CFD EXPERTS

Simulate the Future

WWW.CFDEXPERTS.NET

Ansys 2021 R2
Engineering What's Ahead.

Ansys ICEM CFD User's Manual

ANSYS, Inc.
Southpointe
2600 Ansys Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
http://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

Release 2021 R2
July 2021

Published in the U.S.A.

Table of Contents

List of Figures

Introduction to Ansys ICEM CFD

Ansys ICEM CFD provides advanced geometry acquisition, mesh generation, and mesh diagnostic and repair tools to provide integrated mesh generation for today's sophisticated analyses.

Maintaining a close relationship with the geometry during mesh generation, Ansys ICEM CFD is designed for use in engineering applications such as computational fluid dynamics and structural analysis.

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

- Multi-block 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, you can input geometry in almost any format, whether a commercial CAD design package, third-party universal database, scan data, or point data. Beginning with a robust geometry module that 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.

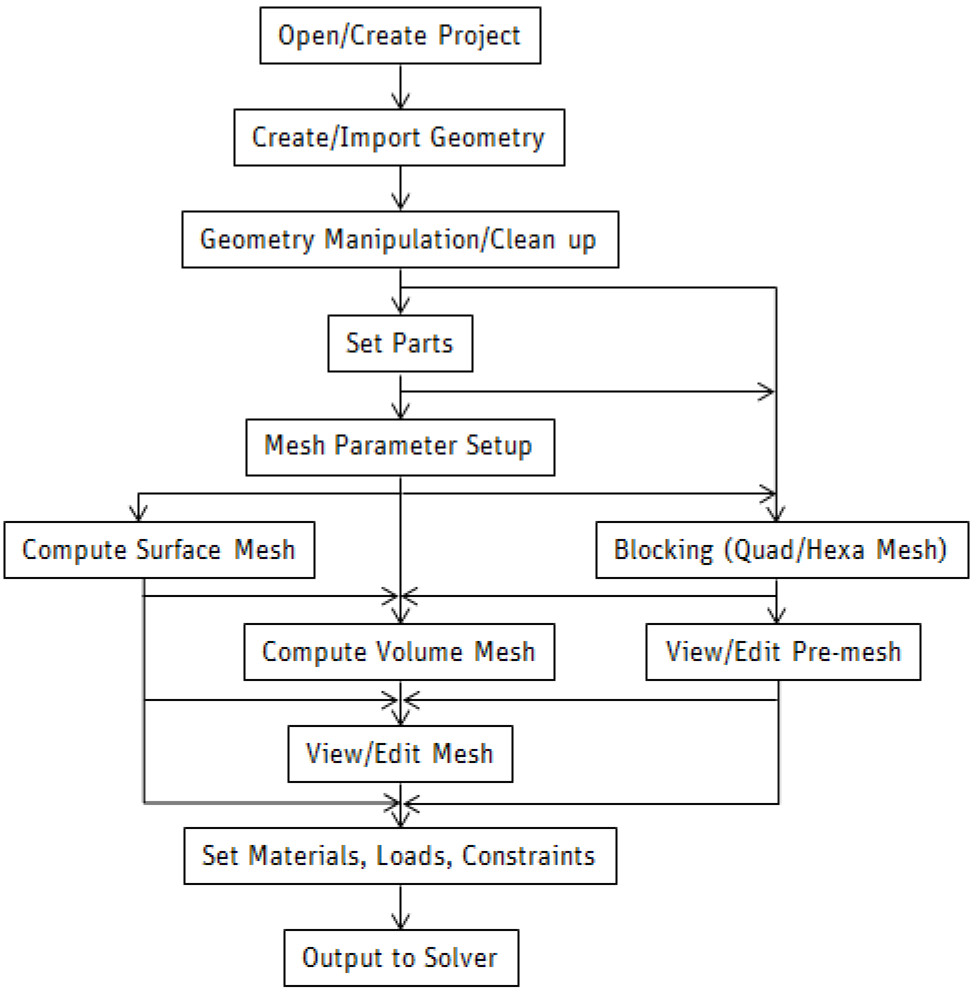
General Meshing Workflow

The general meshing workflow is:

1. Create a working directory.
2. Create a new project (or open an existing project).
3. Create or import the geometry.
4. If necessary, repair the geometry before meshing.
5. If necessary, block the geometry.
6. Compute the mesh.

7. Check the mesh for errors and edit as required.
8. Describe the physics.
9. Generate the input for the solver.

Figure 1: Overall ICEM CFD Process



The workflow is demonstrated in a video on the customer site. See Ansys ICEM CFD Process Overview.

Creating a Project

All the files required for a particular analysis are contained within a *project*. You can either create a new project or open an existing project.

Note:

If you already have a project open, creating a new project will close the current project.

To create a project:

1. Create a working directory where you will keep all of the files associated with your project.

2. Select **File > New Project**.
3. Browse to the working directory that you created.
4. Specify a **File name** for the project and click **Save**.

> **Note:**
>
> The special characters Ä, ä, Ç, Ö, ö, Ü, and ü are not supported in file names on Linux.

The Project directory typically contains one or more of the following file types:

Project Settings (*.prj)

An ICEM CFD project file contains the information necessary to manage the data files associated with your project.

Tetin (*.tin)

Contains geometry entities, material points, parts associations, and global and entity mesh sizes.

Mesh (*.uns)

Includes details of the line, shell, and volume mesh elements of the project. Shell meshes are composed of triangular and/or quadrilateral elements; volume meshes may include tetrahedra, hexahedra, pyramids, and/or prisms.

Blocking (*.blk)

Includes details of the underlying framework used to create a structured hexahedral mesh in your project. Blocking files can also be loaded from, or saved to, an unstructured mesh.

Attributes (*.atr) or (*.fbc)

Maintain the association of user-specified data for parts, element properties, loads, and constraints with the nodes/elements of the mesh for a project.

Parameters (*.par)

Contains mesh-independent data such as material properties, local coordinate systems, solver analysis setup, and run parameters. The data in the parameters file is cross-referred in the attributes file when a set of parameters is associated with the nodes/elements of the mesh.

Cartesian (*crt)

Contains information regarding the Cartesian grid, if one has been created for your project.

Journal (*.jrf)

Contains a record of the operations performed (see General in the *Ansys ICEM CFD Help Manual*).

Replay (*.rpl)

Contains a replay script (see Replay Scripts in the *Ansys ICEM CFD Help Manual*).

Creating or Importing the Geometry

Ansys ICEM CFD includes a wide range of tools for creating a new geometry or manipulating an existing geometry. You can create a simple geometry or alter a complex geometry without having to go back to the original CAD. This can be done for CAD (NURBS surfaces) and triangulated surface data.

ICEM CFD also supports several *geometry interfaces* that enable you to import geometry from other formats (CAD, 3rd party geometry, Faceted Data, and Mesh formats). These 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. Currently supported interfaces include the most popular CAD systems and many legacy formats. For a complete list of supported geometry interfaces, see Import Model and Import Geometry in the Help manual.

Geometry Preparation

Although most of the meshing modules within Ansys ICEM CFD allow minor gaps and holes in the geometry, you should confirm that the geometry is free of any flaws that would inhibit optimal mesh creation. For example, the tetra mesher requires that the model contains a closed volume; if there are any holes in the geometry that are larger than the local tetras, the mesh will not be created. Ansys ICEM CFD provides tools for such operations on either CAD or triangulated surfaces.

The Repair Geometry page in the Help manual provides links to the tools that are used to find and repair such flaws. The typical procedure is to identify possible geometry problems using the **Build Topology** operation and then use the other tools to effect any needed repair.

In cases where feature capture is necessary, geometry curves and points will act as constraints for the edges and nodes, respectively, of the mesh elements. If necessary, **Build Topology** is able to create such curves and points automatically from surface data, thereby capturing certain key features in the geometry.

See Repairing the Geometry before Meshing (p. 11) in the Tetra Meshing chapter for more information on geometry preparation.

Parts

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 into a grouping called a part. With this part association, you can enable or disable all entities within the part, visualize them with a different color, assign mesh sizes on all entities within the part, and apply different boundary conditions by part. More information on the display and management of parts can be found in Parts, in the Help manual.

Blocking the Geometry

The blocking feature in Ansys ICEM CFD provides a projection-based mesh-generation environment. All block faces between different materials are projected to the closest CAD surfaces. Block faces within the same material can also be associated to specific CAD surfaces to enable the definition of internal walls. Generally you do not need to perform any individual face associations to underlying CAD geometry, which reduces the time for the mesh generation.

The blocking step is used when a structured, hexa-mesh is desired in one or more parts (quad mesh if 2D parts). A premesh is generated in the blocked regions which can be refined and improved on a block-by-block basis. The premesh data is converted to structured or unstructured mesh data before it is merged with mesh data from other parts or passed to a solver.

There can be multiple Blocking strategies, depending on the topology you are working with:

- Typically you take a top-down approach, in which you first capture the outer geometry and then split, delete, and merge blocks to capture the minor geometry. The volume can be filled by a variety of methods including mapped hexa blocks, swept blocks, or unstructured mesh zones (which can be filled with a variety of unstructured methods available).
- You may be able to use a bottom-up approach, in which you start by creating a 2D blocking, add blocks to capture more detail, and then extrude the 2D blocking to create the 3D blocking. Again, the volume can be filled by a variety of methods.

For further information on blocking, see Blocking Strategy (p. 30) in the Hexa Meshing chapter.

Computing the Mesh

To proceed from geometry and parts to a completed mesh requires two steps:

1. Set up Mesh Parameters

   You should specify mesh size, type, and method along with several type- and method-specific controls. The parameters can be applied globally and individually to parts, surfaces, curves, or regions. Rotational and translational periodicity is also supported.

2. Compute the Mesh

Meshing methods 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 good element quality. Options are available to automatically refine and coarsen the mesh both on the 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; while the advantage of the Advancing Front method is its ability to generate a smoothly transitioning Tetra mesh with a controlled volume-growth ratio.

Prism

Ansys ICEM CFD Prism generates hybrid meshes 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. When compared to pure tetrahedral meshes, this results in smaller analysis models, better convergence of the solution, and better analysis results.

Hexa

The Ansys ICEM CFD Hexa mesher is a semi-automated meshing module that allows rapid generation of multi-block structured or unstructured hexahedral volume meshes. ICEM CFD Hexa represents a new approach to mesh 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 Ogrids, can also be generated automatically.

Hybrid Meshes

The following types of hybrid meshes can also be created:

- Tetra and Hexa meshes can be united (merged) at a common interface in which a layer of pyramids is automatically created 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) for both 3D and 2D geometries. 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 that aligns better with the geometry curvature.

- **Patch based shell meshing (Patch Dependent)**: Uses a series of "loops" that 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 use on an unclean geometry.

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

Complete descriptions of the meshing processes are included in subsequent chapters.

Checking/Editing the Mesh

The mesh editing tools in Ansys ICEM CFD enable 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, and so on.

