to surface normals) differ by more than 15deg, the surface is marked narrow.

```
.. code-block:: python
```

```
# Check if surface 5 contains any
narrow region smaller than 0.1 units.

narrow = cubit.is_narrow_surface
(5, 0.1)

print("Surface 5 is
narrow:" , narrow)
```

```
@n type of surface_id:  int
```

Parameters

surface_id ID of the surface to check.
type of mesh_size: float

mesh_size Distance threshold for identifying narrow edge pairs.
return type of : boolean

Returns

True if any qualifying edge pair is found; otherwise false.

◆ is_occlusion_on()

def cubit.is_occlusion_on ( )

Check if occlusion is enabled.

Returns true if occlusion is currently active.

```
.. code-block:: python
```

```
occ = cubit.is_occlusion_on
()
```

```
@n return type of :  boolean
```

Returns

true if occlusion (hidden-line removal) is enabled; false otherwise.

◆ is_on_thin_shell()

def cubit.is_on_thin_shell (*bc_type_enum*, *entity_id*)

Check if a boundary condition is applied to a thin shell.

Valid for temperature, convection, and heatflux BCs.

type of bc_type_enum: int

Parameters

bc_type_enum BC type (temperature = 4, convection = 7, heatflux = 8).
type of entity_id: int

entity_id ID of the boundary condition.
return type of : boolean

Returns

True if BC is on a thin shell element.

◆ is_opencascade_engine_available()

def cubit.is_opencascade_engine_available ( )

Determine whether the OpenCASCADE geometry engine is available.

Notes OpenCASCADE is not included in standard Cubit distributions. Only special builds with OpenCASCADE support will return true.

```
.. code-block:: python
```

```
                has_occ =
    cubit.is_opencascade_engine_available
                ()


                print("OpenCASCADE engine
                available:" , has_occ)
```

```
@n return type of :  boolean
```

Returns

True if OpenCASCADE support is compiled in; otherwise false.

◆ is_part_of_list()

def cubit.is_part_of_list (*target_id*, *id_list*)

Check if an ID is present in a list of IDs.

This function checks whether the specified target ID is present in the given list of IDs. Returns `true` if the target ID is found in the list; otherwise returns `false` .

```
.. code-block:: python
```

```
                # Example ID list


                id_list = [1, 2, 3, 5, 8]


                # Check if ID 3 is part of the
                list


                found = cubit.is_part_of_list
                (3, id_list)


                print(f"Is 3 in the list?
                {'Yes' if found else 'No'}" )


                # Check if ID 4 is part of the
                list


                found = cubit.is_part_of_list
                (4, id_list)


                print(f"Is 4 in the list?
                {'Yes' if found else 'No'}" )
```

```
@n type of target_id:  int
```

Parameters

target_id Target ID to search for.
type of id_list: std::vector< int,std::allocator< int > >

id_list List of IDs.
return type of : boolean

Returns

`true` if target_id is a member of id_list, otherwise `false`.

◆ is_performing_undo()

def cubit.is_performing_undo ( )

Query whether an undo operation is currently being performed.

Use this to detect if Cubit is in the middle of executing an undo command.

.. code-block:: python

```
import cubit

cubit.cmd
("undo" )

print(cubit.is_performing_undo
()) # True if still
undoing
```

@n return type of : boolean

Returns
True if an undo is in progress; false otherwise.

◆ is_periodic()

def cubit.is_periodic (*geometry_type*,
entity_id
)

Query whether a specified surface or curve is periodic.

.. code-block:: python

```
cubit.cmd
("create cylinder radius 3
height 10" )

periodic = cubit.is_periodic
("surface" , 1)
# returns True

print(periodic)
```

Parameters

geom_type Specifies the geometry type ("surface" or "curve").
type of entity_id: int

entity_id ID of the entity to query.
return type of : boolean

Returns
True if the entity is periodic; otherwise false.

◆ is_perspective_on()

def cubit.is_perspective_on ( )

Check if perspective projection is enabled.

Returns true if the graphics window is using perspective projection; false if orthographic.

.. code-block:: python

```
import cubit

# Enable perspective projection
and verify

cubit.cmd
("graphics perspective
on" )

persp_on = cubit.is_perspective_on
()

print("Perspective
on?" , persp_on)
```

@n return type of : boolean

Returns
true if perspective projection is enabled; false otherwise.

◆ is_playback_paused()

def cubit.is_playback_paused ( )

Check if journal playback is currently paused.

Returns true if journal playback is currently paused, either manually via **pause_playback()** or due to an error when playback-paused-on-error is enabled.

.. code-block:: python

```
import cubit

# Pause playback

cubit.pause_playback
()


# Check if playback is
paused
```

if **cubit.is_playback_paused()** : print("Playback is currently paused.")

```
# Resume playback

cubit.resume_playback
()
```

@n return type of : boolean

Returns
True if playback is paused; false otherwise.

◆ is_playback_paused_on_error()

def cubit.is_playback_paused_on_error ( )

Query whether playback is paused on error.

Returns the current setting that controls whether Cubit pauses journal playback when an error occurs.

.. code-block:: python

```
import cubit

# Enable pause on error

cubit.set_playback_paused_on_error
(True )


# Run a journal file

cubit.cmd
("play 'test.jou'" )


# Check pause-on-error
status
```

if **cubit.is_playback_paused_on_error()** : print("Playback paused on error.")

@n return type of : boolean

Returns
True if playback will pause on errors; false otherwise.

◆ is_point_contained()

def cubit.is_point_contained (*geometry_type*,
entity_id,
xyz_point
)

Determine if a point is inside, outside, on, or unknown relative to a given entity.

Commonly used for volumes or sheet bodies to test point containment.

.. code-block:: python

```
                # Create a 10x10x10 brick
                centered at the origin; volume 1
      occupies -5 to
                +5.

                cubit.cmd
                ("brick x 10" )


                # Test a point inside the
                volume.

                pt_inside = (0.0, 0.0, 0.0)

                result_inside =
      cubit.is_point_contained
                ("volume" , 1,
                pt_inside)

                # result_inside == 1
                (inside)

                print("Point
                inside:" , result_inside)


                # Test a point outside the
                volume.

                pt_outside = (10.0, 0.0, 0.0)

                result_outside =
      cubit.is_point_contained
                ("volume" , 1,
                pt_outside)

                # result_outside == 0
                (outside)

                print("Point
                outside:" , result_outside)


                # Test a point on the boundary of
                the volume.

                pt_on = (5.0, 0.0, 0.0)

                result_on = cubit.is_point_contained
                ("volume" , 1,
                pt_on)

                # result_on == 2 (on
                boundary)

                print("Point on
                boundary:" , result_on)
```

@n type of geometry_type: string

Parameters

geometry_type Type of the entity ("volume" or "body").
type of entity_id: int

entity_id ID of the entity to test.
type of xyz_point: std::array< double,3 >

xyz_point A 3-element array {x, y, z} defining the point to test.
return type of : int

Returns

-1 on failure, 0 if outside, 1 if inside, 2 if on the boundary.

◆ is_scale_visibility_on()

def cubit.is_scale_visibility_on ( )

Check if the scale annotation is visible in the graphics window.

The scale annotation displays X, Y, and Z axes (with unit markings) at the bounding box of the current objects. It helps judge physical dimensions.

```
.. code-block:: python
```

```
                import cubit

                # Turn on the scale annotation
                and verify visibility

                cubit.cmd
                ("Graphics Scale on"
                )

                scaleVis = cubit.is_scale_visibility_on
                ()

                print("Scale
                visible?" , scaleVis)
```

```
@n return type of :  boolean
```

Returns
: true if the scale annotation is currently visible; false otherwise.

◆ is_select_partial_on()

def cubit.is_select_partial_on ( )

Check if partial selection is enabled.

return type of : boolean

Returns
: true if partial picks are allowed; false otherwise.

```
.. code-block:: python
```

```
                import cubit

                partial = cubit.is_select_partial_on
                ()
```

◆ is_sheet_body()

def cubit.is_sheet_body (*volume_id*)

Determine if a volume is a sheet body (zero thickness).

.. code-block:: python

```
                # Create a solid brick (volume
                1).

                cubit.cmd
                ("brick x 10" )

                # Copy surface 1 and move it by 1
                unit in Z; this creates a new sheet
      body (ID 2).

                cubit.cmd
                ("surface 1 copy move z
                1" )

                # Check if volume 2 is a sheet
                body.

                is_sheet = cubit.is_sheet_body
                (2) # expected: True

                print(is_sheet)
```

```
@n type of volume_id:  int
```

Parameters
: **volume_id** ID of the volume to query.
return type of : boolean

Returns
: True if the volume is a sheet body (zero thickness); otherwise false.

◆ is_special_build()

def cubit.is_special_build (*build_type*)

Check if a special build type is available.

Determines if the specified build_type corresponds to a special Cubit build or engine support compiled into this Cubit build. The comparison is case-insensitive and "acis", "catia", "goodyear", "granite", "machine_learning" ("ml"), "opencascade", and "sgm".

```
.. code-block:: python
```

```
import cubit

has_ml = cubit.is_special_build
("ml" )

print("ML special build
available:" , has_ml)
```

```
@n type of build_type:  string
```

Parameters

build_type Name of the special build to check (e.g., "acis", "catia", "ml"). Case-insensitive.
return type of : boolean

Returns

True if the specified special build type is available; false otherwise.

◆ is_surface_meshable()

def cubit.is_surface_meshable (*surface_id*)

Check if a surface is meshable under the current meshing scheme.

Returns whether the specified surface can be meshed using the active mesh settings. Surfaces have a default scheme if none was explicitly set.

```
.. code-block:: python
```

```
import cubit

# Set the surface scheme to 'map'
(default applies if not set):

cubit.cmd
("surface 17 scheme
map" )

can_mesh = cubit.is_surface_meshable
(17)

print("Surface 17 meshable
under 'map'?" , can_mesh)
```

```
@n type of surface_id:  int
```

Parameters

surface_id ID of the surface to test.
return type of : boolean

Returns

True if the surface is meshable; false otherwise.

◆ is_surface_planar()

def cubit.is_surface_planar (*surface_id*)

◆ is_surface_planer()

def cubit.is_surface_planer (*surface_id*)

Query whether a specified surface is planar.

Some workflows require distinguishing planar faces from curved ones. Both `is_surface_planer` (archaic spelling) and `is_surface_planar` are provided for compatibility; they behave identically.

```
.. code-block:: python
```

```
                    # Create a brick; surfaces 1-6
                    are planar.

                    cubit.cmd
                    ("brick x 10" )

                    flat = cubit.is_surface_planar
                    (1) # returns True

                    print(flat)
```

```
@n type of surface_id:  int
```

Parameters

surface_id ID of the surface to query.
return type of : boolean

Returns

True if the surface is planar; otherwise false.

◆ is_type_filtered()

def cubit.is_type_filtered (*filter_type*)

Check if a specific entity type is currently excluded from picking.

Returns true if the given type is in the pick filter list (i.e., will not be picked).

```
.. code-block:: python
```

```
                    import cubit

                    filtered = cubit.is_type_filtered
                    ("surface" )

                    print("Surface
                    filtered?" , filtered)
```

```
@n type of filter_type:  string
```

Parameters

filter_type **Entity** type to test ("vertex", "curve", "surface", "volume", "node", "edge", "face", etc.).
return type of : boolean

Returns

True if that type is currently filtered out; false otherwise.

◆ is_undo_save_needed()

def cubit.is_undo_save_needed ( )

Check if the model requires an undo checkpoint save.

Returns true if any CAD operation has modified the model since the last call to **set_undo_saved()** .

```
.. code-block:: python
```

```
                    import cubit

                    cubit.cmd
                    ('import "part.sat"'
                    )

                    print(cubit.is_undo_save_needed
                    ()) # True

                    cubit.set_undo_saved
                    ()

                    print(cubit.is_undo_save_needed
                    ()) # False
```

```
@n return type of :  boolean
```

Returns

True if the model has changed since last undo save; false otherwise.

◆ is_virtual()

def cubit.is_virtual (*geometry_type*, *entity_id*)

Query whether a specified geometry entity is virtual.

.. code-block:: python

```
cubit.cmd
("brick x 10" )
# create brick (Volume ID =
1)


cubit.cmd
("composite create surf 1
6" ) # create composite
surface (Surface ID = 7)


is_virt = cubit.is_virtual
("surface" , 7)


print("Surface 7 is
virtual:" , is_virt)


cubit.cmd
("composite delete surf
7" ) # delete composite
surface


is_virt = cubit.is_virtual
("surface" , 1)


print("Surface 1 is
virtual:" , is_virt)
```

```
@n type of geometry_type:  string
```

Parameters

geometry_type The type of geometry ("curve", "surface", "volume", etc.).
type of entity_id: int

entity_id The ID of the entity to query.
return type of : boolean

Returns

True if the specified entity is virtual; false otherwise.

◆ is_visible()

def cubit.is_visible (*geometry_type*, *entity_id*)

Query visibility for a specific geometry entity.

.. code-block:: python

```
# Turn off visibility for volume
4


cubit.cmd
("vol 4 vis off" )
```

if not **cubit.is_visible** ("volume", 4): print("Volume 4 is now hidden.")

```
# Turn visibility back on for
volume 4


cubit.cmd
("vol 4 vis on" )
```

if **cubit.is_visible** ("volume", 4): print("Volume 4 is now visible.")

```
@n type of geometry_type:  string
```

Parameters

geometry_type The type of geometry ("curve", "surface", "volume", etc.).
type of entity_id: int

entity_id The ID of the entity to query.
return type of : boolean

Returns

True if the specified entity is currently visible; false otherwise.

◆ is_volume_meshable()

def cubit.is_volume_meshable (*volume_id*)

Check if a volume is meshable under the current meshing scheme.

Returns whether the specified volume can be meshed using the active mesh settings. Volumes have a default scheme if none was explicitly set.

```
.. code-block:: python
```

```
import cubit

# Set the volume scheme to
'sweep' (default applies if not set):

cubit.cmd
("volume 42 scheme
sweep" )

can_mesh = cubit.is_volume_meshable
(42)

print("Volume 42 meshable
under 'sweep'?" , can_mesh)
```

```
@n type of volume_id:  int
```

Parameters

volume_id ID of the volume to test.
return type of : boolean

Returns

True if the volume is meshable; false otherwise.

◆ is_working_dir_set()

def cubit.is_working_dir_set ( )

Check if the "-workingdir" option was provided at Cubit startup.

Determines whether the user specified a working directory on the Cubit command line using -workingdir <path>.

```
.. code-block:: python
```

```
import cubit

print("Working dir
set?" , cubit.is_working_dir_set
())
```

```
@n return type of :  boolean
```

Returns

true if -workingdir was used when launching Cubit; false otherwise.

◆ journal_commands()

def cubit.journal_commands (*state*)

Enable or disable journaling of Cubit commands.

Controls whether commands issued via **cubit.cmd** or API calls are recorded in the active journal file. Journaling is enabled by default when Cubit starts.

.. code-block:: python

```
import cubit

# Disable journaling using
command

cubit.cmd
("set journal off" )


# Manually disable via API

cubit.journal_commands
(False )


# Re-enable journaling

cubit.journal_commands
(True )
```

@n type of state: boolean

Parameters

state True to enable journaling; false to disable it.

load_ML()

def cubit.load_ML (*args*,
kwargs

)

Load machine learning training data into memory.

This function loads and caches the specified machine learning models for immediate use in predictions or feature extraction. If a model is required by another ML function and not yet loaded, it will be loaded automatically. Calling this function explicitly ensures the model is loaded immediately.

type of model_type: string, optional

Parameters

model_type Specifies which data to load:

- "all" : Load both classification and regression models.
- "classification" : Load classification models only.
- "regression" : Load regression models only.
  return type of : boolean

Returns

True if the data was loaded successfully; false otherwise.

.. code-block:: python

```
# Example usage in Python

success = cubit.loadML("regression" )
```

if success: print("Regression models loaded.") else:

```
print("Failed to load
models." )
```

measure_between_entities()

def cubit.measure_between_entities (*entity_type1*, *entity_id1*, *entity_type2*, *entity_id2*)

Returns the shortest distance between two geometry entities and their closest points.

Computes the closest points between two entities and returns the distance along with the coordinates of these points.

```
.. code-block:: python
```

```
                    # Geometry setup


                    cubit.cmd
                    ("brick x 10" )


                    cubit.cmd
                    ("vol 1 copy mov x 10.1 y
                    5" )



                    dist_info = cubit.measure_between_entities
                    ("Vertex" , 12,
                    "Volume" , 1)


                    dist = dist_info[0]


                    vertex_point = [dist_info[1], dist_info[2],
                    dist_info[3]]


                    volume_point = [dist_info[4], dist_info[5],
                    dist_info[6]]


                    print("Distance:" ,
                    dist)


                    print("Vertex
                    point:" , tuple(vertex_point))


                    print("Volume
                    point:" , tuple(volume_point))


                    # Expected Output:


                    # Distance: 0.1


                    # Vertex point: (5.1, 0.0,
                    5.0)


                    # Volume point: (5.0, 0.0,
                    5.0)
```

```
 @n type of entity_type1:  string
```

Parameters

entity_type1 Type of the first geometry entity.
type of entity_id1: int

entity_id1 ID of the first geometry entity.
type of entity_type2: string

entity_type2 Type of the second geometry entity.
type of entity_id2: int

entity_id2 ID of the second geometry entity.

```
 @n return type of : std::vector< double,std:: allocator< double > >
```

Returns

: A vector containing distance and coordinates [distance, x1, y1, z1, x2, y2, z2].

◆ ML_train()

def cubit.ML_train (*geom_type*)

Force retraining of the ML classification model for a specific geometry type.

This function initiates a new training run for the specified classification operation. Currently, it is supported only for "volume_no_op" and "surface_no_op" models.

type of geom_type: string

Parameters

geom_type Geometry type to train ("volume" or "surface"), or operation name.
return type of : boolean

Returns

True if the training job was successfully triggered; false otherwise.

```
.. code-block:: python
```

```
success = cubit.ML_train
("surface" )
```

if success: print("Training for surface model started successfully.")
else:

```
print("Failed to trigger
training for surface model." )
```

move()

def cubit.move (*entity*,
vector,
preview = `False`
)

Translates an **Entity** by a specified vector.

Moves the given `entity` by the offset defined in `vector` . Use `preview` to display the translation without applying it.

.. code-block:: python

```
# Create a unit cube (center at
0,0,0)


cubit.cmd
("brick x 1 y 1 z 1"
)


# Move the cube (body 1) by (1,
2, 3)


b = cubit.body
(1)


vec = [1.0, 2.0, 3.0]


cubit.move
(b, vec, False )


# Get and print the center point
of the moved body


pt = b.center_point()


print("Center
Point:" , pt) # prints
(1.0, 2.0, 3.0)
```

@n type of entity: :py:class:`Entity`, in

Parameters

entity The **Entity** to translate.
type of vector: std::array< double,3 >, in

vector The translation vector (x, y, z).
type of preview: boolean, in, optional

preview If true, show a preview without applying (default false).

◆ number_undo_commands()

def cubit.number_undo_commands ( )

Query the number of undoable commands in the stack.

.. code-block:: python

```
# Enable undo and perform three
operations.


cubit.cmd
("set undo on" )


cubit.cmd
("brick x 10" )


cubit.cmd
("cylinder rad 3 height
12" )


cubit.cmd
("subtract vol 2 from vol
1" )



# Check the undo stack size
(should be 3).


count = cubit.number_undo_commands
()


print("Undo stack
size:" , count) # prints
3



# Undo the last operation
(subtract).


cubit.cmd
("undo" )



# Check the stack size again
(should be 2).


count = cubit.number_undo_commands
()


print("Undo stack
size:" , count) # prints
2
```

```
@n return type of :  int
```

Returns

The number of commands currently in the undo stack.

◆ parse_cubit_list()

def cubit.parse_cubit_list (*type*,
entity_list_string
)

Parse a Cubit-style entity selection expression into a list of IDs.

Converts a free-form Cubit entity expression?supporting ranges, name patterns, and topological queries?into a flat list of integer IDs for the given type.

Title Supported syntax examples

- "1 to 12" - IDs 1 through 12
- "5,7,9" - IDs 5, 7, and 9
- with name "my*" - all matching names
- in volume 5 to 23 - surfaces or curves within volumes 5?23

.. code-block:: python

```
                # Setup: create 4 bricks and name
                volumes


                cubit.command("brick x
                10" )


                cubit.command("volume 1
                copy move x 10 repeat 3" )


                cubit.command('vol 1 name
                "widget1"' )


                cubit.command('vol 2 name
                "widget2"' )


                cubit.command('vol 3 name
                "gizmo1"' )


                cubit.command('vol 4 name
                "gizmo2"' )


                # 1) Curves 1?4 and 8


                curve_ids = cubit.parse_cubit_list
                ('curve' ,
                '1 to 4 8' )


                print(tuple(curve_ids)) # (1, 2,
                3, 4, 8)


                # 2) Volumes named
                "widget*"


                widget_ids = cubit.parse_cubit_list
                ('volume' ,
                'with name
                "widget*"' )


                print(tuple(widget_ids)) # (1,
                2)


                # 3) Curves in volumes named
                "gizmo*"


                gizmo_curve_ids =
      cubit.parse_cubit_list
                ('curve' ,
                'in volume with name
                "gizmo*"' )


                print(tuple(gizmo_curve_ids)) #
                (25, 26, . . ., 48)
```

@n type of type: string

Parameters

type **Entity** type string (e.g., "vertex", "curve", "surface", "volume").
type of entity_list_string: string

entity_list_string Free-form Cubit selection expression.
return type of : std::vector< int,std::allocator< int > >

Returns

Vector of validated entity IDs matching the expression.

◆ parse_locations()

def cubit.parse_locations (*location_str*)

Parse a Cubit location specification into concrete 3D coordinates.

Accepts any Cubit-style location expression and returns the resulting point(s) as x-y-z triples. See documentation for supported location options.

.. code-block:: python

```
# Create a brick

cubit.command("brick x
10" )

# Single point at absolute
coordinates

pts1 = cubit.parse_locations
("1 2 3" )

print(pts1) # Output: ((1.0, 2.0,
3.0),)

# Coordinates of vertex 5

pts2 = cubit.parse_locations
("at vertex 5" )

print(pts2) # Output: ((5.0, 5.0,
-5.0),)

# Multiple positions

pts3 = cubit.parse_locations
("position 1 2 3 position 4
5 6" )

print(pts3) # Output: ((1.0, 2.0,
3.0), (4.0, 5.0, 6.0))

# Vertices in volume 1

pts4 = cubit.parse_locations
("at vertex in vol
1" )

print(pts4) # Output: ((5.0,
-5.0, 5.0), (5.0, 5.0, 5.0), (-5.0,
5.0, 5.0), (-5.0,
-5.0, 5.0), ...)

# Mesh the volume and get all
nodes on curve 1

cubit.command("vol 1 size
2" )

cubit.command("mesh vol
1" )

pts5 = cubit.parse_locations
("at node in curve
1" )

print(pts5) # Output: ((5.0,
-5.0, 5.0), (5.0, 5.0, 5.0), (5.0, -
3.0, 5.0), (5.0,
-1.0, 5.0), (5.0, 1.0, 5.0), (5.0, 3.0,
5.0))
```

```
@n type of location_str:  string
```

Parameters

location_str Cubit location expression string.
return type of : std::vector< std::array< double,3 >,std::allocator< std:: array< double,3 > > >

Returns

Vector of 3-element arrays (x, y, z) for each parsed location.

◆ pause_playback()

def cubit.pause_playback ( )

Pause journal playback immediately.

Halts any ongoing playback of journal commands. Typically used within a journal file to pause execution for debugging or inspection. Playback can be resumed by calling **resume_playback()** .

.. code-block:: python

```
import cubit

# Perform initial setup

# (Assume these steps are part of
a journal execution context)


# Pause playback for
inspection

cubit.pause_playback
()


# Resume playback after
inspection

cubit.resume_playback
()
```

◆ print_cmd_options()

def cubit.print_cmd_options ( )

Display Cubit's supported startup options.

Prints every command-line flag that the Cubit executable accepts when launched from a terminal, along with its syntax and description.

Example output (first few of many):

.. code-block:: c++

```
-Include <$val>        (Default path to search for input files)
-[no]banner            (Suppress printing of startup information.)
-[no]batch             (Batch Mode--No Interactive Command Input)
-debug <$val>          (Set specified flags on, e.g. 1,3,7-9 enables 1,3,7,8,9))
etc.                   (and many more options)
```

.. code-block:: python

```
import cubit

cubit.print_cmd_options
()
```

◆ print_current_selections()

def cubit.print_current_selections ( )

Print all selected entities and their types.

Outputs each selection in order (index, type, and ID) to the console.

.. code-block:: python

```
import cubit

cubit.print_current_selections
()
```

◆ print_currently_selected_entity()

def cubit.print_currently_selected_entity ( )

Print details of the current selected entity.

Shows the type and ID of the entity at the internal selection pointer.

```
.. code-block:: python
```

```
                import cubit

                cubit.print_currently_selected_entity
                ()
```

◆ print_info()

def cubit.print_info (*message*)

Print a message through Cubit's messaging system.

Send an informational message to the Cubit message handler for display or logging.

```
.. code-block:: python
```

```
                import cubit

                cubit.print_info
                ("Processing
                complete." )
```

```
@n type of message:  string
```

Parameters

message The text to be printed.

◆ print_raw_help()

def cubit.print_raw_help (*input_line*,
order_dependent,
consecutive_dependent
)

Display context-sensitive help while typing commands.

Called when the user presses one of the help keys ('?', '&', or '!') at the prompt. It prints relevant syntax or usage hints based on the current input line.

```
.. code-block:: python
```

```
                import cubit

                # User typed "mesh surface " then
                pressed '&'

                line = "mesh surface
                "

                order_dependent = False# key was
                '&'

                consecutive_dependent = False# key was
                not '?'

                cubit.print_raw_help
                (line, order_dependent,
      consecutive_dependent)
```

```
@n type of input_line:  string
```

Parameters

input_line	Current command-line text being edited. type of order_dependent: int
order_dependent	Set to 1 (true) if the key was '?' or '!'; set to 0 (false) if it was '&'. type of consecutive_dependent: int
consecutive_dependent	Set to 1 (true) if the key was '?' (allows repeated help); otherwise 0 (false).

