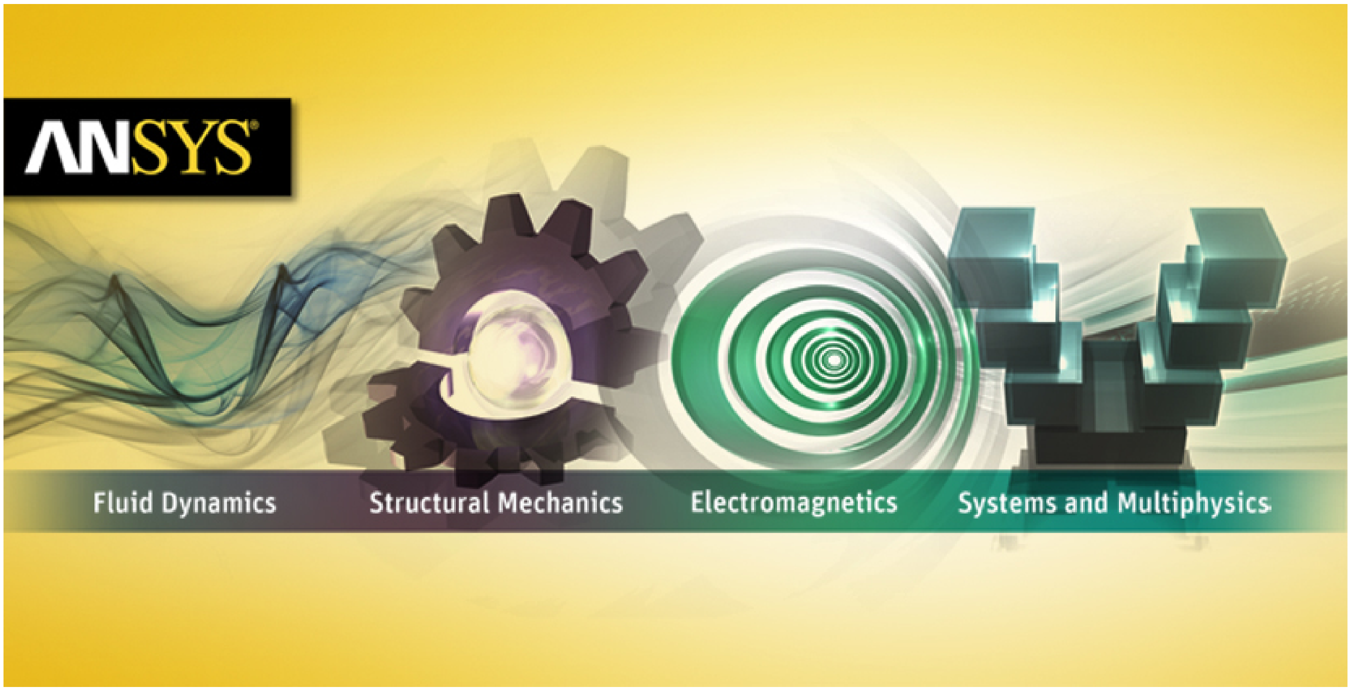


ANSYS ICEM CFD User's Manual

ANSYS, Inc.
Southpointe
275 Technology Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
http://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

ANSYS ICEM CFD 15.0
November 2013

ANSYS, Inc. is certified to ISO 9001:2008.

Published in the U.S.A.

Table of Contents

Introduction to ANSYS ICEM CFD 1
- Overall Process 1
  - Opening/Creating a Project 2
  - Creating/Manipulating the Geometry 3
  - Creating the Mesh 3
  - Checking/Editing the Mesh 4
  - Generating the Input for the Solver 5
- The ANSYS ICEM CFD GUI 5
- Using the Help System 9

CAD Repair 13
- Close Holes 13
- Remove Holes 14
- Fill, Trim and Blend in Stitch/Match Edges 15
- Match in Stitch/Match Edges 16

Tetra Meshing 19
- Introduction 19
  - Tetra Mesh Generation 19
  - Input to Tetra 20
- Tetra Generation Steps 20
  - Repairing the Geometry 21
  - Geometry Details Required 21
  - Sizes on Surfaces and Curves 22
  - Meshing Inside Small Angles or in Small Gaps Between Objects 22
  - Desired Mesh Region 22
  - The Octree Mesh Method 23
- Important Features in Tetra 26
  - Curvature/Proximity Based Refinement 27
  - Tetrahedral Mesh Smoother 27
  - Tetrahedral Mesh Coarsener 27
  - Triangular Surface Mesh Smoother 27
  - Triangular Surface Mesh Coarsener 27
  - Triangular Surface Editing Tools 27
  - Check Mesh 28
  - Smooth Mesh Globally 28
  - Quality Metrics 29
  - Advanced Options for Smoothing Mesh 30

Prism Mesh 33
- Prism Mesh Process 34
  - Prism Mesh Preparation 34
  - Smoothing Tetra/Prism Mesh 35

Hexa 37
- Introduction 37
- Features of Hexa 37
- Mesh Generation with Hexa 38
- The Hexa Database 39
- Intelligent Geometry in Hexa 39
- Unstructured and Multi-block Structured Meshes 39
  - Unstructured Mesh Output 40
  - Multi-Block Structured Mesh Output 40
- Blocking Strategy 40

Hexa Block Types ... 41
Split ... 41
Merge ... 41
Automatic O-grid generation ... 41
Using the Automatic O-grid ... 42
Important Features of an O-grid ... 42
Edge Meshing Parameters ... 43
Smoothing Techniques ... 44
Refinement and Coarsening ... 44
Refinement ... 44
Coarsening ... 44
Replay Functionality ... 44
Generating a Replay File ... 45
Advantage of the Replay Function ... 45
Using Variables in the Replay Script ... 45
Periodicity ... 45
Applying the Periodic Relationship ... 45
Pre-Mesh Quality ... 46
Determinant ... 46
Angle ... 46
Volume ... 46
Warpage ... 46
Most Important Features of Hexa ... 47
Properties ... 49
Create Material Property ... 49
Save Material ... 49
Open Material ... 49
Define Table ... 49
Define Element Properties ... 49
Constraints ... 51
Create Constraint / Displacement ... 51
Define Contact ... 51
Define Single Surface Contact ... 51
Define Initial Velocity ... 51
Define Planar Rigid wall ... 51
Loads ... 53
Force ... 58
Pressure ... 58
Temperature ... 58
Solve Options ... 59
Setup Solver Parameters ... 59
Setup Analysis Type ... 59
Setup Sub-Case ... 59
Write/View Input file ... 59
Submit Solver Run ... 59
FEA Solver Support ... 59
Workbench Integration ... 61
Elements of the ICEM CFD Component ... 62
Creating an ICEM CFD Component ... 63
Updating ICEM CFD Projects ... 64
Interface Differences in the Data-Integrated ICEM CFD ... 67
One-Click Menus ... 67

- Workbench Replay Control Dialog 68
- Setting Parameters 69
  - Setting Input Parameters 70
  - Setting Parameters for All Existing Curves, Surfaces, or Edges 72
  - Setting Workbench Mesh Parameters for Parts 72
  - Setting Parameters for Prism Meshing 73
  - Setting User-Defined Input Parameters 73
  - Setting Output Parameters 74
    - Setting Output Parameters 74
    - Deleting Output Parameters 75
- User-Defined Parameters Example 75

Introduction to ANSYS ICEM CFD

ANSYS ICEM CFD provides advanced geometry acquisition, mesh generation, and mesh optimization tools to meet the requirement for integrated mesh generation for today's sophisticated analyses.

Maintaining a close relationship with the geometry during mesh generation, ANSYS ICEM CFD is used especially in engineering applications such as computational fluid dynamics and structural analysis.

ANSYS ICEM CFD's mesh generation tools offer the capability to parametrically create meshes from geometry in numerous formats:

- Multiblock structured
- Unstructured hexahedral
- Unstructured tetrahedral
- Cartesian with H-grid refinement
- Hybrid meshes comprising hexahedral, tetrahedral, pyramidal and/or prismatic elements
- Quadrilateral and triangular surface meshes

ANSYS ICEM CFD provides a direct link between geometry and analysis. In ANSYS ICEM CFD, geometry can be input from just about any format, whether from a commercial CAD design package, 3rd party universal database, scan data or point data. Beginning with a robust geometry module which supports the creation and modification of surfaces, curves and points, ANSYS ICEM CFD's open geometry database offers the flexibility to combine geometric information in various formats for mesh generation. The resulting structured or unstructured meshes, topology, inter-domain connectivity and boundary conditions are then stored in a database where they can easily be translated to input files formatted for a particular solver.

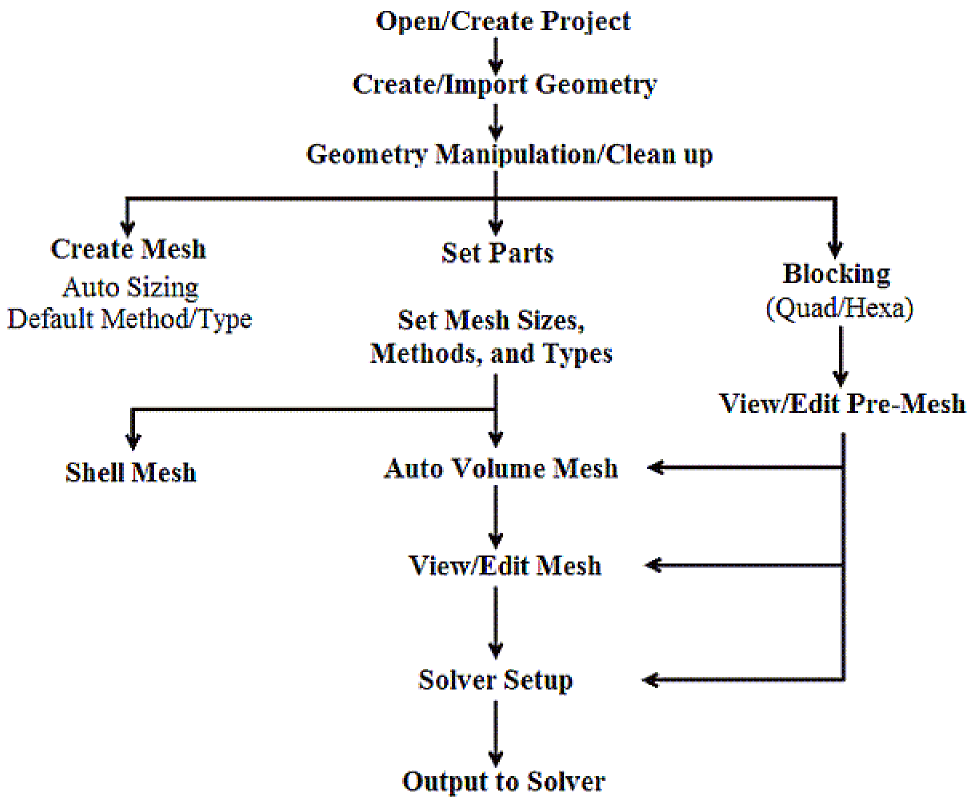
Overall Process
The ANSYS ICEM CFD GUI
Using the Help System

Overall Process

The generic working process involves the following:

1. Open/Create a project.
2. Create/Manipulate the geometry.
3. Create the mesh.
4. Check/Edit the mesh.
5. Generate the input for the solver.

Figure 1: The Overall Process



Opening/Creating a Project

All the files required for a particular analysis are contained within a **Project**. You can either open an existing project or create a new project. The **Project** directory typically contains one or more of the following file types:

Tetin (*.tin)
contains geometry entities, material points, part association, and global and entity mesh sizes.

Project Settings (*.prj)
contains the project settings.

Domain (*.uns)
contains the unstructured mesh.

Blocking (*.blk)
contains the blocking topology.

Boundary Conditions (*.fbc)
contains boundary conditions.

Attributes (*.atr)
contains attributes, local parameters, and element types.

Parameters (*.par)
contains model parameters and element types.

Journal (*.jrf)
contains a record of operations performed (echo file).

Replay (*.rpl)
contains the replay script.

Creating/Manipulating the Geometry

ANSYS ICEM CFD includes a wide range of tools for creating new and/or manipulating existing geometry. You can either alter complex geometry or create simple geometry without having to go back to the original CAD. This can be done for CAD (NURBS surfaces) and triangulated surface data. The ANSYS ICEM CFD Direct CAD Interfaces provide the bridge between parametric geometry creation tools available in CAD systems and the computational mesh generation and mesh optimization tools available in ANSYS ICEM CFD, allowing users to operate in their native CAD systems. ANSYS ICEM CFD currently supports Direct CAD Interfaces for CATIA, I-deas, Creo Parametric, and Unigraphics.

The ANSYS ICEM CFD environment can combine CAD surface geometry and triangulated surface data into a single geometry database (tetin file) using the geometry interfaces. All geometry entities, including surfaces, curves and points are tagged or associated to a grouping called a part. With this part association, you can enable or disable all entities within the parts, visualize them with a different color, assign mesh sizes on all entities within the part and apply different boundary conditions by part.

ICEM CFD also supports several *geometry interfaces* which allow you to import geometry from other formats (CAD, 3rd party geometry, Faceted Data and Mesh formats). For a complete list of supported geometry interfaces, see Import Geometry in the Help manual.

Although most of the meshing modules within ANSYS ICEM CFD allow minor gaps and holes in the geometry, in some cases it is necessary to find and close large gaps and holes without returning to the original CAD software. ANSYS ICEM CFD provides tools for such operations on either CAD or triangulated surfaces. Finally, curves and points can be automatically created to capture certain key features in the geometry. These curves and points will act as constraints for the mesher, forcing nodes and edges of the elements to lie along them, and thus capturing the feature.

Creating the Mesh

The meshing modules available include the following:

Tetra
The ANSYS ICEM CFD Tetra mesher takes full advantage of object-oriented unstructured meshing technology. With no tedious up-front triangular surface meshing required to provide well-balanced initial meshes, ANSYS ICEM CFD Tetra works directly from the CAD surfaces and fills the volume with tetrahedral elements using the Octree approach. A powerful smoothing algorithm provides the element quality. Options are available to automatically refine and coarsen the mesh both on geometry and within the volume. Also included are a Delaunay algorithm and an Advancing Front algorithm to create tetras from an existing surface mesh and also to give a smoother transition in the volume element size. The Delaunay method is robust and fast; the advantage of the Advancing Front method is its ability to generate a smoothly transitioning Tetra mesh with a controlled volume growth ratio.

Hexa
The ANSYS ICEM CFD Hexa mesher is a semi-automated meshing module which allows rapid generation of multi-block structured or unstructured hexahedral volume meshes. ICEM CFD Hexa represents a new approach to grid generation where the operations most often performed by experts

are automated and made available at the touch of a button. Blocks can be built and interactively adjusted to the underlying CAD geometry. This blocking can be used as a template for other similar geometries for full parametric capabilities. Complex topologies, such as internal or external O-grids can also be generated automatically.

Prism

ANSYS ICEM CFD Prism generates hybrid tetrahedral grids consisting of layers of prism elements near the boundary surfaces and tetrahedral elements in the interior for better modeling of near-wall physics of the flow field. Compared to pure tetrahedral grids, this results in smaller analysis models, better convergence of the solution and better analysis results.

Hybrid Meshes

The following types of hybrid meshes can be created:

- Tetra and Hexa meshes can be united (merged) at a common interface in which a layer of pyramids is automatically created at a common interface to make the two mesh types conformal. These meshes are suitable for models where it is preferred to have a "structured" hexa mesh in one part and is easier to create an "unstructured" tetra mesh in another more complex part.

- Hexa-Core meshes can be generated where the majority of the volume is filled with a Cartesian array of hexahedral elements essentially replacing the tetras. This is connected to the remainder of a prism/tetra hybrid by automatic creation of pyramids. Hexa-Core allows for reduction in number of elements for quicker solver run time and better convergence.

Shell Meshing

ANSYS ICEM CFD provides a method for rapid generation of surface meshes (quad and tri), both 3D and 2D. Mesh types can be **All Tri**, **Quad w/one Tri**, **Quad Dominant**, or **All Quad**. The following methods are available:

- **Mapped based shell meshing (Autoblock)**: Internally uses a series of 2D blocks, resulting in a mesh better lined up with geometry curvature.

- **Patch based shell meshing (Patch Dependent)**: Uses a series of "loops" which are automatically defined by the boundaries of surfaces and/or a series of curves. This method gives the best quad dominant quality and capturing of surface details.

- **Patch independent shell meshing (Patch Independent)**: Uses the Octree method. This is the best and most robust method for unclean geometry.

- **Shrinkwrap**: Used for quick generation of mesh. As it is used as the preview of the mesh, hard features are not captured.

Checking/Editing the Mesh

The mesh editing tools in ANSYS ICEM CFD allow you to diagnose and fix problems in the mesh. You can also improve the mesh quality. A number of manual and automatic tools are available for operations such as conversion of element types, refining or coarsening the mesh, smoothing the mesh, etc.

The process typically involves the following:

1. Check the mesh for problems such as holes, gaps, overlapping elements using the diagnostic checks available. Fix the problems using the appropriate automatic or manual repair methods.

2. Check the elements for bad quality and use smoothing to improve the mesh quality.

3. If the mesh quality is poor, it may be appropriate to fix the geometry instead or recreate the mesh using more appropriate size parameters or a different meshing method.

Generating the Input for the Solver

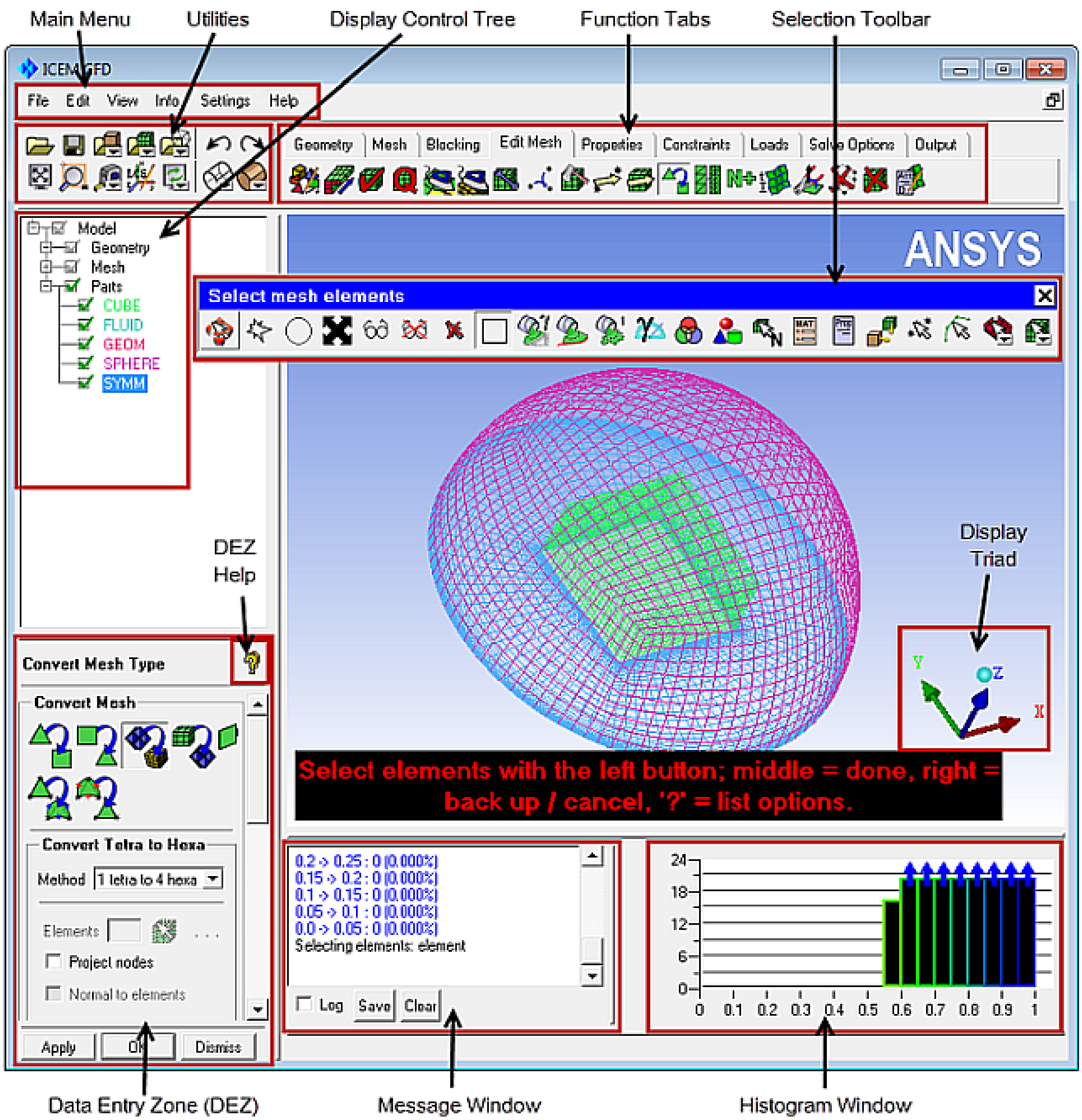
ANSYS ICEM CFD includes output interfaces to various flow and structural solvers, producing appropriately formatted files that contain complete mesh and boundary condition information. After selecting the solver, you can modify the solver parameters and write the necessary input files.