The process of improving the mesh quality typically involves:

1. Check the mesh for problems such as holes, gaps, and 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.

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

Describing the Physics

Ansys ICEM CFD includes tools to describe the physical properties of your model and the type of analysis being performed.

Describing the physics for your project involves:

1. Defining or editing the material properties.

   Several common materials are included in the Material Library; or you can define your own material, including certain non-linear behavior. The data may be saved for later recall to a *.mat file.
2. Assigning the material properties to the Parts of your model.
3. Applying Constraints to the model.

   Constraints include movement restrictions and initial velocity, and may be applied to points, curves, surfaces or parts.
4. Applying Loads to the model.

   Loads may be force, pressure or temperature, and may be applied to points, curves, surfaces, or parts.

Note:

Not all solvers support all available Constraints or Loads.

Generating the Input for the Solver

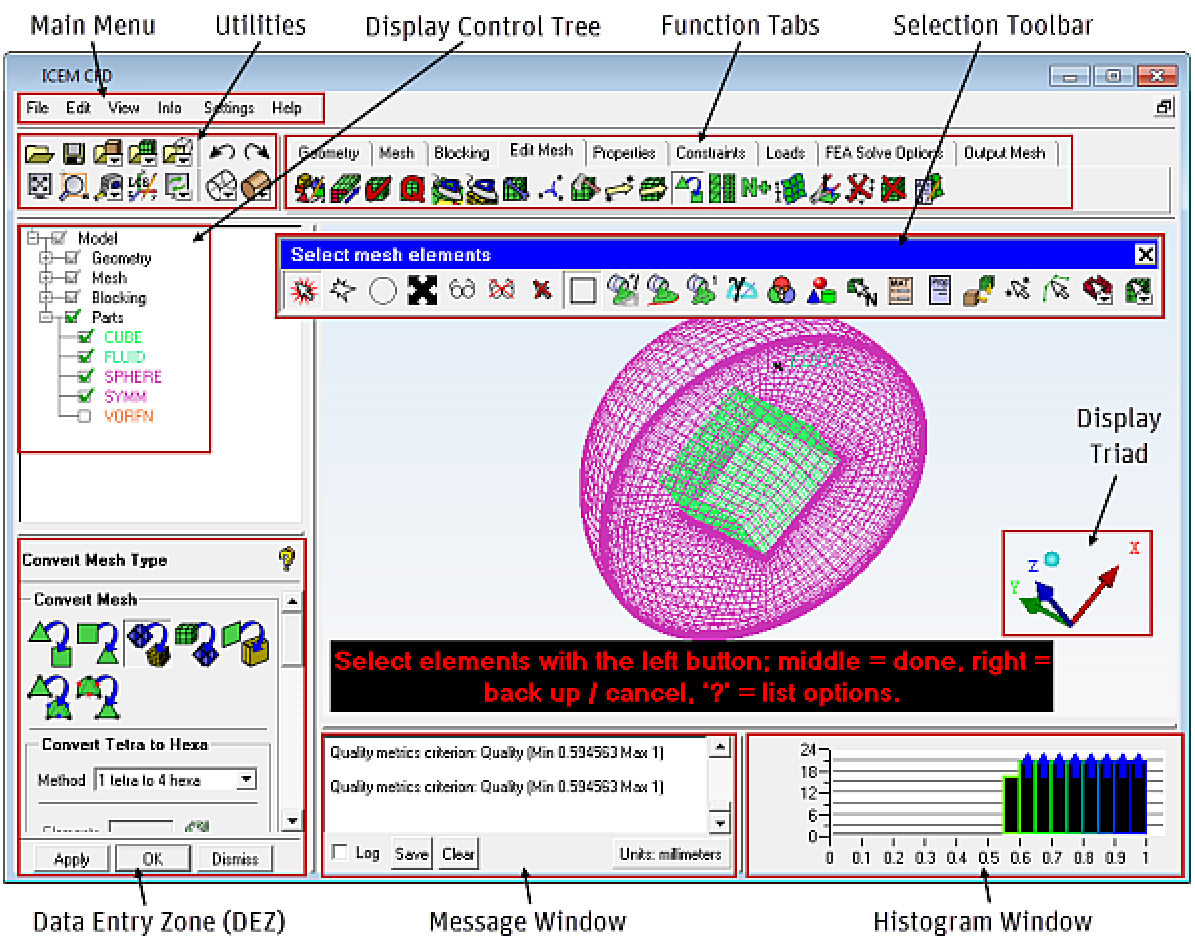
Ansys ICEM CFD includes output interfaces to various flow and structural solvers, producing appropriately formatted files that contain complete mesh and boundary-condition information. After selecting the solver, you can modify the solver parameters and write the necessary input files.

More information about the Ansys ICEM CFD Output Interfaces is available from the Help menu. The **Output Interfaces** feature 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 User Interface

The Ansys ICEM CFD user interface offers a complete environment to create and manage your computational grids. As shown in Figure 2: Ansys ICEM CFD User Interface Components (p. 8), the user interface contains several functional areas. The general meshing workflow is from left to right across the **Function Tabs**.

Figure 2: Ansys ICEM CFD User Interface Components



Note:

The user interface style shown is the default, **Workbench** style. For more information about the user interface (**GUI Style**) features, refer to the Settings menu.

The user interface is completely described in the Ansys ICEM CFD Help Manual.

Tetra Meshing

The Tetra mesher generates tetrahedral volume meshes directly from the CAD geometry or stereolithography (STL) data, without requiring an initial triangular surface mesh—you have only to select the geometry to be meshed.

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.

Features of Tetra Meshing

The **Tetra** mesher is suitable for complex geometries and offers:

- 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
- Automatic detection of holes and easy ways to repair the mesh.

Note:

- Tetra meshing is not efficient for capturing shear or boundary layer physics. To mesh for such situations, choose a method that supports inflation layers and prisms. See Prism Meshing (p. 23).
- Tetra meshing creates an unstructured mesh of tetrahedral cells. For a structured, multi-block approach or an unstructured mesh of hexahedral cells, see Hexa Meshing (p. 27).

The tetra meshing chapter includes the following sections.

Inputs to the Tetra Mesher

The Tetra mesher accepts the following inputs:

- B-Spline Curves and Surfaces (p. 10)
- Triangular Surface Meshes as Geometry Definition (p. 10)
- Full/Partial Surface Mesh (p. 11).

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 enable 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 enable 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 stereo-lithography (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 Mesh Generation Steps

The steps involved in generating a Tetra mesh are:

1. Repair the geometry before meshing.
2. Specify the geometry details.
3. Specify sizes on surfaces/curves.
4. Specify the behavior in small angles or in small gaps between objects.
5. Determine the desired mesh method.
6. Compute the mesh.

The computed mesh should be checked for errors and smoothed for quality as necessary. See Checking and Improving the Mesh (p. 41) in this manual. When satisfactory, it is ready for applying loads, boundary conditions, and so on, and for writing to the desired solver.

Repairing the Geometry before Meshing

Before generating the mesh, you should confirm that the geometry is free of any flaws that would inhibit optimal mesh creation. The Tetra mesher requires that the model contains a closed volume; if there are any holes in the geometry that are larger than the local tetras, the mesh may not be computed. The Build Topology operation is used to check your geometry and represent connectivity between adjacent surfaces with different colored curves. You can correct any identified problem on the mesh or you can repair the geometry in that vicinity and repeat the meshing process. For further information on the process of interactively closing holes, see the following sections.

Repairing Holes in the Geometry

Filling, Trimming, and Blending a Gap

Matching Curves

Note:

If you want to save the changes in the native CAD files, perform any geometry repairs in a direct CAD interface.

Repairing Holes in the Geometry

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.

Closing Holes Bounded by Multiple Surfaces

If the hole is bounded by more than one surface, you can use the **Close Holes** feature. For example, in Figure 3: Hole Bounded by Multiple Surfaces (p. 12), the yellow curves represent the boundary of the hole. It is clear that this hole is bounded by more than one surface.

Figure 3: Hole Bounded by Multiple Surfaces

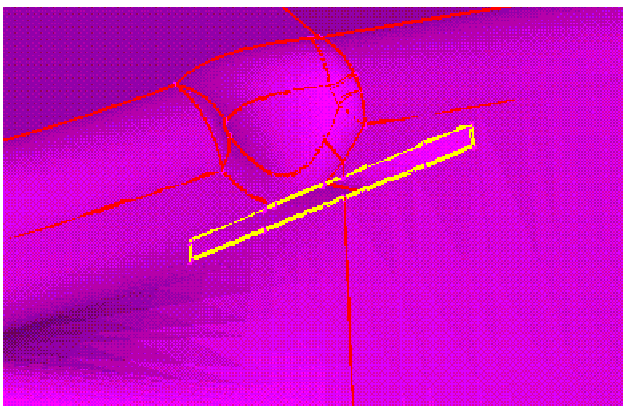
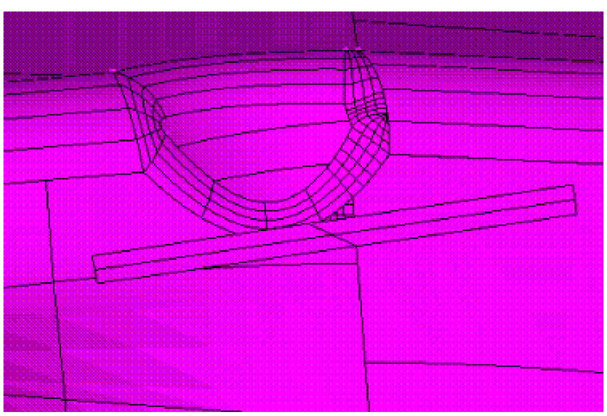


Figure 4: Closed Hole (p. 12) shows the geometry after the **Close Holes** operation is completed. A new surface is created to close the hole.

Figure 4: Closed Hole



Closing Holes on Single Surfaces

If the hole lies entirely within a single surface, such as a trimmed surface, you can use the **Remove Holes** feature. For example, in Figure 5: Holes Within a Single Surface (p. 13), the two yellow curve loops represent the boundaries of holes on a surface.