◆ print_surface_summary_stats()

def cubit.print_surface_summary_stats ( )

Print summary statistics for all surfaces in the model.

Title Reported metrics

- Number of surfaces measured and their average area
- Minimum and maximum surface area, with corresponding surface IDs
- Minimum hydraulic radius (area/perimeter) and the surface ID
- Minimum curve length (smallest edge) with edge ID and surface ID
- Maximum ratio of adjacent curve lengths on a surface and its surface ID
- Minimum distance between loops on multi-loop surfaces (if applicable) and surface ID
- Minimum distance between edges within a single loop and surface ID
- Minimum and maximum interior angles between adjacent curves on a surface, with surface IDs

.. code-block:: python

```
# Create a 10x10x10 brick;
surfaces 1-6 are generated.


cubit.cmd
("brick x 10" )


# Print summary statistics for
all surfaces.


cubit.print_surface_summary_stats
()
```

◆ print_volume_summary_stats()

def cubit.print_volume_summary_stats ( )

Print summary statistics for all volumes in the model.

Title Reported metrics

- Total number of volumes measured and average volume
- Minimum volume value and its volume ID
- Minimum hydraulic radius (6 * volume/total_surface_area) and its volume ID
- Minimum and maximum interior angles between any two faces in a volume, including the two face IDs and volume ID
- Maximum area ratio between adjacent faces in a volume, including the larger and smaller face IDs and volume ID
- Total counts of merged and unmerged surfaces across all volumes and their ratio
- Number of shell groups found among the combined faces of all volumes and the name of the RefGroup containing those shells
- Summary statistics for all surfaces in these volumes (calls **print_surface_summary_stats()**)

.. code-block:: python

```
# Create two bricks and subtract
the smaller from the larger.


cubit.cmd
("brick x 10" )
# volume 1


cubit.cmd
("brick x 5" )
# volume 2


cubit.cmd
("subtract vol 2 from vol
1" )


# Print summary statistics for
all volumes.


cubit.print_volume_summary_stats
()
```

◆ prism()

def cubit.prism (*height*,
sides,
major,
minor
)

Create a prism of the specified dimensions.

Creates an extruded prism from a regular polygon base defined by major and minor radii. The prism height and number of sides specify its shape and extrusion.

.. code-block:: python

```
import cubit

# 1) Create a regular hexagonal
prism: height=3, sides=6, major
radius=2, minor
radius=2.

my_prism = cubit.prism
(3, 6, 2, 2)

print("Prism ID:" ,
my_prism.id())


# 2) Create an elongated
hexagonal prism: height=3, sides=6,
major radius=4,
minor radius=2.

my_prism = cubit.prism
(3, 6, 4, 2)

print("Prism ID:" ,
my_prism.id())
```

@n type of height: float, in

Parameters

height Height of the prism.
type of sides: int, in

sides Number of sides of the base polygon.
type of major: float, in

major Major radius of the base polygon.
type of minor: float, in

minor Minor radius of the base polygon.
return type of : :py:class: **Body**

Returns

Body object representing the newly created prism geometry.

◆ project_unit_square()

def cubit.project_unit_square (*pts*,
surface_id,
quad_id,
node00_id,
node10_id
)

Map points in a unit square (u-v coordinates on a quad face) and project them onto a surface.

For each (u, v) in pts (with values in [0, 1], representing local coordinates on the quad face), this function uses node00_id (u=0, v=0) and node10_id (u=1, v=0) to locate the corresponding point on the quad (quad_id), then projects that 3D point onto the specified surface (surface_id). Returns the 3D coordinates of each projected point.

.. code-block:: python

```
# 1) Create a sphere of radius 1
(surface_id = 1, volume_id = 1).


cubit.cmd
("sphere rad 1" )


# 2) Set mesh size on volume 1
and mesh it.


cubit.cmd
("vol 1 size 1" )


cubit.cmd
("mesh vol 1" )


# 3) Define points in the unit
square (u-v coordinates on quad face).


pts = [(0.25, 0.25), (0.5, 0.5), (0.75, 0.75)]


# 4) Project onto surface 1 using
quad_id = 3, node00_id = 10, node10_id = 15.


positions = cubit.project_unit_square
(pts, 1, 3, 10, 15)
```

for p in positions: print(p) # Each p is a list [x, y, z]

@n type of pts: std::vector< std::vector< double,std::allocator< double > >,std::allocator< std::vector< double,std::allocator< double > > > >

Parameters

pts A list of (u, v) pairs (each a length-2 vector) in [0, 1], representing local u-v coordinates on the quad face.
type of surface_id: int

surface_id ID of the surface onto which points are projected.
type of quad_id: int

quad_id ID of the quad face used to map unit-square coordinates.
type of node00_id: int

node00_id ID of the quad node corresponding to (u=0, v=0).
type of node10_id: int

node10_id ID of the quad node corresponding to (u=1, v=0).
return type of : std::vector< std::vector< double,std::allocator< double > >,std::allocator< std::vector< double,std:: allocator< double > > > >

Returns

A list of 3-element vectors [x, y, z] representing each projected point.

◆ pyramid()

```
def cubit.pyramid (  height,
                     sides,
                     major,
                     minor,
                     top = 0.0
                  )
```

Create a pyramid of specified dimensions.

Creates an extruded pyramid from a regular polygon base defined by major and minor radii. The apex or top plateau size can be specified.

.. code-block:: python

```
                import cubit


                # 1) Create a pointed pyramid:
                height=4, sides=5, major radius=3,
    minor
                radius=2.


                pyramid_point = cubit.pyramid
                (4, 5, 3, 2)


                print("Pointed pyramid
                ID:" , pyramid_point.id())



                # 2) Create a truncated pyramid
                (frustum): height=4, sides=5, major=3,
    minor=2, top
                radius=1.


                pyramid_frustum = cubit.pyramid
                (4, 5, 3, 2, 1)


                print("Frustum pyramid
                ID:" , pyramid_frustum.id())
```

@n type of height: float, in

Parameters

height Height of the pyramid.
type of sides: int, in

sides Number of sides of the base polygon.
type of major: float, in

major Major radius of the base polygon.
type of minor: float, in

minor Minor radius of the base polygon.
type of top: float, in, optional

top Top radius for a truncated pyramid (default = 0, pointed apex).
return type of : :py:class: **Body**

Returns

Body object representing the newly created pyramid geometry.

reflect()

def cubit.reflect (*entity*,
axis,
preview = `False`
)

Reflects an **Entity** about a specified axis (e.g., plane normal).

Reflects the given `entity` across the plane perpendicular to `axis` . The original entity can be previewed without modification using `preview` .

.. code-block:: python

```
# Create a unit cube (Body ID
1)


cubit.cmd
("brick x 1 y 1 z 1"
)


# Move the cube along +X by 10
units


b = cubit.body
(1)


cubit.move
(b, [10.0, 0.0, 0.0], False )


# Reflect across the Y-Z plane
(normal = X-axis)


axis = [1.0, 0.0, 0.0]


cubit.reflect
(b, axis, False )


# Get and print the new center
point


pt = b.center_point()


print("Center
Point:" , pt) # prints
(-10.0, 0.0, 0.0)
```

@n type of entity: :py:class:`Entity`, in

Parameters

entity The **Entity** to be reflected.
type of axis: std::array< double,3 >, in

axis The axis (normal) defining the reflection plane.
type of preview: boolean, in, optional

preview If true, show a preview without applying (default false).

◆ release_interface()

def cubit.release_interface (*instance*)

Release a previously retrieved Cubit interface.

Decrements the reference count (or performs cleanup) for the given interface instance. After calling this, the pointer should not be used.

.. code-block:: python

```
import cubit


iface = cubit.get_interface
("MeshInterface" )
```

if iface is not None:

use iface ...

success = cubit.release_interface(iface) if not success: print("Failed to release interface")

@n type of instance: CubitBaseInterface

Parameters

instance Pointer to the CubitBaseInterface to release.
return type of : boolean

Returns

True if the interface was successfully released; false otherwise.

◆ remove_entity_from_group()

def cubit.remove_entity_from_group (*group_id*, *entity_id*, *entity_type*)

Remove a specific entity from a given group.

.. code-block:: python

```
# Initialize geometry and group
all vertices in volume 1.


cubit.cmd
("brick x 10" )


cubit.cmd
("group \"vert_group\" add
vertex in vol 1" )


vert_group_id = cubit.get_id_from_name
("vert_group" )


# Remove vertex with ID 8 from
the group.


cubit.remove_entity_from_group
(vert_group_id, 8, "vertex" )


# Query remaining vertices in the
group.


verts = cubit.get_group_vertices
(vert_group_id)


print("Remaining
vertices:" , len(verts)) #
prints "7"
```

```
@n type of group_id:  int
```

Parameters

group_id ID of the group from which the entity will be removed.
type of entity_id: int

entity_id ID of the entity to remove.
type of entity_type: string

entity_type Type of the entity (e.g., "vertex", "curve", "surface", "volume"). Only geometric entities are supported.
return type of : void

Returns

None.

◆ remove_filter_type()

def cubit.remove_filter_type (*filter_type*)

Remove an entity type from the graphics pick filter.

Disallows picking of the specified entity type; other filters remain unchanged.

```
.. code-block:: python
```

```
import cubit


# Remove "curve" from the pick
filter


cubit.remove_filter_type
("curve" )
```

```
@n type of filter_type:  string
```

Parameters

filter_type **Entity** type to remove from the pickable set (e.g., "vertex", "curve", "surface", "volume", "node", "edge", "face").

◆ replace_progress_handler()

def cubit.replace_progress_handler (*progress*)

Replace the current progress-bar callback handler and return the old one.

Unregisters the existing handler and replaces it with the given one. Returns the previous handler without deleting it.

.. code-block:: python

```
import cubit

new_handler = MyProgressHandler()

old_handler =
cubit.replace_progress_handler
(new_handler)

# old_handler can be reused or
released by Python garbage collection
```

@n type of progress: :py:class:`CubitProgressHandler`

Parameters

progress Shared pointer to the new CubitProgressHandler instance.
return type of : :py:class: `CubitProgressHandler`

Returns

Shared pointer to the previous handler (may be null if none was set).

◆ reset()

def cubit.reset ( )

Reset Cubit state.

Clears all geometry and mesh data currently loaded in the session, returning Cubit to its initial state.

.. code-block:: python

```
import cubit

# Create geometry and mesh

cubit.cmd
("brick x 10" )

cubit.cmd
("mesh volume 1" )

# Reset Cubit session

cubit.reset
()

print("Cubit session has
been reset." )
```

◆ reset_camera()

def cubit.reset_camera ( )

Reset the camera view and clear auxiliary graphics windows.

Restores the default view, closes dialogs, and clears overlays.

.. code-block:: python

```
import cubit

cubit.cmd
("from 10 5 15" )

cubit.cmd
("at 0 0 0" )

cubit.cmd
("up 0 1 0" )

cubit.reset_camera
()

cubit.cmd
("journal view" )
# display updated
from/at/up
```

◆ resume_playback()

def cubit.resume_playback ( )

Resume a paused journal playback.

Continues journal playback after it has been paused, either manually via **pause_playback()** or automatically due to an error when playback-paused-on-error is enabled.

.. code-block:: python

```
import cubit

# Assume playback was paused
earlier (manually or on error)

cubit.resume_playback
()
```

◆ scale()

def cubit.scale (*entity*,
factor,
preview = `False`
)

Scales an **Entity** uniformly by a specified factor.

Scales the given `entity` by `factor` in all directions. Use `preview` to display the scaled geometry without applying it.

.. code-block:: python

```
# Create a 5x5x5 cube (Body ID
1)

cubit.cmd
("brick x 5" )

# Scale the cube by a factor of
2

b = cubit.body
(1)

cubit.scale
(b, 2.0, False )

# Get and print the new
volume

vol = b.volume()

print("Volume:" ,
vol) # prints 1000
```

@n type of entity: :py:class:`Entity`, in

Parameters

entity	The **Entity** to scale. type of factor: float, in
factor	The uniform scale factor. type of preview: boolean, in, optional
preview	If true, show a preview without applying (default false).

◆ set_copy_block_on_geometry_copy_setting()

def cubit.set_copy_block_on_geometry_copy_setting (*val*)

Set the behavior for block propagation during geometry copy.

Valid settings:

- "ON": Create new blocks on copied geometry/mesh, mirroring the original.
- "USE_ORIGINAL": Add copied geometry/mesh to the original block.
- "OFF": Do not copy block to the new geometry/mesh.

.. code-block:: python

```
                success =
      cubit.set_copy_block_on_geometry_copy_setting
                ("OFF" )


                print("Set to OFF,
                success:" , success)



                setting =
      cubit.get_copy_block_on_geometry_copy_setting
                ()


                print("Current
                setting:" , setting)



                success =
      cubit.set_copy_block_on_geometry_copy_setting
                ("ON" )


                print("Set to ON,
                success:" , success)



                setting =
      cubit.get_copy_block_on_geometry_copy_setting
                ()


                print("Current
                setting:" , setting)
```

@n type of val: string

Parameters

val Desired setting ("ON", "USE_ORIGINAL", or "OFF").
return type of : boolean

Returns

True if setting was successfully applied, false otherwise.

◆ set_copy_nodeset_on_geometry_copy_setting()

def cubit.set_copy_nodeset_on_geometry_copy_setting (*val*)

Set the behavior for nodeset propagation during geometry copy.

Valid settings:

- "ON": Create new nodesets on copied geometry/mesh, mirroring the original.
- "USE_ORIGINAL": Add copied geometry/mesh to the original nodeset.
- "OFF": Do not copy nodeset to the new geometry/mesh.

.. code-block:: python

```
                success =
      cubit.set_copy_nodeset_on_geometry_copy_setting
                ("OFF" )


                print("Set to OFF,
                success:" , success)



                setting =
      cubit.get_copy_nodeset_on_geometry_copy_setting
                ()


                print("Current
                setting:" , setting)



                success =
      cubit.set_copy_nodeset_on_geometry_copy_setting
                ("ON" )


                print("Set to ON,
                success:" , success)



                setting =
      cubit.get_copy_nodeset_on_geometry_copy_setting
                ()


                print("Current
                setting:" , setting)
```

@n type of val: string

Parameters

val Desired setting ("ON", "USE_ORIGINAL", or "OFF").
return type of : boolean

Returns

True if setting was successfully applied, false otherwise.

◆ set_copy_sideset_on_geometry_copy_setting()

def cubit.set_copy_sideset_on_geometry_copy_setting (*val*)

Set the behavior for sideset propagation during geometry copy.

Valid settings:

- "ON": Create new sidesets on copied geometry/mesh, mirroring the original.
- "USE_ORIGINAL": Add copied geometry/mesh to the original sideset.
- "OFF": Do not copy sideset to the new geometry/mesh.

.. code-block:: python

```
                success =
      cubit.set_copy_sideset_on_geometry_copy_setting
                ("OFF" )


                print("Set to OFF,
                success:" , success)



                setting =
      cubit.get_copy_sideset_on_geometry_copy_setting
                ()


                print("Current
                setting:" , setting)



                success =
      cubit.set_copy_sideset_on_geometry_copy_setting
                ("ON" )


                print("Set to ON,
                success:" , success)



                setting =
      cubit.get_copy_sideset_on_geometry_copy_setting
                ()


                print("Current
                setting:" , setting)
```

```
@n type of val:  string
```

Parameters

val Desired setting ("ON", "USE_ORIGINAL", or "OFF").
return type of : boolean

Returns

True if setting was successfully applied, false otherwise.

◆ set_cubit_interrupt()

def cubit.set_cubit_interrupt (*interrupt*)

Enable or disable interruptible operations in Cubit.

When set to true, any interruptible Cubit process will be stopped at the next interrupt check.

.. code-block:: python

```
                import cubit

                # Disable interrupts

                cubit.set_cubit_interrupt
                (False )

                # Enable interrupts

                cubit.set_cubit_interrupt
                (True )
```

```
@n type of interrupt:  boolean
```

Parameters

interrupt True to stop interruptible processes, false to allow them.

◆ set_cubit_message_handler()

def cubit.set_cubit_message_handler (*hdlr*)

Redirect Cubit output to a custom message handler.

Replace the default Cubit message handler so that all subsequent messages are routed to the provided handler instance, which must implement print_message() and/or print_error().

.. code-block:: python

```
import cubit
```

class MyHandler(cubit.CubitMessageHandler): def print_message(self, message):

```
print("[Info]" ,
message)
```

def print_error(self, message): print("[Error]", message, file=sys.stderr)

```
handler = MyHandler()

cubit.set_cubit_message_handler
(handler)
```

@n type of hdlr: :py:class:`CubitMessageHandler`

Parameters

hdlr Shared pointer to a CubitMessageHandler that will receive messages.

◆ set_element_variable()

def cubit.set_element_variable (*element_ids*, *variable_name*, *variables*)

Sets scalar variables on specified mesh elements for Exodus export.

Assigns the element variable named `variable_name` to the elements in `element_ids` , using the corresponding values in `variables` . If `variables` contains a single entry, that value is applied to all elements; otherwise, its length must match `element_ids` . When exporting the mesh to an Exodus file, these element variables will be included for downstream analysis.

.. code-block:: python

```
# Create a rectangular surface
and mesh to generate elements

cubit.cmd
("create surface rectangle
width 10" )

cubit.cmd
("mesh surface 1" )

# Define block 1 using surface 1
to specify global element IDs

cubit.cmd
("block 1 surface 1"
)

# Assign 'compression' variable
to elements in block 1

element_ids = [1, 2, 3, 4] # IDs
derived from block 1

var_name = "compression"

values = [1.2, 1.53, 2.3, 9.5]

cubit.set_element_variable
(element_ids, var_name, values)

# Export mesh to Exodus to
include element variables

cubit.cmd
("export mesh
\"output.e\"" )
```

@n type of element_ids: std::vector< int,std::allocator< int > >, in

Parameters

element_ids List of element IDs to assign variables to.
type of variable_name: string, in

variable_name Name of the element variable.
type of variables: std::vector< double,std::allocator< double > >, in

variables Values for each element (single value applies to all elements).

◆ set_entity_name()

def cubit.set_entity_name (*entity_type*,
entity_id,
new_name
)

Set the name of a specified entity.

Equivalent to the command:

.. code-block:: c++

```
{Body|Volume|Surface|Curve|Vertex} <id_range> name '<name_string>'
```

For example, to name vertex 22 "point_load":

.. code-block:: c++

```
Vertex 22 name 'point_load'
```

.. code-block:: python

```
success = cubit.set_entity_name
("vertex" , 22,
"point_load" )
```

if success: print("Rename succeeded")

```
@n type of entity_type:  string
```

Parameters

entity_type Type of the entity ("body", "volume", "surface", "curve", or "vertex").
type of entity_id: int

entity_id ID of the entity to rename.
type of new_name: string

new_name The new name to assign to the entity.
return type of : boolean

Returns

True if the entity was found and renamed; otherwise false.

◆ set_exit_handler()

def cubit.set_exit_handler (*hdlr*)

Set a custom exit callback for Cubit.

Provide an ExternalExitHandler-derived instance to handle Cubit's exit events.

```
.. code-block:: python
```

```
import cubit
```

class MyExitHandler(cubit.ExternalExitHandler): def handle_exit_event(self, code):

```
print(f"Cubit exiting with
code {code}, cleaning up..." )


handler = MyExitHandler()


cubit.set_exit_handler
(handler)
```

```
@n type of hdlr:  :py:class:`ExternalExitHandler`
```

Parameters

hdlr Pointer to an ExternalExitHandler implementation that will be called when Cubit exits.

◆ set_filter_types()

def cubit.set_filter_types (*num_types*, *filter_types*)

Set multiple pick filter types for graphics selections.

Configures the graphics system to allow picking any entity whose type is in the provided list. Overrides any previously set pick filters.

.. code-block:: python

```
import cubit

# Enable picking of surfaces and
curves

cubit.set_filter_types
(2, ["surface" ,
"curve" ])
```

@n type of num_types: int

Parameters

num_types Number of entity types in filter_types.
type of filter_types: std::vector< std::string,std::allocator< std::string > >

filter_types Vector of entity-type strings to enable for picking. Valid values include "vertex", "curve", "surface", "volume", "node", "edge", "face", etc.

◆ set_label_type()

def cubit.set_label_type (*entity_type*, *label_flag*)

Sets label display type for a given entity type.

Controls how labels are displayed for the specified entity type in the graphics window.

Valid `entity_type` values:

```
"RefVolume" ,
"RefFace" ,
"RefEdge" ,
"RefVertex" ,

"CubitHex" ,
"CubitTet" ,
"CubitFace" ,
"CubitTri" ,

"CubitEdge" ,
"CubitNode" ,
"CubitSphere"
```

Label flag values (`SVUtil::LabelType`):
0=CUBIT_LABEL_NONE, 1=CUBIT_LABEL_ID,
2=CUBIT_LABEL_ELEMENT_ID, 3=CUBIT_LABEL_NAME,
4=CUBIT_LABEL_INTERVAL, 5=CUBIT_LABEL_SIZE,
6=CUBIT_LABEL_MERGE, 7=CUBIT_LABEL_IS_MERGED,
8=CUBIT_LABEL_FIRMNESS, 9=CUBIT_LABEL_SCHEME,
10=CUBIT_LABEL_NAME_ID,
11=CUBIT_LABEL_NAME_ONLY,
12=CUBIT_LABEL_SPHERE_ID

.. code-block:: python

```
cubit.set_label_type
("RefFace" , 1)
# Show face IDs
```

@n type of entity_type: string

Parameters

entity_type **Entity** type to label.
type of label_flag: int

label_flag Label display type.

@n return type of : void

Returns
None.

◆ set_max_group_id()

def cubit.set_max_group_id (*maximum_group_id*)

Reset Cubit's internal maximum group ID to a specified value.

Cubit tracks a monotonically increasing next group ID. GUI power tools may create and delete groups behind the scenes, incrementing Cubit's group ID counter and causing downstream journal files to reference unexpected IDs. This function restores control by resetting Cubit's max group ID, but only when the specified `maximum_group_id` matches Cubit's current highest ID? otherwise no change occurs.

Warning
Use sparingly. An incorrect `maximum_group_id` can desynchronize journal files or lead to ID collisions.

type of maximum_group_id: int

Parameters

maximum_group_id The desired maximum group ID to enforce (must equal Cubit's current max ID).
return type of : void

Returns
None.

◆ set_ML_base_user_dir()

def cubit.set_ML_base_user_dir (*path*,
print_info = `False`
)

Set the root directory for user-provided ML training data (classification only).

Specifies the location of user training files for classification operations. The directory must contain subfolders matching the ML operation names (e.g., "ml/volume_no_op").

type of path: string

Parameters

path Top-level path to the user training data directory.
type of print_info: boolean, optional

print_info If true, output status messages about directory setup.

.. code-block:: python

```
data_dir =
"/path/to/user/training/data"

cubit.set_ML_base_user_dir
(data_dir, True )
```

◆ set_modified()

def cubit.set_modified ( )

Reset the model's modified status to "unmodified".

Clears the internal flag so that **is_modified()** returns false until the next CAD operation.

.. code-block:: python

```
import cubit

cubit.cmd
('import "part.sat"'
)

cubit.set_modified
()

# Now:

print(cubit.is_modified
()) # False

cubit.cmd
("remove surface 10"
)

print(cubit.is_modified
()) # True
```

◆ set_nodal_variable()

def cubit.set_nodal_variable (*node_ids*, *variable_name*, *variables*)

Sets scalar variables on specified mesh nodes.

Assigns the nodal variable named `variable_name` to the nodes in `node_ids` , using the corresponding values in `variables` . If `variables` contains a single value, that value is applied to all nodes; otherwise, its length must match `node_ids` . When exporting to Exodus, these nodal values will be included for analysis.

```
.. code-block:: python
```

```
# Create a rectangular sheet and
mesh it to generate nodes

cubit.cmd
("create surface rectangle
width 10" )

cubit.cmd
("mesh surface 1" )

# Assign compression variable to
nodes 1, 2, 3, and 4

node_ids = [1, 2, 3, 4]

var_name = "compression"

values = [1.2, 1.53, 2.3, 9.5]

cubit.set_nodal_variable
(node_ids, var_name, values)
```

```
@n type of node_ids:  std::vector< int,std::allocator< int > >, in
```

Parameters

node_ids List of node IDs to assign variables to.
type of variable_name: string, in

variable_name Name of the nodal variable.
type of variables: std::vector< double,std::allocator< double > >, in

variables Values for each node (single value applies to all nodes).

◆ set_overlap_max_angle()

def cubit.set_overlap_max_angle (*maximum_angle*)

Set the maximum angle tolerance for calculating surface overlaps.

Updates the threshold that defines how much angular difference between surface normals can exist before surfaces are considered non-overlapping. Smaller values enforce stricter overlap detection by requiring surfaces to be nearly coplanar.

```
.. code-block:: python
```

```
# Set the overlap max angle to 5
degrees.

cubit.set_overlap_max_angle
(5.0)

# Verify the new setting.

new_max_angle =
cubit.get_overlap_max_angle
()

print("New overlap max
angle:" , new_max_angle)
```

```
@n type of maximum_angle:  float
```

Parameters

maximum_angle The new maximum angle tolerance (in degrees) to use when testing surface overlaps.

set_overlap_max_gap()

def cubit.set_overlap_max_gap (*maximum_gap*)

Set the maximum gap tolerance for calculating surface overlaps.

Updates the threshold that defines how large a gap between two surfaces can be before they are considered non-overlapping. Smaller values make the overlap test more stringent, while larger values allow greater discrepancies.

```
.. code-block:: python
```

```
                # Set the overlap max gap to 0.01
                model units.

                cubit.set_overlap_max_gap
                (0.01)

                # Verify the new setting.

                new_max_gap = cubit.get_overlap_max_gap
                ()

                print("New overlap max
                gap:" , new_max_gap)
```

```
@n type of maximum_gap:  float
```

Parameters

maximum_gap The new maximum gap tolerance (in model units) to use when testing surface overlaps.

set_overlap_min_gap()

def cubit.set_overlap_min_gap (*min_gap*)

Set the minimum gap tolerance for calculating surface overlaps.