More information about the ANSYS ICEM CFD Output Interfaces is available from the Help menu. The **Output Interfaces** option opens the ANSYS ICEM CFD Output Interfaces information in a browser. For information about a specific interface, refer to the Table of Supported Solvers and click the name of the interface.

The ANSYS ICEM CFD GUI

The ANSYS ICEM CFD GUI offers a complete environment to create and manage your computational grids. As shown in Figure 2: ANSYS ICEM CFD GUI Components (p. 6), the GUI window contains several functional areas, which are described below.

Figure 2: ANSYS ICEM CFD GUI Components



Note

The GUI style shown is the default, **Workbench** style. For more information about the **GUI Style** options, refer to the Product-Selection settings.

Main Menu

The **Main Menu** is in the top, left corner of the GUI, and provides access to the following pull-down menus. Full details are available in the Main Menu Area of the Help Manual.

File Menu
contains options for creating new or opening existing projects, loading and saving files, importing and exporting geometry, and initializing scripting.

Edit Menu
contains **Undo**/**Redo** options, the option to open a shell window, and various internal mesh/geometry conversion commands.

View Menu
contains options for managing the look of the model in the graphics window. These include standard views, view controls, and annotations.

Info Menu
contains options for presenting statistics and other data about your geometry, mesh or other entities.

Settings Menu
contains options for managing many of the program preferences. Typically the default settings for performance, graphics, and others are suitable more than 90% of the time by a specific user.

Help Menu
contains links to Help Topics, tutorials, other documentation modules, and version information.

Utilities
The **Utilities** are icon representations of some of the most commonly used functions in the **Main Menu**, including opening/closing a project, or managing geometry, mesh or blocking files. Also included are quick access icons for undo/redo, measurement options, local coordinate system controls, and display options. Full details are available in the Graphical Main Menu, Utilities, and Display Options section of the Help Manual.

Display Control Tree
The **Display Control tree**, also referred to as the **Display tree**, along the upper left side of the screen, allows control of the display by part, geometric entity, element type and user-defined subsets. The tree is organized by categories. Each category can be enabled or disabled by selecting the check box. If the check mark is faded, some of the sub-categories are enabled and some disabled. Each category can be expanded or collapsed by selecting the "+" or "-" symbol, respectively.

Since some functions are performed only on the entities shown, the tree is an important feature to use when isolating the particular entities to be modified. Use the right mouse button on a category or type to reveal several display and modification options. Full details are available in the Display Tree section of the Help Manual.

Function Tabs
The **Function Tabs**, across the top of the GUI window, allow you to access the main functionality for the entire grid generation process. The function tabs are laid out from left to right in the order of a typical meshing process – **Geometry**, **Mesh**, **Blocking**, **Edit Mesh**, **Properties**, **Constraints**, **Loads**, **Solve Options**, and **Output**. Full details for each tab are available in the Help Manual.

Selection Toolbar
The **Selection Toolbar** contains icons for the tools and filters applicable to your current function as identified in the **Data Entry Zone**. Some tools and filters are linked to the hotkeys available in the particular selection mode. Full details are available in the Selection Options section of the Help Manual.

If a **Selection Toolbar** is present, you are operating in **Selection Mode** and the mouse is used to identify locations or entities. If there is no **Selection Toolbar**, you are operating in **Dynamic Mode** and the mouse is used to manipulate the graphical model display. See MouseBindings/Spaceball in the Help manual.

The Data Entry Zone (DEZ)

Clicking on a **Function Tab** brings its action icons to the fore front. Clicking on any of these icons will activate the associated **Data Entry Zone** (**DEZ**). The DEZ provides access to the controls and parameters associated with a particular operation.

The controls utilized by the DEZ are described here:

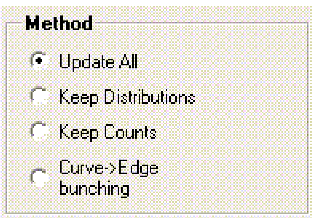
Button

A button is used to perform a function indicated by the button label.

Check Box

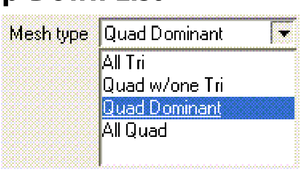
A check box is used to enable/disable an item or action indicated by the check box label.

Radio Buttons



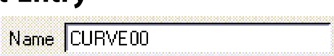
Radio buttons are a set of check boxes with the condition that only one can be enabled at a time. When you click the left mouse button on a radio button, it will be enabled, while all others will be disabled.

Drop-Down List



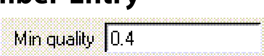
A drop-down list is a hidden single-selection list that shows only the current selection. Click the arrow button to display the list.

Text Entry

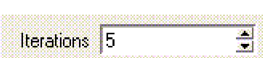


Text entries allow you to enter text associated with the label for the field.

Number Entry

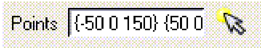


Number entries allow you to enter numerical values for the parameter indicated by the label for the field.



Some number entry fields may have arrow buttons which allow you to increase or decrease the value in entry field.

Selections



Selection fields indicate the entities selected for a particular operation. Click the button adjacent to the selection field to invoke the selection mode. The selection toolbar associated with the operation will appear. After confirming the selections, the selected items will be listed in the selection field.

Apply/OK/Dismiss

At the bottom of the DEZ, these buttons are used to accept or reject the parameters.

Apply
accepts your parameters and leaves the DEZ open.

OK
accepts your parameters and closes the DEZ.

Dismiss
rejects your parameters and closes the DEZ.

The Histogram Window

The **Histogram Window** shows a bar graph representing the mesh quality. The X axis represents element quality (usually normalized to between 0 and 1) and the Y axis represents the number of elements. Other functions which utilize this space will become pop-up menus if the quality or histogram is enabled.

The Message Window

The **Message Window** contains all the messages that ANSYS ICEM CFD writes out to keep the user informed of internal processes. The **Message Window** displays the communication between the GUI and the geometry and meshing functions. Any requested information, such as measure distance, surface area, etc. will be reported in the message window. You can use the scroll bar to review the information from your entire session. Also, internal commands can also be typed and invoked from within the message window.

The **Save** command will write all message window contents to a file. This file will be written to the folder from which ICEM CFD was launched. The **Log** check box allows only user-specified messages to be saved to a file.

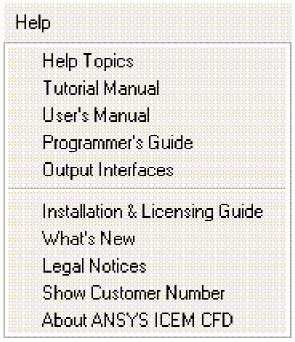
Note

The **Log** file is unique from the file created with the **Save** button. This file will be written to the starting directory, and it automatically updates as more messages are recorded. If the check box is disabled, you can append to a file by enabling **Log** and accepting an existing file name. **Log** will then append to this file.

Using the Help System

Click **Help** on the main menu and select the appropriate option from the menu.

Figure 3: Help Menu Options

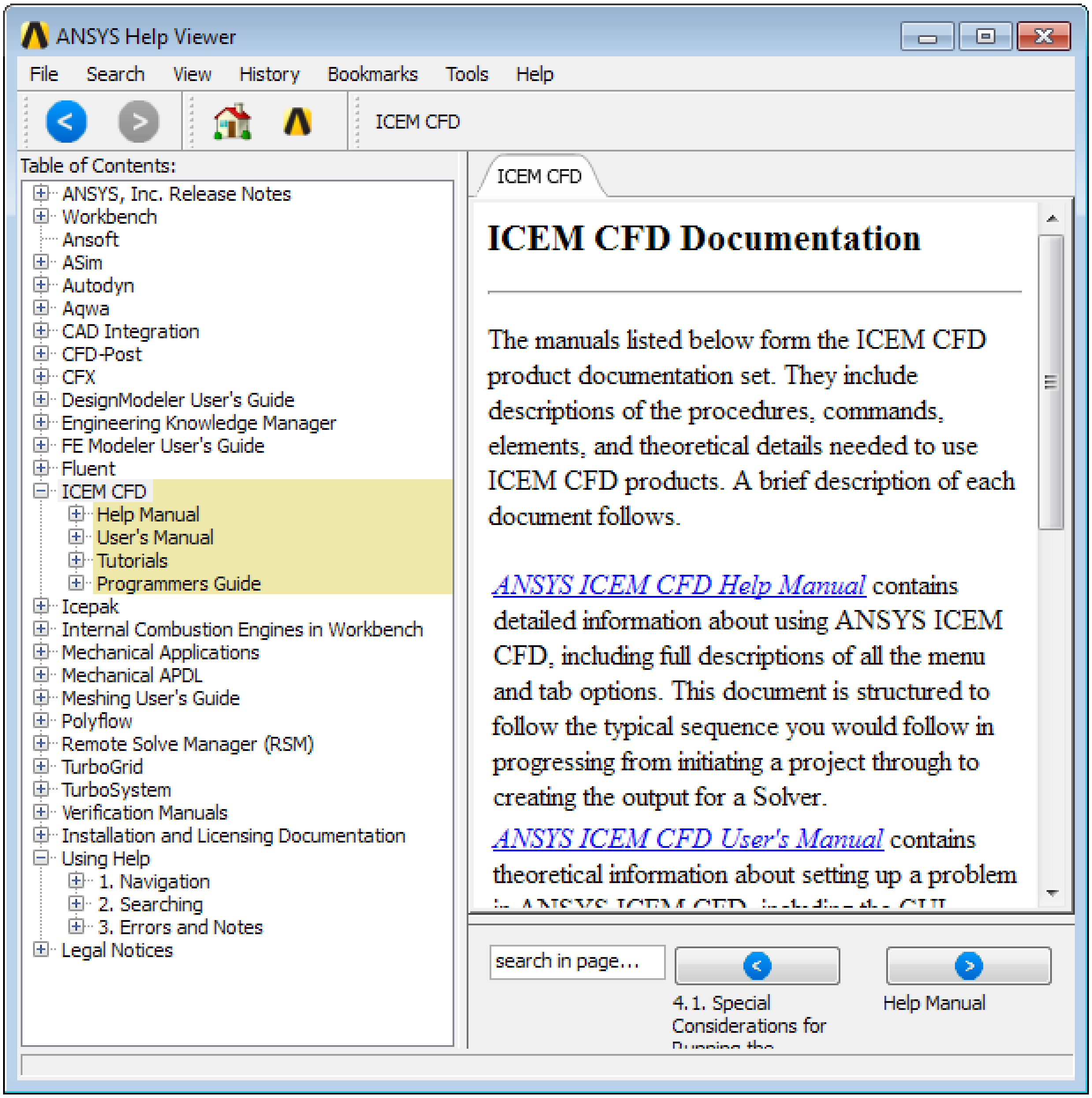


Online Help Interface

The **ANSYS Help Viewer** provides easy access to the documentation for the entire ANSYS family.

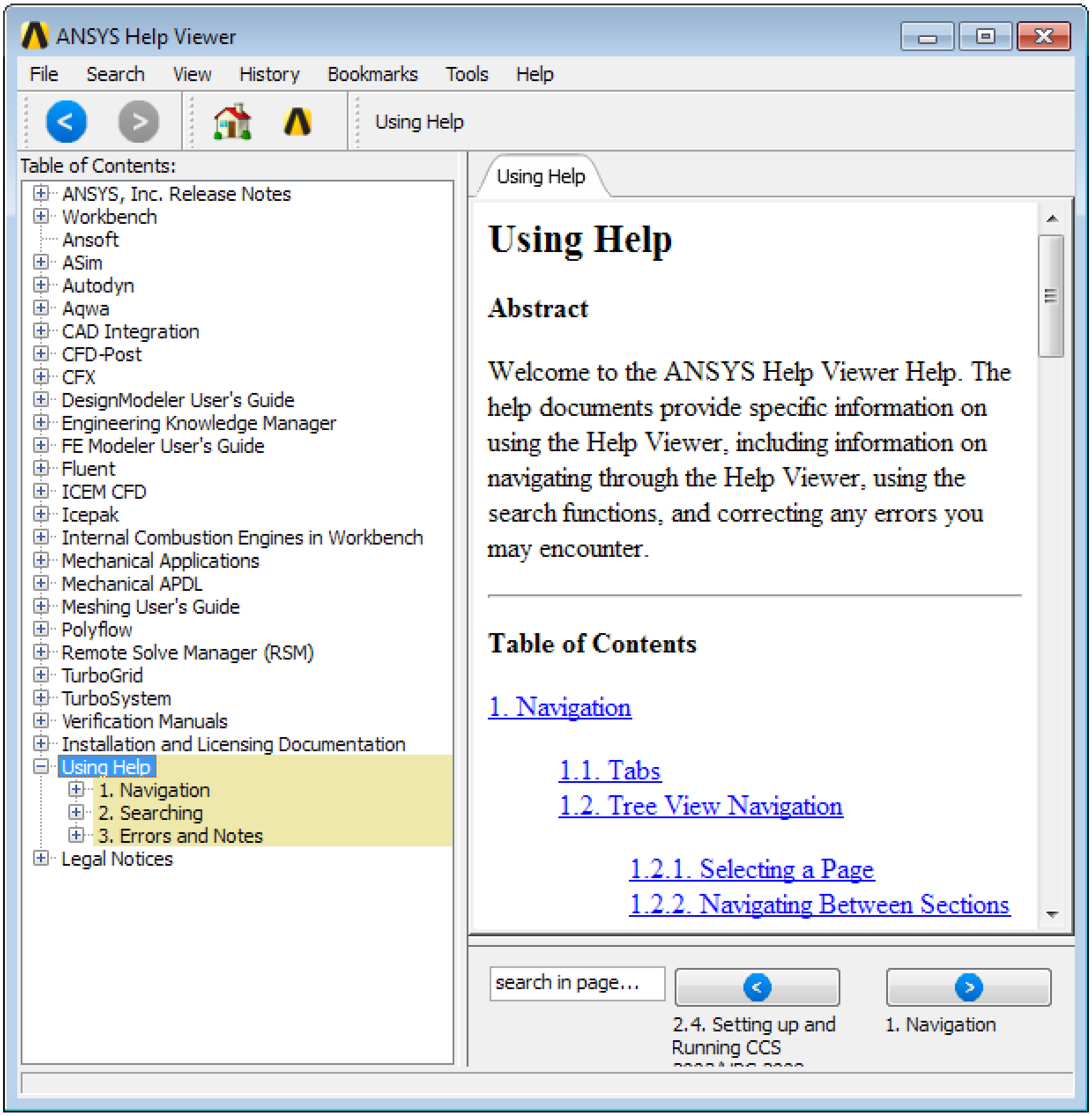
Click **ICEM CFD** on the hierarchical list and select the appropriate option from the menu.

Figure 4: The Online Help Interface



Full instructions for navigating the **ANSYS Help Viewer** are accessed by expanding the **Using Help** hierarchy in the Viewer.

Figure 5: Using the Help Viewer



CAD Repair

Before generating the mesh, you should confirm that the geometry is free of any flaws that would inhibit optimal mesh creation. If you wish to save the changes in the native CAD files, the following checks should be performed in a direct CAD interface:

- To create a mesh, the **Tetra** mesher requires that the model contains a closed volume. If there are any holes (gaps or missing surfaces) in the geometry that are larger than the local tetras, the **Tetra** mesher will be unable to find a closed volume. Thus, if you notice any holes in the model prior to mesh generation, you should fix the surface data to eliminate these holes.

- The **Build Topology** operation will find holes and gaps in the geometry. It should display yellow curves where there are large (in relation to a user-specified tolerance) gaps or missing surfaces.

- During the Tetra process any leakage path (indicating a hole or gap in the model) will be indicated. The problem can either be corrected on a mesh level, or the geometry in that vicinity can be repaired and the meshing process repeated. For further information on the process of interactively closing holes, see the section Tetra > Tetra Generation Steps > Desired Mesh Region.

Close Holes

You can use the **Close Holes** option if the hole is bounded by more than one surface. For example, in Figure 6: Hole Bounded by Multiple Surfaces (p. 13), the yellow curves represent the boundary of the hole. It is clear that this hole is bounded by more than one surface.

Figure 6: Hole Bounded by Multiple Surfaces

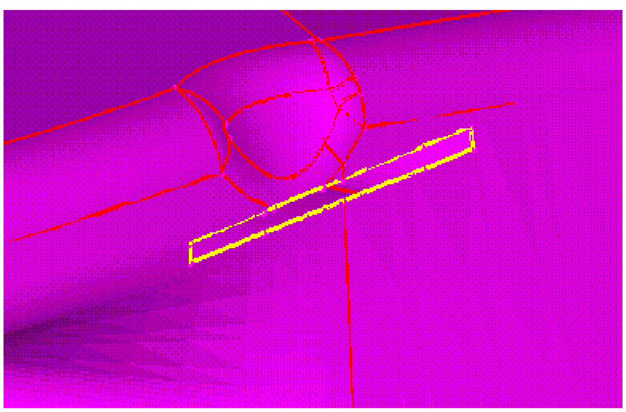
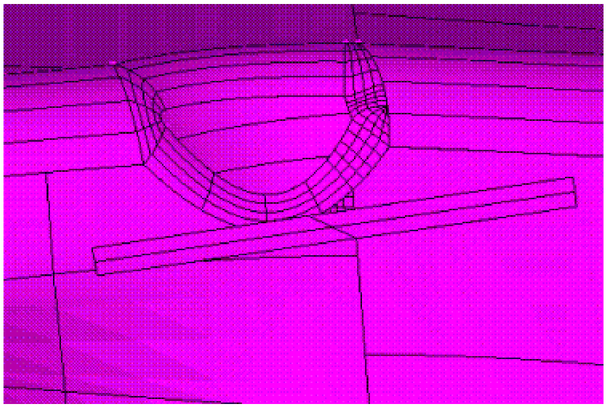


Figure 7: Closed Hole (p. 14) shows the geometry after the **Close Holes** operation is completed. A new surface is created to close the hole.

Figure 7: Closed Hole



Remove Holes

You can use the **Remove Holes** option if the hole lies entirely within a single surface, such as a trimmed surface. For example, in Figure 8: Hole Within a Single Surface (p. 14), the two yellow curve loops represent the boundaries of the holes, which lie entirely in one surface.