Figure 5: Holes Within a Single Surface

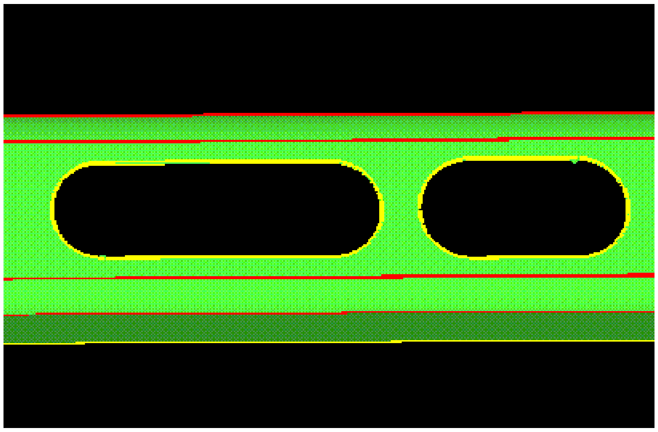
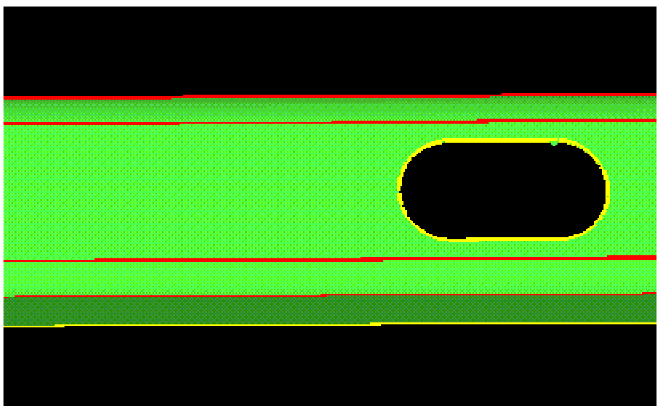


Figure 6: After Removing One Hole (p. 13) shows the geometry after the **Remove Holes** operation is completed for one of the holes. The existing surface is modified by removing the trim definition.

Figure 6: After Removing One Hole



Filling, Trimming, and Blending a Gap

Consider the case of a geometry with a gap shown in Figure 7: Geometry With a Gap (p. 13).

Figure 7: Geometry With a Gap

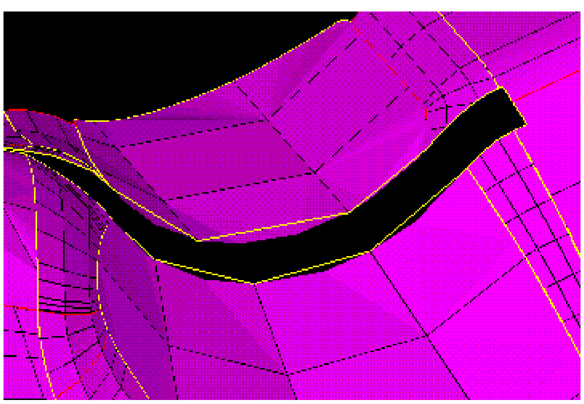


Figure 8: Using the Fill Feature (p. 14) shows the use of the Fill feature.

Figure 8: Using the Fill Feature

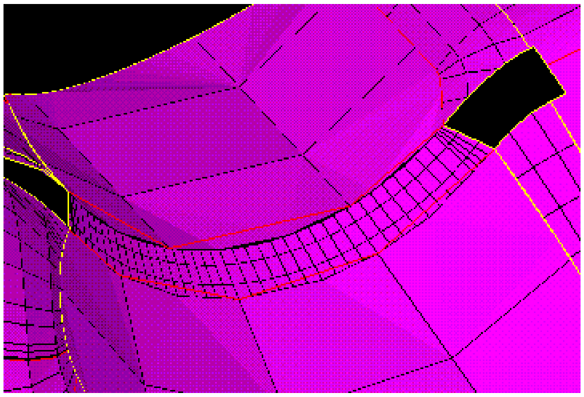


Figure 9: Using the Trim Feature (p. 14) shows the use of the Trim feature.

Figure 9: Using the Trim Feature

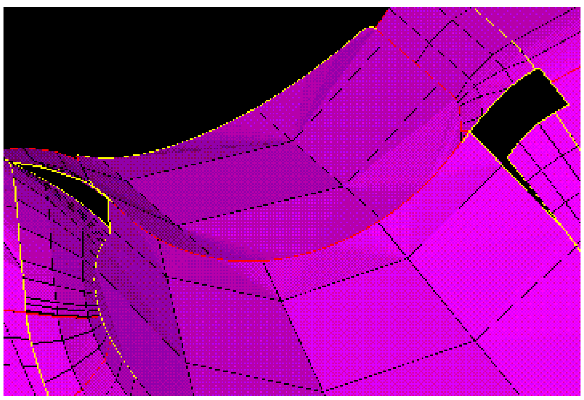
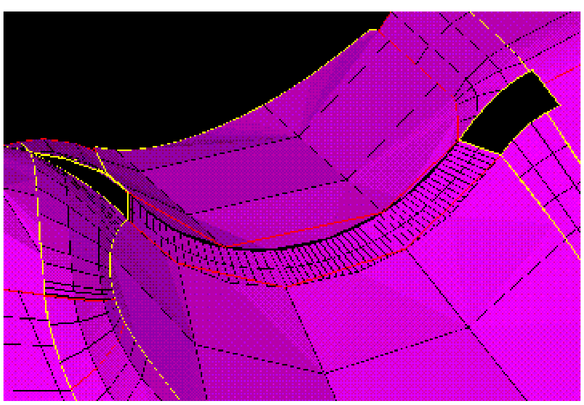


Figure 10: Using the Blend Feature (p. 14) shows the use of the Blend feature.

Figure 10: Using the Blend Feature



Matching Curves

The **Match** feature is used in cases where curves lie very close to each other and have ends that meet (see Figure 11: Geometry With Mismatched Edges (p. 15) and Figure 12: Geometry After Using the Match Edges Option (p. 15)). The gap between the two curves must be within a reasonable tolerance in order for this feature to work.

Figure 11: Geometry With Mismatched Edges

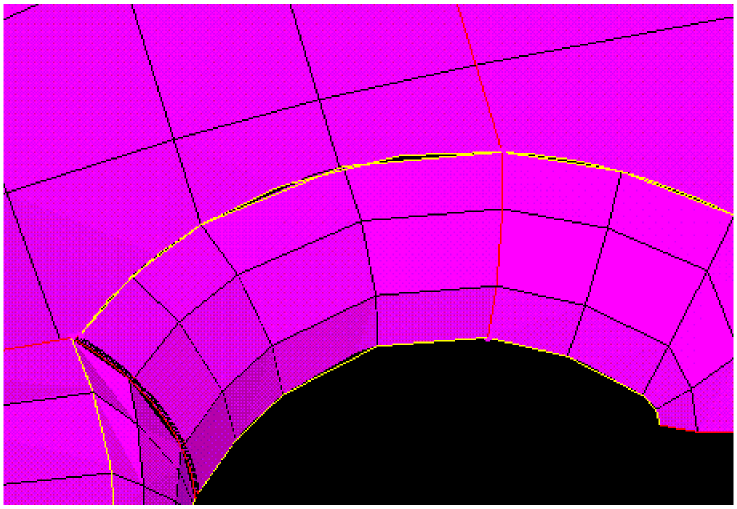
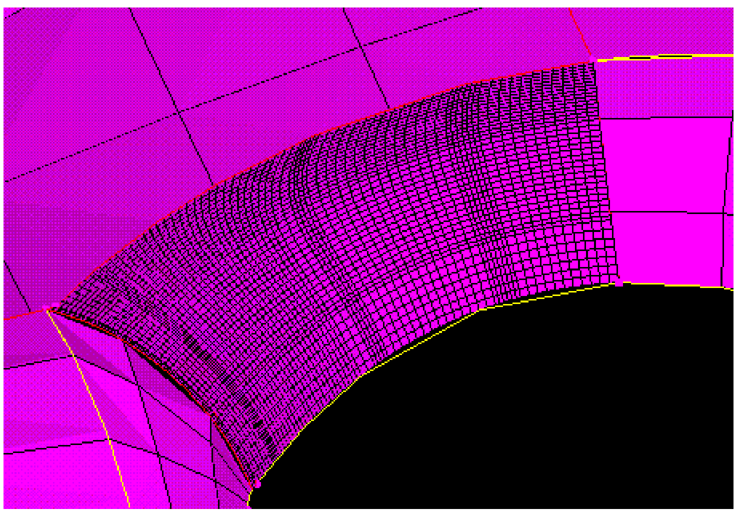


Figure 12: Geometry After Using the Match Edges Option



Specifying the Geometry Details

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 13: Curves and Points Representing Sharp Edges and Corners (p. 16) shows a set of curves and points representing hard features of the geometry.

Figure 13: Curves and Points Representing Sharp Edges and Corners

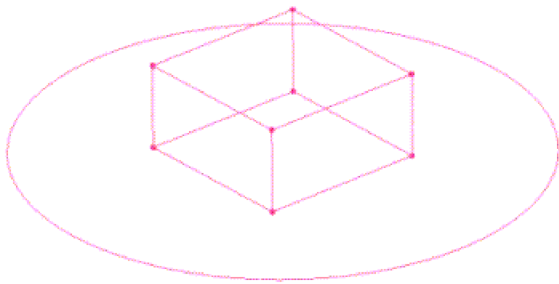


Figure 14: Mesh with Curves and Points (p. 16) shows the resultant surface mesh if the curves and points are preserved in the geometry. Mesh nodes are forced to lie along the curves and points to capture the hard features of the geometry.

Figure 14: Mesh with Curves and Points

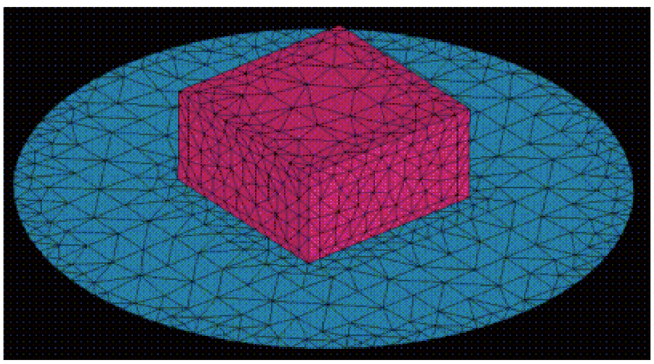
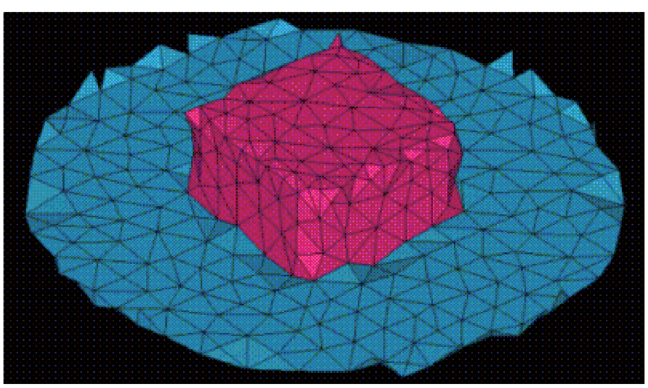


Figure 15: Mesh Without Curves and Points (p. 16) shows the resultant surface mesh if the curves and points are deleted from the geometry. The hard features of the geometry are not preserved, but rather are neglected or chamfered. The boundary mesh nodes lie on the surfaces, but they will lie on the edges of the surfaces only if curves and points are present. Removal of curves and points can be used as a geometry-defeaturing tool.