Updates the threshold that defines how small a gap between two surfaces can be before they are considered overlapping. Smaller values enforce stricter overlap detection by ignoring only very tiny gaps.

```
.. code-block:: python
```

```
                # Set the overlap min gap to
                0.0001 model units.

                cubit.set_overlap_min_gap
                (0.0001)

                # Verify the new setting.

                new_min_gap = cubit.get_overlap_min_gap
                ()

                print("New overlap min
                gap:" , new_min_gap)
```

```
@n type of min_gap:  float
```

Parameters

min_gap The new minimum gap tolerance (in model units) to use when testing surface overlaps.

set_pick_type()

def cubit.set_pick_type (*pick_type*,
silent = `False`
)

Set the current pick mode for entity selection.

Specifies which entity type the graphics system will select on the next pick. This determines the output of **get_selected_ids()** and pick dialogs.

```
.. code-block:: python
```

```
import cubit

# Change pick mode to surface
(prints confirmation)

cubit.set_pick_type
("surface" )

# Change pick mode to volume
silently (no confirmation)

cubit.set_pick_type
("volume" ,
True )
```

```
@n type of pick_type:  string
```

Parameters

pick_type **Entity** type to pick. Valid values include: "vertex", "curve", "surface", "volume", "node", "edge", "face", etc.
type of silent: boolean, optional

silent If true, suppress the console message confirming the change. Defaults to false (confirmation printed).

◆ set_playback_paused_on_error()

def cubit.set_playback_paused_on_error (*pause*)

Configure whether playback pauses on error.

Controls whether Cubit pauses journal playback automatically when an error occurs, allowing inspection before continuing.

```
.. code-block:: python
```

```
# Python example

import cubit

# Enable pause on error during
journal playback

cubit.set_playback_paused_on_error
(True )


# Run a journal file; playback
will pause if an error occurs

cubit.cmd
("play 'test.jou'" )


# Disable pause on error

cubit.set_playback_paused_on_error
(False )
```

```
@n type of pause:  boolean
```

Parameters

pause True to pause playback on error; false to continue automatically.

◆ set_progress_handler()

def cubit.set_progress_handler (*progress*)

Register a progress-bar callback handler with Cubit.

Sets the given handler as the active progress-bar callback. If a handler is already registered, it is released (but not returned).

`.. code-block:: python`

```
import cubit
```

class MyProgressHandler(cubit.CubitProgressHandler): def start(self, title, info_string, hasCancel): print {info_string}") def end(self): print("Progress complete.") def percent(self, pcnt): print {pcnt * 100:.1f}%", end="\r") def check_interrupt(self): return False

```
handler = MyProgressHandler()

cubit.set_progress_handler
(handler)
```

type of progress: :py:class: `CubitProgressHandler`

Parameters

progress Shared pointer to a CubitProgressHandler instance.

◆ set_rendering_mode()

def cubit.set_rendering_mode (*mode*)

Set the current graphics rendering mode (equivalent to "Graphics Mode <option>").

This is the programmatic equivalent of the Cubit command:

- **`cubit.cmd`**`("Graphics Mode <option>")`

where `mode` can be one of `HiddenLine` , `TrueHiddenLine` , `SmoothShade` , `Transparent` , `WireFrame` , or `GeomFacet` . Instead of a string, this function accepts an integer code:

- 0 = `WireFrame`
- 1 = `HiddenLine`
- 2 = `TrueHiddenLine`
- 3 = `SmoothShade`
- 4 = `Transparent`
- 5 = `GeomFacet`

`Python example (for reference):`

`.. code-block:: python`

```
import cubit

# Equivalent to:
cubit.cmd("Graphics Mode SmoothShade")

cubit.set_rendering_mode
(3)
```

type of mode: int

Parameters

mode Integer code for the rendering mode (0-5), corresponding to: 0=WireFrame, 1=HiddenLine, 2=TrueHiddenLine, 3=SmoothShade, 4=Transparent, 5=GeomFacet.

◆ set_undo_saved()

def cubit.set_undo_saved ( )

Clear the undo-needed flag for the model.

Marks the current model state as saved for undo purposes. After calling, **is_undo_save_needed()** returns false until the next modifying operation.

`.. code-block:: python`

```
import cubit

cubit.set_undo_saved
()
```

◆ silent_cmd()

def cubit.silent_cmd (*input_string*)

Execute a Cubit command without echoing or verbose output.

Behaves like **cmd()** , but suppresses prompt echo and messages–ideal for scripting.

```
.. code-block:: python
```

```
import cubit

ok = cubit.silent_cmd
("mesh volume 1" )
```

```
@n type of input_string:  string
```

Parameters

input_string C-string containing the complete Cubit command.
return type of : boolean

Returns

true if the command executed successfully; false otherwise.

◆ snap_locations_to_geometry()

def cubit.snap_locations_to_geometry (*locations*, *entity_type*, *entity_id*, *tol*)

Snaps given XYZ locations to nearest points on specified entity.

Points are first snapped to the closest location on the specified entity. Within the provided tolerance, points are then snapped preferentially to vertices, followed by curves.

```
.. code-block:: python
```

```
# Geometry setup

cubit.cmd
("sphere rad 1" )

cubit.cmd
("split periodic vol
1" ) # creates a curve at
the equator


locations = [[1, 0.001, 0], [1, 1, 1], [-1, 0.1, 0]]

snapped_xyz_vec = cubit.snap_locations_to_geometry
(locations, "body" ,
1, 0.01)

print(snapped_xyz_vec)

# Expected Output:

# ((1.0, 0.0, 0.0), (0.577350,
0.577350, 0.577350), (-0.995037, 0.099504, 0.0))
```

```
@n type of locations:  std::vector< std::array< double,3 >,std::allocator< std::array< double,3 > > >
```

Parameters

locations A vector of XYZ points to snap.
type of entity_type: string

entity_type Type of entity (e.g., "volume", "surface", "curve").
type of entity_id: int

entity_id ID of the target entity.
type of tol: float

tol Tolerance for snapping to vertices and curves.

```
@n return type of : std::vector< std::array< double,3 >,std::allocator< std:: array< double,3 > > >
```

Returns

A vector of snapped XYZ points corresponding to input locations.

◆ sphere()

```
def cubit.sphere ( radius,
                   x_cut = 0,
                   y_cut = 0,
                   z_cut = 0,
                   inner_radius = 0
                 )
```

Create all or part of a sphere.

Creates a spherical geometry of given radius. Optional planar cuts (along the yz, xz, or xy planes) and an inner radius for hollow spheres can be specified.

.. code-block:: python

```
import cubit

# 1) Create a full sphere of
radius 1.

full_sphere = cubit.sphere
(1.0)

print("Full sphere Body
ID:" , full_sphere.id())

print("Sphere
volume:" , full_sphere.volume())

# Expected output:

# Full sphere Body ID: 1

# Sphere volume: 4.18879


# 2) Create a half-sphere (cut by
yz-plane).

half_sphere = cubit.sphere
(1.0, 1)

print("Half-sphere Body
ID:" , half_sphere.id())

print("Half-sphere
volume:" , half_sphere.volume())

# Expected output:

# Half-sphere Body ID: 2

# Half-sphere volume:
2.0940168474871794


# 3) Create a hollow sphere:
outer radius 5, inner radius 3.

hollow_sphere = cubit.sphere
(5.0, 0, 0, 0, 3.0)

print("Hollow sphere Body
ID:" , hollow_sphere.id())

print("Hollow sphere
volume:" , hollow_sphere.volume())

# Expected output:

# Hollow sphere Body ID: 3

# Hollow sphere volume:
410.5014400690663
```

```
@n type of radius:  float, in
```

Parameters

radius	Radius of the sphere. type of x_cut: int, in, optional
x_cut	If 1, cut sphere by the yz-plane (default = 0). type of y_cut: int, in, optional
y_cut	If 1, cut sphere by the xz-plane (default = 0). type of z_cut: int, in, optional
z_cut	If 1, cut sphere by the xy-plane (default = 0). type of inner_radius: float, in, optional
inner_radius	Inner radius for a hollow sphere (default = 0). return type of : :py:class: **Body**

Returns

Body object representing the newly created sphere geometry.

◆ step_next_possible_selection()

def cubit.step_next_possible_selection ( )

Advance to the next entity in the current selection list.

Moves the internal selection pointer forward, updating which entity is considered the "current" selection for **print_currently_selected_entity()** .

.. code-block:: python

```
import cubit

cubit.print_currently_selected_entity
() # first pick

cubit.step_next_possible_selection
()

cubit.print_currently_selected_entity
() # next pick
```

◆ step_previous_possible_selection()

def cubit.step_previous_possible_selection ( )

Move back to the previous entity in the current selection list.

Moves the internal selection pointer backward, so that the "current" selection reverts to the one before the last step.

.. code-block:: python

```
import cubit

cubit.step_previous_possible_selection
()

cubit.print_currently_selected_entity
()
```

◆ stop_playback()

def cubit.stop_playback ( )

Stop journal playback entirely.

Immediately terminates the execution of a journal file. This is typically used within a journal to abort playback due to errors or conditional logic. To restart, use **resume_playback()** or manually reissue commands.

.. code-block:: python

```
import cubit

# Check some condition, then stop
playback if needed

cubit.stop_playback
()


# Playback will not continue past
this point
```

◆ string_from_id_list()

def cubit.string_from_id_list (*ids*)

Convert a list of integers into a compact Cubit-style ID string.

Collapses consecutive IDs into "start to end" ranges, separates entries with commas, and inserts line breaks at 80 characters for readability.

Title Examples:

- {1,2,3,4,5,6,7,8} -> "1 to 8\n"
- {1,2,3,100,5,6,7,8} -> "1 to 3, 5 to 8, 100\n"

```
.. code-block:: python
```

```
                import cubit

                ids = [1, 2, 3, 4]

                s = cubit.string_from_id_list
                (ids)

                # s == "1 to 4\n"
```

```
 @n type of ids:  std::vector< int,std::allocator< int > >
```

Parameters

ids Vector of integer IDs to format.
return type of : string

Returns

String containing ranges and line breaks at column 80.

◆ subtract()

def cubit.subtract (*tool_in*,
from_in,
imprint_in = `False`,
keep_old_in = `False`
)

Performs a boolean subtract operation: removes tool bodies from target bodies.

Subtracts each body in `tool_in` from the corresponding body in `from_in` . The subtraction can imprint shared geometry if `imprint_in` is true, and the original bodies can be preserved if `keep_old_in` is true.

```
   .. code-block:: python
```

```
                # Create a box (Body ID 1) and a
                cylinder (Body ID 2)

                cubit.cmd
                ("brick x 10" )
                # Volume 1000

                cubit.cmd
                ("cylinder radius 3 height
                10" ) # Volume
                ~282.74

                # Define tool and target
                bodies

                tool = cubit.body
                (2)

                target = cubit.body
                (1)

                # Subtract cylinder from
                box

                result = cubit.subtract
                ([tool], [target], False ,
                False )

                b = result[0]

                print("Resulting Body
                ID:" , b.id())

                print("Volume:" ,
                b.volume()) # prints
                ~717.26
```

```
  @n type of tool_in:  std::vector< CubitInterface::Body,std::allocator< CubitInterface::Body > >, in
```

Parameters

tool_in Bodies to subtract (tools).
type of from_in: std::vector< CubitInterface::Body,std::allocator< CubitInterface::Body > >, in

from_in Bodies to subtract from (targets).
type of imprint_in: boolean, in, optional

imprint_in True to imprint shared geometry (default false).
type of keep_old_in: boolean, in, optional

keep_old_in True to retain the original bodies (default false).
return type of : std::vector< CubitInterface::Body,std::allocator< CubitInterface:: Body > >

Returns

A vector of **Body** objects representing the resulting bodies after subtraction.

◆ surface()

def cubit.surface (*id_in*)

Retrieve a surface object by its ID.

Retrieves the surface object corresponding to the provided ID.

```
.. code-block:: python
```

```
import cubit

# Create a 10x10x10 brick
(surface IDs include the outer faces).

cubit.cmd
("brick x 10" )

# Retrieve surface with ID
1.

surface = cubit.surface
(1)

print("Surface ID:"
, surface.id())
```

```
@n type of id_in:  int
```

Parameters

id_in ID of the surface to retrieve.
return type of : :py:class: **Surface**

Returns

Surface object corresponding to the specified ID.

◆ sweep_curve()

```
def cubit.sweep_curve (  curves,
                         along_curves,
                         draft_angle,
                         draft_type,
                         rigid
                      )
```

Sweep one or more curves along a path to create sheet bodies.

Constructs sheet bodies by sweeping the specified cross-section curves along the given path curves. Optional draft angle, draft type, and rigidity control taper and rounding behavior. This creates surface bodies (no solid volume); each returned **Body** contains exactly one **Surface** . Use the body's `surfaces()` method to access it and obtain area.

.. code-block:: python

```
import cubit

# Create section curve.

v0 = cubit.create_vertex
(0, 0, 0)

v1 = cubit.create_vertex
(1, 0, 0)

section = cubit.create_curve
(v0, v1)


# Create path curve.

p0 = cubit.create_vertex
(0, 0, 0)

p1 = cubit.create_vertex
(0, 0, 1)

path = cubit.create_curve
(p0, p1)


# Sweep with default
parameters.

bodies = cubit.sweep_curve
([section], [path])

# We expect one sheet
body.

sheet = bodies[0]

print("Sheet Body
ID:" , sheet.id())

# Retrieve its surface and print
area.

surface = sheet.surfaces()[0]

print("Surface
area:" , surface.area())

# Expected output:

# Sheet Body ID: 1

# Surface area: 1.0
```

@n type of curves: std::vector< CubitInterface::Curve,std::allocator< CubitInterface::Curve > >, in

Parameters

curves	Cross-section curves to sweep. type of along_curves: std::vector< CubitInterface::Curve,std::allocator< CubitInterface::Curve > >, in
along_curves	Path curves defining the sweep trajectory. type of draft_angle: float, in
draft_angle	Angle (in degrees) for draft taper (default = 0). type of draft_type: int, in
draft_type	Draft type: 0=extended, 1=rounded, 2=natural (default = 0). type of rigid: boolean, in
rigid	If true, maintains cross-section rigidity (default = false). return type of : std::vector< CubitInterface::Body,std::allocator< CubitInterface:: Body > >

Returns

Vector of sheet **Body** objects created by the sweep (each contains one **Surface**). :raises: **CubitFailureException** Thrown if input is invalid or sweep fails.

◆ temperature_is_on_shell_area()

def cubit.temperature_is_on_shell_area (*bc_type_enum*, *bc_area_enum*, *entity_id*)

Check if a temperature BC is on a shell area.

type of bc_type_enum: int

Parameters

bc_type_enum BC type (temperature = 4, convection = 7).
type of bc_area_enum: int

bc_area_enum CI_BCEntityTypes: 7 (top), 8 (bottom), 9 (gradient), 10 (middle).
type of entity_id: int

entity_id ID of the BC.
return type of : boolean

Returns

True if the temperature BC is on the specified shell area.

◆ temperature_is_on_solid()

def cubit.temperature_is_on_solid (*bc_type_enum*, *entity_id*)

Check if a temperature BC is applied to a solid region.

Valid for temperature and convection BCs.

type of bc_type_enum: int

Parameters

bc_type_enum BC type (temperature = 4, convection = 7).
type of entity_id: int

entity_id ID of the boundary condition.
return type of : boolean

Returns

True if temperature BC is on a solid.

◆ torus()

def cubit.torus (*center_radius*,
swept_radius
)

Create a torus of specified dimensions.

Creates a toroidal geometry defined by the distance from center to the center of the swept circle (major radius) and the radius of the swept circle (minor radius).

.. code-block:: python

```
import cubit

# 1) Create a torus:
center_radius=5, swept_radius=1.

torus1 = cubit.torus
(5, 1)

print("Torus ID:" ,
torus1.id())

print("Torus
volume:" , torus1.volume())

# Expected output:

# Torus ID: 1

# Torus volume: 98.69604


# 2) Create a thick torus:
center_radius=3, swept_radius=2.

torus2 = cubit.torus
(3, 2)

print("Torus ID:" ,
torus2.id())

print("Torus
volume:" , torus2.volume())

# Expected output:

# Torus ID: 2

# Torus volume: 236.87051
```

@n type of center_radius: float, in

Parameters

center_radius Distance from torus center to center of swept circle (major radius, must be > swept_radius).
type of swept_radius: float, in

swept_radius Radius of the circle to be swept (minor radius, must be < center_radius).
return type of : :py:class: **Body**

Returns

Body object representing the newly created torus geometry.

◆ tweak_curve_offset()

def cubit.tweak_curve_offset (*curves*,
distances,
keep_old = `False`,
preview = `False`
)

2D equivalent of tweak_surface_offset: offsets specified curves on a sheet body.

Offsets each curve in `curves` by the corresponding value in `distances` on a 2D sheet body. The original sheet can be retained via `keep_old` , and a preview generated via `preview` .

.. code-block:: python

```
# Create a rectangular sheet with
width 10 (height defaults to 10)


cubit.cmd
("create surface rectangle
width 10" )


# Offset curve 1 by 10
units


curves = [ cubit.curve
(1) ]


distances = [ 10.0 ]


bodies = cubit.tweak_curve_offset
(curves, distances, False
, False )


# Only one sheet body is
returned


b = bodies[0]


print("Body ID:" ,
b.id())


# Get the first surface from the
sheet and print its area


surf = b.surfaces()[0]


area = surf.area()


print("Surface
Area:" , area) # prints
200
```

@n type of curves: std::vector< CubitInterface::Curve,std::allocator< CubitInterface::Curve > >, in

Parameters

curves List of **Curve** objects to offset.
type of distances: std::vector< double,std::allocator< double > >, in

distances List of distances for each curve (in model units).
type of keep_old: boolean, in, optional

keep_old True to retain the original sheet body (default false).
type of preview: boolean, in, optional

preview True to generate a preview without modifying geometry (default false).
return type of : std::vector< CubitInterface::Body,std::allocator< CubitInterface:: Body > >

Returns

A vector of sheet **Body** objects representing the resulting bodies after offset.

◆ tweak_curve_remove()

def cubit.tweak_curve_remove (*curves*,
keep_old = `False`,
preview = `False`
)

Removes specified curves and extends adjacent surfaces on a sheet body.

2D equivalent of tweak_surface_remove. Removes each curve (edge) in `curves` from a 2D sheet body. Adjacent surfaces are extended to fill the gap created by removal. The original sheet can be retained via `keep_old` , and a preview can be generated without modification when `preview` is true.

.. code-block:: python

```
# Create a rectangular sheet with
width 10

cubit.cmd
("create surface rectangle
width 10" )

# Fillet vertex 1 with radius 1
(creates curve ID 5)

cubit.cmd
("tweak vertex 1 fillet
radius 1" )

# Remove the fillet curve (ID
5)

curves = [ cubit.curve
(5) ]

bodies = cubit.tweak_curve_remove
(curves, False ,
False )

# Only one sheet body is
returned

b = bodies[0]

print("Body ID:" ,
b.id())

# Get the first surface from the
sheet and print its area

surf = b.surfaces()[0]

area = surf.area()

print("Surface
Area:" , area) # prints
100
```

@n type of curves: std::vector< CubitInterface::Curve,std::allocator< CubitInterface::Curve > >, in

Parameters

curves List of **Curve** objects to remove.
type of keep_old: boolean, in, optional

keep_old True to retain the original sheet body (default false).
type of preview: boolean, in, optional

preview True to generate a preview without modifying geometry (default false).
return type of : std::vector< CubitInterface::Body,std::allocator< CubitInterface:: Body > >

Returns

A vector of sheet **Body** objects representing the resulting bodies after curve removal.

◆ tweak_surface_offset()

def cubit.tweak_surface_offset (*surfaces*, *distances*)

Offsets specified surfaces by given distances.

Offsets each surface in `surfaces` by the corresponding value in `distances` , modifying the geometry and returning the resulting bodies (one per set of inputs).

.. code-block:: python

```
# Create a 1x1x1 brick; surfaces
1-6 are generated.


cubit.cmd
("brick x 1 y 1 z 1"
)


# Offset surface 1 by 1
unit


surfaces = [ cubit.surface
(1) ]


distances = [ 1.0 ]


bodies = cubit.tweak_surface_offset
(surfaces, distances)


# Only one body is
returned


b = bodies[0] # returns 1


print("Body ID:" ,
b.id())


vol = b.volume() # returns
2


print("Volume:" ,
vol)
```

@n type of surfaces: std::vector< CubitInterface::Surface,std::allocator< CubitInterface::Surface > >

Parameters

surfaces List of **Surface** objects to offset.
type of distances: std::vector< double,std::allocator< double > >

distances List of distances for each surface (positive values offset outward).
return type of : std::vector< CubitInterface::Body,std::allocator< CubitInterface:: Body > >

Returns

A vector of **Body** objects corresponding to the modified geometry (one **Body** for each offset operation).

◆ tweak_surface_remove()

def cubit.tweak_surface_remove (*surfaces*,
extend_ajoining = `True`,
keep_old = `False`,
preview = `False`
)

Removes specified surfaces from a body, optionally extending adjacent surfaces.

Removes each surface in `surfaces` from its body. If `extend_ajoining` is true, adjacent surfaces are extended to close the gap. The original body can be retained via `keep_old` , and a preview can be generated without modification when `preview` is true.

.. code-block:: python

```
# Create a brick with x-length
10

cubit.cmd
("brick x 10" )

# Add a blend on curve 12 with
radius 1

cubit.cmd
("modify curve 12 blend
radius 1" )

# Remove the blend surface (ID
7)

surfaces = [ cubit.surface
(7) ]

bodies = cubit.tweak_surface_remove
(surfaces, True ,
False , False )

# Only one body is
returned

b = bodies[0]

print("Body ID:" ,
b.id())

# Get the volume of the resulting
brick

vol = b.volume() # returns
1000

print("Volume:" ,
vol) # prints 1000
```

@n type of surfaces: std::vector< CubitInterface::Surface,std::allocator< CubitInterface::Surface > >, in

Parameters

surfaces	List of **Surface** objects to remove. type of extend_ajoining: boolean, in, optional
extend_ajoining	True to extend adjacent surfaces (default true). type of keep_old: boolean, in, optional
keep_old	True to retain the original body (default false). type of preview: boolean, in, optional
preview	True to generate a preview without modifying geometry (default false). return type of : std::vector< CubitInterface::Body,std::allocator< CubitInterface:: Body > >

Returns

A vector of **Body** objects representing the resulting bodies after removal.

◆ tweak_vertex_fillet()

```
def cubit.tweak_vertex_fillet (  verts,
                                 radius,
                                 keep_old = False,
                                 preview = False
                              )
```

2D sheet vertex chamfer: creates chamfers at specified vertices by offsetting adjacent curves.

Performs a chamfer operation at each vertex in `verts` on a 2D sheet body. Adjacent curves are trimmed and extended to form a flat face at distance `radius` from the vertex. The original sheet can be retained via `keep_old` , and a preview generated via `preview` .

```
.. code-block:: python
```

```
# Create a rectangular sheet with
width 10

cubit.cmd
("create surface rectangle
width 10" )

# Chamfer vertex 1 with distance
1 (creates new edges)

verts = [ cubit.vertex
(1) ]

bodies = cubit.tweak_vertex_fillet
(verts, 1.0, False ,
False )

# Only one sheet body is
returned

b = bodies[0]

print("Body ID:" ,
b.id())

# Get the first surface from the
sheet and print its area

surf = b.surfaces()[0]

area = surf.area()

print("Surface
Area:" , area) # prints
99.5
```

```
@n type of verts:  std::vector< CubitInterface::Vertex,std::allocator< CubitInterface::Vertex > >, in
```

Parameters

- **verts** List of **Vertex** objects to chamfer.
  type of radius: float, in
- **radius** Chamfer distance (model units).
  type of keep_old: boolean, in, optional
- **keep_old** True to retain the original sheet body (default false).
  type of preview: boolean, in, optional
- **preview** True to generate a preview without modifying geometry (default false).
  return type of : std::vector< CubitInterface::Body,std::allocator< CubitInterface:: Body > >

Returns

A vector of sheet **Body** objects representing the resulting bodies after chamfering.

◆ unite()

def cubit.unite (*body_in*,
keep_old_in = `False`
)

Performs a boolean unite operation: merges specified bodies into one.

Unites the bodies in `body_in` into a single body. Surfaces in contact remain connected. The original bodies can be retained via `keep_old_in` .

.. code-block:: python

```
# Create a 10x10x10 brick (Body
ID 1) and a cylinder (Body ID 2)


cubit.cmd
("brick x 10" )
# Volume 1000


cubit.cmd
("cylinder radius 3 height
10" ) # Volume
~282.74


# Move the cylinder (Body 2)
along +Z by 10


cubit.cmd
("move volume 2 z
10" )



# Unite bodies 1 and 2 into
one


inputs = [ cubit.body
(1), cubit.body
(2) ]


result = cubit.unite
(inputs, False )


b = result[0]


print("Body ID:" ,
b.id())


print("Volume:" ,
b.volume()) # prints
~1282.74
```

@n type of body_in: std::vector< CubitInterface::Body,std::allocator< CubitInterface::Body > >, in

Parameters

body_in List of **Body** objects to unite.
type of keep_old_in: boolean, in, optional

keep_old_in True to retain original bodies (default false).
return type of : std::vector< CubitInterface::Body,std::allocator< CubitInterface:: Body > >

Returns

A vector of **Body** objects representing the new united bodies (one **Body**).

◆ unload_ML()

def cubit.unload_ML (*args*,
kwargs
)

Unload machine learning training data from memory.

This function clears cached ML models and releases associated resources for the specified type.

type of model_type: string, optional

Parameters

model_type Specifies which data to unload:

- "all" : Unload both classification and regression models.
- "classification" : Unload classification models only.
- "regression" : Unload regression models only.

.. code-block:: python

```
# Example usage in Python

cubit.unloadML("regression" )

print("Regression models
unloaded." )
```

◆ unselect_entity()

def cubit.unselect_entity (*entity_type*, *entity_id*)

Unselect an entity that is currently selected.