Figure 8: Hole Within a Single Surface

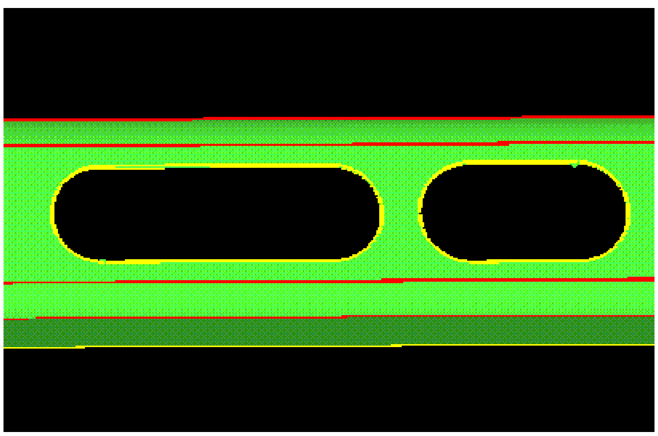
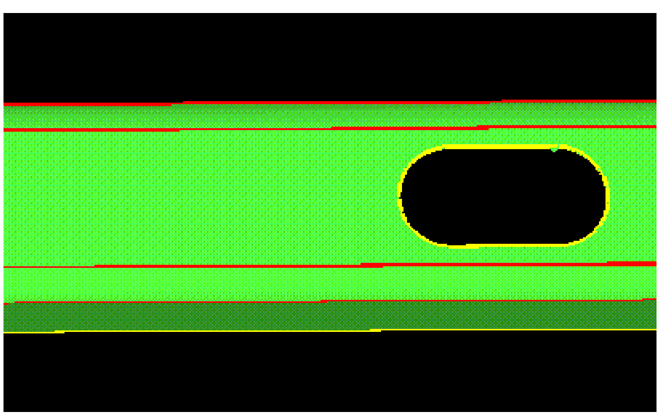


Figure 9: After Remove Holes (p. 14) shows the geometry after the **Remove Holes** operation is completed for one of the holes. The existing surface is modified by removing the trim definition.

Figure 9: After Remove Holes



Fill, Trim and Blend in Stitch/Match Edges

Consider the case of a geometry with a gap shown in Figure 10: Geometry With a Gap (p. 15).

Figure 10: Geometry With a Gap

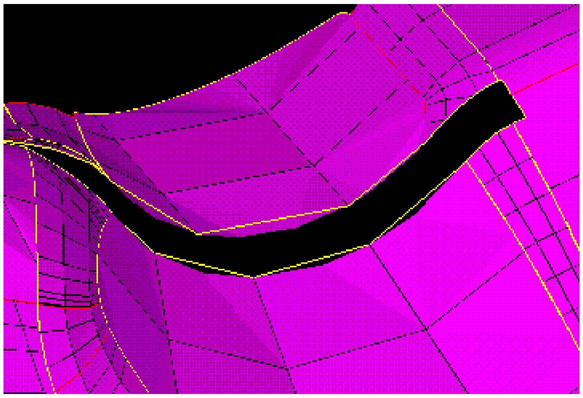


Figure 11: Using the Fill Option (p. 15) shows the use of the Fill option.

Figure 11: Using the Fill Option

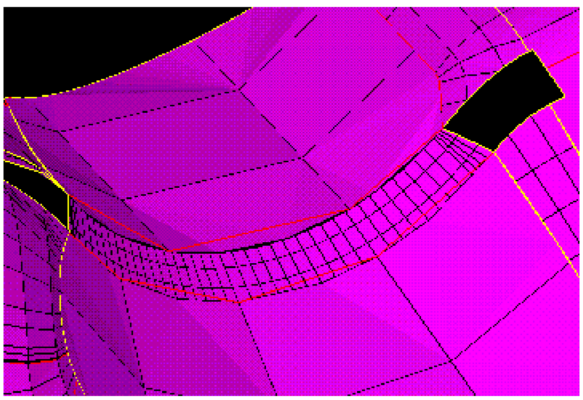


Figure 12: Using the Trim Option (p. 15) shows the use of the Trim option.

Figure 12: Using the Trim Option

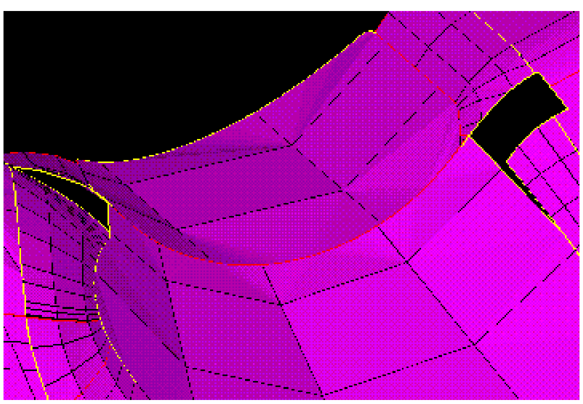
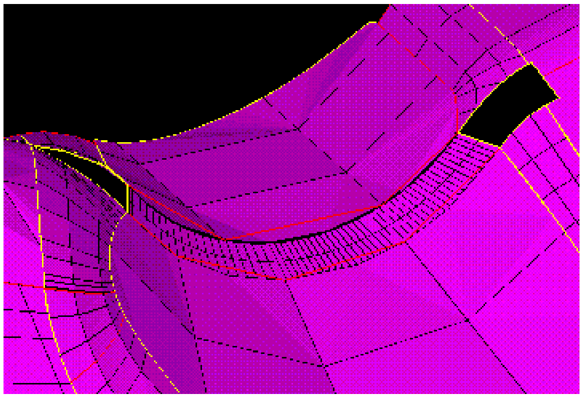


Figure 13: Using the Blend Option (p. 16) shows the use of the Blend option.

Figure 13: Using the Blend Option



Match in Stitch/Match Edges

The **Match** option is generally used in those cases where curves lie very close to each other, specifically when the two ends meet together (see Figure 14: Geometry With Mismatched Edges (p. 16) and Figure 15: Geometry After Using the Match Edges Option (p. 17)). You should have the two sets of curves within some tolerance for this option to work.

Figure 14: Geometry With Mismatched Edges

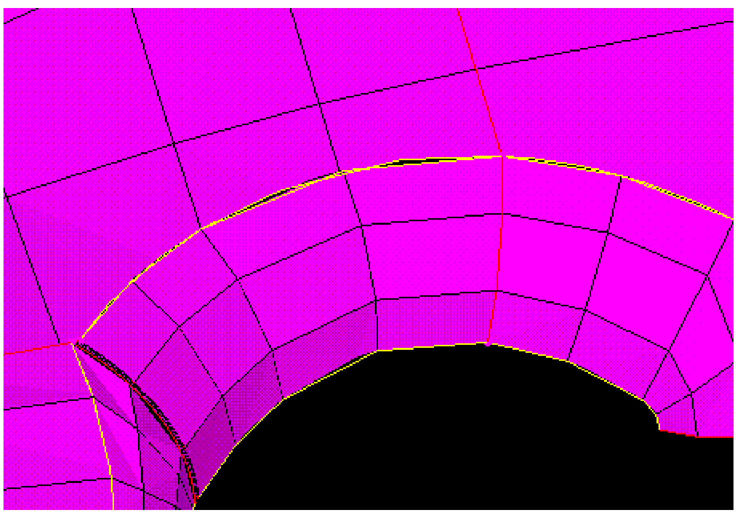
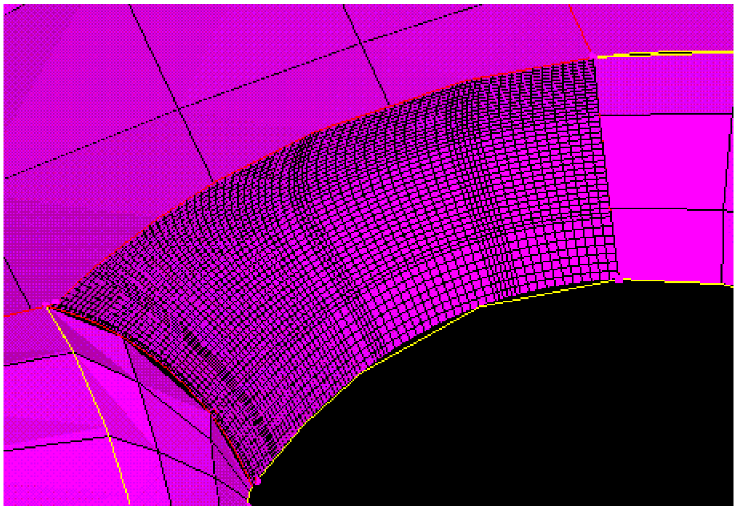


Figure 15: Geometry After Using the Match Edges Option



Tetra Meshing

Automated to the point that you have only to select the geometry to be meshed, the **Tetra** mesher generates tetrahedral meshes directly from the CAD geometry or STL data, without requiring an initial triangular surface mesh.

Introduction
Tetra Generation Steps
Important Features in Tetra

Introduction

The **Tetra** mesher can use different meshing algorithms to fill the volume with tetrahedral elements and to generate a surface mesh on the object surfaces. You can define prescribed curves and points to determine the positions of edges and vertices in the mesh. For improved element quality, the **Tetra** mesher incorporates a powerful smoothing algorithm, as well as tools for local adaptive mesh refinement and coarsening.

Tetra Mesh Generation

The **Tetra** mesher is suitable for complex geometries, and offers several advantages, including:

- Rapid model setup
- Mesh independent of underlying surface topology
- No surface mesh necessary
- Generation of mesh directly from CAD or STL surfaces
- Definition of element size on CAD or STL surfaces
- Control over element size inside a volume
- Nodes and edges of tetrahedra are matched to prescribed points and curves
- **Curvature/Proximity Based Refinement** automatically determines tetrahedra size for individual geometry features
- Volume and surface mesh smoothing, merging nodes and swapping edges
- Tetrahedral mesh can be merged into another tetra, hexa or hybrid mesh and then can be smoothed
- Coarsening of individual material domains
- Enforcement of mesh periodicity, both rotational and translational
- Surface mesh editing and diagnostic tools
- Local adaptive mesh refinement and coarsening

- One consistent mesh for multiple materials
- Fast algorithm: *1500 elements/second*
- Automatic detection of holes and easy way to repair the mesh
- For more details, go to Run Tetra - The Octree Approach

Input to Tetra

The following are possible inputs to the **Tetra** mesher:

- Sets of B-Spline curves and trimmed B-Spline surfaces with prescribed points
- Triangular surface meshes as geometry definition
- Full/partial surface meshes

B-Spline Curves and Surfaces

When the input is a set of B-Spline curves and surfaces with prescribed points, the mesher approximates the surface and curves with triangles and edges respectively; and then projects the vertices onto the prescribed points.

The B-Spline curves allow the **Tetra** mesher to follow discontinuities in surfaces. If no curves are specified at a surface boundary, the **Tetra** mesher will mesh triangles freely over the surface edge. Similarly, prescribed points allow the mesher to recognize sharp corners in the geometry. ANSYS ICEM CFD provides tools (**Build Topology**) to extract points and curves to define sharp features in the surface model.

Triangular Surface Meshes as Geometry Definition

Prescribed curves and points can also be extracted from triangulated surface geometry. This could be stereolithography (STL) data or a surface mesh converted to faceted geometry. Though the nodes of the **Tetra**-generated mesh will not exactly match the nodes of the given triangulated geometry, they will follow the overall shape. A geometry for meshing can contain both faceted and B-Spline geometry.

Full/Partial Surface Mesh

Existing surface mesh for all or part of the geometry can be specified as input to the **Tetra** mesher. The final mesh will then be consistent with and connected to the existing mesh nodes.

Tetra Generation Steps

The steps involved in generating a Tetra mesh are:

- Repairing/cleaning up the geometry
- Specifying geometry details
- Specifying sizes on surfaces/curves
- Meshing inside small angles or in small gaps between objects

- Desired mesh region
- Computing the mesh
- Checking the mesh for errors
- Editing the mesh to correct any errors
- Smoothing the mesh to improve quality

The mesh is then ready to apply loads, boundary conditions, etc., and for writing to the desired solver.

Repairing the Geometry

Refer to the CAD Repair section.

Geometry Details Required

In addition to a closed set of surfaces, the **Tetra** mesher requires curves and points where hard features (hard angles, corners) are to be captured in the mesh. Figure 16: Curves and Points Representing Sharp Edges and Corners (p. 21) shows a set of curves and points representing hard features of the geometry.

Figure 16: Curves and Points Representing Sharp Edges and Corners

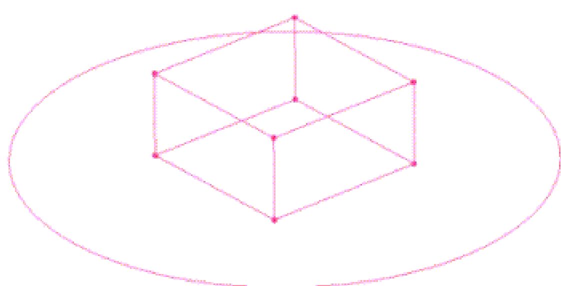


Figure 17: Mesh with Curves and Points (p. 21) shows the resultant surface mesh if the curves and points are preserved in the geometry. Mesh nodes are forced to lie along the curves and points to capture the hard features of the geometry.

Figure 17: Mesh with Curves and Points

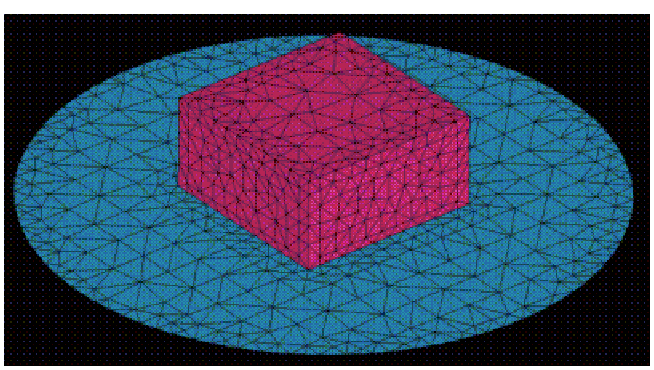
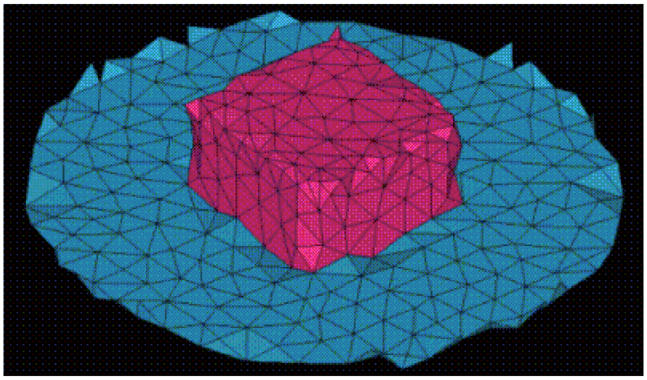


Figure 18: Mesh Without Curves and Points (p. 22) shows the resultant surface mesh if the curves and points are deleted from the geometry. The hard features of the geometry are not preserved, but rather

are neglected or chamfered. The boundary mesh nodes lie on the surfaces, but they will only lie on the edges of the surfaces if curves and points are present. Removal of curves and points can be used as a geometry defeaturing tool.

Figure 18: Mesh Without Curves and Points



Sizes on Surfaces and Curves

To produce the optimal mesh, it is essential that all surfaces and curves have the proper tetra sizes assigned to them. For a visual representation of the mesh size, select **Geometry > Surfaces > Tetra Sizes** from the **Display Tree**. The same can be done with **Curves**. Tetra icons will appear, representing the element size of the mesh to be created on these entities. Using the mouse, you may rotate the model and visually confirm that the tetra sizes are appropriate. If a curve or surface does not have an icon plotted on it, the icon may be too large or too small to see. In this case, modify the mesh parameters so that the icons are visible in a normal display.

To modify the mesh size for all entities, adjust the **Scale Factor**, which is found under **Mesh > Global Mesh Setup**. Note that if the **Scale Factor** is assigned a value of `0`, the **Tetra** mesher will not run.

Meshing Inside Small Angles or in Small Gaps Between Objects

Examine the regions between two surfaces or curves that are very close together or that meet at a small angle. (This would also apply if the region outside the geometry has small angles.) If the local tetra sizes are not small enough so that at least 1 or 2 elements would fit through the thickness, you should define thin cuts. This option is in the **Mesh > Global Mesh Setup > Volume Meshing Parameters** section. To define a thin cut, the two surfaces have to be in different Parts. If the surfaces meet, the curve at the intersection of the surfaces will need to be in a different part.

If the tetra sizes are larger or approximately the same size as the gap between the surfaces or curves, the surface mesh could have a tendency to jump the gap, thus creating non-manifold vertices. These non-manifold vertices would be created during the meshing process. The **Tetra** mesher automatically attempts to close all holes in a model. Since the gap may be confused as a hole, you should either define a thin cut, in order to establish that the gap is not a hole; or make the mesh size small enough so that it will not close the gap when the meshing is performed. A hole is usually considered a space that is greater than 2 or 3 elements in thickness.

Desired Mesh Region

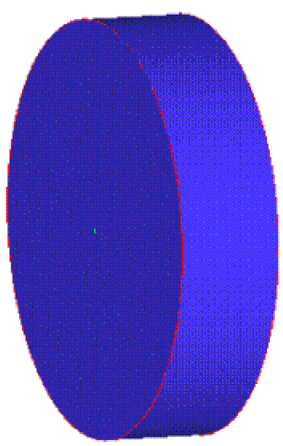
During the process of finding the bounding surfaces to close the volume mesh, the mesher will determine if there are holes in the model. If holes are found, the Message window will display a message like "Material point ORFN can reach material point LIVE." You will be prompted with a dialog box saying, "Your geometry has a hole, do you want to repair it?" A jagged line will display the leakage path from the ORFN part to the LIVE part. The elements surrounding the hole will also be displayed. To repair the hole, select the single edges bounding it - and the mesher will loft a surface mesh to close the hole. Further holes would be flagged and repaired in the same manner. If there are many problem areas, it may be better to repair the geometry or adjust the meshing parameters.

The Octree Mesh Method

The Octree method starts with generating a volume mesh using a top-down approach. Then the mesh is made conformal to the geometry and patch independent surface mesh is created at all the boundaries and internal walls. Expert users often use the robust Octree approach to generate a surface mesh and then regenerate the volume mesh using a bottom-up approach such as the Delaunay method or Advancing Front method. See Tetra/Mixed Mesh Type in the Help document for more information on these methods.

The Octree mesh method is based on the following spatial subdivision algorithm: This algorithm ensures refinement of the mesh where necessary, but maintains larger elements where possible, allowing for faster computation. Once the "root" tetrahedron, which encloses the entire geometry, has been initialized, **Tetra** subdivides the root tetrahedron until all element size requirements are met.

Figure 19: Geometry Input to Tetra



At this point, the Tetra mesher balances the mesh so that elements sharing an edge or face do not differ in size by more than a factor of 2.