Figure 15: Mesh Without Curves and Points



Use the Build Topology option to identify points, curves, surfaces and their connectivity in the geometry. **Build Topology** also includes options to automatically repair some minor geometry faults.

Specifying 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 can 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.

Mesh size can be set for individual curves and surfaces, or by parts, using the **Curve Mesh Setup** and **Surface Mesh Setup** tools on the **Mesh** tab. Modify the mesh size for all entities by adjusting the **Scale Factor**, which is found with other **Global Mesh Setup** tools on the **Mesh** tab.

Important:

If the **Scale Factor** is assigned a value of **`0`**, the Tetra mesher will not run.

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**, accessed in the **Global Mesh Setup** tools, 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.

Specifying Meshing Behavior Inside Small Angles or in Small Gaps

The Tetra mesher automatically attempts to close all holes in a model so special consideration should be given to 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 tetra sizes are larger than, or approximately the same size as, the distance between the surfaces or curves, the surface mesh could have a tendency to jump the gap, thereby creating non-manifold vertices during the meshing process. That is, the small thickness or gap may be interpreted as a hole by the mesher. 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.

If the local tetra sizes are not small enough to allow at least 2 or 3 elements across the material thickness (or gap), you should select **Define thin cuts**, in the **Volume Meshing Parameters** section of the **Global Mesh Setup** tools. 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.

Choosing the Tetrahedral Mesh Method

Ansys ICEM CFD supports four different methods for generating the tetrahedral volume mesh. They are listed as **Tetra/Mixed** as each method includes the option of adding boundary layer prism elements. See Tetra/Mixed Mesh Type in the *Ansys ICEM CFD Help Manual* for more information on these methods.

The **Robust (Octree)** method generates a volume mesh using a top-down approach, then the mesh is made conformal to the geometry and a patch-independent surface mesh is created at all of the boundaries and internal walls. This method supports several options as describe in Robust (Octree). The process is described in detail on the next page.

The **Quick (Delaunay)** and **Smooth (Advancing Front)** methods are bottom up, requiring an existing closed surface mesh to start. If one has not yet been created, ICEM CFD will create the surface mesh from the geometry. Both methods support several options as described in Quick (Delaunay) and Smooth (Advancing Front). Choose a method based on your speed and quality priorities.

Tip:

Expert users often use the Robust approach to generate a surface mesh and then regenerate the volume mesh using one of the bottom-up approaches.

The **Fluent Meshing** method uses Ansys Fluent, in batch, to generate the tetrahedral volume mesh from an input surface mesh, volume mesh or geometry. If no surface mesh input is specified, it will be computed before the Fluent batch process is started. This method is similar to the **Quick (Delaunay)** method with additional options for prism layers and/or hexahedral elements in the core. The pre inflation prism option first creates the boundary layers from the surface mesh and then fills the remaining volume with tetrahedrons, or optionally, hexa elements. The post inflation prism option replaces tetra elements at the surface with prisms and is more familiar to ICEM CFD users. Each prism option has advantages, and you should not expect the same results from both algorithms. Fluent Meshing inflation also allows part-by-part inflation parameter control.

Global volume fill parameters for this method are described and set using the Fluent Meshing DEZ. Both pre inflation and post inflation prism options use most of the same parameters, which are set using the Prism Meshing Parameters DEZ. Options presented when the mesh is computed include pre inflation or post inflation selection and the hexa-core option.

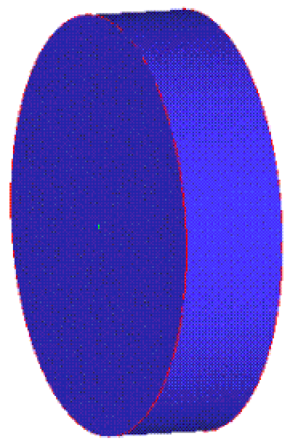
Important:

If enabled, the prisms are grown from a shell mesh (input or created) with the direction of growth determined by the geometry. If no geometry is available, pure tetra meshing will be done without inflation. If the input mesh is a surface mesh, the volume(s) should be marked by material point(s).

The Octree Method

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, enabling 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 16: Initial 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 17: Tetra Enclosing the Full Geometry

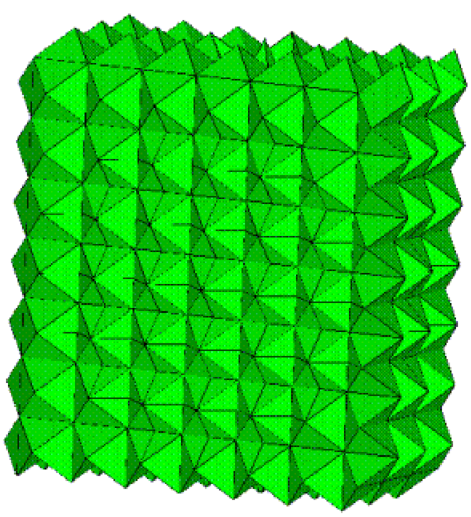


Figure 18: Tetra Enclosing the Full Geometry in Wire Frame Mode

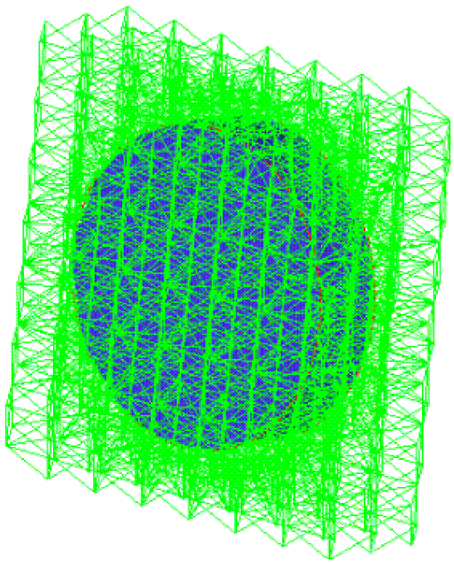
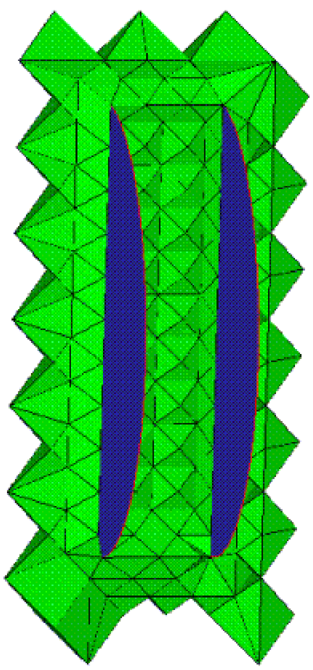


Figure 19: Cross-Section of the Initial Meshing



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 that cannot be reached by a user-defined material point without intersection of a surface.

Figure 20: Mesh after Tetra Captures Surfaces and Separates the Useful Volume

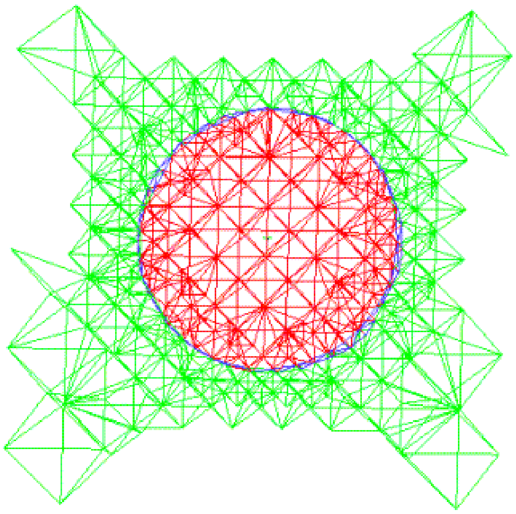
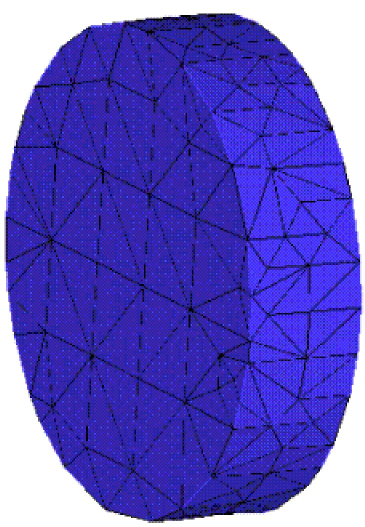
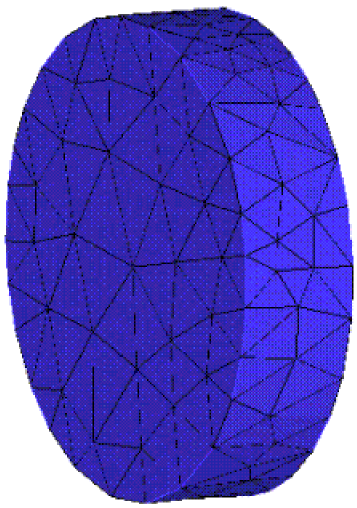


Figure 21: Final Mesh before Smoothing



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

Figure 22: Final Mesh after Smoothing



Computing the Tetra Mesh

Additional options such as disabling the surface mesh display, generating prisms or hexa elements, and selecting specific parts may be set before computing the tetrahedral volume mesh using the parameters specified. See Compute Volume Mesh

Prism Meshing

Tetra meshing is not efficient for capturing shear or boundary layer physics. Using inflation elements (prism, penta-6 or hexa) along surfaces efficiently increases boundary layer resolution perpendicular to the wall, allowing you to capture these effects. With Ansys ICEM CFD, mixed prism and tetra generation is automatic and intelligent, creating prism layers near the surface while maintaining the ease and automation of Tetra mesh. Prism has always been necessary for CFD customers, but now that the feature 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.

To capture the Y+ for a Navier-Stokes mesh, the spacing of the prism layers is the primary concern, with the rate of volume change between cells also important. Calculations are done between nodes or elements, and a 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.

Prism Mesh Process

The prism mesh process generates prism elements near boundary surfaces from tetrahedral volume or triangular surface mesh. There are two main processes:

- The post inflation process creates prisms between the boundary shells and the adjacent tetrahedral elements. The process can remove tetrahedral elements and smooth the mesh to yield the necessary quality as the inflation layer grows into the existing volume mesh. The resulting prisms are made conformal with the existing tetrahedral volume mesh.
- The pre inflation process first creates the boundary layer elements from the surface mesh. The remaining volume is then filled with unstructured mesh. If a volume mesh exists, it is first removed. If no surface mesh exists, it must first be generated from the geometry.

Note:

- This process is available using the Fluent Meshing executable, run in batch from ICEM CFD.
- Pre inflation prism meshing (Fluent Meshing method) requires either a volume mesh or material point to define a direction for prism growth.
- To define a direction for pre-inflation prism growth into a volume containing more than one region of the same material due to an internal wall, an initial volume mesh (for example, tetrahedral by octree method) is needed.

