ANSYS ICEM CFD/AI*Environment 10.0 User Manual

ANSYS ICEM CFD/AI*Environment 10.0 User Manual

Table of Contents

List of Figures

Chapter 1: CAD Repair

Before generating the **Shell/Tetra** mesh, the user should confirm that the geometry is free of any flaws that would inhibit optimal mesh creation. If the user wishes to save the changes in the native CAD files, the following checks should be performed in a direct CAD interface.

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

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

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

For more useful information on CAD Repair topics, please go to http://www-berkeley.ansys.com/faq/faq_topic_8.html.

1.1. How are Close Holes and Remove Holes different?

1.1.1. Close Holes

Use Close Holes if the hole is bounded by more than one surface. For example, look at Figure 1-1 below. The yellow curves represent the boundary of the hole. From the figure it is clear that this hole is bounded by more than one surface. Figure 1-2 shows the geometry after Close Holes is completed. A new surface is created to close the hole.

Figure 1.1 Close Hole

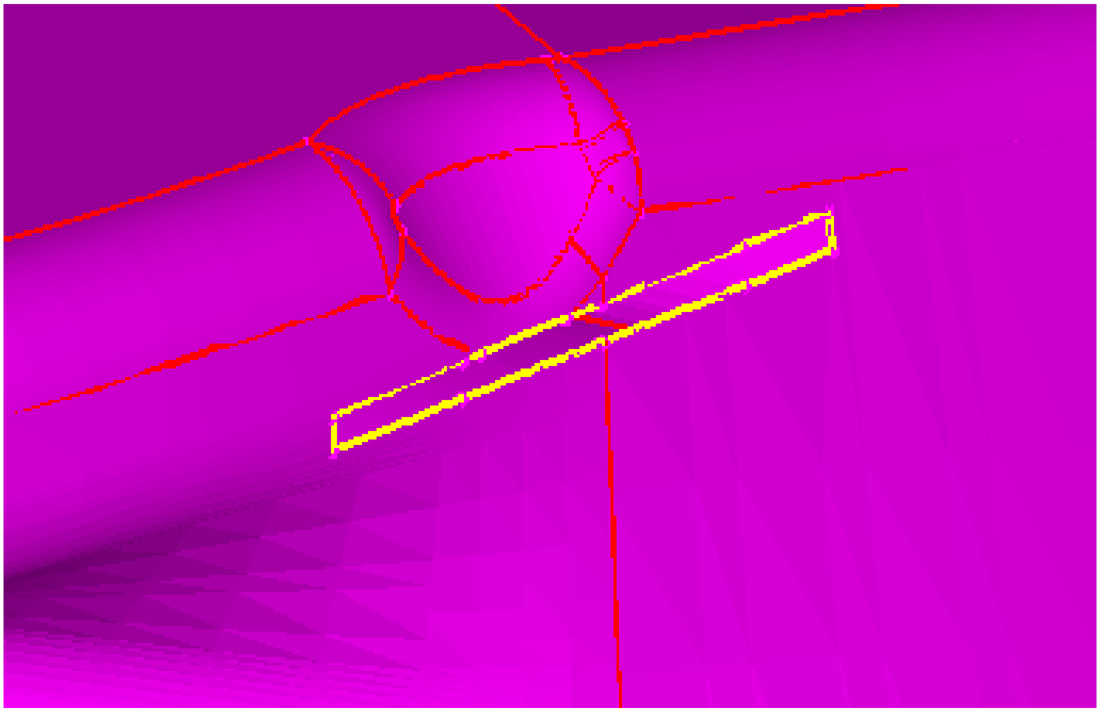
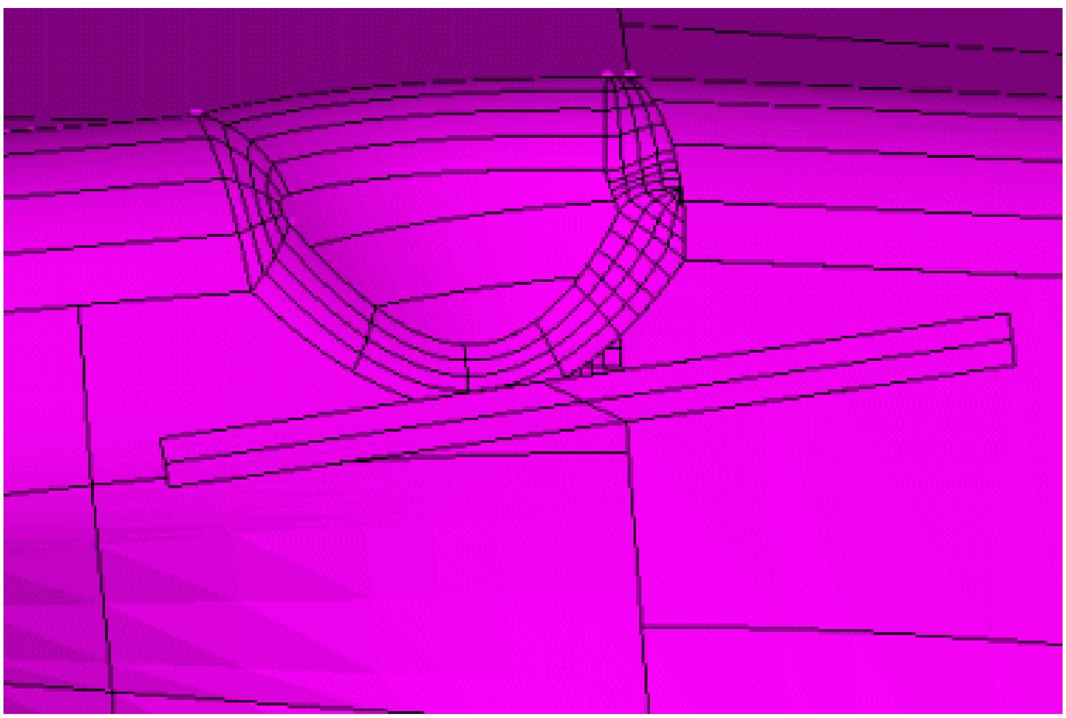


Figure 1.2 After closing the hole



1.1.2. Remove Holes

Use Remove Holes if the hole lies entirely within a single surface, such as a trimmed surface. For example, look at Figure 1-3. The two yellow curve loops represent the boundaries of the holes, which lie entirely in one surface. Figure 1-4 shows the geometry after Remove Holes is completed for one of the holes. The existing surface is modified by removing the trim definition.

Figure 1.3 Before Remove Holes

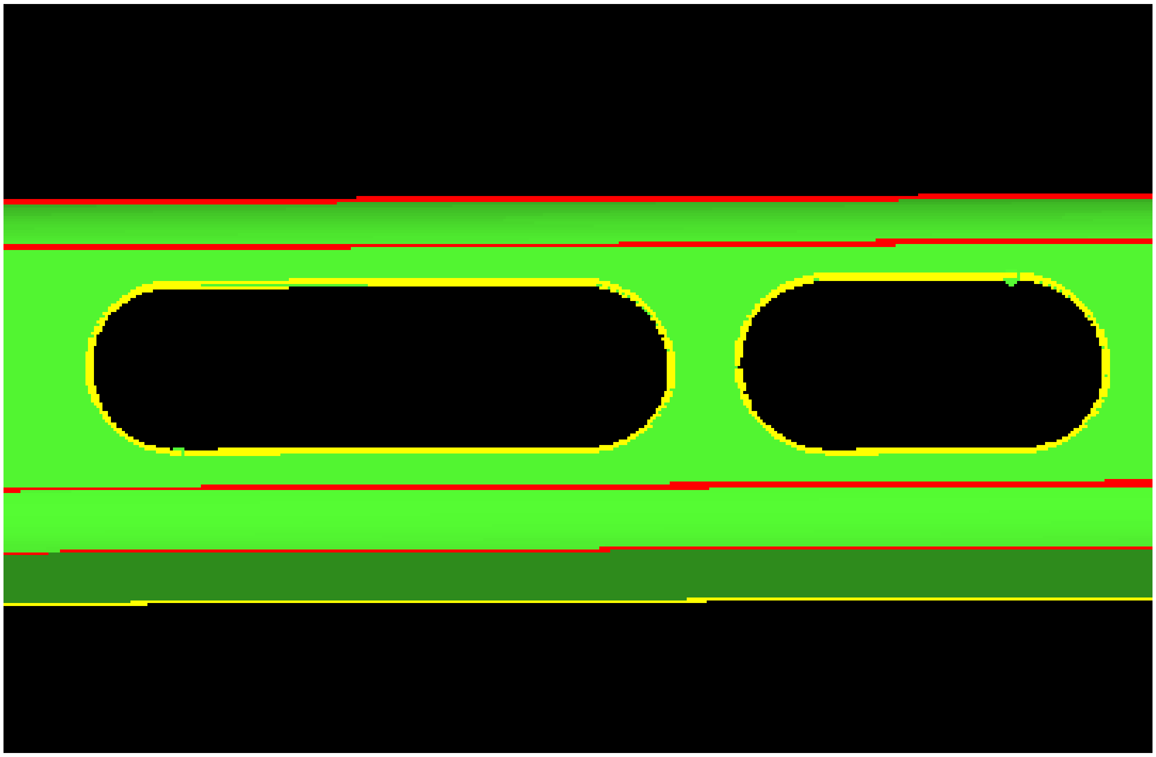
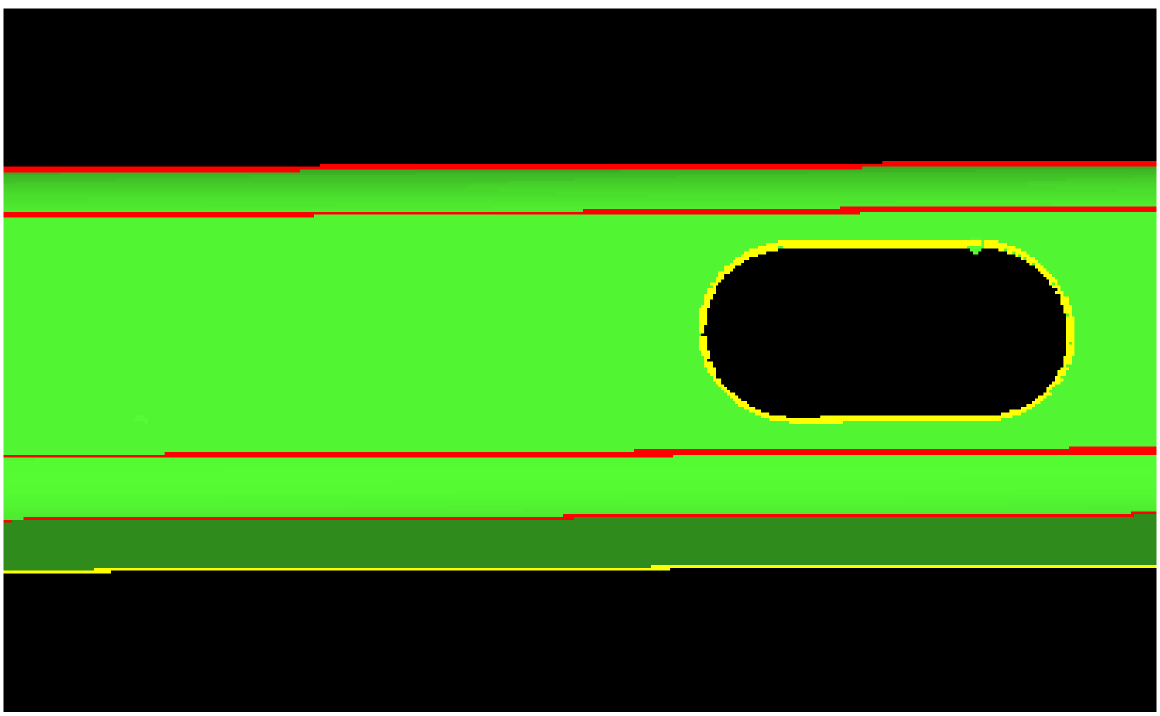


Figure 1.4 After Remove Holes



1.2. How do Fill, Trim and Blend work in Stitch/Match Edges?

Consider the case as shown in Figure 1.5: "Geometry with a gap". The following figures explain how these work:

Figure 1.5 Geometry with a gap

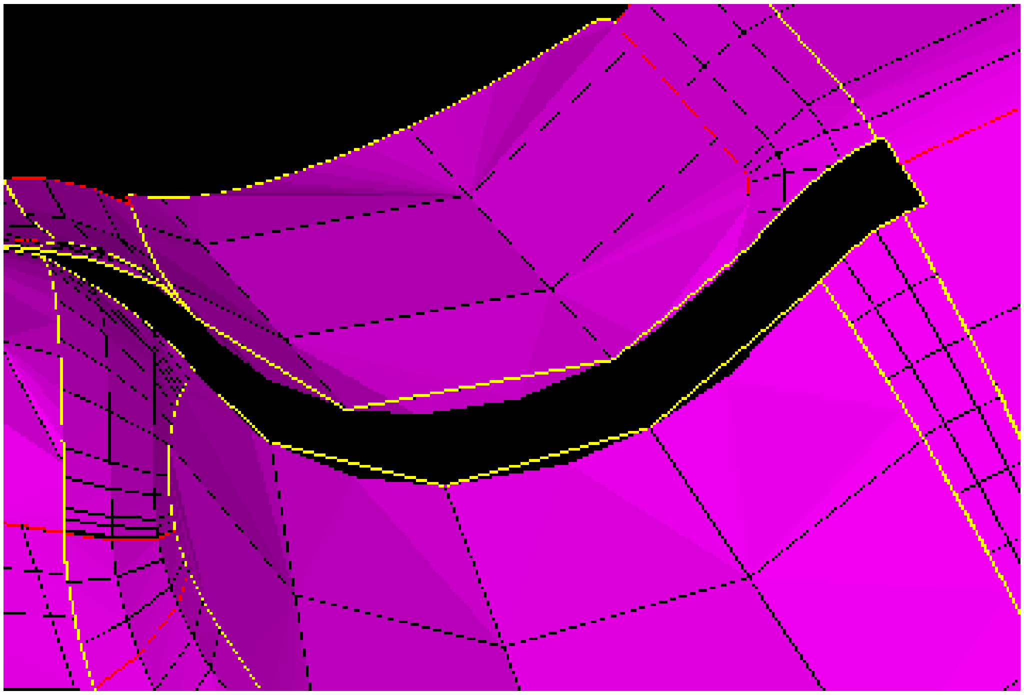


Figure 1.6 Result with Close Gap > Fill

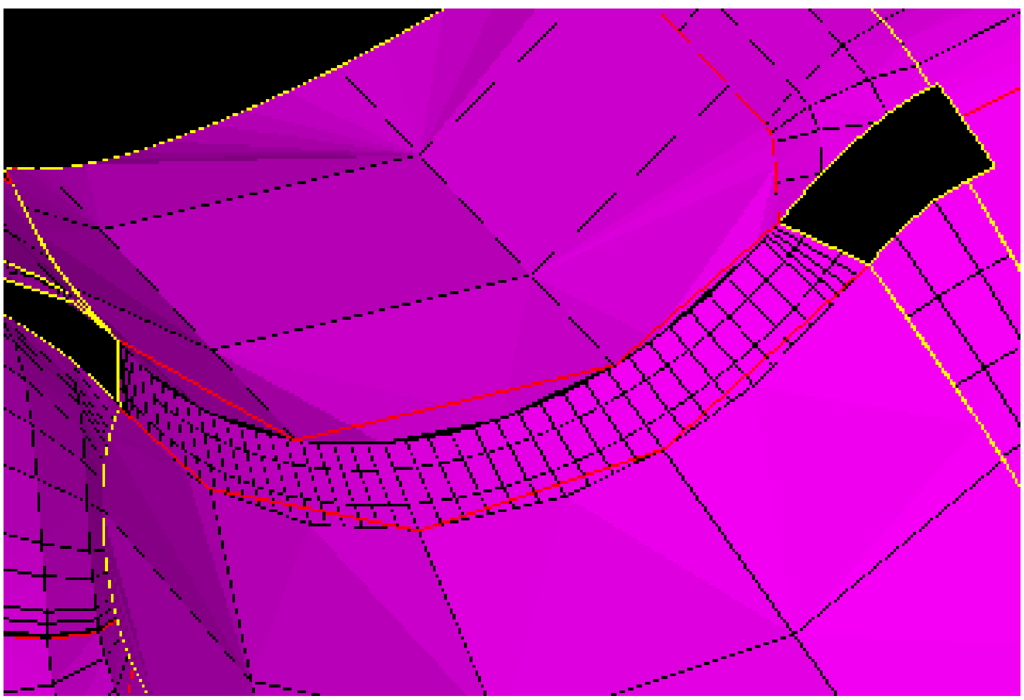


Figure 1.7 Result with Close Gap > Trim

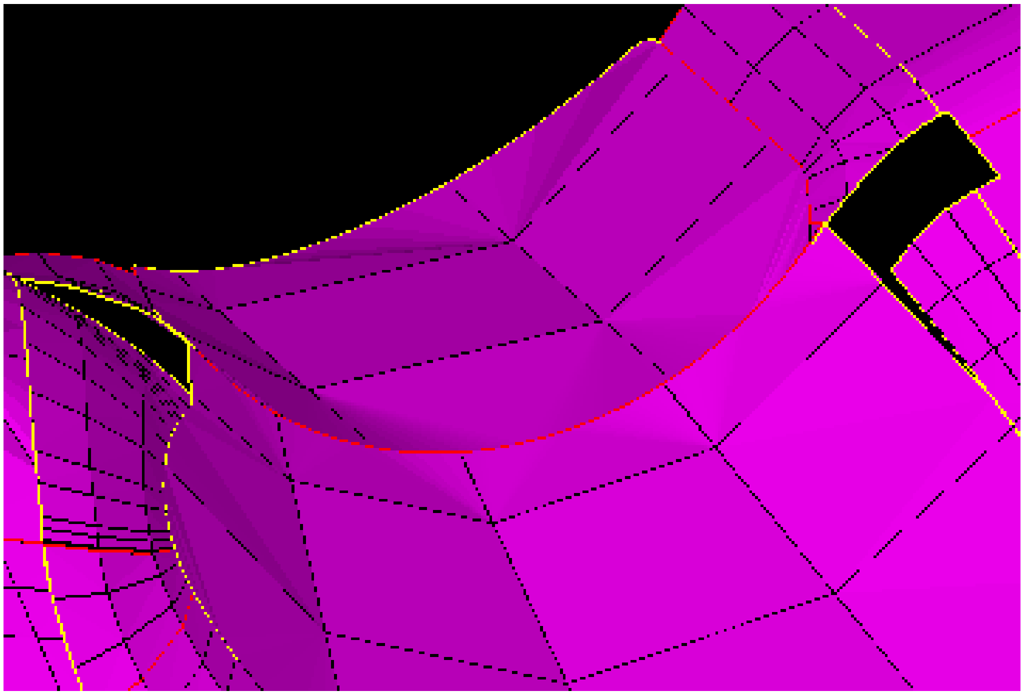
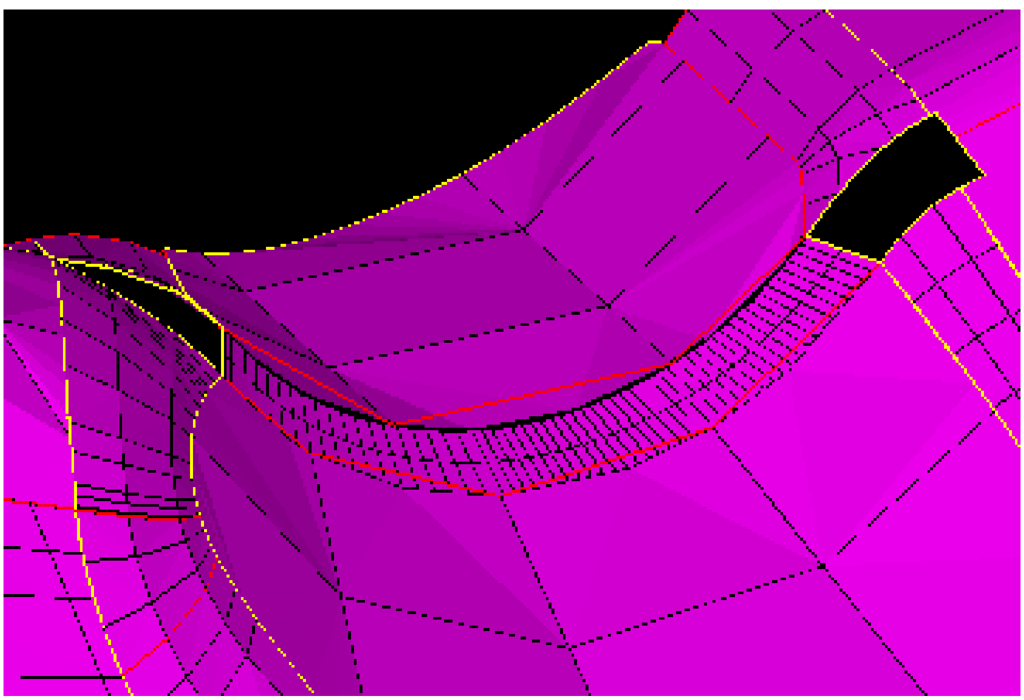


Figure 1.8 Result with Close Gap > Blend



1.3. How does Match work in Stitch/Match Edges?

Match is generally used in those cases where curves lie very close to each other, specifically when the two ends meet together. You should have the two sets of curves within some tolerance for this option to work. Refer to the figures below to get an idea.

Figure 1.9 Geometry with mismatched edges

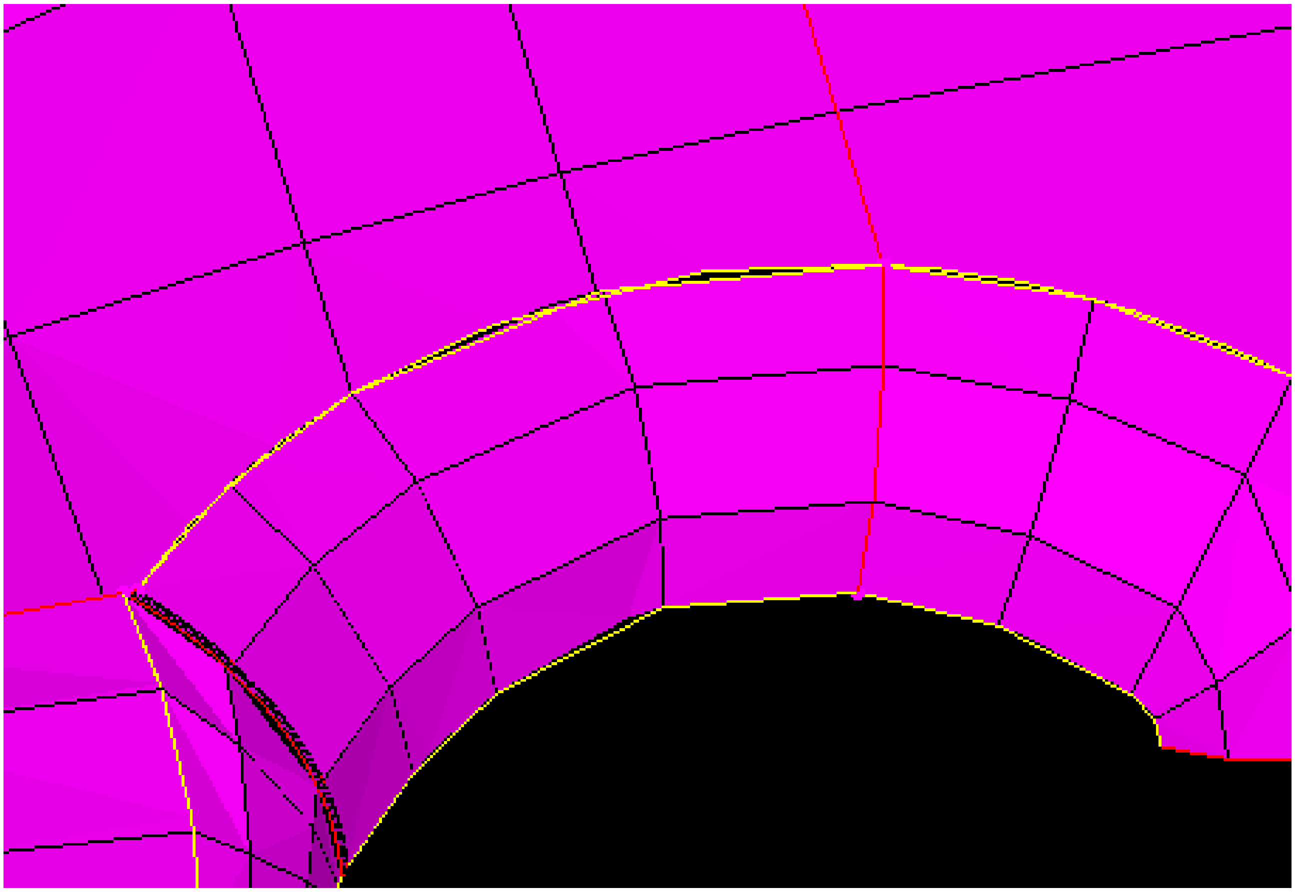
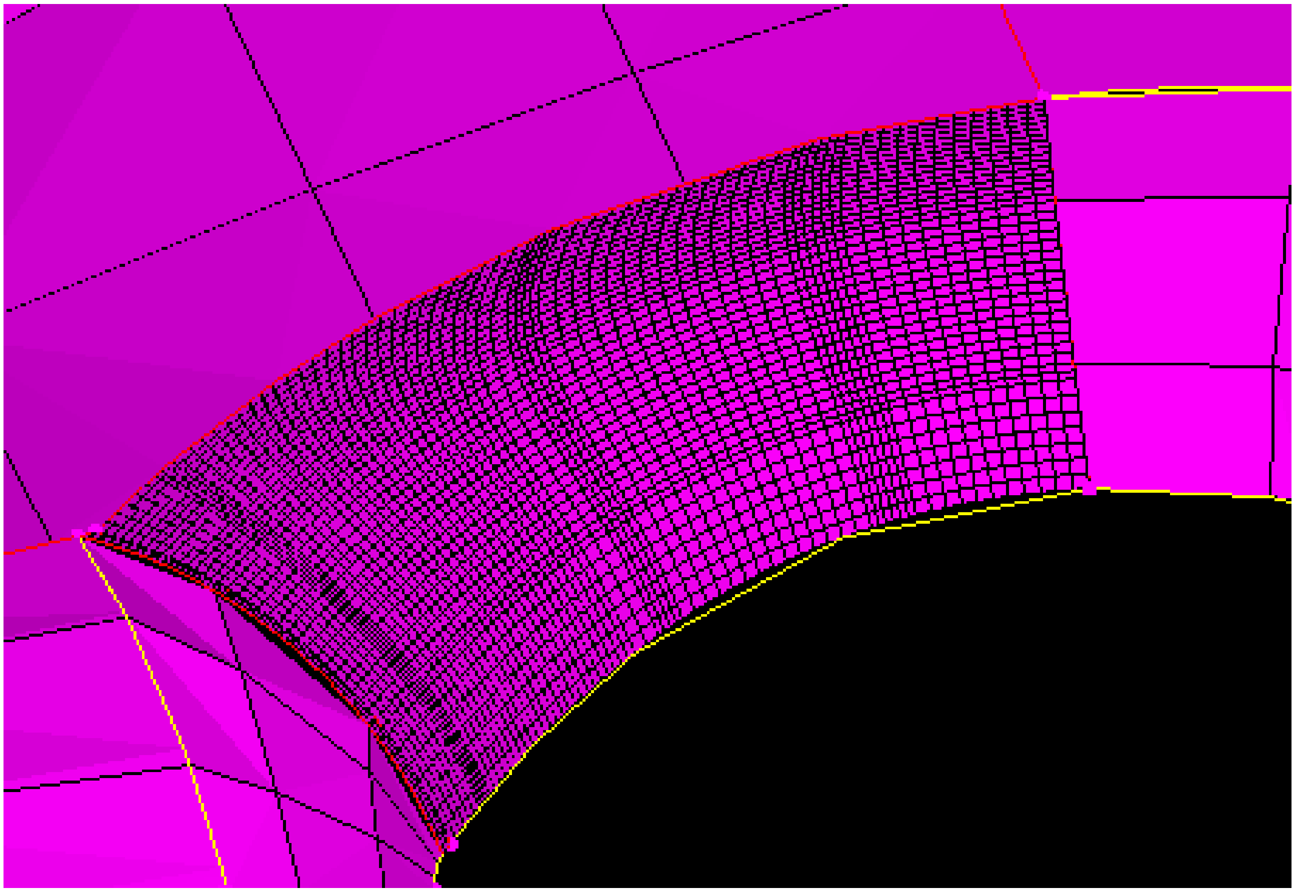


Figure 1.10 Geometry after match edges



Chapter 2: Tetra

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

2.1. Introduction

Tetra uses an Octree-based meshing algorithm to fill the volume with tetrahedral cells and to generate a surface mesh on the object surfaces. The user can define prescribed curves and points to determine the positions of edges and vertices in the mesh. For improved cell quality, **Tetra** incorporates a powerful smoothing algorithm, as well as tools for local adaptive mesh refinement and coarsening.

For more useful information on Tetra, please go to http://www-berkeley.ansys.com/faq/faq_topic_2.html.

2.1.1. Tetra mesh generation

Suitable for complex geometries, **Tetra** offers several advantages, including:

- Rapid model set-up
- Mesh is independent of underlying surface topology
- No surface mesh necessary
- Generation of mesh directly from CAD or STL surfaces
- Definition of cell size on CAD or STL surfaces
- Control over cell size inside a volume
- Nodes and edges of tetrahedra are matched to prescribed points and curves
- *Natural size* automatically determines tetrahedra size for individual geometry features
- Volume and surface mesh smoothing, merging nodes and swapping edges
- Tetrahedral mesh can be merged into another tetra, hexa or hybrid mesh and then can be smoothed
- Coarsening of individual material domains
- Enforcement of mesh periodicity, both *rotational* and *translational*
- Surface mesh editing and diagnostic tools
- Local adaptive mesh refinement and coarsening
- One consistent mesh for multiple materials
- Fast algorithm: *1500 cells/second*
- Automatic detection of holes and easy way to repair the mesh
- For more details, go to Run Tetra - The Octree Approach

.

2.1.2. Input to Tetra

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

- Sets of B-Spline curves and trimmed B-Spline surfaces with prescribed points

- Triangular surface meshes as geometry definition
- Full/partial surface meshes

B-Spline Curves and Surfaces

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

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

Triangular surface meshes as geometry definition

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

Full/partial surface mesh

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

2.2. Tetra Generation Steps

The steps involved in generating a Tetra mesh are:

- Geometry Repair/Clean up
- Geometry details required
- Sizes on Surfaces/Curves
- Meshing inside small angles or in small gaps between objects
- Desired Mesh Region
- Run Tetra - The Octree Approach
- Check the mesh for errors
- Edit mesh to correct any errors
- Smooth the mesh to improve quality

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

2.2.1. Repair Geometry

Refer to the CAD Repair section.

2.2.2. Geometry Details Required

In addition to a closed set of surfaces, **Tetra** requires curves and points where hard features (hard angles, corners) are to be captured in the mesh. The first figure below shows a set of curves and points representing hard features of the geometry, where the second and third figure show the resultant mesh with and without the curves and points preserved.

Figure 2 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 3 shows the resultant surface mesh if the curves and points are deleted from the geometry. The hard features of the geometry are not preserved, but rather are neglected or chamfered. The boundary mesh nodes lie on the surfaces, but they will only lie on the edges of the surfaces if curves and points are present. Removal of curves and points can be used as a geometry defeaturing tool.

Figure 2.1 Curves and Points representing the sharp edges and corners

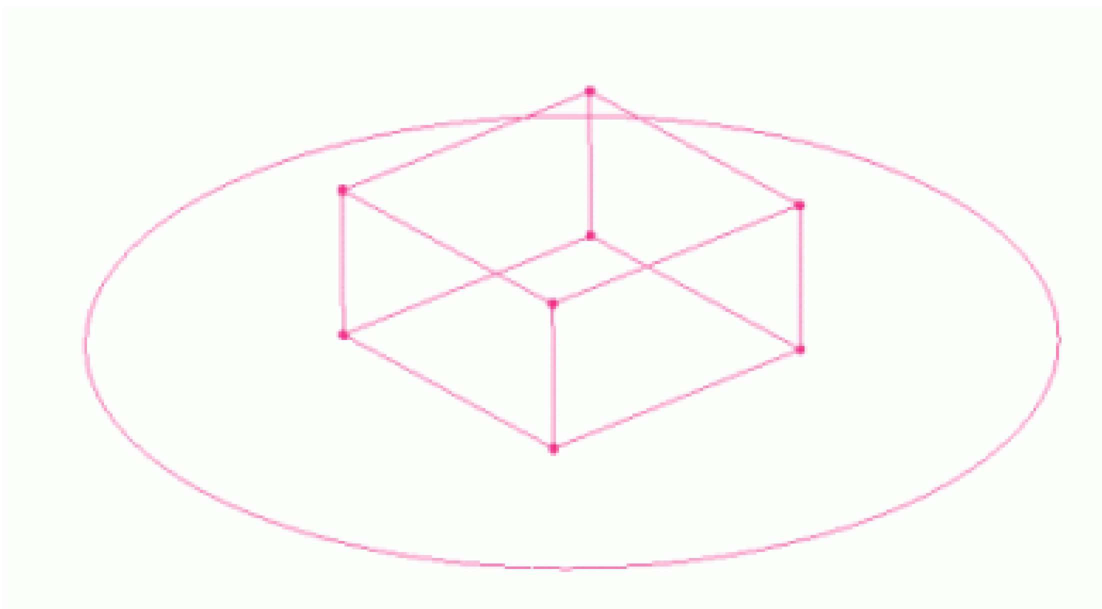


Figure 2.2 Mesh with Curves and points

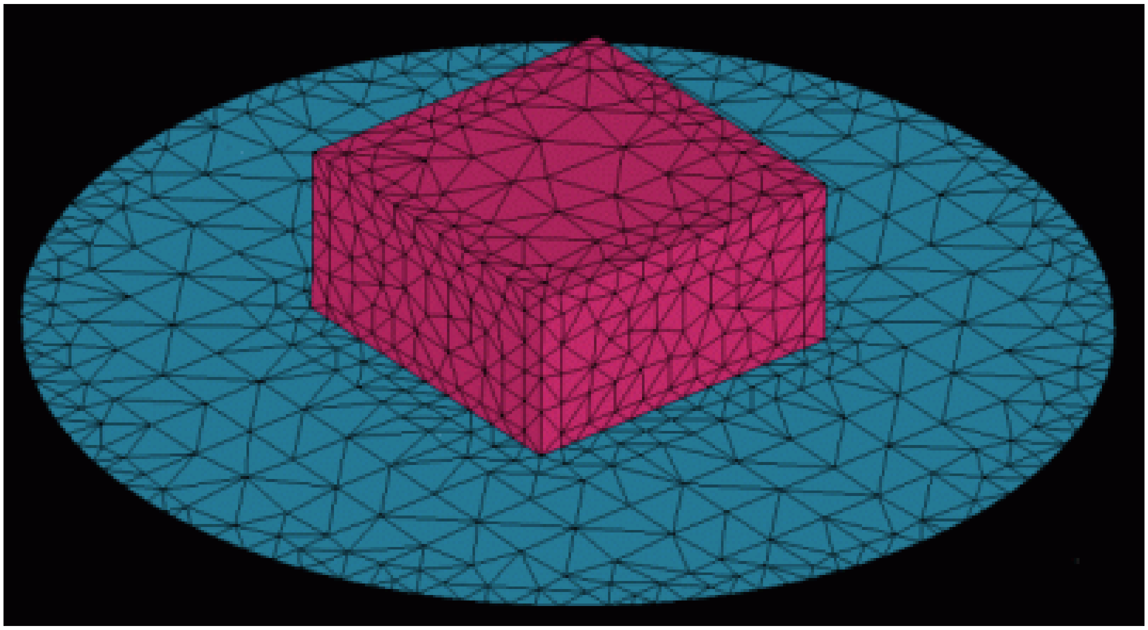
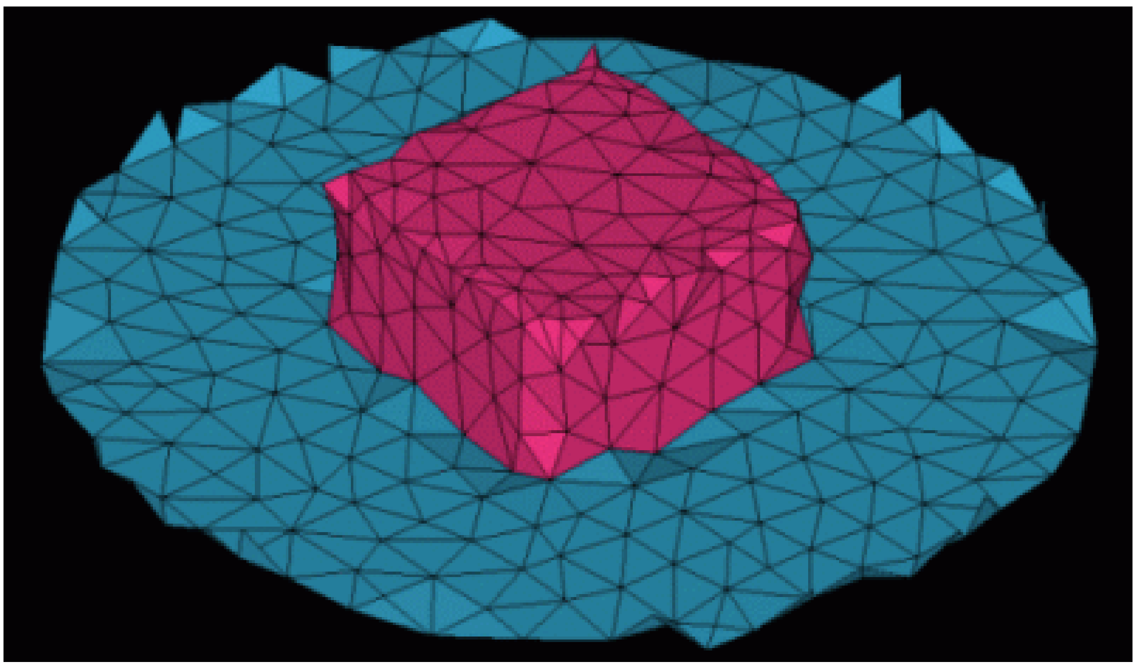


Figure 2.3 Mesh without curves and points



2.2.3. 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 > Surface Tetra sizes** from the **Display Tree**. The same can be done with **Curves**. Tetra icons will appear, representing the cell size of the mesh to be created on these entities. With the mouse, the user may rotate the model and visually confirm that the tetra sizes are appropriate. If a curve or surface does not have an icon plotted on it, the icon may be too large or too small to see. In this case, the user should modify the mesh parameters so that the icons are visible in a normal display.

The user should also make sure that a reference cell size has been defined. To modify the mesh size for all entities, adjust the **Scale Factor,** which is found in **Set Global Mesh Size** window from Mesh Tabbed menubar. Note that if **0** is assigned as the **Scale Factor**, **Tetra** will not run.

2.2.4. Meshing inside small angles or in small gaps between objects

Examine the regions between two surfaces or curves that are very close together or that meet at a small angle. (This would also apply if the region outside the geometry has small angles.) If the local tetra sizes are not small enough so that at least 1 or 2 elements would fit through the thickness, the user should define Thin cuts. This is in the Tet Meshing Parameters section of the Global Mesh Size window. 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 third, different Part.

If the tetra sizes are larger or approximately the same size as the gap between the surfaces or curves, the surface mesh could have a tendency to jump the gap, thus creating non-manifold vertices. These non-manifold vertices would be created during the **Tetra** process. **Tetra** automatically attempts to close all holes in a model. Since the gap may be confused as a hole, the user 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 won't close the gap when the **Tetra** process is performed. A hole is usually considered a space that is greater than 2 or 3 cells in thickness.

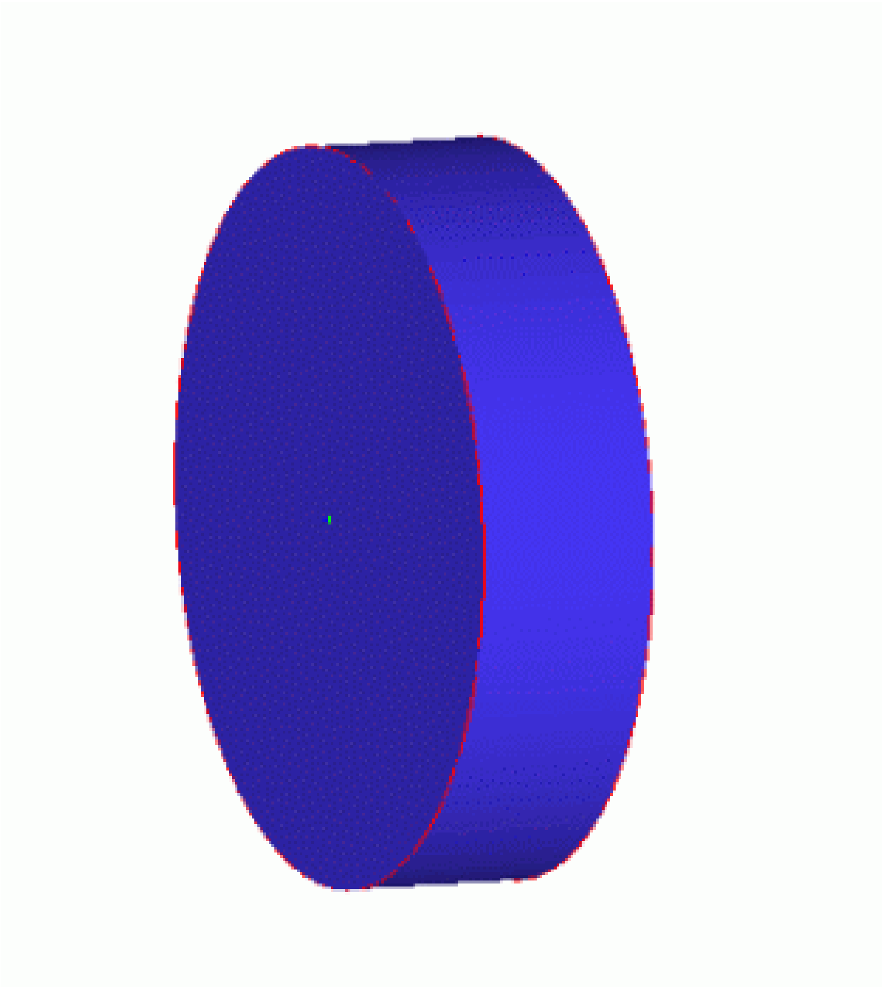
2.2.5. Desired Mesh Region

During the process of finding the bounding surfaces to close the volume mesh, the mesher will determine if there are holes in the model. If there are, the messages 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 cells 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.

2.2.6. Run Tetra - The Octree Approach

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

Figure 2.4 Geometry Input to Tetra



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

Figure 2.5 Full Tetra enclosing the geometry

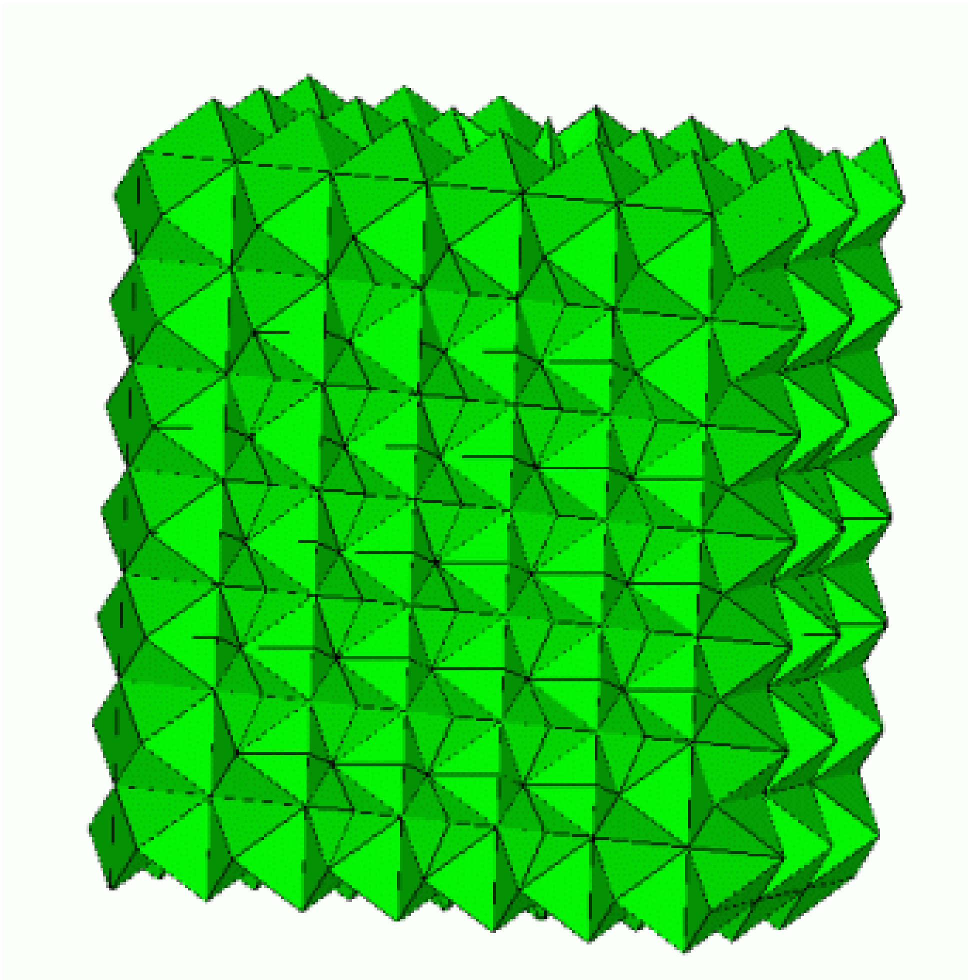


Figure 2.6 Full Tetra enclosing the geometry (In wire frame node)

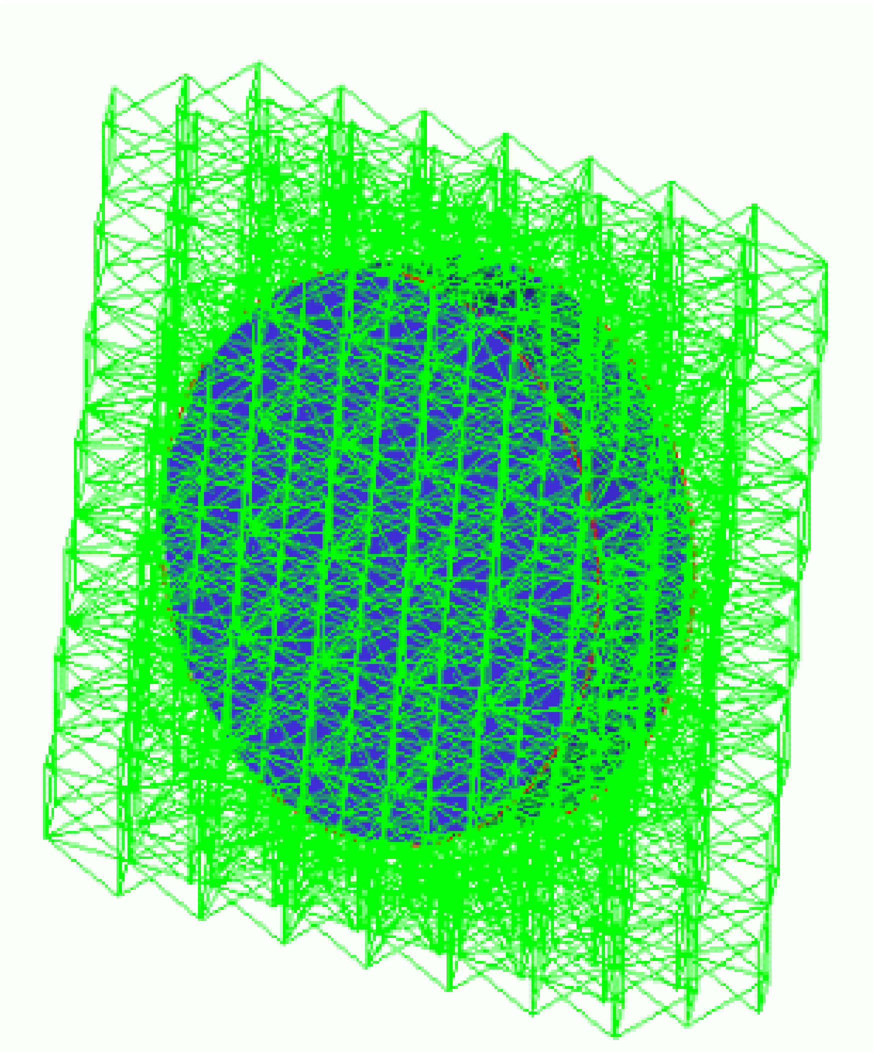
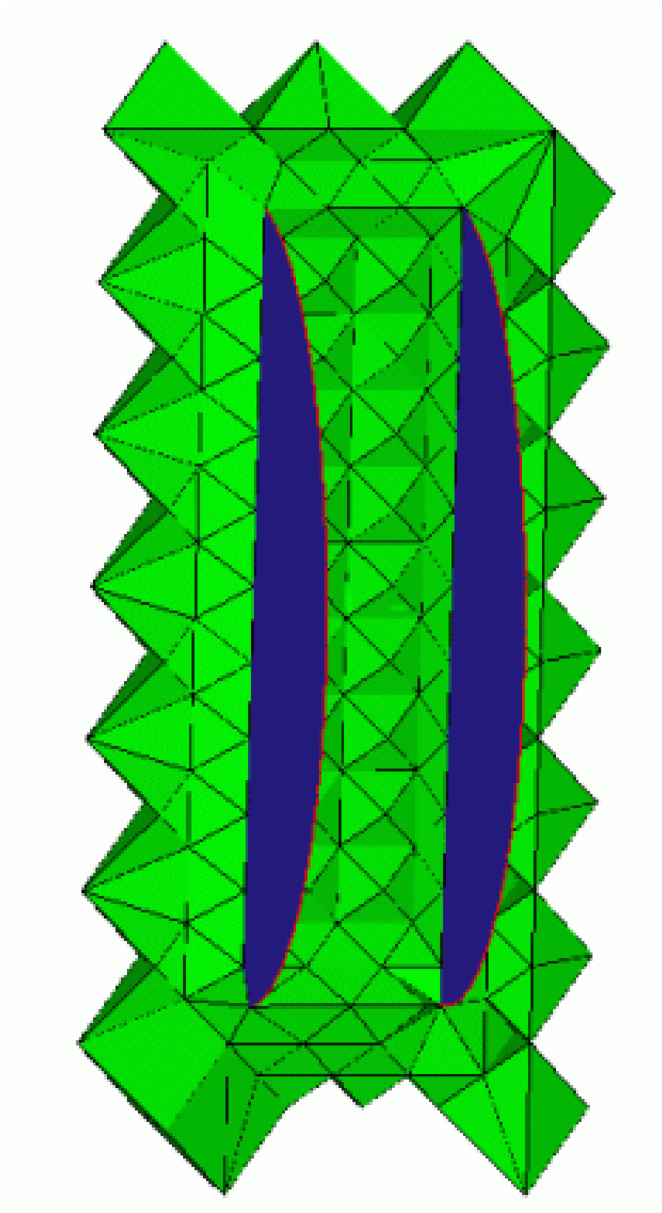


Figure 2.7 Cross-section of the Tetra to show how Tetra are fit in around geometry



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

Figure 2.8 Mesh after it captures surfaces and separation of useful volume

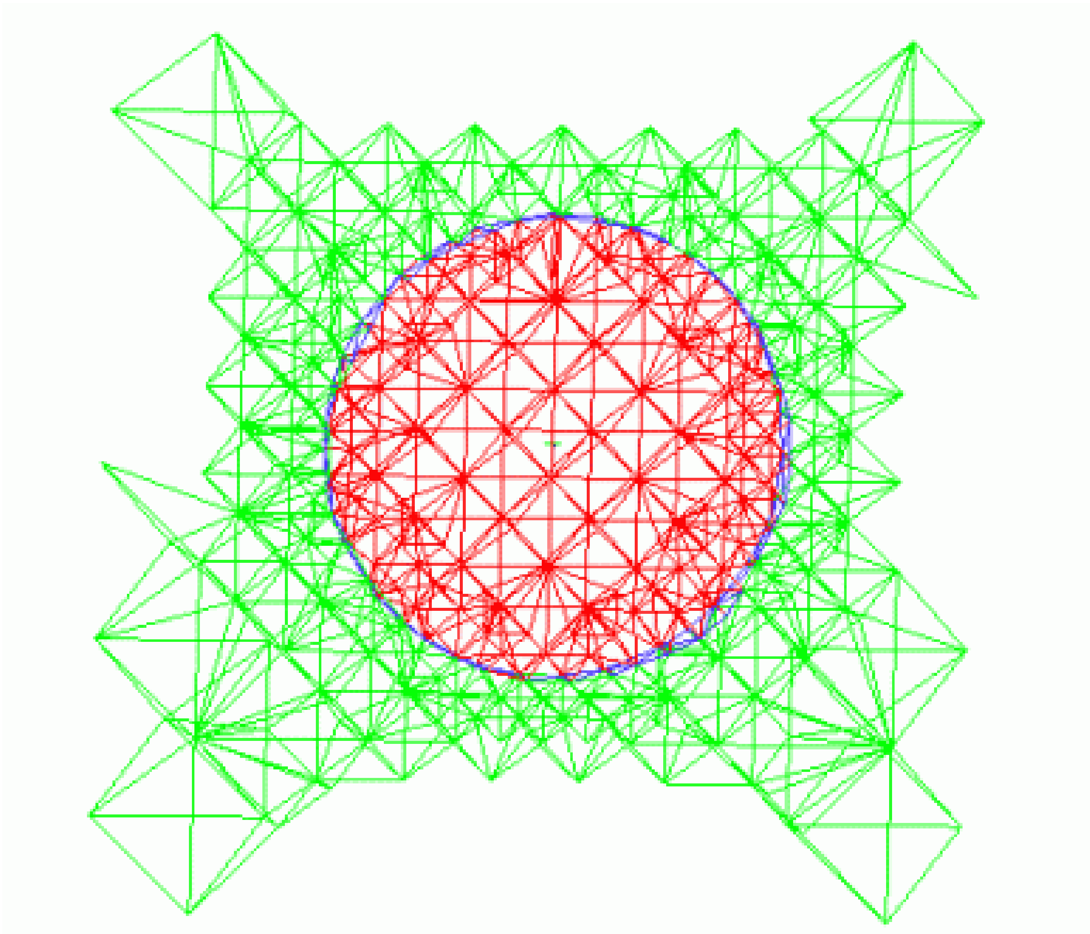
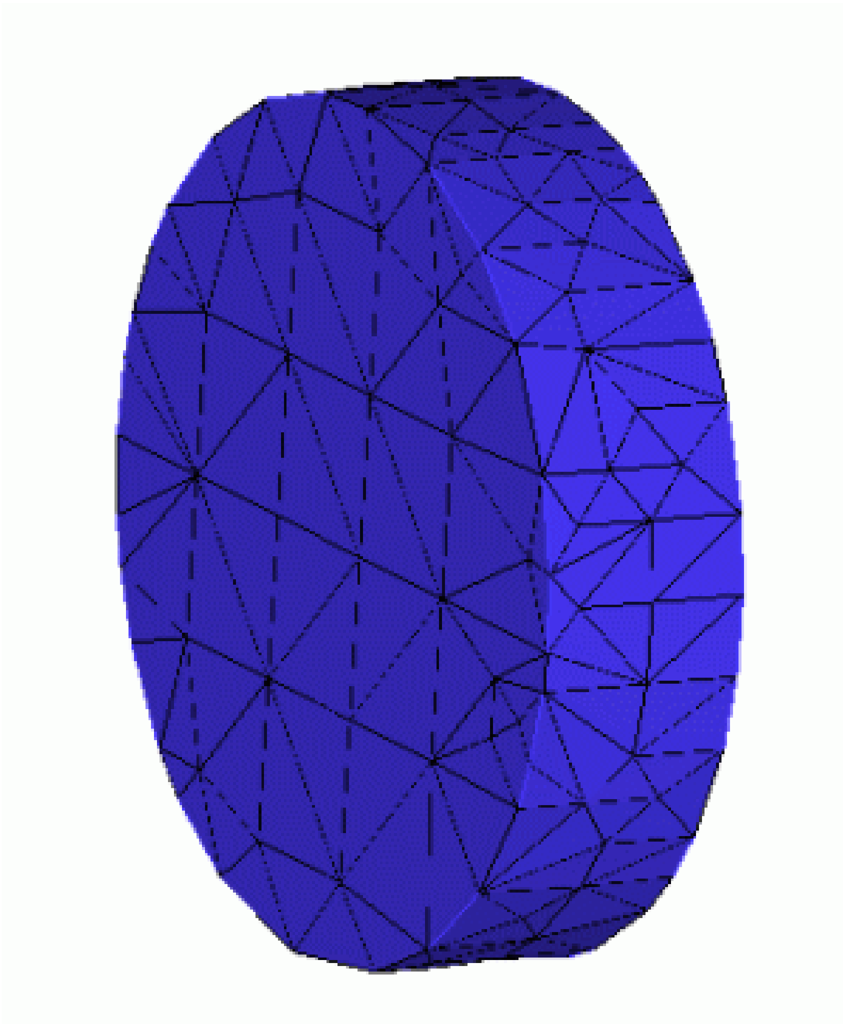
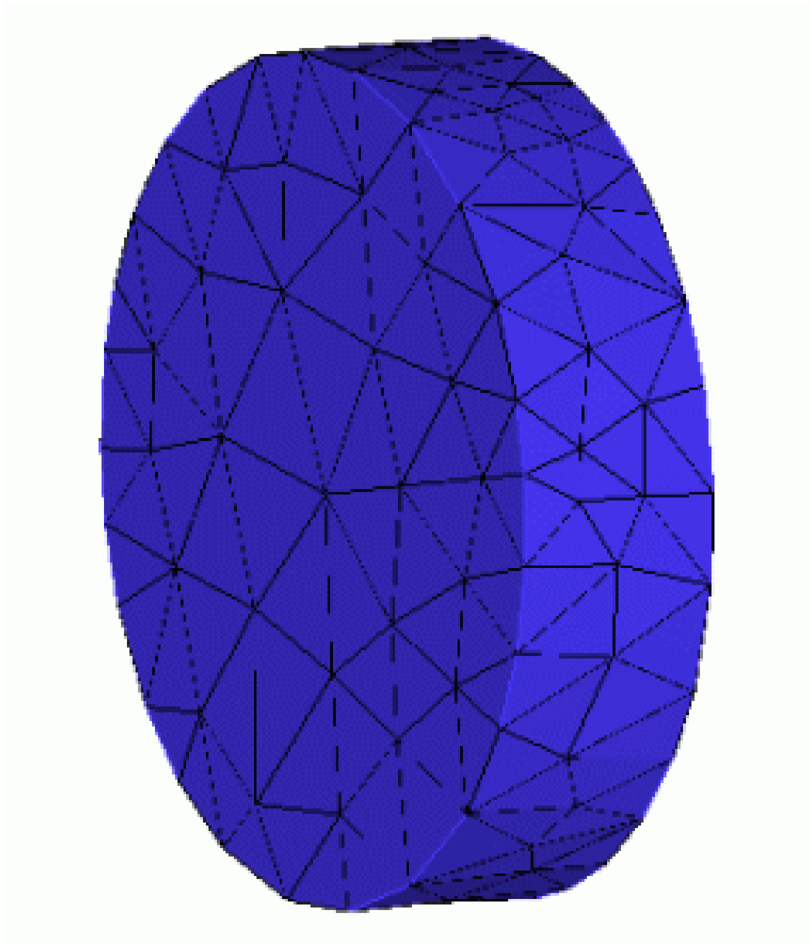


Figure 2.9 Final Mesh before smoothing



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

Figure 2.10 Final Mesh after smoothing



2.3. Important Features in Tetra

2.3.1. Natural Size

If the maximum tetrahedral size defined on a surface is larger than needed to resolve the feature, the user can employ **Natural** size 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 **Natural** size is the minimum element size to be achieved via automatic subdivision. If the maximum size on a geometry entity is smaller than **Natural** size, Tetra will still subdivide to meet that requested size. The effect of the natural size is a geometry- based adaptation of the mesh.

2.3.2. Tetrahedral Mesh Smoother

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

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

2.3.3. Tetrahedral Mesh Coarsener

During the coarsening process the user can exclude surface or material domains by selecting those Parts in the Parts to freeze panel. If the **Maintain surface sizes** option is enabled during coarsening, the resulting mesh satisfies the specified mesh size criteria on the geometric entities.

2.3.4. Triangular Surface Mesh Smoother

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

2.3.5. Triangular Surface Mesh Coarsener

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

2.3.6. Triangular Surface Editing Tools

For the interactive editing of surface meshes, **Tetra** offers a mesh editor in which 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. The user can perform the quality checks, as well as local smoothing.

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

2.3.7. Check Mesh

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

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

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

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

There are several **Errors** as well as **Possible problems** checks. The descriptions of these are as follows:

Duplicate elements

This check locates cells that share all of their nodes with other cells of the same type. These cells should be deleted.

> *Note* — Note that deletion during the automatic fix will remove one of the two duplicate elements, thus eliminating this error without creating a hole in the geometry.

Uncovered faces

This check will locate any face on a volume element that neither touches a surface element nor touches another volume face. This error often indicates a hole in the volumetric domain. It is unlikely that this error would occur in the initial model -- usually, it results during manual editing when the user happens to delete tetra or tri cells.

The automatic **Fix** will cover these uncovered faces with triangles (surface mesh). This may or may not be the proper solution. A better method may be for the user to first **Select** the flawed cells and then decide if the uncovered faces are the result of missing surface mesh or the result of a hole. If it is due to missing surface mesh, the **Fix** option will eliminate the problem (re-run the check and select **Fix**). If the error points out a hole in the model, the user could attempt to correct the grid by manually creating tetras or merging nodes.

Missing internal faces

This check will find pairs of volume elements that belong to different Parts, but do not have a surface element between the shared faces. This error, like **Uncovered faces**, should not occur in the original model and would most likely result from mistakes made during the manual editing process. The tetra cutter will detect this problem as a leakage. The automatic **Fix** will create surface mesh in between these cells.

Periodic problems

The user selects the two Parts that should be one-to-one periodic matches based on the specified periodicity settings. Errors are reported if periodic matches are missing. Slight offsets in node positions are often repaired automatically during the check process. Remaining errors can be repaired manually via Edit Mesh > Repair Mesh > Make/Remove Periodic. You should not get this error ideally unless you have done some editing on the mesh.

Volume orientations

This check will find cells where the order of the nodes does not define a right-handed cell. The automatic **Fix** will re-order the mis-oriented cells' nodes to eliminate this error.

Surface orientations

This check will flag any location where more than one tet element share a single triangle surface element. The tet elements would have 3 common nodes, but the fourth node would be different. These errors, that indicate a major problem in the connectivity in the model, need to be fixed manually. This would normally involve manually deleting and creating elements.

Multiple Edges

This check will find cells with an edge that shares more than two cells. Legitimate multiple edges would be found at a "T"-shaped junction, where more than two geometrical surfaces meet.

Triangle boxes

This check locates groups of 4 triangles that form a tetrahedron, with no actual volume cell inside. This undesirable characteristic is best fixed by choosing **Select** for this region and merging the two nodes that would collapse the unwanted triangle box.

Hanging elements

For a volume mesh, a surface or line element that does not have an attached volume element is flagged as a hanging element.

Penetrating elements

Flags regions where two sets of elements penetrate through each other.

Disconnected bar elements

This flags line elements where one or both nodes are not connected to other elements.

2-Single edges

This locates surface elements with two single edges. These are either corners of baffles or are triangles that are protruding from a surface like a shark's fin and are thus undesirable in the mesh. These elements are a subset of the single edges check and can normally be deleted.

Single-Multiple edges
: This check locates elements that have both single and multiple edges.

Stand-alone surface mesh
: This check locates surface elements that do not share a face with a volume element. This could be an area with an extra surface element to be deleted or a missing volume element to be created.

Single edges
: This check will locate surface cells that have an edge that isn't shared with any other surface cell. This would represent a hanging edge and the cell would be considered an internal baffle. These may or may not be legitimate. Legitimate single edges would occur where the geometry has a zero thickness baffle with a free or hanging edge or in a 2D model at the perimeter of the domain.

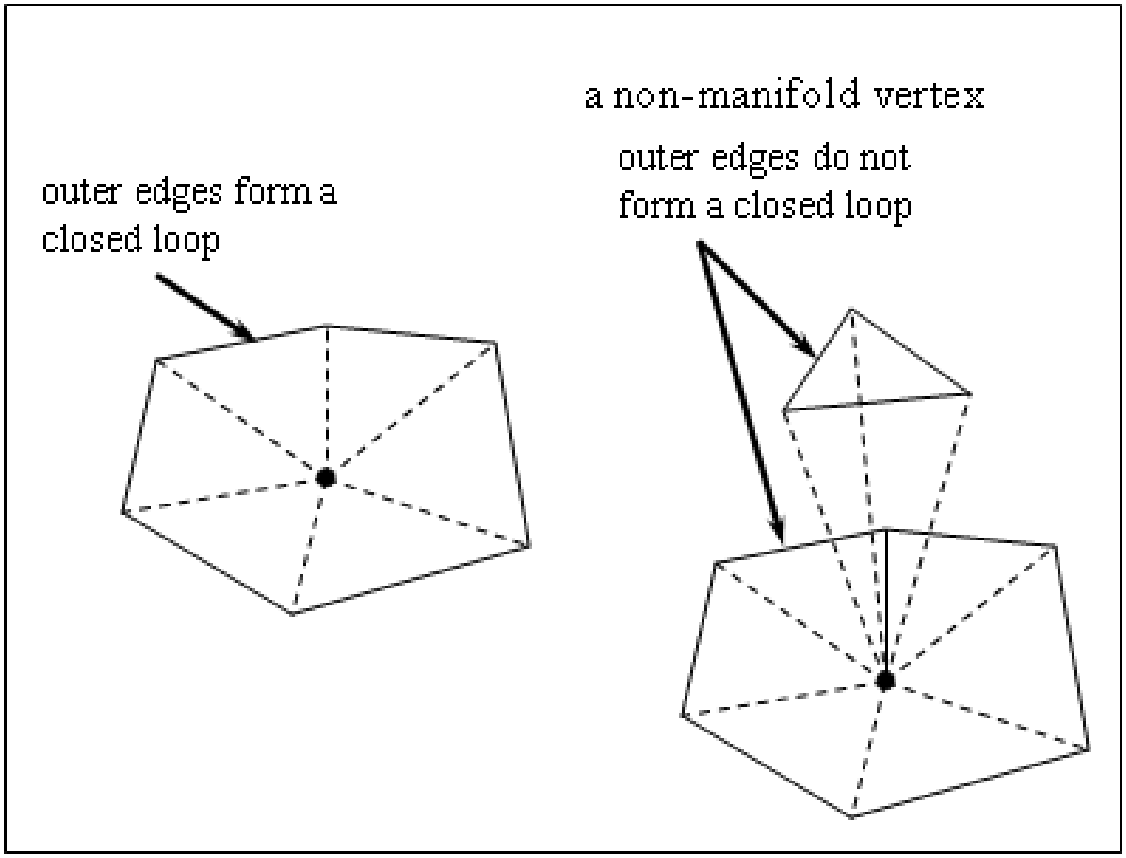
  If the single edges form a closed loop -- a hole in the surface mesh -- the user can select **Fix** when prompted from the appearing menu. A new set of triangles will then be created to eliminate the hole.

Delaunay-violation
: This check finds cells which violate the Delaunay rule, which states that a circumscribed circle around a surface triangle should not enclose any other nodes. These can often be removed by swapping edges of these triangles.

Overlapping elements
: This flags surface elements that occupy part of the same surface area, but don't share the same nodes (so are not duplicates).

Non-manifold vertices
: This check will find vertices who adjacent cells' outer edges don't form a closed loop. Finding this problem usually indicates the existence of cells that jump from one surface to another, forming a "tent"-like structure, as shown in the figure below.

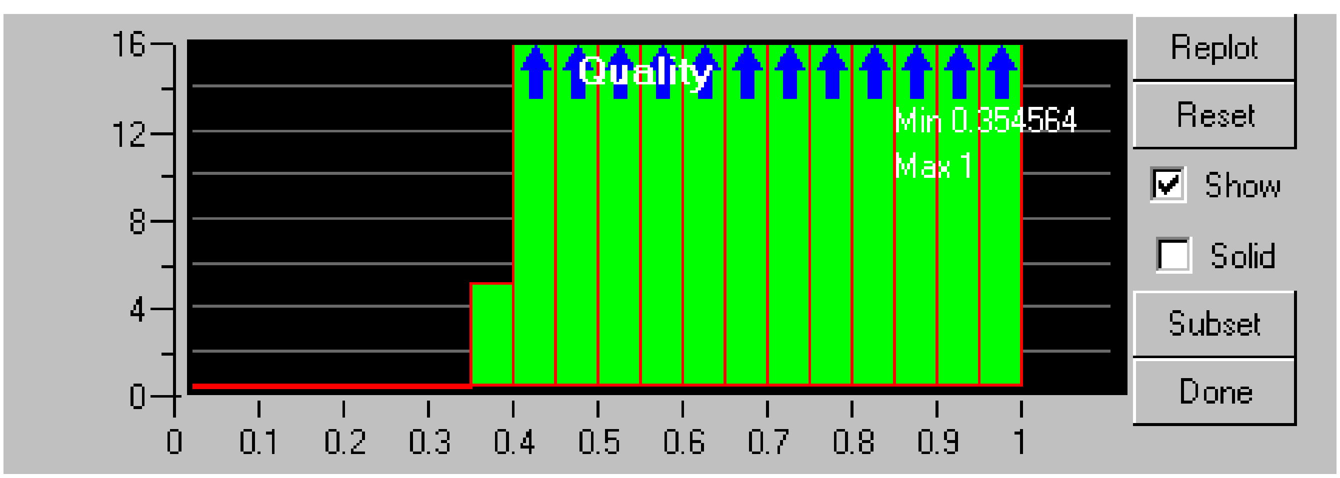
Figure 2.11 Non-Manifold Vertices

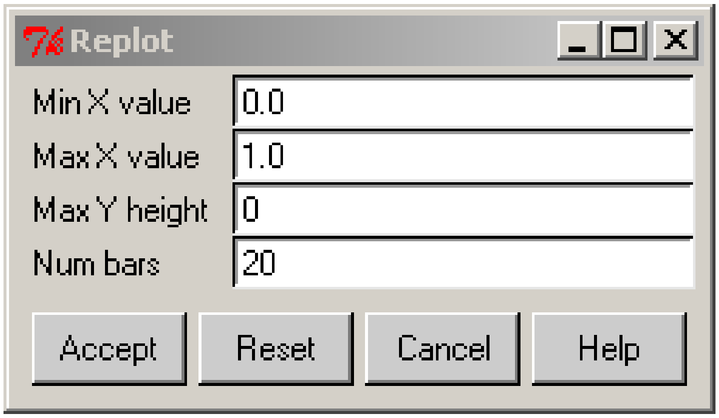


Un-connected vertices
: This check finds vertices that are not connected to any cells. These can generally be deleted.

Smoothing
: After eliminating errors/possible problems from a tetra grid, the user needs to smooth the grid to improve the quality.

Figure 2.12 Quality Histogram





Histogram: The tetrahedral **Quality** will be displayed within this histogram, where 0 represents the worst aspect ratio and 1 represents the best aspect ratio. The user may modify the display of the histogram by adjusting the values of **Min**, **Max**, **Height** and **Bars**.

Replot: If any modification is to be done for displaying the histogram then select Replot which pops ups the Replot window shown above. User can change the following parameters in this window, pressing **Accept** will replot 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 cells.

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 cells can achieve.

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

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

- **Reset**: Selecting this option will return all of the values back to the original parameters that were present when the **Smooth cells** window was first invoked.
- **Show**: The user may press the left mouse button on any of the bars in the histogram and the color will change from green to pink. Toggling **ON Show** will display the cells that fall within the selected range on the model in the main viewing window.
- **Solid**: This toggle option will display the cells as solid tetras, rather than as the default grid representation. The user will have to select **Show**, as well, to activate this option.
- **Subset**: If the user has highlighted bars from the histogram and toggled **ON Show**, the cells displayed in white color are also placed into a **Subset**. The visibility of this subset is controlled by **Subset** from the **Display Tree**. **Add select**: This option allows the user to add cells to an already established subset.

  Smoothing Elements window:

  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 assigned in **Up to quality**.

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

- **Freeze**: If the **Freeze** option is selected for a cell type, the nodes of this cell type will be fixed during the smoothing operation; thus, the cell type will not be displayed in the histogram.
- **Float**: If the **Float** option is selected for a cell type, the nodes of the cell type are capable of moving freely, allowing nodes that are common with another type of cell 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.

2.3.8. Quality metric

Changing this option allows the user to modify what the histogram displays.

Quality: This histogram displays the overall quality of the mesh. The x-axis measures the quality, with 0 representing poor quality and 1 representing high quality. The y-axis measures the number of cells that belong within each quality sub-range. **Aspect ratio:** For **HEXA_8** (hexahedral) and **QUAD_4** (quadrilateral) cells, the **Aspect ratio** is defined as the ratio of the distances between diagonally opposite vertices (shorter diagonal/longer diagonal). For **TETRA_4** (tetrahedral) cells, **MED** calculates the ratio between the radii of an inscribed sphere to a circumscribed sphere for each cell. For **TRI_3** (triangular) cells, this operation is done using circles. An **Aspect ratio** of 1 is a perfect cell and an **Aspect ratio** of 0 indicates that the cell has zero volume. **Determinant:** This histogram is based on the determinant of the Jacobian matrix. The Jacobian value is based on the difference between the internal angles of the opposing edges within the cell.**Min angle:** The **Min angle** option yields a histogram based upon the minimum internal angle of the cell edges.**Max warp:** This histogram is based on the warpage of the quad faces of the prism. This is based on the worst angle between two triangles that make up the quad face. **Skew:** This histogram is based upon calculations of the maximum skewness of a hexahedral or quadrilateral cell. The skewness is defined differently for volume and surface cells. For a volume cell, it is obtained by taking all pairs of adjacent faces and computing the normals. The maximum value thus obtained, is normalized so that 0 corresponds to perpendicular faces and 1 corresponds to parallel faces. **Custom quality:** One can define one's own quality definition by going to **Diagnostics > Quality metrics.** Select the **Diagnostic:** as **Custom quality** and go for **Define custom quality**. One can change the values there to suit his/her needs.

2.3.9. Advanced options

Prism warpage Ratio
: Prisms are smoothed based on a balance between prism warpage and prism aspect ratio. Numbers from 0.01 to 0.50 favor improving the prism aspect ratio, and 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
: This is the default where normally, when a grid is smoothed, the nodes are restricted to the geometry -- surface, curves and points -- and can only be moved along the geometrical entities to obtain a better mesh.

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

Violate geometry Relative Tolerance
: This option works in the similar fashion as above except that the distance is relative here.

Allow refinement
: If the quality of the mesh cannot be improved through normal algebraic smoothing, Allow refinement will allow the smoother to automatically subdivide tetras to obtain further improvement. After smoothing with

Allow refinement selected, it may be necessary to Smooth further with the option turned off. The goal of this option is to reduce the number of cells that are attached to one vertex by refinement in problem regions.

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

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

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

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

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

Ignore pre points
: Selecting this option will allow the smoother to attempt to improve the mesh quality without being bound by the initial points of the geometry. This option is similar to the **Violate geometry** option, but works only for points located on the geometry. This option is available only when the user has hexahedral cells 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 cells and edit this surface mesh to improve the quality.

Chapter 3: Hexa

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

3.1. Introduction

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

Recognized as the fastest hexahedral mesh generation tool in the market, **ICEM CFD 4.CFX** allows users to generate high-quality meshes for aerospace, automotive, computer and chemical industry applications in a fraction of the time required for traditional tools.

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

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

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

For more useful information on Hexa, please go to .

3.2. Features of Hexa

Some of the more advanced features of **Hexa** include:**O-grids:** For very complex geometry, **Hexa** automatically generates body-fitted internal and external O-grids to parametrically fit the block topology to the geometry to ensure good quality meshes.**Edge-Meshing Parameters: Hexa's** edge-meshing parameters offer unlimited flexibility in applying user specified bunching requirements.**Time Saving Methods: Hexa** provides time saving surface smoothing and volume relaxation algorithms on the generated mesh.**Mesh Quality Checking:** With a set of tools for mesh quality checking, cells with undesirable skewness or angles may be displayed to highlight the block topology region where the individual blocks need to be adjusted.**Mesh Refinement/Coarsening:** Refinement or coarsening of the mesh may be specified for any block region to allow a finer or coarser mesh definition in areas of high or low gradients, respectively.**Replay Option: Replay** file functionality enables parametric block topology generation linked to parametric changes in geometry. **Symmetry:** As necessary in analyzing rotating machinery applications, for example, **Hexa** allows the user to take advantage of symmetry in meshing a section of the rotating machinery thereby minimizing the model size. **Link Shape:** This allows the user 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 3-D surface meshes from the 3-D volume mesh and 2-D to 3-D block topology transformation.

3.3. Mesh Generation with Hexa

To generate a mesh within **Hexa** the user will :

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

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

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

> *Note* — 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.

3.4. The Hexa Database

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

The Geometric Data Entities:

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

The Block Topologic Data Entities:

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

3.5. Intelligent Geometry in Hexa

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

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

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

3.6. Unstructured and Multi-block Structured Meshes

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

3.6.1. Unstructured Mesh Output:

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

3.6.2. Multi-Block Structured Mesh Output:

Used for solvers that accept multi-block structured meshes, the multi-block structured mesh output option will produce a mesh output file for every block in the topology model.

For example, if the block model has 55 blocks, there will be 55 output files created in the output directory.

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

3.7. Blocking Strategy

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

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

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

Colors of Edges:

White Edges and Vertices:

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

Blue Edges and Vertices:

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

Green Edges and Vertices:

These edges and the associated vertices are being projected to curves. The vertices can only be moved on the curves to which it is being projected.

Red Vertices:

These vertices are projected to prescribed points.

3.7.1. Split

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

3.7.2. Merge

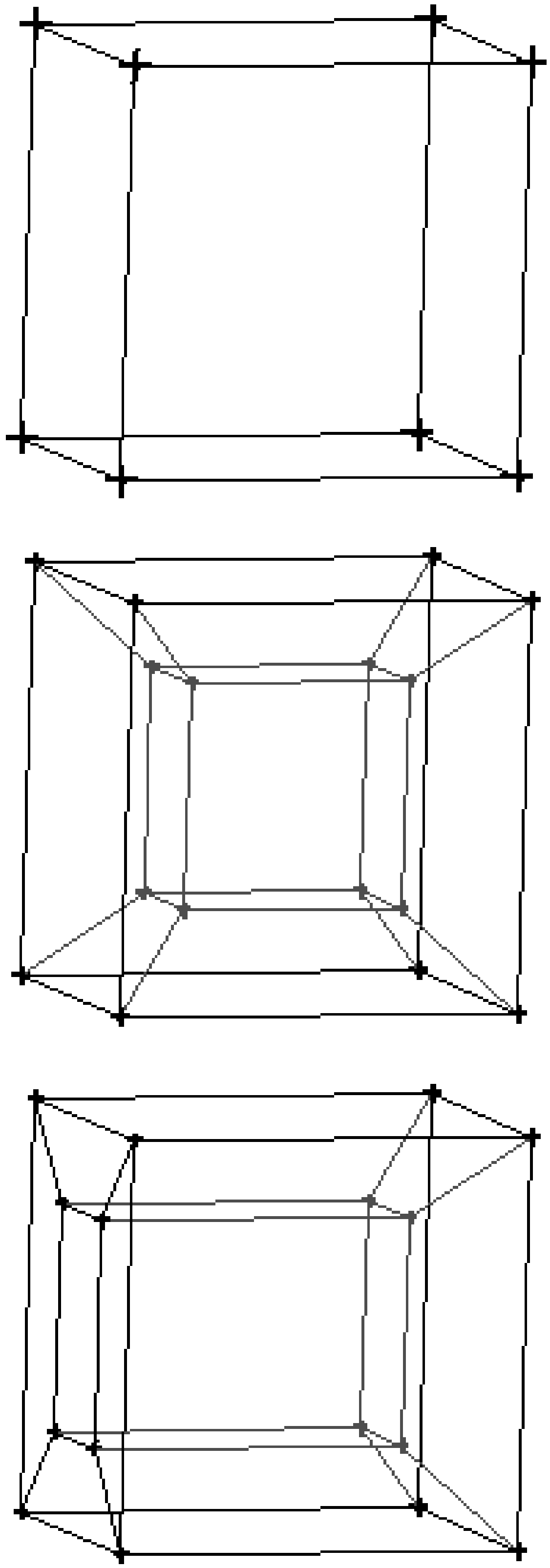
The **Merge** function works similarly to split blocks; one can either merge the whole block or merge only a face or an edge of the block.

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

3.8. Using the Automatic O-grid

The **O-grid** creation capability is simply the modification of a single block or blocks to a 5 sub-block topology as shown in Figure 3.1: “Initial block, block with O-Grid, O-Grid with include a face”. There are several variations of the basic **O-grid** generation technique and the **O-grid** shown below is created entirely inside the selected block.

Figure 3.1 Initial block, block with O-Grid, O-Grid with include a face



Using the **Add face** option, an **O-grid** may also be created such that the **O-grid** passes through the selected block faces. In Figure , the **Add Face** option was used on the right most block to add the bottom face on the block prior to generating the **O-grid**.

Another important feature of the automatic **O-grid** is the ability to re-scale the **O-grid** after generation. When the **O-grid** is generated, the size of the **O-grid** is scaled based upon a factor in the **Blocking > O-grid parameter** window. The **Re scale O grid** option allows the user to re-scale the previously generated **O-grid.**

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

3.9. Most Important Features of Hexa

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

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

3.10. Automatic O-grid generation

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

3.10.1. Important Features of an O-grid

Generation of Orthogonal Mesh Lines at an Object Boundary
: The generation of the O-grid is fully automatic and the user simply selects the blocks needed for O-grid generation. The O-grid is then generated either inside or outside the selected blocks. The O-grid may be fully contained within its selected region, or it may pass through any of the selected block faces.

Rescaling an O-grid After Generation
: When the O-grid is generated, the size of the O-grid is scaled based upon the **Factor** in the **Blocking > O-grid** parameter window. The user may modify the length of the O-grid using the **Blocking > Re- scale O-grid** option. If a value that is less than 1 is assigned, the resulting O-grid will be smaller than the original. If, however, a value is larger than 1, the resulting O-grid will be larger.

3.11. Edge Meshing Parameters

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

The user has access to the following pre-defined bunching laws or **Meshing laws**:

Default (Bi-Geometric Law)**UniformHyperbolicPoissonCurvatureGeometric 1Geometric 2Exponential 1Exponential 2Bi-ExponentialLinearSpline**

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

This option yields these possible functions: **ConstantRampS curveParabola MiddleParabola EndsExponentialGaussianLinearSpline**

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

3.12. Smoothing Techniques

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

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

The criteria for smoothing are: **Determinant:** This criteria attempts to improve the cell's determinant by movement of nodes, which are subject to geometry and association constraints.**Laplace:** The **Laplace** option attempts to minimize abrupt changes in the mesh lines by moving the nodes.**Warp:** The **Warp** method is based upon correcting the worst angle between two cells in the mesh. **Quality:** Like the determinant criteria, the **Quality** criteria attempts to improve the cell's interior angle by repositioning the nodes, which are subject to geometry and association constraints.**Orthogonality:** The **Orthogonality** option attempts to provide orthogonal mesh lines at all boundaries of the model.**Skewness:** The **Skewness** is defined differently for volume and surface cells. For a volume cell, this value is obtained by taking all pairs of adjacent faces and computing the normals. The maximum value thus obtained is normalized so that **0** corresponds to perpendicular faces, and **1** corresponds to parallel faces. For surface cells, the skew is obtained by first taking the ratio of the two diagonals of the face. The skew is defined as one minus the ratio of the shorter diagonal over the longer diagonal. Thus, **0** is perfectly rectangular, and **1** represents maximum skewness.

3.13. Refinement and Coarsening

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

3.13.1. Refinement

The refinement capability is used for solvers that accept non-conformal node matching at the block boundaries. The refinement capability is used to minimize the model size, while achieving proper mesh definition in critical areas of high gradients.

3.13.2. 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.

3.14. Replay Functionality

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

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

3.14.1. Generating a Replay File

The first step in generating a **Replay** file is to activate the recording of the commands needed to generate the initial block topology model. As mentioned above, this function can be invoked through **File > Replay**. All of the steps in the mesh development process are recorded, including blocking, mesh size, edge meshing, boundary condition definition, and final mesh generation. The next step in the process is to make the parametric change in the geometry and then replay the recorded **Replay** file on the changed geometry. All steps in the mesh generation process are automated from this point.

3.14.2. Advantage of the Replay Function

With the **Replay** option, the user is capable of analyzing more geometry variations, thus obtaining more information on the critical design parameters. This can yield optimal design recommendations within the project time limits.

3.15. Periodicity

Periodic definition may be applied to the model in **Hexa**. The **Periodic nodes** function, which is found under **Blocking > Periodic nodes**, plays a key role in properly analyzing rotating machinery applications, for example.

Typically, the user will model only a section of the rotating machinery, as well as implement symmetry, in order to minimize the model size. By specifying a periodic relationship between the inflow and outflow boundaries, the particular specification may be applied to the model -- flow characteristics entering a boundary must be identical to the flow characteristics leaving a boundary.

3.15.1. Applying the Periodic Relationship

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

3.16. Mesh Quality

The mesh quality functions are accessible through **Meshing > Quality check**. Any of the four quality check options will display a histogram plot for the user.

3.16.1. Determining the Location of Cells

By clicking on any of the histogram bars with the left button, the user may determine where in the model these cells are located. The selected histogram bars will change in color to pink. After selecting the bar(s), the **Show** button is pressed to highlight the cells in this range. If the **Solid** button is turned on, the cells marked in the histogram bars will be displayed with solid shading.

3.16.2. Determinant

The **Determinant** check computes the deformation of the cells in the mesh by first calculating of the Jacobian of each hexahedron and then normalizing the determinant of the matrix. A value of 1 represents a perfect hexahedral cube, while a value of 0 is a totally inverted cube with a negative volume. The mesh quality, measured on the x-axis, of all cells will be in the range from 0 to1. If the determinant value of a cell is 0, the cube has one or more degenerated edges. In general, determinant values above 0.3 are acceptable for most solvers.

The y-axis measures the number of cells that are represented in the histogram. This scale ranges from 0 to a value that is indicated by the **Height**. The subdivision among the quality range is determined by the number of assigned **Bars**.

3.16.3. Angle

The **Angle** option checks the maximum internal angle deviation from 90 degrees for each cell. Various solvers have different tolerance limits for the internal angle check. If the cells 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.

3.16.4. Volume

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

3.16.5. Warpage

The **Warpage** check will yield a histogram that indicates the level of cell distortion. Nodes that are in-plane with one another will produce a cell with small warpage. Nodes that make cells twisted or distorted will increase a cells distortion, giving a high degree of warpage. The y-axis is the scale for the number of cells represented in the histogram - a value determined by the assigned **Height**. The x- axis, which ranges from a **Min** of 0 to a **Max** of 90, is the degree of warpage that a cell experiences.

Chapter 4: Properties

Properties menu allows the user to create different materials by specifying the material type that is whether isotropic, Young's Modulus, Poission's ratio. Once the material is created the user can apply those properties to the respected elements.

4.1. Create Material Property

Here the user can define a material by Specifying a name of the material, define whether isotropic, Young's Modulus, Shear modulus, Poission's ratio, Mass Density, Thermal expansion coefficient.

4.2. Save Material

This Option allows the user to save the material which is created, So that user can retrieve the material when ever necessary. The material file will be saved with .mat extension.

4.3. Open Material

This option allows the user to open a material file so that the user can use the same for future or modify the file and save for further usage.

4.4. Define Table

Here user can create different tables by specifying the values for x and y ,and user can even visualize the graph.

4.5. Define Point Element

Although the user has created the mesh, user has to apply the material created to the respected elements. Allows user to define point element, user have to specify Mass type and scalar Mass.

4.6. Define Line Element

This option allows the user to select a part and Specify the material which is already, and property identification number, cross section Area and moment of inertia.

4.7. Define Shell Element

This option allows the user to select a part and specify the property identification number material, thickness etc.

4.8. Define Volume Element

This option allows the user to select a part and specify the material, property identification number etc.

Chapter 5: Constraints

Here user can define the constraints on deferent entities like point, curve, surface, subset & other options like **Contact definition, Velocity** and **Rigid Wall.**

5.1. Displacement on Point

This option allows the user to apply the constraint on point in directional displacement Translation can be constrained in all the three directions by clicking in the checkbox displacement values can be specified, and similarly Rotational displacement.

5.2. Displacement on Curve

This option allows the user to apply the constraint on Curve in directional displacement Translation can be constrained in all the three directions, and similarly rotational displacement

5.3. Displacement on Area

This option allows the user to apply the constraint on Area in directional displacement as well as Rotational displacement

5.4. Displacement on Subset

This option allows the user to apply the constraint on Subset in directional displacement as well as Rotational displacement

5.5. Define Contact

Here there are two options are available for the user

5.5.1. Automatic Detection

User has to specify Contact proximity Factor, It is a range within which all the elements get selected.

5.5.2. Manual Definition

Here user has to manually pick the contact and target surfaces. for the contact creation.

5.6. Define Single Surface Contact

This is mainly used for LS-Dyna Solver, wherein user can pick the contact surface.

5.7. Define Initial velocity

This allows the user to define the rigid wall by specifying the directional and rotational velocity.

5.8. Define Planar Rigid wall

User can define Planar Rigid Wall by Specifying the Head and Tail coordinates.

Chapter 6: Loads

In this tab there are several optional available for applying the Load, Pressure temperature and gravity

Theory

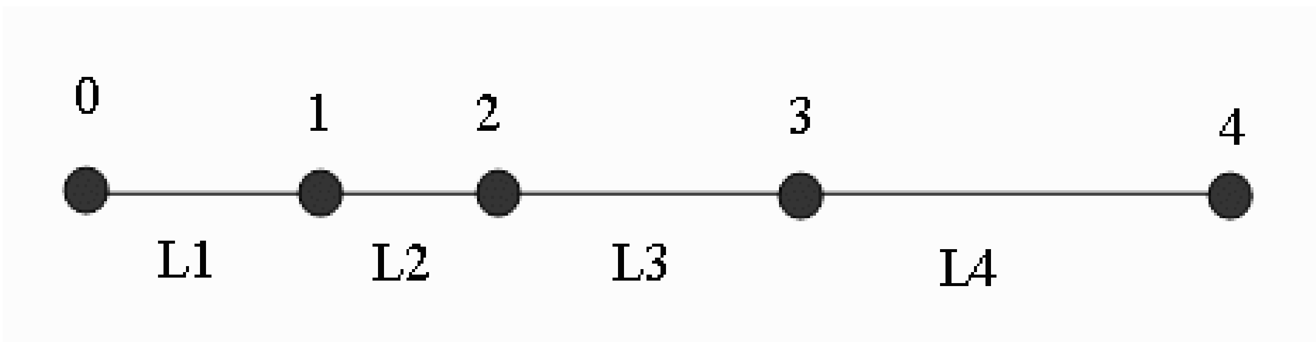
Force Distribution follows the following formulation.

Curve

Total Force '**FT**' is applied on the curve as shown in Figure 6.1: " Elements on Curve".

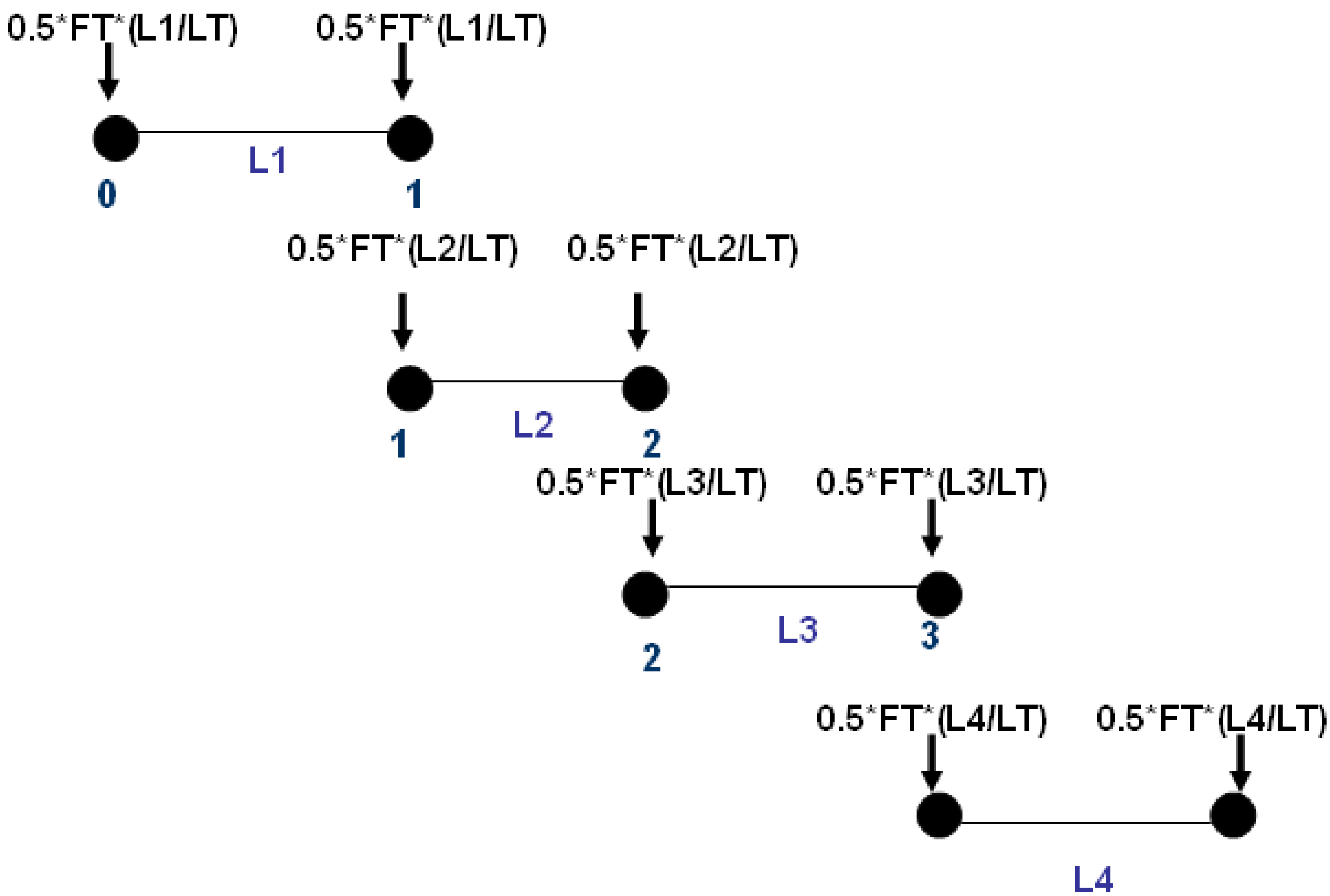
Where L1 is the length of Element 1, '0' and '1' are the node number connecting Element 1.

Figure 6.1 Elements on Curve



Then the force distribution on 'Linear' elements as per the FEA concept is shown in Figure 6.2: "Force Distribution as per the FEA concepts".

Figure 6.2 Force Distribution as per the FEA concepts



The formulation for Linear Element is as follow

Points 0: F0 = **0.5* FT *(L1/LT)**

Points 1: F1 = **0.5* FT *(L1/LT) + 0.5* FT *(L2/LT)**

Points 2: F2 = **0.5* FT *(L2/LT) + 0.5* FT *(L3/LT)**

Points 3: F3 = **0.5* FT *(L3/LT) + 0.5* FT *(L4/LT)**

Points 4: F4 = **0.5* FT *(L4/LT)**

The general formula is as follow

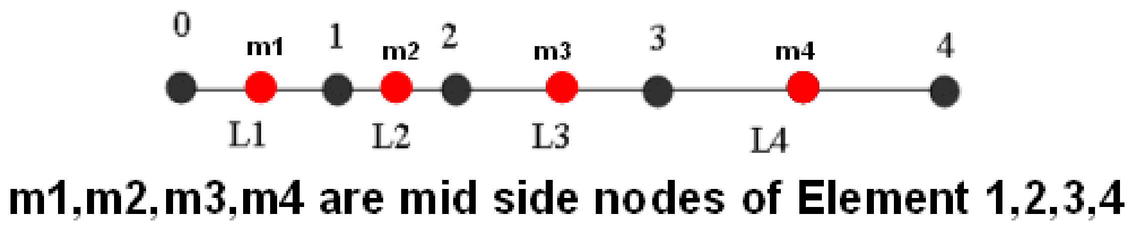
Fi = Sum [FT * (L (attached element)/LT) * (1/number of nodes per element)]

If we sum up **F0+F1+F2+F3+F4** then the resultant comes to be **FT**

It also satisfies the FEA concepts

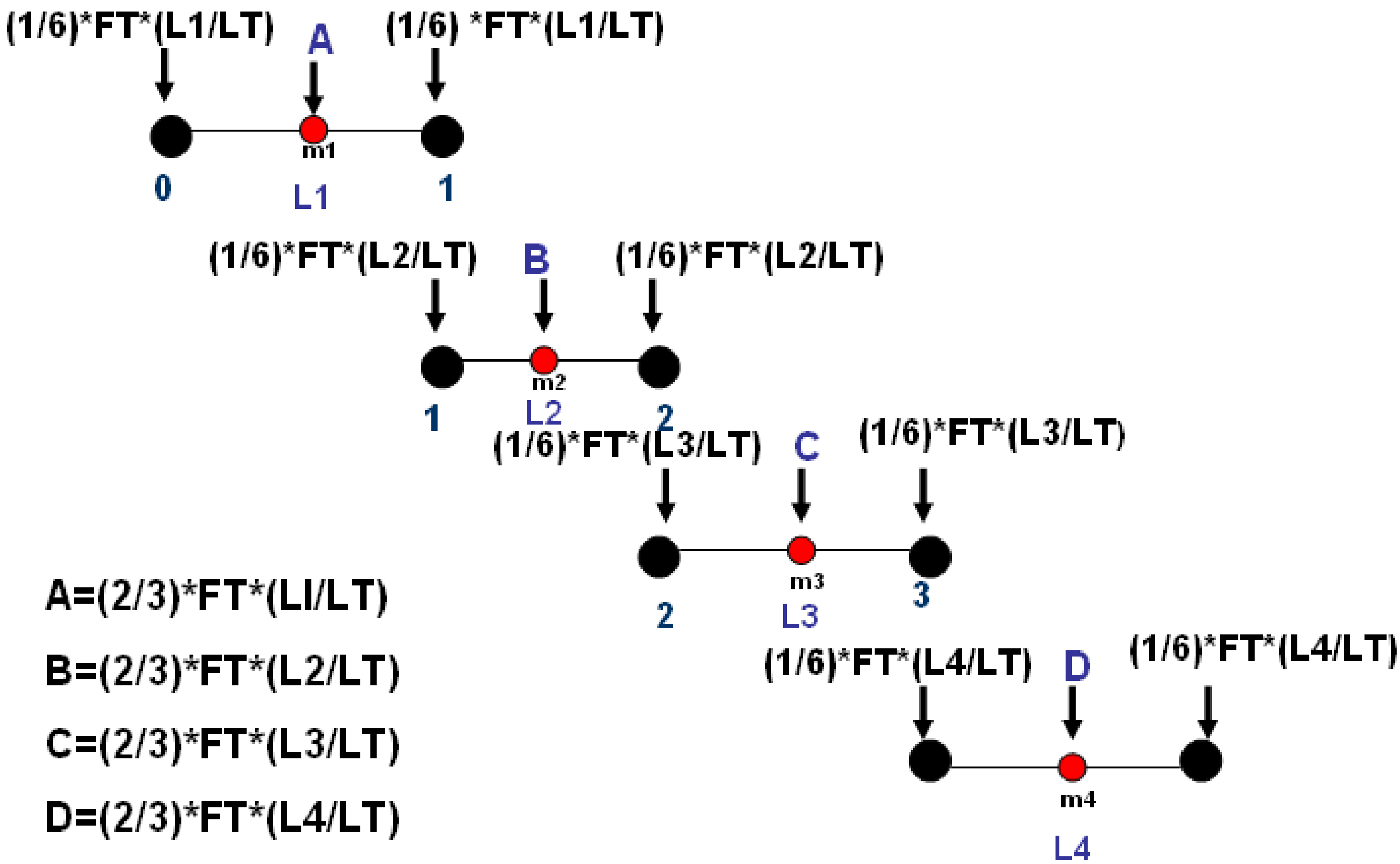
Now if we want to apply the same force on the **Quadratic Element** as shown in Figure 6.3: "Quadratic Element Nodes position".

Figure 6.3 Quadratic Element Nodes position



The Load distribution as per the FEA concept on the Quadratic elements is shown in Figure 6.4: "Load Distribution as per the FEA concepts".

Figure 6.4 Load Distribution as per the FEA concepts



The formulation of **Total Force** at the side nodes is as follow

Points 0:**F0q** = (1/6)* **FT** *(L1/LT)

Points 1:**F1q** = (1/6)* **FT** *(L1/LT) + (1/6)* **FT** *(L2/LT)

Points 2:**F2q** = (1/6)* **FT** *(L2/LT) + (1/6)* **FT** *(L3/LT)

Points 3:**F3q** = (1/6)* **FT** *(L3/LT) + (1/6)* **FT** *(L4/LT)

Points 4: **F4q** = (1/6)* **FT** *(L4/LT)

But the formulation of mid-side node is as follow

Points m1: **Fm1** = (2/3)* **FT** *(L1/LT)

Points m2: **Fm2** = (2/3)* **FT** *(L2/LT)

Points m3: **Fm3** = (2/3)* **FT** *(L3/LT)

Points m4: **Fm4** = (2/3)* **FT** *(L4/LT)

As in the previous case of Linear Element the total Force of Quadratic element is

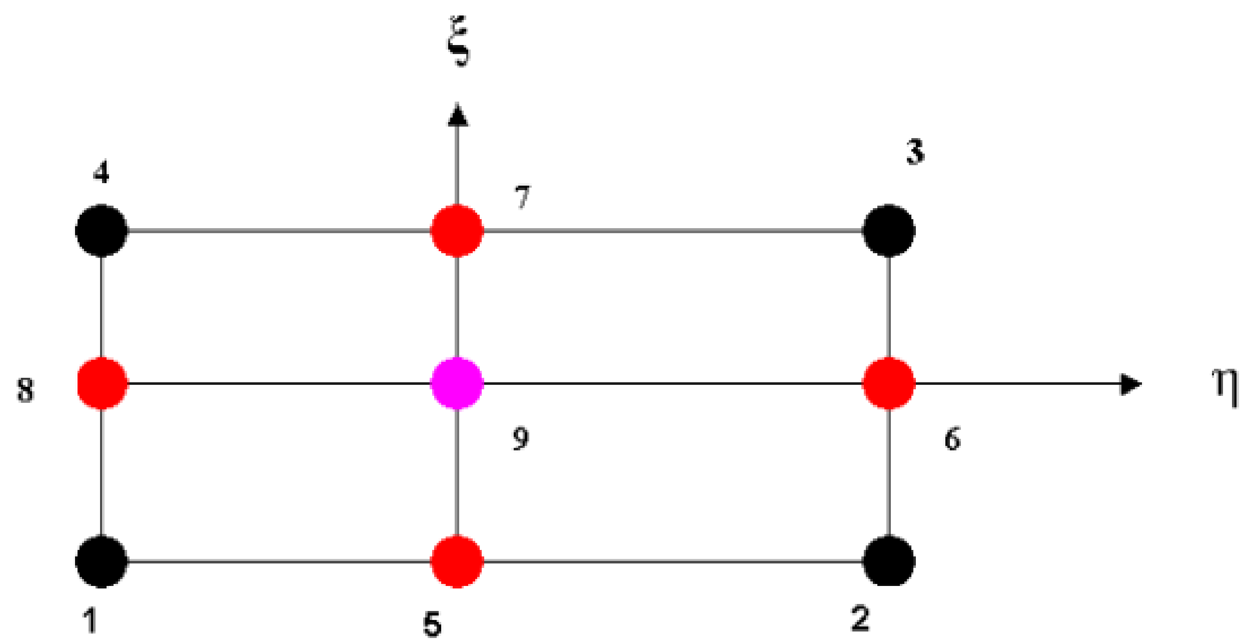
$\mathbf{F_{Total}}$= F0q+ F1q+ F2q+ F3q+ F4q+ Fm1+ Fm2+ Fm3+ Fm4

Which yields $\mathbf{F_{Total}}$=FT

QUAD 9 Consistent Nodal Load Distributions

The Nine Node two dimension Lagrange **QUAD 9** Element is shown in Figure 6.5: "QUAD9 Element".

Figure 6.5 QUAD9 Element



The Shape function for **Corner Node**

N_1 (?, ?) = (1/4)* ??(?-1)(?-1)

The Shape function for **Mid-Side Node**

N_5(?, ?) = (1/2)* ?(1-?^{2})(?-1)

The Shape function for **Middle Node**

N_9(?, ?) = (1-?^{2})(1-?^{2})

Suppose a Force **F** is uniformly distributed over the whole Area the pressure is

p=F/4 (Because in '**??'** coordinate system the **Area of the rectangle is '4'**)

To Find Consistent Load

Consistent Load at Node 1=L1

Consistent Load at Node 5=L5

Consistent Load at Node 9=L9

On Integration we get the following values

L1=p $_{?=-1}?^{?=+1}{}_{?}=-1?^{?=+1}N_1$ d?d?=4p/36

L5=p $_{?=-1}?^{?=+1}{}_{?}=-1?^{?=+1}N_5$ d?d?=4p/9

L9=p $_{?=-1}?^{?=+1}{}_{?}=-1?^{?=+1}N_9$ d?d?=16p/9

Now F=4p

Putting this value in the above equation we get the consistent Nodal Force as

L1=F/36

L5=F/9

L9=4F/9

By symmetry **Consistent Nodal Force** on Node **1, 2, 3&4** are equal.

By symmetry **Consistent Nodal Force** on Node **5, 6, 7&8** are equal.

Now **Total Force =4*L1+4*L5+L9**

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

=F

> *Note* — Note: The same method is employed for calculating the Consistent Nodal Force on **QUAD8** element, which gives correct results

6.1. Force on Point

Using this option user can apply force on points in all the three directions as well as user can apply the moment in x,y,z, directions. Here specifying negative value implies force applied in the negative direction.

6.2. Force on curve

Using this option user can apply force on Curves in all the three directions as well as user can apply the moment about x, y, z directions.

There are two options available Force can be applied on curves uniformly -implies the nodes attached to the curve will be applied the same force

Incase of Total, The force applied on the curve gets distributed on the nodes attached to the curves according to the FEA concepts

6.3. Force on Surface

Using this option user can apply force on Surface in all the three directions; Negative value indicates force acts away from the surface.

6.4. Force on Subset

Using this option user can apply **Force on Subset**.

6.5. Pressure on surfaces

The user can apply the **Pressure on Surfaces**.

6.6. Pressure on subset

The user can apply the **Pressure on Subset**.

6.7. Temperature on Points

This option allows the user to apply the **Temperature on point**.

6.8. Temperature on curves

This option allows the user to apply the **Temperature on Curve**.

6.9. Temperature on Surface

Allows user to apply **Temperature on Surface**

6.10. Temperature on Body

Allows user to apply **Temperature on Body**

6.11. Temperature on Subset

This option allows the user to apply the **Temperature on Subset**

6.12. Set Gravity

This option allows the user to apply the Gravity

Chapter 7: Solver Options

This Menu includes tabs for specifying the Solver. Specify the analysis Solution and Post processing

Fore more useful information about Solver Options, please go to http://www-berkeley.ansys.com/faq/faq_topic_7.html.

7.1. Setup Solver Parameters.

You can select from the following solvers: **ANSYS**, **Nastran**, **ABAQUS**, and **LSDyna**.

7.2. Setup Analysis Type

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

7.3. Setup Sub-Case

To apply the load in different steps, subcases can be created.

7.4. Write/View Input file

To create and view the input file generated for the solver.

7.5. Submit Solver Run

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

7.6. Post Process Results

Allows you to plot the results.

7.7. FEA Solver Support

For more useful information on FEA Solvers Support, please go to http://www-berkeley.ansys.com/faq/faq_topic_7.html

ANSYS ICEMCFD 10.0

Tutorial Manual

August 2005

Table of Contents

The ANSYS ICEMCFD Projects

Each project represents a directory within the $ICEM_ACN/../docu/CFDHelp/CFD_Tutorial_Files directory. Since some of the projects are used as examples in multiple meshing modules, this directory may contain several files. A particular project directory can contain one or more of the following files:

*.prj:	Project settings
*.tin:	Tetin (geometry)
*.uns:	Unstructured mesh
domain.*:	Multiblock structured hexahedral meshes
*.blk:	Block topology
*.fbc:	Boundary conditions (for solver output)
*.atr:	Attributes (for solver output)
*.par:	Parameters (for solver output)
*.rpl:	Replay script
*.jrf:	Journal (echo) file

These extensions are preceded typically with the project name, e.g., project1.tin is the tetin (geometry) file associated with project1. Most of the tutorials will already have a tetin file called geometry.tin (the project has yet to be created).

Some of the tutorials will begin with a 3rd party geometry, e.g., geometry.stl (stereolithography – triangulated surface data), which is then saved to the *.tin format.

The Tutorial Examples

It is recommended that for each chapter, the tutorials be done in sequence. Commands in succeeding tutorials may be referred to or explained in a previous tutorial. Please read through the introduction before beginning the tutorials.

The input files for the tutorials can be found within the ANSYS ICEMCFD installation. For example: ~/Ansys_inc/v100/icemcfd/linux64/../docu/CFDHelp/CFD_Tutorial_Files. They can also be downloaded from http://www-berkeley.ansys.com/icemcfd_ftp/index.html#icemcfd_100_manual. This download also contains these manuals in *.pdf format for hardcopy output.

Tutorial Design

These tutorials provide explanation for each step in the mesh generation process. The user not only learns the sequence of commands, but also comes to understand the concept behind the individual commands. After going through these tutorials, the user will be capable of extending his or her knowledge of the functions into more complicated projects.

Each example will either introduce new features or use familiar features in new ways to ultimately achieve better results for specific geometries. Lessons begin by outlining the functions and operations being introduced in the example. New features will receive the most thorough explanations in the chapter in which they are first introduced.

For specific questions regarding the usage of a command, refer to Help > Help Topics.

Text Conventions

The text conventions of this tutorial are categorized in the following manner:

“>” indicates order of selection. For example, “Edges > Group curve > screen select” means to choose the screen select option of the Group curve function found in the Edges menu.

Italicized font indicates a button selection.

Bold font indicates user input.

ALL CAPS indicates a part/entity name.

Mouse and Keyboard functions

Mouse Button or Keyboard key	**Action Description**
Left mouse button, click and drag	Rotates model
Right mouse button, click and drag up/down	Zooms in or out on the model
Right mouse button, click and drag left/right	Rotates model about screen Z-axis
Press F9, and then use any mouse button. Press F9 again to return to previous operation.	Toggles temporarily to dynamic mode (translate, zoom, rotate)
F11 Key	Emergency Graphics Reset

1: Introduction to ANSYS ICEMCFD

Meeting the requirement for integrated mesh generation and post processing tools for today's sophisticated analysis, ANSYS ICEMCFD provides advanced geometry acquisition, mesh generation, mesh optimization, and post-processing tools.

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

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

Multiblock structured

Unstructured hexahedral

Unstructured tetrahedral

Cartesian with H-grid refinement

Hybrid Meshes comprising hexahedral, tetrahedral, pyramidal and/or prismatic elements

Quadrilateral and triangular surface meshes

ANSYS ICEMCFD provides a direct link between geometry and analysis.

In ANSYS ICEMCFD, geometry can be input from just about any format, whether it is from a commercial CAD design package, 3rd party universal database, scan data or point data.

Beginning with a robust geometry module which supports the creation and modification of surfaces, curves and points, ANSYS ICEMCFD'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.

1.1: The Unified Geometry Concept

The unified geometry input environment in ANSYS ICEMCFD provides rapid geometry evaluation capability for computational mesh generation. This environment can combine CAD surface geometry and triangulated surface data into a single geometry database using the geometry interfaces.

All geometry entities, including surfaces, curves and points are tagged or associated to a grouping called a part. With this part association, the user can quickly toggle off or on all entities within the parts, visualize them with a different color, assign mesh sizes on all entities within the part and apply different boundary conditions by part.

Geometry is collected into a common geometry database (tetin file) which can be used by any of ANSYS ICEMCFD's meshing modules.

Direct CAD Interfaces and Intelligent Geometry

The ANSYS ICEMCFD Direct CAD Interfaces provide the bridge between parametric geometry creation tools available in CAD systems and the computational mesh generation, post-processing and mesh optimization tools available in ANSYS ICEMCFD, allowing users to operate in their native CAD systems. ANSYS ICEMCFD currently supports Direct CAD Interfaces for:

Catia

I-deas

Pro/E

Unigraphics

Solid Works

In an environment that has the look and feel of their native CAD system, users can choose solids, surfaces, curves and points, group these entities into parts and assign mesh sizes for mesh generation.

Further information on ANSYS ICEMCFD's Direct CAD Interfaces is available in the ANSYS ICEMCFD Direct CAD Interface Tutorial Manual.

Since the CAD geometry is tagged with mesh parameters and boundary conditions directly in this interface, the user can recalculate a mesh reflecting these changes in the geometry immediately after having saved the geometry file.

3rd Party Interfaces

Available for STEP/IGES, DXF, GEMS, ACIS, DWG, Parasolid and point data.

Triangulated Surface Data Input

Available for STL, Patran, Nastran, Plot3d (a popular Aerospace format for multiblock structured surface meshes) and VRML.

1.2: The ANSYS ICEMCFD Geometry Interface

Geometry Tools

ANSYS ICEMCFD includes a wide range of tools for creating new and/or manipulating existing geometry. This allows the user to alter complex geometry or create simple geometry without having to go back to the original CAD. This can be done for CAD (NURBS surfaces) and triangulated surface data.

Although most of the meshing modules within ANSYS ICEMCFD are forgiving of minor gaps and holes in the geometry, in some cases it is necessary to find and close large gaps and holes without returning back to the original CAD. ANSYS ICEMCFD provides tools to do both on either CAD or triangulated surfaces.

Finally, curves and points can be automatically created to capture certain key features in the geometry. These curves and points will act as constraints for the mesher, forcing nodes and edges of the elements to lie along them, and thus capturing the hard feature.

1.3: Meshing Modules

Tetra/Auto Volume

ANSYS ICEMCFD Tetra takes full advantage of object-oriented unstructured meshing technology. With no tedious up-front triangular surface meshing required providing well-balanced start meshes, ANSYS ICEMCFD Tetra works directly from the CAD surfaces and fills the volume with tetrahedral elements using the Octree approach. A powerful smoothing algorithm provides the element quality. Options are available to automatically refine and coarsen the mesh both on geometry and within the volume.

A Delauny algorithm is also included to create tetras from surface mesh that already exists and also to give a smoother transition in the volume element size.

Hexa

This ANSYS ICEMCFD semi-automated meshing module presents rapid generation of multi-block structured or unstructured hexahedral volume meshes.

ANSYS ICEMCFD Hexa represents a new approach to grid generation where the operations most often performed by experts are automated and made available at the touch of a button.

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

Prism

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

Hybrid Meshes

Hybrid meshes can be created by several means:

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

Hex 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. Hex core allows for reduction in number of elements for quicker solver run time and better convergence.

Shell Meshing

ANSYS ICEMCFD provides a method for rapid generation of surface meshes (quad and tri), both 3D and 2D. Mesh types can be all quad, quad dominant or all tri. Three methods are available:

Patch based shell meshing: Uses a series of “loops” which are automatically defined by the boundaries of surfaces and/or a series of curves. Gives best quad dominant quality and capturing of surface details

Patch independent shell meshing: Uses the Octree method. Best and most robust on unclean geometry.

Mapped based shell meshing: Internally uses a series of 2d blocks, results in mesh better lined up with geometry curvature.

1.4: Mesh Visualization and Optimization

Mesh visualization tools, including solid/wireframe display, 2D cut planes, color coding and node display is provided.

After initial mesh is created by any of the meshing modules, diagnostics can be performed to determine local and overall mesh quality. Automatic smoothing algorithms are in place to improve overall quality. Local editing can be done using a wide range of automatic re-meshing and manual mesh editing tools.

Output Interfaces

ANSYS ICEMCFD includes output interfaces to over 100 flow and structural solvers, producing appropriately formatted files that contain complete mesh and boundary condition information.

Post Processing

ANSYS ICEMCFD Visual3 provides easy-to-use powerful result visualization features for structured, unstructured and hybrid grids, both steady-state and transient.

Visual3 integrates CAD Geometry, computational grids and the flow solution within one environment. It provides an in-depth view of data with visualization tools such as cut planes, stream ribbons, contours, vectors, grids, iso-surfaces, offset surfaces, result surfaces, integration, XY plots, data probes, function calculator, solution and experimental comparison, scripts, annotations and animation. Results can be interpolated on imported surface meshes and written out to a different solution process, e.g. interpolating fluid results on to a surface mesh and brought in as loads for a structural analysis.

A surface manager tool controls the display status for all surfaces, including any dynamic surface, domain surfaces and user-defined surfaces.

2: ANSYS ICEMCFD GUI

ANSYS ICEMCFD's unified graphical user interface, also known as AI*Environment, offers a complete environment to create and edit your computational grids.

The AI*Environment GUI includes the following:

Main menu

Function Tabs

Utility icons

Data Entry Zone

Display Control Tree

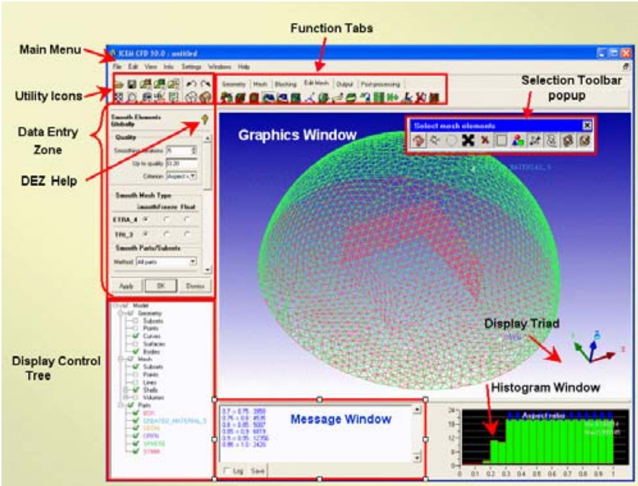
Graphics Window

Message Window

Histogram (Quality) Display Window

Selection Toolbar

Figure 2.1
The ICEM CFDMesh Editor



2.1: Main Menu

Many of the following menu items are accessible as icons in the upper left hand corner.

File

The File menu is used to create new or open existing projects, loading and saving files, importing and exporting geometries and initialize scripting.

Edit

This menu contains Undo/Redo, the option to open a shell window, and various internal mesh/geometry conversion commands.

View

Contains various options for the standard view, controls, and annotations.

Info

This menu allows the user to get various information regarding geometry, mesh and individual entities.

Settings

Contains default settings for performance, graphics, and other settings most likely to be used more than 90% of the time by a specific user.

Help

Contains links to reference material, tutorials, user's guide and version information.

2.2: Utilities

Icon representation of some of the most commonly used functions represented in the Main menu including opening/closing a project, undo/redo, and display options. It also includes measurement and setup of local coordinate systems.

2.3: Function Tabs

The main functionality for the entire grid generation process is accessed through the function tabs which include: **Geometry, Mesh, Blocking, Edit mesh, Output, Post-processing**.

2.3.1: The Geometry menu

The Geometry menu includes functions for the creation, editing and repair of geometry.
For more information on ANSYS ICEMCFD's Tetin files and treatment of geometry entities, refer to the section on Geometry definitions in Help > Help Topics.
Functions and utilities in this menu include:
Create Point
Create/Modify Curve
Create/Modify Surface
Create Body
Create Faceted
Repair Geometry
Transform Geometry
Restore Dormant Entities

Delete Point, Delete Curve, Delete Surface, Delete Body and Delete Any Entity.

2.3.2: The Mesh menu

These tools are the heart of ANSYS ICEMCFD. The Mesh menu contains the ANSYS ICEMCFD meshing modules as well as options to set mesh sizes (parameters).

Depending on the licensing, some users may not be able to access certain meshing modules. Contact customer support or ANSYS ICEMCFD's website for guidance with any licensing questions, or for help with adding any additional modules to the license.

The following buttons would lead to different mesh generation modules, which ANSYS ICEMCFD maintains and develops:
Set Global Mesh Size
Set Surface Mesh Size
Set Curve Mesh Size
Set Meshing Params by Part
Create Mesh Density
Create Elements
Surface Meshing
Volume Meshing
Mesh Prism
Global Cartesian Mesher
Extrude Mesh.
Pressing any of these buttons will invoke the preferred meshing module.

2.3.3: The Blocking menu

The Blocking menu contains the functions necessary to create a topology for block structured hexahedral meshes. Either a block file must be loaded or an initial block created to make all the items active.
Create Block
Split Block
Merge Vertices

Edit Block
Associate
Move Vertex
Transform Blocks
Edit Edge
Pre-Mesh Params
Pre-Mesh Quality
Pre-Mesh Smoothing
Block Checks
Delete Block

2.3.4: The Edit Mesh menu

The Edit mesh menu contains tools necessary for mesh editing, both automated and manual. Operations include:
Create Elements
Check Mesh
Display Mesh Quality
Smooth Mesh Globally
Smooth Hexahedral Mesh Orthogonal
Repair Mesh
Merge Nodes
Split Mesh
Move Nodes
Transform Mesh
Convert Mesh Type
Adjust Mesh Density
Renumber Mesh
Reorient Mesh
Delete Nodes
Delete Elements

2.3.5: The Output menu

The Output menu contains all tools necessary for setting up the model and writing out to the solver:
Select Solver

Boundary Conditions
Edit Parameters
Write Input

2.3.6: The Post Processing menu

The Post Processing menu controls the viewing of solution results. A results file (from various CFD and structural formats) must first be loaded to make this menu active. The functions included in the post processing menu are:
Set Transient Time
Variables
Define Cut Plane
Define Iso Surface
Point Probe on Surface
Import External Surface
Streams
Control All Animations
Annotation
XY or polar

2.4: The Display Control Tree

The Display Control Tree, also referred to as the Display tree, along the lower left side of the screen, allows control of the display by part, geometric entity, element type and user-defined subsets.
The tree is organized by categories. Each category can be turned on or off by selecting the check box. If the check mark is faded, some of the sub-categories are turned on and some off. Each category can be expanded by selecting the "+" symbol to reveal the sub-categories. Select "-" to collapse the tree.
Since some functions are performed only on the entities shown, the model tree is a very important feature to use when isolating the particular entities to be modified.
Right mouse selecting a particular category or type will reveal several display and modification options.

2.4.1: Geometry

Controls display of points, curves, surfaces and bodies (material volumes). Subsets can also be created, displayed and modified. A given subset can contain any number of different geometry types. A given entity can belong to more than one subset.

2.4.2: Mesh

Controls display of all mesh types: points (node elements), lines (bars), shells (tris or quads) and volumes (tetras, pyramids, prisms, hexas). Subsets within this category are the same as for Geometry but contain only mesh element types.

2.4.3: Parts

All entities, geometry or mesh, are associated to a given part. An entity cannot belong to more than one part. With this association, groups of entities, regardless of type can be toggled on and off. Parts have a specific color to discern them from other parts. Parts can be made sub-categories of assemblies, created by right mouse selecting on “Parts.” Individual parts can then be dragged and dropped into the assembly. Toggling the assembly on/off will turn on/off all the parts within the assembly as for any category/sub-category.

2.4.4: The Message window

The Message window contains all the messages that ANSYS ICEMCFD writes out to keep the user informed of internal processes. The Message window displays the communicator between the GUI and the geometry and meshing functions. It is important to keep an eye on the Message window, because it will keep the user informed of the status of operations.

Any requested information, such as measure distance, surface area, etc. will be reported in the message window.

Also, internal commands can also be typed and invoked within the message window.

The Save commands will write all Message window contents to a file. This file will be written to wherever ANSYS ICEMCFD was fired.

The Log toggle switch allows only user specified messages to be saved to a file.

It is important to note that the Log file is unique from the file created with the Save button. This file will be written to the starting directory, and it interactively updates as more messages are recorded. Once the toggle is turned OFF, you can continue to add to the file by turning the toggle back ON and accepting the same file name (which is the default). It will then continue to append this file.

2.5: The Histogram window

The Histogram window shows a bar graph representing the mesh quality. The X axis represents element quality (usually normalized to between 0 and 1) and the Y axis represents the number of elements.

Other functions which utilize this space will become pop-up menus if the quality or histogram is turned on.

3: CFD Tutorials

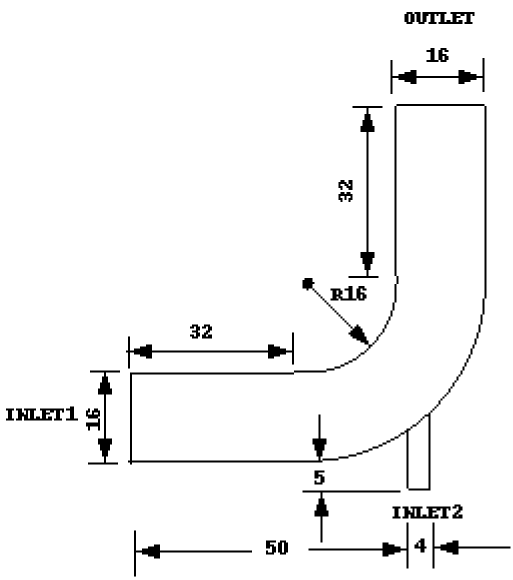
3.1: Geometry Creation

3.1.1: 2D Pipe Junction

Overview

We are going to create geometry for a two-dimensional pipe junction as shown in Figure3.1.

Figure3.1
2D Pipe Junction with Dimensions



a) Summary of steps

Geometry Menu

Creating the points using Explicit Coordinates

Creating the points using Curve-Curve Intersection

Creating the curves using From Points

Creating the curves using Arc through 3 points

Segmentation of curve using Segment Curve

Deleting unused entities

Creating Material Point using Mid Point

File Menu

Saving the geometry

b) Generating the Geometry

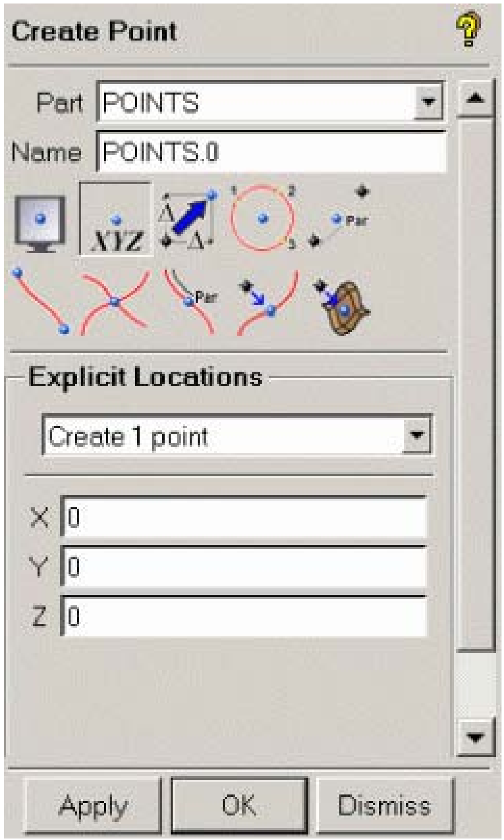
Point Creation

Note: Settings > Selection >Auto pick mode should be turned OFF for ANSYS ICEMCFD to behave exactly as this tutorial describes.

Geometry >Create Point > Explicit Coordinates: Select the 

(Explicit coordinate) to open the Explicit Location window as shown in Figure 3.2.

Figure 3.2
Point creation window



As shown in Figure 3.2, Select Create 1 Point, and assign coordinates (0 0 0).Input the Part name POINTS, and the Name as POINTS.0 and press Apply to create a point.

Switch ON the Geometry > Points in the left side Display Tree window. To see the names of the points, use the right mouse button and select Points > Show Point Names in the Display Tree window. Select Fit Window from the main menu. Use the right mouse button to zoom out if needed. The created point name would be shown as POINTS.0.

Similarly create the other points by entering POINTS.1 in the name field and specifying the coordinates as (32, 0, 0).

Now enter the coordinates as shown below, and press Apply after each one. You will see the names automatically change to the ones shown below:

POINTS.2	(0, 16, 0)
POINTS.3	(32, 16, 0)
POINTS.4	(48, 32, 0)
POINTS.5	(48, 64, 0)
POINTS.6	(64, 32, 0)
POINTS.7	(64, 64, 0)
POINTS.8	(50, -5, 0)
POINTS.9	(54, -5, 0)
POINTS.10	(16, 32, 0)
POINTS.11	(0, 32, 0)
POINTS.12	(50, 16, 0)
POINTS.13	(54, 16, 0)

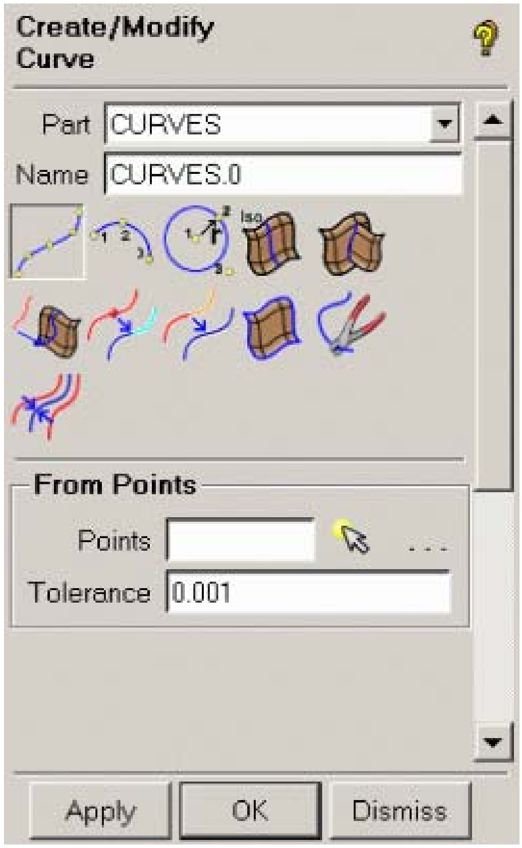
Figure 3.3
Points created thus far

Press **Dismiss** to close the window. The Display window should now show the points as seen in **Figure 3.3**.

Line Creation

Geometry > Create/Modify Curve > From Points: Select the From Points option  to open the window as shown in **Figure 3.4**.

Figure 3.4 :
From points window



To select Points, click on  (select point icon) and then select POINTS.0 and POINTS.1 with the left mouse button. Press the middle mouse button to accept the points. The point names will appear in the selection window. Enter the Part as CURVES, and the Name as CURVES.0. Press **Apply** to create the line.
Switch ON Geometry > Curves in the Display Tree if they are switched off. To see the names of the curves, use the right mouse button and select Curves > Show Curve Names in the Display Tree. Use the right mouse button to zoom out if needed. The created line name would be shown as CURVES.0

Similarly, select the following points, pressing Apply each time. Without changing the Name entry, by default the names of each new curve would appear as shown on the left:
CURVES.1 from POINTS.0 and POINTS.2
CURVES.2 from POINTS.2 and POINTS.3
CURVES.3 from POINTS.4 and POINTS.5
CURVES.4 from POINTS.5 and POINTS.7
CURVES.5 from POINTS.6 and POINTS.7
CURVES.6 from POINTS.8 and POINTS.9
CURVES.7 from POINTS.8 and POINTS.12
CURVES.8 from POINTS.9 and POINTS.13
Press **Dismiss** to close the window.

Arc Creation

Geometry > Create/Modify Curves > Arc Through 3 points: Select the Arc Through 3 Points option  to open the window as seen in Figure 3.5.

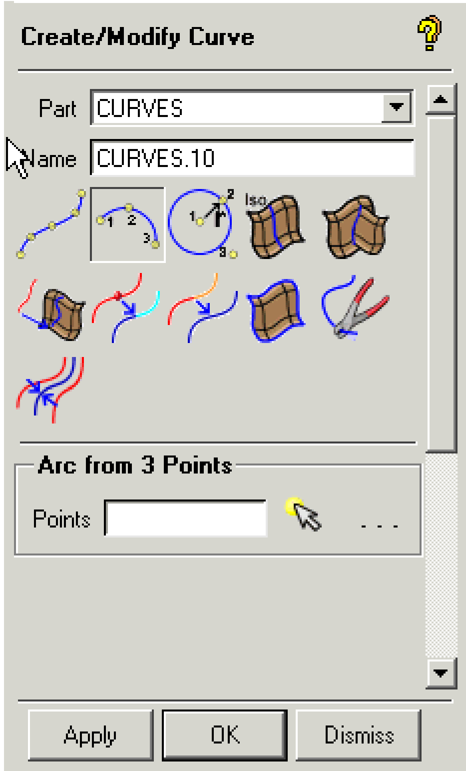
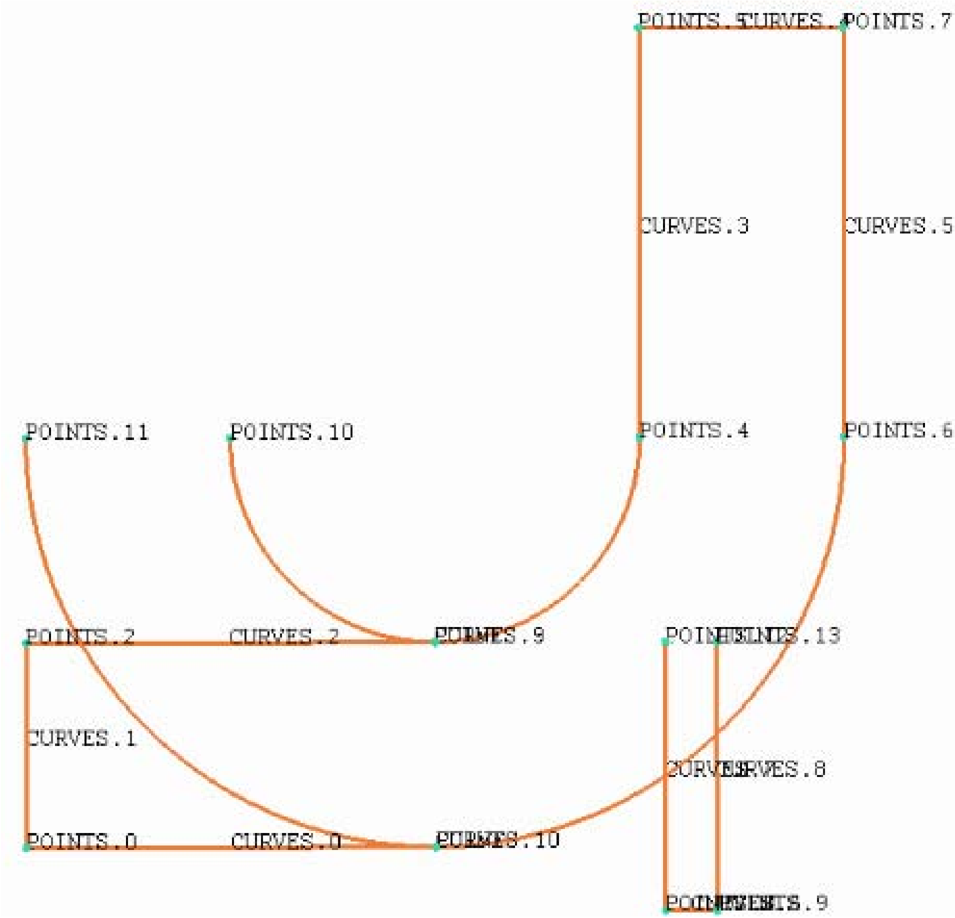


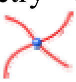
Figure 3.5
Arc from 3 points window

To select Points click on  (select point icon), and select the points **POINTS.4, POINTS.3** and **POINTS.10** with the **left mouse** button. Press the middle mouse button to accept the point. Click on the drop down menu next to the Part field to select an existing Part
Click on **CURVES** to select this Part in the window. Enter the Name as **CURVES.9** and press Apply to create the arc.
Similarly, make another arc named **CURVES.10** out of points **POINTS.6, POINTS.1**, and **POINTS.11**. Press Dismiss to close this window. The geometry after creating the two arcs is shown in Figure 3.6.

Figure 3.6
Geometry after arc creation

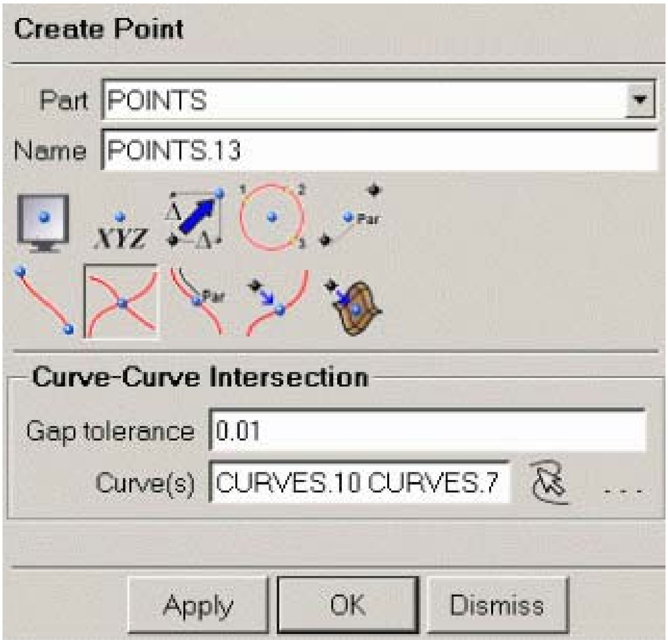


Curve-Curve Intersection

Geometry >Create Point > Curve-Curve Intersection: Selecting Curve-Curve  opens the window as shown in Figure 3.7. Select the Part name **POINTS**. Select **CURVES.10** and **CURVES.7** with the **left mouse button**. Press the **middle mouse** button to accept the selection. Give Gap a Tolerance of **0.01** and enter the name of the last point created

(POINTS.13) in the Name window and press Apply. This will create the intersection point called **POINTS.14**. Repeat the procedure for curves **CURVES.10** and **CURVES.8** and press Apply without changing the name in the Name window to get the intersection point **POINTS.15**. Press **Dismiss** to close the Create Point window.

Figure 3.7 Selection window of Curve-Curve Intersection



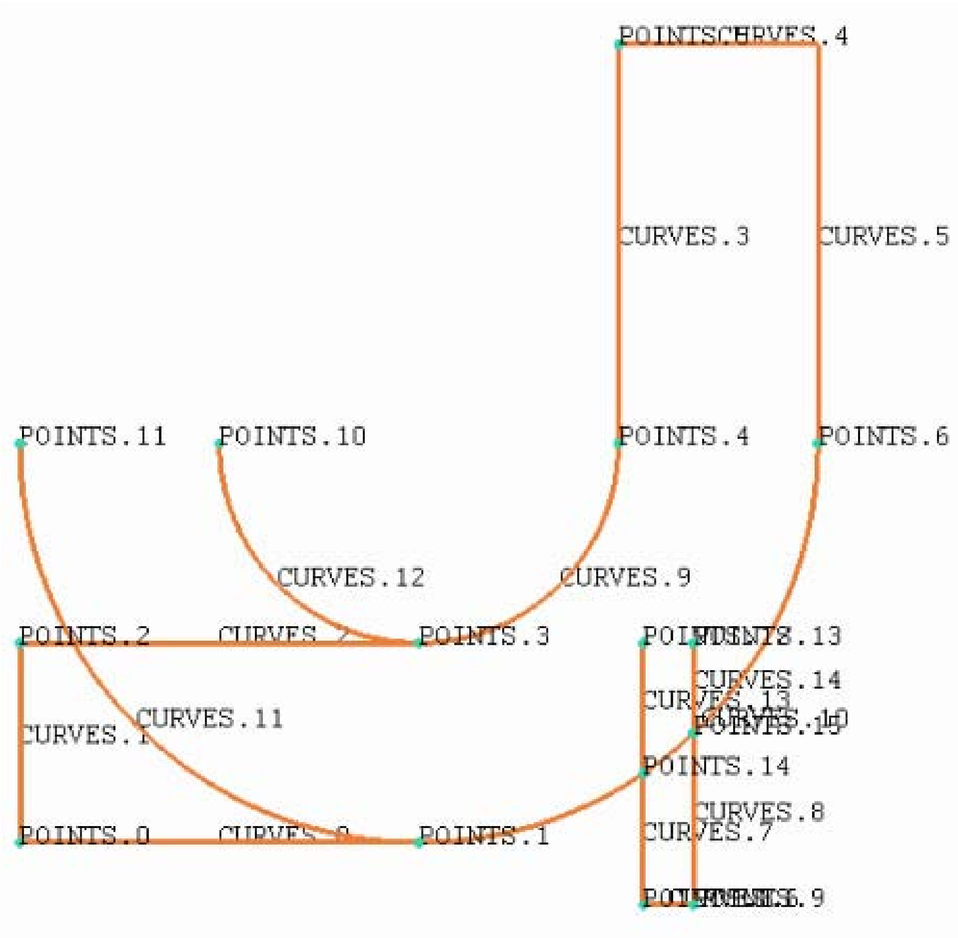
Segmentation of Curves at existing points

Geometry > Create/Modify Curve > Segment curve**:** Select the Segment Curve option . In the dropdown, Segment by Point should be selected. Select the curve selection icon and select **CURVES.10** with the **left mouse button**. Now select the point selection icon and select **POINTS.1** with the left mouse button and then press the middle mouse button to accept the point. Select the Part **CURVES** and enter the name as

CURVES.10 so that the next curves will start with **CURVES.11.** After pressing Apply, the **CURVES.10** segments into two curves, **CURVES.10** and **CURVES.11**.
Similarly segment **CURVES.9** at **POINTS.3** to get **CURVES.9** and **CURVES.12.** Segment **CURVES.7** at **POINTS.14** to get **CURVES.7** and **CURVES.13.** Segment **CURVES.8** at **POINTS.15** to get **CURVES.8** and **CURVES.14.** The geometry after segmenting the curve is shown in Figure 3.8.

Note: After segmenting two Curves at a particular Point the Curves name may be different but user can refer to the Figure 3.8 and select the Curves to be deleted.

Figure 3.8
Geometry after curve segmentations



Deletion of unused entities

Geometry > Delete Curves: Select (Delete Curve) to open the Delete Curve window. Select the curve selection icon and select **CURVES.11, CURVES.12, CURVES.13** and **CURVES.14**. Press the **middle mouse** button to complete the selection. Press Apply to delete these curves.

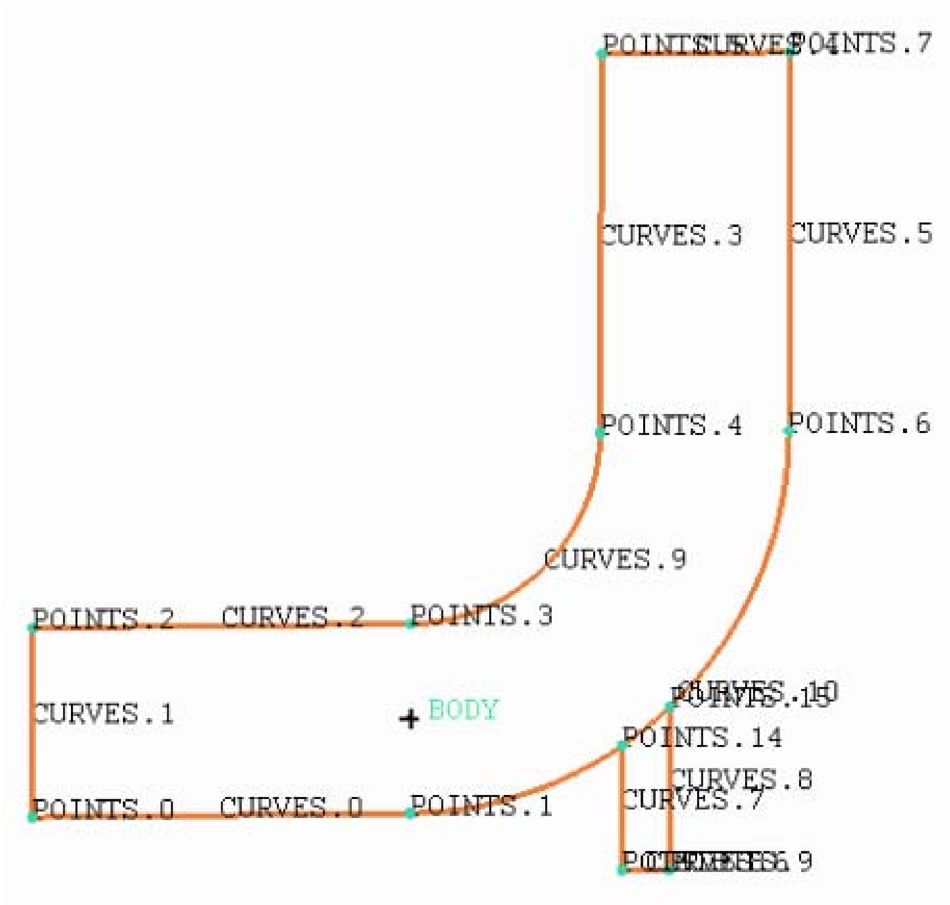
Geometry >Delete Points: Select the (Delete Point) to open the Delete Points window. Select the point selection icon and select **POINTS.10, POINTS.11, POINTS.12** and **POINTS.13**. Press the middle mouse button to complete selection, and press Apply to delete these points.

c) Creating the Material point

Geometry > Create Body > Material Point > Centroid of 2 points:

Select the location selection icon and click close to **POINTS.1** and **POINTS.3** with the left mouse button. Press the middle mouse button to complete the selection. Give the Part name **BODY**, and press Apply to create the material point. Switch on Bodies in the left side Display Tree window to see the body. The Geometry after creating material point is shown in Figure 3.9.

Figure 3.9
Final Geometry



d) Saving Geometry

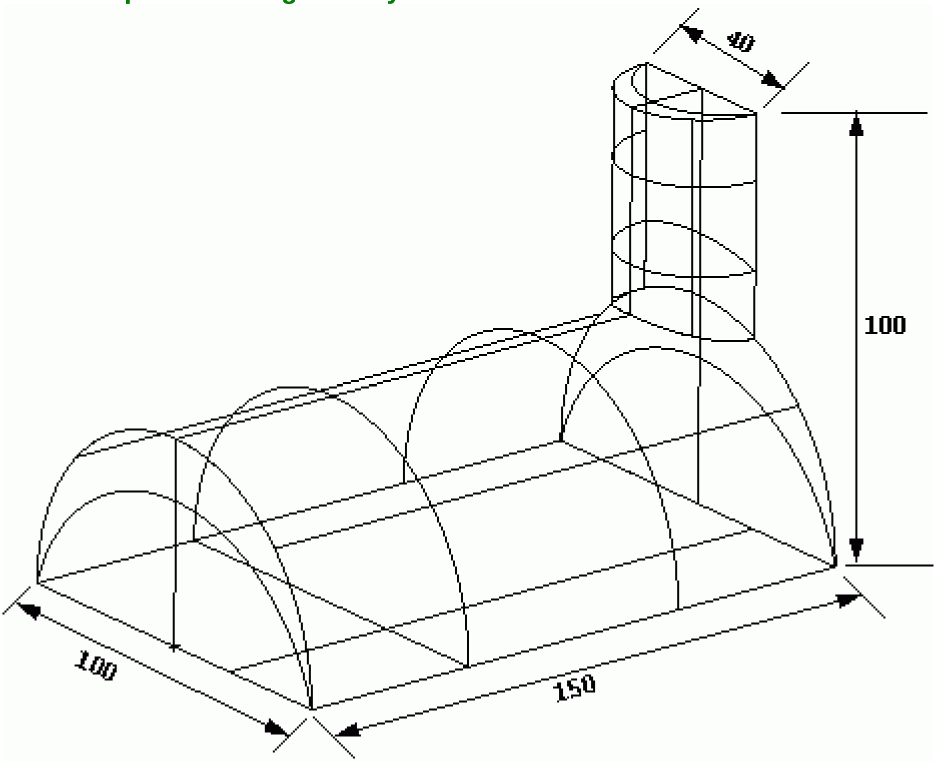
File > Geometry > Save Geometry As: Enter the file name as **Geo_2DPipe.tin** and press **Save** to save the geometry file

3.1.2: 3D Pipe Junction

Overview

We are going to create geometry for a three-dimensional pipe junction as shown in Figure 3.10.

Figure 3.10
The 3D Pipe Junction geometry with dimensions



a) **Summary of steps**

Create points, then curves from points
Create surfaces from curves
Curves from Surfaces-Surface Intersection
Segment surface with the intersection curve

Delete unused segmented surfaces

Create material point

b) Generating the Geometry

Point Creation

Note: Settings > Selection >Auto pick mode should be turned OFF for ICEM CFD to behave exactly as this tutorial describes.

Geometry >Create Point> Explicit Coordinates: Select XYZ to open the Explicit Locations window. Type the Part name **POINTS,** and the Name as **POINTS.0**, and enter the co-ordinates (**0 0 0**). Press Apply to create the point.

Switch on the Geometry > Points in the left side Display Tree window. To see the names of the points, use the **right mouse button** and select **Points > Show Point names** in the Display Tree window. Select **Fit Window** from the main menu. Use the right mouse button to zoom out if needed. The created point name will be shown as **POINTS.0**.

Similarly, enter coordinates as (0,0,50) and enter the name as **POINTS.1** next to Name. Press Apply.

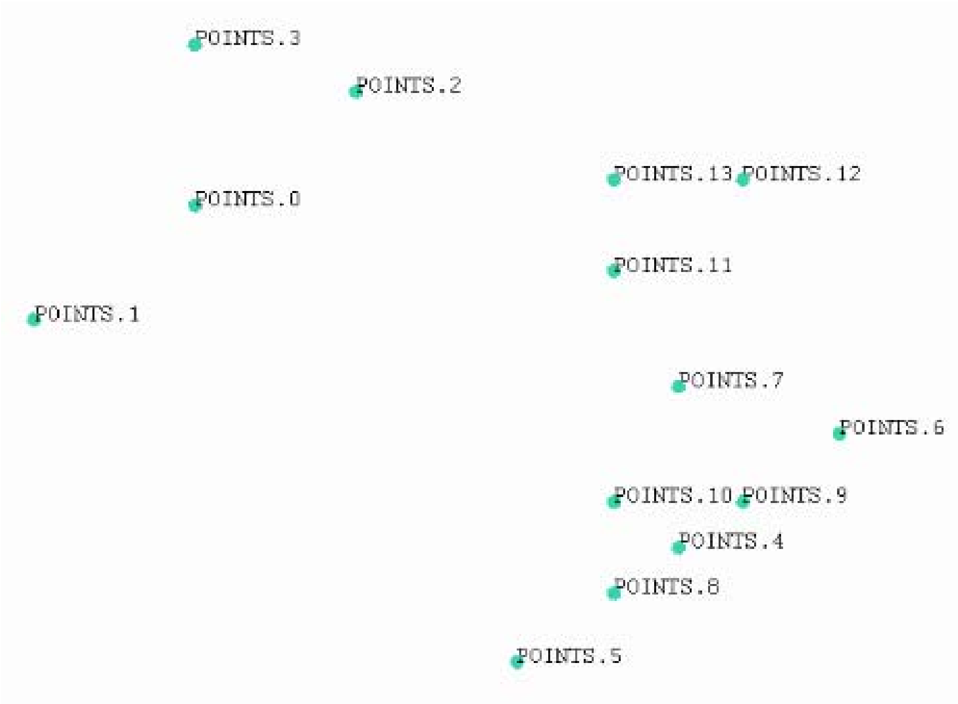
Now, create the rest of the points listed below by just entering the locations. The names continue on from POINTS.1, so they will automatically change as shown below:

POINTS.2	(0, 0, -50)
POINTS.3	(0, 50, 0)
POINTS.4	(150, 0, 0)
POINTS.5	(150, 0, 50)
POINTS.6	(150, 0, -50)
POINTS.7	(150, 50, 0)
POINTS.8	(150, 0, 20)
POINTS.9	(150, 0, -20)
POINTS.10	(130, 0, 0)

POINTS.11	(150, 100, 20)
POINTS.12	(150, 100, -20)
POINTS.13	(130, 100, 0)

Press **Dismiss** to close the **Explicit Location** window. The points should appear as shown in Figure 3.11 when oriented in the **Isometric** view.