Figure 20: Full Tetra Enclosing the Geometry

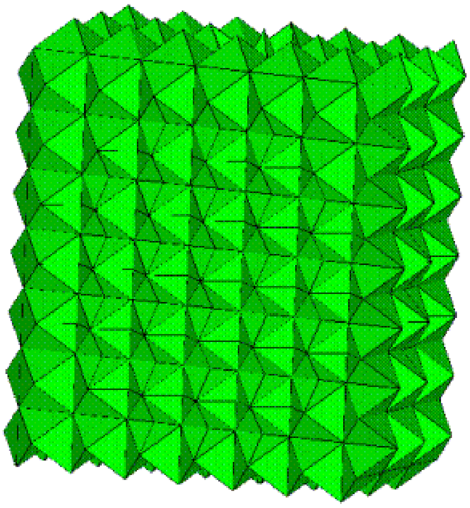


Figure 21: Full Tetra Enclosing the Geometry in Wire Frame Mode

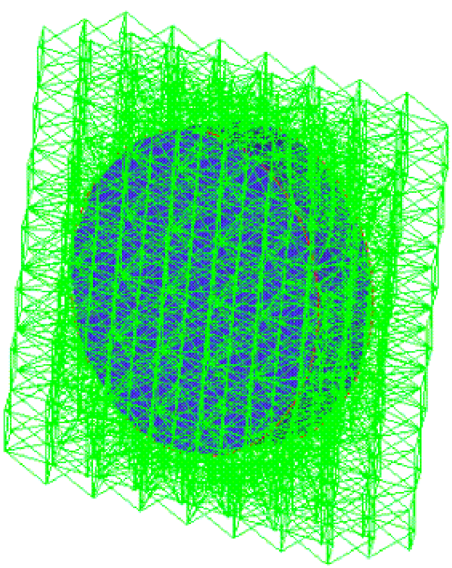
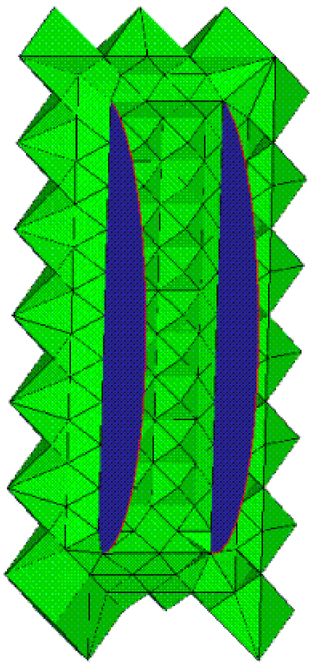


Figure 22: Cross-Section of the Tetra



After this is done, Tetra makes the mesh conformal - that is, it guarantees that each pair of adjacent elements will share an entire face. The mesh does not yet match the given geometry, so the mesher next rounds the nodes of the mesh to the prescribed points, prescribed curves or model surfaces. Tetra then "cuts away" all of the mesh, which cannot be reached by a user-defined material point without intersection of a surface.

Figure 23: Mesh after it captures surfaces and separation of useful volume

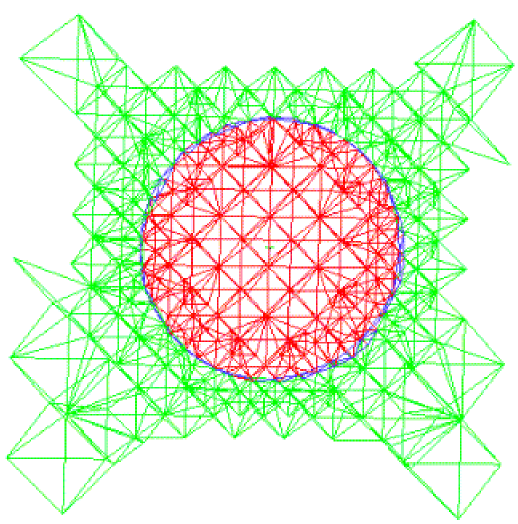
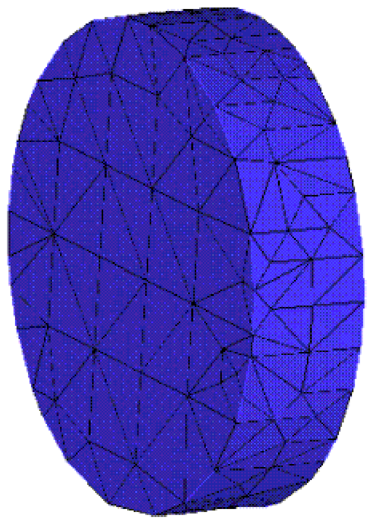
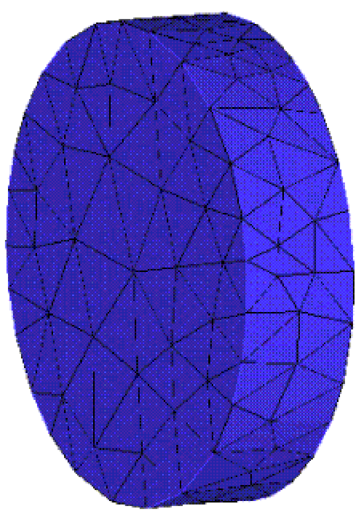


Figure 24: Final Mesh before smoothing



Finally, the mesh is smoothed by moving nodes, merging nodes, swapping edges and in some cases, deleting bad elements.

Figure 25: Final Mesh after smoothing



Important Features in Tetra

The following sections describe important features of tetra meshing.

- Curvature/Proximity Based Refinement
- Tetrahedral Mesh Smoother
- Tetrahedral Mesh Coarsener
- Triangular Surface Mesh Smoother
- Triangular Surface Mesh Coarsener
- Triangular Surface Editing Tools
- Check Mesh
- Smooth Mesh Globally
- Quality Metrics

Advanced Options for Smoothing Mesh

Curvature/Proximity Based Refinement

If the maximum tetrahedral size defined on a surface is larger than needed to resolve the feature, you can employ **Curvature/Proximity Based Refinement** to automatically subdivide the mesh to capture the feature. The value specified is proportional to the global scale factor, and is the smallest size to be achieved through automatic element subdivision. Even with large sizes specified on the surfaces, the features can be captured automatically.

The **Curvature/Proximity Based Refinement** value is the minimum element size to be achieved via automatic subdivision. If the maximum size on a geometry entity is smaller than the **Curvature/Proximity Based Refinement** value, the Tetra mesher will still subdivide to meet that requested size. The effect is a geometry-based adaptation of the mesh.

Tetrahedral Mesh Smoother

In smoothing the mesh, the tetrahedral smoother calculates individual element quality based on the selection from the list of available criteria.

The smoother modifies the elements with quality below the specified **Up to quality** value. Nodes can be moved and/or merged, edges are swapped, and in some cases elements are deleted. This operation is then repeated on the improved grid, up to the specified number of iterations. You can choose to smooth some element types while freezing others.

Tetrahedral Mesh Coarsener

During the coarsening process you can exclude surface or material domains. If the **Maintain surface sizes** option is enabled during coarsening, the resulting mesh satisfies the specified mesh size criteria on the geometric entities.

Triangular Surface Mesh Smoother

The triangular surface mesh inherent in the **Tetra** mesh generation process can also be used independently of the volume mesh. The triangular smoother marks all elements that are initially below the quality criterion and then runs the specified number of smoothing steps on the elements. Nodes are moved on the actual CAD surfaces to improve the quality of the elements.

Triangular Surface Mesh Coarsener

In the interest of minimizing grid points, the coarsener reduces the number of triangles in a mesh by merging triangles. This operation is based on the maximum deviation of the resultant triangle center from the surface, the aspect ratio of the merged triangle and the maximum size of the merged triangle.

Triangular Surface Editing Tools

There are tools available under the **Edit Mesh** menu for interactive mesh editing, where nodes can be moved on the underlying CAD surfaces, merged or even deleted. Individual triangles of the mesh can be subdivided or tagged with different names. You can perform quality checks, as well as local smoothing.

Diagnostic tools for surface meshes allow you to fill holes easily in the surface mesh. Also there are tools for the detection of overlapping triangles and non-manifold vertices, as well as detection of single/multiple edge and duplicate elements.

Check Mesh

Check the validity of the mesh using **Edit Mesh > Check Mesh**.

You can opt to use the **Create subsets** option for each of the problems so that they can be fixed later or can opt to use the **Check/fix each** option to check and fix each one of them. Using subset manipulation and mesh editing techniques, you can diagnose the problem and resolve it through merging nodes, splitting edges, swapping edges, delete/create elements, etc.

For ease of use when working with subsets, it is usually helpful to add elements to the subset in order to see what is happening around the problem elements. To do this, right-click on the Subset name in the Display tree and then add layers of elements to the subset. It is also useful to display the element nodes and/or display the elements slightly smaller than actual size. Both of these options can be accessed by right-clicking on Mesh in the Display Tree.

Keep in mind that after mesh editing, the diagnostics should be re-checked to verify that no mistakes were made.

There are several checks for **Errors** as well as **Possible problems**. The descriptions of each of these checks can be found in the Edit Mesh > Check Mesh section of the Help Manual.

Smooth Mesh Globally

After eliminating errors/possible problems from a tetra mesh, you need to smooth the grid using **Edit Mesh > Smooth Mesh Globally** to improve the quality.

Smoothing iterations
: This value is the number of times the smoothing process will be performed. Models with a more complicated geometry will require a greater number of iterations to obtain the desired quality, which is specified for **Up to quality**.

Up to quality
: The **Min** value represents the worst quality, while the **Max** value represents the highest quality elements. Usually, the **Min** value is set to 0.0 and the **Max** value is set to 1.0. The **Up to quality** value gives the smoother a quality to aim for. Ideally, after smoothing, the quality of the elements should be higher than or equal to this value. If this does not happen, you should employ other methods of improving the quality, such as merging nodes and splitting edges. For most models, the elements should all have ratios of greater than 0.3, while a ratio of 0.15 for complicated models is usually sufficient.

Freeze
: If the **Freeze** option is selected for an element type, the nodes of this element type will be fixed during the smoothing operation. As a result, this element type will not be displayed in the histogram.

Float
: If the **Float** option is selected, the nodes of the specified element type will be capable of moving freely, allowing nodes that are common with another type of element to be smoothed. The quality of elements set to float is not tracked during the smoothing process and so the quality is not displayed in the histogram.

The tetrahedral quality will be displayed within the **Quality Histogram** , where 0 represents the worst aspect ratio and 1 represents the best aspect ratio. You may modify the display of the histogram by adjusting the values of **Min**, **Max**, **Height**, and **Bars**. Right-click on the histogram to access the following options for modifying its display attributes.

- The **Replot** option opens a small window that allows you to change the following parameters. Clicking **Accept** replots the histogram to the newly set values.

  Min X Value
  : This minimum value, which is located on the left-most side of the histogram's x-axis, represents the worst quality elements.

  Max X Value
  : This maximum value, which is located on the right-most side of the histogram's x-axis, represents the highest quality that elements can achieve.

  Max Y height
  : You can adjust the number of elements that will be represented on the histogram's y-axis. Usually a value of 20 is sufficient. If there are too many elements displayed, it is difficult to discern the effects of smoothing.

  Num bars
  : This represents the number of subdivisions within the range between the **Min X** and **Max X** values. The default **Bars** have widths of 0.05. Increasing the number of displayed bars will decrease their width.

- The **Reset** option will return all of the values back to their original settings.
- **Show**: Click the left mouse button on any of the bars in the histogram to select elements that fall within that selected Quality range. If **Show** is enabled, the selected elements on the model will become visible in the main viewing window. The following options control how the selected elements are displayed.
- **Solid**: Enabling this option will display the elements as solid, rather than the default grid representation. (**Show** must be enabled.)
- **Color by Quality**: If available, enabling this option will display the elements in the same color as the selected Quality bar in the histogram. (**Show** must be enabled.)
- **Highlight**: If available, this option allows you to display one or two additional layers adjacent to the selected elements. (**Show** must be enabled.)
- **Subset**: Allows you to create a **Subset** containing only the elements chosen from the Quality histogram. The visibility of this subset is controlled by **Subset** in the **Display Tree**. The **Add select** option allows you to add elements to an already established subset.

Quality Metrics

This option allows you to modify the histogram display.

The histogram displays the overall quality of the mesh. The x-axis measures the quality, with 0 representing poor quality and 1 representing high quality. The y-axis measures the number of elements that belong within each quality sub-range.

For descriptions of all the quality metrics, refer to the Edit Mesh > Display Mesh Quality section in the Help Manual.

Advanced Options for Smoothing Mesh

Prism Warpage Ratio
Prisms are smoothed based on a balance between prism warpage and prism aspect ratio. Values from 0.01 to 0.50 favor improving the prism aspect ratio, while those from 0.50 to 0.99 favor improving prism warpage. A value of 0.5 favors neither. The farther the value is from 0.5, the greater the effect.

Stay on geometry
The default is, when a grid is smoothed, the nodes are restricted to the geometry -- surface, curves and points -- and can be moved only along the geometrical entities to which they are associated.

Violate Geometry
Enabling this option allows the smoothing operation to yield a higher quality mesh by violating the constraints of the geometry. The nodes can be moved off the geometry to obtain better mesh quality, as long as the movement remains within the absolute distance specified.

Relative Tolerance
This option works in the similar fashion as **Violate Geometry** except that the distance is relative here.

Allow refinement
If the quality of the mesh cannot be improved through normal algebraic smoothing, the **Allow refinement** option will allow the smoother to automatically subdivide elements to obtain further improvement. After smoothing with **Allow refinement** enabled, it may be necessary to smooth further with the option disabled. The goal is to reduce the number of elements that are attached to one vertex by refinement in problem regions.

Laplace smoothing
This option will solve the Laplace equation, which will generally yield a more uniformly spaced mesh.

Note

This can sometimes lead to a lower determinant quality of the prisms. Also, this option works only for the triangular surface mesh.

Allow node merging
This option will collapse and remove the worst tetra and prism elements when smoothing in order to obtain a higher quality mesh. This is enabled by default, and is often very useful in improving the grid quality.

Not just worst 1%
This option will smooth all of the geometry's elements to the assigned quality (specified under **Up to quality**) not just focus on the worst 1% of the mesh. Typically, when a mesh is smoothed, the smoother concentrates on improving the worst regions; this option will allow the smoother to continue smoothing beyond the worst regions until the desired quality is obtained.

Surface fitting
This option will smooth the mesh, keeping the nodes and the new mesh restricted to the surface of the geometry. Only **Hexa** models will utilize this option.

Ignore PrePoints

This option will allow the smoother to attempt to improve the mesh quality without being bound by the initial points of the geometry. This option is similar to the **Violate geometry** option, but works only for points located on the geometry. This option is available only when there are hexahedral elements in the model. Usually, the best way to improve the quality of grids that cannot be smoothed above a certain level is to concentrate on the surface mesh near the bad elements and edit this surface mesh to improve the quality.

Prism Mesh

Tetra meshing is not efficient for capturing shear or boundary layer physics. Prism mesh efficiently captures these effects near the surface while maintaining the ease and automation of Tetra mesh. Prism has always been necessary for CFD customers, but now that the option is more widely available, many other branches of CAE have started using prisms to better resolve the physics perpendicular to the surfaces of their models. With ANSYS ICEM CFD, Prism and Tetra generation is automatic and intelligent.

The spacing of the prism layers to capture the Y+ for Navier-Stokes mesh is the primary concern. The rate of volume change between cells is also important. Calculations are done between nodes or elements, and Prism mesh gives you more elements perpendicular to the surface. This is an efficient way to achieve better resolution (more calculations per unit distance) of the solution normal to the surface, without increasing the number of elements along the surface. This gives you a quicker and more accurate solution than can be achieved with a very fine tetra mesh.

The height and direction of the prism layer extrusion are calculated on an element by element basis and may vary due to global or local controls, or for improved quality. You may choose to set the initial height, number of layers and growth ratio, which are then used to determine the Prism height limit factor. Or you may prefer to set only the number of layers and growth ratio, which then allows Prism to adjust the initial height and locally optimize the volume transition between the prisms and tetras. Users concerned about Y+ can then adjust the first cell height using **Edit Mesh > Split Mesh > Split Prisms**.

Prism parameters are set globally, but can then be adjusted on a part by part or entity by entity basis. Entity settings override global settings, and between entities the smaller size overrides the larger. For instance, 3 layers could be set for a growth rate of 1.2 globally, but a certain part could be set for 5 layers. Setting a specific parameter on a single entity within that part or another part is handled intelligently. For example, if you set a local parameter such as height on a single curve entity, Prism will interpolate that parameter smoothly across the surface between curves.

You may notice that you can also select volume parts for prism. If no volume parts are selected, it will assume that you want to grow prisms into all volumes bordering the prism surfaces. If you select specific volume parts, then prism will be grown only into those volumes.

After each layer is extruded, smoothing is done according to the global settings. The layers are grown one at a time. This continues until all the requested layers are grown. You can add prisms to exiting layers or you can subdivide and redistribute layers at a later date. You can save time by growing only a few thicker layers and then subdividing them into many layers. The smoothing is the most time consuming part, so for simple configurations, it may be best to turn off all smoothing but grow all the layers one at a time. This allows you to take advantage of the variable height feature.

There are two ways to generate prism mesh:

Compute Mesh > Prism Mesh
This option is used to grow prism layers next to wall geometries. You can define the local initial height, growth ratio, and number of layers at **Mesh > Global Mesh Setup > Global Prism Settings**. This option can create prisms from existing volume or surface mesh.

Note

If the existing volume mesh is tet/hexa mesh, on the hexa side the prisms will be added within the first hexa layer.

Compute Mesh > Volume Mesh > Create Prism Layers
This option allows you to directly create tetra mesh with prisms next to wall geometries. You can choose whether to create prism mesh from the geometry and/or the surface mesh.

Note

- If many prism layers are needed, it is faster and can be more robust to create initial thick prism layers and then split them to create the total desired number of prism layers using **Edit Mesh > Split Mesh > Split Prisms**.
- You can compute a prism mesh without an input surface model loaded. Prism will generate a temporary faceted surface model from the input mesh.

Prism Mesh Process

The prism mesh process generates prism elements near boundary surfaces from tetrahedral volume or triangular surface mesh. This batch process creates prisms by extrusion of the surface mesh, and the resulting prisms are made conformal with any existing tetrahedral volume mesh. The prism mesh can be smoothed to yield the necessary quality.

Prism Mesh Preparation

When generating prism mesh, preparation is key. It is easier to edit a tetra mesh than a tetra prism mesh. Prism mesh can also be difficult to smooth, so it will save time to start with good quality tetra or tri surface mesh.

Start with the best possible initial hybrid mesh quality
Hybrid mesh is generally difficult to smooth.

Start with good Tetra or Tri surface mesh

- Choose prism options carefully
- Check aspect ratios / quality.
- Check and fix all diagnostics. Single/multiple edges, Non-manifold vertices, and Duplicate elements will crash the prism mesher.
- Visually scan the surface mesh. Look for any surface discrepancies or sharp tent-like structures in the mesh.

- Make sure part associations are correct.

  Look for a few elements of one part scattered among another part. Extruding from a few isolated elements will likely crash the prism mesher. Modify part assignments of such elements.

- Use the Smoothing Options for Tetra and Tri surface mesh under Mesh > Prism before creating prism mesh.

- Laplace Triangle Quality type is typically best for eventual prism quality.

Smoothing Tetra/Prism Mesh

First smooth the tetra and tri elements (set PENTA_6 to **Freeze**). Once the tetra and tri elements are as smooth as possible, then smooth all elements at the same time (including PENTA_6). Decrease the **Up to quality** value, so that the prism elements are not distorted too much.

Hexa

Hexa is a 3D object-based, semi-automatic, multi-block structured and unstructured, surface and volume mesher.

Introduction
Features of Hexa
Mesh Generation with Hexa
The Hexa Database
Intelligent Geometry in Hexa
Unstructured and Multi-block Structured Meshes
Blocking Strategy
Automatic O-grid generation
Edge Meshing Parameters
Smoothing Techniques
Refinement and Coarsening
Replay Functionality
Periodicity
Pre-Mesh Quality
Most Important Features of Hexa

Introduction

Hexa represents a new approach to hexahedral mesh generation. The block topology model is generated directly on the underlying CAD geometry. Within an easy-to-use interface, those operations most often performed by experts are readily accessible through automated features.

There is access to two types of entities during the mesh generation process in **Hexa**: block topology and geometry. After interactively creating a 3-D block topology model equivalent to the geometry, the block topology may be further refined through the splitting of edges, faces and blocks. In addition, there are tools for moving the block vertices, either individually or in groups, onto associated curves or CAD surfaces. You may also associate specific block edges with important CAD curves to capture important geometric features in the mesh.

Moreover, for models where you can take advantage of symmetry conditions, topology transformations such as translate, rotate, mirror and scaling are available. The simplified block topology concept allows rapid generation and manipulation of the block structure and, ultimately, rapid generation of the hexahedral meshes.