Prism Mesh Preparation

When generating prism mesh, preparation is essential. It is easier to edit a tetra mesh than a tetra/prism mesh. A 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 a Tetra-volume mesh or Tri-surface mesh.
- Check aspect ratios and quality.

  Highly skewed or low quality elements are not suitable for prism growth.
- 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 and repair 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** > **Global Mesh Setup** > **Prism Meshing Parameters** when creating the prism mesh.

  Laplace is typically the best option for **Triangle Quality type** for the eventual prism quality.

Choosing Prism Options

All necessary parameters are specified globally on the **Mesh** > **Global Mesh Setup** > **Prism Meshing Parameters** DEZ. A full description is contained at Prism Meshing Parameters in the Help manual.

Certain parameters can then be adjusted on a part-by-part or entity-by-entity basis using **Part Mesh Setup**, **Surface Mesh Setup** and **Curve Mesh Setup**. For instance, you can specify 3 prism layers with a growth rate of 1.2 globally, but set 5 layers on a certain part. Part settings override global settings and entity (surface or curve) settings override part settings. Between entities the smaller size overrides the larger. Setting a specific parameter on a single entity within a part or between parts is handled intelligently. For example, if you set a local parameter, such as height, on a single curve entity, the prism mesher will interpolate that parameter smoothly across the surface between curves.

You may also select volume parts for prism growth. If no volume parts are selected, ICEM CFD assumes that you want to grow prisms into all volumes bordering the prism surfaces. If you select specific volume parts, then prism layers will be grown into only those volumes.

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 last layer height and prisms total height (unless limited by 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. If you are concerned about Y+, you can then adjust the first-cell height using **Edit Mesh** > **Split Mesh** > **Split Prisms**.

Before the first layer is created and after the last layer is created, surface and/or volume smoothing is done according to the global settings and selected process. The layers are grown one at a time with only directional smoothing applied before extruding each layer. This continues until all the requested layers are grown. The smoothing is the most time-consuming operation, so for simple configurations, it may be best to turn off all smoothing and grow all the layers one at a time. This enables you to take advantage of the variable height feature.

Computing the Prism Mesh

There are two ways to initiate the prism mesh computation:

Compute Mesh > **Volume Mesh** > **Create Prism Layers**

You enable this option to create prisms next to wall geometries as part of the volume meshing process. You can choose whether to create prism mesh from the geometry, or from the surface or volume mesh depending on the **Mesh Method** chosen.

All volume meshing parameters are set globally using **Mesh** > **Global Mesh Setup** before clicking **Compute.**

Compute Mesh > **Prism Mesh**

You can use this option to grow prism layers next to wall geometries, from an existing volume mesh or surface mesh. If the existing volume mesh is tet/hexa mesh, on the hexa side the prisms will be added within the first hexa layer.

Prism growth parameters are set globally using **Mesh** > **Global Mesh Setup** > **Global Prism Settings**. Local parameters, which override the global parameters, can be set as described in the previous section (p. 24) or using **Select Parts for Prism Layer**.

The inflation process (p. 23) is chosen before clicking **Compute**.

Note:

- You can add prisms to existing layers or you can subdivide and redistribute layers. If many prism layers are needed, it can be faster and 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.

Smoothing a Hybrid Tetra/Prism Mesh

Smoothing options are described in Smoothing the Mesh Globally (p. 43) in this manual.

It is easier to smooth a triangle or tetrahedral mesh than a hybrid tetra/prism mesh. First smooth only the **Tetra** and **Tri** elements by selecting **Freeze** for other element types.

Once the tetra and tri elements are as smooth as possible, then all elements are smoothed concurrently. Decrease the quality **Up to value** if the prism elements are severely distorted by smoothing.

Hexa Meshing

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

Ogrids

For very complex geometry, Hexa automatically generates body-fitted internal and external Ogrids to parametrically fit the block topology to curved 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 can 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 can 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.

Periodicity

Applicable when analyzing rotating machinery applications, for example, Hexa allows you to define the periodic behavior of a section of the system and mesh only that section, thereby minimizing the model size. Translational periodicity is also supported.

Link Shape

This enables 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.

The Hexa Database

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

The Geometric data entities are:

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

The Block Topological data entities are:

- **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

Color coding in Hexa

The topological entities are color-coded based on their properties:

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:**
>
> If you have chosen a light-colored background for the graphics display window, these display in black.

Blue Edges and Vertices

These are internal edges within a material 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 edges will take the shape of the curves when meshed, and 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 Mesh Generation with Blocking

The process to generate a Hexa mesh is outlined below, and described in detail in the following sections.

1. Import a geometry file using any of the direct, indirect, or faceted data interfaces.
2. Interactively define the block model through split, merge, Ogrid definition, edge/face modifications, and vertex movements.
3. Check the block quality to ensure that the block model meets specified quality thresholds.
4. Assign edge meshing parameters—such as maximum element size and initial element height at the boundaries—and set the expansion ratios.
5. Generate the mesh with or without projection parameters specified.
6. Check the mesh quality to ensure that specified mesh quality criteria are met.
7. Write output files to the desired solvers.

If necessary, you can 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 can be saved at any time, 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.

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 primary 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.

Blocking Strategy

With Hexa, the basic steps necessary to generate a hexahedral mesh are the same, regardless of model complexity.

1. Create an initial blocking (Initialize Blocks) using one of the various approaches to blocking.

2. Edit the blocking topology using splitting, merging and shaping techniques so that the blocking topology represents the simulation model, composed with hex shapes wherever possible.

3. Fine tune mesh sizes and tweak topology get a good quality premesh.

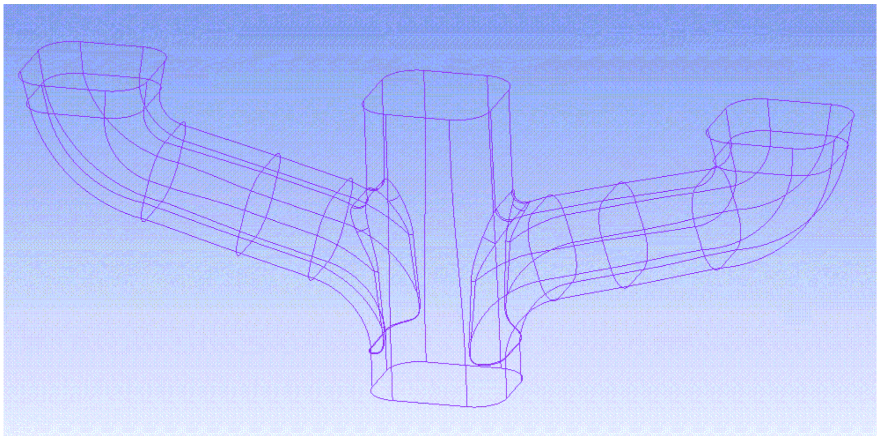
When you are satisfied with the pre-mesh quality, convert the pre-mesh to the appropriate type of mesh - unstructured or multiblock.

There are two general strategies for blocking a 3D model. These are commonly referred to as Top Down (p. 30) or Bottom Up (p. 31).

Top Down

The top down blocking approach starts from one block generated from the 3D bounding box. The blocking topology is shaped to the simulation model using split, delete, merge and move tools.

Figure 23: Top Down Blocking Strategy



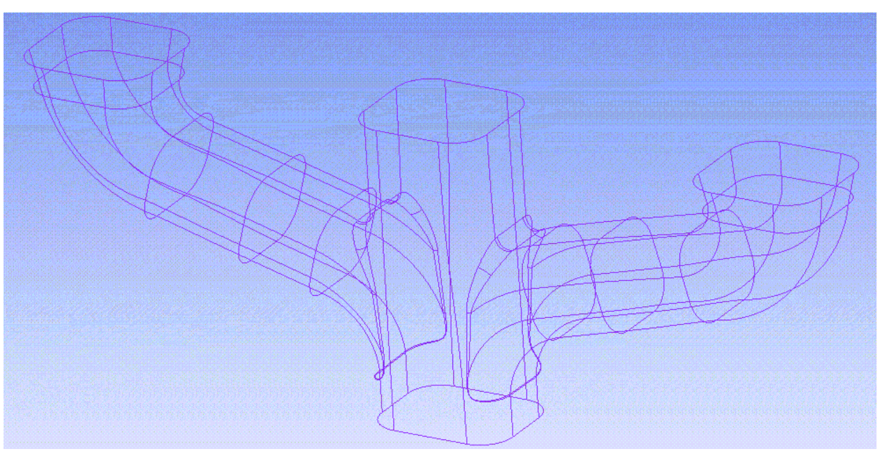
This approach has the following advantages:

- It is often easier to get an all hex mesh using this approach since the global topology is a hex mesh. When splits are made, the blocking remains as all 3D mapped blocks and there is an index control which maintains the connectivity for fast mesh calculations.
- The global topology helps avoid getting stuck in a corner, but requires some upfront thought on how the global topology should be constructed to match the CAD topology.
- There are no direct associations to the CAD when the blocking is initialized. Such associations are controlled manually. The mesh can be generated at any time and the blocking projected to the surfaces, allowing you to check and add shape as the blocking progresses.

Bottom Up

The bottom up blocking approach, also called 3D MultiZone, starts from the surface structure of the 3D CAD. For each surface it constructs a 2D blocking face, and then it tries to fill those 2D faces with 3D blocks.

Figure 24: Bottom Up Blocking Strategy



This approach has the following advantages:

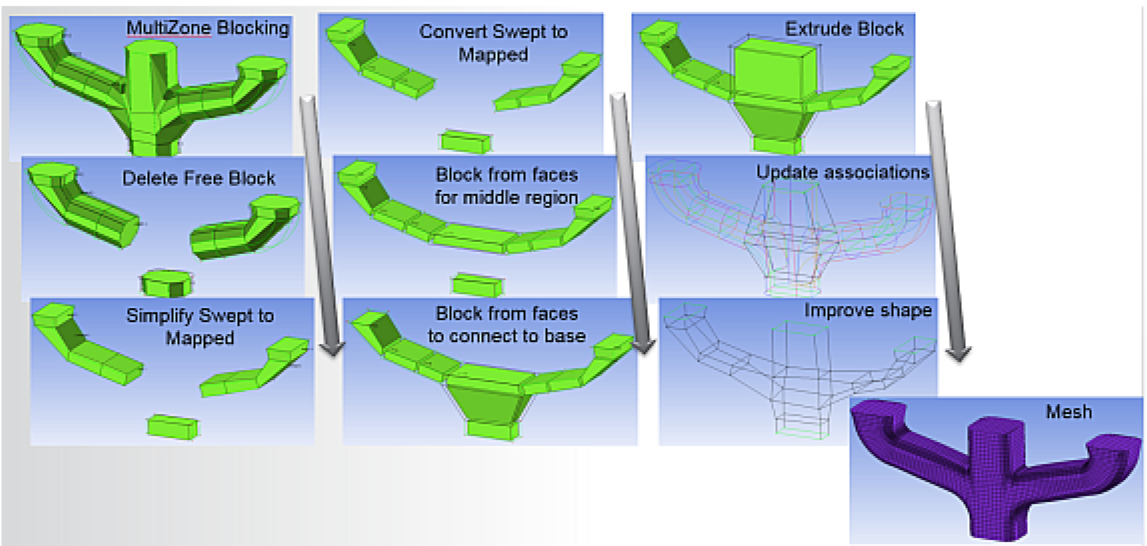
- It should always produce a mesh, but it may not be a hexahedral mesh.