Figure 3.11
Points created



Arc Creation

Geometry > Create/Modify Curve > Arc through 3 points: Select  (Arc Through 3 Points) to open window, then select the location selection icon  and select points **POINTS.1, POINTS.3** and **POINTS.2** with the **left mouse button**. By default the Part name is **CURVES.** Enter the Name as **CURVES.0.** Press Apply to create the arc.

Switch **ON** the **Curves** in the left side Display Tree window. To see the names of the curves, use the right mouse button and select **Curves > Show Curve names** in the Display Tree window. The newly created curves name will display as CURVES.0.

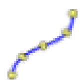
Similarly, select POINTS.5, POINTS.7 and POINTS.6 and enter the name as **CURVES.1.** Press Apply to create the arc.

Now, make two more arcs by just selecting the points as specified below and pressing Apply each time. The curve names will be generated as shown below:

CURVES.2: POINTS.8, POINTS.10 and POINTS.9
CURVES.3: POINTS.11, POINTS.13 and POINTS.12

Press Dismiss to close the window.

Line Creation

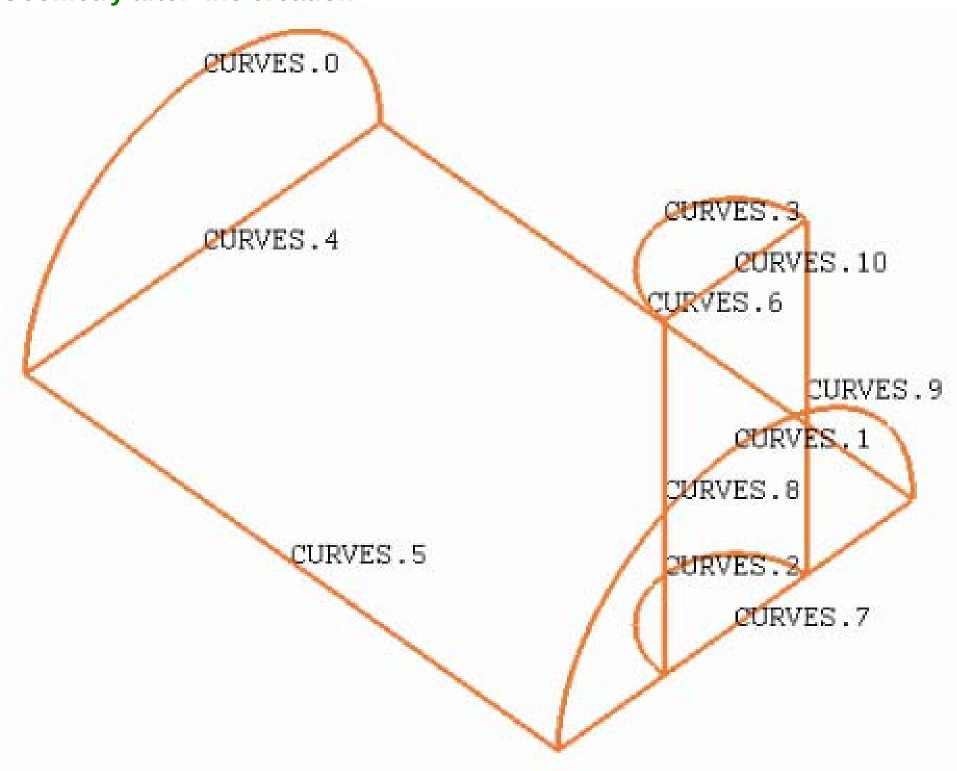
Geometry > Create/Modify Curves > From Points: Select  (From Points). Press the location selection icon , select the Points **POINTS.1** and **POINTS.2** with the **left mouse button**, and press the **middle mouse button** to complete the selection. Enter the Part name **CURVES** and Name **CURVES.4**. Press Apply to create the line.

Similarly, create six more lines using the points listed below for each curve. The curve names will adjust consecutively for each curve:

CURVES.5: POINTS.1 and POINTS.5
CURVES.6: POINTS.2 and POINTS.6
CURVES.7: POINTS.5 and POINTS.6
CURVES.8: POINTS.8 and POINTS.11
CURVES.9: POINTS.9 and POINTS.12
CURVES.10: POINTS.11 and POINTS.12

Press **Dismiss** to close the window. The Geometry after curve creation is shown in Figure 3.12. Switch OFF the Points in the Display Tree window to avoid clutter on the screen.

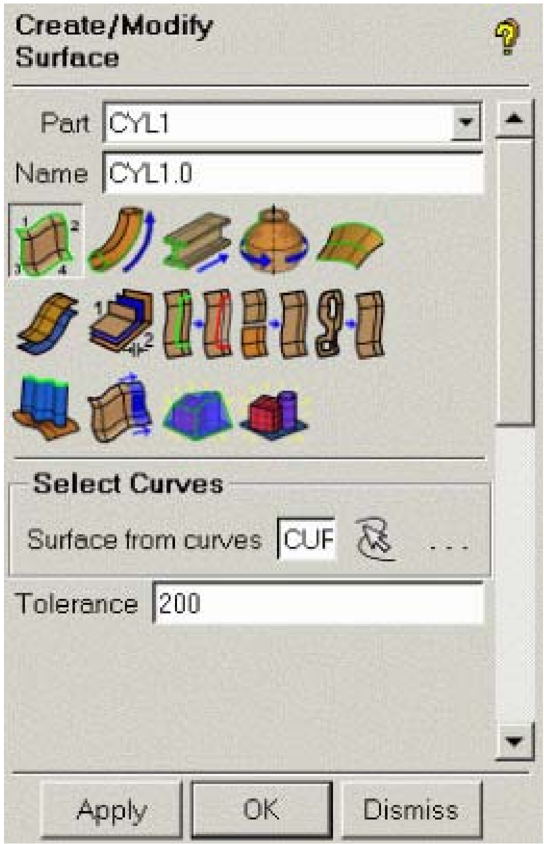
Figure 3.12
Geometry after line creation



Surface Creation

Geometry >Create/Modify Surface > From curves: Select (From Curves) icon to open the window shown in Figure 3.13.

Figure 3.13
Surface creation from curve



Press the curve selection icon and select the curves CURVES.0 and CURVES.1 with the left mouse button. Press the middle mouse button to complete the selection. The distance between these two curves that the surface must cross is 150, so enter a Tolerance bigger that than, such as 200. Enter the Part name CYL1 and Name CYL1.1. Press Apply to create the surface.

Note: Pressing the right mouse button while in selection mode will cancel each previous selection.

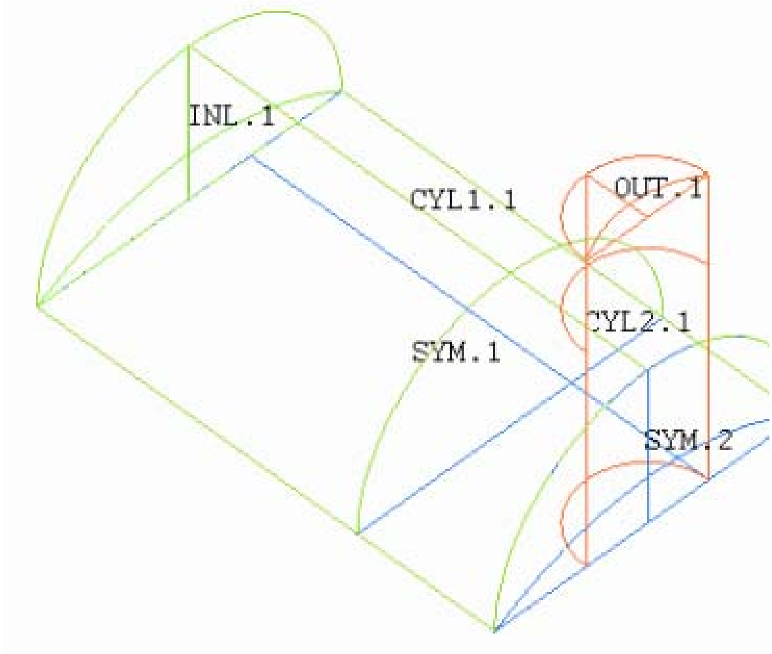
Switch **ON** the **surfaces** from the left side Display Tree window. To see the names of the surfaces, using the right mouse button, select Surface > Show Surface names in the Display Tree window. Use the right mouse button to zoom out if needed. The newly created surface name would display as CYL1.1.

Similarly, create the other surfaces as follows, entering the part names and names each time:

PART	**NAME**	**SELECTED CURVES**
INL	INL.1	CURVES.0, CURVES.4
CYL2	CYL2.1	CURVES.2, CURVES.3
OUT	OUT.1	CURVES.3, CURVES.10
SYM	SYM.1	CURVES.4, CURVES.7
SYM	SYM.2	CURVES.1, CURVES.7

Press **Dismiss** to close the window. The Geometry after surface creation is shown in Figure 3.14. Switch **OFF** the Curves from the Display Tree window to avoid clutter on the screen.

Figure 3.14
Geometry after Surface creation



Surface-Surface Intersection

Geometry >Create/Modify Curves > Surface-Surface Intersection: Select (Surface-Surface Intersection. Press the surface selection icon to select Set1 surfaces as **CYL1.1** and Set2 surfaces as **CYL2.1** with the **left mouse button**, pressing the **middle mouse button** to complete the selection each time. Press Apply to create the intersection curve.

Segmentation of Surface

Geometry > Create/Modify Surface > Segment/Trim surface: Select (Segment/Trim Surface), and choose the Method **by Curves,** which is the default. Press the surface selection icon and select the surface **CYL1.1** using the left mouse button and press the middle mouse button to complete the selection. Press the curve selection icon and using the left mouse button, select the intersection curve that was created in the previous step. Press the middle mouse button to complete the selection. Press Apply to segment the surface CYL1.1 into two parts. Similarly, segment the surface CYL2.1 with the same intersection curve. If the two previous curves have been split into two, then select both curves.

Deleting unused entities

Geometry > Delete Surface: Select (Delete Surface) to open the Delete Surface window, Select the surface **CYL1.1** and **CYL2.1.cut.0** with the left mouse button. Press the middle mouse button to complete the selection and press Apply to delete these surfaces.

Note: The curves and points will need to be deleted, so the next step, which is "build topology," will not segment the surfaces where the curves span them.

Geometry > **Delete Curve:** Select the (Delete Curve) to open the Delete Curves window. Check ON **Delete permanently**. Press "a" on the keyboard to select all curves, and press Apply to delete.

Geometry > **Delete Point:** Select the (Delete Point) to open the Delete Point window. Check ON **Delete permanently.** Press "a" on the keyboard to select all points, and press Apply to delete them.

Build topology

Geometry > Repair Geometry > Build Diagnostic Topology: Select (Build Diagnostic Topology) from the Geometry tab. This will extract all the curves from the surfaces, and the points from the curves. But the new curves will only span the boundary of the new surfaces after segmenting and deleting. The **tolerance** should be 0.1, and **Filter points** and **Filter curves** should be turned off. Press Apply.

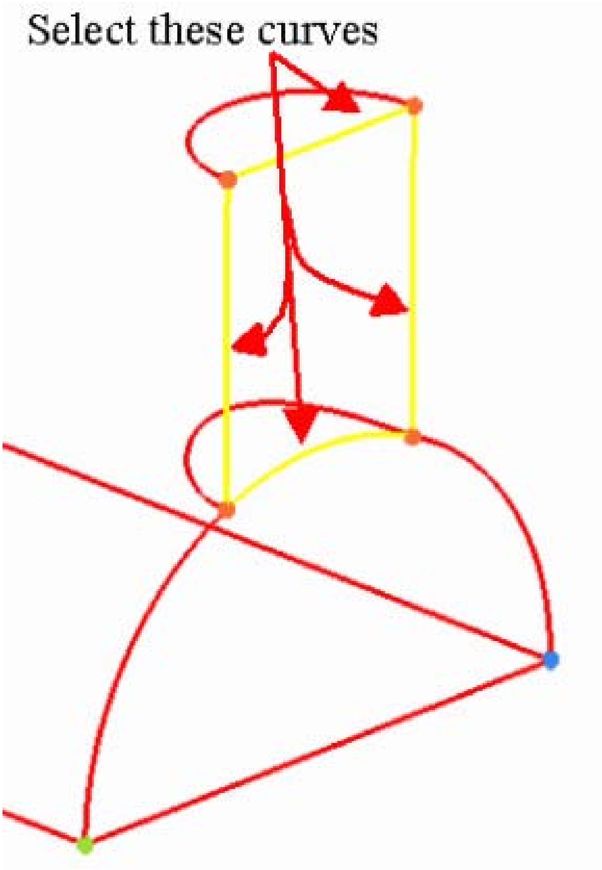
Surface creation

First, make sure **Curves** are ON in the Display Tree.

Geometry > Create/Modify Surface > From curves: Select  (From Curves) to open the Create/Modify Surface window. Select the curves shown in Figure 3.15 below with the left mouse button and press the middle mouse button to complete the selection. Make sure the Part name is SYM and the Name is SYM.3. Press Apply to create the surface.

Press **Dismiss** to close the window.

Figure 3.15
Curves for surface



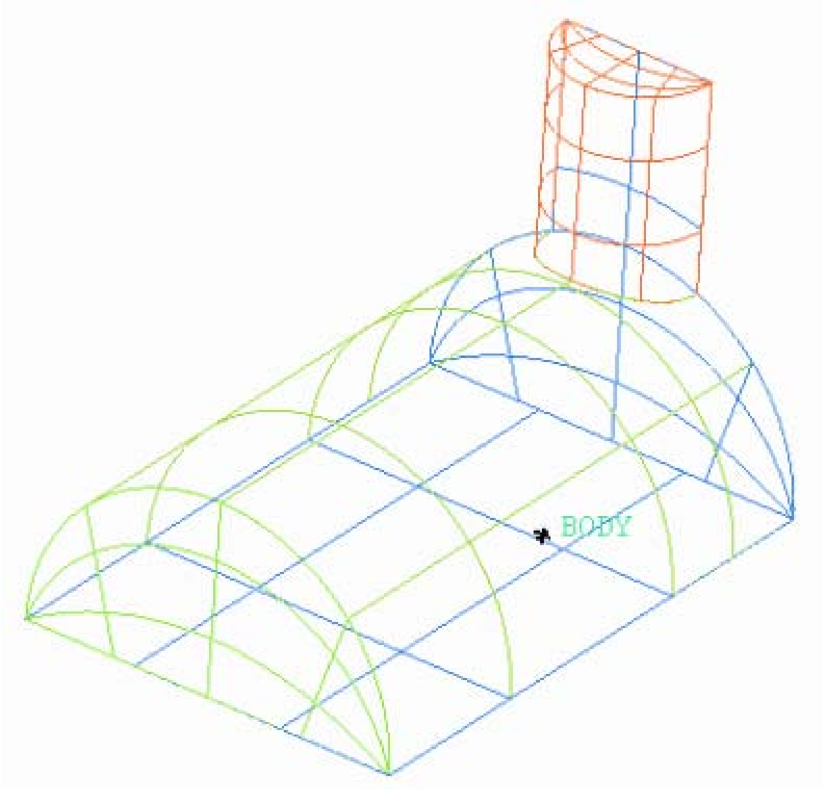
c) Creating the Material point

Geometry > Create Body >Material Point >Centroid of 2 points:

Select (Create Body). Type **BODY** for the new Part name. Select any two locations on any surfaces, curves or points so that the midpoint will be within the pipe junction. Press the middle mouse button to accept, then press Apply. Switch **ON Bodies** in the Display Tree to see the material point. The final geometry is shown in Figure 3.16.

Figure 3.16

Final Geometry



d) Saving Geometry

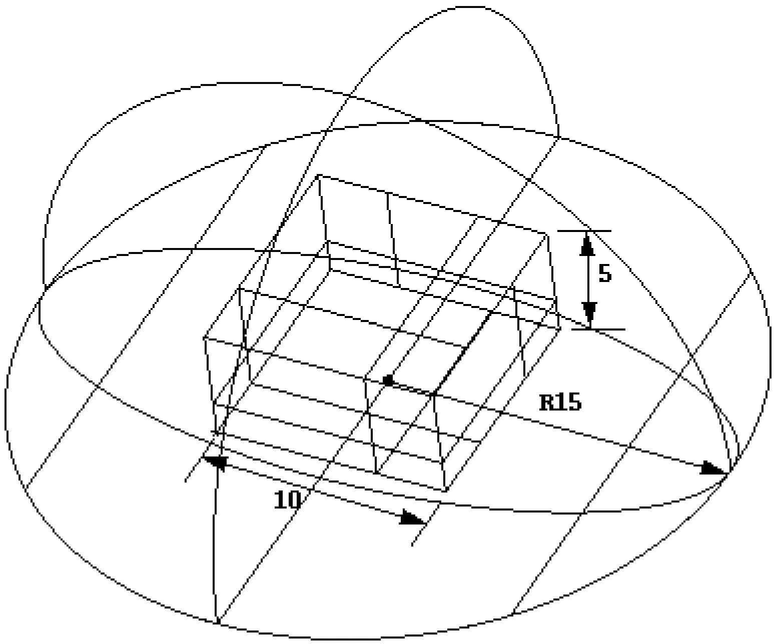
File > Geometry > Save Geometry As: Enter the file name as Geo_3DPipe.tin and press **Save** to save the geometry.

3.1.3: Sphere Cube

Overview

We will create geometry for a sphere cube as shown in Figure 3.17.

Figure 3.17
The sphere cube with dimensions



a) **Summary of steps**

Create Cube by **Standard Shapes**
Create Hemisphere (**Surface of Revolution**)
Create points at Parameter along curve
Create arcs to use to create the symmetry surfaces

b) Generating the Geometry

Note: Settings >Selection>Auto pick mode should be turned OFF for ICEM CFD to behave exactly as this tutorial describes.

Point Creation

Geometry >Create Point> Explicit Coordinates: Select XYZ (Explicit Coordinates) to open the Explicit Location window, Give the Part name **POINTS,** and the Name **POINTS.0**. Enter the co-ordinates (**5, -10, 0**), and press Apply to create the point.

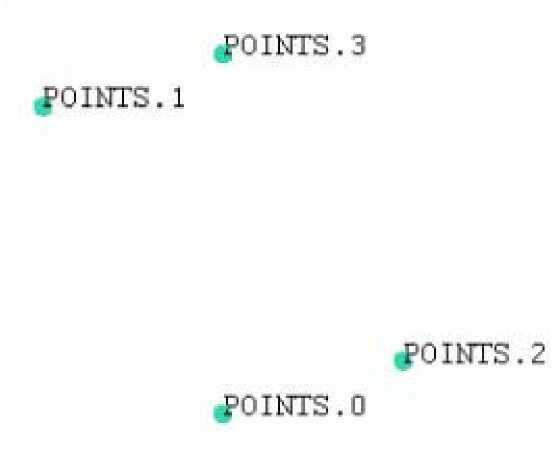
Switch on the **Points** in the Display Tree window. To see the names of the points, use the right mouse button to select **Points > Show Point names** in the Display Tree window. Select **Fit Window** from the main menu. Use the right mouse button to zoom out if needed. The newly created point name would be displayed as POINTS.0.

Similarly, enter the coordinate as (-10,5,0) and enter the name as **POINTS.1.** Then press Apply. Then create 2 more additional points at the following locations. The names will automatically adjust as shown below:

POINTS.10 (20, 5, 0)
POINTS.11 (5, 20, 0)

Press **Dismiss** to close the window. The Geometry after point creation is shown in Figure 3.18 in **Isometric** view.

Figure 3.18
Points created so far



Arc Creation

Geometry > Create/Modify Curve > Arc through 3 points: Select  (Arc Through 3 Points) to open the Arc from 3 Points window. Then select **POINTS.1, POINTS.0** and **POINTS.2** with the **left mouse button**, and press the **middle mouse button** to complete selection. Enter the Part as **CURVES** and Name as **CURVES.0**. Press Apply to create the arc.

Similarly, create another arc called **CURVES.1** from points **POINTS.1, POINTS.3**, and **POINTS.2**. Press Dismiss to close the window.

Note: Turn on **Curves** in the Display tree to see the curves.

Cube Creation

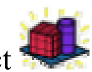
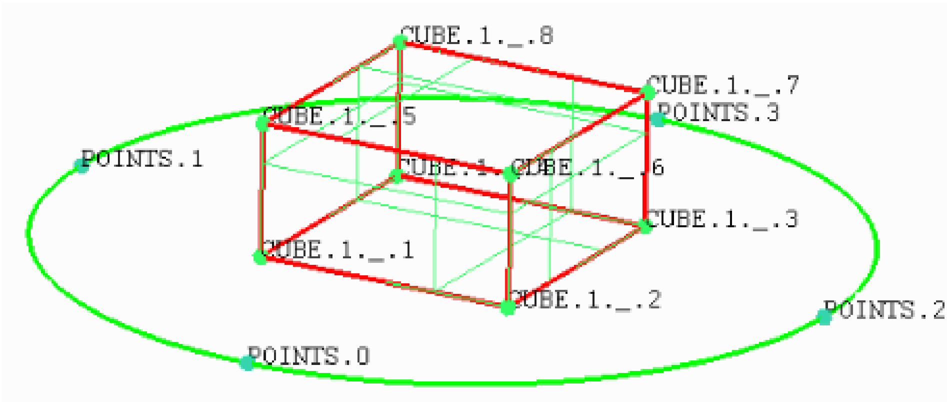
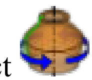
Geometry > Create/Modify Surface > Standard Shapes: Select  (Standard Shapes) to open the Create Std Geometry window and choose the Box about a Point icon . Change the Part name to **CUBE,** and the Name to **CUBE.1**. Enter the XYZ size as "**10 10 5". These values will be in X, Y and Z directions as 10, 10 and 5 respectively. Type “**0 0 0**” for the Box Origin coordinates. Press Apply to create the cube. The geometry so far should look as in Figure 3.19.

Figure 3.19: Geometry so far



Hemisphere Creation

Note: Turn ON Geometry > Curves > Show Curve Names in the Display Tree, to see which curve to select in this step.

Geometry > Create/Modify Surface> Surface of Revolution: Select  (Surface of Revolution) to open the window shown in Figure 3.20. Change the Part to **SPHERE**, Name to **SPHERE.1**. Enter the Start angle **0** and the End angle as **180. S**elect Axis Points as **POINTS.1** and **POINTS.2**. Select curves as **CURVES.0** and press Apply to create the hemisphere.

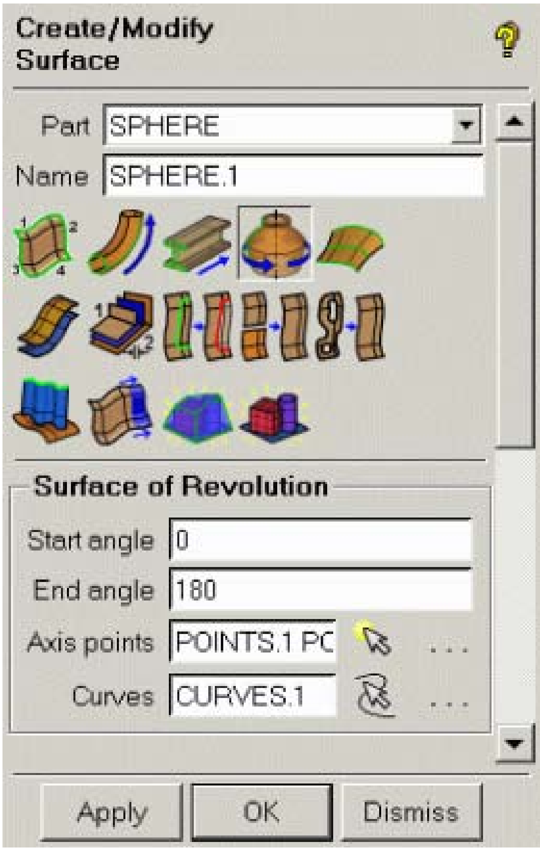
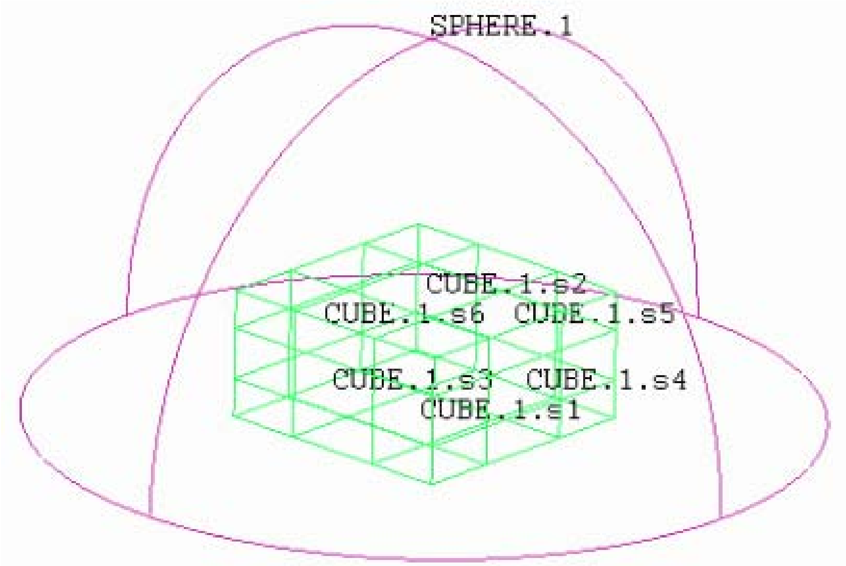


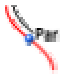
Figure 3.20: Surface of revolution window

Switch **ON** the **Surfaces** in the Display Tree window. To see the names of the surfaces, select **Surfaces > Show Surface Names** in the Display Tree window using the right mouse button. The geometry should resemble Figure 3.21 with the Points and Curves OFF.

Figure 3.21 Geometry after revolution

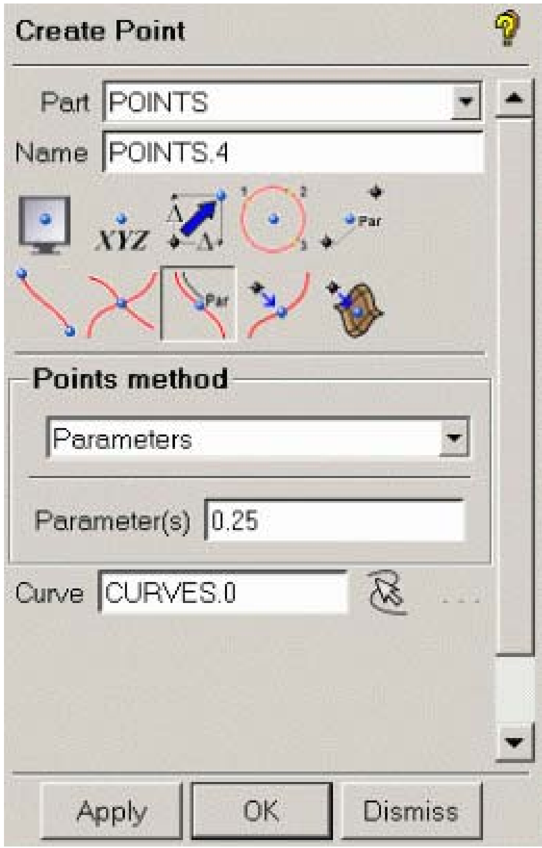


Point Creation

Geometry >Create Point> Parameter along a Curve: Select 

(Parameter along a curve) to open the window as seen in Figure 3.22. For a clearer view, the user can turn **OFF** Surfaces in the Display Tree, and make sure that Curves and Points are ON.

Figure 3.22: Point Parameter on curve window



Enter the Part as **POINTS**. And enter the Name as **POINTS.4.** Then select the curve, **CURVES.0**. Enter Curve Parameter **0.25** and press Apply to create **POINTS.4**. Then change the parameter to **0.75,** and press Apply again to create **POINTS.5.**

Next, select the curve, **CURVE.1**. You will need to turn off the part **SPHERE** in the Display Tree to be able to select CURVE.1. Also turn OFF Points > Show Point Names in the Display Tree to be able to see the curve names better. With the parameter left at **0.75**, press Apply to create POINTS.6. Then change the parameter to **0.25,** and press Apply again to create POINTS.7.

Press **Dismiss** to close the selection window.

Geometry > Delete Curve: Select (Delete Curves), and toggle ON **Delete permanently**. Select the curves, **CURVES.0** and **CURVES.1**, and press Apply.

Arc Creation

Geometry > Create/Modify Curve > Arc through 3 points: Select (Arc through 3 Points) to open the Arc from 3 Points window. Make sure Point Names are being displayed by right clicking in the Display Tree on Points > Show Point Names. Select the points, **POINTS.5, POINTS.2,** and **POINTS.6**. Enter the Part as **CURVES** and the Name as **CURVES.0**. Press Apply to create the arc.

Similarly create three other arcs by using the following points:
CURVES.1: POINTS.6, POINTS.3 and POINTS.7
CURVES.2: POINTS.7, POINTS.1 and POINTS.4
CURVES.3: POINTS.4, POINTS.0 and POINTS.5

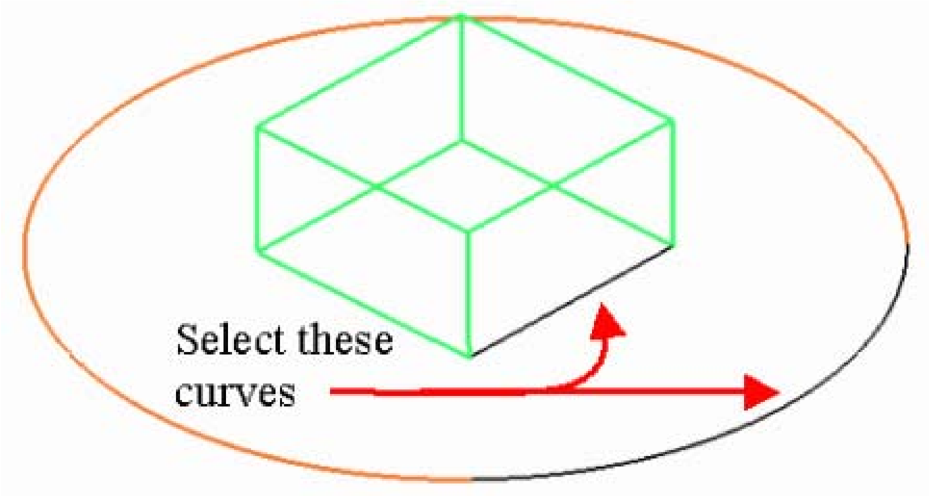
Press **Dismiss** to close the window.

Surface Creation

Geometry >Create/Modify Surface > From curves: Select (From Curves) to open the Select Curves window. Turn **OFF** the **Points** (Geometry) for a better view. Also turn OFF the curve names for a better view: Curves > Show Curve Names. Select the two curves shown in Figure 3.23 with the left mouse button, and press the middle mouse button to complete the selection. Assign the Part as **SYM** and Name as **SYM.1.** Enter a tolerance greater than the gap that the surface must jump across. 10 will work fine here. Press Apply to create the surface.

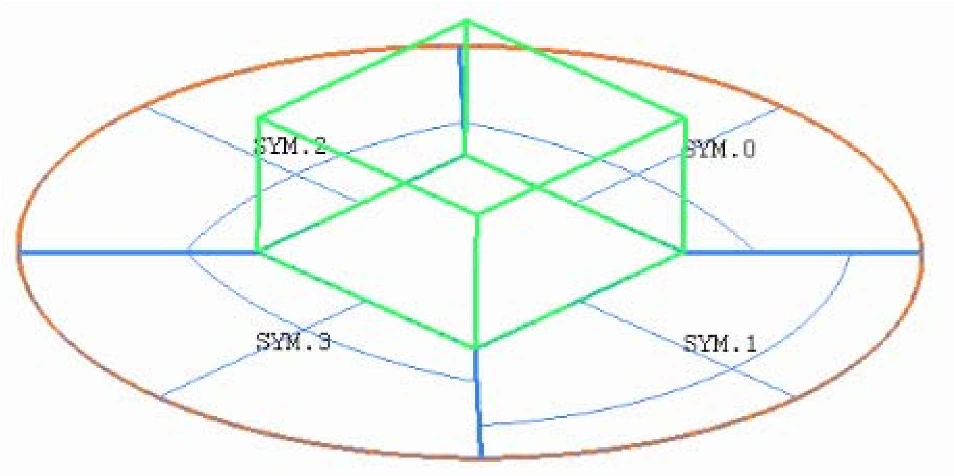
Press Dismiss to close the window.

Figure 3.23 Curves for Surface



Similarly, create the other three surfaces around the cube. The result is shown in Figure 3.24:

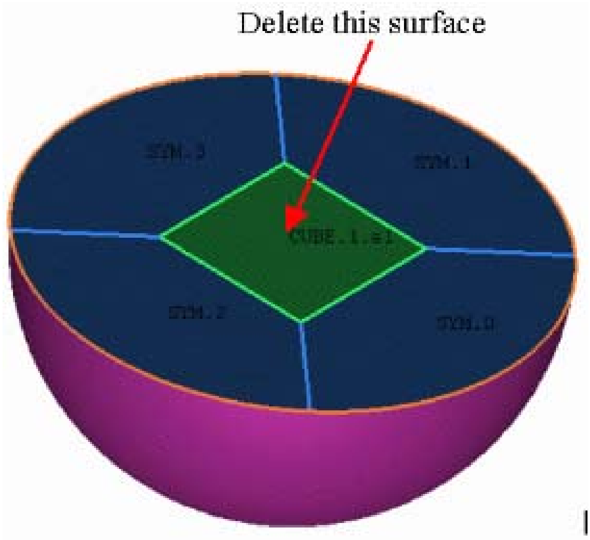
Figure 3.24
Symmetry Surfaces



c) Deleting unused entities

Geometry > **Delete Surface:** Select (Delete Surface) to open the Delete Surface window. Select the surface shown in Figure 3.25 with the **left mouse button**. If there is too much clutter, the user can switch **OFF** all other Parts except CUBE. Press the **middle mouse button** to complete the selection, and press Apply to delete the surface.

Figure 3.25 Surface to delete

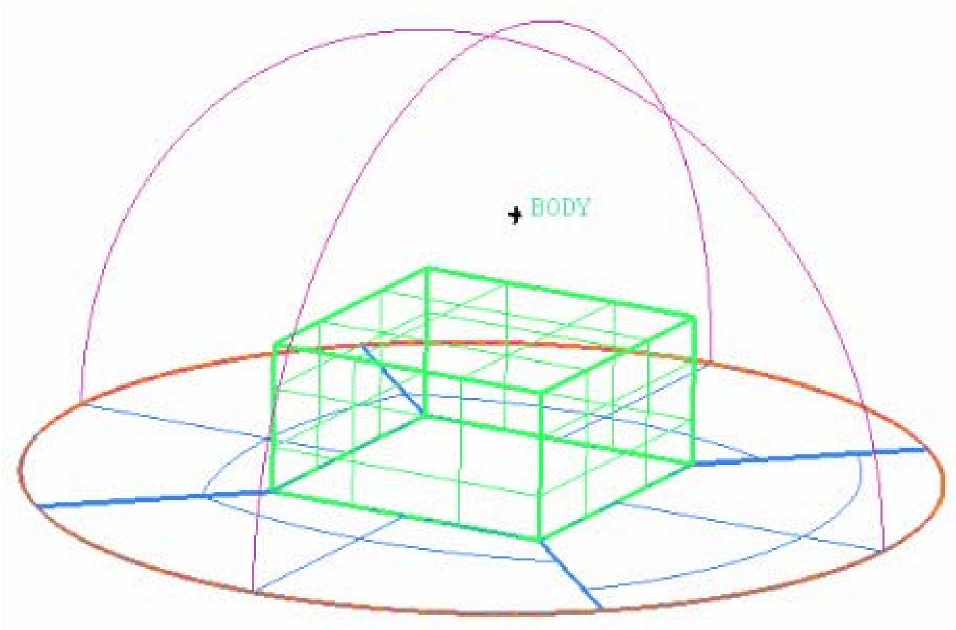


d) Creating the material point

Geometry > Create Body >Material Point >Centroid of 2 Point: Select 

(Create Body) and assign the name **BODY** to a new Part. Select one of the corners of the CUBE that do not lie inside the flat plane of the SYM surfaces. Select the second point on the surface of the sphere. Then press Apply. Switch **ON Bodies** in the Display Tree window to see the material point. It should appear inside the hemisphere of the SPHERE surface but outside of the CUBE. The final geometry is shown in Figure 3.26

Figure 3.26 Final Geometry



e) Saving Geometry

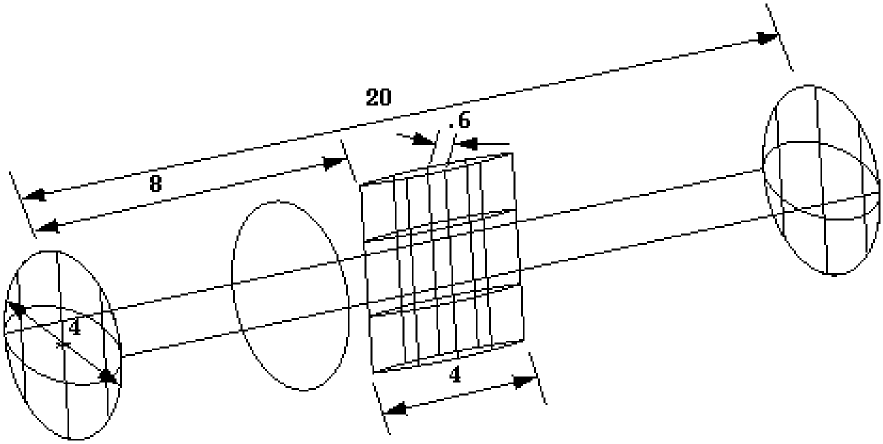
File > Geometry > Save Geometry As: Enter the file name **Geo_SphereCube.tin** and press **Save** to save the geometry.

3.1.4: Pipe Blade

Overview

We are going to create the geometry for a pipe blade as shown in Figure 3.27 .

Figure 3.27 : The Pipe Blade with dimensions



a) Summary of steps

Geometry Menu

Create points
Create arcs for the blade
Create Cylinder from Standard Shapes
Create surfaces for the blade and inlets and outlets.
Intersect surfaces, and trim surfaces by those intersection curves.

b) Generating the Geometry

Note: Settings >Selection > Auto pick mode should be turned OFF for ICEM CFD to behave exactly as this tutorial describes.

Point Creation

Geometry >Create Point> Explicit Coordinates: Select XYZ (Explicit Coordinates) to open the window. Assign the Part name POINTS, and the Name POINTS.0. Enter the co-ordinates **(0, 2, 8),** and press **Apply** to create the point.

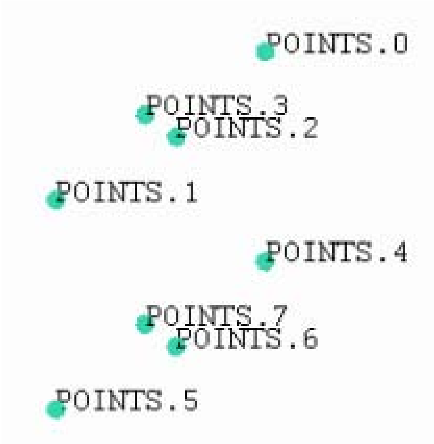
Switch on the **Points** in the Display Tree window. To see the names of the points, use the right mouse button and select **Points > Show Point names** in the Display Tree window. Select Fit Window from main menu. Use the right mouse button to zoom out if needed. The newly created point name would be displayed as POINTS.0.

Similarly, create another point by entering the coordinate **(0,2,12)** and the Name as POINTS.1 and press **Apply.** Enter the following coordinates, pressing **Apply** each time, and the names will automatically adjust to the names shown below:

POINTS.2	(0.3, 2, 10)
POINTS.3	(-0.3, 2, 10)
POINTS.4	(0, -2, 8)
POINTS.5	(0, -2, 12)
POINTS.6	(0.3, -2, 10)
POINTS.7	(-0.3, -2, 10)

Press **Dismiss** to close the window. The points should appear as shown in Figure 3.28 when viewed in the **Isometric** view:

Figure 3.28 Created Points



Arc Creation

Geometry > Create/Modify Curve > Arc through 3 points: Select (Arc Through 3 Points) to the window. Enter the Part CURVES and the name as CURVES.0. Select POINTS.0, POINTS.2 and POINTS.1. Press Apply to create the arc.

Switch on the **Curves** in the Display Tree window. To see the names of the curves, right mouse click on **Curves > Show Curve Names** in the Display Tree window. The newly created curve name would be displayed as CURVES.0.

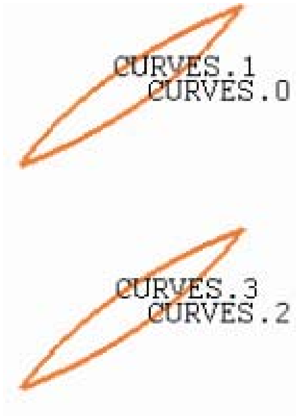
Similarly, create several more arcs using the following points. The curve names will automatically follow the first curve name to adjust to the names seen below:

CURVES.1: POINTS.0, POINTS.3 and POINTS.1
CURVES.2: POINTS.4, POINTS.6 and POINTS.5
CURVES.3: POINTS.4, POINTS.7 and POINTS.5

Press **Dismiss** to close the window. To reduce clutter on the screen switch off the **Points** from the Display Tree window.

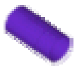
The geometry after arc creation creation is shown in Figure 3.29

Figure 3.29 Geometry After Arc Creation



Cylinder Creation

Geometry > Create/Modify Surface > Standard Shapes  **>**

Cylinder: Select  (Cylinder) to open the **Create Std Geometry** window as shown in Figure 3.30. Enter the Part name CYL and Name CYL.1. Enter a **Radius** of 2. Next to the Two axis Points, enter "{0 0 0} {0 0 20}". Press **Apply** to create the cylinder.

Press **Dismiss** to close the window.

Figure 3.30 Cylinder Creation

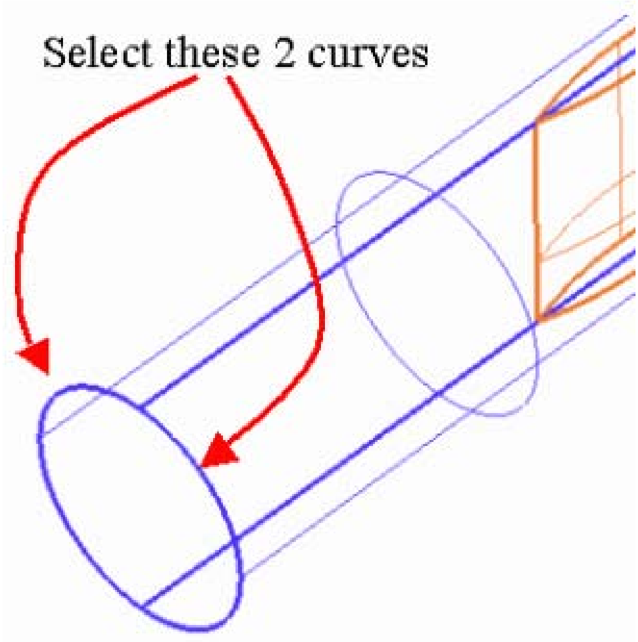
Surface Creation

Geometry >Create/Modify Surface > From curves: Select  (From Curves). Enter the Part name as BLADE and the Name as BLADE.1. Enter the **tolerance** as a number larger than the distance the surface must cross. Use 5 here. Select CURVES.0 and CURVES.2 with the left mouse button. Press the middle mouse button to complete selection, and press **Apply** to create the surface.

Similarly, create the other blade surface by selecting CURVES.1 and CURVES.3.

To create the OUTLET surface, enter OUTLET for the Part name and OUTLET.1 for the Name. Select the two curves shown in Figure 3.31:

Figure 3.31 Surface Creation

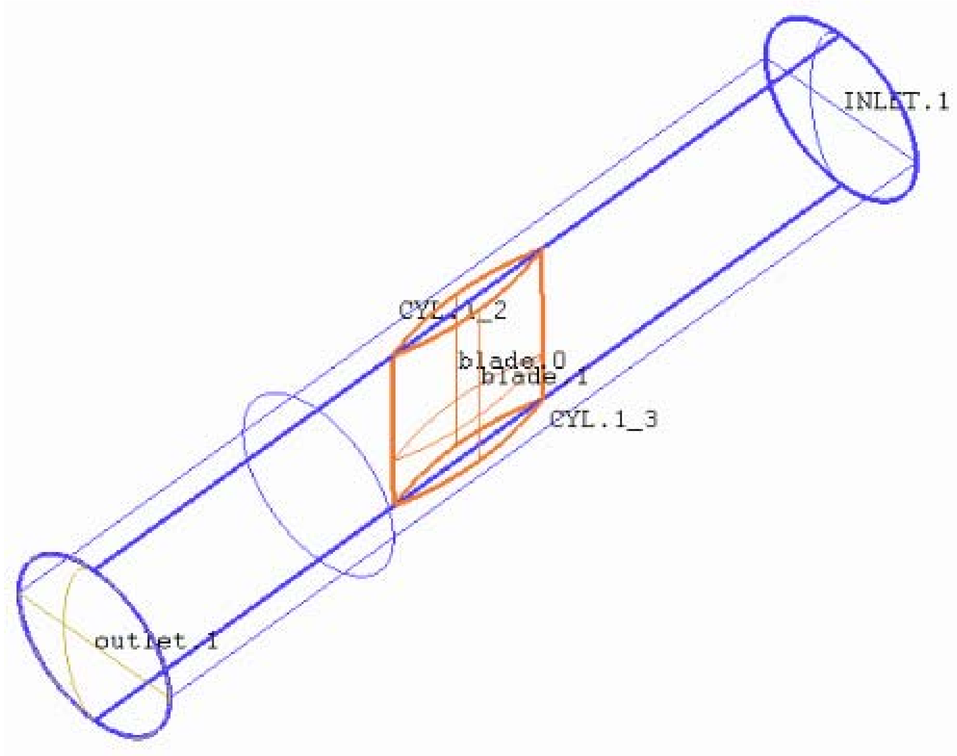


Create the INLET surface on the other side by selecting the two half circles on the other side. Assign the Part name INLET, and the Name INLET.1.

Switch ON the Surfaces in the Display Tree window. To see the names of the surfaces, use the right mouse button and select **Surface** > **Show Surface Names** in the Display Tree window. The geometry after surface creation is shown in Figure 3.32.

Press **Dismiss** to close the window.

Figure 3.32 Geometry After Surface Creation

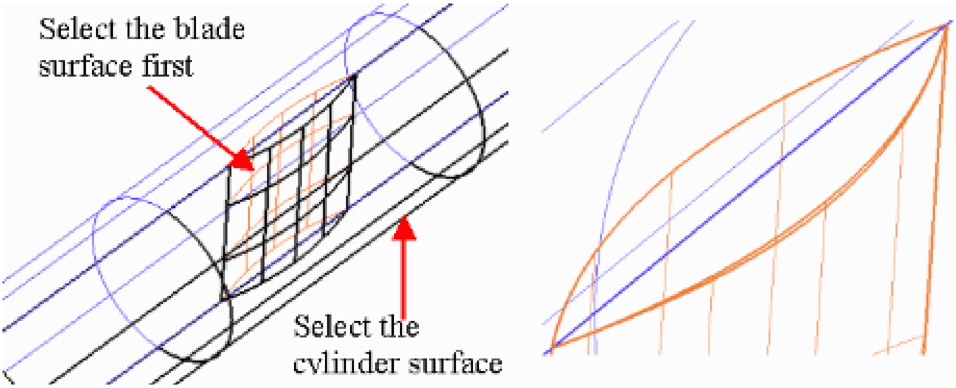


Surface-Surface Intersection

Geometry >Create/Modify Curve > Surfaces-Surface Intersection:

Select  (Surface-Surface Intersection), and choose the B-spline option. Select the two surfaces shown in Figure 3.33 Select the blade surface for Set1 Surfaces and the cylinder surface for Set2 Surfaces, pressing the middle mouse button each time. Press **Apply**. Repeat this for the other side of the blade.

Figure 3.33 First intersection curve



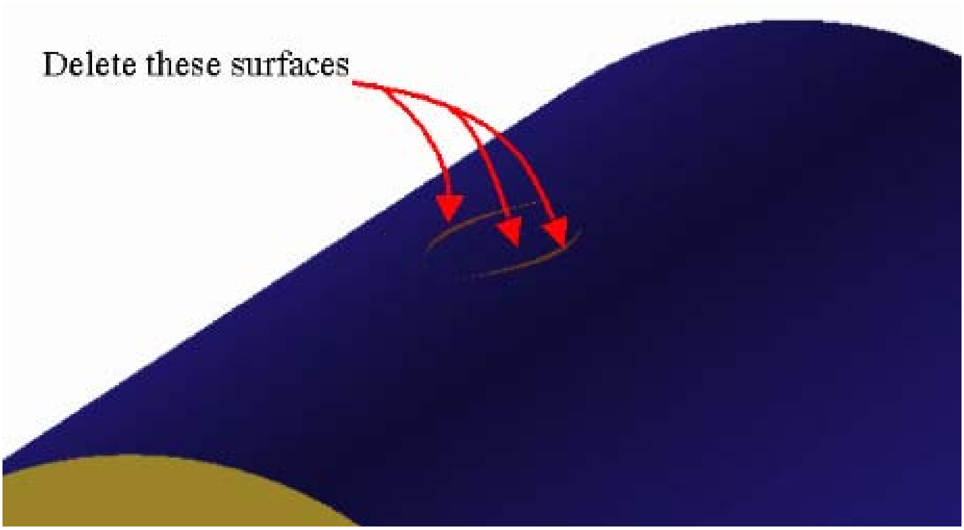
Build topology

Geometry > Repair Geometry **> Build Diagnostic Topology**: Select (Build Diagnostic Topology) from the geometry tab. This will extract all the curves from the surfaces, and the points from the curves, and delete any duplicates. It will also automatically segment the surfaces by the previously created intersection curves. Set the **tolerance** to 0.002, and **Filter points** and **Filter curves** should be turned off. Press Apply.

Deleting unused entities

Geometry > Delete Surface: Select (Delete Surface) icon. Delete the surfaces shown in Figure 3.34. Repeat this for the other side of the tube.

Figure 3.34 Surfaces to delete



Build topology

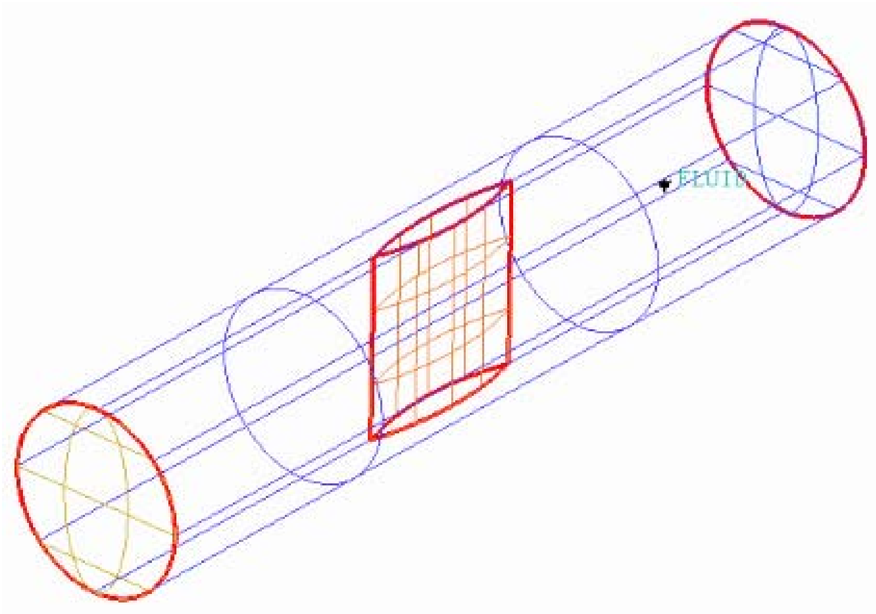
Geometry > Repair Geometry > Build Diagnostic Topology: Select (Build Diagnostic Topology) from the geometry tab. Build topology once more, but this time turn ON **Filter points** and **Filter curves.** Use a tolerance of 0.002.

c) Creating the material point

Geometry > Create Body > Material Point >Centroid of 2 points:

Select (**Create Body**) to open the window. Enter a new Part name of FLUID and select one location on the blade and one location on the INLET or OUTLET so that the midpoint will be inside the tube but outside the blade. Press the middle mouse button to complete the selection process. Press **Apply** to create the material point. The final geometry is as shown in Figure 3.35.

Figure 3.35 Final Geometry



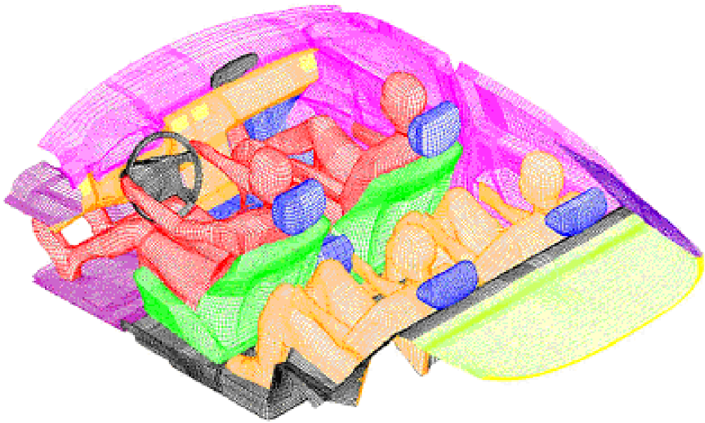
d) **Saving geometry**

File > Geometry >Save Geometry As: Enter the file name as **Geo_PipeBlade.tin** and press **Save** to save the geometry file.

3.2: Hexa Meshing

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

Figure 3.36 This mesh for the Mercedes SLK (model courtesy of Daimler-Chrysler) was generated with ICEM CFD Hexa combining the exterior and cabin flow



3.2.1: Introduction

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

Recognized as the fastest hexahedral mesh generation tool in the market, ANSYS ICEMCFD allows users to generate high-quality meshes for aerospace, automotive, computer and chemical industry applications in a fraction of the time required for traditional tools.

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

For symmetric models, 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 mesh.

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

a) Features of ANSYS ICEMCFD Blocking

O-grids: For complex geometry, ANSYS ICEMCFD Blocking automatically generates body-fitted internal and external O-grids for creating good quality meshes.

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

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

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

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

Symmetry: Can be used in analyzing rotating machinery applications. For example, Hexa allows the user to take advantage of symmetry in meshing a section of the rotating machinery thereby minimizing the model size.

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

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

2D Surface Meshing: Automatic 2D blocks creation for mapped surface meshing.

b) Mesh Generation with Blocking – Overall Process

First, create or import geometry using any of the direct, indirect or facetted data interfaces.

Interactively split blocks, discard unused blocks to capture underlying shape: "top down" approach else create blocks and extrude blocks: "bottom up" approach. Blocks are at first created "independently" of the geometry.

Associate edges to curves to capture hard features. Move vertices to position block corners on geometry.

Assign mesh sizes such as maximum element size, initial element height and expansion ratio to surfaces and/or curves. Assign edge meshing parameters for better control of node distributions.

Automatically generate mesh. Boundary nodes will project on to geometry, volumes are interpolated. Check mesh quality to ensure that specified mesh quality criteria are met.

Write Output files to the desired solvers.

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

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

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.

Afterwards, a block file can be used as a template for similar geometries, such as parametric design changes. Necessary alterations can either be done manually, automatically update projection or running a replay script; depending on the nature of the change.

c) The Blocking Database

The blocking database (block file) will have an extension of *.blk. It contains all the information necessary for defining and computing the block structured mesh including block definitions, part associations, and mesh size parameters. Block definitions include the following block topology types:

Vertices

Edges

Faces

Blocks

All of the block entities are defined by I, J, K index.

d) Unstructured and Multi-block Structured Meshes

The computed mesh stored internally within Blocking is termed “Pre-Mesh.” Pre-Mesh is then converted to either multi-block or unstructured files for eventual output to the solvers.

Unstructured Mesh Output

The unstructured mesh output option will produce a single mesh output file (*.uns) where all common nodes on the block interfaces are merged, independent of the number of blocks in the model. Unstructured elements are defined by node number definition.

Multi-Block Structured Mesh Output

The multi-block structured mesh output option will produce a mesh output file for every block in the topology model. For example, if the block model has 55 blocks, there will be 55 output files created in the output directory. Elements are defined by I, J, K indices rather than node numbers.

The number of blocks upon output can be reduced by an automatic internal merge of blocks (Output Blocks).

e) Main Blocking Functions

Here are some of the most often used functions within the Blocking menu:

Initialize block

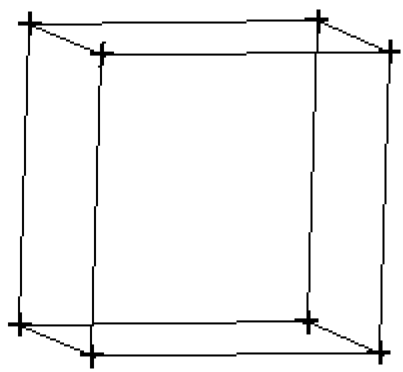
This is under Blocking > Create Block. First, a block is defined that encompasses the entire or selected portions of the geometry. This block is associated to a part (SOLID is the default part name). This volume part should be different than any part containing geometry. This initial block is then modified by splitting, discarding unused blocks, and creating O-grids.

Split
This option is under Blocking > Split Block. The most common way of "creating" blocks is to split existing blocks. The Split function, which divides the selected block interactively, may propagate across all visible blocks, selected blocks or selected faces. An edge is selected and the split (new) edges will propagate perpendicular to the selected edge. Blocks may be visually blanked/unblanked by using the Index control which toggles the blocks in I, J, K or radial (if o-grids exist) directions. Any new split will create a new I, J, K or radial (if splitting an o-grid) index.

O-grid Creation
Subdivides selected blocks into a configuration of one central block surrounded by radial blocks. Accessed through Blocking > Split Block > O-grid Block. Recommended for cylindrical type geometries to avoid bad internal angles at block corners.

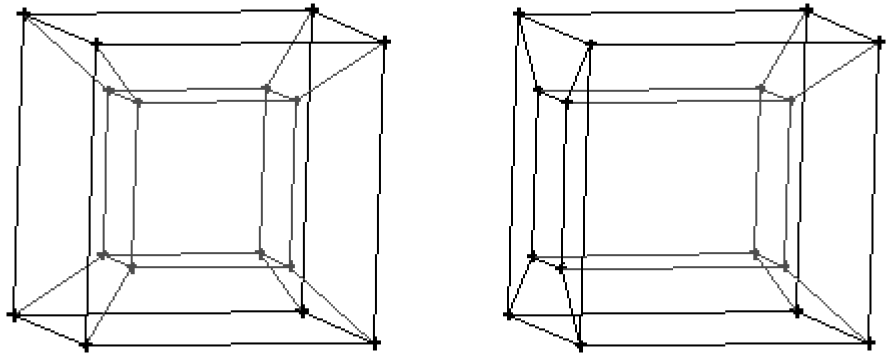
Figure 3.37
OGrid Block

The initial block



The initial block with an O grid

The initial block with an O grid to include a face



Adding faces will create an O-grid that "passes through" the selected block faces creating a "C-grid" configuration. (Figure 3.37) shows the result of adding a face on the left side of the block.

O-grids can be scaled before or after O-grid creation. The scale factor (Offset) is the ratio of the radial edge to the shortest edge of the initial block. The larger the factor, the longer the radial edge and the smaller the central block.

Associate

The next step is to associate block entities to geometric entities. Most of the time, this means associate edges to curves. This will make sure hard features are captured. Other options are to associate vertices to points to capture sharp corners and faces to surfaces if the default face projection, where nodes are projected to the nearest point in the normal direction to the nearest surface, fails to give proper results.

Move vertices

Vertices are typically moved on to the geometry. Computation will automatically move vertex nodes to the nearest point in the normal direction on the geometry. It's always best to manually position the node on to the geometry rather than leave it up to the default projection.

Color Coding

All vertices and edges are color coded depending on their constraint to the geometry. Vertex movement depends on this constraint:
White Edges and Vertices: These edges are either on the boundary or between two material volumes. The edge and the associated vertices will be projected to the closest CAD during pre-mesh computation. White vertices can only be moved on active surfaces.
Blue (Cyan) Edges and Vertices: Internal, between blocks of the same volume. Blue vertices can be moved by selecting the edge just before it and can be dragged along that edge direction.
Green Edges and Vertices: Associated to curves. The vertices can only be moved on the curves to which they have been projected.
Red Vertices: Vertices projected to prescribed points. They are fixed and cannot be moved unless projection type is changed.
All vertices can also be constrained by fixing x, y and or z coordinates. When thus constrained, the vertex movement ignores the above color coded geometric constraints.

Set Pre-Mesh Parameters

Mesh sizes (parameters) can be set globally, or on the surfaces, curves or parts. These operations constitute the first four icons in the Mesh menu. These sizes then have to be applied to the blocking: Pre-Mesh Params > Update Sizes. Selecting Pre-Mesh in the Display Tree will then prompt the user to (re)compute the pre-mesh.
The user may also fine-tune the node distributions within Pre-Mesh Params > Edge Params. The Number of nodes, initial and final node spacing, expansion ratios and mathematical meshing laws can be prescribed on individual edges. These distributions can be copied to opposing parallel edges down and upstream of the selected edge.

Pre-Mesh Quality

Before converting the pre-mesh to unstructured or multi-block, the quality should be checked. Blocking > Pre-Mesh Quality will create a histogram (bar graph) of element quality in the same manner as for Edit Mesh >

Quality described in Section 2.6 (Histogram Window). Different criteria such as determinant, angle and warpage can be checked and displayed.

Delete Blocks

One of the main functions in the "top down" approach. After splitting blocks, some may need to be discarded by Blocking > Delete Block. By default, these "deleted" blocks are actually moved to the VORFN part.

Vorfn Blocks

The VORFN part is a default part that is automatically created when blocking is first initialized. The initial block will actually consist of 27 blocks, a 3x3x3 arrangement in I, J, K index directions. Since VORFN is turned off by default, only the central block will be displayed and activated within the designated part.

If Delete permanently within Delete Blocks is turned on, selected blocks will be removed, not just moved to the VORFN. The VORFN blocks will then be reconfigured in a radial (o-grid) manner instead of the initial Cartesian arrangement.

Other Functions

Besides the main functions listed above, many other tools are available for building and fine-tuning the blocking topology:

Create blocks

Besides initializing, Create Block allows the user to build blocks by selecting existing vertices and/or screen locations. Blocks can also be built by extruding from existing block faces.

Besides regular hex blocks, degenerate (wedge) blocks, unstructured and swept blocks can be created. Swept blocks (3D) and unstructured blocks (2D) will allow you to have a different number of nodes across opposing edges.

Merge vertices

Vertices can be merged to create degenerate blocks. If "propagate merge" is turned on, all vertices up and downstream of those selected will also be merged, essentially removing the split.

Edit block
Various block editing commands including merge blocks, re-scale o-grids, etc.

Move Vertices
Besides manually moving vertices on the geometry, other options allow you to align vertices and to set coordinate locations of vertices.

Transform Blocks
Copy or move blocks either by translation, rotation, mirror or scale. Allows the user to build blocking on one portion of the model and copy and move to capture other portions that are topologically similar.

Edit Edge
Allows the user to "shape" the edges, either by manually splitting the edge or linking the edge shape with that of another edge. This gives the user better control of the flow of the mesh which can fix projection, skewness or other quality issues.

Pre-Mesh Smooth
Smoothing algorithms are available to automatically improve mesh quality before it is converted to either unstructured or multi-block mesh.

Block Checks
Check/fix is used to try and automatically fix the database if any serious errors arise. Also, left-handed (inverted) blocks can be automatically detected and fixed.

Visibility Controls

Most of the visibility controls, such as toggling objects on/off and right mouse clicking for display options, are discussed in the Introduction. The same applies for the Blocking tree and its sub-categories.
Once a blocking is initialized or a block file is loaded (File > Blocking > Open Blocking…) a new category, Blocking, is created in the model tree. Sub-categories within blocking are:
Subsets
Vertices
Edges
Faces
Blocks
Topology
Pre-Mesh
Edges are turned on by default. Most of the time, edges are the only type that needs to be displayed in order to perform the majority of the functions. A crosshairs representing the vertices (block corners) will also be displayed. Vertices are only necessary to display when certain information is desired.
Turning on Pre-Mesh will display the surface mesh. The user will be asked to compute the mesh if any changes have been made since the previous calculation.

Projection Options
One of the Pre-Mesh display options is projection type:
No Projection: Will simply interpolate all nodes without projecting to geometry. Useful if a quick mesh preview is desired, for example to visually check distribution patterns.
Project Vertices: Will project vertex nodes onto geometry. All other nodes are interpolated.
Project Edges: Will project all nodes along edges. All interior face nodes are interpolated. Required for final output of 2D planar grids.
Project Face: The default setting. Projects all boundary nodes, including those in the face interior. Only internal volumetric nodes are interpolated. Required for final output of 3D volumetric grids.

Scan Planes

Another display option within Pre-Mesh. With this function, the user can visualize the interior volume mesh by "scanning" or scrolling a logical (I, J, K) index plane through the model.

Note: The scan plane control displays I, J, K index dimensions as 0, 1, 2 respectively. O-grid index dimensions begin with 3. Additional o-grids will have an index of 4, 5, etc.

Blanking

This display option under the Blocking Display Tree simply blanks or turns off selected blocks to reduce screen clutter or if one wants to focus on a smaller set of blocks.

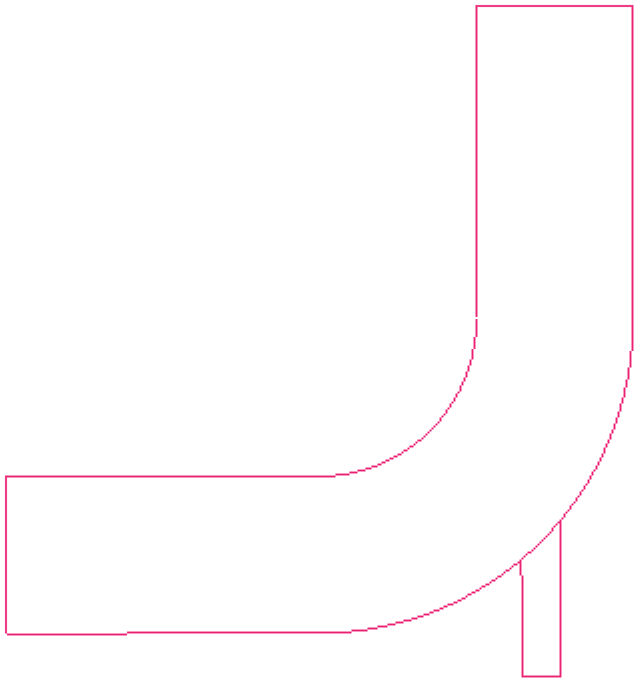
Index Control

As a display option within the Blocking Display tree, this turns blocks on and off by toggling up and down one or more of the I, J, K or radial (o-grid) indices. The index control menu will appear in the lower right hand corner of the screen in the same area where the quality histogram is displayed. If the histogram is turned on, it will take precedence and the index control will be displayed as a pop-up menu.

3.2.2: 2D Pipe Junction

Overview

In this first tutorial example, the user will generate a mesh for a two-dimensional pipe junction, composed of two Inlets and one Outlet. After generating an initial mesh, the user will check the quality of the Mesh, and refine it for a **Navier-Stokes** solution.



a) Summary of Steps

The Blocking Strategy

Starting the Project

Splitting the Blocking Material

Creating Composite Curves

Projecting the Edges to curves

Moving the Vertices

Generating the Mesh

Refining the Mesh with Edge Meshing

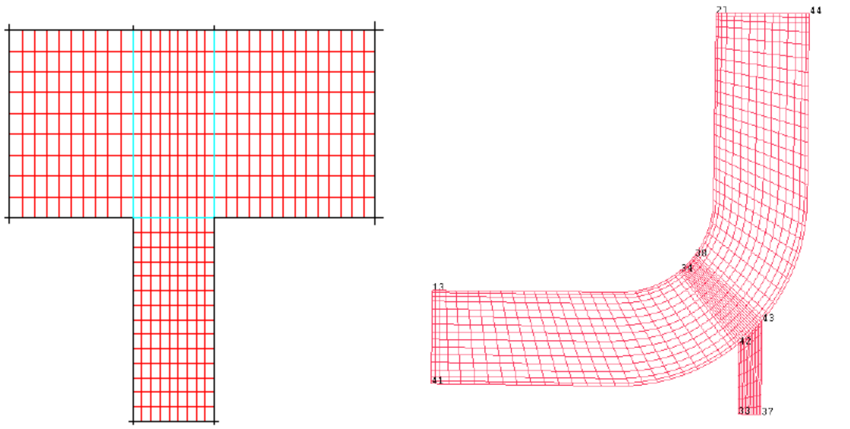
Saving the Blocking and Mesh before Quitting

b) The Blocking Strategy

The first step in generating a Mesh with Blocking is to decide on a blocking strategy.

Note: The geometry is equivalent to a "**T**" (Figure 3.38). The right side of the blocking crossbar needs only to be bent upward to resemble the geometry.

Figure 3.38 The mesh and its topology



Fitting the "T"-shaped Blocking Material to the geometry is accomplished by creating Associations between the Edges of the Blocks and the Curves in the geometry, and then moving the Vertices of the Blocks onto the corners of the geometry.

Once this is done, mesh sizes are set and the mesh is computed. The program will automatically project the edge nodes onto the curves of the geometry and the internal 2D volume mesh will be interpolated.

c) Starting the Project

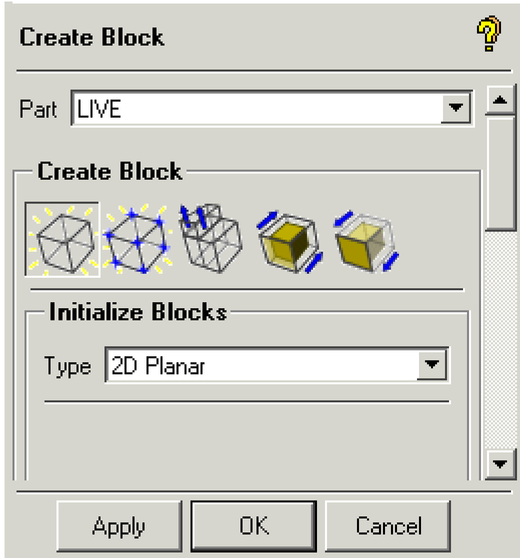
From a UNIX or DOS window, start ANSYS ICEMCFD. Select File > Change working directory and browse to the $ICEM_ACN/../docu/CFDHelp/CFD_Tutorial_Files>2DPipeJunction directory. Select File > Geometry > Open and select the tetin file, geometry.tin.

For this tutorial, the geometry and part information has already been pre-defined for the user.

Turn on Curves in the Display tree so that the geometry of the pipe is visible.

Initialize the 2D blocking: Select Blocking > Create Block  > Initialize Blocks  and change the type to 2D Planar as shown in Figure 3.39. Enter LIVE in the Part field and Apply.

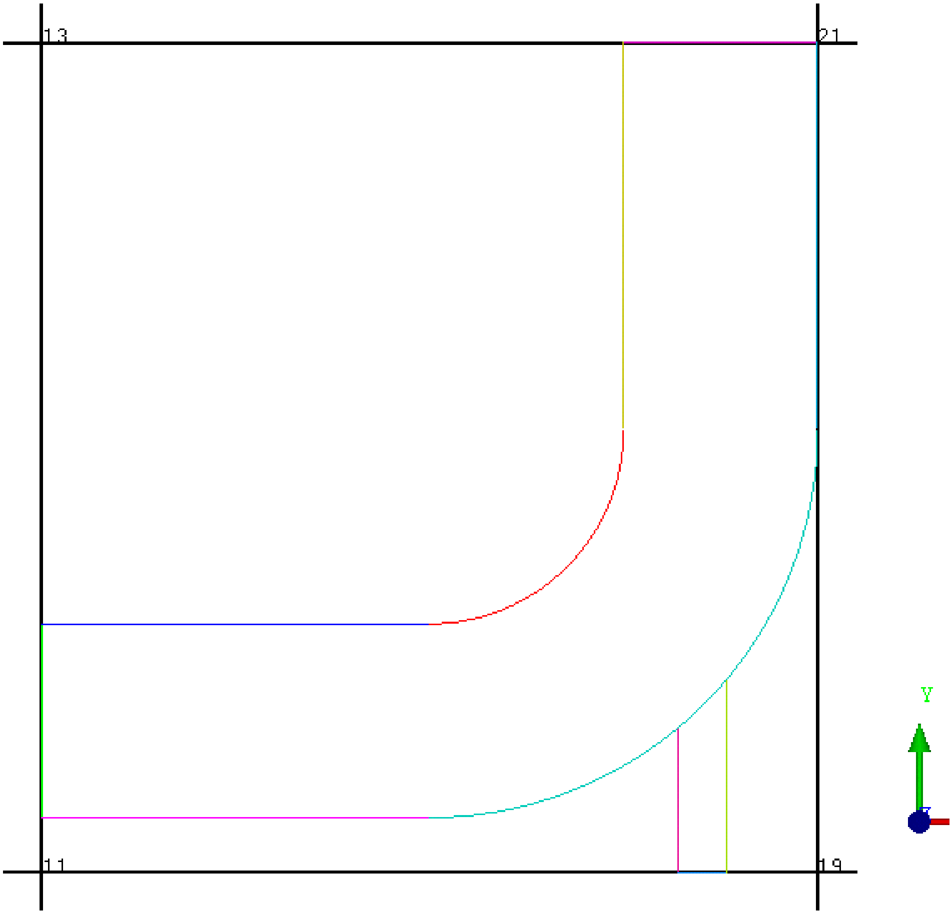
Figure 3.39
The Create Block Menu



Note the white block that encloses the geometry, as shown in Figure 3.40. This is the initial block that will be used to create the topology of the model.

Also note that the curves are now colored separately instead of by Part. This is so that the individual curve entities can be distinguished from each other, which is necessary for some of the blocking operations. This color coding can be turned on/off by right mouse selected Curves in the model tree and toggling Show Composite.

Figure 3.40 Initial LIVE block



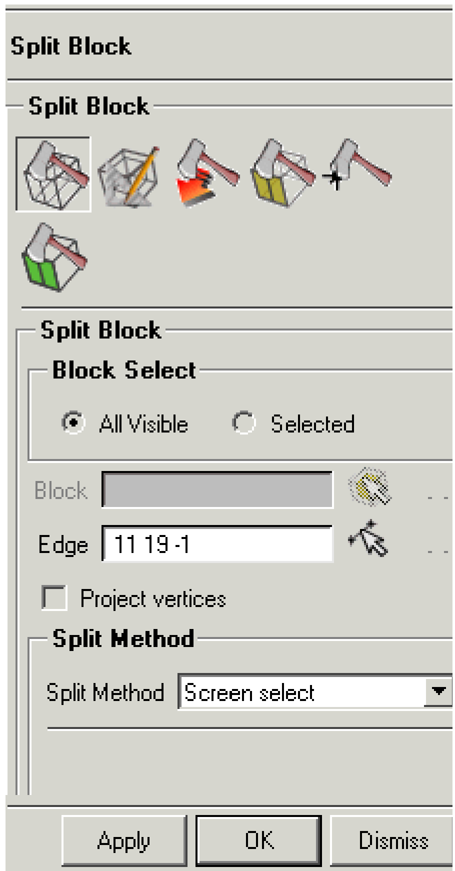
Turn on Vertices in the model tree. Then, right mouse select Vertices > Numbers. The following operations will refer to these numbers.

d) Block Splitting

First, two vertical splits and then one horizontal split will be made.

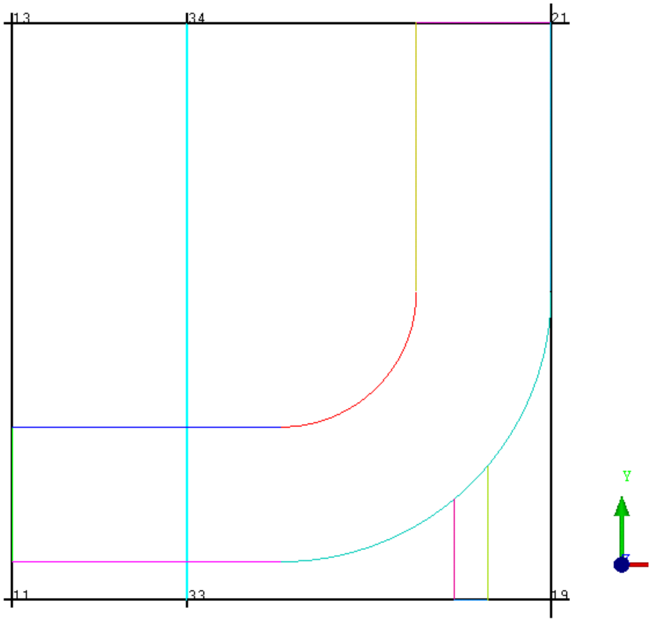
Select Blocking > Split Block  > Split Block  .Note the Split Method is set to Screen Select by default as in (Figure 3.41). In this case, the split may be done by approximation, as it is only the topology of the "T" that is essential, not the exact proportion.

Figure 3.41
The Split Window



In the Data Entry Panel, select Split Block  once again or the Select Edge  icon. You will be prompted to select an edge (note red text at the bottom of the view screen). With the left mouse key, select the edge defined by vertices 11 and 19 (or 13, 21) as shown in Figure 3.42.Keeping the left mouse key depressed, slide the new edge to the desired location and middle mouse key to perform the operation. The split is shown in Figure 3.42

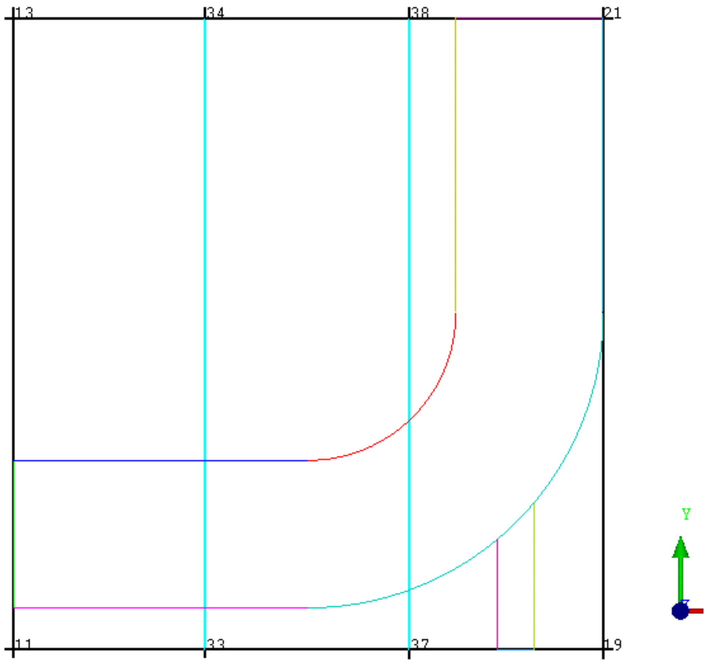
Figure 3.42
First Split Edge 11-19



Note: Pressing the right mouse button while in selection mode will cancel the previous selection. Also, note the color of the edge: blue (cyan) designates an internal edge.

Repeat for edge 33 & 19 (or 34, 21). The results are shown in Figure 3.43

Figure 3.43 Second Split Edge 33-19



Create the horizontal split, this time changing Split Method to Relative as in Figure 3.44.Enter 0.5 (mid-point of selected edge), select any one of the four vertical edges and press the middle mouse button or Apply.

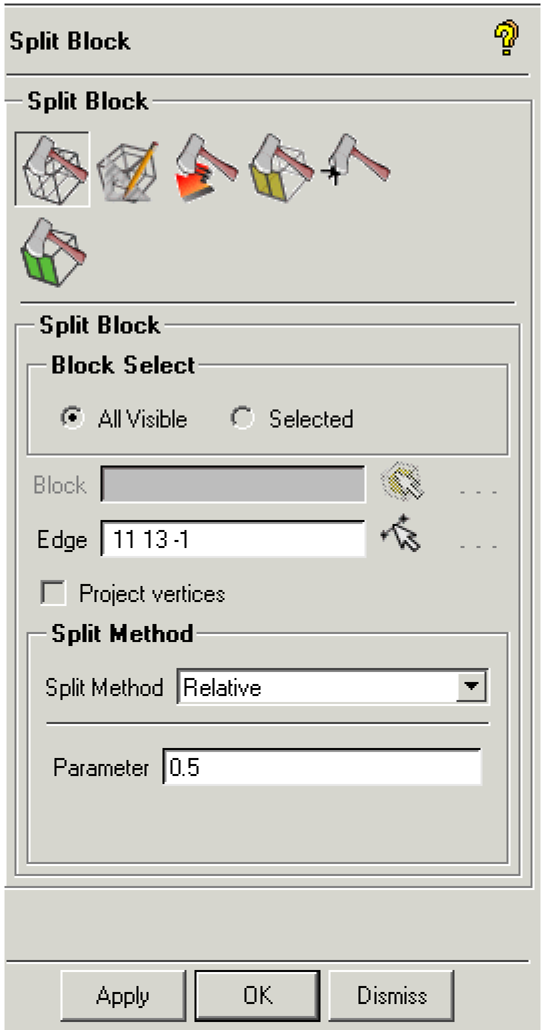
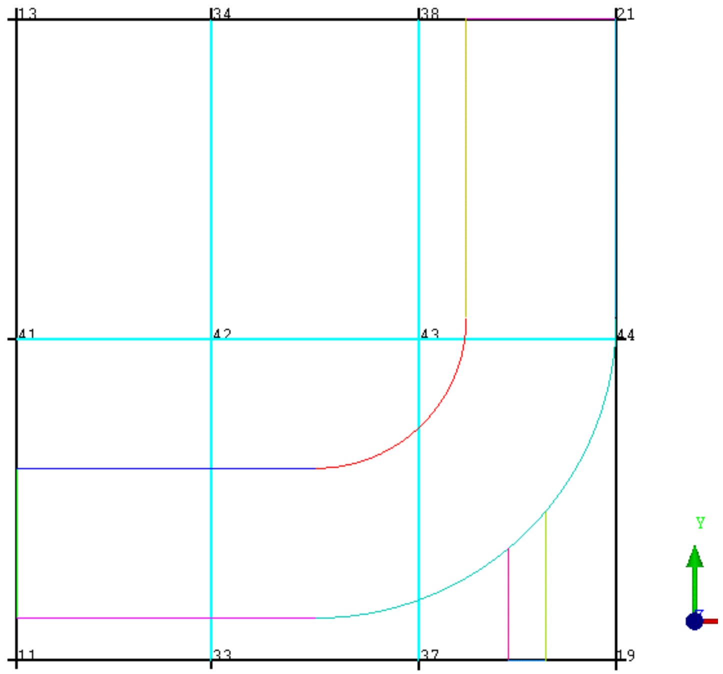


Figure 3.44
Split Method Relative

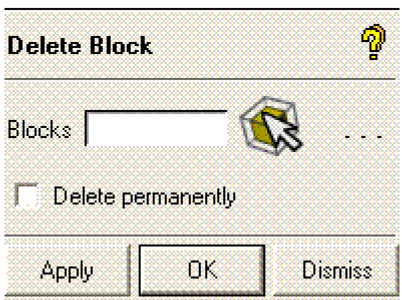
This horizontal split is shown in Figure 3.45

Figure 3.45 Display of the curves and LIVE block after making three splits



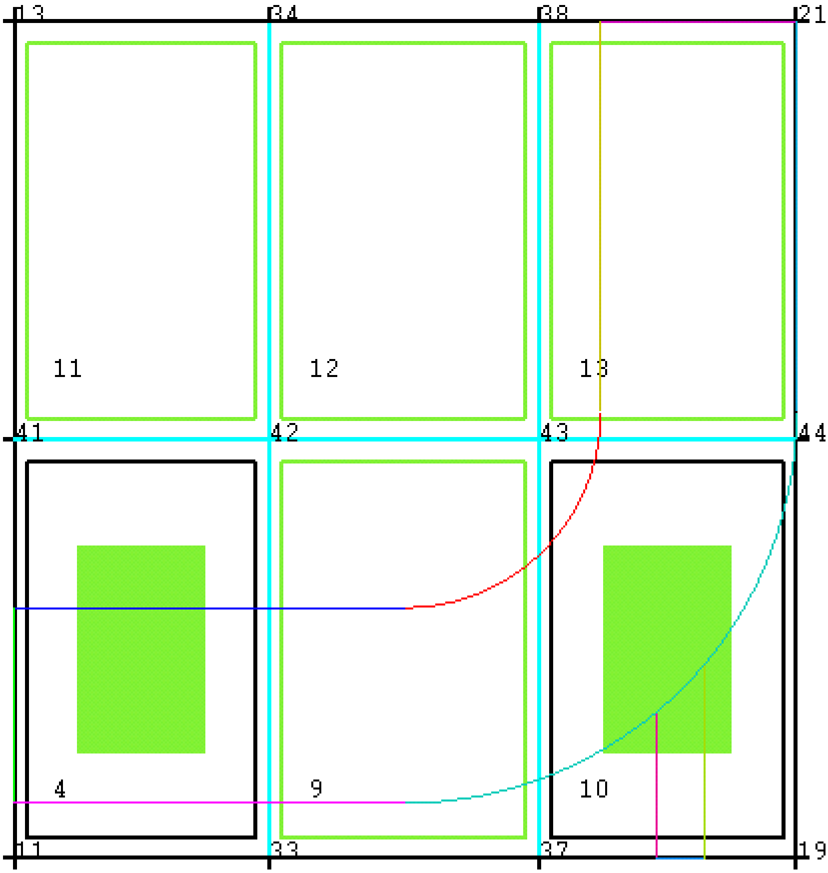
e) Discard Blocks

The next step in this "top down" approach is to remove or discard the unneeded blocks. Select Blocking > Delete Blocks .



Select blocks as shown in Figure 3.46and press the middle mouse button or Apply.

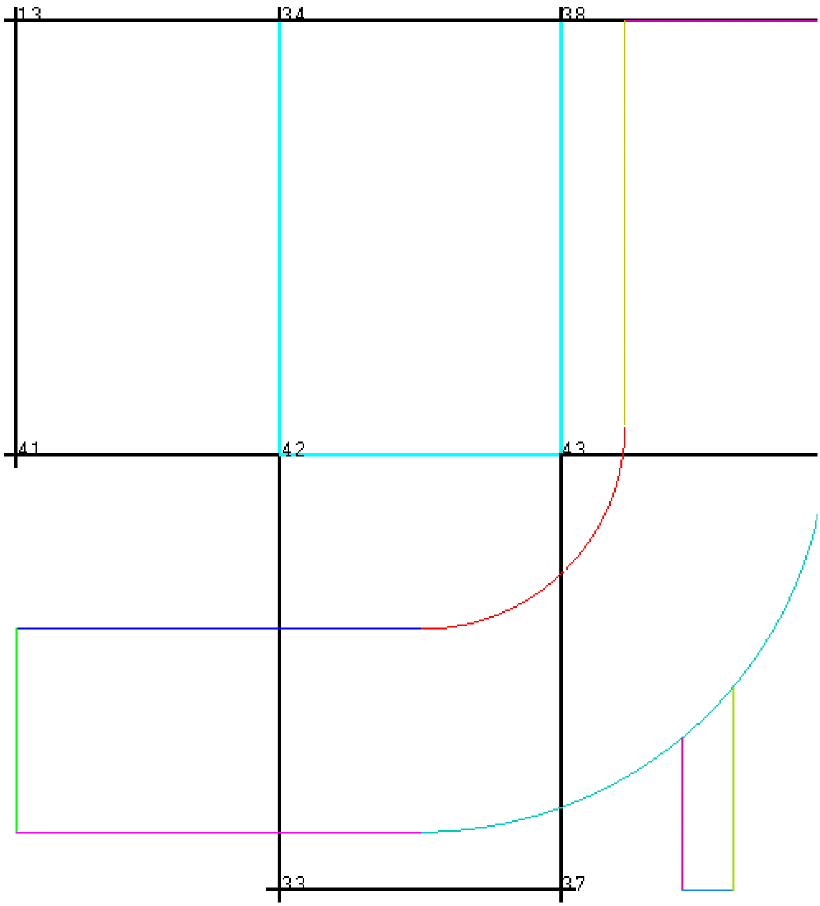
Figure 3.46 Select blocks to delete



Note: Deleted blocks with Delete Permanent turned off (default) are actually put into the VORFN part, a default dead zone that is usually deactivated.

The geometry and blocking of the model should now resemble that shown in Figure 3.47.

Figure 3.47 Final T Shape Topology

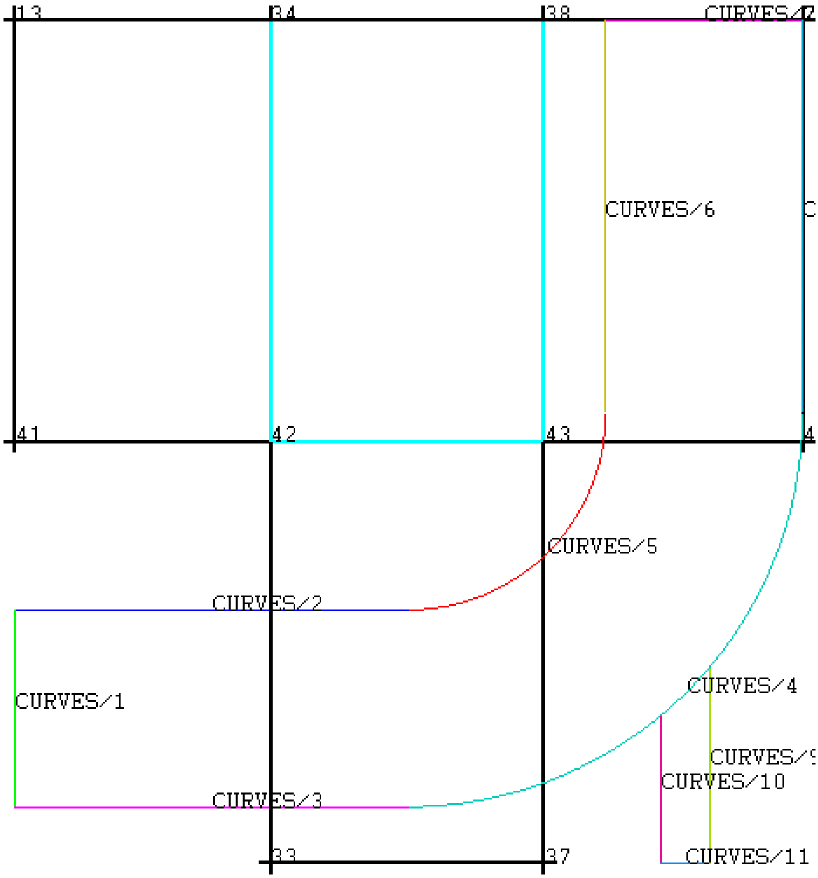


f) Associating to Geometry

The edges of the blocking will now be associated to the curves of the CAD geometry. First select the edges, then the curves to which you want to associate the edges. If two or more curves are selected per operation, those curves will automatically be grouped (concatenated).

For reference turn on Vertices > Numbers (right mouse option) and Curves > Show Curve Names in the Display tree.

Figure 3.48 Vertex numbers and Curve names



Select Blocking > Associate  > Associate Edge to Curve  as shown in Figure 3.49.

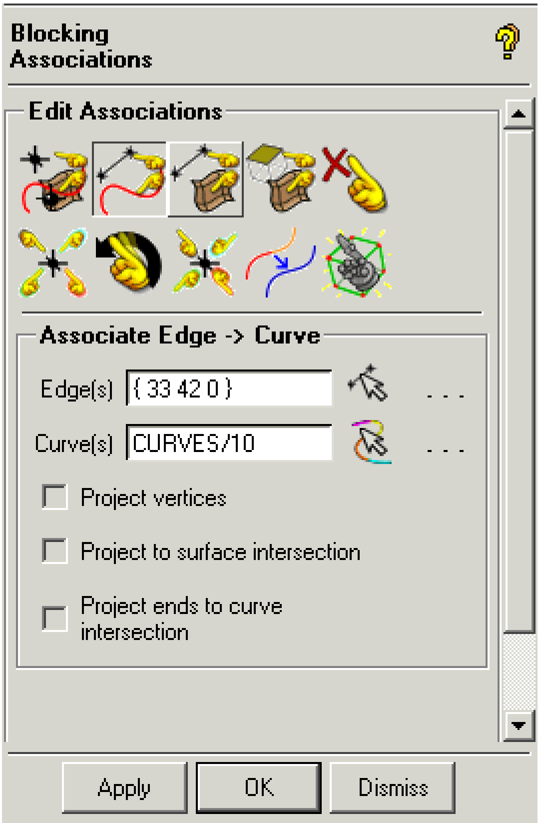


Figure 3.49 Blocking Association window

Note: Project Vertices should be disabled (default).

First capture the "inlet," the leftmost end of the large pipe. Select Associate Edge to Curve  once again or the

Select Edge  icon, and select Edge 13-41 with the left mouse button. Press the middle mouse button to accept the selection.

Then select the curve, CURVES/1 with the left mouse button and press the middle mouse button or select Apply to perform the association. The edge will turn green when associated.

Note: This operation runs in “continuation mode”, which allows the user to select the next set of edges and curves without reinvoking the function. Selecting the middle mouse button when no entities are selected or selecting Dismiss will cancel the function.

In a similar manner, associate the following edge/curve combinations to make the “T” fit the geometry:

Small pipe: Edge 33-42 to curve CURVES/10; 33-37 to CURVES/11; 37-43 to CURVES/9.

Outlet (top horizontal end of large pipe): Edge 21-44 to curve CURVES/7. This vertical edge will eventually be moved to capture the horizontal curve.

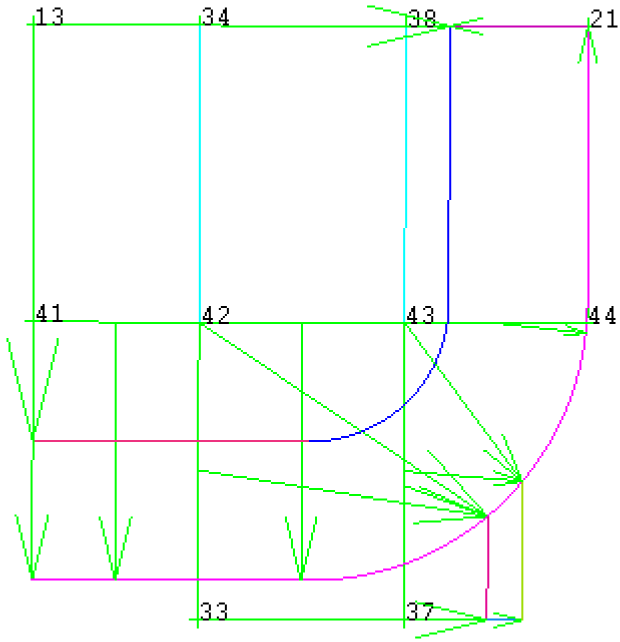
Note: It may help to toggle entity types off and back on to identity the right entity, if they overlap other entities. For example, turn off Vertices and Edges to verify the curve names. Turn Edges back on to proceed with the selection.

Sides of large pipe: Edges 13-34, 34-38, 38-21 to curves CURVES/2, /5 and /6. Select all three edges first, press the middle mouse button to confirm, then select the three curves, and press the middle mouse button again. The three curves will automatically be grouped as one logical composite entity. Geometrically, they are still three separate curves. Edges 41-42, 43-44 to curves CURVES/3, /4, /8.

The blue (cyan) edges (42-43, 34-42, 38-43) do not have to be associated. They are internal and will interpolate instead of project on to geometry when the mesh is computed.

The associations may be verified by selecting Edges >Show Association in the Display tree. As in Figure 3.50, the green arrows in the display point from an edge to its associated curve. Nodes and vertices of these edges will project on to the associated geometry.

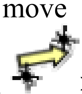
Figure 3.50 Projection of edges to Curve



Note: If, once completed, the associations do not appear as in Figure 3.50, the steps of operation may be retraced with the Undo and Redo buttons. Also edges can be re-associated to their proper curves. It is not necessary to disassociate and then re-associate. The re-association will overwrite the previous association.

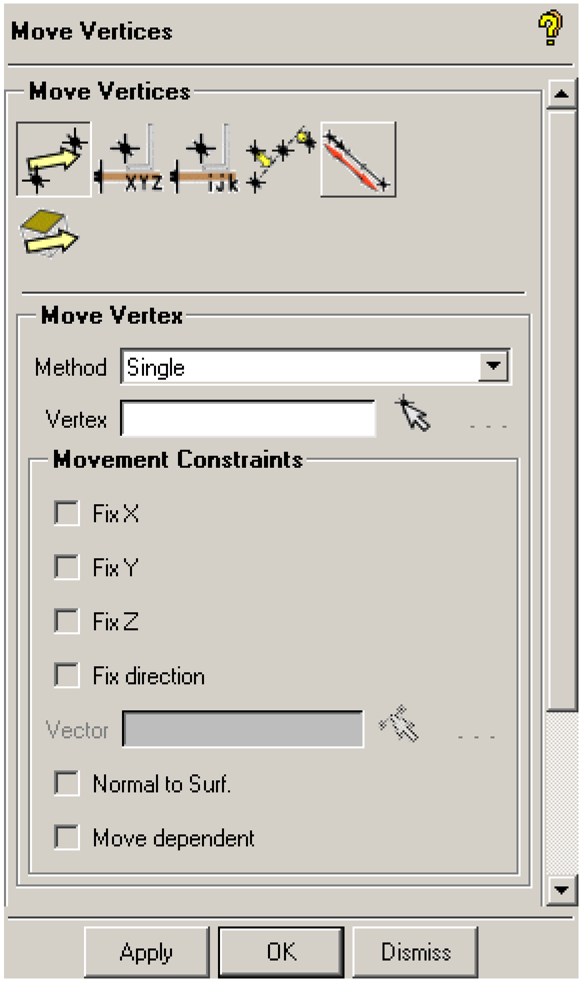
Turn off Edges > Show Association after verifying.

g) Moving the Vertices

Next, move vertices on to the geometry. Select Move Vertex  > Move Vertex  (if not already selected) as shown in Figure 3.51.

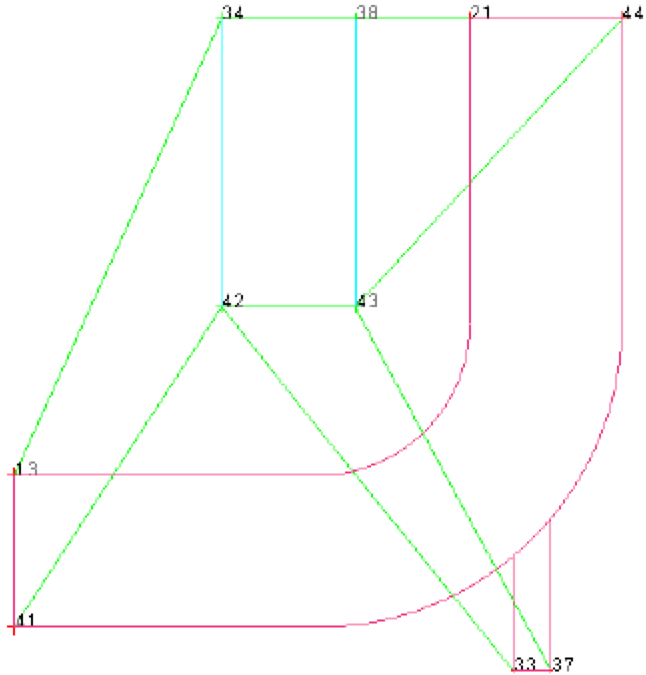
Note: Selecting Move Vertex from the Menu tab will immediately prompt you to select from the screen. It is usually not necessary to select Move Vertex from the **Data Entry Panel** unless another option was previously selected.

Figure 3.51
Move Vertex Window



Move the vertices of the Inlets and Outlet (ends of large pipe) as shown in. Keeping the left mouse key depressed, one can “drag” the vertex along the curve.

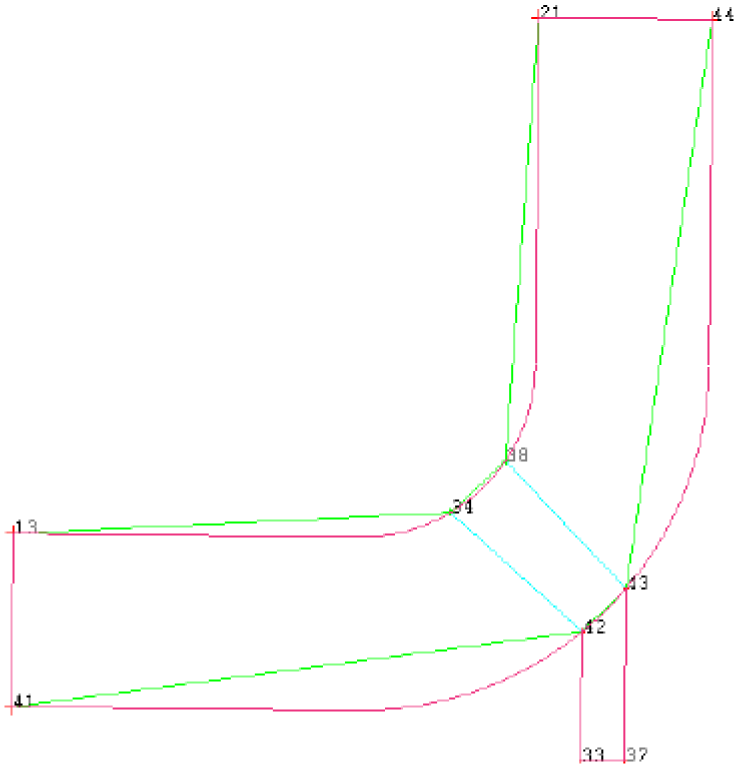
Figure 3.52 Associate Edges to Curve



Note: Due to the associations made between the edges and curves, many of these vertices will “snap” to the correct position. Vertices may, however, be moved along the curve by dragging the mouse. To capture the ends of the curves, drag with the left mouse key depressed until the vertex can be moved no further: position the cursor beyond the end of the curve will assure that the end is captured.

Move the remaining vertices to their appropriate positions on the geometry until the blocking resembles Figure 3.53 Try to make the blocks as orthogonal (good internal angles) as possible.

Figure 3.53 Move the rest of the vertices to their position



When finished, complete the operation by selecting the middle mouse button or Dismiss to exit the Move Vertices window. Right mouse key will undo the previous vertex movement, NOT exit the function.

Save the current work to a file by choosing File > Blocking > Save Blocking As. Provide a filename (such as blk1) so that the file may be reloaded at a later time, using File > Blocking > Load blocking.

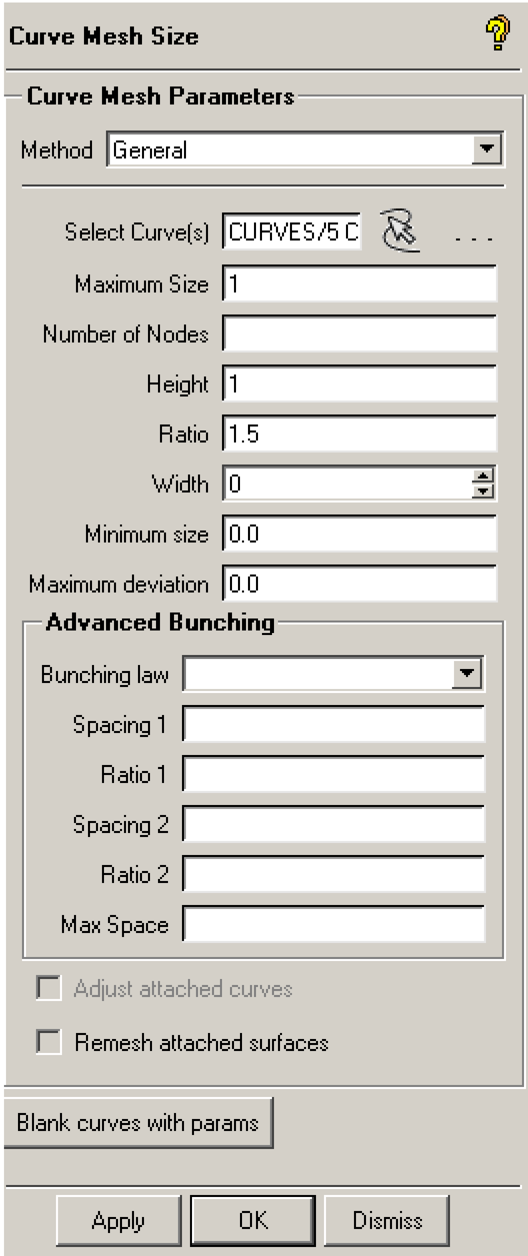
h) Generating the Mesh

First, Mesh parameters (sizes) must be set on the geometry (curves in this 2D case).

Select Mesh > Set Curve Mesh size to invoke the window seen in Figure 3.54 Keep Method as General. Set Maximum Size to 1. Ignore all other parameters.

Select the visible Curves (can select “v” for visible or “a” for all or select the appropriate icons from the selection tool bar) and Apply.

Figure 3.54
Curve Mesh Parameter Window

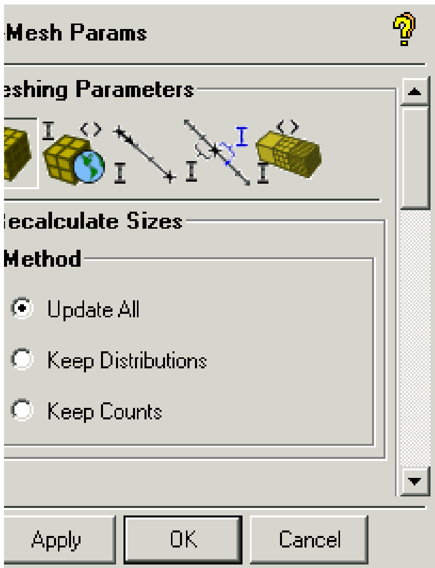


Note: Maximum Size determines the length of the edges on the curve (or surface for 3D). Height determines the length of the edge of the first layer normal to the curve. Ratio determines the normal heights of the subsequent layers. In this case, height and ratio are determined by the perpendicular curves whose Maximum Size will override any height or ratio settings.

Initial Mesh Generation:

Select Blocking > Pre-mesh Params  > Update Sizes  as shown in Figure 3.55.

Figure 3.55
Pre Mesh Param Window



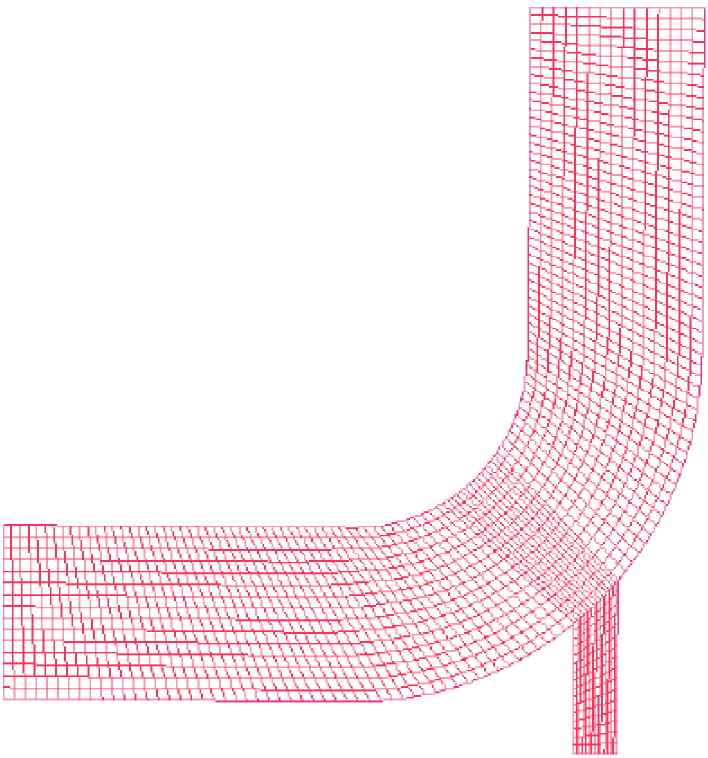
Toggle on Update All and Apply.

Note: This will automatically determine the number of nodes on the edges from the mesh sizes set on the curves.

Turn on Blocking > Pre-Mesh in the Display tree. Select Yes when prompted to recompute.

Switch off Edges and Vertices from the Display tree to view the mesh as in Figure 3.56.

Figure 3.56 The initial mesh



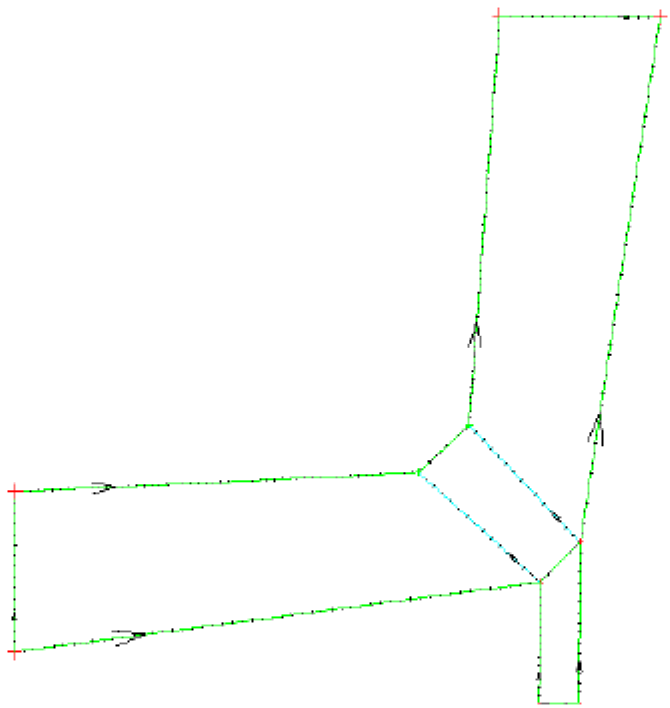
Refining the Mesh with Edge Meshing

Now the user will employ advanced edge meshing features to re-distribute grid points to resolve the salient features of the flow.

Turn off Pre-Mesh in the Display tree, and re-display Curves and Edges. Right mouse select Edges and select Bunching from the pull down options to see the distribution of grid points along the edges (Figure 3.57).

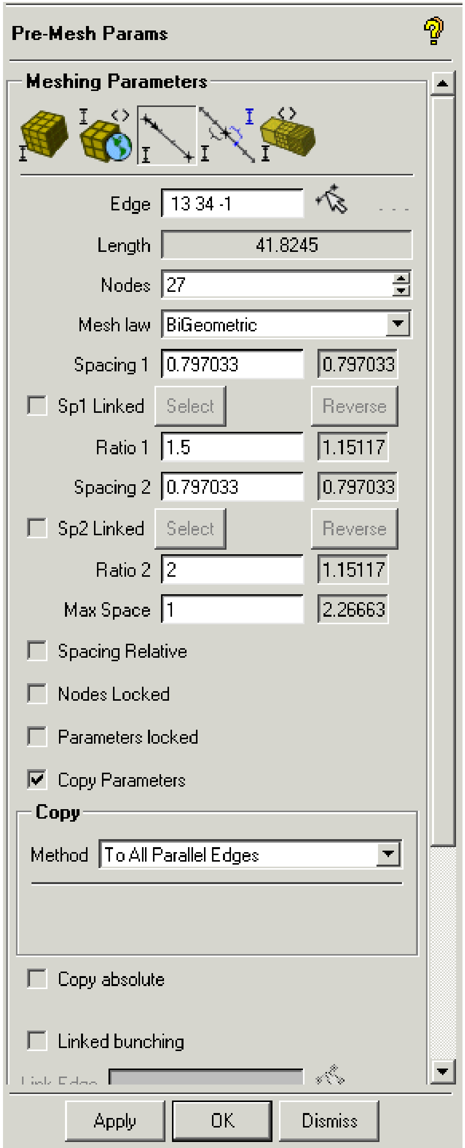
First, we'll reduce the number of nodes along the length of the large pipe.

Figure 3.57 The bunching on the edges



Turn on Vertices > Numbers (right mouse) in the Display tree again for reference. Select Blocking > Pre-mesh Params  > Edge params  to display the Edge meshing parameters window as shown in Figure 3.58 Select  again or  and select edge 13-34 when prompted. In the panel, change the number of Nodes to 27 then Apply.

Figure 3.58 Edges Parameter Window



Similarly, re-select  or , select edge 21-38, change Nodes to 27 and Apply.

Toggle on Pre-Mesh and recompute to view the new mesh.

Note: This is a structured grid. When the number of nodes is changed on one edge, all parallel opposing edges will automatically have the same number of nodes. In this case, edges 41-42 and 43-44 will have the same number of nodes as 13-41 and 21-38 respectively.

Next, we'll bias the nodes closer to the wall boundaries of the large pipe.

Still within the Edge Parameters menu, select edge 13-41, and change the Spacing 1 and Spacing 2 to 0.5. Change Ratio 1 and 2 to 1.2 and Apply.

Note: Spacing 1 refers to the node spacing at the beginning of the edge, and Spacing 2 refers to the spacing at the end of the edge. The beginning of the edge is shown by the white arrow after the edge is selected.

Requested values for spacing and ratio are typed in the first column. Actual values are displayed in the second column. Note that due to the number of Nodes, the Mesh Law and Spacing, the requested Ratios cannot be attained. Increase the number of Nodes using the arrow toggles until the Ratios are close to the requested value, 1.2.

Note: The Mesh Law is by default set to BiGeometric. This allows the nodes to be biased towards both ends of the edge. The expansion rate from the end is a linear progression. Several other mathematical progression functions (laws) are available.

Toggle on Copy Parameters. Set Copy >Method > To All Parallel Edges (default) and press Apply. This will ensure that the parallel Edges 34-42, 38-43, and 21-44 have the same spacing.

Next, select Edge 21-38 and change Spacing 1 and 2 to 0.5.

This will concentrate grid points toward the outlet and toward the small pipe. To have these changes reflected in edge 43-44 as well, be sure that Copy Parameters > Copy > Method > To All Parallel Edges is selected Apply.

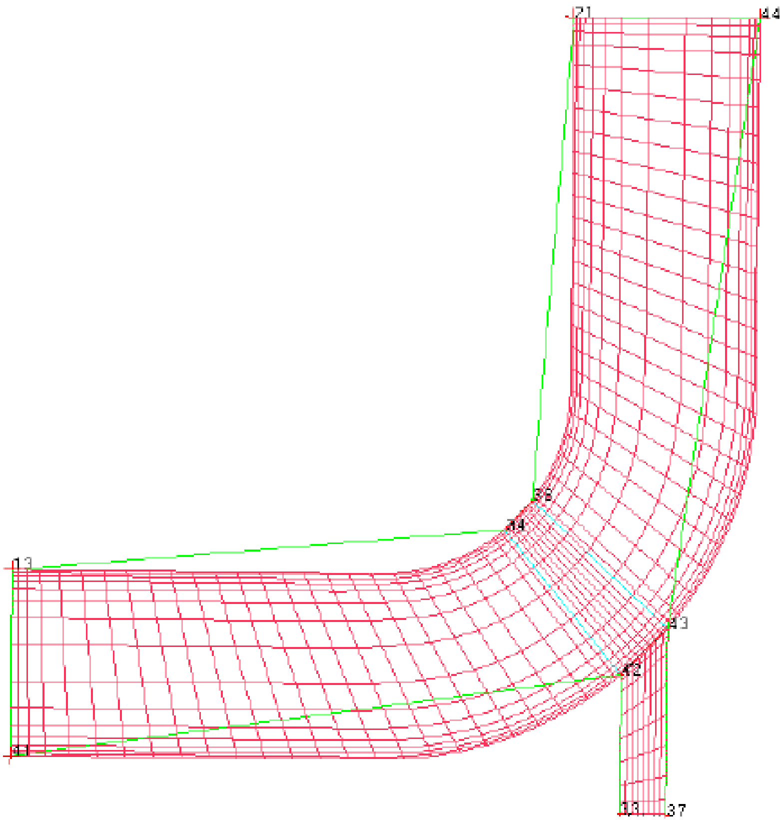
Next, we'll copy the same distribution to the other section of the large pipe. Still in the Edge Parameters menu, change Copy Parameters > Copy > Method > To Selected Edges Reversed. Select the Select edge(s)  icon immediately underneath the Method field and select Edge 13-34. Press the middle mouse button or Apply.

Refine the nodes along the small pipe. Select Edge 33-42 (make sure to select the icon toward the top of the menu, not the one beneath the Method field), change Nodes to 9, Spacing 1 to 1.0, and Spacing 2 to 0.5. Change Copy Parameters > Copy > Method back to To All Parallel Edges and Apply.

Change the number of Nodes of edge 34-38 to 9.

Toggle off/on Pre-mesh and recompute to view the refined mesh shown in Figure 3.59.

Figure 3.59 The Final Refined Mesh



i) Saving the Mesh and Blocking

Save the mesh in unstructured format: Right mouse select Pre-Mesh and select Convert to Unstruct Mesh to generate the domain file.

Select File > Blocking > Save Blocking As and input a filename for the blocking, after the project name. This block file can be loaded in a future session (File > Blocking > Open Blocking…) for additional modification or to mesh a similar geometry.

It is recommended to save each blocking to a separate file instead of overwriting a previous one. In more complex models, the user may have to back track and load a previous blocking.

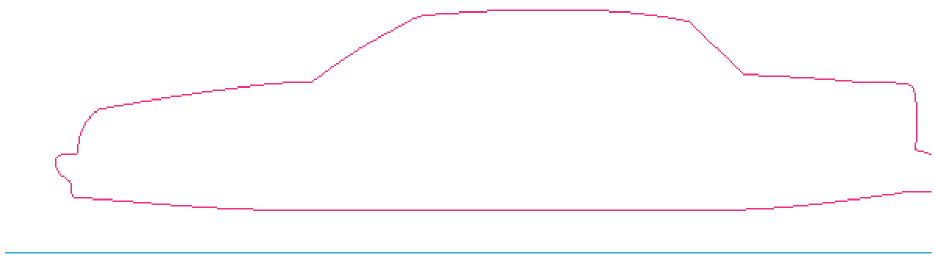
Select File > Save Project As… and type in a project name. All files: tetin, blocking and unstructured mesh will be saved.

File > Exit or continue with the next tutorial.

3.2.3: 2D Car

Overview

In this tutorial, the user will generate a Mesh for external flow over a simple 2D Car residing in a wind tunnel. The Replay will be employed for recording all the blocking steps This replay (script) file will be run to model a modified geometry.



a) Summary of Steps

The Blocking Strategy

Starting the Project

Splitting the Blocks with Prescribed Points

Splitting Blocks using the Index Control

Reassigning the Material Domains

Body Fitting the Blocking

Aligning the Vertices

Meshing with Curve Parameters

Creating an O-grid around the Car

Meshing with Edge Parameters

Saving your Replay File and Quitting Hexa

Using Replay for the Design Iteration

b) The Blocking Strategy

For an external flow model in a wind tunnel, the following steps are usually taken when blocking the model to obtain the desired results.

The Split function is a common technique when beginning blocking by carving a Cartesian set of blocks around the object.

The vertices are then moved onto the geometry in order to fit the shape of the car with all its features: front bumper, hood, etc.

An O-grid block is created around the car to give an orthogonal grid.

The following Parts that have been defined in the geometry (Figure 3.60):

CAR: Vehicle geometry

GROUND: Ground surface of the wind tunnel

INLET: Inlet face of the wind tunnel

OUTLET: Outlet face of the wind tunnel

PNTS: Prescribed points associated with the Car.

TOP: Top surface of the wind tunnel.

Figure 3.60 The Parts of the 2D Car

A modification to this Geometry called "car_mod.tin" is also available in the Project directory. Use the Replay file for this geometry. The parts are the same in both the base and modified geometries, allowing the "Replay file" to be run on each identically.

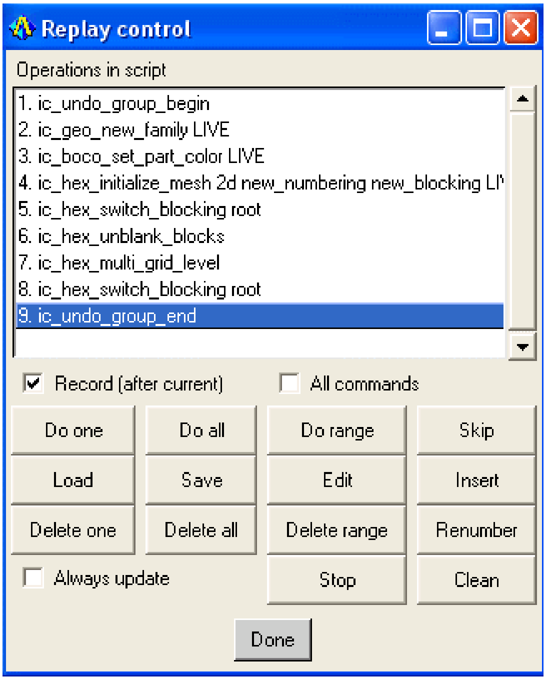
c) Starting the Project

After opening, select File > Change working directory and browse to `$ICEM_ACN/../../docu/CFDHelp/CFD_Tutorial_Files/2Dcar`. Select File > Geometry > Open and load `car_base.tin`.

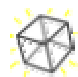
Before proceeding, note that the names of Parts that are listed are located in the Display Tree. As in the previous tutorial, the geometry and Parts have already been defined for the user.

Start the Replay File. The Replay function allows the user to record all the steps necessary to complete the mesh. Select File > Replay Scripts > Replay Control to bring up the Replay control window (Figure 3.61).

Figure 3.61 The Replay control Window

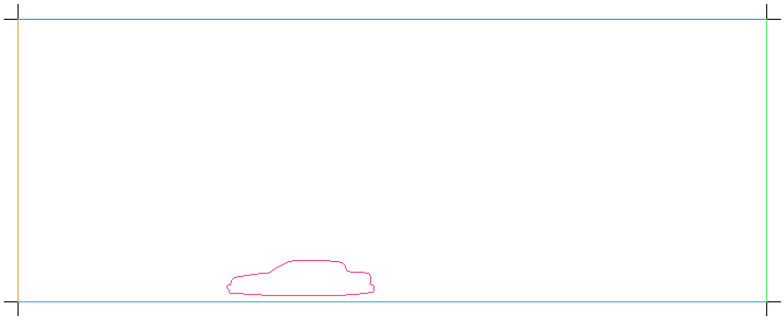


Note: The option of loading is not recorded in the replay script. Record (after current) is turned on by default. It will record all of the commands until this button is turned off or the user selects Done. The Replay control window may be moved aside or minimized while recording, but the window should be kept active until recording is complete.

Select Blocking > Create Block  > Initialize Block  > Type 2D planar.

Name the Part as LIVE and press Apply.

Figure 3.62 The Initialized Blocks



d) Splitting the Blocks with Prescribed Points

Make sure Curves are turned on (default) in the Display tree. Edges should also be displayed (default) showing the initial block as in Figure 3.62.

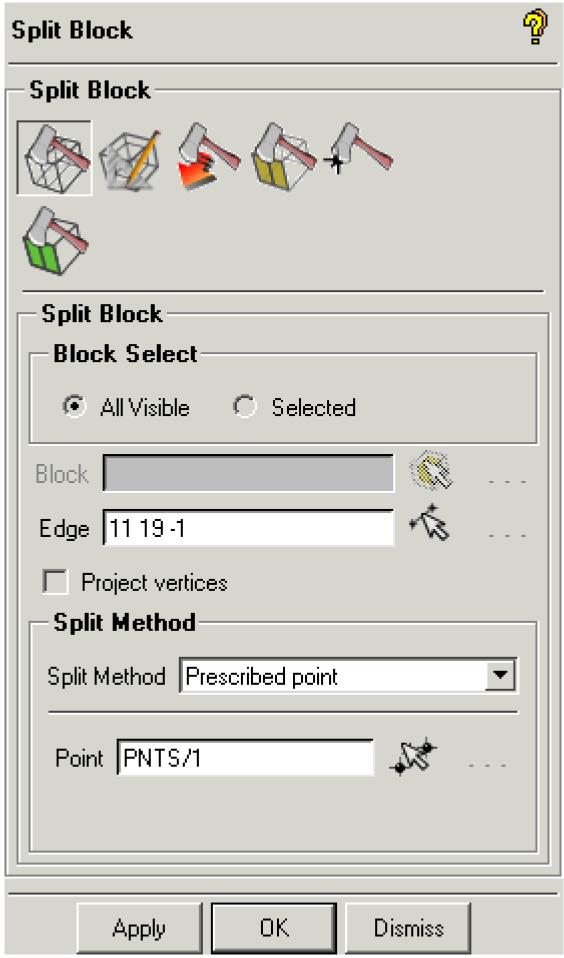
Turn on Points > Show Point Name in the Display tree. The name of the Points will appear on the screen. Zoom in to the bumper.

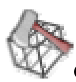
Select Blocking > Split Block  > Split Block  in the Data Entry Panel will be active by default. Don't select at this time.

Note: Many functions, including Split Block in **the Data Entry Panel** will automatically prompt the user to select from the screen. This mode can be turned off/on by selecting Settings > Selection > Auto Pick Mode. If turned on (default) it will sometimes be necessary to exit selection mode (right or middle mouse key) in order to change some options. The selections in this and other tutorials are based on Auto Pick Mode being turned on. Please leave on for the remainder of this tutorial.

In the Split Block panel, change Split Method to Prescribed Point as shown in Figure 3.63.

Figure 3.63
Split Block Window

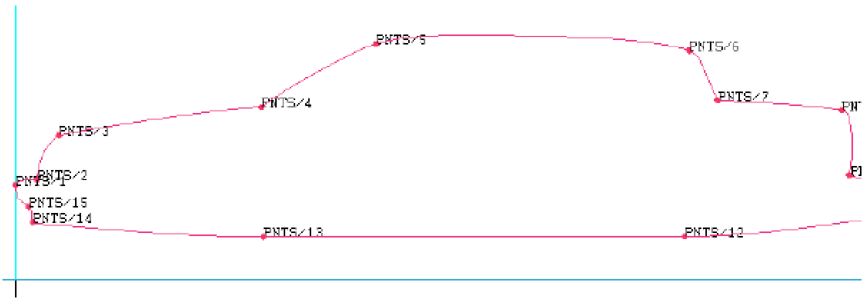


Now select either Split Block  or Select Edge(s)  and select any horizontal edge (top or bottom edge). Then select (PNTS/1) at the front of the bumper. The new edge will automatically be created as in Figure 3.64.

Note how the new edge intersects the point.

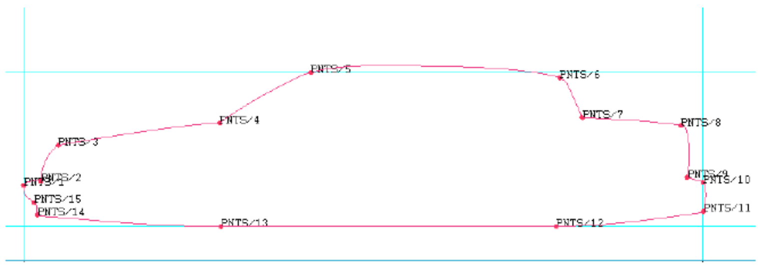
Note: At any point in time while in selection mode, you can toggle on dynamic mode by selecting F9. This may be necessary in order to zoom in to get a closer view of the points. Toggling F9 again will return to selection mode.

Figure 3.64 First Split



In the same manner, make one more vertical split at the rear of the car (choose prescribed point PNTS/10), and two horizontal splits at the top and bottom of the vehicle (PNTS/5 and PNTS/12) as in Figure 3.65.

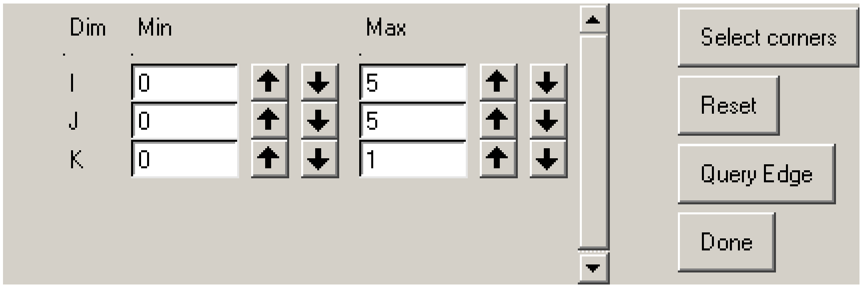
Figure 3.65 Additional Split



e) Overview of the Index Control

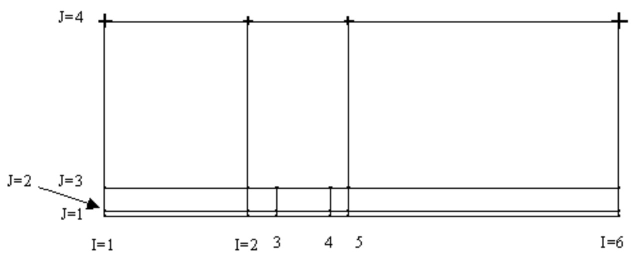
By default, splits only propagate through the displayed blocks. Blocks can be blanked by right mouse selecting Blocking (in the Display tree) > Index control (Figure 3.66) which will appear in the lower right hand corner.

Figure 3.66 Index Control



All block edges and vertices are assigned an I, J, K value. For example, in Figure 3.67, the first edge perpendicular to the x- axis of the global coordinate system has an index of I = 1, while the first edge perpendicular to the y- axis has an index of J=1. For 2D cases, such as this, the K index is undefined.

Figure 3.67 Blocking Indices



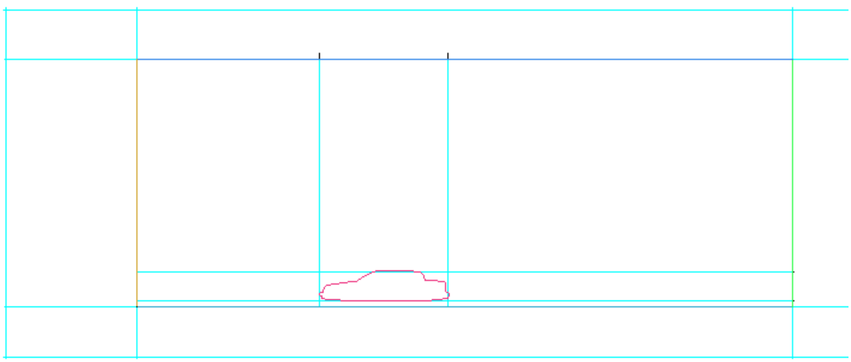
The Index control panel has two columns, Min and Max (left and right columns respectively). The range can be changed by toggling the arrows or entering an integer value in the appropriate field. Only the blocks within this range are displayed. Selecting Reset will turn all of the block indices back 'on'.

f) Splitting the Blocks

Display only the blocks containing the vehicle and those underneath the vehicle: Change the Index range to I: 2-3, J: 1-3.

Note: Notice that incrementing the Index control from 0 to 1 in the "minimum" left column does not result in any change in the block/edge display. Likewise, no change occurs when the maximum number, Nmax, is decreased to Nmax-1, in the right column. The index ranges 0 to 1 and Nmax to Nmax-1 are used by blocks in the VORFN part that form an "outer perimeter" around the initial, central block. These outer blocks are visible (Figure 3.68) if the VORFN part is turned on in the **Index control** panel. The outer blocks are used for O-grid propagation to be explained later in this manual. To simplify the display, leave the VORFN volume family turned off for now.

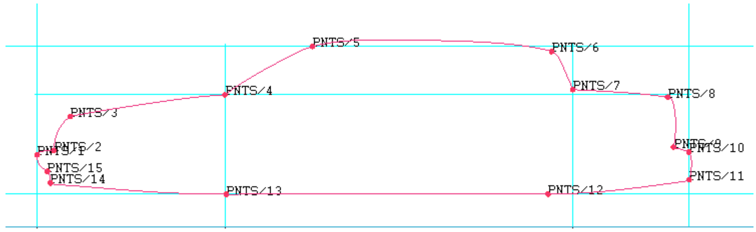
Figure 3.68 VORFN Blocks



Create two vertical splits, one through PNTS/4, the other through PNTS/7. If needed, adjust the Index control to I: 2-5, while keeping the J index the same. Create a horizontal split through PNTS/4.

Select Reset in the Index control panel so that the block appears as shown in Figure 3.69. Note how these new splits don't propagate through all of the blocks.

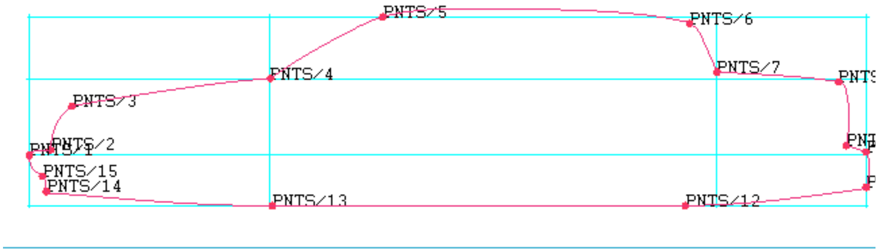
Figure 3.69 Additional Split



Next, carve out a block above each bumper. First set the Index control to I: 2-5, J: 2-3.

Create a horizontal split through PNTS/1. Reset the Index Control of J:2-5 as in Figure 3.70.

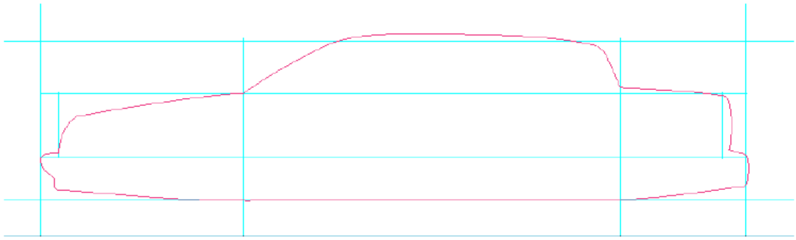
Figure 3.70 The Mid Block



Change Index control to I:2-5 and J:3-4. Proceed to create two vertical splits through PNTS/2 and PNTS/8.

Reset the indices and turn off Points. Your blocks should appear as in Figure 3.71.

Figure 3.71 The Block Indices



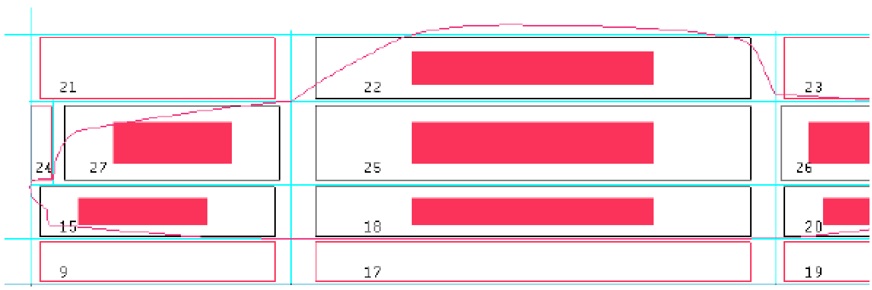
g) Discarding Blocks

For flow analysis, only the blocks outside of the car need be retained. So far, all of the blocks are in the LIVE volume part. The blocks representing the car's interior must be reassigned into a different volume part.

Select Delete Blocks  and select all interior blocks as shown in Figure 3.72 The blocks are 15, 18, 20, 22, 25, 26 and 27. After selection press the middle mouse button or Apply. These blocks will actually be put in the VORFN part since Delete Permanent is turned off (default).

Save blocking

Figure 3.72 OGrid Block



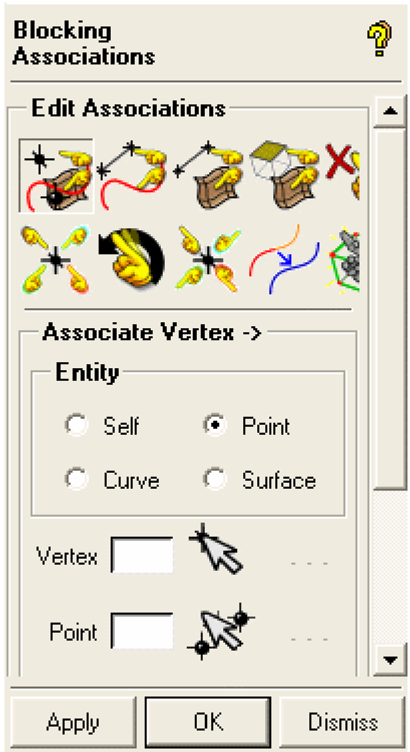
h) Body Fitting the Blocking

To ensure proper projection of the blocking edges onto the geometry, the user will project block vertices to the prescribed points and block edges to the curves.

Turn on Points.

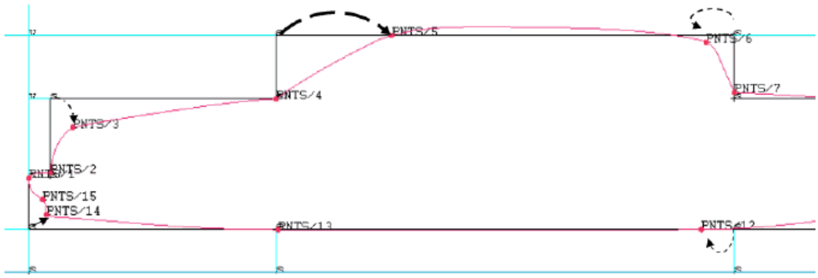
Select Blocking > Associate  > Associate Vertex  as in Figure 3.73.The Entity type Point is toggled on by default.

Figure 3.73
Associate Vertex Panel



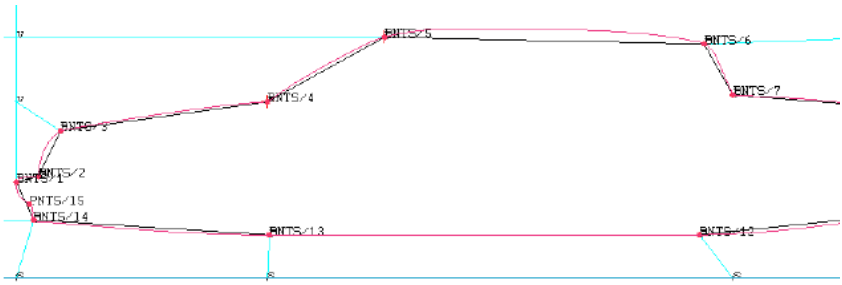
Select Associate Vertex  again or the Select vert(s) icon  and first select the vertex and then the appropriate point as shown in Figure 3.74. The vertex will immediately jump to the selected point. Make sure you associate the vertices that are right on top of their respective points (e.g. PNTS/4).

Figure 3.74 Associating Vertex to Point



Note: The vertices will turn red indicating they are fixed to the prescribed point. The blocks should now better represent the geometry of the car (Figure 3.75).

Figure 3.75 The Blocking fit to the Car



i) Edge-Curve Association.

Turn Points off in the Display tree. Turn off the internal edges: right mouse select Edges and toggle off Volume. Turn off all outer edges: Set Index control to I:2-6, J:2-5.

Select Blocking > Associate  Associate Edge to Curve  Select all the edges that lie on the car body either by dragging a selection box or selecting "v" (or the  icon in the Select blocks toolbar) for all visible. Then select all curves making up the car body individually or by dragging a selection box. Turn back on Edges > Volume and Reset the Index control.

Temporarily turn off Curves and Points in the Display tree to confirm that all the edges around the car body are associated – colored green.

Check to make sure the association is correct by selecting Edges > Show Association from the Display tree and switch on Curves. The projection on the front bumper will resemble Figure 3.76.

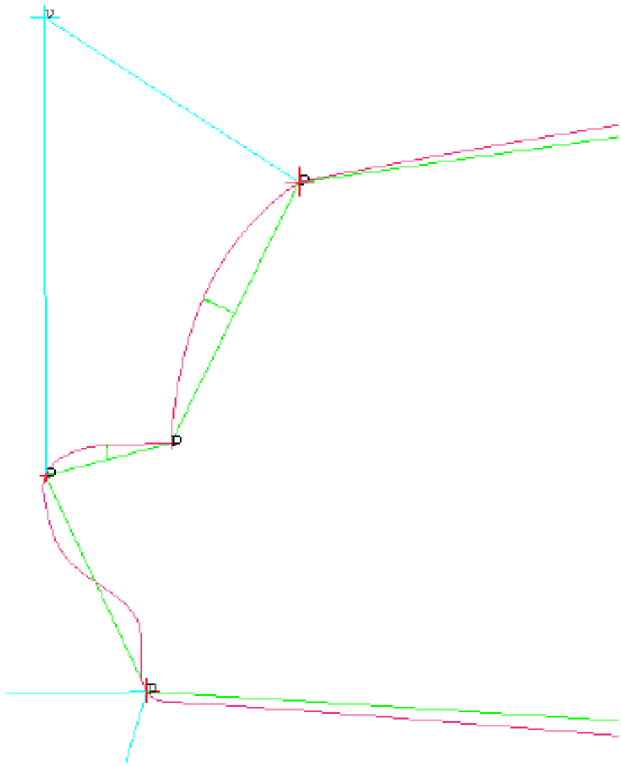


Figure 3.76 Display of Edge Projection

j) Aligning the Vertices

To obtain optimal mesh quality, it is sometimes necessary to line up the block vertices.

Note: As for Split Blocks command, Align Vertices only acts upon the blocks displayed; thus, it is important to use the **Index control** to isolate those blocks.

First line up the vertices of the three blocks underneath the car. To more quickly isolate the blocks, select Blocking (In the Display tree) > Index control > Select corners and select the two diagonally opposing vertices (corners) as shown in

Figure 3.77. Note the change in the I, J ranges within the Index control panel.

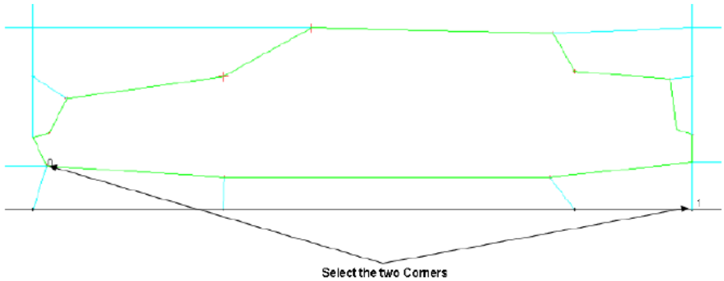


Figure 3.77 Adjusting the index control using From Corners

Turn on Vertices > Indices for reference.

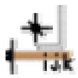
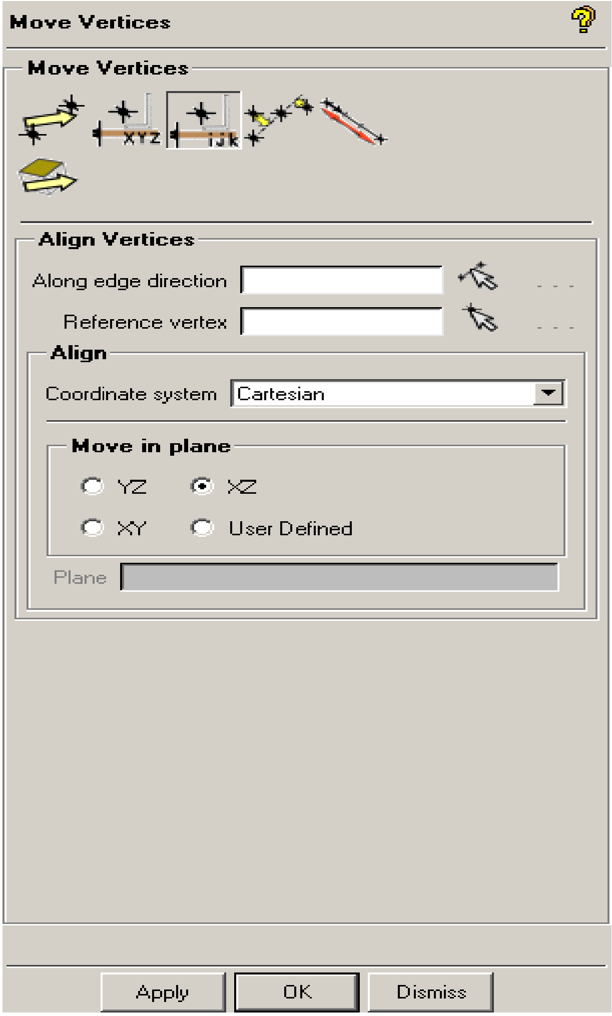
Select Blocking > Move Vertex  > Align Vertices  to obtain the window shown in Figure 3.78.

Figure 3.78
Aligning Vertices panel



First select any one of the vertical (J) edges to define the index align direction. Then select any of the top four vertices as shown in Figure 3.79 and Apply.

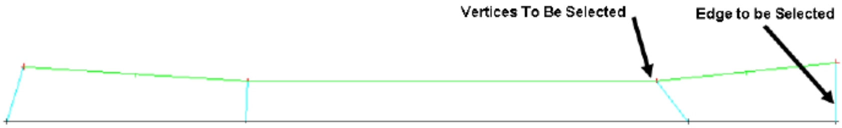


Figure 3.79 Edge and Reference Vertices Selection

Note that the bottom vertices are adjusted to line up with those at the top. By selecting one of the top vertices (e.g. 5 **2** 1), all J=2 vertices will be fixed and all other visible vertices will be adjusted. Also note Move in plane > XZ is automatically toggled on. By selecting a J edge, the program assumes the alignment to be along Y of the active coordinate system, so only the X and Z (in this case Z is undefined) coordinates will be adjusted.

Select Index control > Reset to turn on all blocks. In the model tree turn on Points and turn off Vertices.

Vertex positions can also be adjusted by setting location of coordinates. In this case, we'll line up one of the vertices near the front bumper. Select Move Vertex > Set location and select a reference point, PNTS/3, as shown in Figure 3.81. The coordinates will appear in the Modify fields within the Move Vertex panel. Toggle on Modify Y only. Towards the bottom of the panel, select Vertices to Set > Select vert(s) and select the vertex corresponding to '0' as shown in Figure 3.80 Then Apply.

Figure 3.80 Using Set location to align Vertices

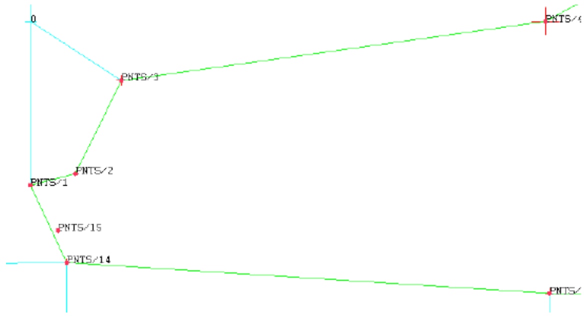
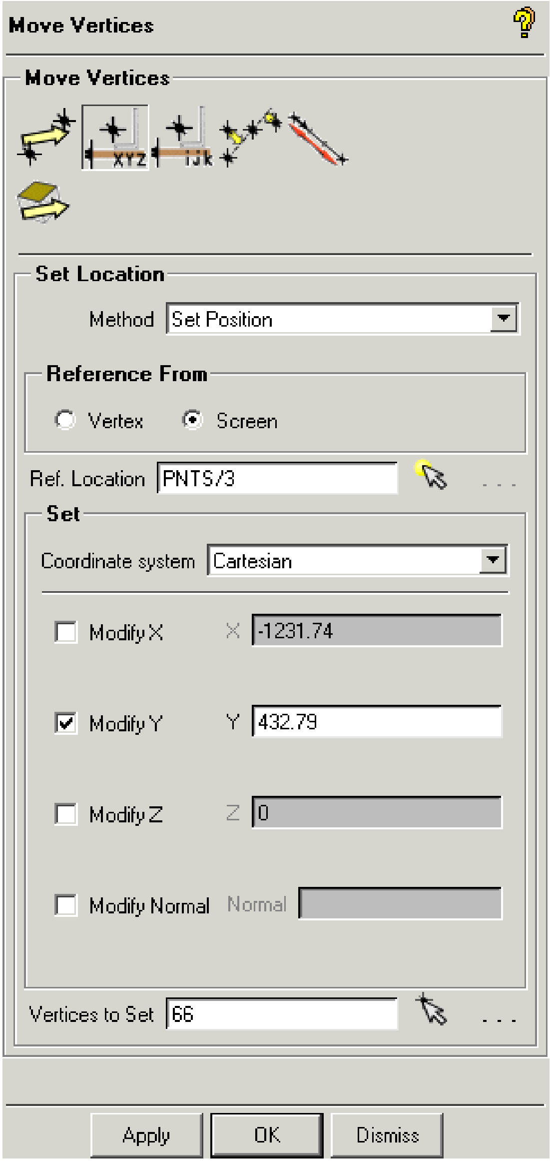
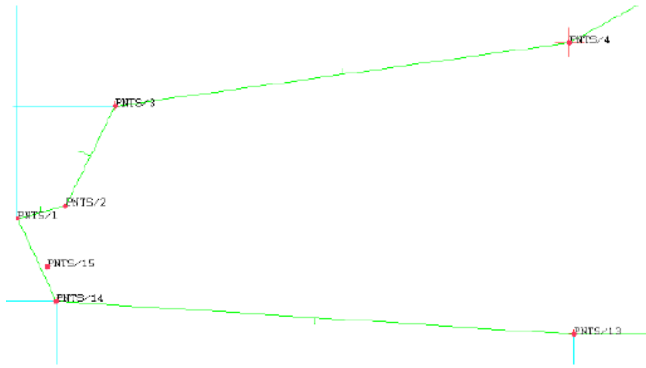


Figure 3.81
Setting the Vertex Location



The vertex will line up with the other one based on the y-coordinate as shown in Figure 3.82.

Figure 3.82
After Performing The Set Location the Vertex will line up



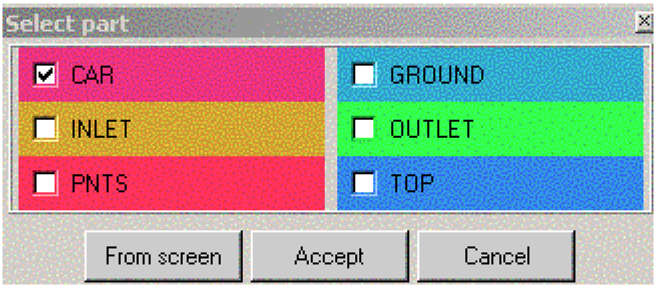
k) Meshing with Curve Parameters

Currently, there is one node on the end of each edge, so the number of elements is equal to the number of blocks. As in the previous tutorial, appropriate node distributions for the edges must be made.

Select Mesh > Set Curve Mesh Size , set Maximum Size to 25

Select curve(s) and either type Shift P or select the Select items in Part icon from the Select geometry toolbar.(Figure 3.83) In the Select part window turn on CAR and Accept.

Figure 3.83 Select Part 'Car'



Back in the Curve Mesh Size panel, set Maximum Size to 25 and Apply.

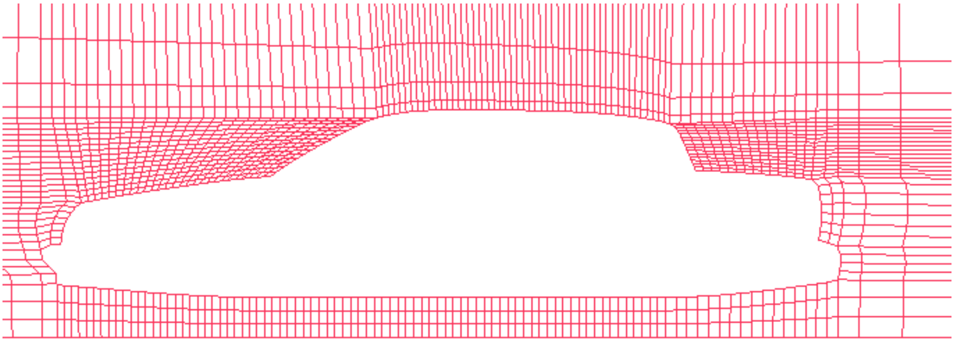
Repeat this procedure but toggle on INLET, OUTLET, TOP and GROUND, set Maximum Size to 500 and Apply.