Hexa provides a projection-based mesh generation environment where, by default, all block faces between different materials are projected to the closest CAD surfaces. Block faces within the same material may also be associated to specific CAD surfaces to allow for definition of internal walls. In general, there is no need to perform any individual face associations to underlying CAD geometry, which further reduces the difficulty of mesh generation.

Features of Hexa

Some of the more advanced features of **Hexa** include:

O-grids: For very complex geometry, **Hexa** automatically generates body-fitted internal and external O-grids to parametrically fit the block topology to the geometry to ensure good quality meshes.

Edge-Meshing Parameters: Hexa's edge-meshing parameters offer unlimited flexibility in applying user specified bunching requirements.

Time Saving Methods: Hexa provides time saving surface smoothing and volume relaxation algorithms on the generated mesh.

Mesh Quality Checking: With a set of tools for mesh quality checking, elements with undesirable skewness or angles may be displayed to highlight the block topology region where the individual blocks need to be adjusted.

Mesh Refinement/Coarsening: Refinement or coarsening of the mesh may be specified for any block region to allow a finer or coarser mesh definition in areas of high or low gradients, respectively.

Replay Option: Replay file functionality enables parametric block topology generation linked to parametric changes in geometry.

Symmetry: As necessary in analyzing rotating machinery applications, for example, **Hexa** allows you to take advantage of symmetry in meshing a section of the rotating machinery thereby minimizing the model size.

Link Shape: This allows you to link the edge shape to existing deforming edge. This gives better control over the grid specifically in the case of parametric studies.

Adjustability: Options to generate 3D surface meshes from the 3D volume mesh and 2D to 3D block topology transformation.

Mesh Generation with Hexa

To generate a mesh within **Hexa** you need to:

- Import a geometry file using any of the direct, indirect or faceted data interfaces.
- Interactively define the block model through split, merge, O-grid definition, edge/face modifications and vertex movements.
- Check the block quality to ensure that the block model meets specified quality thresholds.
- Assign edge meshing parameters such as maximum element size, initial element height at the boundaries and expansion ratios.
- Generate the mesh with or without projection parameters specified. Check the **Mesh quality** to ensure that specified mesh quality criteria are met.
- Write **Output** files to the desired solvers.

If necessary, you may always return to previous steps to manipulate the blocking if the mesh quality does not meet the specified threshold or if the mesh does not capture certain geometry features. The blocking may be saved at any time, thus allowing you to return to previous block topologies.

Additionally, at any point in this process, you can generate the mesh with various projection schemes such as full face projection, edge projection, point projection or no projection at all.

Note

In the case of no projection, the mesh will be generated on the faces of the block model and may be used to quickly determine if the current blocking strategy is adequate or if it must be modified.

The Hexa Database

The Hexa database contains both geometry and block topology data, each containing several sub-entities.

The Geometric Data Entities:

- **Points:** x, y, z point definition
- **Curves:** trimmed or untrimmed NURBS curves
- **Surfaces:** NURBS surfaces, trimmed NURBS surfaces

The Block Topologic Data Entities:

- **Vertices:** corner points of blocks, of which there are at least eight, that define a block
- **Edges:** a face has four edges and a block twelve
- **Faces:** six faces make up a block
- **Blocks:** volume made up of vertices, edges and faces

Intelligent Geometry in Hexa

Using **ANSYS ICEM CFD's** Direct CAD Interfaces, which maintain the parametric description of the geometry throughout the CAD model and the grid generation process, hexahedral grids can be easily remeshed on the modified geometry.

The geometry is selected in the CAD system and tagged with information (made intelligent) for grid generation such as boundary conditions and grid sizes, and this intelligent geometry information is saved with the master geometry.

In **Hexa**, by updating all entities with the update projection function, blocking vertices projected to prescribed points in the geometry are automatically adapted to the parametric change and one can recalculate the mesh immediately. Additionally, with the use of its **Replay** functionality, **Hexa** provides complete access to previous operations.

Unstructured and Multi-block Structured Meshes

The mesh output of **Hexa** can be either unstructured or multi-block structured, and need not be determined until after you have finished the whole meshing process when the output option is selected.

Unstructured Mesh Output

The unstructured mesh output option will produce a single mesh output file where all common nodes on the block interfaces are merged, independent of the number of blocks in the model.

Multi-Block Structured Mesh Output

Used for solvers that accept multi-block structured meshes, this output option will produce a mesh output file for every block in the topology model. For example, if the block model has 55 blocks, there will be 55 output files created in the output directory.

Additionally, without merging any of the nodes at the block interfaces, the Output Block option allows you to minimize the number of output files generated with the multi-block structured approach.

Blocking Strategy

With **Hexa**, the basic steps necessary to generate a hexahedral model are the same, regardless of model complexity. The blocking topology, once initialized, can then be modified by splitting and merging the blocks, as well as through the use of an operation called **O-grid** (Refer to the next section). While these operations are performed directly on the blocks, the blocks may also go through indirect modification by altering the sub-entities of the blocks (i.e.: the vertices, edges, faces).

Upon initialization, **Hexa** creates one block that encompasses the entire geometry. The subsequent operations under the **Blocking** menu of developing the block model, referred to as "blocking the geometry," may be performed on a single block or across several blocks.

Note

The topologic entities in Hexa are color-coded based on their properties.

Colors of Edges:

White Edges and Vertices

These edges are between two material volumes. The edge and the associated vertices will be projected to the closest CAD surface between these material volumes. The vertices of these edges can move only on the surfaces.

> **Note**
>
> These display in black if you have chosen a light colored background for the graphics display window.

Blue Edges and Vertices

These edges are in the volume. The vertices of these edges, also blue, can be moved by selecting the edge just before it and can be dragged on that edge.

Green Edges and Vertices

These edges and the associated vertices are being projected to curves. The vertices can be moved only on the curve(s) to which they are being projected.

Red Vertices

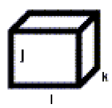
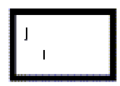
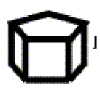
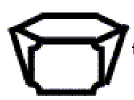
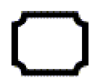
These vertices are projected to prescribed points.

Hexa Block Types

When blocking a model, it is important to note that the block type affects many operations within Hexa and the entire approach to mesh generation. For example, if you split a model with mapped blocks, the split will propagate through faces that have a mapped relationship on the opposite side. For free blocks, a split will terminate at the free face. Similarly, if you set edge parameters on a mapped face edge, opposite edges will have a similar number of nodes. If however, that edge is attached to a free face, the number of nodes on the opposite side will not be adjusted. Using this free/mapped relationship, you can shape the blocking and resulting mesh.

The ability to convert blocks from free to mapped or vice versa imposes constraints on the blocking and resulting mesh. By imposing more constraints, you can enforce a greater number of hexa elements, while reducing the constraints can sometimes improve mesh transitioning.

Figure 26: Hexa Block Types

Dimension	3D	2D
Mapped / Structured	 i, j, k are mapped	 i, j are mapped
Swept	 j is mapped	N/A
Free/Unstructured	 All edges are free from mapping	 All edges are free from mapping

Split

The **Split** function, which divides the selected block interactively, may be applied across the entire block or to an individual face or edge of a block by using the **Split face** or **Split edge** options, respectively. Blocks may be isolated using the **Index control**.

Merge

The **Merge** function works similar to split blocks; one can either merge the whole block or merge only a face or an edge of the block.

While some models require a high degree of blocking skill to generate the block topology, the block topology tools in **Hexa** allow you to quickly become proficient in generating a complex block model.

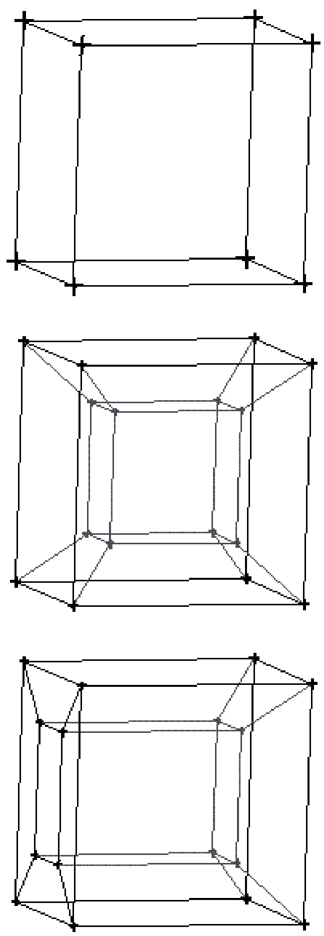
Automatic O-grid generation

Generating **O-grids** is a very powerful and quick technique used to achieve a quality mesh to model geometry when the you desire a circular or "O"-type mesh either around a localized geometric feature or globally around an object. This process would not have been possible without the presence of O-grids.

Using the Automatic O-grid

The **O-grid** creation capability is simply the modification of a single block or blocks to a 5 sub-block topology as shown below. There are several variations of the basic **O-grid** generation technique and the **O-grid** shown below is created entirely inside the selected block.

Figure 27: Initial block, Block with O-Grid, O-Grid with Add Face



Using the **Add face** option, an **O-grid** may be created such that the **O-grid** passes through the selected block face(s). In Figure 27: Initial block, Block with O-Grid, O-Grid with Add Face (p. 42), the **Add Face** option was used on the last block to include one face on the block prior to generating the **O-grid**.

Important Features of an O-grid

Generation of Orthogonal Mesh Lines at an Object Boundary

The generation of the **O-grid** is fully automatic and you simply select the blocks needed for **O-grid** generation. The **O-grid** is then generated either inside or outside the selected blocks. The **O-grid** may be fully contained within its selected region, or it may pass through any of the selected block faces.

Rescaling an O-grid After Generation

When the **O-grid** is generated, its size is scaled based upon a factor in the **Blocking** > **O-grid parameter** window. The **Re scale O-grid** option allows you to re-scale the previously generated **O-grid.**If a value less than 1 is assigned, the resulting O-grid will be smaller than the original. Conversely, a value larger than 1, will result in a larger O-grid.

The blocks may also be modified by moving the vertices of the blocks and by defining specific relationships between the faces, edges and vertices to the geometry.

Edge Meshing Parameters

The edge meshing parameter task has been greatly automated to provide you with unlimited flexibility in specifying bunching requirements. Assigning the edge meshing parameters occurs after the development of the block topology model. This option is accessible by selecting **Meshing > Edge params**.

You can use the following pre-defined bunching laws or **Meshing laws**:

- Default (Bi-Geometric Law)
- Uniform
- Hyperbolic
- Poisson
- Curvature
- Geometric 1
- Geometric 2
- Exponential 1
- Exponential 2
- Bi-Exponential
- Linear
- Spline

You may modify these existing laws by applying pre-defined edge meshing functions, accessible through the **Meshing > Edge Params > Graphs** option in **Hexa**.

This option yields these possible functions:

- Constant
- Ramp
- S curve
- Parabola Middle
- Parabola Ends
- Exponential
- Gaussian
- Linear

- Spline

Note

By selecting the **Graphs** option, you may add/delete/modify the control points governing the function describing the edge parameter settings. Additional tools such as **Linked Bunching** and the multiple **Copy** buttons provide you with the ability to apply the specified edge bunching parameters quickly to the entire model.

Smoothing Techniques

In **Hexa**, both the block topology and the mesh may be smoothed to improve the overall block/mesh quality either in a certain region or for the entire model. The block topology may be smoothed to improve the block shape prior to mesh generation. This reduces the time required for development of the block topology model.

The geometry and its associative surfaces, curves, and points are all constraints when smoothing the block topology model. Once the block topology smoothing has been performed, you may smooth the mesh after specifying the proper edge bunching parameters.

The quality criteria for smoothing are described in the Help Manual, under Blocking > Pre-Mesh Quality.

Refinement and Coarsening

The refinement function, which is found through **Blocking > Pre-Mesh Params > Refinement**, can be modified to achieve either a refined or a coarsened result. The refinement/coarsening may be applied in all three major directions simultaneously, or they may be applied in just one major direction.

Refinement

The refinement capability is used for solvers that accept non-conformal node matching at the block boundaries. The refinement capability is used to minimize the model size, while achieving proper mesh definition in critical areas of high gradients. Entering a scale factor greater than 1 will result in refinement.

Coarsening

In areas of the model where the flow characteristics are such that a coarser mesh definition is adequate, coarsening of the mesh may be appropriate to contain model size. Entering a scale factor less than 1 will result in coarsening.

Replay Functionality

Parametric changes made to model geometry are easily applied through the use of Hexa's replay functionality, found in **File > Replay Scripts**. Changes in length, width and height of specific geometry features are categorized as parametric changes. These changes do not, however, affect the block topology. Therefore, the **Replay** function is capable of automatically generating a topologically similar block model that can be used for the parametric changes in geometry.

If any of the Direct CAD Interfaces are used, all geometric parameter changes are performed in the native CAD system.

You can also use variables in the replay script to parameterize edge parameters. Refer to Using Variables in the Replay Script in the User's Guide for details.

Generating a Replay File

The first step in generating a **Replay** file is to activate the recording of the commands needed to generate a custom meshing process. All of the steps in the mesh development process are recorded, including blocking, mesh size, edge meshing, boundary condition definition, and final mesh generation. The next step in the process is to make the parametric change in the geometry and then replay the recorded file on the changed geometry. All steps in the mesh generation process are automated from this point.

Advantage of the Replay Function

With the **Replay** option, you may analyze more geometry variations, thus obtaining more information on the critical design parameters. This can yield optimal design recommendations within the project time limits.

Using Variables in the Replay Script

You can use variables in the replay script as a means to parameterize edge parameters. An example of the use of variables in a replay script is as follows:

```
#variables
set n 10
set h1 0.01
set r1 1.2
ic_load_tetin myfile.tin
ic_hex_surface_blocking -inherited -swept -min_edge 0.0
ic_geo_new_family SOLID
ic_hex_twod_to_threed SOLID -swept
ic_hex_set_mesh 19 18 n $n h1 $h1 h2rel 0.0 r1 $r1 r2 2 lmax 0 default unlocked
ic_hex_create_mesh SURFS SOLID proj 2 dim_to_mesh 3
ic_hex_write_file hex.uns SURFS SOLID proj 2 dim_to_mesh 3 -family_boco family_boco.fbc
ic_uns_load hex.uns 3 0 {} 2
```

The variables for the edge parameters are set at the top of the replay file. Within the script, the '$' indicates a variable. To parameterize the edge parameters, you may update the variables at the top of the script and then rerun the script.

Periodicity

Periodic definition may be applied to the model in **Hexa**. The **Periodic nodes** function, which is found under **Blocking > Periodic nodes**, plays a key role in properly analyzing rotating machinery applications, for example. Typically, you will model only a section of the rotating machinery, as well as implement symmetry, in order to minimize the model size. By specifying a periodic relationship between the inflow and outflow boundaries, the particular specification may be applied to the model—flow characteristics entering a boundary must be identical to the flow characteristics leaving a boundary.

Applying the Periodic Relationship

The periodic relationship is applied to block faces and ensures that a node on the first boundary have two identical coordinates to the corresponding node on the second boundary. You will be prompted to select corresponding vertices on the two faces in sequence. When all vertices on both flow boundaries have been selected, a full periodic relationship between the boundaries has been generated.

Pre-Mesh Quality

The pre-mesh quality functions are accessible through **Blocking > Pre-Mesh Quality**. Applying any of the quality checks will yield a histogram plot.

Determining the Location of Elements

By clicking on any of the histogram bars with the left button, you may determine where in the model these elements are located. The selected histogram bars will be highlighted by a change in color. After selecting the bar(s), the **Show** option is selected to highlight the elements in this range. If the **Solid** option is enabled, the elements marked in the histogram bars will be displayed with solid shading.

Some of the quality metrics are explained below.

Determinant
Angle
Volume
Warpage

Determinant

The **Determinant** check computes the deformation of the elements in the mesh by first calculating of the Jacobian of each hexahedron and then normalizing the determinant of the matrix. A value of 1 represents a perfect hexahedral cube, while a value of 0 is a totally inverted cube with a negative volume.

The mesh quality, measured on the x-axis, of all elements will be in the range from 0 to1. If the determinant value of an element is 0, the cube has one or more degenerated edges. In general, determinant values above 0.3 are acceptable for most solvers. The subdivision across the quality range is determined by the number of assigned **Bars**

The y-axis measures the number of elements that are represented in the histogram. This scale ranges from 0 to a value that is indicated by the **Height**.

Angle

The **Angle** option checks the maximum internal angle deviation from 90 degrees for each elements. Various solvers have different tolerance limits for the internal angle check. If the elements are distorted and the internal angles are small, the accuracy of the solution will decrease. It is always wise to check with the solver provider to obtain limits for the internal angle threshold.

Volume

The **Volume** check will compute the internal volume of the elements in the model. The units of the volume will be displayed in the unit that was used to create the model.

Warpage

The **Warpage** check will yield a histogram that indicates the level of element distortion. Nodes that are in-plane with one another will produce an element with small warpage. Nodes that make elements twisted or distorted will increase a element's distortion, giving a high degree of warpage. The y-axis is the scale for the number of elements represented in the histogram - a value determined by the assigned

Height. The x- axis, which ranges from a **Min** of 0 to a **Max** of 90, is the degree of warpage that an element experiences.

Most Important Features of Hexa

Hexa has emerged as the quickest and most comprehensive software for generating large, highly accurate, 3D geometry-based hexahedral meshes. Now, in the latest version of Hexa, it is also possible to generate 3D surface meshes with the same speed and flexibility.

- CAD- and projection-based hexahedral mesh generation
- Easy manipulation of the 3D object-based topology model
- Modern GUI and software architecture with the latest hexahedral mesh technology
- Extensive solver interface library with over 100 different supported interfaces
- Automatic O-grid generation and O-grid re-scaling
- Geometry-based mesh size and boundary condition definition
- Mesh refinement to provide adequate mesh size in areas of high or low gradients
- Smoothing/relaxation algorithms to quickly yield quality meshes
- Generation of multi-block structured, unstructured, and super- domain meshes
- Ability to specify periodic definitions
- Extensive replay functionality with no user interaction for parametric studies
- Extensive selection of mesh bunching laws including the ability to graphically add/delete/modify control points defining the graph of the mesh bunching functions
- Link bunching relationships between block edges to automate bunching task
- Topology operations such as translate, rotate, mirror, and scaling to simplify generation of the topology model
- Automatic conversion of 3D volume block topology to 3D surface mesh topology
- Automatic conversion of 2D block topology to 3D block topology
- Block face extrusion to create extended 3D block topology
- Multiple projection options for initial or final mesh computation
- Quality checks for determinant, internal angle and volume of the meshes
- Domain renumbering of the block topology
- Output block definition to reduce the number of multi-block structured output mesh files
- Block orientation and origin modification options

Properties

The **Properties** menu allows you to create different materials by specifying material or element properties, such as type, the Young's Modulus, and Poisson's ratio. Once the material is created, you can apply those properties to the elements.

Create Material Property
Save Material
Open Material
Define Table
Define Element Properties

Create Material Property

You can define a material by specifying a name of the material, define whether isotropic, enter in values for Young's Modulus, Shear modulus, Poisson's ratio, Mass Density, and Thermal expansion coefficient.

Save Material

The **Write Material File** option allows you to save the material specification so that it may be reused whenever necessary. The material file will be saved with the .mat extension.

Open Material

The **Load Material From File** option allows you to open a material file to be used in your design, or modified and saved for future use.

Define Table

The **Create Table** option allows you to create your material property empirically by entering values for x and y ,You may even visualize the graph of the property.

Define Element Properties

These options allow you to apply your material properties to their respective elements. Different types of elements that can be defined include: Point, Line, Shell, and Volume. After choosing the part, the various properties are applied to its elements.