Where it can it generates mapped blocks. If this is not possible it will try to generate swept blocks. If this is not possible, a free block is generated. The free block can be filled with a tet, hex-core, or hex-dominant mesh.

- This automated decomposition provides a good starting point to meshing. A user can then do manual decomposition to get more hex mesh. This flexibility is often helpful as a user doesn't always know how long it will take to get a hex mesh, so it is reassuring to know that a mesh can be generated at any time.
- The blocking topology model is automatically associated (shaped) by the CAD. A user can gradually release those constraints as the blocking progresses to get more hex mesh.

While the two general approaches are identified as top down and bottom up, a user may use a combination to successfully block any given geometry.

Figure 25: Mixed Blocking Strategy

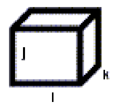
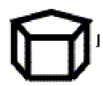
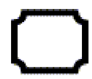


Hexa Block Types

A description of Mapped, Swept and Free block types is shown in Figure 26: Hexa Block Types (p. 33).

- Mapped blocks are required to have an equal number of nodes on parallel edges. In 3D blocking, this will result in hexahedral cells; quadrilateral cells in 2D blocking.
- Free blocks may have different numbers of nodes on opposite sides, resulting in non-hexa (quad if 2D) cells.
- Swept blocks are identified by having two free faces opposite each other; and mapped faces on the perpendicular sides between.

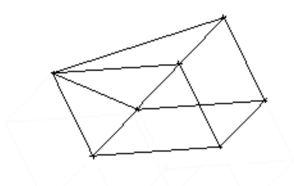
Figure 26: Hexa Block Types

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

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 having mapped blocks, the split will propagate through faces that have a mapped relationship to 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 necessarily have the same 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.

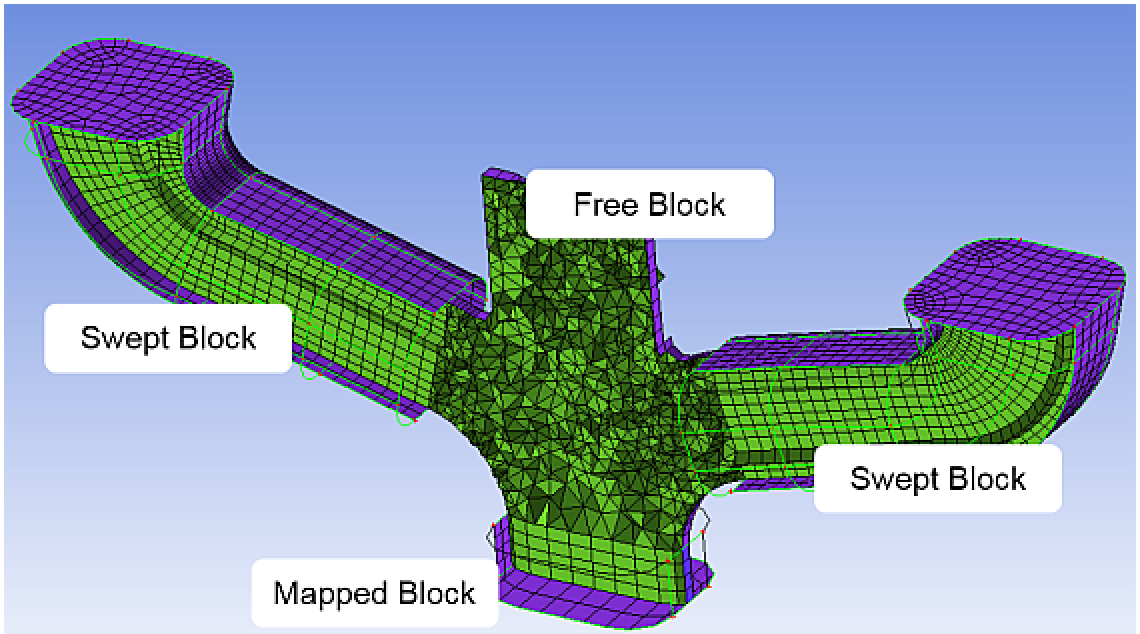
Some operations may result in a **degenerate** block - a 7-sided mapped block. Degenerate blocks have special behavior for mapped blocking and are not allowed in swept or free blocking.

Figure 27: Degenerate Block



The ability to Convert Block Type 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 28: Mesh Generation within Block Types



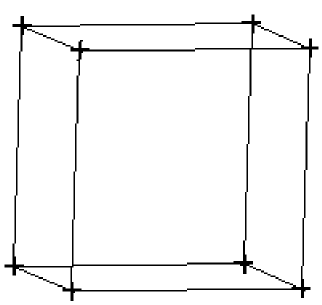
Automatic Ogrid Generation

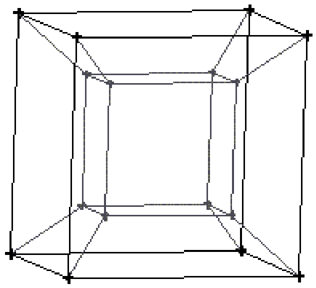
Generating **Ogrids** is a very powerful and quick technique used to achieve a quality mesh to model geometry when you desire a circular or "O"-type mesh, either around a localized geometric feature or globally around an object. This would not be possible without Ogrids.

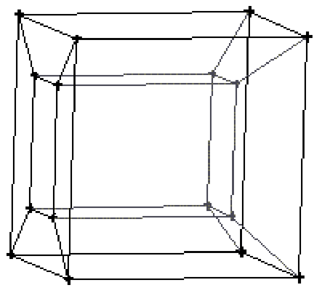
Using the Automatic Ogrid

The **Ogrid** 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 **Ogrid** generation technique; the **Ogrid** shown below is created entirely inside the selected block.

Figure 29: Initial block, Block with Ogrid, Ogrid with Add Face







Using the **Add face** feature, an **Ogrid** can be created such that the **Ogrid** passes through the selected block faces. In Figure 29: Initial block, Block with Ogrid, Ogrid with Add Face (p. 34), the **Add face** feature was used on the last block to include one face on the block prior to generating the **Ogrid**.

Important Features of an Ogrid

Generation of Orthogonal Mesh Lines at an Object Boundary

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

Rescaling an Ogrid After Generation

When the **Ogrid** is generated, its size is scaled based upon a factor in the **Blocking >Edit Block > Modify Ogrid** window. The **Rescale Ogrid** feature enables you to re-scale the previously generated **Ogrid**. If a value less than 1 is assigned, the resulting Ogrid will be smaller than the original. Conversely, a value larger than 1, will result in a larger Ogrid.

The blocks can 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 is 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 feature is accessible by selecting **Blocking > Edge params**.

You can use the following predefined **Mesh laws**. These are described in Bunching Laws:

- Bi-Geometric (default)

- Bi-Exponential
- Curvature
- Exponential 1
- Exponential 2
- FullCosinus
- Geometric 1
- Geometric 2
- HalfCosinus1
- HalfCosinus2
- Hyperbolic
- Linear
- Poisson
- Spline
- Uniform

In addition to the predefined edge meshing functions, there are several interactive or user-defined Mesh laws available in the drop-down list.

The interactive or user-defined mesh laws include:

- FromGraphs

Note:

By selecting this option, a dialog box opens in which you select a source file or function and then add, delete, or otherwise modify the control points describing the edge parameter settings.

- FromEdgePoints
- OnScreen

Additional tools such as **Linked Bunching** and multiple **Copy** options provide you with the ability to apply the specified edge bunching parameters quickly to the entire model.

Tip:

You can use variables in a replay script as a means to parameterize edge parameters. See Parameterizing Edge Parameters (p. 38) for details.

Smoothing Techniques

In Hexa, both the block topology and the mesh can be smoothed to improve the overall block/mesh quality either in a certain region or for the entire model. The block topology can 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 can smooth the mesh after specifying the proper edge-bunching parameters.

The quality criteria for smoothing are described in Pre-Mesh Quality in the *Ansys ICEM CFD Help Manual*.

Refinement and Coarsening

Use **Blocking > Pre-Mesh Params > Refinement** to refine or coarsen the mesh. You can apply the refinement/coarsening in all three major directions simultaneously or in just one major direction.

Refinement

You can use refinement for solvers that accept non-conformal node matching at the block boundaries. The refinement capability minimizes the model size while achieving proper mesh definition in critical areas of high gradients. To refine the mesh, enter a scale factor greater than 1.

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. To coarsen the mesh, enter a scale factor less than 1.

Making Parametric Changes to the Geometry

With ICEM CFD's **Replay** feature, you can analyze geometry variations. Changes in length, width and height of specific geometry features are known as *parametric changes*. Parametric changes do not affect the block topology, so the **Replay** function can automatically generate 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.