Select Blocking > Pre-mesh Params  > Update Size  > Update All and Apply.

Turn on Pre-mesh in the Display tree and recompute. (Figure 3.84).

What has been created so far is a body-fitted blocking that is aligned with indices I and J. This is known as a Cartesian or H-grid type of blocking.

Figure 3.84 The H Grid Blocking



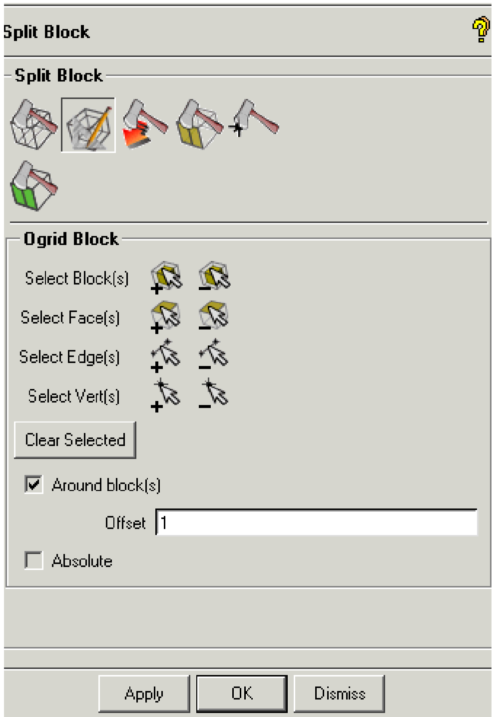
l) Creating an O-grid around the Car

Next, create an O-grid, where the mesh “radially” propagates from the surface of the car towards the outer domain. This will result in an orthogonal mesh to better capture near-wall or boundary layer flow.

First, turn off Pre-Mesh and turn on Edges. Also turn on the VORFN part for we’re going to select the interior blocks

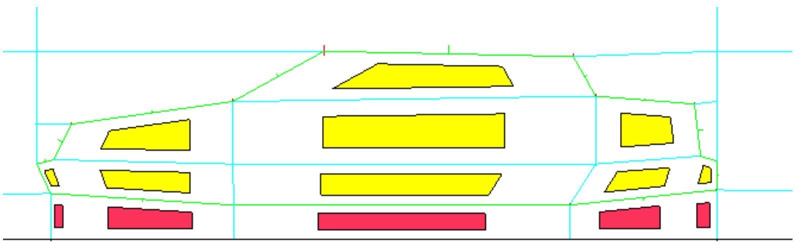
Select Blocking > Split Block  > O Grid  to obtain the panel shown in Figure 3.85.

Figure 3.85
Creating an O grid in the Blocking



Using Select Blocks , select the blocks as in Figure 3.86 and press the middle mouse button to accept selection. Turn off VORFN. The selected blocks will disappear.

Figure 3.86
Select the blocks for the O grid



Turn on Around Block(s) and Apply. The blocking will appear as below in Figure 3.87.

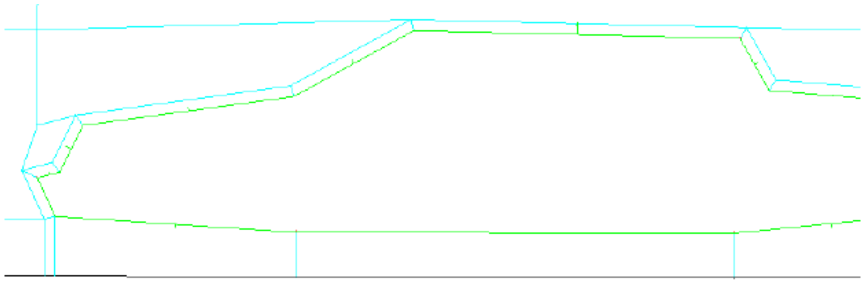


Figure 3.87 Your external O grid of the car

m) Meshing with Edge Parameters

Select Pre-mesh Params > Edge params and select one of the radial edges of the O-grid as in Figure 3.88.

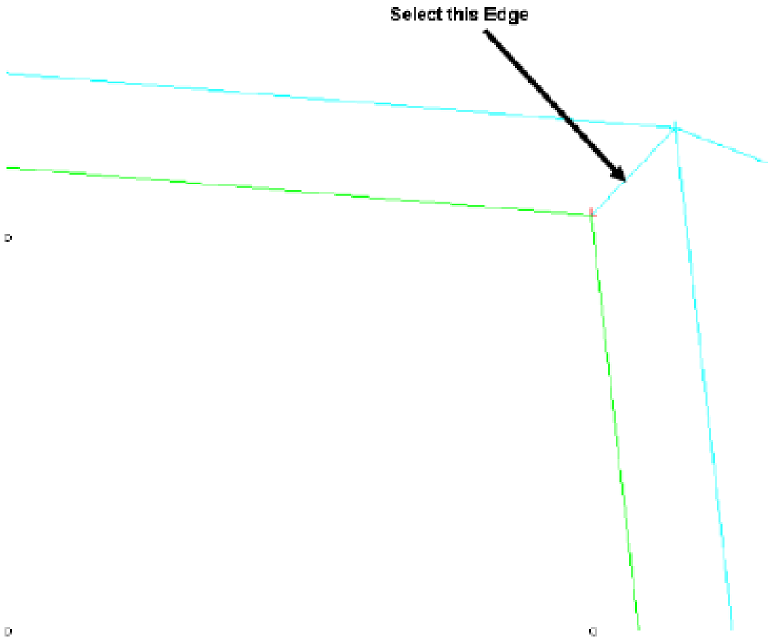


Figure 3.88 Setting the meshing parameters on the edge

Increase Nodes to 7. To bunch the elements close to the car, decrease Spacing 2 to 1 and change Ratio 2 to 1.5. Toggle on Copy Parameters, set Method > To All Parallel Edges (default) and Apply. This node distribution will be applied throughout the O-grid.

Select one of the vertical edges between the car and the ground. Change Nodes to 15, Spacing 1 and 2 to 1, Ratio 1 and 2 to 1.5 and Apply. Note that the ratios presented in the second column (actual) were not attained. Increase the number of Nodes until both ratios are near 1.5.

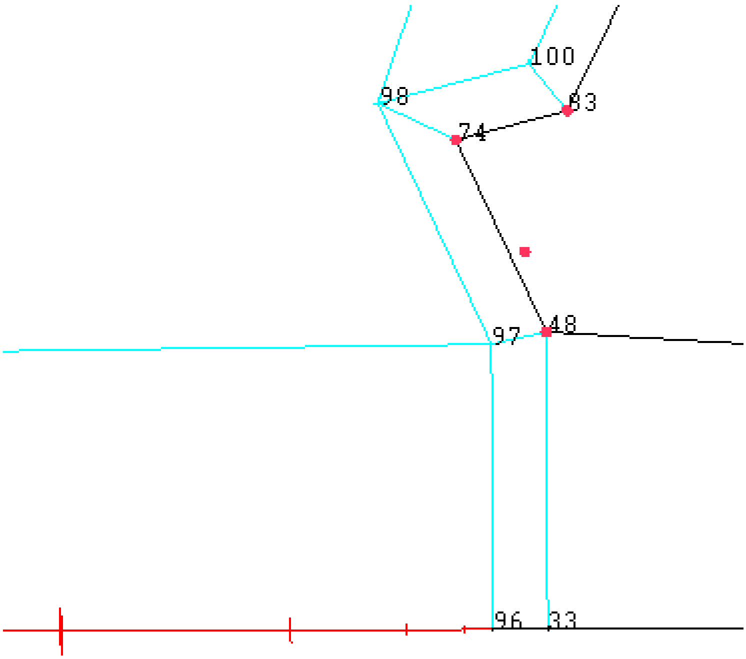
Select Edges > Bunching in the Display tree.

Turn on Pre-Mesh and recompute. Note the large gradients in mesh size just before and behind the vehicle. We will now match the node spacing of one edge to the other.

Turn off Pre-Mesh and, for reference turn on Vertices > Numbers.

Select Pre-Mesh Params > Match Edges. Select the radial edge on the ground plane in front, 33-96 for the Reference Edge and then immediately select the edge just before it as the Target Edge(s) as in Figure 3.89. Press the middle mouse button to complete.

Figure 3.89 Display of the bunching using the Edge>Bunching

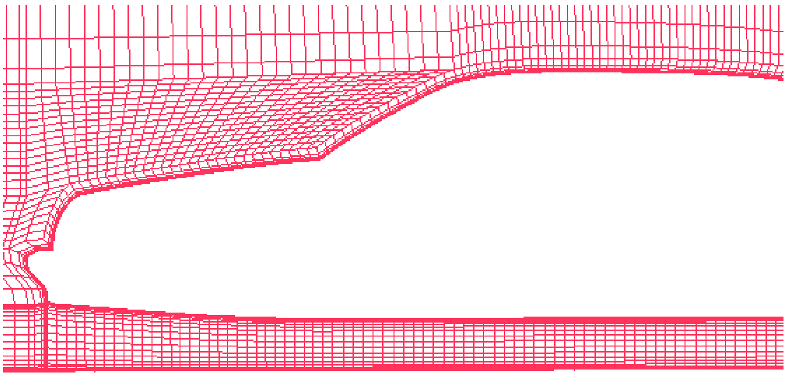


Next, back to Edge Params, select the previous target edge, and make sure Copy Parameters > To All Parallel Edges is on and Apply.

Repeat for the edge behind the car, using edge 37-111 as the reference.

Turn on Pre-Mesh and recomputed.

Figure 3.90 The Final Mesh of the baseline model



n) Saving your Replay File.

Bring the Replay control window to the foreground and select Save. Accept the default filename “replay_file.rpl” and Save from the Save Script File browser.

Select Done to close the Replay control window.

Select File > Close Project and type in any suitable name.

o) Using Replay for the Design Iteration

The user is now ready to rebuild the block topology on a similar geometry, or design iteration. Instead of repeating the same commands manually, run the Replay file.

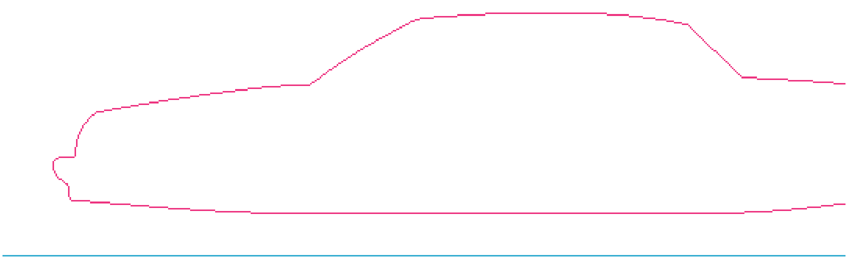
To load the iteration, select File > Geometry >Open Geometry, choose `car_mod.tin` and Replace the original geometry when prompted.

In car_mod.tin the trunk or deck-lid has been extended rearward, the rear windshield (backlight) angle has been changed and the windshield has been moved slightly rearward.

Since the replay file will act on the prescribed points (which have been moved but carry the same name), all of those operations performed with respect to prescribed points will be valid.

Display the Curves and zoom in so the box representing the wind tunnel fills the window (Figure 3.91). Notice the differences in the geometry from the car_base subproject.

Figure 3.91 The car model geometry

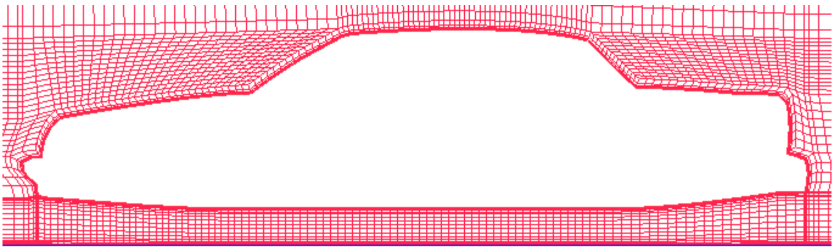


Select File > Replay Script > Replay Control. The Replay control window will show all the commands you previously saved in the subproject car_base. If a new session, you would have to select Load from the Replay control window and select the saved `replay_file.rpl`.

In the Replay control window, scroll all the way to the top and highlight line no. 1. Select Do all.

Turn on Pre-Mesh and recompute (Figure 3.92).

Figure 3.92 Final Mesh



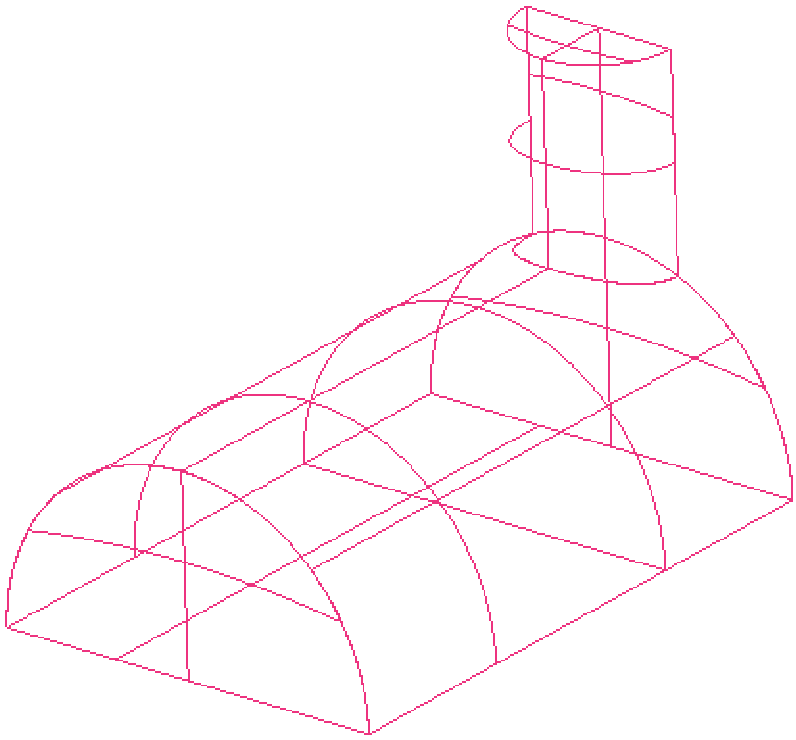
This mesh will have been generated using exactly the same parameters as the first, so the differences in solutions may be attributed to the changes in the geometry, rather than to any dissimilarity in the grids.

When finished looking at the results, save the unstructured mesh: Pre-Mesh > Convert to Unstruct.

Save the project and File > Exit or continue with the next tutorial.

3.2.4: 3D Pipe Junction

Overview



In this tutorial example, the user will generate a mesh for a three-dimensional pipe junction. After checking the quality of the first mesh, the user will create an O-grid in the blocking to improve mesh quality.

a) Summary of Steps

The Blocking Strategy

Starting the Project

Creating Parts

Starting Blocking

Blocking the Geometry

Projecting the Edges to the Curves

Moving the Vertices

Generating the Mesh

Checking the Mesh Quality

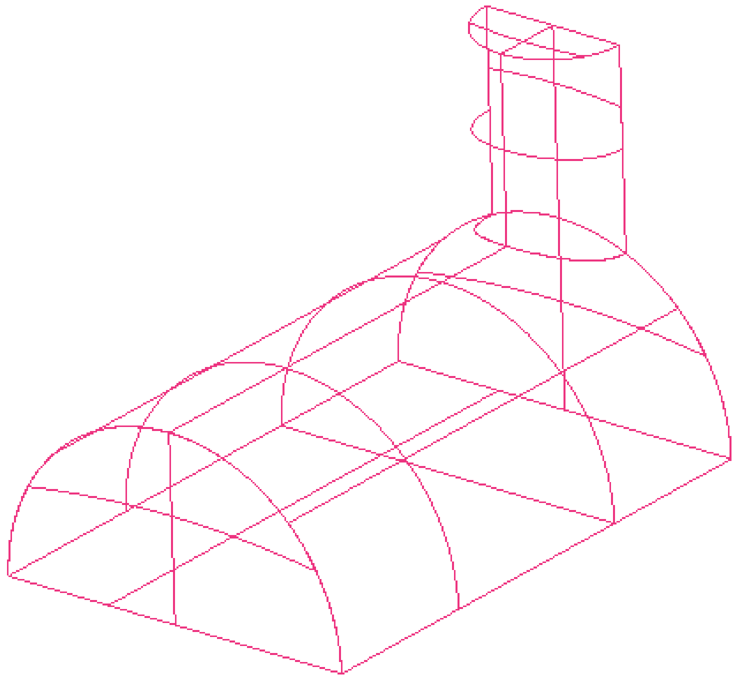
Creating an O-grid in the Blocking

Verifying and Saving the Mesh

b) The Blocking Strategy

The strategy for this first three-dimensional example is fairly simple. First, cut twwo blocks from the initial block, one each for each half cylinder forming an L-shaped configuration. Then, create an O-grid to improve the mesh quality.

Figure 3.93 3D Pipe Geometry



c) Starting the Project

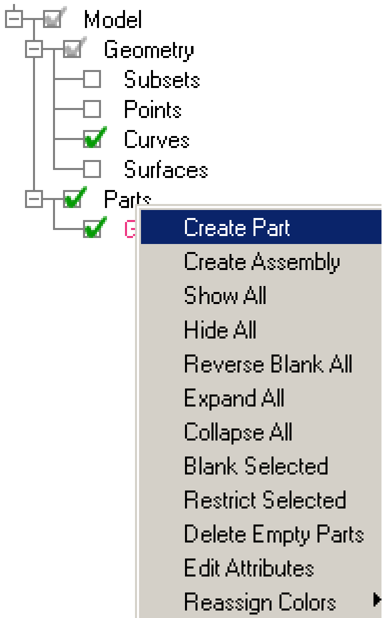
Start ANSYS ICEMCFD and Change working directory to $ICEM_ACN/../../docu/CFDHelp/CFD_Tutorial_Files/3DPipeJunction. Select File > Geometry > Open and select the tetin file, geometry.tin.

d) Creating Parts

In the first two tutorials, the parts were pre-defined. For this and the remaining tutorials, the initial geometry is in a single part. Geometry will be put into different parts to define different boundary regions. First expand the Parts tree and turn on Surfaces.

Right mouse select Parts and select Create Part as shown in Figure 3.94.

Figure 3.94
Create Part option

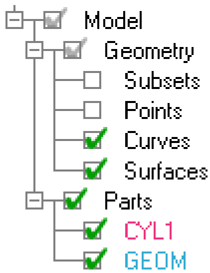


In the Create Part panel type in CYL1 for the Part name. Select Create Part by Selection  or Select entities . To avoid selecting entities other than surfaces, turn off Toggle selection of points , Toggle selection of curves  and Toggle selection of bodies (material region definition). Leave on Toggle selection of surfaces  as shown in Figure 3.94. Entity types can also be deactivated (unselectable) by turning them off in the Display tree.

Figure 3.95
Select Geometry tool bar

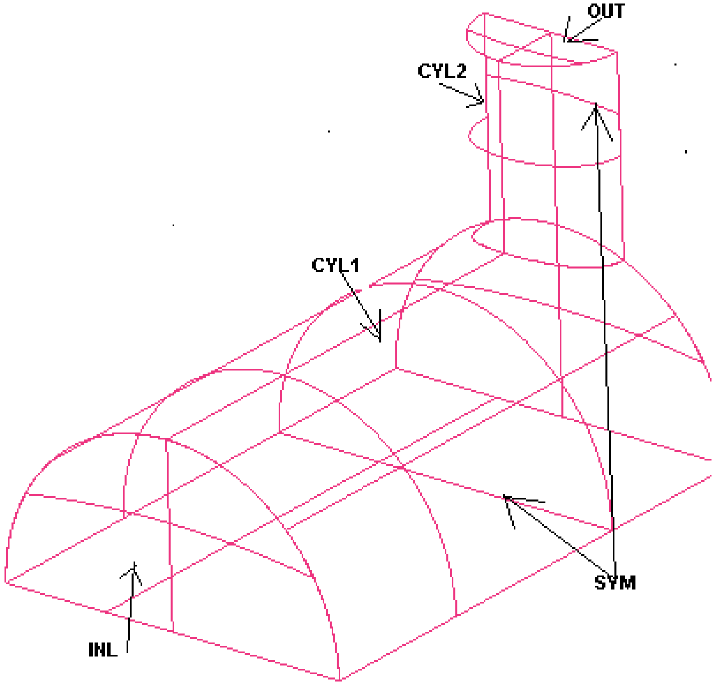
Select the largest semi cylinder with the left mouse button and press middle mouse button or Apply in the panel. Note the new part in the model tree as shown in Figure 3.96.

Figure 3.96
Part CYL1 added in display tree



Similarly create new parts for the smaller semi cylinder (CYL2), cylinder ends (INL and OUT), and symmetry planes SYM, as shown in Figure 3.97. When in continuation mode after pressing the middle mouse button or Apply, you can simply type in a new Part name and continue to select the surface(s) without having to re-invoke the function.

Figure 3.97 The 3DPipeJunct geometry and its Surface Parts



Now turn off Toggle selection of surfaces  and turn on Toggle selection of curves  in the toolbar as shown in Figure 3.98.

Figure 3.98 Curve selection

Type in CURVE for the Part name and select all curves. Either type “a” for all (icon in the toolbar), “v” for all visible ( ) or click and drag a box selection. For “a” and “v” selection options you don’t need to hit the middle mouse button or Apply to complete the operation.

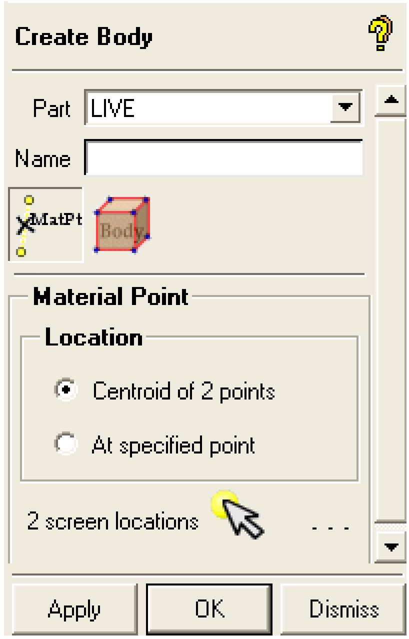
Similarly, put all points in a POINT part. Turn off Toggle selection of curves and turn on Toggle selection of points . Type “a” for all or select in the toolbar as shown in Figure 3.99.

Figure 3.99
Geometry Part Point or Node Selection

e) Creating a Material point.

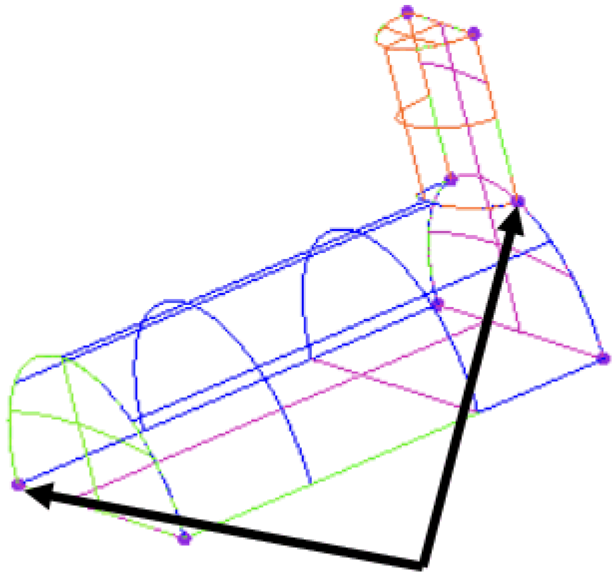
Select Geometry > Create Body from the geometry tab (Figure 3.100). Type in LIVE for the Part name.

Figure 3.100
Create Body panel



Select Material Point  or Select location(s)  and select two locations such that the center lies within the volume as in Figure 3.101.

Figure 3.101 Selection of points for Material point creation



Press the middle mouse button or Apply.

Right select Parts > Delete Empty Parts in the Display tree. The empty GEOM part should be deleted. If not, right mouse select GEOM > Delete.

Save the geometry file: using File > Geometry > Save Geometry As or save the project.

f) Blocking the Geometry

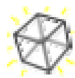
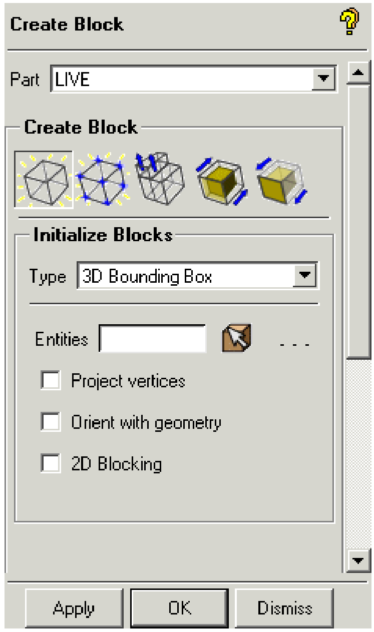
Select Blocking > Create Block  > Initialize block  .Refer to Figure 3.102. Select the LIVE Part, make sure Type > 3D Bounding Box is selected (default) and Apply.

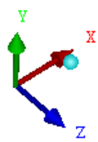
Figure 3.102
Create Block Window



It isn't necessary to select entities for a bounding box around the entire geometry.

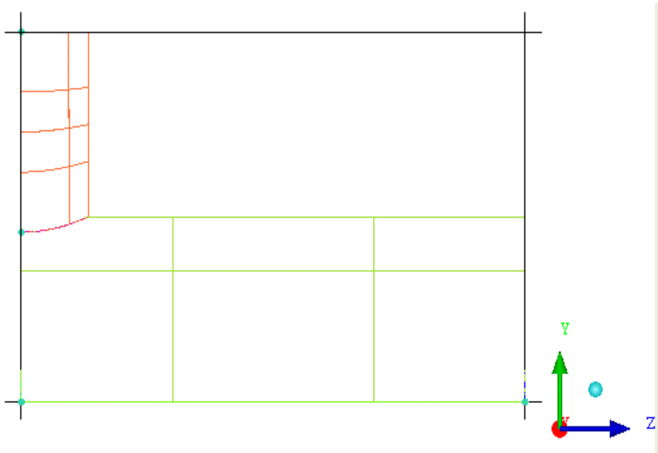
The next step is to split the block into four sub-blocks. Begin by turning on Curves and Surfaces from the Display tree.

The L-shaped topology is best seen in a side view. Select View > Left or select the X axis in the Triad Display



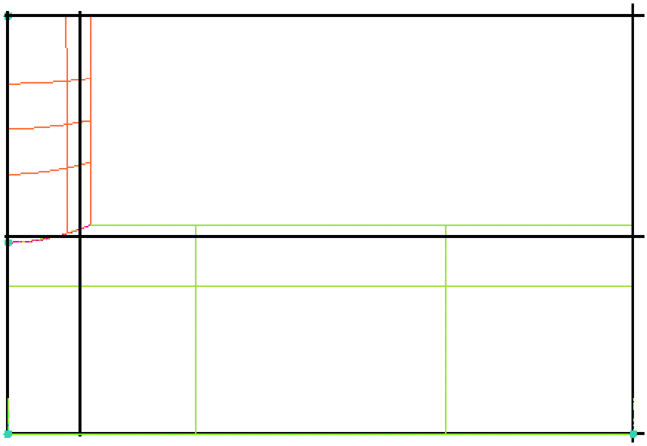
in the lower right hand corner to re-orient the model as it appears in Figure 3.103.

Figure 3.103 Geometry showing the spilt locations



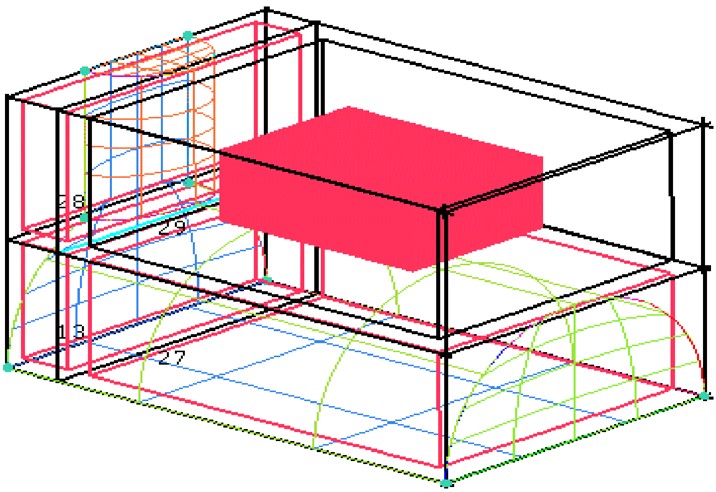
Select Blocking > Split Block  > Split Block . Select any horizontal edge with the left mouse button; try to position the new edge near the front end of the small cylinder, and press the middle mouse button to accept. Next select any of the vertical edges and position the new edge near the top of the large cylinder (CYL1). Splits should appear as shown in Figure 3.104.

Figure 3.104 Block Splits



Next, discard the upper large block. Select Delete Blocks and remove block shown in Figure 3.105.

Figure 3.105 Delete Block



g) Projecting the Edges to the Curves

Turn off the Surfaces, displaying Curves only.

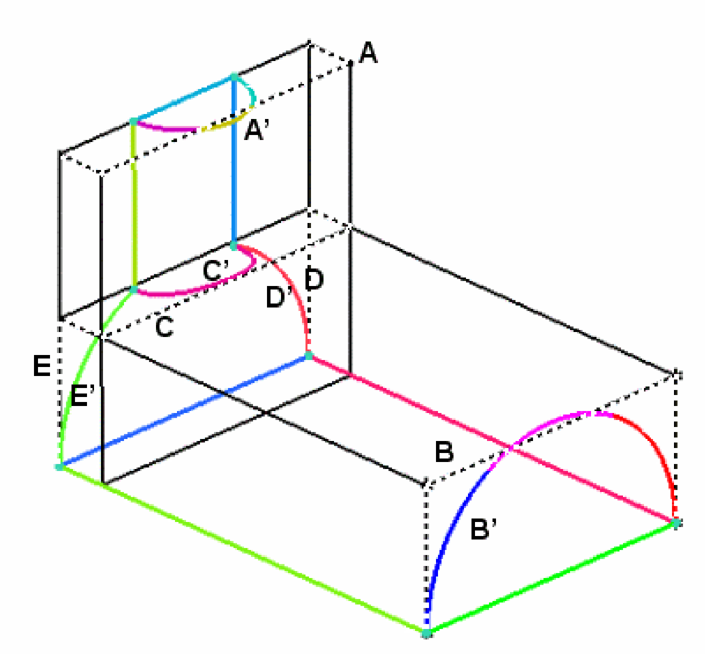
Select Associate  > Associate Edge to Curve  .Select the three edges at the top (A) indicated with dashed lines in Figure 3.106. Press the middle mouse button, then select the three curves (A’) making up the small semicircle. Press the middle mouse button to complete the operation.

In continuation mode, you’ll be prompted to select the next set of edges/curves. Select the three edges (B) at the front of the large cylinder, accept with the middle mouse button, and then select the three curves making up the large semicircle (B’). Again, press the middle mouse button to complete.

Associate the three edges on the Y-plane near the cylinder intersections (C), then the semicircle curve making up the intersection (C’).

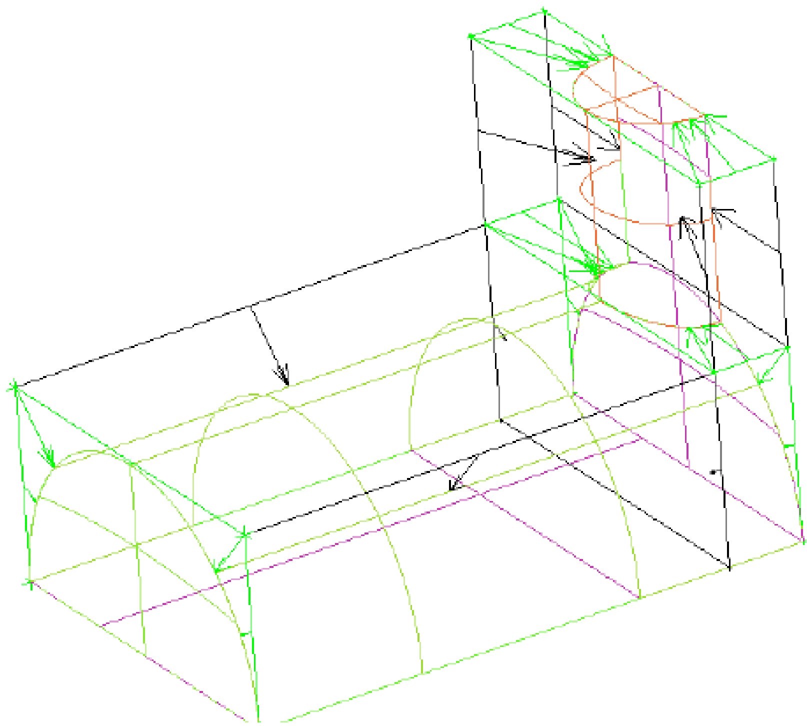
Finally, associate the side rear edges to the curves making up the backend of the large cylinder, D->D’ and E -> E’, as shown in Figure 3.106.

Figure 3.106 Associating edges to curves



Verify that the correct associations have been set: right mouse select Edges > and select Show Association in the Display tree (Figure 3.107). The arrow originates from the edge center and points to the geometry entity it's associated to. Note that white edges point to the nearest point normal to the nearest surface for they're not directly associated to a specific surface.

Figure 3.107 Display of the projections of the edges to the associated curve

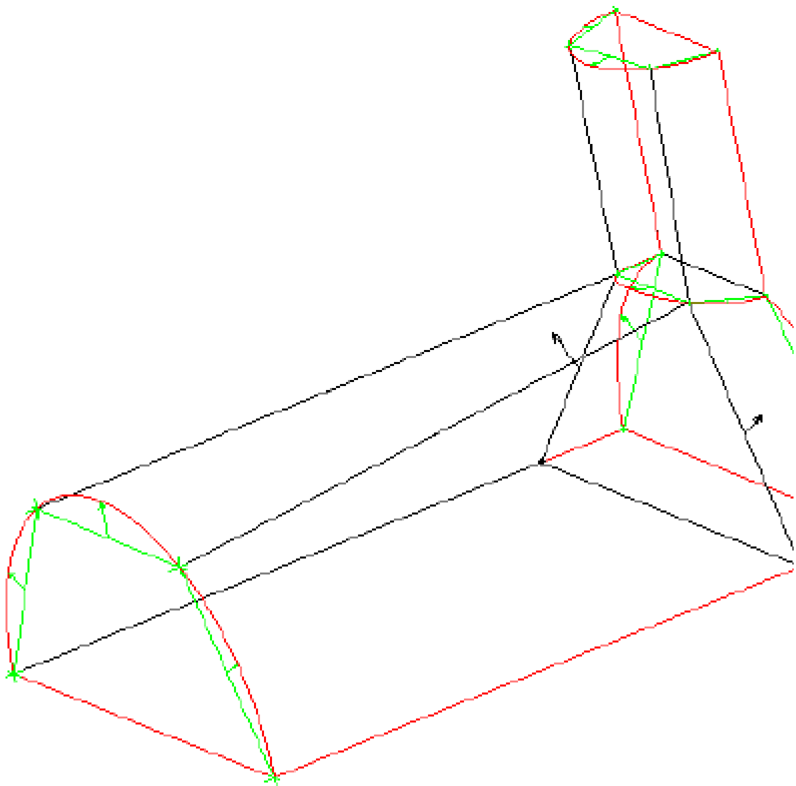


h) Moving the Vertices

You can automatically snap all vertices on to the geometry with Select Associate > Snap Project Vertices Toggle on Vertex Select > All Visible (default) and Apply.

Manually move the vertices, Move Vertex > Move Vertex and position the vertices as in Figure 3.108. For now, only move green vertices on their associated curves. Primarily, make the edges along the ends of the small cylinder more or less equidistant.

Figure 3.108 After moving the vertices to the appropriate locations on the geometry



Turn off Edges > Association in the Display tree.

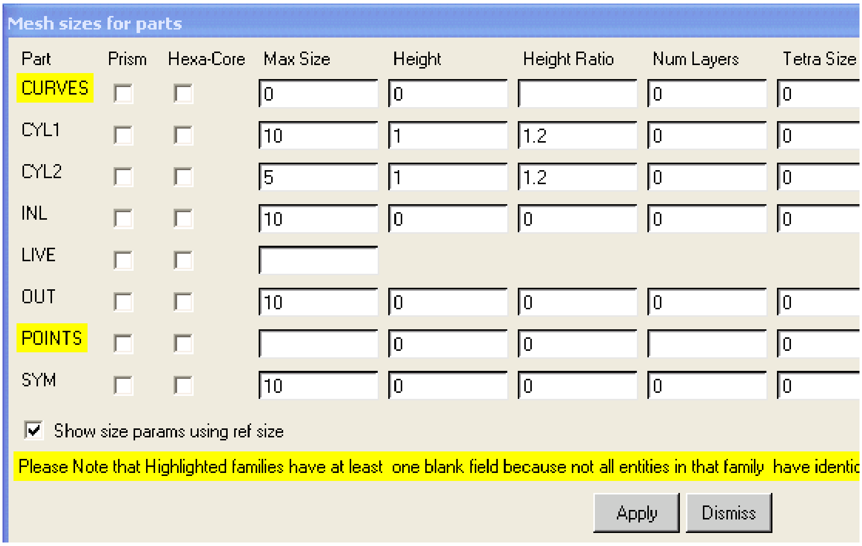
Save the Blocking!

i) Generating the Mesh

Next, specify mesh parameters, this time on surfaces for a 3D model. For this model, we'll set the sizes on the parts, rather than individual surfaces or curves.

Select Mesh > Set Meshing Params by Parts to get the menu shown in Figure 3.109.

Figure 3.109 Entering new mesh parameters

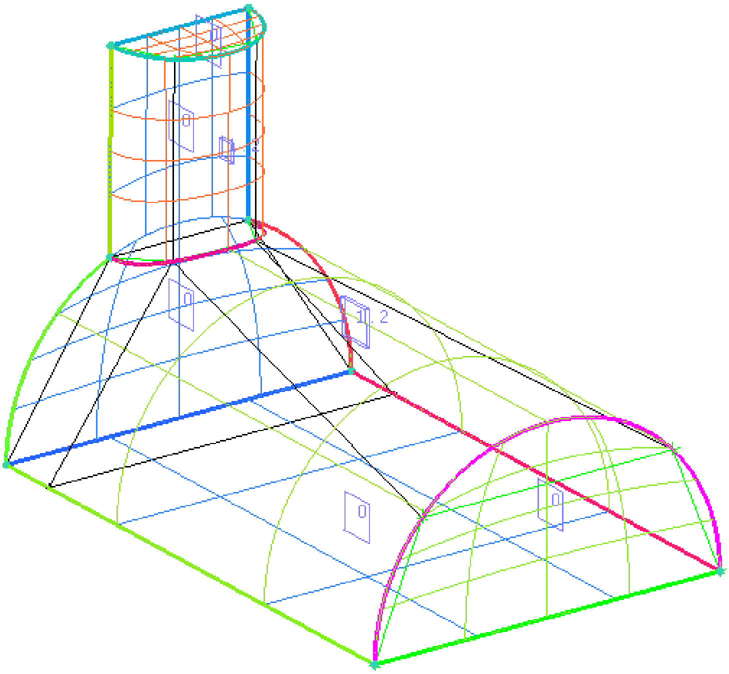


Set sizes as shown: Max Size of 10 on all surface parts except the small cylinder (CYL2) which can be 5, Height of 1 only on the wall boundaries (CYL1 and CYL2) and a Height Ratio of 1.2 on those same walls. Apply and Dismiss.

Turn on Surface > Hexa size in the Display tree. View the meshing parameters for each surface as in Figure 3.110.

Note:The “quad” perpendicular to the surface represents the Max Size, the thickness represents the Height and the number is the Height Ratio.

Figure 3.110 Hexa Mesh sizes



Select Blocking > Pre-Mesh Params  > Update Size  , make sure Method > Update all (default) is selected and press Apply.

Turn on Pre-Mesh and (re) compute.

Turn off the Edges, Surfaces and other geometry types. Turn on (right mouse select) Pre-Mesh > Solid & Wire. View this initial mesh as in Figure 3.111.

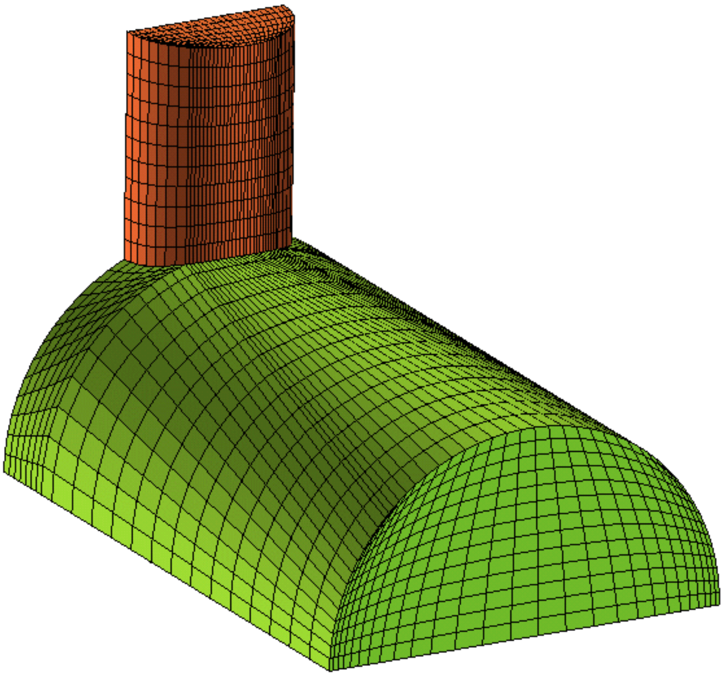


Figure 3.111 The initial Mesh

j) Checking the Mesh Quality

After generating the mesh, the user should check the mesh quality. For a more complete description of the Mesh quality criteria, refer to Help > Help Topics. The main criteria affecting a hexa mesh are

Angle

This checks the minimum internal angle, in degrees, for each element.

Determinant:

This calculates the determinant of all elements in the mesh, which is a volume measurement calculated from a Jacobian matrix.

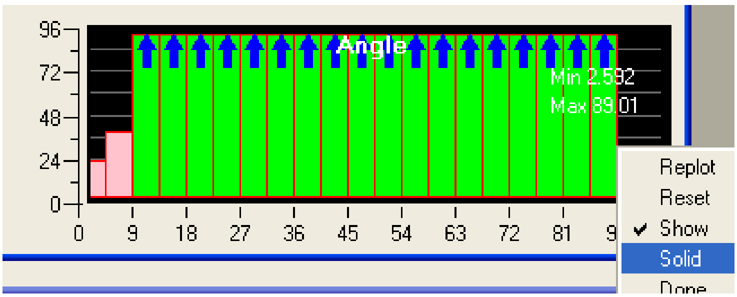
Warpage:

This is the angle between two virtual tri faces making up a quad face. Reported value is the worst angle of the “tri” faces within a given element.

Select Blocking > Pre-mesh Quality  .For the Criterion, select Angle. A histogram (bar graph) of the values will be displayed as in Figure 3.112 Select the two worst ranges (bars). They will be highlighted in pink.

Select with the right mouse button anywhere within the histogram window. Make sure Show (default) is turned on in the pull down. You may wish to turn Solid off.

Figure 3.112 Histogram of Angle



View the highlighted elements as in Figure 3.113. Pre-Mesh should be turned off. Turn on Geometry > Curves for reference. Note that most of the bad elements (those with the worst angles) are on the block corners. This is due to the H-grid nature of the mesh within a curved geometry. Select Done from the pull down (after right mouse selecting in the window).

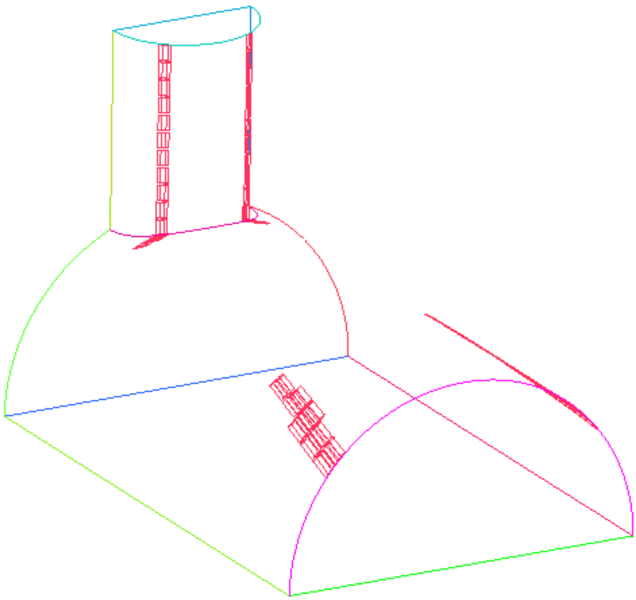


Figure 3.113 The highlighted elements in mesh

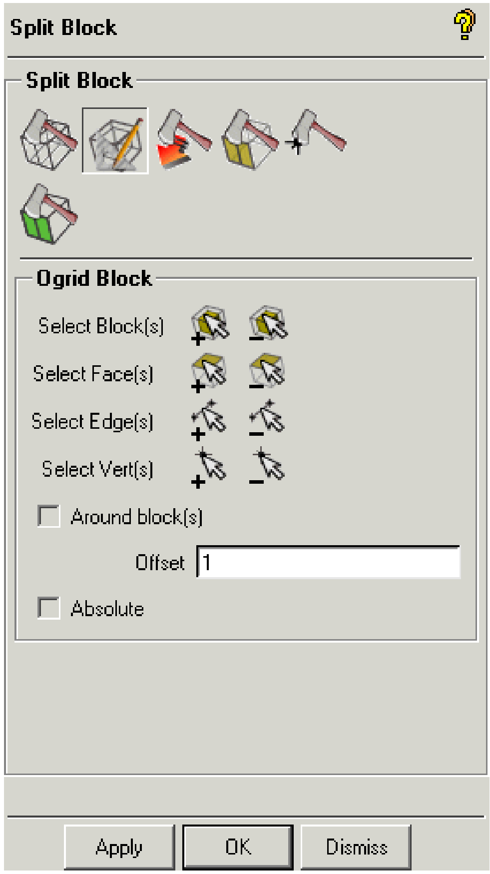
k) Creating an O-grid in the Blocking

The best method for fixing bad angles in block corners within cylindrical geometry is to create an internal o-grid which will radially propagate from a central block.

First, turn back on Edges, Surfaces and Curves.

Select Blocking > Split Block  > Ogrid Block  .This will bring up the Ogrid Block panel as in Figure 3.114.

Figure 3.114
Creating an O-grid

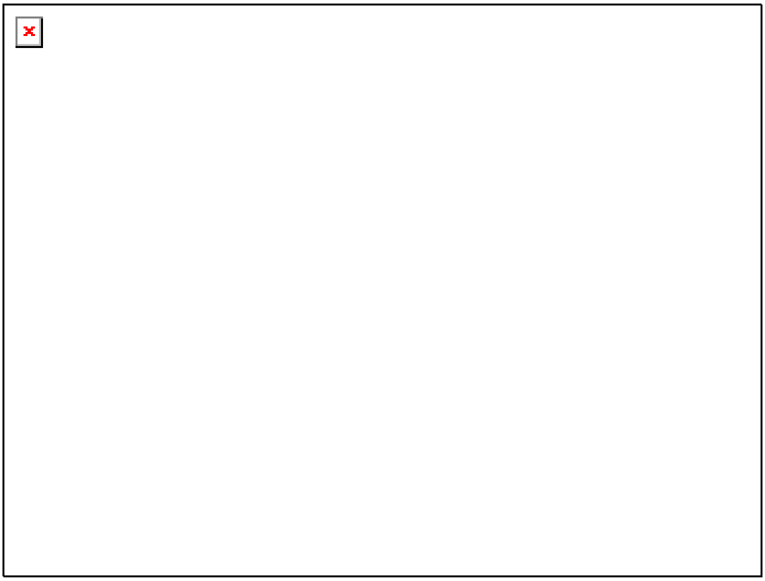


First Select Block(s)  and either type “v” for all visible or drag a box to select all the blocks. Note, “a” for all is not available for blocking.

Then, select faces representing all planar geometry: INL, SYM and OUT. Select Face(s)  and select the face icons as shown in Figure 3.115.

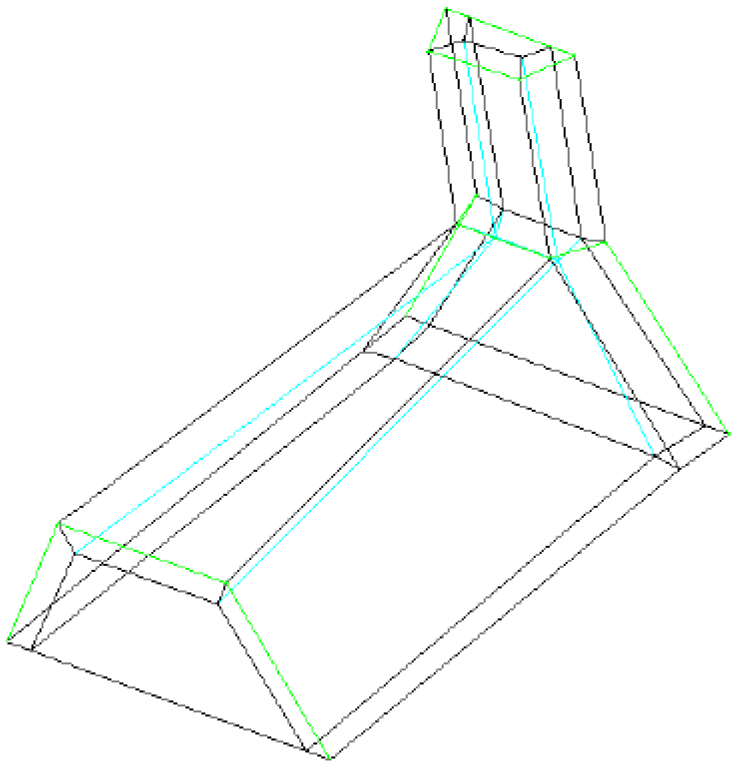
If there is difficulty in seeing the face icon, one can select a face (or block for that matter) selecting Select diagonal corner vertices from the Select blocks toolbar or typing Shft-D on the keyboard. This will allow you to select two diagonally opposing corners that make up the face.

Figure 3.115 Selected Blocks and Faces



Use the default Offset and Apply. An o-grid structure will be created as in Figure 3.116. Note the o-grid "passing through" the selected faces. Radial blocks are only adjacent to the cylinder surfaces.

Figure 3.116 The blocking with O grid structure



To re-size the o-grid after it's been created, select Edit Block  > Modify Ogrid , this will open the Modify Ogrid panel as shown in Figure 3.117. Select edge(s)  and select one of the radial edges as in Figure 3.118. Enter the Offset as 0.5, toggle off Absolute distance (default) and Apply. The radial edge will be shrunk in half reducing the size of the radial blocks and increasing the size of the central block.

Figure 3.117 Modify OGrid panel

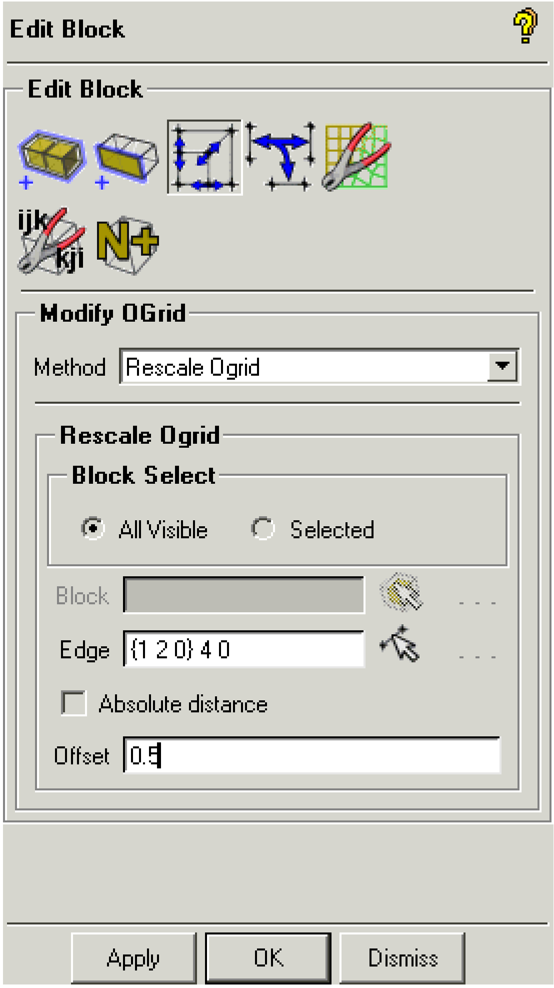
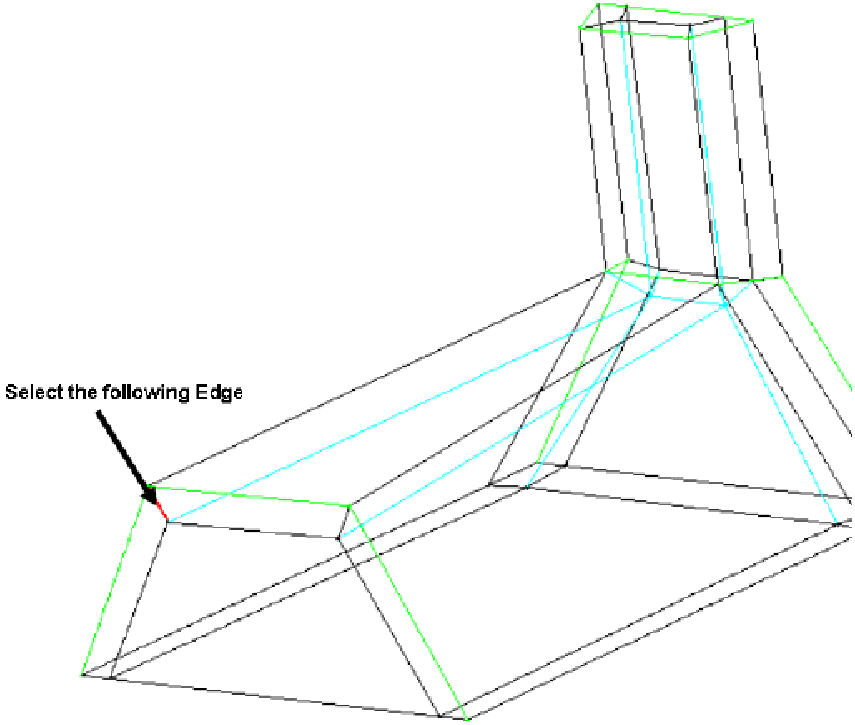


Figure 3.118 Modify Ogrid edge



Update surface mesh sizes on the blocking: Select Pre-mesh Params > Update Size and Apply.

Turn on Pre-Mesh and recompute.

l) Further refinement with Edge Parameters

Again, turn off Pre-Mesh. Select Select Pre-Mesh Params > Edge Params > Select edge(s) and again select one of the radial edges.

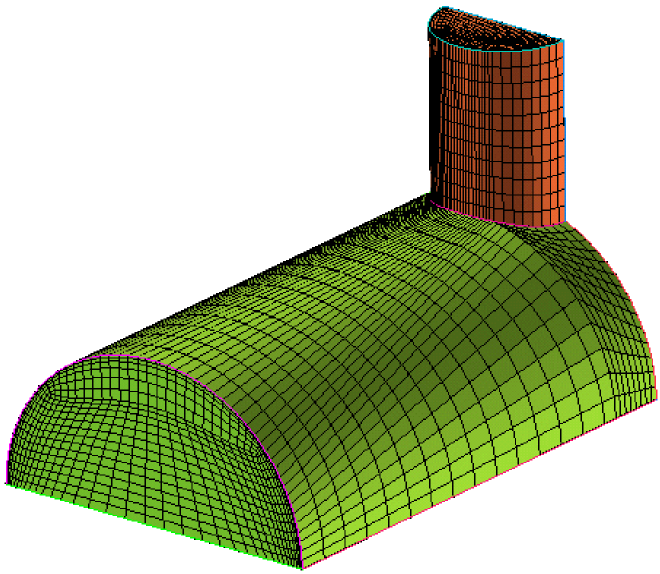
Increase the number of Nodes to 7. Change Spacing 1 (end near the wall) to 0.2. Turn on Copy Parameters and select

Copy > Method > To All Parallel Edges (default). Turn on Copy Absolute. Apply. This will carry a 0.2 near wall spacing throughout all of the cylinder surfaces. Make any other node distribution changes you see fit.

Turn on Pre-Mesh and recompute.

Turn off Curves, Surfaces, and Edges and view the final mesh as in Figure 3.119.

Figure 3.119
A Solid display of the Mesh



m) Verifying and Saving the Mesh

Select Pre-mesh Quality  change the Criterion to Angle and Apply. Note the improved mesh quality in the histogram in the right hand window. Also check Criterion >

Determinant 2 x 2 x 2. Re-Apply and note the quality in the histogram.

Save to unstructured: Right mouse select Pre-Mesh and select Convert to Unstruct Mesh from the pull down.

Save the blocking using File > Blocking > Save as… and/or save the Project.

Use File > Exit to quit or continue with the next tutorial.

3.2.5: Sphere Cube

Overview

In this example, the user will employ an O-grid to fit the topology of the region between a Cube and a Sphere. The O-grid forms a topological bridge between the dissimilar topologies and provides excellent element quality.

a) Summary of Steps

The Blocking Strategy

Starting the Project

Creating Parts

Starting Blocking

Creating the Composite Curve

Projecting the Edges to Curves

Moving Vertices

Creating the O-grid

Fitting the O-grid Using Prescribed Points

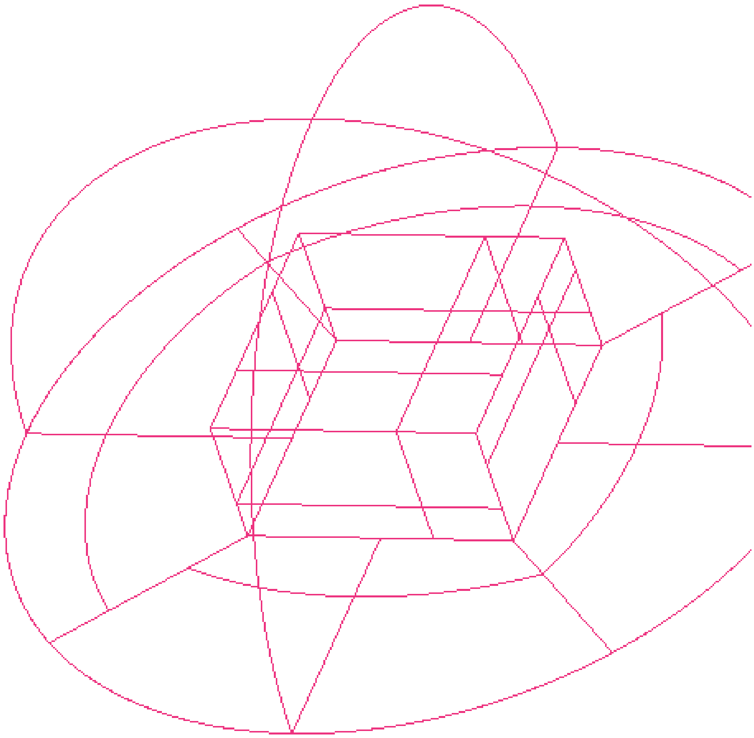
Setting the Inner block to VORFN

Generating the Mesh

b) The Blocking Strategy

The topology for this geometry is quite simple. The user will first create an O-grid around the cube and then fit the inside of the O-grid to the cube using the prescribed points of the model which is shown in Figure 3.120.

Figure 3.120 The Sphere Cube Geometry



c) Starting the Project

From UNIX or DOS window, start ANSYS ICEMCFD. File > Change working directory to the $ICEM_ACN/../docu/CFDHelp/CFD_Tutorial_Files/. Open the Sphere Cube project and load geometry.tin.

d) Creating Parts

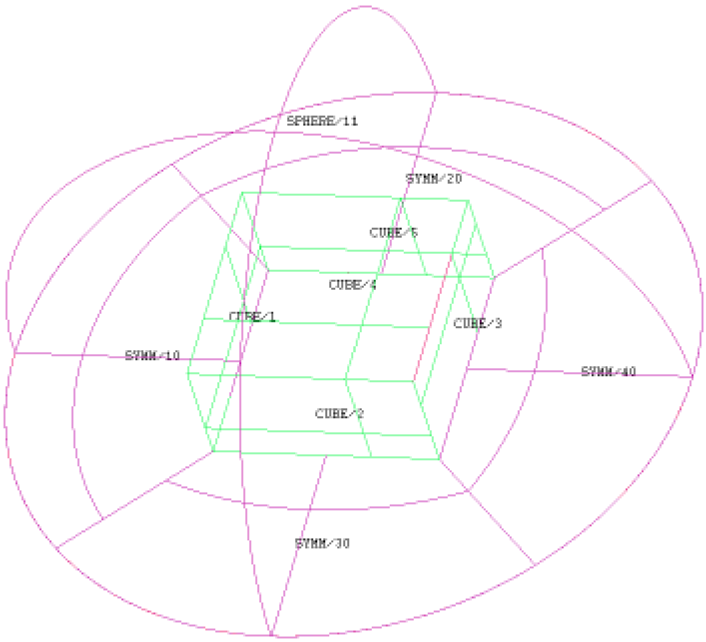
As in the 3D Pipe Junction tutorial, associate the geometry into different Parts before proceeding with the blocking.

In the Display tree, turn on Surfaces. Right mouse select Parts and select Create Part.

Type in SYMM in the Part field and select Create Part by Selection  or Select entities . Select the four surfaces on the bottom of the geometry as in Figure 3.121. Press the middle mouse button or Apply.

Similarly create new parts, SPHERE and CUBE referring to Figure 3.121 as a guide.

Figure 3.121 The Sphere cube with labeled Surfaces



For this tutorial, we will leave the curves and points in the GEOM.

e) Creating the Material Point

Select Geometry > Create Body > Material Point, enter LIVE in the Part field, select either Material Point or Select location(s) and click on two locations on the displayed geometry so that the midpoint will be located inside the volume. Press the middle mouse button or Apply. Right or middle mouse again, or Dismiss to exit the function.

Note: The use of a Material point is not actually required. However, creating one will "fix" the volume part name within the tetin file. This will avoid any problems caused by the volume name in the block file not being recognized by the tetin (geometry) file in future sessions.

Save the geometry or the project.

f) Starting Blocking

Select Blocking > Create Block > Initialize Blocks .Change Type to 3D Bounding Box (default).

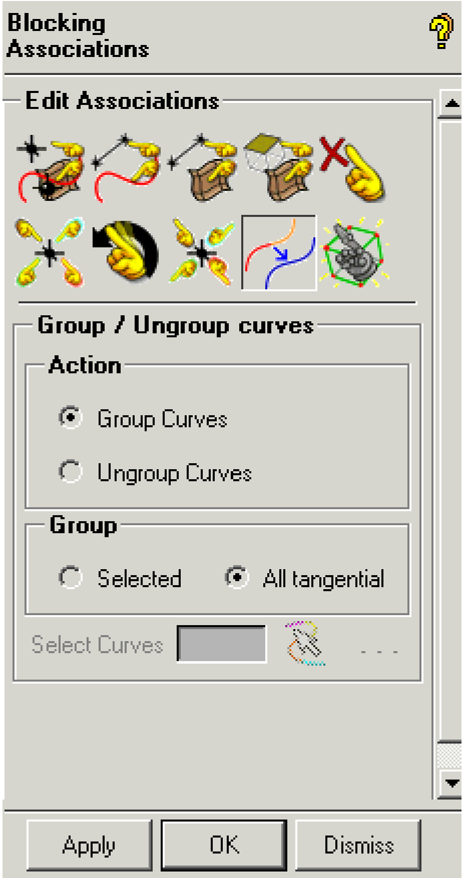
Select Part > LIVE (select the down arrow to get a pull down list of parts) and Apply.

g) Creating the Composite Curves

Even though curves can be automatically grouped while associating edges to curves, sometimes it beneficial to group them first. One such benefit is the ability to group all curves that tangentially meet (smooth transition at the ends of two adjacent curves).

Select Blocking > Associate > Group curves> All tangential as in Figure 3.122. Apply.

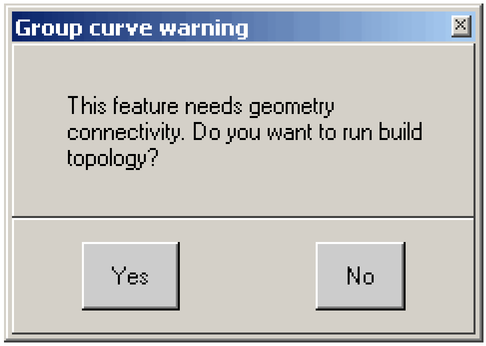
Figure 3.122
Group Ungroup curve window



This feature needs geometry connectivity so it will ask to run build topology as shown in Figure 3.123 Select Yes.

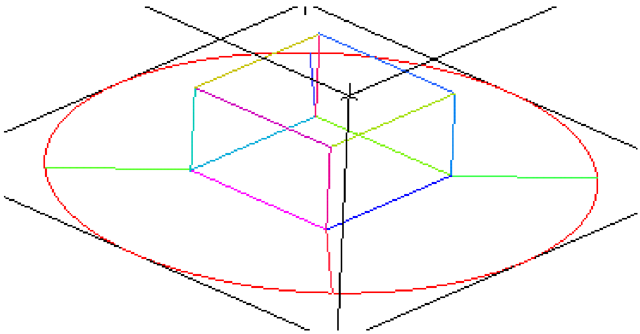
Note: Build Topology will generate a series of curves along all shared edges of surfaces. It is meant as a geometry diagnostic tool but is also used to determine logical connectivity between surfaces and to build curves and points to capture sharp features. To invoke independently, use Geometry > Repair > Build Diagnostic topology (not necessary for this tutorial).

Figure 3.123
Group curve warning window

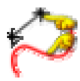


Re-Apply from the panel to group all tangential curves. All four base curves forming the circular perimeter of the hemisphere will be grouped as in Figure 3.124.

Figure 3.124
Grouping of all tangential Curves



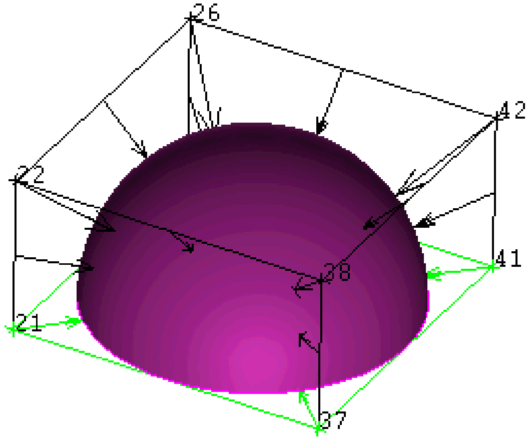
h) Projecting the Edges to Curves

Select Associate  > Associate Edge to Curve  .

Select the four bottom edges press the middle mouse button and then select the grouped circular curve. Press the middle mouse button or Apply. The selected edges will turn green.

Verify association: In the model tree turn on Surface > Solid and Edges > Show Association and view as in Figure 3.125.

Figure 3.125 Projection to the curve and sphere surface

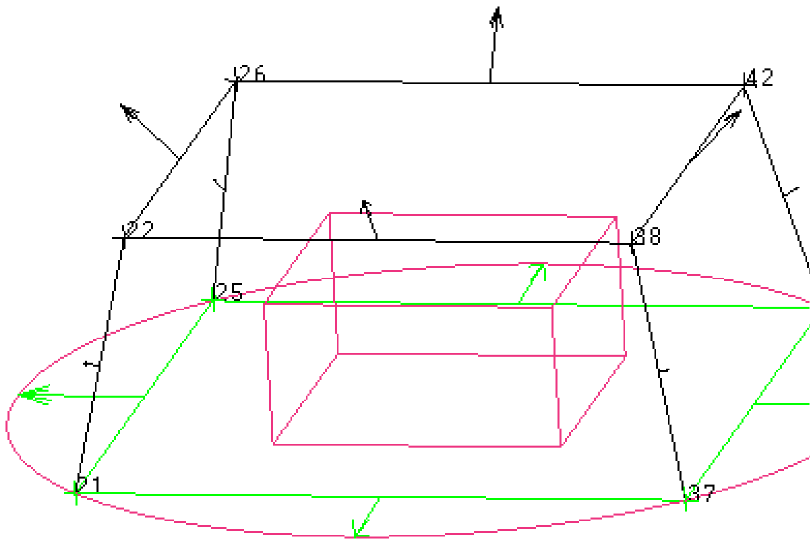


i) Moving Vertices

Select Blocking > Associate  > Snap Project Vertices  (All Visible) and Apply.

Turn off the surfaces to better view the new vertex positions as in Figure 3.126.

Figure 3.126 Vertices moved on the geometry



j) Creating the O-grid

An O-grid will be used to capture the cube as well as radially propagate the mesh onto the sphere.

Select Blocking > Split Block  > Ogrid Block .

Select face(s)  select the bottom face of the block and press the middle mouse button. Note that the block will be selected as well (Figure 3.127). Selecting the face selects both blocks on either side. The VORFN block beneath the face is not active, so a flat icon is shown instead of the block underneath.

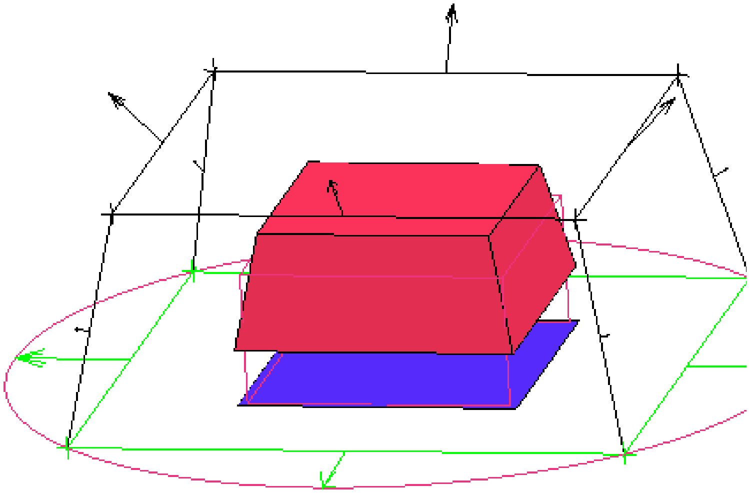
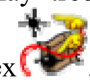


Figure 3.127 Selecting the Face for the O-Grid

Press Apply to create the half O-Grid.

k) Fitting the O-grid Using Prescribed Points

Use the central block of the o-grid to represent the cube.

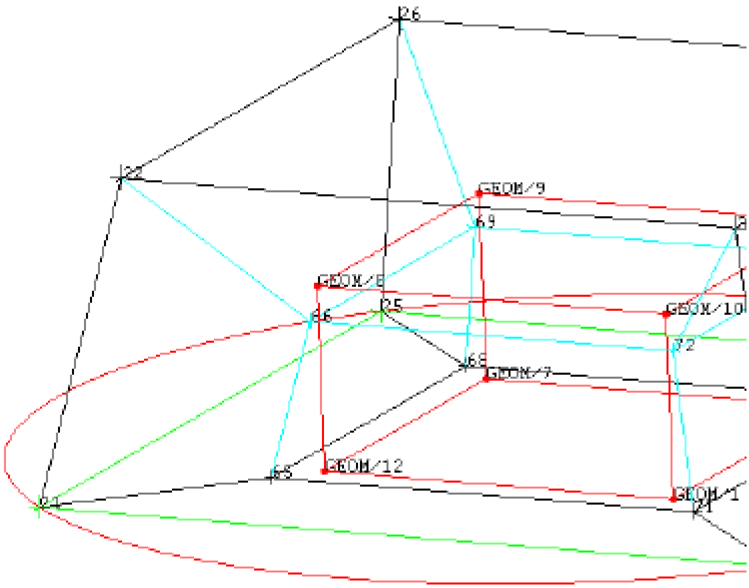
Turn on Geometry > Points in the Display tree. Select Blocking > Associate  > Associate Vertex .

Make sure Point is selected under Associate Vertex -> Entity options in the Blocking Associations panel. Select a corner vertex of the central block then select the nearest corner point to that vertex on the cube geometry. The vertex will immediately snap to the selected point.

Note: When the vertex snaps to the point selected, the point will turn red. Red designates a fixed vertex which can't be moved unless the association is changed.

Repeat to capture all eight corners. Thus, make the block fit the cube as shown in Figure 3.128. Use F9 repeatedly to toggle between selection mode and dynamic mode to reorient the view (translate, rotate, zoom) whenever necessary.

Figure 3.128 Fitting the inner block to the cube with Prescribed Points

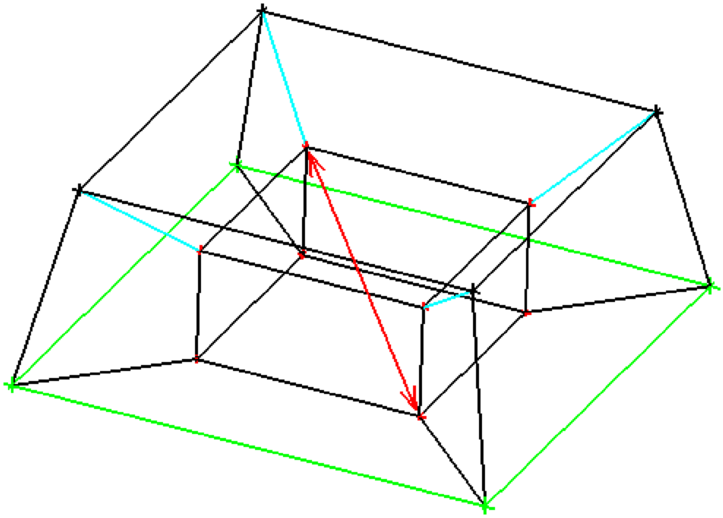


l) Discarding the Inner block

For this example, the mesh will only be generated in the volume between the SPHERE and CUBE; therefore the central block must be removed. Quite often, when multiple blocks are displayed, it is difficult to select the icon representing the block(s). One option is to select the block by selecting a pair of diagonally opposing corners.

Select Delete Block  and either select diagonal corner vertices  from the Select blocks toolbar or type Shft-D on the keyboard. Proceed to select two corner vertices as in Figure 3.129. Press the middle mouse button or Apply.

Figure 3.129 Removing the central block

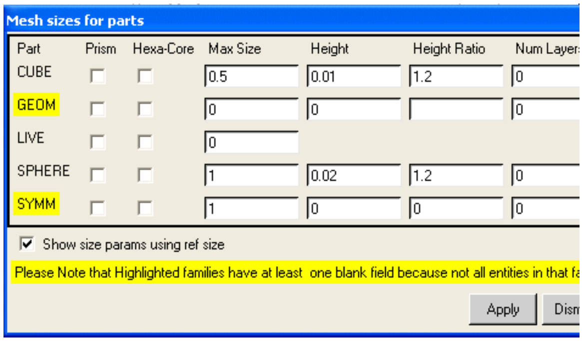


m) Generating the Mesh

In the Display tree turn off Blocking > Edges and turn on Geometry > Surfaces > Wireframe.

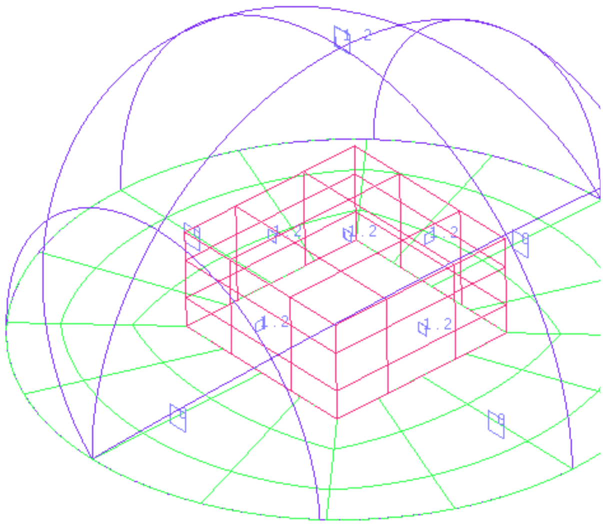
Select Mesh > Set Meshing Params by Parts . Type in the values as shown in Figure 3.130. Set a Max Size of 1 for SPHERE and SYMM, 0.5 for CUBE; Height of 0.01 for CUBE, 0.02 for SPHERE and Height Ratio of 1.2 for CUBE and SPHERE. Apply.'

Figure 3.130 Mesh Size for Part



Verify the sizes by right mouse selecting Surfaces and turn on Hexa Sizes (Figure 3.131).

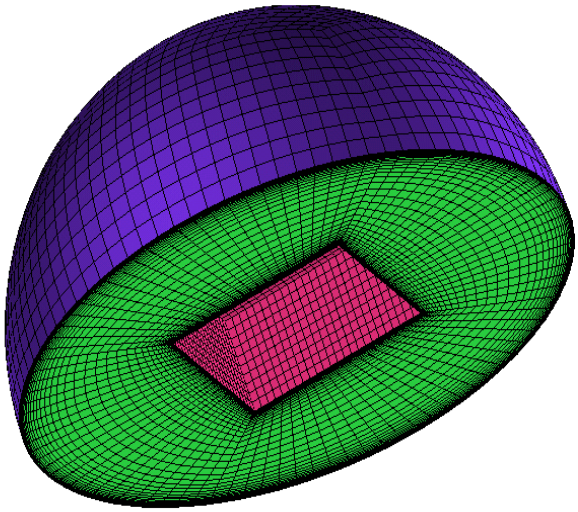
Figure 3.131 Verifying Hexa Sizes



Select Blocking > Blocking > Pre-Mesh Params > Update Sizes Make sure Method > Update All is selected (default) and Apply.

In the Display tree turn on Blocking > Pre-Mesh and (re)compute the mesh when prompted. View the mesh as shown in Figure 3.132.

Figure 3.132 Mesh after Recompute Operation



n) Viewing with Scan Planes

Viewing the volume mesh can provide another good visual diagnostic. Within blocking, this is done by means of a scan plane, where an I, J, K or radial o-grid index plane is scrolled through the volume.

Turn on Edges and Curves for reference. Turn off Pre-Mesh, then right mouse select Pre-Mesh and select Scan planes. The Scan Plane Control window will appear in the lower right hand corner of the screen as shown in Figure 3.133.

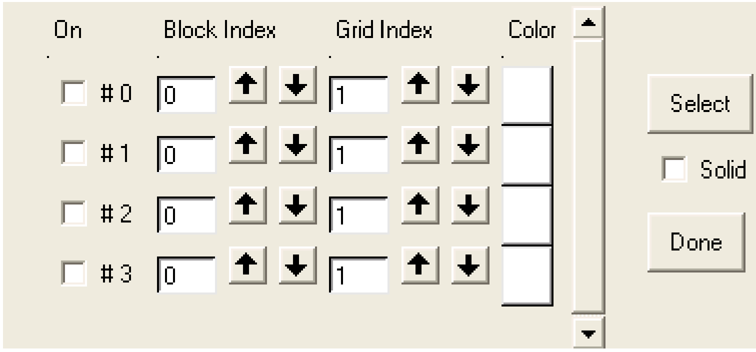


Figure 3.133 The Scan Plane Control window

First, select which index plane you wish to turn on. # 0, # 1, # 2 represents I, J, K respectively. # 3 represents the radial (o-grid) direction. In this model, I, J, K is more or less lined up with the global X, Y, Z coordinates respectively.

The scan plane isn’t planar! For instance, turning on # 0 will display all the nodes of constant I index, not constant X coordinates.

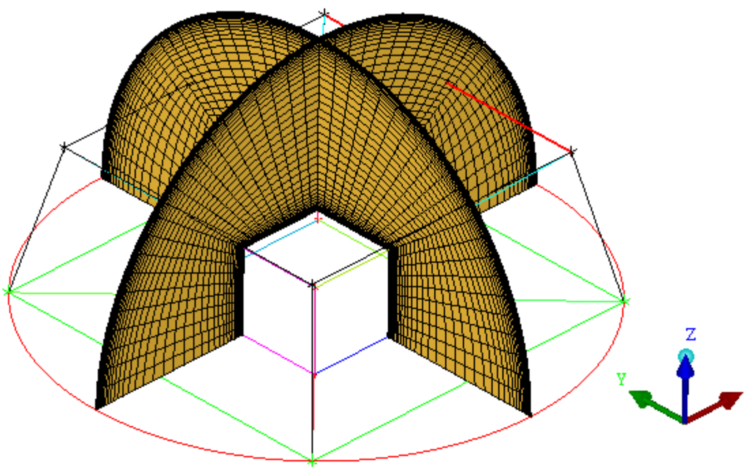
To move the scan plane toggle the up/down arrows underneath either Block Index or Grid Index. Block Index will increment one block at a time whereas Grid Index will increment one node at a time.

Select will turn on the index plane perpendicular to any selected edge.

Turn on # 0. Select the up arrow within # 0 row, Grid Index column and keep toggling until the scan plane appears about half way through the model Note: When you toggle from 0 to 1, notice how two planes are visible at the same time. Along the radial (o-grid) block, I is equal to 1 throughout the entire block.

Pick Select from the Scan Plane Control window and select one of the edges parallel to the current Scan plane, an edge lined up along Y. This will select a J edge and the resulting scan plane will be perpendicular to that edge and will display constant J nodes (Figure 3.134). Note that the # 1 column is automatically turned on in the window.

Figure 3.134 Scan planes of the final mesh



Continue to select and toggle back and forth through the other planes as well.

Press 'Done' to exit the Scan Plane functions.

Check Pre-Mesh Quality  and once satisfied convert to unstructured mesh: Right mouse select Pre-Mesh and select Convert to Unstruct Mesh.

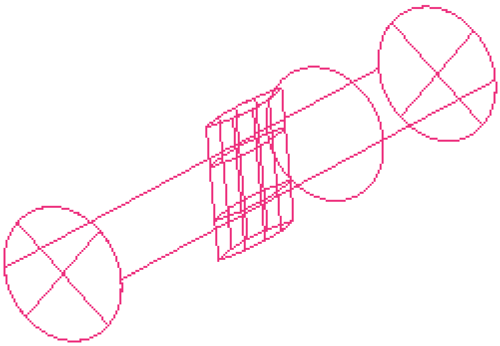
Save the project within the File menu. This will save the tetin, unstructured mesh, blocking and project settings files all beginning with the project name.

Exit or continue on with the next tutorial

3.2.6: Pipe Blade

Overview

This tutorial example uses the “Collapse” function to create a degenerate topology in a Conjugate Heat transfer problem around a blade located in the center of a cylindrical pipe.



a) Summary of Steps

The Blocking Strategy

Starting the Project

Creating Parts in the Mesh Editor

Starting Blocking

Using Prescribed Points to Fit the Blocking

Splitting the Topology Using Prescribed Points

Collapsing Blocks to Represent the Blade Material

Edge to Curve Association on the Blade

Moving the Vertices

Generating the O-grid

Defining Surface Parameters for the Mesh

Defining Edge Parameters to Adjust the Mesh

Checking mesh quality for determinants and angle

Saving before Quitting

b) The Blocking Strategy

In this lesson, the blade is regarded as a Solid region, while the region surrounding the blade is regarded as the Fluid region. Using Block Splitting at “Prescribed point”, the user will generate a Hexahedral Mesh for both of the regions, so that the topology of the solid region is a degenerate ‘Hexahedral’ mesh.

Before the user employs the Collapse function for his/her own applications, confirm that the solver accepts degenerated hexas (for a structured solver) or penta_6 elements (prism) for an unstructured solver.

Note: Settings >Selection>Auto pick mode should be turned OFF for ANSYS ICEM CFD to behave exactly as this tutorial describes.

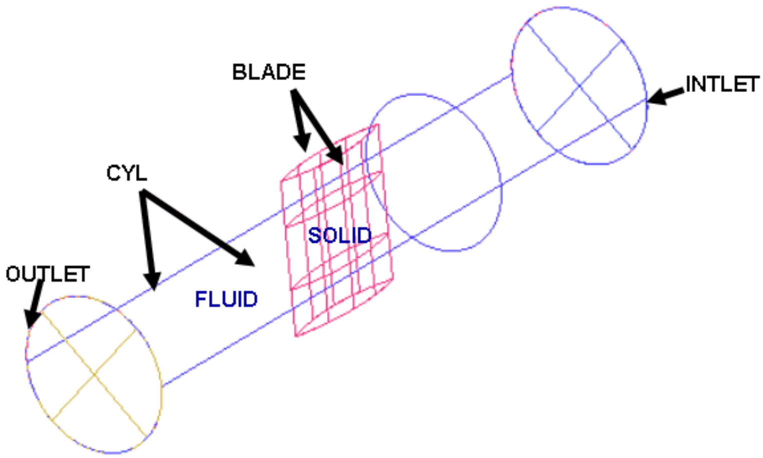
c) Starting the Project

From UNIX or DOS window, start ANSYS ICEMCFD. Go to File > Change working directory. Change the directory to $ICEM_ACN/../docu/CFDHelp/CFD_Tutorial_Files>PipeBlade. The top ICEM Installation directory is referred to as “$ICEM_ACN” here. Go to File>Geometry>Open geometry and choose the tetin file geometry.tin.

d) Creating Parts in the Mesh Editor

Right click in the Display Tree on Parts > Create Part to create different Parts and assign the different surface of the geometry to the appropriate part. Refer to Figure 3.135 for the Surface part assignments.

Figure 3.135 The Pipe Blade configuration



e) Surface Parts

After the Pipe Blade project is open, activate the Points and Surfaces from the Display Tree. Switch on Points > Show Points Names.

Begin the Surface part reassignment by changing the region enclosed by GEOM/4 - GEOM/7 to the part INLET.

The region that is denoted by GEOM/0 - GEOM/3 should be reassigned to the part OUTLET.

The Surface defining the Cylinder pipe will be placed in the Surface part, CYL.

The surfaces belonging to the solid blade in the middle of the cylinder should be classified as BLADE.

When all of the Surface parts have been assigned (INLET, OUTLET, CYLIN, BLADE), press the middle mouse button to exit from continuous mode.

f) Curve Parts and Point Parts

For this tutorial, we will leave the curves and points assigned to the initial part **GEOM**.

g) Creating the Material Points

Select Geometry > Create Body > Material Point
Enter FLUID in the Create Body window that appears. The material point that will be created will help us to keep the FLUID region separate from the SOLID region, but is not necessary since blocks can simply be created in the FLUID part rather than creating a material point.

With the left mouse button, select two locations on the opposite sides of the cylinder, shown in Figure 3.135. Note that the FLUID material point should not be within the BLADE. If tetra meshing, this location would be important. With Hexa meshing, it is not. Press the middle mouse button to accept the selection, and press Apply and the Body name FLUID should appear within the geometry (midway between the selected locations). Rotate the model to confirm that FLUID is in an appropriate location.

Now enter SOLID as the new Part Name in the Create Body window.

Press the location selection icon and select two locations on the blade surfaces so that the midpoint will be inside of the blade. Press the middle mouse button to accept, and press Apply. After accepting this Parts assignment, dynamically rotate the model to confirm that SOLID is inside the blade.

When this is complete, all components of the Geometry should now have part name assignments.

Delete any Empty Parts: From the Display Tree, right mouse select on Parts > Delete empty Parts.

File > Save Project As to save the updated model before continuing on in this tutorial. Give the project any name you choose.

h) Blocking

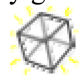
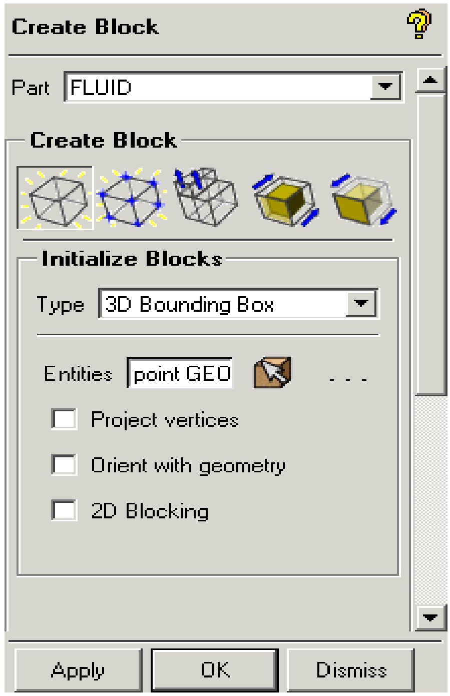
Initialize blocking, which will create the first block, by going to Blocking > Create Block  > Initialize Block . The Create Block window will open as shown in Figure 3.136

Figure 3.136
Create block window



Select the block Type as 3D Bounding Box from the pull down arrow. Name the Part as Fluid. Press Apply without selecting anything, and the initial block will be created around the whole model.

To fit the Initialized Blocking more closely to the geometry, the user will associate vertices to points.

Select Blocking > Associate  > Associate Vertex  and the window shown in Figure 3.137 will open. Toggle ON Geometry > Points and right mouse click on Vertices > Numbers under Blocking from Display Tree. Then toggle ON Blocking > Vertices.

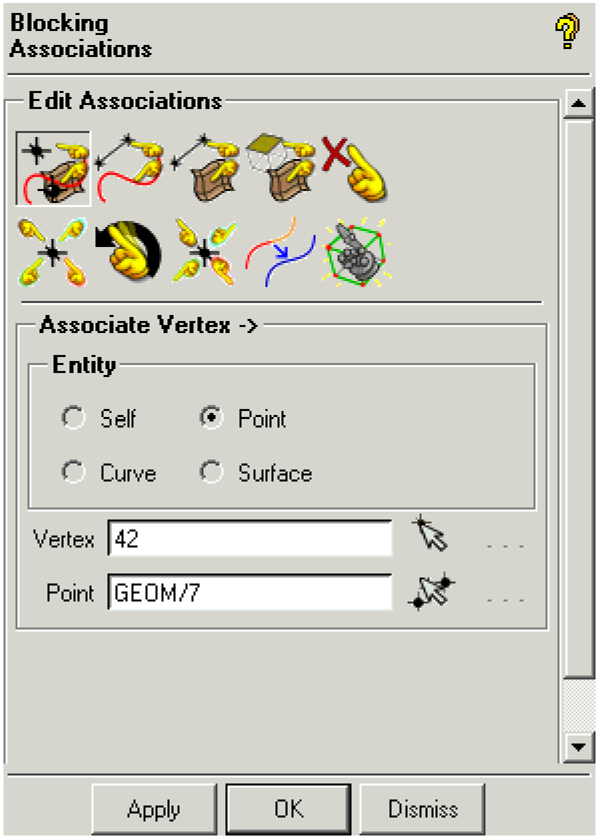
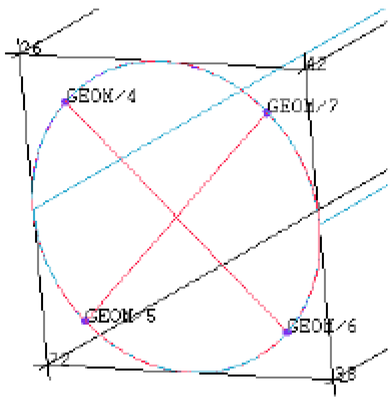


Figure 3.137
Associate vertex window

Select Point under Entity.

Press the vertex selection icon  and select Vertex 42. Press the point selection icon  and select Point GEOM/7 and press Apply to associate them as shown in Figure 3.138. Similarly, associate the other vertices and points for the inlet and outlet so that after completion the geometry should look like as shown in Figure 3.139

Figure 3.138 Moving the vertices



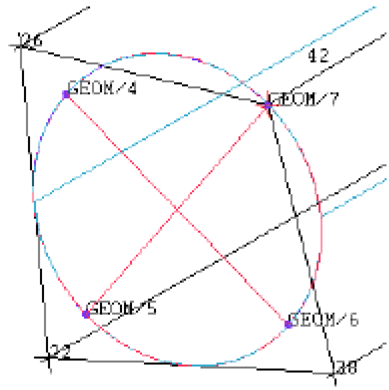
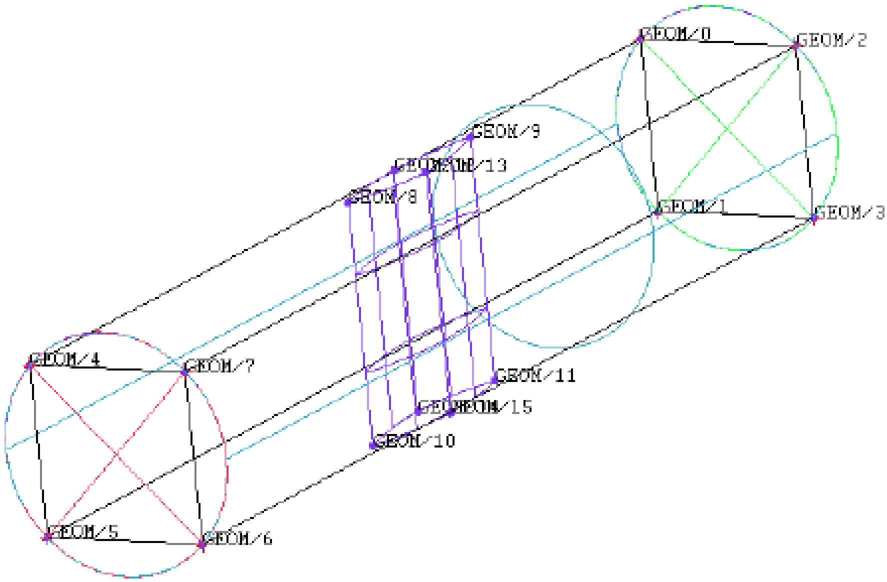


Figure 3.139
Geometry after associating all vertices to corresponding points

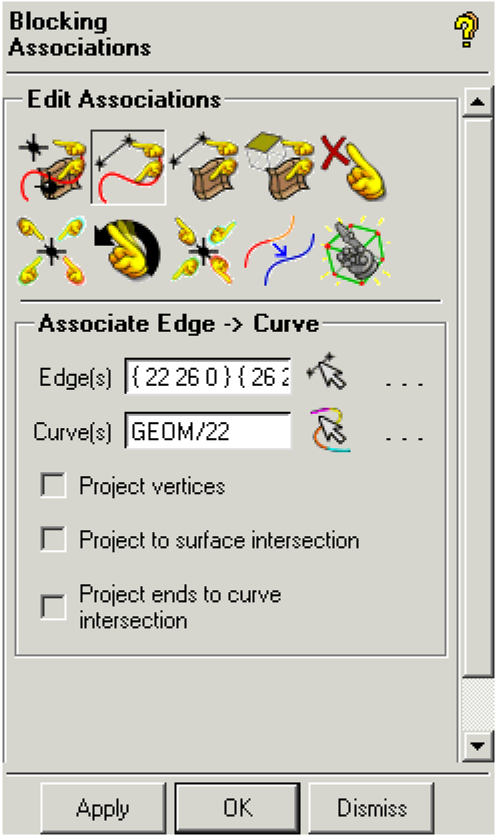


Note: When possible, the Block vertices on any circular geometry should be placed so that edges are equal in length and the angles between edges are 90 degrees. This amounts to vertices being placed at 45, 135, 225, and 315 degrees around the circle. This results in the best mesh quality.

i) Associating edges to curves

Select Associate >Associate Edge to Curve .The window shown in Figure 3.140 will open. Press the edge selection icon then select the four edges shown in Figure 3.141 and press the middle mouse button. Then press the curve selection icon and select the four curves shown in Figure 3.140 and press the middle mouse button. Then press Apply. Notice that the block edges then transform from “white’ to ‘green’, confirming their association with the curve. Also notice that the four curves become one color, indicating that they have been grouped into one curve.

Figure 3.140
Association window



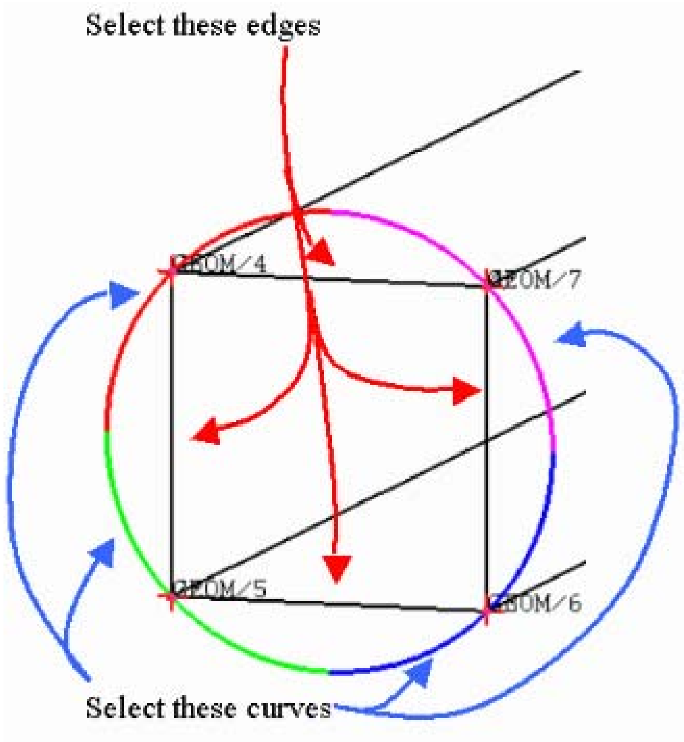
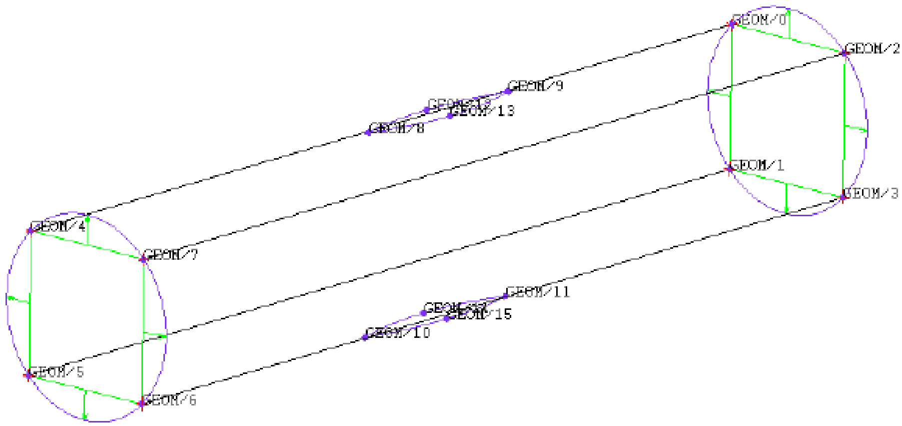


Figure 3.141 Edges and Curve selection for association

Similarly, associate the four edges on the other circle to the corresponding four curves. To see a confirmation of these associations, right mouse click on Blocking > Edges > Show Association in the Display Tree. The geometry should look as shown in Figure 3.142

Figure 3.142 The Edge Projection



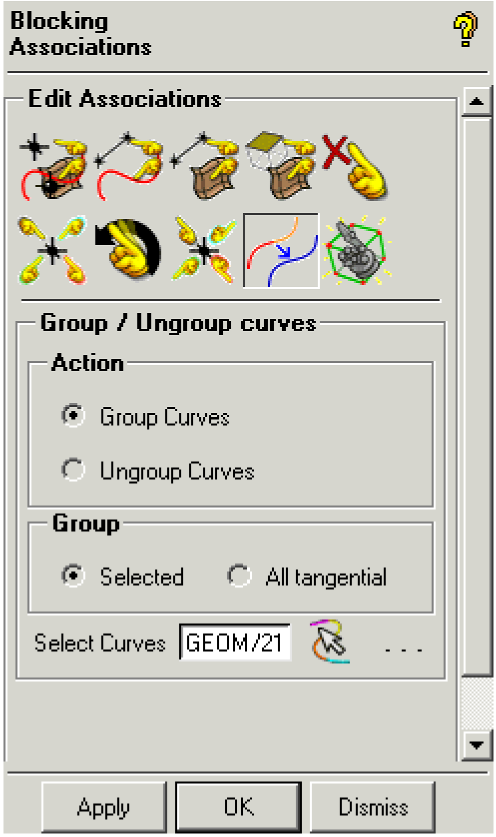
Note: If the edges lie on the geometry, as is the case with longitudinal edges, the projection arrows are not shown. By default, all external edges are surface-associated to the nearest active surface and appear as white. The association can be set to this default using Associate  > Associate Edge to Surface . This operation is useful to correct any Edge to Curve Association mistakes. All internal edges, by default, have no association, and appear as blue. You can set this association, which is really deleting an association, by pressing .

j) Grouping curves

Note: This section does not need to be performed on the model, but it shows the user how to manually group curves.

Select Blocking > Associate  > Group curves. It will open the window as shown in Figure 3.143.

Figure 3.143
Group curve window



Select the four curves corresponding to OUTLET as shown in the figure and press Apply to group them.

k) Splitting the Topology Using Prescribed Points and Screen Select

The following steps instruct the user to split the block in the 'k' and 'j' directions around the blade, thus creating further blocking topology for the blade. The k-direction splits will be created through the prescribed point method, while the j-direction splits will be made by visual judgment.

Press View > Top, then Fit Window 

Turn off Vertices at this stage.

Choose Blocking > Split Block  >Split Block  and it will open the window as shown in the Figure 3.144. Choose All visible and Split method as Prescribed Point.

Select the edge selection icon  then select one of the two vertical edges. Press the point selection icon  then select the Prescribe point, GEOM/9 and press Apply.

Similarly, make another split using the vertical edge and Prescribed Point, GEOM/8.

Similarly, make another horizontal split through the prescribed point GEOM/12. The final result will have three horizontal splits as shown in Figure 3.145.

Note: Make sure that the Edge that is selected lies within the range of the Prescribed Point that will be selected.

Figure 3.144
Split block window

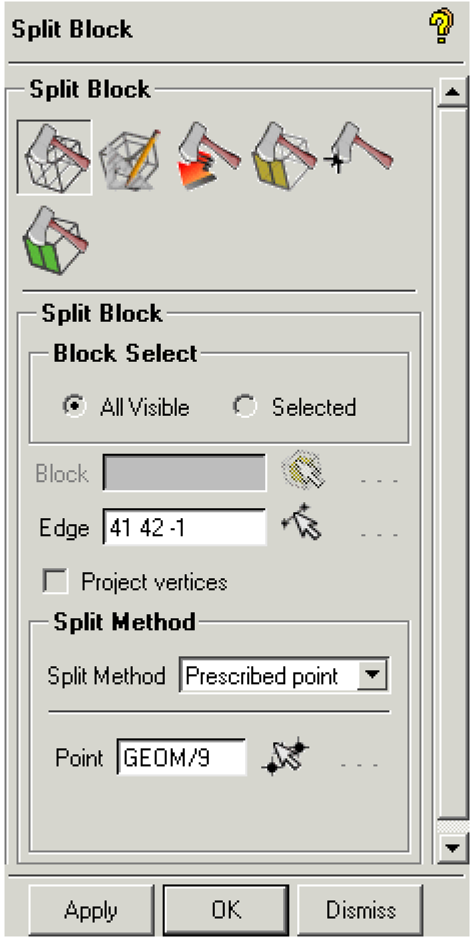
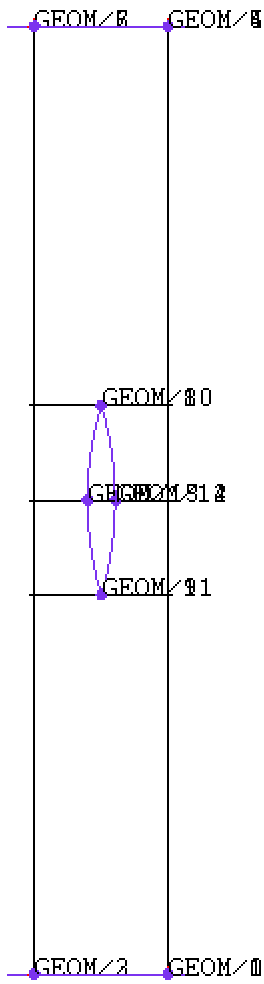


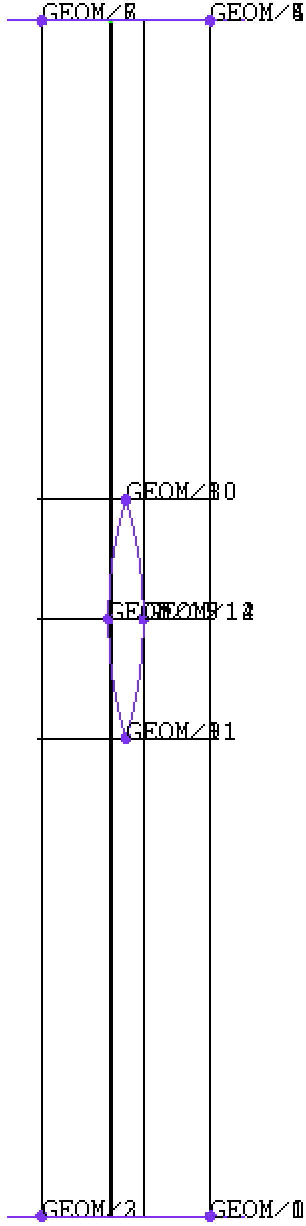
Figure 3.145 Make the horizontal splits in the block



These are the splits in the 'k'-direction. The next set of splits will be in the 'i' direction.

Now select the Split method as Screen select. Press the edge selection icon and select any of the horizontal edges to create a vertical split. If Settings>Selection>Auto pick mode is OFF, press Apply, and it will ask for a location on the screen to split through. Select on a curve or edge on any location that is vertically in line with the right side of the blade. If Auto pick mode is ON, you should left mouse click on the edge and hold the button while dragging the split to where you want it. Press the middle mouse button to complete the split operation. Then use the same method to create another vertical split on the left side of the blade. These two splits should show as in Figure 3.146.

Figure 3.146
Horizontal splits on blade sides



Note: Every time a block Split is performed, the Index control is updated. After the splits are complete, the new range of the K index will be from 0-6.

l) Collapsing Blocks to Display the Blade

In this section, the Collapse feature is introduced to create degenerate blocks for the blade.

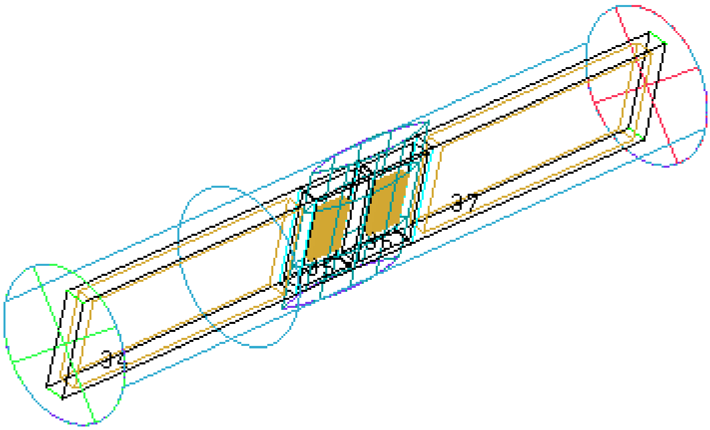
For clarity in these operations, right mouse click in the Display tree on Blocking>Index Control. Change the Index control for the 'I' dimension so that the Min is 2 and the Max is 3. Turn OFF the Points from the Display window. The restricted topology consists of four blocks, where the two center blocks belong to the blade.

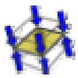
Before collapsing the blocks, change the Part family of the two center blocks to SOLID, the material representing the blade.

Right mouse click on SOLID>Add to part underneath Parts in the Display Tree, and it will open the Add to Part window.

Select Blocking Material, Add Blocks to Part , and select the blocks of the blade as shown in Figure 3.147, then press the middle mouse button. Press Apply to complete the operation.

Figure 3.147 Assigning the blade blocks



Now select Blocking > Merge Vertices  > Collapse Block .

Choose the edge that should be collapsed. In this case it is the shortest edge (shown in Figure 3.148) of the selected blocks. Select the two blocks shown in Figure 3.148. Press Apply to Collapse the blocks.

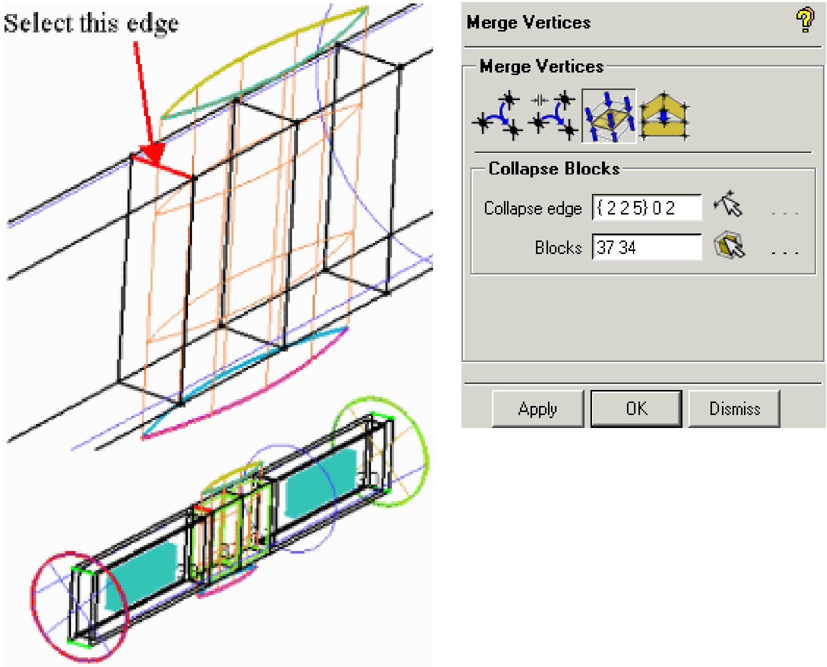


Figure 3.148 Collapsing the blade Blocks

After collapsing we get the model as shown in Figure 3.149.

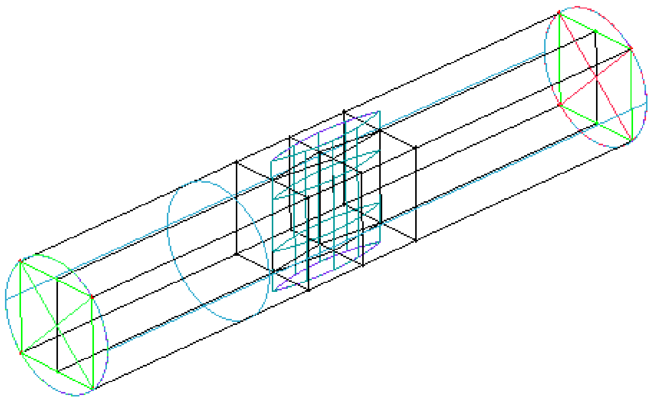


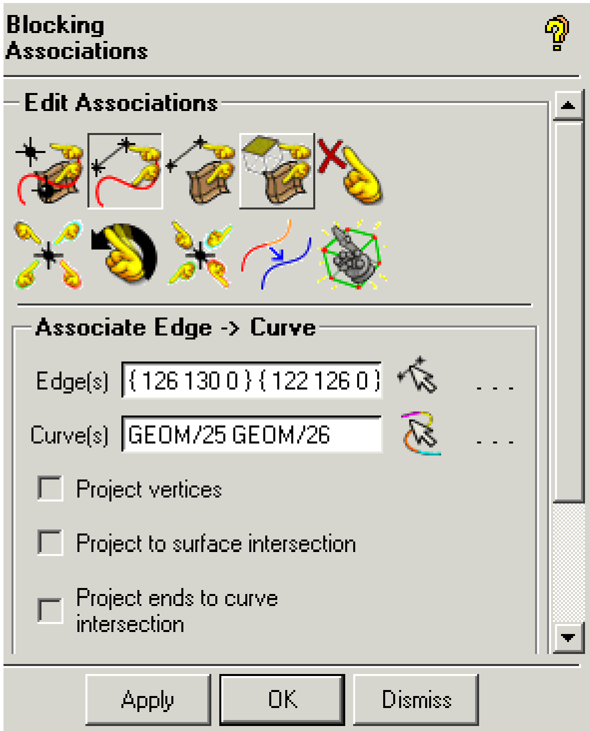
Figure 3.149 The Collapsed Blocking

m) Edge to Curve Association on the Blade

Choose Blocking> Associate  >Associate Edge to Curve  .The Associate edge to curve window will open as shown in Figure 3.150.

Note: Make sure Project Vertices is disabled.

Figure 3.150 Association Edge to Curve Window



You should associate the Edges and corresponding blade curves as shown in Figure 3.151.Do this to the top and bottom of the blade, on both sides.

After associating, Switch on Blocking > Edge > Show Association from the Display Tree. The geometry should look as shown in Figure 3.152

Figure 3.151
Blade edges to be association to curves

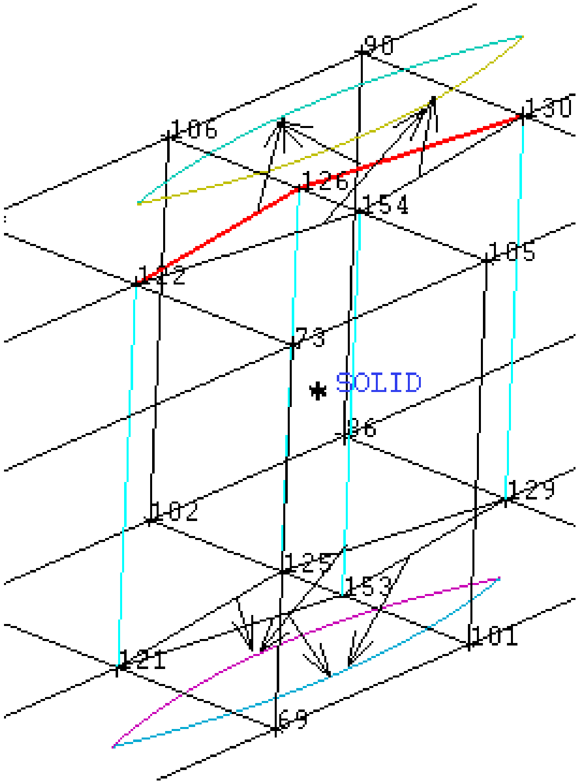
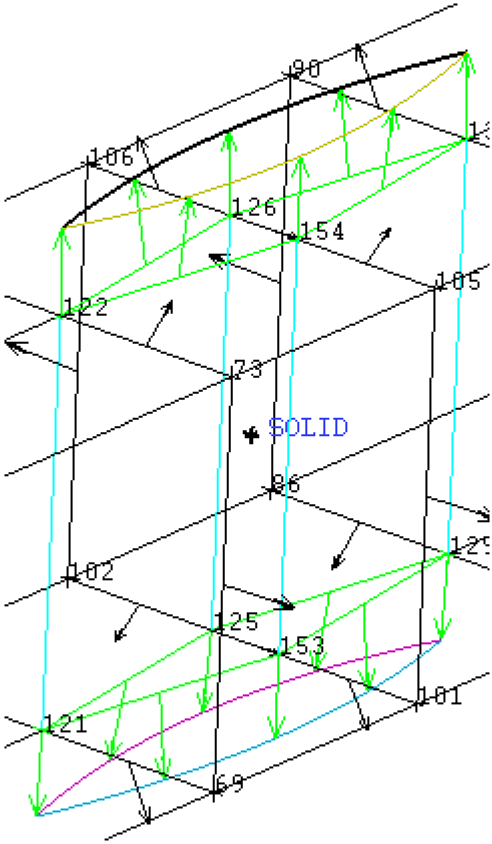


Figure 3.152
Blade edges
Associated to curves



n) Moving the Vertices

This section shows the user how to move all the associated vertices onto the geometry in one step.

Snap the appropriate block vertices onto the geometry by selecting Associate > Snap Project Vertices .All Visible should be toggled ON. Then Press Apply.

Switch off Edges > Show Association. All the vertices belonging to blade, inlet and outlet are moved to the locations as shown in Figure 3.153.

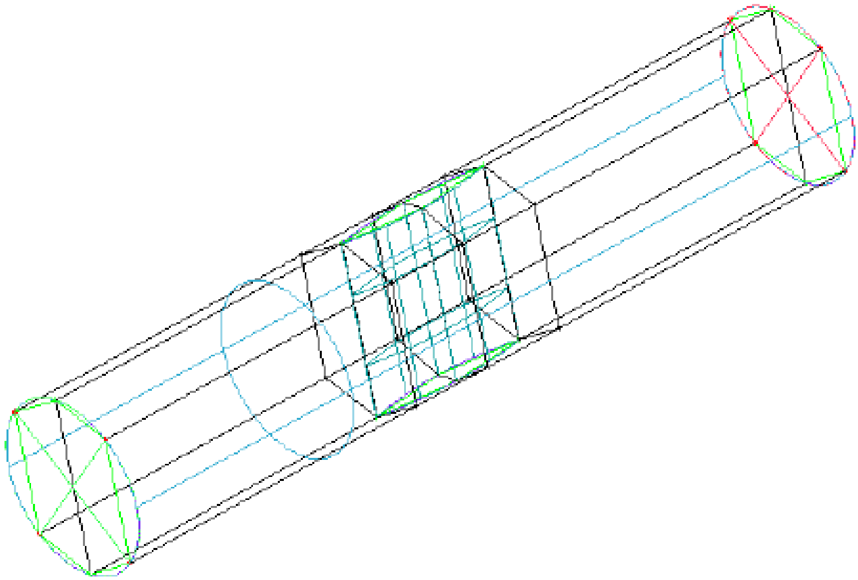


Figure 3.153 The final positions of the vertices before the O grid

o) Vertex Color Distinction

Notice from this lesson and from previous lessons, that the movement of the vertices is restricted to the associated Curve. The colors of the vertices indicate their associations and degrees of freedom.

Vertices associated with Prescribed Points are red and are fixed at a point.

Vertices associated to a curve are green and can be moved on the associated curve.

By default, all the vertices lying on the block material boundary are white and are free to move on any surface.

Additionally, internal surfaces are blue and can be moved along the blue block edges to which they are connected.

p) Generating the O-Grid

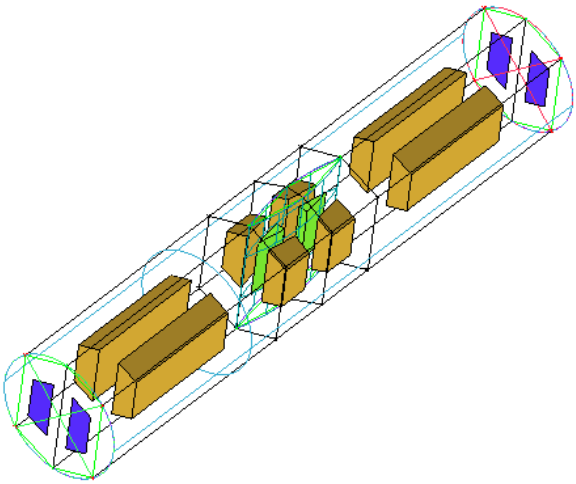
If the pre-mesh is generated at this point, the existing blocking would result in skewed cells on the four 'corners' of the pipe. Converting the existing H-Grid type topology to an O-grid type topology inside the pipe will produce a mesh that is low in skewness, with orthogonal grid on the pipe walls. The following steps will improve the overall mesh quality.

Press Blocking >Split Block > O grid Block

Press and select all the Blocks of both the FLUID and SOLID regions since the O-grid will be added in the entire pipe as shown in Figure 3.154. Press the middle mouse button to accept.

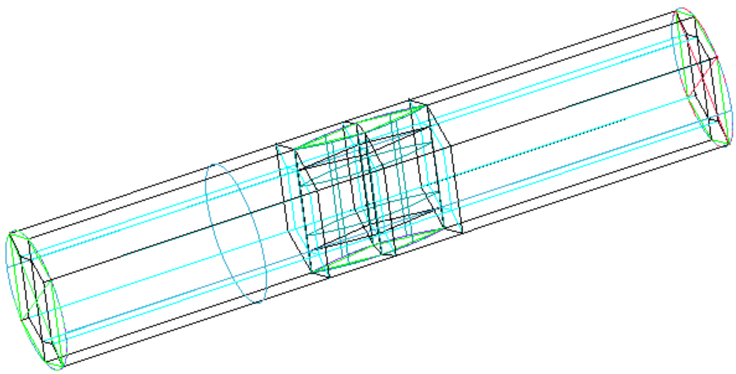
Similarly, press and select the two INLET faces and two OUTLET faces as shown in Figure 3.154. Press the middle mouse button to accept, and Press Apply to create the O-grid.

Figure 3.154 Add the faces of the outlet and inlet to O-grid



After creating the O-Grid, the blocking will appear as shown in Figure 3.155.

Figure 3.155 The O-grid

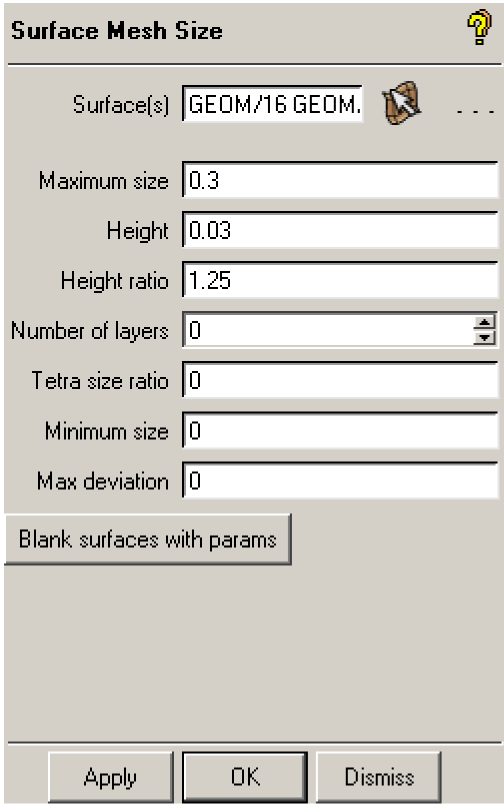


q) Defining Surface Parameters for the Mesh

In this step, the user will define node distributions on the blocking using surface parameters. Surfaces should be turned ON in the Display Tree so they can be selected from the screen.

Select Mesh> Set Surface Mesh Size  and select the surface selection icon . Then select all the surfaces by box selecting the entire model or pressing "a." Enter the Maximum Element size as 0.3, Height as 0.03 and Ratio as 1.25, as shown in Figure 3.156

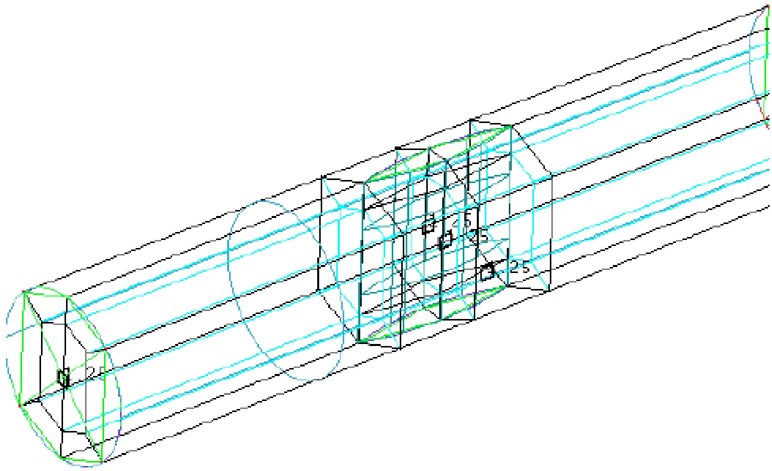
Figure 3.156 Surface mesh size window



Press Apply to assign the surface parameters. Display the surface parameters by right mouse clicking in the Display

Tree on Geometry> Surface > Hexa Sizes. The surfaces will show hexa icons as shown in Figure 3.157.

Figure 3.157 The surface parameters



Switch OFF Surface > Hexa Sizes.

r) Defining Edge Parameters to Adjust the Mesh

Although it may be enough to define the meshing with surface parameters, the mesh quality of more complex models can be improved by defining additional edge parameters. Perform these next steps to redistribute points along the diagonal (radial) edge of the O-grid.

For the convenience of selecting the edges, right mouse click in the Display Tree to turn ON Vertices > Numbers and Edges > Bunching. Then make sure Vertices in ON. Zoom-in on the OUTLET area of the blocking.

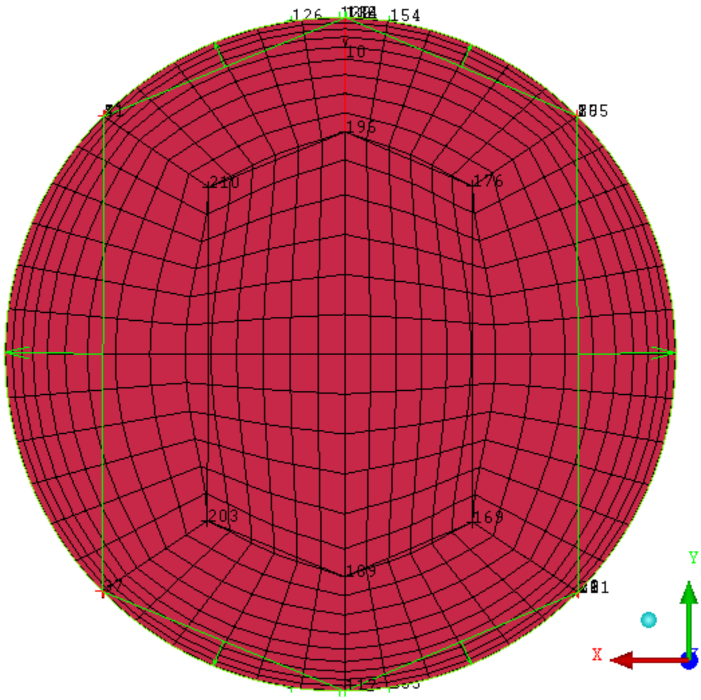
Select Blocking >Pre-mesh Params >Update Sizes .Make sure Update All is toggled on, and Press Apply.

This will compute the node distributions on the blocking edges from the surface parameters.

Turn 'ON' Blocking > Pre-Mesh from the Display Tree. Press Yes, when it says, Mesh is currently out of date – recompute?

Right click on Blocking > Pre-Mesh>Solid & wire in the Display Tree to display the mesh in Solid/Wire for better Visualization. The mesh will look like as shown in Figure 3.158 when viewing the OUTLET.

Figure 3.158 Mesh before changing mesh parameters



The mesh is denser at the walls. The near wall elements will have the same initial height that was set on the surface parameters, which was 0.03. It may be desirable to have denser near-wall spacing.

Select Blocking >Pre-mesh Params >Edge Params and you will see the window shown in Figure 3.159. Turn OFF Blocking > Pre-Mesh so the edges can be easily seen and selected. Select any of the “radial” edges. These are the edges created by the O-grid that are oriented radially in relation to the grid lines that run circumferentially around the tube. Or you can select the same edge shown in Figure 3.159, which is the blocking Edge 196-118. Set Spacing1 to 0.015, which is half the previous value. Set Spacing2 to 0, which will allow it to go as large as possible. Increase the number of nodes to 13 so the Ratio1 (1.25) can be met. Enable ‘Copy Parameters’ and select Method ‘Copy to Parallel edges’ to duplicate these settings on parallel edges in the blocking. Then press Apply.

Figure 3.159
Setting edge meshing parameters

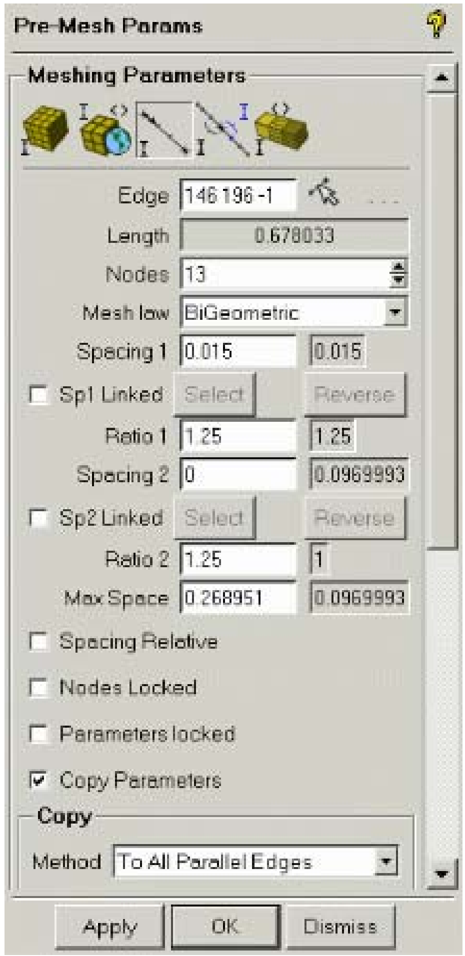
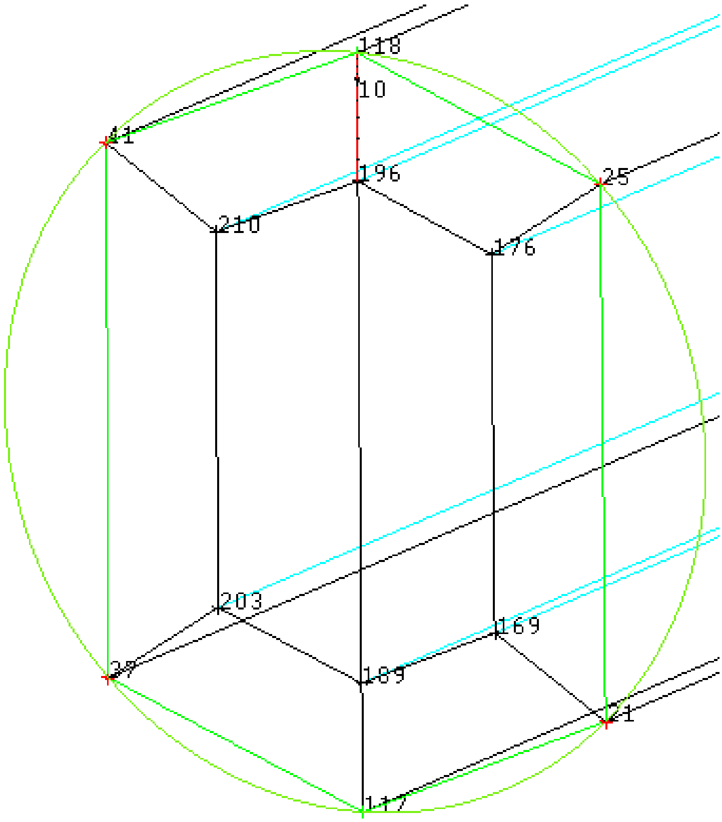


Figure 3.160 Selection of edge for changing Parameters



Note: Spacing1 is the first element size at vertex 118 while spacing2 is the first element size at vertex 196. Side 1 and Side 2 are indicated by the direction arrow that displays on the edge after it is selected.

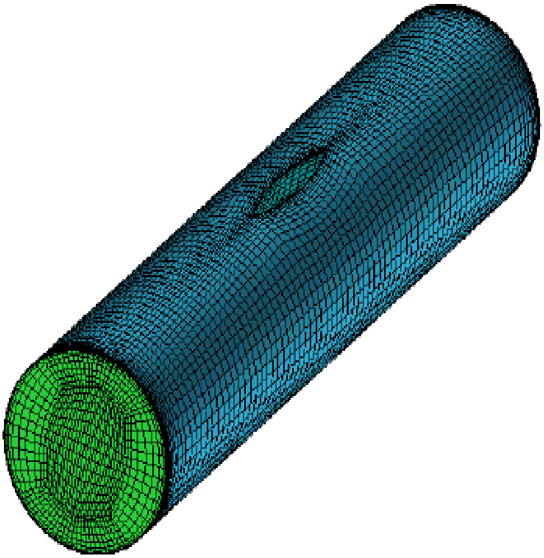
Switch OFF Edges > Bunching in the Display Tree.

Switch ON Blocking > Pre-Mesh in the Display Tree. If you right click on Blocking > Pre-mesh, you should see Project Faces checked ON by default. Choose Yes when asked to recompute the mesh. Switch OFF Geometry, Vertices and Edges in the Display Tree.

Turn off the SOLID volume part name from the Display Tree and right click in the Display Tree to turn on Blocking

> Pre-mesh > Solid & Wire if it is not already on. See Figure 3.161.

Figure 3.161 The final mesh displayed in Solid & Wire



s) Checking mesh quality for determinants and angle

To check the mesh quality, select Blocking >Pre-mesh Quality Histogram . The window shown in the Figure 3.162 will open. Select the criterion as Determinant (3x3x3) and enter the Min-X value 0, Max-X value 1, Max-y height 12 and Num of bars 20. Press Apply. The histogram shown in Figure 3.162 will display at the lower right.

A value of determinant greater than 0.2 is acceptable for most commercial solvers.

Figure 3.162 Pre-mesh quality window while selecting Determinant 2x2x2

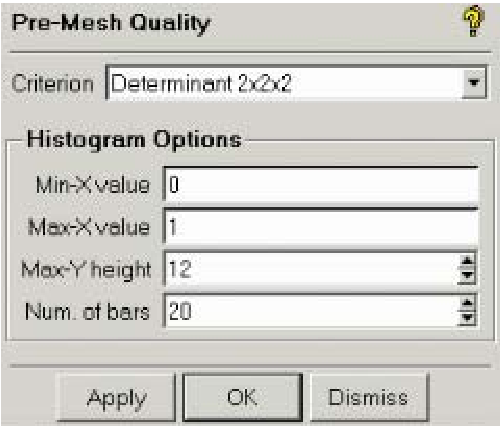
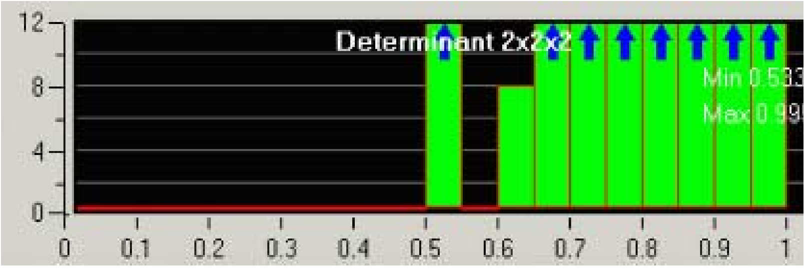
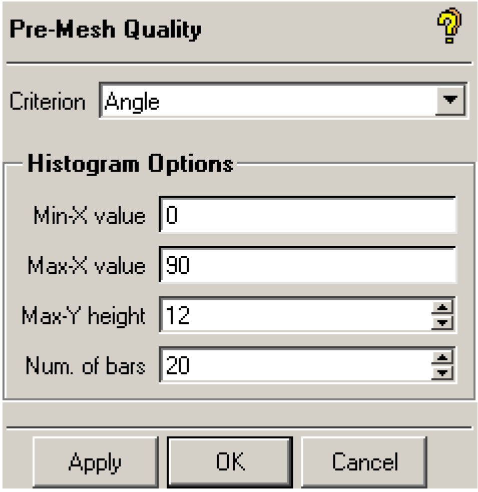


Figure 3.163 Histogram showing Determinant 2x2x2



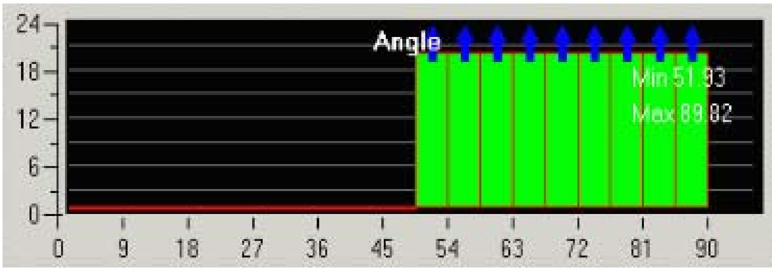
Then, in the Pre-Mesh Quality window at the upper left, select Angle from the Criterion pull down. Enter the values as shown in Figure 3.164 and press Apply. A new histogram will appear for the internal angles of elements as shown in Figure 3.165.

Figure 3.164 Pre-mesh quality Window while selecting Angle



An angle greater than 18 degrees is acceptable for most commercial solvers.

Figure 3.165 Histogram showing Angle



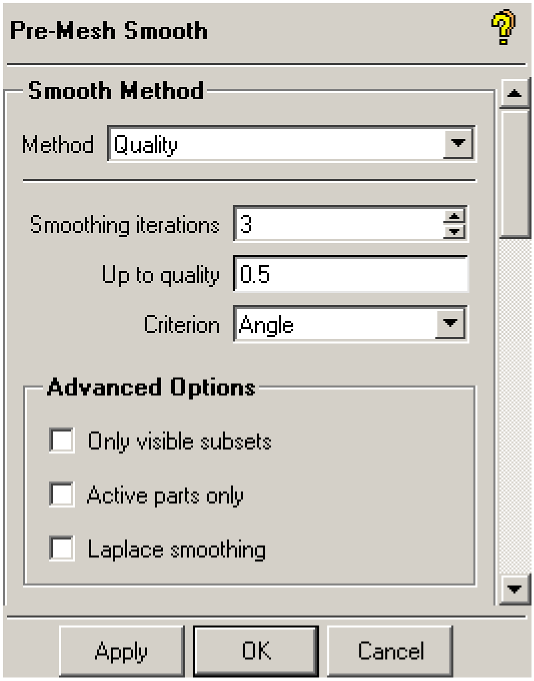
Note: As taught in the 3DPipeJunct example, to display cells of a particular determinant or angle value, select a histogram bar and then select **Show**. Cells within that range will be highlighted. The user should then inspect the elements and decide on a solution. In most cases, block vertices can be moved or edge parameters can be changed to improve the area.

t) Running Pre-mesh smoother

Before converting the Pre-mesh to an unstructured or structured mesh, the user may choose first to smooth the mesh.

Select Blocking > Pre-mesh Smooth.The Pre-mesh smooth window will then appear. Select the Method as Quality. Select the Criterion as Angle and enter Smoothing iterations 3 and Up to quality 0.5 as shown in Figure 3.166

Figure 3.166
Pre-mesh smooth window



Press Apply to smooth mesh. Changes in the minimum angle of the mesh can be seen in the histogram as shown in Figure 3.167. The node position changes made by the pre-mesh smoother will not be saved to the blocking. So reloading the blocking and computing the mesh will always produced the mesh before smoothing. So at this point, you should not recompute the mesh.

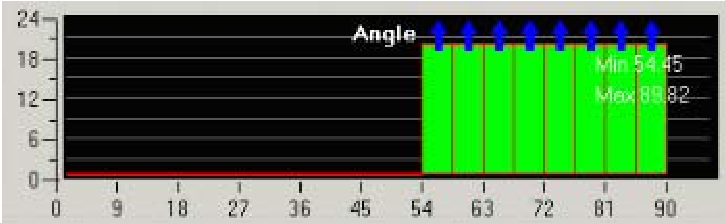


Figure 3.167 Histogram after running smoother

u) Saving

Select File > Blocking > Save blocking As and enter a name, such as b1.blk. Saving the blocking will allow the user to change any meshing parameters in the future by reloading the blocking onto the geometry.

To write the mesh in an unstructured format, right mouse click in the Display Tree on Blocking>Pre-mesh>Convert to Unstruct Mesh. This will write the default name "hex.uns" to the working directory, and immediately load the mesh. To save the mesh to a different name, the user can then select File>Mesh>Save Mesh As.

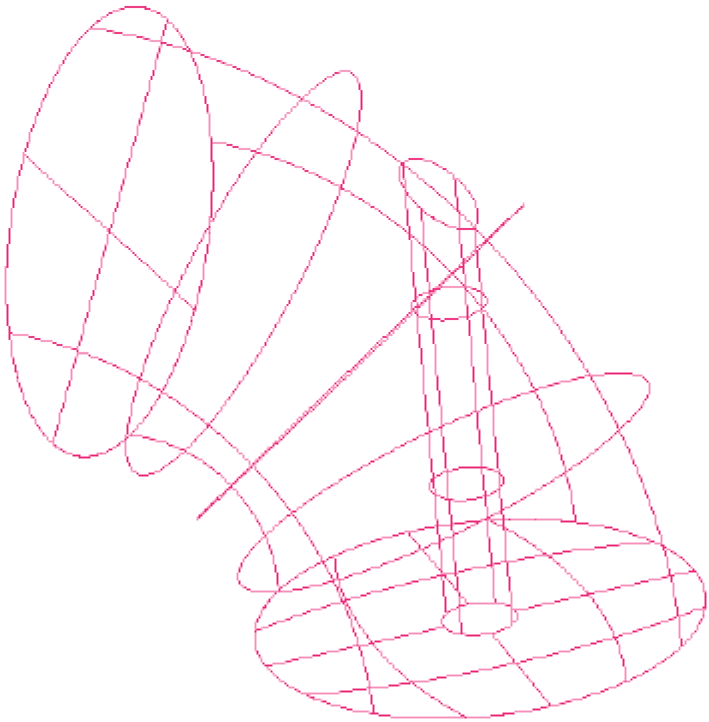
To write the mesh in a structured format, right mouse click in the Display Tree on Blocking>Pre-mesh>Convert to MultiBlock Mesh.

Finally, save the project.

3.2.7: Elbow Part

Overview

In this tutorial example, the user will generate a hexa mesh for a three-dimensional elbow intersected internally by a Cylinder.



a) Summary of Steps

The Blocking Strategy

Starting the Project

Creating Parts

Creating Material Point

Starting Blocking

Splitting the Blocking Material

Fitting the Computational Domain to the Geometry

Creating the First O-grid

Creating the Second O-grid

Generating the Mesh

b) The Blocking Strategy

For this model, the user will make two internal O-grids inside of an "L" shaped blocking.

The first O-grid will create the internal cylinder hole.

The second O-grid will improve the mesh quality within the main elbow-pipe.

c) Starting the Project

Start ANSYS ICEMCFD. File > Change working directory to $ICEM_ACN/../docu/CFDHelp/CFD_Tutorial_Files>Elbow Part. Use File > Geometry > Open geometry and choose geometry.tin.

Note: Settings >Selection>Auto pick mode should be turned OFF for ICEM CFD to behave exactly as this tutorial describes.

d) Creating Parts

Like the previous two tutorials, the user will need to assign and create the Parts before blocking is to be performed. Use the Part > Create Part functions within the Display Tree to change the Part for the surfaces. The following steps will lead the user through this process.

In the Display Tree turn ON Surfaces and right click on Parts > Create Part. Type IN next to the Part name. The Create

Part by Selection icon should be selected by default.

Select the entity selection icon and select the surface of the geometry labeled as IN in Figure 3.168, and then press the middle mouse button to accept. Press Apply to create new part.

Refer to Figure 3.168 as a guide to assign the other surfaces to the Parts IN, ELBOW, CYL, and OUT.

When all of the Surfaces have been assigned to their respective PARTS, press the middle mouse button to end selection, and press Dismiss to exit from the 'Create Part' Window.

For this tutorial, we will leave the Curves and Points assigned to the initial family, GEOM.

e) Creating the Material Points

Select Geometry > Create Body > Material Point . The Centroid of 2 Points location should be selected.

Enter FLUID for the Part name.

With the left mouse button, select two locations on opposite sides of the elbow, so that the midpoint is inside the ELBOW and outside the CYL, as shown in Figure 3.168. Press the middle mouse button to accept the selection, and press Apply to create the material point. The Body name FLUID should appear within the geometry. Rotate the model to confirm that FLUID is in the appropriate location.

In a similar way, create a material point with the Part name DEAD inside the cylinder.

File > Geometry >Save Geometry As (geometry) file to save the updated model before continuing on in this tutorial.

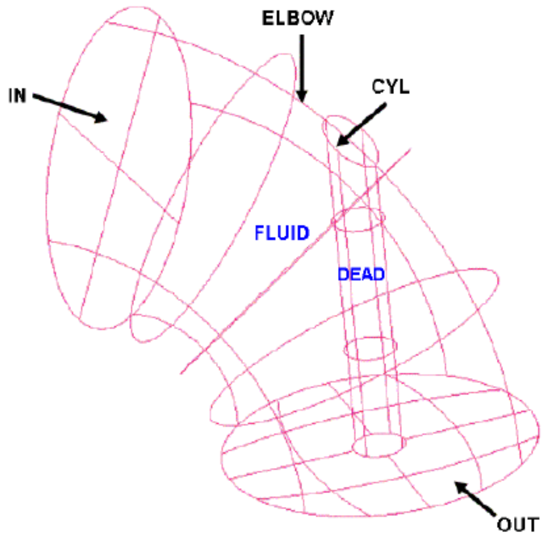


Figure 3.168 The geometry of the Elbow Part with the labeled Surfaces and Material

f) Blocking

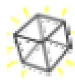
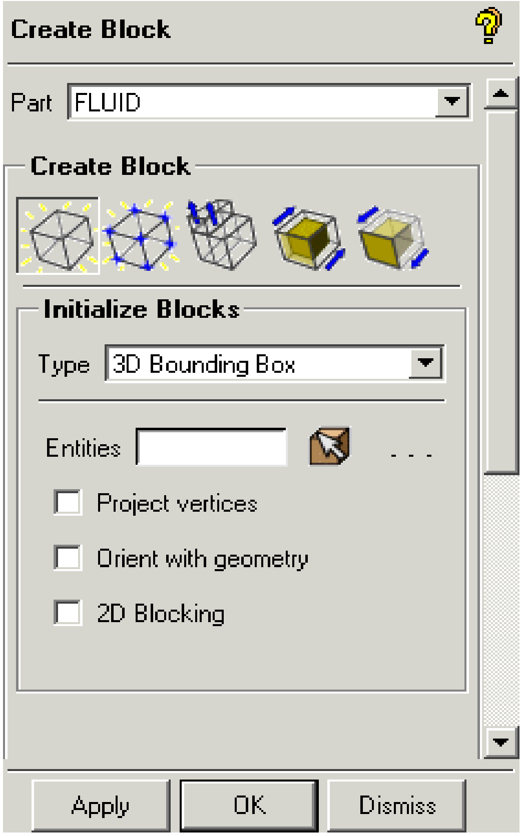
Select Blocking > Create Block  > Initialize Block  .The window is shown in Figure 3.169. Choose 3DBounding Box from the Type pull down if not already set. Enter FLUID in the Part name, and make sure Orient to geometry is OFF. Press Apply to initialize the first block around everything.

Figure 3.169
Initialize Block Window



Note: If nothing is selected in the entities window. Then by default it takes all the entities.

To achieve the “L”-shaped blocking topology shape for the elbow, the user will make two block splits and discard a block. Deleting a block without “permanently” checked will only move the block to VORFN.

Turn ON Curves from the Display Tree and select Blocking > Split Block  > Split Block  to open the window shown Figure 3.170. The Split method should be set as Screen select by default. Create splits as in Figure 3.171 by selecting any edge that you want the split to run perpendicular through. Press Apply, and then select a location for the split to run through, and press the middle mouse button to accept. Make two splits as shown in Figure 3.171.

Figure 3.170
Split blocking window

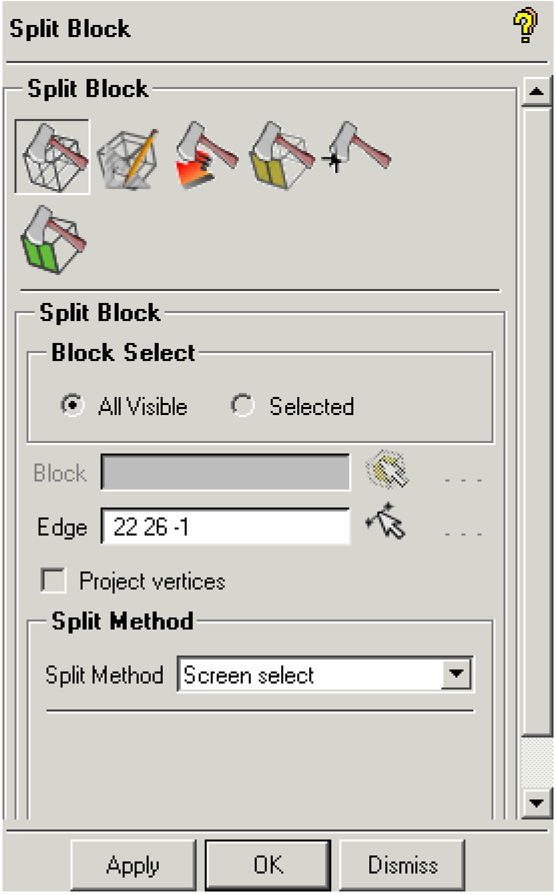
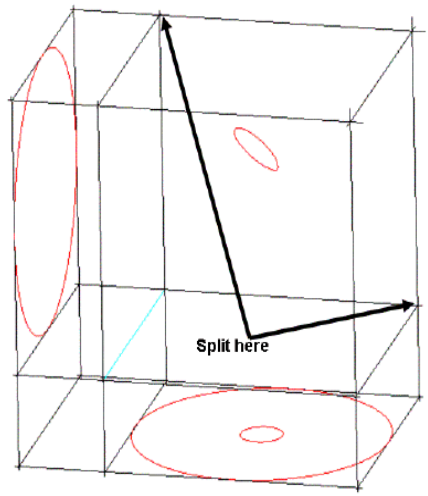
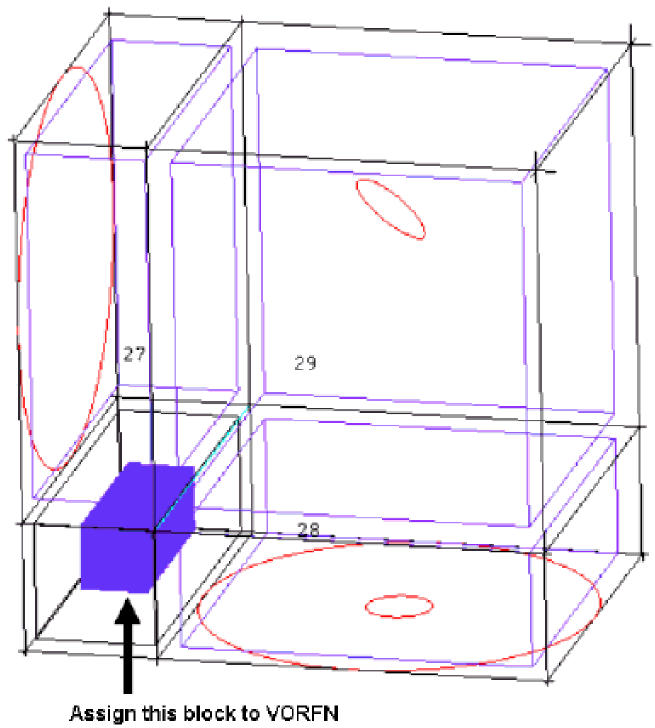


Figure 3.171
The two Block Splits



Next, select Blocking > Delete Block, and select the block shown in Figure 3.172. Delete permanently should be turned OFF, then press Apply.

Figure 3.172 Deleting a block



g) Fitting the Blocking to the Geometry

Here, the user will associate blocking edges to geometry curves, and move vertices onto the geometry.

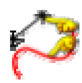
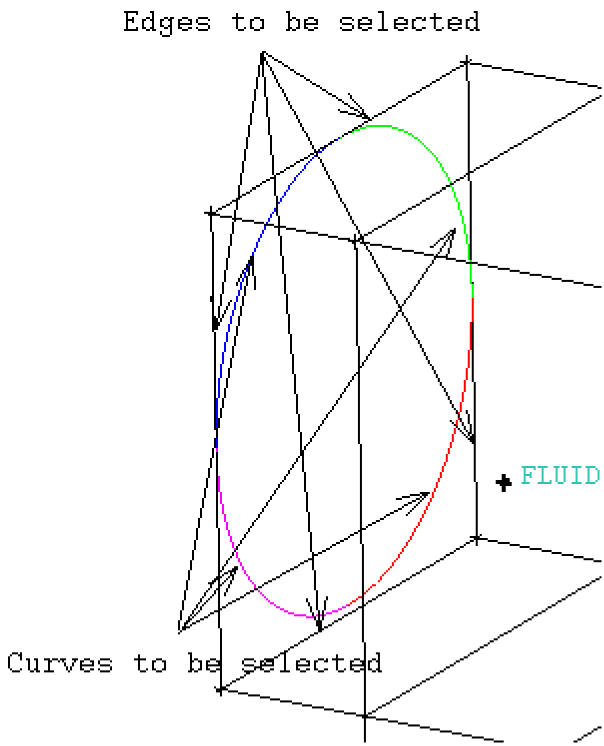
Select Associate  > Associate Edge to Curve  and turn ON Project vertices. Select the four edges surrounding the IN part as shown in Figure 3.173, and press middle mouse button to complete selection. Next, select the four curves shown in Figure 3.173, and press the middle mouse button to complete selection. Press Apply to associate the edges to the curves.