Constraints

From the **Constraints** tab, you can define the motion restrictions on different entities such as points, curves, surfaces, or subsets, as well as define other options such as **Contact definition, Velocity** and **Rigid Wall.**

Create Constraint / Displacement
Define Contact
Define Single Surface Contact
Define Initial Velocity
Define Planar Rigid wall

Create Constraint / Displacement

This option allows you to apply a directional or rotational constraint on an entity, in any direction.

Define Contact

This option allows you to define contacts by Automatic Detection or Manual Definition.

Define Single Surface Contact

This is mainly used for LS-DYNA Solver, where you can pick the contact surface.

Define Initial Velocity

This allows you to define initial nodal point velocities by specifying the translational and rotational velocity for nodal sets.

Define Planar Rigid wall

You can define a Planar Rigid Wall by specifying the Head and Tail coordinates.

Loads

In this tab, there are several options available for applying internal and external loads, such as force, pressure, and temperature. How ICEM CFD treats the load depends on how the load is applied - to a curve, a surface, or a mesh. In all cases, the load information is not calculated until the user is creating the output files for one of the supported "Common Structural" solvers, – ANSYS, Autodyn, LS-DYNA, ABAQUS, or NASTRAN. That is, the output file is generated thru the Solve Options Tab, and at this time, the loads will be written out according to the selected solver's published format.

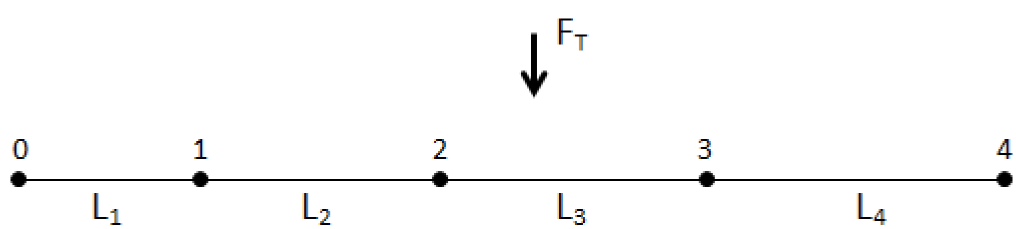
Theory

Applied forces are distributed as follows.

By Curve

The total Force '$\mathbf{F_T}$' may be applied on a curve as shown below. Applying the load to a geometry entity simplifies the process for you and keeps the load information at the geometry level so the mesh can be regenerated without losing the setup information.

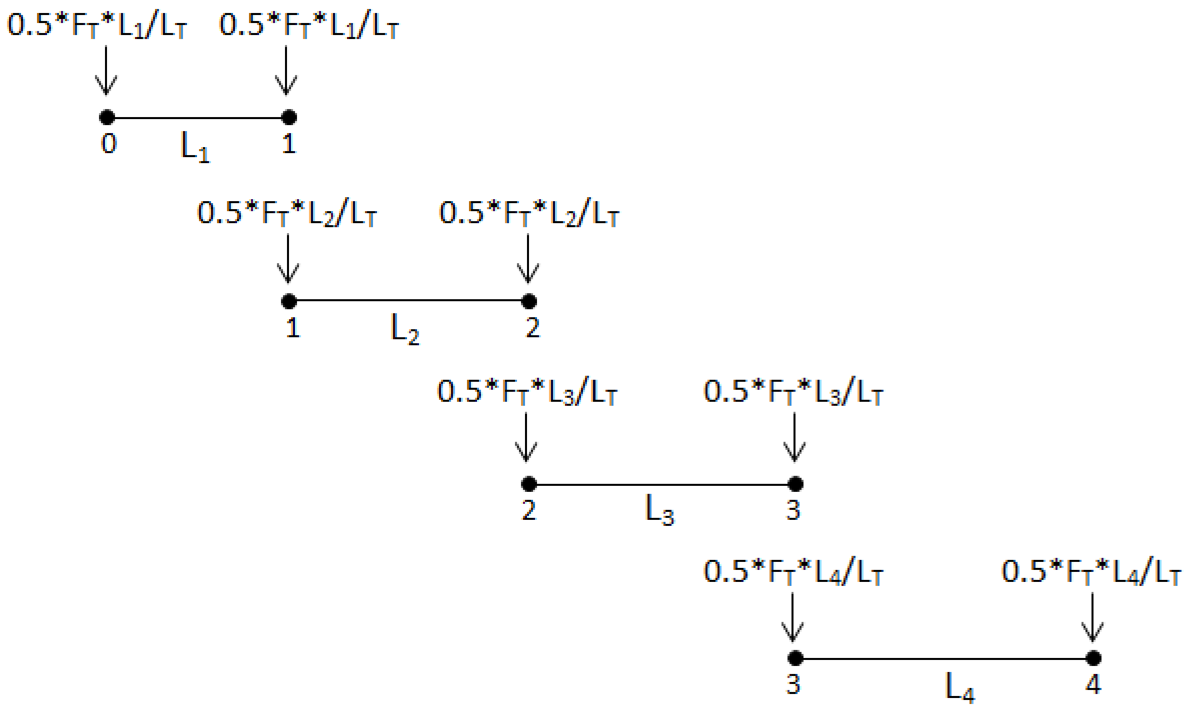
Figure 28: Force on a Curve



Nodes are numbered 0, 1, 2, and so on. Elements between the nodes have lengths L_1, L_2 and so on. The total length is L_T.

Then the force on the Nodes, as per FEA concepts, is distributed linearly in proportion to the Element length as shown in the figure below.

Figure 29: Linear Force Distribution



For a Linear distribution, the load at each node is calculated as follows.

Node 0: $F_0 = 0.5 * F_T * (L_1 / L_T)$

Node 1: $F_1 = 0.5 * F_T * (L_1 / L_T) + 0.5 * F_T * (L_2 / L_T)$

Node 2: $F_2 = 0.5 * F_T * (L_2 / L_T) + 0.5 * F_T * (L_3 / L_T)$

Node 3: $F_3 = 0.5 * F_T * (L_3 / L_T) + 0.5 * F_T * (L_4 / L_T)$

Node 4: $F_4 = 0.5 * F_T * (L_4 / L_T)$

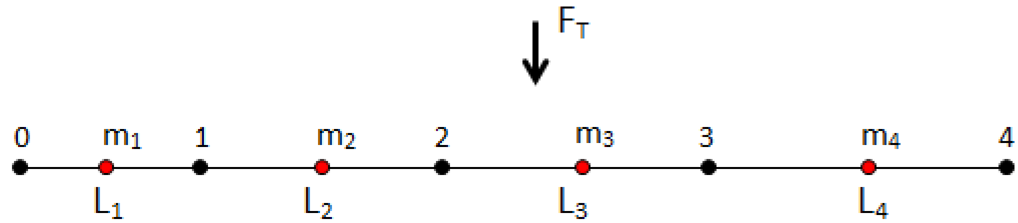
In general, the force at any Node is: $\mathbf{F_i = Sum\ [\ F_T * (L_j / L_T) * (1 / N_j)\ ]}$, where *i* is the node number, *j* is the element number, L_j is the length of element j, and N_j is the number of Nodes attached to element j.

As a check, if you add the individual node forces, $\mathbf{F_0 + F_1 + F_2 + F_3 + F_4}$, then the result equals $\mathbf{F_T}$.

By Mesh Elements

You may also apply loads directly to the elements (select the elements directly rather than the geometry entities).This may occur, for example, if you don't have the geometry in a particular model or area of a model. In this case, the load is distributed element-by-element using a **Quadratic** distribution as shown.

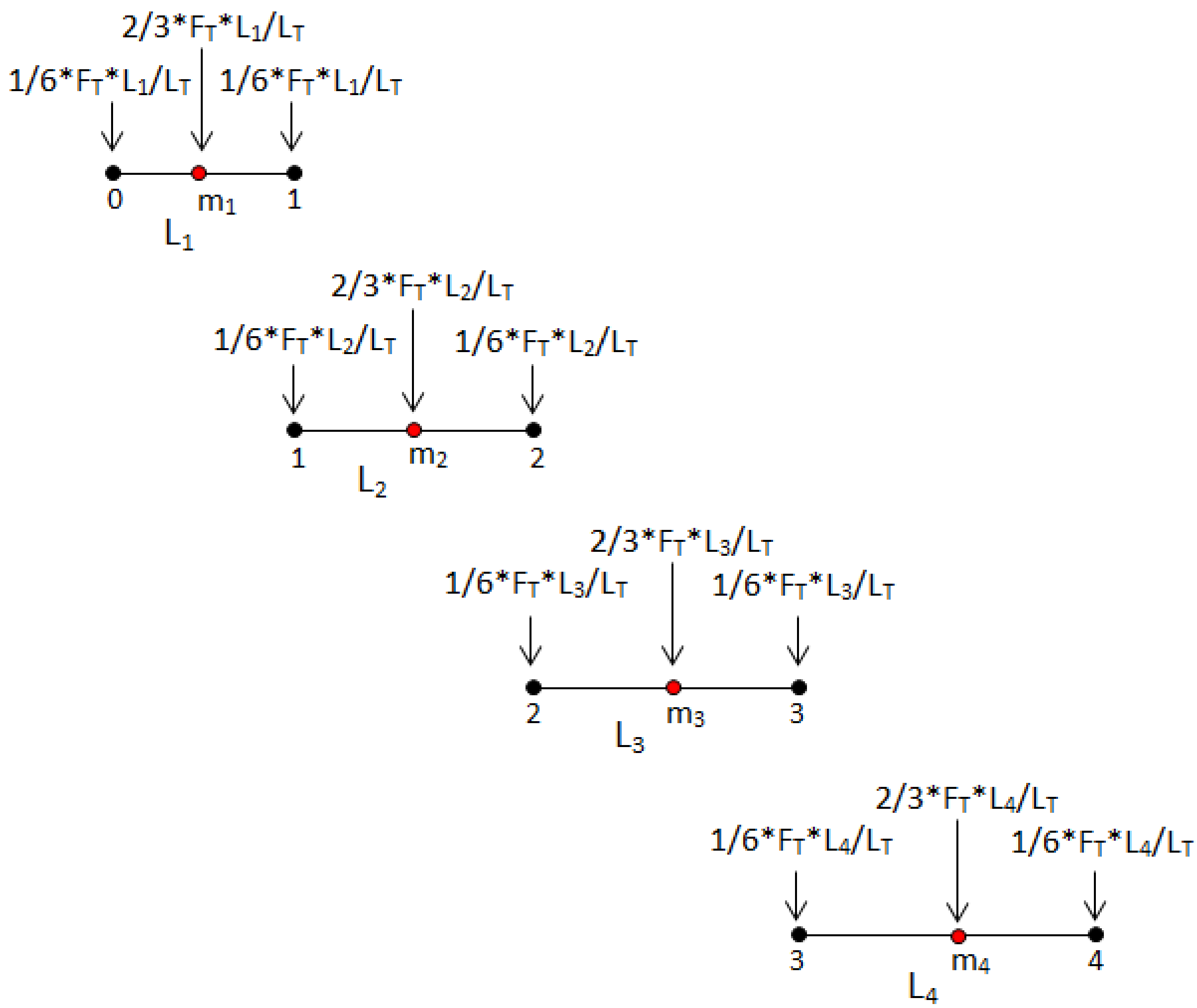
Figure 30: Force on Elements with mid-side nodes



Node numbers and element lengths are defined as before. In addition, each element has a mid-side node labelled m_1, m_2, and so on.

The Quadratic Load distribution, as per FEA concepts on an element-by-element basis is shown in Figure 31: Quadratic Load Distribution (p. 55).

Figure 31: Quadratic Load Distribution



The distribution of the Total Force, F_T, at the boundary nodes is as follows:

Node 0: $F_{0q} = 1/6 * F_T * (L_1 / L_T)$

Node 1: $F_{1q} = 1/6 * F_T * (L_1 / L_T) + 1/6 * F_T * (L_2 / L_T)$

Node 2: F_{2q} = 1/6 * F_T * (L_2 / L_T) + 1/6 * F_T * (L_3 / L_T)

Node 3: F_{3q} = 1/6 * F_T * (L_3 / L_T) + 1/6 * F_T * (L_4 / L_T)

Node 4: F_{4q} = 1/6 * F_T * (L_4 / L_T)

And the distribution at mid-side nodes is:

Node m_1: F_{m1} = 2/3 * F_T * (L_1 / L_T)

Node m_2: F_{m2} = 2/3 * F_T * (L_2 / L_T)

Node m_3: F_{m3} = 2/3 * F_T * (L_3 / L_T)

Node m_4: F_{m4} = 2/3 * F_T * (L_4 / L_T)

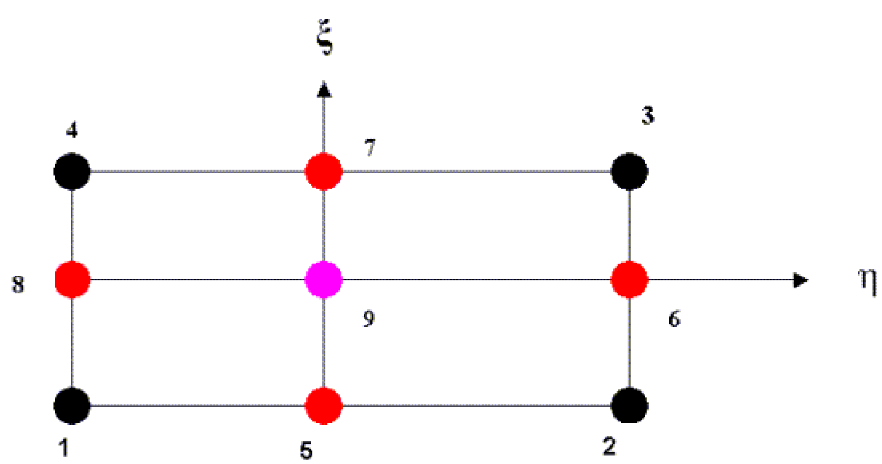
As in the previous case of linear distribution, you can check the total Force by adding the individual nodal forces:

$\mathbf{F_T} = F_{0q} + F_{1q} + F_{2q} + F_{3q} + F_{4q} + F_{m1} + F_{m2} + F_{m3} + F_{m4}$.

By Surface

If you choose to set a load on a surface entity, then the load distribution follows the Nine Node, Two Dimension, Lagrange distribution. See the Figure 32: QUAD 9 Element (p. 56) for an illustration.

Figure 32: QUAD 9 Element



Nine nodes are identified: N_1-N_4 at the corners; N_5-N_8 at the mid-sides; and N_9 in the middle. The symbols ξ and η define a local coordinate system for the development of the load distribution, and vary from −1 to +1 over the surface of the element.

The Shape function for the distribution at a Corner Node is:

$$N_I(\xi, \eta) = \frac{1}{4} \cdot \xi \cdot \eta \cdot \left(\xi - 1\right) \cdot \left(\eta - 1\right)$$

At a Mid-Side Node, the Shape function for the distribution is:

$$N_5\,(\xi\,,\eta)\;=\frac{1}{2}\cdot\eta\cdot\left(1-\xi^2\right)\cdot\left(\eta-1\right)$$

And finally, the Shape function for the Middle Node load distribution is:

$$N_9\,(\xi\,,\eta)\;=\;(1-\xi^2)\;\cdot\;(1-\eta^2)$$

Suppose a Force **F** is uniformly distributed over the whole Area. Then the pressure is **P = F / 4**. (Because in the ξ-η coordinate system the area of the surface is 4.)

To find the Consistent Load at each Node, we must integrate the Shape function over the surface area:

Consistent Load at Node 1 = L_1:

$$L_1 = P\cdot\int_{\xi\,=\,-1}^{\xi\,=\,1}\int_{\eta\,=\,-1}^{\eta\,=\,1} N_1 d\xi d\eta = \frac{4P}{36}$$

Consistent Load at Node 5 = L_5:

$$L_5 = P\cdot\int_{\xi\,=\,-1}^{\xi\,=\,1}\int_{\eta\,=\,-1}^{\eta\,=\,1} N_5 d\xi d\eta = \frac{4P}{9}$$

Consistent Load at Node 9 = L_9:

$$L_9 = P\cdot\int_{\xi\,=\,-1}^{\xi\,=\,1}\int_{\eta\,=\,-1}^{\eta\,=\,1} N_9 d\xi d\eta = \frac{16P}{9}$$

Now F = 4 P,.

Substituting this value in the above equations we get the Consistent Load as

L_1 = F / 36

L_5 = F / 9

L_9 = 4F / 9

By symmetry, the **Consistent Load** on all corner nodes N_1, N_2, N_3, and N_4 are equal.

Similarly, the **Consistent Load** on all mid-side nodes N_5, N_6, N_7, and N_8 are equal.

As with the Linear and Quadratic distributions, a check against the Total can be performed.

The sum of the Consistent Loads = 4 * L_1 + 4 * L_5 + L_9

= F / 9 + 4F / 9 + 4F / 9

$= F_T$

Note

A similar process for calculating the Consistent Load on a QUAD8 element load distribution is available. Again, the output file is generated thru the "Solve Options" Tab, and during output, the distributed load information will be written out according to the selected solver's published format.

Force

Using this option, you can apply translational (force) or rotational (moment) loads on entities in all three directions.

Forces can be applied by two different options. The Uniform option applies the stated force at all selected entities. For example with curves, the Uniform option will apply the full force to all nodes attached to the curve. The Total option means that the force gets distributed among all the nodes of the selected entities according to FEA concepts.

Pressure

You can apply pressure loads to surfaces, subsets, or parts. See the Help for more detail.

Temperature

This option allows you to apply temperature to points, curves, surfaces, bodies, and subsets.See the Help for more Detail.

Solve Options

Setup Solver Parameters has options for specifying the solver parameters. You can also specify the analysis solution parameters.

Setup Solver Parameters

You can select from the following solvers: **ANSYS**, **Nastran**, **ABAQUS**, **Autodyn**, and **LS-DYNA**.

Setup Analysis Type

Depending on the selected solver, different options are available. For the ANSYS solver, you can select either Structural or Thermal. If Nastran solver is selected, then you have the choice of more **Analysis** types.

Setup Sub-Case

You can create subcases to apply the load in different steps.

Write/View Input file

You can create and view the input file generated for the solver.

Submit Solver Run

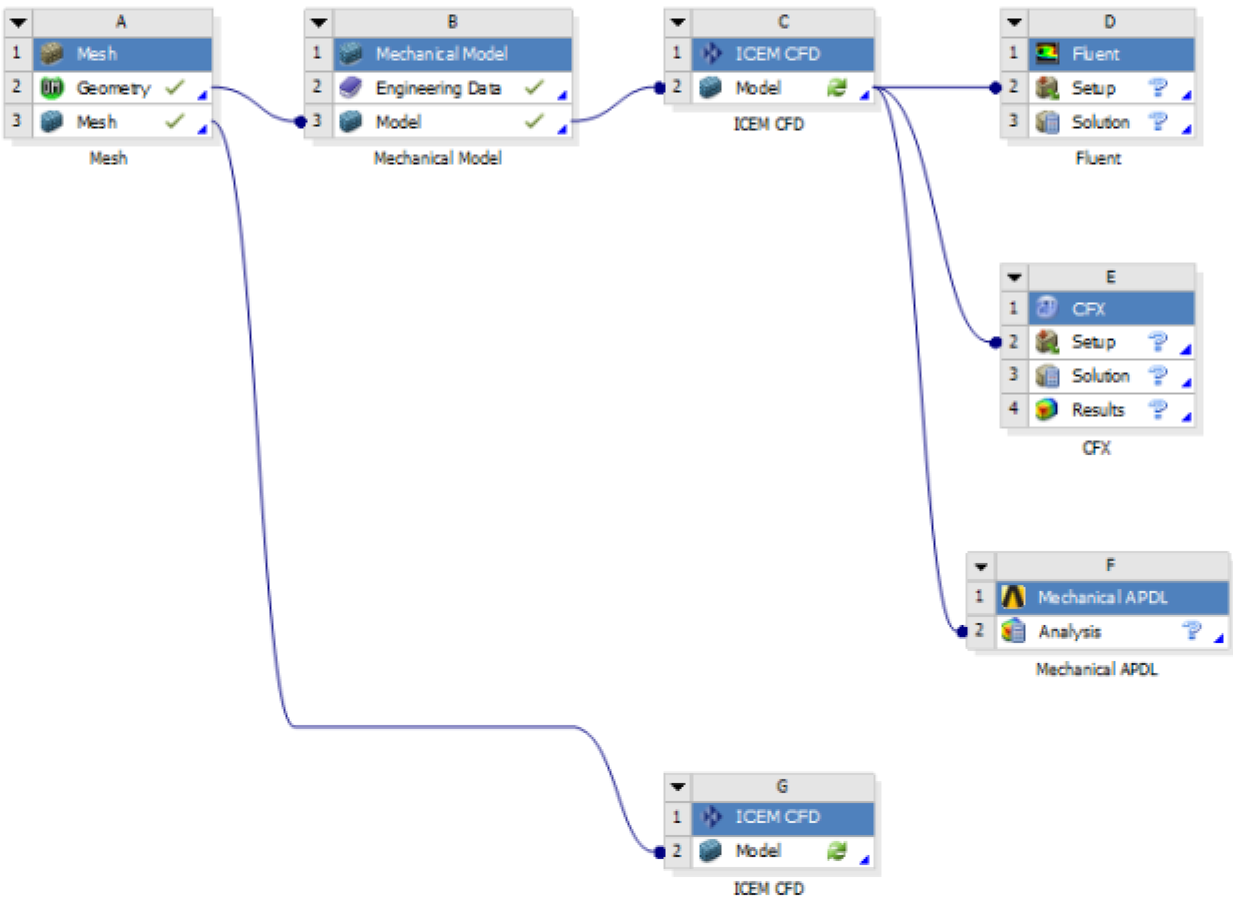
Using this option, you can solve the input file generated for a particular solver.