To generate a **Replay** file:

1. From **File > Replay Scripts**, 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.

2. Make a 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.

Parameterizing Edge Parameters

You can use variables in the replay script as a means to parameterize edge parameters. Here is an example:

```
#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 can update the variables at the top of the script and then rerun the script.

Analyzing Rotating Machinery

Periodic definition can be applied to the blocking in ICEM CFD. Typically, you will model only a section of the rotating machinery and implement symmetry in order to minimize the model size. By specifying a periodic relationship between the inflow and outflow boundaries, the particular specification can be applied to the model; flow characteristics entering a boundary must be identical to the flow characteristics leaving a boundary.

The periodic relationship is applied to block faces; and ensures that a node on the first boundary has two identical coordinates to the corresponding node on the second boundary. Using the **Blocking > Edit Block > Periodic Vertices** option, the application prompts you 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.

Determining the Pre-Mesh Quality

The pre-mesh quality functions are accessible through **Blocking > Pre-Mesh Quality**.

Applying any of the quality checks produces a histogram plot. The quality **Criterion**, typically in the range of 0 to 1, is subdivided and assigned to **Bars** on the x-axis. The y-axis measures the number of elements in a given range and is scaled from 0 to a value that is indicated by the **Height**.

Determining the Location of Elements

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

Some of the quality metrics are:

Determinant

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. In general, determinant values above 0.3 are acceptable for most solvers.

Angle

Checks the maximum internal angle deviation from 90° for each element. 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. The histogram x-axis will cover the range of 0 to 90°.

Volume

Computes the internal volume of the elements in the model. The x-axis will show volume in the unit that was used to create the model.

Warpage

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

A full list of pre-mesh quality **Criteria** is available in Pre Mesh Quality Options

Unstructured and Multi-block Structured Meshes

The hexa mesh can be output in either unstructured or multi-block structured format. The format decision can be delayed until after you have finished the whole meshing process when the output file format is selected.

Unstructured Mesh Output

The unstructured mesh output feature 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 feature 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** feature enables you to minimize the number of output files generated with the multi-block structured approach. See Pre-Mesh > Output Blocks.

The mesh output file(s) is (are) created from the context-sensitive menu under **Blocking > Pre-Mesh** in the Display tree.

Checking and Improving the Mesh

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.

Checking the Mesh

Check the validity of the mesh using **Edit Mesh** > **Check Mesh**. ICEM CFD includes several checks for **Errors** as well as **Possible problems**. The descriptions of each of these checks can be found in Check Mesh in the *Ansys ICEM CFD Help Manual*.

You can opt to use the **Create subsets** feature for each of the problems so that they can be fixed later or can opt to use the **Check/fix each** feature 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, and so on.

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 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 features can be accessed by right-clicking on Mesh in the Display Tree.

> **Note:**
>
> After mesh editing the diagnostics should be re-checked to verify that no mistakes were made.

Quality Metrics

The **Display Mesh Quality** feature runs a diagnostic check of individual element quality and displays the mesh quality as a histogram. 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. Click a bar in the histogram to identify all the cells within that quality sub-range, in the graphics display.

For descriptions of all the quality metrics, refer to Display Mesh Quality in the *Ansys ICEM CFD Help Manual*.

You can modify the display of the histogram by adjusting the values of **Min**, **Max**, **Height**, and **Bars**. Right-click the histogram to access the following features for modifying its display attributes.

- The **Replot** feature opens a small window that enables 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** feature 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 features control how the selected elements are displayed.
- **Solid**: Enabling this feature will display the elements as solid, rather than the default grid representation. (**Show** must be enabled.)
- **Color by Quality**: If available, enabling this feature will display the elements in the same color as the selected Quality bar in the histogram. (**Show** must be enabled.)
- **Highlight**: If available, this feature enables 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** feature enables you to add elements to an already established subset.

Smoothing the Mesh Globally

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

The smoother modifies the elements with quality below the specified **Up to quality** value, based on the selection from the list of available quality criteria. 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.

The triangular surface mesh smoother operates independently of the volume mesh smoother. Initially, all elements that are below the quality threshold of the specified criterion are marked, and then the specified number of smoothing steps are run on those elements. Nodes movement is constrained to the actual CAD surfaces during the smoothing process.

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** feature 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** feature is selected, the nodes of the specified element type will be capable of moving freely, enabling 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.

Advanced Options for Smoothing the 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 feature 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 feature works much like **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** feature will enable 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 feature disabled. The goal is to reduce the number of elements that are attached to one vertex by refinement in problem regions.

Laplace smoothing

This feature 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 feature works only for the triangular surface mesh.

Allow node merging

This feature 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 feature 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 feature will allow the smoother to continue smoothing beyond the worst regions until the desired quality is obtained.

Surface fitting

This feature will smooth the mesh, keeping the nodes and the new mesh restricted to the surface of the geometry. Only **Hexa** models will use this feature.

Ignore PrePoints