Removes the highlight and pick status of the specified entity.

```
.. code-block:: python
```

```
import cubit

# First select curve 221

cubit.cmd
("select curve 221"
)

# Now unselect the same
curve

cubit.unselect_entity
("curve" , 221)
```

```
@n type of entity_type:  string
```

Parameters

entity_type Type of the entity ("vertex", "curve", "surface", "volume").
type of entity_id: int

entity_id ID of the entity to unselect.

◆ vertex()

def cubit.vertex (*id_in*)

Retrieve a vertex object by its ID.

Retrieves the vertex object corresponding to the provided ID.

```
.. code-block:: python
```

```
import cubit

# Create a 10x10x10 brick (vertex
IDs include corners).

cubit.cmd
("brick x 10" )

vertex = cubit.vertex
(1)

print("Vertex ID:" ,
vertex.id())
```

```
@n type of id_in:  int
```

Parameters

id_in ID of the vertex to retrieve.
return type of : :py:class: `Vertex`

Returns

Vertex object corresponding to the specified ID.

◆ volume()

def cubit.volume (*id_in*)

Retrieve a volume by its ID.

Retrieves the volume object corresponding to the provided ID.

```
.. code-block:: python
```

```
import cubit

# Create a 10x10x10 brick (volume
ID 1).

cubit.cmd
("brick x 10" )

volume = cubit.volume
(1)

print("Volume ID:" ,
volume.id())
```

```
@n type of id_in:  int
```

Parameters

id_in ID of the volume to retrieve.
return type of : :py:class: `Volume`

Returns

Volume object corresponding to the specified ID.

◆ volume_contains_tets()

def cubit.volume_contains_tets (*volume_id*)

Determine whether a specified volume contains any tetrahedral elements.

.. code-block:: python

```
cubit.cmd
("brick x 10" )
# create brick (Volume ID =
1)

cubit.cmd
("vol 1 scheme
tetmesh" )

cubit.cmd
("mesh vol 1" )
# mesh volume 1 with tets

has_tets = cubit.volume_contains_tets
(1)

print("Volume 1 contains
tets:" , has_tets)
```

```
@n type of volume_id:  int
```

Parameters

volume_id ID of the volume to check.
return type of : boolean

Returns

True if the volume contains any tetrahedral elements; false otherwise.

◆ was_last_cmd_undoable()

def cubit.was_last_cmd_undoable ( )

Report whether the last executed command was undoable.

Returns true if the most recent Cubit command supports undo; false otherwise.

```
.. code-block:: python
```

```
import cubit

cubit.cmd
("create brick x 5"
)

undoable = cubit.was_last_cmd_undoable
()

print("Last command
undoable?" , undoable)
```

```
@n return type of :  boolean
```

Returns

true if the last command supports undo; false otherwise.

◆ write_to_journal()

def cubit.write_to_journal (*words*)

Append a custom entry to Cubit's journal and recording streams.

Forces the given text to be recorded as if it were a Cubit command. This marks the model as modified and, if journaling and recording are enabled, writes the entry to the active journal and recording files.

```
.. code-block:: python
```

```
                import cubit

                # Write an import command
                directly into the journal

                cubit.write_to_journal
                ('import "part.sat"'
                )
```

```
 @n type of words:  string
```

Parameters

words Text to append to the journal and recording streams.

Variable Documentation

◆ BAR

BAR = _cubit3.BAR

◆ BAR2

BAR2 = _cubit3.BAR2

◆ BAR3

BAR3 = _cubit3.BAR3

◆ BEAM

BEAM = _cubit3.BEAM

◆ BEAM2

BEAM2 = _cubit3.BEAM2

◆ BEAM3

BEAM3 = _cubit3.BEAM3

◆ CI_BCENTITY_CC

CI_BCENTITY_CC = _cubit3.CI_BCENTITY_CC

◆ CI_BCENTITY_CC_SHELL_BOTTOM

CI_BCENTITY_CC_SHELL_BOTTOM = _cubit3.CI_BCENTITY_CC_SHELL_BOTTOM

◆ CI_BCENTITY_CC_SHELL_TOP

CI_BCENTITY_CC_SHELL_TOP = _cubit3.CI_BCENTITY_CC_SHELL_TOP

◆ CI_BCENTITY_FIRST

CI_BCENTITY_FIRST = _cubit3.CI_BCENTITY_FIRST

CI_BCENTITY_LAST

CI_BCENTITY_LAST = _cubit3.CI_BCENTITY_LAST

CI_BCENTITY_SHELL_AREA_BOTTOM

CI_BCENTITY_SHELL_AREA_BOTTOM = _cubit3.CI_BCENTITY_SHELL_AREA_BOTTOM

CI_BCENTITY_SHELL_AREA_GRADIENT

CI_BCENTITY_SHELL_AREA_GRADIENT = _cubit3.CI_BCENTITY_SHELL_AREA_GRADIENT

CI_BCENTITY_SHELL_AREA_MIDDLE

CI_BCENTITY_SHELL_AREA_MIDDLE = _cubit3.CI_BCENTITY_SHELL_AREA_MIDDLE

CI_BCENTITY_SHELL_AREA_TOP

CI_BCENTITY_SHELL_AREA_TOP = _cubit3.CI_BCENTITY_SHELL_AREA_TOP

CI_BCENTITY_SOLID_HEATFLUX

CI_BCENTITY_SOLID_HEATFLUX = _cubit3.CI_BCENTITY_SOLID_HEATFLUX

CI_BCENTITY_SOLID_TEMP

CI_BCENTITY_SOLID_TEMP = _cubit3.CI_BCENTITY_SOLID_TEMP

CI_BCENTITY_TEMP

CI_BCENTITY_TEMP = _cubit3.CI_BCENTITY_TEMP

CI_BCENTITY_UNKNOWN

CI_BCENTITY_UNKNOWN = _cubit3.CI_BCENTITY_UNKNOWN

CI_BCTYPE_ACCELERATION

CI_BCTYPE_ACCELERATION = _cubit3.CI_BCTYPE_ACCELERATION

CI_BCTYPE_AXIS

CI_BCTYPE_AXIS = _cubit3.CI_BCTYPE_AXIS

CI_BCTYPE_CONSTRAINT

CI_BCTYPE_CONSTRAINT = _cubit3.CI_BCTYPE_CONSTRAINT

CI_BCTYPE_CONTACT_PAIR

CI_BCTYPE_CONTACT_PAIR = _cubit3.CI_BCTYPE_CONTACT_PAIR

CI_BCTYPE_CONTACT_REGION

CI_BCTYPE_CONTACT_REGION = _cubit3.CI_BCTYPE_CONTACT_REGION

◆ CI_BCTYPE_CONVECTION

CI_BCTYPE_CONVECTION = _cubit3.CI_BCTYPE_CONVECTION

◆ CI_BCTYPE_DISPLACEMENT

CI_BCTYPE_DISPLACEMENT = _cubit3.CI_BCTYPE_DISPLACEMENT

◆ CI_BCTYPE_EXHAUSTFAN

CI_BCTYPE_EXHAUSTFAN = _cubit3.CI_BCTYPE_EXHAUSTFAN

◆ CI_BCTYPE_FAN

CI_BCTYPE_FAN = _cubit3.CI_BCTYPE_FAN

◆ CI_BCTYPE_FIRST

CI_BCTYPE_FIRST = _cubit3.CI_BCTYPE_FIRST

◆ CI_BCTYPE_FORCE

CI_BCTYPE_FORCE = _cubit3.CI_BCTYPE_FORCE

◆ CI_BCTYPE_GAP

CI_BCTYPE_GAP = _cubit3.CI_BCTYPE_GAP

◆ CI_BCTYPE_HEATFLUX

CI_BCTYPE_HEATFLUX = _cubit3.CI_BCTYPE_HEATFLUX

◆ CI_BCTYPE_INLETVENT

CI_BCTYPE_INLETVENT = _cubit3.CI_BCTYPE_INLETVENT

◆ CI_BCTYPE_INTAKEFAN

CI_BCTYPE_INTAKEFAN = _cubit3.CI_BCTYPE_INTAKEFAN

◆ CI_BCTYPE_INTERFACE

CI_BCTYPE_INTERFACE = _cubit3.CI_BCTYPE_INTERFACE

◆ CI_BCTYPE_INTERIOR

CI_BCTYPE_INTERIOR = _cubit3.CI_BCTYPE_INTERIOR

◆ CI_BCTYPE_LAST

CI_BCTYPE_LAST = _cubit3.CI_BCTYPE_LAST

◆ CI_BCTYPE_LUMPMASS

CI_BCTYPE_LUMPMASS = _cubit3.CI_BCTYPE_LUMPMASS

◆ CI_BCTYPE_MASSFLOWINLET

CI_BCTYPE_MASSFLOWINLET = _cubit3.CI_BCTYPE_MASSFLOWINLET

◆ CI_BCTYPE_MATERIAL

CI_BCTYPE_MATERIAL = _cubit3.CI_BCTYPE_MATERIAL

◆ CI_BCTYPE_MEDIA

CI_BCTYPE_MEDIA = _cubit3.CI_BCTYPE_MEDIA

◆ CI_BCTYPE_MPC

CI_BCTYPE_MPC = _cubit3.CI_BCTYPE_MPC

◆ CI_BCTYPE_OUTFLOW

CI_BCTYPE_OUTFLOW = _cubit3.CI_BCTYPE_OUTFLOW

◆ CI_BCTYPE_OUTLETVENT

CI_BCTYPE_OUTLETVENT = _cubit3.CI_BCTYPE_OUTLETVENT

◆ CI_BCTYPE_PERIODIC

CI_BCTYPE_PERIODIC = _cubit3.CI_BCTYPE_PERIODIC

◆ CI_BCTYPE_PERIODICSHADOW

CI_BCTYPE_PERIODICSHADOW = _cubit3.CI_BCTYPE_PERIODICSHADOW

◆ CI_BCTYPE_POROUSJUMP

CI_BCTYPE_POROUSJUMP = _cubit3.CI_BCTYPE_POROUSJUMP

◆ CI_BCTYPE_PRESSURE

CI_BCTYPE_PRESSURE = _cubit3.CI_BCTYPE_PRESSURE

◆ CI_BCTYPE_PRESSUREFARFIELD

CI_BCTYPE_PRESSUREFARFIELD = _cubit3.CI_BCTYPE_PRESSUREFARFIELD

◆ CI_BCTYPE_PRESSUREINLET

CI_BCTYPE_PRESSUREINLET = _cubit3.CI_BCTYPE_PRESSUREINLET

◆ CI_BCTYPE_PRESSUREOUTLET

CI_BCTYPE_PRESSUREOUTLET = _cubit3.CI_BCTYPE_PRESSUREOUTLET

◆ CI_BCTYPE_RADIATOR

CI_BCTYPE_RADIATOR = _cubit3.CI_BCTYPE_RADIATOR

◆ CI_BCTYPE_RIGIDBODY

CI_BCTYPE_RIGIDBODY = _cubit3.CI_BCTYPE_RIGIDBODY

◆CI_BCTYPE_SPRING

CI_BCTYPE_SPRING = _cubit3.CI_BCTYPE_SPRING

◆CI_BCTYPE_SUPERSONICINFLOW

CI_BCTYPE_SUPERSONICINFLOW = _cubit3.CI_BCTYPE_SUPERSONICINFLOW

◆CI_BCTYPE_SUPERSONICOUTFLOW

CI_BCTYPE_SUPERSONICOUTFLOW = _cubit3.CI_BCTYPE_SUPERSONICOUTFLOW

◆CI_BCTYPE_SYMMETRY

CI_BCTYPE_SYMMETRY = _cubit3.CI_BCTYPE_SYMMETRY

◆CI_BCTYPE_TEMPERATURE

CI_BCTYPE_TEMPERATURE = _cubit3.CI_BCTYPE_TEMPERATURE

◆CI_BCTYPE_UNKNOWN

CI_BCTYPE_UNKNOWN = _cubit3.CI_BCTYPE_UNKNOWN

◆CI_BCTYPE_VELOCITY

CI_BCTYPE_VELOCITY = _cubit3.CI_BCTYPE_VELOCITY

◆CI_BCTYPE_VELOCITYINLET

CI_BCTYPE_VELOCITYINLET = _cubit3.CI_BCTYPE_VELOCITYINLET

◆CI_BCTYPE_WALL

CI_BCTYPE_WALL = _cubit3.CI_BCTYPE_WALL

◆CI_CONDUCTIVITY

CI_CONDUCTIVITY = _cubit3.CI_CONDUCTIVITY

◆CI_DENSITY

CI_DENSITY = _cubit3.CI_DENSITY

◆CI_ERROR

CI_ERROR = cvar.CI_ERROR

◆CI_MATERIAL_FIRST

CI_MATERIAL_FIRST = _cubit3.CI_MATERIAL_FIRST

◆CI_MATERIAL_LAST

CI_MATERIAL_LAST = _cubit3.CI_MATERIAL_LAST

◆CI_MATERIAL_UNKNOWNPROPERTY

CI_MATERIAL_UNKNOWNPROPERTY = _cubit3.CI_MATERIAL_UNKNOWNPROPERTY

CI_MODULUS

CI_MODULUS = _cubit3.CI_MODULUS

CI_POISSON

CI_POISSON = _cubit3.CI_POISSON

CI_SHEAR_MODULUS

CI_SHEAR_MODULUS = _cubit3.CI_SHEAR_MODULUS

CI_SPECIFIC_HEAT

CI_SPECIFIC_HEAT = _cubit3.CI_SPECIFIC_HEAT

CI_THERMAL_EXPANSION

CI_THERMAL_EXPANSION = _cubit3.CI_THERMAL_EXPANSION

CI_ULTIMATE_STRAIN

CI_ULTIMATE_STRAIN = _cubit3.CI_ULTIMATE_STRAIN

CI_ULTIMATE_STRENGTH

CI_ULTIMATE_STRENGTH = _cubit3.CI_ULTIMATE_STRENGTH

CI_YIELD_STRENGTH

CI_YIELD_STRENGTH = _cubit3.CI_YIELD_STRENGTH

CUBIT_TRI

CUBIT_TRI = _cubit3.CUBIT_TRI

CURVE

CURVE = _cubit3.CURVE

cvar

cvar = _cubit3.cvar

HEX

HEX = _cubit3.HEX

HEX20

HEX20 = _cubit3.HEX20

HEX21

HEX21 = _cubit3.HEX21

HEX26

HEX26 = _cubit3.HEX26

HEX27

HEX27 = _cubit3.HEX27

◆ HEX8

HEX8 = _cubit3.HEX8

◆ HEX9

HEX9 = _cubit3.HEX9

◆ HEXSHELL

HEXSHELL = _cubit3.HEXSHELL

◆ INVALID_ELEMENT_TYPE

INVALID_ELEMENT_TYPE = _cubit3.INVALID_ELEMENT_TYPE

◆ PYRAMID

PYRAMID = _cubit3.PYRAMID

◆ PYRAMID13

PYRAMID13 = _cubit3.PYRAMID13

◆ PYRAMID18

PYRAMID18 = _cubit3.PYRAMID18

◆ PYRAMID5

PYRAMID5 = _cubit3.PYRAMID5

◆ PYRAMID8

PYRAMID8 = _cubit3.PYRAMID8

◆ QUAD

QUAD = _cubit3.QUAD

◆ QUAD4

QUAD4 = _cubit3.QUAD4

◆ QUAD5

QUAD5 = _cubit3.QUAD5

◆ QUAD8

QUAD8 = _cubit3.QUAD8

◆ QUAD9

QUAD9 = _cubit3.QUAD9

◆ SHARED_PTR_DISOWN

SHARED_PTR_DISOWN = _cubit3.SHARED_PTR_DISOWN

SHEL

SHEL = _cubit3.SHEL

SHELL4

SHELL4 = _cubit3.SHELL4

SHELL8

SHELL8 = _cubit3.SHELL8

SHELL9

SHELL9 = _cubit3.SHELL9

SPHERE_EXO

SPHERE_EXO = _cubit3.SPHERE_EXO

SPRING

SPRING = _cubit3.SPRING

SUPERELEMENT_TOPOLOGY

SUPERELEMENT_TOPOLOGY = _cubit3.SUPERELEMENT_TOPOLOGY

SURFACE

SURFACE = _cubit3.SURFACE

TETRA

TETRA = _cubit3.TETRA

TETRA10

TETRA10 = _cubit3.TETRA10

TETRA14

TETRA14 = _cubit3.TETRA14

TETRA15

TETRA15 = _cubit3.TETRA15

TETRA4

TETRA4 = _cubit3.TETRA4

TETRA8

TETRA8 = _cubit3.TETRA8

TRI3

TRI3 = _cubit3.TRI3

TRI4

TRI4 = _cubit3.TRI4

TRI6

TRI6 = _cubit3.TRI6

TRI7

TRI7 = _cubit3.TRI7

TRISHELL

TRISHELL = _cubit3.TRISHELL

TRISHELL3

TRISHELL3 = _cubit3.TRISHELL3

TRISHELL4

TRISHELL4 = _cubit3.TRISHELL4

TRISHELL6

TRISHELL6 = _cubit3.TRISHELL6

TRISHELL7

TRISHELL7 = _cubit3.TRISHELL7

TRUSS

TRUSS = _cubit3.TRUSS

TRUSS2

TRUSS2 = _cubit3.TRUSS2

TRUSS3

TRUSS3 = _cubit3.TRUSS3

VERTEX

VERTEX = _cubit3.VERTEX

VOLUME

VOLUME = _cubit3.VOLUME

WEDGE

WEDGE = _cubit3.WEDGE

WEDGE15

WEDGE15 = _cubit3.WEDGE15

WEDGE16

WEDGE16 = _cubit3.WEDGE16

WEDGE20

WEDGE20 = _cubit3.WEDGE20

WEDGE21

WEDGE21 = _cubit3.WEDGE21

WEDGE6

WEDGE6 = _cubit3.WEDGE6

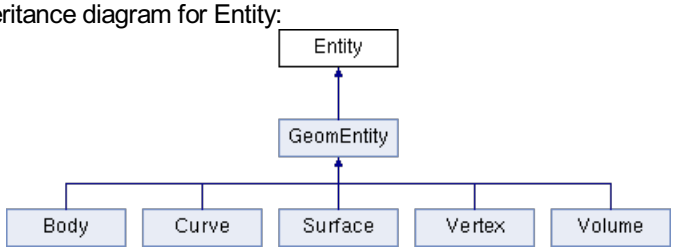
Cubit Python API 17.05

Entity

Public Member Functions

The base class of all the geometry and mesh types. More...

Inheritance diagram for Entity:



Public Member Functions

def **bounding_box** (self)
Returns the axis-aligned bounding box of the **Entity**. More...

def **center_point** (self)
Returns the geometric center point of the **Entity**. More...

def **id** (self)
Retrieves the unique identifier of the **Entity**. More...

Detailed Description

The base class of all the geometry and mesh types.

.. code-block:: python

```
import cubit


br = cubit.brick
(1,1,1)


cubit.scale
(br,2)


cubit.cmd
('delete body 1' )
```

Member Function Documentation

◆ bounding_box()

def bounding_box (*self*)

Returns the axis-aligned bounding box of the **Entity** .

Computes the minimum and maximum coordinates of the Entityxs geometry in X, Y, and Z directions.

.. code-block:: python

```
                # Create a half-sphere of radius
                1 cut by the YZ-plane


                halfSphere = cubit.sphere
                (1.0, 1)


                # Compute bounding box as a list
                [minX, minY, minZ, maxX, maxY, maxZ]


                bbox = halfSphere.bounding_box()


                print(f"Bounding Box:
                [{bbox[0]}, {bbox[1]}, {bbox[2]}] to
    [{bbox[3]},
                {bbox[4]}, {bbox[5]}]" ) #
                prints [0, -1, -1] to [1, 1, 1]
```

@n return type of : std:: array< double,6 >

Returns

: An array (or list) of six values minX, minY, minZ, maxX, maxY, maxZ.

◆ center_point()

def center_point (*self*)

Returns the geometric center point of the **Entity** .

Computes the centroid of the Entityxs geometry based on its current position.

```
.. code-block:: python
```

```
                # Create a unit cube (center at
                origin)


                b = cubit.brick
                (1.0, 1.0, 1.0)


                # Check original center


                center = b.center_point()


                print(f"Original Center:
                ({center[0]}, {center[1]},
    {center[2]})" )
                # prints (0, 0, 0)


                # Move the cube by (1, 2,
                3)


                cubit.move
                (b, [1.0, 2.0, 3.0], False
                )


                # Check new center


                center = b.center_point()


                print(f"New Center:
                ({center[0]}, {center[1]},
    {center[2]})" )
                # prints (1, 2, 3)
```

```
@n return type of : std:: array< double,3 >
```

Returns

: An array (or list) of three values x, y, and z coordinates of the Entityxs center.

◆ id()

def id (*self*)

Retrieves the unique identifier of the **Entity** .

Returns the integer ID assigned to the **Entity** upon creation.

```
.. code-block:: python
```

```
# Create a unit cube

b = cubit.brick
(1.0, 1.0, 1.0)

# Get and print its ID

id = b.id()

print("Entity ID:" ,
id) # prints 1
```

```
@n return type of :  int
```

Returns

The integer ID of the **Entity**.

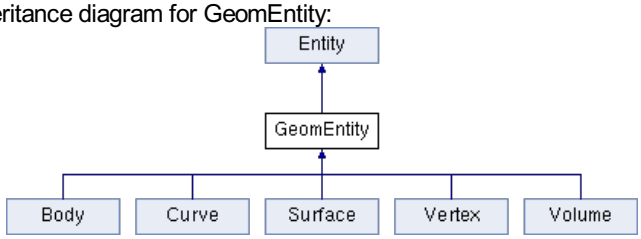
Cubit Python API 17.05

GeomEntity

Public Member Functions

The base class for specifically the Geometry types (**Body** , **Surface** , etc.) More...

Inheritance diagram for GeomEntity:



Public Member Functions

def **bodies** (self)
Retrieve bodies contained within the geometry entity. More...

def **curves** (self)
Retrieve curves (edges) associated with the geometry entity. More...

def **dimension** (self)
Get the topological dimension of the geometry entity. More...

def **entity_name** (self)
Retrieve the primary name of the geometry entity. More...

def **entity_names** (self)
Retrieve all names assigned to the geometry entity. More...

def **is_meshed** (self)
Returns the current mesh state of the **GeomEntity**. More...

def **is_transparent** (self)
Get the transparency state of the geometry entity. More...

def **is_visible** (self)
Get the visibility state of the geometry entity. More...

def **mesh** (self)
Generates mesh on a **GeomEntity** and verifies meshing status. More...

def **num_names** (self)
Get the count of names assigned to the geometry entity. More...

def **remove_entity_name** (self, name)
Remove a specific name from the geometry entity. More...

def **remove_entity_names** (self)
Remove all non-default names from the geometry entity. More...

def **remove_mesh** (self)
Remove any mesh associated with the geometry entity. More...

def **set_entity_name** (self, name)
Assign a name to the geometry entity. More...

def **set_transparent** (self, transparency_flag)
Set the transparency state of the geometry entity. More...

def **set_visible** (self, visibility_flag)
Set and verify the visibility state of the geometry entity. More...

def **smooth** (self)
Smooths the mesh on a **GeomEntity** to improve element quality. More...

def **surfaces** (self)

Retrieve surfaces associated with the geometry entity. More...

def **vertices** (self)
Retrieve vertices (points) associated with the geometry entity. More...

def **volumes** (self)
Retrieve volumes associated with the geometry entity. More...

▸ **Public Member Functions inherited from Entity**

Detailed Description

The base class for specifically the Geometry types (**Body** , **Surface** , etc.)

Member Function Documentation

◆ bodies()

def bodies (*self*)

Retrieve bodies contained within the geometry entity.

Returns a list of **Body** objects associated with this **GeomEntity** , such as sheet bodies from a surface or volumes from a body.

.. code-block:: python

```
# Create a 2D rectangular sheet
via a Cubit command


cubit.cmd
("create surface rectangle
width 1" )



# Get the surface entity by its
ID


surf = cubit.surface
(1)



# Retrieve sheet bodies
associated with this surface


bodies = surf.bodies()


print("Number of
bodies:" , len(bodies)) #
Expected output: 1
```

@n return type of : std::vector< CubitInterface::Body,std::allocator< CubitInterface:: Body > >

Returns
A vector (list) of **Body** objects contained within the **GeomEntity**.

◆ curves()

def curves (*self*)

Retrieve curves (edges) associated with the geometry entity.

Returns a list of **Curve** objects that bound this **GeomEntity** , such as the edges of a brick or boundary curves of surfaces.

.. code-block:: python

```
# Create a brick which has edges
as curves


b = cubit.brick
(1)



# Retrieve all curves (edges) of
the brick


curves = b.curves()


print("Number of
curves:" , len(curves)) #
Expected output: 12



# Access the first curve and
query its dimension


c = curves[0]


print("Curve
dimension:" , c.dimension()) # Expected output: 1
```

@n return type of : std::vector< CubitInterface::Curve,std::allocator< CubitInterface:: Curve > >

Returns

A vector (list) of **Curve** objects associated with this **GeomEntity**.

dimension()

def dimension (*self*)

Get the topological dimension of the geometry entity.

Returns the dimension (0 for **Vertex** , 1 for **Curve** , 2 for **Surface** , 3 for Volume/Body) of this **GeomEntity** .

```
.. code-block:: python
```

```
                # Create a brick and query its
                dimension


                b = cubit.brick
                (1)


                print("Body
                dimension:" , b.dimension()) # Expected
    output: 3



                # Create a cylinder and query its
                dimension


                cyl = cubit.cylinder
                (0.5, 2.0)


                print("Cylinder
                dimension:" , cyl.dimension()) #
    Expected output: 3



                # Retrieve boundary curves of the
                cylinder and query their dimensions


                curves = cyl.curves()
```

for c in curves: print("Curve dimension:", c.dimension()) # Expected output: 1

```
@n return type of :  int
```

Returns

The topological dimension of the **GeomEntity** (0x3).

◆ entity_name()

def entity_name (*self*)

Retrieve the primary name of the geometry entity.