Figure 3.173 Selection of edges and curves for association

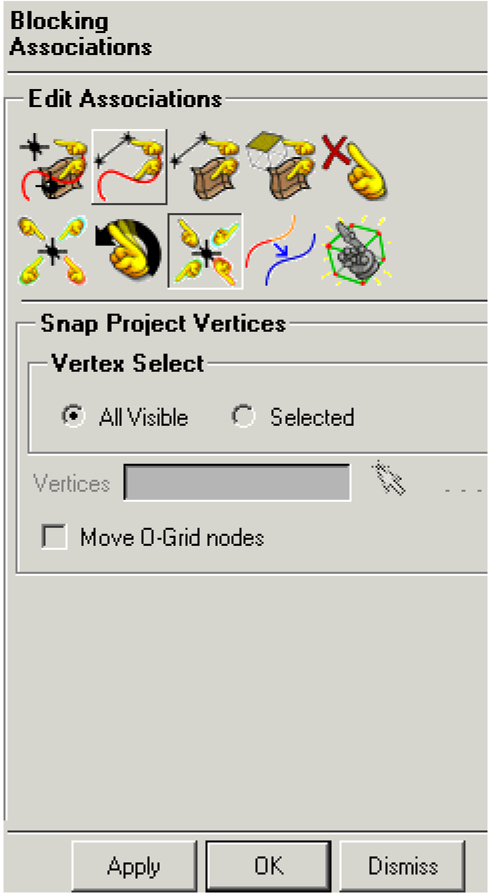


In the same way, select the surrounding four edges and curves of the OUT surface for association.

Note: With those edges associated to the appropriate curves, the other (surface associated) vertices can be automatically moved onto the geometry to the nearest active surface.

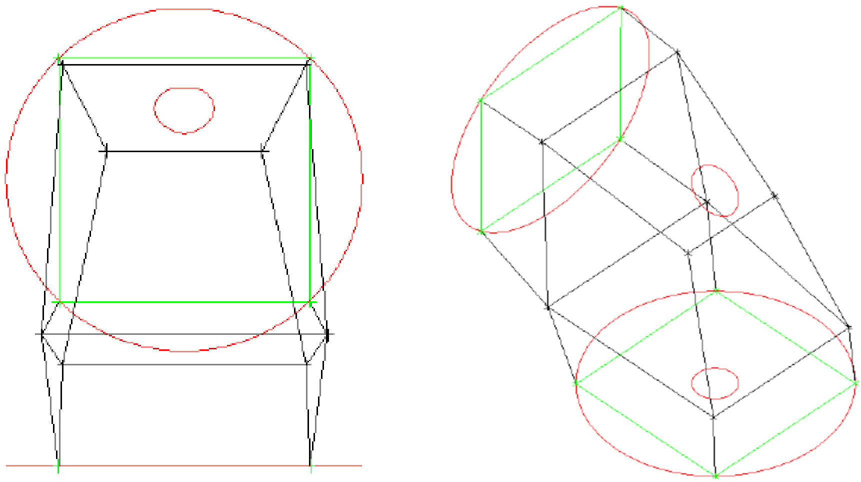
Select Blocking > Associate > Snap Project Vertices to open the window shown in Figure 3.174. All Visible should be toggled on by default. Then Press Apply. Figure 3.175 represents the completion of these operations.

Figure 3.174
Snap Project vertices window



Note: View > Right can be used to orient the model as seen on the left in Figure 3.175. View > Isometric can be used to orient the model as show on the right inFigure 3.175.

Figure 3.175 Project the displayed edges



Before creating the two O-grids, it will be necessary to move two of the vertices slightly from their present position.

Select Blocking >Move Vertex  > Move Vertex  to open the window shown in Figure 3.176. Orient the model as shown in Figure 3.177, and move the vertices to their new position as indicated in Figure 3.178 and Figure 3.179. You'll need to left mouse click on the vertex and hold the button while you slide the vertex on the surface.

Figure 3.176
Movement constraints window

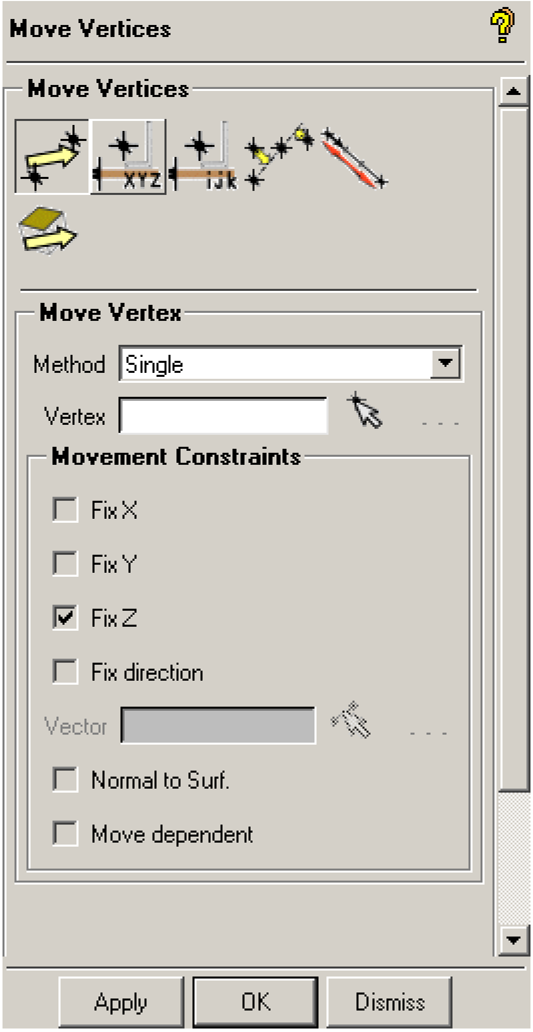


Figure 3.177 Vertices to be moved

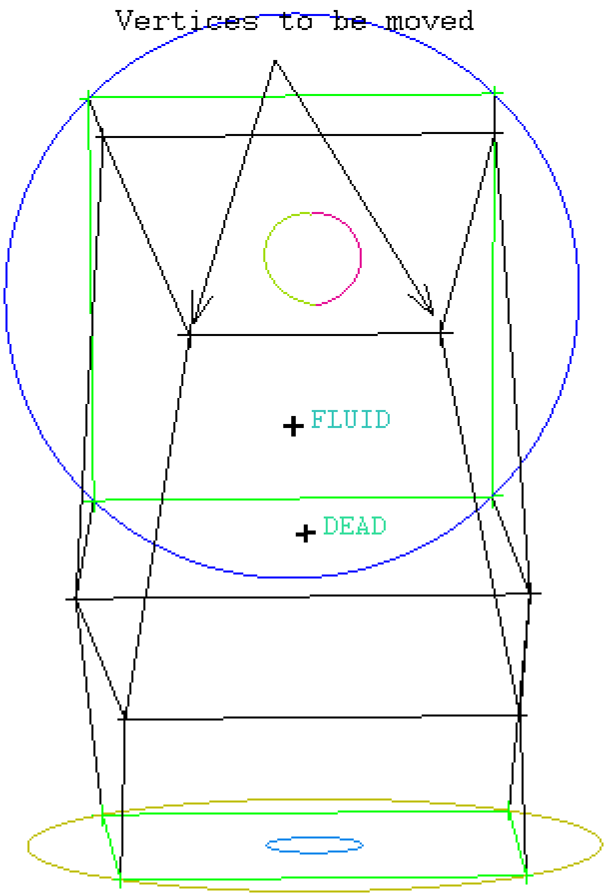


Figure 3.178 Vertex positions after moving

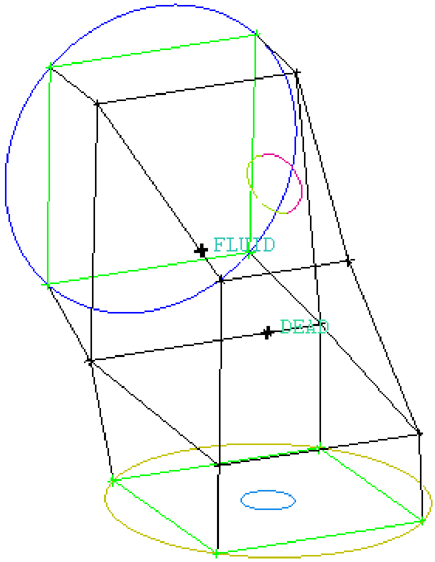


Figure 3.179 Vertex positions after moving

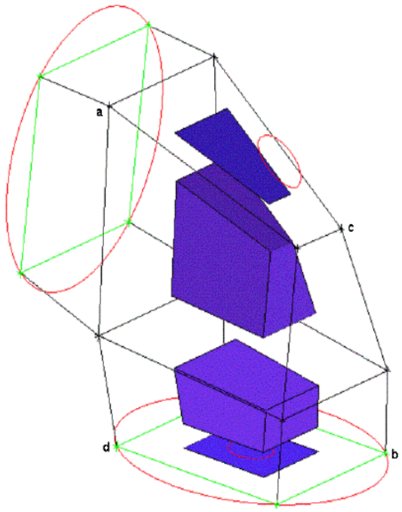
h) Creating the First O-grid

This tutorial focuses on the flow outside of the internal pipe. Thus, in generating the first O-grid, the user will essentially partition the volume around the small internal pipe so that part of the blocking may be removed.

Select Blocking >Split Block > Ogrid Block .Within the O-grid Block window, press the Select Block(s) button. A long, horizontal selection window will appear at the upper right. Press the last button, called "Select diagonal corner vertices" , and select two corners diagonally spanning the blocks in which we want the O-grid. Selecting the vertices 'a' and 'b', as specified in Figure 3.180 will work well. Press the middle mouse button to finish selection.

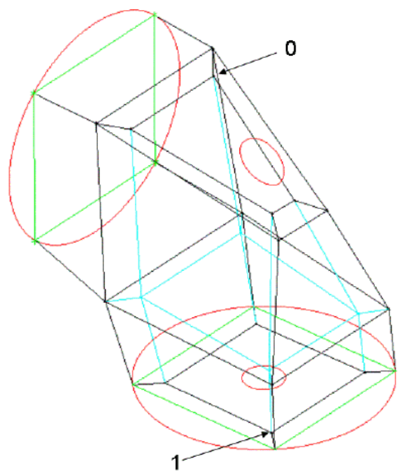
Since we want the cylinder to pass through the top and bottom of the geometry, press the Select Face(s) button. Again, press the "Select Diagonal corner vertices" button and select the face defined by vertices 'a' and 'c'. Then press the button again and then select vertices 'b' and'd'. Refer to Figure 3.180. Press the middle mouse button to finish selection, and press Apply to create the first O-grid.

Figure 3.180 Creating the first O grid



After creating the first O-grid, the geometry will appear as shown in Figure 3.181.

Figure 3.181 Assigning the block to DEAD



Next, the user will assign the material inside the cylinder to the DEAD part. This will remove this region from the mesh if it is computed with the DEAD part turned off.

Right mouse click in the Display Tree on Blocking > Index control. You'll see a window appear at the lower right. Change the Min for O3 from 0 to 1. You can press the up arrow or type the number and press the enter key on the keyboard. You should then only see the blocking shown in Figure 3.182.

Right mouse click in the Display Tree on Part>Dead>Add to part. Select the Blocking Material, Add blocks to Part  button. Press the Select Blocks  button, and then box select the entire model or press 'v' on the keyboard. Press the middle mouse button, and then press Apply. The selected blocks will then be assigned to the DEAD part.

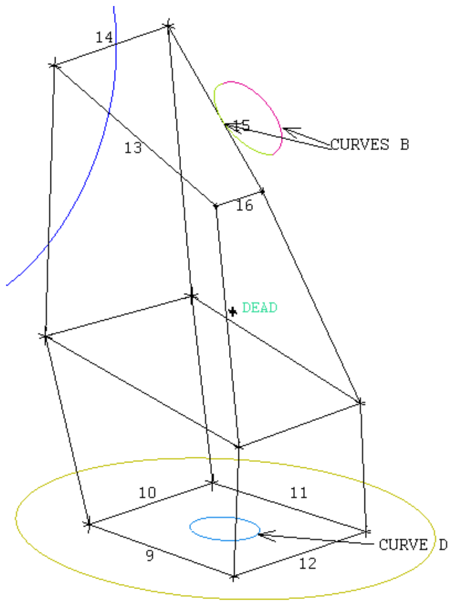
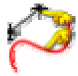


Figure 3.182
Projecting the inner block to the small pipe curves

Press Associate  > Associate Edge to Curve  to open the window as shown in Figure 3.183. Make sure that Project vertices are ON.

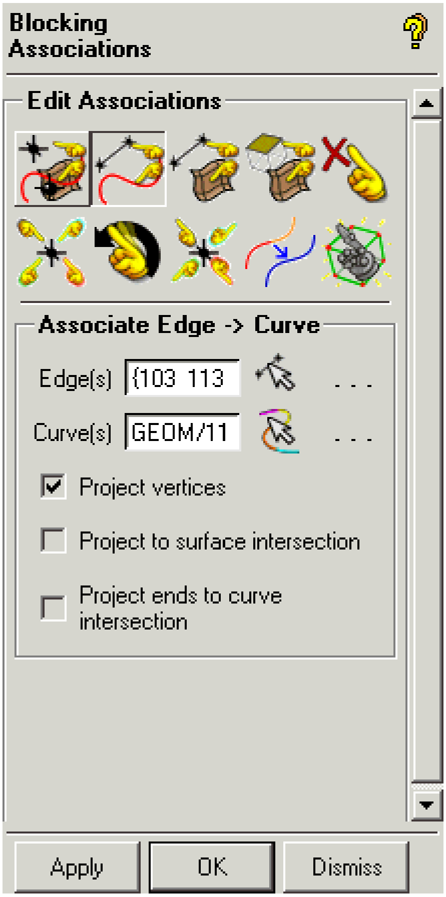


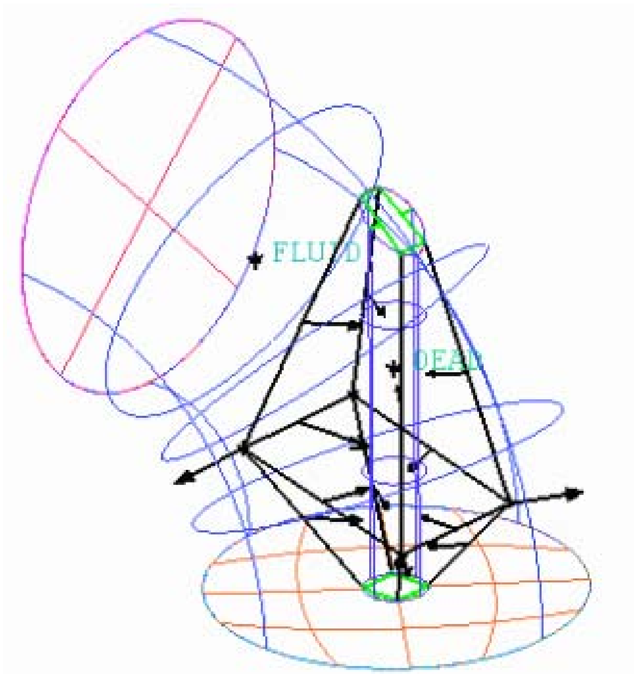
Figure 3.183
Associate edge to curve window

Now Associate Edges 9, 10, 11, and 12, to CURVE D using Figure 3.182 as a guide.

Referring to Figure 3.182, associate Edges 13, 14, 15, and 16 to CURVES by selecting both curves. Displaying the

Blocking > Edges > Show Association in the Display Tree should look like Figure 3.184

Figure 3.184
The edges to Curve projection



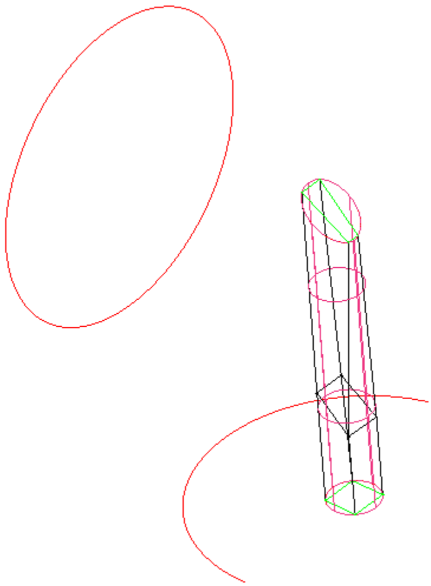
From the Display Tree widget, turn on Surfaces.

i) Moving the remaining vertices.

Notice the association arrows pointing to the outside surfaces of the elbow part in Figure 3.184. If we were to snap project vertices now, these vertices would move outward to the elbow part surfaces. So you must turn OFF the ELBOW part from the Display Tree, so that the vertices will not go to that part.

Go to Association > Snap Project Vertices .All Visible should be toggle on by default. Press Apply. The model should look like Figure 3.185. Then press Reset at the lower right where the Index Control window is located.

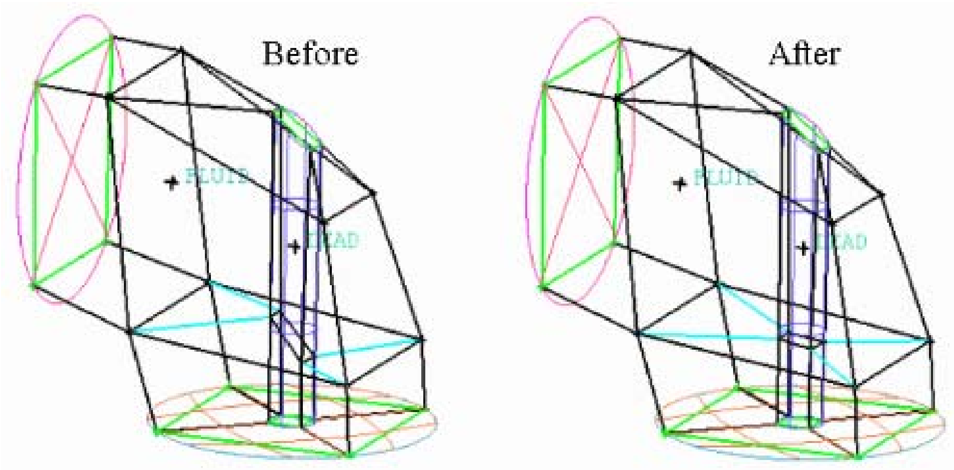
Figure 3.185
After the projection



Use Blocking > Move Vertex > Move Vertex to improve the placement of the vertices on the cylinder. See Figure 3.186.

Turn the ELBOW part back on.

Figure 3.186
Vertex positions after moving



j) Creating the Second O-grid

The following steps instruct on how to add the second O-grid.

Choose Blocking > Split Block >Ogrid Block .

Press Select Block(s), and then select all the blocks by box selecting over the entire model and clicking the middle mouse button or pressing “v” on the keyboard.

Now add the faces on the inlet and outlet by pressing Select Faces, and selecting all the faces on the IN and OUT as shown in Figure 3.187. There are five faces on the OUT part and one face on the IN part. If you select the wrong face, right mouse click to deselect the last face. Press the middle mouse button after selecting the correct faces.

Press Apply to create the O-grid, which should appear as shown in Figure 3.188

.

Figure 3.187
Select the FLUID material and add faces for the O grid

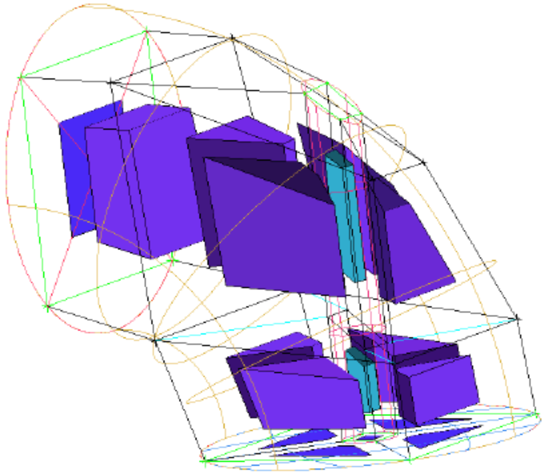
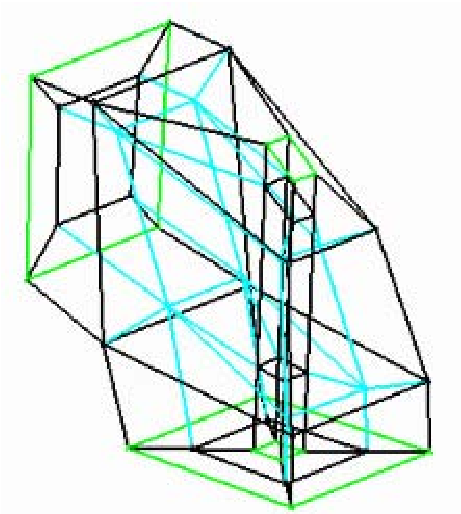
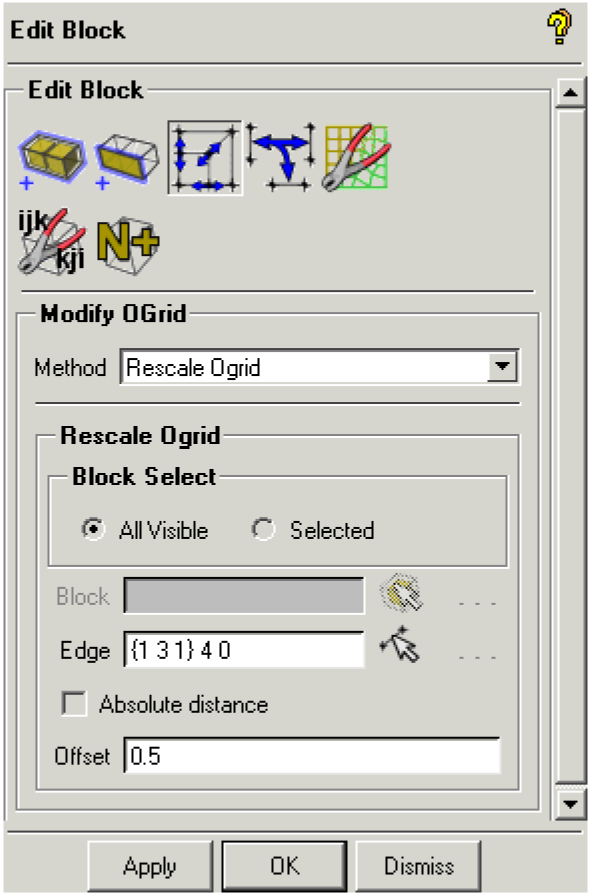


Figure 3.188
The second O-grid



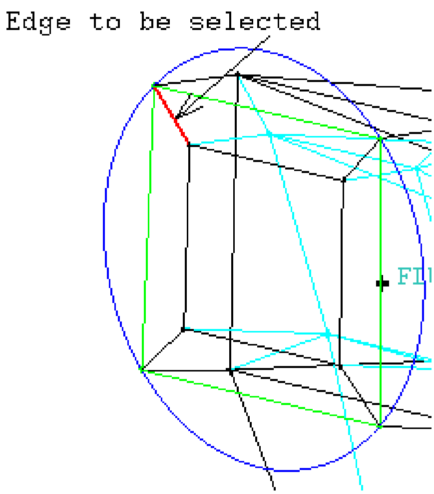
To resize the O-grid, select Blocking > Edit Block  > Modify O-Grid  to open the window shown in Figure 3.189. Choose Rescale O grid from the dropdown.

Figure 3.189
Rescale O-grid window



Select any of the small radial edges of the second O-grid. Figure 3.1903 shows one of these radial edges that you could select. Enter an Offset value of 0.5. With Absolute distance turned OFF, this value is a relative distance. This means it is a multiple of the original edge length, which is given as 1. Setting the offset to 0.5 will reduce the selected edge to half the length. It will do this for all the radial edges of the O-grid. This is why it doesn't matter which radial edge is first selected. Press Apply to rescale the O-grid. This will result in a better element quality.

Figure 3.190
Edge to be selected for rescaling

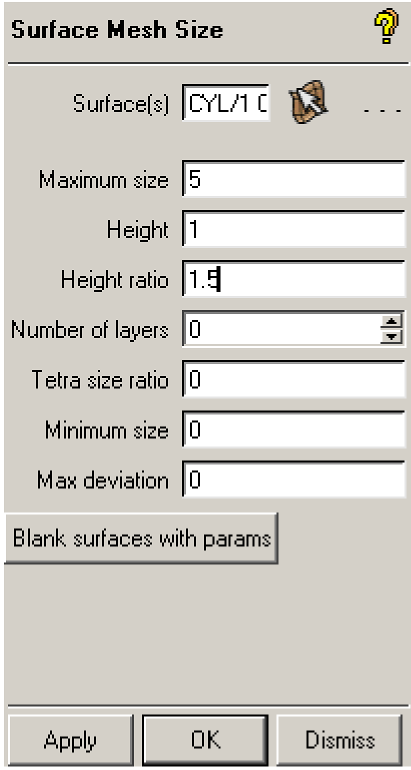


k) Generating the Mesh

Select Mesh > Set Surface Mesh Size  and box select all surfaces followed by clicking the middle mouse button or press "v" on the keyboard. Enter the following parameters as shown in Figure 3.191.

Max Element size 5, Height 1, and Ratio 1.5. Then press Apply.

Figure 3.191
Surface mesh size window

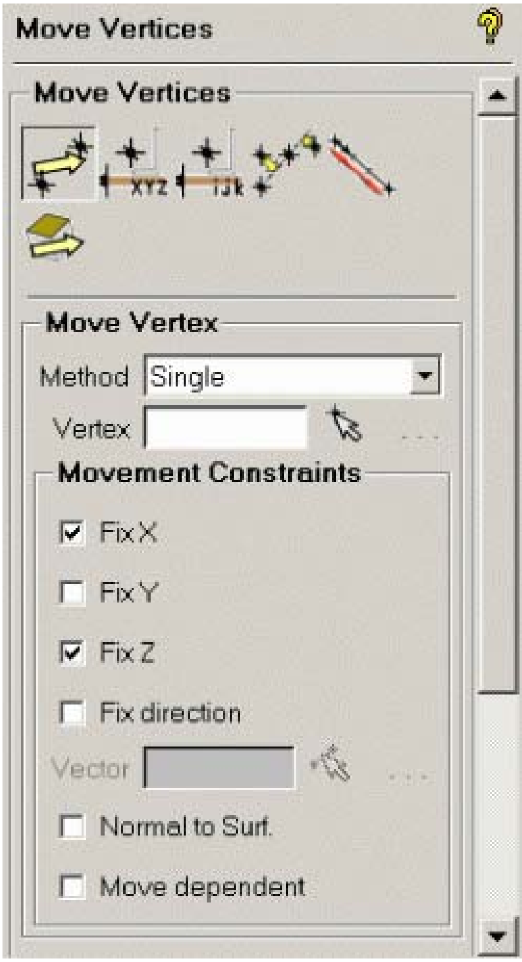


Before generating the mesh, there is an additional step that will improve the quality of the mesh.

Select Blocking > Move Vertex > Move Vertex and reposition the vertices indicated in Figure 3.190 to improve the denoted angle. Under Movement constraints, select Fix X and Fix Z. Then press the vertex selection button and left mouse click and hold to move the vertex down the CYL tube. Figure 3.192shows the before and after pictures of the vertex positions. Notice that the vertices only move up and down the Y-axis.

Now turn OFF Fix X and Fix Z. Then move the two vertices shown in the second part of Figure 3.192, by selecting on the small blue radial edge close to the vertex, but do not select on the vertex itself. Internal vertices (blue) move in the direction of the edge selected on that is connected to the vertex.

Figure 3.192
Move vertices window



Press the middle mouse button to finish the operation.

Note: These two minor vertex adjustments will decrease the acuteness of angle in the blocking and improve the overall quality of the mesh.

Figure 3.193
Moving the vertices

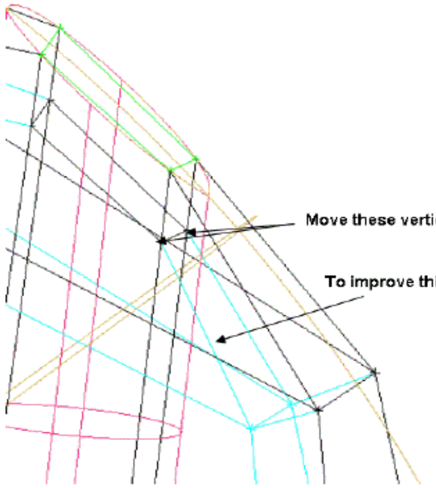
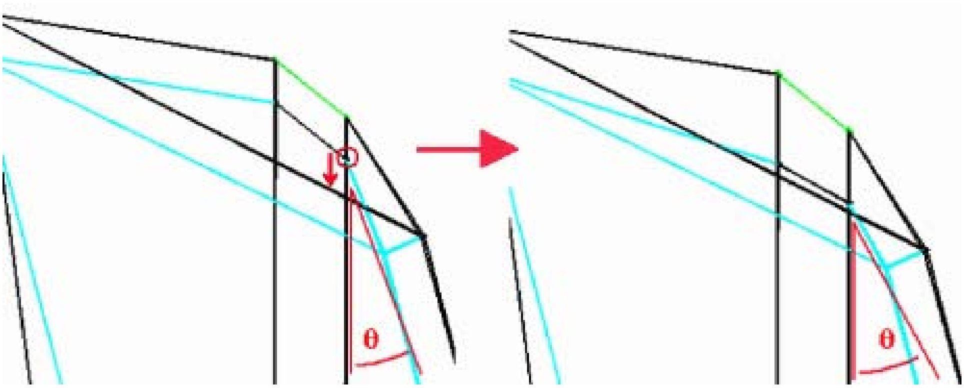
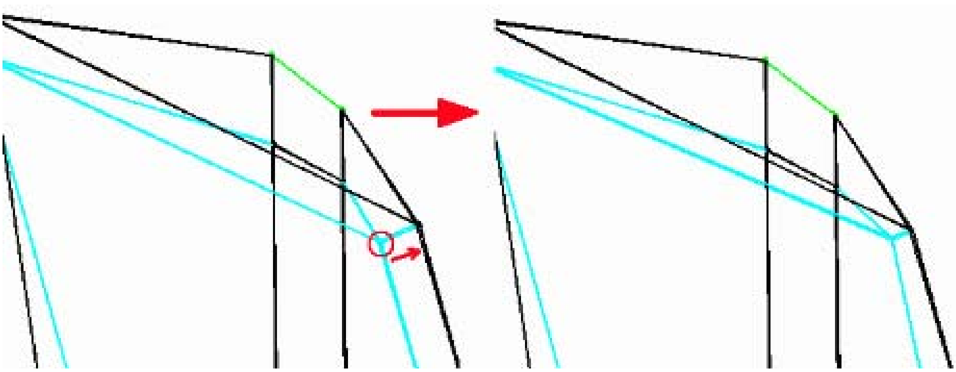


Figure 3.194
Vertex positions after moving, which shows the improved angle





Select Blocking > Pre-mesh Params > Update Size. For Method, ensure that Update all is toggled ON. Then press Apply. This will reapply the any surface and curve parameters to the blocking edges, which is necessary after any new edges are created through blocking splits or O-grid creation.

Before computing the Pre-mesh, turn the DEAD part off so that the mesh is not computed for that part.

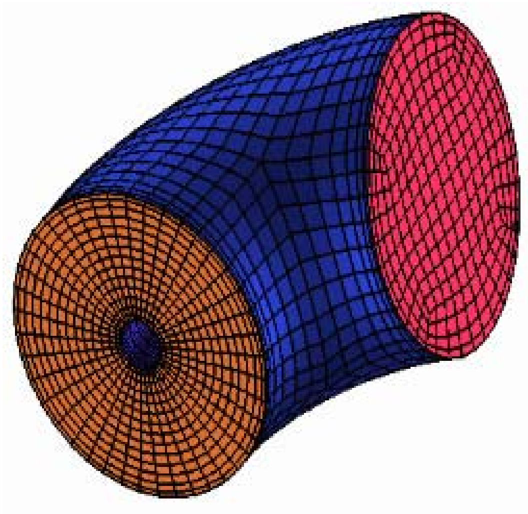
In the Display Tree, right click on Blocking > Pre-mesh and ensure that Project Faces is checked. Then turn ON the Pre-mesh, and choose Yes when asked to recompute mesh.

Switch off Edges and all geometry in the Display Tree to view only the pre-mesh.

The final mesh should look similar to Figure 3.195.

To get a good quality mesh, check angles and determinants, view the lowest histogram bars, then inspect the lowest quality elements and decide which blocks they exist in and which vertices need to be moved and what direction. After adjusting vertices, turn the Pre-mesh OFF and ON again to recompute the mesh. Try to get determinants above 0.3 and angles above 15 degrees.

Figure 3.195
The final mesh



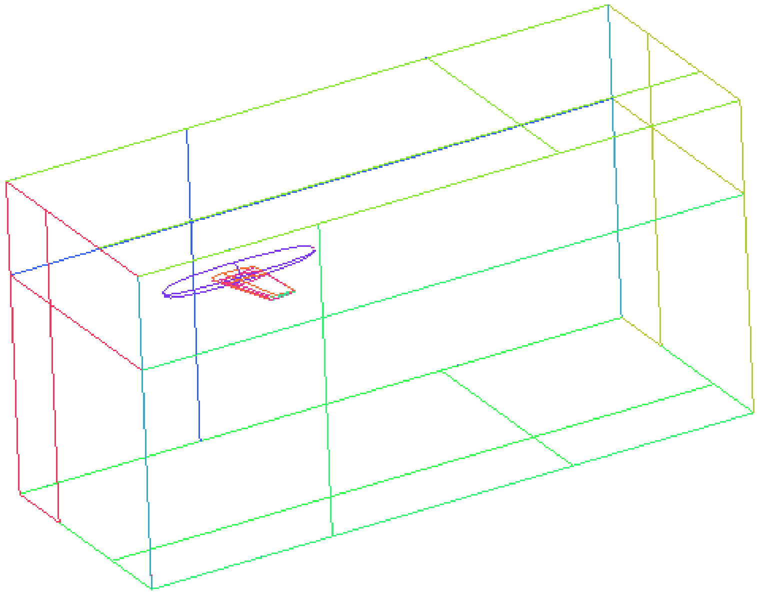
Save the blocking. File > Blocking > Save Blocking As.

Right click in the Display Tree on Blocking > Pre-mesh > Convert to Unstruct Mesh. This will write out the unstructured mesh to the default name hex.uns to the working directory. Then it will automatically load the mesh. You can resave (File > Mesh > Save Mesh As) to a different name if you'd like after that.

3.2.8: Wing Body

Overview

This tutorial example will focus on generating a mesh with a replay file for a three-dimensional wing body configuration, as shown in the diagram below. The geometry consists of a simple cigar-shaped body with a tapered wing.



a) Summary of Steps

Geometry and Blocking Strategy

Starting the Project

Starting Blocking

Splitting the Blocking around Fuselage

Splitting the Blocking around Wing

Assigning the Material

Fitting the blocking to the fuselage and wing

Creating the O-grid around the block

Setting Mesh Parameters with Linked bunching

Improving Mesh Quality

b) Geometry and Blocking Strategy

For this model, the user will execute blocking methods by employing functions such as Split, Set location and O-grid. The main fuselage and wing will be modeled by simple blocks. An O-grid will be added around the entire body near the end in order to improve element quality and allow grid lines to be aligned normal to the surfaces in order to set a fine boundary layer distribution.

c) Starting the Project

Start ANSYS ICEMCFD. Select File > Change working directory, and change the current working directory to $ICEM_ACN/../docu/CFDHelp/CFD_Tutorial_Files/WingBody. Open geometry.tin by selecting File > geometry > Open geometry.

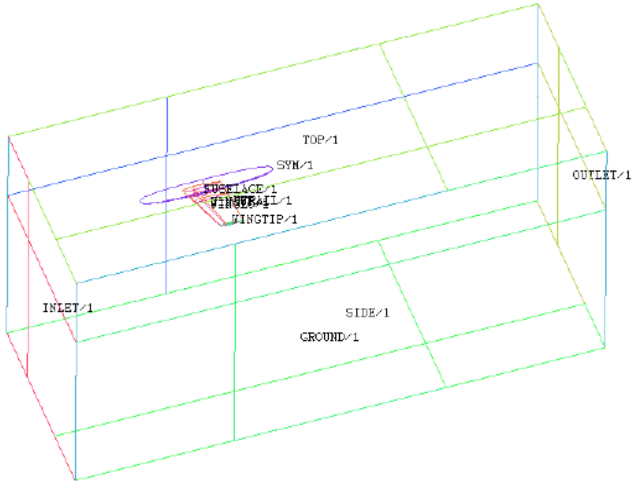


Figure 3.196
The Wing Body Far field Surface parts

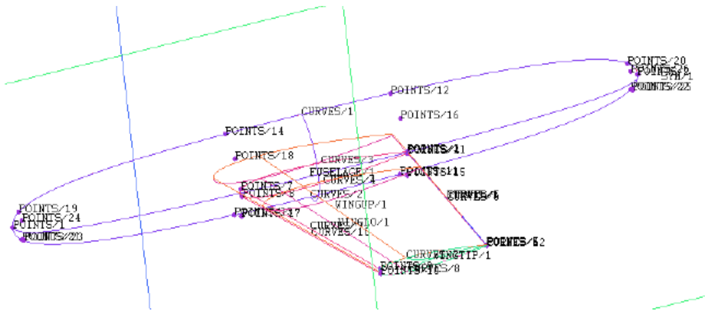
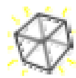


Figure 3.197
Curves and points on the fuselage and wing

In this geometry, the points, curves, and surfaces have already been placed into separate part names. Thus, the user can go directly to the blocking process.

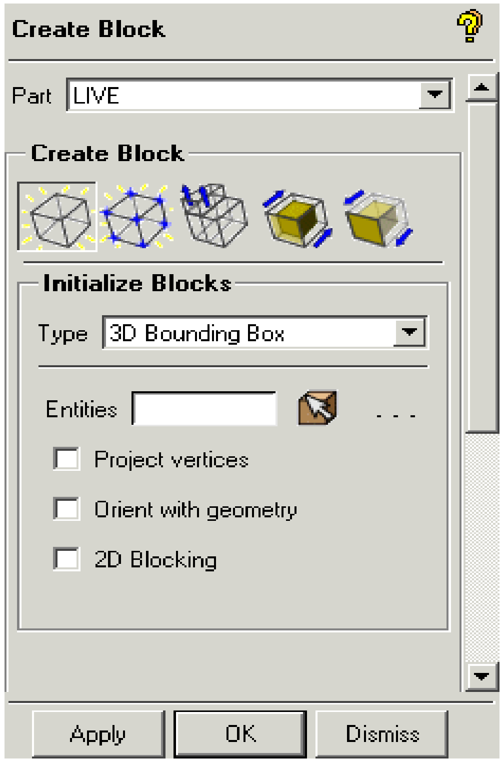
d) Blocking

Select File > Replay Scripts > Replay Control to start recording all the commands executed while blocking.

Press Blocking > Create Block  > Initialize Block  to will open the Create Block window as shown in Figure

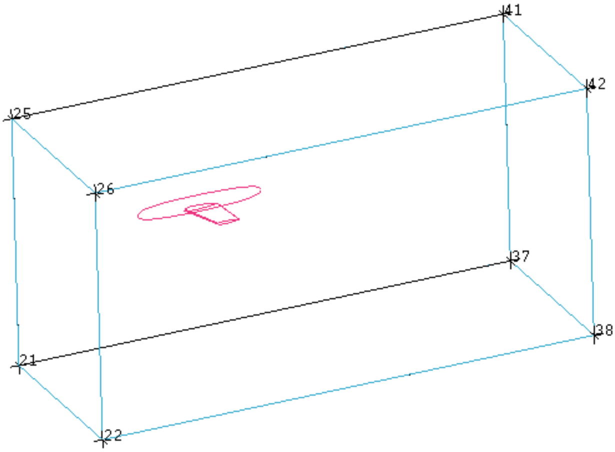
3.198. The default Type is 3D Bounding Box. Verify that this is shown by the Type. Enter the Part name as LIVE, and press Apply without selecting anything. This will create the initial block around everything.

Figure 3.198
Create blocks window



From the Display Tree, make sure that Curves are turned ON and curve names are turned OFF. Right click on Geometry > Curves > Show Curve Names to turn off the curve names. Also make sure that Surfaces are turned OFF as well. Turn ON the Blocking > Vertices and right mouse click on Vertices > Numbers to display the vertex numbers. The initialized blocking is shown in Figure 3.199.

Figure 3.199 The Initialized blocking with vertices



Switch ON Points > Show Point Names in the Display Tree and turn Points ON.

Select Blocking > Split Block  > Split Block . Next to Split Method, select Prescribed Point from the pull down menu. You'll see the window shown in Figure 3.201.

Press the select Edge  icon and select the edge connecting vertices 21 and 25 with the left mouse button. Its end vertex numbers defines an edge. You'll have to temporarily turn off the Points to see the vertex numbers.

Then press the select Point icon  and select POINTS/14 at the top of the fuselage as shown in the figure below. Once POINTS/14 appears in the window, press Apply to get the split through the prescribed point.

Figure 3.200
Split points

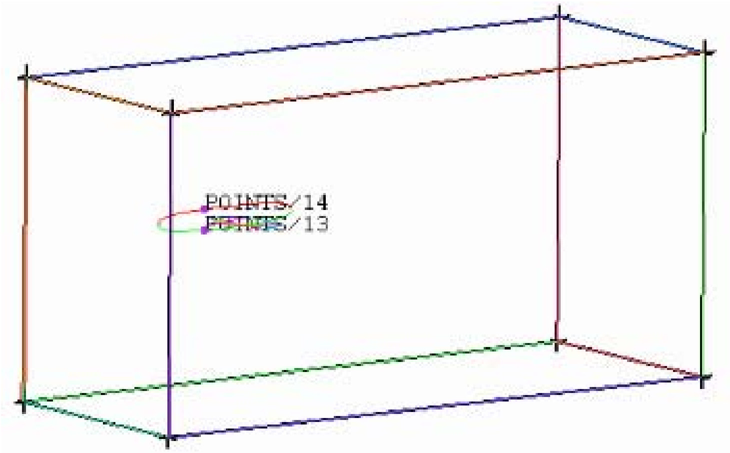
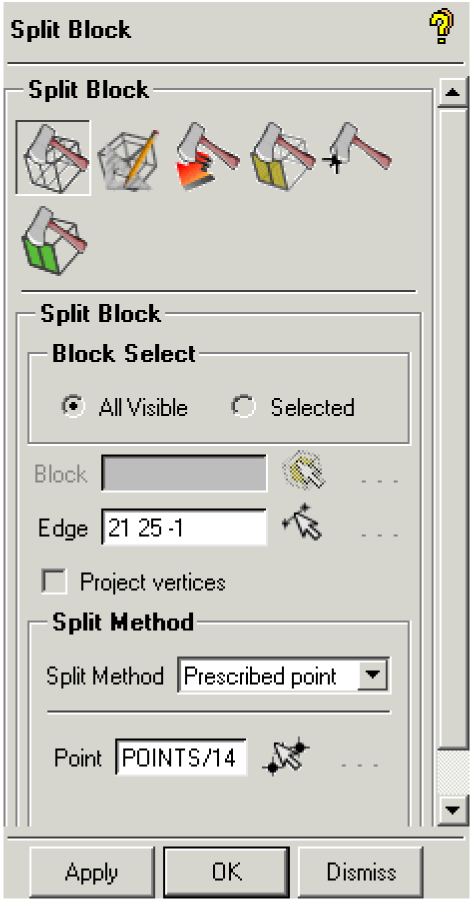
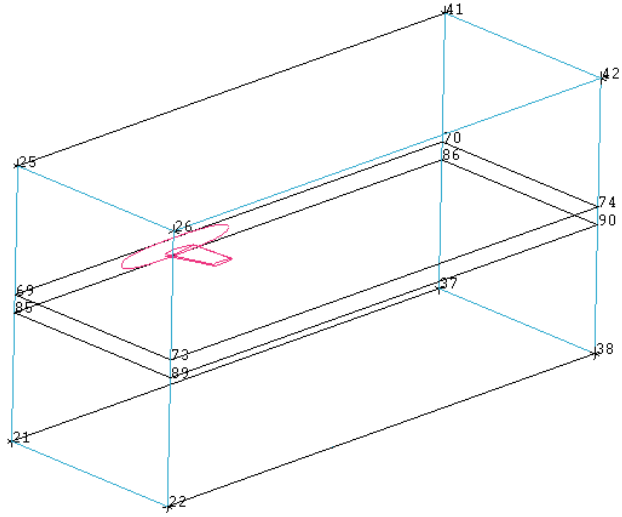


Figure 3.201
The Split block window



Now, select the edge defined by vertices 21 and 69 and split this edge by the Prescribed point, POINTS/13 at the bottom of the fuselage as shown in Figure 3.202. Switch off Points to have a better view. The blocking should now look like Figure 3.202.

Figure 3.202 Splitting around the fuselage



Right mouse click in the Display Tree on Blocking > Index control to display the index control in the lower right corner. Press Select corners, and select vertices 89 and 70 with the left mouse button. The blocking will restrict to the blocks that connect the diagonal of this selection. Switch on Points to Proceed Further.

Blocking > Split Block  > Split Block  : Select the edge connecting vertices 69 and 73 and split this edge by the Prescribed point, POINTS/5 at the tip of the wing.

In the Index control, use Select corners to further restrict the blocking by selecting vertices 105 and 70.

Blocking > Split Block  > Split Block  . Select the edge connecting vertices 69 and 70 and split this edge by the Prescribed point, POINTS/19 at the front of the fuselage.

Blocking > Split Block > Split Block: Select the edge connecting vertices 129 and 70 and split this edge by the Prescribed point POINTS/20 at the tail of the fuselage.

Switch off Points. The blocking should look like Figure 3.203.

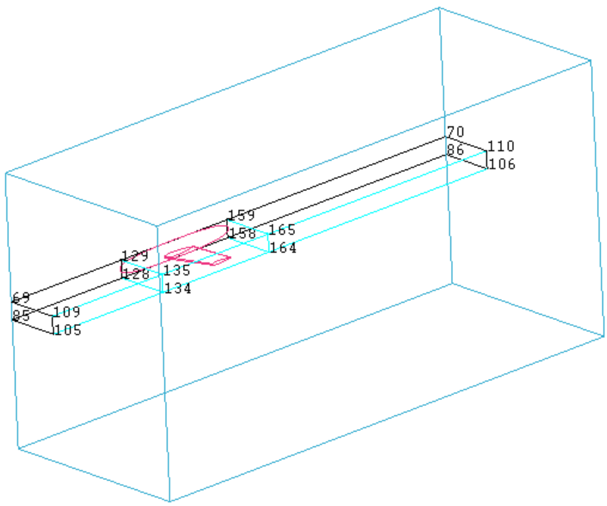


Figure 3.203 More splitting around the fuselage

e) Splitting the Blocking around Wing

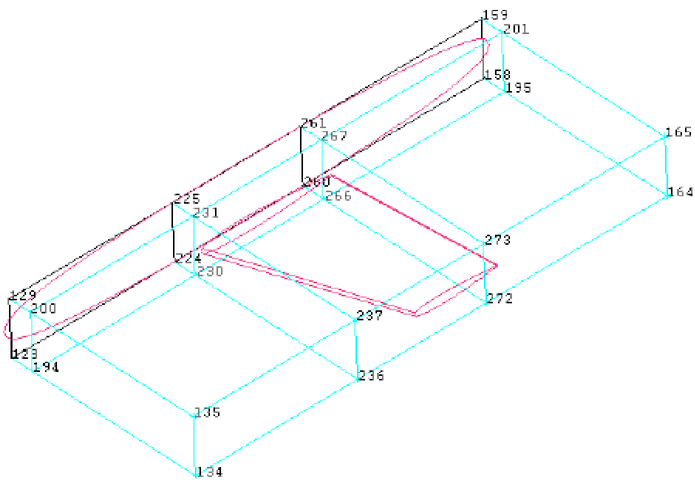
To further restrict the display around the fuselage, use the Index Control and press Select corners, and select the vertices 134 and 159.

Blocking > Split Block > Split Block : Select the edge connecting vertices 129 and 135 and split this edge by the prescribed point, POINTS/18, which is near the base of the wing.

Blocking > Split Block > Split Block : Select the edge connecting vertices 135 and 165. Make sure that the Max K in the Index Control is 3. Then split this edge by the same prescribed point, POINTS/18.

Blocking > Split Block  > Split Block  : Select the edge connecting vertices 237 and 165 and split this edge by the Prescribed point, POINTS/16, which is near the trailing edge and base of the wing. Switch off the Points. The blocking at this stage should appear as shown in Figure 3.204.

Figure 3.204 Splitting at the wing tip



In the Index control, press Select corners, and select the vertices 236 and 267 to restrict the display to the one block around the wing. And also switch on Points.

Blocking > Split Block  > Split Block  : Select the edge connecting vertices 230 and 231. Then turn Points back on and split this edge by the Prescribed point, POINTS/7, which is at the leading edge base of the wing.

Blocking > Split Block > Split Block : Select the edge connecting vertices 230 and 307 and split this edge by the prescribed point, POINTS/8 at the leading edge base of the wing.

Switch off Points. Then press Reset in the Index Control to display the full blocking again. The blocking should look as in Figure 3.205 at this stage.

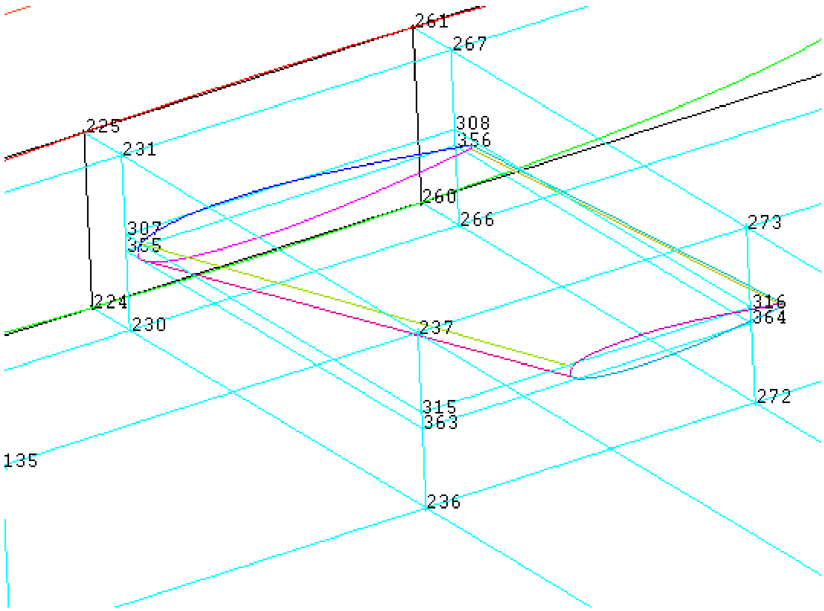


Figure 3.205 Splits around the wing

f) Assigning the Material

In the Index control, press From corners, and select the vertices 134 and 159 to restrict the display. Switch off Vertices.

Right mouse click on Parts > Create Part from Display Tree, and it will open the window as shown in Figure 3.206.

Rename the Part as SOLID. Then select the last icon, .

Then press the Select Block button, and select the four blocks for the fuselage and wing as shown in Figure 3.207. Press the middle mouse button to complete the selection, and press Apply to move the blocks into the new part. The edges at the interface between the new block material and the surrounding material will automatically become surface-associated, and the color will change to indicate that.

Figure 3.206
Create part window

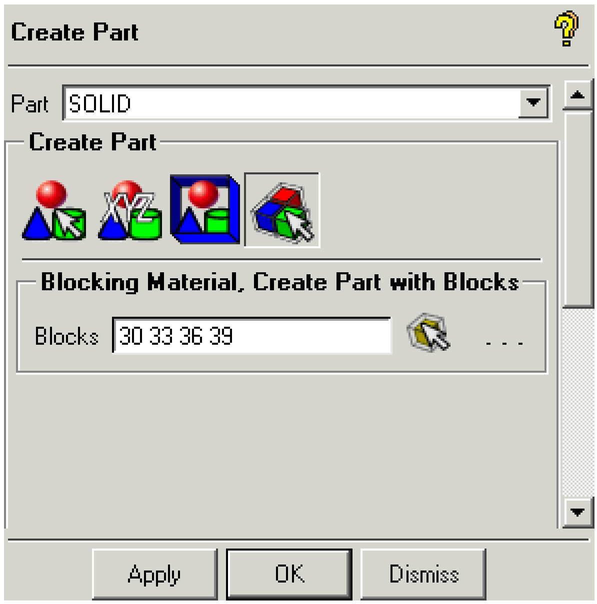
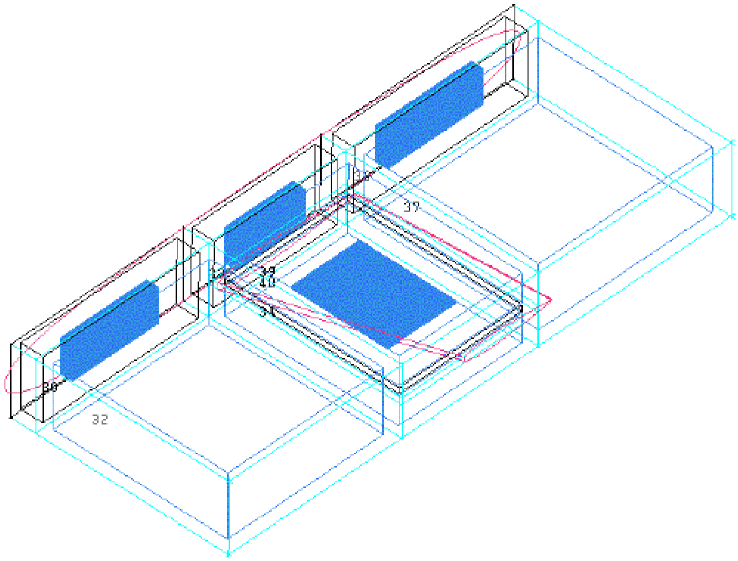


Figure 3.207 Assign these blocks to SOLID



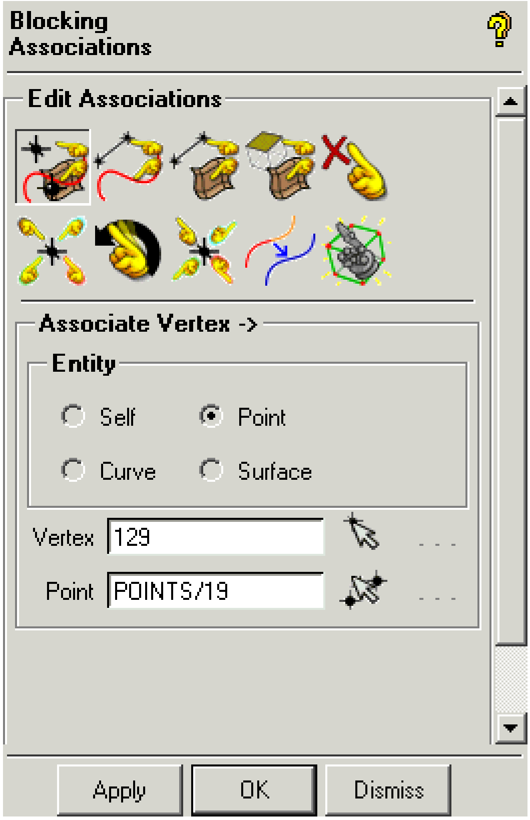
g) Fitting the blocking to the fuselage and wing

To ensure proper association of the blocking edges onto the geometry, the user will project block vertices to the prescribed points first, then and block edges to the curves.

Right click in the Display Tree to switch ON Blocking > Vertices > Numbers. Then turn ON Blocking > Vertices and Geometry > Points.

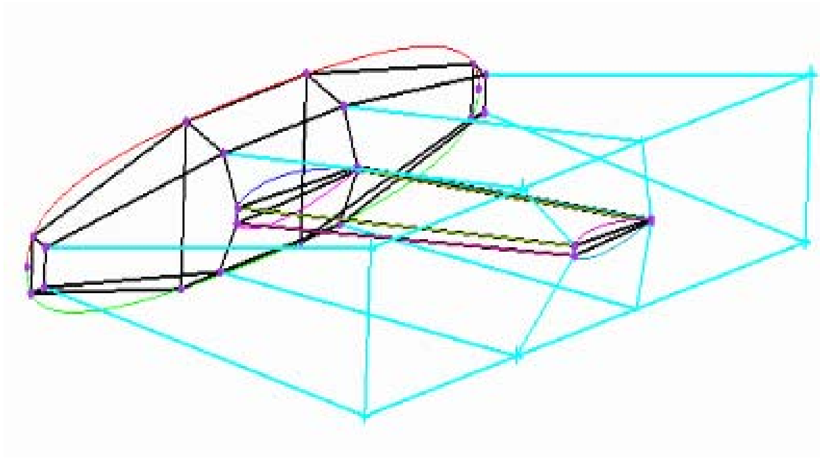
Select Blocking > Associate  > Associate Vertex  and you should see the window as shown in Figure 3.208. Make sure the Entity type to associate to its Point. Select the vertex 129. Then select the point POINTS/19. Press Apply. This will assign the association and move the vertex to the point all in the same step.

Figure 3.208
Associate vertex to entity window



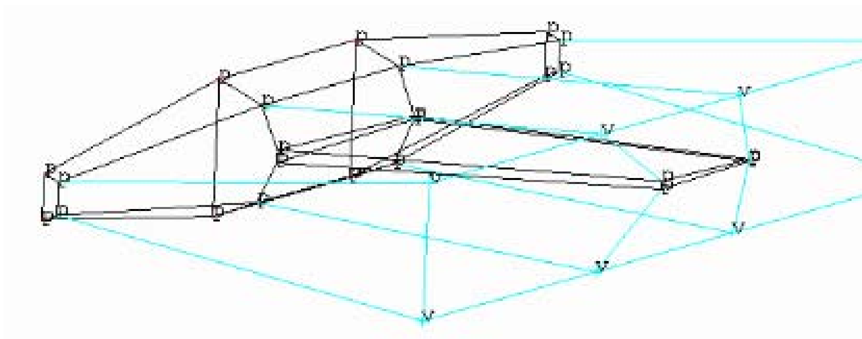
Similarly place other vertices to the corresponding points as shown in Figure 3.209.

Figure 3.209 Projecting the vertices to points



Make sure all the Vertices in the Fuselage and Wings are properly associated to a point. To view this, Switch off Points and Switch on Vertices > Pro Type in the Display Tree. Then turn ON Vertices. You should see a “p” next to each point-associated vertex as shown in Figure 3.210. A “v” stands for a volume vertex while a “c”means a curve vertex and an “s” stands for a surface-associated vertex.

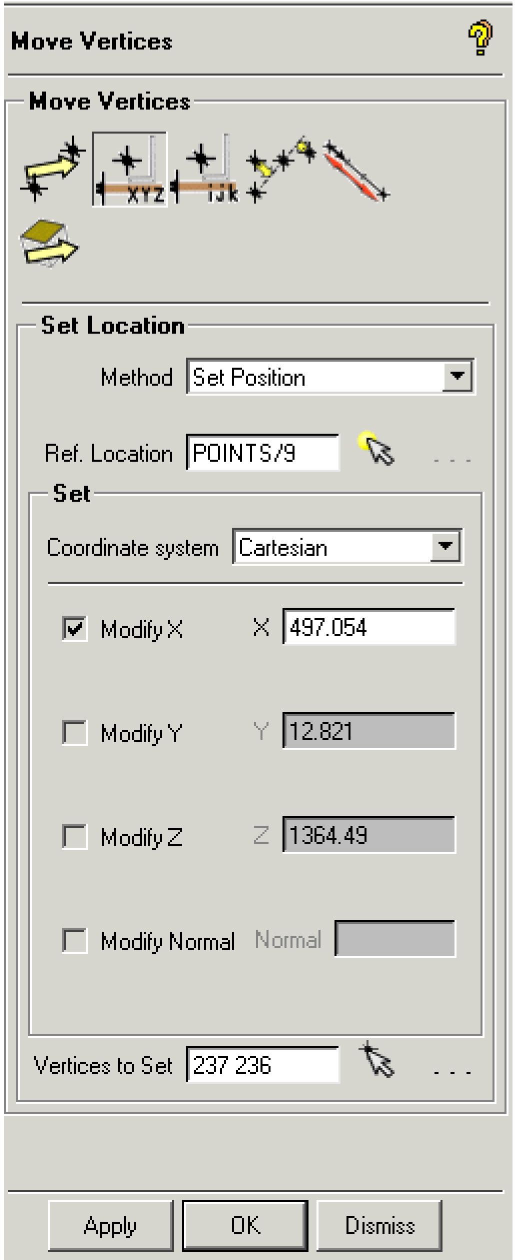
Figure 3.210 Display Proj Type



To align the volume vertices near the wing tip, select Blocking > Move Vertex > Set Location. You should see the window shown in Figure 3.211.

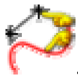
Switch on Vertices > Numbers and Switch on Geometry > Points > Show Point Names, and turn ON Points. Select POINTS/9 at the wing tip as the Ref. Point. Toggle ON the Modify X and for the Vertices to Set selection, select the vertices 236 and 237 with the left mouse button and press the middle mouse button to accept the selection. Press Apply to move the vertices which will match the X-coordinate of the selected vertices to the reference vertex.

Figure 3.211
Set location window



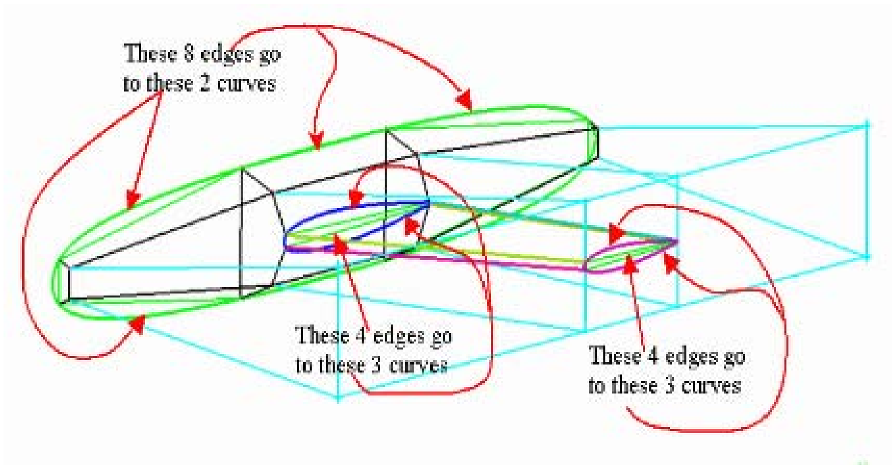
Similarly, set the location for vertices 272 and 273 using the Reference Point as POINTS/5.

Switch On Geometry > Curves and switch OFF Geometry > Points from the Display Tree.

Select Blocking > Association  > Associate Edge to Curve .

Associate the edges to the curves as shown in Figure 3.212. The green colors of the edges indicate that they are associated to a curve.

Figure 3.212 Fuselage, Wing Root and Wing Tip curve and corresponding edges



h) Creating the O-grid around the fuselage and wing

Now we will create an O-grid around the body (around the volume part SOLID) to refine the boundary layer around the geometry.

In the Index Control, press Reset to display the entire blocking.

Select Blocking > Split Block  > O grid Block . Toggle ON Around Block(s). Press add to Select block(s) icon  then press the part selection icon  in the selection menu that appears to the upper right. This will bring up a list of the current parts. Select SOLID, and press Accept. This will select all the blocks in the part, SOLID as shown in Figure 3.213. Then press Apply to create the O-grid. The O-grid should appear as shown in Figure 3.214.

Figure 3.213 O-grid selection

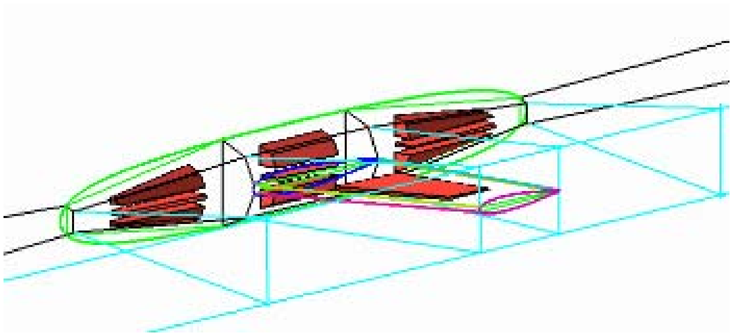
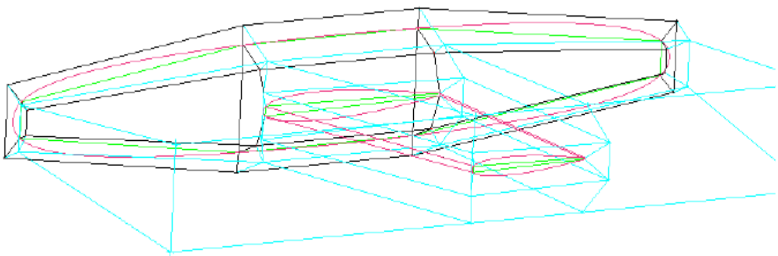


Figure 3.214 Blocking after creating O-grid



i) Setting Mesh Parameters on Surfaces for an Initial Mesh

- Press **Mesh** > **Set Surface Mesh Size** . The window shown in Figure 3.215 should appear. Enter surface selection and box selects all the surfaces of the model. Turn **Surfaces** ON in the Display Tree, and right click on **Geometry** > **Surfaces** > **Hexa sizes** to display the Hexa icons. Set the **Maximum size** to 300, **height** to 300, and the **height ratio** to 1. For a Hexa mesh, all 3 of these need to be filled in. Press Apply, and you will see the icons update.
- Now zoom in closer to the fuselage and body. Box select around these surfaces, but not the outer surfaces. The box select should be set to "entire" selection mode by default, so it will only select what is completely enclosed within the box. If it was left in "partial" mode from a previous selection, press  in the popup selection option window to switch to entire selection mode. Set the **Maximum size** to 50, **height** to 50, and **height ratio** to 1.4. Press Apply.

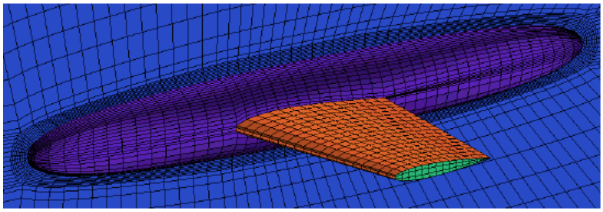
Figure 3.215
Setting mesh sizes on surfaces

- In the Display Tree, turn OFF the part SOLID, as only the LIVE part is required for meshing. Then right click in the Display Tree on **Blocking**

> Pre-mesh > Project edges. Turn on the **Pre-mesh**. The mesh distribution on the symmetry plane with the fuselage and wing is shown in Figure 3.2167.

Note: Project edges do not do any face projections. Thus, it is a good way to save time when first computing the mesh, even in a 3D model. This allows the user to detect any problematic edge projections and distributions and fix them quickly.

Figure 3.216
Mesh distribution obtained from setting the surface meshing parameters for all the surfaces



j) Setting Mesh Parameters with Linked bunching

The next step is to better define the mesh size parameters on the individual edges of the blocking.

The mesh is distorted in the farfield region. The Linked Bunching function will be utilized to link the mesh distribution.

Switch off Pre-mesh in the Display Tree.

Select Blocking > Pre-Mesh params  >Edge Params . The Meshing Parameters window will open as shown in Figure 3.217 Select the edge to be modified indicated in Figure 3.218. The selected edge has an arrow displayed on it, which indicates side 1 and side 2 of the edge. Side 1 is the back of the arrow, while side 2 is the front on the arrow.

Figure 3.217
Edge meshing parameters window

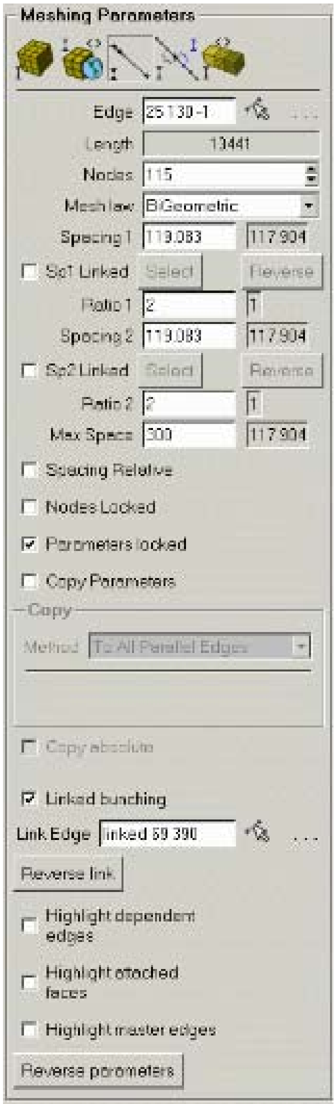
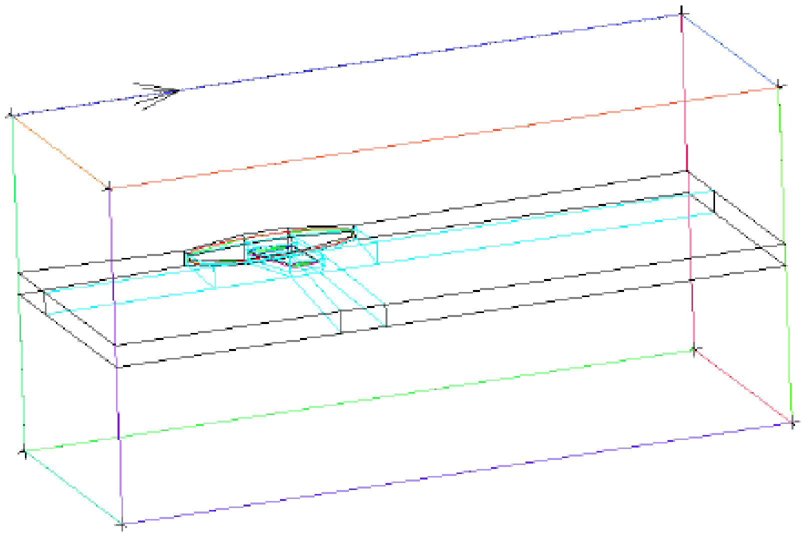
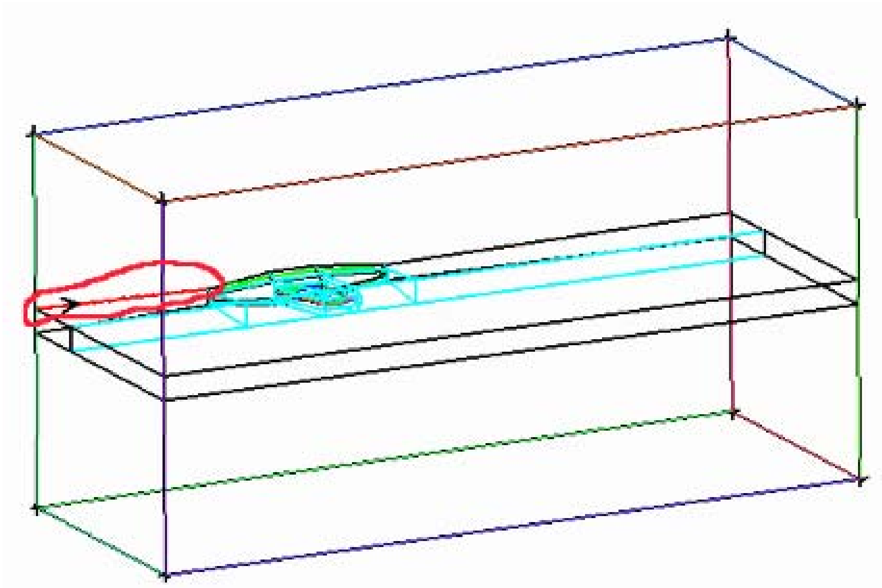


Figure 3.218 Select this edge for setting edge parameters



Click on Linked bunching. Then next to link edge, select the edge selection icon  and select the first edge on the same side as side 1 of the main edge. The main edge will link its node distribution to all the edges connected to this edge that spans the main edge. This edge is shown in Figure 3.219. Remember that the beginning of the larger edge is shown by a white arrow.

Figure 3.219 Select the edges to link



Toggle on the Copy Parameters and under the Method select To All Parallel edges and press Apply to achieve results similar to those shown in Figure 3.220. All the edges that are copied to have an arrow displayed on them. This will fix the mesh distortion in the farfield for the TOP, SIDE, and GROUND. You can also do this for the INLET and OUTLET,

Figure 3.220 Select the edges to link

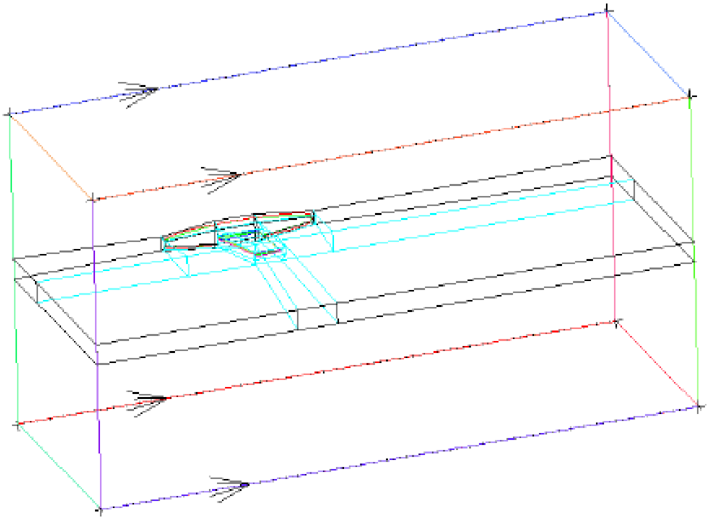
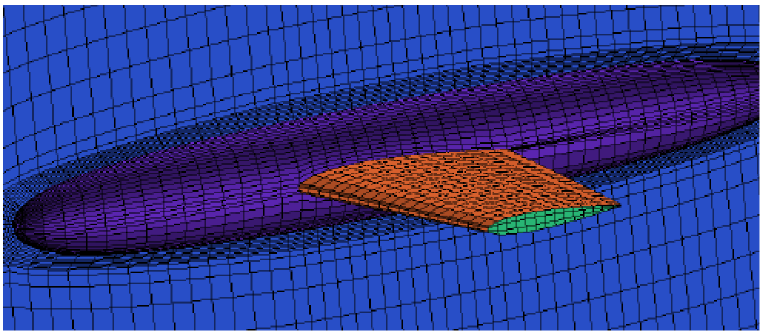


Figure 3.221 New mesh distribution after Linked bunching



Notes on Linked Bunching: Linked bunching allows the distribution of nodes on a single edge to be identical to the distribution of nodes on a series of smaller parallel edges. Linking defines a permanent relationship, called a link, between these edges. The node distribution can only be modified on the smaller edges. The user will not be able to specify any node distribution on the larger edge. The node distribution on the larger edge will automatically be updated to reflect the node distribution on the smaller edges. Note that the index space of the larger edge and all the smaller edges must be identical (the ends must meet and the same spit); otherwise the relationship cannot be defined.

k) Improving Mesh Quality

To check the general quality of the block shapes, use the Worst Blocks function.

Without toggling on the Blocks option, right click in the Display Tree on Blocks > Worst. This will highlight the worst block and give its determinant in the message window.

Based on these results, make any necessary adjustments to the blocking (using particularly Split edge and Move Vertex).

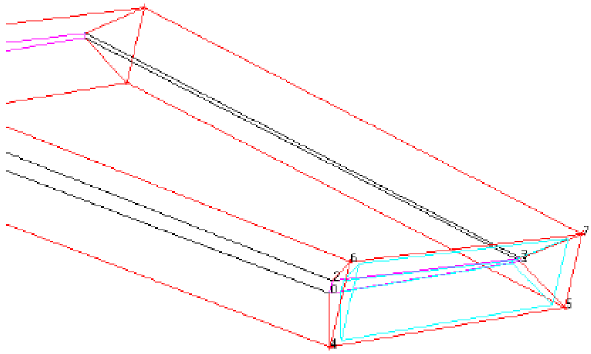
The highlighted worst block will automatically change to a different block as you edit the blocks to improve them. This is just a rough check for any badly distorted blocks. Checking the mesh quality is far more important, and checking the worst block can easily be skipped.

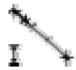
Compute the Pre-mesh with Project faces turned ON from now on, so that everything projects as in the final mesh.

Check the quality of the mesh using two metrics, Determinant 2x2x2 and Angle: Press Blocking > Pre-mesh Quality Histograms. First check the determinant with a Min-X value of 0, a max-X value of 1, and a small Max-Y height of 30 so you can see the smallest histogram bars. Select the worst bar, and right click to select Show if it is not already ON.

Use the Move Vertex and Edge Parameters to improve the blocking. Recheck your changes with the Determinant check.

Figure 3.222
Find the worst block and fix it first



A change of bunching can be accomplished using Blocking > Pre-mesh params > Edge params  .

You can recompute the mesh by toggling the Pre-mesh OFF then ON again, or go straight to the determinant check, and it will recognize the change and ask to recompute the mesh. Try to improve the Determinant to greater than 0.3. After this try to improve the Angle to better than 18 degrees.

When you are satisfied with the mesh distributions and element quality, save the replay file and blocking, and write out the mesh. Select Save from the Replay Control window, then Done after you save the file.

Save the blocking using File > Blocking > Save Blocking As.

Write out the mesh by right clicking in the Display Tree on Pre-mesh > Convert to Unstruct Mesh.

3.3: Hexa Meshing Appendix

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

3.3.1: The Most Important Features of Blocking

- CAD- and projection-based hexahedral mesh generation
- Easy manipulation of the 3D object-based topology model
- Modern GUI and software architecture with the latest hexahedral mesh technology
- Extensive solver interface library with over 100 different supported interfaces
- Automatic O-grid generation and O-grid re-scaling
- Geometry-based mesh size and boundary condition definition
- Mesh refinement to provide adequate mesh size in areas of high or low gradients
- Smoothing/relaxation algorithms to quickly yield quality meshes
- Generation of multi-block structured, unstructured, and super-domain meshes
- Ability to specify periodic definitions
- Extensive replay functionality with no user interaction for parametric studies
- Extensive selection of mesh bunching laws including the ability to graphically add/delete/modify control points defining the graph of the mesh bunching functions
- Link bunching relationships between block edges to automate bunching task
- Topology operations such as translate, rotate, mirror, and scaling to simplify generation of the topology model

- Automatic conversion of 3D volume block topology to 3D surface mesh topology
- Automatic conversion of 2D block topology to 3D block topology
- Block face extrusion to create extended 3D block topology
- Multiple projection options for initial or final mesh computation
- Quality checks for determinant, internal angle and volume of the meshes
- Domain renumbering of the block topology
- Output block definition to reduce the number of multi-block structured output mesh files
- Block orientation and origin modification options

3.3.2: Automatic O-grid Generation

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

3.3.3: Important Features of an O-grid

Generation of Orthogonal Mesh Lines at an Object Boundary
The generation of the O-grid is fully automatic and the user simply selects the blocks needed for O-grid generation. The O-grid is then generated either inside or outside the selected blocks. The O-grid may be fully contained within its selected region, or it may pass through any of the selected block faces.

Rescaling an O-grid After Generation
When the O-grid is generated, the size of the O-grid is scaled based upon the Factor in the Blocking > O-grid parameter window. The user may modify the length of the O-grid using the Blocking > Re-scale O-grid option. If a value that is less than 1 is assigned, the resulting O-grid will be smaller than the original. If, however, a value is larger than 1, the resulting O-grid will be larger.

3.3.4: Edge Meshing Parameters

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

The user has access to the following pre-defined bunching laws or Meshing laws:

Default (Bi-Geometric Law)

Uniform

Hyperbolic

Poisson

Curvature

Geometric 1

Geometric 2

Exponential 1

Exponential 2

Bi-Exponential

Linear

Spline

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

This option yields these possible functions:

Constant

Ramp

S curve

Parabola Middle

Parabola Ends

Exponential

Gaussian

Linear

Spline

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

3.3.5: Smoothing Techniques

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

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

The criteria for smoothing are:

Determinant: these criteria attempt to improve the element's determinant by movement of nodes, which are subject to geometry and association constraints.

Laplace: The Laplace option attempts to minimize abrupt changes in the mesh lines by moving the nodes.

Warp: The Warp method is based upon correcting the worst angle between two elements in the mesh.

Quality: Like the determinant criteria, the Quality criteria attempts to improve the element's interior angle by repositioning the nodes, which are subject to geometry and association constraints.

Orthogonality: The Orthogonality option attempts to provide orthogonal mesh lines at all boundaries of the model.

Skewness: The Skewness is defined differently for volume and surface elements. For a volume element, this value is obtained by taking all pairs of adjacent faces and computing the normals. The maximum value thus obtained is normalized so that 0 corresponds to perpendicular faces, and 1 corresponds to parallel faces. For surface elements, the skew is obtained by first taking the ratio of the two diagonals of the face. The skew is defined as one minus the ratio of the shorter diagonal over the longer diagonal. Thus, 0 is perfectly rectangular, and 1 represents maximum skewness.

3.3.6: Refinement and Coarsening

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

Refinement

The refinement capability is used for solvers that accept non-conformal node matching at the block boundaries. The refinement capability is used to minimize model size, while achieving proper mesh definition in critical areas of high gradients.

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.

3.3.7: Replay Functionality

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

Note: If any of the Direct CAD Interfaces are used, all geometric parameter changes are performed in the native CAD system. If any of the indirect interfaces are used, however, the parametric geometry changes are performed in ICEM DDN.

Generating a Replay File

The first step in generating a Replay file is to activate the recording of the commands needed to generate the initial block topology model. As mentioned above, this function can be invoked through File > Replay. All of the steps in the mesh development process are recorded, including blocking, mesh size, edge meshing, boundary condition definition, and final mesh generation.

The next step in the process is to make the parametric change in the geometry and then replay the recorded Replay file on the changed geometry. All steps in the mesh generation process are automated from this point.

Advantage of the Replay Function

With the Replay option, the user is capable of analyzing more geometry variations, thus obtaining more information on the critical design parameters. This can yield optimal design recommendations within the project time limits.

3.3.8: Periodicity

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

Applying the Periodic Relationship

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

3.3.9: Mesh Quality

The mesh quality functions are accessible through Meshing > Quality check. Any of the four quality check options will display a histogram plot for the user.

Determining the Location of Elements

By clicking on any of the histogram bars with the left button, the user may determine where in the model these elements are located. The selected histogram bars will change in color to pink.

After selecting the bar(s), the Show button is pressed to highlight the elements in this range. If the Solid button is turned on, the elements marked in the histogram bars will be displayed with solid shading.

Determinant

The Determinant check computes the deformation of the elements in the mesh by first calculating of the Jacobian of each hexahedron and then normalizing the determinant of the matrix. A value of 1 represents a perfect hexahedral cube, while a value of 0 is a totally inverted cube with a negative volume. The mesh quality, measured on the x-axis, of all elements will be in the range from 0 to1. If the determinant value of an element is 0, the cube has one or more degenerated edges. In general, determinant values above 0.3 are acceptable for most solvers.

The y-axis measures the number of elements that are represented in the histogram. This scale ranges from 0 to a value that is indicated by the Height. The subdivisions among the quality range are determined by the number of assigned Bars.

Angle

The Angle option checks the maximum internal angle deviation from 90 degrees 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.

Volume

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

Warpage

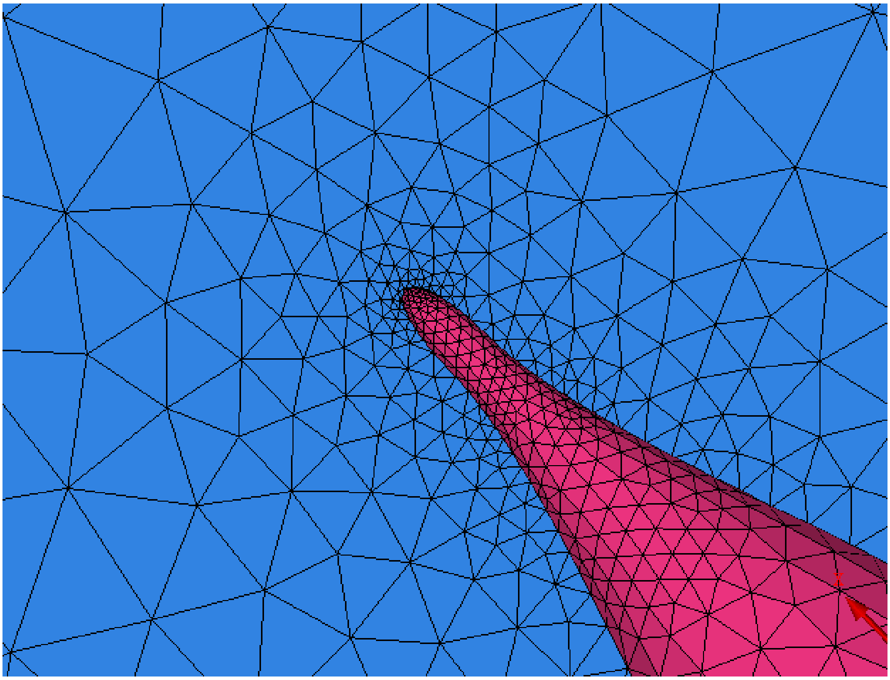
The Warpage check will yield a histogram that indicates the level of element distortion. Nodes that are in-plane with one another will produce an element with small warpage. Nodes that make elements twisted or distorted will increase an elements distortion, giving a high degree of warpage. The y-axis is the scale for the number of elements represented in the histogram -- a value determined by the assigned Height. The x-axis, which ranges from a Min of 0 to a Max of 90, is the degree of warpage that an element experiences.

3.4: Tetra

Tetra Meshing

Automated to the point that the user has only to select the geometry to be meshed, ANSYS ICEMCFD Tetra generates tetrahedral meshes directly from the CAD geometry or STL data, without requiring an initial triangular surface mesh.

Figure 3.223 This mesh was generated using ANSYS ICEM CFD Tetra. The model has approximately 550,000 tetrahedral elements.



3.4.1: Introduction

Tetra uses an Octree-based meshing algorithm to fill the volume with tetrahedral elements and to generate a surface mesh on the object surfaces. The user can define prescribed curves and points to determine the positions of edges and vertices in the mesh. For improved element quality, Tetra incorporates a powerful smoothing algorithm, as well as tools for local adaptive mesh refinement and coarsening.

Suitable for complex geometries, ANSYS ICEMCFD Tetra offers several advantages, including:

Octree-based mesh generation

Rapid model set-up

Mesh is 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 tetrahedral are matched to prescribed points and curves

Natural size automatically determines tetrahedral size for individual geometry features

Volume and surface mesh smoothing, node merging, and edge swapping

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 translation

Surface mesh editing and diagnostic tools

Local adaptive mesh refinement and coarsening

One consistent mesh for multiple materials

Fast algorithm: 1500 cells/second

Automatic detection of holes and easy way to repair the mesh

Tetrahedral mesh from a completely closed surface mesh using the Delauney meshing algorithm

Extrusion of Prism layers from the surface mesh for boundary layer calculations

Hex-core meshing from a tetra, tetra/prism, or surface mesh

a) Input to ANSYS ICEMCFD Tetra

The following are possible inputs to ANSYS ICEMCFD Tetra:

B-Spline Curves and Surfaces

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

The B-Spline curves allow Tetra to follow discontinuities in surfaces. If no curves are specified at a surface boundary, the Tetra will mesh triangles freely over the surface edge. Similarly, the prescribed points allow the mesher to recognize sharp corners in the prescribed curves. ANSYS ICEMCFD provides tools to extract points and curves automatically from the surface model at sharp features.

Triangular surface meshes as geometry definition

For triangular surface representation, prescribed curves and points can automatically be extracted from the geometry. Though the nodes of the Tetra-generated mesh will not match exactly to the nodes of the given mesh, it will follow the given geometric shape. This is especially useful when importing geometry from existing mesh databases or from systems which output stereo lithography (STL) data. The user can combine faceted geometry input with the B-Spline input.

Full/partial surface mesh

If the surface mesh is available for full/part of the geometry, the user might want to make use of that in the final mesh. This can be provided as an input to the Tetra and it makes sure that the rest of the volume and surface mesh is connected to the provided mesh.

b) Intelligent Geometry in ANSYS ICEMCFD Tetra

Using ANSYS ICEMCFD's Direct CAD Interfaces, which maintain the parametric description of the geometry throughout the CAD model and the grid generation process, unstructured grids can be directly remeshed on the modified geometry.

The geometry is selected in the CAD system and tagged with information for grid generation such as boundary conditions and mesh element sizes. This intelligent geometry information is saved with the master geometry. Parametric changes in the geometry simply require the user to write the updated geometry file for grid generation. The user can then immediately re-calculate the unstructured tetrahedral grids.

c) The Octree Approach

Tetra's mesh generation from surfaces is based on the following spatial subdivision algorithm: This algorithm ensures refinement of the mesh wherever necessary, but maintains larger elements wherever possible, allowing for faster computation.

Once the "root" tetrahedron, which encloses the entire geometry, has been initialized, Tetra subdivides the root tetrahedron until all element size requirements are met.

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.

After this is done, Tetra makes the mesh conformal – i.e., 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 geometry surfaces, curves, and prescribed points.

The mesher then determines which portion of the mesh is enclosed by surfaces bounding a Body or Material Point (based on mesh connectivity). The remainder of the mesh is deleted.

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

d) Parts Creation, Material Points, & Prescribed Points

The grouping of the geometric entities into parts in the mesher interface allows the user to define different parameters on the individual parts. Aside from assigning unique boundary condition information to the various parts, the user can define the parameters, which govern the element size for each part: maximum size, initial height and height ratio. Additionally, users can define element size on individual curves and surfaces.

With the definition of prescribed points and curves in the mesher interface, the user can control the locations of tetrahedral nodes and edges in critical areas of the mesh. As described above in the mesh generation process (c)The Octree Approach), when the mesher rounds the nodes of the mesh to match the given geometry, it first tries to project them onto the nearest prescribed points and curves.

For the cutting step of the mesh generation, Tetra requires that a material point be defined for each distinct material that is needed for analysis. (The mesher can create these automatically if none are defined.) A material point might be used to define a fluid region for CFD analysis, a solid region for FEA analysis or both fluid and solid regions for conjugate heat transfer analysis.

e) Important Features in ANSYS ICEMCFD Tetra

Natural Size

If the maximum tetrahedral size defined on surface parts is larger than a geometric entity in the specified part, the user must employ the natural size limit. The user can specify a Natural size that is proportional to the scale factor. It should be assigned a value that is slightly smaller than the smallest gap in the model, so that the mesher will further subdivide the tetrahedral to match this geometric feature.

The Natural size is the minimum size of any tetrahedral achieved via automatic subdivision for the entire model. If the user defines a smaller max element size on a geometry entity, Tetra does continue to subdivide until it meets the maximum size request. The effect of the natural size is a geometry-based adaptation of the mesh based on feature curvature and proximity.

Tetrahedral Mesh Smoother

In smoothing the mesh, the tetrahedral smoother calculates individual element quality – based on the selected criterion.

Referring then to the user specified element quality lower bound, the smoother modifies all elements below this quality criterion --nodes are moved and 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.

To exclude particular parts from the smoothing, ICEM CFD offers the utility to smooth the mesh only on visible parts. Also, the user can smooth only specific element without affecting the others

Tetrahedral Mesh Coarsener

The mesh coarsener allows the element count to be decreased while still capturing the major features of the geometry. Users can choose to freeze surface elements during the coarsening process.

If the mesh has multiple material domains and the user does not want to coarsen some of them, he/she can exclude individual material domains by specifying them in the frozen parts option. If the size checks option is used during coarsening, the resulting mesh does satisfy the selected mesh size criteria on all of the geometric entities.

Furthermore, Tetra includes a complete set of projection and smoothing tools, as well as tools for element creation, deletion, and splitting, swapping and uniform enforcement of orientation.

Triangular Surface Mesh Smoother

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

Triangular Surface Mesh Coarsener

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

Triangular Surface Editing Tools

For the interactive editing of surface meshes, ANSYS ICEMCFD Tetra offers a mesh editor in which nodes can be moved on the underlying CAD surfaces, merged, or even deleted. Individual triangles of the mesh can be subdivided or added to different parts. The user can perform the quality checks, as well as local smoothing.

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

Mesh Periodicity

Periodicity definition for ANSYS ICEMCFD Tetra meshes is well suited for rotating turbomachinery flow solutions. Meshes for any rotational or translational cyclic geometry can be generated with ease.

Mesh Density Control

The mesh Density definition for ANSYS ICEMCFD Tetra allows users to control the tetra size locally where no geometry is present. Densities can be of different shapes: point, sphere, line, arbitrary volume.

Smooth Transition

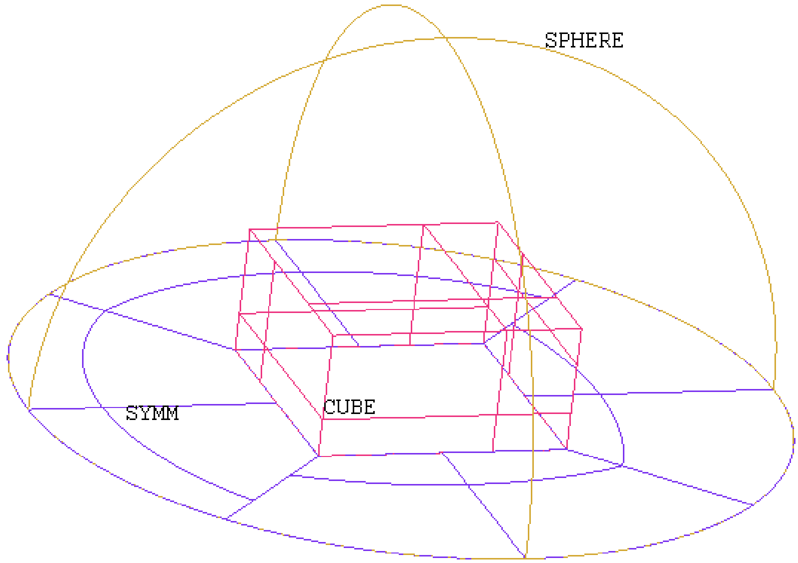
Smooth transition allows the user to fill the volume with the Delauney approach.

3.4.2: Sphere Cube

Overview

After generating a tetrahedral mesh for a hemisphere containing a cube, the user will check the mesh quality. The geometry of the Sphere Cube is shown in the figure below.

Figure 3.224 The geometry of the Sphere Cube.



a) Summary of Steps

Starting the project

Repairing the geometry.

Assigning mesh sizes

Generating the tetrahedral mesh.

Diagnostics

Saving the project

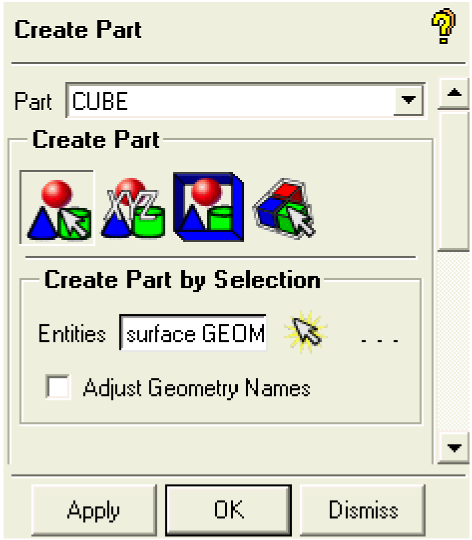
Assigning mesh sizes

b) Parts Creation

All points, curves and surfaces are initially assigned to one part, GEOM. The user needs to create and assign separate parts for surfaces, curves and bodies. The parts for the surfaces (SPHERE, CUBE, and SYMM) are labeled in Figure 3.224. (If the parts are already defined, then please go to section Reassigning Mesh sizes.)

To change the part names of surfaces, in the Display Tree widget right-click on **Parts** > **CreatePart > CreatePart by Selection**.  It will open the window as shown in Figure 3.225. Click on **Select entities**  to select the desired Surfaces (if not already in selection mode). The **Select geometry** toolbar opens. Toggle **OFF** points and curves selection and keep the toggle **ON** for surfaces selection. Enter CUBE as the Part name and select the five surfaces of the cube as shown in Figure 3.225 with the left mouse button. Press Apply (or middle-click) to create the CUBE part. The new part will appear in the Parts list in the Display Tree widget. Similarly create the SYMM (four surfaces) and SPHERE (one surface) parts. Select **Dismiss** when finished. For this example, leave the points and curves assigned to the GEOM part.

Figure 3.225
Create part window

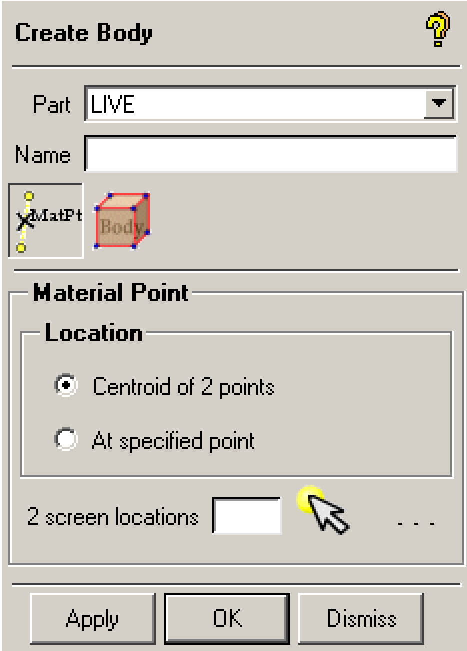


c) Creating Body

The body of the model – which will hold the tetrahedral elements - will be placed into the part, LIVE. Select

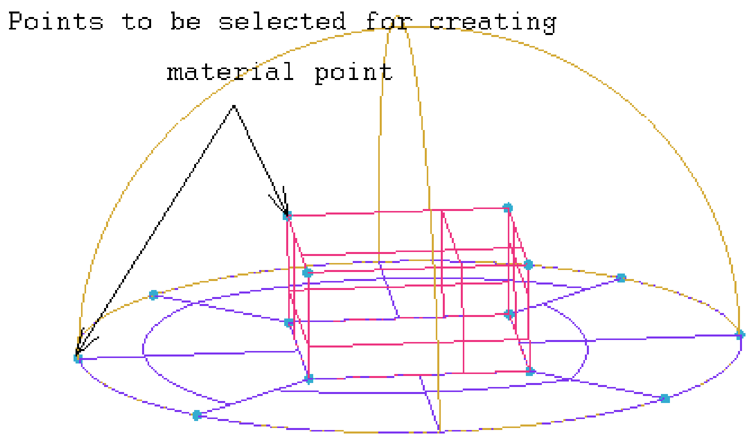
Geometry > **Create Body**.  A window will as shown in Figure 3.226. In the Create Body window, use the Material Point option. Enter Part as LIVE in the window and then in Location, enable Centroid of 2 points with the left mouse button.

Figure 3.226
Creating body window



Select two points as shown in Figure 3.227 and middle-click. LIVE should appear in the model – the small cross marks the location. The user might have to make Bodies visible in the Tree. Dynamically rotate the model to confirm that LIVE is located within the region to be meshed – outside the cube but inside the sphere.

Figure 3.227 Points to be selected for creating material point



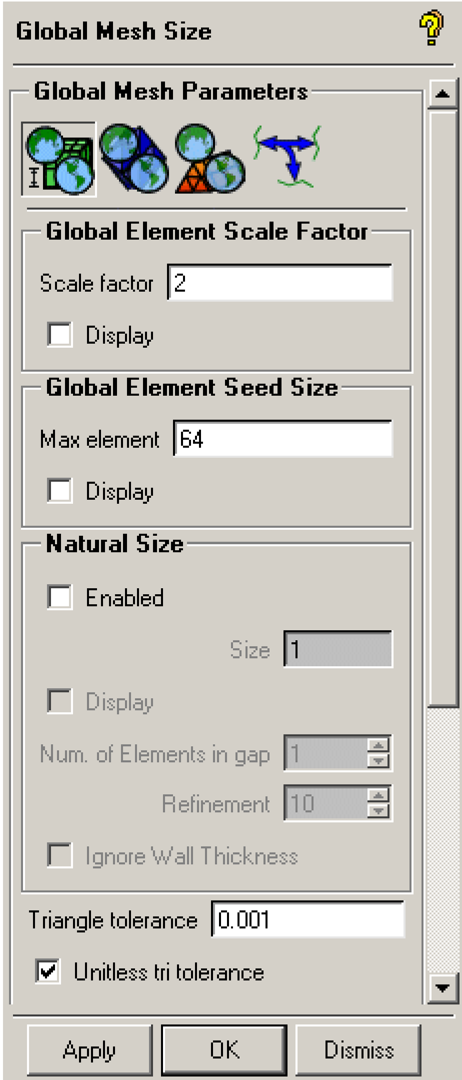
d) Set global mesh size

The user must define mesh sizes before mesh generation.

Select Mesh > Set Global Mesh Size > General Parameters to obtain the Global Mesh Size window as shown inFigure 3.228. Enter **2** for **Scale factor** and **64** for **Max element** (Figure 3.228). Press Apply followed by Dismiss to close the window.

Note: To visualize the size defined in the Global Mesh Size window, toggle **ON Display** under **Scale factor** and **Max element**. These options will provide tetra icons on the display labeled as **scale** and **max.**

Figure 3.228 Setting the Global mesh sizes for the model



From the Display Tree widget, right click on Surfaces > Tetra Sizes and Curves > Curve Tetra sizes. This displays icons representing the maximum element sizes specified on the entities.

e) Set surface mesh size

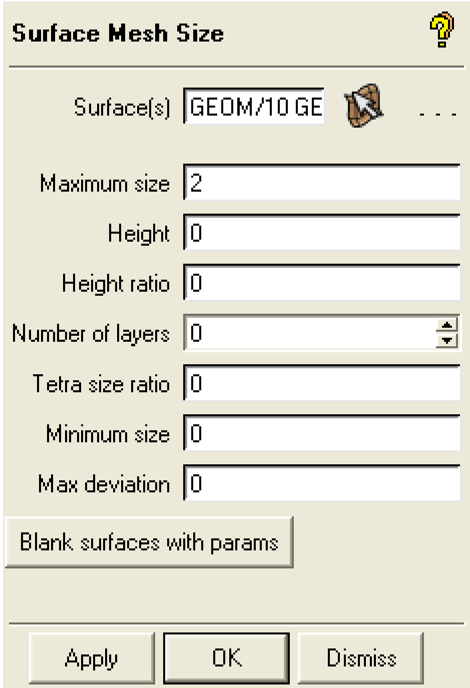
The meshing can be adjusted on the different parts of the model via Mesh > Set Surface Mesh Size.  Make only the SYMM part visible from the Display Tree widget. Select  and click 'v' on the keyboard to select the visible surfaces. Set Maximum size = 2 as shown in. Figure 3.229 Press Apply. Make only CUBE and SPHERE visible from the Display Tree widget. Select and click 'v' on the keyboard to select the visible surfaces. Set Maximum size = 1.

Press Apply followed by Dismiss to close the window

The effects of the modification of the values in the params screen can be seen on the model.

Make all Parts visible from the Display Tree widget.

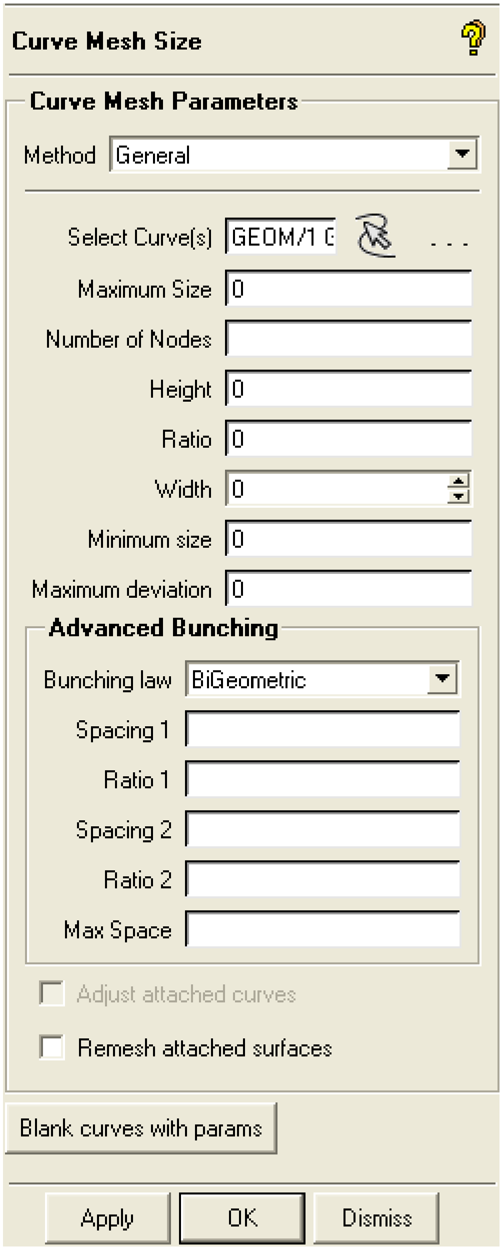
Figure 3.229
Setting the mesh sizes for the selected surface parts



f) Set Curve Mesh Size

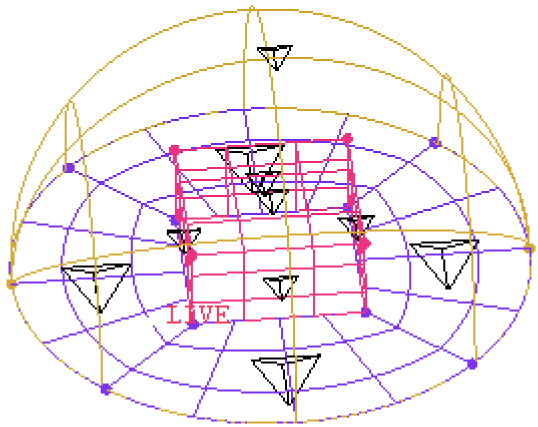
Similarly, select Mesh > Set Curve Mesh Size to open the window as seen in Figure 3.230. Select and choose all the curves by clicking 'a' on the keyboard. All the curves will be highlighted in the display. Set all the parameters in the Curve Mesh Size window to 0 and then press Apply followed by Dismiss.

Figure 3.230 Curve mesh size window



The assigned Tetra sizes are represented on the geometry as shown in Figure 3.231.

Figure 3.231
Tetra sizes on the geometry



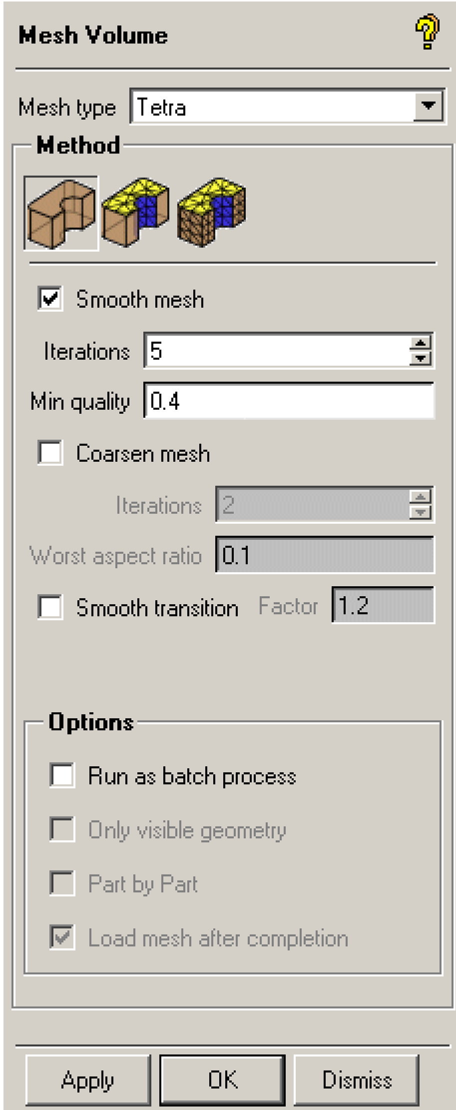
Make these displayed tetra invisible by right clicking on Surfaces > Tetra Sizes and Curves >Curve Tetra Sizes from the Display Tree widget.

When satisfied with the results, press File > Save Project to save the tetin file. Use the default project name.

g) Generating the tetrahedral mesh

Choose Mesh > Volume Meshing  > Tetra > From geometry.  The Mesh Volume window will appear as shown in Figure 3.232.

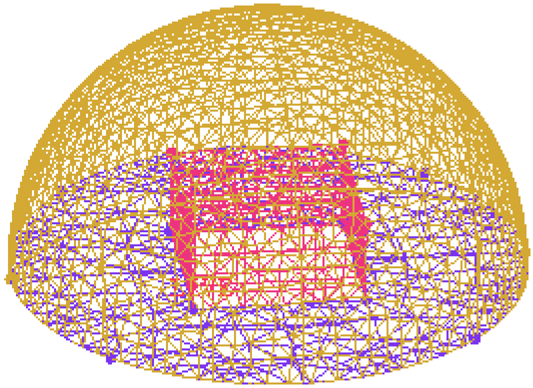
Figure 3.232
Mesh with Tetrahedral window



Press Apply to start the meshing process. The mesh opens automatically once the meshing process is complete.

Once the meshing process is completed, make triangles visible from the Display Tree widget under Mesh > Shell, so that the mesh appears as in Figure 3.233.

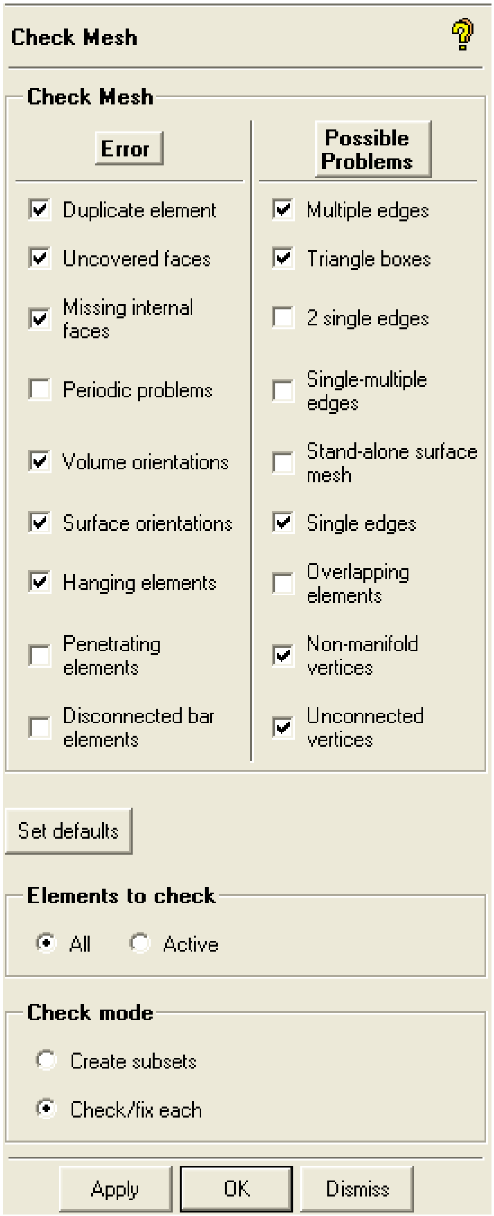
Figure 3.233
The smoothed mesh



h) Diagnostics

The user should check the mesh for any errors or problems that may cause problems for analysis. The Check Mesh window, shown in Figure 3.234 is accessible under Edit Mesh > Check Mesh. 

Figure 3.234
Check mesh window



Use the default set of checks and press Apply to check for Errors and Possible Problems in the mesh. Once the check for each possible problem is over, ICEM CFD creates subsets of the bad elements, and displays the number of elements for the particular problem in the messages area. If there are errors reported for any unconnected vertices, choose Yes to delete them. Press Dismiss to close the window. Refer to the Tetra Appendix for a detailed description of the various errors and possible problems.

i) Saving the Project

Save the mesh and geometry by selecting File > Save Project

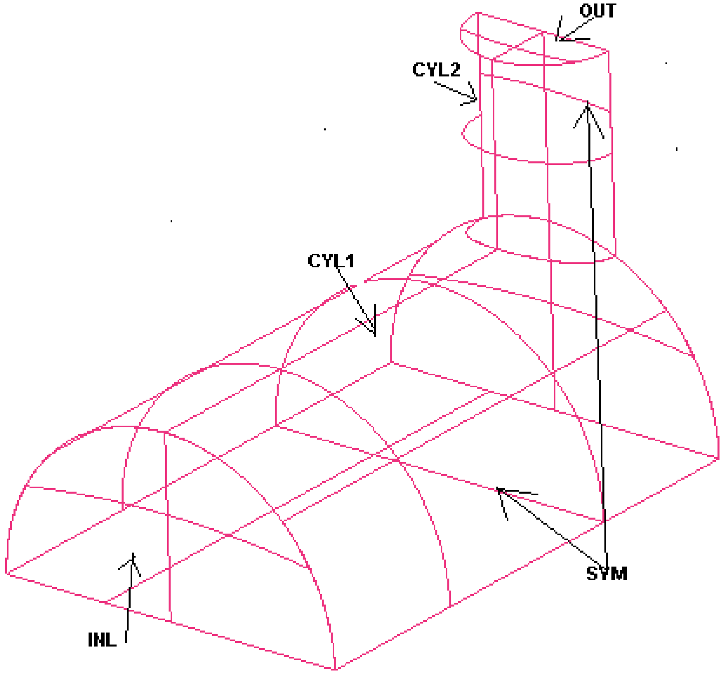
Close the project by selecting File > CloseProject.

3.4.3: 3D Pipe Junction

Overview

In this tutorial example, the user will generate a tetrahedral mesh for the three-dimensional pipe junction geometry. Prism layers will then be added. The 3D Pipe geometry is shown in Figure 3.235.

Figure 3.235 The 3D Pipe geometry with the labeled surfaces



a) **Summary of Steps**

Starting the project

Repairing the geometry

Assigning mesh sizes

Generating the tetrahedral mesh with Smooth Transition

Diagnostics

Saving the project

b) Starting the Project

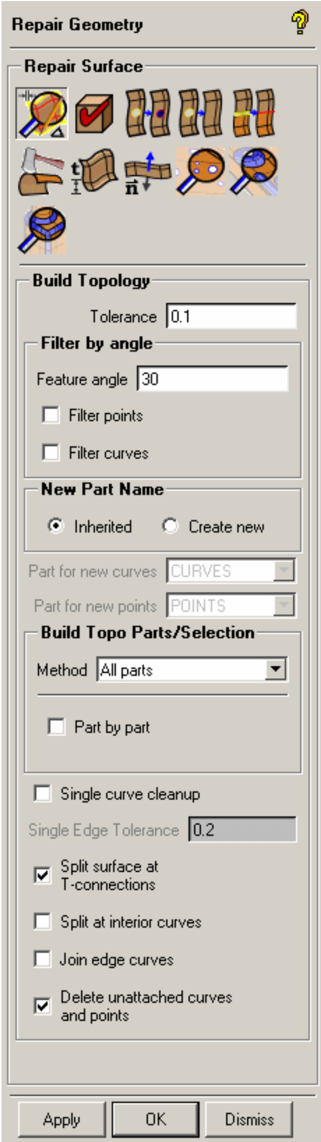
From UNIX or DOS window, start ANSYS ICEMCFD. File > Change Working Dir… $ICEM_ACN/../docu/CFDHelp/CFD_Tutorial_Files>3DpipeJunct. Open the tetin file geometry.tin.

c) Repairing the Geometry:

Select Geometry > Repair Geometry  > Build Diagnostic Topology  .The Repair Geometry window will open as shown in Figure 3.236.

Use the default values and click Apply.

Figure 3.236
Repair Geometry Window



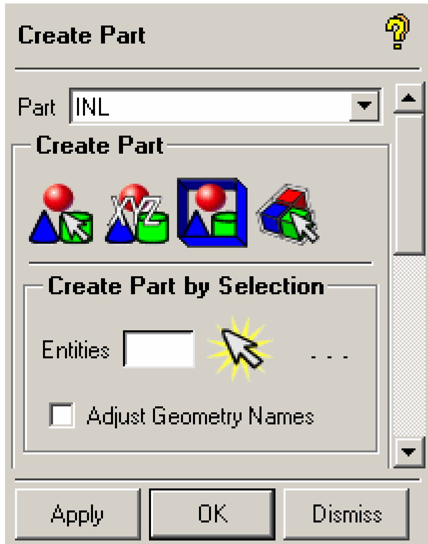
Note: Build Topology creates the curves and points necessary for Mesh generation

d) Parts Creation

If the Parts are already defined, please go to the section Reassigning Mesh Parameters.

If the Parts are not defined, create new Parts and add the appropriate surfaces to the parts. Initially, all the geometry is grouped into the GEOM part. Referring to Figure 3.235; create and add the appropriate surfaces to the Parts INL, OUT, CYL1, CYL2, and SYM.

Figure 3.237
Create part window



To create Parts for the surfaces, right-click on Parts > Create Part > Create Part by Selection.  The Create Part window is shown in Figure 3.237. Enter the part name INL and select the appropriate surface with the left mouse button. Accept the selection with the middle mouse button

Similarly create the OUT, SYM (3 surfaces), CYL1, and CYL2 Parts. Select Dismiss when finished.

Create a Part called CUR and all of the curves to it.

Note: To change the part names of curves, Right click on Parts >Create part. Make only Curves visible from the Display Tree widget. Use Create Part by Selection and make sure Curves are selectable from the **Select geometry** pop-up window. Box-select the entire model with a left-click and drag, rather than individually selecting the curve entities.

All of the points that compose the geometry will be placed in the part PTS.

Note: To change the part names of points, Right click on Parts >Create part. Make only Points visible from the Display Tree widget. Use Create Part by Selection and make sure Points are selectable from the **Select geometry** pop-up window. Once in selection mode, press ‘v’ on the keyboard to select the points.

e) Creating Bodies

The Body of the model will be assigned to the part LIVE. This will be the region that lies within the cylinders.

Select Geometry > Create Body > Material Point. Enter LIVE as the Part. Use the Centroid of 2 points option.

Click on Select locations to choose 2 screen locations in the geometry between which the material point will be created. LIVE should then appear inside the model. Dynamically rotate the model to ensure that LIVE is located within the interior of the volume, and not outside.

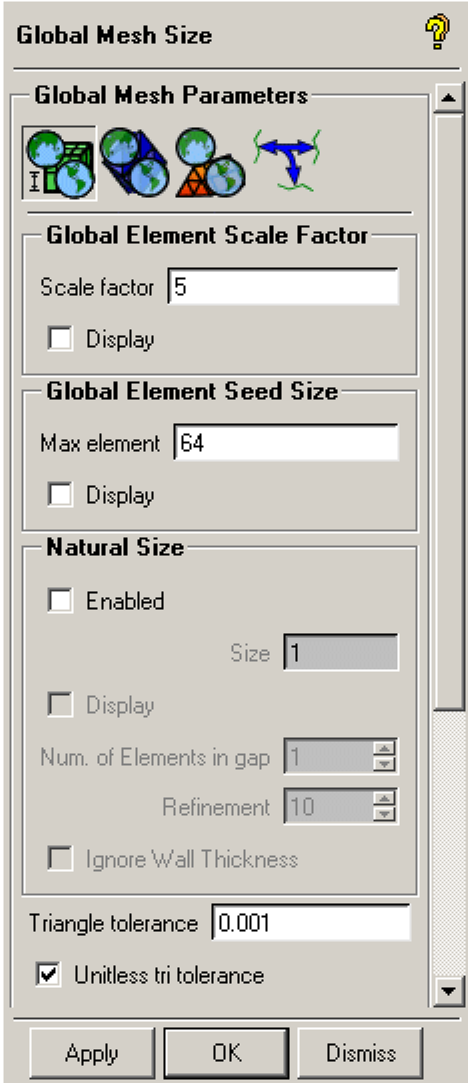
To delete empty parts, the user can right click on Parts > Delete Empty Parts from the Display Tree widget. The list should then modify itself so that empty parts are no longer included and the messages area states that GEOM has been deleted.

f) Reassigning Mesh Parameters

The user will now specify the mesh size on the entire model with Mesh > Set Global.Mesh Size > General Parameters (Figure 3.238). Change the Scale factor to

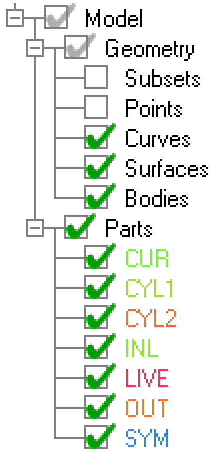
5 and Max element to 64. Click Apply. The scale factor is a multiplier for all size specifications applied to parts or individual curves and surfaces. The local element size will be equivalent to the local size applied to that entity, multiplied by the Scale factor.

Figure 3.238 Assigning Global mesh sizes to the entire model



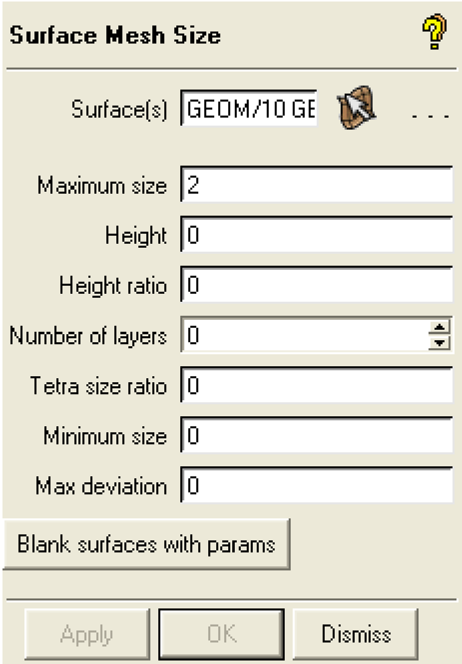
The user can make the parts visible from the Display Tree widget that appears in Figure 3.239.

Figure 3.239
Select parts to modify



To change the mesh size on specific surfaces, select Mesh > Set Surface Mesh Size and use Select surfaces to choose the required surfaces. Click 'a' on the keyboard to select all the surfaces. In the Surface Mesh Size window Figure 3.240 assign Maximum element size of 2. Press Apply and Dismiss.

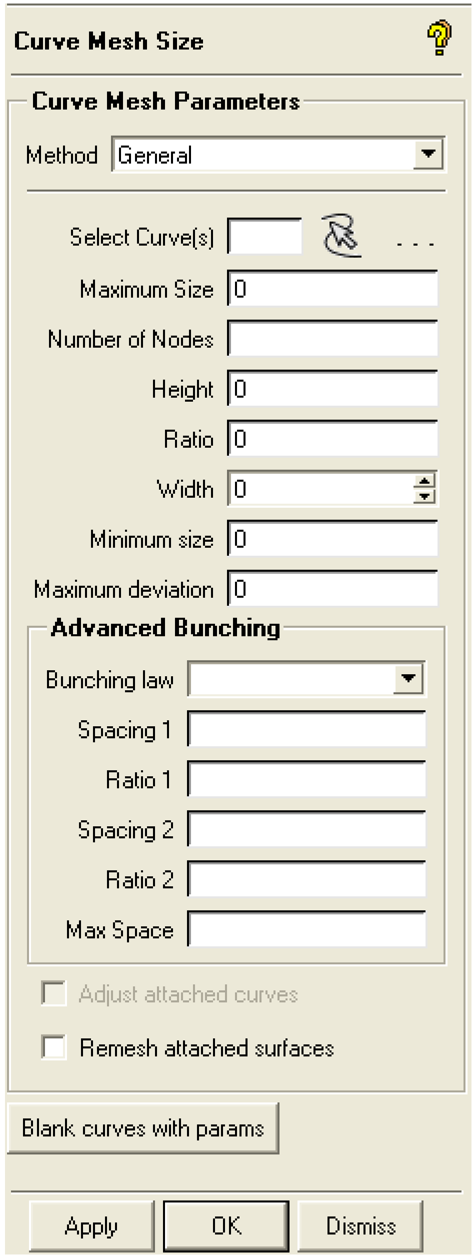
Figure 3.240 Adjusting the surface mesh sizes Associated to the selected surfaces



g) Setting curve mesh size

Select Mesh > Set Curve Mesh Size. Select and click 'a' on the keyboard to select all curves. Set Maximum Size = 0 to all the curves as shown in Figure 3.241. Select Apply and Dismiss.

Figure 3.241
Adjusting the curves Mesh sizes

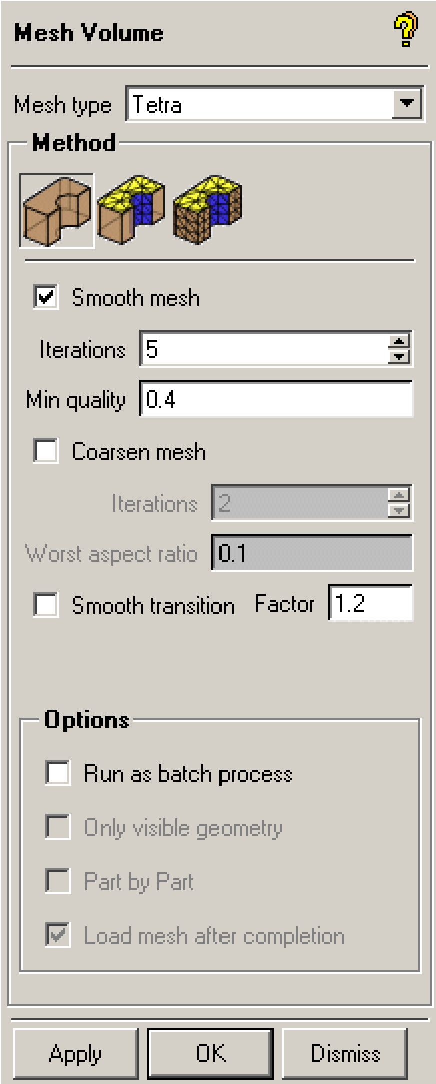


When satisfied with the meshing parameters, press File > Save Project to save the changes made to the model before proceeding further.

h) Generating tetrahedral Mesh

Choose Mesh > Volume Meshing > Tetra > From geometry . A Mesh Volume window will open as shown in Figure 3.242.

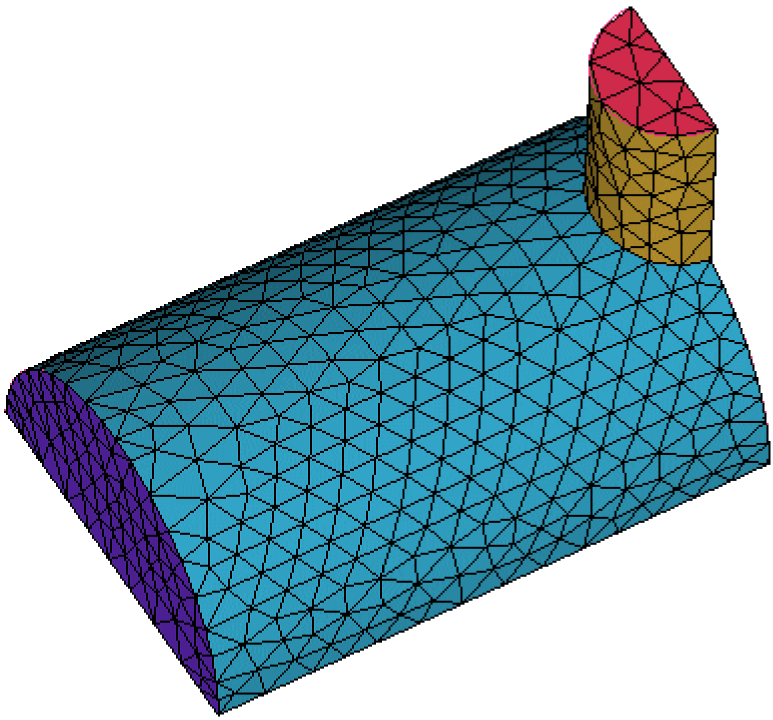
Figure 3.242 Mesh with tetrahedral



Enable Smooth transition and set Factor = 1.2.
Press Apply. This will initially use the Octree tetra mesher to create the volume and surface mesh.

With Smooth transition active, the volume (tetra) elements will be discarded. The Delauney method will then be used to re-fill the volume with tetra elements.
The mesh will appear in the display when the meshing process is finished. Make sure that the Mesh type Shells in the Display Tree widget is active so that the mesh, represented by its triangular surface elements, should appear as in Figure 3.243

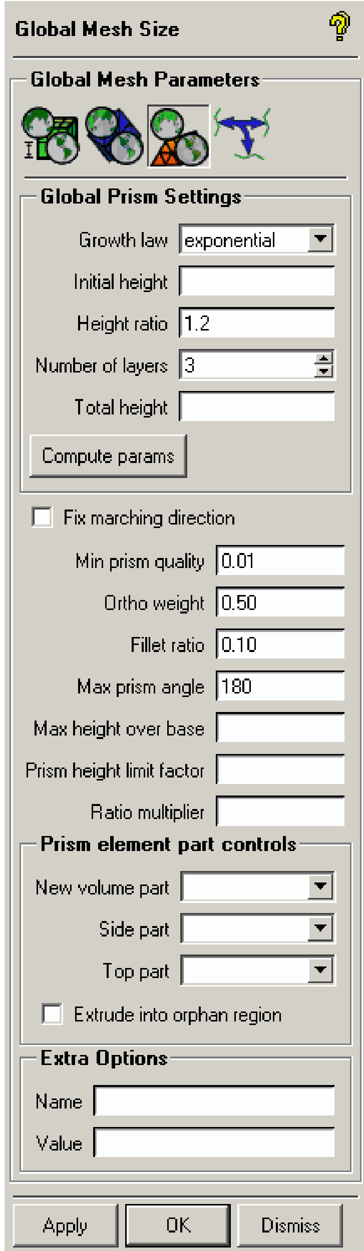
Figure 3.243 The tetrahedral mesh



i) Assigning Prism Parameters

Select Mesh > Set Global Mesh Size > Prism Meshing Parameters to open the Global Prism Settings panel as show in Figure 3.244. Use the defaults to grow 3 layers with a Height ratio of 1.2. Rather than specifying an Initial height, the thickness of the prism layers will be based on the size of the local surface triangle. Press Apply.

Figure 3.244
Global mesh size window



j) Generating the Prism Mesh

Select Mesh > Mesh Prism.  (Save and overwrite the project as prompted.) This opens the Mesh with Prisms window as in Figure 3.245.

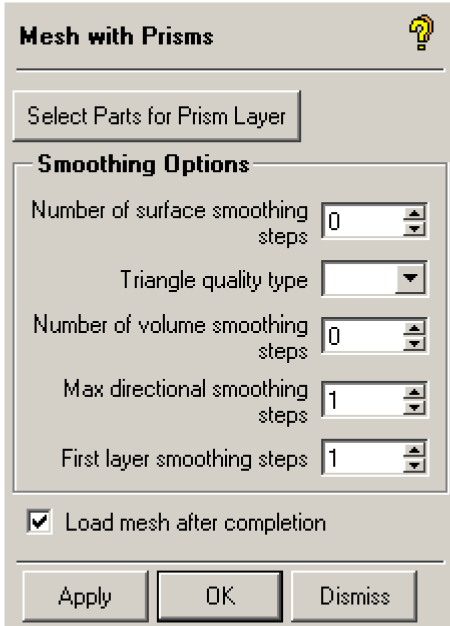
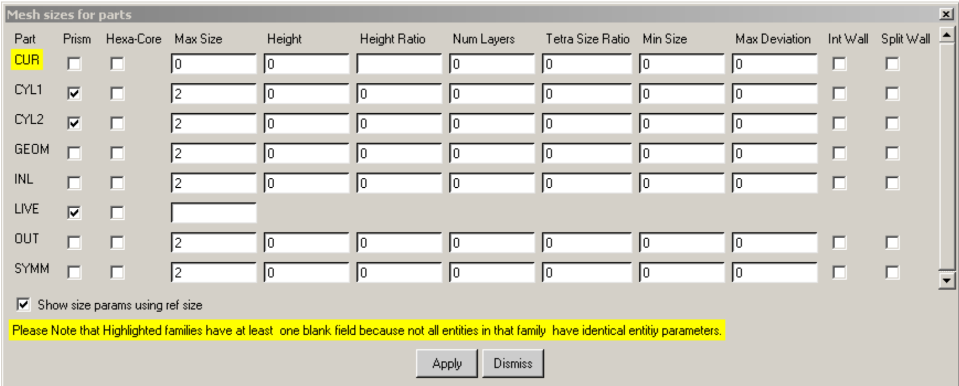


Figure 3.245
Mesh with prism window

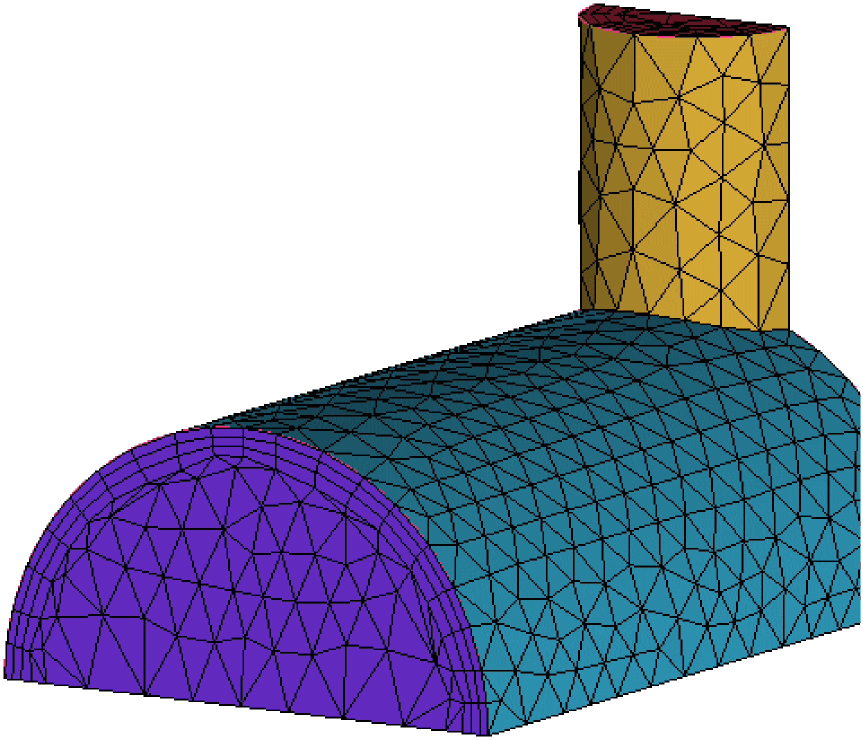
Click on Select Parts for Prism Layer. In the Mesh sizes for parts window, enable Prism for the CYL1 and CYL2 Parts, as shown in Figure 3.246. Click Apply and Dismiss.

Figure 3.246
Selecting parts for prism mesh generation



In the Mesh with Prisms window (see Figure 3.245) enable Load mesh after completion and select Apply to start the prism mesher.
The resultant tet/prism mesh is shown in Figure 3.247.

Figure 3.247 Tetra with prism mesh



k) Diagnostics

Check mesh

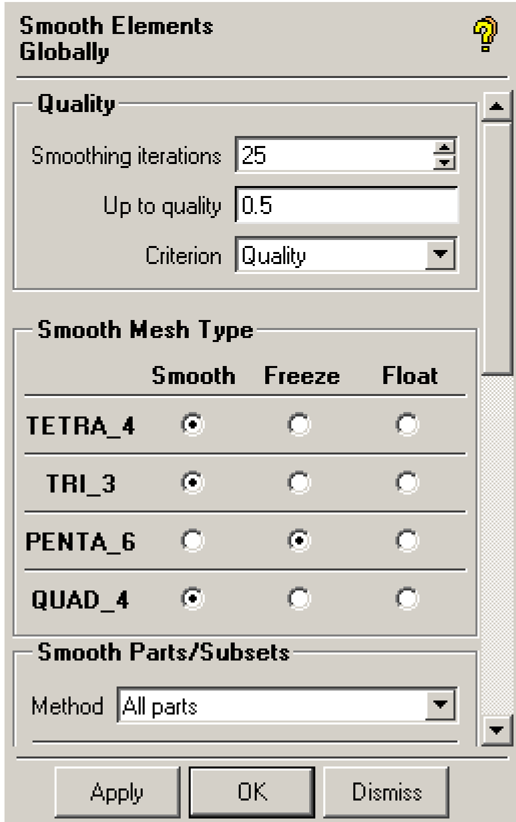
As done in the previous example, the user should go through all of the checks for Errors and Possible problems to ensure that the mesh does not contain any flaws that would cause problems for analysis. If a question box pops up asking whether to delete disconnected vertices, respond by pressing Yes.

Smoothing the Mesh

Once the ICEM CFD Mesh Editor has reported no Errors or Possible problems, the user may continue by smoothing the generated Tetra/Prism mesh.

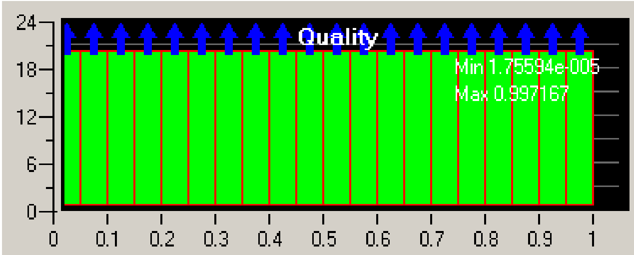
Press Edit mesh > Smooth Mesh Globally  to start the Smooth Elements Globally window as shown in Figure 3.248

Figure 3.248
Smooth elements globally



Set Smoothing iterations to 25, Up to quality to 0.5, and Criterion to Quality. With a Tet/Prism mesh, first smooth the interior elements without adjusting the prisms. Under Smooth Mesh Type set PENTA_6 (the prisms) to Freeze. The mesh quality histogram is next to the messages area as shown in Figure 3.249. Press Apply to start the smoother.

Figure 3.249 The Quality histogram before smoothing

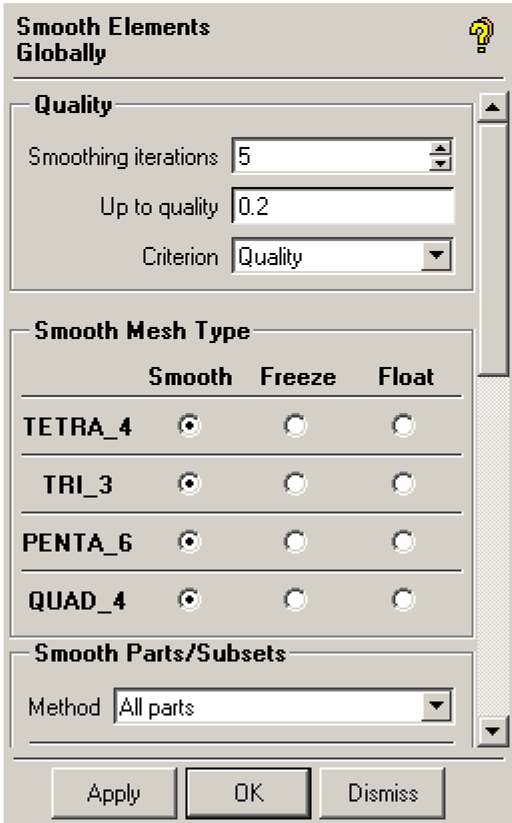


The improvements are noticeable in the histogram seen in Figure 3.250. There is no element below the quality of 0.2. Note that only element types set to Smooth are included in the histogram.

Figure 3.250 The Quality histogram after smoothing

Now set PENTA_6 back to Smooth to allow the prisms to adjust as well. So as not to modify them drastically, set Smoothing iterations to 5 and Up to quality to 0.2 as in Figure 3.251. Select Apply to do the final smoothing.

Figure 3.251
Smooth elements globally window



l) Saving the project

Save the mesh by selecting File > Save Project. If a question box pops up to delete disconnected vertices, respond by saying Yes.

Close the project by selecting File > Close Project.

3.4.4: Fin Configuration

Overview

In this tutorial example, the user will generate and smooth a combined tet/prism mesh. The mesh will be for the fluid region surrounding a general fin configuration, as well as for the surface of the fin. The user will define a mesh density region for mesh control around the fin. Finally after meshing user will perform Laplace smoothing which generally provides more uniformly spaced mesh. The Hex-Core utility will then be used to obtain a bulk of hex elements in the main volume.

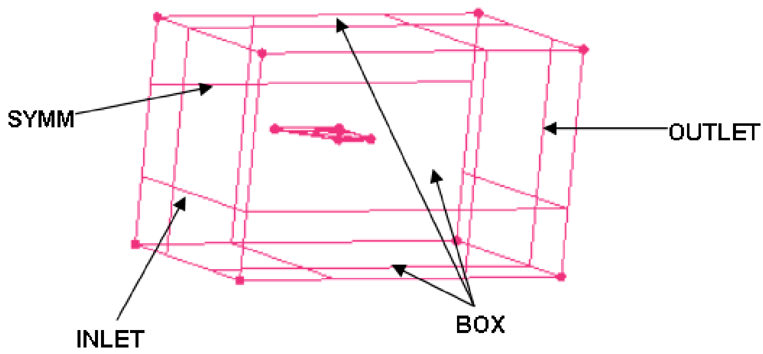
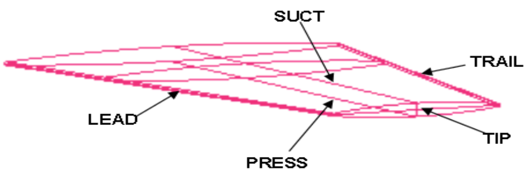


Figure 3.252 The geometry with the labeled Surfaces of the exterior domain (Top). The labeled Surfaces of the fin(Bottom)



a) **Summary of steps**

Starting the project

Repairing the Geometry

Assigning Mesh sizes

Generating Tetrahedral/Prism mesh

Diagnostics

Generating Hex-Core mesh

Smoothing

Saving the project

b) Starting Project

From UNIX or DOS window, start ANSYS ICEMCFD. File > Change Working Dir… to $ICEM_ACN/../docu/CFDHelp/CFD_Tutorial_Files>FinConfig. Open the geometry file geometry.tin.

c) Repairing the geometry

Repair the geometry via Geometry > Repair Geometry  > Build Diagnostic Topology  .

Run Build Topology with the default parameters. Press Apply

d) Parts Creation

If the project only contains one part, the user needs to create and assign separate parts for surfaces, curves, and material/body. The surface parts (BOX, INLET, LEAD, OUTLET, PRESS, SUCT, SYMM, TIP, and TRAIL) are indicated in Figure 3.252.

Enable the display of Surfaces from the Display Tree widget. To change the part names of surfaces, right-click on Parts > Create Part. In the Create Part window enter the appropriate Part name. Choose Create part by selection, toggle OFF Selection of points, curves and bodies from the Select geometry toolbar. Click on Select entities  to select the required surfaces with the left mouse button. After selection is over press middle mouse button to complete the selection process. Continue to create the other Parts for the surfaces. Then Press Apply followed by Dismiss to close the window.

Leave the curves and points in the GEOM part.

e) Defining the Material Point

The material of the model will be assigned to the material point, LIVE. The LIVE material is the region that lies within the BOX, surrounding the fin. Select Geometry > Create Body > Material Point. Enter Part as LIVE. Click on Select location(s) to select two locations. Select two locations (e.g. one on the fin tip and one at a box corner) and middle-click such that the LIVE material point will appear within the volume of interest.

f) Setting Global Mesh Size

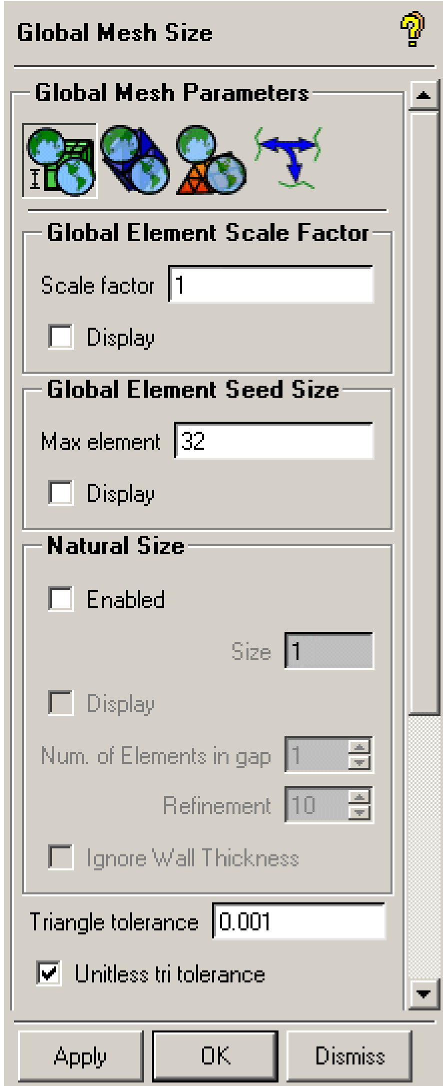
Choose Mesh > Set Global Mesh Size > General Parameters to open the Global Mesh Size window (Figure 3.253).

Enable Natural size. Enter the value of 32 for Max element

Enter a Scale factor of 1. This value is a parameter that is referred to by other mesh parameters. The Scale factor allows the user to globally control the mesh size instead of changing the mesh size on each and every entity. For further description of this option, refer to the on-line help.

Press Apply followed by Dismiss to close the window.

Figure 3.253
Editing the Global Mesh sizes



g) Setting Surface mesh size

Select Mesh > Set Surface Mesh Size  > Select surface(s).  Assign a Maximum size of 1 to the fin surfaces: i.e. LEAD, TRAIL, PRESS, SUCT and TIP. For the outer box, define Maximum size of 4: i.e. for parts BOX, SYMM, INLET and OUTLET.

h) Setting curve mesh size

Select Mesh > Set Curve Mesh Size > Select curves. All curves are in the GEOM part. Assign size 0 to all curves. Press 'shift-P' to get the list of parts Select GEOM from the list and Accept. Set Maximum Size to 0 and Apply.

The mesh density will be a region in which one can prescribe a certain maximum element size. This is useful for refining the mesh in a volumetric region.

i) Creating the density regions

Select Mesh > Create Mesh Density. A Create Density window Figure3.254 will appear.

Use the Points option and click on Select location(s) to select two points from the screen. Select one point at the base of the fin, centered on the leading edge; and one point at the tip of the fin, also centered on the leading edge. See Figure 3.255. Middle-click to complete the selection. Set Size = 0.0625, Ratio = 0, and Width = 4. Press Apply in the Create Density window. Create a similar density at the trailing edge of the fin.

Figure3.254
Create density window

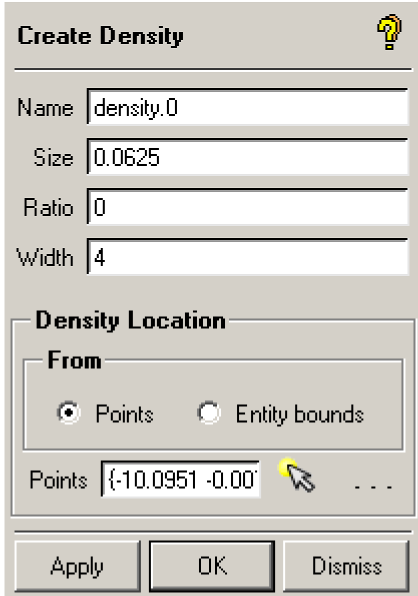
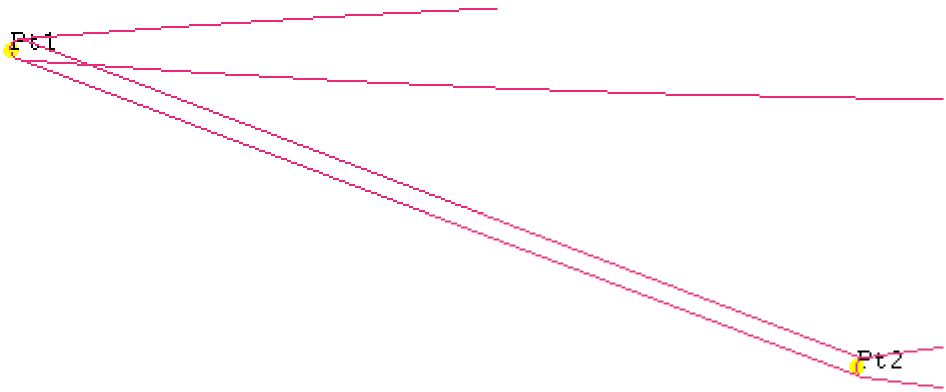


Figure 3.255 Density Creation at the Leading Edge

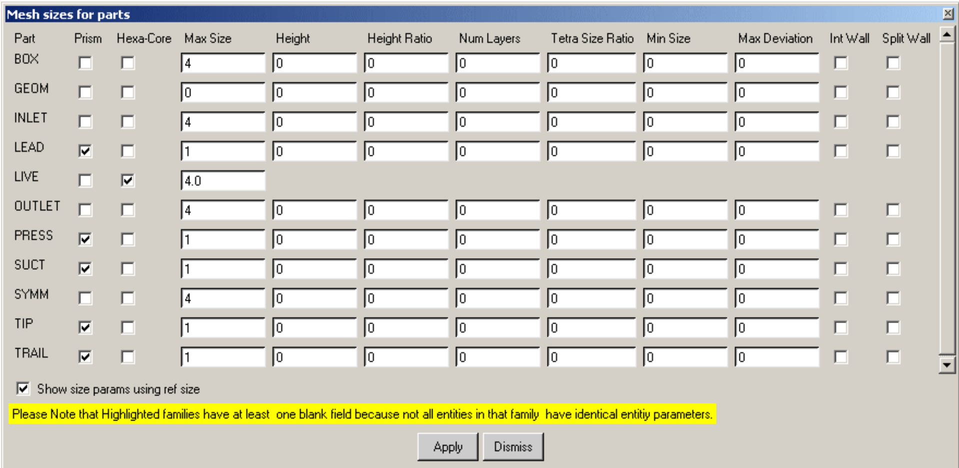


j) Defining Parts for prisms and hexa-core

Select Mesh > Set meshing Params By Parts to open the Mesh sizes for parts window. Enable Prism for the LEAD, PRESS, SUCT, TIP and

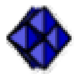
TRAIL parts as in Figure 3.256. Enable Hexa-Core for LIVE and set Max Size to 4.0. Select Apply and Dismiss.

Figure 3.256
Selecting parts for Prism and Hexa-core mesh



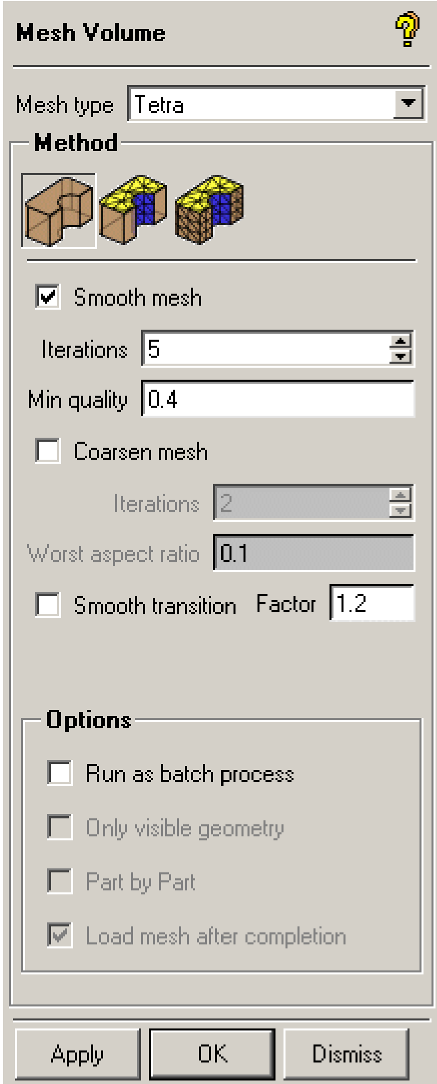
Choose File > Save Project to save the changes in the Tetin file. Accept the default project name.

k) Generating the Tet/Prism Mesh

Press Mesh > Volume Meshing  > From Geometry . Set Mesh type to Tetra + Prism Layers as shown in Figure 3.257. Press Apply to generate the mesh using the default parameters.