This feature will allow the smoother to attempt to improve the mesh quality without being bound by the initial points of the geometry. This feature is similar to the **Violate geometry** feature, but works only for points located on the geometry. This feature 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.

Mesh Editing Tools

In addition to the automated mesh improvement options, there are several manual diagnostic and repair tools that enable you to build mesh topology, remesh bad elements and fill holes easily. 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. Other tools allow you to merge or split the mesh, move elements, convert mesh type, change density, or otherwise manipulate the mesh.

These are fully described at Edit Mesh.

Using the Properties Menu

The **Properties** menu enables 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

Enables you to 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.

Write Material File

Saves the material specification so that it can be reused whenever necessary. The material file will be saved with the `.mat` extension.

Load Material From File

Opens a material file to be used in your design or to be modified and saved for future use.

Create Table

Enables you to create your material property by entering values for x and y. You can even graph the property.

Define Element Properties

These features enable 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 you choose the part, the various properties are applied to its elements.

Using the Constraints Tab

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 features such as **Contact definition**, **Velocity**, and **Rigid Wall.**

Create Constraint / Displacement

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

Define Contact

Enables you to define contacts by Automatic Detection or Manual Definition.

Define Single Surface Contact

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

Define Initial Velocity

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

Define Planar Rigid Wall

Enables you to define a Planar Rigid Wall by specifying the Head and Tail coordinates.

Working with Loads

On the **Loads** tab, there are several features available for applying internal and external loads, such as force, pressure, and temperature.

Force

Pressure

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 you are 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.

Force

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

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

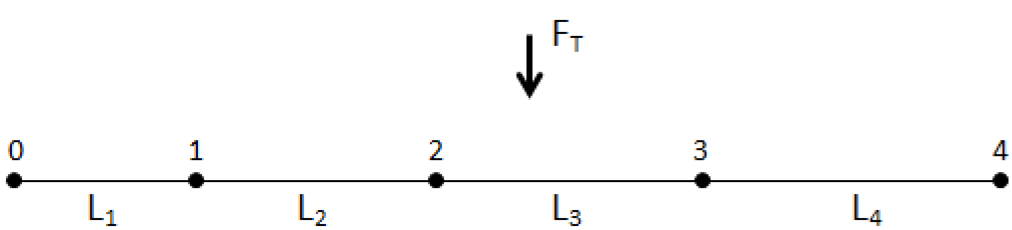
Distributed Forces Theory

Total forces are distributed as described below, depending on the entity type.

- **By Curve**

  The total Force '$\mathbf{F_T}$ ' can 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.

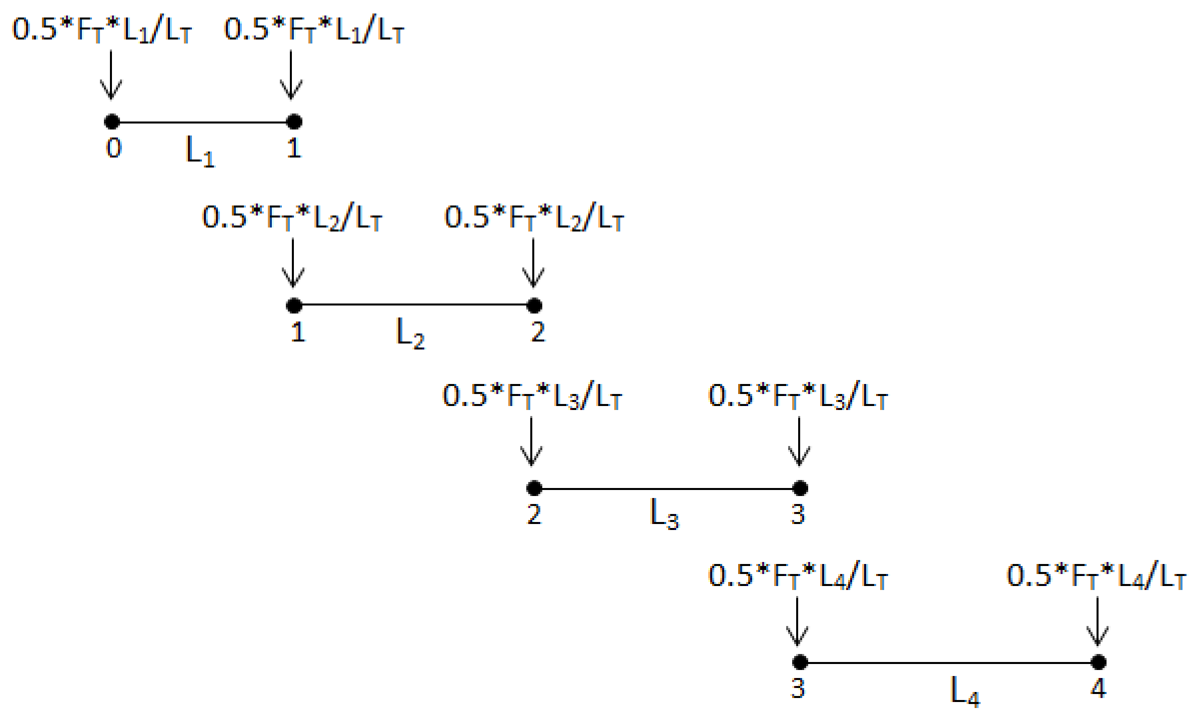
  Figure 30: Force on a Curve



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

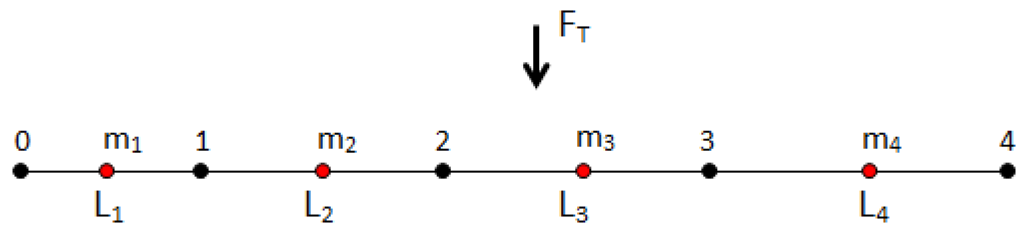
In general, the force at any Node is: $\mathbf{F_i = Sum\ [F_T * (L_j / L_T) * (1 / N_j)]}$, where *i* is the node number, *j* is the element number, L_j is the length of element j, and N_j is the number of Nodes attached to element j.

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 can also apply loads directly to the elements (select the elements directly rather than the geometry entities). This can occur, for example, if you do not 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.

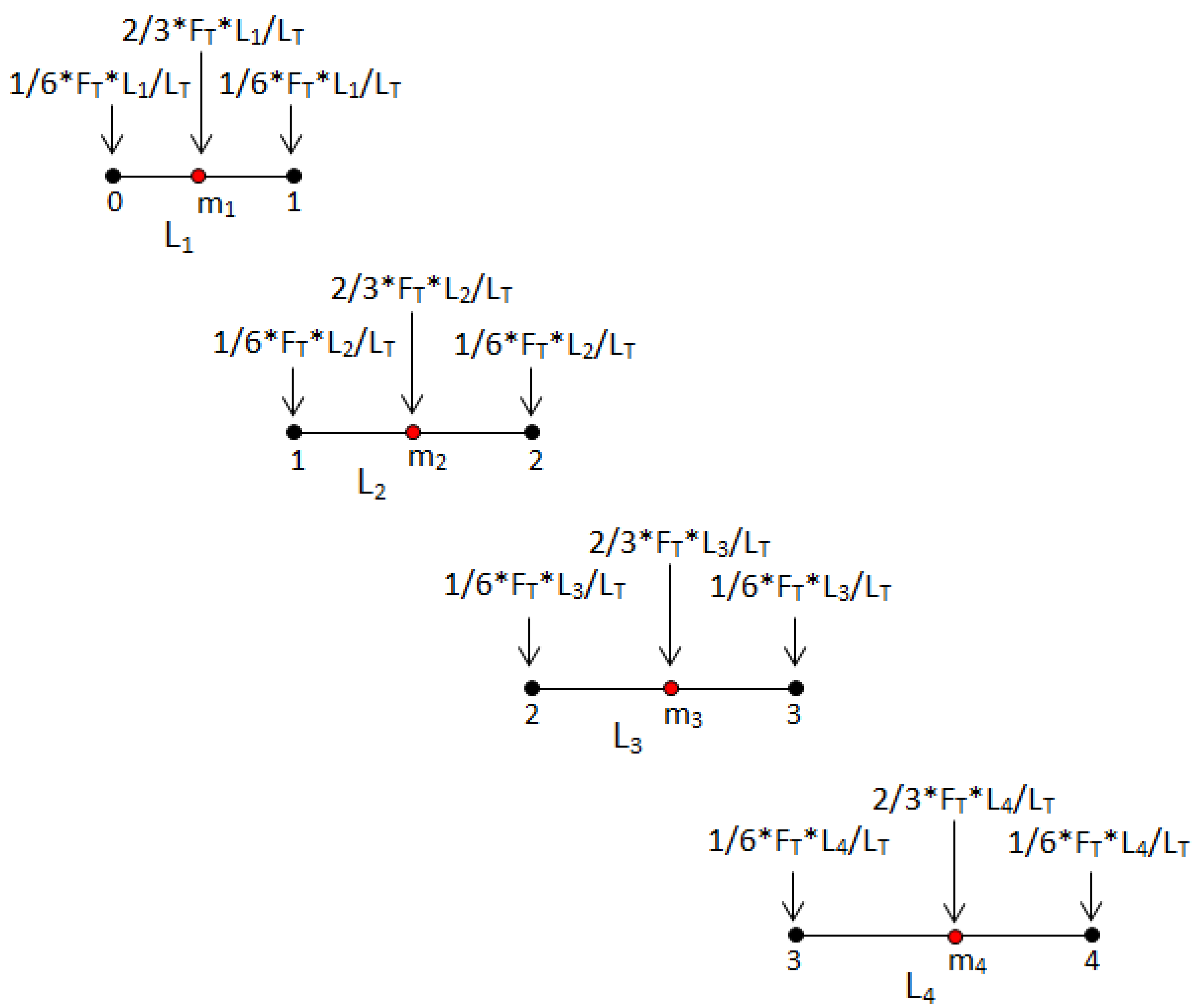
Figure 32: Force on Elements with mid-side nodes



Node numbers and element lengths are defined as before. In addition, each element has a mid-side node labeled 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 33: Quadratic Load Distribution (p. 53).

Figure 33: 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: $F_{m4} = 2/3 * F_T * (L_4 / L_T)$

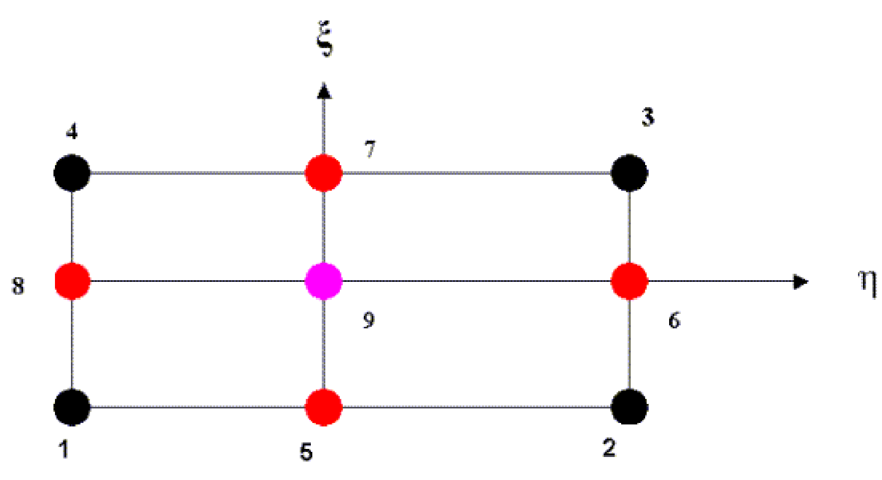
As in the previous case of linear distribution, you can check the total Force by adding the individual nodal forces:

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

- **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 34: QUAD 9 Element (p. 54) for an illustration.

  Figure 34: 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:

$$N_1(\xi,\eta)=\frac{1}{4}\cdot\xi\cdot\eta\cdot(\xi-1)\cdot(\eta-1)$$

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

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

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 = 4P.

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.

Pressure

You can apply pressure loads to Surfaces, Subsets, or Parts. Pressure loads may be uniformly applied or described using a set of vectors. See Place Pressure for more detail.

Temperature

This feature enables you to apply temperature to Points, Curves, Surfaces, Bodies, Subsets or Parts. Typically, the temperature load is applied uniformly to the selected entity. See Create Temperature Boundary Condition for more detail.

Sending Data to a Solver

The **FEA Solve Options** tab and **Output Mesh** tab have tools for selecting an output solver and then specifying the type of analysis and other parameters necessary to compute a solution.

The **FEA Solve Options** tab is used to setup for structural analysis. Choose from one of the following common solvers: **Ansys**, **Nastran**, **ABAQUS**, **Autodyn**, and **LS-DYNA**.

Setup Solver Parameters

Your choice of solver determines which additional parameters are required. ABAQUS and Autodyn do not require additional solver parameters.

Setup Analysis Type

Depending on the selected solver, different features are available. For the Ansys solver, you can select either Structural or Thermal. If Nastran solver is selected, then you can choose from more **Analysis** types.

Setup Sub-Case

Enables you to create subcases to apply the load in different steps.

Write/View Input File

Enables you to create and view the input file generated for the solver.

Submit Solver Run

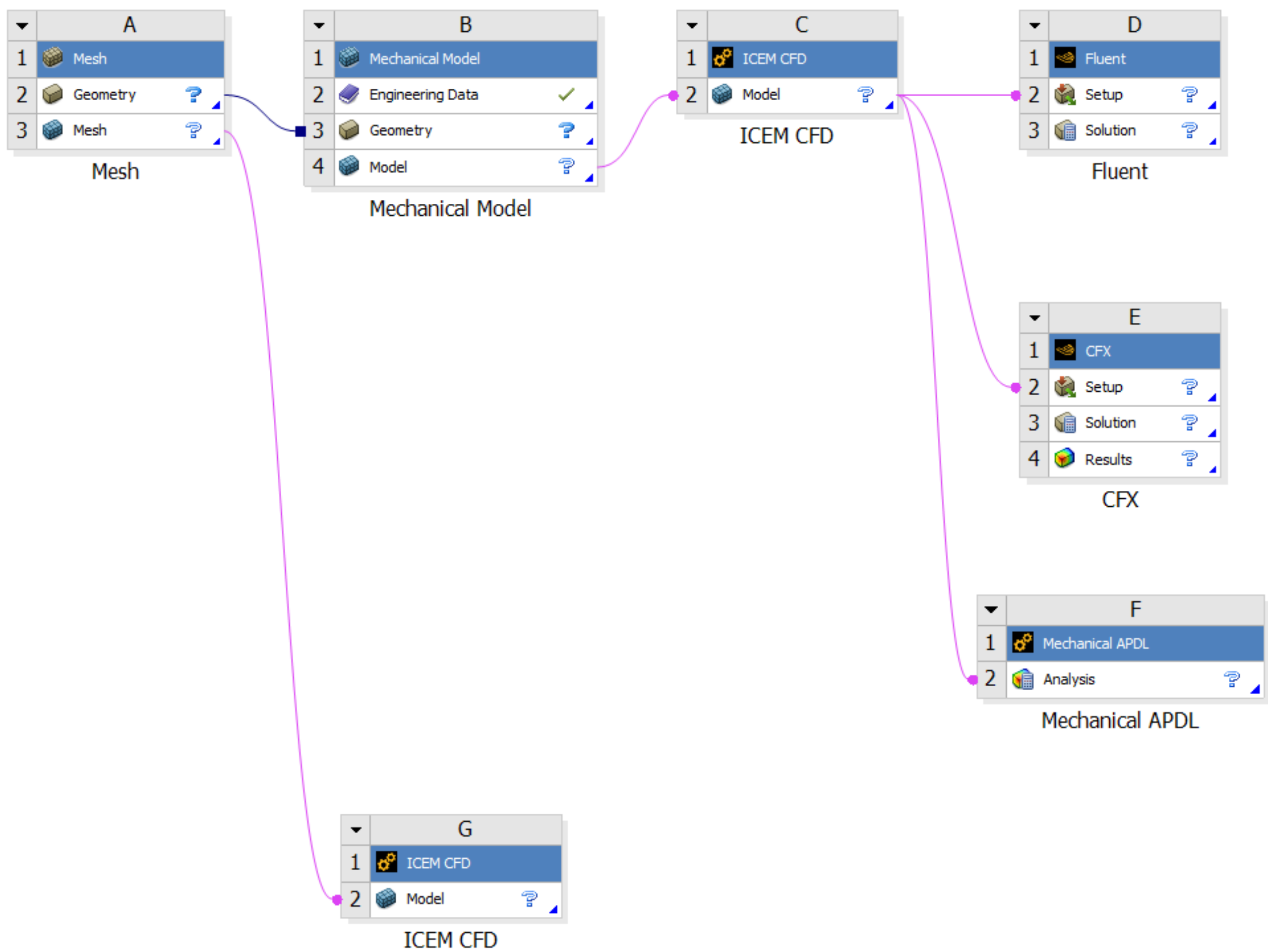
Enables you to send an input file to a particular solver.

The **Output Mesh** tab is used for CFD analysis and exporting the mesh in supported formats. Choose from one of the supported **Output Solvers**, and then setup **Boundary Conditions** and other **Parameters** before writing the solver input file.

More information about the supported solvers is available from the **Help** menu. The **Output Interfaces** feature 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, FENSAP-ICE, and Mechanical APDL.



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

- For a description of the ICEM CFD component, see Elements of the ICEM CFD Component.
- For instructions on how to initiate an ICEM CFD component, see Creating an ICEM CFD Component.
- For a description of how an ICEM CFD component iteracts with the Workbench environment, see Updating ICEM CFD Projects.

- For a description of the user interface is changed, see Interface Differences in the Data-Integrated ICEM CFD.
- For instructions on how to pass parameters into or out of an ICEM CFD component, see Setting Parameters.
- For an example of setting a user-defined input parameter, see User-Defined Parameters Example.
- For special instructions on how to interface to Static Structural, see Transferring an ICEM CFD Project to Static Structural.