Returns the first name assigned to this **GeomEntity** , useful for identification and labeling.

.. code-block:: python

```
# Create a brick, set its name,
and retrieve it

b = cubit.brick
(1)

b.set_entity_name("MyBrick" )


name = b.entity_name()

print("Entity name:"
, name)
```

@n return type of : string

Returns

The first name of the **GeomEntity**.

◆ entity_names()

def entity_names (*self*)

Retrieve all names assigned to the geometry entity.

Returns a list of every name that has been assigned to this **GeomEntity** , allowing enumeration of aliases or prior identifiers.

.. code-block:: python

```
# Create a brick, assign two
names, and list them

b = cubit.brick
(1)

b.set_entity_name("BrickA" )

b.set_entity_name("PrimaryBrick" )


names = b.entity_names()
```

for n in names: print("Name:", n)

@n return type of : std::vector< std::string,std::allocator< std:: string > >

Returns

A vector (list) of all names assigned to the **GeomEntity**.

◆ is_meshed()

def is_meshed (*self*)

Returns the current mesh state of the **GeomEntity** .

Indicates whether the **GeomEntity** has mesh elements defined.

```
.. code-block:: python
```

```
# Create and mesh a 10x10x10
brick


b = cubit.brick
(10.0, 10.0, 10.0)


b.mesh()


# Check mesh state


meshed = b.is_meshed()


print("Meshed:" ,
meshed) # prints True
```

```
@n return type of :  boolean
```

Returns

True if the **GeomEntity** is meshed, false otherwise.

◆ is_transparent()

def is_transparent (*self*)

Get the transparency state of the geometry entity.

Returns whether this **GeomEntity** is currently rendered as transparent (`1`) or opaque (`0`).

```
.. code-block:: python
```

```
# Create a brick and query
default transparency


b = cubit.brick
(1)


print("Is brick
transparent?" , b.is_transparent()) #
Expected output: 1
```

```
@n return type of :  int
```

Returns

The current transparency state of the **GeomEntity** (`1` if transparent, `0` if opaque).

◆ is_visible()

def is_visible (*self*)

Get the visibility state of the geometry entity.

Returns whether this **GeomEntity** is currently visible in the viewport (1) or hidden (0).

.. code-block:: python

```
# Create a brick and hide
it

b = cubit.brick
(1)

b.set_visible(False )

# Query visibility state

vis = b.is_visible()

print("Is brick
visible?" , vis) #
Expected output: False
```

@n return type of : int

Returns

The current visibility state of the **GeomEntity** (1 if visible, 0 if hidden).

◆ mesh()

def mesh (*self*)

Generates mesh on a **GeomEntity** and verifies meshing status.

Discretizes the geometry of the given **GeomEntity** (e.g., surface, volume) into mesh elements. Use `is_meshed` to verify if meshing has been performed on the entity.

.. code-block:: python

```
# Create a 10x10x10 brick

b = cubit.brick
(10.0, 10.0, 10.0)

# Generate mesh on the
volume

b.mesh()

# Verify that meshing has
occurred

meshed = b.is_meshed()

print("Mesh
generated:" , meshed) #
prints True
```

◆ num_names()

def num_names (*self*)

Get the count of names assigned to the geometry entity.

Returns the number of names currently assigned to this **GeomEntity** for tracking aliases or identifiers.

.. code-block:: python

```
# Create a brick, assign two
names, and get the name count


b = cubit.brick
(1)


b.set_entity_name("BrickA" )


b.set_entity_name("PrimaryBrick" )


count = b.num_names()


print("Name count:"
, count)
```

@n return type of : int

Returns

The number of names assigned to the **GeomEntity**.

◆ remove_entity_name()

def remove_entity_name (*self*,
name
)

Remove a specific name from the geometry entity.

Deletes one of the names previously assigned to this **GeomEntity** , preserving any other names.

.. code-block:: python

```
                # Create a brick, assign two
                names, remove one, and list remaining
    names


                b = cubit.brick
                (1)


                b.set_entity_name("BrickA" )


                b.set_entity_name("PrimaryBrick" )


                # Remove the first assigned
                name


                b.remove_entity_name("BrickA" )


                # Verify remaining names


                names = b.entity_names()
```

for n in names: print("Remaining name:", n)

@n type of name: string, in

Parameters

name The specific name to remove from this **GeomEntity**.

remove_entity_names()

def remove_entity_names (*self*)

Remove all non-default names from the geometry entity.

Deletes every custom name assigned to this **GeomEntity** , preserving its inherent default name.

.. code-block:: python

```
                # Create a brick, assign two
                custom names, then remove them and
    verify


                b = cubit.brick
                (1)


                b.set_entity_name("BrickA" )


                b.set_entity_name("PrimaryBrick" )


                # Initial name count (default + 2
                custom names)


                print("Before removal, name
                count:" , b.num_names())


                # Expected output: 3


                # Remove all custom names


                b.remove_entity_names()


                # Verify only default name
                remains


                print("After removal, name
                count:" , b.num_names())


                # Expected output: 1
```

◆ remove_mesh()

def remove_mesh (*self*)

Remove any mesh associated with the geometry entity.

Deletes the mesh on a **GeomEntity**

```
.. code-block:: python
```

```
# Create a brick and mesh
it

b = cubit.brick
(1)

b.mesh()

# Check the number of hexes that
were created

print("Hex count:" ,
cubit.get_hex_count
())

# Delete the mesh on the
brick

b.remove_mesh()

# Check that the number of
elements is 0

print("Hex count:" ,
cubit.get_hex_count
())
```

◆ set_entity_name()

def set_entity_name (*self*,
name
)

Assign a name to the geometry entity.

Sets the primary name of this **GeomEntity** for identification and labeling.

```
        .. code-block:: python
```

```
                    # Create a brick and set its
                    name

                    b = cubit.brick
                    (1)

                    b.set_entity_name("Brick1" )


                    # Retrieve and print the assigned
                    name

                    name = b.entity_name()

                    print("Entity name:"
                    , name)
```

```
        @n type of name:  string, in
```

Parameters

name The name to be assigned to the **GeomEntity**.

◆ set_transparent()

def set_transparent (*self*,
transparency_flag
)

Set the transparency state of the geometry entity.

Toggles whether this **GeomEntity** is rendered with transparency. Passing `1` makes it transparent; `0` makes it opaque. The default state upon creation is transparent.

```
        .. code-block:: python
```

```
                # Create a brick (default is
                transparent)

                b = cubit.brick
                (1)


                # Turn transparency off

                b.set_transparent(0)

                print("Is brick
                transparent?" , b.is_transparent()) #
    Expected output: 0


                # Turn transparency back
                on

                b.set_transparent(1)

                print("Is brick
                transparent?" , b.is_transparent()) #
    Expected output: 1
```

```
        @n type of transparency_flag:  int, in
```

Parameters

transparency_flag The flag (`1` for transparent, `0` for opaque) to set the render transparency state.

set_visible()

def set_visible (*self*,
visibility_flag
)

Set and verify the visibility state of the geometry entity.

Toggles whether this **GeomEntity** is rendered in the viewport. Passing `true` makes it visible; `false` hides it. Use **`is_visible()`** to query the current state.

.. code-block:: python

```
                # Create a brick


                b = cubit.brick
                (1)



                # Hide the brick and
                verify


                b.set_visible(False )


                print("Is brick
                visible?" , b.is_visible()) # Expected
    output: False



                # Show the brick and
                re-verify


                b.set_visible(True )


                print("Is brick
                visible?" , b.is_visible()) # Expected
    output: True
```

@n type of visibility_flag: boolean, in

Parameters

visibility_flag The flag to set whether the **Entity** is visible (`true`) or hidden (`false`).

◆ smooth()

def smooth (*self*)

Smooths the mesh on a **GeomEntity** to improve element quality.

Applies mesh smoothing to the existing mesh on the **GeomEntity** . Requires that the entity has already been meshed (`is_meshed()` returns true). Smoothing method is based on the scheme currently set on the geomEntity (ie. Laplacian, Mean Ratio, etc.)

.. code-block:: python

```
# create a sphere with radius
1

sphere_body = cubit.sphere
(1)

# get the surface from the sphere
body

sphere_surf = sphere_body.surfaces()[0]

# set the meshing scheme to
trimesh

cubit.cmd
(f"surface
{sphere_surf.id()} scheme trimesh" )

# set the smoothing scheme to
mean ratio

cubit.cmd
(f"surface
{sphere_surf.id()} smooth scheme mean
ratio" )

# mesh the surface with
triangles

sphere_surf.mesh()

# check the quality and display
the result

cubit.cmd
("quality surf all draw
mesh" )

# smooth the surface mesh

sphere_surf.smooth()

# display the new quality after
smoothing

cubit.cmd
("quality surf all draw
mesh" )
```

◆ surfaces()

def surfaces (*self*)

Retrieve surfaces associated with the geometry entity.

Returns a list of **Surface** objects that bound this **GeomEntity** , such as the faces of a brick.

```
.. code-block:: python
```

```
# Create a brick which generates
a volume


b = cubit.brick
(1)



# Retrieve boundary surfaces from
the body


surfaces = b.surfaces()


print("Number of
surfaces:" , len(surfaces)) # Expected output: 6



# Access the first surface and
query its dimension


s = surfaces[0]


print("Surface
dimension:" , s.dimension()) # Expected output: 2
```

```
@n return type of : std::vector< CubitInterface::Surface,std::allocator< CubitInterface:: Surface > >
```

Returns

A vector (list) of **Surface** objects associated with this **GeomEntity**.

◆ vertices()

def vertices (*self*)

Retrieve vertices (points) associated with the geometry entity.

Returns a list of **Vertex** objects that bound this **GeomEntity** , such as the corners of a brick.

.. code-block:: python

```
# Create a brick which generates
a volume


b = cubit.brick
(1)



# Retrieve vertices from the
body


vertices = b.vertices()


print("Number of
vertices:" , len(vertices)) # Expected output: 8



# Access the first vertex and
query its dimension


v = vertices[0]


print("Vertex
dimension:" , v.dimension()) # Expected output: 0
```

@n return type of : std::vector< CubitInterface::Vertex,std::allocator< CubitInterface:: Vertex > >

Returns

A vector (list) of **Vertex** objects associated with this **GeomEntity**.

volumes()

def volumes (*self*)

Retrieve volumes associated with the geometry entity.

Returns a list of **Volume** objects associated with this **GeomEntity** , such as the volume created by a brick.

.. code-block:: python

```
# Create a brick which generates
a volume


b = cubit.brick
(1)



# Retrieve volumes from the
body


volumes = b.volumes()


print("Number of
volumes:" , len(volumes)) # Expected output: 1



# Access the first volume and
query its dimension


vol = volumes[0]


print("Volume
dimension:" , vol.dimension()) # Expected output: 3
```

@n return type of : std::vector< CubitInterface::Volume,std::allocator< CubitInterface:: Volume > >

Returns

A vector (list) of **Volume** objects associated with this **GeomEntity**.

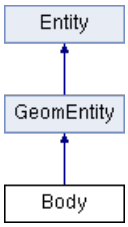
Cubit Python API 17.05

Body

Public Member Functions

Defines a body object that mostly parallels Cubit's **Body** class. More...

Inheritance diagram for Body:



Public Member Functions

def **get_mass_props** (self)
Get the center of gravity of the **Body**. More...

def **is_sheet_body** (self)
Get whether the **Body** is a sheet body or not. More...

def **point_containment** (self, loc_in)
Determine point containment relative to the **Body**. More...

def **volume** (self)
Get the volume of the **Body**. More...

- **Public Member Functions inherited from GeomEntity**
- **Public Member Functions inherited from Entity**

Detailed Description

Defines a body object that mostly parallels Cubit's **Body** class.

Member Function Documentation

◆ get_mass_props()

def get_mass_props (*self*)

Get the center of gravity of the **Body** .

Computes and returns the mass properties of this **Body** , specifically the center of gravity coordinates.

.. code-block:: python

```
# Create and unite two bricks: a
cube and a long rectangular brick


cubit.cmd
("brick x 1 y 1 z 1"
)


cubit.cmd
("brick x 10 y 1 z
1" )


cubit.cmd
("vol 2 move x 10" )


cubit.cmd
("unite vol all" )



# Retrieve the unified
body


b = cubit.body
(1)



# Get geometric center and center
of gravity


center_pt = b.center_point()


cg = b.get_mass_props()



print("Geometric
center:" , center_pt)


print("Center of
gravity:" , cg)


# Expected: geometric center !=
center of gravity
```

@n return type of : std:: array< double,3 >

Returns
: A std::array of three double values the x, y, and z coordinates of the **Body**'s center of gravity.

◆ is_sheet_body()

def is_sheet_body (*self*)

Get whether the **Body** is a sheet body or not.

Determines if this **Body** represents a sheet (2D) geometry rather than a volume (3D).

.. code-block:: python

```
# Create a 3D brick
(volume)


cubit.cmd
("reset" )


vol_body = cubit.brick
(1)


print("Is sheet
body?" , vol_body.is_sheet_body()) #
Expected output: False



# Create a 2D rectangular sheet
via command


cubit.cmd
("reset" )


cubit.cmd
("create surface rectangle
width 1" )


surf = cubit.surface
(1)


# Extract the sheet body from the
surface


sheet_bodies = surf.bodies()


sheet_body = sheet_bodies[0]


print("Is sheet
body?" , sheet_body.is_sheet_body())
# Expected output: True
```

@n return type of : boolean

Returns

True if the **Body** is a sheet body, False otherwise.

◆ point_containment()

def point_containment (*self*,
loc_in
)

Determine point containment relative to the **Body** .

Checks whether a given point is inside, on, or outside this **Body**

.. code-block:: python

```
                # Create a unit cube body (1x1x1)
                centered at origin (extends -0.5 to
0.5)


                b = cubit.brick
                (1)



                # Point strictly inside the
                volume


                print("Inside:" ,
                b.point_containment([0.0, 0.0, 0.0])) #
Expected output: 1



                # Point on the boundary


                print("On:" ,
                b.point_containment([0.5, 0.0, 0.0])) #
Expected output: 2



                # Point outside the volume


                print("Outside:"
                ,b.point_containment([1.0, 0.0, 0.0]))
# Expected output: 0
```

@n type of loc_in: std::array< double,3 >, in

Parameters

loc_in A three-element array specifying the point coordinates (x, y, z).
return type of : int

Returns

: An integer code -1 if unknown, 0 if outside, 1 if inside, or 2 if on the boundary of the **Body**.

◆ volume()

def volume (*self*)

Get the volume of the **Body** .

Notes This function is only available on **Body** objects.

```
Returns the computed geometric volume of this Body.



.. code-block:: python
```

```
# Create a unit cube body
(1x1x1)

b = cubit.brick
(1)

# Compute and print its
volume

vol = b.volume()

print("Body volume:"
, vol) # Expected output:
1.0
```

```
@n return type of :  float
```

Returns

The volume of the **Body**.

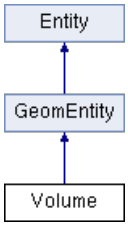
Cubit Python API 17.05

Volume

Public Member Functions

Defines a volume object that mostly parallels Cubit's RefVolume class. More...

Inheritance diagram for Volume:



Public Member Functions

def **centroid** (self)
Get the centroid of the **Volume**. More...

def **color** (self)
Get the RGBA color of this **Volume** entity. More...

def **principal_axes** (self)
Retrieve the principal axes of this **Volume** entity. More...

def **principal_moments** (self)
Get the principal moments of the **Volume**. More...

def **set_color** (self, value)
Set the RGBA color of this **Volume** entity. More...

def **volume** (self)
Compute the three-dimensional volume of this **Volume** entity. More...

Public Member Functions inherited from GeomEntity

Public Member Functions inherited from Entity

Detailed Description

Defines a volume object that mostly parallels Cubit's RefVolume class.

Member Function Documentation

◆ centroid()

def centroid (*self*)

Get the centroid of the **Volume** .

.. code-block:: python

```
centroid = volume.centroid()
```

```
@n return type of : std:: array< double,3 >
```

Returns

A vector (or list) of the coordinates of the centroid of the volume with the indices of the vector corresponding to the values as follows:

0 x coordinate

1 y coordinate

2 z coordinate

color()

def color (*self*)

Get the RGBA color of this **Volume** entity.

Retrieves the current display color of the volume as RGBA channels.

Note

if a specific color has not been explicitly applied, this function returns (0.0, 0.0, 0.0, 0.0).

Example create a sphere with radius 1, apply a yellow color (via command or API), and retrieve its color.

.. code-block:: python

```
# Create a sphere of radius
1


cubit.cmd
("sphere rad 1" )


# Obtain the Volume object for ID
1


vol_obj = cubit.volume
(1)


# Method 1: apply yellow via
command interface


cubit.cmd
("color vol 1
yellow" )


# Method 2: apply yellow via API
call


vol_obj.set_color([1.0, 1.0, 0.0, 1.0])


# Get the RGBA color


col = vol_obj.color()


print("Color RGBA:"
, col) # Expected output: [1.0,
1.0, 0.0, 1.0]
```

```
@n return type of : std:: array< double,4 >
```

Returns

Array of four doubles representing the RGBA channels of the volume's color.

◆ principal_axes()

def principal_axes (*self*)

Retrieve the principal axes of this **Volume** entity.

Returns the three principal directions (axes) of the volume, corresponding to its principal moments of inertia, as a flattened 3x3 matrix.

Example create a unit sphere and get its principal axes (unit vectors along X, Y, Z).

```
.. code-block:: python

                # Create a sphere of radius
                1


                cubit.cmd
                ("sphere rad 1" )


                # Obtain the Volume object for ID
                1


                vol_obj = cubit.volume
                (1)


                # Retrieve principal axes


                axes = vol_obj.principal_axes()


                print("Axes:" ,
                axes) # Expected: [1.0, 0.0, 0.0,
                0.0, 1.0, 0.0, 0.0, 0.0, 1.0]

      @n return type of : std:: array< double,9 >
```

Returns

: Array of nine doubles representing the principal axes indicesx0x2 = axisx1 (x,y,z), 3x5 = axisx2, 6x8 = axisx3.

◆ principal_moments()

def principal_moments (*self*)

Get the principal moments of the **Volume** .

.. code-block:: python

```
moments = volume.principal_moments()
```

```
@n return type of : std:: array< double,3 >
```

Returns

A vector (or list) of the principal moments of the volume with the indices of the vector corresponding to the values as follows:

0 x moment

1 y moment

2 z moment

◆ set_color()

def set_color (*self*,
value
)

Set the RGBA color of this **Volume** entity.

Applies a display color using red, green, blue, and alpha (opacity) channels.

Note

the alpha channel controls opacity (0.0 = fully transparent, 1.0 = fully opaque).

Example create a sphere with radius 1, apply yellow color (full and semi-transparent) via command and API, and force graphics updates.

.. code-block:: python

```
# Create a sphere of radius
1


cubit.cmd
("sphere rad 1" )


# Obtain the Volume object for ID
1


vol_obj = cubit.volume
(1)


# Method 1: set yellow via
command interface (opaque)


cubit.cmd
("color vol 1
yellow" )


# Force graphics update


cubit.cmd
("display" )


# Method 2: set semi-transparent
yellow via API (alpha = 0.5)


vol_obj.set_color([1.0, 1.0, 0.0, 0.5])


# Force graphics update


cubit.cmd
("display" )
```

```
@n type of value:  std::array< double,4 >, in
```

Parameters

value Array of four doubles [R,G,B,A] in [0,1], where A controls opacity.

◆ volume()

def volume (*self*)

Compute the three-dimensional volume of this **Volume** entity.

Calculates the geometric volume enclosed by this **Volume** object.

Example create a sphere with radius 1 via command-line, obtain its **Volume** object, and measure its volume.

```
.. code-block:: python

                # Create a sphere of radius
                1


                cubit.cmd
                ("sphere rad 1" )


                # Obtain the Volume object for ID
                1


                vol_obj = cubit.volume
                (1)


                # Compute the geometric
                volume


                geo_vol = vol_obj.volume()


                print("Computed
                volume:" , geo_vol) #
                Expected output: 4.18879
```

```
@n return type of :  float
```

Returns

The computed three-dimensional volume.

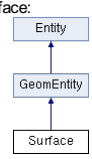
Cubit Python API 17.05

Surface

Public Member Functions

Defines a surface object that mostly parallels Cubit's RefFace class.
More...

Inheritance diagram for Surface:



Public Member Functions

def **area** (self)
Get area of the **Surface**. More...

def **closest_point_along_vector** (self, location, along_vector)
Get the nearest point on the **Surface** to point specified along the specified vector. More...

def **closest_point_trimmed** (self, location)
Get the nearest point on the **Surface** to point specified. More...

def **color** (self)
Get the color of the surface. More...

def **get_param_range_U** (self)
Get range of u for the **Surface**. More...

def **get_param_range_V** (self)
Get range of v for the **Surface**. More...

def **is_cylindrical** (self)
Get whether the **Surface** is cylindrical or not. More...

def **is_planar** (self)
Get whether the **Surface** is planar or not. More...

def **normal_at** (self, location)
Get the normal at a particular point on the **Surface**. More...

def **ordered_loops** (self)
Get the ordered loops of the **Surface**. More...

def **point_containment** (self, point_in)
Get whether a point is on or off of the **Surface**. More...

def **position_from_u_v** (self, u, v)
Get the Cartesian coordinates from the uv coordinates on the **Surface**. More...

def **principal_curvatures** (self, point)
Get the principal curvatures of the **Surface**. More...

def **set_color** (self, value)
Set the color of the surface. More...

def **u_v_from_position** (self, location)
Get the uv coordinates from the supplied Cartesian coordinates on the **Surface**. More...

▸ **Public Member Functions inherited from GeomEntity**

▸ **Public Member Functions inherited from Entity**

Detailed Description

Defines a surface object that mostly parallels Cubit's RefFace class.

Member Function Documentation

◆ area()

def area (*self*)

Get area of the **Surface** .

.. code-block:: python

```
area = surface.area()
```

```
@n return type of :  float
```

Returns
The area of the **Surface**.

◆ closest_point_along_vector()

def closest_point_along_vector (*self*,
location,
along_vector
)

Get the nearest point on the **Surface** to point specified along the specified vector.

.. code-block:: python

```
nearest =
surface.closest_point_along_vector([0,0,0],
[1,1,1])
```

```
@n type of location:  std::array< double,3 >, in
```

Parameters

location A vector containing three values that are the coordinates of a point.
return type of : std:: array< double,3 >

Returns

A vector (or list) of doubles representing values of nearest point as follows:

0 x coordinate

1 y coordinate

2 z coordinate

◆ closest_point_trimmed()

def closest_point_trimmed (*self*,
location
)

Get the nearest point on the **Surface** to point specified.

.. code-block:: python

```
nearest =
surface.closest_point_trimmed([0,0,0])
```

```
@n type of location:  std::array< double,3 >, in
```

Parameters

location A vector containing three values that are the coordinates of a point.
return type of : std:: array< double,3 >

Returns

A vector (or list) of doubles representing values of nearest point as follows:

0 x coordinate

1 y coordinate

2 z coordinate

◆ color()

def color (*self*)

Get the color of the surface.

.. code-block:: python

```
col = surface.color()
```

```
@n return type of : std:: array< double,4 >
```

Returns

The color value associated with the surface's current color.

◆ get_param_range_U()

def get_param_range_U (*self*)

Get range of u for the **Surface** .

.. code-block:: python

```
bounds = surface.get_param_range_U()
```

```
@n return type of : std:: array< double,2 >
```

Returns

The curvature values:

0 The lowest value in the u direction

1 The highest value in the u direction

◆ get_param_range_V()

def get_param_range_V (*self*)

Get range of v for the **Surface** .

.. code-block:: python

```
                lower_bound, upper_bound =
    surface.get_param_range_V()
```

```
      @n return type of : std:: array< double,2 >
```

Returns

The curvature values:

0 The lowest value in the v direction

1 The highest value in the v direction

◆ is_cylindrical()

def is_cylindrical (*self*)

Get whether the **Surface** is cylindrical or not.

.. code-block:: python

```
                cyl = surface.is_cylindrical()
```

```
  @n return type of :  boolean
```

Returns

Whether the **Surface** is cylindrical or not.

◆ is_planar()

def is_planar (*self*)

Get whether the **Surface** is planar or not.

.. code-block:: python

```
                planar = surface.is_planar()
```

```
  @n return type of :  boolean
```

Returns

Whether the **Surface** is planar or not.

◆ normal_at()

def normal_at (*self*,
location
)

Get the normal at a particular point on the **Surface** .

.. code-block:: python

```
                norm = surface.normal_at([0,0,0])
```

```
    @n type of location:  std::array< double,3 >, in
```

Parameters

location A vector containing three values that are the coordinates of a point.
return type of : std:: array< double,3 >

Returns

A vector (or list) of doubles representing values of normal vector as follows:

0 x value

1 y value

2 z value

◆ ordered_loops()

def ordered_loops (*self*)

Get the ordered loops of the **Surface** .

.. code-block:: python

```
                loops = surface.ordered_loops()
```

```
    @n return type of : std::vector< std::vector< CubitInterface::Curve,std::allocator< CubitInterface::Curve > >,std::allocator< std::vector< CubitInterface::Curve,std::
```

Returns
A vector of vectors (or list of lists) of Curves in loops:

0, 0 - loop 1 curve 1

0, 1 - loop 1 curve 2

1, 0 - loop 2 curve 1

Returns
etc...

◆ point_containment()

def point_containment (*self*,
point_in
)

Get whether a point is on or off of the **Surface** .

.. code-block:: python

```
                on_off =
    surface.point_containment([0,0,0])
```

```
  @n type of point_in:  std::array< double,3 >, in
```

Parameters
point_in A vector containing three values that are the coordinates of a point.
return type of : int

Returns
A python boolean representing whether the point is off (0) or on (1) the **Surface**.

◆ position_from_u_v()

def position_from_u_v (*self*,
u,
v
)

Get the Cartesian coordinates from the uv coordinates on the **Surface** .