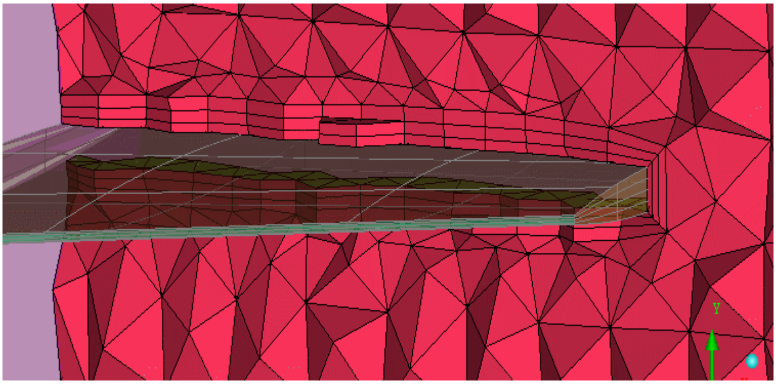
Note: The default prism parameters (Mesh > Set Global Mesh Size > Prism Meshing Params) indicate 3 prism layers using the local prism thickness based on the surface triangle size (as no initial height is specified).

Figure 3.257
Mesh withTetrahedra parameters



A cut plane through the complete tetra/prism mesh should appear as in Figure 3.258.

Figure 3.258 Tetra/Prism mesh



l) Diagnostics

As in the SphereCube example, the user should go through all of the checks for Errors and Possible problems to ensure that the mesh does not contain any flaws that would cause problems for analysis. For checking, Edit Mesh > Check Mesh

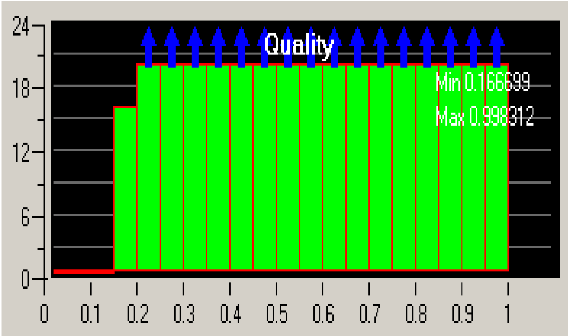
m) Smoothing

After confirming that no serious problems or errors exist, continue by smoothing the generated tetra/prism mesh.

Select Edit Mesh > Smooth Mesh Globally. 

Several elements have a lower quality than the "acceptable" value of 0.3 (as shown in Figure 3.259). Set the Smoothing iterations to 5 and the Up to quality to 0.4. Make sure Criterion is set to Quality.

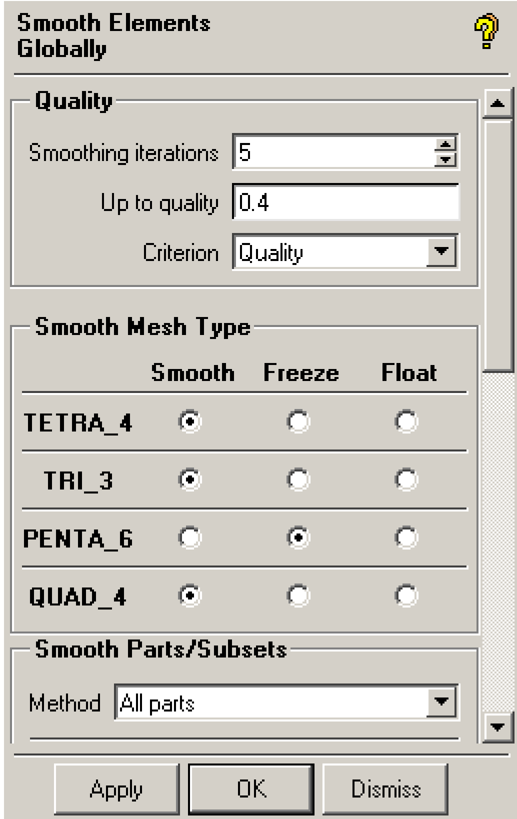
Figure 3.259 Quality of the mesh before smoothing



Select the Smooth option for TETRA_4 and TRI_3 and QUAD_4. Select the Freeze option for PENTA_6 (the prisms) as shown in Figure 3.260.

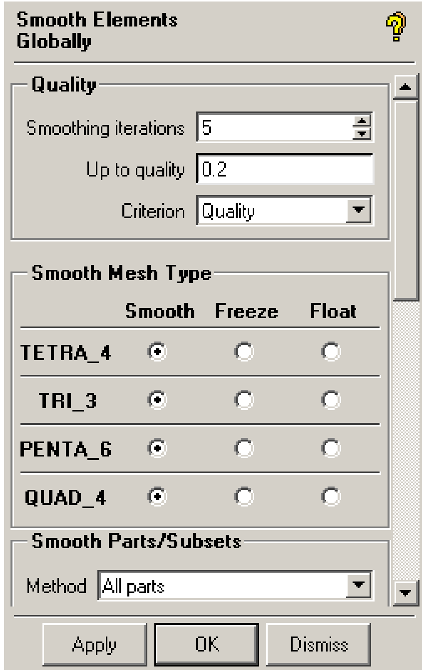
Press Apply when the operation is complete, a new histogram will be displayed.

Figure 3.260 Smooth Elements Globally



The histogram doesn't change much, indicating quality can't be improved without allowing the prism elements to smooth as well. Now set PENTA_6 to Smooth as well. Set Up to quality to 0.2 as shown in Figure 3.261 to prevent dramatic warpage of the prism layers. Press Apply.

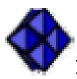
Figure 3.261 Smooth Elements Globally



Select Info > Mesh Info. Scan the messages area to find the number of elements in the LIVE part (the volume elements belong to this part – tets/prisms). The information indicates there are roughly 800,000 elements in the LIVE part.

Save the project.

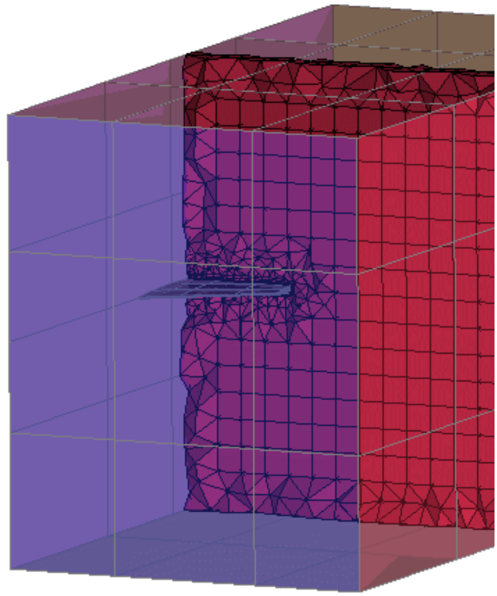
7.4.8 Building the Hex-Core mesh

Select Mesh > Volume Meshing  > From surface mesh.  Set Mesh type to Hexa-Core. This will replace the core volume elements with Hex elements of a size = 4.0, according to the parameters set in the Mesh sizes for parts window.

Click Apply to start the mesher.

A cut plane through the mesh is shown in Figure 3.262

Figure 3.262: Cut-plane showing volume mesh



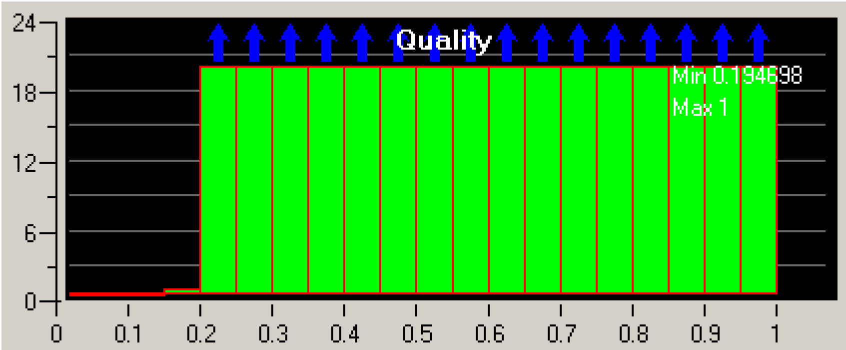
Select File > Save Project As… and give the project a new name. This preserves the existing tet/prism mesh in one project, and the hex-core mesh in another.

Go through the smoothing steps again – as in the base tet/prism mesh. Remember to Freeze the prism elements until the core has been smoothed as much as possible. The final histogram should be similar to that shown in Figure 3.263.

Select Info > Mesh Info. Now the information in the messages window indicates the LIVE part has roughly

200,000 elements. The Hex-Core operation cut the mesh by roughly 75%.

Figure 3.263 Final quality histogram



n) Saving the project

Save the mesh by selecting File > Save Project. If a question box pops up asking whether to delete disconnected vertices, respond by saying Yes.

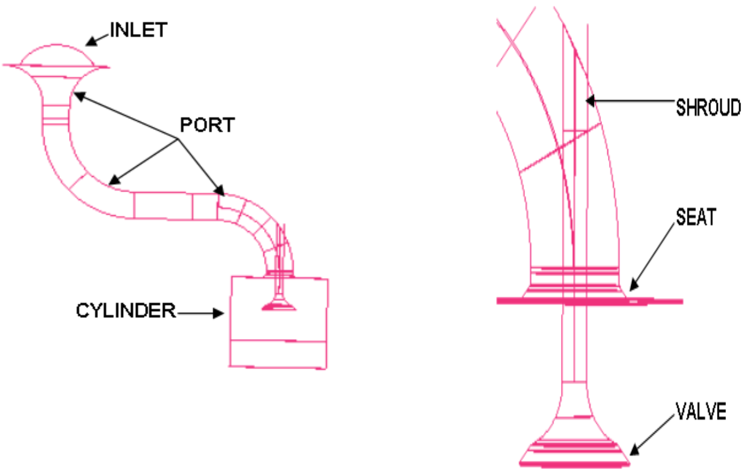
Then close the project by selecting File > Close Project.

3.4.5: Piston Valve

Overview

In this tutorial example, the user will define a thin cut in the Geometry to mark a region where ANSYS ICEMCFD Tetra will generate a thin layer of elements. The user will then generate and smooth a tetrahedral mesh for a piston valve configuration

Figure 3.264 Piston valve figure with labeled surfaces



a) Summary of Steps

Starting the project

Repairing the geometry

Assigning the mesh sizes

Generating the tetrahedral mesh

Conversion from Linear to quadratic

Diagnostics

Saving the project

b) Starting the Project

From UNIX or DOS window, start ANSYS ICEMCFD. File > Change working directory $ICEM_ACN/../docu/CFDHelp/CFD_Tutorial_Files>piston valve project. Choose its Tetin file geometry.tin.

c) Repairing the geometry

To repair this geometry, select Geometry > Repair geometry > Build Diagnostic Topology using the tolerance value of 0.3. Press Apply in this window with the default parameters.

d) Parts Creation

After running repair geometry, define separate parts for surfaces (CYL, INLET, PORT, SEAT, SHROUD and VALVE). Likewise, define parts for curves (CUR) and points (PTS) as indicated in Figure 3.264.

e) Defining the Material Point

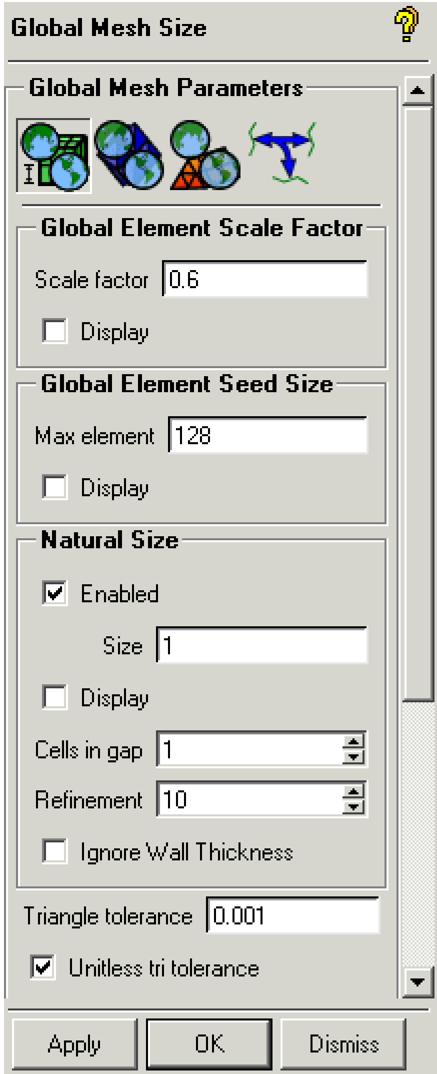
The material of the model will be assigned to the material point, LIVE.

Utilize the Geometry> Create Body. This will invoke a Create body window. Here press Material point function to assign this material point. Then press Choose an item button chooses option to select 2 Screen locations. With the left mouse button, select two locations on the port. Press the middle mouse button, and LIVE should appear in the model. Dynamically rotate the model to confirm that LIVE is located within the geometry, and not outside the geometry.

f) Setting global mesh size

Press Mesh > Set Global Mesh Size > General parameters to bring up the global mesh size as seen in Figure 3.265. Enter 0.6 as the Scale factor and 128 for Max Element. Switch ON Natural size limit by providing the value of 1 as shown in Figure 3.265.

Figure 3.265
Global Mesh Size Window



Natural size allows ICEM CFD Tetra to determine local tetrahedral sizes based on the size of the features in the model. The mesher will compare the size of the elements to the radius of curvature of the curves and

surfaces and the distance between the non-intersecting curves and surfaces. Like other size parameters, Natural size is a multiplier of the scale factor. The value given by Natural size multiplied with the scale factor represents a minimum element size.
The Natural size > Refinement parameter defines the number of edges along a radius of curvature. Refinement parameter is used to compute the Natural size, consequently, the larger this parameter, the smaller will be the computed the Natural size. Refinement should always be a positive integer value.

Note: For more information on Natural size, see the ICEM CFD on-line help.

The value entered for **Natural size** limit is a factor multiplied by the **scale factor**. The N**atural size** limit will be the minimum size of any tetrahedral for entire model. Only if the user defines a smaller max size on the geometry entity, the geometry will be having the smaller size. These values will be used for the entire model by default, but the user can also define specific natural sizes for each part by defining Minimum size for individual entities.

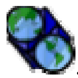
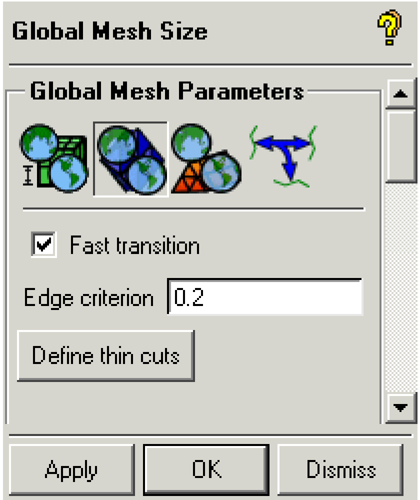
Select Mesh > Global Mesh Parameters  > Tetra Meshing Parameters , it will open the window as shown in Figure 3.266.

Figure 3.266
Tetra Meshing Parameters Window

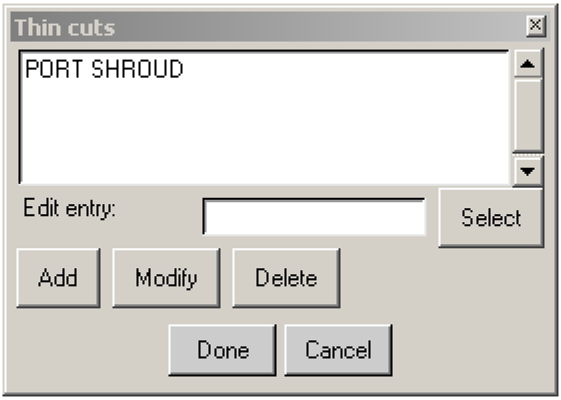


To add a thin cut to the model, consider a region between any two parts that may be thinner than the tetrahedral size defined on then, in which a fine layer(s) of tetrahedral elements may be created.

Note: For more information on Thin cuts, refer to the ICEM CFD on-line Help.

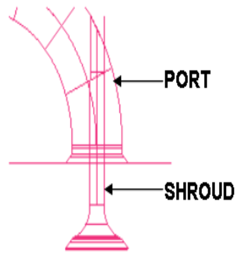
Select Define thin cuts, the Thin cuts window will appear, press Select. The first part is PORT and the second part is SHROUD. Press Add, and the two part names will appear in the 'Thin cuts' window, as shown in Figure 3.267.

Figure 3.267
The Thin cuts window



Using parts in the Display Tree widget, the user is able to browse the parts of the model. The close-up view of the PORT and SHROUD part is shown in Figure 3.268, between which the thin cut will be defined.

Figure 3.268
The PORT and SHROUD parts



When finished, press Done.

Select Apply in the Global Mesh Size window to activate the modifications.

Press Dismiss to Close the Global Mesh Size window.

g) Setting the surface mesh size

Select Mesh > Set Surface Mesh size to set the meshing parameters on the surfaces of the model. Select and Press the "a" keyboard key to select all surfaces.

In the Set Surface Mesh Size window (Figure 3.269), enter Maximum Element size of 16 and press Apply.

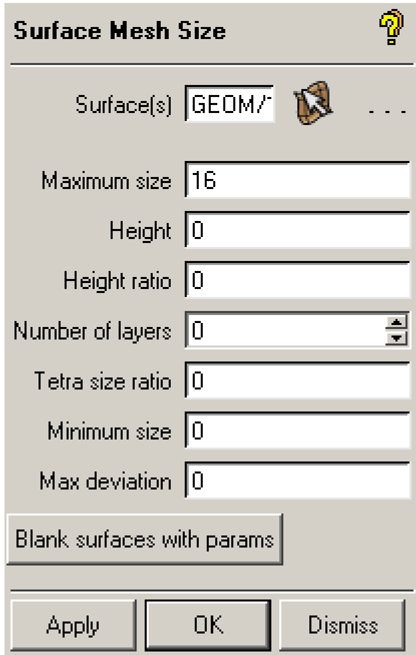


Figure 3.269
Edit the surface meshing sizes

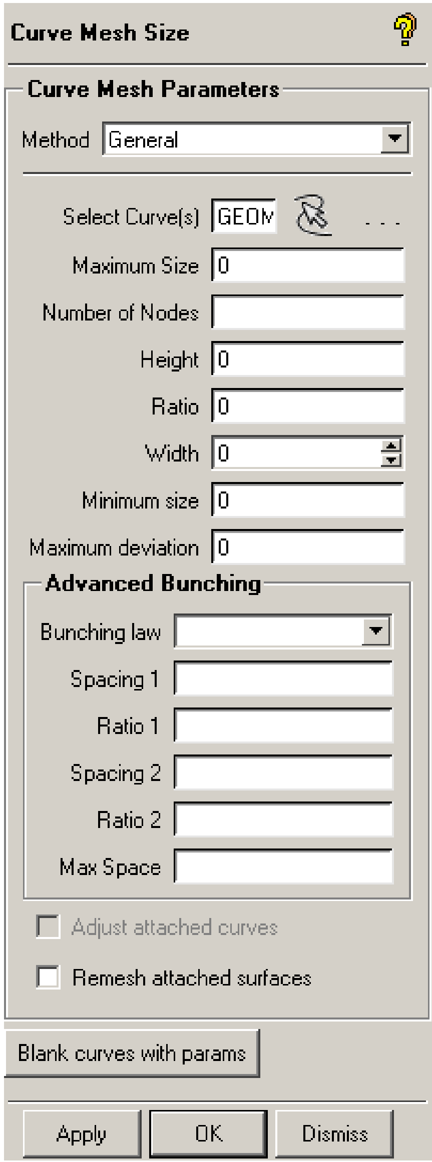
h) Setting curve mesh size

Next, select Mesh > Set Curve Mesh size to set the meshing parameters on the curves of the model.

Select Press the "a" keyboard key to select all curves.

In the Curve Mesh Size window (Figure 3.270), enter all the parameters 0. Press Apply followed by Dismiss to close the window.

Figure 3.270
Curve mesh size window



Make sure that Surfaces and Curves are visible in Display Tree widget. Right click on Surfaces > Tetra sizes and

Curves >Tetra sizes. Check that all the surfaces have the Maximum element size of 16, and curves have size of 0.

Choose File > Save project to save the additions to the tetin file.

i) Generating the Tetrahedral Mesh

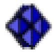
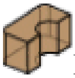
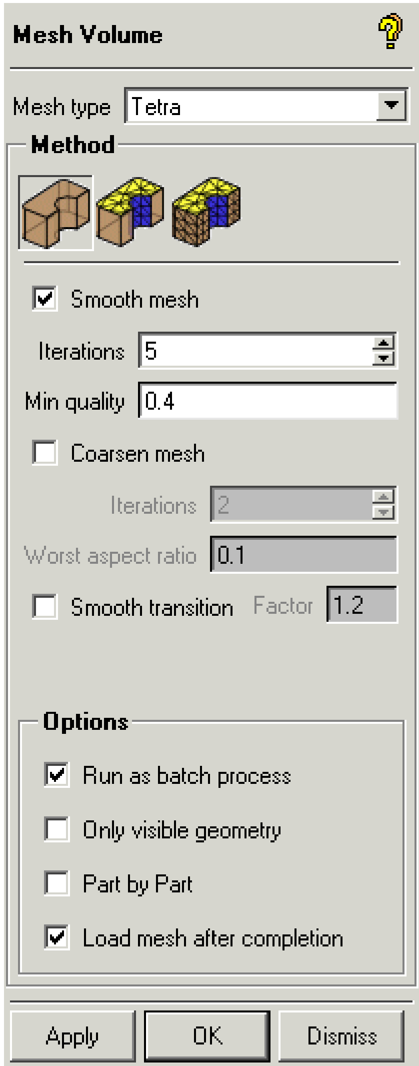
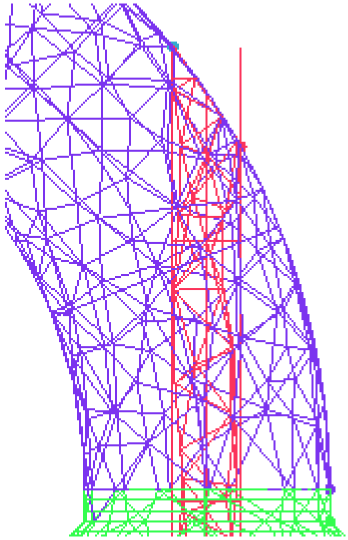
Click on Mesh > Volume Meshing  > From Geometry  Press Apply to create the mesh

Figure 3.271
Mesh with Tetrahedral window



When the meshing process is complete, the user should make sure that the element type Triangle is highlighted in the Display Tree widget. Zoom in on the region between PORT and SHROUD where the thin cut was defined, the mesh should resemble Figure 3.272.

Figure 3.272
The mesh in the Thin cut region



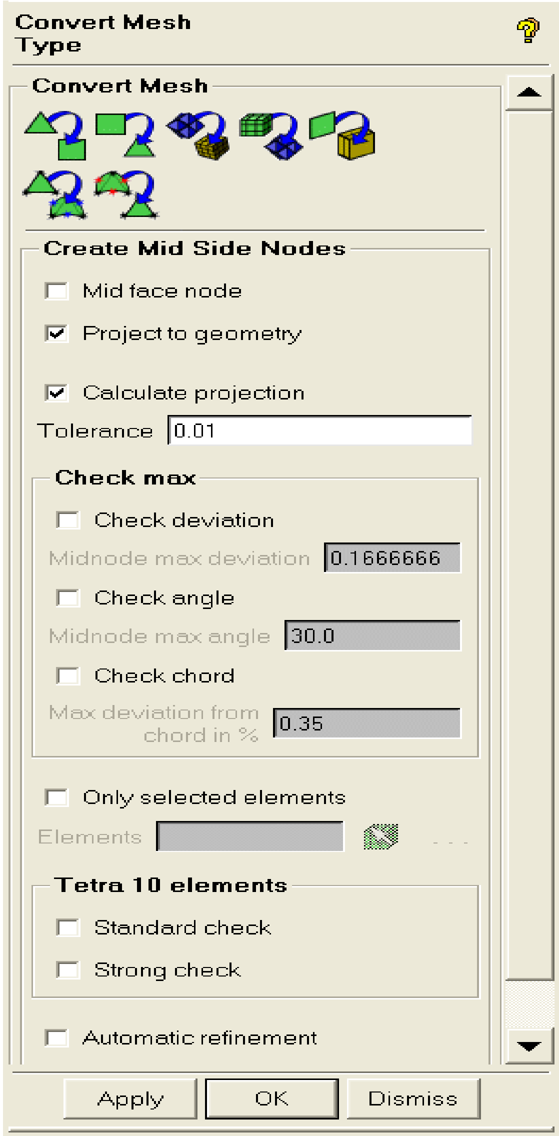
j) Checking the mesh

Check the mesh for different errors and possible problems with Edit Mesh > Check Mesh.

k) Conversion of Elements from Liner to Quadratic

Choose Edit mesh >Convert Mesh Type > Create Mid Side Node . A new window will appear as seen in Figure 3.273. Switch ON **Mid Face node** and choose all elements to be converted to quadratic before selecting Apply.

Figure 3.273 Linear to quadratic window



The TRI_3 elements get converted to TRI_6 and TETRA_4 get converted to TETRA_10 as seen in the Figure 3.274.

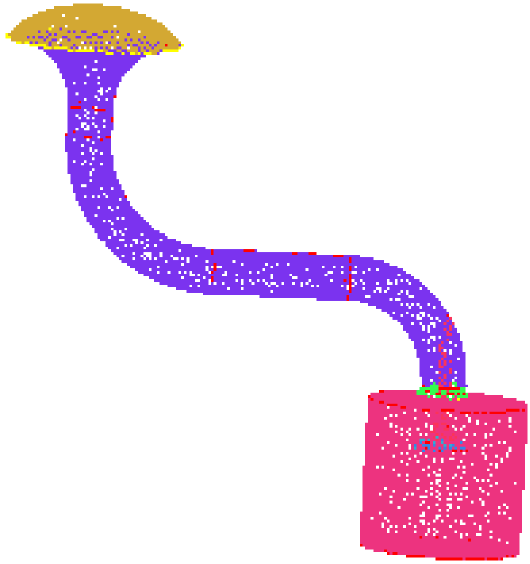


Figure 3.274 The mesh after conversion

l) Diagnostics

As in the SphereCube example, the user should go through all of the checks for Errors and Possible problems to ensure that the mesh does not contain any flaws that would cause problems for analysis.

m) Saving the Project

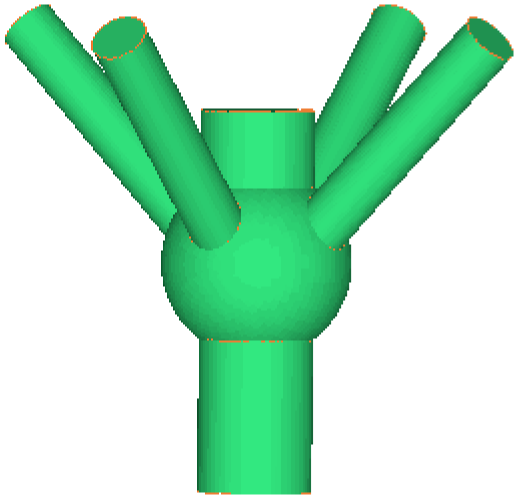
Save the mesh by selecting File > Save Project.

Close the project by selecting File > Close Project.

3.4.6: STL Configuration

Overview

In this tutorial example, the user will import STL data to the ANSYS ICEMCFD Mesh Editor. After extracting a single curve from the model, the user will segment this one curve into multiple curves to be used for segmenting the surfaces. The surfaces and material points will then be defined according to parts. From there, the user can set meshing parameters for the model for input to Tetra. Lastly, the user will generate a tetrahedral mesh for the configuration.



a) Summary of Steps

Converting STL file to Geometry file
Extracting the curves
Segmenting the curves
Segmenting the surfaces
Assigning the parts
Generating the tetrahedral mesh
Adding prism a layer

Subdividing the prism layer
Saving the Project

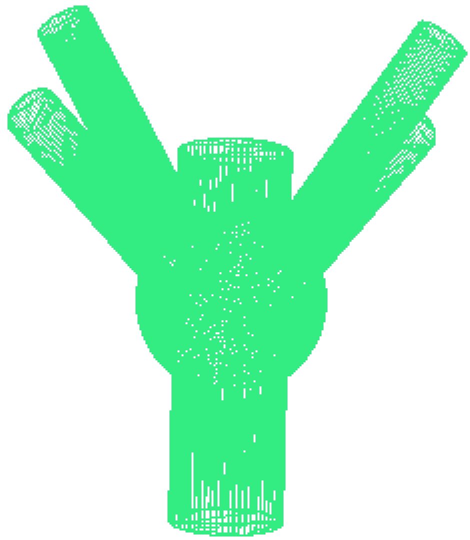
b) Converting the STL File to a Geometry File

Start the ANSYS ICEMCFD Mesh Editor.

Select File > Import Geometry > STL to translate the STL data into a triangular surface mesh. From the File selection window, choose the geometry.stl file and press Open.

When the import is complete, the extracted surface will be displayed in its Simple form.

Figure 3.275 Detailed display of the surface



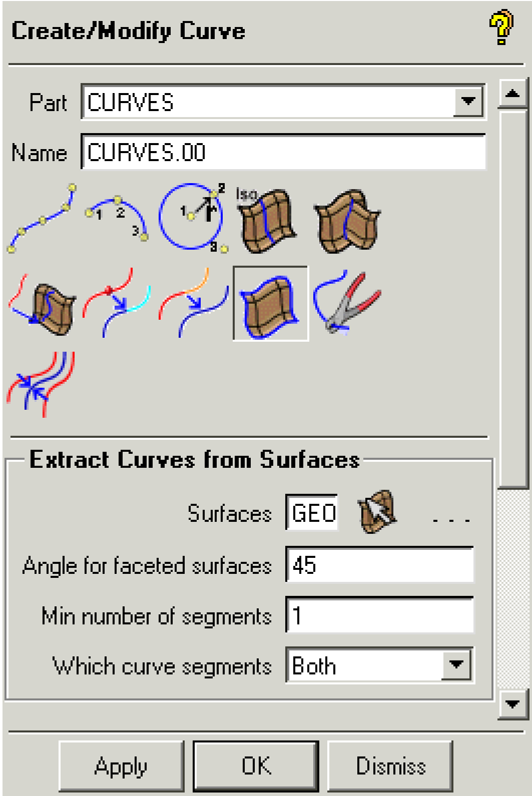
In the Display Tree widget, Select Surfaces >Show Full to see the surface. Reset the display to Simple before preceding to speed-up the display.

c) Extracting the Curve

At this point, the geometry is defined by only one surface in one part. In order to proceed, the user need to extract the curves from the surface and rename all of the entities.

Select Geometry > Create/Modify Curve > Extract curves from Surface . Click on select surfaces option for surfaces and Select the surface with the left mouse button. Complete the selection by pressing the middle mouse button. Enter 45 as the Angle for Faceted surface and 1 as the Min. number of segments and enter Both under Which curve segments as seen in Figure 3.276.

Figure 3.276
Settings for extracting curve



Press Apply to perform the extraction.

The curve extraction procedure is based on the Angle. If the Angle between parts of the surface is greater than a threshold angle, a curve segment will be created along the common edge. The extraction of curves can be based on the boundary, the interior or both.

Only interior: Where only curves on the interior are extracted, provided the feature angles between the triangles are greater than the threshold value.

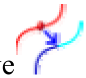
Only boundary: Where all curves are extracted from the boundary of the surface family.

Both: Both interior and exterior regions are extracted.

d) Segmenting the Curves

The curve, extracted so far, includes several closed-loop curves in distinct regions of the model, but is still considered one curve. Now user needs to segment this curve into unique entities.

To do so, the user is asked to see the curve names by right clicking on Curves > Show Curve Names in the Display Tree widget. Notice that the curve is named CURVES/0.0 is the first curve in CURVES.

Select Geometry > Create/ Modify Curve  > Segment Curve  .

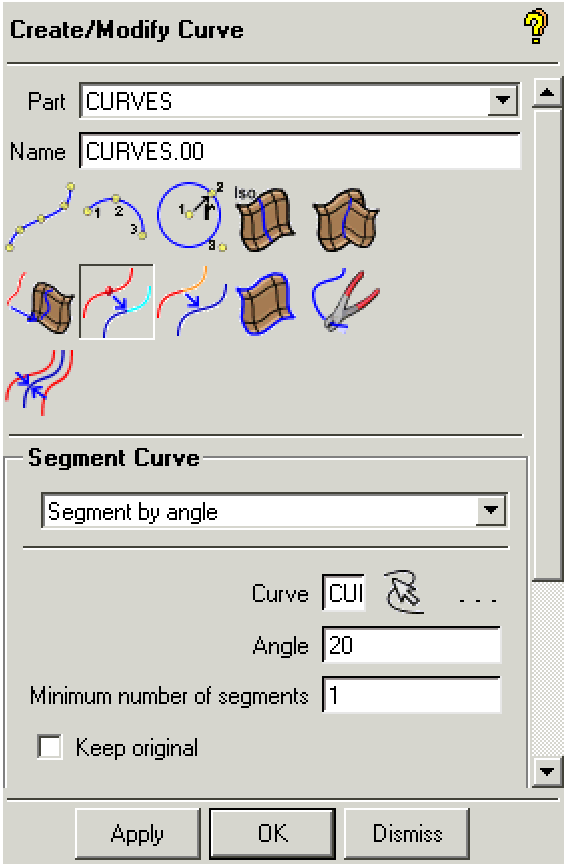


Figure 3.277 Segmenting curves

Select the CURVE/0.1 curve and complete the selection. A Segment curve window will appear as shown in Figure 3.277.

In the Segment Curve window, select segment faceted by angle of 45 as the criterion. Press Apply to perform the segmentation.

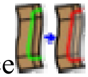
This will create a set of 22 new curves whose names will label them in the display (CURVES/0.0.1 to CURVES/0.0.22).

Note: If the user is experimenting with the angle, and is unsure of the outcome of the operation; use **Keep Original**, in order to try different operation parameters on the curve.

In some models, particularly those with sharp features where the angle of extraction was set very small, the user would next extract points from the curves.

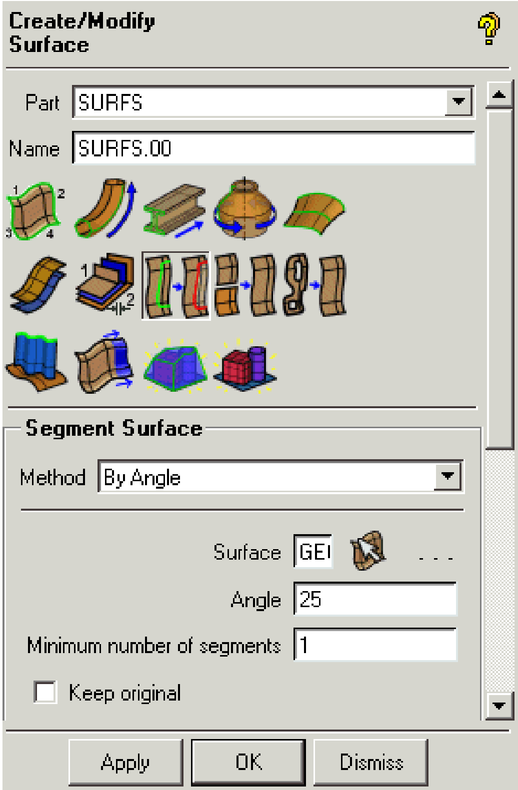
e) Segmenting the Surface

Now, based on these new curves, the user can segment the surface into regions on which to generate mesh.

Select Geometry >Create/Modify Surface  > Segment/Trim Surface 

Select the surface with the help of the left mouse button and complete the selection using the middle mouse button. A Segment Surface window will appear as seen in Figure 3.278. In the Method select Faceted Surfaces by Angle.

Figure 3.278 Segmenting the surfaces



To segment the surface with Faceted Surfaces by Angle, select surface from screen.

Enter Angle as 25 and keep other option as default.

Press Apply to create the new surfaces (FAM.1/0.1 to FAM.1/0.8).

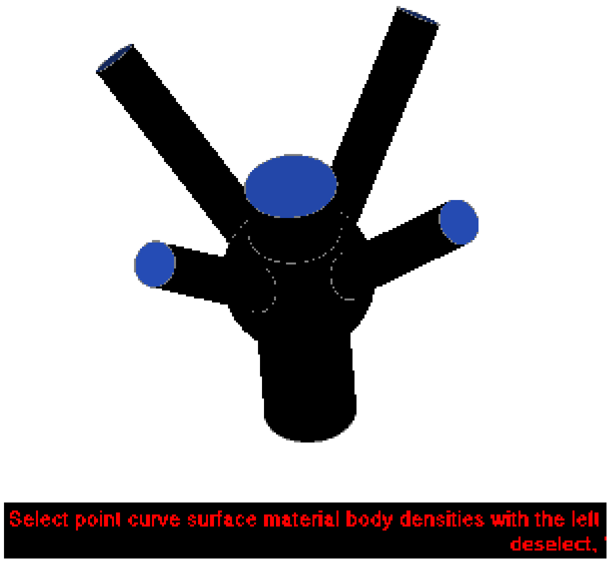
Turn OFF the Curves names by right clicking on Curves > Show curve names and turn ON Surfaces > Show surface names in the Display Tree widget to see the new surfaces labeled.

f) Parts creation

Rename new surfaces in order to create distinct parts in the model.

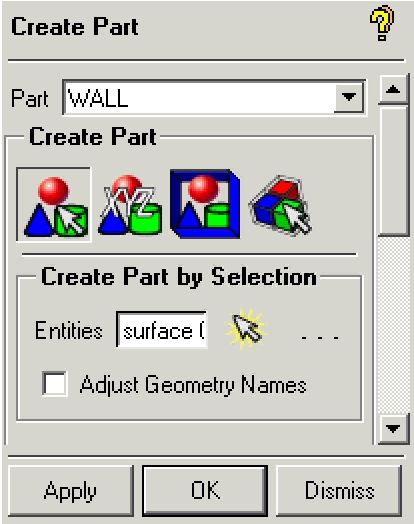
Right click on Parts >Create part to open the create part window. In that window click on Create Part by selection. Then Click on  to select the desired, surfaces, curves, points and material. Select the two surfaces that make up the walls of the model as seen in Figure 3.279 and complete the selection.

Figure 3.279
Selecting WALL entities



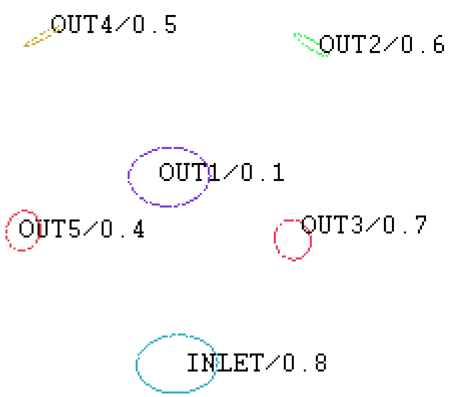
In the Create Part window, enter the part as WALL and press Apply.

Figure 3.280
Creating the WALL part



Make the WALL part invisible in the Display Tree widget for easy selection of the remaining surfaces. Using the same procedure as when creating the WALL part, create parts for the remaining surface segments as seen in Figure 3.281.

Figure 3.281
Part definition of inlet and outlet surfaces



Similarly, add each curve to the part of the surface they circumscribe by right clicking on Parts> Create part > Create Part by Selection. Then click on the to select the desired option. Now the toolbar selection window will pop up on the screen. Toggle OFF selection for points, surfaces and materials

Display only curves in the Display Tree widget and select the remaining curves by drawing a box around them using the left mouse button Continue by adding them to the WALL part.

g) Defining the Material Point

To perform the cutter operation in the tetra mesh generation process, the user needs to define material points inside and outside the volume. Since user is interested in flow within the model, the material inside the volume will be called LIVE and the material outside the volume ORFN.

Select Geometry > Create body > Material point .

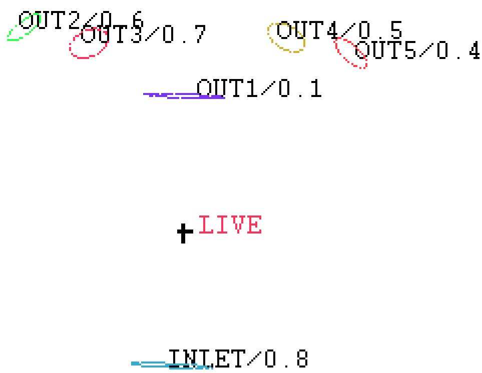


Figure 3.282 Creating the LIVE volume part

Select a point on INLET and a second point diagonally opposite the first point on OUT1 and Press Apply.

The material point LIVE will appear inside the volume as shown in Figure 3.282. Rotate the model to make sure that it's within the volume.
If the material point is in the wrong position, choose Geometry > Delete Bodies select it and then Apply. The Material point will be removed from the screen. Now redefine the material point using the same procedure described above.

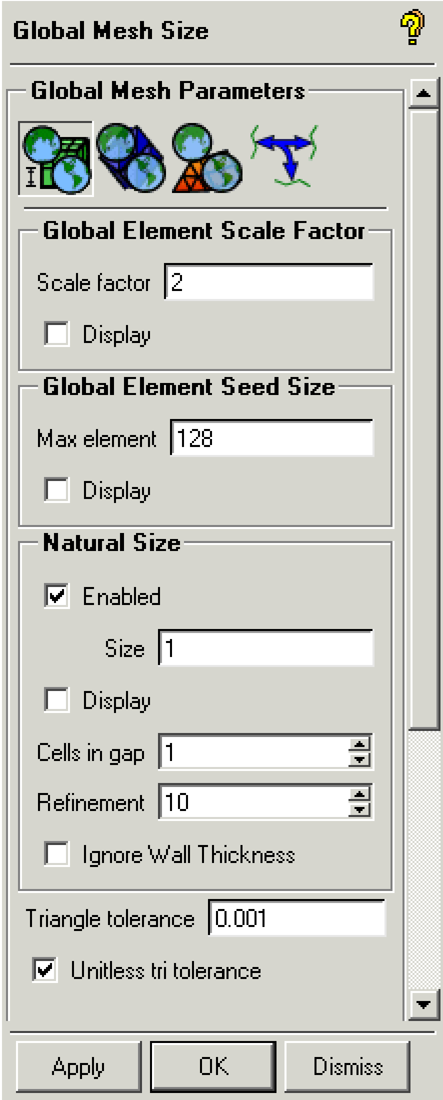
h) Assigning the Mesh Sizes

The User can define meshing parameters in several ways. In this example, the user will define them on the model, on the surfaces, and on the curves.

i) Setting Global mesh size

Select Mesh > Set Global Mesh Size .> General parameters , it will open the Global Mesh Size window, (Figure 3.283), enter a Scale factor of 2.0, a Max Element of 128, Natural size of 1, Natural size > Refinement of 10, and Tri tolerance of 0.001. Leave the other parameters at their default settings. Press Apply and then Dismiss.

Figure 3.283
Edit the Global mesh sizes



Select Mesh > Set Global Mesh Size > Tet Meshing Parameters, it will open the window as shown in Figure 3.284. Turn ON Fast transition and press Apply.

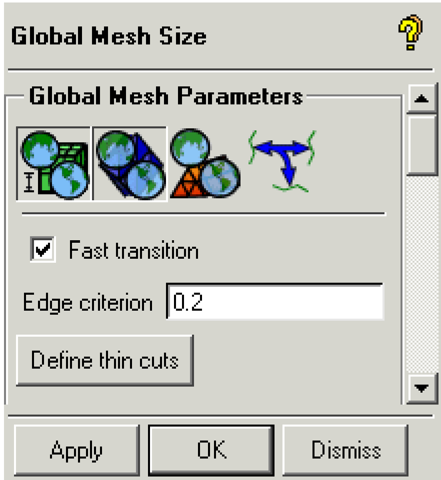


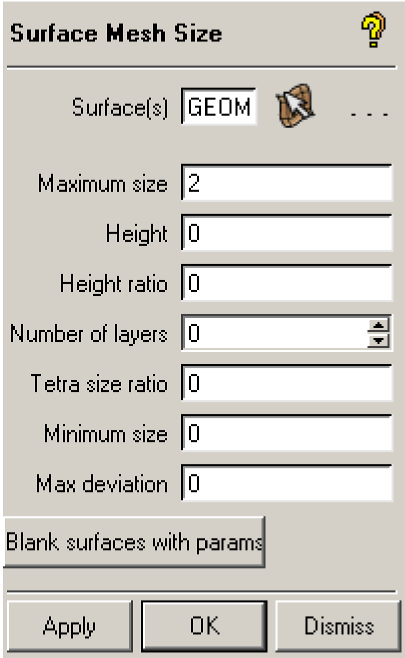
Figure 3.284 Tetra Meshing Parameters Window

j) Setting surface mesh size

Next, Select Mesh > Set Surface Mesh Size to set the meshing parameters on the surfaces of the model.

Select and Press the "a" keyboard key to select all surfaces.

Figure 3.285
Edit the surface mesh sizes



In this Surface Mesh Size window (Figure 3.285), enter a Maximum element size of 2 and press Apply followed by Dismiss.

k) Setting the curve mesh size

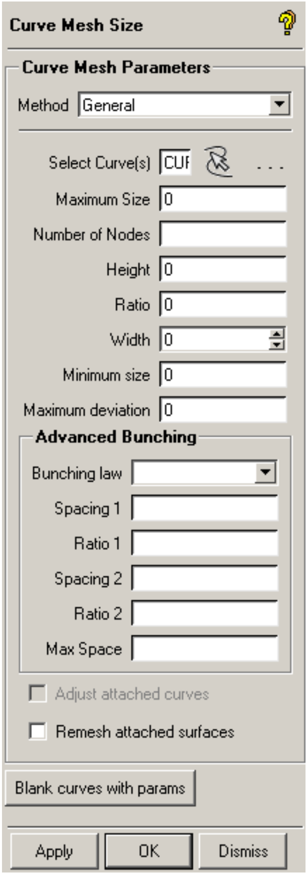
By default, for a new geometry, the mesh size on all curves is zero and therefore need not be set. If, however, the user needs to set mesh size on some or all of the curves the following procedure may be used.

Select Mesh> Set Curve Mesh Size  to set the meshing parameters on the curves of the model.

Select  and use one of the selection methods to pick some or all curves.

In the Curve Mesh Size window (Figure 3.286), enter a value for Max Size parameters. Press Apply followed by Dismiss to close the window.

Figure 3.286
Edit the curve mesh sizes



Next, save this configuration as a Tetin file, be sure that all entities are displayed so that they will be written to the file.

Press File > Save Project to save this data.

l) Generating the Tetrahedral Mesh

Select Mesh > Volume Meshing > From Geometry Press Apply in Mesh with tetrahedral window (Figure 3.287). After the mesh is generated, it will be as seen in Figure 3.288.

Figure 3.287
Mesh with Tetrahedral window

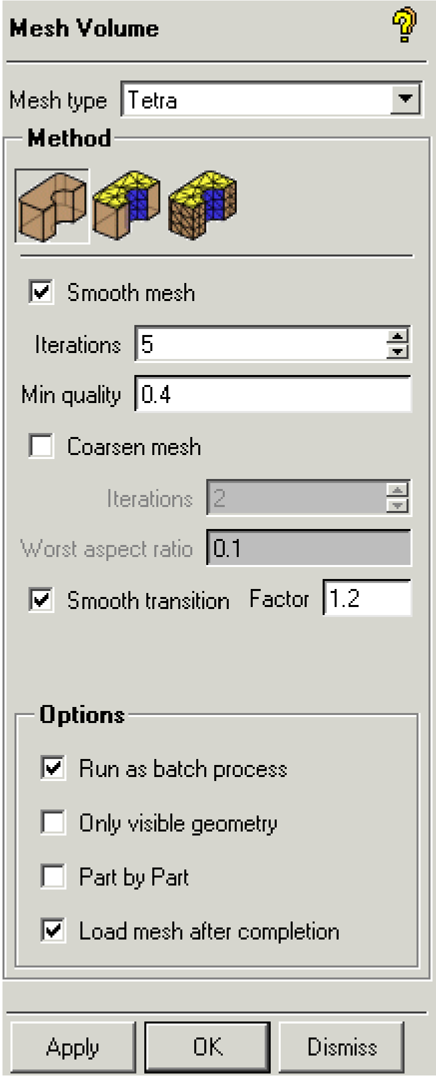
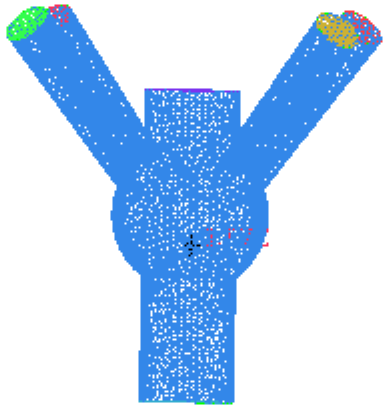


Figure 3.288
The complete mesh



As in the SphereCube example, the user should go through all of the checks for Errors and Possible problems to ensure that the mesh does not contain any flaws that would cause problems for analysis.

m) Saving the Project

Save the mesh by selecting File > Save Project. If a question box pops up asking whether to delete disconnected vertices, respond by saying Yes.

Close the project by selecting File > Close Project.

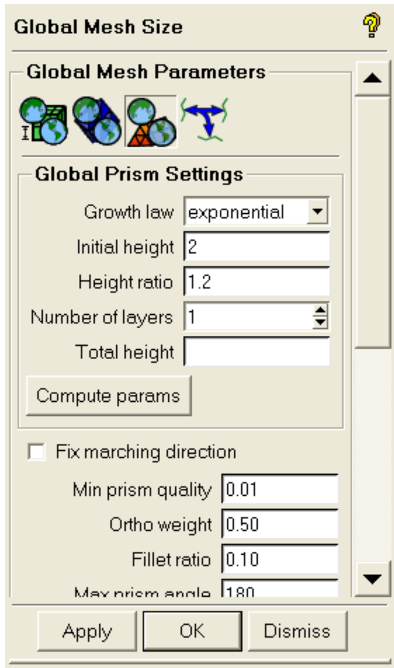
n) Adding Prism Layers

In this example, adding prism layers as a seperate process will be demonstrated. To make sure the prism mesh is computed with highest quality, the user must check the quality of the tetra mesh for smoothness. In general, one of two strategies may be taken. The user can grow the desired number of layers, or grow a single layer and subdivide later. The single layer to can be optimally divided for a desired initia height and growth ratio. For this example we demonstrate the latter method.

select Edit Mesh > Smooth Mesh. Set Up to Quality parameter to 0.4. Press Apply. After three trials, the mesh quality increases to 0.37.

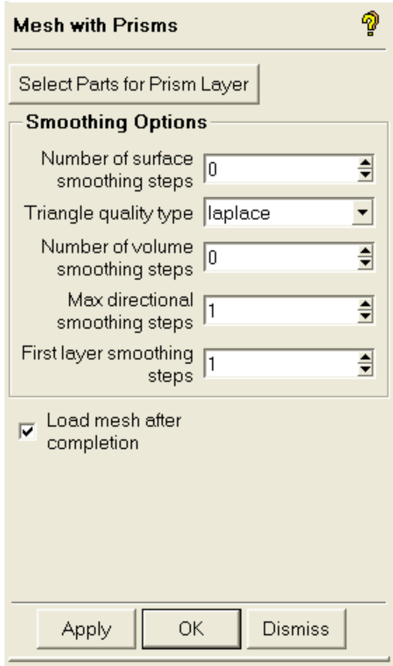
on Global parameters > Global Prism Parameters, Set Initial height to 2 and the Number of leyers to 1 as shown in Figure 3.289.

Figure 3.289 Global Prism Parameters



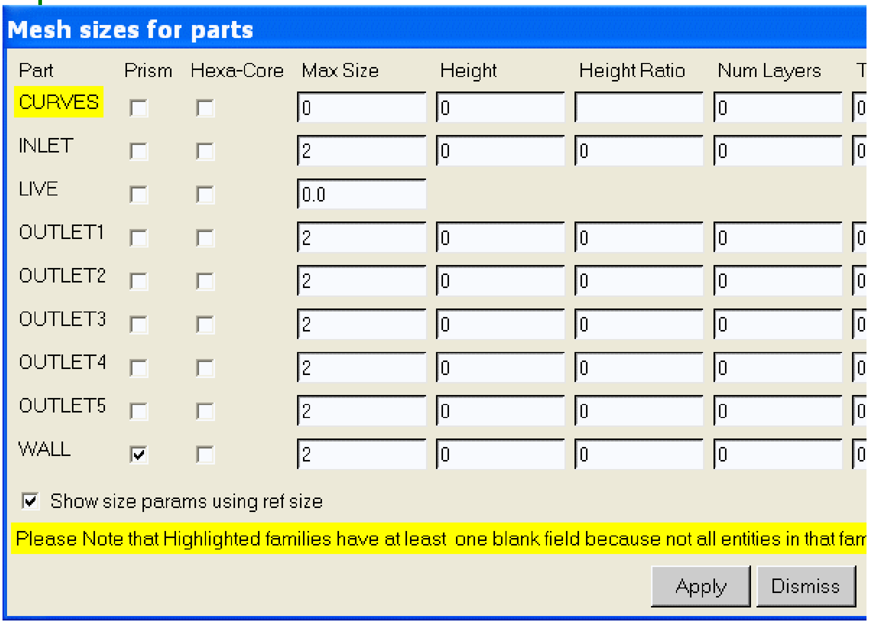
select Mesh > Prism to set parts to grow prism from and specify detailed prism mesh size parameters as shown in Figure 3.290. Click on Select Parts for Prism Layer.

Figure 3.290 Prsim mesh paramaters



In the ensuing table, Figure 3.291, click on walls check box. Note also the individual parameters, such as Initial height, Ratio and Number of layers, that can be defined any part in the list. Here we leave these parameters blank. Press Apply and follow with Dismiss.

Figure 3.291
Prism parts table

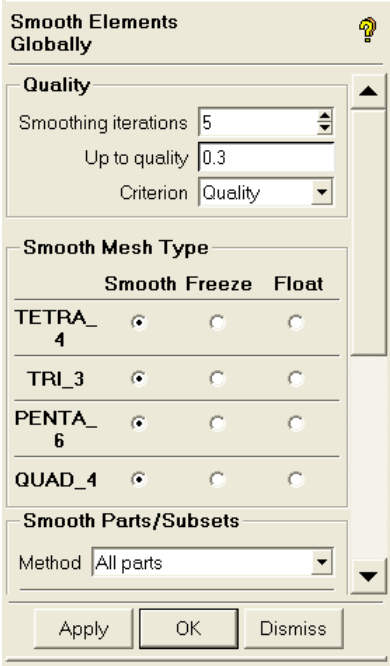


On the prism form (Figure 3.290) press Apply to start prism mesh computation. By default the prism domain will be loaded.

Generally it is a good idea to check the quality of the hybrid mesh (tet + prism). Select Smooth mesh icon on Edit Mesh to invoke the smooth mesh panel. Figure 3.292 shows this panel. Set Up to Quality to 0.3, and press Apply. Repeat the smoothing several times until the quality approaches 0.3.

Save the mesh by selecting File > Save Project. If a question box pops up asking whether to delete disconnected vertices, respond by saying Yes

Figure 3.292 Smooth hybrid mesh



o) Subdividing the prism layer

On model tree > mesh, select Cut Plane and observe the single layer of prism, Figure 3.293.

Select Edit Mesh Tab > Split Mesh > Split Prism, it will open a window as shown in Figure 3.294, press Apply,

The single prism layer breakes into 5 layers, Figure 3.295.

Figure 3.293 Cut plane showing a layer of prism

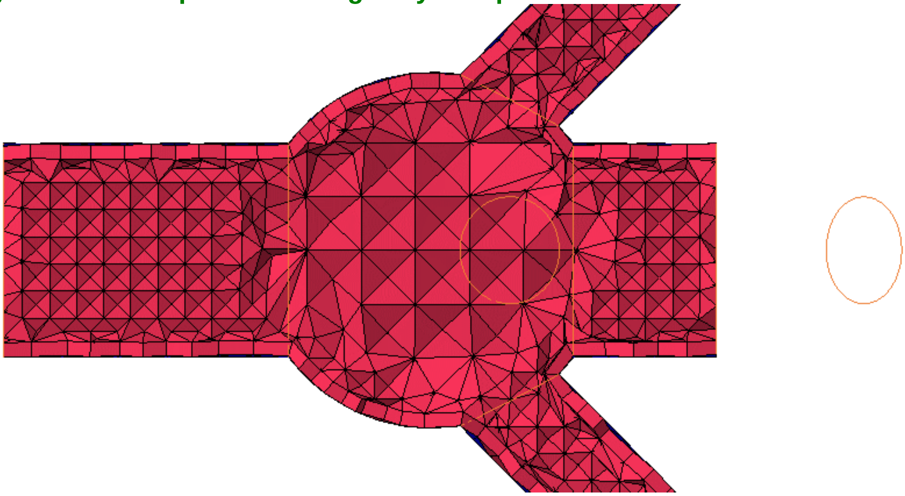


Figure 3.294
Split prism window

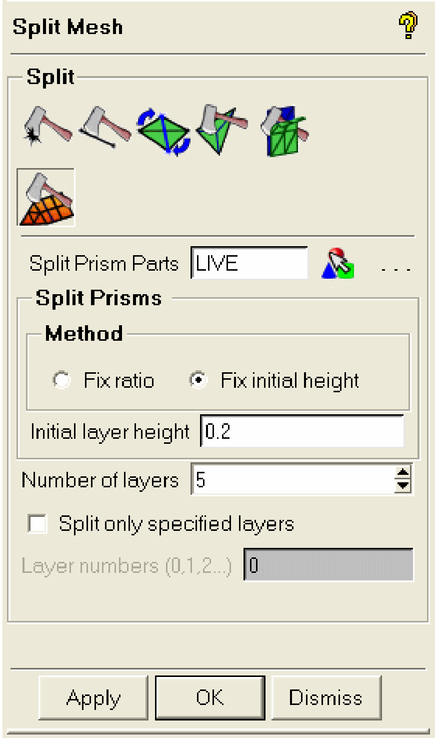
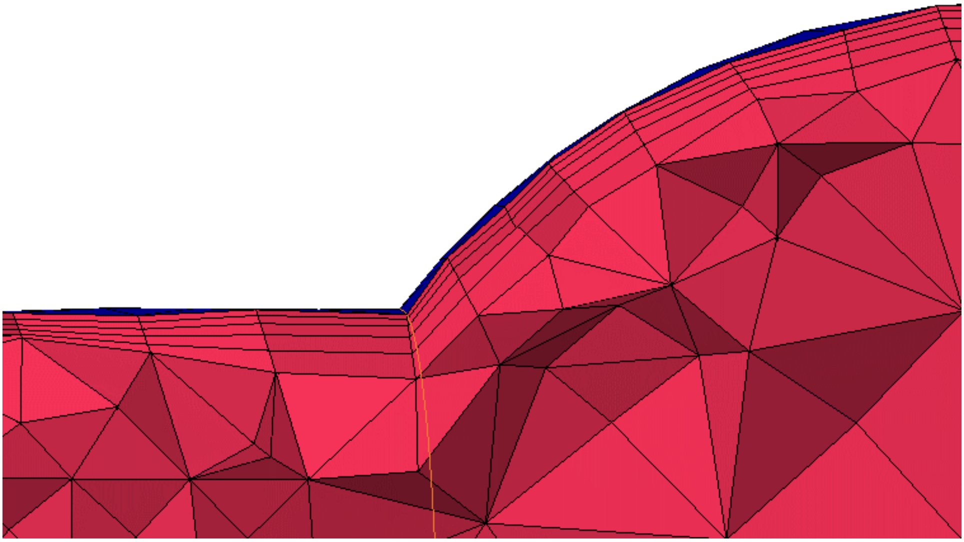


Figure 3.295
A single was subdivided into 5 layers



For a fixed of number of layers (5) and the total thickness, the layers can be redistributed to achieve the optimum initial height.

Select Edit Mesh Tab > Move Nodes > Redistribute prism edg,, Set Initia height to 0.1 and press Apply. The ratios will be adjusted.

p) Saving project

Save the mesh by selecting File > Save Project.

Close the project by selecting File > Close Project.

3.5: Tetra Meshing Appendix

3.5.1: Mesh Editor - Before Creating the Tetra Mesh

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

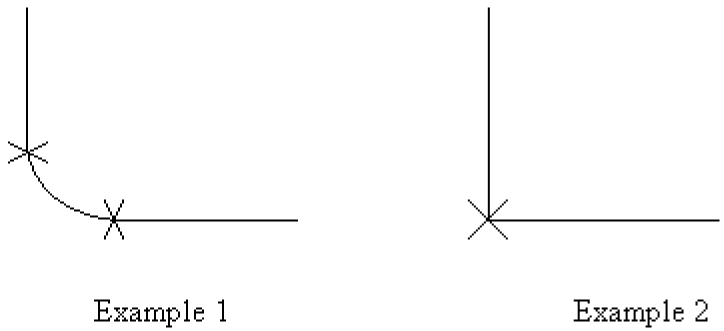
Missing surfaces or holes

To create a mesh, ANSYS ICEMCFD Tetra requires that the model contains a closed volume. If, however, there are any holes (gaps or missing surfaces) in the geometry that are larger than the reference tetras in that particular location, Tetra will be unable to find a closed volume. Thus, if the user notices any holes in the model prior to mesh generation, the surface data should be fixed to eliminate these holes.
Mostly the holes can be found using the **Geometry > Repair Geoemtry > Build Topology** option. It should give you yellow curves for the regions where there are big cracks or missing surfaces.
If holes are not apparent to the user, but they are unsure of the model's integrity, they can still run Tetra. After the run Tetra automatically figures out the holes and prompts the user to close the holes interactively. For further information on the process of interactively closing holes, refer to the description later in this section or see the Mesh Editor on-line Help.

Curves and points on sharp edges

ANSYS ICEMCFD Tetra requires curves at locations where the user is interested in capturing geometric features where two surfaces intersect. Curves in Tetra indicate to the mesher that nodes of the mesh should be aligned along a feature. Refer to the two examples in Figure 3.296.

Figure 3.296 Example-1 illustrates two flat surfaces, with a fillet surface going between the two. In Example 2, the two flat surfaces meet.



In Example 1, the tetra mesh will round along the filleted region. The mesh does not have to conform to the edges of the surfaces. In fact if the mesh did have to conform to the edges of the surface, the mesh could be over-constrained, since if the mesh size is large, the mesh might jump from one edge to the other edge and neglect to model the fillet region. Thus, the curves at the top and bottom of the filleted surface shouldn't be included in the model.

It is important to include the curve in Example 2, since it will force Tetra to locate nodes along this corner.

Points are also necessary to capture the corners of curves. If two curves intersect at a sharp angle and the user wishes to capture this feature, a point must be created in the corner of intersection.

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 **Surfaces** > **Tetra sizes** from the Display Tree

widget... 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, the user may rotate the model and visually confirm that the tetra sizes are appropriate. If a curve or surface does not have an icon plotted on it, the icon may simply be too large or too small to see. In this case, the user should modify the mesh parameters so that the icons are visible in a normal display.
The user should also make sure that a reference element size has been defined. To modify the mesh size for all entities, adjust the Scale factor, which is found through **Mesh** > **Set Global mesh size**. Note that if 0 is assigned as the scale factor, Tetra will not run.
To make sure that sizes are defined for all of the surfaces, activate all Parts and Surfaces in the Display Tree widget. Then, select **Mesh** > **Set surface mesh size**. Press the right mouse button to Dismiss the selection mode and in the params window, check the box for Blank surfaces already done. This will blank all surfaces in the model. Any surfaces that remain visible need to have proper tetra sizes defined. The same check should be done for Curves.

Material point(s)

From the Display Tree widget, Make bodies visible and right click on **Bodies** > **By name**. Dynamically rotate the model to confirm that each closed volume has a material assigned to it. If a closed volume does not have a material assignment, provide one for the region.
The user need not define material point ORFN for every dead region as Tetra automatically finds the dead regions and throws them out. When periodicity is defined; however it is preferable to assign material point to speed up the meshing process.

Converging or thin regions between objects

Examine the regions between two surfaces or two curves that are very close together or converging and check whether the tetra sizes (refer to the tetra icons) are small enough so that at least 1 or 2 tets would fit through the thickness. If the tetra sizes are not small enough, the user should select to define thin cuts between the two surfaces. To define a thin cut, the two surfaces have to be in different parts; if the surfaces are converging, the curve at the intersection of the surfaces will need to be in a third, different part.

If the tetra sizes are larger or approximately the same size as the gap between the surfaces or curves, the surface mesh could jump the gap, thus creating non-manifold vertices. These non-manifold vertices would be created during the meshing process. Tetra automatically attempts to close all holes in a model. Since the gap may be confused as a hole, the user 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 won't close the gap when the meshing process is performed. A space that is greater than 2 or 3 elements in thickness is usually considered as a hole.

Density control

3.5.2: Tetra

The Three modes for Tetra are: **From geometry**, **From geometry and surface mesh** and **From surface mesh**.

From Geometry

In this mode, the user can accept the default parameters by selecting Apply from the Mesh with Tetrahedral window. Additionally; the user may also modify any of the parameters before selecting Apply.

There are some options to mesh:

Smooth mesh

This will attempt to improve the quality of the Tetra mesh to Min. quality in a particular number of iterations. If you give iterations as 0, the smoother will not run on the tetra mesh.

Coarsen mesh

This will attempt to coarsen the mesh for the elements whose aspect ratio is below the specified value.

Smooth transition

This option is used to have the smooth transition of tetra height.

Factor

This is the ratio of the height of the tetra elements in (inner) layer to that of the next outer layer

Additional Options:

Run as batch process

This option is used to run the stand-alone tetra mesher. In order to run tetra in batch, we need to save the problem before we start the tetra mesher.

Only visible geometry

This will mesh the only visible geometry on the screen.

Part by part

This option will do meshing part by part

Load mesh after completion

Loading the tetra mesh after meshing will automatically invoke the domain file (named tetra_mesh.uns by default), when the Tetra batching process is complete. If this option is not selected, then tetra will not load the mesh in the screen.

From Geometry and From surface mesh.

An Existing surface mesh file should be provided in order to select the triangular mesh for different parts. The Part window is displayed so that you can select the parts you want for which the surface mesh should be used.

From surface mesh

This option uses only surface mesh to create tetra mesh

Once the mesh is generated, the Mesh Editor automatically tries to figure out if there are any holes in the model. If there are, it displays a message like "Material point ORFN can reach material point [volume part name, e.g. LIVE]" in the messages window. You will be prompted also with a dialog box saying "Your geometry has a hole, do you want to repair it?" If there is leakage in the geometry, a jagged line will appear in the display. All elements attached to the hole would also be displayed. Additionally, a window would appear to help the user fix the problem. Go ahead and accept the defaults by pressing Accept. Select the single edges in the rectangular box when prompted and that should fix the problem. If there were additional holes, it would keep the user in a loop until the problem is fixed.

3.5.3: Editing the Tetra Mesh

The two main criteria in validating a Tetra mesh are Check mesh and Smooth mesh globally, both of which are found under the Edit mesh menu.

Check Mesh

From the **Edit mesh> Check mesh**, Then, Press Apply**.**
The user can Check/fix each of the problems at this time, or can opt to create subsets for each of them so that they can be fixed later. Using subset manipulation and mesh editing techniques diagnose the problem and resolve it by merging nodes, splitting edges, swapping edges, delete/create elements, etc.
For subset manipulation, Right click on **Subset > Modify** in the mesh under Display Tree widget. Ordinarily, the user will select to
Add layer from the **Modify subset** window.
Keep in mind that after editing the mesh diagnostics should be re-checked to verify that no mistakes were made.
There are several Errors as well as possible problems checks. The descriptions of these are as follows:

Errors > Duplicate elements

This check locates elements that share all of their nodes with other elements of the same type. These elements should be deleted.
Please note that deleting elements during the automatic fix procedure will remove one of the two duplicate elements, thus eliminating this error without creating a hole in the geometry.

Errors > Uncovered faces

This check will locate any face on a volume element that neither touches a surface boundary nor touches another internal face. This error usually

indicates that there is a hole in the volume domain. It is unlikely that this error would occur in the initial model -- usually, it results during manual editing when the user happens to delete tetra or tri elements.
The automatic Fix Feature will cover these uncovered faces with triangles. This may or may not be the proper solution. A better method may be for the user to first Select the flawed elements and then decide if the uncovered faces are the result of missing surface mesh or the result of a hole. If it is due to missing surface mesh, the Fix option will eliminate the problem (re-run the check and select Fix). If the error points out a hole in the model, the user can attempt to correct the grid by creating tetras or merging nodes manually.

Errors > Missing internal faces

This check will find pairs of volume elements that belong to different families, but do not have a surface element between the shared face. This error, like uncovered faces, should not occur in the original model and would most likely result from mistakes made during the manual editing process. The tetra cutter will detect this problem as leakage. The automatic Fix Feature will create a surface mesh in between these cells.

Errors > Periodic problems

This check will compare the families that were selected to have periodic nodes and would report an error if they mismatch or if there is a missing connection. This should be repaired by hand using **Edit nodes > Periodic > Make periodic** or **Remove periodic**. The user should not get this error unless they have edited on the mesh.

Errors > Volume orientations

This check will find elements where the order of the nodes does not define a right-handed element. The automatic Fix feature will re-order the mis-oriented elements' nodes to eliminate this error.

Errors > Surface orientations

This checks the direction of the face normal to the elements. This check will indicate any location where tetras share the same volume, but not the same nodes (duplicate elements are elements that occupy both the same

volume and the same nodes). The error that indicates a major problem in the connectivity in the model, need to be fixed manually. Usually this can be done by clearing a subset and adding specific elements to it in the location where the orientation problem was found.
The orientation errors will be displayed in the messages window with the location. The user can then select View > Add marker and enter in the coordinates reported in the message window. This will place a marker, with the name assigned by the user, at the assigned location. Then, select Modify from the subset menu. Proceed to select Add > Specific from the Modify subset window and then enter the coordinates in the box and select near position. The user then has to fix the orientation errors by editing the volume elements so that the criss-crossing of elements is eliminated. This is typically done by merging nodes and/or splitting edges.

Note that Diagnostics > which elements doesn’t pertain to this check.

Possible problems > Multiple edges

This check will find elements with an edge that shares more than two elements. Legitimate multiple edges would be found at a "T"-shaped junction, where more than two geometric surfaces meet.

Possible problems > Triangle boxes

This check locates groups of four triangles that form a tetrahedron, with no actual volume element inside. This undesirable characteristic is best fixed by choosing Select for this region and merging the two nodes that would collapse the unwanted triangle box.

Possible problems > Single edges

This check will locate surface elements that have an edge that isn’t shared with any other surface element. This would represent a hanging edge and the element would be considered an internal baffle. These may or may not be legitimate. Legitimate single edges would occur where the geometry has a zero thickness baffle with a free or hanging edge or in a 2D model at the perimeter of the domain.

If the single edges form a closed loop -- a hole in the surface mesh -- the user can select Fix when prompted by the corresponding menu. A new set of triangles will then be created to eliminate the hole.

Possible problems > 2-Single edges

This check will locate surface elements that have two edges as single edges. Mostly these elements should be thrown out.

Possible problems > Single-Multiple edges

This check will locate surface elements that have an edge which is single and another which is multiple.

Possible problems > Stand-alone surface mesh

This check locates surface elements that do not share a face with a volume element. These can generally be deleted, in the case of a volume mesh.

Possible problems > Delaunay-violation

This check finds the elements if they are violating the Delaunay rule. Delaunay rule says that a circumscribed circle around a surface triangle should not have any additional node in the circle. Often this can be removed by doing the diagonal swapping at that location.

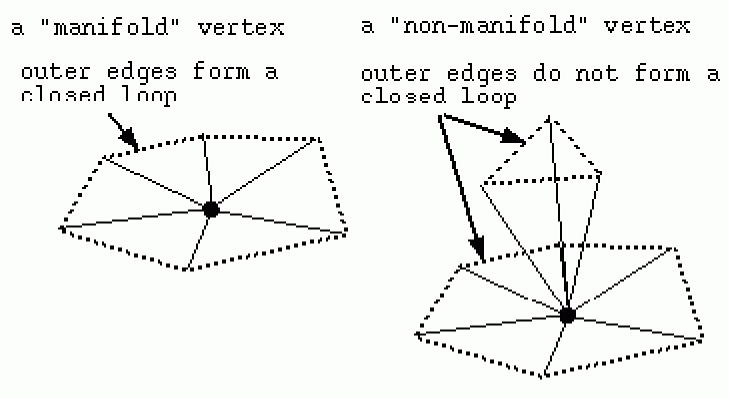
Possible problems > Overlapping triangles

It depects triangles which lie on top of each other though they may not be sharing any nodes or edges. These can be taken care of by moving/merging the nodes or splitting/swapping the edges.

Possible problems > Non-manifold vertices

This check will find vertices whose adjacent elements' outer edges don't form a closed loop. This problem usually indicates the existence of elements that jump from one surface to another, forming a "tent"-like structure, as shown in Figure 3.297.

Figure 3.297 In a "manifold" vertex, the outer edges form a closed loop, thus posing no problems to the mesh quality (Left). With a " non-manifold" vertex, the outer edges do not form a closed loop, acting as a barrier in the free domain (Right)



The automatic Fix option for possible problems should only be employed if the non-manifold volumetric mesh is within the surface mesh that is disconnected. If there is volume mesh on both sides of either surface mesh, do not use the automatic Fix Option. Instead, choose Select and split one of the normal edges.

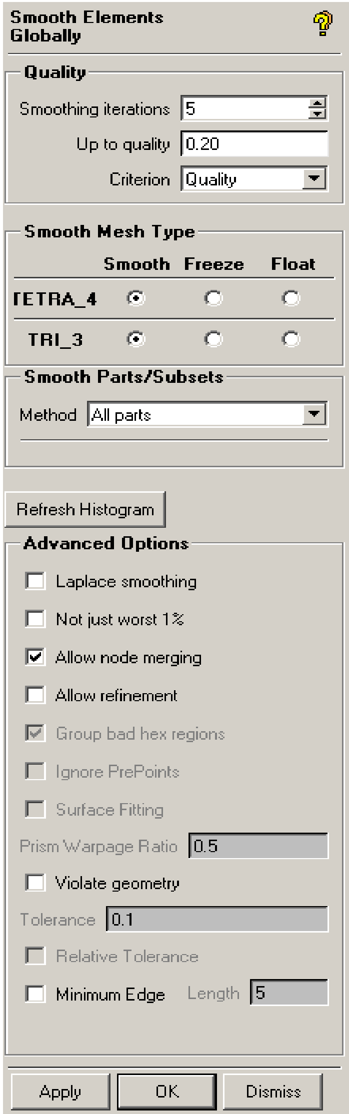
Possible problems > Un-connected vertices

This check finds vertices that are not connected to any elements. These can generally be deleted.

Smoothing

After eliminating errors/possible problems from a tetra grid, the user needs to smooth the grid to improve the quality. To do this, select **Edit mesh > Smooth mesh Globally**

Figure 3.298 Smooth mesh globally window



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

Up to quality: As mentioned previously, the Min value represents the worst quality of elements, while the Max value represents the highest quality elements. Usually, the Min is set at 0.0 and the Max is set at 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, the user should find 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.

Criterion: User can select any criterion to display from pull down menu.

Smooth: If the Smooth option is toggled on for a particular element type, then this element will be smoothed in order to produce a higher grid quality. Element types that have the Smooth option selected will have their qualities appear in the associated histogram.

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

Float: If the Float option is selected for an element type, the nodes of the element type are capable of moving freely, allowing nodes that are common with another type of element to be smoothed. The nodes of this type of element, however, are not affected during the smoothing process and so the quality of these elements is not displayed in the histogram.

Advanced options:

Only visible subsets

This smooth only visible subsets.

Active parts only

This will smooth only active parts from the screen.
Laplace smoothing: This option will solve the Laplace equation, which will generally yield a more uniformly spaced mesh.

All elements

This will smooth all the visible and invisible elements**.**

Violate geometry: Selecting this option allows the smoothing operation to yield a higher quality mesh by violating the constraints of the geometry. When this option is activated; however, the smoothing operation has a greater degree of freedom. The nodes can be moved off of the geometry to obtain better mesh quality, as long as it remains within the absolute distance that is specified by the user.

Tolerance: Allowance to so that smoother can violate geometry by mentioned distance.

Min edge: Minimum edge that is allowed to occur after smoothing.

Length: Value of the minimum edge.

If the user has highlighted bars from the histogram and selected to Show them on the model, choosing Select will modify the display so that only those elements are visible. These elements are also placed into a Subset. The visibility of this subset is controlled by toggling Display subset from the Display window. The contents of the subset may also be altered with the Modify option.

Add select: This option allows the user to add elements to an already established subset.

Quality metric

Changing this option allows the user to modify what the histogram displays.

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

Quality (4.3 version): This will calculate the quality as per the 4.3 version.

Aspect ratio: For HEXA_8 (hexahedral) and QUAD_4 (quadrilateral) elements, the Aspect ratio is defined as the ratio of the distances between diagonally opposite vertices (shorter diagonal/longer diagonal). For TETRA_4 (tetrahedral) elements, MED calculates the ratio between the radii of an inscribed sphere to a circumscribed sphere for each element. For TRI_3 (triangular) elements, this operation is done using circles. An Aspect ratio of 1 is a perfect cell and an Aspect ratio of 0 indicates that the element has zero volume.

Determinant: This histogram is based on the determinant of the Jacobian matrix. The Jacobian value is based on the difference between the internal angles of the opposing edges within the element.

Min angle: The Min angle option yields a histogram based upon the minimum internal angle of the element edges.

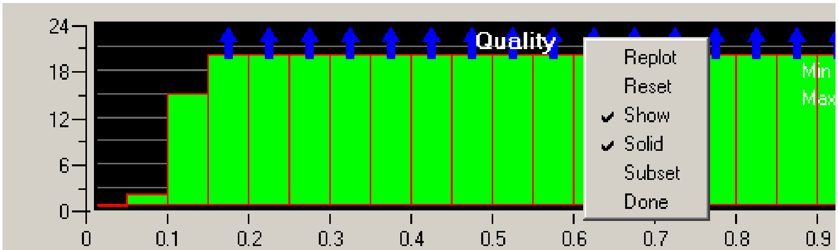
Max othogls

Max warp: This histogram is based on the warpage of the quad faces of the prism. This is based on the worst angle between two triangles that make up the quad face.

Skew: This histogram is based upon calculations of the maximum skewness of a hexahedral or quadrilateral element. The skewness is defined differently for volume and surface elements. For a volume element, it is obtained by taking all pairs of adjacent faces and computing the normals. The maximum value thus obtained, is normalized so that 0 corresponds to perpendicular faces and 1 corresponds to parallel faces.

Custom quality: One can define one's own quality definition by going to Diagnostics > Quality metrics. Select the Diagnostic: as custom quality and go for Define custom quality. One can change the values there to suit his/her needs.

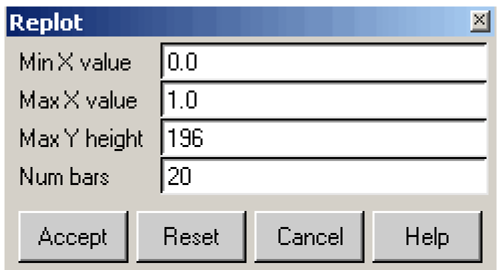
Figure 3.299 Histogram window



Histogram: The element Quality will be displayed within this histogram, where 0 represents the worst aspect ratio and 1 represents the best aspect ratio. The user may modify the display of the histogram by adjusting the values of Min X value, Max X Value, Max Y Height andNum Bars by pressing the Replot.

Replot: If any modifications have been made to any of the parameters within the Smooth mesh globally window or to the model, selecting Replot will display an updated histogram.

Figure 3.300 Replot window



Min X value: This minimum value represents the worst quality elements on the histogram's x-axis.

Max X value: This maximum value represents the highest quality that elements can achieve.

Max Y Height: The user 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 and the Max. The default Bars have widths of 0.05. Increasing the amount of displayed bars, however, will decrease this width.

Reset: Selecting this option will return all of the values back to the original parameters that were present when the Smooth mesh globally window was first invoked.

Show: The user may press the left mouse button on any of the bars in the histogram and the color will change from green to pink. Selecting Show will display the elements that fall within the selected range on the model in the main viewing window.

Solid: This toggle option will display the elements as solid tetras, rather than as the default grid representation. The user will have to select Show, as well, to activate this option.

Subset: This will create the subset of the selected elements.Selected elements will be placed in a subset.

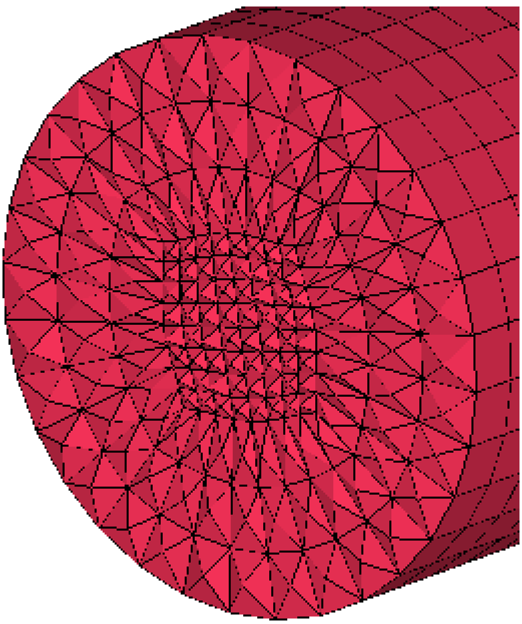
Done: If this button is pressed then it will close the hostogram window.

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 cells and edit this surface mesh to improve the quality.

3.6: Advanced Meshing Tutorials

ANSYS ICEMCFD is tuned to help users create advance operations. They include at times:
Complex operations of topology transformations inside HEXA
Reducing the number of blocks for Multiblock mesh output
Merging HEXA and TETRA meshes in all possible ways to get a conformal hybrid mesh
Getting a tetra mesh out of a case where the surface mesh data is available additionally for some of the regions
Getting Quad surface mesh on complex objects
Getting rid of leakages in Tetra mesh using geometry repair tools
Setting boundary conditions and writing output for solvers

Figure 3.301
Hybrid Mesh

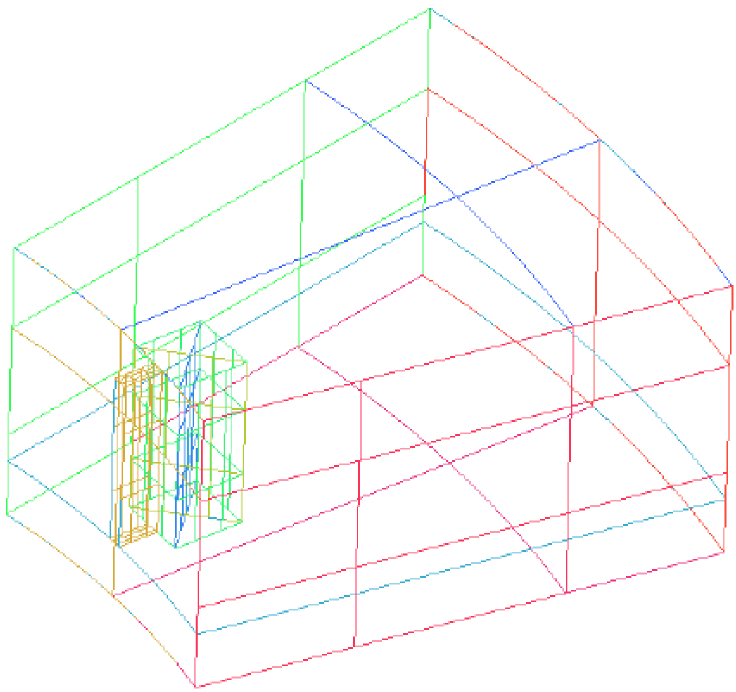


The tutorials in this section are related to these advanced steps only and will try to take the user to deal with real world applications.

3.6.1: Hexa Mesh in a Grid Fin

Overview

In this Tutorial example, the user will generate a hexa mesh for a Grid Fin. Since the mesh is very similar in the Z-direction, a 2D blocking can first be made, which is easier than a 3D blocking. Then the 2D blocking can be extruded into a 3D blocking.



a) Summary of Steps

The Blocking Strategy

Starting the Project in ICEM CFD

Generating the 2-D Blocking

Creating the O-grid

Resolving other grids

Creating remaining Blocks

Step involved achieving Complete Blocking

Placing all nodes to one plane

3-D Blocking

Resolving zero thickness walls

Defining Periodicity

Generating the Mesh

Checking the Mesh Quality

Multiblock mesh

Saving the files

b) The Blocking Strategy

Since the geometry is mostly an extruded model in the Z-direction, the blocking can also easily be done by extruding a 2D blocking in the Z-direction to create a 3D blocking. Generally, the blocking process starts by capturing the outer geometry, and then proceeds to capture the minor parts of geometry by means of splitting the blocks. This is the "top-down" approach. But in this example, the strategy is exactly the opposite of the standard strategy. First, the minor geometry will be captured. Then "Transform/Copy/Merge Blocks" will be employed to get a wider repetitive portion and then "create block" will be used to capture the rest. This is known as the "bottom-up" approach.

c) Starting the Project in ANSYS ICEMCFD

Start ANSYS ICEMCFD. Go to File > Change working directory, and set the current working directory to $ICEM_ACN/../docu/CFDHelp/CFD_Tutorial_Files>Gridfin. Go to File > Geometry > Open Geometry, and open geometry.tin.

For this tutorial, the part grouping has already been pre-defined. Thus, the user can immediately proceed to blocking.

Curves should be ON in the Display Tree.

d) Generating the 2-D Blocking

Inside the mesh tab, press Blocking > Create Block  > Initialize Block .

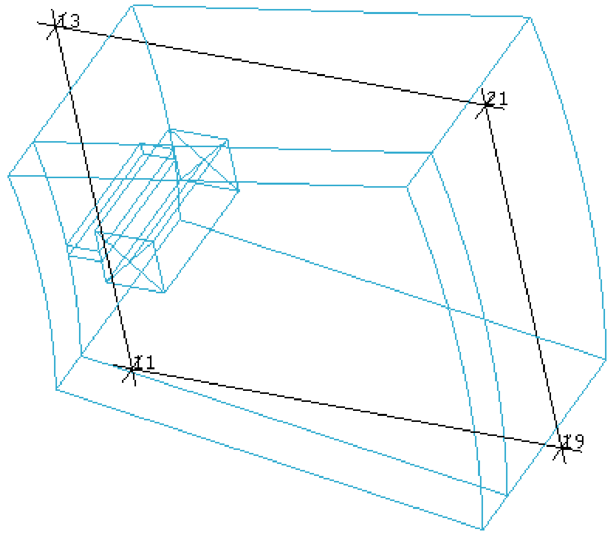
Enter LIVE for the Part.

Next to the Type, choose 2D Planar from the pulldown.

Press Apply. The 2D blocking will be created in the XY plane. This is the orientation that the 2D Planar blocking is meant to work with. If the 2D part of your geometry is not parallel to the XY plane, it is recommended to orient the geometry in the XY plane, or some blocking operations may be difficult.

The Initial block should look like Figure 3.302.

Figure 3.302 Initial 2D Blocking

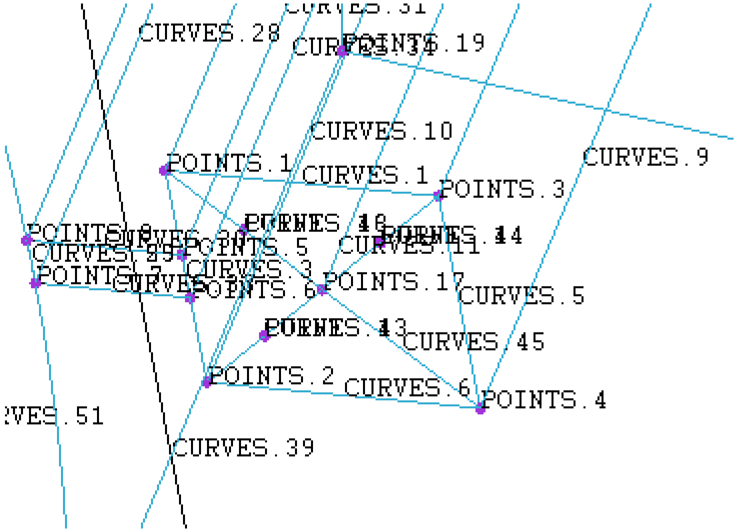


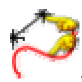
Turn on the Vertices and their numbers from the Display Tree by right mouse clicking on Blocking > Vertices > Numbers. The vertex numbers are shown in Figure 3.302.

Right click in the Display Tree to turn on Geometry > Curves > Show Curve Names.

Zoom in toward the bottom of the inner blocked-shaped geometry, and it should look like Figure 3.303.

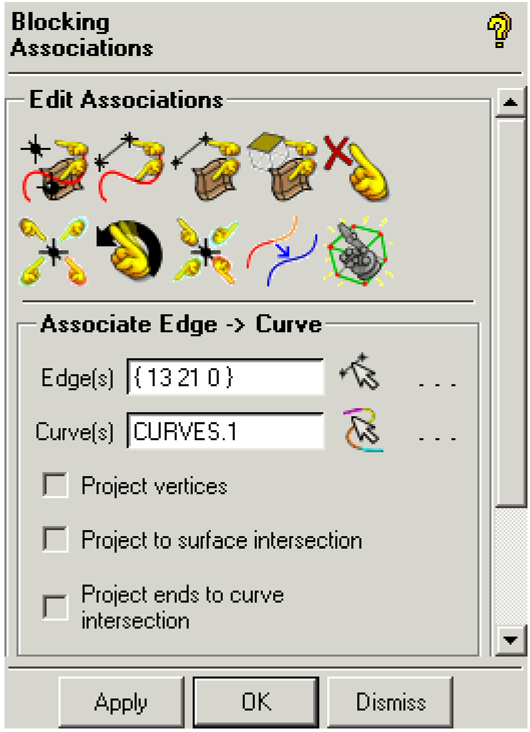
Figure 3.303 Bottom of the geometry



Associate  > Associate Edge to Curve .Select the edge 13-21 with the left mouse button and press middle mouse button to accept the selection. Select the curve, CURVES.1 with the left mouse button and press middle mouse button to accept the selection.

Project Vertices should be switched OFF as shown in Figure 3.304.

Figure 3.304
Associate Edge to Curve Window

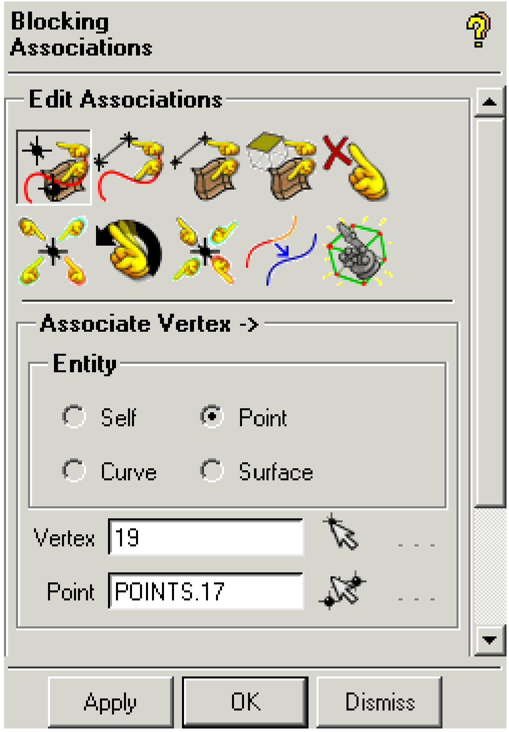


Press Apply. Similarly, associate the edge 11-13 to the curve, CURVES.3.

Note: The user can toggle Off and ON the **Curves** and **Points** to better see what needs to be selected.

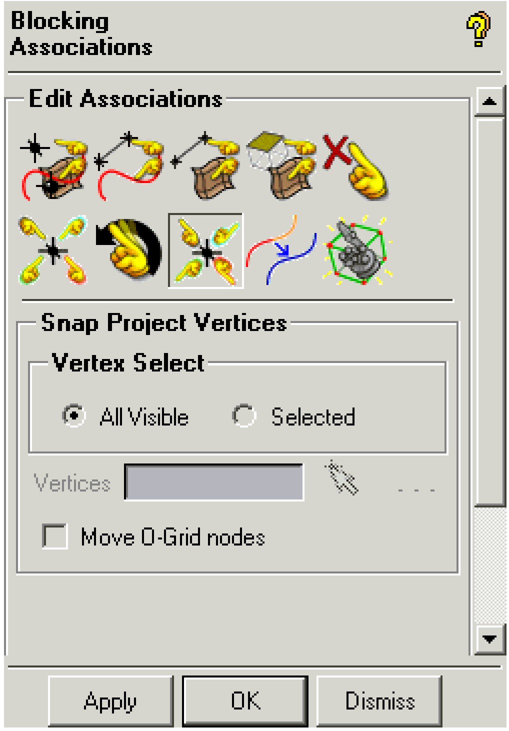
Select Associate  > Associate Vertex.  The Entity to associate to should be set as Point. Select the vertex 19. Then select the point, POINTS.17 as shown in Figure 3.305. Press Apply to associate the vertex to the point.

Figure 3.305
Associate Vertex to Point



Select Associate  > Snap Project Vertices  Toggle on 'All visible' as shown in Figure 3.306. Press Apply.

Figure 3.306 Snap Project Vertices Window



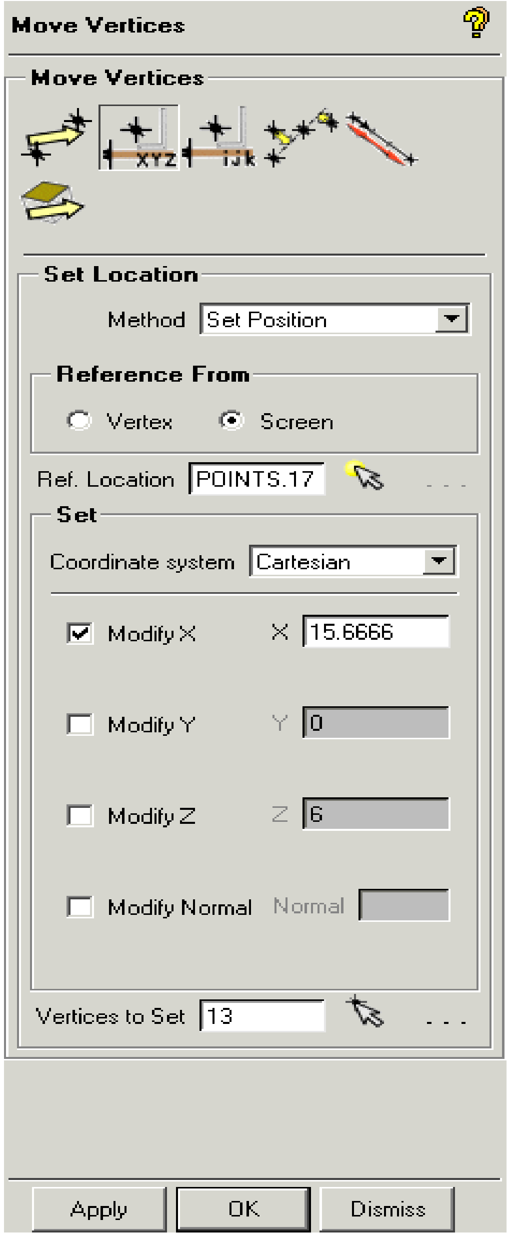
Switch OFF Curves.

Press Move Vertex  > Set Location.  The Set Location window will appear as shown in Figure 3.307. The Reference From should be set to Vertex. Toggle on Modify X. Select vertex 19 for the Ref. Vertex. Select Vertex 13 for the Vertices to Set. Press Apply. The final image is shown in Figure 3.308.

Press Associate > Associate Vertex. The Entity type should be set to Point. For the Vertex, select vertex 11. Turn on Points. For the Point, select POINTS.1. Press Apply.

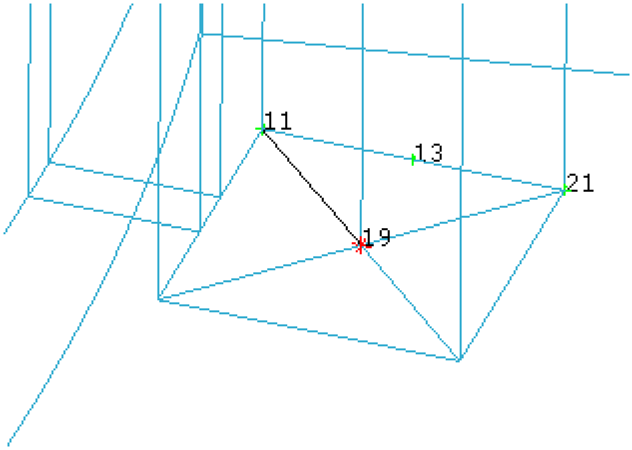
Associate  > Associate Edge to Curve  .Turn OFF Points. Select the edge 11-13. Turn ON Curves, and select the curve, CURVES.1. Press Apply.

Figure 3.307
Set Position of Vertex 13



Note: The user should switch off Curves>Show Curves Names and Points > Show Points Name for most of the time to reduce clutter. They should be turned on only when it's required and then should be turned off again. For the rest of the tutorial, it is assumed that the user would do that to find the location of the Curves or Points.

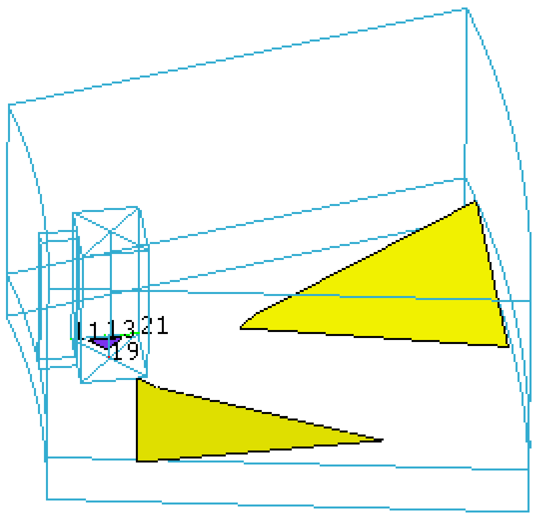
Figure 3.308
Initial Blocking after vertex placement



e) Creating the O-grid

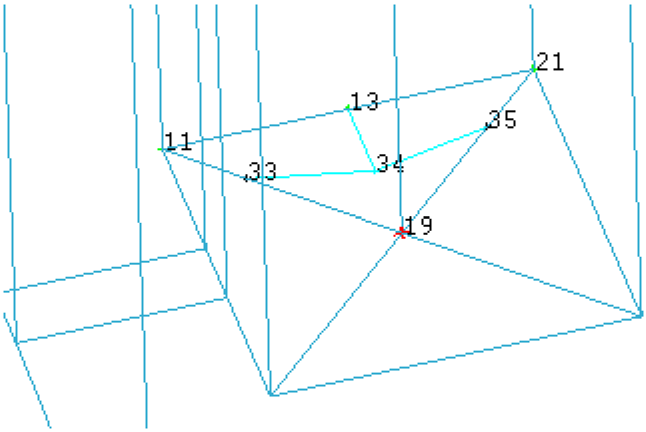
Select Blocking > Split Block  > O Grid Block . Select the Face and then select the Edges 11-19 and 19-21as shown in Figure 3.309 and Press Apply.

Figure 3.309 Selection of edges and Faces for the O-Grid



The Blocking after O-grid creation is shown in Figure 3.310.

Figure 3.310 Blocking after O-grid creation

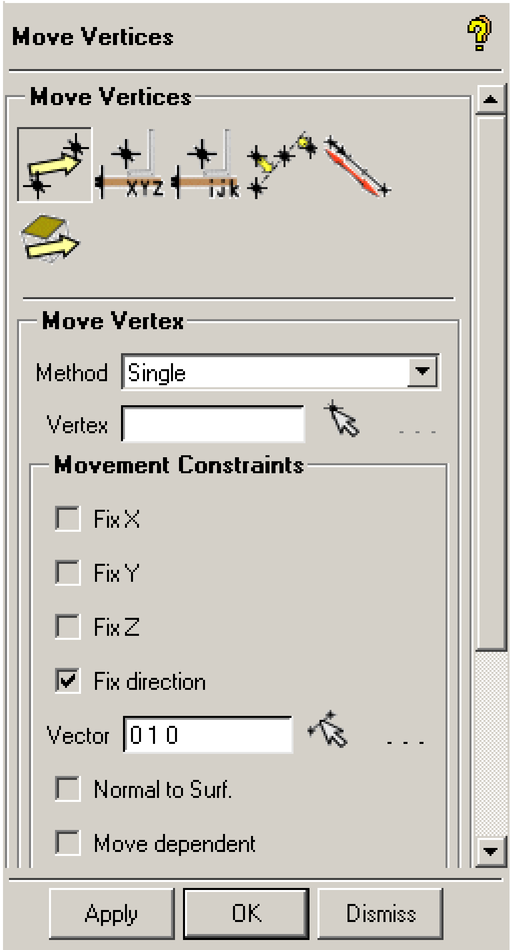


Switch on Points > Show Points Name, select, Association > Associate Vertex to Point project vertex 33 to POINTS.18 and vertex 35 to POINTS.14.

Set the Screen position to View > Front and then by using Blocking > Move Vertex > Move Vertex

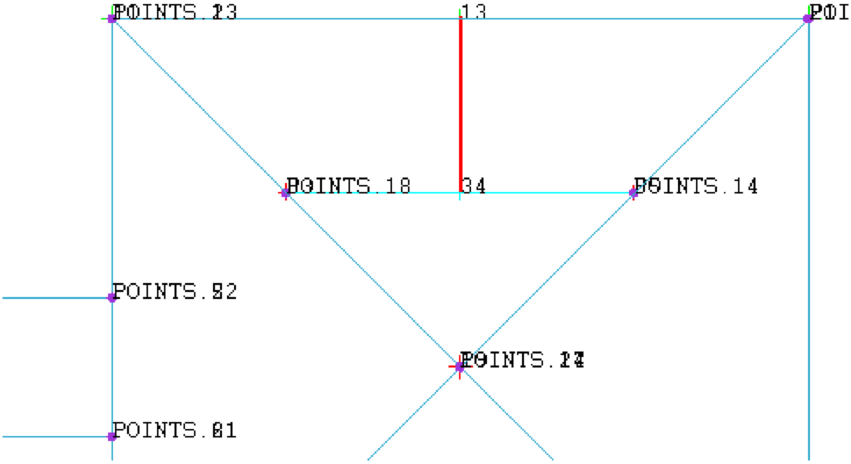
Enable Fix Direction as shown in Figure 3.311.

Figure 3.311
Fix Direction Window



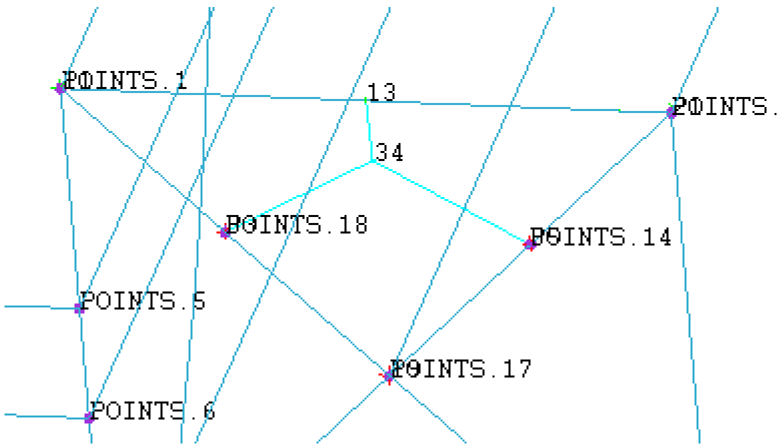
Select the Direction highlighted as shown in Figure 3.312.

Figure 3.312 Move Vertex Fix Direction Option



Place the vertex 34 closers to vertex 13 as shown in Figure 3.313 so that all the Blocks are of Good Quality.

Figure 3.313 Blocking after vertices placements

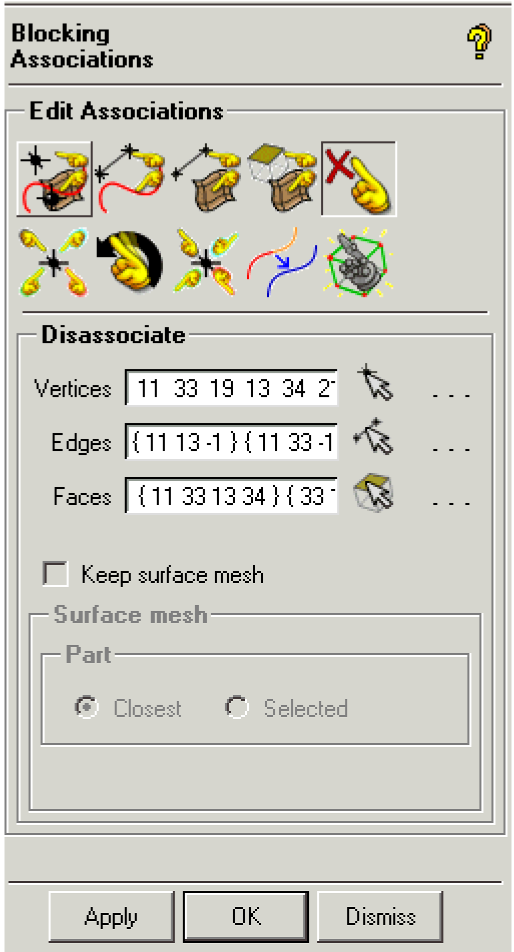


f) Resolving Other Grids

The user will do Copy/Rotate to resolve other Triangular portion of the grid. However, since it copies all the associations too, it's better to first remove all the associations.

Blocking > Association > Disassociate from Geometry . A Dissaccosiate window appears as shown in Figure 3.314.

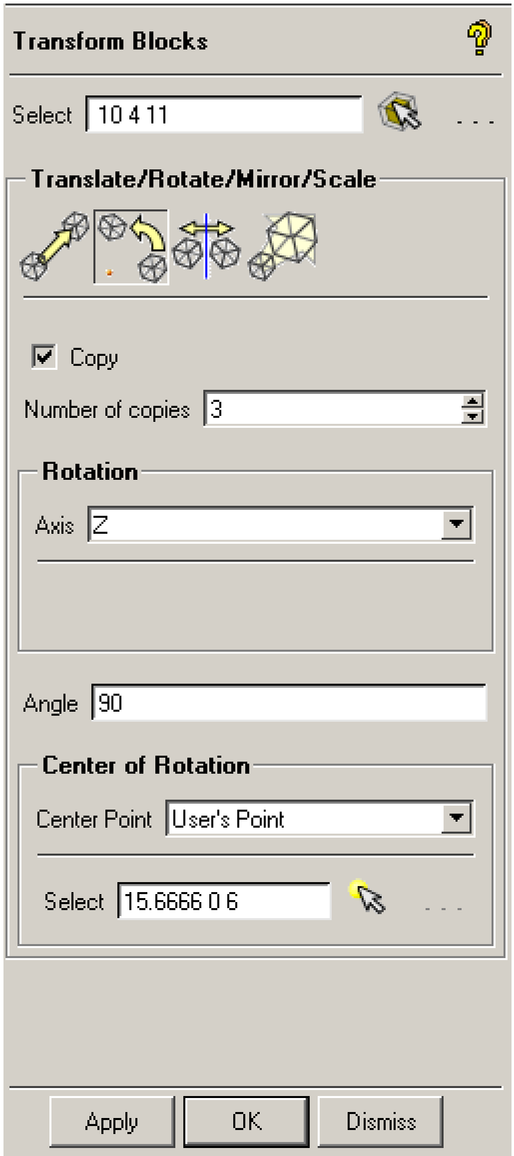
Figure 3.314
Disassociation Window



Select all the Edges, Faces and Vertex and Press Apply.

Go To Blocking > Transform Block  > Rotate Block . A new window will open as shown in Figure 3.315.

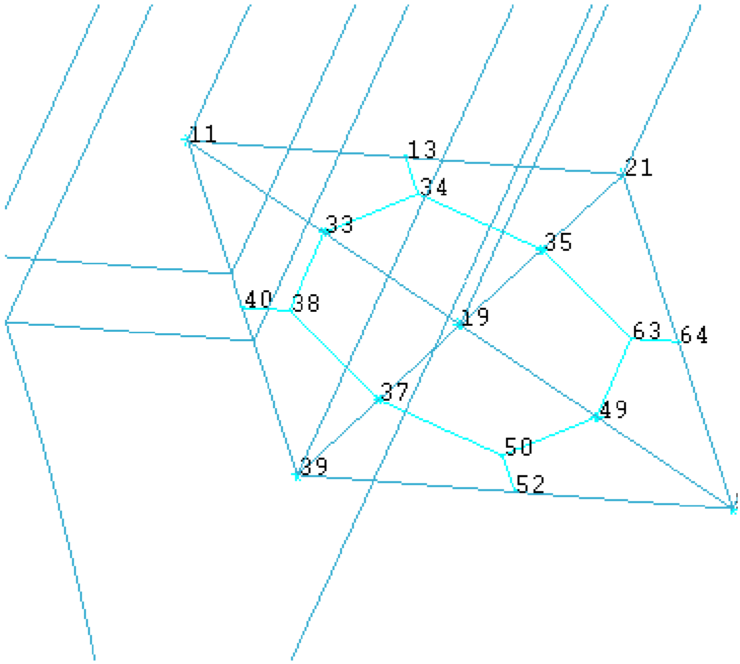
Figure 3.315
Translate Topology window



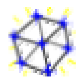
Toggle on ‘Copy’ and enter the value 3 in Number of Copies. In the angle enter the value 90 and select Rotation

Axis as Z. In the Center of Rotation Select 'User Point' and select POINTS.17 which is the center point of the GRID. Select all the Blocks and Press Apply to transform the blocking. The blocking after transformation is shown in Figure 3.316.

Figure 3.316 Blocking after Transformation

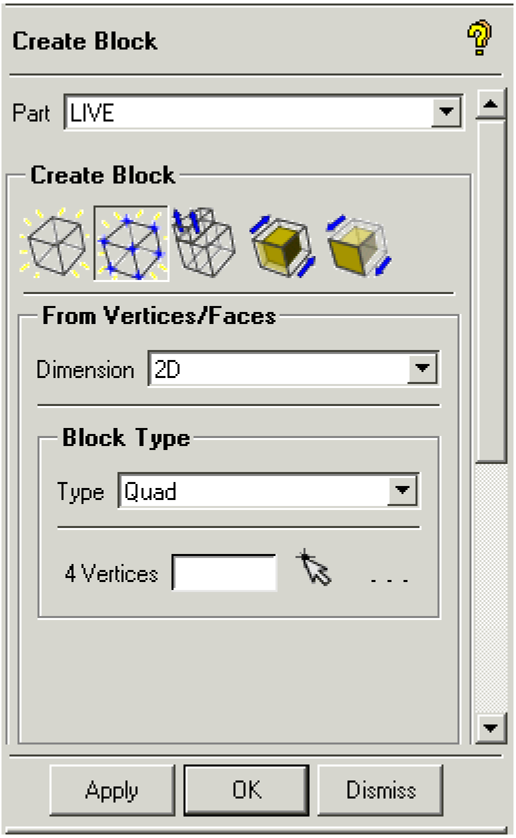


g) Creating remaining Blocks

Blocking > Create Block  > From Vertices/Faces , in the dimension select 2D and in the Block Type select Quad as shown in Figure 3.317.

Note: Part Name will be LIVE by default.

Figure 3.317
Create Vertices/Faces



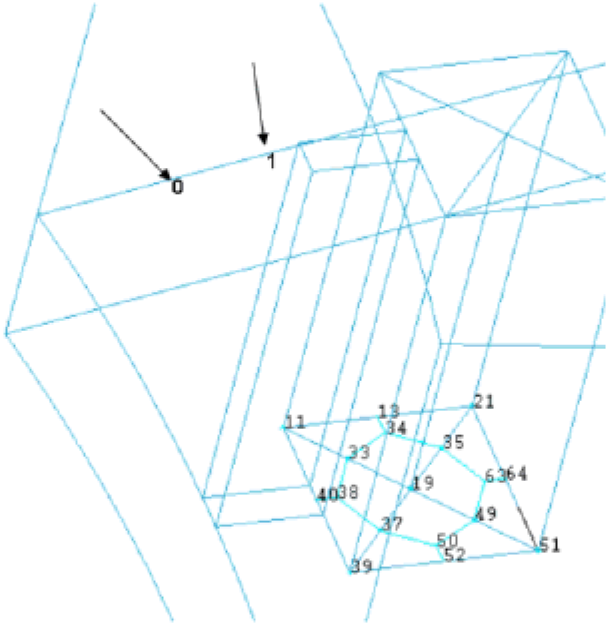
Select vertices 11 and 13 with the left mouse button in that order and press middle mouse button.

Note: As two more position needs to be selected it comes in geometry selection mode.

Proceed to select other two places 0 and 1 by screen select with the left mouse button as shown in Figure 3.318. Press middle mouse button to accept the selection and press Apply.

Figure 3.318

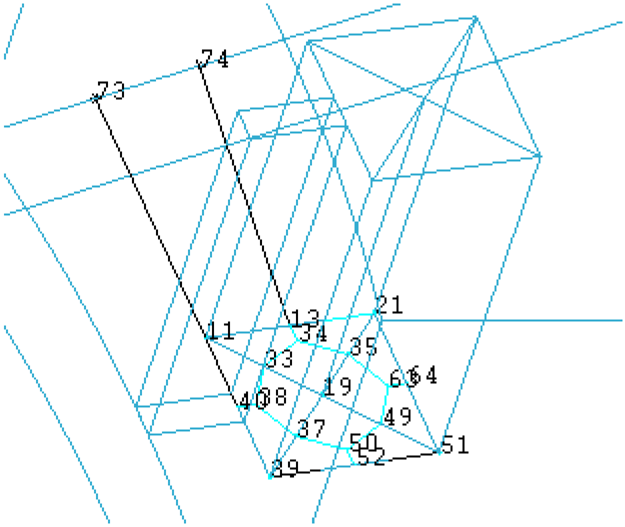
Vertex location for block and Material Selection Window



Note: The User should select the vertices/points in an order that should be in Z shape. First, all the existing vertices should be selected and once they are done, then middle mouse button should be pressed to proceed to select the screen locations.

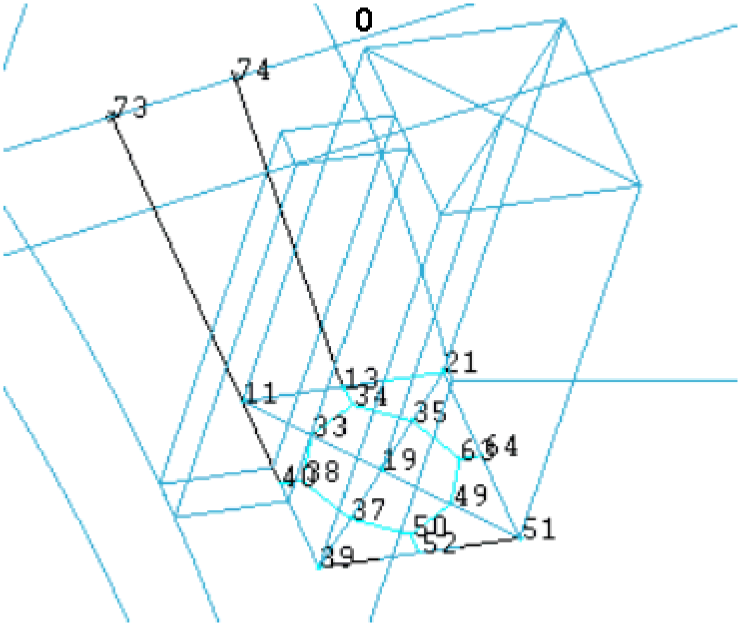
The blocking after this operation should like as shown in Figure 3.319.

Figure 3.319
Blocking after creation of block



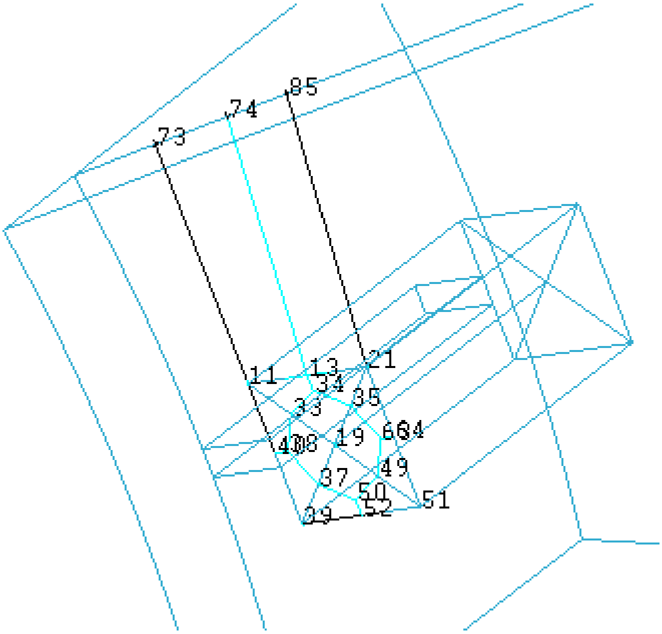
Similarly create the Block by selecting the vertices 13, 21 and 74 (in that order) and press middle mouse button. Screen select for vertex 0 as shown in Figure 3.320.

Figure 3.320
Selecting vertices for another block creation



The blocking after creation of second Block is shown in Figure 3.321.

Figure 3.321
Blocking after creation of second block



h) Step involved to achieve complete blocking

Note: User has to choose Blocking > Index Control > Reset at time to time to update the Blocking if some Blocking disappears.

Finally to achieve the complete 2D Blocking with the same vertex numbers as shown in Figure 3.322, the following steps need to be performed.

Select vertex 39 and 52 and then select two points corresponding to the position of vertex 96 and 97 as shown in Figure 3.322.

Select vertex 52, 51 and 97 (in order) and then select the point corresponding to the position of vertex 110.

Select vertex 73, and 11 then select the point corresponding to the position of vertex 123 and 124.

Select vertex 11, 40 and 124 and then select the point corresponding to the position of vertex 139.

Select vertex 40, 39 and 139 and then select the point corresponding to the position of vertex 154.

Select vertex 39, 96 and 154 and then select the point corresponding to the position of vertex 169.

Select vertex 21, and 85 and then select the point corresponding to the position of vertex 184 and 185.

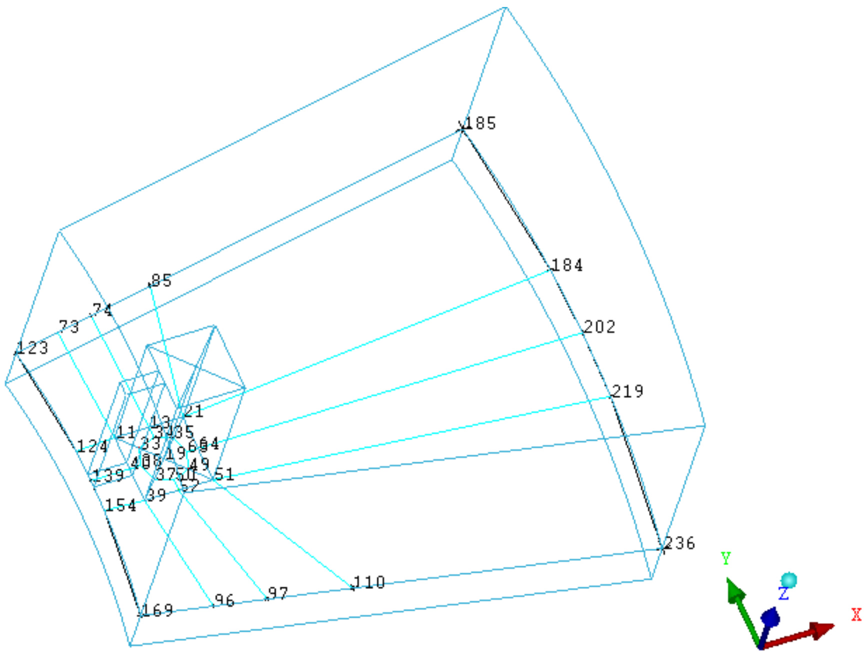
Select vertex 21, 64 and 184 and then select the point corresponding to the position of vertex 202.

Select vertex 64, 51 and 202 and then select the point corresponding to the position of vertex 219.

Select vertex 51, 110 and 219 and then select the point corresponding to the position of vertex 236.

Performing the above steps we get the Blocking as shown in Figure 3.322.

Figure 3.322
Complete 2D Blocking



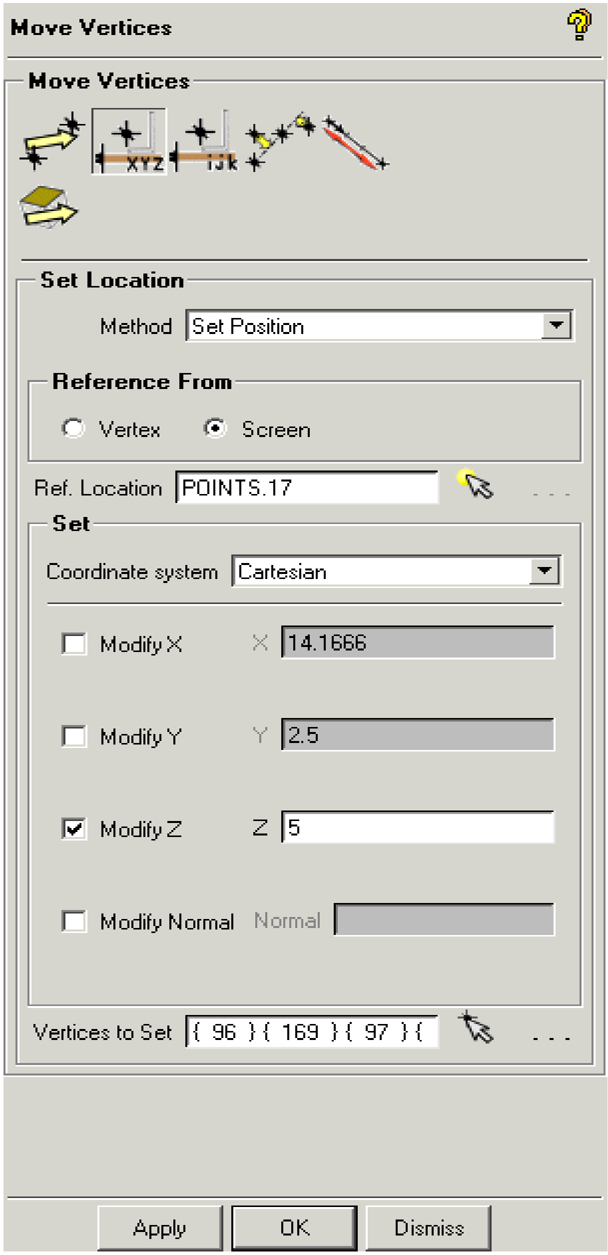
Note: It is advisable to switch off the Points when position of vertex is to be selected.

i) Placing all nodes to one plane

The user should now move the topology to the bottom most planes and then extrude it to get 3D blocking, which would then be split to get further planes.

Blocking > Move Vertex > Set location. A new window will appear as shown in the Figure 3.323. Switch On Points > Show Point name. Toggle on Modify Z, select all the vertices. Enter 5 and press Apply to move complete topology to Z=5 plane. Click on Dismiss to close the panel.

Figure 3.323 Vertex Positions Window



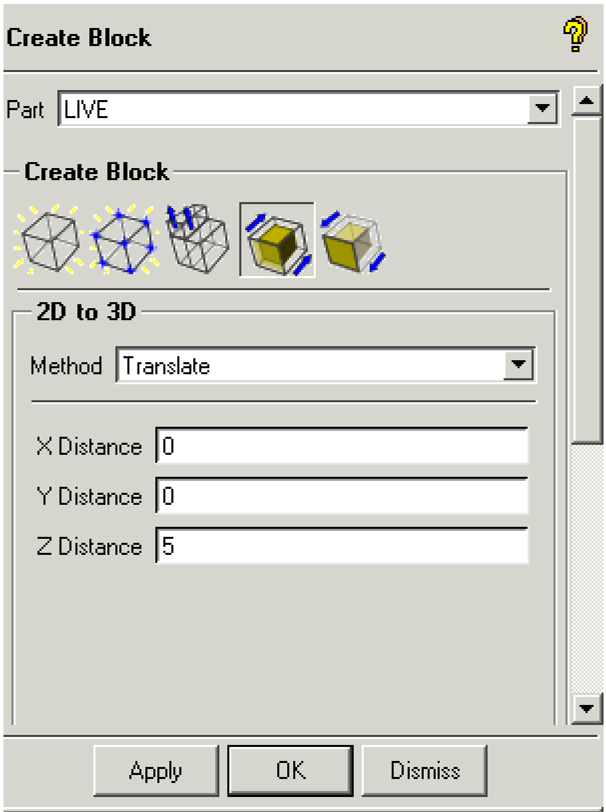
j) 3-D Blocking

Extruding 2D blocking

Blocking > Create Block > 2D to 3D.

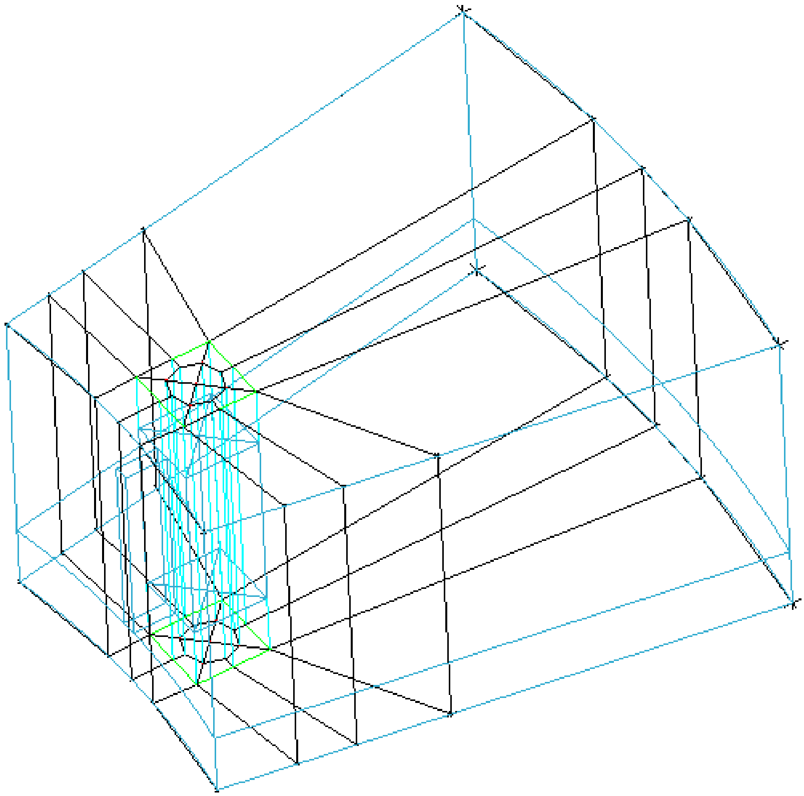
For the method, select Translate. A window will appear as shown in Figure 3.324. Enter the value 5 and press Apply.

Figure 3.324 Extrusion Window

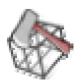


Switch off Vertices and Points. The extruded 3-D blocking is shown in Figure 3.325.

Figure 3.325
3D Blocking after Extrusion



k) Getting other regions Resolved

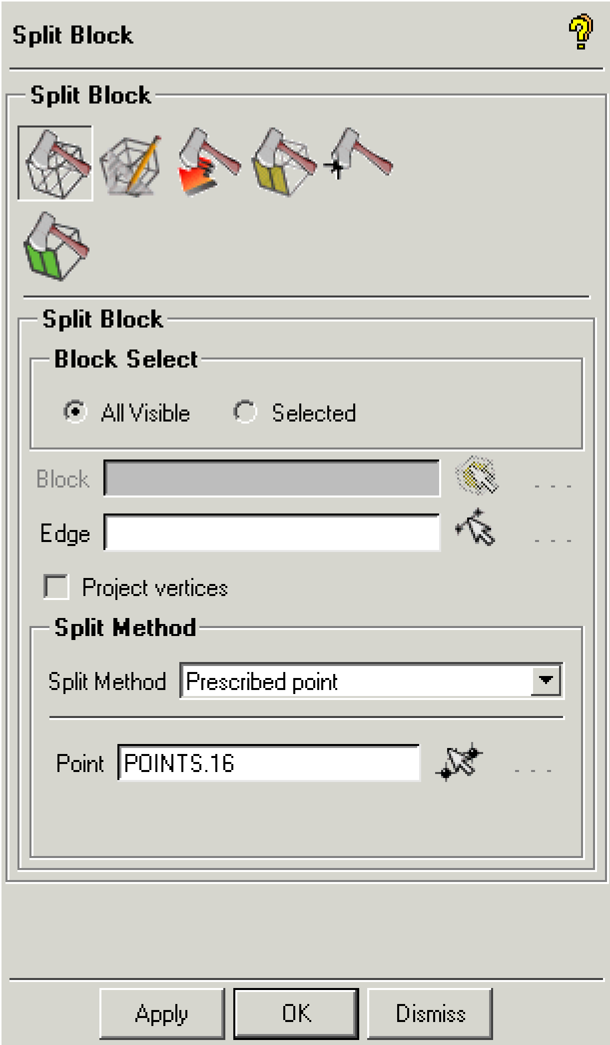
Go to Blocking > Split Block  > Split Block .

Switch on Point > Show Point Names.

Now select Prescribed point. Click on Screen select and accept POINTS.16 using the left mouse button, and press the middle mouse button to accept the selection as shown in

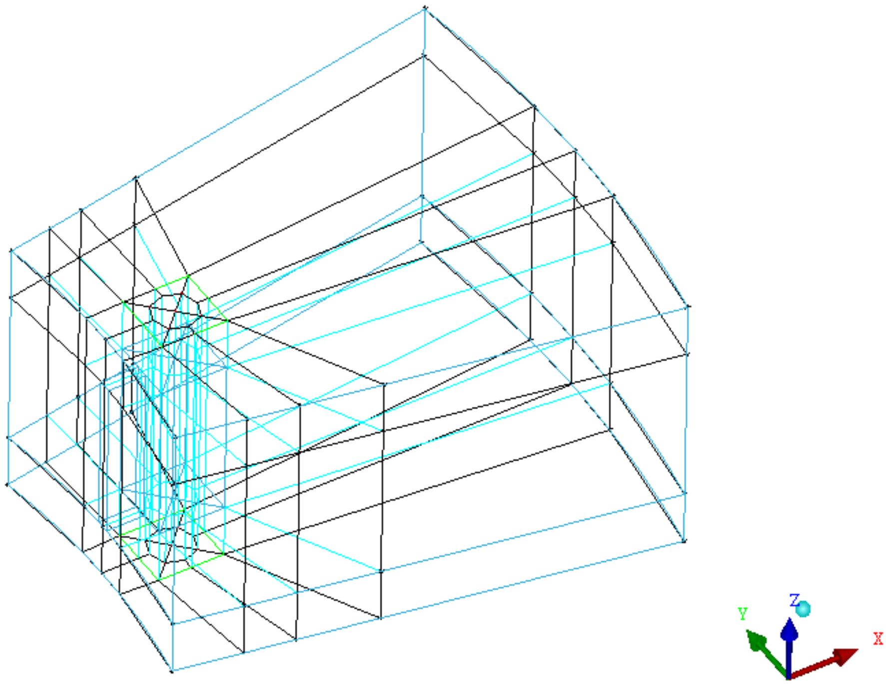
Figure 3.326. Select an edge representing the Z-direction with the left mouse button and Press Apply.

Figure 3.326
Split block window



Similarly, select any of the remaining Z-direction edge and split this edge by Prescribed point POINTS.8. Then switch off Points, to view the blocking shown in Figure 3.327.

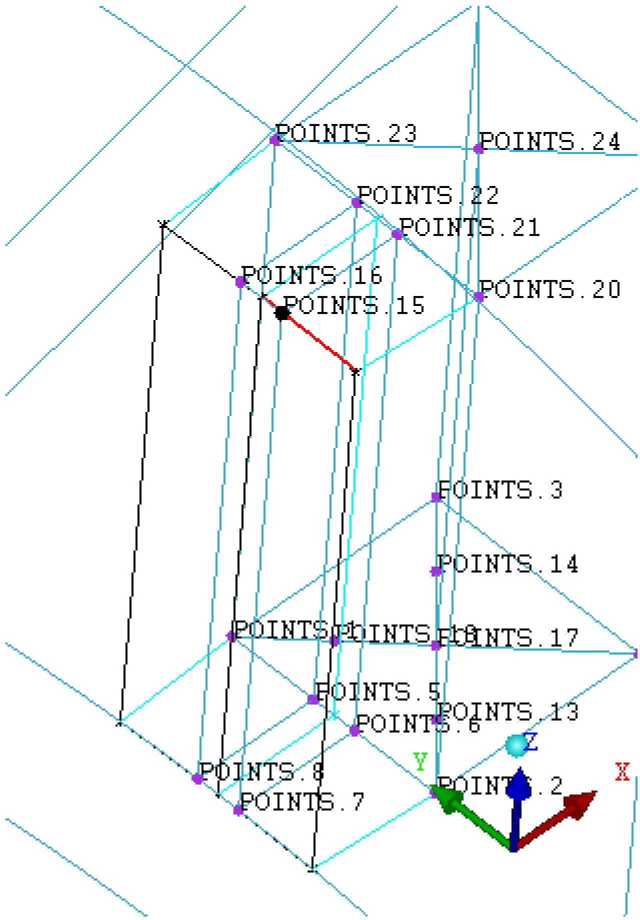
Figure 3.327
Blocking after splitting



To model the HUB region, we first need to split some blocks. Use the Blocking > Index control > Select Corner . Select the vertices to restrict the display to blocks as displayed in solid in Figure 3.330. You will need to readjust the index control so that the ranges are I:0-1, J:1-1, K:2-3, O3:0-0, O4:0-0, O5:0-1, O6:0-1, O7:0-1 and O8: 0-1. But if there is a discrepancy it can also be set by checking the edges using Query Edges and then readjusting the Index Control.

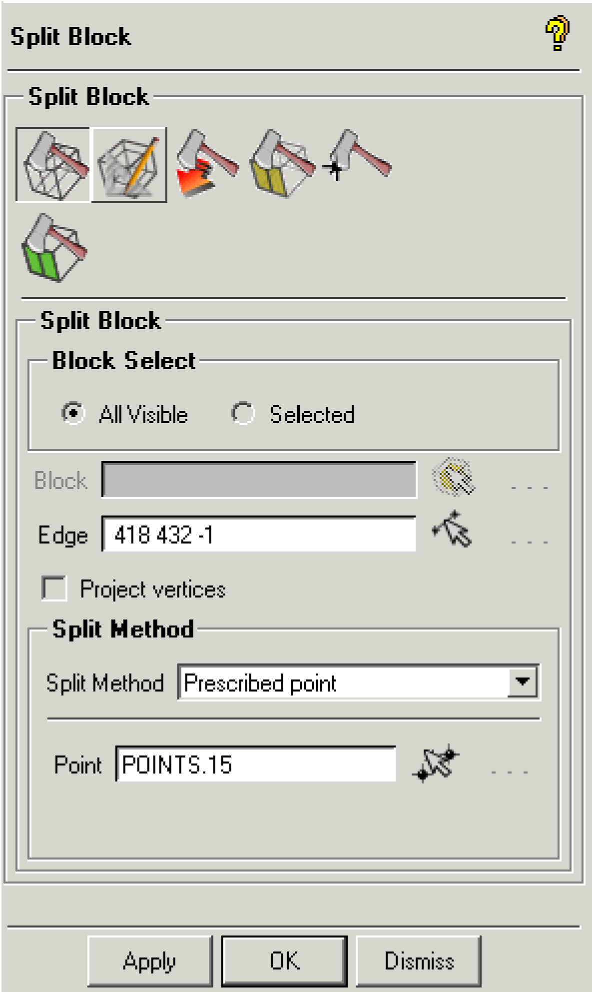
Blocking > Split Block  > Split Block . Switch on Points > Show Point Names. Select one of the circumferential edges on the right block as shown in Figure 3.328.

Figure 3.328
Edge Selected



Select POINTS.15 as shown in Figure 3.329.

Figure 3.329
POINTS.15 selected



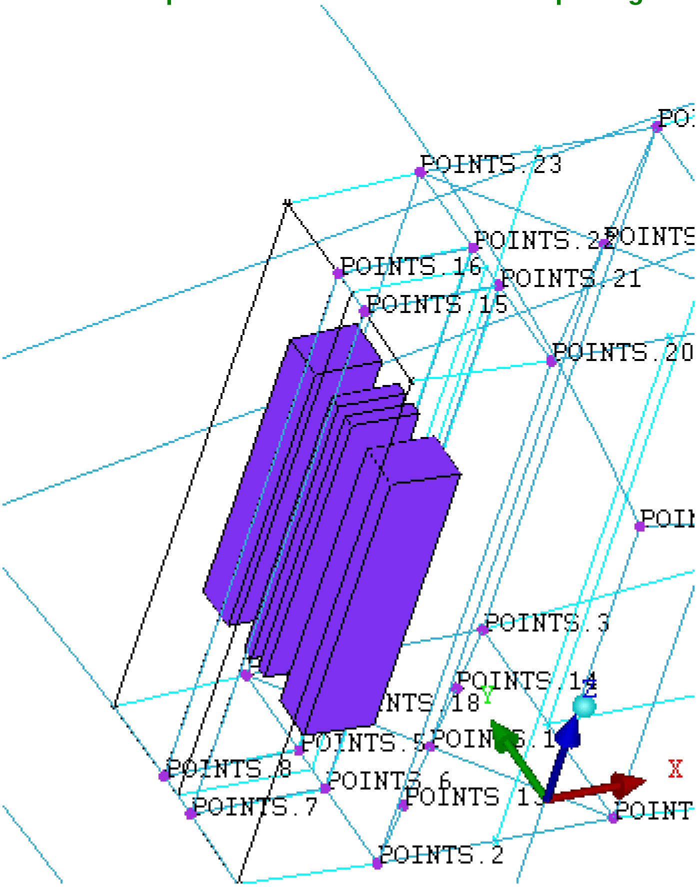
Select the method 'Prescribed Point' and select POINTS.15. It is common that upon split, one may start seeing extra blocks/edges. In that case, blocking should be restricted as explained in the previous step.

Similarly split the other Block by selecting the Circumferential edge at ‘Prescribed point’ POINTS.16.

The Final Block can be seen using Blocking > Blocks > Solid as shown in Figure 3.330.

Note: Don’t use the Whole Block option in Blocking > Blocks (Display Tree widget).

Figure 3.330
Block after Split in Selection of blocks for Splitting



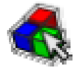
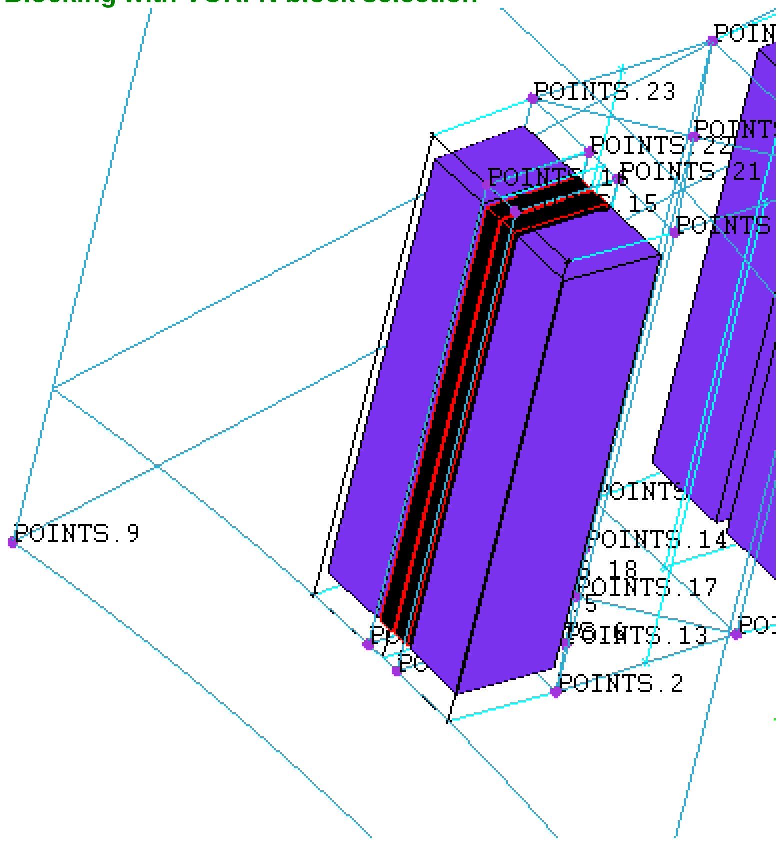
Switch OFF Blocking > Blocks. All blocks to the VORFN part by going to Parts >VORFN (keep it in Off Mode) > Add to Part > Blocking Material, Add to Part by Selection  with Blocks, and selecting the Block as shown in Figure 3.331. Press the middle mouse button and then Apply.

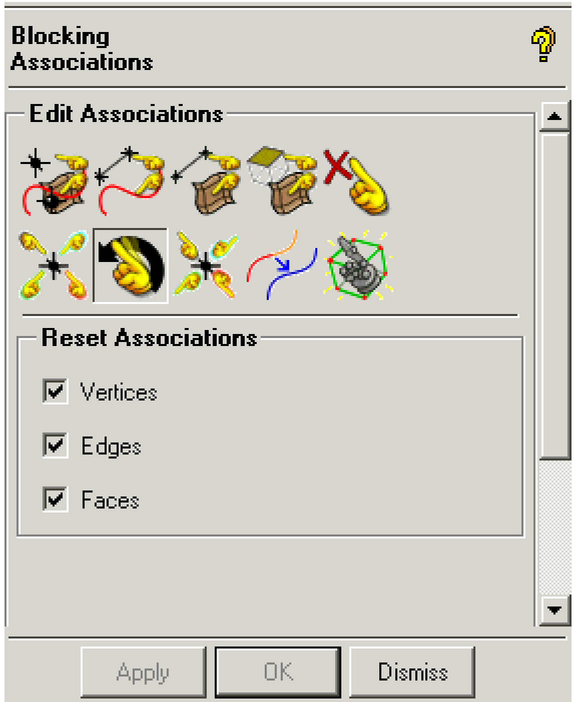
Figure 3.331
Blocking with VORFN block selection



Select Blocking > Association  > Reset Association .

Enable Vertices, Edges and Curves and Faces as shown in Figure 3.332.

Figure 3.332
Reset Association Window



To resolve the HUB accurately, the user needs to associate the edges lying on top of curves CURVES.31, CURVES.36, CURVES.28 and CURVES.34 using Association  > Associate Edge to Curve .

Note: Associate the edge to their respective Curves as mush possible.

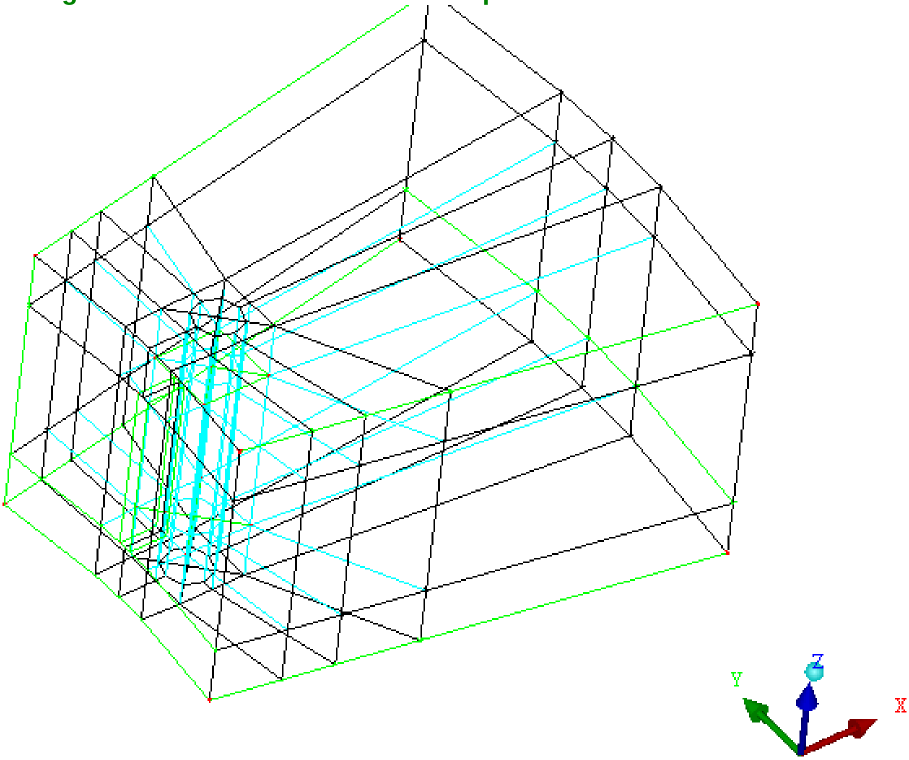
Associate all the circumferential edges to their respective curves wherever the curve exists.

Similarly, associate the vertex to its nearest point whereever possible.

Switch ‘On’ Points.

The Blocking should look like Figure 3.333.

Figure 3.333
Blocking after associations and vertices placement



l) Resolving zero thickness walls

Select Associate >Associate Face to Surface . For Method, select Part. A new window will open as shown in Figure 3.334.

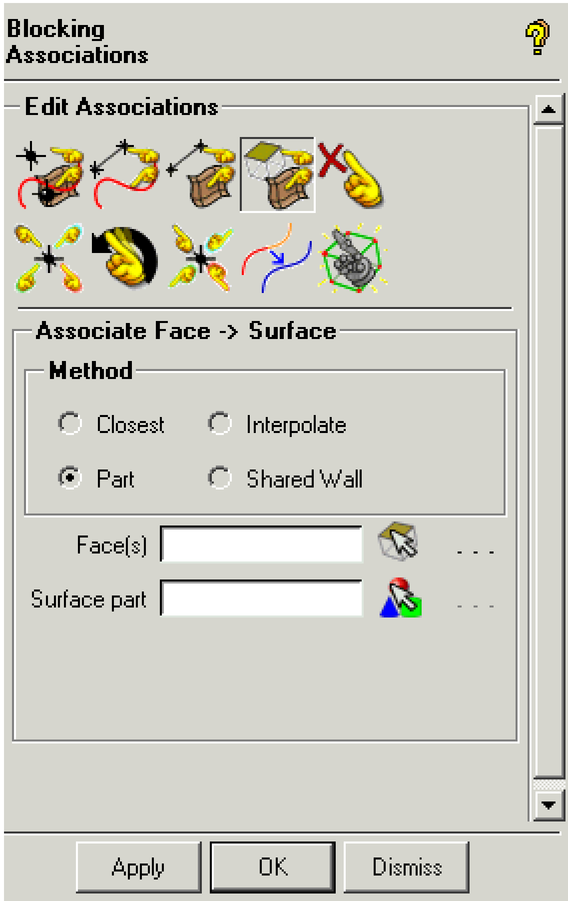


Figure 3.334
Project face window

It would be good to reduce the clutter of the Block display before selecting the faces. Use Index Control to change to

I:0-3, J:1-1, K:2-3,O4:0-0, O5:0-3, O6:0-0, O7:0-1, and O8:0-0.

In the Display Tree widget switch ‘Off’ All Parts except PLATE 1, PLATE 2, SHELL and LIVE.

Press hotkey ‘h’.

Select the FACES and its corresponding Part.

Note: Make sure that “**Toggle between all and partial enclosure**” is enabled as shown in Figure 3.335

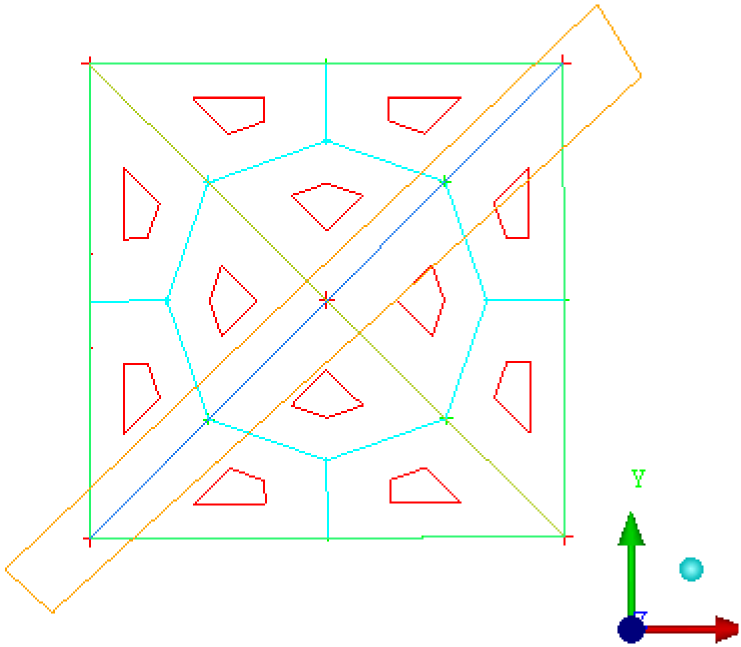
Figure 3.335 Toggle Between All and Partial option

Use Polygon  selection to select the Faces.

As shown in Figure 3.336 we can easily select the Face to be associated to PLATE 1.

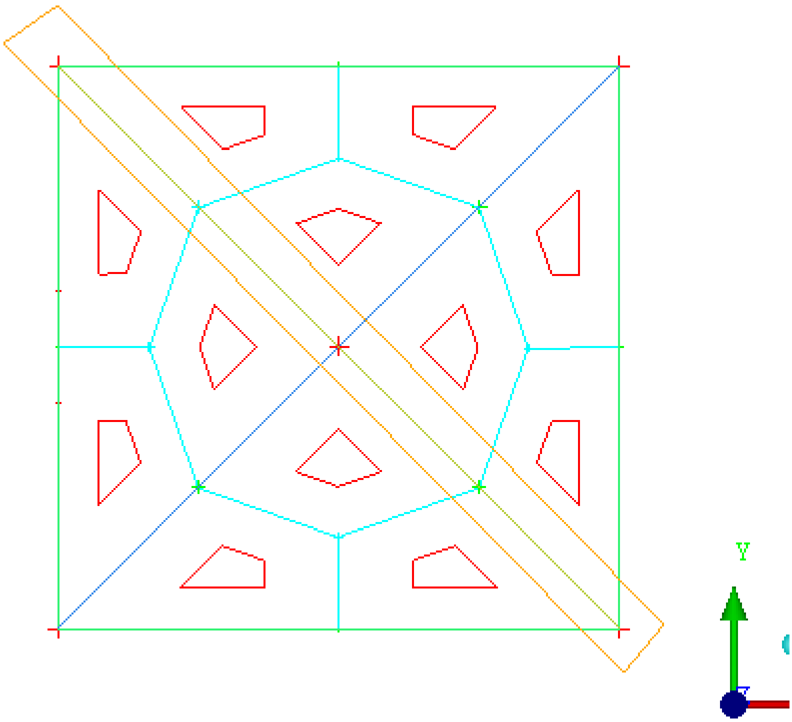
Now in the Surface Part window select Plate 1.

Figure 3.336 Faces selected to be Associated to PLATE 1



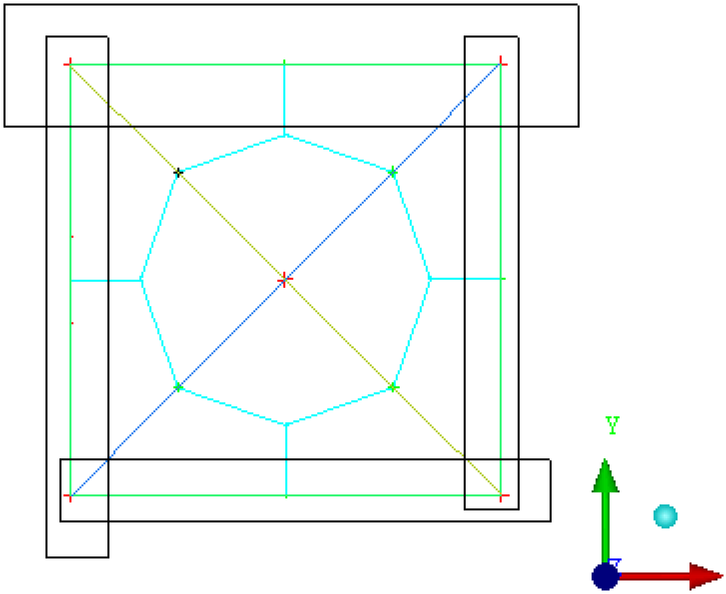
Similarly select the Face to be associated to PLATE2 as shown in Figure 3.337.