FEA Solver Support

More information about the supported solvers is available from the **Help** menu. The **Output Interfaces** option opens the ANSYS ICEM CFD Output Interfaces information in a browser. For information about a specific solver, refer to the Table of Supported Solvers and click the name of the solver.

Workbench Integration

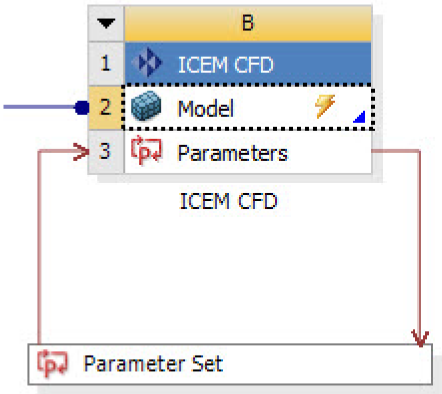
The data-integrated ICEM CFD component system, or "ICEM CFD Add-in", enables you to launch ICEM CFD from ANSYS Workbench and use it to build a project, with the option of adding upstream data from Geometry, Mesh, Mechanical Model, or combined Geometry and Mesh system components. You can also use ICEM CFD to provide data to downstream component systems, such as ANSYS Fluent, ANSYS CFX, ANSYS Polyflow, Mechanical APDL, and FE Modeler.



The additional features available through the ICEM CFD add-in are described in the following sections:

- Elements of the ICEM CFD Component (p. 62)
- Creating an ICEM CFD Component (p. 63)
- Updating ICEM CFD Projects (p. 64)
- Interface Differences in the Data-Integrated ICEM CFD (p. 67)
- Setting Parameters (p. 69)
- User-Defined Parameters Example (p. 75)

Elements of the ICEM CFD Component



The ICEM CFD Component system contains the following cells:

- **ICEM CFD system header**. The System Header identifies the component type and provides access to Workbench context menu options. The ICEM CFD system header context menu options include:
  - **Refresh**
  - **Update**
  - **Duplicate**
  - **Delete**
  - **Rename**
  - **Properties**
  - **Add/Edit Note**

These standard actions are described in the *System Header Context Menu Options* and *Duplicating, Moving, Deleting, and Replacing Systems* sections of the *Workbench User's Guide*.

Note

If available, **Update** will use the **ICEM CFD Replay** file to update the ICEM CFD project.

Note

Scripts written in ICEM CFD may not be parametric with upstream or downstream projects. Care should be taken to write scripts whose functions do not exceed the capabilities of the upstream or downstream component systems.

- **Model cell**. The Model Cell is associated with the ICEM CFD application. You can use the Model Cell to modify some aspects of the project. You can also double-click the Model Cell to open the project in ICEM CFD. The Model cell context menu items include the following:
  - **Edit**: Opens the ICEM CFD application and loads an existing Geometry/ICEM CFD file.

- **Duplicate**: Copies the entire geometry and mesh data, enabling you to edit the Model cell in the duplicate system to investigate an alternative modeling approach.
- **Transfer Data From New**: Enables the transfer of data from upstream Geometry, Mesh, Mechanical Model, or combined Geometry and Mesh components.

Note

If the geometry cell option **Use Associativity** is **On** (its default setting), Part Reference IDs are stored in a Workbench database and remain persistent in ICEM CFD even if the topology of the geometry is changed.

- **Transfer Data to New**: Enables the transfer of data from an ICEM CFD project to downstream data-integrated system projects, such as:

  → Fluent

  → CFX

  → FE Modeler

  → Polyflow

  → Mechanical APDL

- **Update**, **Refresh**, **Reset**, **Rename**, **Properties**, and **Add/Edit Note**. These standard actions are described in *System Header Context Menu Options* and *Duplicating, Moving, Deleting, and Replacing Systems* sections of the *Workbench User's Guide*.

  An additional property is **Create Subset(s) from Named Selection.** If this option is set, then overlapping named selections are transferred to multiple subsets instead of to exclusive parts. This enables the creation of geometry subsets instead of parts, allowing you to decide which part the geometry entity (point/curve/surface) should be associated to. When this option is disabled each geometry is assigned to only one part, thereby losing association with other Named Selections (which are mapped to parts).

- **Parameters cell (optional)**. The Parameters Cell enables you to see and edit Input and Output parameters for ICEM CFD.

Creating an ICEM CFD Component

You can create an ICEM CFD component system in Workbench using any of these methods:

- Double-click the ICEM CFD system template in the **Toolbox**.
- Drag-and-drop the ICEM CFD system template onto the **Project Schematic**.
- Right-click on a Geometry or Mesh project and select **Transfer Data to New> ICEM CFD**.
- Choose **File**>**Import** to import a standalone ICEM CFD project into Workbench. When the **Import** dialog opens, choose ICEM CFD Project File (*.prj) and navigate to the project you want to import.

Updating ICEM CFD Projects

Updating a project in Workbench brings the entire ICEM CFD system up to the most current status, including upstream and downstream data.

Changes that require you to update your ICEM CFD project include changes to upstream data and changes you make in the ICEM CFD editor. When changes are made, the cell in the **Project Schematic** window indicates that an update is required. Note, however, that changes made in the ICEM CFD editor will not cause the system to go out of date until the project, tetin file (geometry), blocking file, and/or Replay file are saved.

Note

Named selections defined in Mesh systems are available only within the Mesh system. They are not available to downstream systems like ICEM CFD.

The actions taken by Workbench when the system is updated depends on whether the following conditions are met:

- Blocking exists.
- A Replay file exists.
- Blocking parameters are set.
- Other input parameters are set.

The following table describes the actions performed by ICEM CFD according to these conditions:

Table 1: Updating ICEM CFD Projects

Blocking	Replay File	Blocking Input Parameters	Other Input Parameters	Actions performed by ICEM CFD
No	No	No	No	1. Runs tetra default meshing. 2. Saves the unstructured mesh. 3. Saves the project
Yes	No	No	No	1. Runs hexa default meshing. 2. Saves the unstructured mesh. 3. Saves the project.
No	No	No	Yes	1. Sets all input parameters.

Blocking	Replay File	Blocking Input Parameters	Other Input Parameters	Actions performed by ICEM CFD
				2. Runs tetra meshing. Runs prism meshing if any `Part PART_NAME: Prism` input parameters exist. (See Setting Parameters for Prism Meshing (p. 73)) 3. Saves the unstructured mesh. 4. Saves the project.
No	Yes	No	Yes	1. Sets all input parameters. 2. Runs the Replay file. 3. Saves the unstructured mesh. 4. Saves the project.
No	Yes	Yes	Yes	1. Sets all input parameters except blocking parameters. 2. Runs the Replay file. 3. If blocking now exists: a. Sets blocking input parameters. b. Runs hexa meshing. c. Converts pre-mesh to unstructured. d. Saves the unstructured mesh 4. Saves the project.
Yes	No	Yes	Yes	1. Sets all input parameters.

Blocking	Replay File	Blocking Input Parameters	Other Input Parameters	Actions performed by ICEM CFD
				2. Sets blocking input parameters. 3. Runs hexa meshing. 4. Converts pre-mesh to unstructured. 5. Saves the unstructured mesh. 6. Saves the project.
Yes	Yes	Yes	Yes	1. Sets all the input parameters except blocking. 2. Runs the Replay file. 3. If blocking still exists: a. Sets blocking input parameters. b. Runs hexa meshing. c. Converts pre-mesh to unstructured. d. Saves the unstructured mesh. 4. Saves the project.

ICEM CFD saves the unstructured mesh and project only if the ICEM CFD GUI is closed and you update the project from Workbench. If the ICEM CFD is open, you will need to manually save the unstructured mesh and project.

The order of operations is Meshing input parameters are set before the **Replay** file is run; Blocking Input parameters are set after a replay file is run, but only if blocking exists after the replay file is run.

If no Replay file exists, the default mesher is determined by the presence or absence of blocking: if no blocking exists in the project, **tetra** is the default. Conversely, if blocking exists in the project, **hexa** is the default mesher.

Updating a Project

To update a project and refresh upstream and downstream data, right-click on the **Model** cell and choose **Update**.

Interface Differences in the Data-Integrated ICEM CFD

The data-integrated ICEM CFD interface has been modified to provide additional functionality that enhances the integration of ICEM CFD and Workbench:

- One-click menus that enable you to control basic project and **Replay Recording** functions.

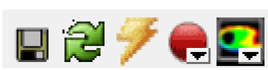
  > **Note**
  >
  > If you create Workbench Input Parameters and want to step through the script line-by-line or using a line range, you must open the **Workbench Replay Control** dialog from this menu.

- The **Workbench Replay Control** dialog enables you to test and edit Replay scripts created within Workbench.

You can ensure that you are working within the data integrated environment by checking the **Message** window within ICEM CFD. The first line identifies that the application is integrated in the Workbench Framework.

```
ICEM CFD 14.5 in Workbench 2.0 Framework
Checked out ansyslmd feature aienv (product ANSYS ICEM CFD) from server 1055@pghlnxlicense
Loading project settings file: ICM.prj...
Loading geometry file "ICM.tin"
Current Coordinate system is global
Loading domain "ICM.uns" ...
Loading family boco data from ICM.fbc
Current Coordinate system is global
```

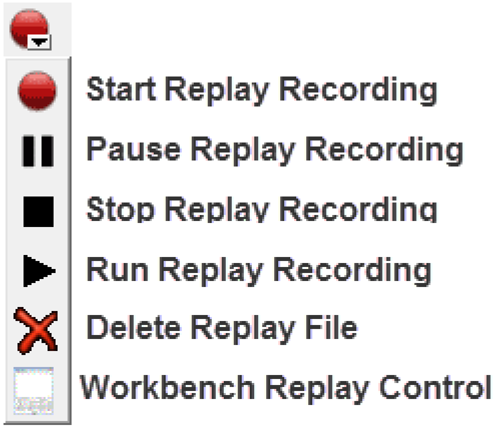
One-Click Menus



The following one-click menu options are available in the **Toolbar** when you open ICEM CFD from Workbench:

- **Save Project**: Saves the entire project, including Workbench data.
- **Refresh Project**: Refreshes the upstream data in the ICEM CFD project.
- **Update Project**: Brings the entire ICEM CFD system up to the most current status, including upstream and downstream data.
- **Start Replay Recording**: Begins recording the commands needed to generate a custom meshing process. All of the steps in the mesh development process are recorded, including blocking, mesh size, edge meshing, boundary condition definition, and final mesh generation. See *Replay Functionality* in the *ANSYS ICEM CFD User's Manual*. After you click the **Start Replay Recording** Icon, the icon changes to the **Stop Replay Recording** icon. You can click this icon to stop recording.

  You can also click on the arrow to choose **Pause Replay Recording**, **Run Replay File**, **Delete Replay File**, and **Replay Control**, which opens the **Workbench Replay Control** dialog.



Note

If you create Workbench Input Parameters and want to step line by line or using a line range through a replay file, you must use the **Replay Control** item from this menu to start the **Workbench Replay Control** dialog.

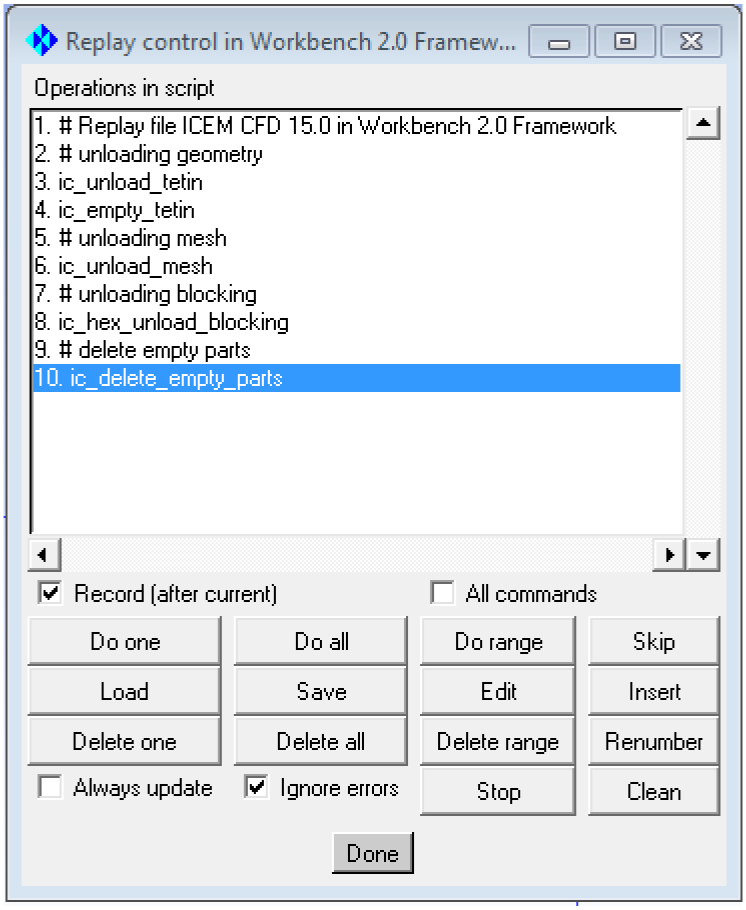
- **Output Mesh**: You can choose to save the ICEM CFD mesh output to Fluent, CFX, or Polyflow projects.

Workbench Replay Control Dialog

The **Workbench Replay Control** dialog helps you create, test, and edit script files by performing operations in ANSYS ICEM CFD and recording the equivalent Tcl/Tk commands in a Replay file. You can then use the dialog to step through and edit the script.

The **Workbench Replay Control** dialog works exactly the same as the standalone **Replay Control** dialog that you open from the **File** menu, with two key exceptions:

- The **Workbench Replay Control dialog** supports Workbench Input Parameters and allows you to step through them.
- The dialog automatically loads the current Replay script file (`ICM.rpl`) instead of opening a file browser.



For more information, see Replay Scripts in the *ANSYS ICEM CFD Help Manual*.

Setting Parameters

Setting Input parameters in Workbench enables you to pass parameters to ICEM CFD and other downstream analysis tools. The interaction of parameters between applications provides you with greater flexibility and capabilities to run optimization and what-if scenarios. For more information about using parameters in Workbench, see *Working with Parameters and Design Points* in the Workbench User's Guide.

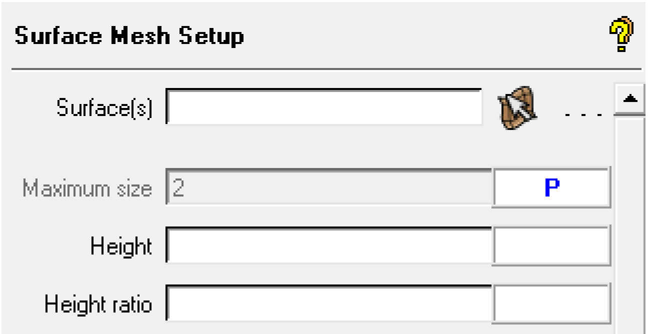
Parameters may be set globally or individually, with individual parameters taking precedence over global values.

The parameters you can set are:

- Input Parameters
- Parameters for all existing curves, surfaces, or edges
- Mesh parameters for parts
- User-defined Input Parameters
- Output Parameters

Setting Input Parameters

Clicking the box to the right of certain **Meshing Input** parameters enables you to select whether the parameter is controlled from within ICEM CFD or from within Workbench. A “**P**” in the check box indicates that it has been selected as a Workbench Input parameter. If the check box is empty, you can control the input from within ICEM CFD.



You can set the following input parameters in Workbench:

- **Global Mesh Size** (See Global Mesh Size in the *ANSYS ICEM CFD Help Manual*).
- **Shell Meshing** (See Shell Meshing Parameters in the *ANSYS ICEM CFD Help Manual*).
- **Volume Meshing** (See Volume Meshing Parameters in the *ANSYS ICEM CFD Help Manual*).
- **Prism Meshing** (See Prism Meshing Parameters in the *ANSYS ICEM CFD Help Manual*).
- **Surface Mesh Setup** (See Surface Mesh Setup in the *ANSYS ICEM CFD Help Manual*).
- **Curve Mesh Setup** (See Curve Mesh Setup in the *ANSYS ICEM CFD Help Manual*).
- **Edge Params** (See Edge Params in the *ANSYS ICEM CFD Help Manual*).

For **Surface Mesh Setup**, **Curve Mesh Setup**, and **Edge Params**, you can set parameters either on all existing surfaces or curves and edges at once, or for a single curve, surface, or edge.

Note

If you create Workbench Input Parameters and want to step line by line or using a line range through a replay file, you must use the **Workbench Replay Control** item from the **One-Click** menu to start the **Workbench Replay Control** dialog.

Setting Input Parameters

To set input parameters in Workbench:

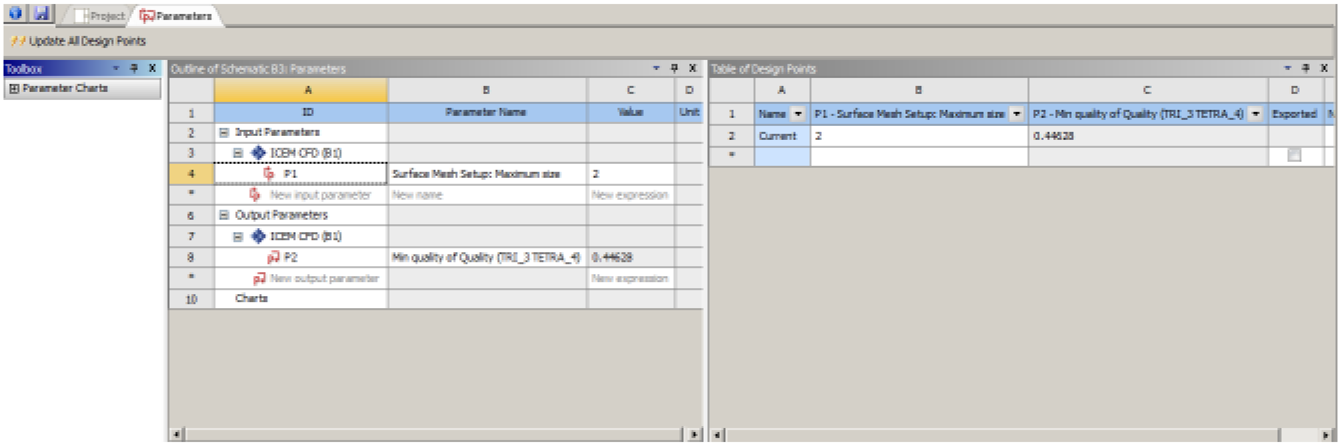
1. Within ICEM CFD, choose any of the input parameters listed above.
2. Select the check box next to the parameter.