.. code-block:: python

```
                pos = surface.position_from_u_v(0, 0)
```

```
    @n type of u:  float, in
```

Parameters
u The u parameter
type of v: float, in
v The v parameter
return type of : std:: array< double,3 >

Returns
The Cartesian coordinates of the supplied uv coordinates as a vector:
0 x coordinate
1 y coordinate
2 z coordinate

◆ principal_curvatures()

def principal_curvatures (*self*,
point
)

Get the principal curvatures of the **Surface** .

.. code-block:: python

```
curvatures =
surface.principal_curvatures([0,0,0])
```

`@n type of point: std::array< double,3 >, in`

Parameters

point A tuple containing 2 values that are the curvature in the principle u and v directions on the surface
return type of : std:: array< double,2 >

Returns

A list of two floats representing the curvatures.

0 curvature 1

1 curvature 2

◆ set_color()

def set_color (*self*,
value
)

Set the color of the surface.

.. code-block:: python

```
surface.set_color(0)
```

`@n type of value: std::array< double,4 >, in`

Parameters

value The color value that the surface will have.

◆ u_v_from_position()

def u_v_from_position (*self*,
location
)

Get the uv coordinates from the supplied Cartesian coordinates on the **Surface** .

.. code-block:: python

```
uv = surface.position_from_u_v([0,0,0])
```

`@n type of location: std::array< double,3 >, in`

Parameters

location A vector containing the Cartesian coordinates.
return type of : std:: array< double,2 >

Returns

The curvature values:

0 The u parameter

1 The v parameter

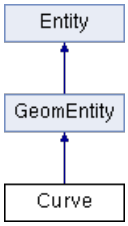
Cubit Python API 17.05

Curve

Public Member Functions

Defines a curve object that mostly parallels Cubit's RefEdge class. More...

Inheritance diagram for Curve:



Public Member Functions

def **arc_center_radius** (self)
Get the center and radius of an arc curve, otherwise returns 0 for non-arc. More...

def **closest_point** (self, point)
Get the curvature of the **Curve** at a particular point. More...

def **closest_point_along_vector** (self, location, along_vector)
Get the nearest point on the **Curve** to point specified along the specified vector. More...

def **closest_point_trimmed** (self, point)
Get the closest location on the **Curve** from a particular point. More...

def **color** (self)
Get the color of the **Curve**. More...

def **curvature** (self, point)
Get the curvature of the **Curve** at a particular point. More...

def **curve_center** (self)
Get the center point of the **Curve**. More...

def **end_param** (self)
Get the highest value of the **Curve** in uv space. More...

def **fraction_from_arc_length** (self, root_vertex, **length**)
Get the fraction along the **Curve** a specified arc length is away from a given **Vertex**. More...

def **is_periodic** (self)
Get whether the **Curve** is periodic or not. More...

def **length** (self)
Get the length of the **Curve**. More...

def **length_from_u** (self, parameter1, parameter2)
Get the length between two specified parameters on a **Curve**. More...

def **point_from_arc_length** (self, root_param, arc_length)
Get the position on a **Curve** that is a specified arc length away from the specified root parameter. More...

def **position_from_fraction** (self, fraction_along_curve)
Get the position of the point a specified fraction along the **Curve**. More...

def **position_from_u** (self, u_value)
Get the position of a particular u value for the **Curve**. More...

def **set_color** (self, value)
Set the color of the **Curve**. More...

def **spline_params** (self)
Retrieve the full set of spline parameters for this curve. More...

def **start_param** (self)
Get the lowest value of the **Curve** in uv space. More...

def **tangent** (self, point)
Get the tangent to the **Curve** at a particular point. More...

def **u_from_arc_length** (self, root_param, arc_length)
Get the u value for a point a specified arc length away from a specified root parameter on the **Curve**. More...

def **u_from_position** (self, position)
Get the u value of a particular position on the **Curve**. More...

▸ **Public Member Functions inherited from GeomEntity**

▸ **Public Member Functions inherited from Entity**

Detailed Description

Defines a curve object that mostly parallels Cubit's RefEdge class.

Member Function Documentation

◆ arc_center_radius()

def arc_center_radius (*self*)

Get the center and radius of an arc curve, otherwise returns 0 for non-arc.

.. code-block:: python

```
center = curve.arc_center_radius()
```

```
@n return type of : std:: array< double,4 >
```

Returns

A vector containing the coordinates of the **Curve**'s center according to the following:

0 x coordinate

1 y coordinate

2 z coordinate

3 radius

◆ closest_point()

def closest_point (*self*,
point
)

Get the curvature of the **Curve** at a particular point.

.. code-block:: python

```
close = curve.closest_point([0,0,0])
```

```
@n type of point:  std::array< double,3 >, in
```

Parameters

point A vector containing 3 doubles representing coordinates of a location on the **Curve**.
return type of : std:: array< double,3 >

Returns

The closest point to the **Curve** from the location specified.

◆ closest_point_along_vector()

def closest_point_along_vector (*self*,
location,
along_vector
)

Get the nearest point on the **Curve** to point specified along the specified vector.

.. code-block:: python

```
nearest =
curve.closest_point_along_vector([0,0,0],
[1,1,1])
```

```
@n type of location:  std::array< double,3 >, in
```

Parameters

location A vector containing three values that are the coordinates of a point.
return type of : std::pair< bool,std:: array< double,3 > >

Returns

A vector (or list) of doubles representing values of nearest point as follows:

0 x coordinate

1 y coordinate

2 z coordinate

◆ closest_point_trimmed()

def closest_point_trimmed (*self*,
point
)

Get the closest location on the **Curve** from a particular point.

.. code-block:: python

```
close = curve.closest_point([0,0,0])
```

```
@n type of point:  std::array< double,3 >, in
```

Parameters

point A vector containing 3 doubles representing coordinates of a location on the **Curve**.
return type of : std:: array< double,3 >

Returns

The closest point to the **Curve** from the location specified

◆ color()

def color (*self*)

Get the color of the **Curve** .

.. code-block:: python

```
col = curve.color()
```

```
@n return type of : std:: array< double,4 >
```

Returns

The color value associated with the curve's current color.

◆ curvature()

def curvature (*self*,
point
)

Get the curvature of the **Curve** at a particular point.

.. code-block:: python

```
curvature = curve.curvature([0,0,0])
```

```
@n type of point:  std::array< double,3 >, in
```

Parameters

point A curvature vector containing 3 doubles. This is related to the 2nd derivative of the curve.
return type of : std:: array< double,3 >

Returns

The curvature of the **Curve** at the location specified

◆ curve_center()

def curve_center (*self*)

Get the center point of the **Curve** .

.. code-block:: python

```
center = curve.curve_center()
```

```
@n return type of : std:: array< double,3 >
```

Returns

A vector containing the coordinates of the **Curve**'s center according to the following:

0 x coordinate

1 y coordinate

2 z coordinate

◆ end_param()

def end_param (*self*)

Get the highest value of the **Curve** in uv space.

.. code-block:: python

```
end = curve.end_param()
```

```
@n return type of :  float
```

Returns

The ending value of the parameter.

◆ fraction_from_arc_length()

def fraction_from_arc_length (*self*,
root_vertex,
length
)

Get the fraction along the **Curve** a specified arc length is away from a given **Vertex** .

.. code-block:: python

```
fraction =
curve.fraction_from_arc_length(vertex, 0.5)
```

```
@n type of root_vertex:  :py:class:`Vertex`, in
```

Parameters

root_vertex The **Vertex** to start from (vertex object)
type of length: float, in

length The length along the **Curve** away from the root **Vertex**.
return type of : float

Returns

The fraction of the **Curve** that is the specified length away from the specified **Vertex**.

◆ is_periodic()

def is_periodic (*self*)

Get whether the **Curve** is periodic or not.

.. code-block:: python

```
periodic = curve.is_periodic()
```

```
@n return type of :  boolean
```

Returns

Whether the **Curve** is periodic or not.

◆ length()

def length (*self*)

Get the length of the **Curve** .

.. code-block:: python

```
len = curve.length()
```

```
@n return type of :  float
```

Returns

The length of the **Curve**.

◆ length_from_u()

def length_from_u (*self*,
parameter1,
parameter2
)

Get the length between two specified parameters on a **Curve** .

.. code-block:: python

```
length = curve.length_from_u(0, 0.5)
```

@n type of parameter1: float, in

Parameters

parameter1 The beginning parameter.
type of parameter2: float, in

parameter2 The ending parameter.
return type of : float

Returns

The length between the two specified paramters along the **Curve**.

◆ point_from_arc_length()

def point_from_arc_length (*self*,
root_param,
arc_length
)

Get the position on a **Curve** that is a specified arc length away from the specified root parameter.

.. code-block:: python

```
position =
curve.point_from_arc_length(0, 0.5)
```

@n type of root_param: float, in

Parameters

root_param The root parameter from which the arc length is added to.
type of arc_length: float, in

arc_length The arc length along the **Curve** away from the root parameter.
return type of : std:: array< double,3 >

Returns

A vector that contains the coordinates of a position a specified arc length away from the root parameter.

◆ position_from_fraction()

def position_from_fraction (*self*,

fraction_along_curve

)

Get the position of the point a specified fraction along the **Curve** .

.. code-block:: python

```
pos = curve.position_from_fraction(0.5)
```

```
@n type of fraction_along_curve:  float, in
```

Parameters

fraction_along_curve A decimal value between 0 and 1 to determine a particular position along the **Curve**.
return type of : std:: array< double,3 >

Returns

A vector containing the coordinates of the position a specified fraction along the **Curve**:

0 x coordinate

1 y coordinate

2 z coordinate

◆ position_from_u()

def position_from_u (*self*,
u_value
)

Get the position of a particular u value for the **Curve** .

.. code-block:: python

```
position = curve.position_from_u(0.5)
```

```
@n type of u_value:  float, in
```

Parameters

u_value The u value of the position along the **Curve**.
return type of : std:: array< double,3 >

Returns

A vector containing the coordinates of the output position

◆ set_color()

def set_color (*self*,
value
)

Set the color of the **Curve** .

.. code-block:: python

```
curve.set_color(0)
```

@n type of value: std::array< double,4 >, in

Parameters

value The color value that the curve will have.

◆ spline_params()

def spline_params (*self*)

Retrieve the full set of spline parameters for this curve.

This method wraps the underlying geometry engine's spline-parameter routine (e.g., ACIS), extracting control point positions, weights, and knot data into a uniform flat vector. It throws **InvalidEntityException** for invalid entities and reports failure on non-spline curves.

Let

N = number of control points, M = number of weights, D = polynomial degree, K = number of knots (should equal D + N - 1). The total vector length S satisfies:

```
S = 6 + 3*N + M + K
```

which expands as: [0] : rational flag (1.0 if rational, 0.0 otherwise) [1] : D (polynomial degree) [2] : N (control points count) [3 ... 3+3*N-1] : control-point coordinates (x, y, z) for each of the N points [3+3*N] : M (weight count) [3+3*N+1 ... 3+3*N+M] : weight values [3+3*N+M+1] : K (knot count) [3+3*N+M+2 ... 3+3*N+M+1+K] : knot values [S-1] : reverse flag (1.0 if reversed, 0.0 otherwise)

.. code-block:: python

```
import cubit


# Build a simple spline curve
(Curve ID = 1)


cubit.cmd
("create vertex 0 0
0" )


cubit.cmd
("create vertex 1 1
0" )


cubit.cmd
("create vertex 2 0
0" )


cubit.cmd
("create curve vertex 1 3 2
spline" )


# Retrieve and unpack spline
```

```
# Retrieve and unpack spline
parameters


spline_curve = cubit.curve
(1)


params = spline_curve.spline_params()


idx = 0


# Rationality


rational = (params[idx] == 1.0); idx += 1


# Degree of Spline


degree = int(params[idx]); idx += 1


# Control Points


n_pts = int(params[idx]); idx += 1


pts = []


for _ in range(n_pts):
pts.append(tuple(params[idx:idx+3])); idx += 3


# Weights


m_w = int(params[idx]); idx += 1


weights = params[idx:idx+m_w]; idx += m_w


# Knots


k_n = int(params[idx]); idx += 1


knots = params[idx:idx+k_n]; idx += k_n


# Reversed Flag


reversed_flag = (params[idx] == 1.0)


# Print nicely


print(f"Rational:
{rational}" )


print(f"Degree:
{degree}" )


print(f"ControlPts({n_pts}):" )
```

for p in pts: print(f" {p}") print(f"Weights({m_w}): {weights}") print {knots}") print {reversed_flag}")

```
                # Expected output:

                # Rational: False

                # Degree: 3

                # ControlPts(5):

                # (0, 0, 0)

                # (0.333333, 0.5, 0)

                # (1, 1.5, 0)

                # (1.666667, 0.5, 0)

                # (2, 0, 0)

                # Weights(0): ()

                # Knots(9): (0, 0, 0, 0, 1.41421,
                2.82843, 2.82843, 2.82843, 2.82843)

                # Reversed: False
```

```
@n return type of : std::vector< double,std:: allocator< double > >
```

Returns

Packed spline parameters as a float sequence (std::vector<double> in C++, tuple in Python).

◆ start_param()

def start_param (*self*)

Get the lowest value of the **Curve** in uv space.

.. code-block:: python

```
                start = curve.start_param()
```

```
@n return type of :  float
```

Returns

The beginning value of the parameter.

◆ tangent()

def tangent (*self*,
point
)

Get the tangent to the **Curve** at a particular point.

.. code-block:: python

```
tan = curve.tangent([0,0,0])
```

```
@n type of point:  std::array< double,3 >, in
```

Parameters

point A vector containing 3 doubles representing coordinates of a location on the **Curve**.
return type of : std:: array< double,3 >

Returns

The tangent to the **Curve** at the location specified

◆ u_from_arc_length()

def u_from_arc_length (*self*,
root_param,
arc_length
)

Get the u value for a point a specified arc length away from a specified root parameter on the **Curve** .

.. code-block:: python

```
u = curve.u_from_arc_length(0, 0.5)
```

```
@n type of root_param:  float, in
```

Parameters

root_param The beginning parameter from which the arc length is added to.
type of arc_length: float, in

arc_length The length away from the root parameter of the output parameter.
return type of : float

Returns

The u value of the **Curve** the arc length away from the root parameter.

◆ u_from_position()

def u_from_position (*self*,
position
)

Get the u value of a particular position on the **Curve** .

.. code-block:: python

```
u = curve.u_from_position([0,0,0])
```

```
@n type of position:  std::array< double,3 >, in
```

Parameters

position A vector containing the coordinates of the input position
return type of : float

Returns

The u value of the position along the **Curve**.

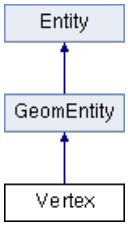
Cubit Python API 17.05

Vertex

Public Member Functions

Defines a vertex object that mostly parallels Cubit's RefVertex class. More...

Inheritance diagram for Vertex:



Public Member Functions

def **color** (self)
Get the color of the **Vertex**. More...

def **coordinates** (self)
Get the Cartesian coordinates of the **Vertex**. More...

def **set_color** (self, value)
Set the color of the **Vertex**. More...

▸ **Public Member Functions inherited from GeomEntity**

▸ **Public Member Functions inherited from Entity**

Detailed Description

Defines a vertex object that mostly parallels Cubit's RefVertex class.

Member Function Documentation

◆ color()

def color (*self*)

Get the color of the **Vertex** .

.. code-block:: python

```
col = vertex.color()
```

```
@n return type of : std:: array< double,4 >
```

Returns
The color value associated with the vertex's current color.

◆ coordinates()

def coordinates (*self*)

Get the Cartesian coordinates of the **Vertex** .

.. code-block:: python

```
position = vertex.coordinates()
```

```
@n return type of : std:: array< double,3 >
```

Returns

A vector containing the coordinates of the **Vertex** with indices corresponding to the coordinates as follows:

0 x coordinate

1 y coordinate

2 z coordinate

◆ set_color()

def set_color (*self*,
value
)

Set the color of the **Vertex** .

.. code-block:: python

```
vertex.set_color(0)
```

```
@n type of value:  std::array< double,4 >, in
```

Parameters

value The color value that the vertex will have.

Cubit Python API 17.05

CubitFailureException

Public Member Functions

An exception class to alert the caller when the underlying Cubit function fails. More...

Public Member Functions

def **what** (self)

Detailed Description

An exception class to alert the caller when the underlying Cubit function fails.

Member Function Documentation

◆ what()

def what (*self*)

InvalidEntityException

Public Member Functions

An exception class to alert the caller that an invalid entity was attempted to be used. More...

Public Member Functions

def **what** (self)

Detailed Description

An exception class to alert the caller that an invalid entity was attempted to be used.

Likely the user is attempting to use an **Entity** who's underlying CubitEntity has been deleted.

Member Function Documentation

what()

def what (*self*)

Cubit Python API 17.05

InvalidInputException

Public Member Functions

An exception class to alert the caller of a function that invalid inputs were entered. More...

Public Member Functions

def **what** (self)

Detailed Description

An exception class to alert the caller of a function that invalid inputs were entered.

Member Function Documentation

◆ what()

def what (*self*)

MeshErrorFeedback

Public Member Functions

Class to implement mesh command feedback processing. More...

Public Member Functions

def **get_entity_id** (self)
def **get_entity_type** (self)
def **get_error_code** (self)
def **get_error_text** (self)
def **set_entity_id** (self, anid)
def **set_entity_type** (self, atype)
def **set_error_code** (self, acode)
def **set_error_text** (self, atext)

Detailed Description

Class to implement mesh command feedback processing.

Member Function Documentation

◆ get_entity_id()

def get_entity_id (*self*)

◆ get_entity_type()

def get_entity_type (*self*)

◆ get_error_code()

def get_error_code (*self*)

◆ get_error_text()

def get_error_text (*self*)

◆ set_entity_id()

def set_entity_id (*self*,
anid
)

)

◆ set_entity_type()

```
def set_entity_type ( self,
                      atype
                    )
```

◆ set_error_code()

```
def set_error_code ( self,
                     acode
                   )
```

◆ set_error_text()

```
def set_error_text ( self,
                     atext
                   )
```

Cubit Python API 17.05

AssemblyItem

Public Member Functions | Properties

Class to implement assembly tree interface. More...

Public Member Functions

def **get_id** (self)
def **get_instance** (self)
def **get_level** (self)
def **get_name** (self)
def **get_path** (self)
def **get_type** (self)
def **set_id** (self, aid)
def **set_instance** (self, ainstance)
def **set_level** (self, alevel)
def **set_name** (self, aname)
def **set_path** (self, apath)
def **set_type** (self, atype)

Properties

volume_id_list = property(_cubit3.AssemblyItem_volume_id_list_get, _cubit3.AssemblyItem_volume_id_list_set)

Detailed Description

Class to implement assembly tree interface.

Member Function Documentation

◆ get_id()

def get_id (*self*)

◆ get_instance()

def get_instance (*self*)

◆ get_level()

def get_level (*self*)

◆ get_name()

```
def get_name ( self )
```

◆ get_path()

```
def get_path ( self )
```

◆ get_type()

```
def get_type ( self )
```

◆ set_id()

```
def set_id ( self,
             aid
           )
```

◆ set_instance()

```
def set_instance ( self,
                   ainstance
                 )
```

◆ set_level()

```
def set_level ( self,
                alevel
              )
```

◆ set_name()

```
def set_name ( self,
               aname
             )
```

◆ set_path()

```
def set_path ( self,
               apath
             )
```

◆ set_type()

```
def set_type (   self,
                 atype
               )
```

Property Documentation

volume_id_list

volume_id_list = property(_cubit3.AssemblyItem_volume_id_list_get, _cubit3.AssemblyItem_volume_id_list_set)

static

CFD_BC_Entity

Public Member Functions

Class to implement cfd bc data retrieval. More...

Public Member Functions

def **get_id** (self)
def **get_name** (self)
def **get_type** (self)
def **set_id** (self, id)
def **set_name** (self, name)
def **set_type** (self, type)

Detailed Description

Class to implement cfd bc data retrieval.

Member Function Documentation

◆ get_id()

```
def get_id ( self )
```

◆ get_name()

```
def get_name ( self )
```

◆ get_type()

```
def get_type ( self )
```

◆ set_id()

```
def set_id ( self,
             id
           )
```

◆ set_name()

```
def set_name (  self,
                name
              )
```

set_type()

```
def set_type (  self,
                type
              )
```

Cubit Python API 17.05

Dir

Public Member Functions | Properties

Defines a direction object. More...

Public Member Functions

def **angle** (self, vec_in)
Returns the interior of two vectors return the angle is in radians. More...

def **cross** (self, vec2)
Returns the cross product of this X vec2 :rtype: :py:class: `Dir`More...

def **dir_print** (self)
Print the output. More...

def **distance** (self, vec_in)
get the distance between two points More...

def **dot** (self, vec2)
Returns the dot product. More...

def **get_xyz** (self)
Get an array of the vector. More...

def **length** (self)
Returns the length. More...

def **normalize** (self)
Normalize 'this' vector :rtype: void. More...

def **orthogonal_vectors** (self)
Finds 2 (arbitrary) vectors that are orthogonal to this one. More...

def **set** (self, **x**, **y**, **z**)
Set the xyz values of the vector. More...

def **set_x** (self, x_in)
Set the x location of the point. More...

def **set_y** (self, y_in)
Set the y location of the point. More...

def **set_z** (self, z_in)
Set the z location of the point. More...

def **x** (self)
Get the x location of the point. More...

def **y** (self)
Get the y location of the point. More...

def **z** (self)
Get the z location of the point. More...

Properties

xVal = property(_cubit3.Dir_xVal_get, _cubit3.Dir_xVal_set)
yVal = property(_cubit3.Dir_yVal_get, _cubit3.Dir_yVal_set)
zVal = property(_cubit3.Dir_zVal_get, _cubit3.Dir_zVal_set)

Detailed Description

Defines a direction object.

Member Function Documentation

◆ angle()

```
def angle ( self,
            vec_in
          )
```

Returns the interior of two vectors return the angle is in radians.

◆ cross()

```
def cross ( self,
            vec2
          )
```

Returns the cross product of this X vec2 :rtype: :py:class: `Dir`

Returns
cross product

◆ dir_print()

```
def dir_print ( self )
```

Print the output.

◆ distance()

```
def distance ( self,
               vec_in
             )
```

get the distance between two points

◆ dot()

```
def dot ( self,
          vec2
        )
```

Returns the dot product.

◆ get_xyz()

def get_xyz (*self*)

Get an array of the vector.

◆ length()

def length (*self*)

Returns the length.

◆ normalize()

def normalize (*self*)

Normalize 'this' vector :rtype: void.

Returns
    void

◆ orthogonal_vectors()

def orthogonal_vectors (*self*)

Finds 2 (arbitrary) vectors that are orthogonal to this one.

◆ set()

def set (*self*,
x,
y,
z
)

Set the xyz values of the vector.

:rtype: void

Returns
    void

◆ set_x()

```
def set_x ( self,
            x_in
          )
```

Set the x location of the point.

:type x_in: float

Parameters
- **x_in** the new x value

◆ set_y()

```
def set_y ( self,
            y_in
          )
```

Set the y location of the point.

:type y_in: float

Parameters
- **y_in** the new y value

◆ set_z()

```
def set_z ( self,
            z_in
          )
```

Set the z location of the point.

:type z_in: float

Parameters
- **z_in** the new z value

◆ x()

```
def x ( self )
```

Get the x location of the point.

:rtype: float

Returns
- x value
- See **y()**, **z()**

◆ y()

def y (*self*)

Get the y location of the point.

:rtype: float

Returns
y value
See **x()**, **z()**

◆ z()

def z (*self*)

Get the z location of the point.

:rtype: float

Returns
z value
See **y()**, **x()**

Property Documentation

◆ xVal

xVal = property(_cubit3.Dir_xVal_get, _cubit3.Dir_xVal_set) static

◆ yVal

yVal = property(_cubit3.Dir_yVal_get, _cubit3.Dir_yVal_set) static

◆ zVal

zVal = property(_cubit3.Dir_zVal_get, _cubit3.Dir_zVal_set) static

Cubit Python API 17.05

Loc

Properties

Defines a location object. More...

Properties

xVal = property(_cubit3.Loc_xVal_get, _cubit3.Loc_xVal_set)
yVal = property(_cubit3.Loc_yVal_get, _cubit3.Loc_yVal_set)
zVal = property(_cubit3.Loc_zVal_get, _cubit3.Loc_zVal_set)

Detailed Description

Defines a location object.

Property Documentation

◆ xVal

xVal = property(_cubit3.Loc_xVal_get, _cubit3.Loc_xVal_set) static

◆ yVal

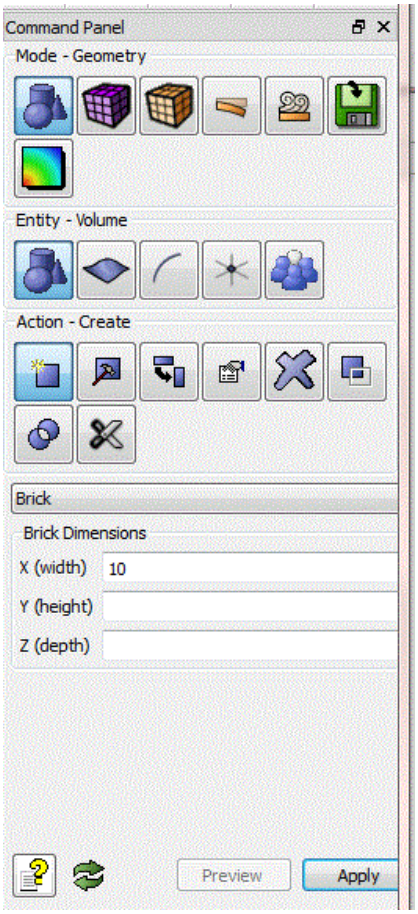
yVal = property(_cubit3.Loc_yVal_get, _cubit3.Loc_yVal_set) static

◆ zVal

zVal = property(_cubit3.Loc_zVal_get, _cubit3.Loc_zVal_set) static

Navigation XML Files

The Cubit GUI includes a section referred to as the Command Panel. It is comprised of a hierarchy of buttons used to navigate to panels that accept user input and generate Cubit command strings. The following example shows the command panel used to create a brick. The user navigates to the command panel by pressing the "**Mode - Geometry**" button, then the "**Entity - Volume**" button, followed by the "**Action - Create**" button, then finally selecting the "**Brick**" option from the pull-down menu.