Figure 3.337 Faces selected to be Associated to PLATE 2



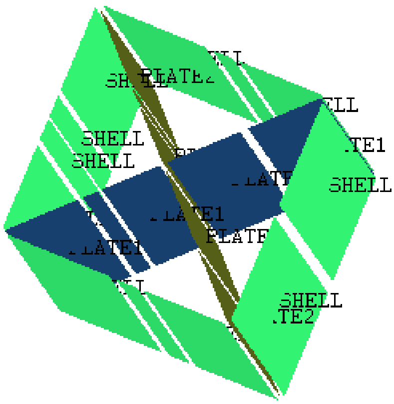
Note: Select the following region using Box Selection. Select one by one the four regions as shown in Figure 3.338. The Part must be **Shell.**

Figure 3.338 Faces selected to be Associated to SHELL



To see the face projection toggle on the Faces > Face Projection. The Face projection is shown in Figure 3.339.

Figure 3.339
Blocking with face projection on family PLATE1, SHELL and PLATE2



Switch on all Parts and switch Off Faces in the Display Tree widget.

Go To Blocking > Index Control and Reset.

Note: The two faces corresponding to the HUB location remain unprojected.

m) Defining Periodicity

Note: Defining periodicity in ICEM CFD requires periodic definitions such as Axis location and Angle that are already defined in the tetin file.

Select Blocking > Edit Block >Periodic Vertices , which will open up a panel as shown in Figure 3.340.

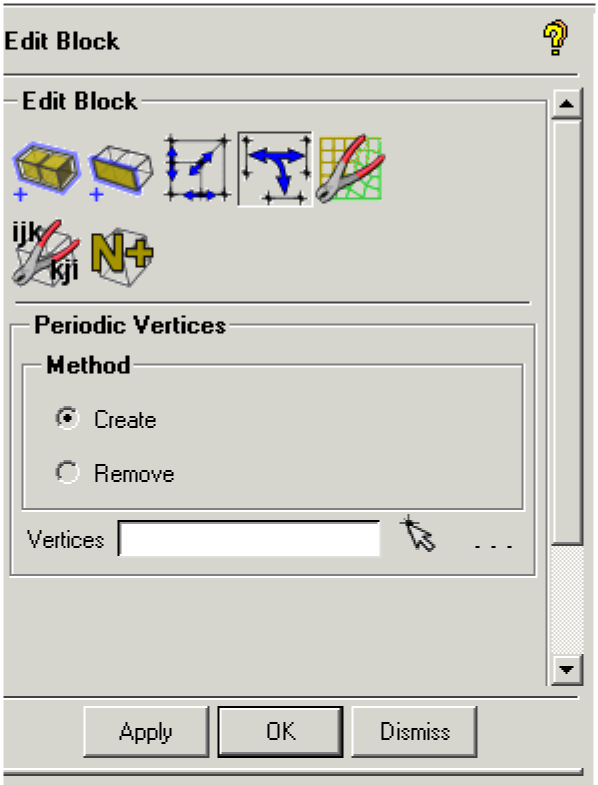


Figure 3.340
Periodic Vertices Panel

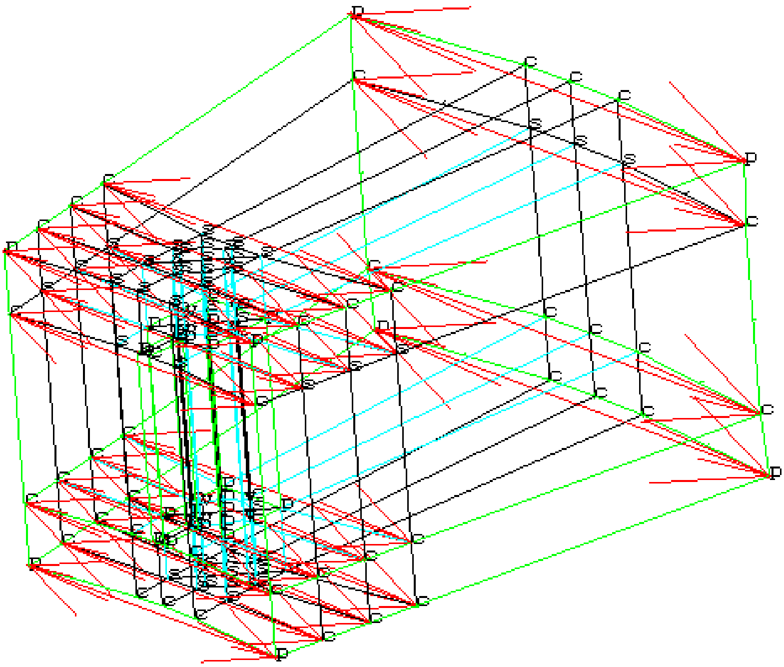
Toggle on Create, select a vertex lying on the periodic plane PERIODIC1 and the corresponding vertex on periodic plane PERIODIC2. This will define the periodicity between two vertices. One can see the periodicity by switching on, Vertices > Periodic from the Display Tree widget.

Do this for the rest of the vertices lying on the periodic planes, PERIODIC1 and PERIODIC2.

Click Dismiss to close the panel. At the end of this process, select Verticies > Periodic we get the display as shown in Figure 3.341.

Figure 3.341

Periodicity in the blocking

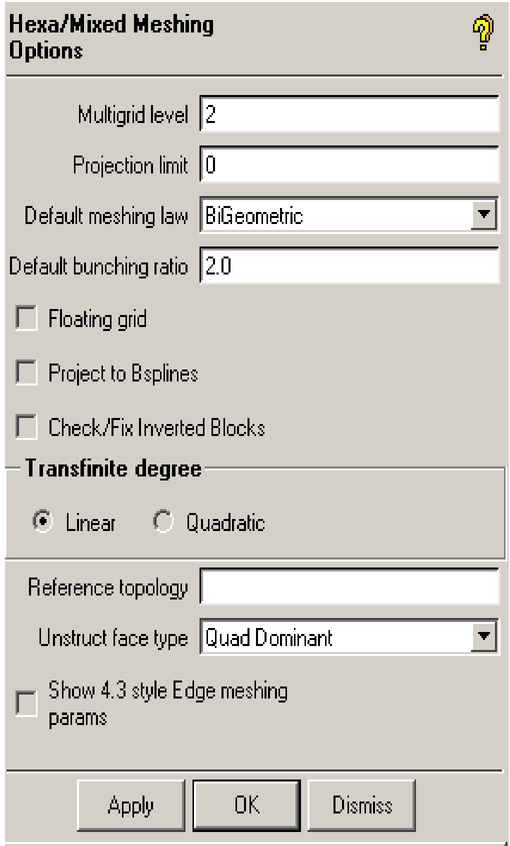


n) Generating the Mesh

Before generating the mesh, the user needs to set the meshing parameters.

Define multigrid for this mesh. The multigrid mesh is required for quite a few solvers and needs to have certain restrictions on the number of nodes that can be defined on an edge. For example, multigrid level 2 would require 5, 9, 13, 17 and similar numbers of nodes on an edge. Hexa allows only these numbers of nodes to be defined on an edge. To activate, select Setting > Meshing > Hexa/Mixed. Enter 2 for Multigrid level in the Meshing options window as shown in Figure 3.342, and press Apply.

Figure 3.342
Meshing option window

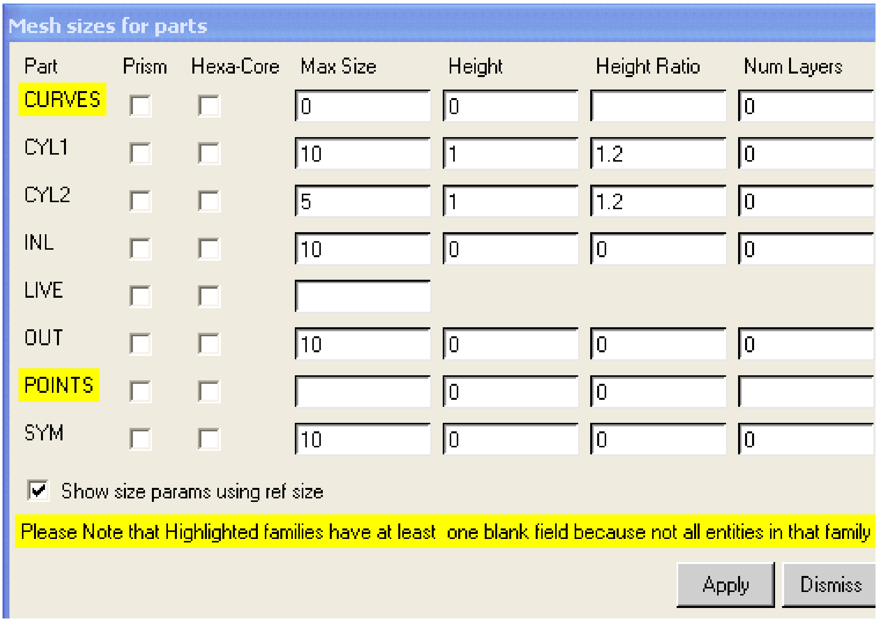


Press Mesh > Set Surface Mesh Size  to open the Mesh parameters window (Figure 3.343).

Select all the Surface Parts and then set the Max Element size to 0.4, Height to 0.4 and Height ratio to 1.2. Press Apply.

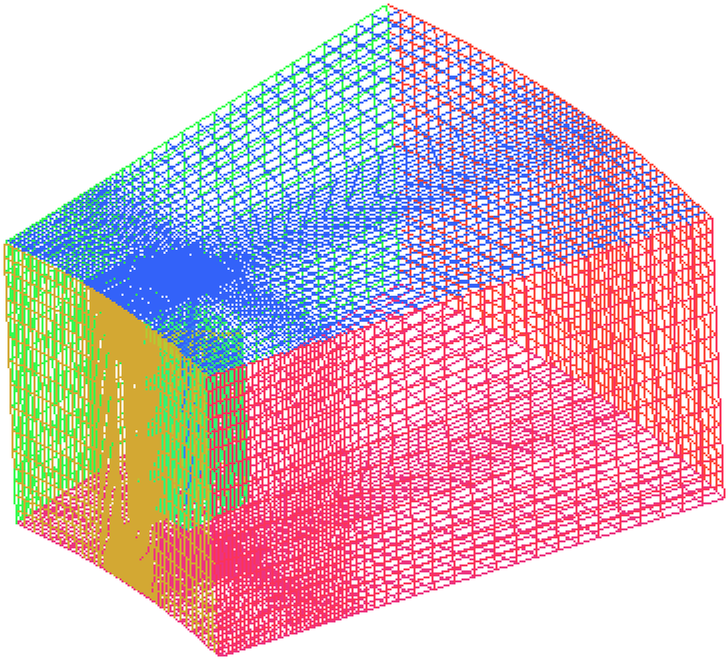
Press Dismiss to close the window.

Figure 3.343 Meshing Parameter window



Blocking > Pre-mesh Params > Update Size toggle on 'Update All' and press Apply.

Figure 3.344 Mesh in geometry

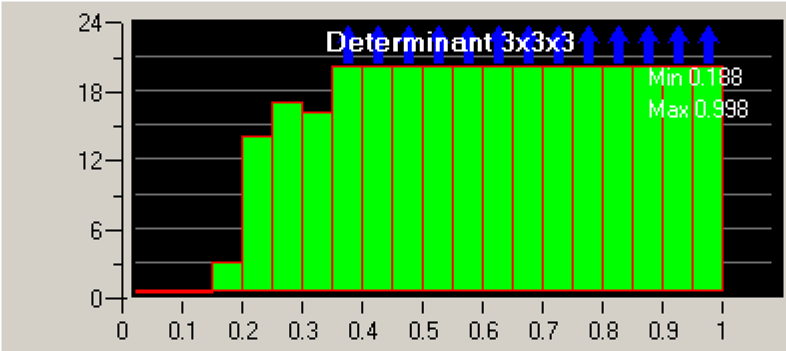


In the Display Tree widget turn on Project faces and answer 'Yes' when asked whether to recompute the mesh. Turn on the Mesh in the Display Tree widget to see the mesh as shown in Figure 3.344.

o) Checking the Mesh Quality

Select Blocking > Pre-mesh Quality. For the Criterion, select Determinant (2x2x2 stencil) to view the histogram as shown in Figure3.345.

Figure3.345
Determinants histogram

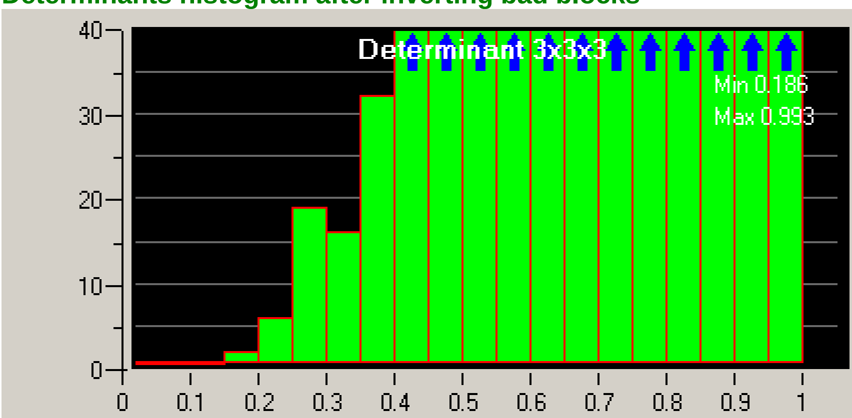


As is shown, there are many bad determinants in first bar from 0-0.05. This happens because inverted blocks were created while creating the block.

Select Blocking > Block Check. Select the method Fix inverted Block > Apply. That will change the direction of inverted blocks.

Again select Blocking > Pre-mesh Quality > Determinant (2x2x2 stencils). In the Mesh window select 'Yes' to recompute the mesh. Now the histogram appears as shown in Figure3.346, without bad determinants.

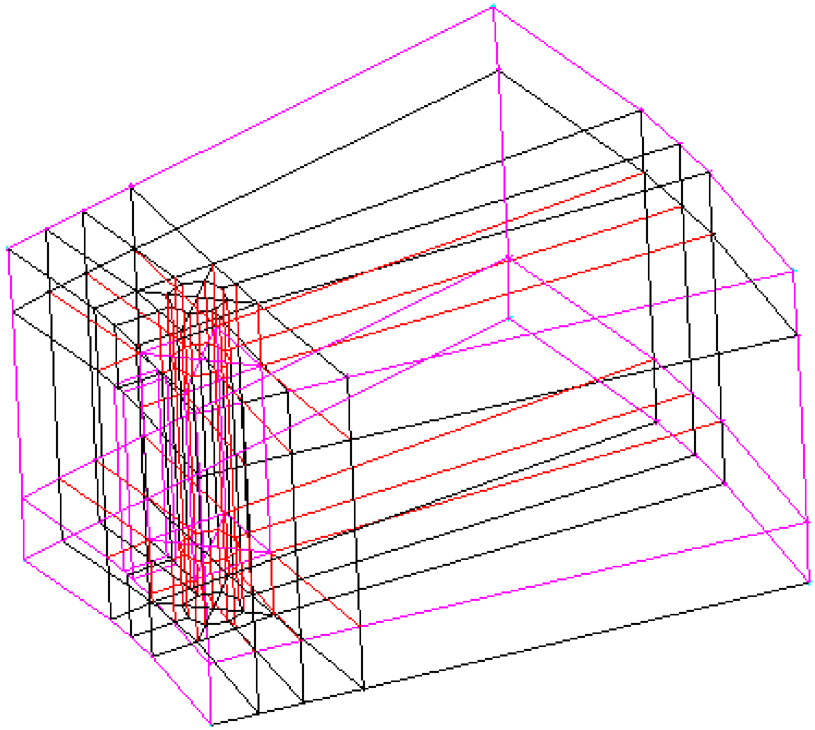
Figure3.346
Determinants histogram after inverting bad blocks



p) Multiblock mesh

For blocking a complicated geometry, the user will end up with many splits, thereby producing many unnecessary blocks. To write a Multiblock mesh output for some solvers, it is better to have as least amount of blocks possible. You can reduce the number of blocks as explained below.

Figure3.347
Blocking before reduction of number of blocks

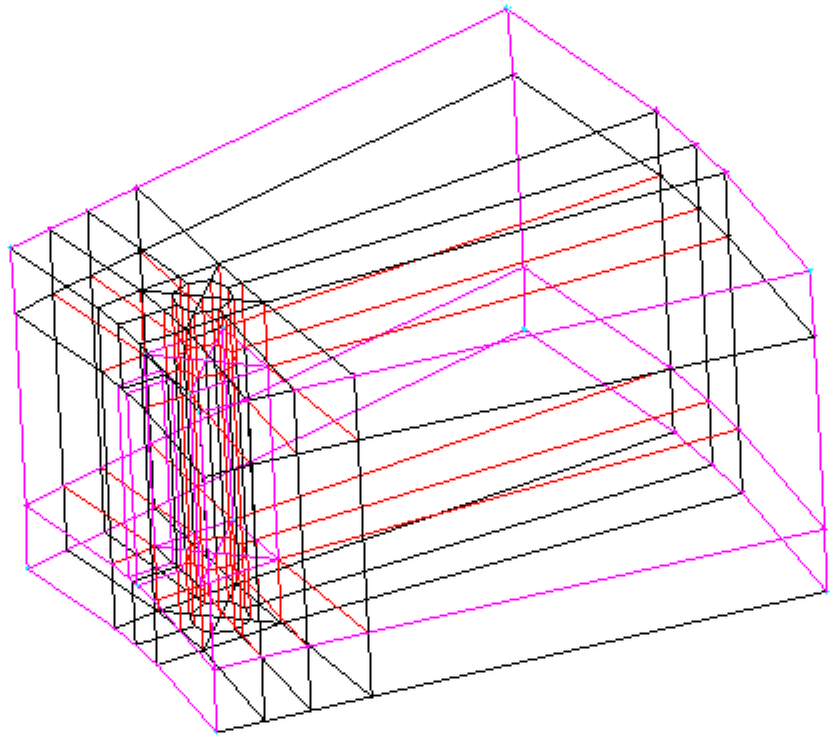


Select Blocking (from the Display Tree widget) > Init Output blocks. This will initialize the output topology for Multiblock mesh.

Toggle on Pre-mesh > Output blocks in the Display Tree widget.

Select Blocking > Edit Block > Merge blocks > In the Join Block Toggle on Automatic. This will merge the unnecessary blocks as shown in Figure 3.348.

Figure 3.348
Blocking after Auto merge



q) Saving the files

Save the blocking, using File > Blocking > Save blocking.

Save the Multiblock mesh with File > Blocking > Write Multiblock domains and select Volume when asked to select the type of domain.

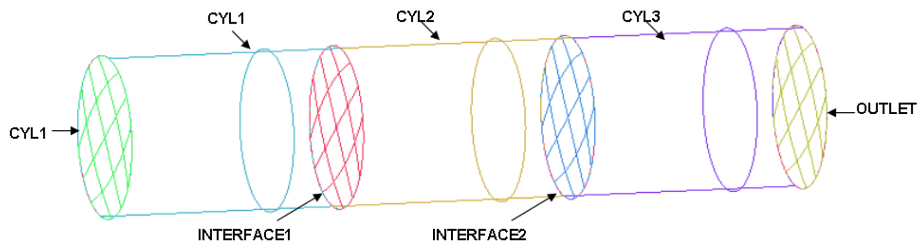
Finally File > Exit to quit ANSYS ICEMCFD

3.6.2: Hybrid tube

Overview

In this tutorial, the user will generate a hybrid mesh for the Hybrid Tube geometry shown in Figure 3.349. The tube is comprised of three regions (CYL1, CYL2 and CYL3) separated by the two interfaces INTERFACE1 and INTERFACE2. The user will first generate 2 separate tetra domains in CYL1 and CYL2, and a hexa domain in CYL3. The three domains will be made conformal at the two interfaces.

Figure 3.349
Hybrid Tube with three sections



a) Summary of Steps

Starting the Project

Generating the Hybrid Mesh

Generating the Tetra Mesh in Middle Section

Merging the Tetra Mesh between Left and Middle Section

Generating the Hexa Mesh in Right Section

Merging the Resultant Mesh with Hexa Mesh at Interface2

Saving the Project

b) Starting the Project

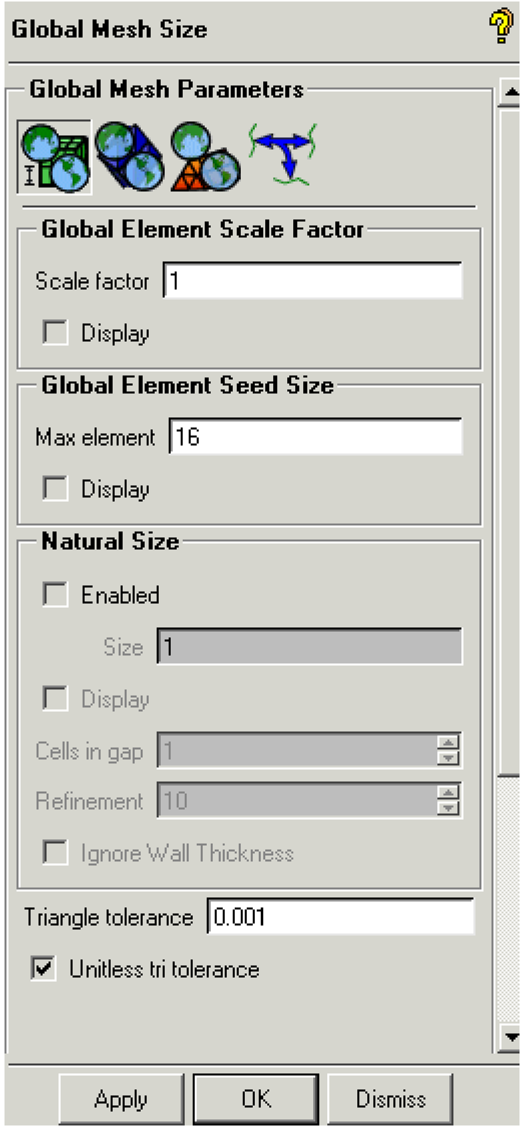
From UNIX or DOS window, start ANSYS ICEMCFD. File > Change working directory $ICEM_ACN/../docu/CFDHelp/CFD_Tutorial_Files>Hybrid tube project. Choose its Tetin file geometry.tin.

c) Generating the Hybrid Mesh

Generating the Tetra Mesh in Left Section

Select Mesh > Set Global Mesh Size  > General Parameters  Enter 16 as Maximum size in the Global mesh size window as shown in Figure 3.350. Press Apply followed by Dismiss to close the window.

Figure 3.350
Global Mesh Size window

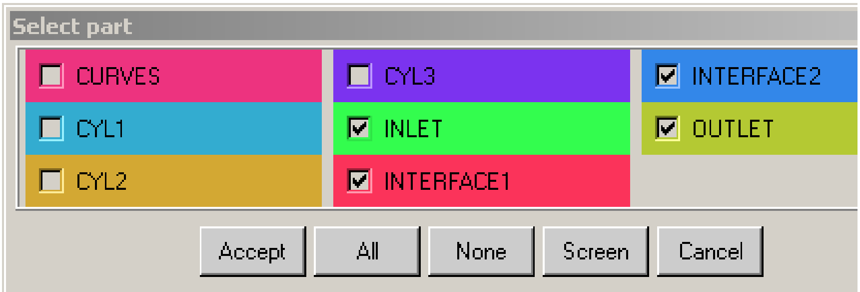


Select Mesh >Set Surface Mesh Size . A Surface mesh size window will appear.

Press Select Surf(s)  .

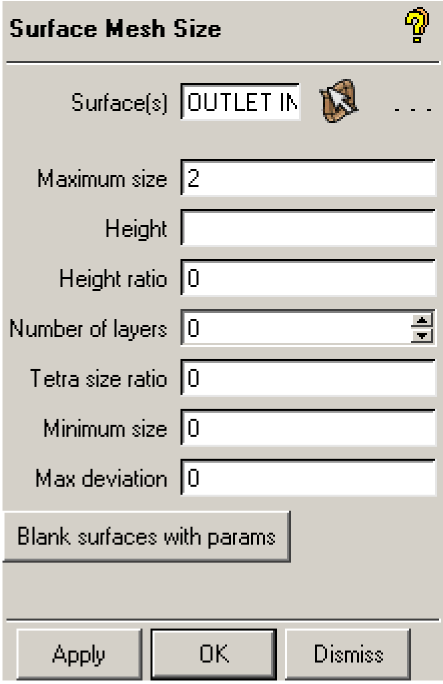
Press Select Item in Part  .
A window appears as shown in Figure 3.351, select INLET, INTERFACE1, INTERFACE2 and OUTLET. Press Accept.

Figure 3.351 Select Part Window



Enter Maximum size as ‘2’ as shown in Figure 3.352.

Figure 3.352
Surface Mesh Size window



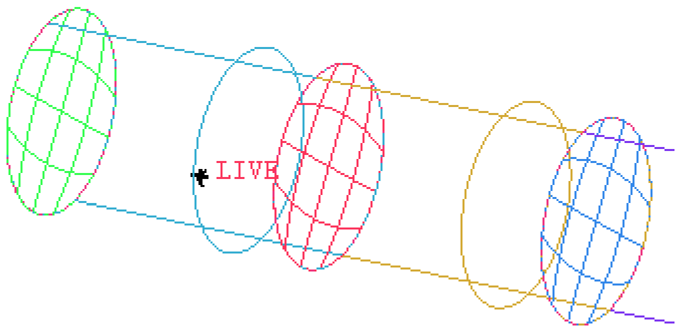
In the same procedure select and select the surfaces CYL1, CYL2 and CYL3. Enter Maximum element size 4. Press Apply followed by Dismiss to close the window.

Select Mesh >Set Curve Mesh size . A Curve mesh sizes window will appear. Press "a" to select all curves of the model. Enter a value 4 for Maximum Size in the Curve mesh params window and press Apply followed by Dismiss to close the window.

Select the Orient > Home option. Select Geometry > Create Body >Material Point . A window will appear. Select the family name as LIVE. Press Accept. Click on two opposite corners of the CYL1 using the left mouse button. Press the middle mouse button to complete the operation.

Turn on Materials. Rotate the model to ensure that LIVE lies inside the left section as shown in Figure 3.353.

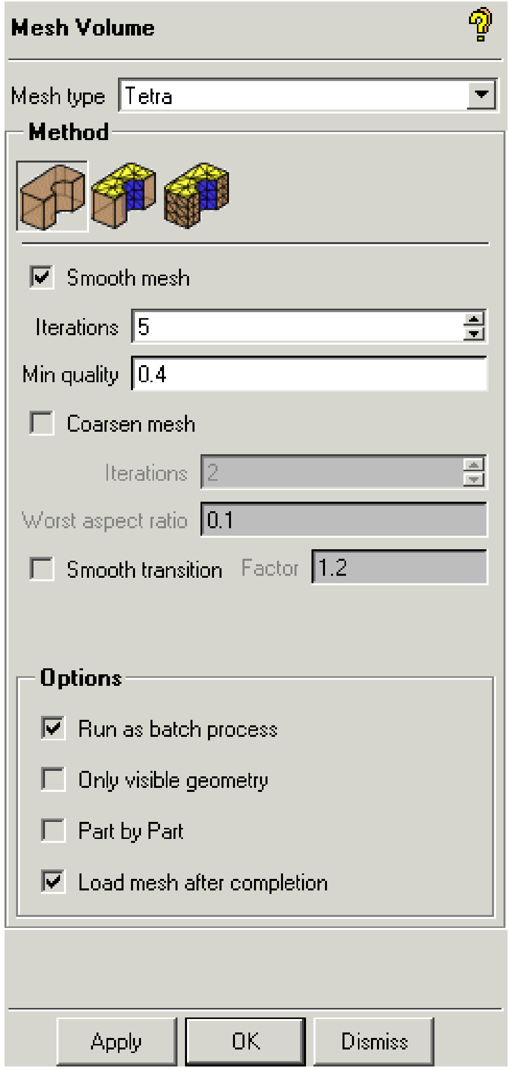
Figure 3.353 LIVE Body Created



Select File > Save project Enter any Name.

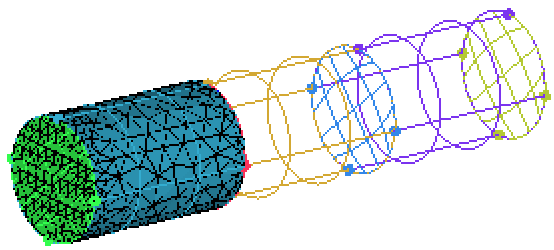
Select Mesh> Volume Meshing  > From Geometry  Window Mesh with tetrahedral parameters will appear (Figure 3.354). Press Apply to generate the tetra mesh.

Figure 3.354
Mesh with Tetrahedra window



Tetra mesh will be generated as shown in Figure 3.355.

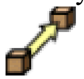
Figure 3.355
Tetra Mesh in Left Section with Solid/wire mode



Note: Even though Tetra Mesh is created we again save by another name.

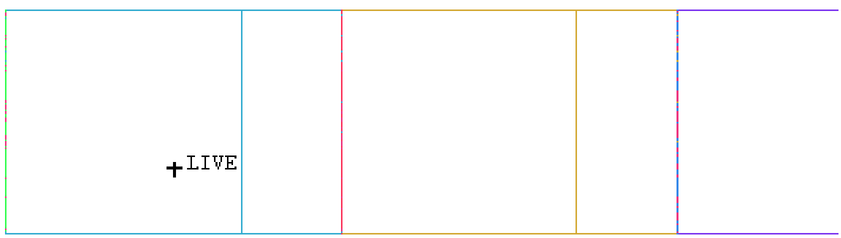
From Main menu, select File > Mesh > Save mesh as tetra_mesh1.uns followed by File > Mesh > Close Mesh.

d) Generating the Tetra Mesh in Middle Section

Select Geometry >Transform Geometry  > Translate Geometry 

Select the LIVE (Body) with the left mouse button. Press middle mouse button to accept as shown in Figure 3.356.

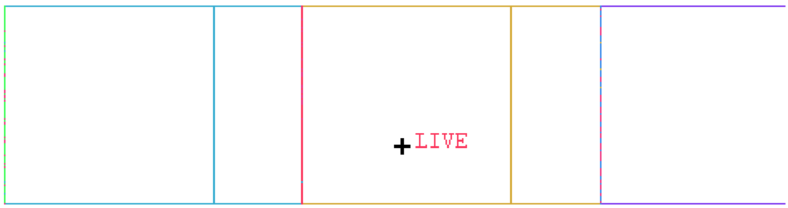
Figure 3.356 Live region selected



Enter some value in the ‘X’ direction so that it is located in CYL2 and then press Apply.

It should be repositioned as shown in Figure 3.357.

Figure 3.357 LIVE region repositioned



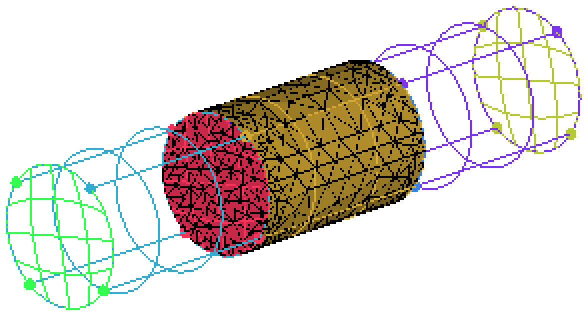
Go to Part > LIVE>Rename and Rename it as LIVE1.

From Main menu, select File > Save project.

From Main menu, select Mesh> Volume Meshing  >

From Geometry  .Press Apply with the default setting the Tetra mesh will be generated in the middle region as shown in Figure 3.358.

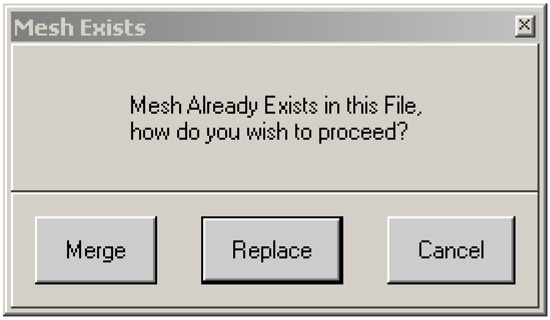
Figure 3.358
Tetra Mesh in the middle section with Solid/Wire model



e) Merging the Tetra Mesh between Left and Middle Section

From Main menu, select File >Mesh > Save Mesh as tetra_mesh2.uns followed by File > Mesh > Open mesh A window will appear as shown in Figure 3.359 with Merge button. Press Merge. A selection window will appear as shown in Figure 3.360.

Figure 3.359
Window with Merge Option



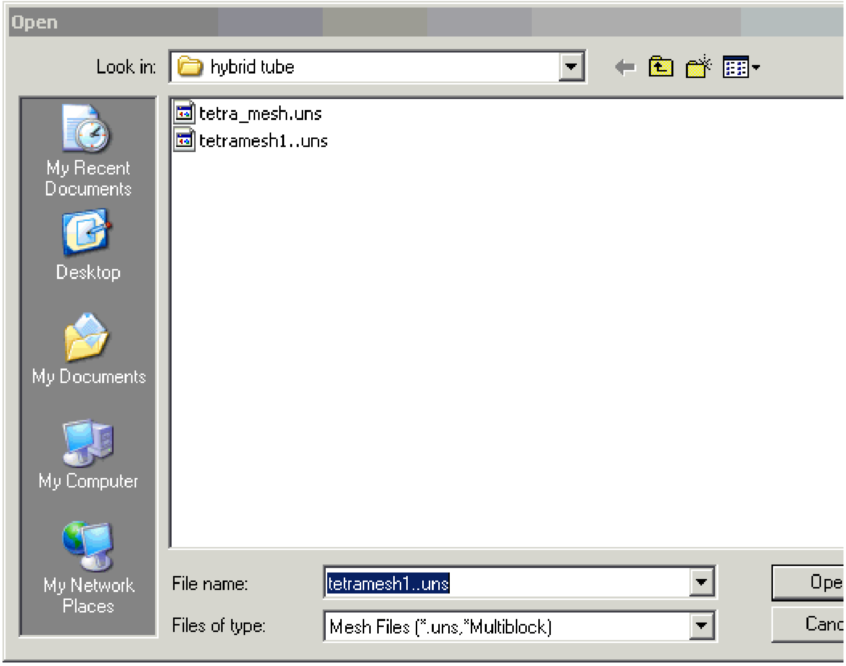
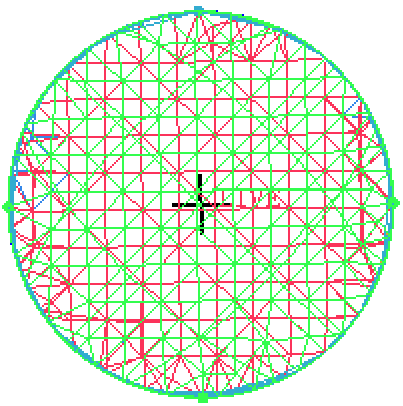


Figure 3.360 File selection window

Figure 3.361 Tetra Mesh before Merging



Before merging, turn on CYL1, CYL2 and INTERFACE1. The surface mesh at the INTERFACE1 will look as shown in Figure 3.361

Select Edit Mesh > Merge Node > Merge Meshes . A window will open as shown in Figure 3.362 select merge volume meshes and press merge surface mesh parts

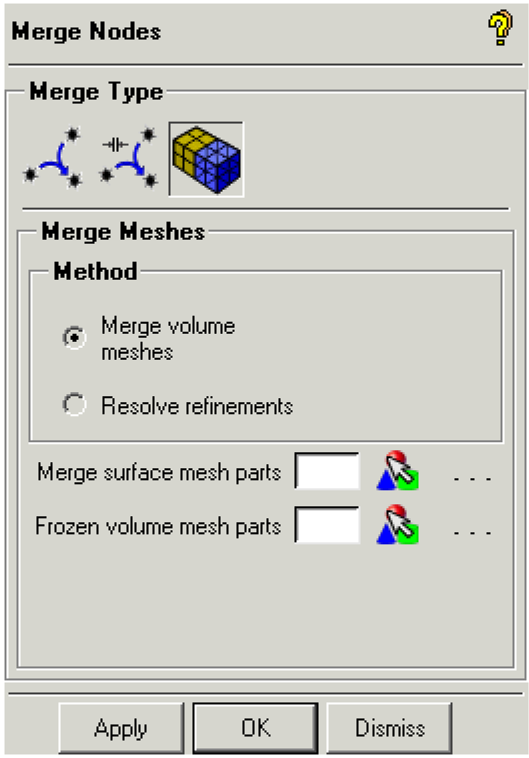
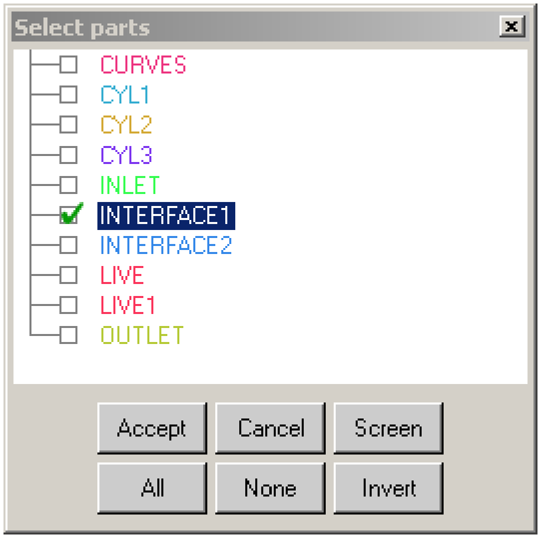


Figure 3.362
Merge meshes Window

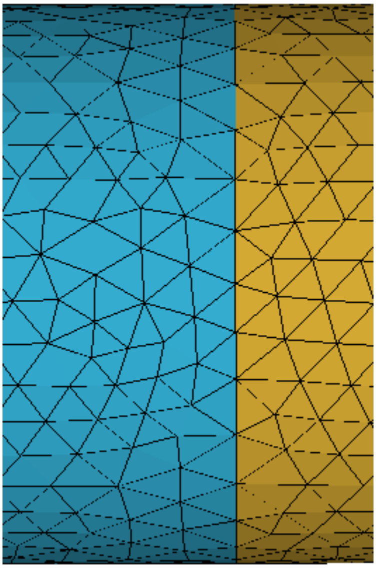
A selection window will appear as shown in Figure 3.363. Select INTERFACE1. Press Accept and then press Apply.

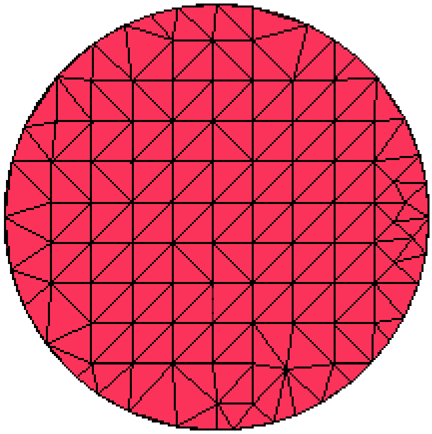
Figure 3.363
Select parts to Merge Meshes window



After merging, the surface mesh at the INTERFACE1 will look as shown in Figure 3.364.

Figure 3.364
Tetra Mesh after Merging





Switch OFF lines and triangles elements to see just the geometry. Switch on all the parts if they are turned OFF. Select the Orient > Home option.

From Main menu, select File > Save project.

f) Generating the Hexa Mesh in Right Section

Switch Off the Mesh in The Display Tree.

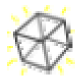
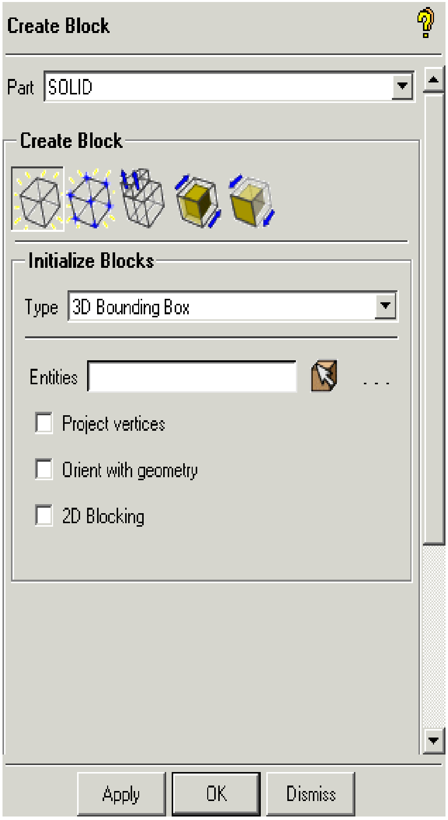
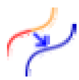
Select Blocking > Create Block  > Initialize Block  as shown in (Figure 3.365), select all the entity and Press Apply.

Figure 3.365
Create Block window



Turn on Vertices and their numbers with Vertices > Numbers and Curves and their names with Curves > Show Curves Names from Display Tree

Select Association  > Group Curve  Toggle on Group Curve and select the Curves corresponding to CURVES.4 and CURVES.3 as shown in Figure 3.367.

Select Association  > Associate Edge to Curve  .

Enable Project Vertices.

Select CURVES.4 and corresponding Edges 37-38, 38-42, 42-41 and 41-37 as shown in Figure 3.367 by using the left mouse button. Click the middle mouse button to accept the selection and then as shown in press 'Apply' as shown in Figure 3.366.

Figure 3.366 Blocking Association window

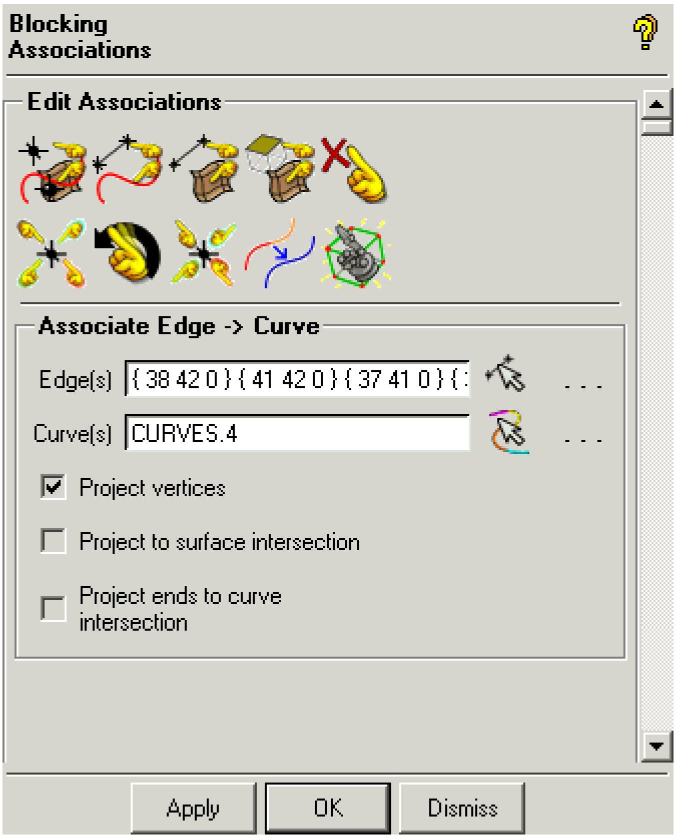
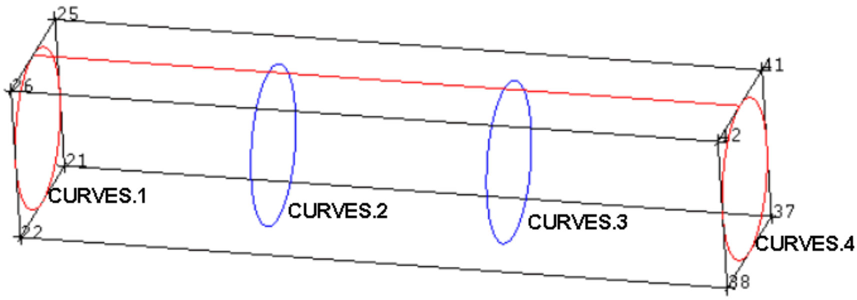


Figure 3.367 Projecting the edges on curves

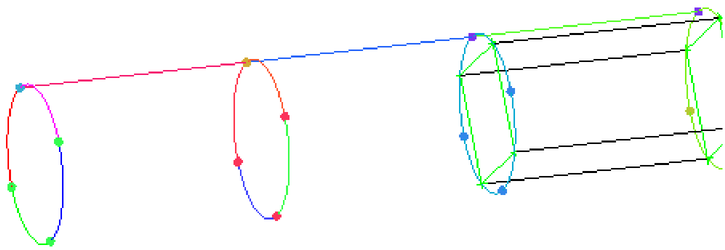


Note: Make sure that Project Vertices is enabled.

Repeat the same procedure for CURVES.3 and corresponding Edges 21-22, 22-26, 26-25 and 25-21.

After Projecting vertices, geometry will look like Figure 3.368

Figure 3.368 Blocking After projecting vertices

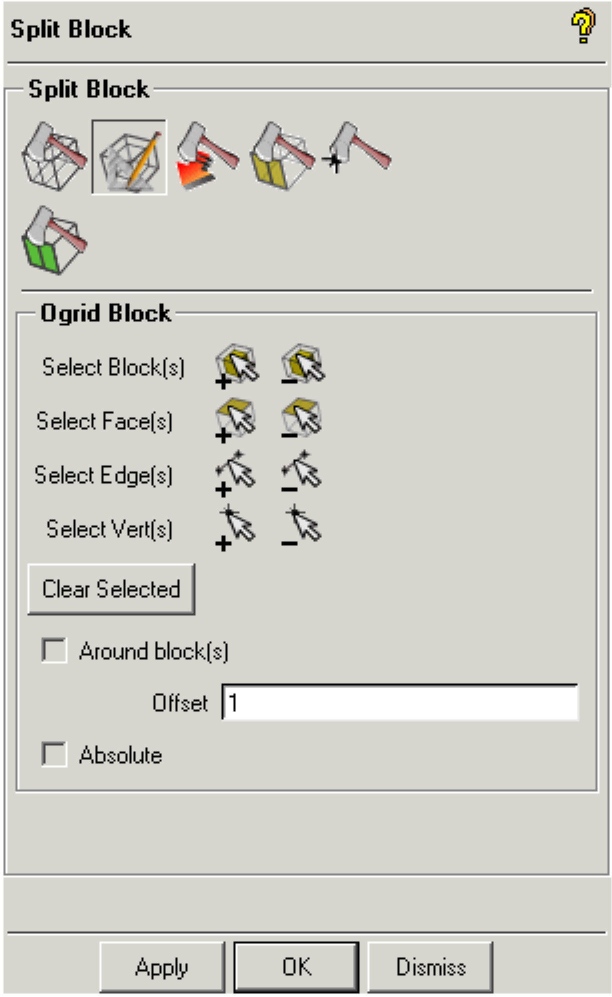


Note: -User has to turn off and again turn on vertices for updating the position of the vertex numbers.

Select Blocking >Split Block  > OGrid.  A window will appear as shown in Figure 3.369.

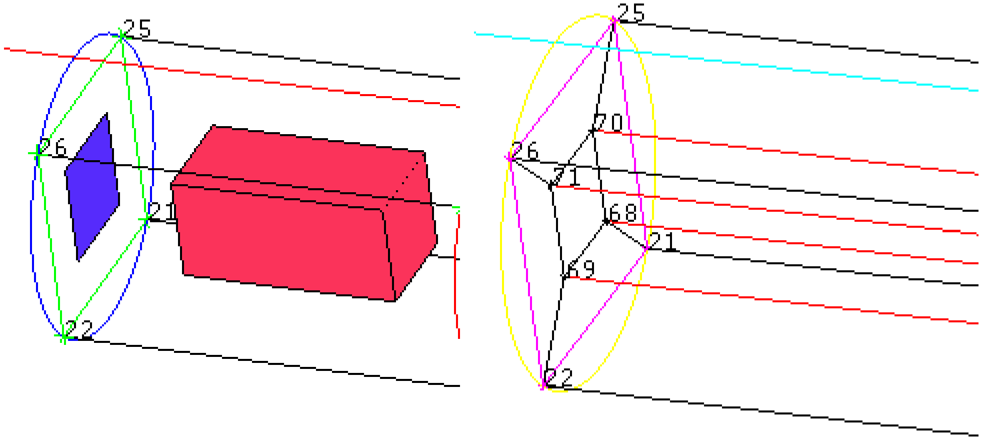
Press  Add in front of Select Block and select existing block by using the left mouse button and then middle mouse button to accept the block.

Figure 3.369
Inner O-grid creation window



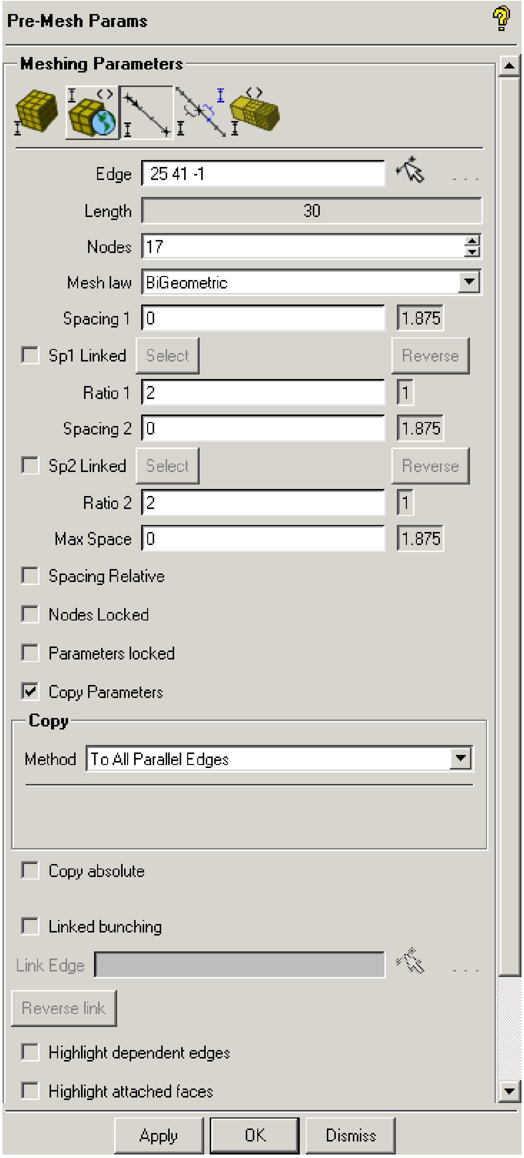
Similarly select two faces by  FACES as shown in Figure 3.370 press Apply and will get the O-Grid shown in the right of figure Figure 3.370 and Press Apply.

Figure 3.370
Before creation of O-grid (Left) & after creation of O-grid (Right)



Select Blocking> Pre-mesh Params > Edge params. A window will appear. Select Edge 25-41, give Nodes as 17. Toggle ON Copy Parameters and select "To All Parallel Edges" as shown in Figure 3.371.and press Apply.

Figure 3.371
Edge meshing parameters window



Note: Make sure Copy Parameters and Method to All Edges is enabled

Similarly Click 'Select Edge'  Select new edge and select Edge 42-41, give Nodes as 9. Click Apply to accept the changes.

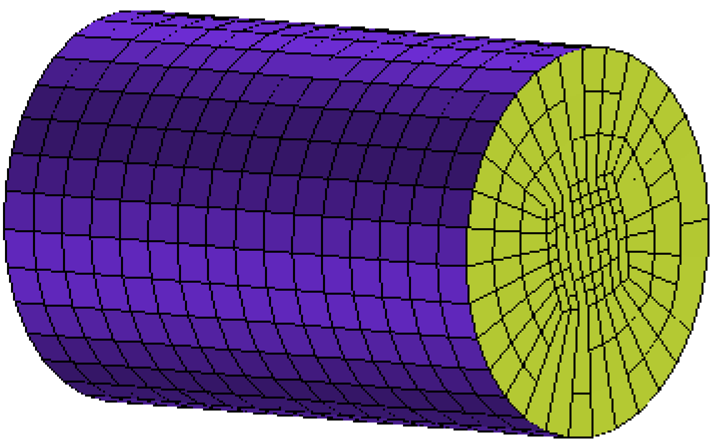
Repeat the procedure for Edge 37-41, give Nodes as 9

For or Edge 37-72, give Num points 4. Press Apply to accept the changes followed by Dismiss to close the window.

Turn on Pre-mesh in the Display Tree. Press 'Yes' to recompute the mesh.

Turn on the Pre- Mesh > Solid. The display will resemble Figure 3.372. The user might have to switch off the Vertices, Edges and Curves to reduce clutter on the screen.

Figure 3.372 Hexa Mesh in Cylinder 3



File > Save blocking will save the Blocking File

File >Mesh > Load from Blocking.

In the Mesh Exist window press Merge as shown in Figure 3.373.

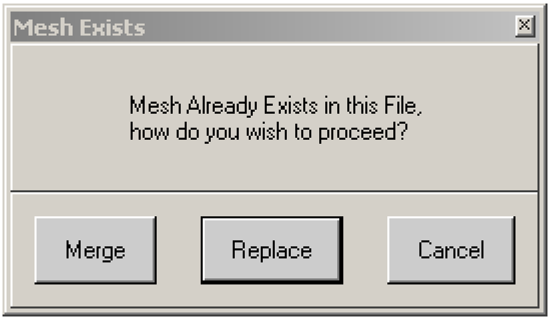


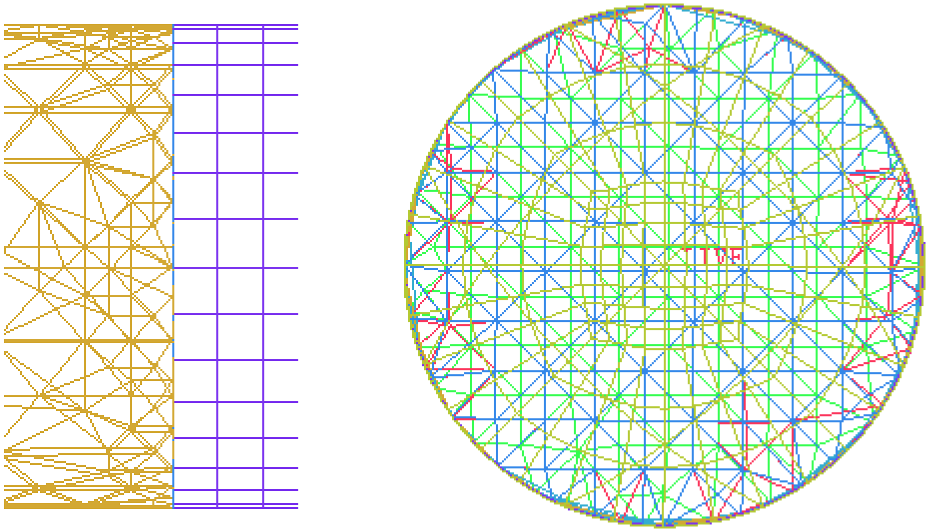
Figure 3.373
Window Asking for Merging New Mesh

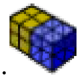
Switch Off Blocking in the Display Tree.

g) Merging the Resultant Mesh with Hexa Mesh at Interface2

Before merging the surface mesh at the INTERFACE2 will look as shown in Figure 3.374. The user might have to switch off all the families except INTERFACE2, CYL2 and CYL3.

Figure 3.374
Hexa Mesh before Merging



Select Edit mesh > Merge Node  >Merge meshes. 
A selection window will appear as shown in Figure 3.363.

In the Merge Surface Part Mesh select INTERFACE2 Press Accept.

Press Apply in the Merge Meshes window

After merging the surface mesh at the INTERFACE2 will look as shown in Figure 3.375. And pyramid at INTERFACE2 will be as shown in Figure 3.376. You can see the pyramids by switching on pyramids with LIVE family switched ON.

Figure 3.375
Hexa Mesh after Merging

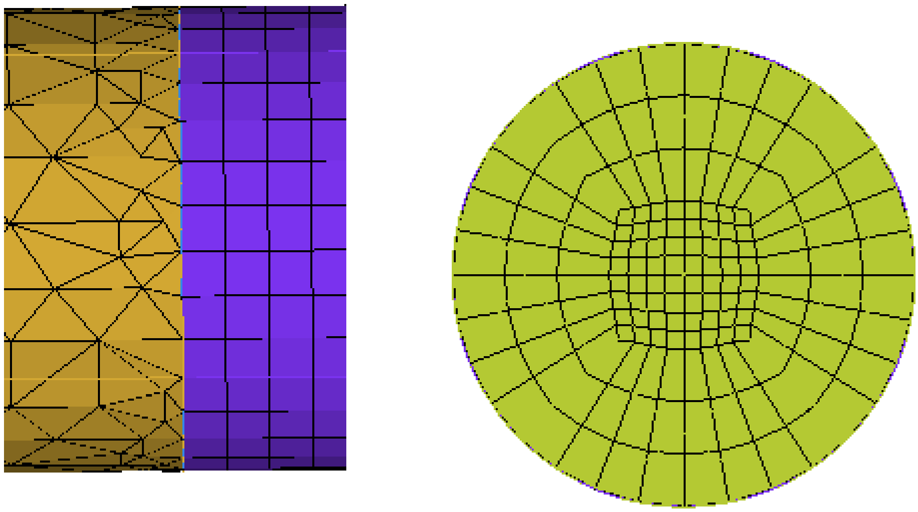
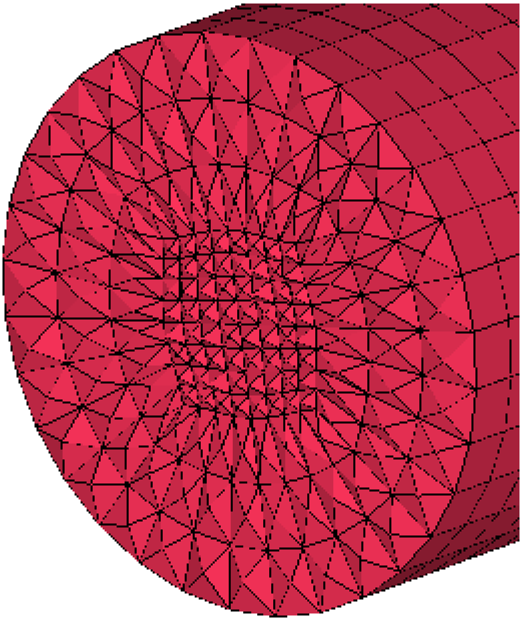


Figure 3.376
Pyramid at INTERFACE2



Smoothing the Hybrid Mesh

Select Edit mesh > Smooth Mesh globally  to start the smoother interface.

A Smooth elements window will appear. The quality of the hybrid mesh before smoothing is shown in Figure 3.377.

Modify the display of the histogram to have a Height of 20 elements. Click on Replot to replot the Histogram. To improve the quality of hybrid mesh, change the Number of smoothing iterations to 10. Assign Up to quality value to 0.4. Press Apply.

Figure 3.377
Quality before Smoothing

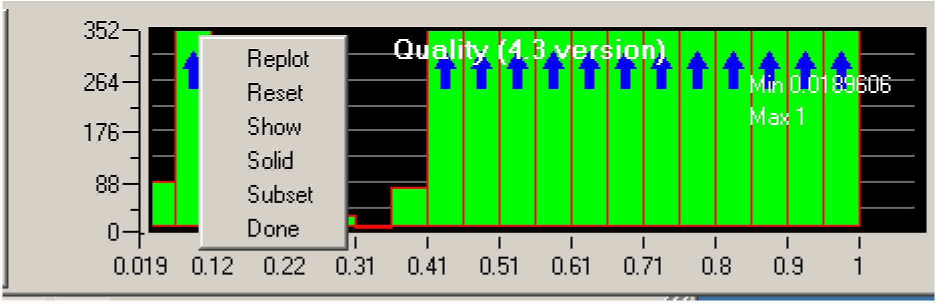

352
264
176
88
0
Replot
Reset
Show
Solid
Subset
Done
Quality (4.3 version)
Min 0.0189606
Max 1
0.019
0.12
0.22
0.31
0.41
0.51
0.61
0.71
0.8
0.9
1

Figure 3.378
Smooth globally window

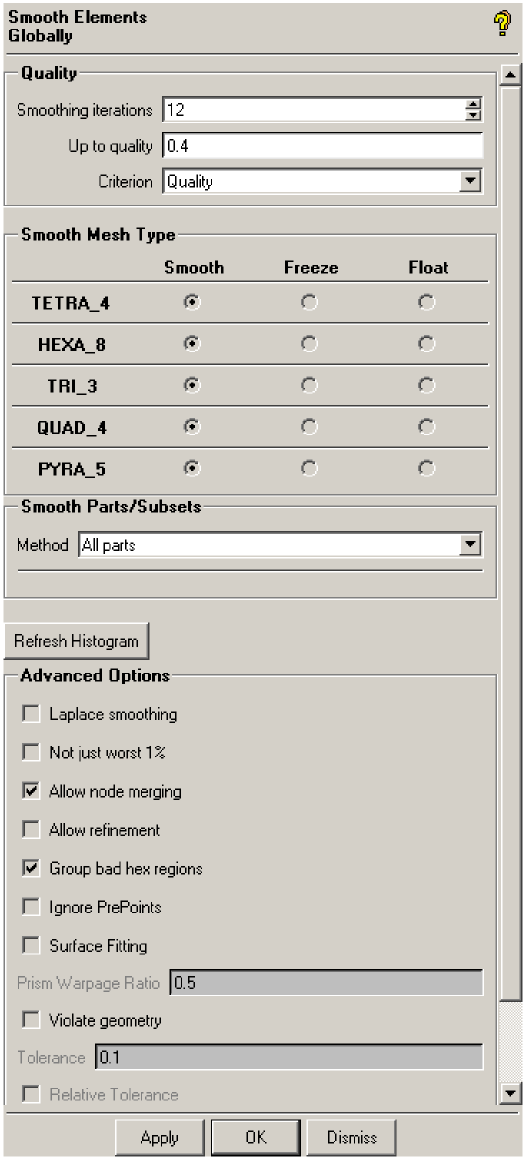
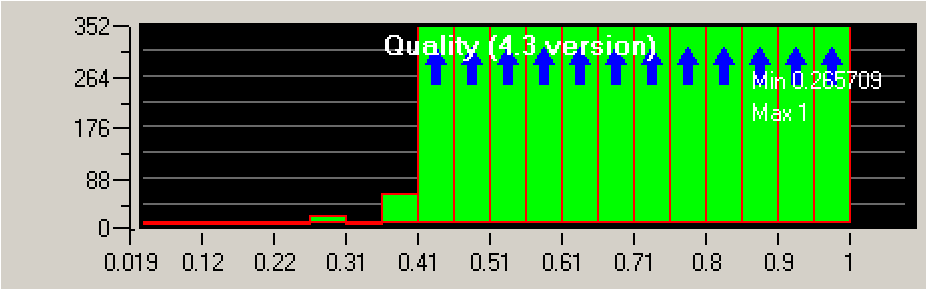


Figure 3.379
Quality after Smoothing



The quality of the hybrid mesh after smoothing is shown in Figure 3.379. Select 'Done' to quit the smooth histogram window.

h) Saving the Project

Select File >Mesh >Save Mesh as. Input file name as merge_domain.uns. Press Accept. It may ask what to do about the disconnected vertices. Say Yes.

Select File > CloseProject.

3.6.3: Tetra mesh for Submarine

Overview

In this example, the objective is to generate a tetra mesh for a submarine by providing partial geometry and partial surface mesh from other sources. The configuration consists of half of a submarine, including a SUBMARINE, FIN (Surface mesh) and TAIL (Surface mesh), are all cut in half by the symmetry plane as shown in Figure 3.380. A cylindrical water channel, that extends a few body lengths upstream and downstream, contains the entire geometry as shown in Figure 3.381.

Figure 3.380
Surface parts and surface mesh of the regions composing the submarine

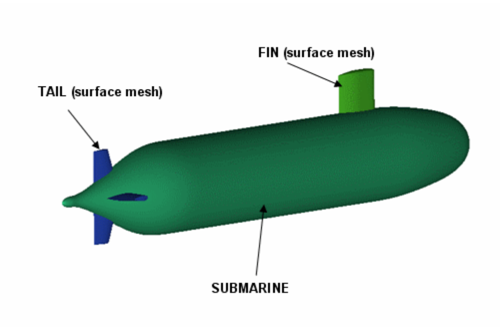
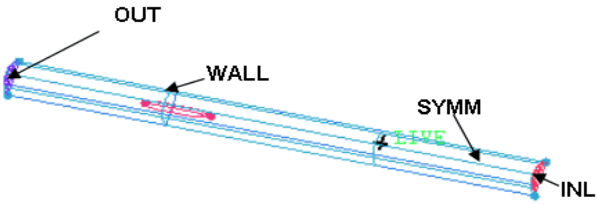


Figure 3.381
Surface parts of the region surrounding the submarine



a) Summary of steps

Starting the project

Assigning the mesh sizes

Create density box in the wake of submarine

Generating tetrahedral mesh from partial surface mesh

Diagnostics

Smoothing the mesh

Saving the project

b) Starting the Projects

From UNIX or DOS window, start ANSYS ICEMCFD. File > Change working directory $ICEM_ACN/../docu/CFDHelp/CFD_Tutorial_Files > submarine project. Choose its geometry file (geometry.tin) and domain surface_mesh.uns.

c) Setting Global mesh size

Choose Mesh > Set Global Mesh Size .

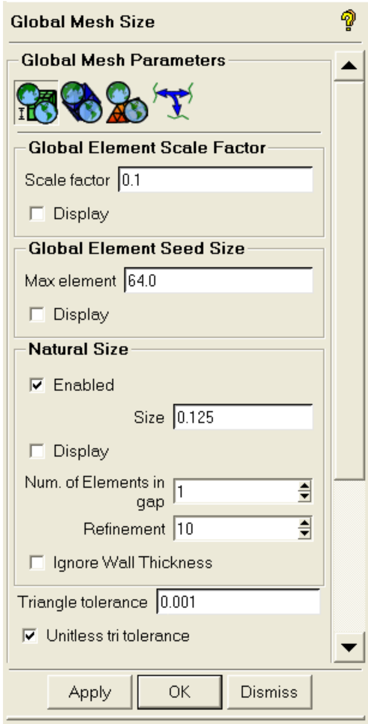


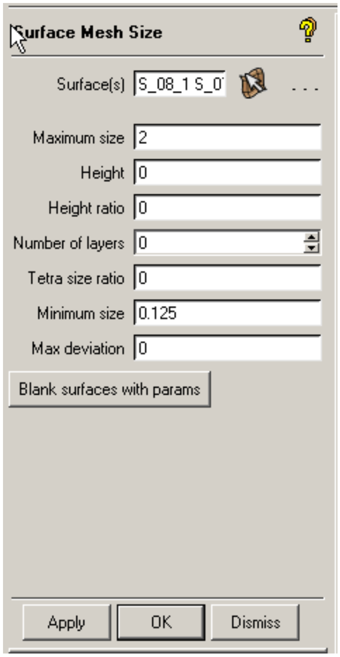
Figure.3.382 Global Mesh Parameters window

In the Global mesh size window, enter a scale factor of 0.25, a Maximum size of 64, Natural size of 0.125, Natural size > Refinement of 10, and Tri tolerance of 0.001 as shown Figure.3.382. Leave the other parameters at their default settings. Press Apply followed by Dismiss.

d) Setting surface mesh size

Choose Mesh > Set Surface Mesh Size  to set the meshing size parameters on the surfaces of the model. Select all surfaces in the model by "a" on the key board, and enter the Maximum element Size of 8. Next, repeat the step and from the selection filter, click on by parts icon and select the part SUBMARINE. The user can make other parts invisible from Display Tree if it's too much clutter on the screen. In the Surface Mesh Size window, enter a Maximum element size of 2 for part SUBMARINE as shown in Figure 3.383 and press Apply.

Figure 3.383 Surface Mesh Parameter



e) Setting curve mesh size

Enter Maximum element size of 0 on all the curves through Mesh > Set Curve Mesh size, Press "a" to select all the curves from the screen. Press Dismiss to close the window.

f) Creating Mesh density

To create density, select Mesh > Create density. However, before using this tool, we need to create another point behind the submarine geometry. Starting from the back point of the submarine on the axis of symmetry, create a point 5 unit further downstream in +X direction. In Create Density box, Figure 3.384, enter size as 1 (a scale factor multiplier), ratio as 1.2 and width as 15. Choose option Points, and Press to pick the two axis points mentioned above.

Figure 3.384
Create density box

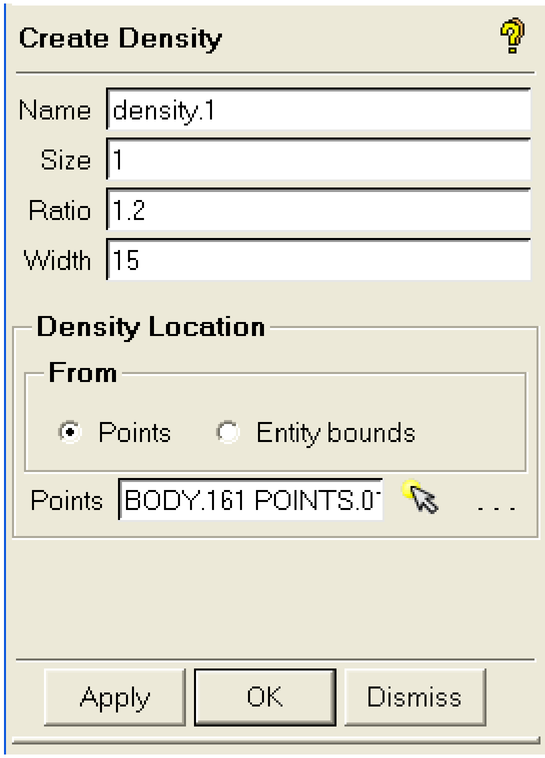
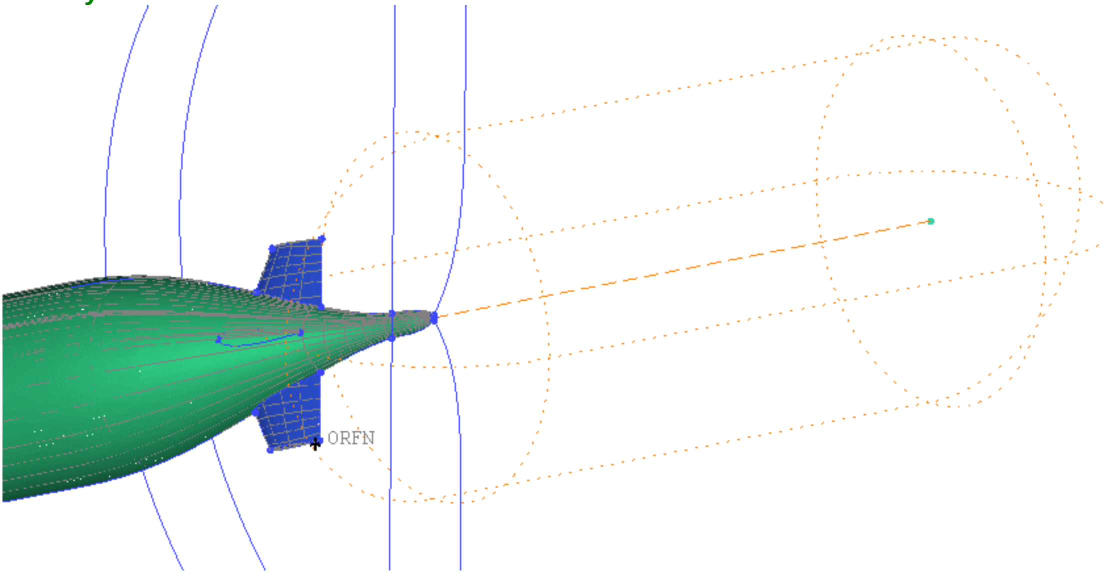


Figure Figure 3.385, shows the density box after creation.

Figure 3.385
Density box in the wake of submarine



Select File > Save project to save the changes made to the model before proceeding further.

g) Generating Tetra Mesh from partial surface mesh

Tetra sizes for the parts FIN and TAIL will be taken from the existing surface mesh by default.

Press Mesh > Volume Meshing  > From Geometry and Surface mesh  to start generating the mesh.

In Mesh with tetrahedral window Figure 3.386, Press on Use existing mesh, a new window select subpart will open. Select the parts FIN and TAIL for existing surface mesh as shown in Figure 3.387. Press Apply to generate the tetrahedral mesh.

Figure 3.386
Mesh with tetrahedral window

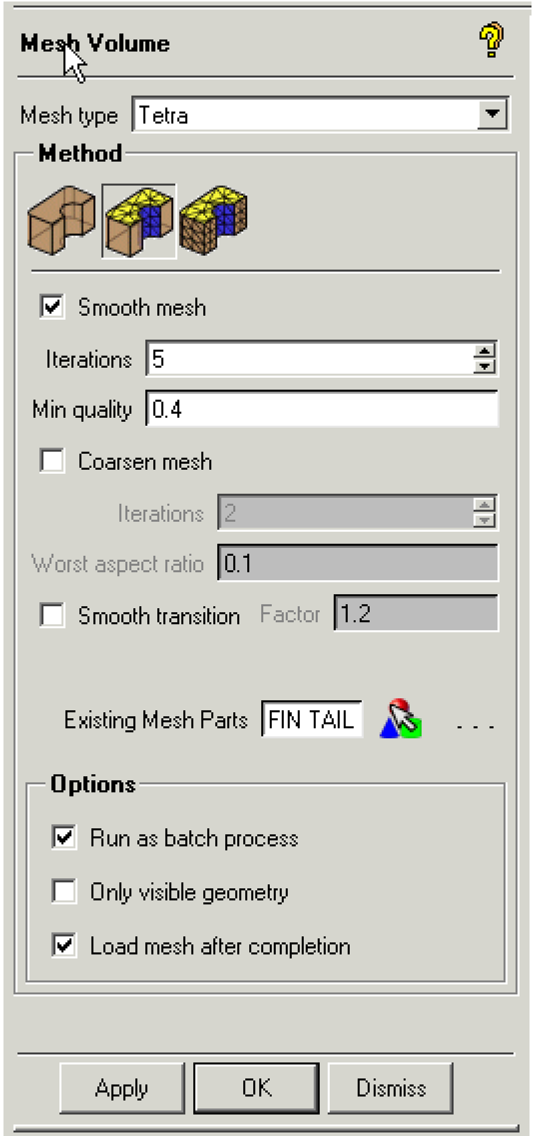
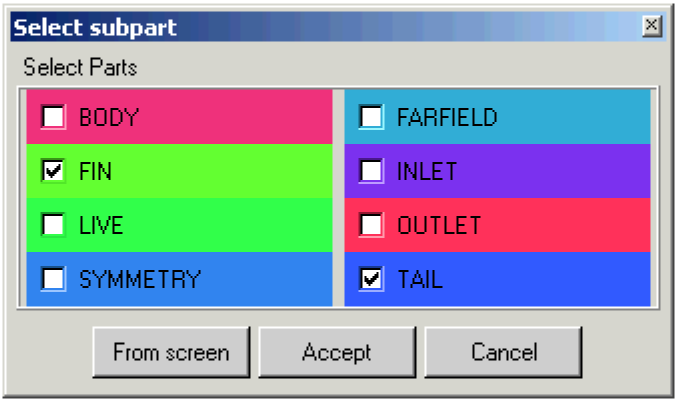
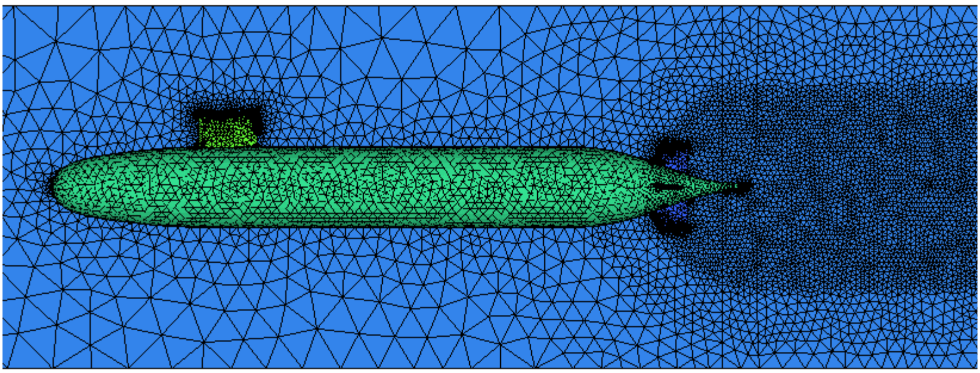


Figure 3.387
Select subpart for existing surface mesh



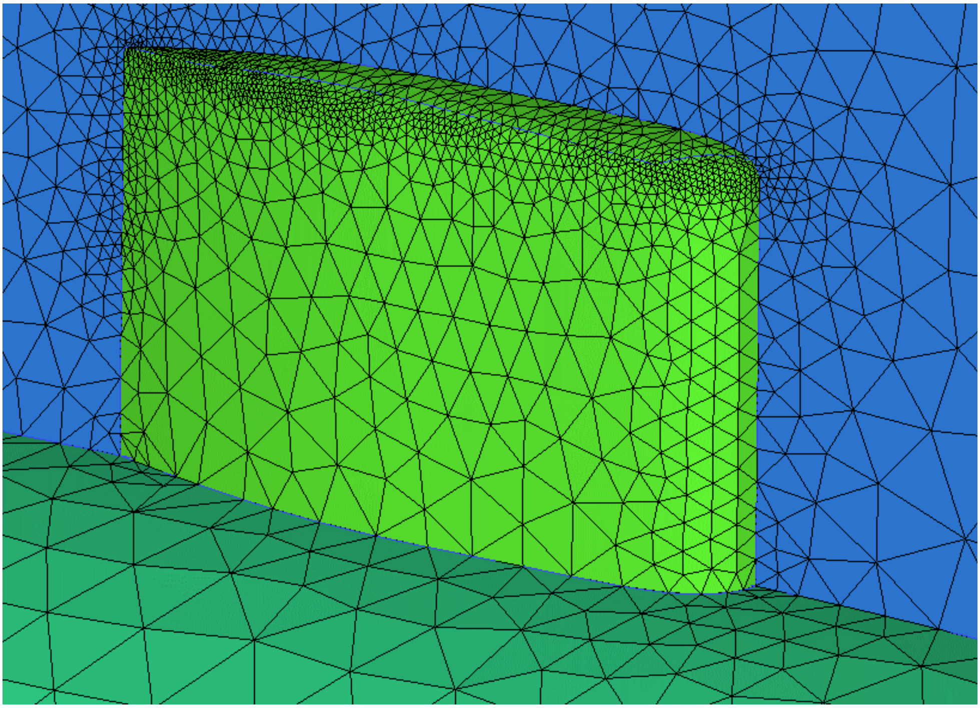
When the tetra process has finished, the complete tetra mesh should be visually examined as in Figure 3.388.

Figure 3.388
Complete Tetra Mesh on symmetry plane



The tetra mesh for submarine with symmetry plane is shown in Figure 3.389.

Figure 3.389
Mesh in the Fin area



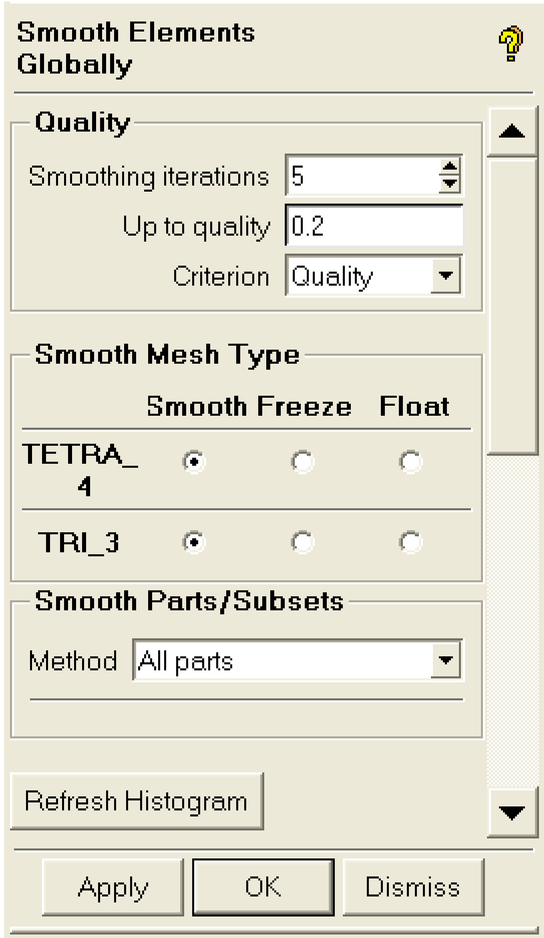
h) Diagnostics

As with the tetra tutorials, the user will need to go through all of the checks for Errors and Possible problems. Select Edit Mesh > Check Mesh to ensure that the mesh does not contain any flaws that would cause problems for analysis.

i) Smoothing

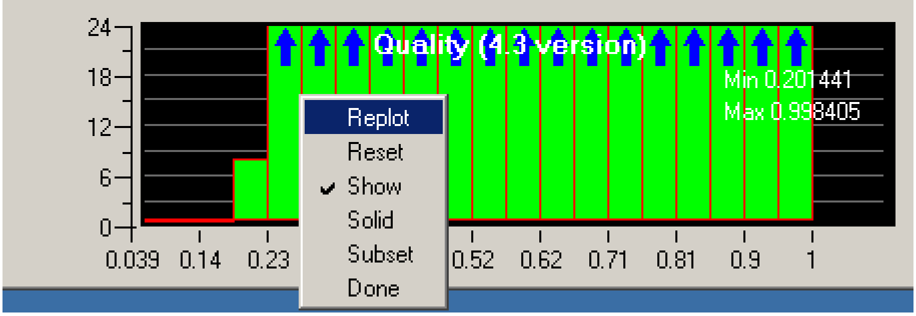
After the generation of tetra mesh smoothing was done automatically. After eliminating errors/possible problems from a tetra grid, the user should re-examine grid quality, and if necessary, smooth the grid to improve the quality. To do this, select Edit Mesh > Smooth Mesh globally . Set the Smoothing iterations to 5 and the Up to quality to 0.4. When all of the parameters have been modified as in Figure 3.390, select Apply. The smoother histogram is shown in Figure 3.391.

Figure 3.390 Smooth mesh globally window



Modify the display of the histogram to have a Height of 20 elements. Right mouse click on any of the histogram bar and press Replot to update the histogram.

Figure 3.391
The smooth elements histogram window



When the quality of the mesh is at an acceptable level, press Dismiss to close the Smooth elements histogram window.

j) Saving the project

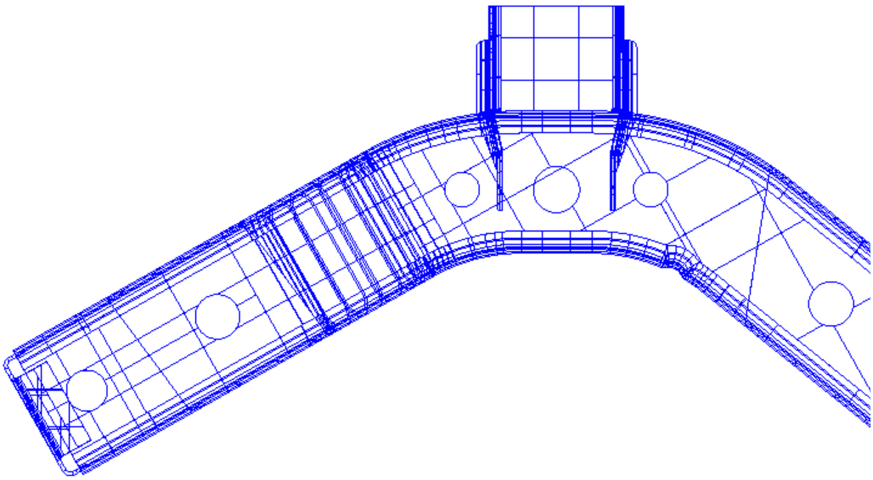
Save the mesh by selecting File > Save project. If a question box pops up to delete disconnected vertices, respond by saying Yes. Then close the project by selecting File > Close Project

3.6.4: Quad Mesh on a Frame

Overview

In this example, the user will generate a Quad Surface mesh on a frame. The frame, shown below in Figure 3.392 , represents a part fabricated from a sheet metal having a thickness of about 2.5 units. In order to set up this geometry for surface meshing, the mid surface utility will be used to get rid of the thin parts.

Figure 3.392 Quad Frame



a) Summary of steps

Starting the project

Determining the mid surface

Repairing the geometry

Defining sizes on curves

Meshing on surface

Checking the quality

Saving the project

b) Starting the Project

From UNIX or DOS window, start ANSYS ICEMCFD.File > Change working directory $ICEM_ACN/../docu/CFDHelp/CFD_Tutorial_Files>Frame project. Choose its Tetin file geometry.tin.

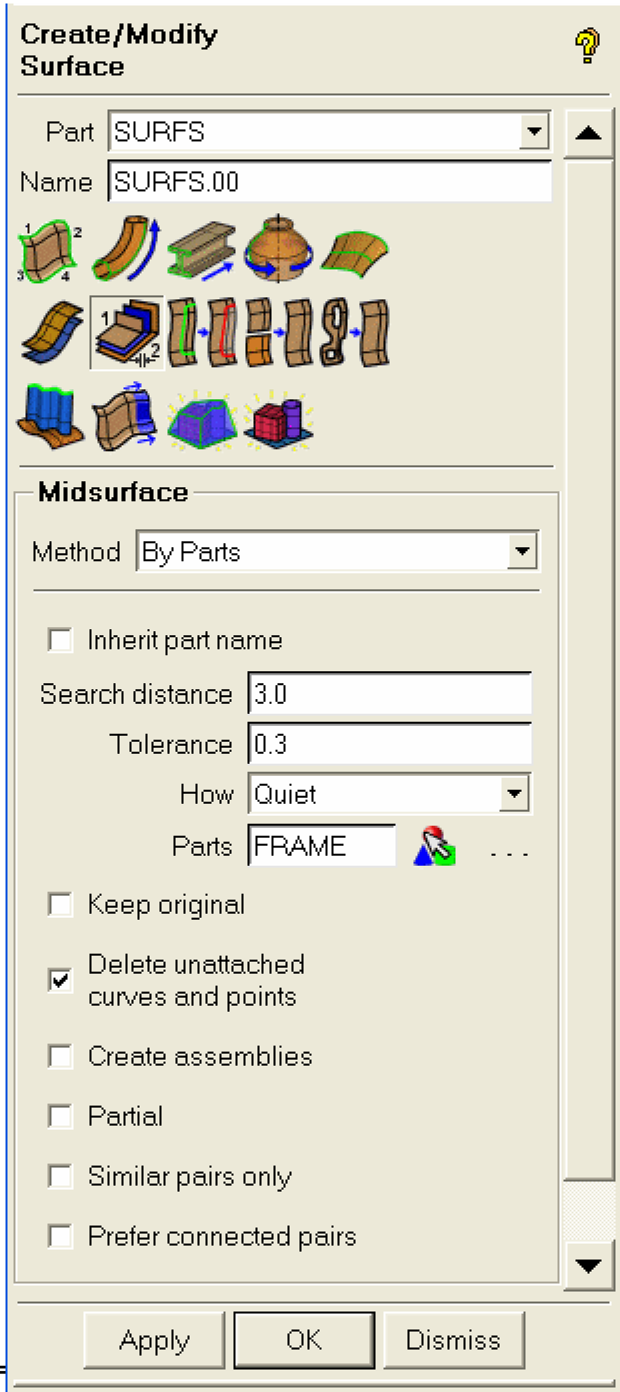
C) Determining mid surface

The user should verify the thickness by View > Distance or distance icon by clicking on two sides spanning the thickness of the geometry. The part will be modeled by the quad shell elements that are computed on the mid surface of the geometry. However, mid surface creation depends on the gap between the two spanning surfaces. The user can safely take a value of **3** to determine the mid surface.

- Display Surfaces from the model tree. The user may type "**h**" from the keyboard to get the home view.

- Geometry >Create/Modify Surface **.** This will open up a window as shown in Figure 3.393. Press Mid Surface. In the panel, Uncheck **Inherit part name**, put the **Search distance** as **3** and accept the default tolerance for subsequent Build Topology. Change **Part** name for mid surfaces to ‘SURFS’ and in the **Name** window enter SURFS.0. Next, select **Quiet** in the **How** window, press on Parts picker to select the all parts by hotkey ‘**a**’.

Figure 3.393
Mid Surface

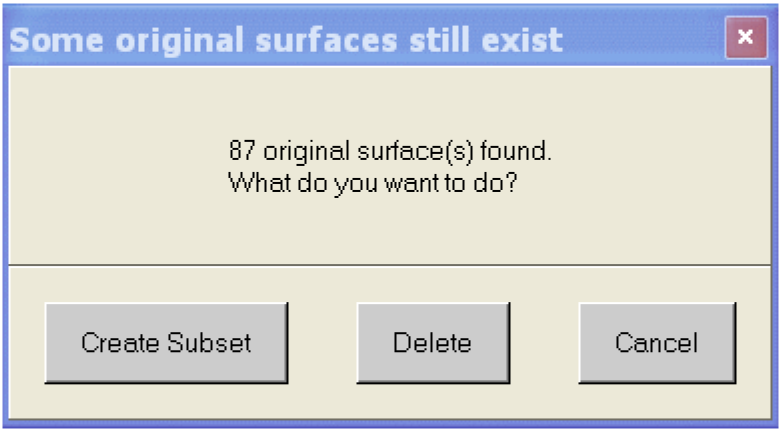


Click **Apply**.

Note: Once the operation is done, all the new surfaces will be in FRAME and the corresponding original surfaces will automatically be deleted from the family SURF. However, SURF family is still not empty.and it needs to be deleted.

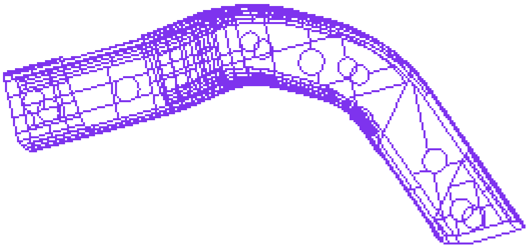
When prompted by the delete window, Figure 3.394, press Delete to delete all the original entities. This operation will also remove the original part name.

Figure 3.394 Delete Original surfaces



The geometry would now look as shown in Figure 3.395 after making all the parts visible. Right mouse click on Curves and select **Color by count**. The yellow and red curves are indicative of the Build Topology that mid surfacing operation did.

Figure 3.395 Geometry After Mid Surfacing



d) Repairing Geometry

The user can notice several yellow color curves in the model. Those represent the free edges of the surfaces that are not connected to a neighboring surface. Since this is an FEA model, it is Ok to have open boundaries. Note that the user can reduce the display clutter by switching **OFF** Show Unattached, Show Double and Show Multiple options for the Curves by right clicking on Curves in the Display Tree.
Now the circular holes in the frame can be removed

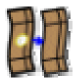
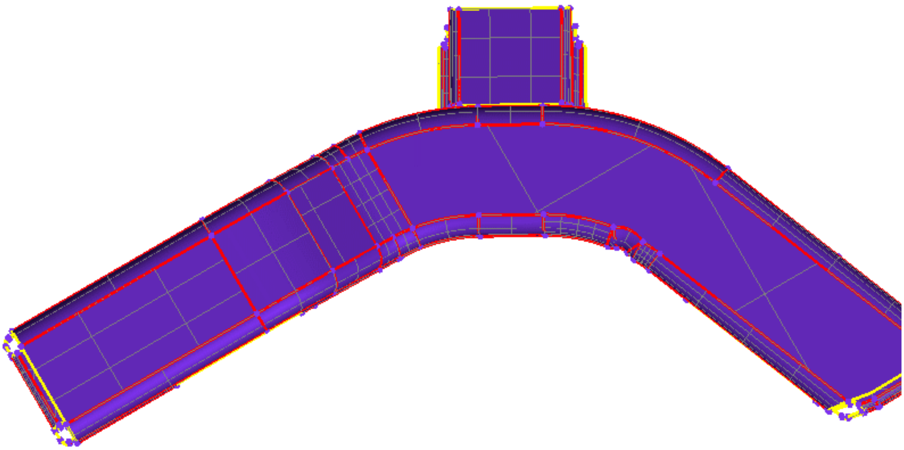
Select Geometry > Repair geometry > Remove Holes  .Pick all the circular holes curves, one by one, and press middle mouse key, Figure 3.396.

Figure 3.396
Remove holes in Repair geometry

Figure 3.397
Display after removing holes

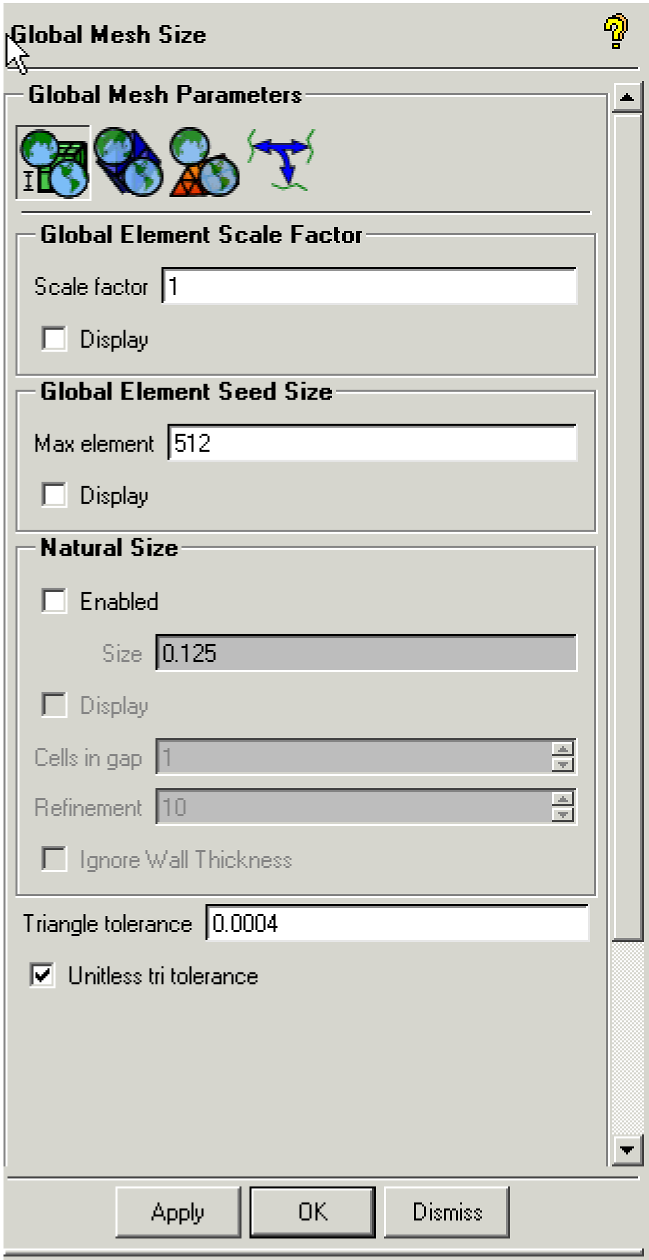


e) Defining sizes on curves

The user should now make all the curves visible by switching ON Unattached, Double and Multiple options for the Curves by right clicking on curves from the model tree.
Options like Color by count and Show Wide could be switched **OFF** from the model tree by right clicking on Curves from the model tree to reduce the clutter.

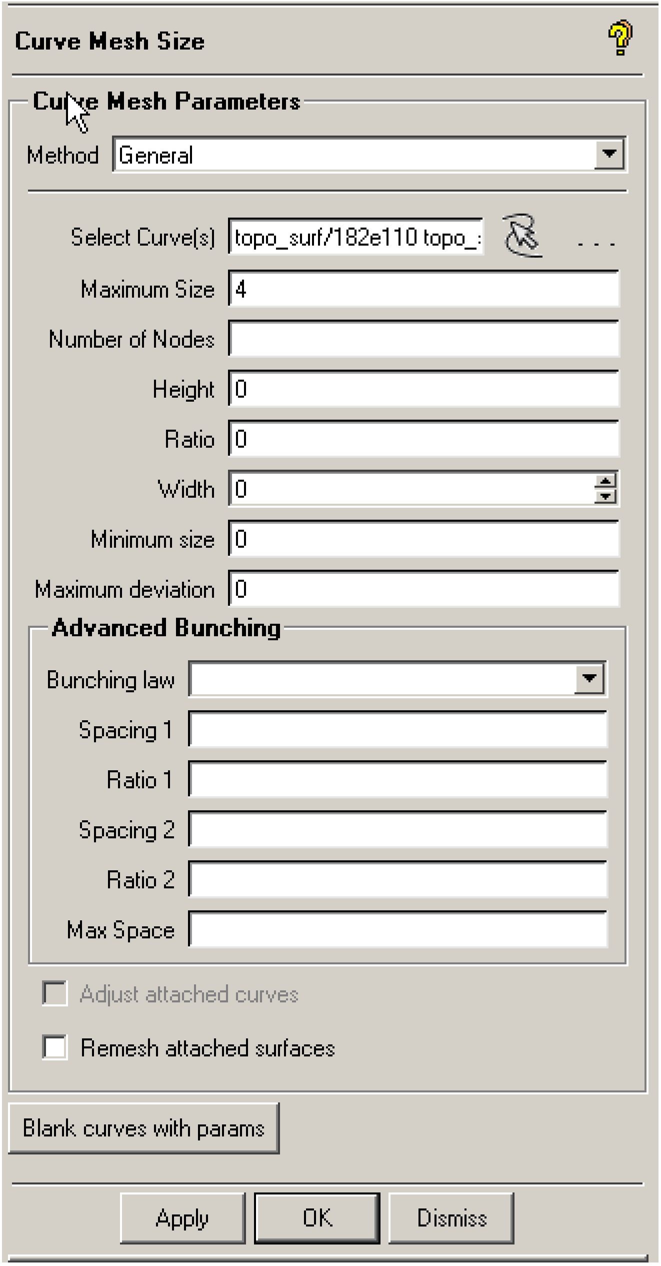
The user must define mesh sizes before mesh generation. Under Mesh > Set Global mesh size  > Global mesh parameters , Set **Scale factor** to **1**and **Max Element** to **512** as Figure 3.398. Press Apply followed by Dismiss to close the window.

Figure 3.398 Global mesh size window



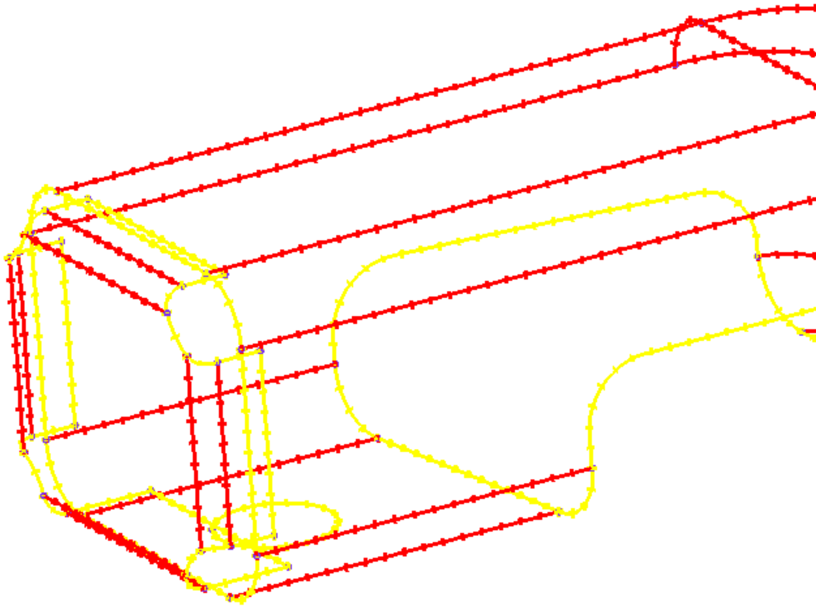
- Select Mesh > Set Curve mesh size , this will open a window as shown in Figure 3.399. Click on picker and select all the curves with the hot key ‘a’. For **Maximum element size,** enter **4** and press Apply. Press Dismiss to close the window.

Figure 3.399 Curve mesh size window



Switch **ON** Quad sizes by right clicking Curves >Curve Node Spacing from the Display tree. This will display the nodes on all the curves in the model as shown in **Figure 3.400.**

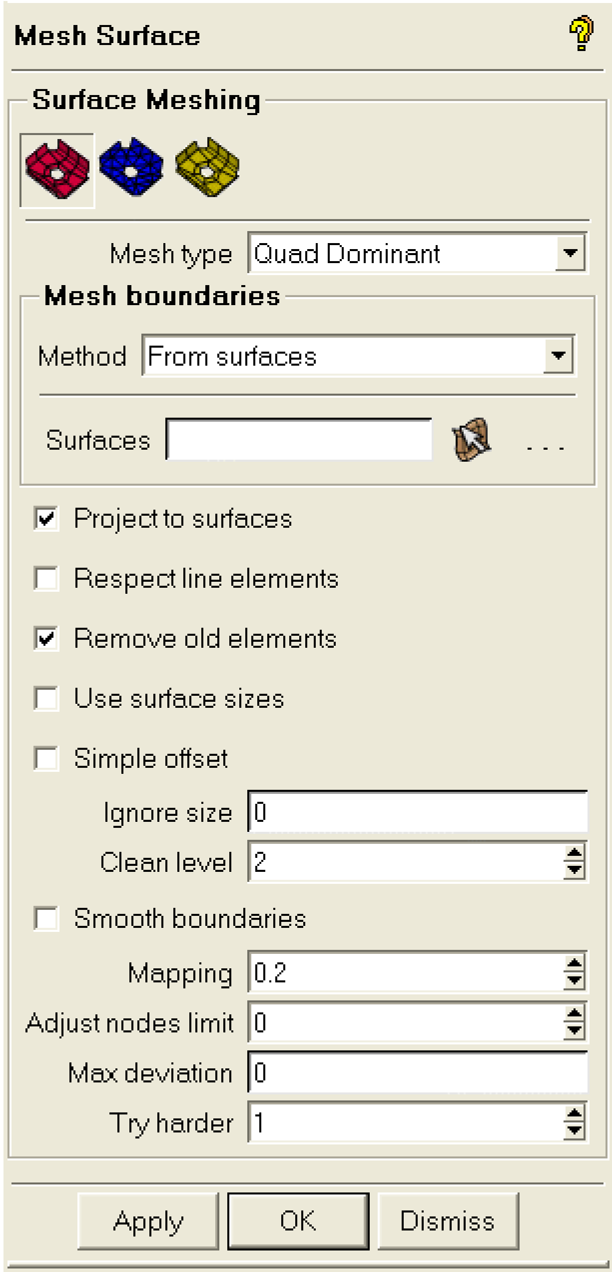
Figure 3.400 Quad sizes on the curves



f) Surface Meshing

Mesh > Shell Mesh  > Patch Based  : This would open up a window like in Figure 3.401.

Figure 3.401
Surface mesher window

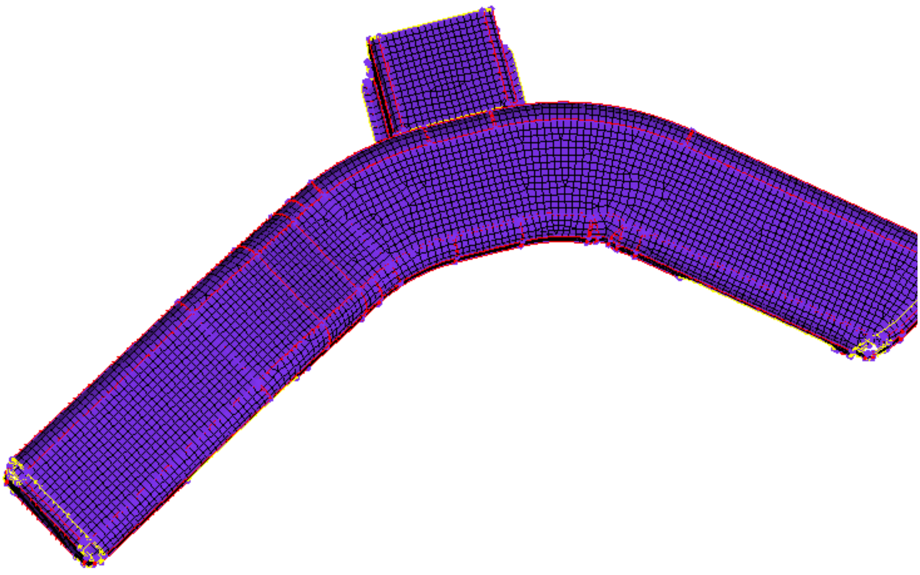


Select Quad Dominant in the **Mesh Type** option, increase the **Clean Level** to **2,** and switch **ON** the **Project to surfaces** & **Respect Bar Element.** Finalize by pressing Apply.

This would create a mesh as shown in **Figure 3.402**.

Note: User can see the mesh by Mesh>Shell>Solid & Wire in theDisplay tree

Figure 3.402
The final mesh

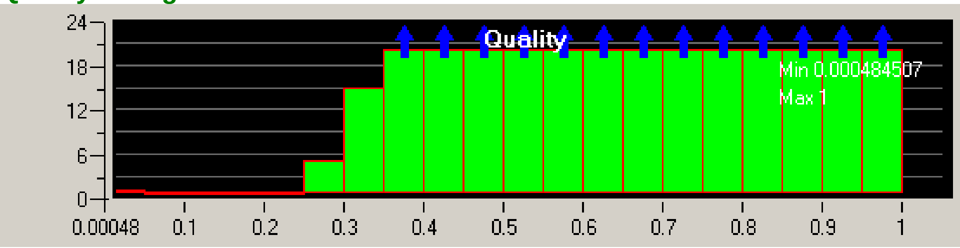


g) Checking quality

The user should always check for the quality of the existing mesh.

Go to Edit mesh > Display Mesh Quality, in the Criterion window select **Quality**, the window pops up as shown in Figure 3.403. The minimum quality stands at 0.00048 and therefore the mesh must be smoothed.

Figure 3.403
Quality histograms window



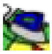
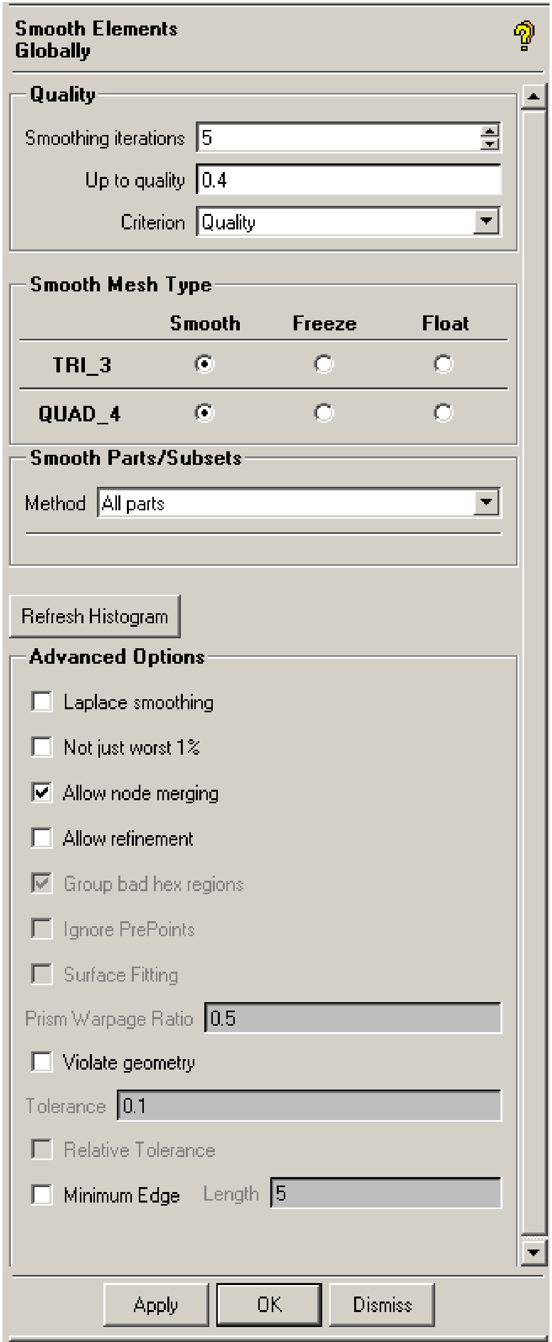
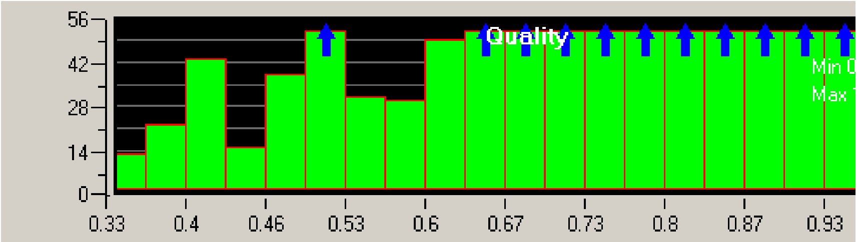
- Go to Edit Mesh>Press Smooth Mesh Globally  and in the **Smooth Mesh Globally** window in the **Up to quality** option change the value to ‘0.4’ as shown in Figure 3.404 and press Apply. Keep the default setting as it is)

Figure 3.404
Smooth Element Globally Window



- After smoothening is performed right click in Histogram window and select Reset, the final histogram is shown in Figure 3.405.

Figure 3.405 Smooth Mesh Globally



h) Saving the project

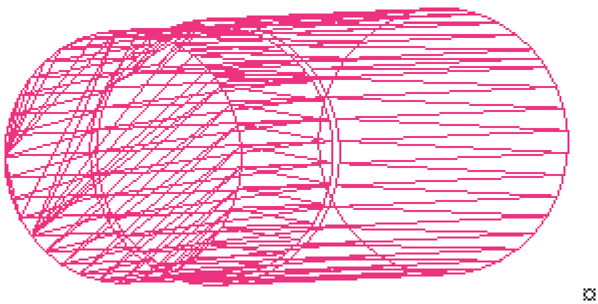
For saving the project select File > Save Project. Accept the default file names when asked.

3.6.5: STL Repair with Tetra meshing

Overview

In this tutorial, the user will generate the tetra mesh in a pipe configuration. The pipe has different problems in the geometry which might cause leakage (holes) in the Tetra mesh. This example focuses on how to deal with leakage and corresponding geometry repair.

Figure 3.406
stl start



a) Summary of steps

Starting the project
Repairing the geometry
Saving the project

b) Starting the Project

Start ANSYS ICEMCFD. Select File > Change working directory and browse for $ICEM_ACN/../docu/CFDHelp/CFD_Tutorial_Files/STL_Repair. Load the tetin file, geometry.tin.

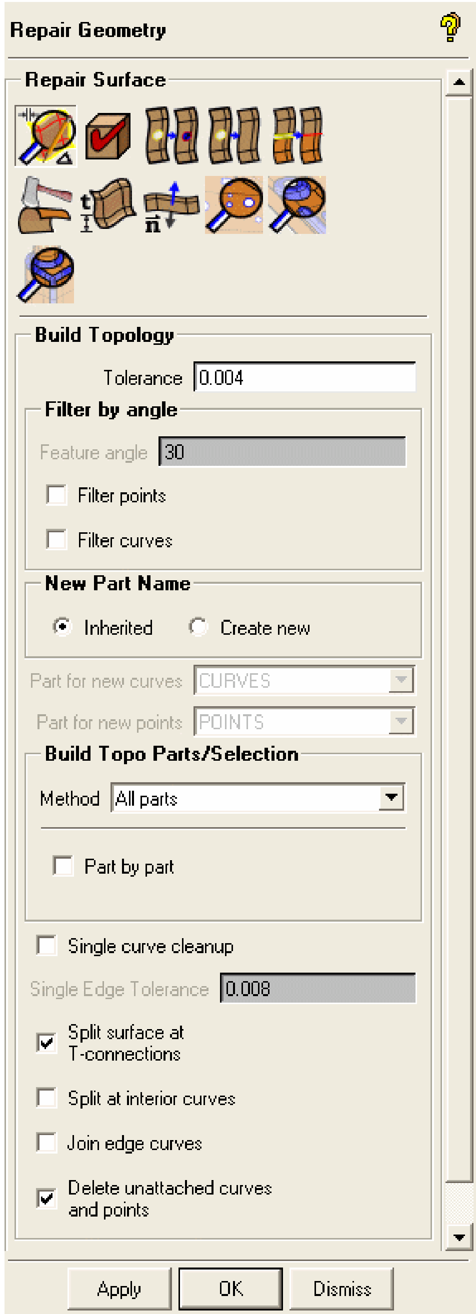
Right mouse select Geometry > Surfaces in the Display tree and select Show Full to see the full triangulation of the surfaces.

c) Repair Geometry

First run initial topology to find any possible problems with the geometry.

Select Geometry > Repair Geometry  > Build Diagnostic Topology . This will open up a window as shown in Figure 3.407.

Figure 3.407
Repair Geometry/Build Topology window



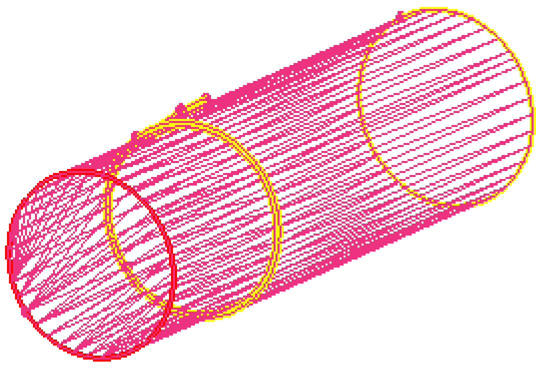
The more important settings are:

Tolerance – maximum gap distance between surface edges not considered to be a problem. Typically set to one order of magnitude smaller than smallest projected mesh size or geometry feature. A default is calculated based on a fraction of the model size.

Filter Curves/Points – If turned on, will remove or "filter out" curves and points of between surfaces and curves that meet at a smooth transition. The Feature angle defines a maximum angle between two surfaces (or curves) that would be considered "smooth." Any curves generated between surfaces whose angle is less than this value would be removed. Generally recommended only for creating a set of curves for meshing constraints, not geometry diagnostics.

Use all default settings, including Tolerance, and Apply. Note the curves as in Figure 3.408.

Figure 3.408
Geometry after Build topology



After building topology, the new curves are automatically turned on and options changed to Show Wide and Color by

Count. These options can be turned on or off by right mouse selecting Geometry > Curves. Color by Count will display curves in the following colors:

Red – Curve is shared by two surfaces. This is desired and would indicate clean, water-tight geometry.

Yellow – Curve is shared by only one surface. This indicates a gap or hole greater than the tolerance. Usually has to fix.

Blue – Curve is shared by three or more surfaces. Usually indicates a t-junction or a sliver surface that's thinner than the tolerance. Most likely okay but in some cases may cause potential problems.

Green – Free curves that are not logically associated to the surface. Usually curves that are imported or manually created. Build topology, by default will remove these curves. Can also be removed manually.

First close the hole for the little stick out portion as shown in Figure 3.409.

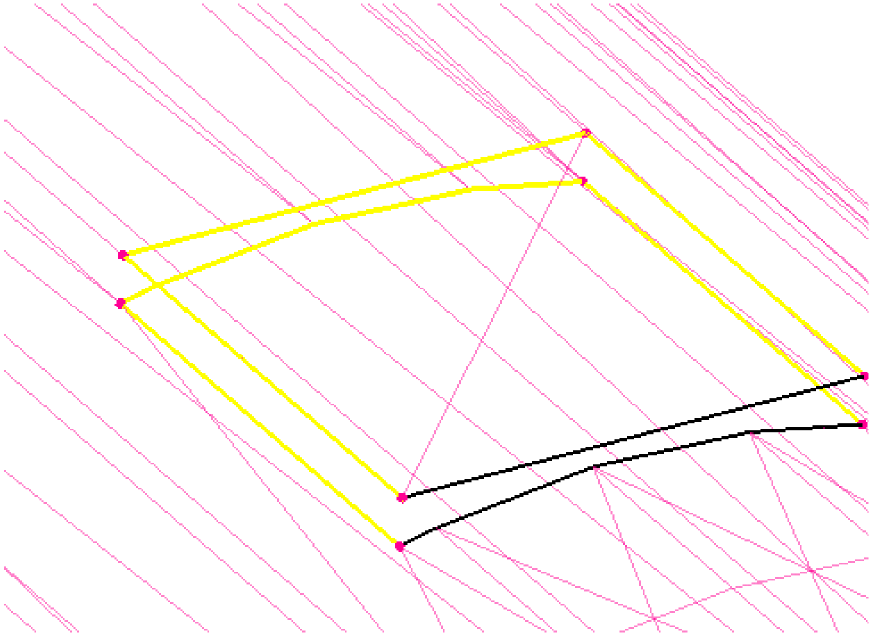


Figure 3.409 Square portion before repair

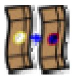
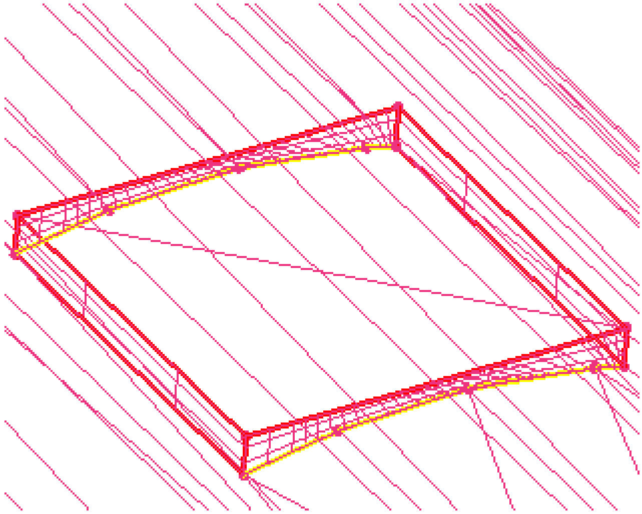
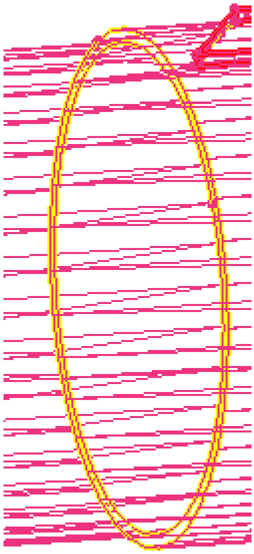
Select Repair Geometry > Close Holes . Select the curves, one pair at a time and press the middle mouse button. Repeat for all four pairs. Triangles will be created to fill in the gaps and the curves will be automatically updated to red as seen in Figure 3.410.

Figure 3.410
Square portion after repair



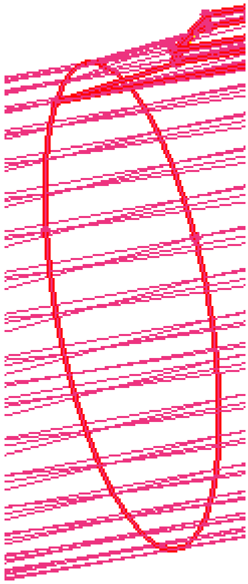
The user will now focus on the two concentric circles in the center (Figure 3.411). Perhaps this feature is small enough to ignore, so rather than fill in the gap, we'll match or stitch the edges.

Figure 3.411 Circular portion before repair



Select Repair Geometry > Stitch/Match Edges . Select the two concentric curves and press the middle mouse button or Apply. Note that the edges of the second curve will be moved to match the edges of the first selected curve. See. Figure 3.412 .

Figure 3.412
Circular portion after repair



The large hole at the end of the pipe will be fixed on the mesh level to follow.

d) Assigning Mesh Sizes.

Select Mesh > Set Global Mesh Size  > General Parameters  . This will open up the window as shown in Figure 3.413.

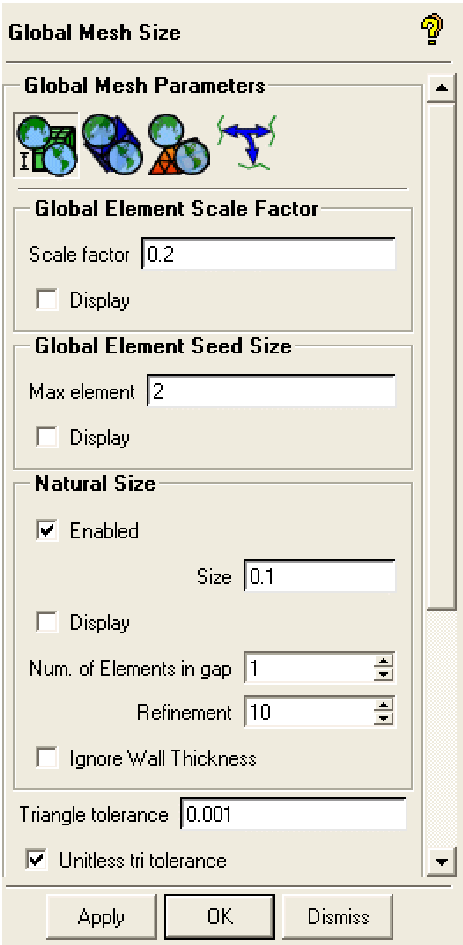


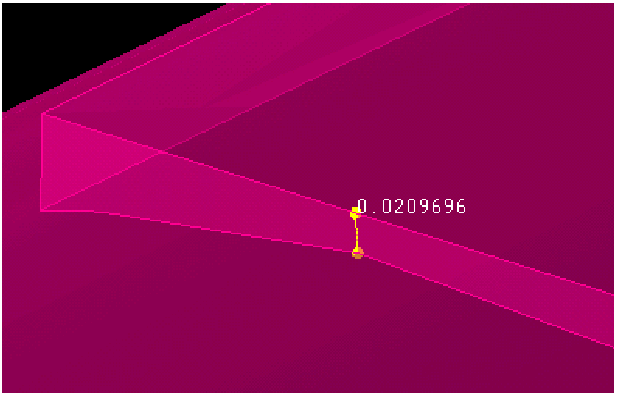
Figure 3.413 Global Mesh Sizes window

Set Scale factor to 0.2, Global Element Seed Size (Max element) to 2.

All surfaces and curves will take on the Global Element Seed Size (Max element) of 2 x 0.2 (scale factor) = 0.4.

Zoom in to the stick out square portion as shown in Figure 3.414. Select Measure distance  in the upper left hand Utility Menu and then select two locations along the lower and upper curves of the square stick out. Note the prescribed elements size is too large to capture this feature.

Figure 3.414
Tetra sizes on surfaces



Back to Mesh > Set Global Mesh Size  > General Parameters . Turn on Natural Size (check Enabled) and change the size to 0.1. This value is multiplied by the Scale factor whose product is the global minimum size. Thus 0.1 x 0.2 = 0.02 is the lower limit of subdivision. With Natural Size enabled, the tetra mesher will automatically subdivide to smaller elements in this area.

e) Defining Material point

Select Geometry > Create body > Material point .

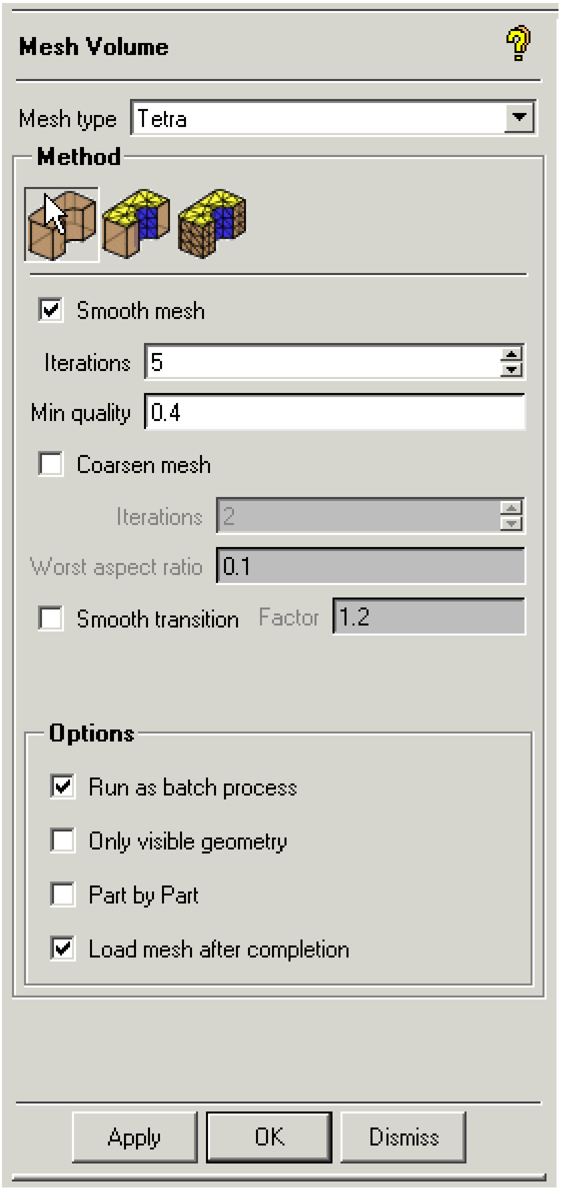
Enter the Part name as LIVE. Select again or Select location(s) and select two locations on the geometry such that the mid point is inside the pipe. Verify by turning on Geometry > Bodies in the Display tree. Rotate the model to ensure that LIVE lies inside the pipe.

Select File > Save Project.

f) Generating Tetra mesh

Select Mesh > Volume Meshing > From geometry The Mesh Volume panel will appear as in(Figure 3.415).

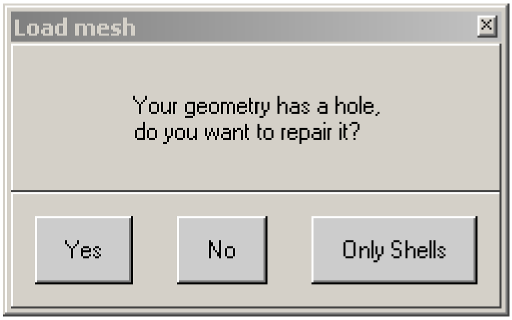
Figure 3.415 Mesh with Tetrahedral window



Make sure Mesh type > Tetra is set, leave all other options as is and press Apply to generate the tetra mesh.

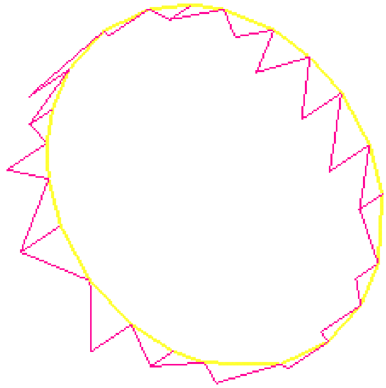
Due to the open end, a window will warn you of leakage (hole) as shown in Figure 3.416.

Figure 3.416 Leakage warning window



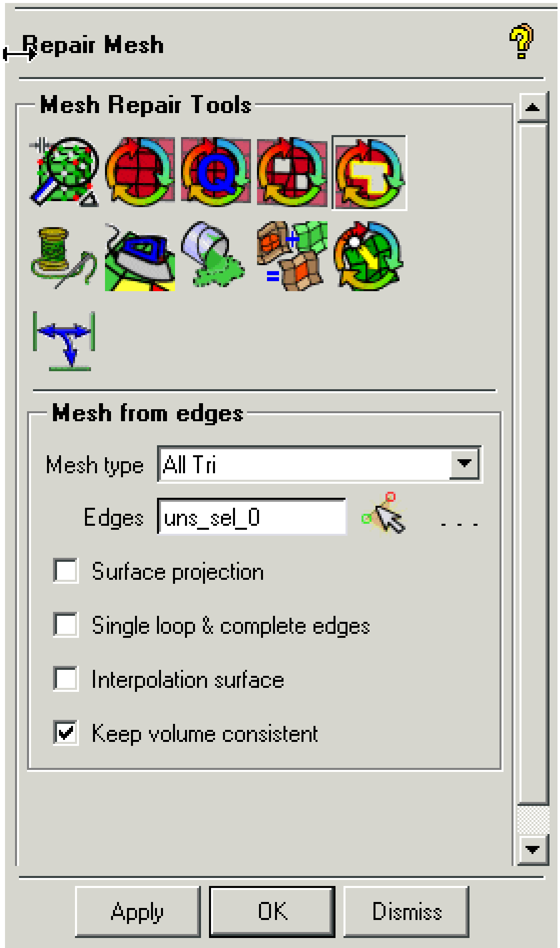
Select Yes to repair the mesh. This will display single, yellow edges forming the perimeter of the hole and their adjacent surface elements. These elements are automatically put in to a subset which is turned on in the Display tree: Mesh > Subset > Leak Location. Turn off all Geometry in the Display tree and view the hole as in Figure 3.417.

Figure 3.417
Leakage in display



Selecting Yes to repair will also bring up the mesh repair window shown in Figure 3.418. This panel contains several tools for automatic mesh repair, one being to fill or create surface mesh within a closed loop of single edges, Mesh from Edges  which is used in this case.

Figure 3.418 Surface mesh repair options window



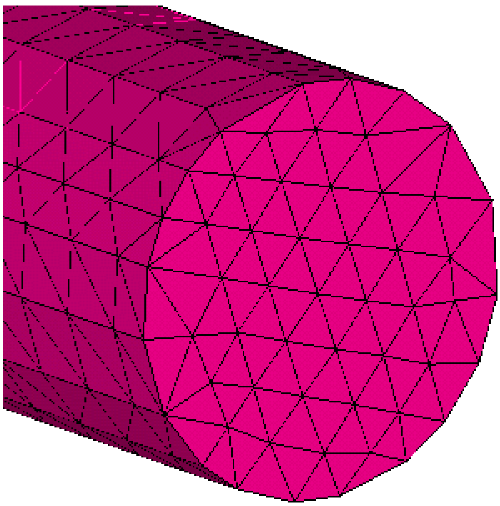
You will be immediately prompted to select edges – no need to select  from the Repair Mesh panel. Leave everything in the panel as default and drag a selection box (keeping the left mouse key depressed) around the displayed edges. Selection will be verified by display of nodes and black edges. Press the middle mouse button or Apply. The yellow edges will eventually disappear indicating the closing of the hole was successful.

Note: Notice that Keep volume consistent is selected. This will restructure the tetras so that they match up with the newly created surface mesh. This is recommended only if there is one hole as in this case. If more than one hole, Keep volume consistent should be turned off, Mesh from Edges should be done manually, one hole at a time, then select Flood Fill/Make Consistent  also from the **Repair Mesh** panel.

Flood fill is also part of the Make consistent process. After the tetras are fixed, Flood Fill is automatically run to determine which elements to retain (those inside the closed volume) and which to throw away. Scroll up in the **Message Window** and note the number of elements assigned to LIVE and those put into ORFN (default dead zone).

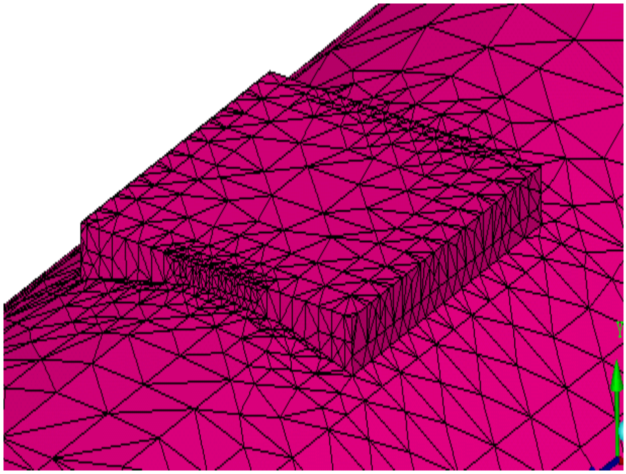
Turn off Mesh > Subsets in the Display tree and turn on Mesh > Shells. Right mouse select Shells and select Solid & Wire. View the corrected surface mesh as in Figure 3.419.

Figure 3.419 Mesh in circular region after repair



Also note the refined mesh in the square stick out portion as a result of Natural Size (Figure 3.420).

Figure 3.420
Final mesh detail



g) Final Steps

Smooth the mesh: Select Edit Mesh > Smooth Mesh Globally. Note the current bad quality in the Histogram Window. Use the defaults in the Smooth Elements Globally panel and Apply. Note the improvement in quality in the histogram.

Select File > Save project and Exit

3.6.6: Workbench Integration

This tutorial will give user the idea about parametric changes in the blocking with the geometry.

Figure 3.421 Blocking geometry

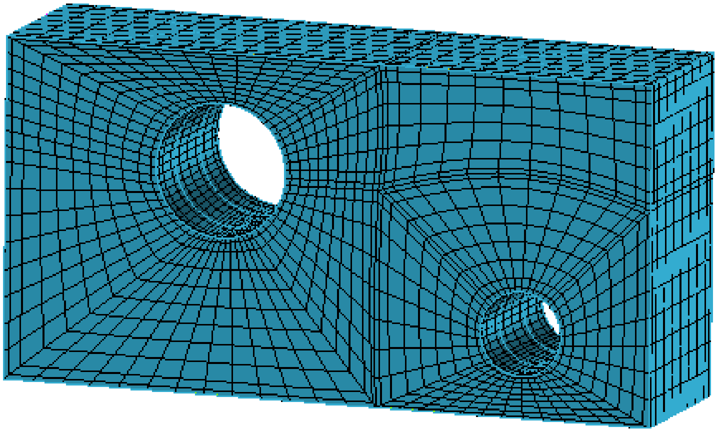
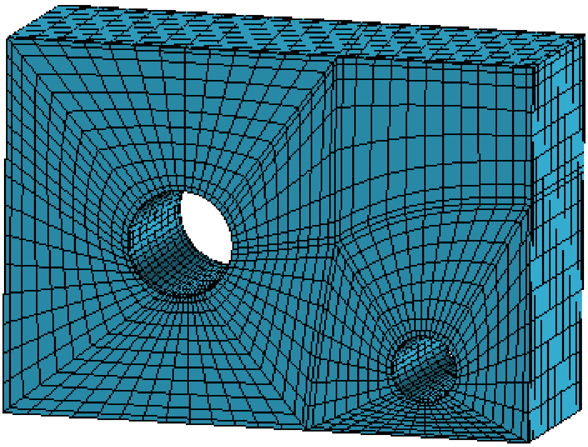


Figure 3.422 Blocking after modifying geometry



a) Summary of the steps

Loading geometry in DM

The Blocking strategy

Create composite curves

Splitting the blocking material

Associating edges to curves

Generating the blocking

Modifying the geometry

Updating the association

Saving the blocking

b) Starting the project

From the Windows, fire Ansys workbench integration. Then one window as shown in Figure3.423will launch. Select **Geoemtry.** This will open a DesignModeler graphics user interface (GUI) as shown in.Figure 3.424 Press **ok** for desired length unit window. This will keep units to the default SI unit system.

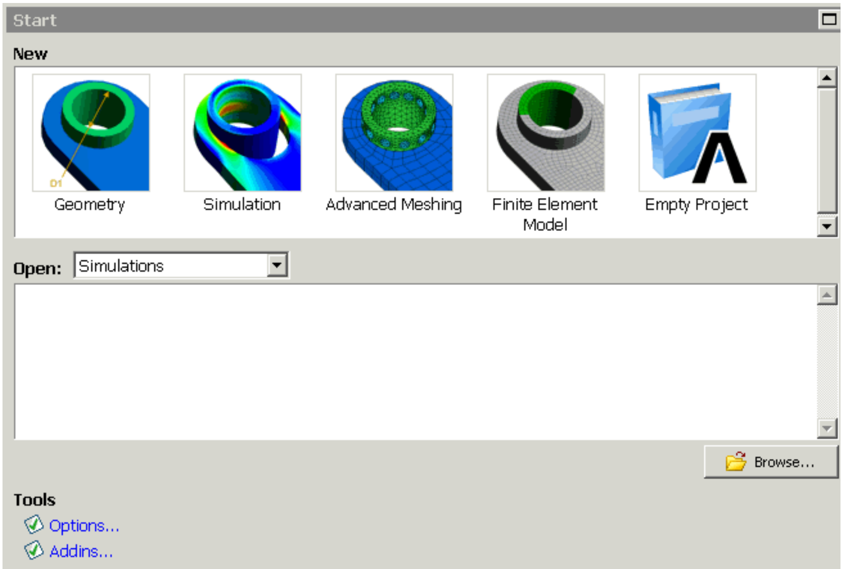


Figure3.423 Workbench main window

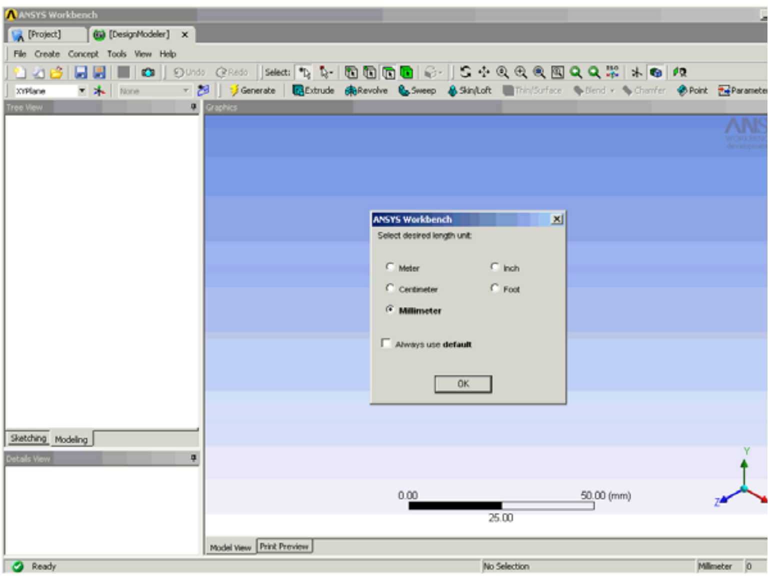
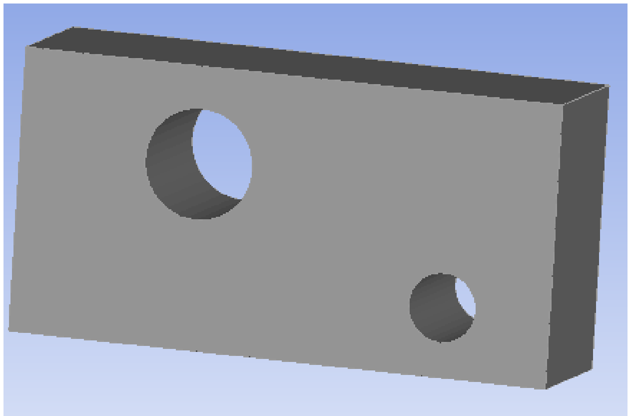


Figure 3.424 DesignModeler interface

c) Loading Geometry in DM

For loading geometry in the DM, go to **File > Open**. Select the Piping.agdb file from the desired location. This will show geometry in the GUI as shownFigure 3.425

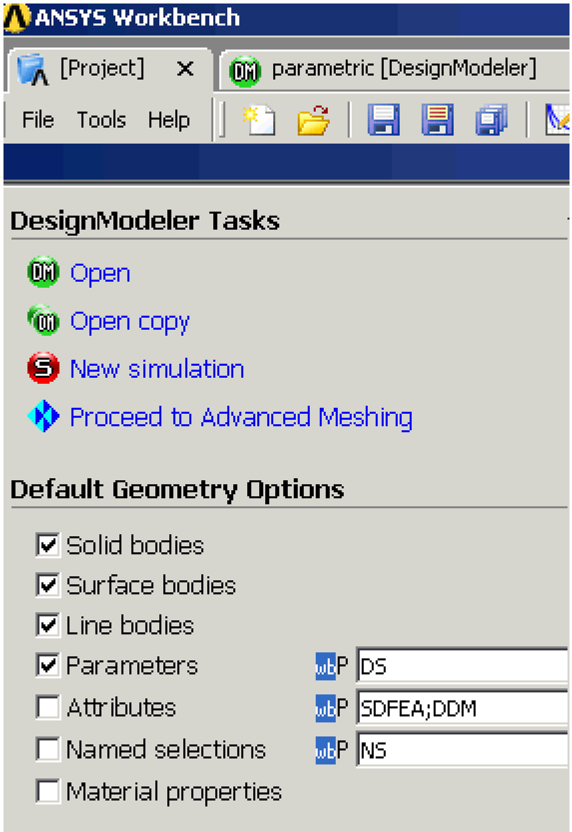
Figure 3.425
Loaded geometry in the workbench environment



d) Proceeding to the advance meshing

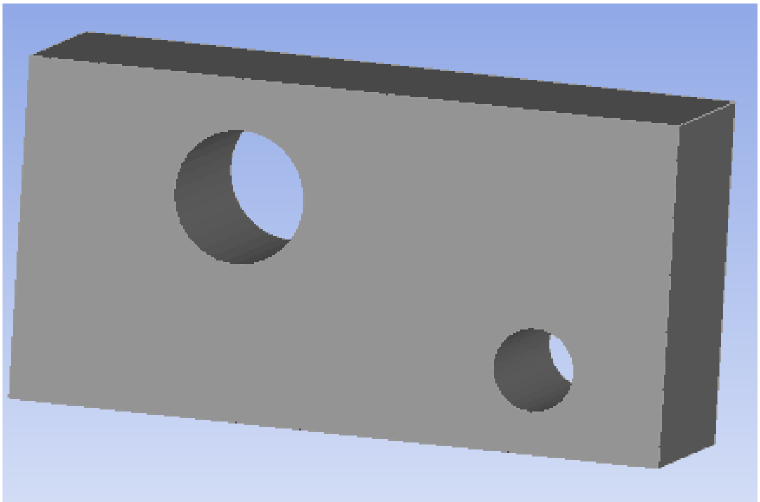
For creating the blocking, user has to go the advance meshing tab. Go to the Project window, select **Proceed to advance meshing** as shown Figure

Figure 3.426 Proceeding to advance meshing



Pressing Proceed to advance meshing will invoke the Advance meshing gui, Select File > Geometry >Update geometry > Merge geometry, it will open geometry in Advance meshing as shown in Figure 3.427

Figure 3.427 Geometry in the Advance meshing



e) Blocking

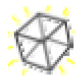
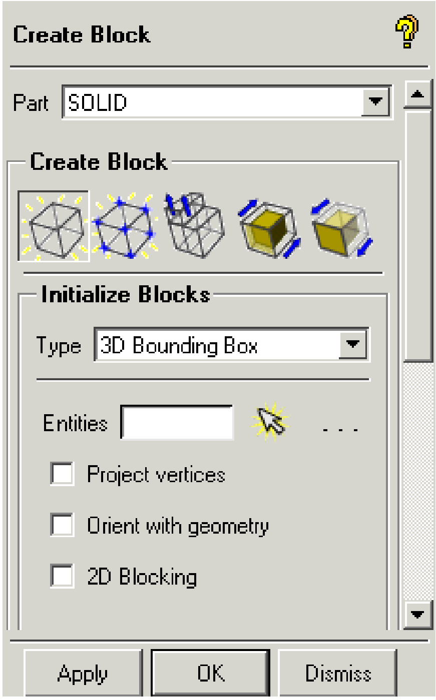
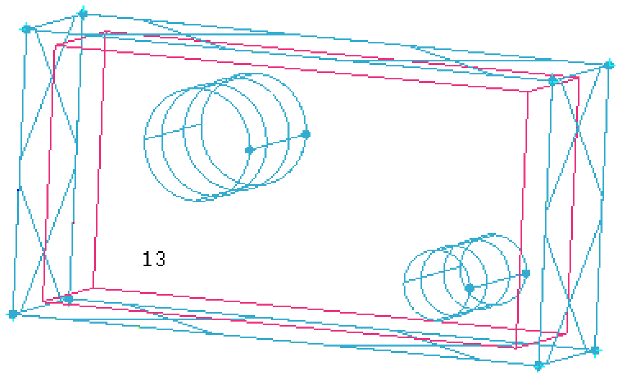
Select Blocking > Create Block > Initialize block , it will open the window. Select Initialize block and 3D for Type of the block as shown in Figure 3.428. Select all entities and press Apply to create blocking.

Figure 3.428
Create block window



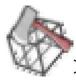
For vertices number, turn ON Blocking > Vertices > Number from Display Tree widget. After creating the block geometry will look like as shown in Figure 3.429.

Figure 3.429 Geometry after creating the block.



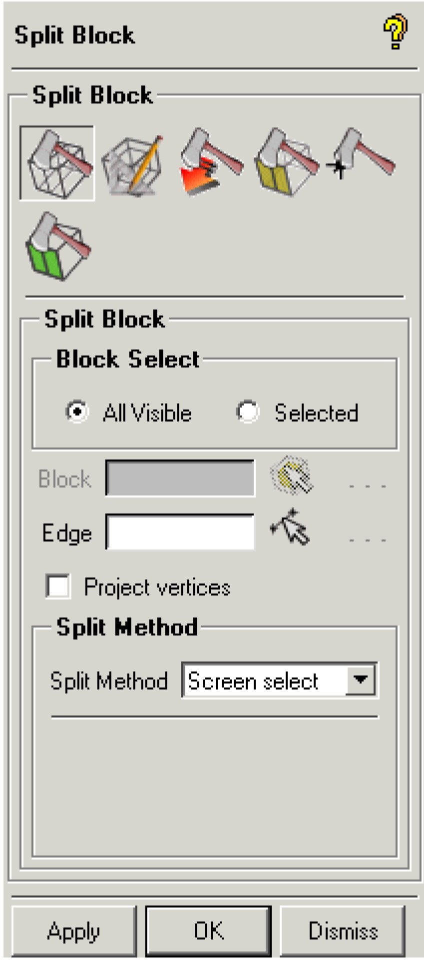
f) Split block

Now user will split the blook in i j and k direction in order to capture the shape of the geometry.

Select Blocking > Split Block  > Split Block , it will open the window as shown in .Figure 3.430

Turn ON the vertices number from Blocking > Vertices > Number from Display Tree widget.

Figure 3.430
Split block window



Select Split method as Screen select and select 26 – 42 and make a split at the location as shown in split it edge as shown in Figure 3.431.

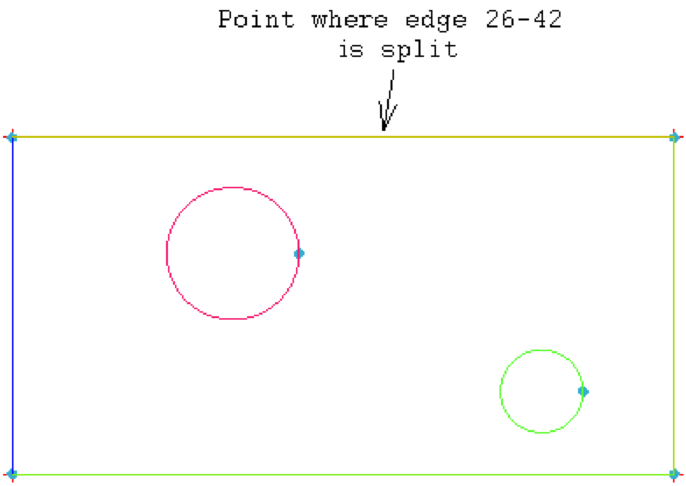


Figure 3.431 Locations where 26-42 edge to be splitted

After splitting the edge geometry will look like as shown in Figure 3.432

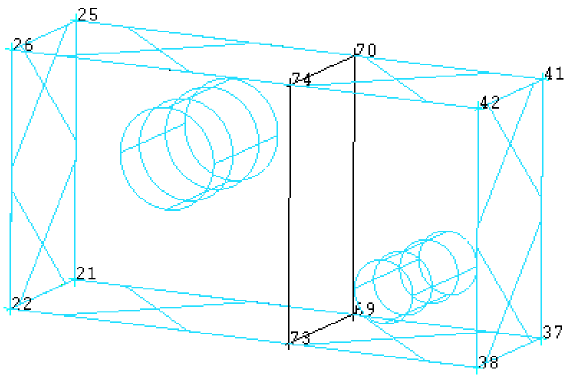


Figure 3.432: Geometry after splitting the 26-42 edges.

g) Creation of first O-grid

To capturing the first hole user will create O-grid and defines the corresponding block to Vorfn family.

Select Blocking > Split block  > O-grid , it will open the window as shown in Figure 3.433

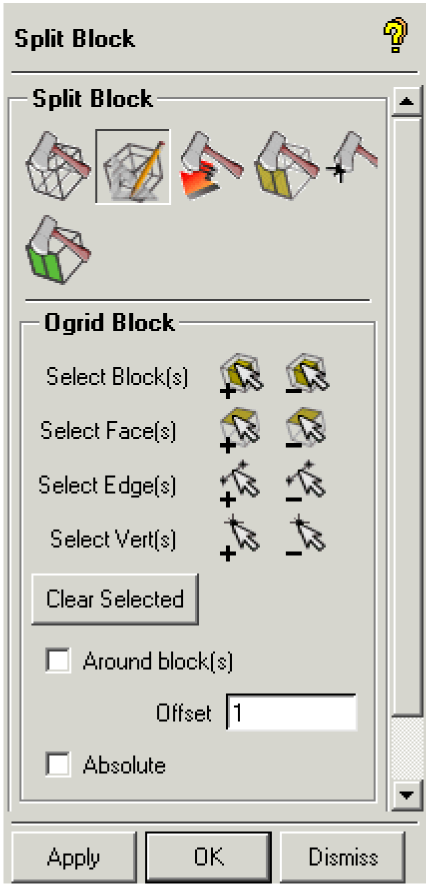


Figure 3.433
O-grid block window

Now select block 13 and it`s two corresponding faces as shown in Figure 3.434, after selection press Apply to create first O-grid. After creation of first O-grid blocking will look like as shown

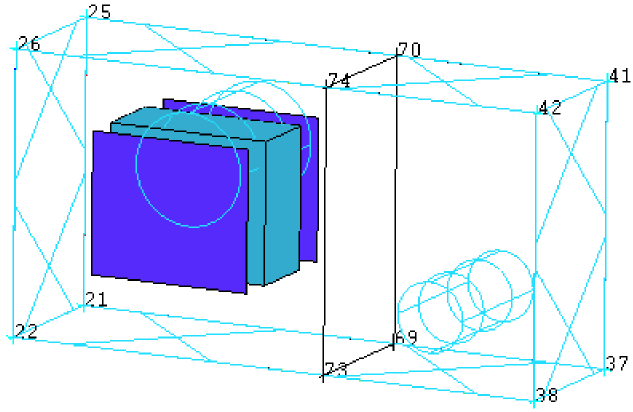


Figure 3.434 Block and faces selection for first O-grid selection.

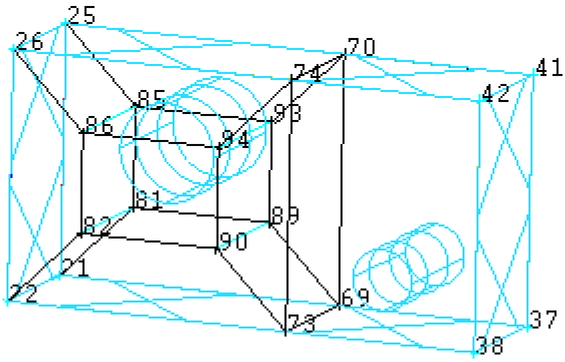


Figure 3.435 Blocking after first O-grid creation.

h) Second O-grid creation.

Now to capture second hole in geometry user will create another O-grid and corresponding block will defines it to Vorfn family.
Now before O-grid creation user will split the block

Select Blocking > Split block > Split block , it will open the window as shown in Figure 3.430. Now select edge 38-42 and split it at the location as shown in Figure 3.436.