3. Click the **Yes** button in the pop-up dialog to confirm the selection. A **P** in the check box indicates that the parameter has been created for Workbench.

> **Note**
>
> You will not be able to edit the parameter within ICEM CFD unless you click the check box again and deselect the parameter.

4. In **Workbench**, double-click on the project's Parameters cell.

5. Edit the parameter values in the **Outline of Schematic: Parameters** window.



6. Click the **Project** tab to return to the **Project Schematic** window.

   You can now update the project using the new parameter settings.

Setting the parameters for a single curve, surface, or edge

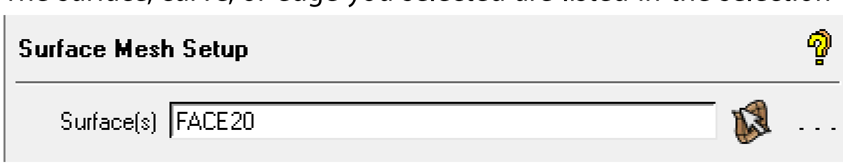
1. Within ICEM CFD, open the **Surface Mesh Setup**, **Curve Mesh Setup**, or **Edge Params** parameters from the **Tab** menu.

2. Click the **Select** button at the top of the **Parameters** window.

3. Click the **Left Mouse** button to select the curve, surface, or edge for which you want to set parameters.

4. Click the **Middle Mouse** button to complete the selection.

   The surface, curve, or edge you selected are listed in the selection entry.



5. Select the check box next to the parameter you want as the input parameter.

6. Click the **Yes** button in the pop-up dialog to confirm the selection. A **P** in the check box indicates that the parameter for the single surface, curve or edge has been created for Workbench.

Setting Parameters for All Existing Curves, Surfaces, or Edges

1. Within ICEM CFD, open the **Surface Mesh Setup**, **Curve Mesh Setup**, or **Edge Params** parameters from the **Tab** menu.
2. Leave the Surface, curve, or edge selection field empty (do not select any surface, curve or edge).
3. Select the check box next to the parameter you want as the input parameter
4. Click the **Yes** button in the pop-up dialog to confirm the selection. A **P** in the check box indicates that the parameter for all existing surfaces, curves or edges has been created for Workbench.

Setting Workbench Mesh Parameters for Parts

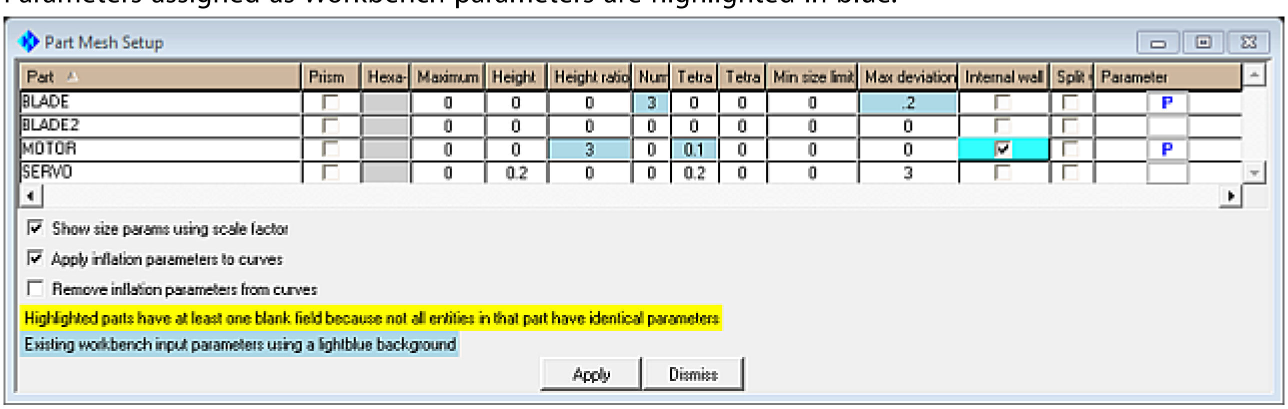
The **Part Mesh Setup** parameters enable you to specify the mesh parameters for different parts of a geometry. Part Mesh Setup is described in the *ANSYS ICEM CFD Help Manual*.

To create a Workbench mesh parameter for a part:

1. Click the **Mesh** tab.
2. Click the **Part Mesh Setup** icon.
3. In the **Part Mesh Setup** dialog, choose a part and assign non-zero values to one or more of its parameters.
4. Click in the **Parameter** column for the part.
5. A dialog asks you to confirm that you want to create a Workbench parameter for each value you've changed.

> **Note**
>
> If any Workbench parameters are already set for the part, a dialog will ask if you want to delete that Workbench parameter.

Parameters assigned as Workbench parameters are highlighted in blue.



To delete a parameter, click in the **Parameter** column. A dialog will ask you to confirm that you want to delete each individual Workbench parameter in the row.

As with ICEM CFD, the **Apply inflation parameters to curves** and **Remove inflation parameters from curves** options affect Workbench parameter behavior as well. The current value of **Apply inflation parameters to curves** is saved to the `project file/.aienv_options` file, so it is always available in **GUI** or **Batch** mode.

Setting Parameters for Prism Meshing

You can add prism meshing parameters using the **Part Mesh Setup** dialog:

1. Click the **Mesh** tab.
2. Click the **Part Mesh Setup** icon.
3. In the **Part Mesh Setup** dialog, choose a part and check the check box in the **Prism** column.
4. Click in the **Parameter** column for the part.
5. A dialog asks you to confirm that you want to create a Workbench parameter for **Part PARTNAME: Prism**.

> **Note**
>
> If any Workbench parameters are already set for the part, a dialog will ask if you want to delete that Workbench parameter.

Setting User-Defined Input Parameters

Setting User-Defined Input Parameters allows for greater flexibility and control over the meshing operation. For example, parameters that can not be applied as single input parameters, may be individually set with User-Defined Input Parameters.

1. Within **ICEM CFD**, choose **Settings>Workbench Parameters> Workbench Input Parameters (User-defined)**.
2. In the **User-Defined Workbench Input Parameters** window, check the **Create User Defined Input Parameter** check box.
3. Enter a value for the **Parameter name** (for example, MY_PARAMETER).
4. Enter a value for the **Parameter** (for example, 1.343). This value must not be empty.
5. Click **Apply** or **OK**.

You can edit this value in the **Outline of Schematic: Parameters** window.

> **Note**
>
> See the *ANSYS ICEM CFD Programmer's Guide* for information about using **User-Defined Parameters** with **Replay Scripting**.

Deleting User-defined Input Parameters

1. Within **ICEM CFD**, choose **Settings>Workbench Parameters> Workbench Input Parameters (User-defined)**.

2. In the **User-Defined Workbench Input Parameters** window, check the **Delete User Defined Input Parameter** check box.

3. Use the drop-down menu to choose the name of the parameter you want to delete.

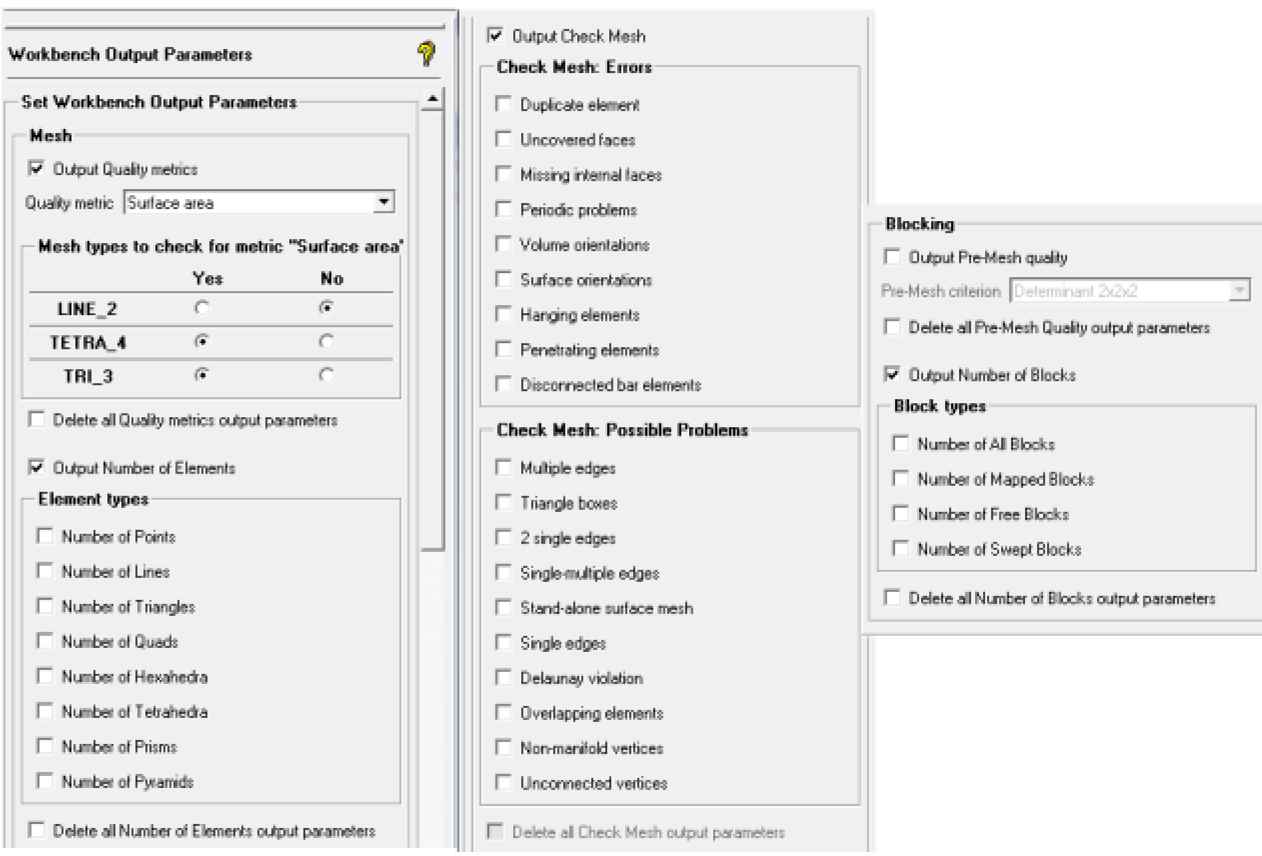
4. Click **Apply** or **OK**.

Setting Output Parameters

You can set Workbench Output parameters to:

- unstruct mesh and pre-mesh blocking quality metrics
- list the number of mesh errors and possible problems
- list the number of element and block types created

You set the output parameters within the ICEM CFD application, then view them in either Workbench or ICEM CFD.



Setting Output Parameters

1. Within **ICEM CFD**, choose **Settings>Workbench Parameters> Workbench Output Parameters**.

2. Optionally, Click the **Output quality metrics** check box to select quality metrics.

   a. Use the **Quality Metrics** drop-down menu to choose the metric you want to set.

b. Click the radio buttons to select the mesh types to check for the metric.

3. Optionally, click the **Output number of elements** check box.

    a. Check the boxes next to the element types for which you want an output number.

4. Optionally, click the **Output Check Mesh** check box.

    a. Check the boxes next to the errors you want to have tallied in the output results. ICEM CFD will perform an analysis as part of the meshing process and list the results in the **Output** window and in the **Workbench Parameters** window.

5. Similarly, check the boxes for **Output Pre-Mesh Quality** and **Output Number of Blocks** to set the Blocking Output parameters.

    For more information about pre-mesh quality, see Pre-Mesh Quality (p. 46).

    For more information about Hexa Block types, see Hexa Block Types (p. 41).

Deleting Output Parameters

1. Within **ICEM CFD**, choose **Settings>Workbench Parameters> Workbench Output Parameters**.

2. In the **Workbench Output Parameters** window, check any applicable check box for **Delete all quality metrics output parameters**, **Delete all number of elements output parameters** , **Delete all Pre-Mesh Quality output parameters**, and **Delete all Number of Blocks output parameters**.

3. Click **Apply** or **OK**.

User-Defined Parameters Example

This example illustrates how you can use user-defined parameters to test different meshing scenarios for a simple box:

1. In the **Workbench Toolbox**, double-click the **ICEM CFD** component to start the data-integrated ICEM CFD component system.

2. Double-click the **Model** cell to open ICEM CFD.

3. First, create an Input parameter, **ZSIZE**, that you can manipulate from Workbench.

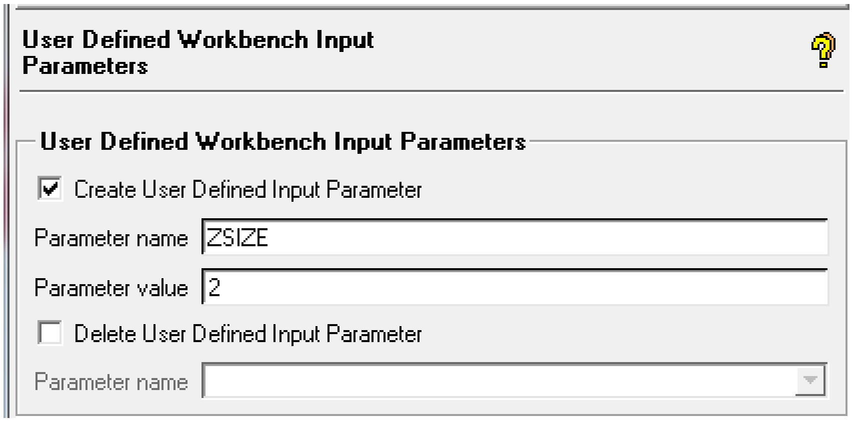
    a. Choose **Settings>Workbench Parameters> Workbench Input Parameters (User-Defined)**.

    b. Click **Create User-Defined Input Parameter**.

    c. Name the Parameter **ZSIZE** and set the **Parameter Value** as **2**

    d. Click **OK** to finish.

    .



4. Now create a box model upon which you will perform a meshing operation. Record the process so the operation can be performed again when you update it from Workbench.

   a. Click the **Start Replay** button.

   b. Click the **Geometry** tab and choose **Create/Modify Surface**.

   c. Choose **Standard Shape** from the **Create/Modify Surface** window.

   d. Choose **Box** and click **Apply**.

   e. Click the **Mesh** tab and choose **Compute Mesh**.

   f. Choose **Volume Mesh** and click **Compute**.

   g. Click **Yes** on the pop-up dialog to use autosizing for the mesh size.

   h. Click the **Stop Replay Recording** button.

   i. Choose **File>Save Project** to save the project.

5. Next, edit the script to use the input parameter.

   a. Choose **Workbench Replay Control** from the **One-Click Replay Recording** menu.

   b. Click the **Edit** button.

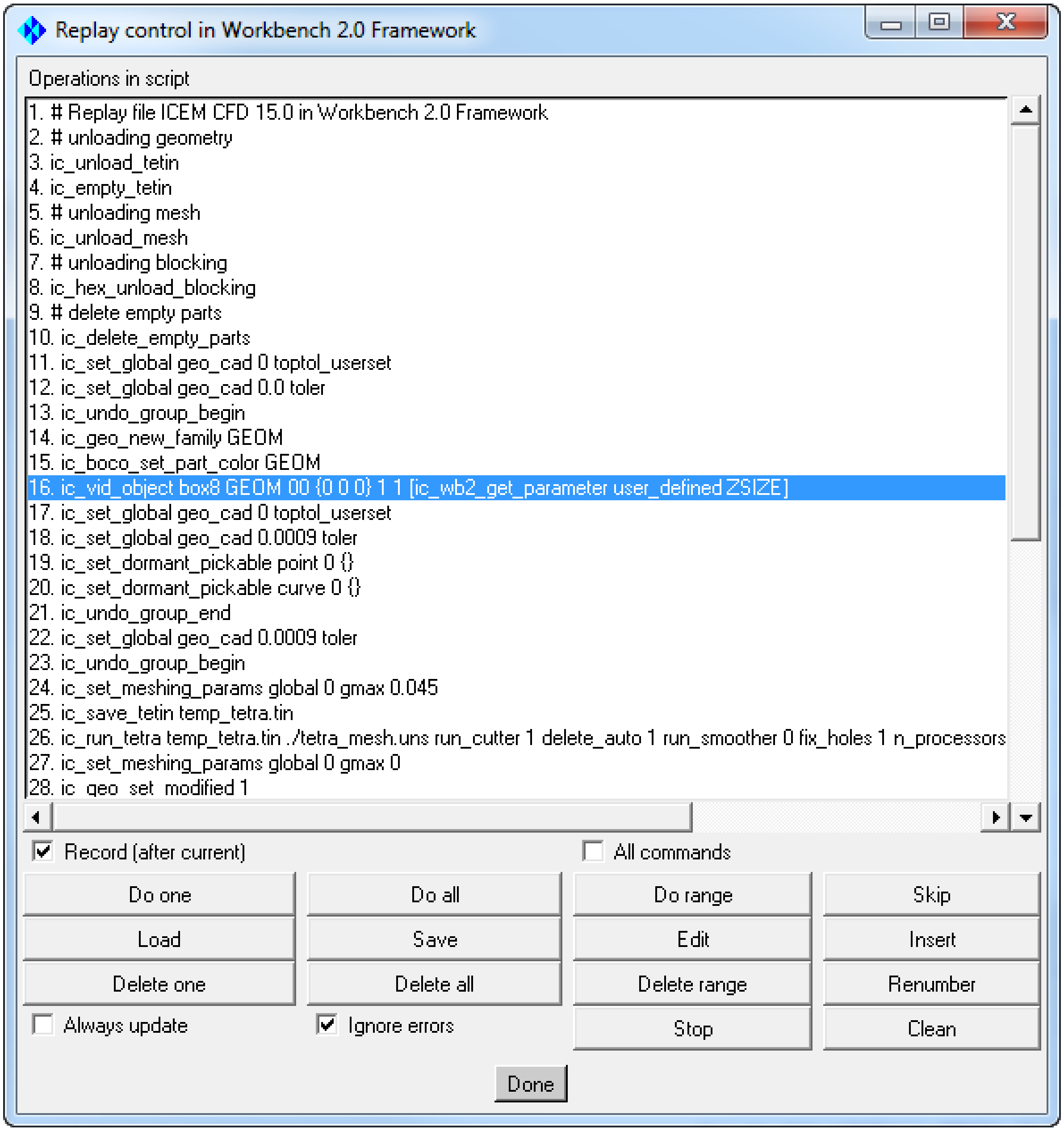
   c. Edit **Line 16** (or whichever line is appropriate), `ic_vid_objectbox8 GEOM 00 {0 0 0} 1 1 1`, to `ic_vid_object box8 GEOM 00 {0 0 0} 1 1 [ic_wb2_get_parameter user_defined ZSIZE]`

   d. Choose **File>Save**.

   e. Close the **Edit** window.

   f. Click the **Save** button and save `ICM.rpl`.

   g. Click the **Done** button. Do not close ICEM CFD.



6. Now you can change the Input parameter through Workbench.

   a. In the Workbench **Project Schematic** window, double-click on the **Parameters** cell of the ICEM CFD component.

   b. Change the value of **ZSIZE** to **5** and choose **Return to Project**.

   c. Right click on the Model cell and choose **Update**.

   You can watch the Replay script run using the new parameter in the ICEM CFD interface. Each time you change the **ZSIZE** parameter in Workbench, will be generated and meshed with the new **ZSIZE** parameter value.