Before Cubit 14.0, this hierarchy was not modifiable by any third party. With the release of Cubit 14.0, any user can modify the contents of the button hierarchy by adding, deleting, or modifying buttons and command panels. The button hierarchy is expressed in a series of XML files located in the directory 'bin/xml.'

The controlling XML file is named, "CubitNavigationRoot.xml." A snippet from the file is shown below:

```
<?xml version="1.0" encoding="UTF-8"?>
<Navigation version="1.1">
  - <NavigationNode title="Entity" name="Geometry">
        <Label>Geometry</Label>
        <ToolTip>Manage Geometry</ToolTip>
        <Icon url=":/clarogui/images/i32geometry_cl.png"/>
      - <NavigationNode name="Volume">
            <Label>Volume</Label>
            <ToolTip>Manage Volumes</ToolTip>
            <Icon url=":/cubitgui/i32gvolume.png"/>
        </NavigationNode>
      - <NavigationNode name="Surface">
            <Label>Surface</Label>
            <ToolTip>Manage Surfaces</ToolTip>
            <Icon url=":/cubitgui/i32gsurface.png"/>
        </NavigationNode>
      - <NavigationNode name="Curve">
```

The first two levels of the hierarchy are managed in this file. Subsequent levels of the hierarchy are managed in more specific XML files. For example, the remaining hierarchy associated with geometry volumes is managed in the file named, "GeometryVolumeNavigation.xml." A snippet from that file is shown below:

```
<?xml version="1.0" encoding="UTF-8"?>
<Navigation version="1.1">
  - <NavigationReference path="Geometry/Volume">
      - <NavigationNode marker="CreateGeometryVolume" name="Create">
            <Label>Create</Label>
            <ToolTip>Create volumes</ToolTip>
            <Icon url=":/cubitgui/i32gcreate.png"/>
          - <NavigationNode marker="CreateVolumeBrick" name="Brick">
                <Label>Brick</Label>
                <ToolTip>Create a brick</ToolTip>
            </NavigationNode>
          - <NavigationNode marker="CreateVolumeCone" name="Cone">
                <Label>Cone</Label>
                <ToolTip>Create a cone</ToolTip>
            </NavigationNode>
```

Users may modify the **Label**, **ToolTip**, or **Icon url**. Users may remove entire categories if necessary. Users should not modify **NavigationNode** or **NavigationReference** tags.

Users may create their own command panels using Qt and add them to the hierarchy.

Periodic Space Filling Models (Tile) 

This appendix describes commands for producing good-quality meshes of models that tile space, such as polycrystalline materials models. Such models are often referred to as "periodic", but since that term already has a different meaning in Cubit, the keyword "tile" is used instead. Meshes may be smoothed across periodic boundaries. Periodic boundary conditions can be automatically set up, according to ALEGRA conventions (SAND99-2698).

Tile commands are alpha features and should be used with caution.

Initial setup

First import the model and merge the surfaces. Then mesh it with any method that will create meshes that match across the tile (periodic) boundary, say with scheme polyhedron or sweep. Once the mesh is created, specify the "tile vectors", which lets Cubit know that the nodes across the periodic boundaries are actually the same node:

```
Tile {x <period> | y <period> | z <period>}
[x <period>] [y <period>] [z <period>]
```

The 'period' you specify is actually the vector offset from one boundary to its match. Specify one tile command for each coordinate axis that the model is periodic in. E.g.

```
Tile x 1
Tile y 1
Tile z 1
```

You can see which nodes are matched to a given node by some combination of tile vectors with the following command: Tile Debug Node <id>

If you later need to delete these tile vectors, use the following command:

```
Tile Off
```

Creating Nodesets

Once the tile vectors are specified, you can set up periodic boundary conditions that meet ALEGRA specifications. The command is:

```
Tile Nodeset <start_id>
```

This will create a nodeset for all combinations of tile vectors that actually connect nodes. The nodesets created will be reported to you. The nodesets will be consecutive starting with the given 'start_id', except that if there are no nodes for a particular combination there will be no nodeset and the id space will have a hole. To delete these nodesets, use the

```
Tile Off
```

command rather than the usual commands to delete nodesets.

Smoothing

Once a mesh has been created and the tile vectors have been specified, you can smooth the mesh and keep the periodic boundaries exactly offset by the tile vectors. Only hex meshes are currently supported. A variety of 3d smoothing schemes are supported, including laplacian, equipotential,

untangle, and condition number.

```
Smooth Volume <volume_id_range> [Global [Float
<dim>] ]
```

Use "Global" if you are smoothing a collection of volumes. Use "float 3" if you want nodes on surfaces, curves, and vertices to be able to move off of their geometric owner. Use "float 2" if you want just nodes on curves and vertices to be able to move off of their owner (but stay on an owning surface). It is often useful to specify that some of the nodes are fixed using the "node position fixed" command.

Example

```
# make the geometry
#{brick_size=500}
brick wid {brick_size}
brick wid {brick_size}
body 2 move {brick_size} 0 0
brick wid {brick_size}
body 3 move {brick_size} {brick_size} 0
brick wid {brick_size}
body 4 move 0 {brick_size} 0
brick wid {brick_size}
body 5 move 0 0 {brick_size}
brick wid {brick_size}
body 6 move {brick_size} 0 {brick_size}
brick wid {brick_size}
body 7 move {brick_size} {brick_size}
{brick_size}
brick wid {brick_size}
body 8 move 0 {brick_size} {brick_size}
merge all

# mesh it
vol all int 3
mesh vol all

# set the tiling vectors
tile x {brick_size*2}
tile y {brick_size*2}
tile z {brick_size*2}
tile debug node 256
tile debug node 245

# set the tiling nodesets
tile nodeset

# mess up the mesh quality
# volume all smooth scheme randomize
# smooth volume all
surface all smooth scheme randomize
smooth surface all
draw hex all

# fix the mesh quality
node in volume all position fixed
node in surface all position free
volume all smooth scheme laplac
# volume all smooth scheme untangle beta 0.08
smooth volume all global float 3
draw hex all
```

References

Attaway, Stephen W.; Mello, Frank J.; Heinstein, Martin W.; Swegle, Jeffrey W.; Ratner, Julie A.; Zadoks, Rick Ian, "PRONTO3D users' instructions: a transient dynamic code for nonlinear structural analysis," Sandia Report SAND 98-1361 Sandia National Laboratories, Albuquerque, NM (1998)

Attaway S. W., unpublished, (1993)

Blacker, T. D., FASTQ Users Manual Version 1.2, SAND88-1326, Sandia National Laboratories, (1988)

Blacker, Ted D. "An Adaptive Finite Element Technique Using Element Equilibrium and Paving", American Society of Mechanical Engineers, Annual Meeting Dallas Texas, November 25-30, 1990, ASME, Nov 1990

Blacker, Ted D., "Paving: A New Approach To Automated Quadrilateral Mesh Generation", International Journal For Numerical Methods in Engineering, John Wiley, Num 32, pp.811-847, 1991

Blacker T.D. and Meyers R.J,."Seams and Wedges in Plastering: A 3D Hexahedral Mesh Generation Algorithm", Engineering with Computers, Springer Verlag, Vol 2, Num 9, pp.83-93, 1993

Brewer, M., L. Diachin, P. Knupp, T. Leurent, and D. Melander, "The Mesquite Mesh Quality Improvement Toolkit", Proceedings, 12th International Meshing Roundtable, 2003

Brewer, M., "Geometry-Tolerant Meshing Using Advancing-Front Techniques", SAND Report, (6-2008)

Butlin, Geoffrey and Clive Stops, "CAD Data Repair", 5th International Meshing Roundtable, pp.7-12, 1996

Clark Brett W., "Removing Small Features with Real Solid Modeling Operations", Submitted to 16th International Meshing Roundtable, 2007

Cook, W. A. and W. R. Oakes (1982) Mapping methods for generating threedimensional meshes, Computers In Mechanical Engineering, CIME Research Supplement:67-72, August 1982

Folwell, Nathan T. and Scott A. Mitchell, "Reliable Whisker Weaving via Curve Contraction", Proceedings, 7th International Meshing Roundtable, Sandia National Lab, pp.365-378, October 1998

Freitag, Lori A. and Patrick M. Knupp , "Tetrahedral Element Shape Optimization via the Jacobian Determinant and Condition Number", Proceedings, 8th International Meshing Roundtable, South Lake Tahoe, CA, U.S.A., pp.247-258, October 1999

George, P.L., F. Hecht and E. Saltel, "Automatic Mesh Generator with Specified Boundary", Computer Methods in Applied Mechanics and Engineering, Vol. 92, pp. 269-288, 1991

Hardwick, Mike, "DART System Analysis Presented to Simulation Sciences Seminar", June 28, 2005

Jones, R.E., QMESH: A Self-Organizing Mesh Generation Program, SLA - 73 - 1088, Sandia National Laboratories, (1974).

Knupp, Patrick M., "Winslow Smoothing On Two-Dimensional Unstructured Meshes", Proceedings, 7th International Meshing Roundtable, Sandia National Lab, pp.449-457, October 1998

Knupp, Patrick M., "Matrix Norms & The Condition Number: A General Framework to Improve Mesh Quality Via Node-Movement", Proceedings, 8th International Meshing Roundtable, South Lake Tahoe, CA, U.S.A.

8th International Meshing Roundtable, South Lake Tahoe, CA, U.S.A.,

pp.13-22, October 1999

Knupp, P., "Achieving Finite Element Mesh Quality via Optimization of the Jacobian Matrix Norm and Associated Quantities, Part I", Int. J. Num. Meth. Engr.. 2000

Lovejoy, S. C. and R. G. Whirley, DYNA3D Example Problem Manual, UCRL-MA--105259, University Of California and Lawrence Livermore National Laboratory, (1990).

Melander, Darryl J., Timothy J. Tautges, Steven E. Benzley "Generation of Multi-Million Element Meshes for Solid Model-Based Geometries: The Dicer Algorithm" AMD-Vol. 220 Trends in Unstructured Mesh Generation, ASME, pp.131-135, July 1997

Mezentsev, Andrey A., "Methods and Algorithms of Automated CAD Repair For Incremental Surface Meshing", Proceedings, 8th International Meshing Roundtable, pp.299-309, 1999

Murdoch, Peter and Steven E. Benzley, "The Spatial Twist Continuum", Proceedings, 4th International Meshing Roundtable, Sandia National Laboratories, pp.243-251, October 1995

Oddy, A., J. Goldak, M. McDill, and M. Bibby "A Distortion Metric for Isoparametric Finite Elements" Transactions of the Canadian Soc. Mech. Engr., pp213-217, Vol 12, No 4, 1988.

Owen, Steven J. and David R. White, "Mesh-Based Geometry: A Systematic Approach to Constructing Geometry from the Nodes and Elements of a Finite Element Mesh", 10th International Meshing Roundtable, Sandia National Laboratories, pp. 83-96, October 2001

Owen, Steven J., Clark, B.W., Melander, D.J., Brewer, M.B., Shepherd, J.F., Merkley, K., Ernst, C., Morris, R., "An Immersive Topology Environment for Meshing", Accepted to 16th International Meshing Roundtable, 2007

Parthasarathy V. N. et al, "A comparison of tetrahedron quality measures", Finite Elem. Anal. Des., Vol 15, 1993, 255-261.

Price, M.A. and C.G. Armstrong, "Hexahedral Mesh Generation by Medial Surface Subdivision: Part I, Solids With Convex Edges, International Journal for Numerical Methods in Engineering, Vol. 38, No. 19, pp. 3335-3359, 1995

W. Quadros, V. Vyas, M. Brewer, S. Owen, and K. Shimada, “A Computational Framework for Generating Sizing Function in Assembly Meshing”, *Proceedings*, *14 th International Meshing Roundtable*, 2005

W. R. Quadros, K. Shimada, and S. J. Owen, “Skeleton-based computational method for the generation of a 3D finite element mesh sizing function”, *Engineering with Computers*, Springer Verlag, Vol 20, Num 3, pp.249-264, 2004

W. R. Quadros, S. J. Owen, M. Brewer, and K. Shimada, “Finite Element Mesh Sizing for Surfaces using Skeleton”, *Proceedings*, *13 th International Meshing Roundtable*, 2004

Robinson, J., "CRE method of element testing and Jacobian shape parameters, Eng. Comput., Vol. 4 (1987).

Ruppert, Jim , "A New and Simple Algorithm for Quality 2-Dimensional Mesh Generation". Technical Report UCB/CSD 92/694, University of California at Berkely, Berkely California (1992)

Scott, Michael A., Matthew N. Earp, Steven E. Benzley, and Michael B. Stephenson, "Adaptive Sweeping Techniques," Proceedings of the 14th International Meshing Roundtable, Springer, pp. 417-432, 2005.

Schoof, L. A.and Victor R. Yarberry, "EXODUS II A Finite Element Data Model", SAND92-2137, Sandia National Laboratories, (1995).

Sheffer, A., "Model simplification for meshing using face clustering", Computer-Aided Design, Vol. 33, No. 13, pp. 925-934(10), 2001

Staten, Matthew L., Steven J. Owen, Ted D. Blacker, "Unconstrained Paving and Plastering: A New Idea for All Hexahedral Mesh Generation", Proceedings, 14th International Meshing Roundtable, pp.399-416, 2005

Staten, Matthew L., Robert A. Kerr, Steven J. Owen, Ted D. Blacker, "Unconstrained Paving and Plastering: Progress Update", Proceedings, 15th International Meshing Roundtable, pp.469-486, 2006

Staten, Matthew L., Brian Carnes, Corey McBride, Clint Stimpson, Jim Cox, "Mesh Scaling for Affordable Solution Verification", Proceedings, 25th International Meshing Roundtable, pp. 46-58, 2016

Stimpson, CJ, Ernst, CD, Knupp, P, Pebay; P, and Thompson, D. "The Verdict Geometric Quality Library", Sandia Report SAND2007-175, 2007

Tautges, Timothy J. and Scott A. Mitchell, "Whisker Weaving: Invalid Connectivity Resolution and Primal Construction Algorithm", Proceedings, 4th International Meshing Roundtable, Sandia National Laboratories, pp.115-127, October 1995

Tautges, Timothy J., Ted Blacker, Scott A. Mitchell, "The Whisker Weaving Algorithm: A Connectivity-Based Method for Constructing All-Hexahedral Finite Element Meshes", International Journal for Numerical Methods in Engineering, Wiley, Vol 39, pp.3327-3349, 1996

Tautges, Timothy J., "The Common Geometry Module (CGM): A Generic, Extensible Geometry Interface", Proceedings, 9th International Meshing Roundtable, pp. 337-348, 2000

Tautges, Timothy J., "Automatic Detail Reduction for Mesh Generation Applications", Proceedings, 10th International Meshing Roundtable, pp.407-418, 2001

Taylor, L. M. and D. P. Flanagan, "Pronto 3D--A Three-Dimensional Transient Solid Dynamics Program", SAND87-1912, Sandia National Laboratories, (1989).

Tipton ,R. E., "Grid Optimization by Equipotential Relaxation", unpublished, Lawrence Livermore National Laboratory, (1990)

Walton, D. J. and D. S. Meek, "A Triangular G1 Patch from Boundary Curves," Computer-Aided Design, Vol. 28 No. 2 pp. 113-123 (1996)

Watson, David F. , "Computing the Delaunay Tessellation with Application to Voronoi Polytopes", The Computer Journal, Vol 24(2) pp.167-172 (1981)

Wellman, Gerald W., "MAPVAR : a computer program to transfer solution data between finite element meshes", Sandia Report SAND 99-0466 Sandia National Laboratories, Albuquerque, NM (1999)

White, David R. and Paul Kinney, "Redesign of the Paving Algorithm: Robustness Enhancements through Element by Element Meshing", Proceedings, 6th International Meshing Roundtable, Sandia National Laboratories, pp.323-335, October 1997

White, David R. and Sunil Saigal (2002) Improved Imprint and Merge for Conformal Meshing, Proceedings, 11th International Meshing Roundtable, pp.285-296

White, David R. and Timothy J. Tautges, "Automatic Scheme Selection for Toolkit Hex Meshing", International Journal for Numerical Methods in Engineering, Vol. 49, No. 1, pp. 127-144, 2000

Engineering, Vol. 15, No. 4, pp. 427-444, 2000

Whiteley, M., D. White, S. Benzley and T. Blacker, "Two and Three-Quarter Dimensional Meshing Facilitators", Engineering with Computers, Springer-Verlag, Vol 12, pp.155-167, December 1996

Yong Lu, Rajit Gadh, and Timothy J. Tautges, "Volume decomposition and feature recognition for hexahedral mesh generation", Proceedings, 8th International Meshing Roundtable, pp. 269-280, 1999

Available Colors

In addition to color specification by RGB values, all color commands in CUBIT allow the specification of a color name. The following table lists the colors available by ID or name in CUBIT. The table lists the color number (#), color name, and the red, green, and blue components corresponding to each color, for reference.

Number	Color Name	Red	Green	Blue
0	black	0.000	0.000	0.000
1	grey	0.500	0.500	0.500
2	orange	0.000	1.000	0.000
3	red	1.000	1.000	0.000
4	green	1.000	0.000	0.000
5	yellow	1.000	0.000	1.000
6	magenta	0.000	1.000	1.000
7	cyan	0.000	0.000	1.000
8	blue	1.000	1.000	1.000
9	white	1.000	0.647	0.000
10	brown	0.647	0.165	0.165
11	gold	1.000	0.843	0.000
12	lightblue	0.678	0.847	0.902
13	lightgreen	0.000	0.800	0.000
14	salmon	0.980	0.502	0.447
15	coral	1.000	0.498	0.314
16	pink	1.000	0.753	0.796

17	purple	0.627	0.125	0.941
18	paleturquoise	0.686	0.933	0.933
19	lightsalmon	1.000	0.627	0.478
20	springgreen	0.000	1.000	0.498
21	slateblue	0.416	0.353	0.804
22	sienna	0.627	0.322	0.176
23	seagreen	0.180	0.545	0.341
24	deepskyblue	0.000	0.749	1.000
25	khaki	0.941	0.902	0.549
26	lightskyblue	0.529	0.808	0.980
27	turquoise	0.251	0.878	0.816
28	greenyellow	0.678	1.000	0.184
29	powderblue	0.690	0.878	0.902
30	mediumturquoise	0.282	0.820	0.800
31	skyblue	0.529	0.808	0.922
32	tomato	1.000	0.388	0.278
33	lightcyan	0.878	1.000	1.000
34	dodgerblue	0.118	0.565	1.000
35	aquamarine	0.498	1.000	0.831
36	lightgoldenrodyellow	0.980	0.980	0.824
37	darkgreen	0.000	0.392	0.000

38	lightcoral	0.941	0.502	0.502
39	mediumslateblue	0.482	0.408	0.933
40	lightseagreen	0.125	0.698	0.667
41	goldenrod	0.855	0.647	0.125
42	indianred	0.804	0.361	0.361
43	mediumspringgreen	0.000	0.980	0.604
44	darkturquoise	0.000	0.808	0.820
45	yellowgreen	0.604	0.804	0.196
46	chocolate	0.824	0.412	0.118
47	steelblue	0.275	0.510	0.706
48	burlywood	0.871	0.722	0.529
49	hotpink	1.000	0.412	0.706
50	saddlebrown	0.545	0.271	0.075
51	violet	0.933	0.510	0.933
52	tan	0.824	0.706	0.549
53	mediumseagreen	0.235	0.702	0.443
54	thistle	0.847	0.749	0.847
55	palegoldenrod	0.933	0.910	0.667
56	firebrick	0.698	0.133	0.133
57	palegreen	0.596	0.984	0.596
58	lightyellow	1.000	1.000	0.878
59	darksalmon	0.914	0.588	0.478

60	orangered	1.000	0.271	0.000
61	palevioletred	0.859	0.439	0.576
62	limegreen	0.196	0.804	0.196
63	mediumblue	0.000	0.000	0.804
64	blueviolet	0.541	0.169	0.886
65	deeppink	1.000	0.078	0.576
66	beige	0.961	0.961	0.863
67	royalblue	0.255	0.412	0.882
68	darkkhaki	0.741	0.718	0.420
69	lawngreen	0.486	0.988	0.000
70	lightgoldenrod	0.933	0.867	0.510
71	plum	0.867	0.627	0.867
72	sandybrown	0.957	0.643	0.376
73	lightslateblue	0.518	0.439	1.000
74	orchid	0.855	0.439	0.839
75	cadetblue	0.373	0.620	0.627
76	peru	0.804	0.522	0.247
77	olivedrab	0.420	0.557	0.137
78	mediumpurple	0.576	0.439	0.859
79	maroon	0.690	0.188	0.376
80	lightpink	1.000	0.714	0.757
81	darkslateblue	0.282	0.239	0.545

82	rosybrown	0.737	0.561	0.561
83	mediumvioletred	0.780	0.082	0.522
84	lightsteelblue	0.690	0.769	0.871
85	mediumaquamarine	0.400	0.804	0.667
86	proe_background	0.000	0.000	0.300
87	win2kgray	0.831	0.815	0.784
88	proe_2001_top	0.000	0.365	0.608
89	proe_2001_bottom	0.000	0.067	0.196
90	wildfire_top	0.620	0.608	0.569
91	wildfire_bottom	0.890	0.882	0.835

Element Numbering

This appendix describes the element node and side numbering conventions used in Exodus II files written by CUBIT. This information is located here for convenience, but is identical to the information presented in the Exodus II manual; citation Schoof, 95

Node Numbering

The node numbering used for the basic elements is shown Figure 1. Specific element types of lower order just contain the number of nodes needed for those elements; for example, QUAD4 or QUAD elements use just the first four nodes shown for quadrilaterals in Figure 1.

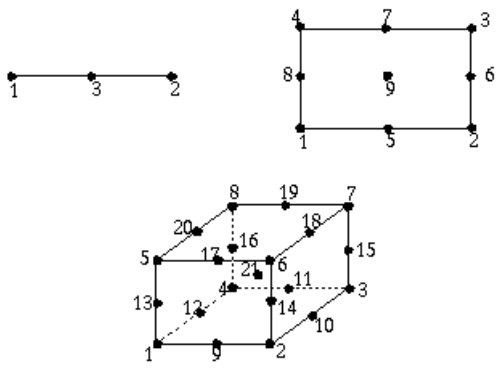


Figure 1. Local Node Numbering for CUBIT element types

Side Numbering

Element sides are used to specify boundary conditions that act over a length or area, for example pressure- or flux-type boundary conditions. Each element side is represented in the Exodus II format by an element number and the local side number for that element. The local side numbering for the basic elements is shown in Figure 2.

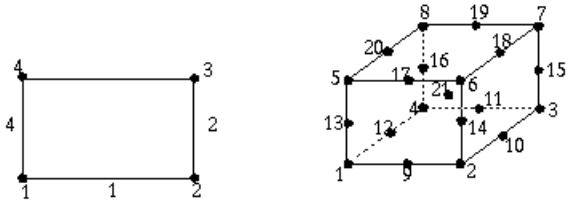


Figure 2. Local side numbering for CUBIT element types

Triangular Shell Element Numbering

A three-dimensional shell element with triangular topology will have the element type 'TRISHELL'. This type can be modified for different element orders by appending the number of nodes onto the end of the type. For example, a 6-node shell could have the element type 'TRISHELL6'. However, any element whose type begins with the 8 letters 'TRISHELL' in upper, lower, or mixed case will refer to an element with a triangular topology. The element can exist in either three-space or two-space.

Attributes:
1. If the element exists in two-space, there are no required attributes.

2. If the element exists in three-space, there is one required attribute which is the thickness of the shell.

3. If the number of attributes is equal to the number of nodes in the connectivity of the element, then the attributes are assumed to specify the thickness of the element at each of the elements nodes. The ordering of the attributes matches the ordering of the elements nodes.

Node Ordering

The node ordering of the 3D triangle matches the node ordering of the 2D triangle as shown in Figure 3.

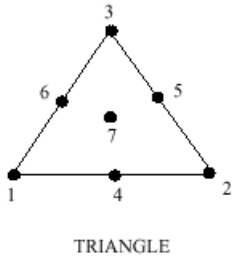


Figure 3. Local Node Numbering for CUBIT triangular element types

Side Set Side Ordering

The sideset side ordering is different for the element in the 2D and 3D instances.

In 2D, the sideset side ordering matches what is shown in Figure 4.

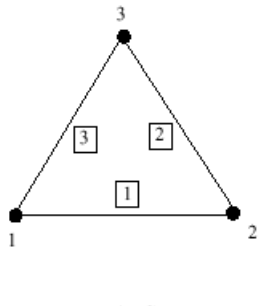


Figure 4. Local sideset numbering for CUBIT triangular element types

In 3D, the sideset side and node ordering is the same as for a quad shell except that there are only 3 or 6 nodes.

Then:

side 1 == {1,2,3}
side 2 == {3,2,1}
side 3 == {1,2}
side 4 == {2,3}
side 5 == {3,1}

If it is a higher order triangular shell (6 [or 7 nodes]), then the higher-order nodes are added on to the end of the above:

side 1 == {1,2,3,4,5,6[,7]}
side 2 == {3,2,1,6,5,4[,7]}
side 3 == {1,2,4}
side 4 == {2,3,5}
side 5 == {3,1,6}

FASTQ

FASTQ is a program developed to create geometry and two-dimensional mesh. The user may choose to upload FASTQ files and work with the files in an environment that accepts a limited number of FASTQ commands.

Table 1. FASTQ Commands Executable in Cubit

Syntax	Description
set fastq on	Cubit is in FASTQ mode.
set fastq off	Cubit exits FASTQ mode.
nine	Mesh will be generated using nine-node quadrilateral elements.
eight	Mesh will be generated using eight-node quadrilateral elements.
five	Mesh will be generated using five-node quadrilateral elements.
import fastq " *.fsq "	Imports FASTQ files into Cubit.