Figure 3.436
Location where edge 38-42 to be splitted

is

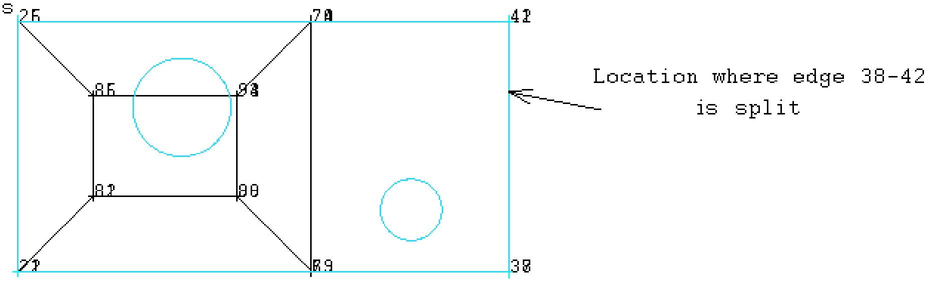
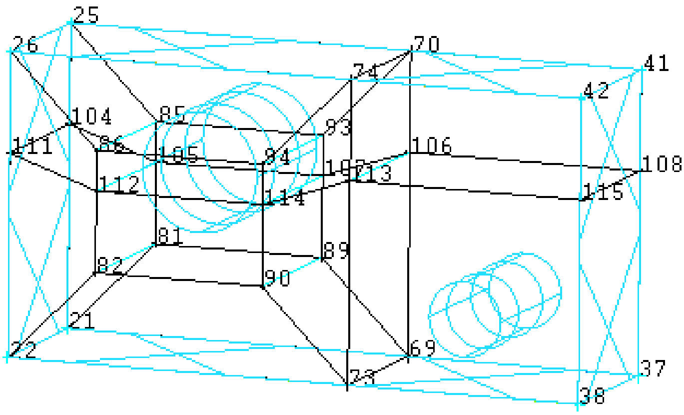


Figure 3.437 After splitting edge 38-42.



Select Blocking > Split block  > O-grid  , it will open the O-grid block window. Select the block and it's two corresponding faces as shown in Figure 3.438. After creation of second O-grid blocking will look like as shown in

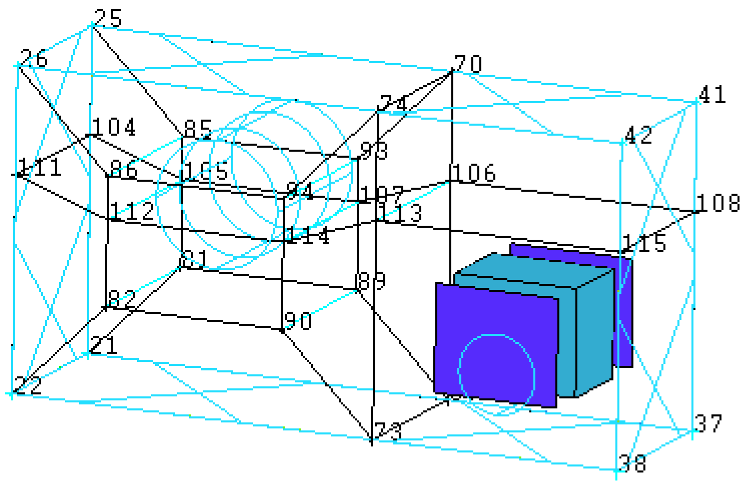


Figure 3.438 Selection of block and faces selection for second O-grid creation.

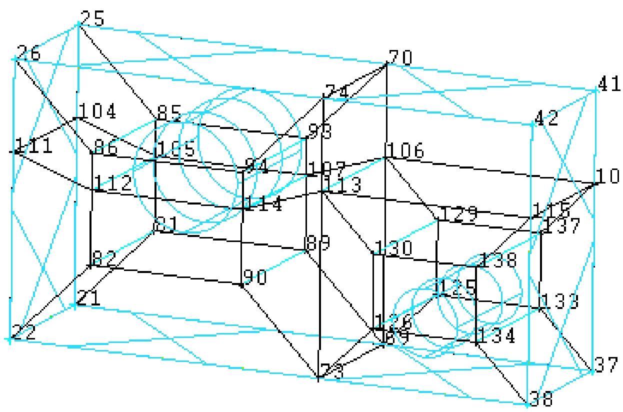


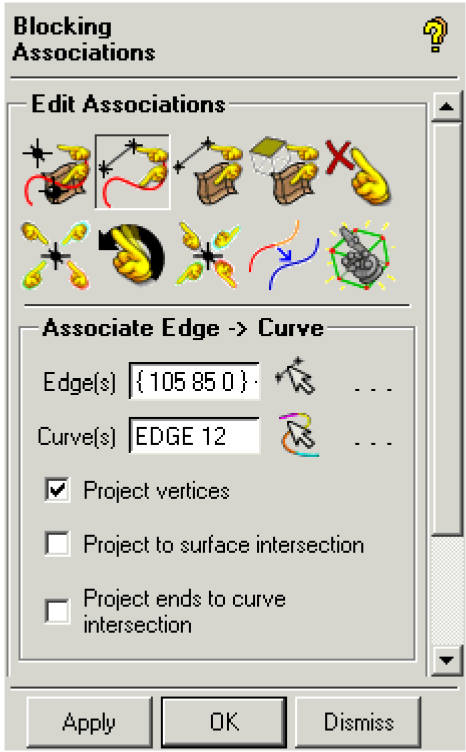
Figure 3.439 Blocking after second O-grid creation.

i) Association of Edges

Now user will associate edges to corresponding curves to capture the geometry.

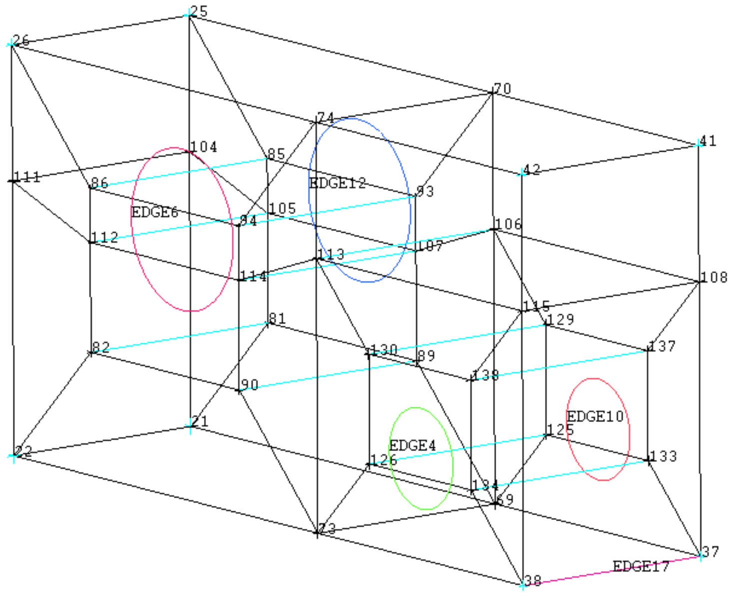
Select Blocking > Association  > Associate edge to curve  , it will open the window as shown in Figure 3.440

Figure 3.440 Associate edges to curve window



Select edges 86-94, 94-114, 114-90, 90-82, 82-112, 112-86 and associate it to EDGE6 as shown in Figure 3.441

Figure 3.441 Association of edges to curves

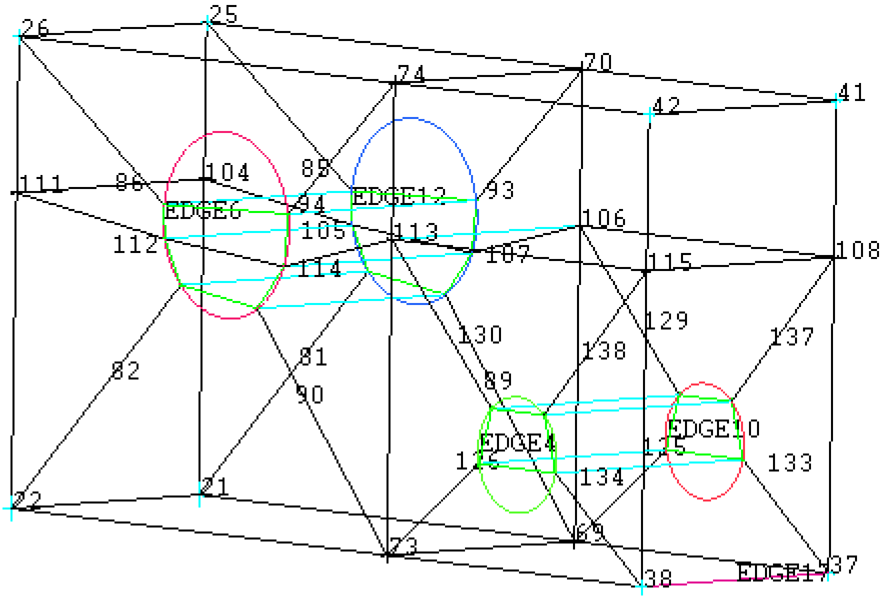


Select edges 105-85, 85-93, 93-107, 107-89, 89-81, 81-105 and associate it to EDGE12.

Select edges 126-130, 130-138, 138-134, 134-126 and associate it to EDGE4.

Similarly select edges 125-129, 129-137, 137-133, 133-125 and associate it to EDGE10. After association of edges it will look like as shown in

Figure 3.442
Association of edges to curves.



j) Moving vertices

Now user will move the vertices to to improve the quality of blocks.

Select Blocking > Move vertices  > Move vertex  , it will open the Move vertex window as shown in Figure 3.443 and move the vertices from EDGES 4, EDGES6, EDGES10, EDGES12 so that after turning the display of blocks ON and turning it to SOLID, blocking will look like as shown in Figure 3.444

Figure 3.443
Move vertex window

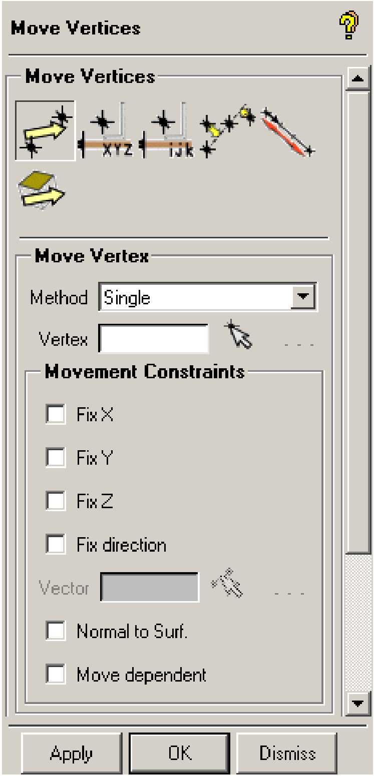
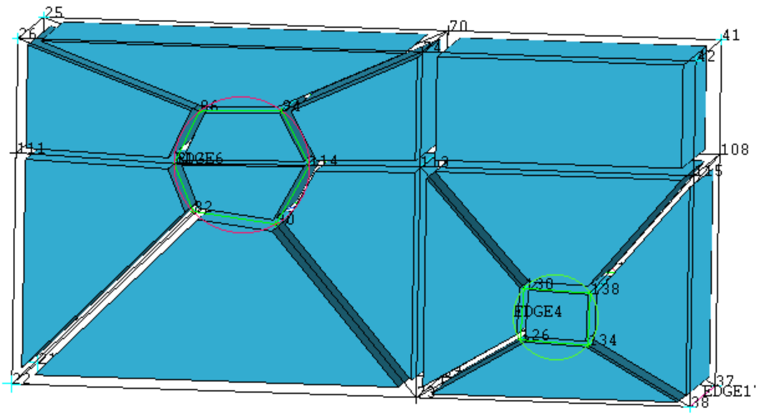


Figure 3.444
Blocking after moving vertices



k) Adding blocks to VORFN

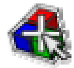
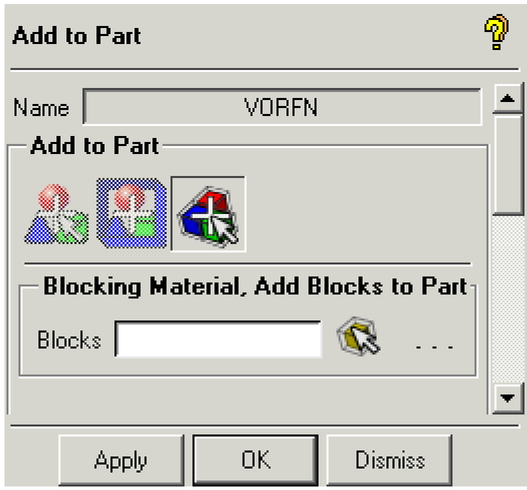
Now user will assign unrequired blocks to VORFN family. Select Parts > VORFN > Add to part, it will open Add to part window as shown in Figure, select Blocking material, Add blocks to Part  and add un required blocks as shown in Figure 3.446 to VORFN. Turn ON Blocking > Blocks > Solid, so that after adding parts to VORFN geometry should look like as shown in Figure 3.446

Figure 3.445 Add to part window



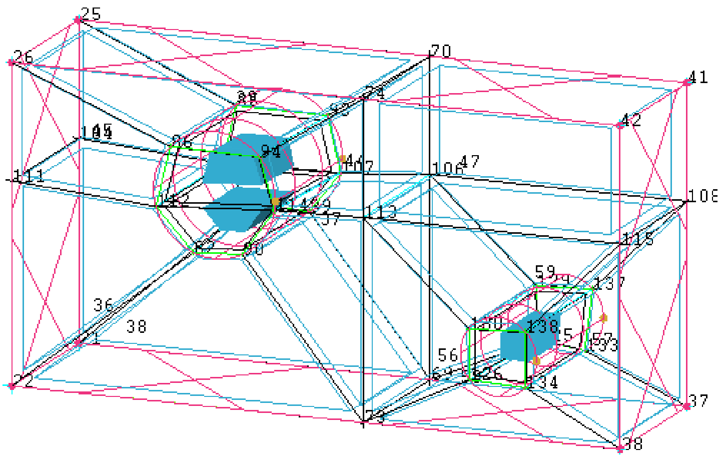


Figure 3.446: Blocks to be selected in vorfn family

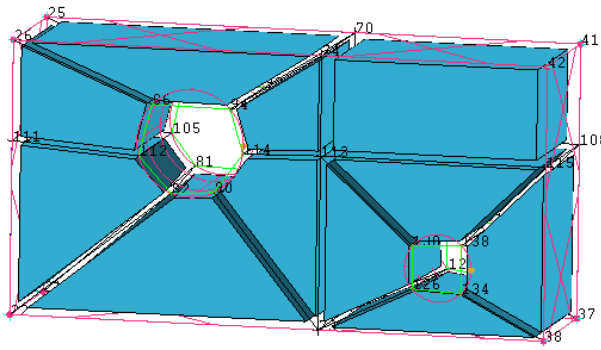


Figure 3.447 Geometry after adding blocks to VORFN

Turn OFF Blocking > Blocks.

l) Association of edges to curves

Now user will associate edges to corresponding curves.

Select Blocking > Associate > Associate Edges to curve ; it will open the window as shown in Figure 3.448. Now associate edges to corresponding curves so that after association and turning Curves OFF from Display Tree widget geometry should look like as shown in Figure 3.449

Figure 3.448 Associate edge to curve window

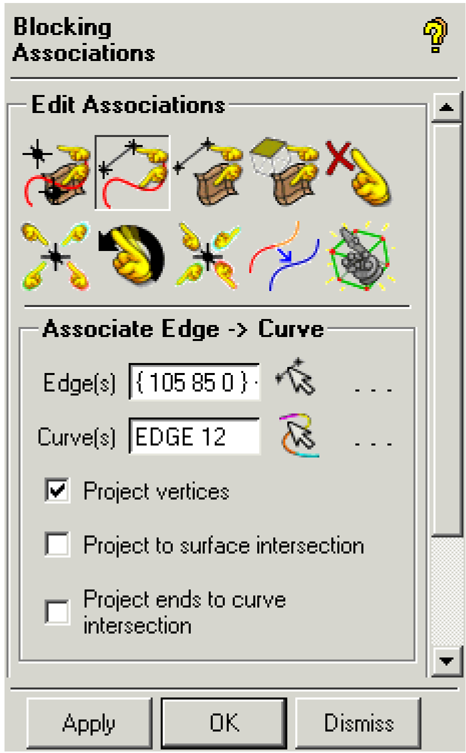
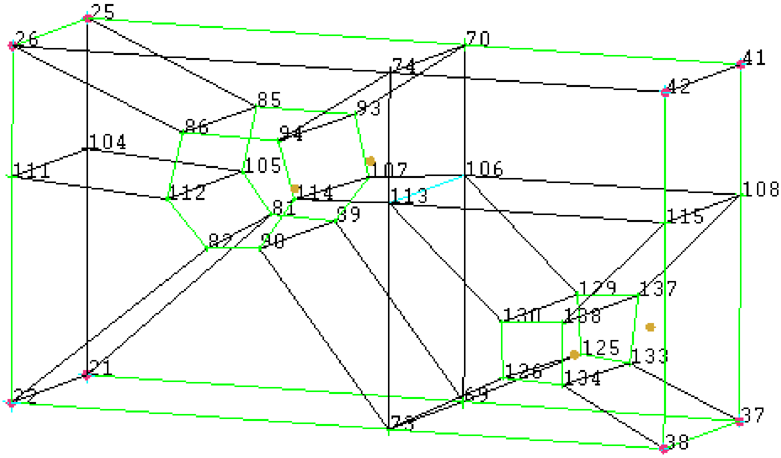


Figure 3.449 Association of edges to curve



m) Surface mesh size

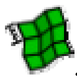
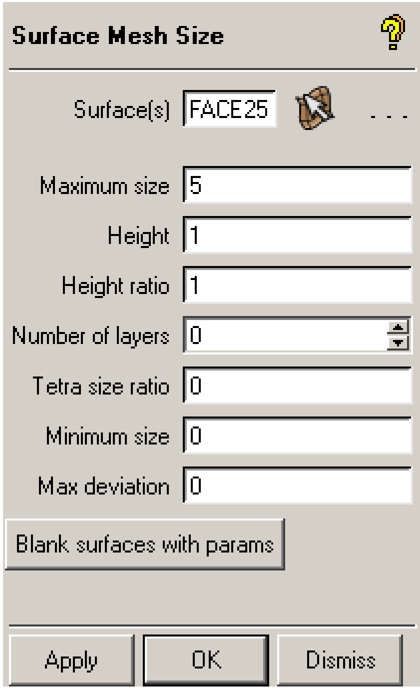
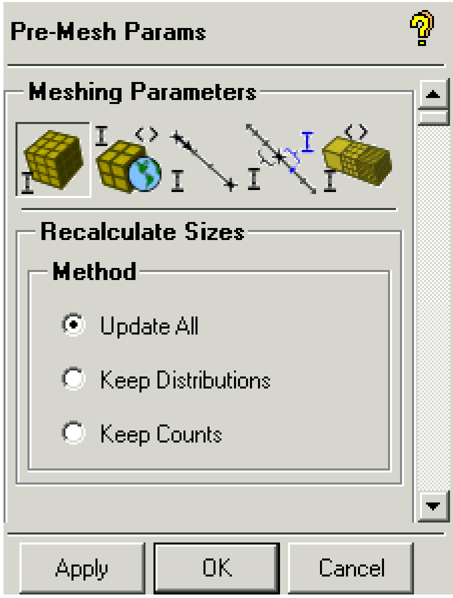
Mesh > surface mesh size , it will open surface mesh size window, enter Maximum element size as 25, Height 5 and Height ratio 1 as shown in Figure 3.450. Select  all surfaces and Press Apply.

Figure 3.450
surface mesh size window



Now select Blocking > Pre-mesh params  > Update Sizes, it will open window as shown Figure. Select Update All and press Apply.

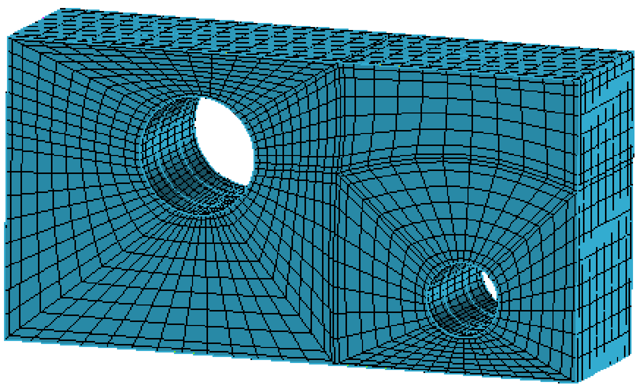
Figure 3.451 Recalculate sizes window



Now turn ON Blocking > Pre-mesh from Display Tree widget, it will ask for Recompute. Select Yes to recompute.

Turn the display of Pre-mesh to Solid from Display Tree widget Pre-mesh > Solid. After turning it to solid blocking will look like as shown in

Figure 3.452 Blocking after completing meshing



n) Checking Quality and running Pre-mesh smoother

Now user will check the quality of mesh which will be created from blocking and run Pre-mesh smoother to improve its quality.

Choose Blocking > Pre-mesh quality  , and criteria as Determinant 3*3*3 and enter the parameters as shown in Figure 3.453 . It will show the quality of mesh in histogram similar to quality shown in

Figure 3.453 Pre-Mesh quality window

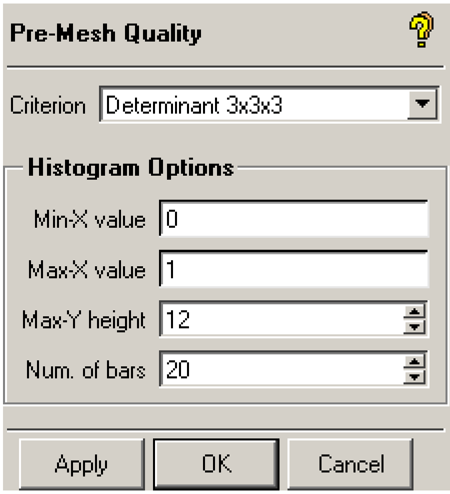
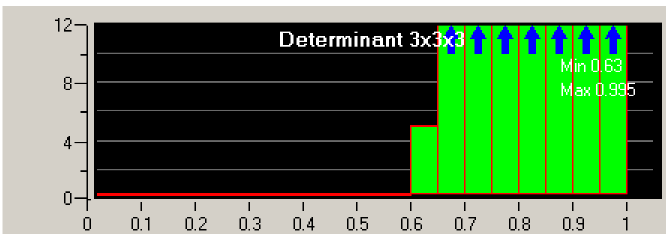


Figure 3.454 Histogram showing Determinant 3*3*3 quality



Now user will save project, Select File > Save project as and enter name as Parametric110.

o) Modifying geometry in DM

Now user will modify the geometry in DM. Select Tools >Parameters from main menu as shown in Figure 3.455 . It will open the parameter window as shown in Figure 3.456.

Figure 3.455 Selection of parameters from main menu.

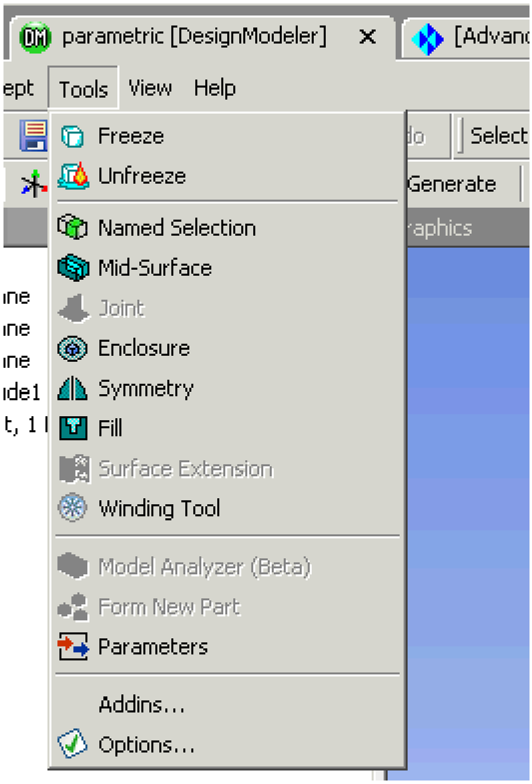
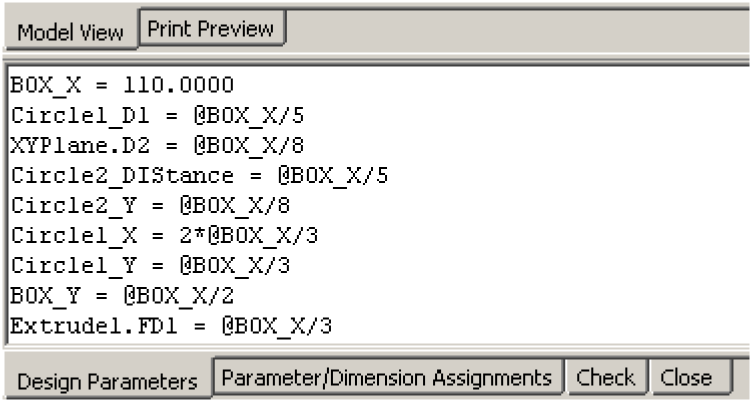
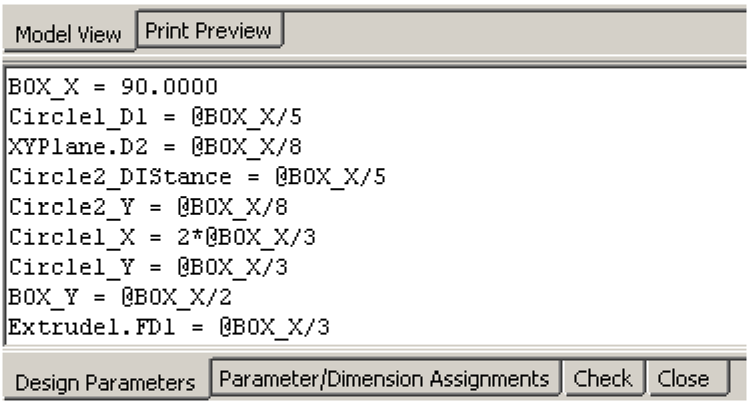


Figure 3.456 Parameter window



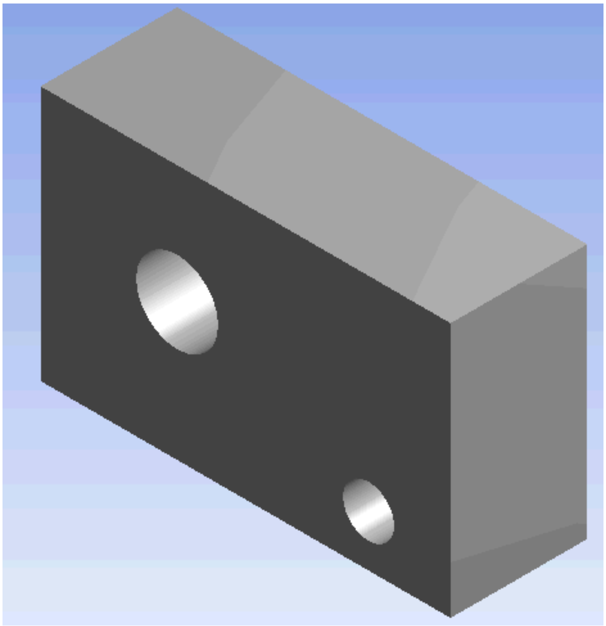
Now user will change the width of block change the dimension of BOX_X = 90. Parameter window after changing length is shown in Figure 3.457. Now close the parameter window.

Figure 3.457 Parameter manager after changing width of the block.



Now select Generate to make changes in the geometry. The Geometry after making changes is as shown in Figure 3.458

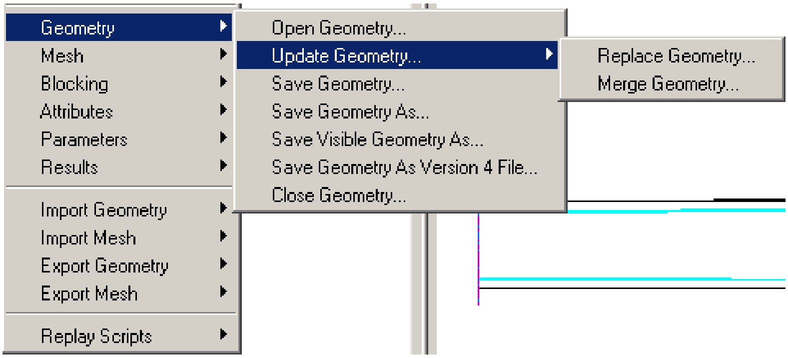
Figure 3.458 Geometry after modifying dimensions



p) Updating blocking in Advance meshing

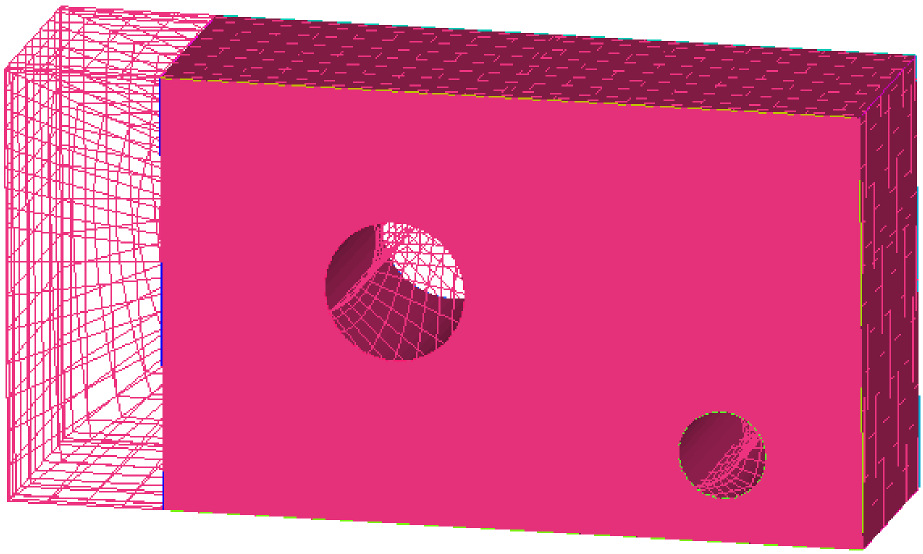
Now user will update blocking for changes made in original geometry. To open modified geometry in Advance meshing select File > Geometry > Update Geometry > Replace Geometry as shown in Figure. It will ask for saving the changes in geometry, select Yes to make changes.

Figure 3.459: Updating geometry in Advance Meshing



It will open the modified geometry file and merge with the original blocking file as shown in Figure 3.460

Figure 3.460
Modified geometry merged with original blocking



Now user will update the projection of original blocking onto modified geometry in order to capture modified changes.

Select Blocking > Associate > Update Association , it will open the window as shown in Figure 3.461. After updating the blocking, new blocking will look like as shown in Figure 3.462

Figure 3.461
Update association window

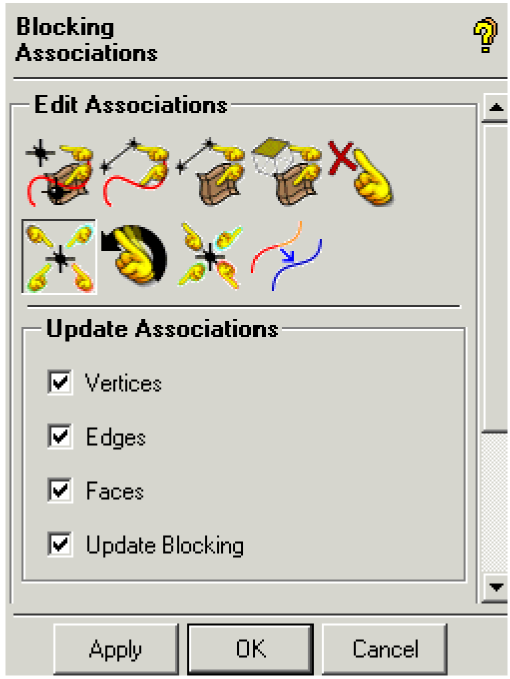
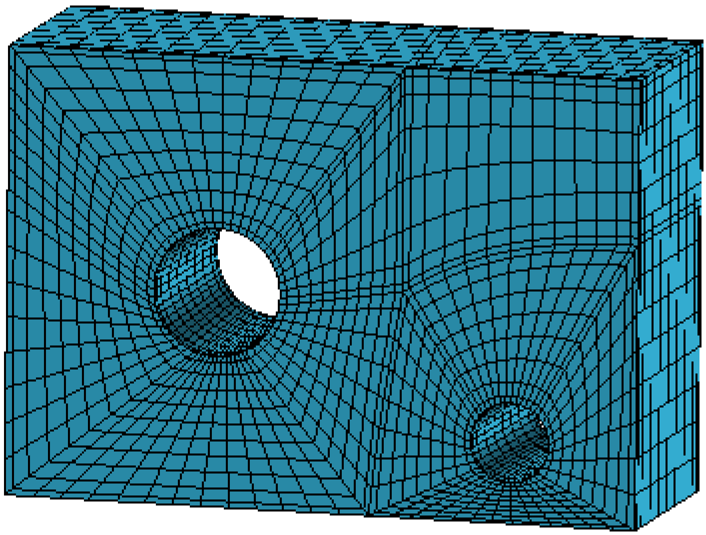


Figure 3.462 Blocking after updating associations



q) Saving the Project

Now user will save project, Select File > Save project as and enter name as Parametric90.

3.7: Cart3D

The main menu contains project and file related options and some settings options. The main menu is shown in Figure 3.463.

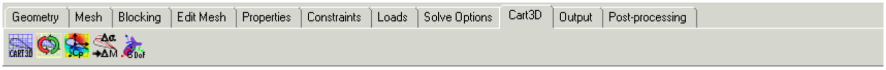


Figure 3.463 Main Menu

The main menu has the following options

1) **Volume Mesher** 

2) **Solve** 

3) **Integrate Cp** 

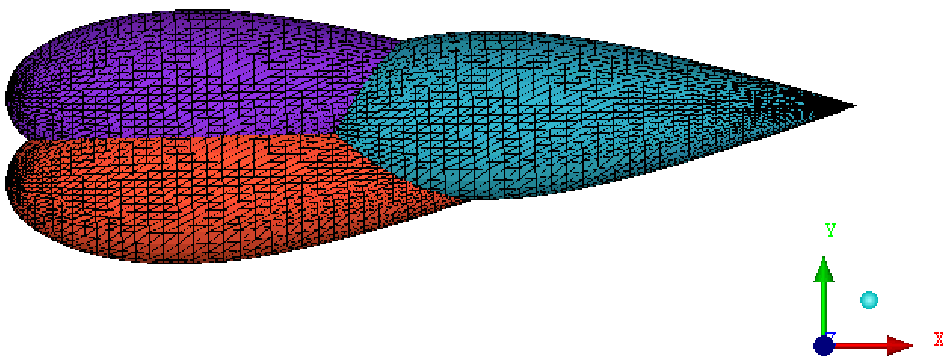
4) **Run Trials** 

5) **Run 6 DOF** 

3.7.1: Tutorial Three Plugs

Overview

This tutorial illustrates how to generate a grid in **Cart3D** around a set of three plugs.



This tutorial introduces the following operations.
i). Use of the Cart3D mesher for mesh generation
ii). Multigrid preparation – running mgPrep

a) Starting the Project

Load **ANSYS ICEM CFD**. Change the working directory using File > Change Working Dir…and set the location to the folder **plugs** (**plugs.uns** is located in that folder).

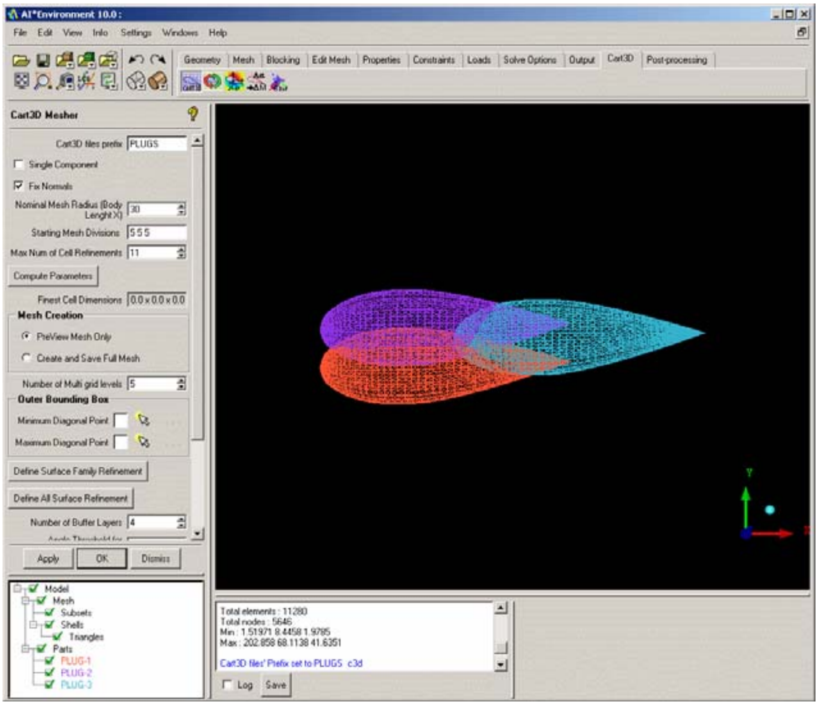
Note: It is preferable to create a separate folder **plugs** and put only the **plugs.uns** (domain/mesh) file in that folder before performing this tutorial.

Select Open Mesh  from the main menu and select '**plugs.uns**.' The model contains three closed triangulated components. Press the '**h**' key to fit the view in the screen if the model is not visible.

b) Mesh Generation Preview only

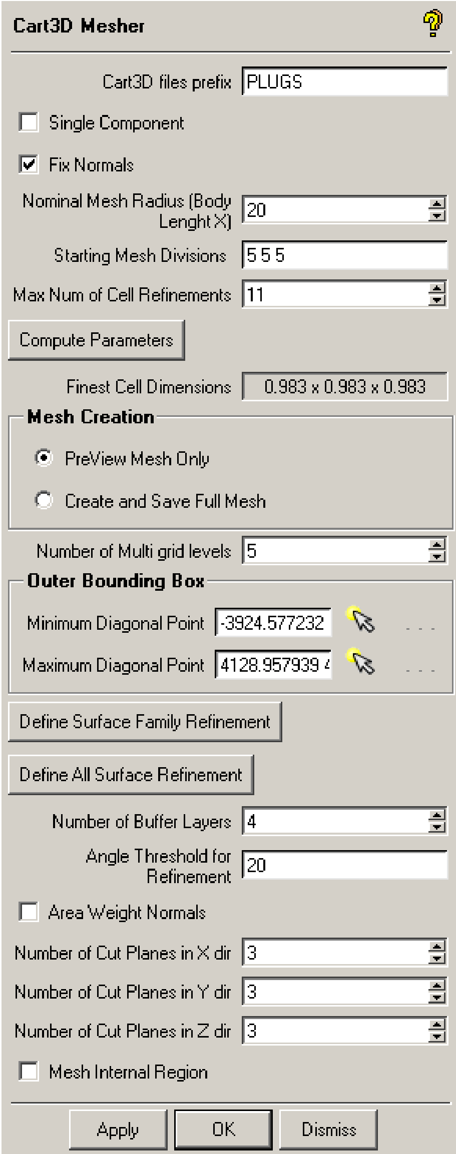
Click on the Cart 3D tab. Select the Volume Mesher icon . We get the Cart 3D Mesher window as shown in Figure 3.464.

Figure 3.464 Cart3D main window



Leave Fix Normals enabled as this will fix the orientation of the triangles such that their normals point outwards.
Choose Nominal Mesh Radius (Body Length X)=20, Starting Mesh Divisions = 5 5 5 and Max Num of Cell Refinements = 11.
Click Compute Parameters. This saves the mesh in the local directory, converts in into Cart3D format, and determines the intersections if any. This step is required even if there is only one component - to convert the triangulation to Cart3D tri format. At the end, it displays the Finest Cell Dimension as shown in Figure 3.465.

Figure 3.465 Cart3D Mesher window



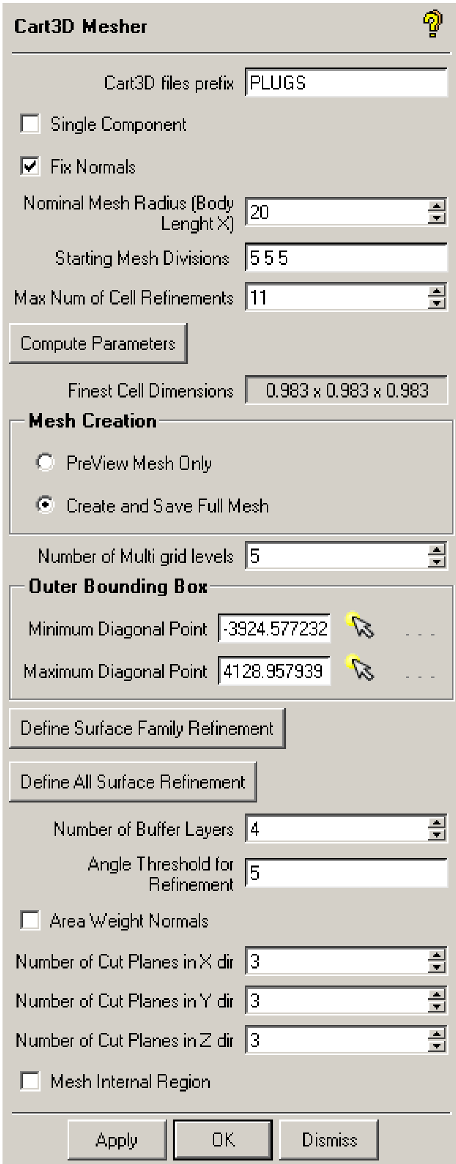
This will create 4 density polygons for mesh density control, which can be viewed in the Display Tree widget by switching on the Geometry > Densities.

This also computes the finest cell size: **0.983 x 0.983 x 0.983**. Varying the starting mesh division and/or Max number of cell refinements can vary the finest cell size.

The diagonal points displayed under the ‘Outer Bounding Box’ are the Minimum and Maximum points of the bounding box/Mesh region (refer to Figure 3.465). They can be changed if desired.

Set the Angle Threshold for Refinement to 5 as shown in Figure 3.466.

Figure 3.466 Change Angle of Refinement

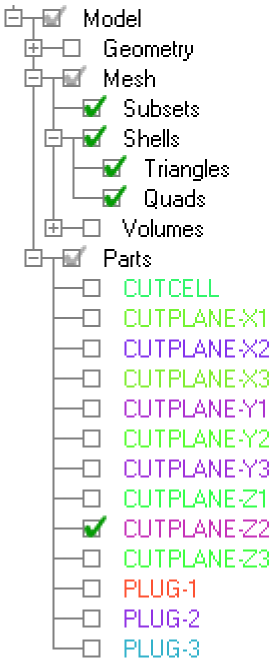


Click Apply to run the mesher. This will create a domain file with 3 Cut Planes (Quad Elements) in each coordinate direction and Cut Cells (Hex

Elements) through which the defining surface triangles pass. This Preview Mesh will be loaded automatically.

In the Part Menu under the Display Tree widget right-click on Parts and select Hide All. Then turn on only the Part **CUTPLANE-Z2** as shown in Figure 3.467.

Figure 3.467 Display Tree widget



The mesh is shown in Figure 3.468. This is the projected mesh on the middle plane in the Z-direction **CUTPLANE-Z2**.

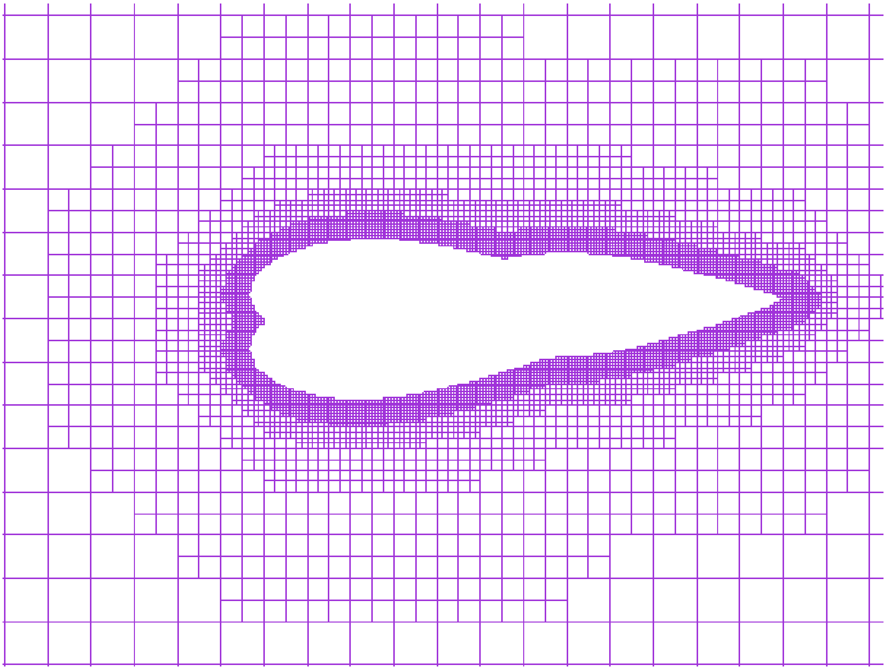


Figure 3.468 Cut Plane Z2 Mesh

Right-click in the Display Tree widget and select Parts > Show All after viewing the mesh.

c) Mesh Generation Full Mesh

Now in the Cart3D Mesher window enable Create and Save Full Mesh as shown in.Figure 3.469.

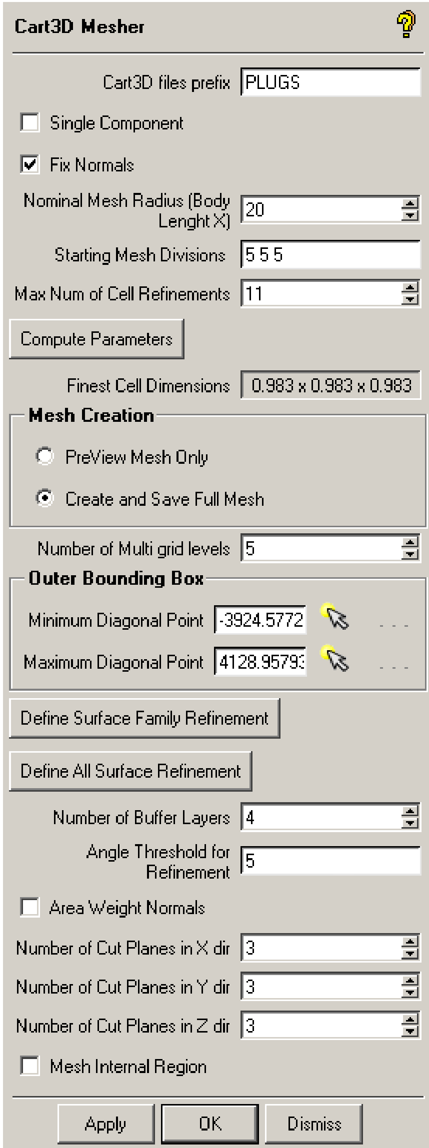


Figure 3.469 Create and Save Full Mesh

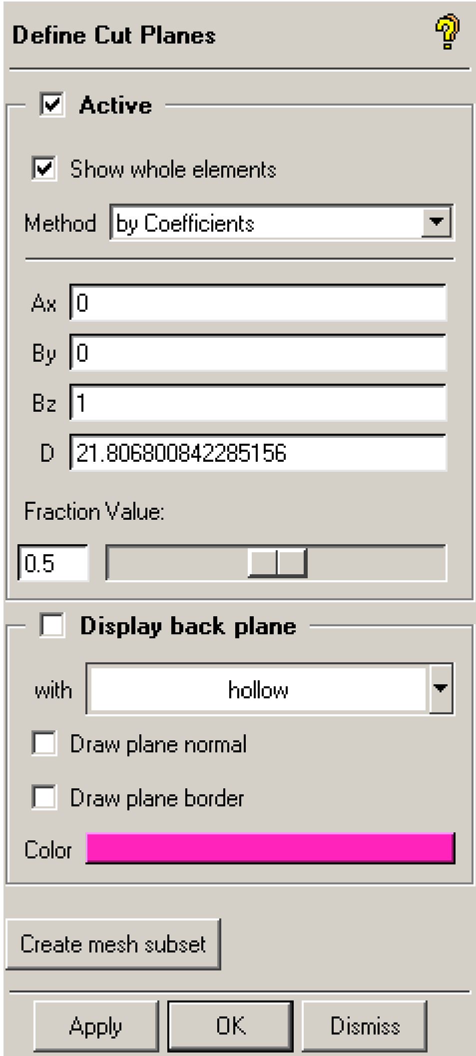
Leave the Number of Multi grid levels to 5. This will create 5 levels of coarsened mesh, which can be read by the solver.

Press Apply. The Cart3D Mesh window appears which asks us about loading the cart3D full mesh as shown in Figure 3.470, press Yes.

Figure 3.470
Cart3D Mesh window

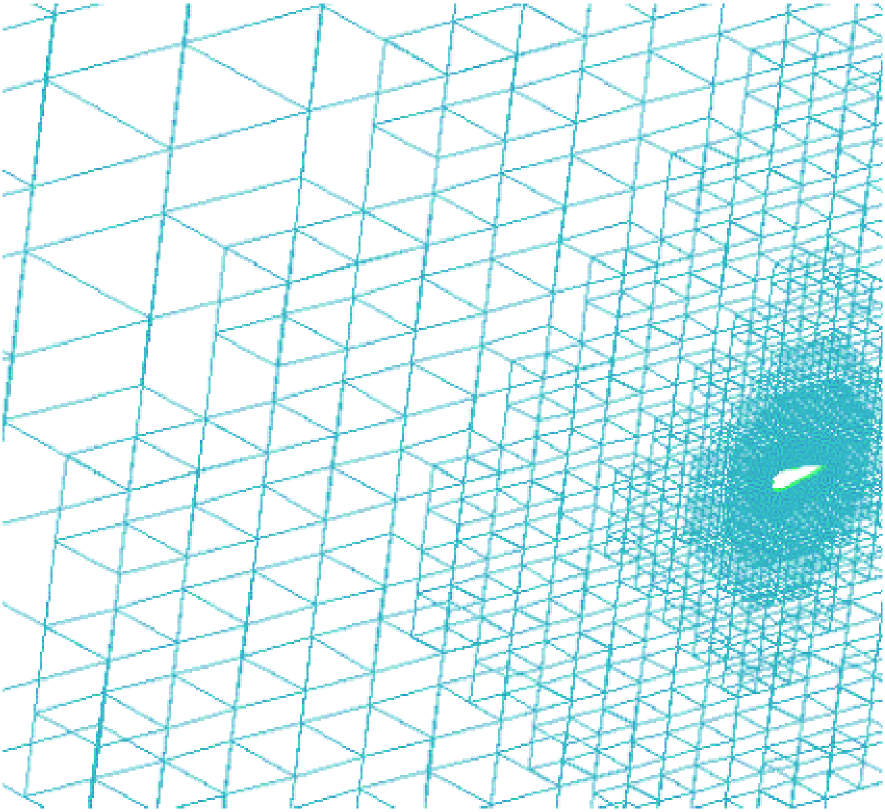
Switch on Mesh > Volumes in the Display Tree widget.
The final mesh generated can be examined through Mesh > Cut plane. The Define Cut Planes window appears as shown. Accept the default settings as shown in Figure 3.471.

Figure 3.471
Cut Plane Display



The mesh viewed using the above parameters is shown in Figure 3.472.

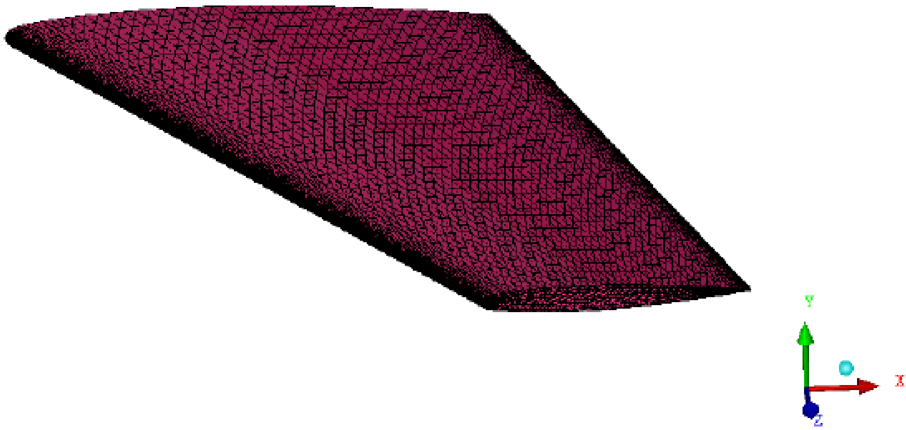
Figure 3.472 Cut Plane mesh



3.7.2: Tutorial Opera M6 Wing with 0.54 M

Overview

This tutorial illustrates how to generate a grid in Cart3D around a Wing and how to solve the problem in flowCart. Post processing the results is also explained.



This tutorial introduces the following operations:
Use of the Cart3D mesher for mesh generation
Multi grid preparation with mgPrep
Running the solver for AOA=3.06 and Mach=0.54
Computing Forces and Moments using Clic.
Visualizing the result in Post Processing

a) Starting the Project

Load **ANSYS ICEM CFD**. Change the working directory using File>Change Working Dir… and set the location to the folder **wing1** (**oneraM6.uns** is located in that folder).

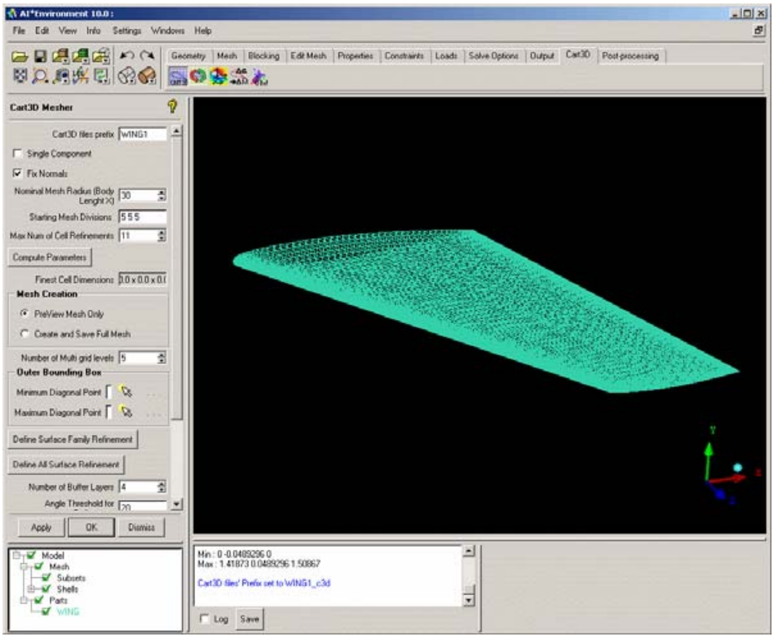
Note: It is preferable to create a separate folder **wing1** and put the **oneraM6.uns** (domain) file in that folder before performing this tutorial.

Select Open Mesh  from the main menu and select **oneraM6.uns**.

b) Mesh Generation-Preview only

Click on Cart3D from the main menu. Select the Volume Mesher  icon. We get the cart 3D Mesher window as shown in Figure 3.473.

Figure 3.473 Cart3D GUI window

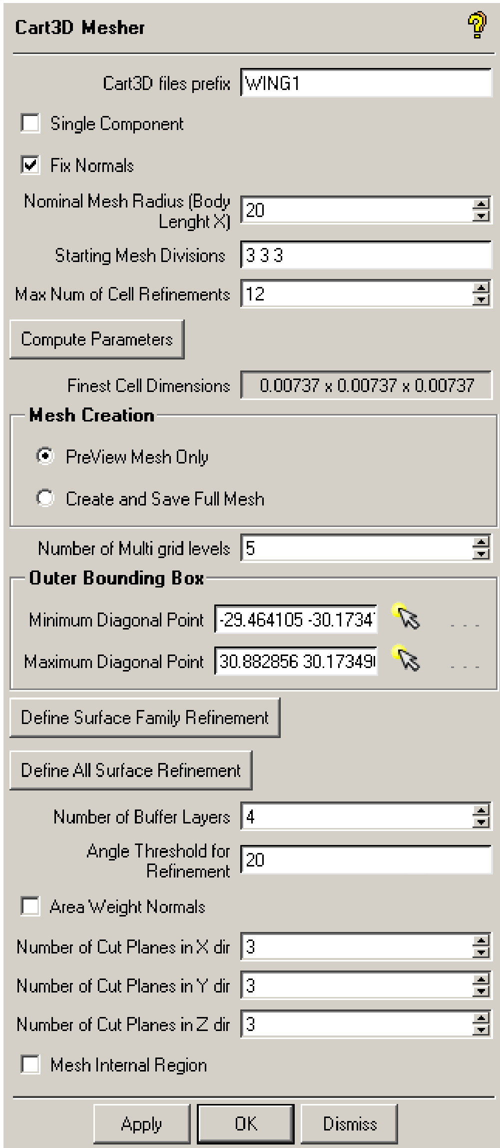


Leave Fix Normals enabled. This will fix the orientation of the triangles such that their normals are pointing outward.

Choose Nominal Mesh Radius (Body Length X) = 20, Starting Mesh Divisions = 3 3 3 and Max number of Cell Refinements = 12 .

Click Compute Parameters. This saves the mesh in the local directory, converts in into Cart3D format, and finds the intersections if any. This is required to convert the triangulation to Cart3D tri format even if there is only one component present. At the end, it displays the Finest Cell Dimensions as shown in Figure 3.474

Figure 3.474
Cart3D Mesher window



This will create 2 density polygons for mesh density control that can be seen by activating Geometries>Densities in the Display Tree widget.
This also computes the Finest Cell Dimensions**: 0.00737 x 0.00737 x 0.00737**. Varying the Starting Mesh Divisions and/or Max Num of Cell Refinements can vary these values.
The diagonal points displayed under the 'Outer Bonding Box' are the maximum and minimum points of the bounding box of the Mesh region. They can be changed if desired.
Set the Angle Threshold for Refinement to 5

Note: In this case we wish to run the case with symmetry in the Z direction. Specify the bounding box minimum Z coordinate as 0.00001 (slightly inside the model). Refer to Figure 3.475. If the model itself is symmetric, turn on Half-Body Mesh (Symmetric in Z).

Click Apply (after specifying minimum Z coordinates as 0.00001) as shown in Figure 3.475 to run the mesher. This will create a domain file with 3 Cut Planes (Quad Elements) in each coordinate direction and Cut Cells (Hex Elements). The Preview Mesh will be loaded automatically.

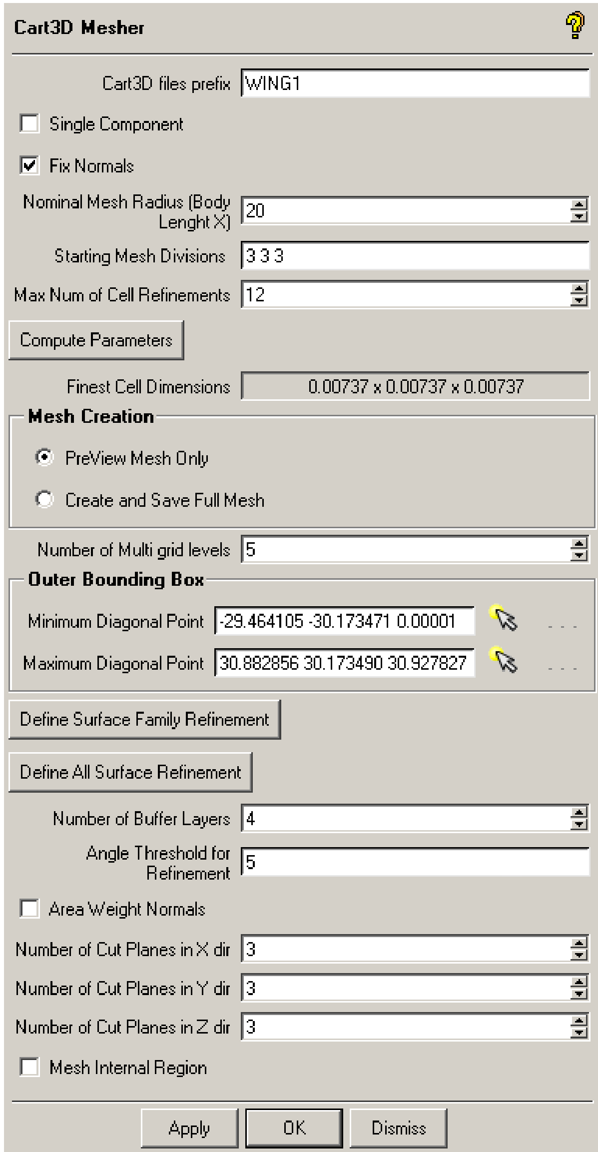
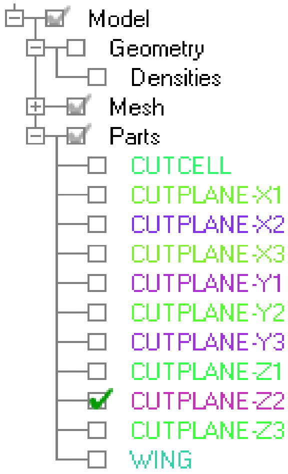


Figure 3.475 Change Angle of Refinement

In the Parts menu under the Display Tree widget perform the operation Parts>Hide All (right-click on Parts to access) and then turn on only the Part **CUTPLANE-Z2** as shown in Figure 3.476.

Figure 3.476 Display Tree widget



The mesh projected onto the middle z-direction plane (in Part **CUTPLANE-Z2**) is shown in Figure 3.477.

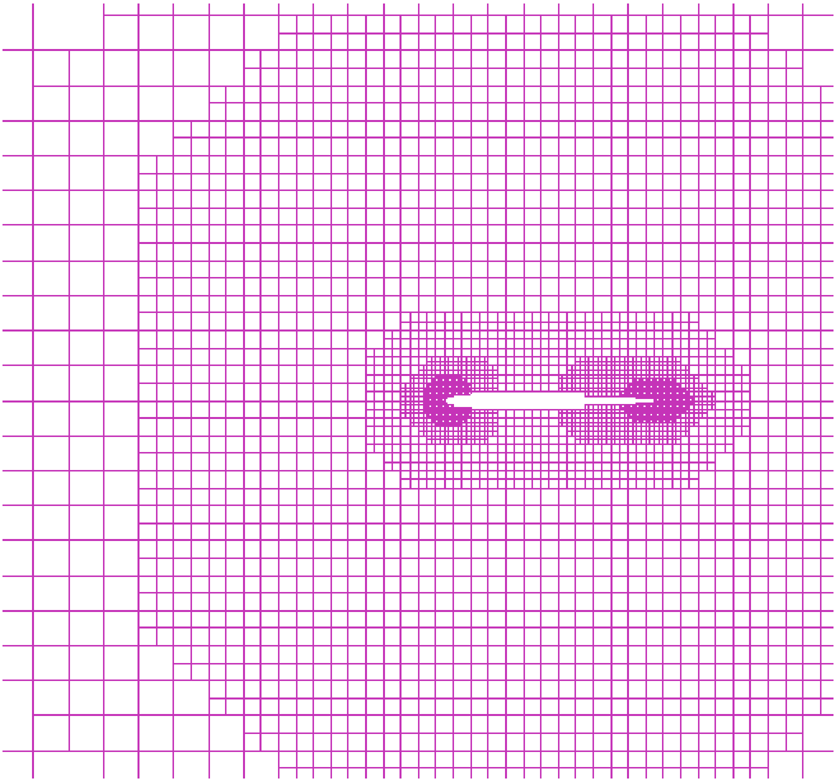


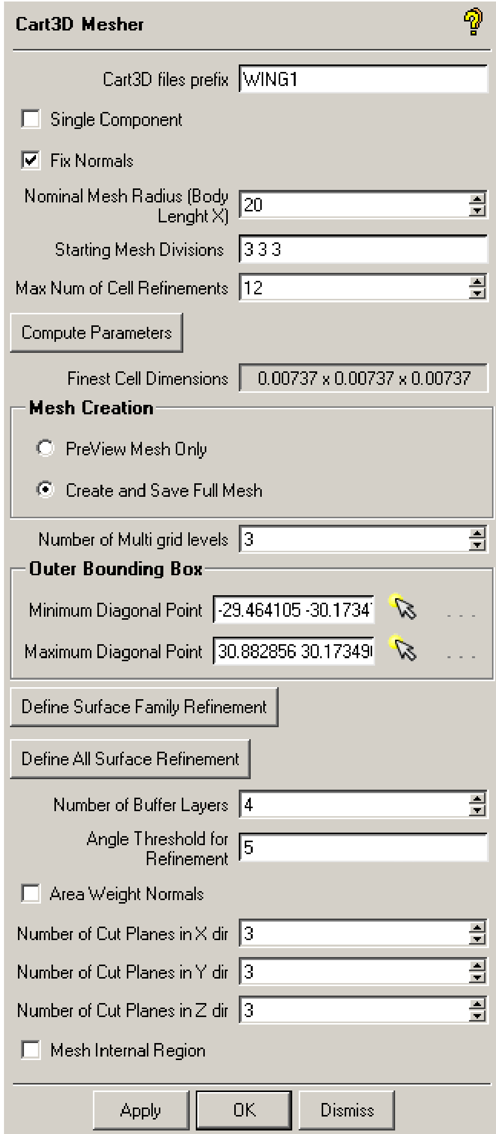
Figure 3.477 CUTPLANE-Z2 Mesh

Perform the operation Parts>Show All by a right-click on Parts in the Display Tree widget after viewing the mesh.

c) Mesh Generation-Full Mesh

Now in the Cart3D mesher window enable Create and Save Full Mesh as shown in Figure 3.478.

Figure 3.478 Create and Save Full Mesh



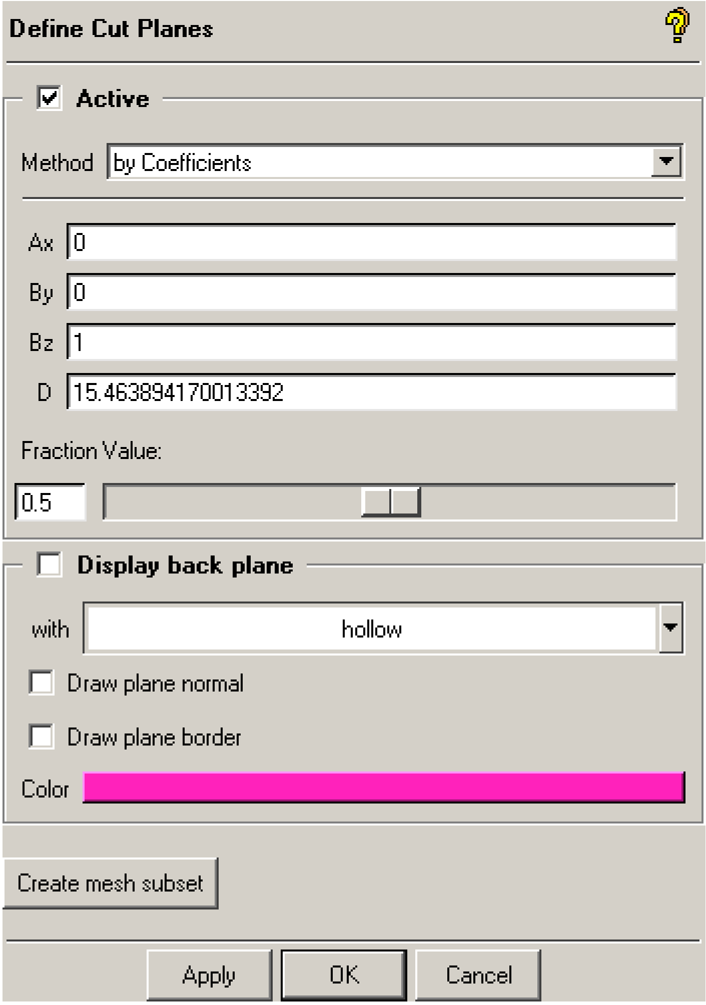
Set the Number of Multi grid levels to 3. This will create 3 levels of coarsened mesh, which can be read by the solver.

Press Apply. The Cart3D Mesh window appears which asks about loading the cart3D Full Mesh as shown in Figure 3.479. Press Yes.

Figure 3.479 Cart3D Mesh window

The final mesh generated can be examined through Mesh>Cutplane. The **Define Cut Planes** window appears as shown. Accept the default settings as shown in Figure 3.480.

Figure 3.480 Define Cut Planes Window



The mesh cut plane using the above parameters is shown in Figure 3.481.

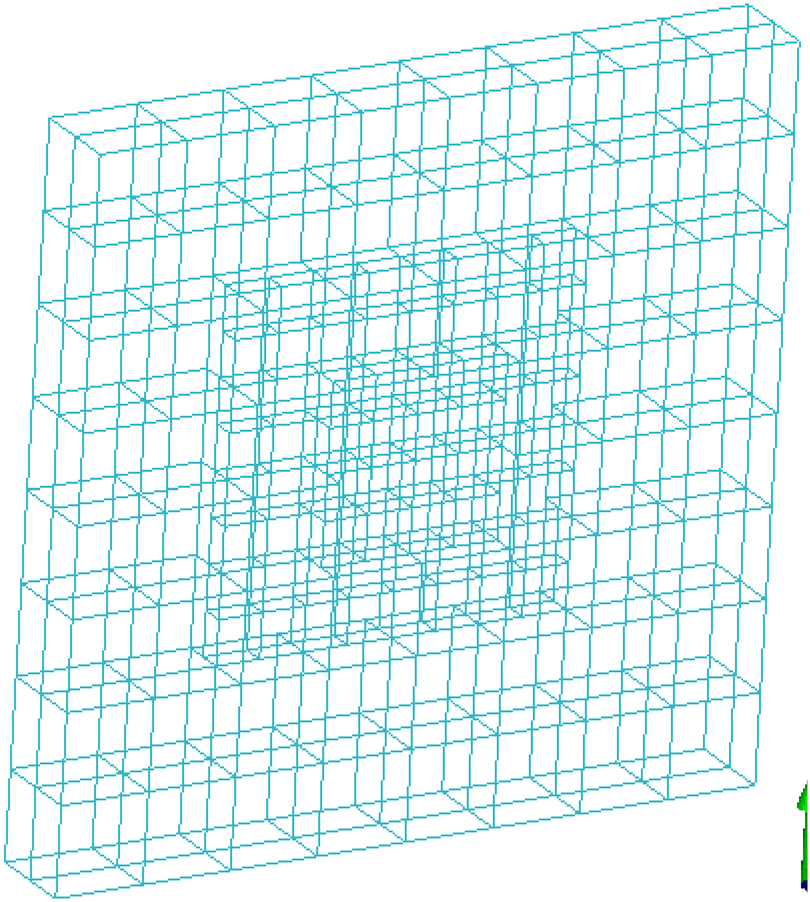
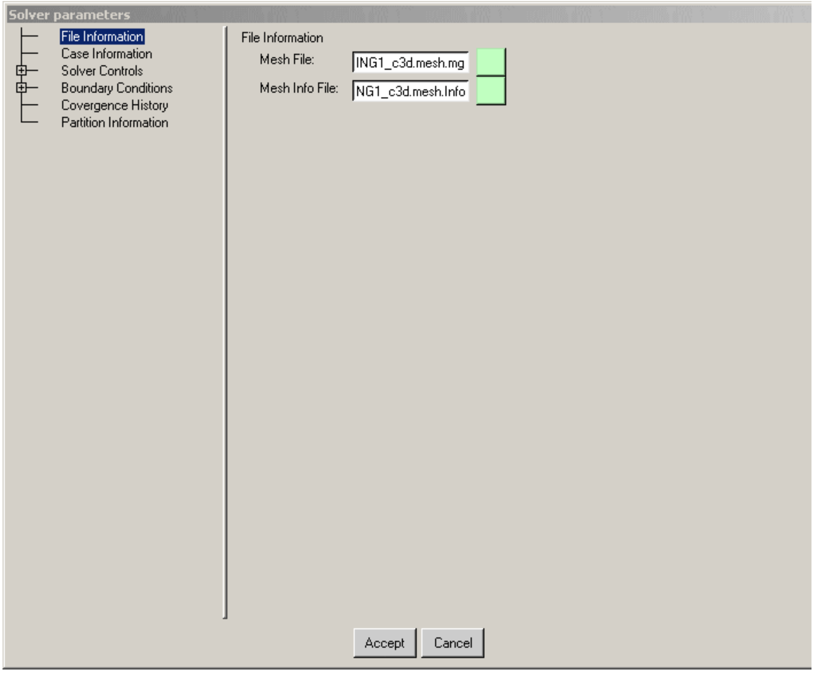


Figure 3.481 Cut Plane Mesh

d) Setup Flow Cart Parameters

In the Cart3D Menu select Solver . Click on Define Solver params  icon (if the panel doesn't open automatically). A **Solver parameters** window appears as shown in Figure 3.482.

Figure 3.482 Solver parameters window



Set File Information>Mesh File as **WING1_c3d.mesh.mg** (should be default).
Click on Case Information and enter the following parameters:
Mach number = 0.54
Angle of Attack = 3.06
Side Slip angle = 0.0
Free Stream Density = 1.0
Free Stream Sound Speed = 1.0
The values are shown in Figure 3.483.

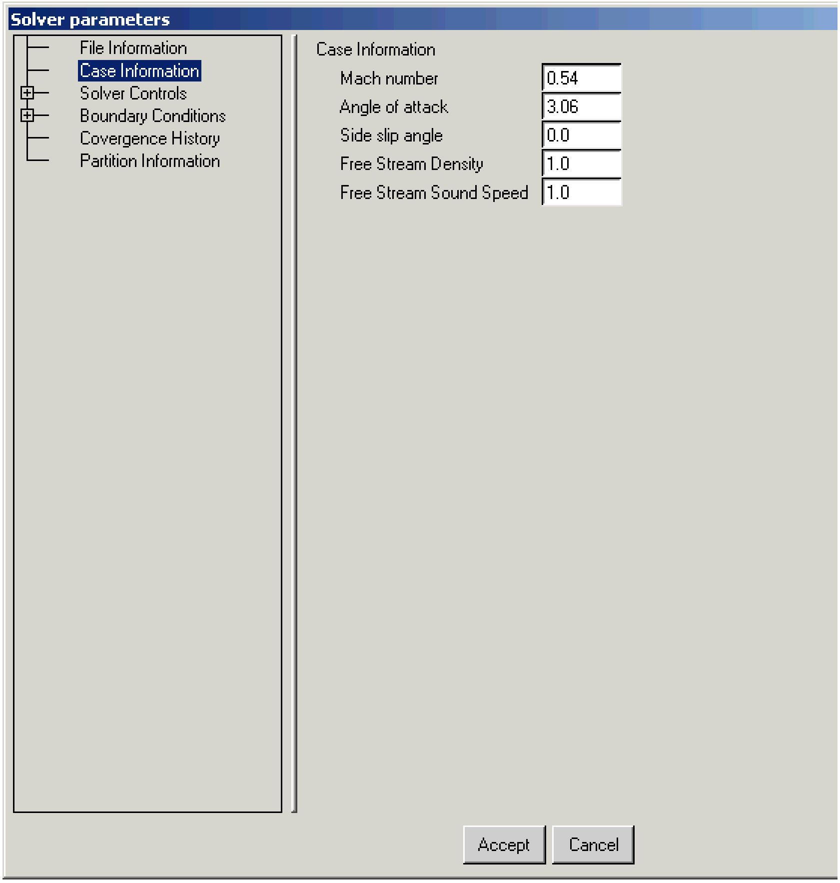
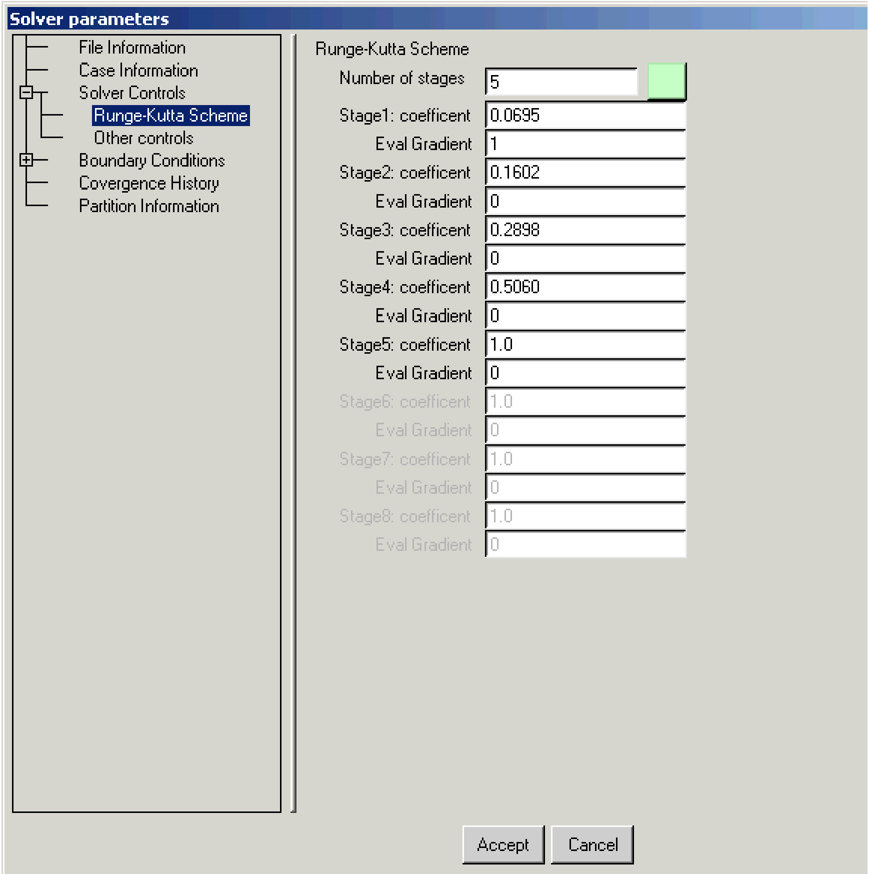


Figure 3.483 Case Information window

Expand Solver Controls –Runge-Kutta Scheme in the Display Tree widget as shown in Figure 3.484 and accept the default settings.

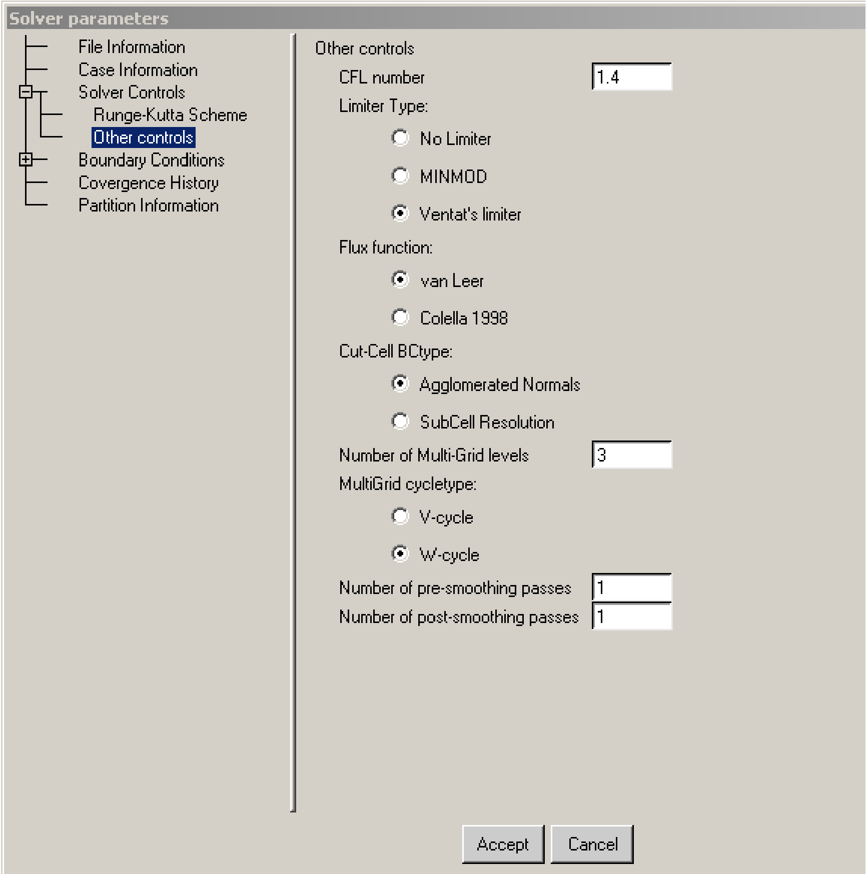
Figure 3.484 Runge-Kutta Scheme window



In **Other controls** specify the following values for the parameters as shown in Figure 3.485.

CFL number: 1.4
Limiter Type: Ventat's limiter
Flux function: van Leer
Cut-Cell BCtype: Agglomerated Normals
Number of Multi-Grid levels: 3
MultiGrid cycle type: W-cycle
Number of pre-smoothing passes:1
Number of post-smoothing passes:1

Figure 3.485 Other controls window



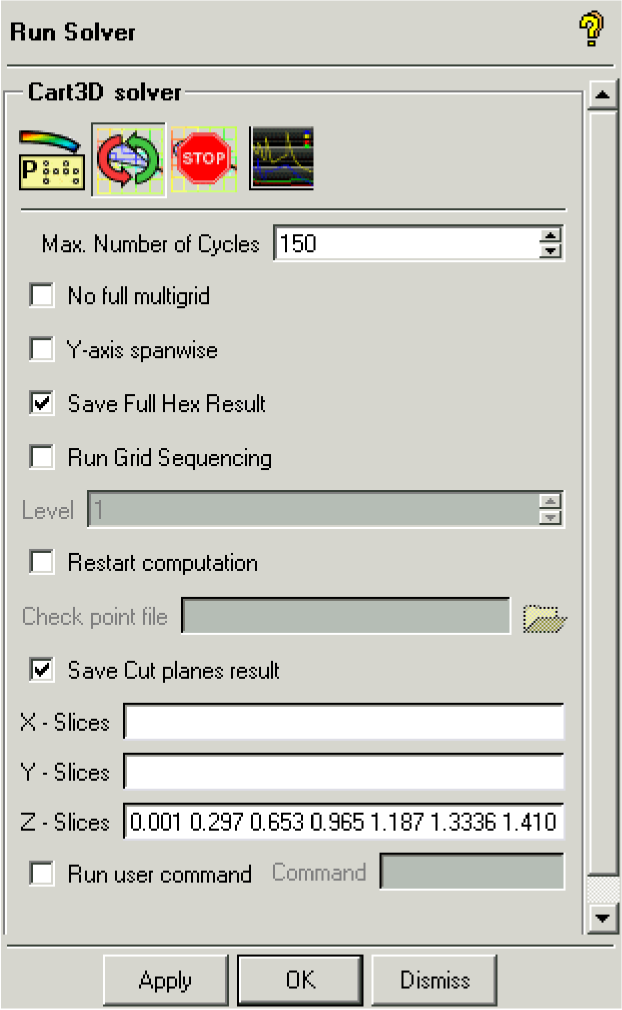
Keep defaults for Boundary Conditions, Convergence History, and Partition Information and press Accept.

e) Running the FlowCart Solver

Select Solver >Run Solver to open the flow chart solver panel (refer to Figure 3.486).
Specify Max. Number of Cycles = 150.
Turn on Save Full Hex Result.

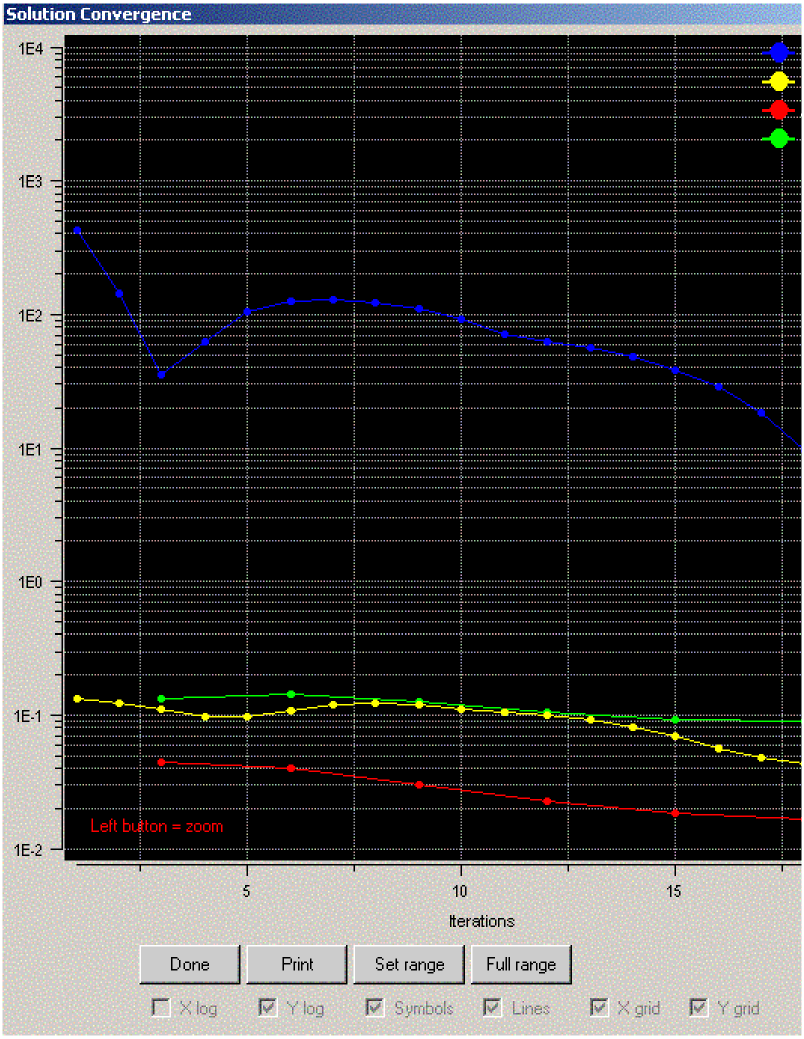
Turn on Save Cut planes result and specify Z-Slices as 0.001 0.297 0.653 0.965 1.187 1.3336 and 1.410.
Click Apply to run the solver.

Figure 3.486 Run Solver window



The user can view the convergence by selecting the Convergence monitor icon  as shown in Figure 3.487. (The monitor may open automatically.)

Figure 3.487 Solution Convergence Window



f) Computing Force and Moments

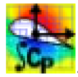
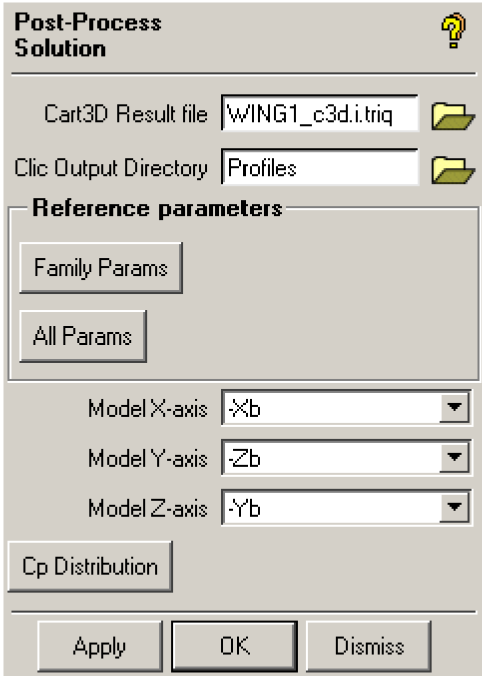
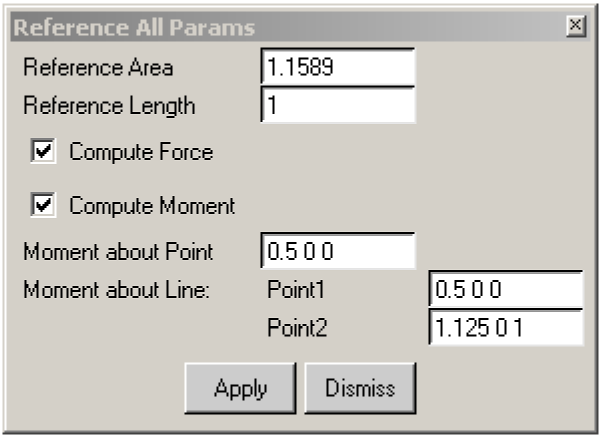
In the Cart3D main menu select Integrate Cp . The Post-Process Solution window appears as shown in Figure 3.488.

Figure 3.488 Post-Process Solution window



Click **All Params** in the Post-Process Solution window.
In the Reference All Params window specify **Reference Area** as 1.1589, **Reference Length** as 1
Enable **Compute Force** and **Compute Moment**.
Set **Moment about Point** = 0.5 0 0, **Point1** = .5 0 0, and **Point2** = 1.125 0 1.
Click Apply in the Reference All Params window and then Dismiss to close as shown in Figure 3.489.

Figure 3.489 Reference All Params Window

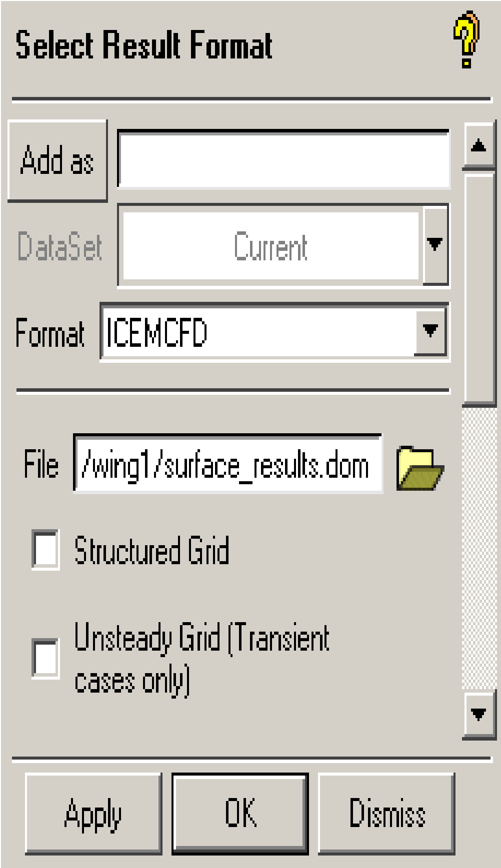


Press Apply in the Post-Process Solution window. The results appear in the GUI messages area.

g) Visualizing the results

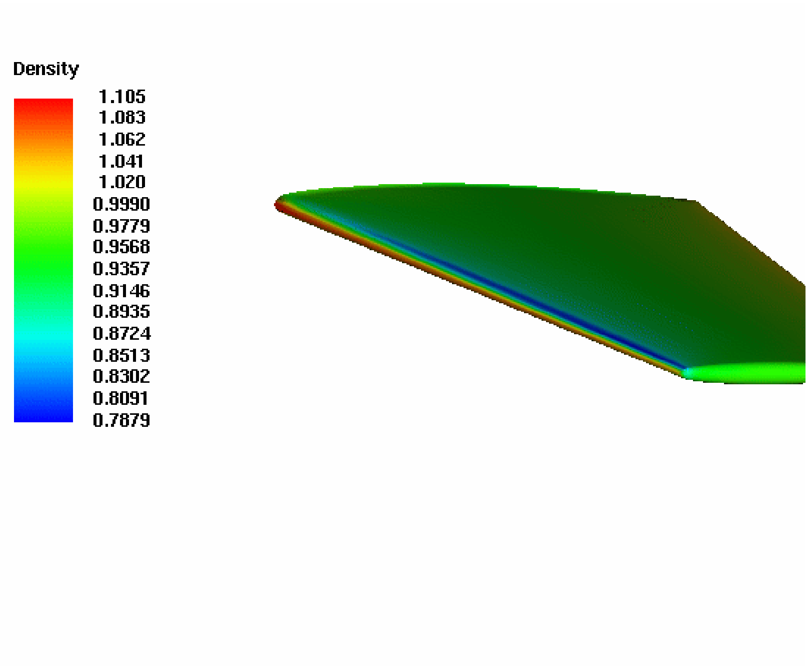
FlowCart writes three output files:
i). WING1_c3d.i.triq - Contains Pressure, Velocity and Density extrapolated to the Surface triangles. This can be converted to a domain file by Edit>Cart3D Tri File->Domain file. The default resultant domain file will be WING1_c3d.uns
ii). **slicePlanes.dom** - Cut Plane results
iii). **results.dom -** Full mesh results
Go to File >Results >Open Results…
Select Format as ICEMCFD.
Specify surface_results.dom as the File as shown in Figure 3.490.

Figure 3.490 Select Result File Window

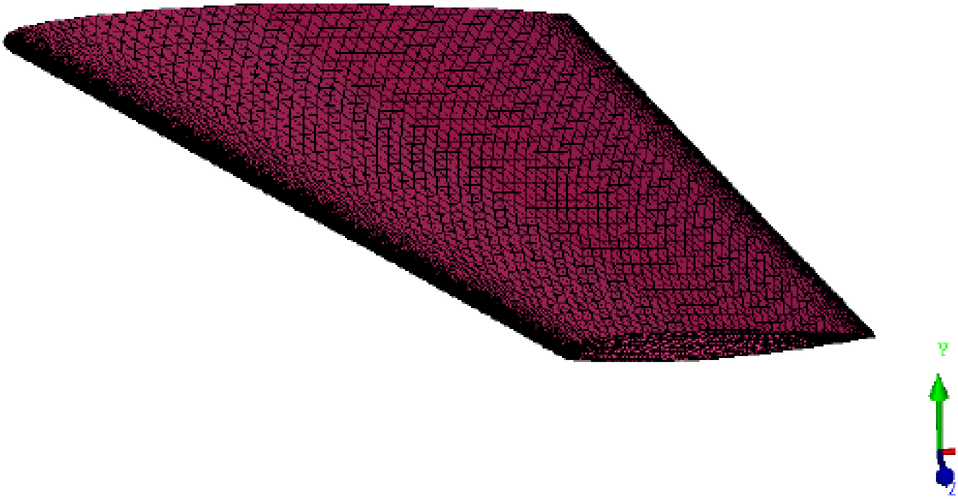


Select Apply from the panel to get the default result as shown in Figure 3.491. Right click on Colormap from the Display Tree widget and select Modify Entries to adjust the Min and Max values for the displayed variable.

Figure 3.491 Visualization of Results



3.7.3: Onera M6 Wing with 0.84 M



Overview
This tutorial illustrates grid generation in Cart3D around a Wing and solving the problem in flowCart. Post processing the results is also explained.
The tutorial introduces the following operations:
Use of the Cart3D mesher for mesh generation.
Multi grid preparation - running mgPrep.
Running the solver for AOA=3.06 and Mach=0.84.
Computing force and moment information.
Visualizing the result in the post processor.

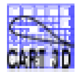
a) Starting the Project

Load **ANSYS ICEM CFD**. Change the working directory by File>Change Working Dir… and set the location to the folder **wing2** (**oneraM6.uns** is located in that folder).

Note: It is preferable to create a separate folder **wing2** and put only the **oneraM6.uns** domain file in that folder before performing this tutorial.

2. Select Open Mesh  from the main menu and select **oneraM6.uns**.

b) Mesh Generation-Preview only

1. Click on Cart 3D from the main menu. Select the Volume Mesher 

icon. We get the cart 3D Mesher window as shown in Figure 3.492.

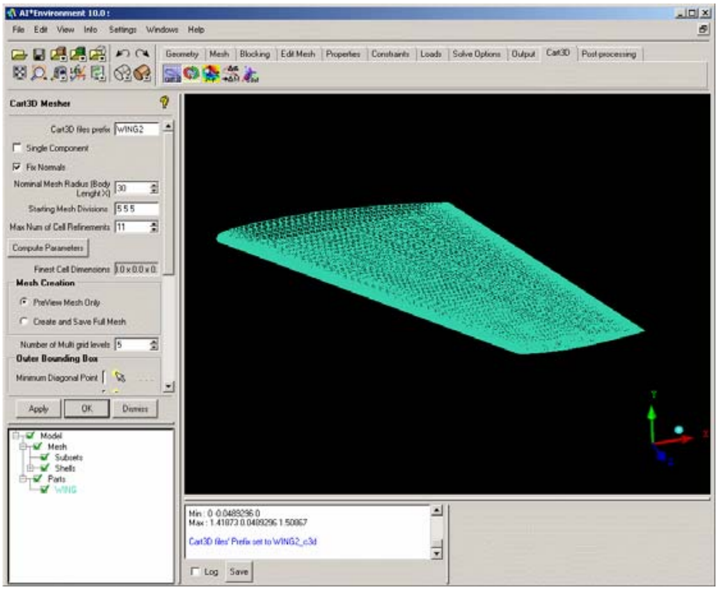
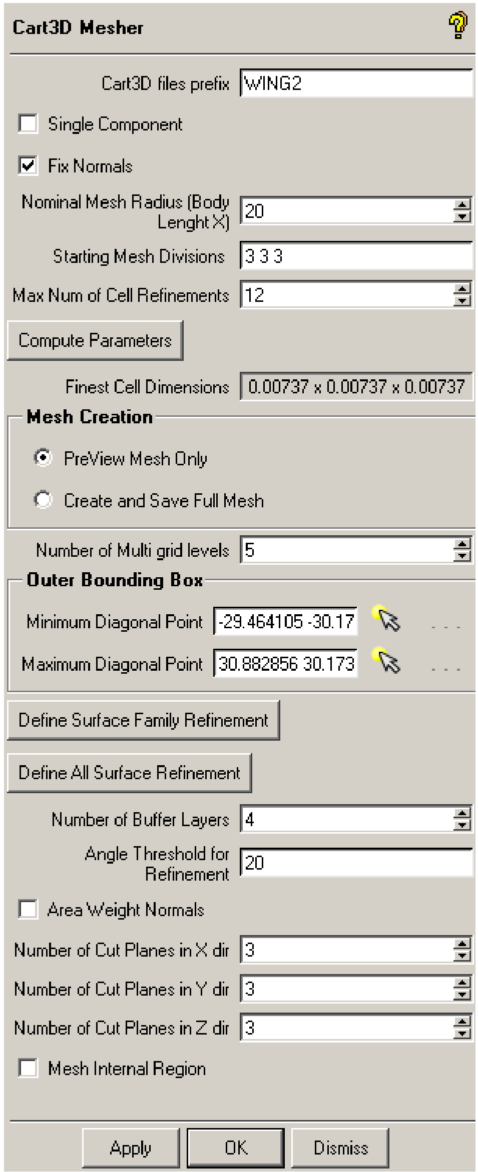


Figure 3.492 Cart 3D Mesher window

2. Leave Fix Normals enabled. This will fix orientation of the triangles such that their normals are pointing outward.
3. Choose Nominal Mesh Radius (Body Length X) = 20, Starting Mesh Divisions = 3 3 3 and Max number of Cell Refinements = 12
4. Click Compute Parameters. This saves the mesh in the local directory, converts in into Cart3D format, and finds the intersections if any. This is required to convert the triangulation to Cart3D tri format even if there is

only one component present. At the end, it displays the Finest Cell Dimensions as shown in Figure 3.493

Figure 3.493
Cart3D Mesher window



5. This will create 2 density polygons for mesh density control that can be seen by activating Geometries>Densities in the Display Tree widget.
6. This also computes the Finest Cell Dimensions**: 0.00737 x 0.00737 x 0.00737**. Varying the Starting Mesh Divisions and/or Max Num of Cell Refinements can vary these values.
7. The diagonal points displayed under the 'Outer Bonding Box' are the maximum and minimum points of the bounding box of the Mesh region. They can be changed if desired.
8. Set the Angle Threshold for Refinement to 5

Note: In this case we wish to run the case with symmetry in the Z direction. Specify bounding box minimum Z coordinate as 0.00001 (slightly inside the model). Refer to Figure 3.494. If the model itself is symmetric, turn on Half-Body Mesh (Symmetric in Z).

9) Click Apply (after specifying minimum Z coordinates as 0.00001) as shown in Figure 3.494 to run the mesher. This will create a domain file with 3 Cut Planes (Quad Elements) in each coordinate direction and Cut Cell (Hex Elements) The PreView mesh will be loaded automatically.

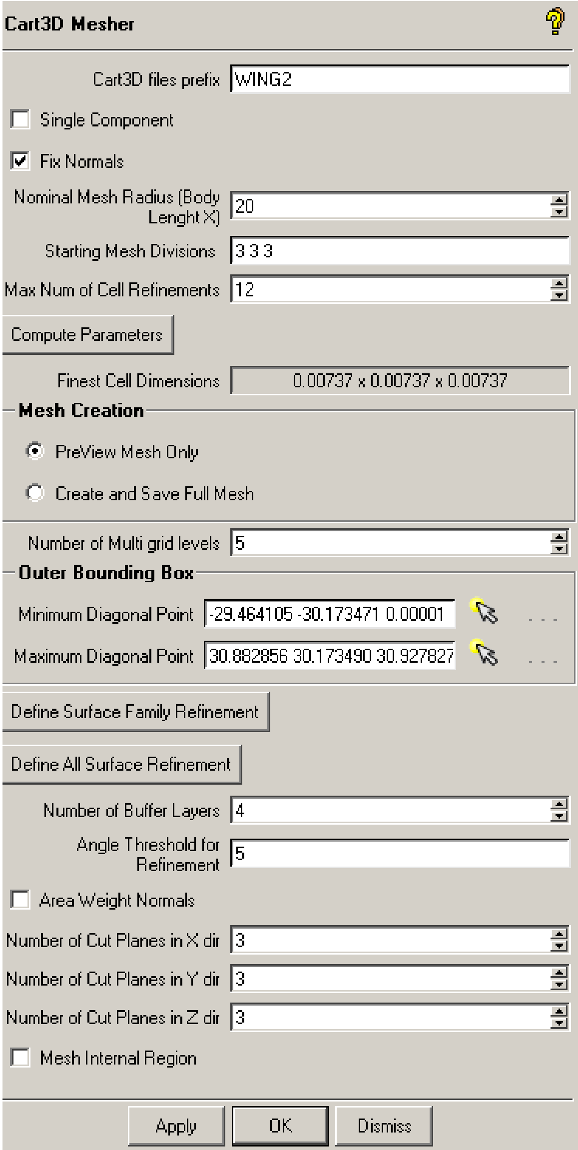
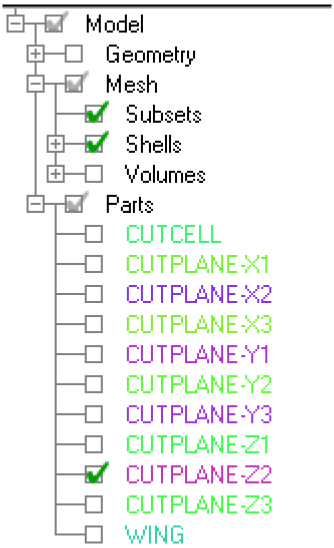


Figure 3.494 Change Angle of Refinement

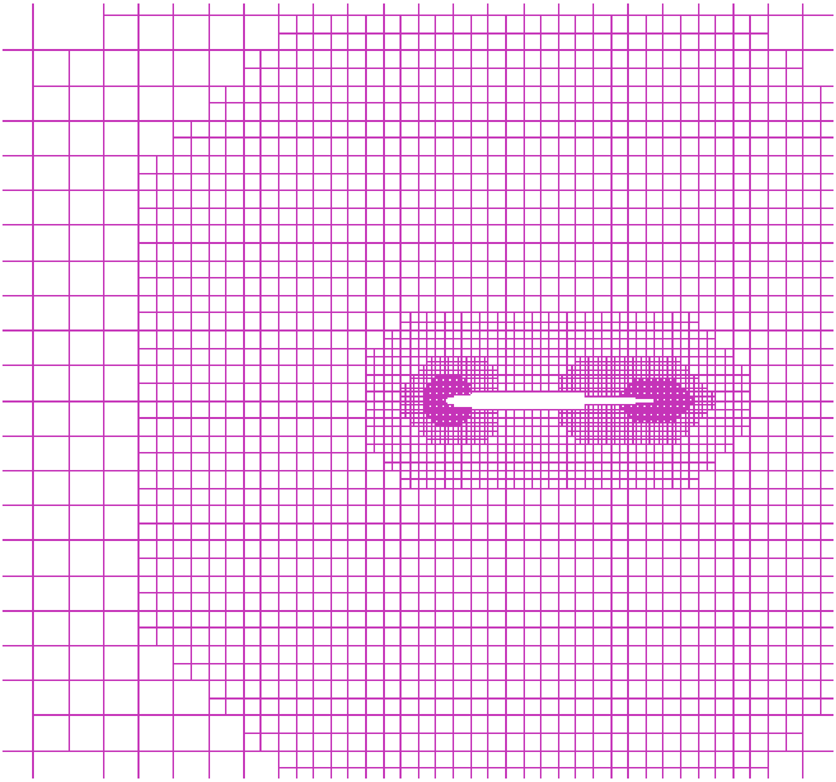
10. In the Parts menu under the Display Tree widget perform the operation Parts>Hide All (right-click on Parts to access) and then turn on only the Part **CUTPLANE-Z2** as shown in Figure 3.495.

Figure 3.495 Display Tree widget



11 The mesh projected onto the middle z-direction plane (in Part **CUTPLANE-Z2**) is shown in Figure 3.496.

Figure 3.496 CUTPLANE-Z2 Mesh

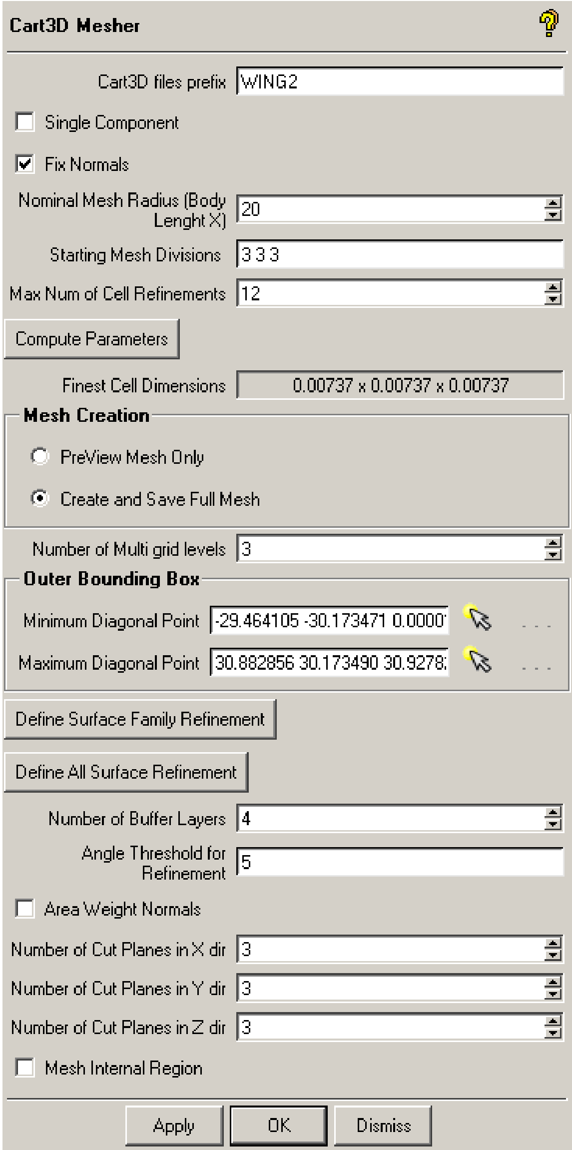


12. Perform the operation Parts>Show All by a right-click on Parts in the Display Tree widget after viewing the mesh.

c) Mesh Generation-Full Mesh

1. Now in the Cart3D mesher window enable Create and Save Full Mesh as shown in Figure 3.497.

Figure 3.497
Create and Save Full Mesh



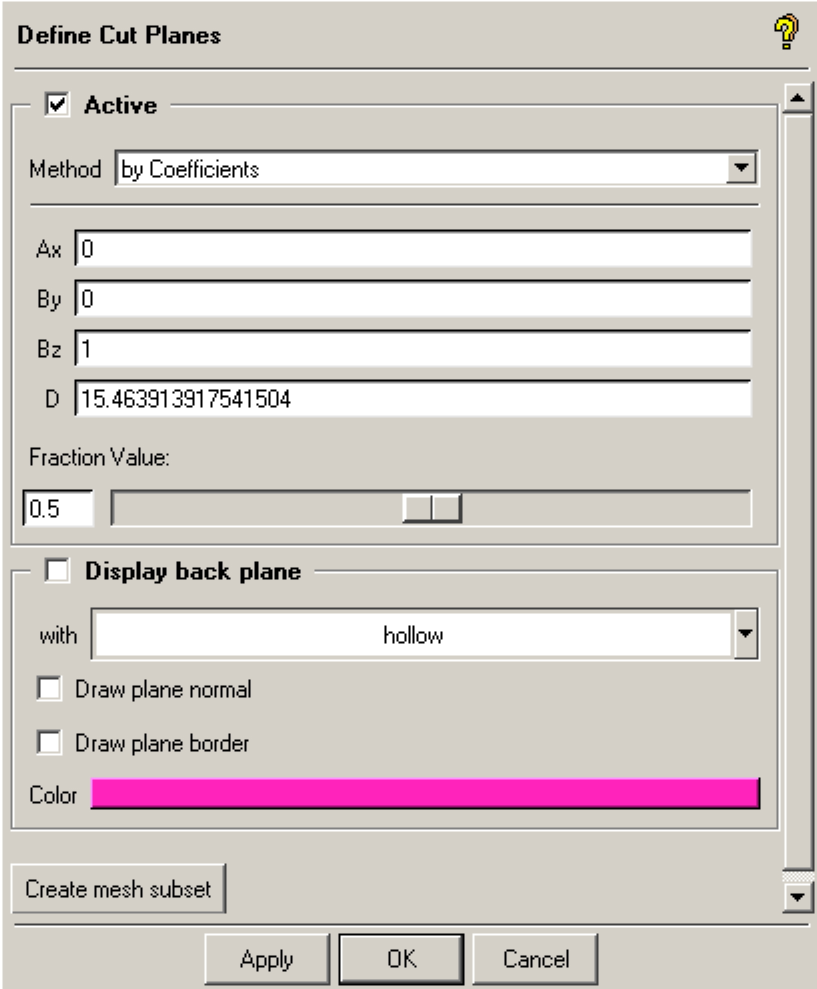
2. Set the Number of Multi grid levels to 3. This will create 3 levels of coarsened mesh, which can be read by the solver.

3. Press Apply. The Cart3D Mesh window appears which asks about loading the cart3D Full Mesh as shown in Figure 3.498. Press Yes.

Figure 3.498 Cart3D Mesh window

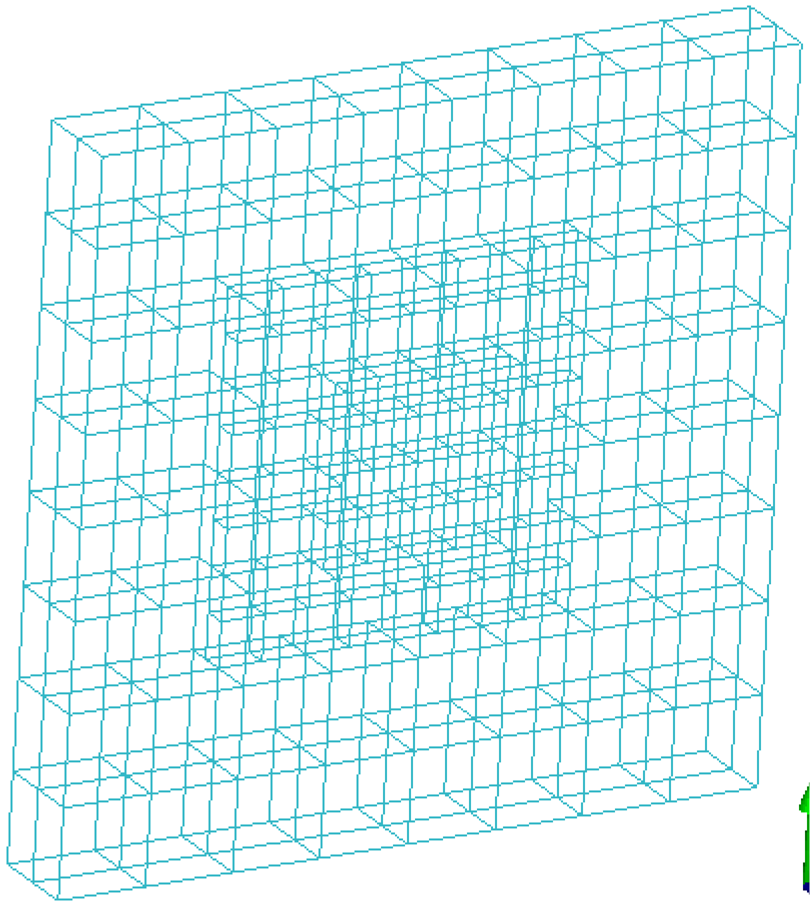
4 The final mesh generated can be examined through Mesh >Cutplane. The **Define Cut Planes** window appears as shown. Accept the default settings as shown in Figure 3.499.

Figure 3.499 Define Cut Planes Window



5. The mesh cut plane using the above parameters is shown in Figure 3.500.

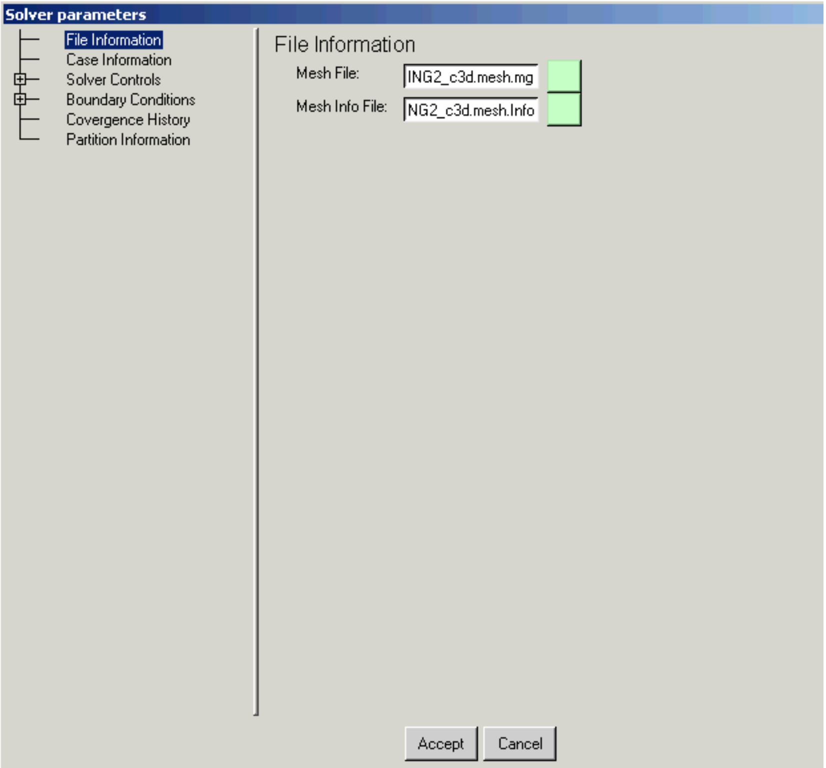
Figure 3.500 Cut Plane Mesh



d) Setup Flow Cart Parameters

1 In the Cart3D Menu select Solver . Click on Define Solver params  icon (if the panel doesn't open automatically). A **Solver parameters** window appears as shown in Figure 3.501.

Figure 3.501 Solver parameters window



2. Set File Information>Mesh File as **WING2_c3d.mesh.mg** (should be default)

3. Click on Case Information window and enter the following parameters

Mach Number = 0.84

Angle of Attack = 3.06

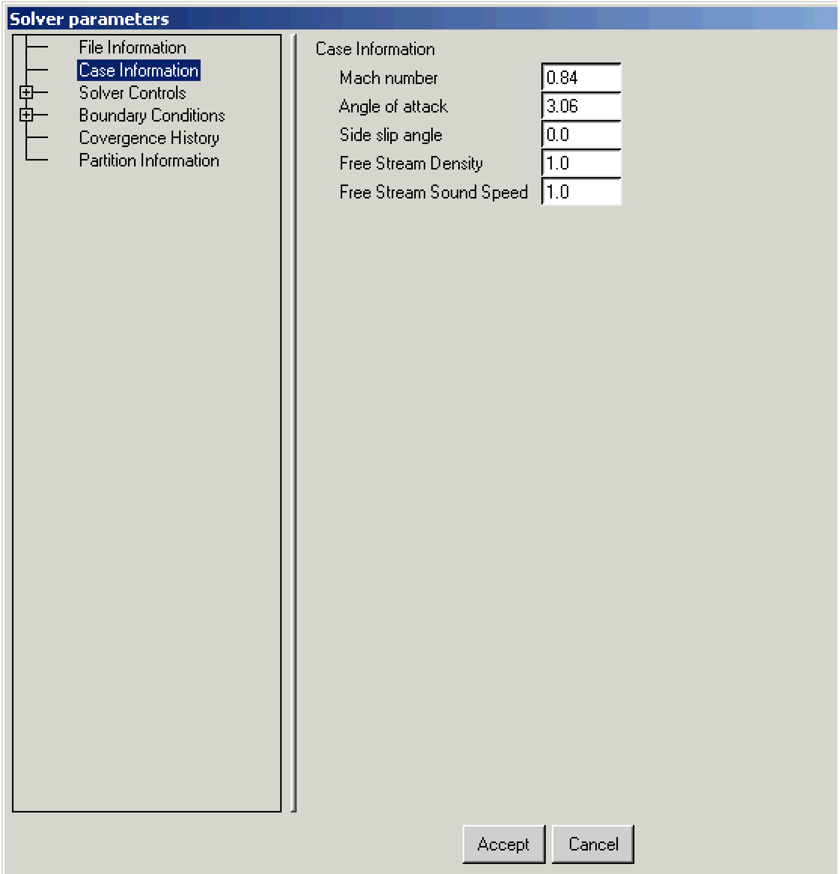
Side Slip angle = 0.0

Free Stream Density = 1.0

Free Stream Sound Speed = 1.0

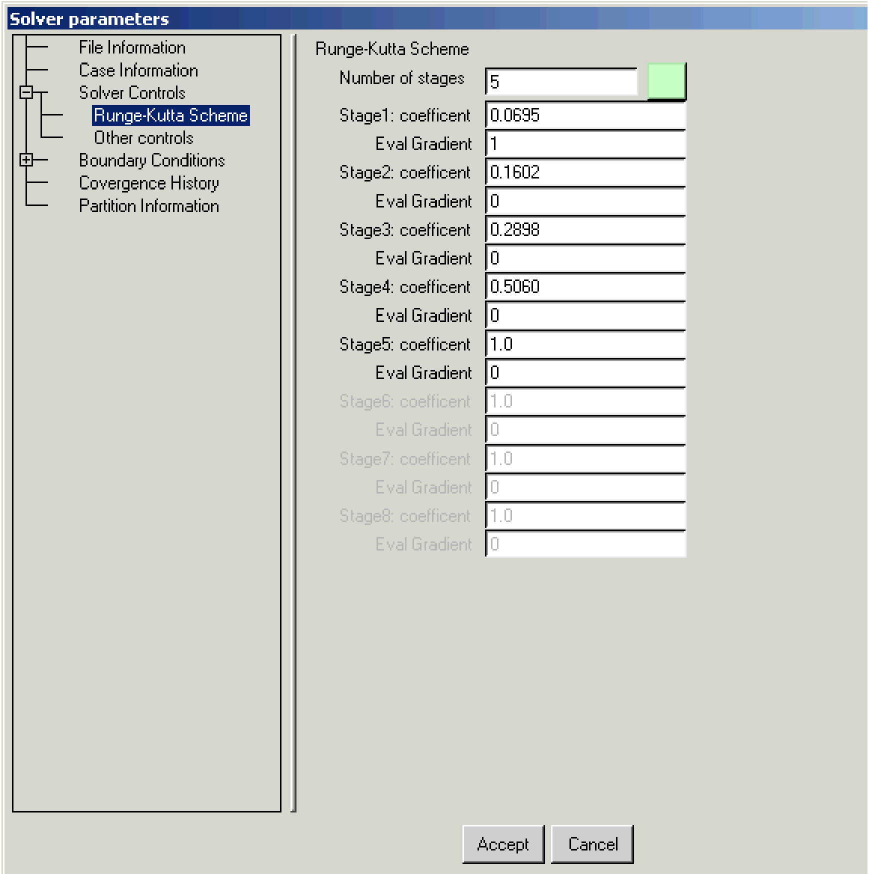
The values are shown in Figure 3.502.

Figure 3.502 Case Information window



4. Expand Solver Controls –Runge-Kutta Scheme in the Display Tree widget as shown in Figure 3.503 and accept the default settings.

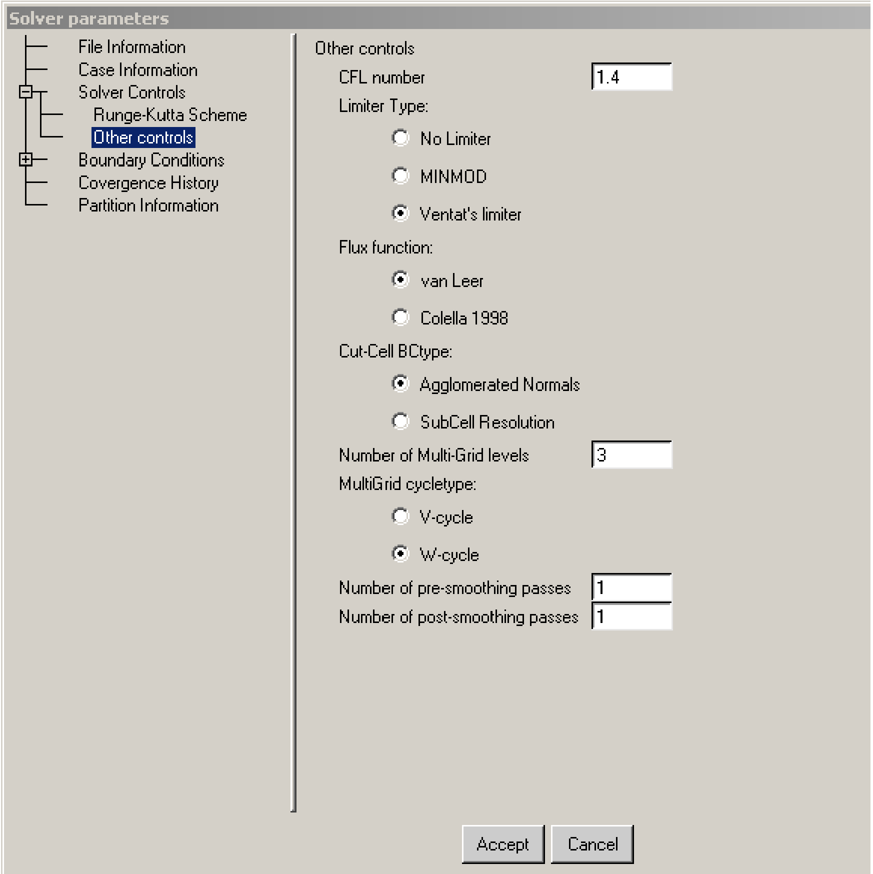
Figure 3.503 Runge-Kutta Scheme window



5. In **Other controls** specify the following values for the parameters as shown in Figure 3.504.

CFL number:	1.4
Limiter Type:	Ventat's Limiter
Flux function:	van Leer
Cut-Cell BCtype:	Agglomerated Normals
Number of Multi-Grid levels	3
MultiGrid cycletype	W-cycle
Number of pre-smoothing passes	1
Number of post-smoothing passes	1

Figure 3.504 Other Control window



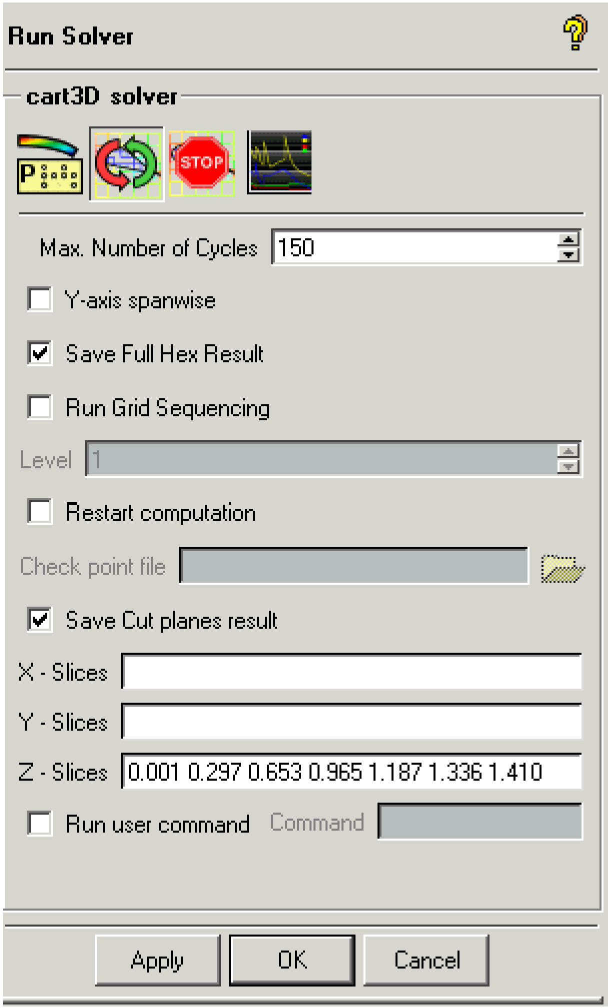
6. Keep defaults for Boundary Conditions, Convergence History and Partition Information and press Accept.

e) Running the FlowCart Solver

1. Select Solver  >Run Solver to open the flow chart solver panel (refer to Figure 3.505).
2. Specify Max. Number of Cycle=150
3. Turn on Save Full Hexa Result

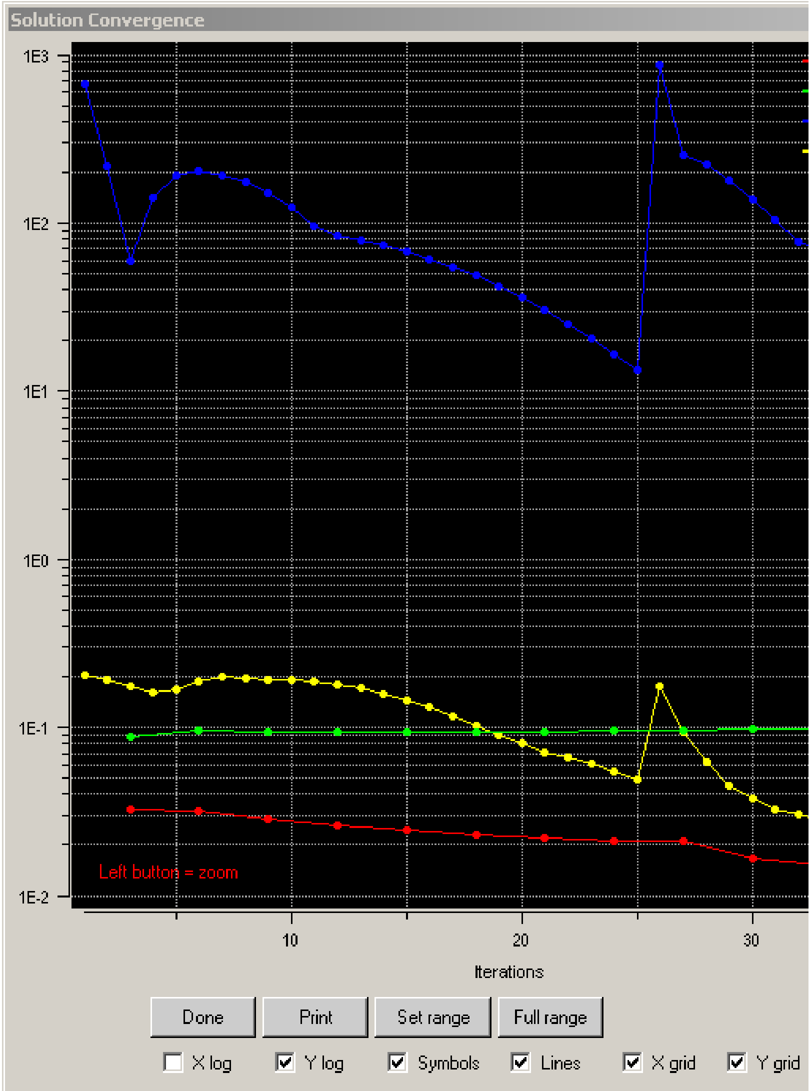
4. Turn on Save Cut planes result and specify Z-Slices as 0.001 0.297 0.653 0.965 1.187 1.3336 and 1.410
5. Click Apply and run the solver

Figure 3.505 Run Solver window



6. The user can view the convergence via the Convergence Monitor icon  as shown in Figure 3.506. (The monitor may open automatically.)

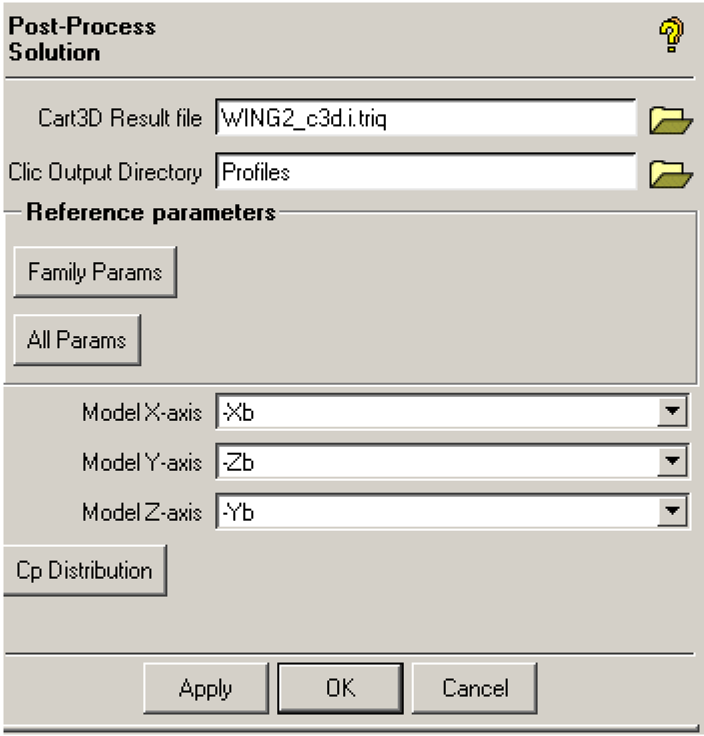
Figure 3.506 Solution Convergence Window



f) Computing Force and Moments

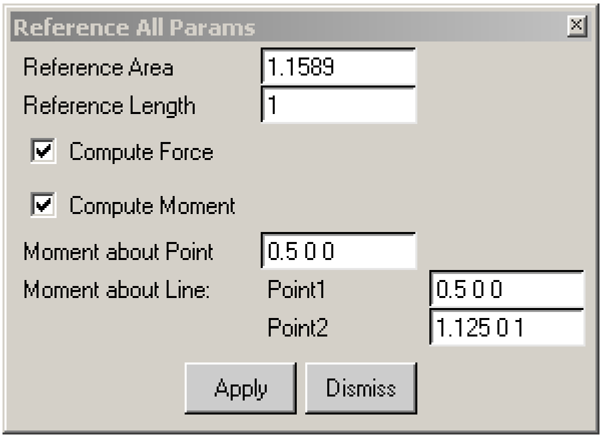
1. In the Cart3D main menu select Integrate Cp.  The Post-Process Solution window appears as shown in Figure 3.507.

Figure 3.507 Post-Process Solution window



2. Click **All Params** in the Post-Process Solution window.
3. In the Reference All Params window specify **Reference Area** as 1.1589, **Reference Length** as 1
4. Enable **Compute Force** and **Compute Moment**.
5. Set **Moment about Point** = 0.5 0 0, **Point1** = .5 0 0, and **Point2** =1.125 0 1.
6. Click Apply in the Reference All Params window and then Dismiss to close as shown in Figure 3.508

Figure 3.508 Reference All Params window



7. Press Apply in the Post-Process Solution window. The results appear in the GUI messages area.

g) Visualizing the results

1. Flow charts write three output files

i). WING2_c3d.i.triq - Contains Pressure, Velocity and Density extrapolated to the Surface triangles. This can be converted to a domain file by Edit>Cart3D Tri File->Domain file. The default resultant domain file will be WING2_c3d.uns

ii). **slicePlanes.dom** - Cut Plane results

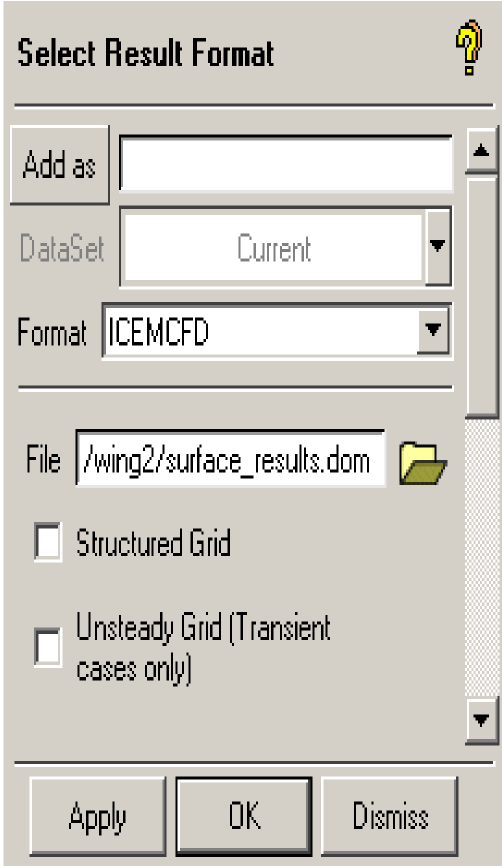
iii). **results.dom** - Full mesh result

2. Go to File > Results > Open Results…

3. Select Format as ICEMCFD.

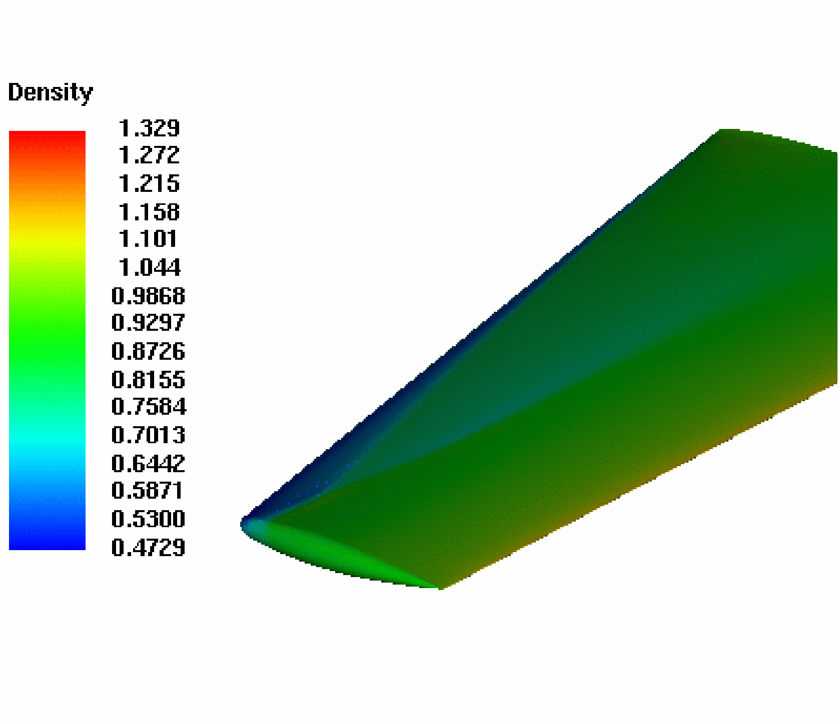
4. Specify surface_results.dom as the File as shown in Figure 3.509.

Figure 3.509
Result File Format Window



5. Select Apply from the panel to get the default result as shown in Figure 3.510. Right click on Color map from the Display Tree widget and select Modify Entries to adjust the Min and Max values for the displayed variable.

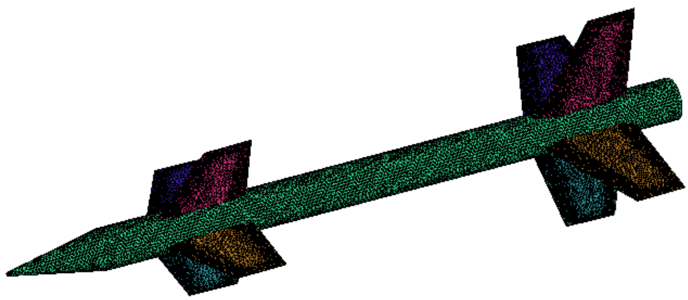
Figure 3.510 Post Processing Result



3.7.4: Supersonic Missile

Overview

This example illustrates how to generate a grid in Cart3D around a supersonic missile and to solve the problem in flowCart. Post-processing the results is also explained.



The tutorial introduces the following operations:

Multigrid preparation - running reorder and mgPrep.

Obtaining surface triangles from geometry data.

Running the solver for AOA = 5 and Mach = 3

Computing force and moment information.

Visualizing the results in Post-processing.

a) Starting the Project

Load **ANSYS ICEM CFD**. Change the working directory by File > Change Working Dir… and set the location to the folder **missile**.

Note: It is preferable to create a separate folder **missile** and put the missile.tin (geometry) file in that folder before performing this tutorial.

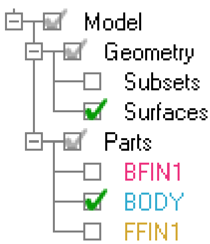
b) Creating Faceted Data from Geometry

The model has a **Fuselage**, **Front Fins** and **Back Fins**. The fuselage can be considered as one component and each fin as one component. First, the surface triangulation for the fuselage will be created. The tetra mesher can be run separately for each component. Thus, any unforeseen difficulty in creating the surface triangulation the model as a whole will be avoided. The user has to create a separate tetin file for each component.

Note: To skip the surface triangle generation process, proceed the Mesh Generation Preview Only. Also put the missile.uns (domain) file into the **missile** directory.

Load the tetin file missile.tin.
In the Display Tree, switch on Surface > Solid and Wire.
Under Parts in the Display Tree, select Parts > Hide All and switch on the Part Body as shown in Figure 3.511.

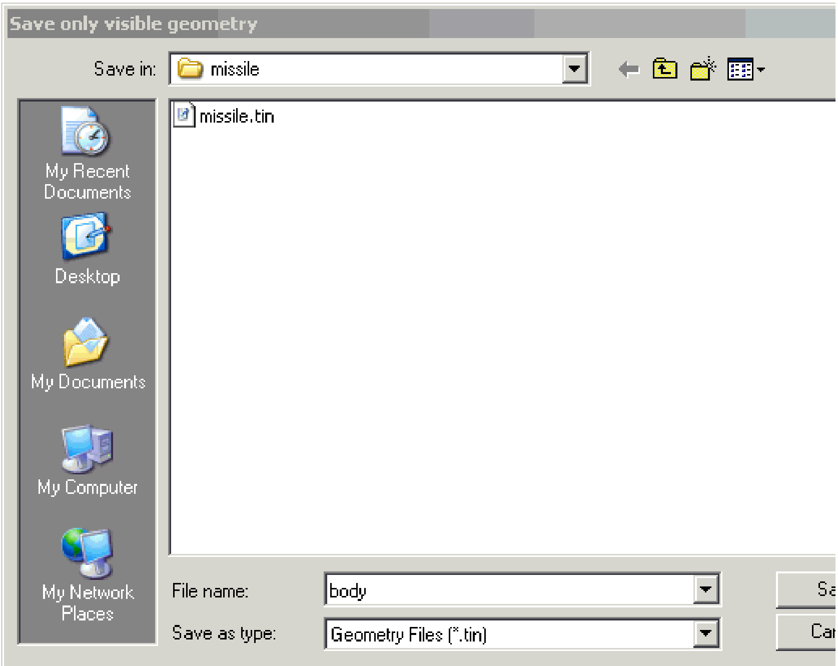
Figure 3.511 Display Tree widget



Save only the visible geometry to a new tetin file using File > Geometry > Save Visible Geometry As… A window appears as shown in Figure 3.512. Specify the file name as body.tin. Select Save.

Note: Don't save it as missile.tin as we will lose the rest of the geometry data.

Figure 3.512 Save Only Visible Geometry window



Similarly save the front fins and back fins under the file names ffin.tin and bfin.tin by displaying the FFIN1 and BFIN1 parts respectively.

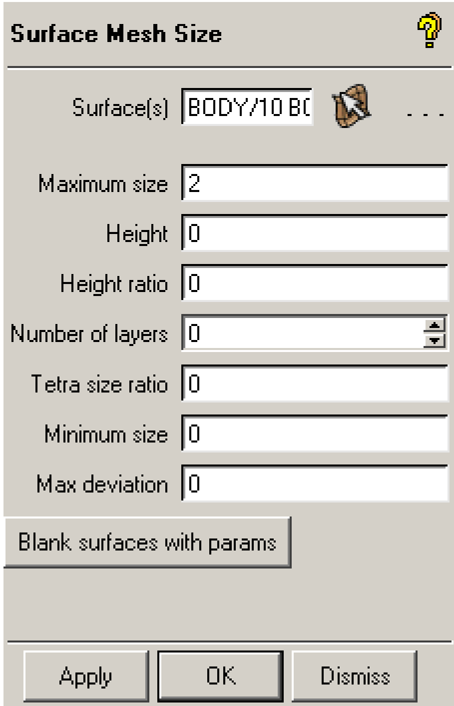
6. Now close the current tetin file using File > Geometry > Close Geometry.

c) Generating Surface mesh on Body

Load the tetin file body.tin (the fuselage) with File > Geometry > Open Geometry.

From the Mesh tab, select Set Surface Mesh Size.  Click the Select surface(s) icon and click 'a' on the keyboard to select all the surfaces. Specify a Maximum size of 2 as shown in Figure 3.513.

Figure 3.513
Surface Window



Click the Select surface(s) icon and select the hemispherical surface at the tip (see Figure 3.514) with the left mouse button. Middle-click to accept the selection. Specify a **Maximum size** of 0.25 as shown in Figure 3.515.

Figure 3.514 Surface Selected

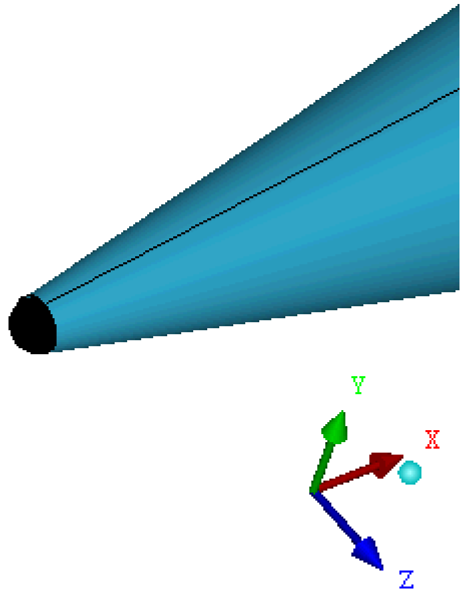
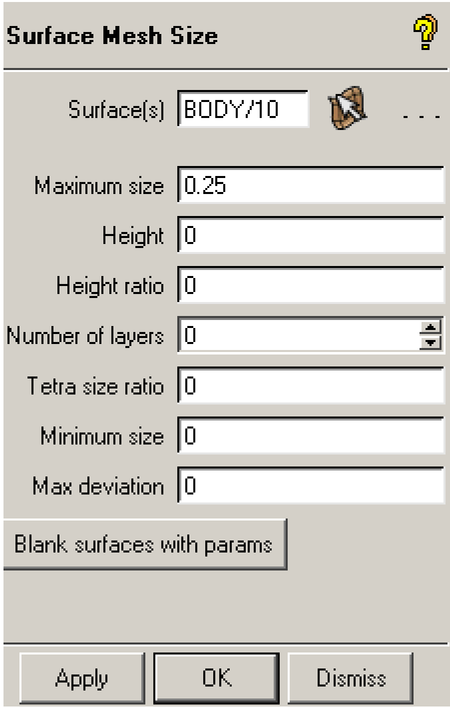
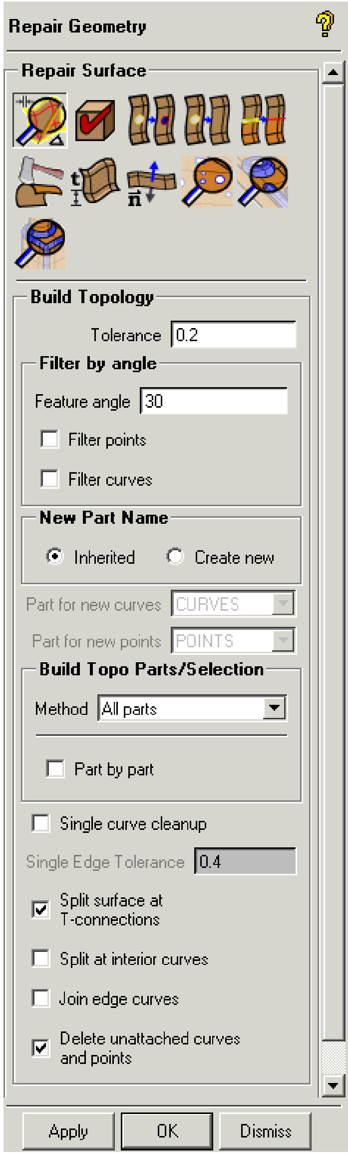


Figure 3.515 Surface Mesh Size window



Extract the hard curves and points on the geometry using Build Diagnostic Topology. Select Geometry>Repair Geometry.  The Repair Geometry window opens in the default option of Build Diagnostic Topology as shown in Figure 3.516. Use the defaults and press Apply.

Figure 3.516
Build Topology Window



Save the tetin file using File > Geometry > Save Geometry.

Note: By default it saves the geometry file to body.tin.

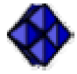
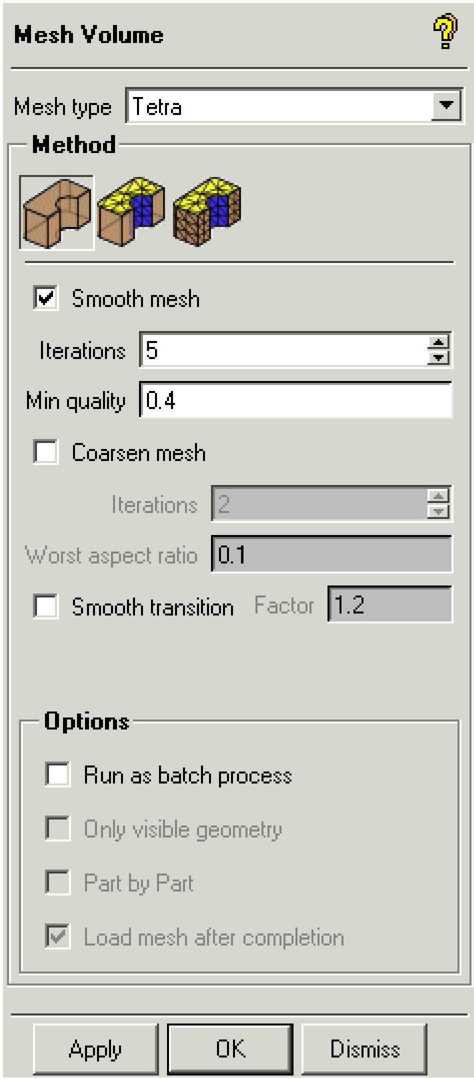
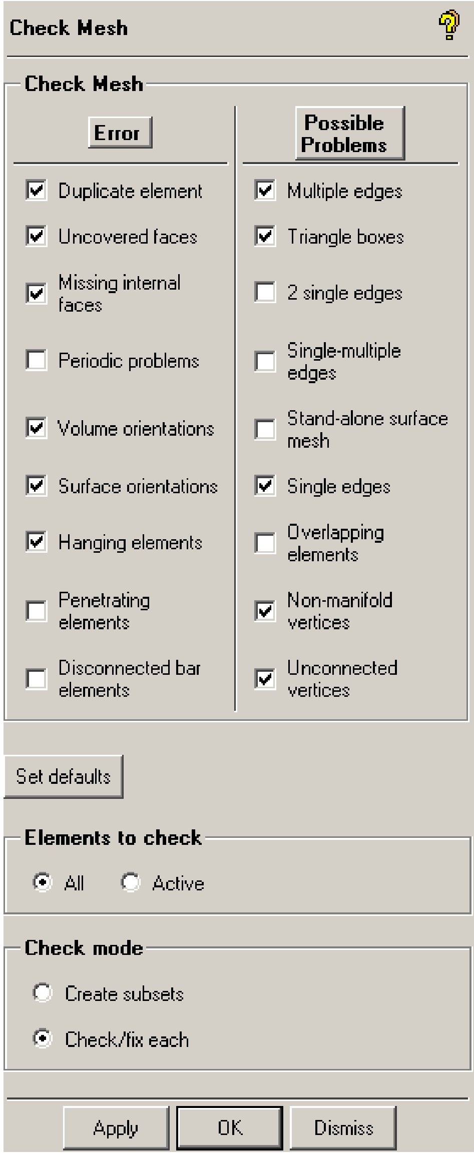
Run Tetra from Mesh > Volume Meshing . Select Tetra as the Mesh type and From geometry . The **Mesh Volume** window appears as shown in Figure 3.517. Accept the default settings and press Apply.

Figure 3.517
Mesh Volume window



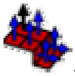
From the Edit Mesh tab select Check Mesh.  The Check Mesh window appears as shown in Figure 3.518. Accept the default settings and press Apply.

Figure 3.518
Check Mesh window



In the Diagnostic window it asks to Delete the unconnected vertices. Press Yes.

Expand the Mesh branch in the Display Tree widget. Right-click on Shells and select Solid and Wire. Similarly select Face Normals for Shells.

From the Edit Mesh tab select Reorient Mesh . Select Reorient Consistent. . The user is automatically placed into selection mode and prompted to select a shell element. Select one element whose normal is facing outward (or an element colored by the Part name color) and middle-click to accept.
Under the Mesh branch of the Display Tree widget, make sure all types are active EXCEPT Shells.

Note: Note that Cart3D requires only Triangles in the Mesh file so other mesh entities like Points, Lines and Volumes need to be deleted.

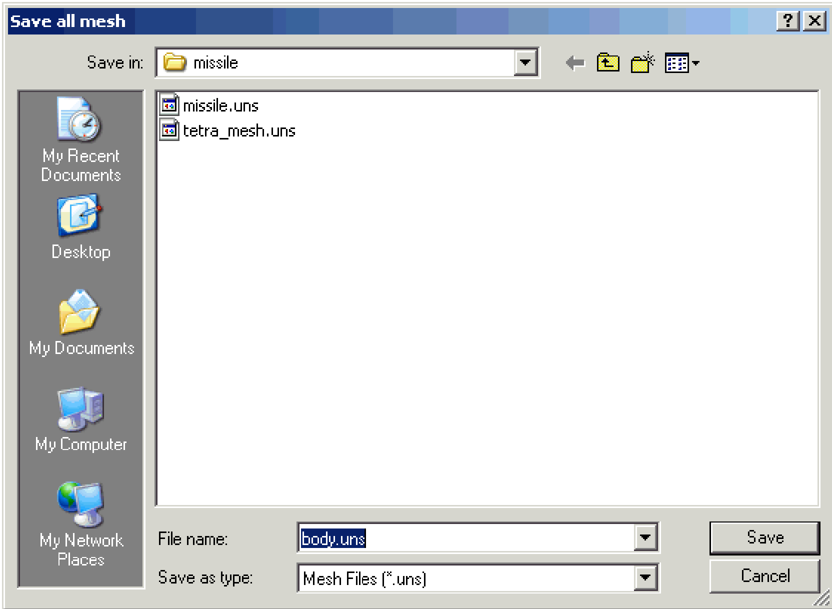
From the Edit Mesh tab select Delete Elements . If not already placed into selection mode, from the Delete Elements window click Select Element(s).  In the Select mesh elements window click on Select all appropriate visible objects . All elements but Shells are deleted. Either middle-click or press Apply to finish.
Activate Shells from the Display Tree widget, and de-select Shells>Face Normals.

Note: Now there are only triangular Surface Mesh Elements as required for Cart3D.

Go To File>Mesh>Save Mesh As. Specify the name as body.uns in the **Save all Mesh** window as shown in Figure 3.519 and press Save.

Note: User should only use the Save Mesh As option

Figure 3.519 Save All Mesh window



Select File>Geometry>Close Geometry and File>Mesh >Close Mesh.