Table 2. Brief List of Importable FASTQ Commands Supported in Cubit

Syntax	Description
point <point_id> <x-coord> <y-coord> [<z-coord>]	This creates a point at the specified coordinates with the id given by the user. The z-coordinate is optional because FASTQ is a two-dimensional meshing tool.
line <line_id> str <begin_pt> <end_pt> 0 [interval] [factor]	This creates a straight line with the given beginning and end points and an id is assigned to the line. The interval option determines the number of intervals or subdivisions of the line for mesh generation. The factor option is the ratio of the interval lengths as the intervals progress towards the end point of the line. For example, if a factor of 2 is specified, each interval will be 2 times longer than the interval before it. If a factor is not specified, the default factor is 1.
line <line_id> circ <begin_pt> <end_pt> <center_pt> [interval] [factor]	The command creates a circular arc (or logarithmic spiral) about a center point. The beginning and ending points specify where to position the circular arc. The third point in the command specifies the center of the circular arc. Interval and factor are defined in the explanation for the Line (STR) Command.

line <line_id> cirm <begin_pt> <end_pt> <center_pt> [interval] [factor]	The CIRM line is similar to the CIRC line. The difference between the CIRM line and the CIRC line is the function of the third point. The third point on a CIRM line is between the beginning and end points and becomes a part of the circular arc. The arc will be drawn through all three points.
line <line_id> cirr <begin_pt> <end_pt> <center_pt> [interval] [factor]	The command creates a circular arc. The beginning and end points function the same as the other commands to create a circular arc, but the third point is used differently. The x value of the third point will be used as the radius of the arc to be created. If the x value is positive, the center point is placed on the left of a straight line drawn through the beginning and end points. If the x value is negative, the center is placed on the right side of the line.
line <line_id> para <begin_pt> <end_pt> <center_pt> [interval] [factor]	This command creates the tip of a parabolic arc. The third point is the peak of the parabola. The beginning and end points must be equidistant from the third point.
line <line_id> corn <begin_pt> <end_pt> <center_pt> [interval] [factor]	The command creates a corner formed by two line segments. The first segment is created by connecting the first and third points. The second segment is created by connecting the third and second points. The line segments can have their interval size set as if the two lines were one.
side <side_id> <list_of_lines>	This creates a group made up of the given lines and assigns the id given by the user.
region <region_id> <block_id> <list_of_lines_or_sides>	A region is a list of lines/sides that enclose an area to be meshed. The region is formed from the list of lines and/or sides; the region is given the id specified by the user.
barset <barset_id> <block_id> <inside> <list_of_lines>	The basis for two and three node element generation is the barset. The barset id is the identifying number for the barset. The block id is the id assigned to all elements in the barset. The inside point is a point on the inside of all lines in the barset. All lines specified at the end of the command will be included in the barset.
interval <interval> <list_of_lines>	This sets the number of intervals on a given line or lines.
factor <factor> <list_of_lines>	This command sets the ratio of the interval lengths as the intervals progress towards the end point of the line. For example, if a factor of 2 is specified, each interval will be 2 times longer than the interval before it. If a factor is not specified, the default factor is 1.

poinbc <node_bc_id> <list_of_points>	This command attaches boundary conditions to the nodes that are created at point locations.The first number to be entered is the id of the flag. After that a list of all points to be flagged is entered.
linebc <node_bc_id> <list_of_lines>	This command attaches boundary conditions to nodes created along certain lines. The first number entered is the id of the flag. Following the id, all lines to be flagged should be entered.
sidebc <side_bc_id> <list_of_lines>	This command attaches boundary conditions to all nodes created on certain lines. The first number entered is the id of the flag. All numbers entered after that point are the ids of the sidesets included in the flag.
scheme <region_id> {m|t|b|c|u}	The letters after the region id indicate the meshing scheme. Schemes specify a meshing algorithm for mesh generation is a regionThe letter 'm' indicates a general rectangle primitive, 't' indicates a triangle primitive, 'b' indicates a transition primitive, 'c' indicates a semicircle primitive, and 'u' indicates a pentagon primitive.

FullHex vs. NodeHex Representation

CUBIT has two different internal representations of hexes: FullHexes and NodeHexes. The NodeHex is a lighter weight data structure, but occasionally nodeset and sideset shortcomings can be overcome by using FullHexes. The user can select which type of hexes get created when generating or importing a volume mesh with the following command:

```
Set FullHex [Use] [on|OFF]
```

Using the FullHex representation increases the memory used to store a mesh by a factor of approximately five.

Generating a Finite Element Mesh from Level-set Data

This documentation will describe how to generate a finite element mesh from an Exodus file that contains level-set data defined as nodal variables. This process was developed to support mesh generation for geometric designs resulting from Adaptive Topological Optimization (ATO). The output format from ATO in this case is an Exodus file containing a tetrahedral mesh with a scalar nodal variable defining a level-set that represents the bounding surfaces of the optimized volume. The process below extracts the bounding surfaces of the optimized volume by evaluating the level-set at a value of zero. These bounding surfaces are in the form of a triangulation that can then be used for generating a finite element mesh. Three methods for generating the finite element mesh will be described using a simple example model.

Hex Mesh Generation Using Sculpt

This section will describe the process for generating a sculpted hex mesh of the ATO design.

1. This process utilizes beta capabilities in Cubit so activate the use of beta commands.

set dev on

2. Import the ATO Exodus file called **"small_bracket.exo"** into Cubit and use the import option that tells Cubit to import the level-set nodal variable called "LSD" (level set data).

import mesh "small_bracket.exo" nodal_var "LSD" no_geom

3. Extract the boundary surface triangulations from the level-set data. Because this creates triangles we use a variation of the "create tri ..." command. We will use the "iso" option telling the command to generate the iso-surfaces from the level-set data defined by the nodal variable "LSD". We will specify the tets to consider when doing the extraction. In this case we use "tet all" meaning all of the tets in the model. When this command is complete there will be two new blocks defined in Cubit. They will be the two blocks with the highest IDs. One of these blocks (the larger of the two block IDs) contains triangles that represent the optimized portions of the design and the other block (the smaller of the two block IDs) contains triangles on the fixed portions of the design.

create tri iso tet all nodal_var "LSD"

4. Look at the extracted iso-surface triangulations by drawing the two new blocks (blocks 3 & 4) created in step 2.

draw block 3 4

5. At this point we could smooth the new triangulations if desired but because the sculpting process will have the same effect we will skip that step here. The smoothing will be demonstrated in the next sections. We will now export the new blocks to an STL file that can then be used by the sculpt algorithm. Specify only the triangles in the new blocks. Also specify the "mesh free_mesh" options to tell the command that the triangles are not owned by geometry.

export stl "small_bracket.stl" tri in block 3 4 mesh free_mesh

6. Exit Cubit and start a Linux command prompt from which to run sculpt.
7. From the command prompt load the sierra module (this will be used later).

module load sierra

8. From the directory where your new stl file is launch the sculpt program. See the sculpt documentation for more details on the options for running sculpt. Here is an example of a simple sculpt command that specifies the sculpt cell size and the number of processors. The number or processors is 8 and the cell size is 0.0007.

sculpt –j 8 –cs 0.1 --stl_file "small_bracket.stl"

9. When sculpt finishes the resulting mesh will be spread across 8 files in our case since we used 8 processors. In our example the files will be named something like "small_bracket.stl_results.e.8.0" where the last number in the filename refers to which processor the file came from. To concatenate all of the files into one use the "epu" command from the Sierra suite of tools.

epu –auto small_bracket.stl_results.e.8.0

10. Start Cubit up and load the mesh file created by sculpt.

import mesh "small_bracket.stl_results.e" no_geom

Tet Mesh Generation by Remeshing Mesh Based Geometry (MBG)

This section will describe the process for generating a tet mesh by meshing a mesh based geometry (MBG) representation of the optimized part.

1. This process utilizes beta capabilities in Cubit so activate the use of beta commands.

set dev on

2. Import the ATO Exodus file called **"small_bracket.exo"** into Cubit and use the import option that tells Cubit to import the level-set nodal variable called "LSD" (level set data).

import mesh "small_bracket.exo" nodal_var "LSD" no_geom

3. Extract the boundary surface triangulations from the level-set data. Because this creates triangles we use a variation of the "create tri …" command. We will use the "iso" option telling the command to generate the iso-surfaces from the level-set data defined by the nodal variable "LSD". We will specify the tets to consider when doing the extraction. In this case we use "tet all" meaning all of the tets in the model. When this command is complete there will be two new blocks defined in Cubit. They will be the two blocks with the highest IDs. One of these blocks (the larger of the two block IDs) contains triangles that represent the optimized portions of the design and the other block (the smaller of the two block IDs) contains triangles on the fixed portions of the design.

create tri iso tet all nodal_var "LSD"

4. Look at the extracted iso-surface triangulations by drawing the two new blocks (blocks 3 & 4) created in step 2.

draw block 3 4

5. Export the new blocks to an Exodus file so that they are disconnected from the original tet mesh.

export mesh "small_bracket_iso.e" block 3 4

6. Reset Cubit.

reset

7. Load the new file that just contains the new blocks.

import mesh "small_bracket_iso.e" no_geom

8. Smooth the optimized part of the triangulation. This will be the triangles in the block with the larger id (4 in our case). We will do this by first fixing the node positions of all of the nodes in the non-optimized part of the triangulation and then smoothing the triangles in the optimized portion. When smoothing we need to use the "target free mesh" option to tell the command to project the smoothed results back to the original triangulation to try to preserve volume. We use the "iteration" option to limit the number of iterations the smoother does.

node in tri in block 3 position fixed

smooth tri in block 4 target free mesh iteration 5

9. Create mesh based geometry from the triangulation. For small models this can be done immediately with the commands below (first command creates surfaces and the second command stitches them together to form a closed volume). For larger models it may be faster to export the mesh to a file, reset Cubit, and then re-import the mesh with the "geom" option on so that mesh based geometry is generated on import. Current limitations in Cubit result in this performance difference.

create mesh geom tri all

create vol surf all

10. Set the size and scheme on the new volume and mesh it.

volume all size .5

volume all scheme tetmesh

mesh volume all

Tet Mesh Generation Using Level-set Triangulation

This section will describe the process for generating a tet mesh using the triangulation from the level-set extraction.

1. This process utilizes beta capabilities in Cubit so activate the use of beta commands.

set dev on

2. Import the ATO Exodus file called **"small_bracket.exo"** into Cubit and use the import option that tells Cubit to import the level-set nodal variable called "LSD" (level set data).

import mesh "small_bracket.exo" nodal_var "LSD" no_geom

3. Extract the boundary surface triangulations from the level-set data. Because this creates triangles we use a variation of the "create tri …" command. We will use the "iso" option telling the command to generate the iso-surfaces from the level-set data defined by the nodal variable "LSD". We will specify the tets to consider when doing the extraction. In this case we use "tet all" meaning all of the tets in the model. When this command is complete there will be two new blocks defined in Cubit. They will be the two blocks with the highest IDs. One of these blocks (the larger of the two block IDs) contains triangles that represent the optimized portions of the design and the other block (the smaller of the two block IDs) contains triangles on the fixed portions of the design.

create tri iso tet all nodal_var "LSD"

4. Look at the extracted iso-surface triangulations by drawing the two new blocks (blocks 3 & 4) created in step 2.

draw block 3 4

5. Export the new blocks to an Exodus file so that they are disconnected from the original tet mesh.

export mesh "small_bracket_iso.e" block 3 4

6. Reset Cubit.

reset

7. Load the new file that just contains the new blocks.

import mesh "small_bracket_iso.e" no_geom

8. Smooth the optimized part of the triangulation. This will be the triangles in the block with the larger id (4 in our case). We will do this by first fixing the node positions of all of the nodes in the non-optimized part of the triangulation and then smoothing the triangles in the optimized portion. When smoothing we need to use the "target free mesh" option to tell the command to project the smoothed results back to the original triangulation to try to preserve volume. We use the "iteration" option to limit the number of iterations the smoother does.

node in tri in block 3 position fixed

smooth tri in block 4 target free mesh iteration 5

9. Sometimes we will also want to smooth the edges on the "curves" in-between the optimized and non-optimized regions. To do this we will first "un-fix" the node positions we fixed in the previous step. Then we will create a group with all of the edges in the optimized region and a group with all of the edges in the non-optimized region. Then we will intersect these two groups to get the edges that are in-between these two regions. Then we can smooth those edges. Finally, we will smooth the surfaces in the optimized region again. You may also wish to smooth the tris in the non-optimized region but this will require some more sophistication in fixing node positions so as not to lose sharp features in the model. This example shows one specific smoothing sequence. You may prefer other approaches.

node in tri in block 3 position free

group "opt_edges" add edge in tri in block 4

group "non_opt_edges" add edge in tri in block 3

group "int_edges" intersect opt_edges with non_opt_edges

smooth edge in int_edges target free mesh

node in edge in int_edges position fixed

smooth tri in block 4 target free mesh iteration 5

10. Once we get a decent surface triangle mesh we can tet mesh the interior.

tetmesh tri all

Credits





Sandia National Laboratories is a multi-mission laboratory managed and operated by National Technology & Engineering Solutions of Sandia, LLC., a wholly owned subsidiary of Honeywell International, Inc., for the U.S. Department of Energy's National Nuclear Security Administration under contract DE-NA0003525.

Manager

- Michael Skroch, Manager, Computational Simulation Infrastructure Department (Org. 1543), Sandia National Laboratories

Project Board

- **Principal Investigator**: Ryan Viertel, Org. 1543

Research and Development

> Computational Simulation Infrastructure Department, Org. 1543, Sandia National Laboratories, Albuquerque, NM
>
> - Matthew L. Staten
> - Steven J. Owen
> - Roshan W. Quadros
> - Byron Hanks
> - Trevor Hensley
> - Clinton Stimpson
> - Corey Ernst
> - Corey McBride
> - Scott Mitchell
> - Brian Carnes
> - Kevin Copps
> - Paul Stallings
>
> External Contributors
>
> - Mark Richardson
> - Karl Merkley

Administrative Assistant

- Janeane Sanchez, 1545, Sandia National Laboratories

Quick Reference

Geometry | **File Import** | **Meshing** | **Genesis** | **Program** | **Entity Parsing** | **Groups** | **Graphics** | **Settings**

The following is a brief overview of some of the most used command-line CUBIT commands.

GEOMETRY

Primitives

Brick X <> [Y <> Z <>]
Cylinder Radius <> Height <>
Frustum Z <> Radius <> [Top <>]
Frustum Z <> Maj Rad <> Min Rad <>
Prism Z <> Sides <> Rad <> [Maj <> Min <>]
Pyramid Height <> Sides <> Radius <>
Sphere Rad <> [Xpos] [Ypos] [Zpos] [Inn <>]
Torus Major Rad <> Minor Rad <>

Booleans

Unite <> [With <>] [keep]
Subtract <> From <> [keep]
Intersect <> [With <>] [keep]

Transformations

Body <> [Copy] Move <dx> <dy> <dz>
Move {} <> location {} <> [except [x] [y] [z]]
Rotate {} <> About {x| y| z|<> <> <>} Angle <>
Rotate {} <> About Vert <> Vert <> Angle <>
Rotate {} <> About Nor Of Surf <> Angle <> Body <> [Copy] Scale <>
Body <> [Copy] Reflect {x| y| z|< x> <y> <z>}

Decomposition

Webcut {} <> Pla Vert <> [Vert]<> [Vert]<> ()
Webcut {} <> Plane Surf <> ()
Webcut {} <> Plane {xpla| ypla| zpla} [offs <>]
Webcut {} <> Tool [Body] <>
Webcut {} <> With Sheet {Body| Surf} <>
Webcut {} <> With Sheet Ext Fr Surf <>
Webcut {} <> Cyl Rad <> Axis {x| y| z| Vert <> Vert <>| <x><y><z>} [cent]
Options: [Noimprint| Imprint(default)], [Nomerge(default)| Merge], [group_ results] Section {} <> {{ xpla| ypla| zpla} [offs <>]} | Surf <>} [keep] [normal(default)| reverse]

FILE IMPORT

Import Acis 'filename'
Export Acis 'filename' [Body <>]
Import Mesh Geometry 'filename' (options)

MESHING

Mesh {} <>
Delete Mesh {} <> [Propagate]

Intervals

{} <> Interval {<> | Hard | Soft | Default}
{} <> Size {<> | Auto}
Match Intervals {} <> [Ass Grou [Onl| Infea]] [Seed Cur <>] [Map| Pave]

Mesh schemes

Mesh schemes

{} <> Scheme ...
Curve: bias, copy, curvature, equal, stretch
Surface: auto, circle, copy, hole, map, mirror, pave, pentagon, qtri, submap, triprimitive, trimap, trimesh, tripave
Volume: auto, copy, map, sphere, submap, sweep, tetmesh, tetprimitive, thex
Smooth {} <>
{} <> Smooth Scheme ...

Smooth schemes

Curves: laplacian, randomize
Surface: centroid area pull, equipotential, laplacian, condition number, randomize, untangle, winslow
Volume: equipotential, laplacian, condition number, untangle, randomize

GENESIS

Block <> {Group| Vol| Surf| Curv} <> [Remove]
SideSet <> {Group| Curve} <> [Remove]
NodeSet <> {} <> [Remove]
Export Genesis 'filename'
Block <> Attribute <>
Block <> Element Type <type_>
Curves: bar[| 2| 3]| beam[| 2| 3]| truss[| 2| 3]
Surfaces: quad[| 4| 8| 9]| shell[| 4| 8| 9]| tri[| 3| 6| 7]
Volumes: hex[| 8| 20| 27]| pyr| tetra[| 4| 8| 10| 14] hexshell
SideSet <> Surf <> [Rem|[She][For| Rev| Both]]
SideSet <> Surf <> wrt Volume <>
Reset {Genesis | Nodesets | Sidesets | Blocks}

PROGRAM

Play 'filename'
Record {' filename' | stop}
Logging {off|on file <'filename'> [resume]}
Reset
Reset Genesis
Quit

ENTITY PARSING

Examples

Surface 1 2 3 4 to 6 by 2 ...
Curve all in Volume 2 ...
Draw Edge all in Hex 32
List Curve 1 to 50 except 2 4 6
Draw Sideset 1 2 3 Curve 3 to 5 Hex 2 4 6

GROUPS

Group <> {add| equals| remove| xor} {} <>
Group <> {inters| unite} grou <> with grou <>
Group <> subtract group <> from group <>

GRAPHICS

Default mouse buttons (command line)

B1 - rotate; B2 - zoom; B3 - pan
Control-B1: pick entity (In graph win: 0,1,2,3,4 - Pick vert, curv, surf, vol, body)

Shortcuts (focus in Graphics Window)

a Add to selection group
b Toggle Bounding Box on Click
c Clear "picked" Group

c Clear 'picked' Group

d Display 'picked' group, make it the selection
e Echo ID of selection to command line
f Assign function to mouse button
g List geometry of selection
h Print help
i Toggle visibility of selection
j/k Move slicing plane down/up
l List current selection (as if you typed 'list ...')
control-l Give focus to the command prompt
m/n List picked group/selection contents
p Toggle Persistent Wireframe
q Quit Current Mode (Exit slicing if slicing)
r Remove from 'picked' Group
s Toggle save-mesh on slice move
u Toggle mouse circle visibility
v Reset view
w Toggle Wireframe on click
x/y/z Slice along x/y/z-axis
Shift-Z Zoom on current selection
F1 Save view 1 Numbers: set what you're picking.
ESC Cancel current Action
Tab Next possible selection
Shift-Tab Previous possible selection

Shift-S Activates graphics clipping plane controls

SETTINGS

Set AcisOption {string|double|integer} 'OptionName' <value>
Set Attribute <attrib_type> Auto {actuate|update} {on|off}
[Set] Auto Size Default
[Set] Auto Size Function [1|2]
Set AutoUniqueId {on|OFF}
Set Auto Sweep Scheme {Sw|Proj|Trans|Rot} Set Boolean Regularize [ON|off] Set Block Mixed Element Output {offset|degenerate|explicit} Set Block Triangle Offset <value> Set Block Tetrahedron Offset <value> Set Block Pyramid Offset <value> Set Catch Interrupt [on|off] Set Cleanup Angle <val> (default = 179.0) Set {curve|surface} Imprint Cleanup Tolerance <value> Set Continue Meshing [ON|off] Set Core [on|off] set {Corner|End} Angle <degrees>
set Corner Weight <value> Set Crash Save [on|off]
[set] Diagnostic {on|off} [set] Geometry Version <> (1400, 1500, 1600, 1700, 1800, 1900)
[set] Debug <index> {on|off}
[set] Debug <index> File <'filename'>
[set] Debug <index> Terminal
set Default Blocks {on|off|Volumes|Surfaces}
set Default Names {on|off} Set Default Element [tri|tet|QUAD|HEX|None] Set Default Autosize [ON|off] Set Digits [<number_to_list = -1>] Set Deletion Off Set Developer [commands] [on|off] Set Detail Periodic Fraction <value> Set Duplicate Block Elements {on|OFF}
[set] Echo [on|off] Set Exodus Single Precision [on|off] [Set] [Export Mesh] Nodeset Associativity [on|OFF] [Set] [Export Mesh] Nodeset Associativity Complete [on|OFF] [Set] Facet BBox [ON|off] [Set] Facet_modify [ON|off] Set Fastq {on|off] Set File Overwrite [Check] [ON|off] set FPE {divbyzero|invalid|underflow|overflow|all} [<toggle>]
set Fix Duplicate Names {on|off}
set FullHex [Use] [on|OFF] [Set] Geometry Accuracy <value> Set Geometry Engine {acis|catia|facet} Set Group Edge Visibility [on|OFF] Set Hex Relative Size Metric <value> [set] Info {on|off} set Interval Weight <value>
Set Import Mesh [vertex] [curve] [surface] Tolerance <distance> [Set] Import Mesh NodeSet Associativity [ON|off] Set Import Mesh NodeSet Order [On|Off] Set Imprint Groups {ON|off} Set Keep Invalid Mesh [on|off]
[set] Journal {on|off}}
[set] Journal [Graphics|Names|Aprepro|Errors] [on|off]\

[set] Journal [Graphics|Names|Aprepro|Errors] [on|off]}

[set] Journal idless [on|off|reverse]}
set Keep Invalid Mesh {on|off} [Set] Laminate Tolerance <double> set Large Angle Weight <value> Set Large Exodus [ON|Off] Set Exodus NetCDF4 [On|OFF] [set] Logging {off|on file <'filename'> [resume]} [Set] Logging Errors {off|on file <'filename'> [resume]} Set Mapping Constraint [ON|off]
set Match Intervals Rounding {on|off}
set Match Intervals Fast {on|off} Set Match Intervals Delta <interval_difference = 0.0> Set Maximum Arc_span {<angle>|default} Set Maximum Interval <int> Set Maximum Memory [on|off|value(in MB)] Set Merge Test BBox {on|OFF} Set Merge Test InternalSurf {on|OFF|Spline} Set Merge Base Names [on|off] Set Measure Small Tolerance <value> Set Metrics [on|OFF] Set Mesh Autodelete [ON|off] [Set] Morph Smooth [ON|off] Set Multisweep [ON|off] Set Nastran Exporter Params Add '<param_string>' Set Nastran Exporter Params Remove '<param_string>' Set Nastran Exporter Params Clear Set New Ids [on|off] Set Node Coincident Tolerance [<value>]
set Node Constraint [ON|off] Set Overlap [Facet] {Angle|Absolute} <value> Set Overlap {Minimum|Maximum} {Gap|Angle} <value> Set Overlap Normal {ANY|opposite|same} Set Overlap Tolerance <value> Set Overlap Group {on|OFF} Set Overlap {List|Display} {ON|off} Set Overlap [Within] {Body|Volume} {on|OFF} Set Overlap Imprnt {on|OFF} Set Parallel Meshing [on|OFF} [Set] Paver Cleanup {ON|off|extend} [Set] Paver Diagonal Scale <factor> (default = 0.9) [Set] Paver Grid Cell <factor> (default = 2.5) [Set] Paver Size Limits {default|minimum <value>|maximum<value>}
[set] Paver Smooth Method { Default | Smooth Scheme|Old}
[set] Paver Linearsizing {off|on} Set Persistent Ids {off|ON} set Patran Export Autogroups [on|OFF] Set Patran Export Groups {ON|off] Set Play History {on|OFF} [set] Project Smooth {on|off} Set Push Attribs {on|off} Set Print Quality {WARNING|error|off} Set QTri Test {angle|diagonal} Set Qtri Split <2|4> (default = 2)
Set Quad Relative Size Metric <value> Set Quality Threshold <double> (default = 0.2)
set Replacement character '.|_|@'
Set ReverseZoom [on|off] Set Save [Exodus|Cubit] [backups <number>]
[set] Scheme Auto Fuzzy [Tolerance] <degrees>
Set Sculpt Refine {on|OFF}[set] Smooth Iterations {default|<value>}
Set Separate After Webcut [ON|off] [set] Smooth Method {laplacian | isoparametric}
[set] Smooth Tol <value> (Default = 0.05) set {source|target} surface pattern '<pattern>' Set Split Surface Tolerance <value> Set Split Surface Parametric {on|OFF} Set Split Surface Auto Detect Triange {ON|off} Set Split Surface Point Angle Threshold <value> Set Split Surface Side Angle Threshold <value> Set Split Surface Extend Gap Threshold <value> Set Split Surface Extend Tolerance <value> Set Split Surface Extend Normal {on|OFF} Set Stop Error {on|OFF} Set Submap CornerPicking {ON|off}
set Suffix character '.|_|@'
Set Tight [[Bounding] [Box] [{Surface|Curve|Vertex} {on|off}]] [Set] Tridelaunay Point Placement [{asp|gq}] (Advancing Steiner Point,Guaranteed Quality) [Set] Trimesher Advancing Front [Set] Tolerant Mesh Feature Size <value> [Set] Tolerant Mesh MBG {OFF|on|only} Set Tri Relative Size Metric <value> Set Tet Relative Size Metric <value> set Turn Weight <value>
Set Unite Mixed [ON|off] [Set] Unmerge Duplicate_mesh {on|off} [Set] Unmerge New Ids [{on|off}]
Set Verbose Errors [on|off] Set Verbose Mesh [on|off] [set] Warning {on|off}
Set WorkingDirectory 'directory_path'

Glossary