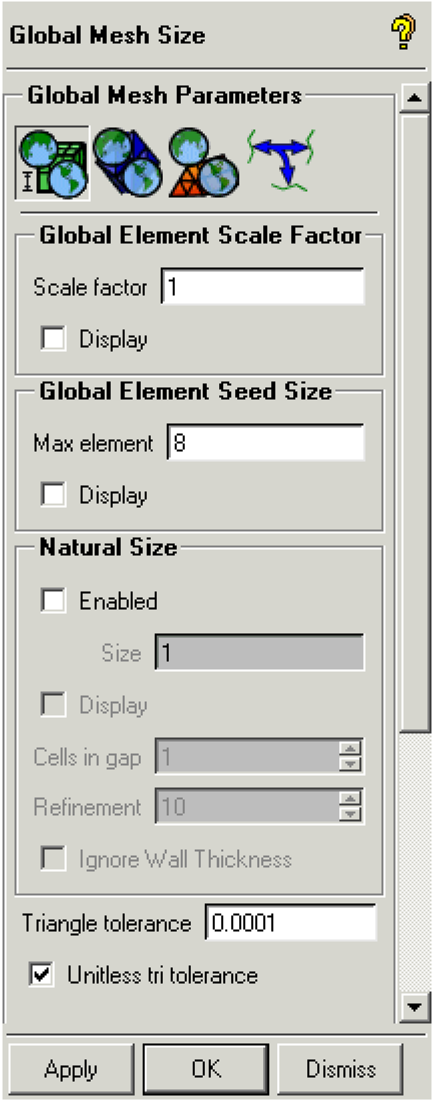
d) Generating Surface Mesh on Front and Back Fins

Load the geometry file ffin.tin via File > Geometry > Open Geometry, and select ffin.tin.

Repair the Geometry (Build Diagnostic Topology) in the same way as for body.tin (Figure 3.516). Use the default values (which may be different from the previous geometry).

To remove the possibility of elements jumping from one side of the fin to the other, from the Mesh tab choose Set Global Mesh Size  and General Parameters. . Enter Max element as 8 (see Figure 3.478) and press Apply. Select Tet Meshing Parameters  and set Edge criterion to 0.05 as shown in Figure 3.521. Press Apply.

Figure 3.520: Global mesh size window



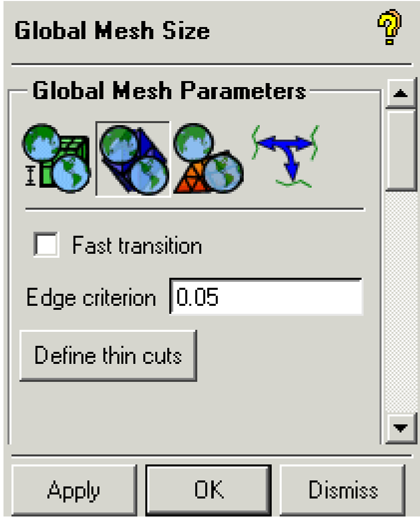
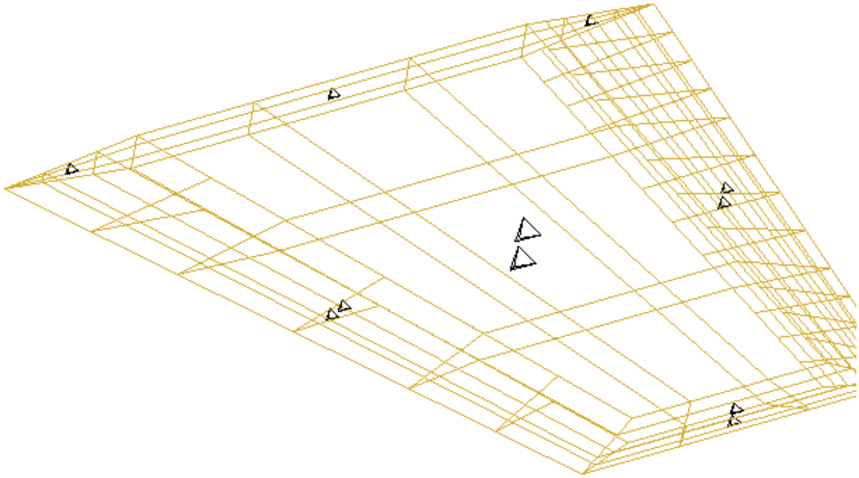


Figure 3.521 Global Mesh Size:Tetra meshing parameters window

From the Mesh tab select Set Surface Mesh Size.  Use **Maximum size** of 1 for the tip/leading/trailing surfaces and 2 for the remaining surfaces.

Note: To display the applied mesh sizes, right-click on Surfaces in the Display Tree widget and select Tetra Sizes as shown in Figure 3.522.

Figure 3.522 Tetra Size



Run the Tetra Volume Mesher and save only the Surface Triangular mesh as done previously for body.uns, assign the name ffin.uns. (Remember to delete the other mesh elements and to run the mesh checks before saving the Shell element mesh. Also be sure to align the element normals to point outwards.) Close the geometry and mesh files.

Repeat the same process for bfin.tin and sand save the surface triangular mesh as bfin.uns. Close the geometry and mesh files.

e) Working on the entire display

Load the following domain files: body.uns, bfin.uns, and ffin.uns. Select Merge as prompted to merge all together.

There is only one fin at the front and one at the back. We will copy and rotate these fins to get the remaining three sets. Normally it is easier to do this at geometry level.

Select Edit > Mesh > Facets in the main menu. The Create sharp features window appears as shown in Figure 3.523. Press No.

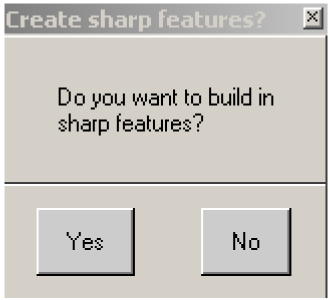


Figure 3.523 Create Sharp Features window

Then the **Merge** window appears which asks us to Merge close nodes in the surfaces as shown in Figure 3.524. Press No.

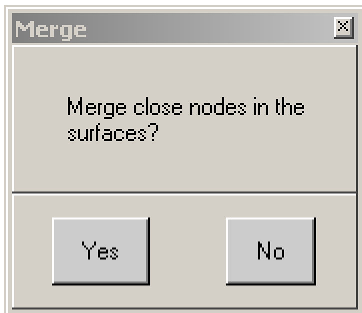


Figure 3.524 Merge window

In the Display Display Tree widget switch off Mesh and switch on Surfaces > Solid.

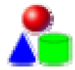
6. Select File > Mesh > Close Mesh and say No in the **Save Mesh** window.

Note: The main purpose of this tutorial is to create faceted surface, which can be converted to mesh triangles at the end of the process. It is advisable to perform the operation Parts > Reassign Colors > ”Good” colors for a good view of the geometry

From the Geometry tab select Transform Geometry  and Rotate Geometry .

In the window press Select entities.  A Select Geometry window pops up as shown in Figure 3.525.

Figure 3.525 Select Geometry window

In the Select geometry window press Select items in a part  . The Select part window appears as shown in Figure 3.526. Select BFIN1 and FFIN1 and press Accept.

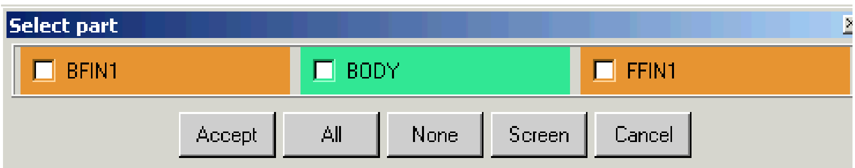
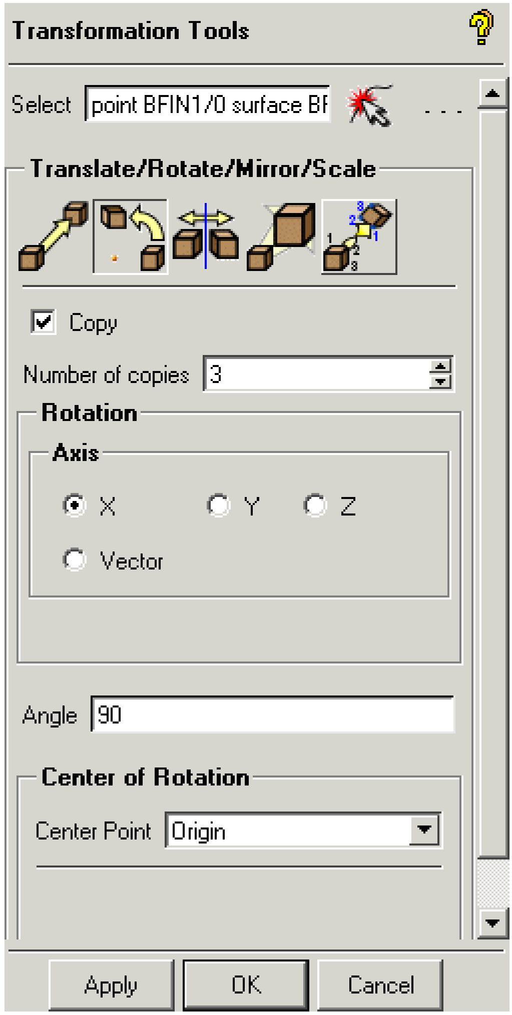


Figure 3.526 Select Part window

In the **Transformation Tools** window enable **Copy**; enter Number of copies as 3, select X for Axis, Angle = 90 and Center Point as Origin as shown in Figure 3.527. Then press Apply.

Figure 3.527 Transformation Tool window



The geometry of BFIN1 and FFIN1 gets rotated as shown in Figure 3.528.

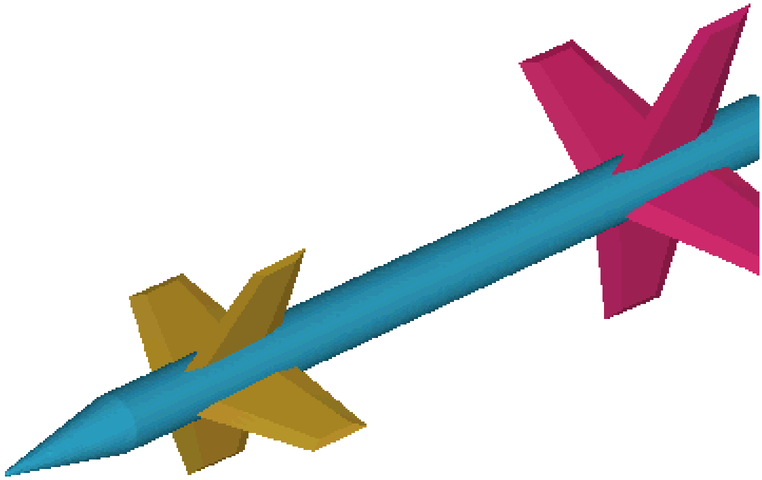


Figure 3.528 After Rotation.

Switch off BODY and BFIN1 from the Display Display Tree widget and select View > Left from the main menu.
In the Display Tree right-click on Parts and select Create Part. Enter FFIN2 as the Part name as shown in Figure 3.530. Select Create Part by Selection,  and select the region as shown in Figure 3.529. Middle-click to accept.

Figure 3.529 Region selected

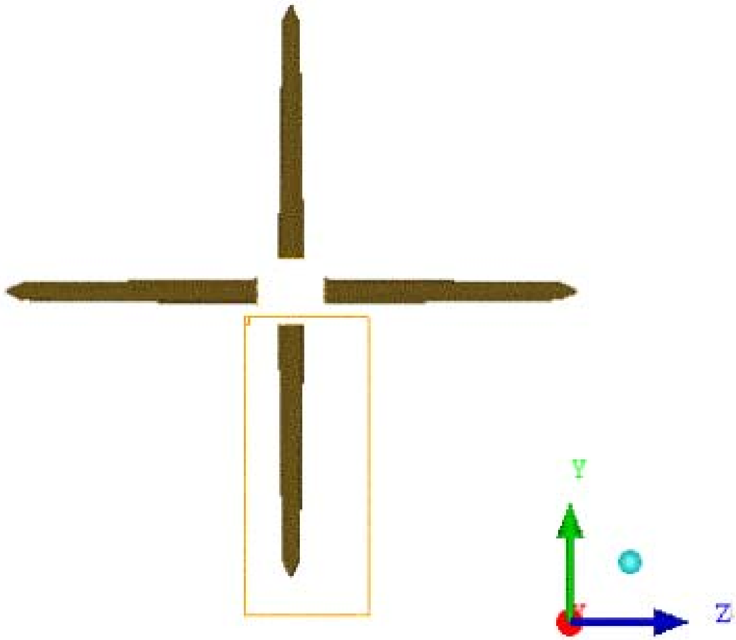
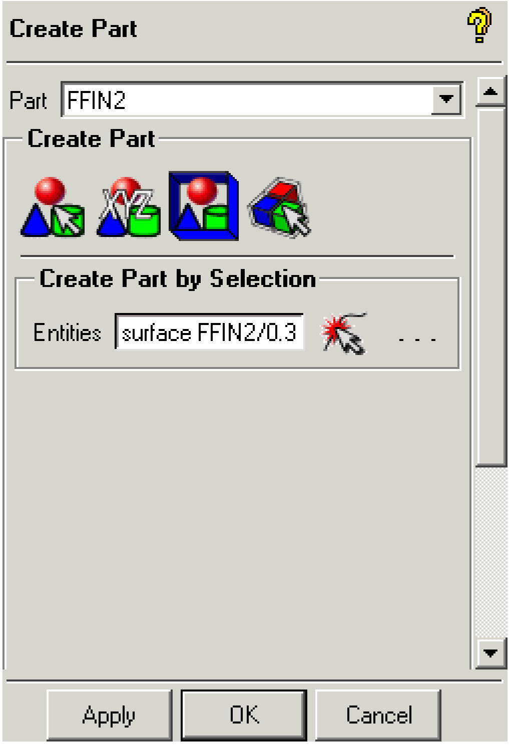


Figure 3.530
Create Part window



Similarly move each fin into a different Part with the front fins in FFIN1, FFIN2, FFIN3, and FFIN4; and the back fins in BFIN1, BFIN2, BFIN3, and BFIN4.

Note: It is better to keep each component in separate Parts.

The final geometry image, which we get after the Part assignments, is shown in Figure 3.531.

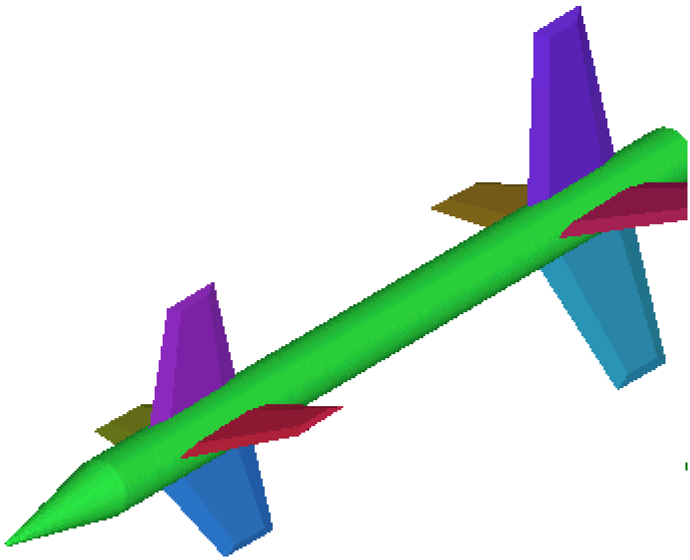
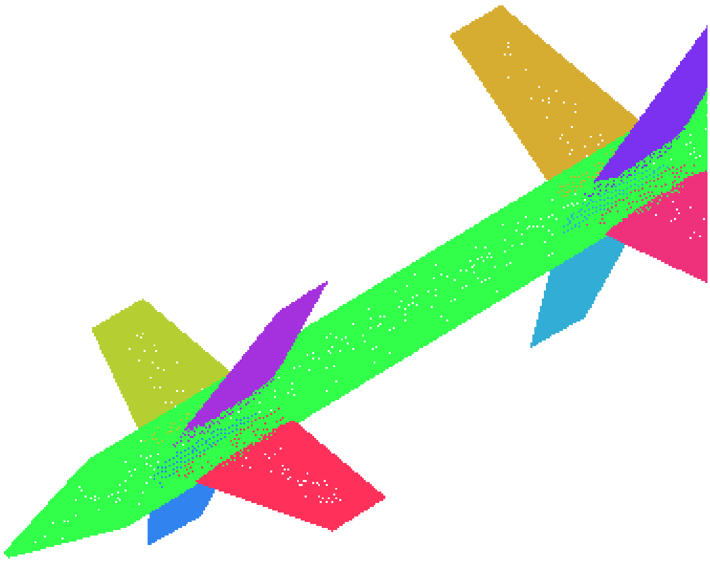


Figure 3.531 After Part Assignment

From the main menu select Edit > Facets > Mesh that would give us the desired mesh as shown in Figure 3.532.

Figure 3.532 Faceted Mesh

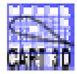


Note: Keep the original missile.uns at some other location so that user who wants to start this tutorial from the **Mesh Generation Preview Only** step can load the missile.uns file without disturbing the original file.

Save the mesh under the name missile.uns and close the geometry.

f) Mesh Generation Preview only

Note: Users are encouraged to use the domain file created in the above section to run Cart3D. Otherwise they can use the domain file **missile.uns** available with the tutorial.

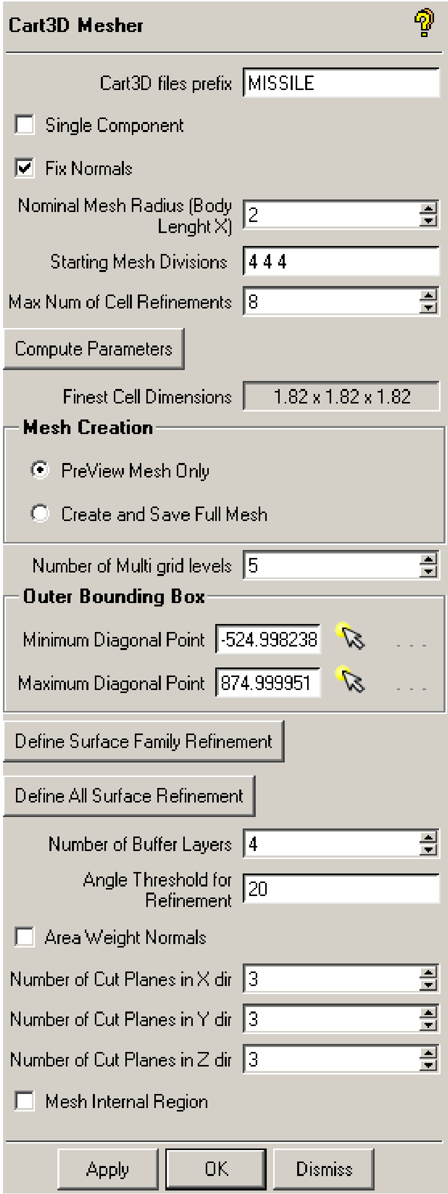
Click on Cart3D from the main menu. Select the Volume Mesher  Icon.

Leave Fix Normals enabled to ensure the triangle normals point outwards.

Set Nominal Mesh Radius (Body Length X) = 2, Starting Mesh Divisions = 4 4 4 and Max Num of Cell Refinements = 8.

Click Compute Parameters. This saves the mesh, converts it to Cart3D format, and finds the intersections. At the end, it displays the Finest Cell Dimensions as shown in Figure 3.533.

Figure 3.533 Cart3D Mesher window



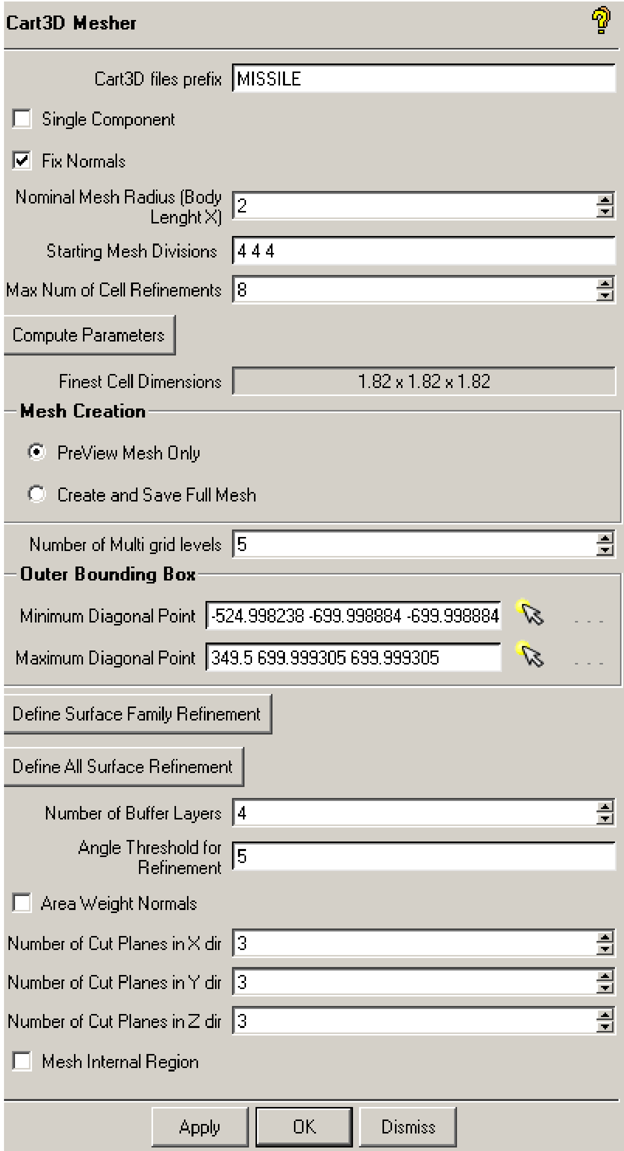
This will create 10 density polygons for mesh density control, which can be viewed in the Display Tree widget by activating Geometry >Densities.

This also computes the finest cell size: **1.82 x 1.82 x 1.82**. Varying the starting mesh division and/or Max number of cell refinements can vary this.

The diagonal points displayed under Outer Bounding Box are the minimum and maximum points of the mesh region bounding box (refer to Figure 3.533). For supersonic computations, choose the downstream boundary at the end of the body. This better represents the experimental setup as in most wind tunnel tests, the missile will be supported at the back of the body. The fuselage expands from 0 to 350 in the X direction so change the X coordinate in Maximum Diagonal Point to 349.5.

Set the Angle Threshold for Refinement to 5 as shown in Figure 3.534.

Figure 3.534 Angle of refinement changed



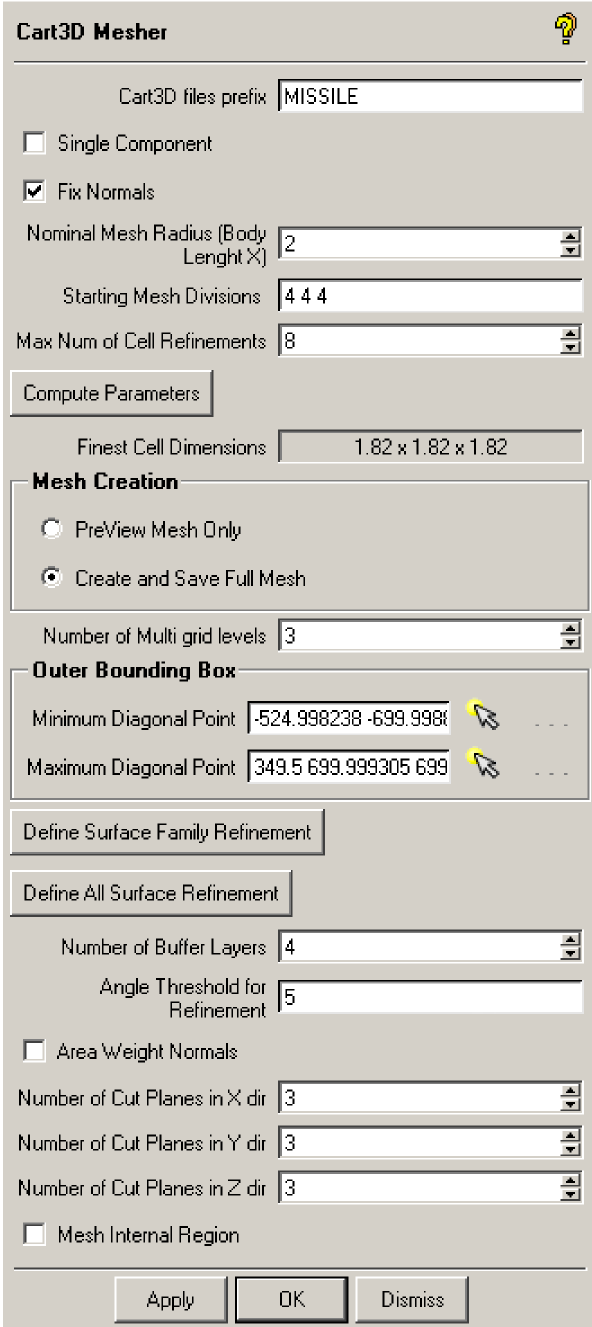
Click Apply to run the mesher. This will create a domain file with 3 Cut Planes (Quad Elements) in each coordinate direction and Cut Cells (Hex Elements). The Preview mesh will be loaded automatically.

Note: As in previous tutorials the mesh can be viewed by switching on the Cut Plane to be viewed.

g) Mesh Generation Full Mesh

Now in the Cart3D Mesher window enable Create and Save Full Mesh and change the Number of Multi grid levels to 3 as shown in Figure 3.535. This will create 3 levels of coarsened mesh, which can be read by the solver.

Figure 3.535 Create and Save Full View.



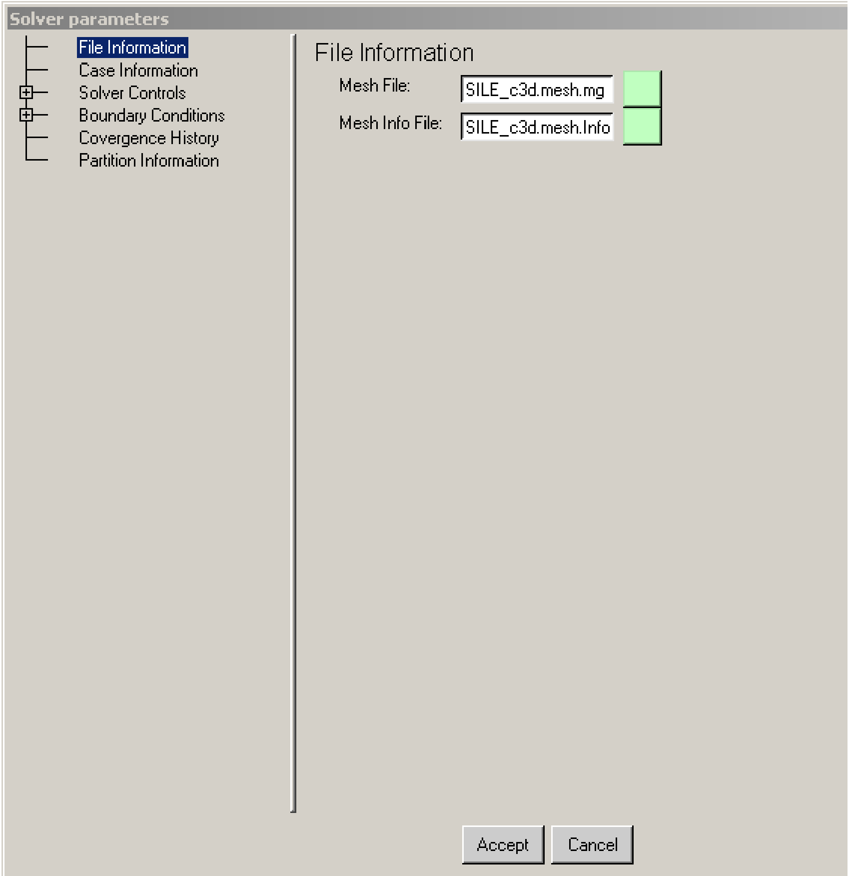
Press Apply. The Cart3D Mesh window appears which asks us to load the cart3D Full Mesh. Press Yes

Note: The final mesh generated can be examined through Mesh > Cut Plane as in the previous tutorials.

h) Setup Flow Cart Parameters

In the Cart3D Menu select Solver.  Click on Define solver params  icon (if the panel doesn't open automatically). The Solver parameters window appears as shown in Figure 3.536.

Figure 3.536 Solver Parameter window



Choose File Information > Mesh File as **MISSILE_c3d.mesh.mg** (this should be the default).

Click on Case Information and enter the parameters as shown in Figure 3.537.

Mach number = 3

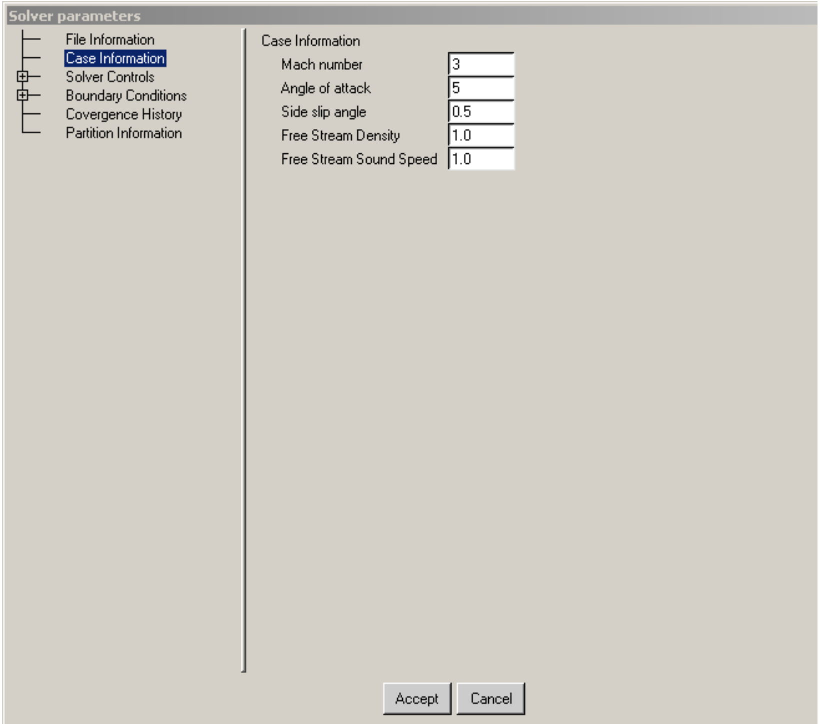
Angle of attack = 5

Side slip angle = 0.5

Free Stream Density = 1.0

Free Stream Sound Speed = 1.0

Figure 3.537 Case Information window



Expand Solver Controls > Runge-Kutta Scheme and evaluate the coefficient only at the first stage as show in Figure 3.538.

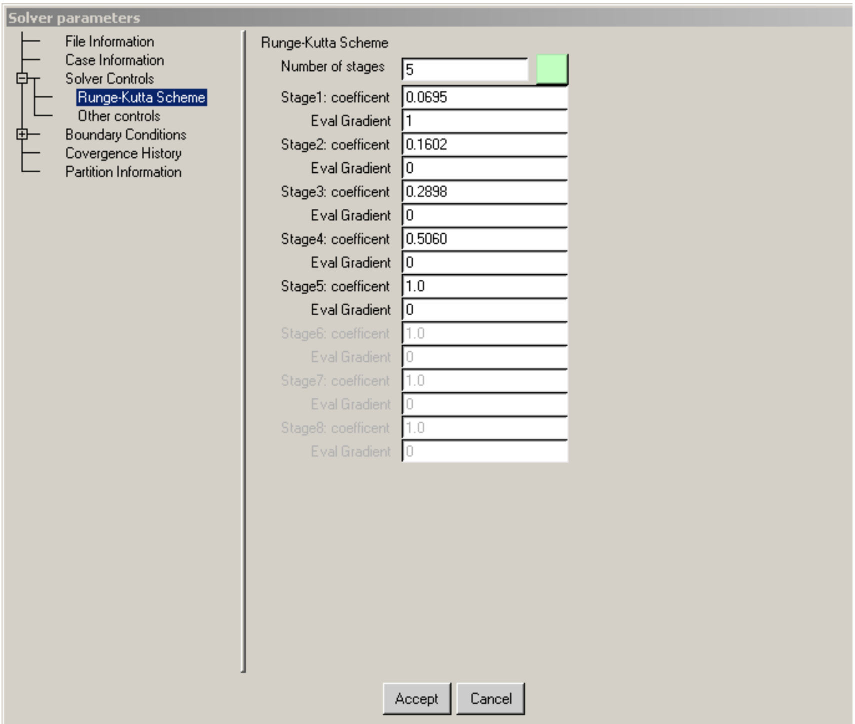
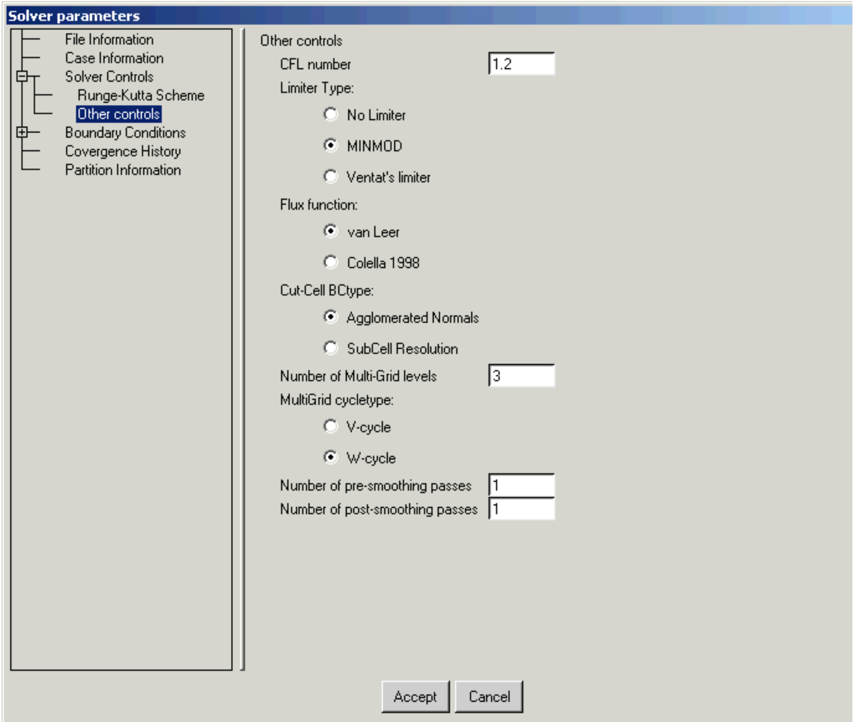


Figure 3.538 Runge Kutta Scheme window

In Other controls specify the following values for the parameters as in Figure 3.539.

CFL number: 1.2
Limiter Type: MINMOD
Flux function: van Leer
Cut-Cell BCtype: Agglomerated Normals
Number of Multi-Grid levels: 3
Multi Grid cycle type: W-cycle
Number of pre-smoothing passes: 1
Number of post-smoothing passes: 1

Figure 3.539 Other Control Window



Expand Boundary Conditions and choose Directional BC for the enclosing Cartesian box. In this case all six faces have will have the FARFIELD boundary condition as shown in Figure 3.540.

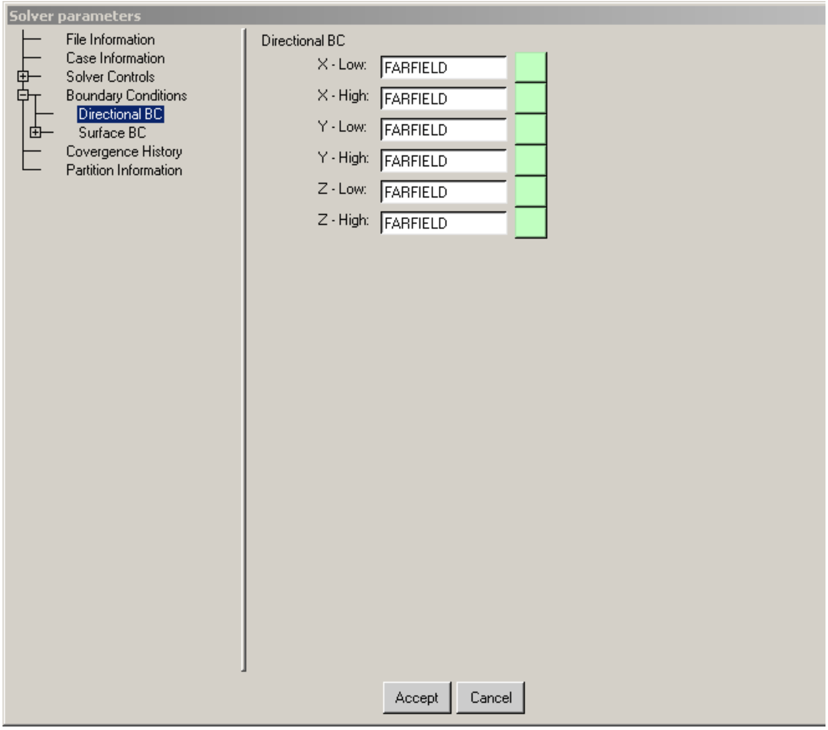


Figure 3.540 Directional BC window

Keep the defaults for Convergence History and Partition Information
Click Accept to save the parameters.

i) Running the FlowCart Solver

Select Solver  >Run Solver  to open the FlowCart solver panel (refer to Figure 3.541).
Specify **Max. Number of Cycles** = 200.
Enable Run Grid Sequencing and set Level = 3.
Enable Save Full Hex Result.
Turn on Save Cut planes result and specify X-Slices as 50, 100, 200, and 300; and Y- and Z-Slices at 0.

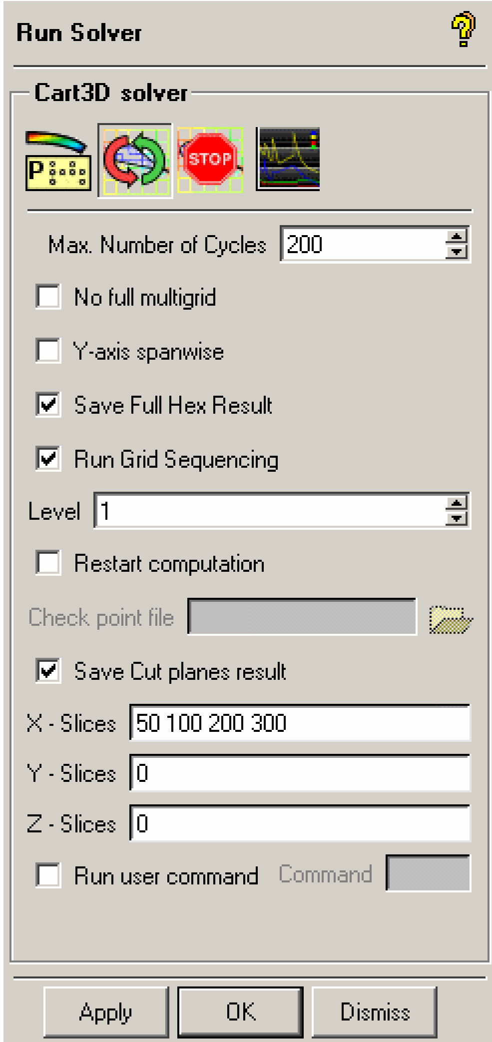
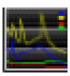


Figure 3.541
Run Solver Window

Click Apply and run the solver
The user can view the convergence by clicking on the Convergence Monitor icon  and the window pops up as shown in Figure 3.542. (The monitor may open automatically.)

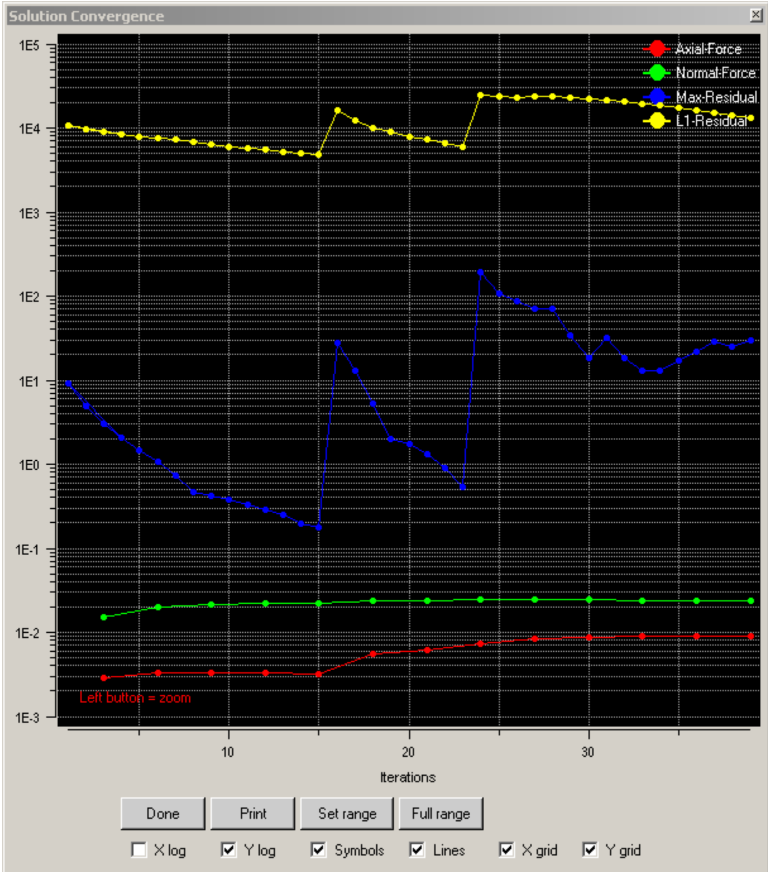
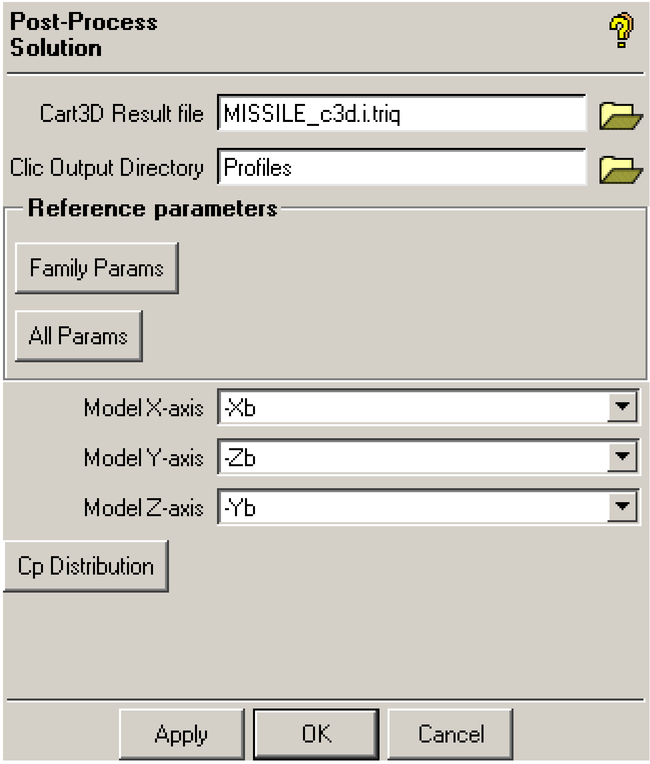


Figure 3.542 Solution Convergence Window

j) Computing Force and Moments

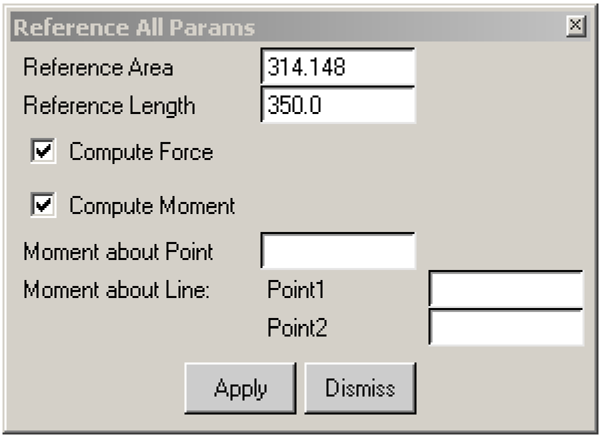
1. In the Cart3D main menu select Integrate Cp  . The Post-Process Solution window appears as shown in Figure 3.543.

Figure 3.543 Post Process Solution Window



Click All Params icon in the Post-Process Solution window.
In the **Reference All Params** window specify **Reference Area** = 314.148 and **Reference Length** = 350.0.
Enable **Compute Force** and **Compute Moment**.
Click Apply in the Reference All Params window and then Dismiss to close as shown in Figure 3.544.

Figure 3.544 Reference All Param window

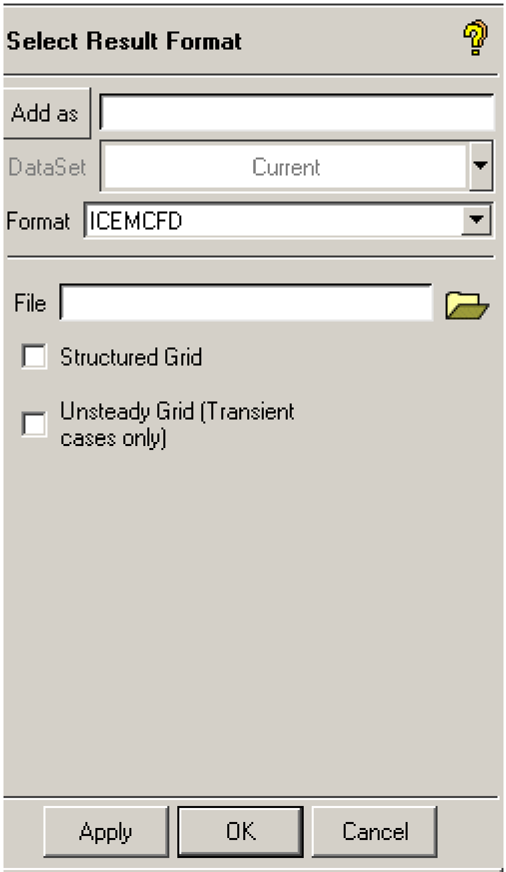


Press Apply in Post-Process Solution window
The computed force and moment coefficients will be displayed in the Messages area.

k) Visualizing the results

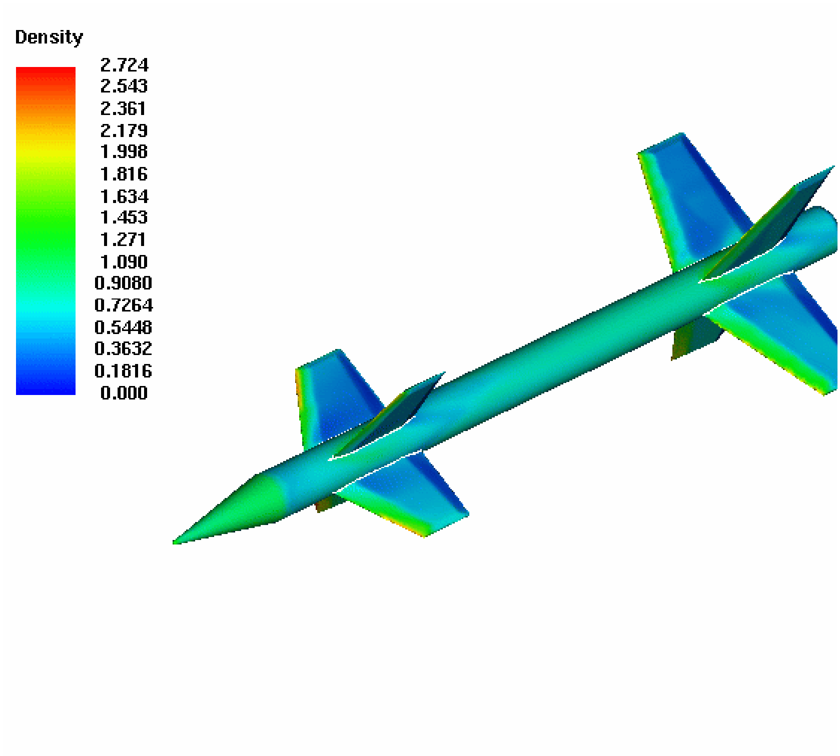
FlowCart writes three output files
i.) **MISSILE_c3d.i.triq** - Contains Pressure, Velocity and Density extrapolated to the surface triangles. This can be converted to the domain file format via Edit> Cart3D Tri File->Domain file. The default resultant domain file will be MISSILE_c3d.uns.
ii). **slicePlanes.dom** - Cut Plane results
iii). **results.dom** - Full mesh result Slice Plane.dom-Cut Plane results
Go to File > Results > Open Results. A Select Result Format window opens as shown in Figure 3.545. Select ICEMCFD as the Format.

Figure 3.545 Select Result Window



Select the result file **surface_results.dom** to get the default result as shown in Figure 3.546.

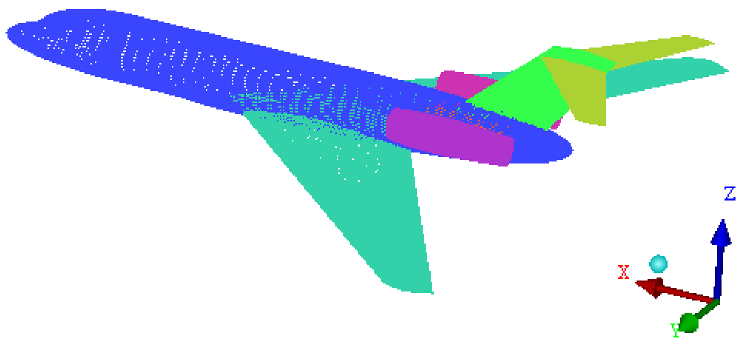
Figure 3.546 Post Processor Display



3.7.5: Business Jet

Overview

This tutorial illustrates how to generate grid in Cart3D around a business jet with multiple components. The flow problem is solved in flowCart and the results are examined.



This tutorial introduces the following operations
Compute force and moment information using Clic.
Visualize the results.

a) Starting the Project

Load **ANSYS ICEM CFD**. Change the working directory by File > Change Working Dir and set the location to the folder **bjet** (**bjet.uns** is located in that folder).

Note: It is preferable to create a separate folder **bjet** and put only **bjet.uns** (domain file) in that folder before performing this tutorial.

Select Open Mesh  from the main menu and select bjet.uns. The model contains several components defining a business jet. Press ‘h’ key to fit the view in the screen if the model is not visible.

b) Mesh Generation Preview only

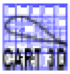
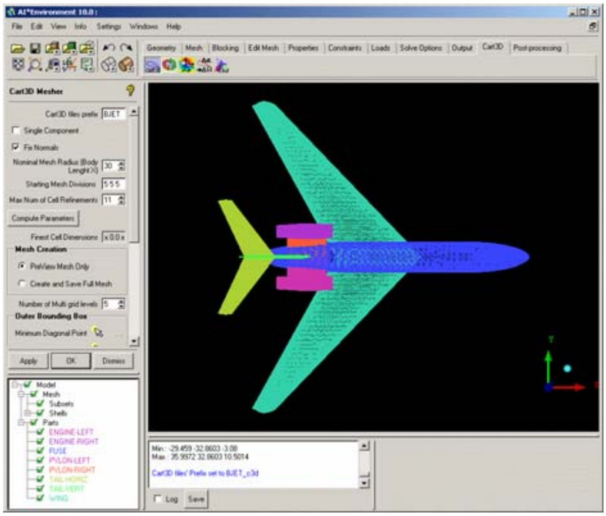
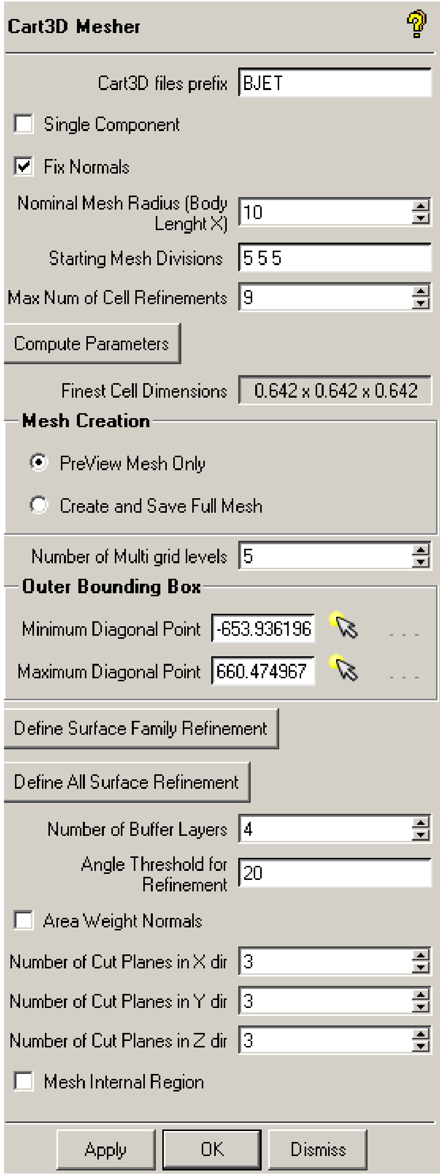
Click on Cart3D from the main menu. Select the Volume Mesher button.  We get the Cart3D Mesher window as shown in Figure 3.547.

Figure 3.547
Cart3D mesher window



Leave Fix Normals enabled to fix the orientation of the triangles such that their normals point outwards.
Set Nominal Mesh Radius (Body length X) = 10, Starting Mesh Divisions = 5 5 5 and Max Num of Cell Refinements = 9.
Click Compute Parameters. This saves the mesh in the local directory, converts in to Cart3D format and determines the intersections. At the end, it displays the Finest Cell Dimensions as shown in Figure 3.548.

Figure 3.548
Cart3D mesher window



This will create 9 density polygons by default for mesh density control, which can be viewed in the Display Tree widget by Geometry>Densities.

This also computes the finest cell size: **0.642 x 0.642 x 0.642**. Varying the starting mesh division and / or Max number of cell refinement can vary finest cell size.

The diagonal points displayed under **Outer Bounding Box** are the minimum and maximum points of the mesh region. These points can be changed if desired.

Leave the Angle Threshold for Refinement as 20.

Click Apply to run the mesher. This will create a domain file with 3 Cut Planes (Quad Elements) in each coordinate direction and Cut Cells (Hex Elements). The Preview mesh will be loaded automatically.

In Parts under the Display Tree widget turn on only CUTPLANE-Y2 as shown in Figure 3.549.

Note: It is advisable to use Parts > Reassign Colors > "Good Colors" to see the results.

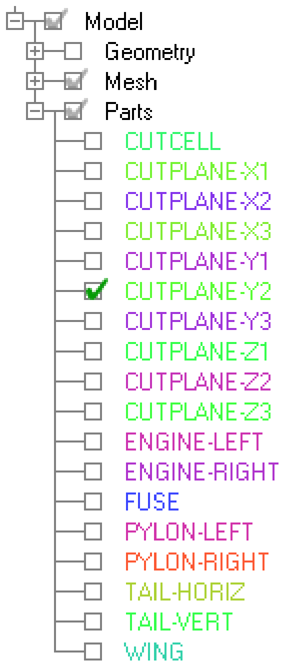
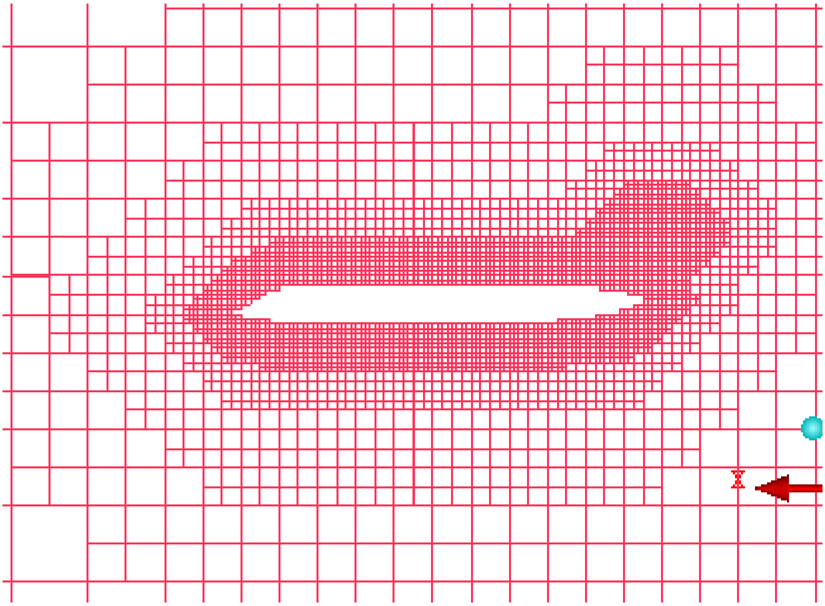


Figure 3.549 Display Tree widget

The mesh is shown in Figure 3.550. Go to View > Top. This is the projected mesh on the middle plane in the Z direction, **CUTPLANE-Y2**.

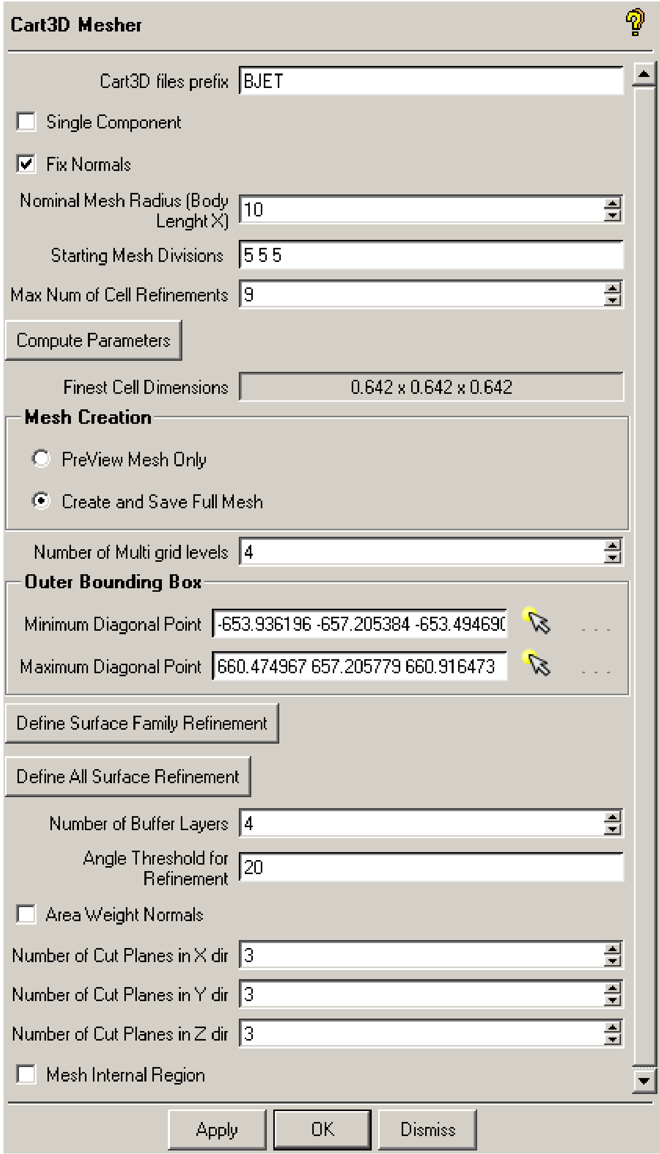
Figure 3.550 Projected mesh CUTPLANE-Y2



c) Mesh Generation Full Mesh

In the Cart3D Mesher window enable Create and Save Full Mesh as shown in Figure 3.551.

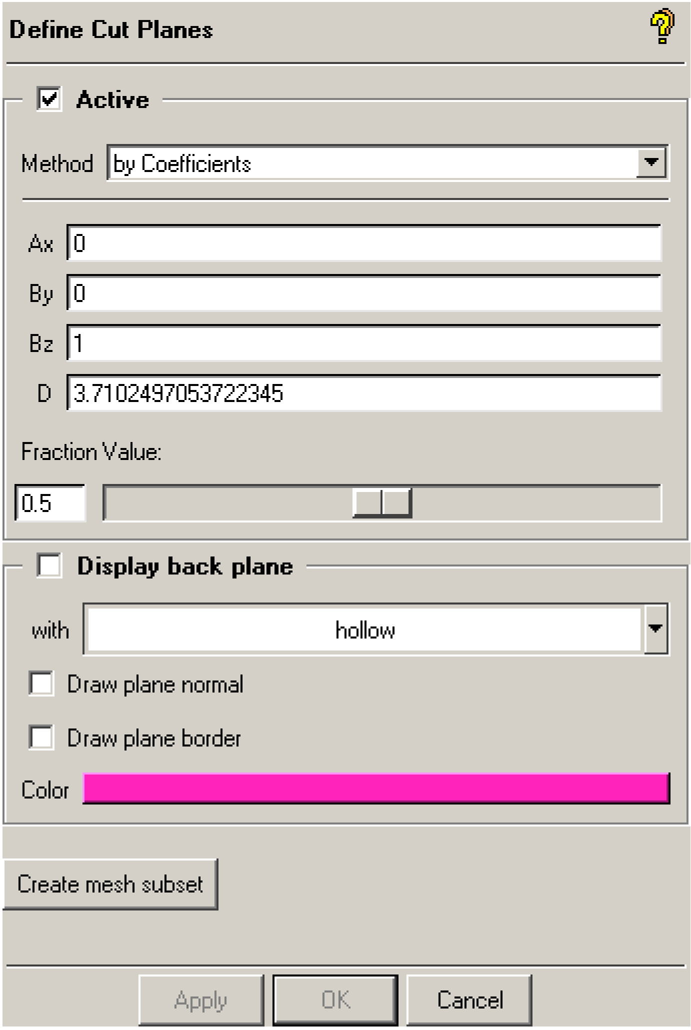
Figure 3.551 Create and Save Full Mesh



Set the **Number of Multi grid levels** to 4. This will create 4 levels of coarsened mesh, which can be read by the solver.

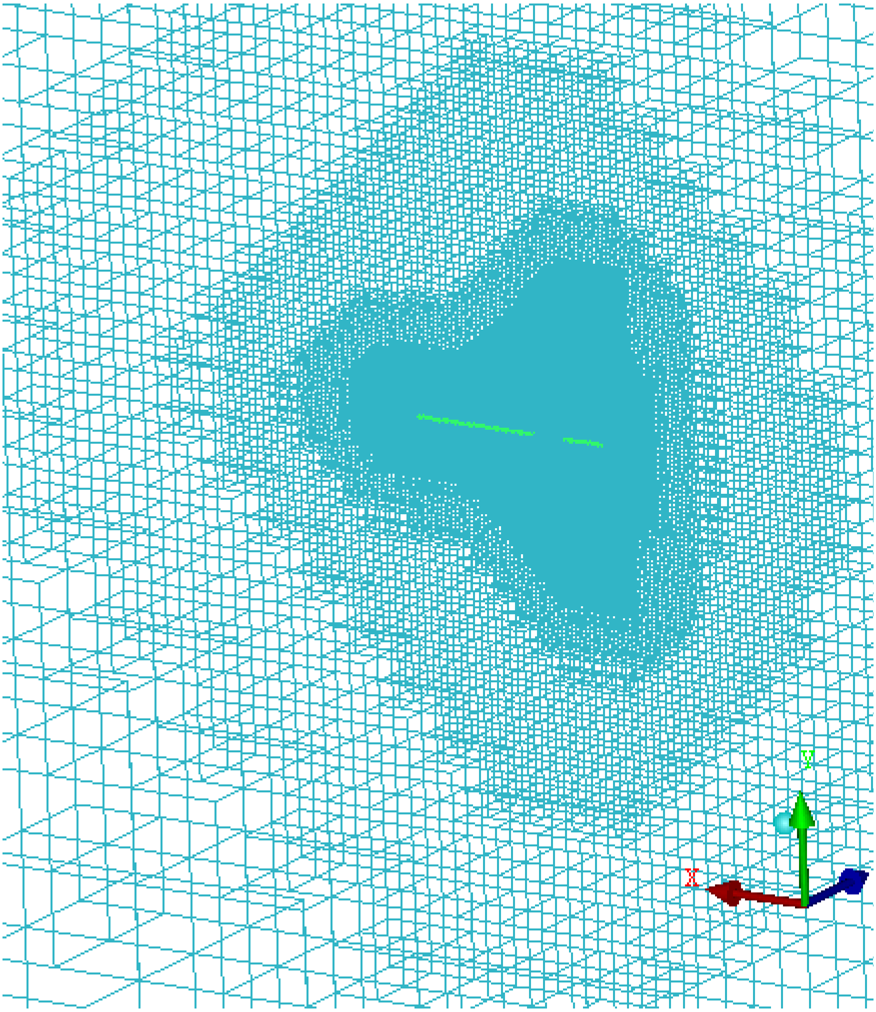
Press Apply. The Cart3D Mesh window appears which asks us to load the cart3D Full Mesh. Press Yes.
The final mesh generated can be examined through Mesh > Cutplane in the Display Tree widget. Accept the defaults in the Define Cut Planes window as shown in Figure 3.552.

Figure 3.552 Cutplane



Enable Volumes from the Mesh branch in the Display Tree widget. The mesh viewed using the above parameters is shown in Figure 3.553.

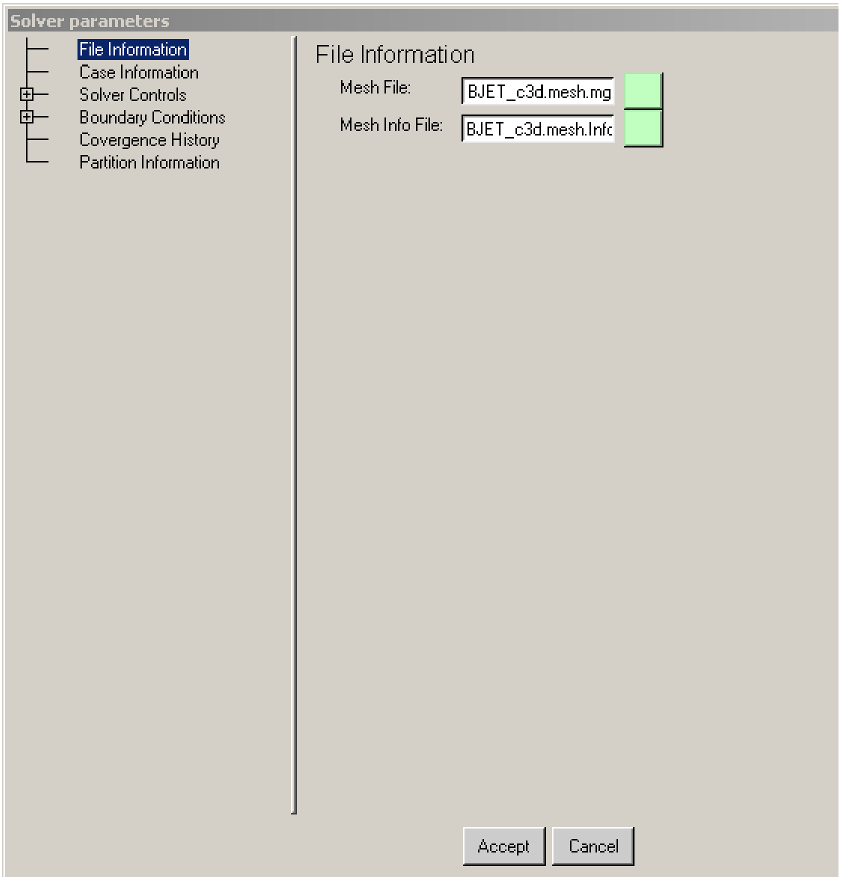
Figure 3.553 View Mesh



d) Setup Flow Cart Parameters

In the Cart3D Menu select Solver.  Click on Define solver params  if the panel doesn't open automatically. A Solver parameters window appears as shown in Figure 3.554.

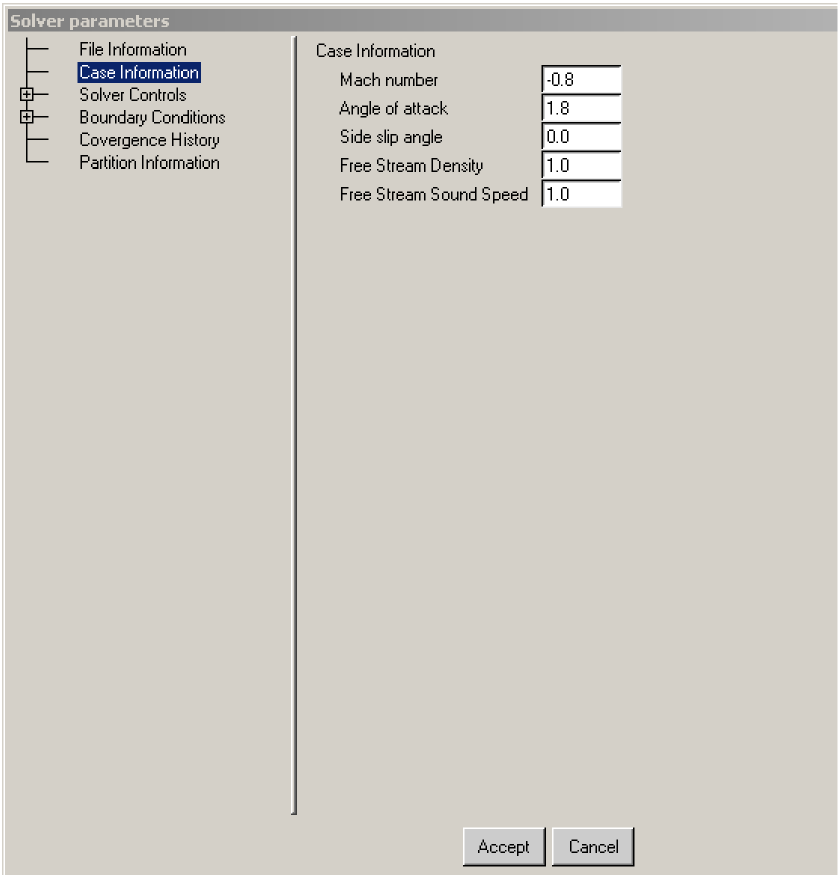
Figure 3.554 Solver Parameters Window



Choose File Information>Mesh File as **BJET_c3d.mesh.mg** (this should be the default).

Click on Case Information and enter the following parameters as shown in Figure 3.555.
Mach number = -0.8
Angle of attack = 1.8
Side slip angle = 0.0
Free Stream Density = 1.0
Free Stream Sound Speed = 1.0

Figure 3.555 Case Information Window



Expand Solver Controls > Runge-Kutta Scheme in the Display Tree as shown in Figure 3.556 and accept the default settings.

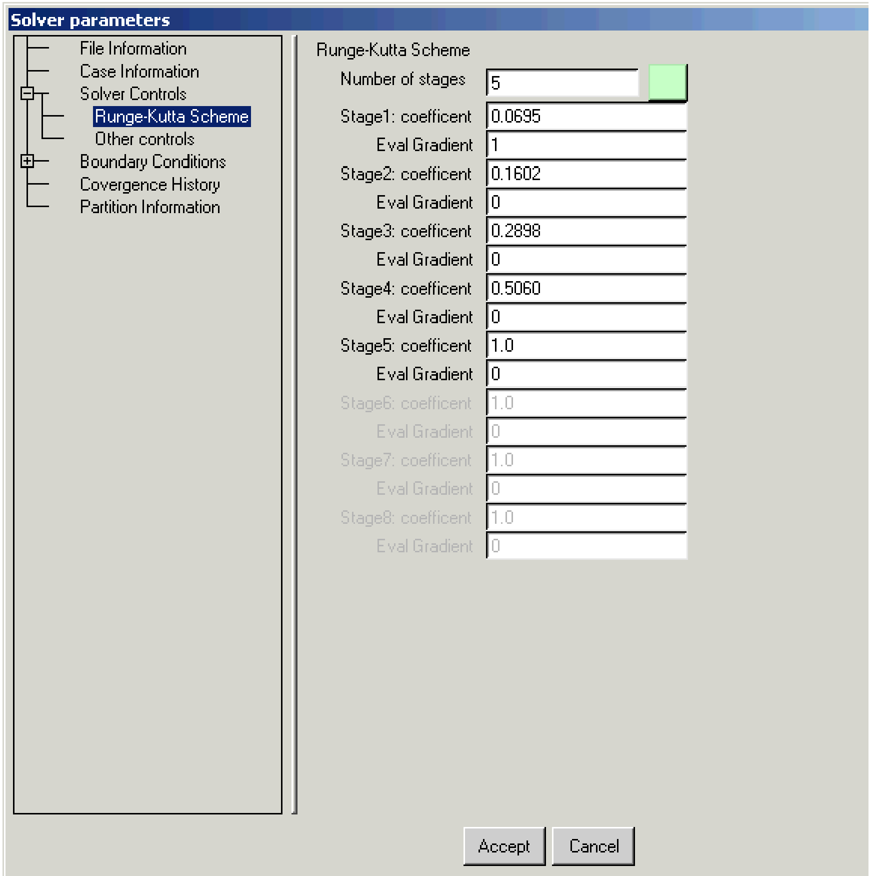


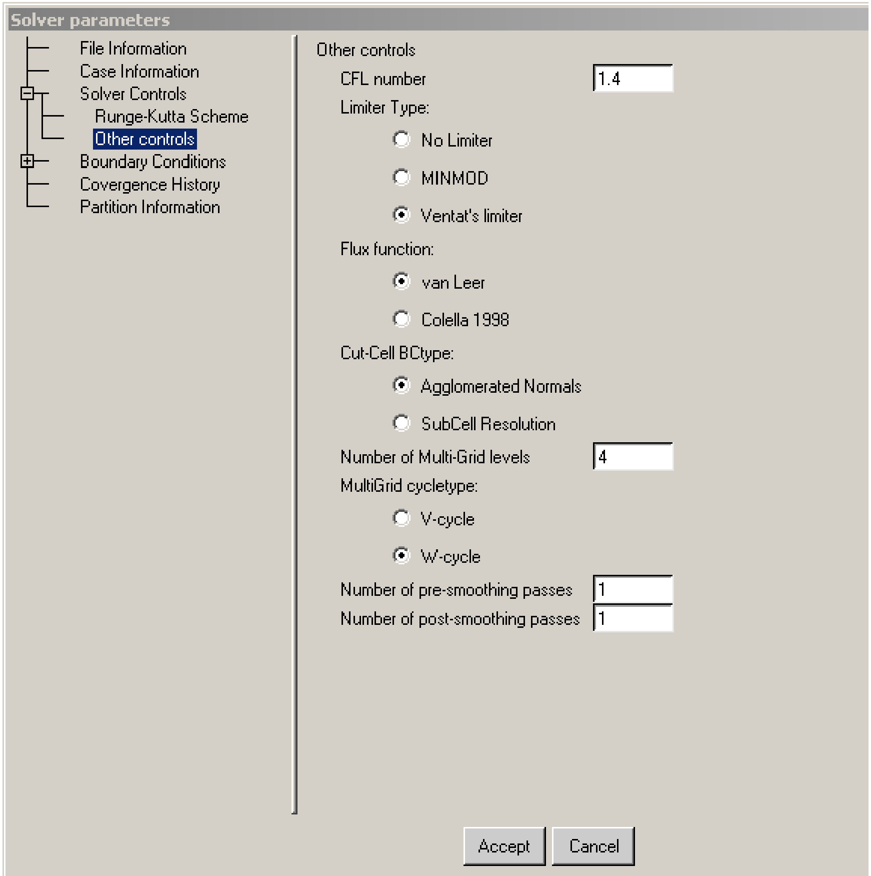
Figure 3.556 Runge-Kutta Window

In Other controls specify the following values for the parameters as in Figure 3.557.

CFL number: 1.4
Limiter Type: Ventat's limiter
Flux function: van Leer
Cut-Cell BCtype: Agglomerated Normals
Number of Multi-Grid levels: 4
MultiGrid cycletype: W-cycle
Number of pre-smoothing passes: 1

Number of post-smoothing passes: 1

Figure 3.557 Other Control Window

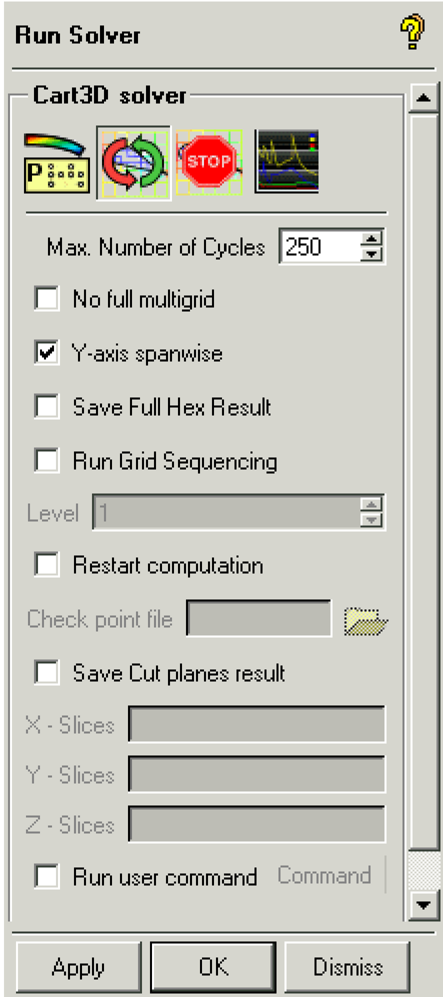


Keep the defaults for Convergence History and Partition Information.and press Accept.

e) FlowCart Solver

Select Solver  > Run Solver  to open the Run Solver panel (refer to Figure 3.558).

Figure 3.558
Run Solver window



Specify Max. Number of Cycles = 250.
Enable Y-axis spanwise.
Enable Save Full Hex Result.
Click Apply and run the solver.

The user can view the convergence by clicking on the Convergence Monitor icon  and the window pops up as shown in Figure 3.559. (This may open automatically.)

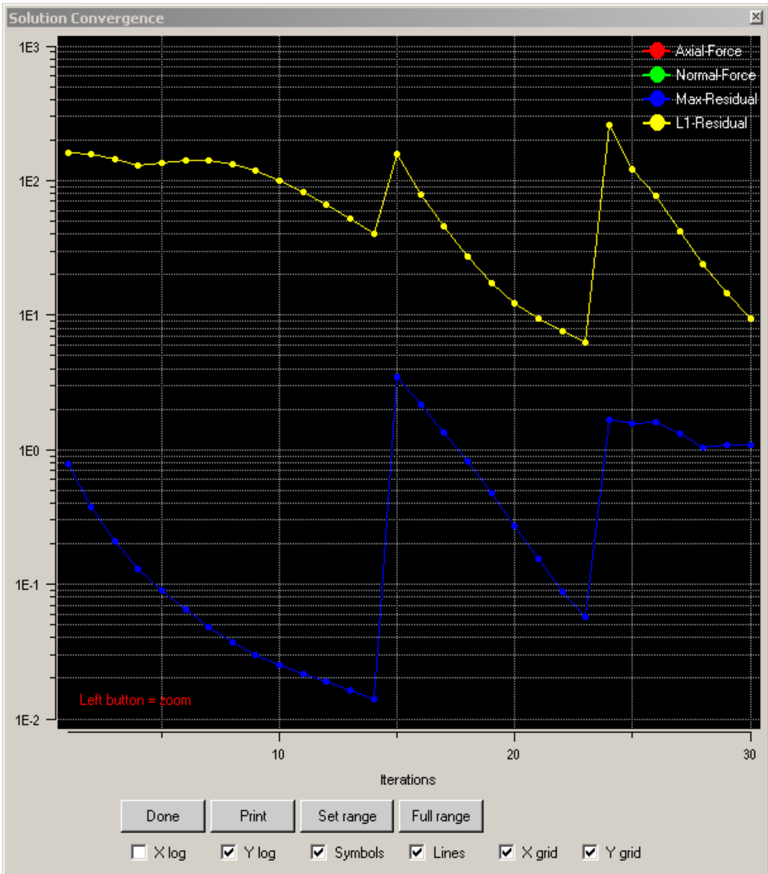
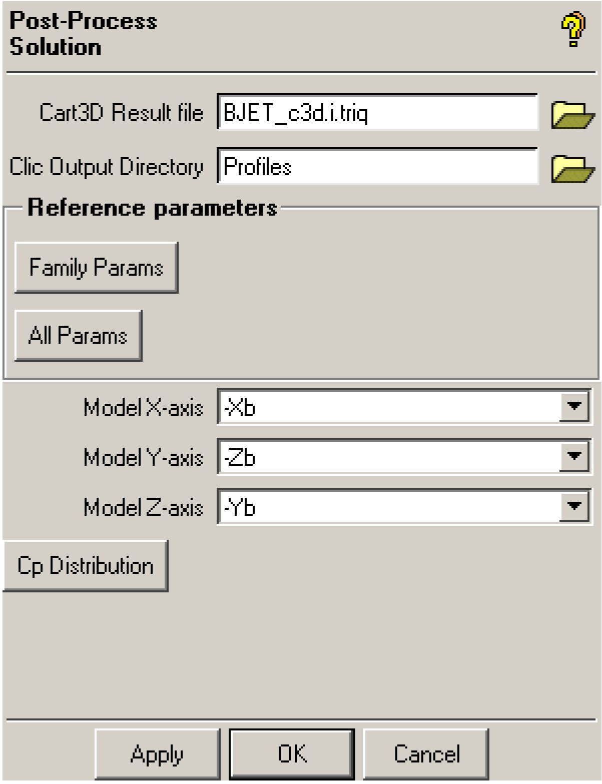


Figure 3.559 Solution Convergence window

f) Computing Force and Moments

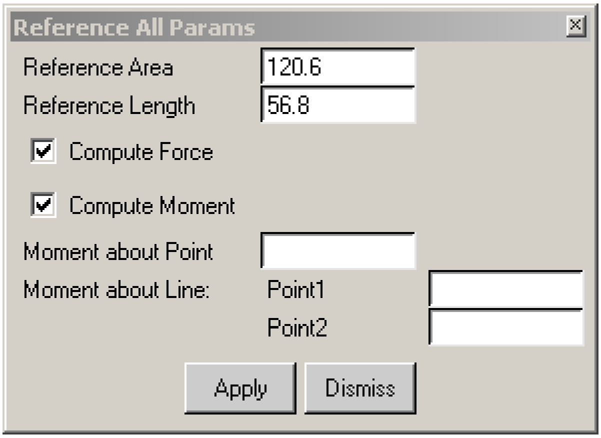
In the Cart3D main menu select Integrate Cp  The Post-Process Solution window appears as shown in Figure 3.560.

Figure 3.560 Post Process Solution window



Click All Params in the window.
In the Reference All Params window set Reference Area = 120.6 and Reference Length = 56.8
Enable Compute Force and Compute Moment.
Click Apply in the Reference All Params window and then Dismiss as shown in Figure 3.561.

Figure 3.561 Reference All Param window

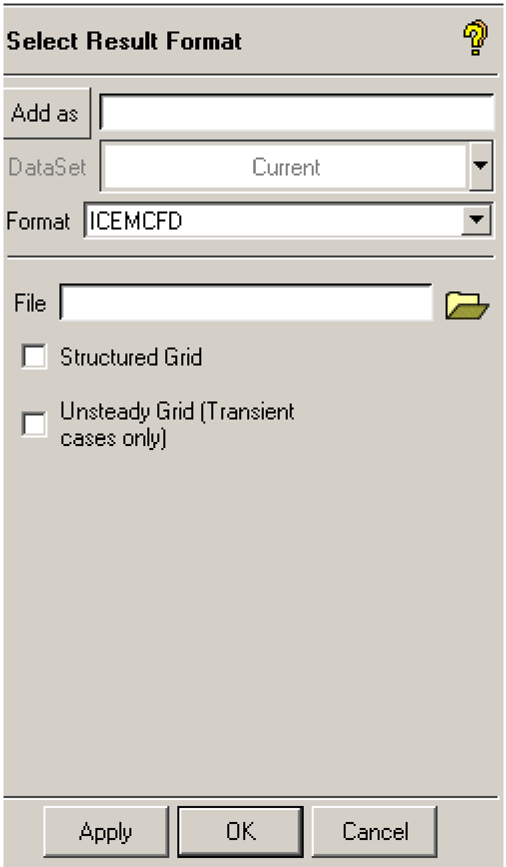


Press Apply in the Post-Process Solution window
The computed force and moment coefficients will be displayed in the Messages area.

g) Visualizing the results

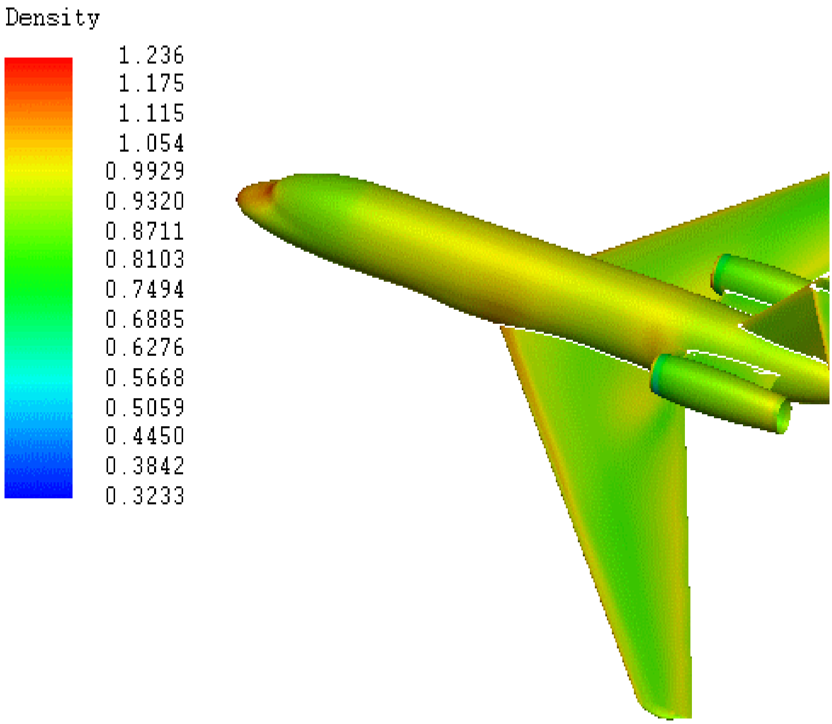
FlowCart writes two output files.
i.) **BJET_c3d.i.triq** - Contains Pressure, Velocity and Density extrapolated to the surface triangles. This can be converted to the domain file format via Edit> Cart3D Tri File->Domain file. The default resultant domain file will be BJET_c3d.uns.
ii). **BOMBER_c3d.dom** - Full mesh result
Go to File > Results > Open Results.
The Select Result Format window pops up as shown in Figure 3.562. Select ICEMCFD as the Format.

Figure 3.562 Result File Window



Select the result file surface_results.dom and press Apply to get the default result as shown in Figure 3.563.

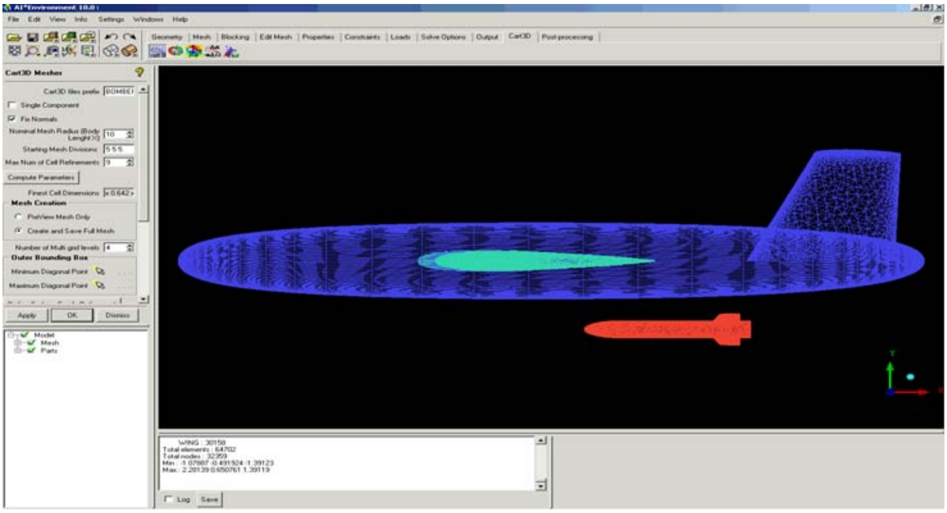
Figure 3.563 Post Processing Result



3.7.6: Bomber

Overview

This tutorial illustrates how to generate grid in Cart3D around a bomber with a missile and solving the problem in flowCart. Post-processing of the results is explained. Use of the SixDOF tool for missile separation is also explained.



This tutorials introduces the following operations:
Running the solver for **AOA** = 5 and **Mach** = 0.65.

Computing Force and Moment information.

Visualizing the results in Post-Processing.
Running the 6DOF tool.

a) Starting the Project

Load **ANSYS ICEM CFD**. Change the working directory by File>Change Working Dir and set the location to the folder **bomber** (**bomber.uns** is located in that folder).

Note: It is preferable to create a separate folder **bomber** and put only the **bomber.uns** (mesh) file in that folder before performing this Tutorial.

Select **Open Mesh**  from the main menu and select **bomber.uns**. Press the ‘h’ key to fit the view in the screen if the model is not visible.

b) Mesh Generation Preview only

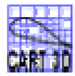
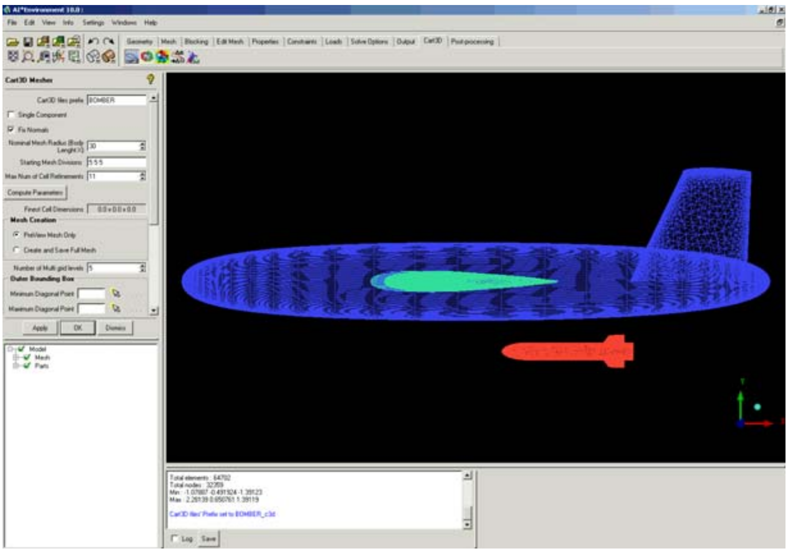
Click on Cart3D from the main menu. Select Volume mesher  and the Cart 3D window pops up as shown in Figure 3.564.

Figure 3.564 Cart3D Main GUI



Leave Fix Normals enabled.

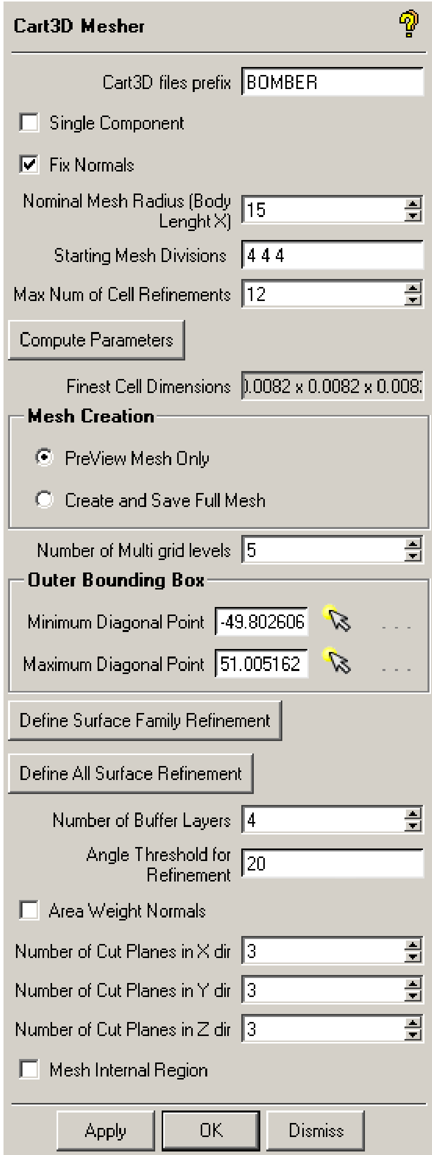
This case is to be run with Mach Number = 0.65. For such subsonic flow, the far field can be 15 times the body length, so specify 15 for Nominal

Mesh Radius (Body Length X), Starting Mesh Divisions = 4 4 4 and Max Num of Cell Refinements = 12.

Note: Though this is a symmetric model the case will not be run with a symmetry boundary condition.

Click Compute Parameters. This saves the mesh in the local directory, converts in into Cart3D format and determines the intersections. At the end, it displays the Finest Cell Dimensions as shown in Figure 3.565.

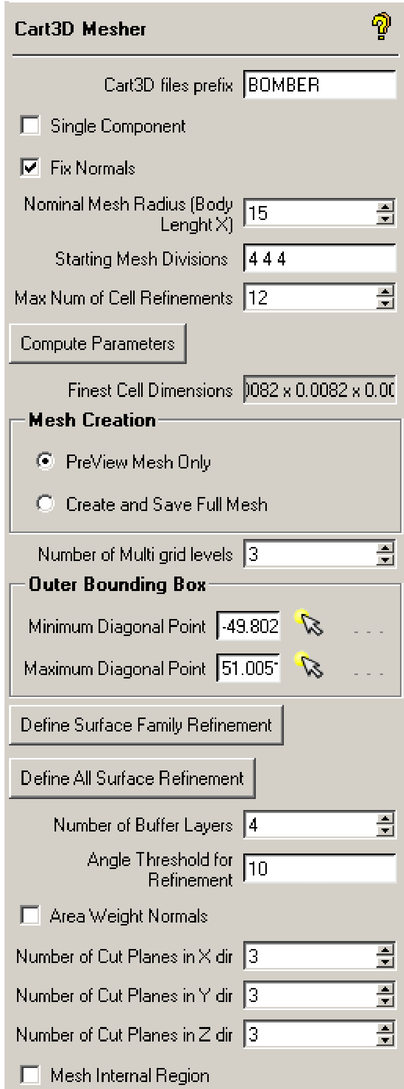
Figure 3.565
Compute Parameter



This will create 5 density polygons by default for mesh density control, which can be viewed via Geometry > Densities in the Display Tree widget.

This also computes the finest cell size: **0. 0082 x 0.0082 x 0.0082**. Varying the starting mesh division and/or Max number of cell refinement can vary the finest cell size
The diagonal points displayed under **Outer Bounding Box** are the Minimum and Maximum points of the mesh region (refer to Figure 3.565).
Set the Angle Threshold for Refinement to 10 as shown in Figure 3.566.

Figure 3.566
Angle Threshold for Refinement Window

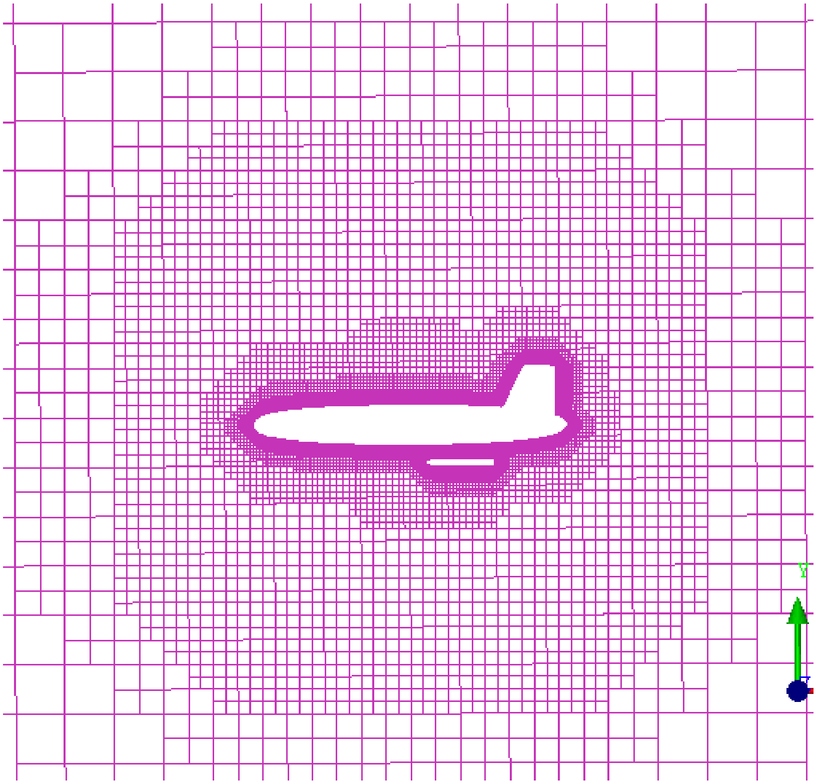


Click Apply to run the mesher. This will create a domain file with 3 Cut Planes (Quad Elements) in each coordinate direction and Cut Cells (Hex Elements). The PreView mesh will be loaded automatically.

Note: As in the case of previous tutorials the mesh can be viewed by switching on the CutPlane to be viewed

One such view is shown in Figure 3.567.

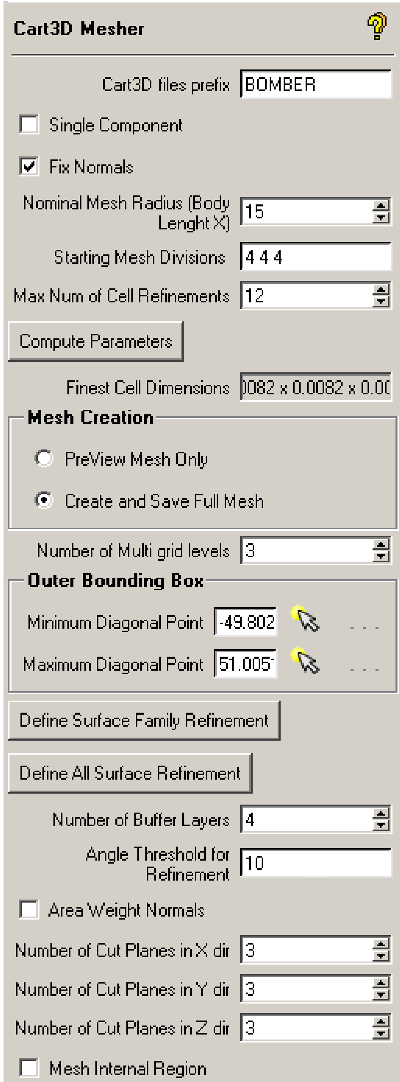
Figure 3.567 CUTPLANE-Z2 View



c) Mesh Generation Full Mesh

Enable Create and Save Full Mesh as shown in Figure 3.568 and change the Number of Multi grid levels to 3. This will create 3 levels of coarsened mesh, which can be read by the solver.

Figure 3.568
Create and Save Full Mesh



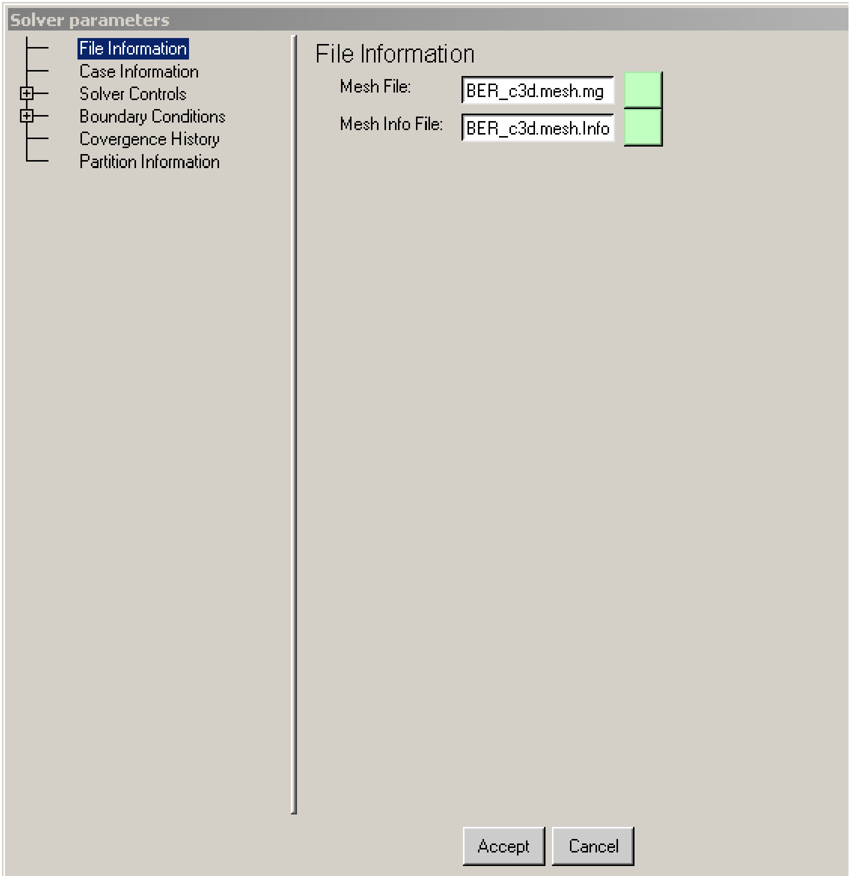
Press Apply. The Cart3D Mesh window appears which asks us about loading the cart3D Full Mesh. Press Yes**.**

Note: The final mesh generated can be examined through Mesh > Cut Plane as in the case of the previous Tutorials

d) Setup Flow Cart Parameters

In the Cart3D Menu select Solver  and Define solver params  (if the panel doesn't open automatically). A Solver parameters window appears as shown in Figure 3.569.

Figure 3.569 Solver Parameter Window



Choose File Information>Mesh File as **BOMBER_c3d.mesh.mg** (this should be default).

3. Click on Case Information and enter the following parameters as shown in Figure 3.570.

Mach number = 0.65
Angle of Attack = 5.0
Side Slip angle = 0.0
Free Stream Density = 1.0
Free Stream Sound Speed = 1.0

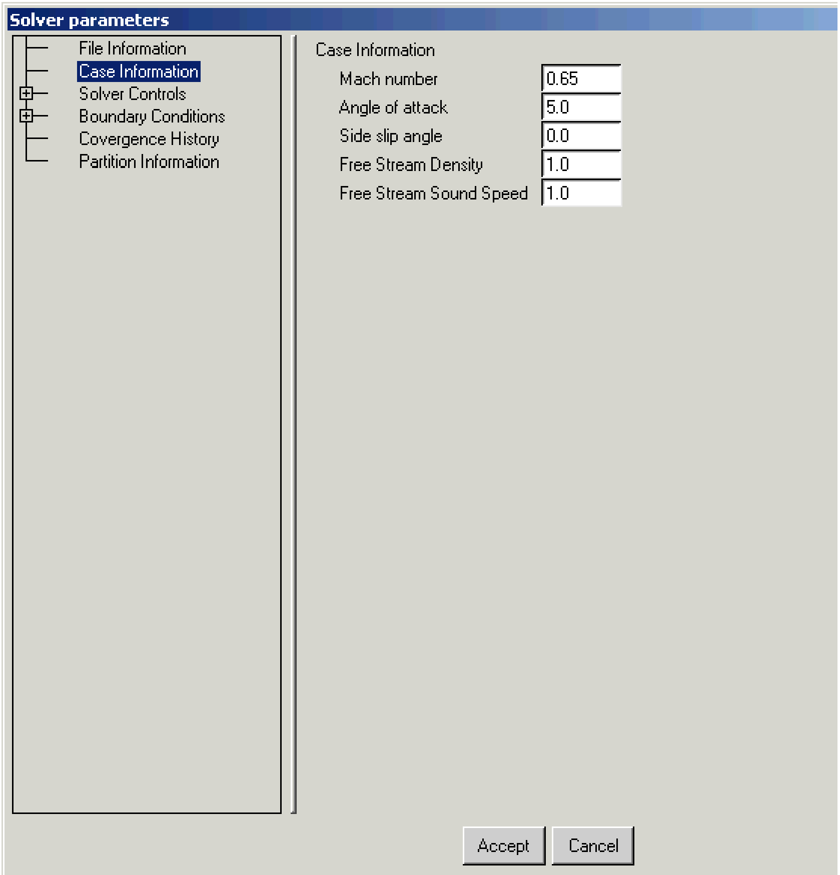
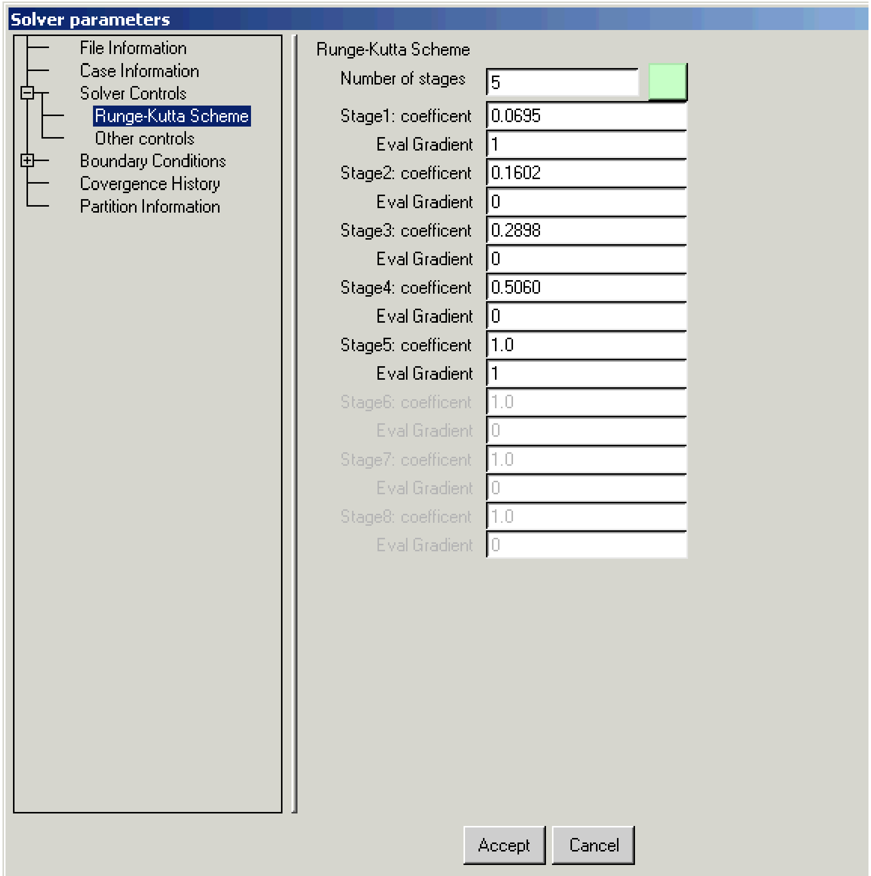


Figure 3.570 Case Information Window

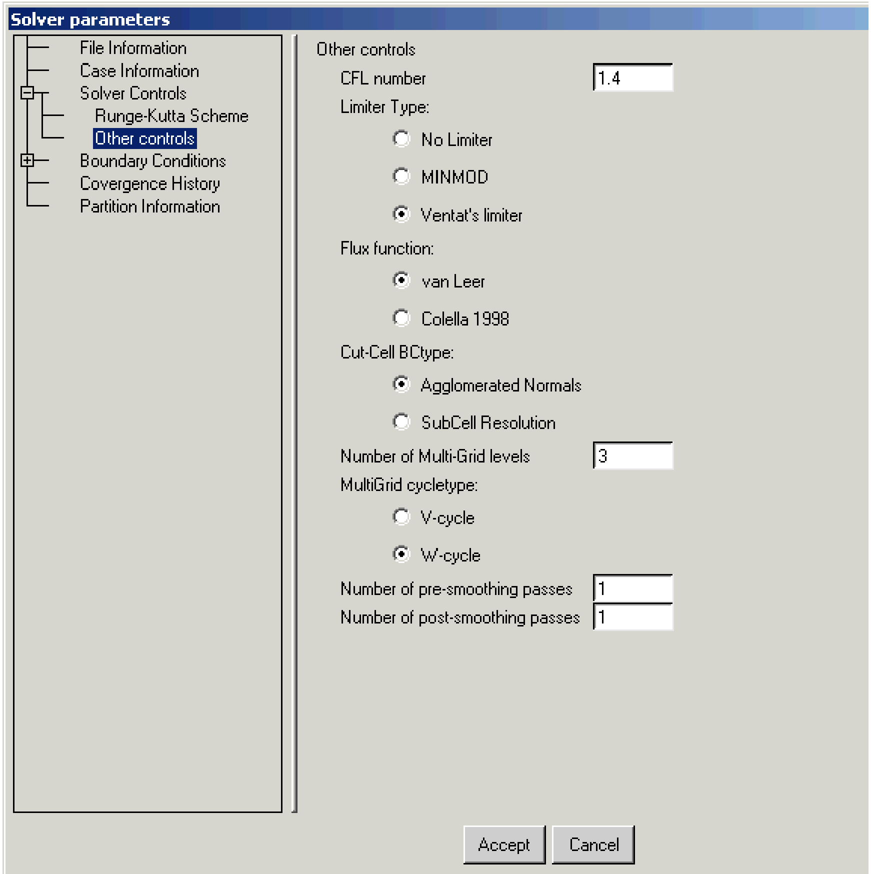
Click on Solver Controls > Runge-Kutta Scheme. Evaluate the gradients only at Stages 1 and 5 as shown in Figure 3.571.

Figure 3.571 Runge-Kutta Scheme



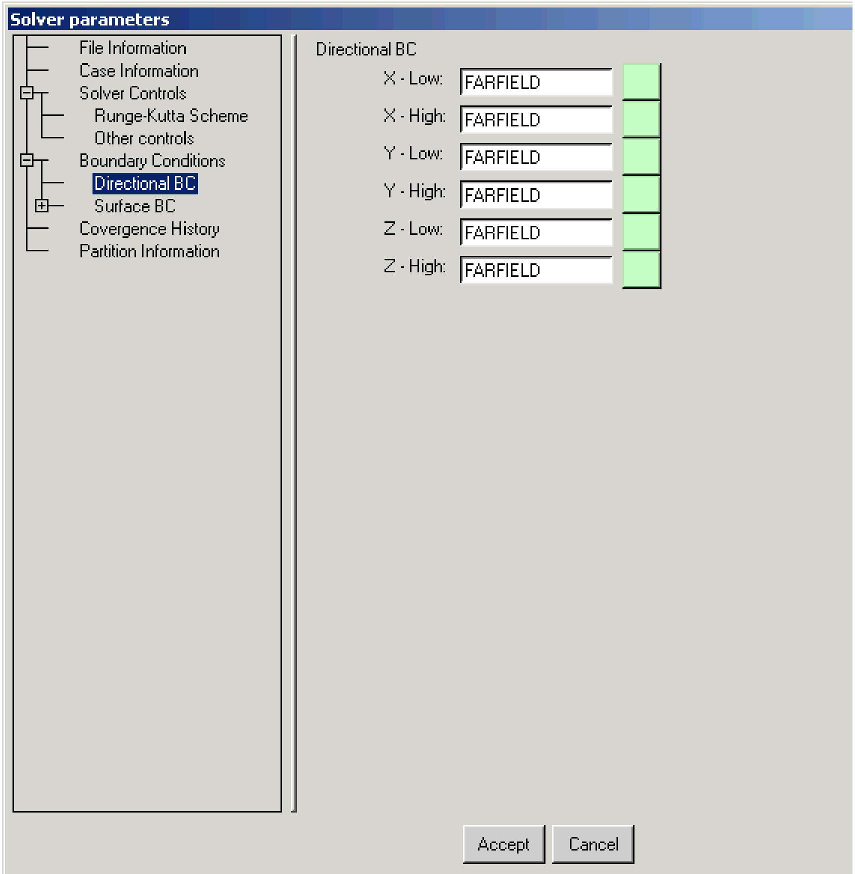
Click on Other controls. Set CFL number to 1.4 and Number of Multigrid levels to 3. Leave the remaining values as default as shown in Figure 3.572.

Figure 3.572 Other Control Window



Click on Boundary Conditions > Directional BC and leave the boundary condition for the six faces of the enclosing Cartesian Box as FAIRFIELD as shown in Figure 3.573.

Figure 3.573 Boundary Condition Window



Leave the Convergence History and Partition Information as default. Click on Accept in the Solver parameters window.

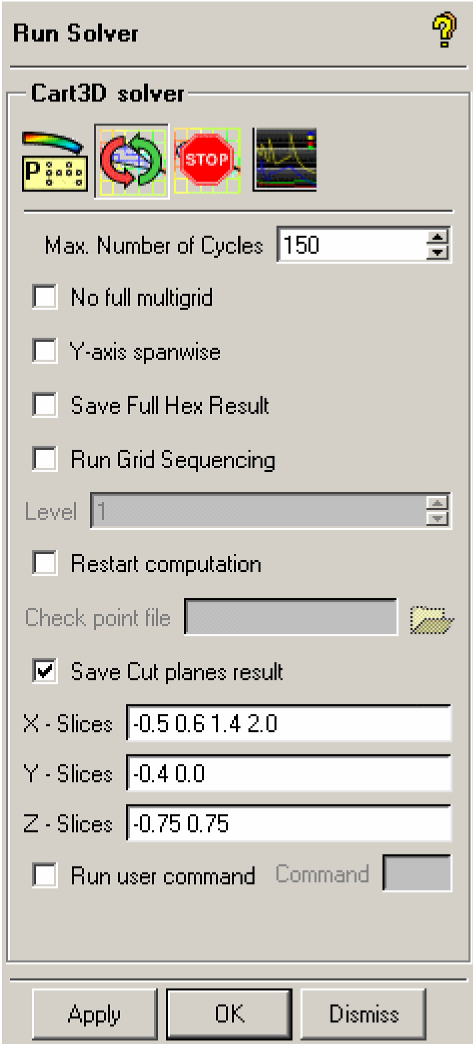
e) Running the FlowCart Solver

Now the case is ready to start the inviscid computation.

Select Solver  >Run Solver  to open the FlowCart solver panel.
Specify Max. Number of Cycles = 150.
Enable Save Full Hex Result.

Turn on Save Cut planes Result. Specify X-Slices as -0.5, 0.6, 1.4, and 2.0; Y-slices as -0.4 and 0.0 and Z-slices as -0.75 and 0.75 as shown in Figure 3.574.

Figure 3.574
Flow Cart Solver Window



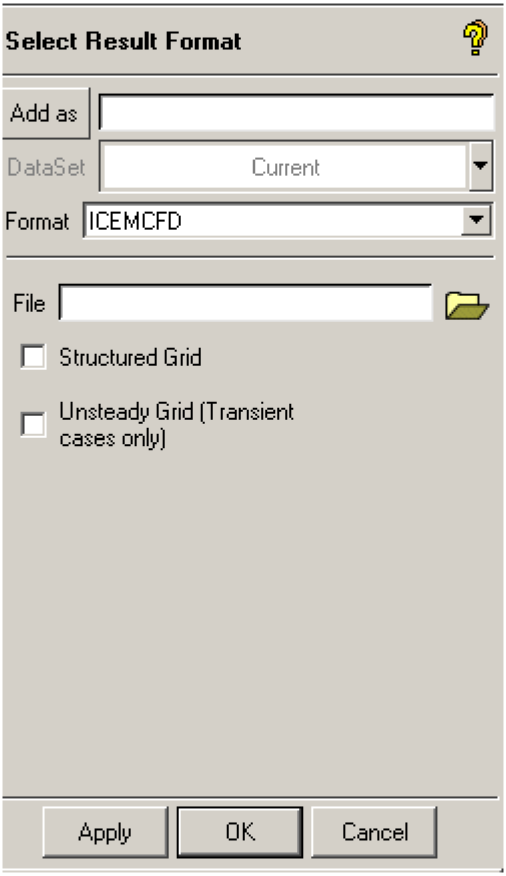
5). Click on Apply to start the solver and output the results files.

6). The convergence history plot window should automatically open.

f) Visualizing the results

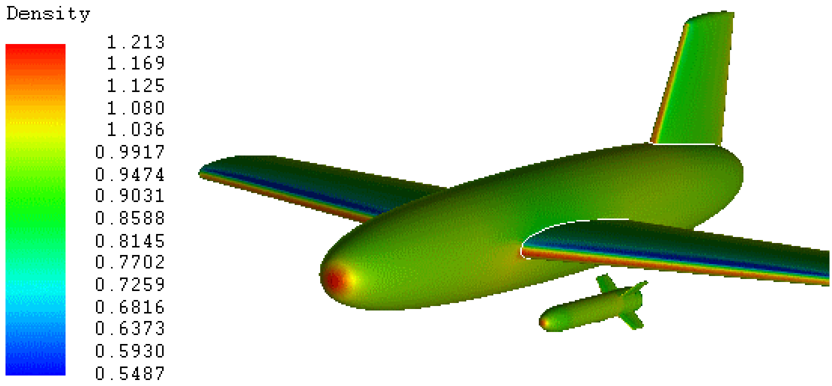
FlowCart writes two output files:
i). **BOMBER_c3d.i.triq** - Contains Pressure, Velocity and Density extrapolated to the surface triangles. This can be converted to domain file format via Edit> Cart3D Tri file->Domain file. The resultant file will be BOMBER_c3d.uns.
ii). **slicePlanes.dom** – cut plane results.
Go to File > Results > Open Result File.
A Select Result Format window pops up as shown in Figure 3.575, Select ICEMCFD as the Format.

Figure 3.575 Results Window



Select the result file **surface_results.dom** and Apply to get the default result as shown in Figure 3.576.

Figure 3.576 The result Generated

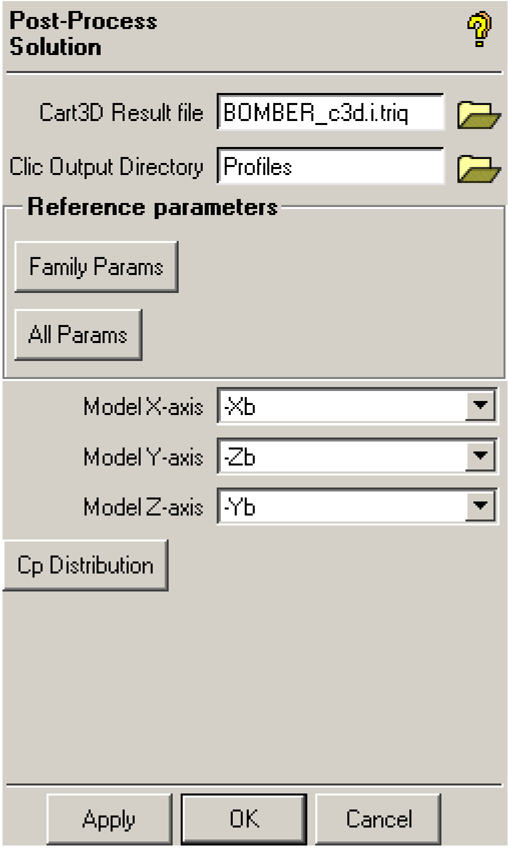


g) Clic - Computing Force and Moment

Aerodynamics and Body forces can be calculated by using the Clic utility.

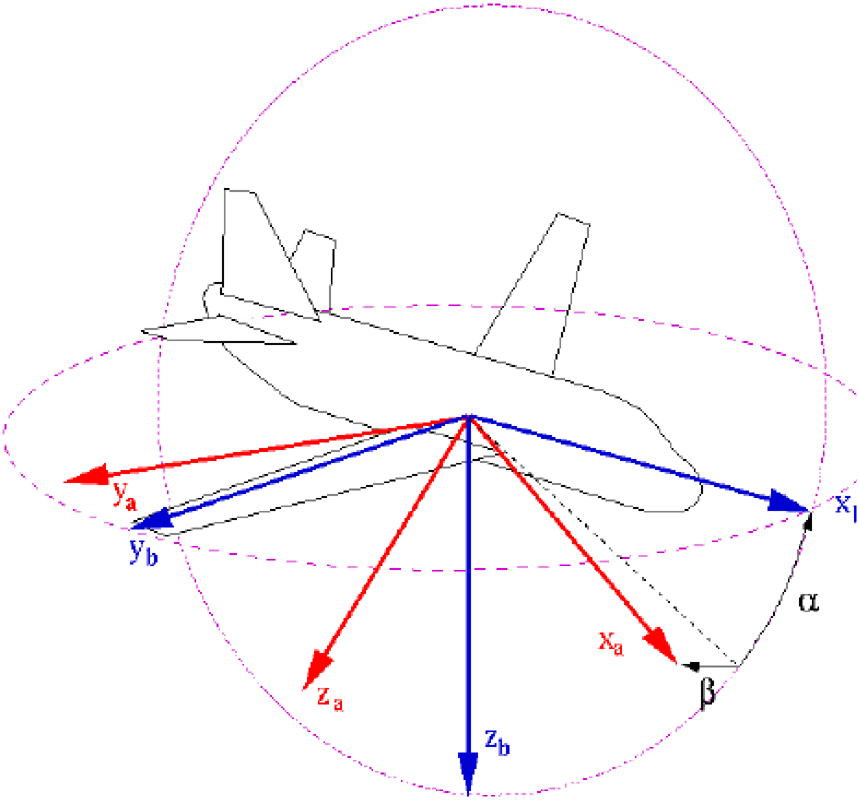
Select Cart3D > Integrate Cp  to open the Post-Process Solution window as shown in Figure 3.577.

Figure 3.577
Post Process Solution Window



Pressure coefficients can be extracted on a cut plane for the component. The results are written in the Clic Output Directory. Specify the directory in which the Cp Distributions are to be stored.
Clic uses its own system of coordinates shown in Figure 3.578. Our model coordinate should be mapped on to Clic's coordinates. In this case, the Display X-axis is –Xb in Clic's coordinates. Similarly Display Y-axis is –Yb in Clic's coordinates & Display Z-axis is –Zb is Clic's coordinates.

Figure 3.578 Axes



Note: These direction are automatically taken, remember Cart3D assumes airflow in positive X-direction. So tail will be negative X direction always.

Note: Reference Area and Reference Length can be specified for each component. This can be specified by click on All Params under Reference parameter option in Figure 3.577, the window is shown in Figure 3.579

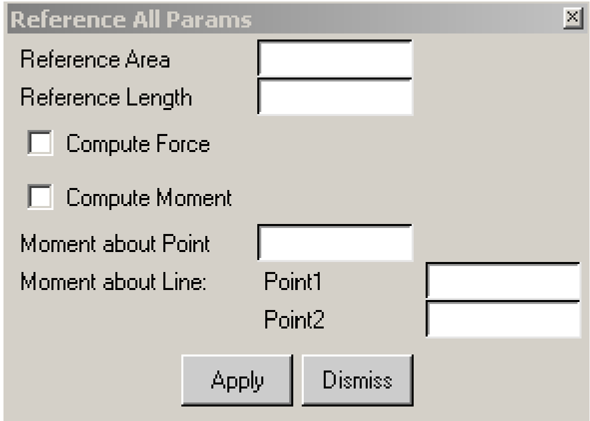


Figure 3.579 Reference All Param Window

For this case Force and Moment are only calculated for the **Bomb** component.

Click on Family Params under Reference parameters as seen in Figure 3.577. The Clic Reference family Params window opens as shown in Figure 3.580.

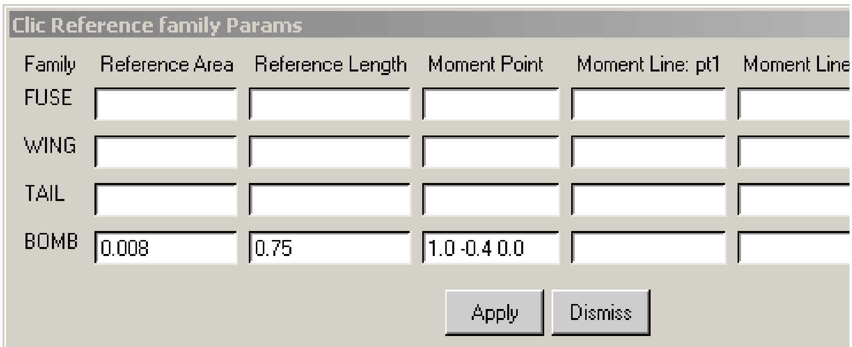


Figure 3.580 Clic Reference family Params window

For the BOMB component specify Reference Area = 0.008 and Reference Length = 0.75. The Moment Point is specified at the center of mass at (1.0, -0.4, 0.0)

Note: Use Floating points instead of integer (1.0 instead of 1). Specify Moment point as [1.0 -0.4 0.0].

Enable Force and Moment for BOMB; then click on Apply and Dismiss. Click Apply in the Post-Process Solution window. This will calculate the Body Force and Aerodynamics Force and Moment about the Center of mass for the Bomb component. The result will be reported in the messages area.

h) Six Degrees of Freedom

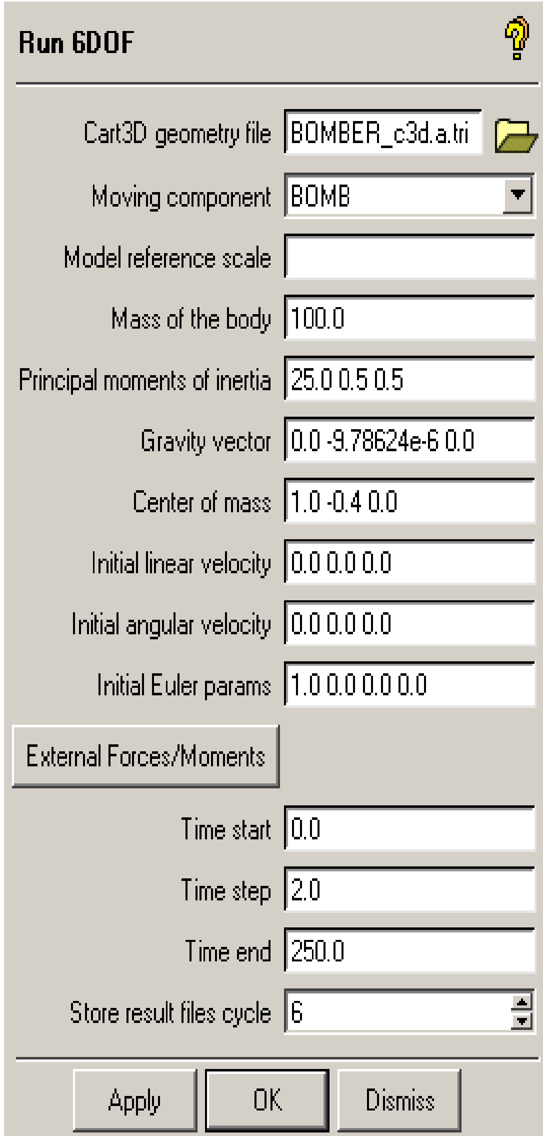
Given the external and Aerodynamic Force and Moment, the 6DOF program calculates the position of the component at the next time step. The 6DOF utility computes the mesh, runs the solver, calculates forces and moves the component accordingly. It goes to the next time step and repeats the process until the end time is reached. 6DOF uses the same parameters defined during the initial meshing and post-processing for the Cart3D initial solution. The user goes through the initial process; then 6DOF can be started.

While running the Clic post-processor, only the component that moves should be used to calculate forces and moments; otherwise, errors will be reported while running 6DOF.

Click on Run 6DOF  from the Cart3D menu.

The Run 6DOF window is shown in Figure 3.581

Figure 3.581 Run SixDOF window



Choose the un-intersected surface tri file **BOMBER_c3d.a.tri** as the Cart3D geometry file.
Select BOMB as the Moving component. Set Mass of the body = 100.0.

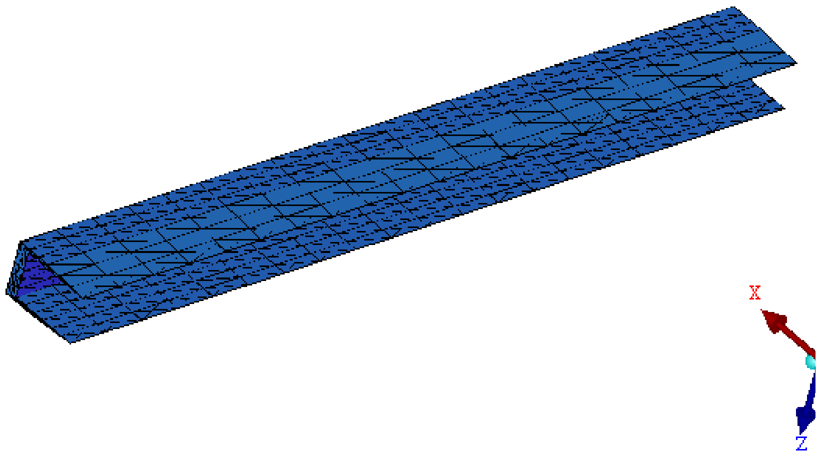
The Principal moments of Inertia have to be specified for three Components. Specify Ixx = 25.0, Iyy = Izz = 0.5.
Use the non-dimensional value for 9.81 m/s^2 (0.0, -9.78624e-6, 0.0) for the Gravity vector.
Specify the BOMB Center of mass as (1.0, -0.4, 0.0)
Leave the default Initial velocity and Initial Euler values
For the Bomber configuration: at time t = 0.0, the missile is dropped from the vehicle, so Time start = 0.0. Set Time step = 2.0. Calculate the trajectory until Time end = 250.
Set Store result files cycle = 6, so results are saved after every 6 steps.
Now click on Apply in the Run 6DOF window. This will run intersect, Cart3D mesher, reorder, multigrid, FlowCart and Clic. Then it calculates the new position for the bomb in the next time step and moves the component to the new position in the geometry file (*. a.tri). Then again goes to intersect. The result is stored intermittently as specified by the Store result files cycle value. The time step will be included in the name of the results files.

3.7.7: Advanced Pitot Intake Tutorial

Overview

This is a benchmark verification case for the Cart3D inlet boundary condition.

The purpose is to study the shock behavior of a classical pitot tube intake for a variety of back pressures.



The Tutorial introduces the following operations:

i). Running the solver for three different cases: **Critical, Sub-Critical** and **Super-Critical.**

ii). Visualize the results in Post Processing

a) Starting the Project

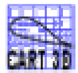
Load **ANSYS ICEM CFD**. Change the working directory by File>Change Working Dir… and set the location to the folder **pitot** (**pitot.uns** is located in that folder).

Note: It is preferable to create a separate folder **pitot** and put **pitot.uns** (domain file) and **Density_boxes.tin** (geometry file) in that folder before performing this tutorial.

Select Open Mesh  from the main menu and select **pitot.uns**.

Select Open Geometry  to load **Density_boxes.tin** which contains density boxes set up for bow and lip shock resolution.
Turn on Geometry > Densities in the Display Tree widget to see the density regions.

b) Mesh Generation Preview Only

Click on Cart3D from the main menu. Select Volume Mesher . We get the Cart3D Mesher window as shown in Figure 3.582.

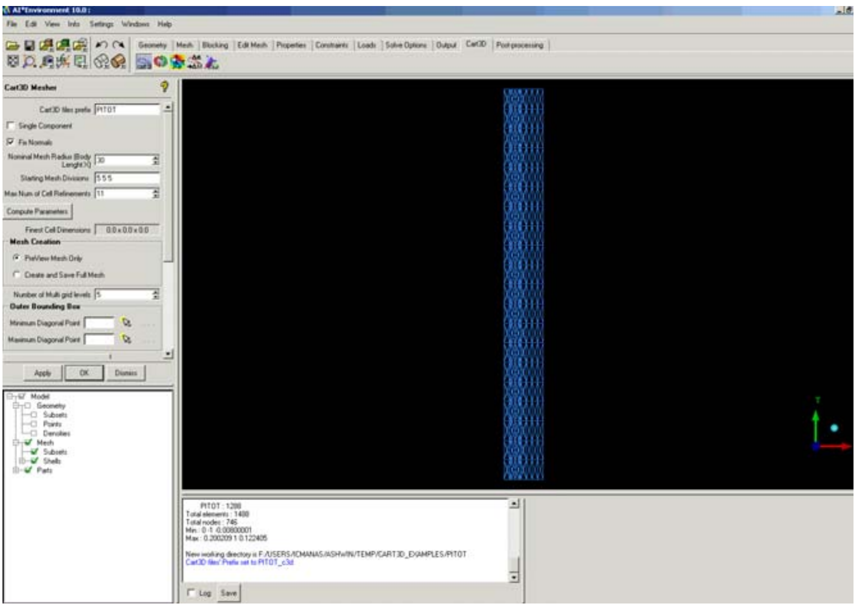


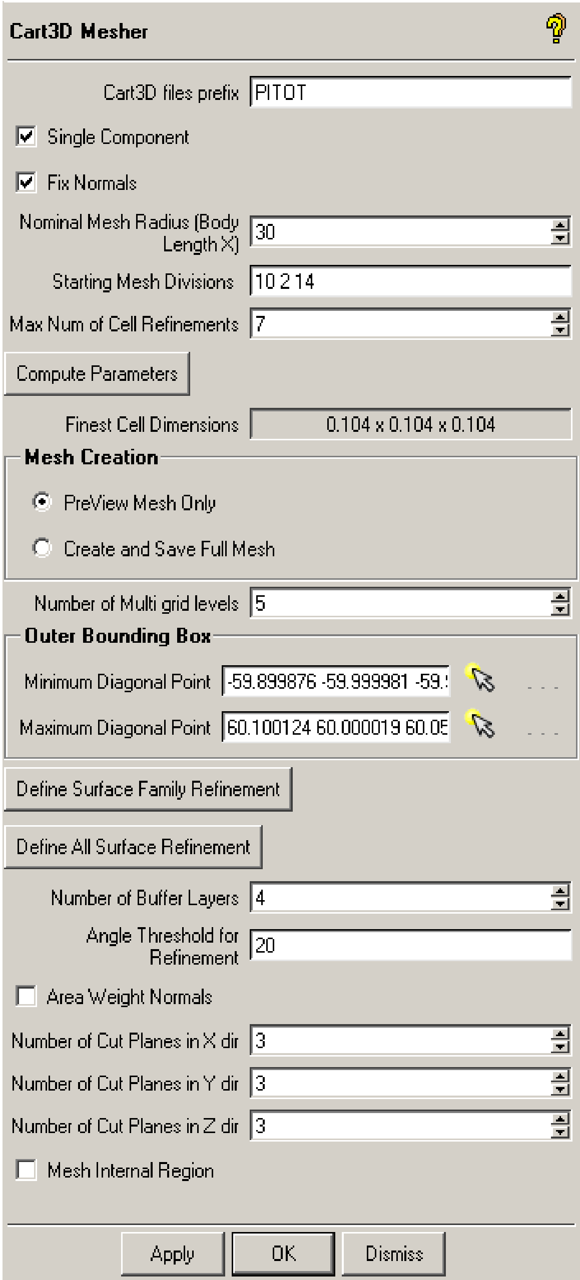
Figure 3.582 Cart 3D GUI window

Leave Fix Normals enabled. This will fix the triangle orientations such that their normals are pointing outward.
Enable Single Component
4Accept default value of Nominal Mesh Radius (Body Length X) = 30, Starting Mesh Divisions = 10 2 14 and Max Num of Cell Refinements = 7

Leave the other values as default

Click Compute Parameters. This saves the mesh in the local directory and converts in into Cart3D format. At the end, it displays the Finest Cell Dimensions as shown in Figure 3.583.

Figure 3.583 Cart3D Mesh window



This will create 3 density polygons by default for mesh density control, which can either be kept or deleted.

This also computes the finest cell size: 0.104 x 0.104 x 0.104. Varying the Starting Mesh Divisions and/or Max Num of Cell Refinements can vary this.

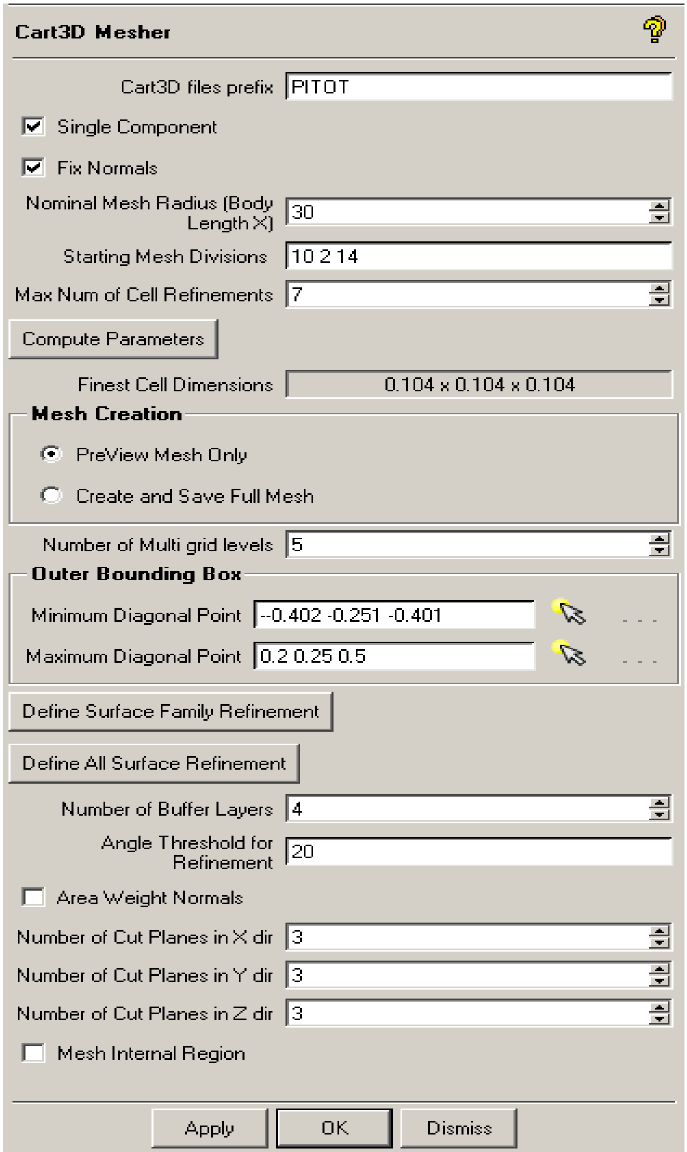
The diagonal points displayed under **Outer Bonding Box** are the maximum and minimum points of the mesh region

Leave the Angle for Threshold Refinement = 20

Specify Minimum Diagonal Point as [-0.402 -0.251 -0.401] and Maximum Diagonal Point as [0.2 0.25 0.5] as shown in Figure 3.584.

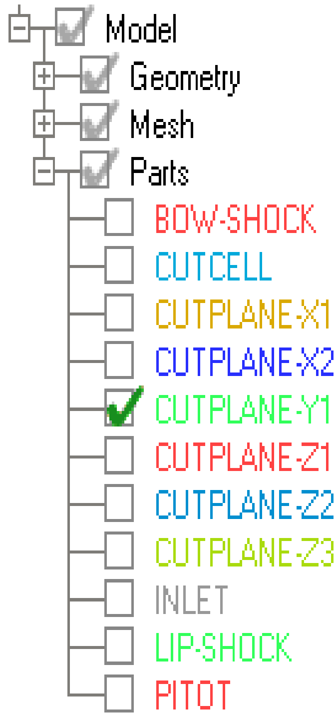
Click Apply to run the mesher. This will create a domain file with 6 Cut Planes (Quad Elements)

Figure 3.584 Change Maximum/Minimum Diagonal Point



In the Part Menu under the Display Tree widget perform the operation Parts>Hide All and the turn on only the Part CUT PLANE Y1 as shown in Figure 3.585.

Figure 3.585
Display Tree widget



The mesh projected onto CUTPLANE-Y1 is shown in Figure 3.586.

Note: The mesh in Figure 3.586 can be view by View > Top.

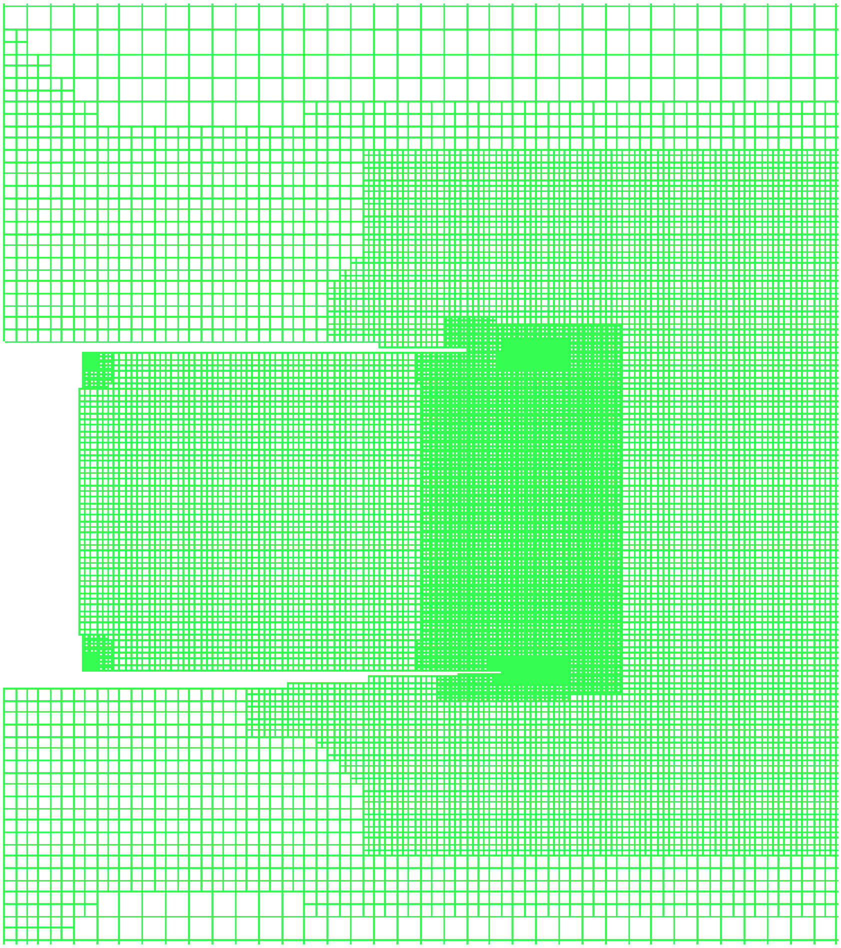


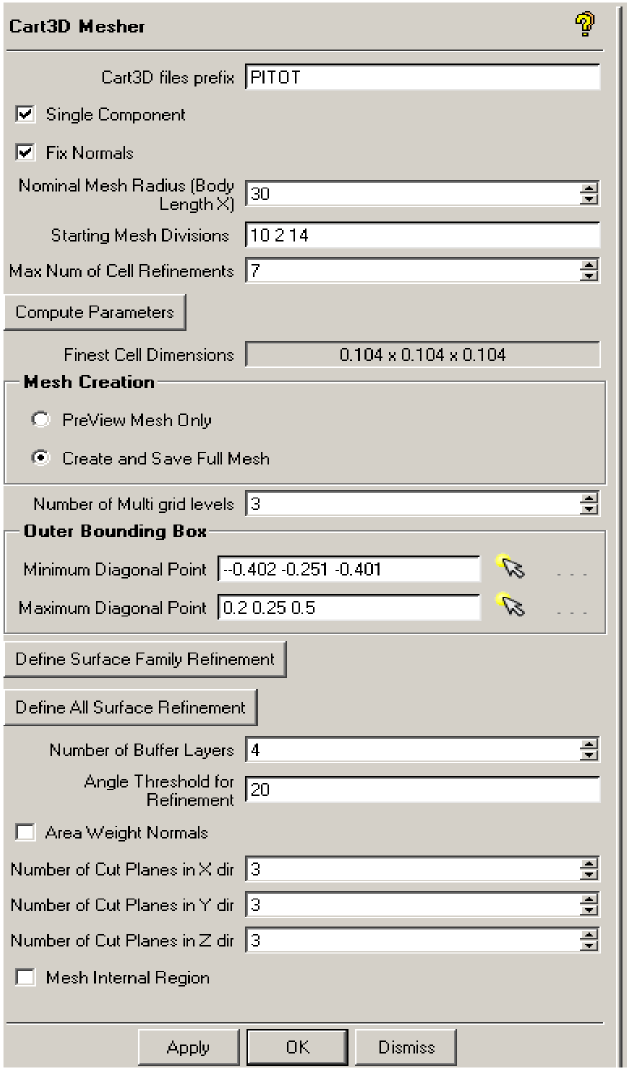
Figure 3.586 CUT PLANE Y1 Mesh

Perform the operation Parts>Show All after viewing the mesh.

c) Mesh Generation-Full Mesh

Now in the Cart3D Mesher window enable Create and Save Full Mesh as shown in Figure 3.587.

Figure 3.587 Create and Save Full Mesh



Set the Number of Multi grid levels to 3. This will create 3 levels of coarsened mesh, which can be read by the solver.

Press Apply. The Cart3D Mesh window appears which asks us about loading the cart3D Full Mesh as shown in Figure 3.588. Press Yes.

Figure 3.588 Cart 3D Mesh window

The final mesh can be examined through Mesh>Cutplane as discussed for the previous tutorials.

Note: There are three cases to be performed for this tutorial so it is advisable to make three copies of the current directory and run the cases in the respective directory. The three cases to be computed are

i) Critical

ii) Sub-Critical

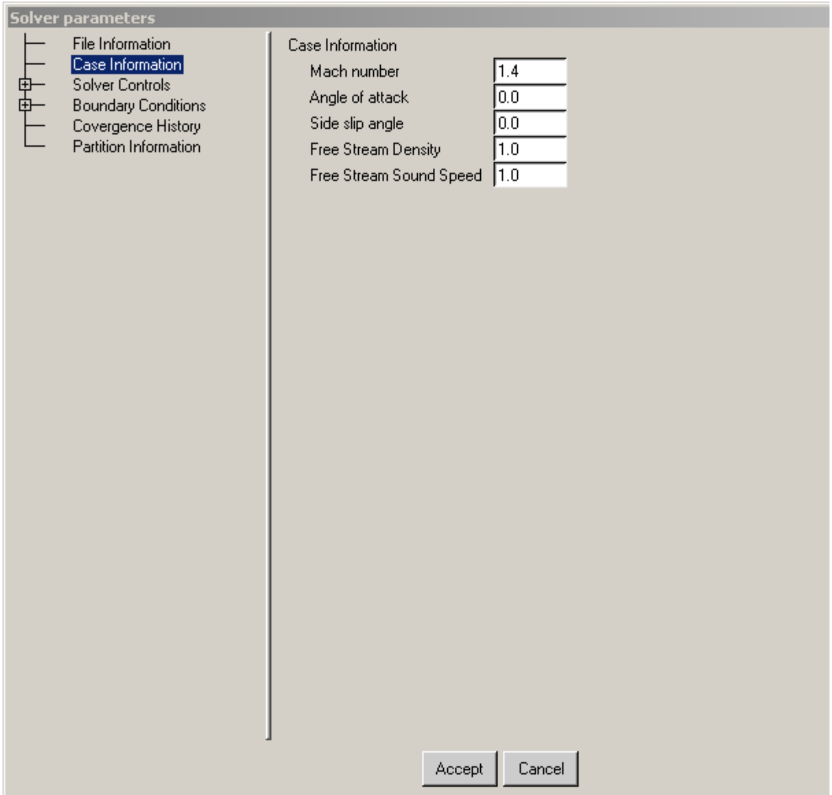
iii) Super-Critical

d) Case 1: CRITICAL

Change the working directory by File > Change Working Dir… and set the location to the folder **Critical** into which the current files were copied.

Go to Solver  > Define solver parameters  (The parameters panel may open automatically.)
In the Solver parameters window do the following:
i) In Case information: set Mach number = 1.4. Use the other defaults as shown in Figure 3.589.

Figure 3.589 Case Information Window



ii) In Solver Controls > Other controls: set Number of Multi-Grid levels = 3. Use the other defaults as shown in Figure 3.590.

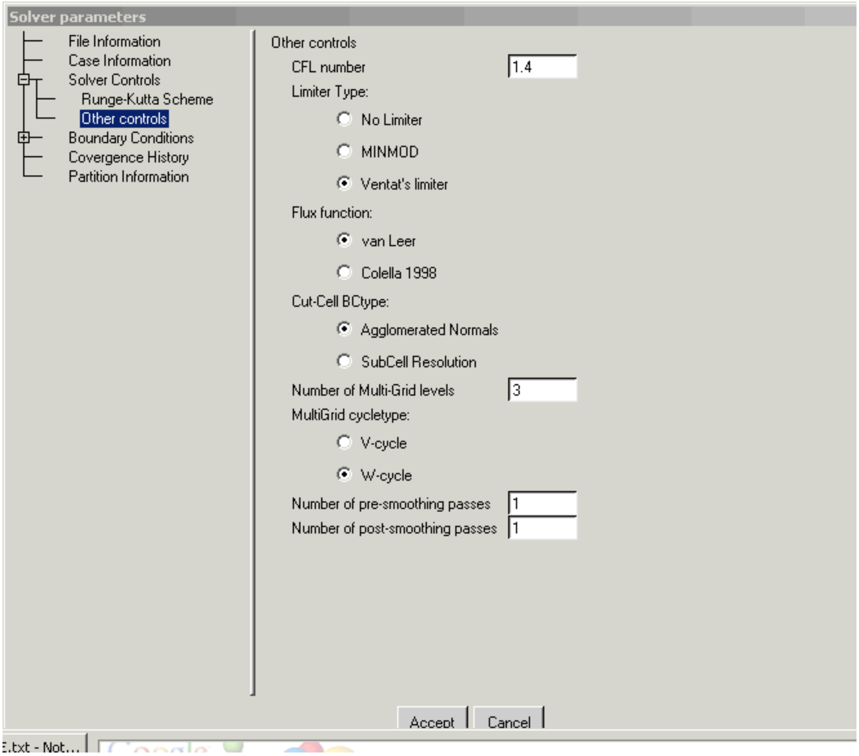
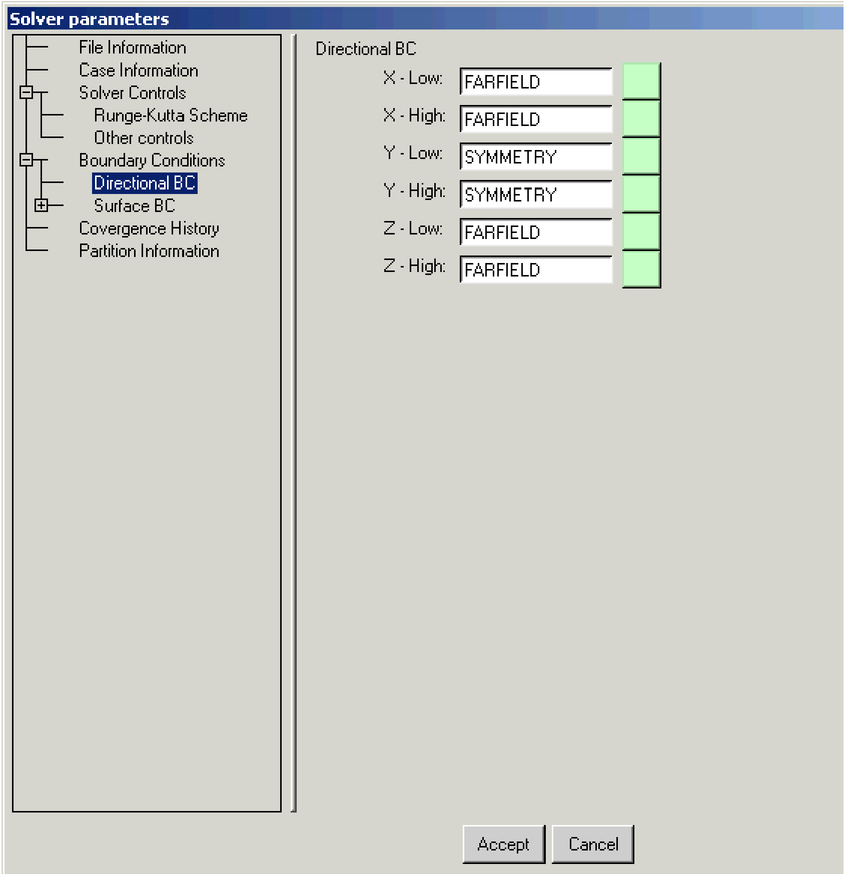


Figure 3.590 Other Control Window

iii) In Boundary Conditions>Directional BC: set Y-Low and Y-High to SYMMETRY. Leave the others as default as shown in Figure 3.591.

Figure 3.591 Directional Boundary Condition Window



iv) In Surface BC select Create new and specify the following:

Name	INLET
Surface family	INLET
Density	1.690
X-Velocity	0.831784
Pressure	1.53

The others should be default as shown in Figure 3.592.
Click Accept from the Solver parameters window.

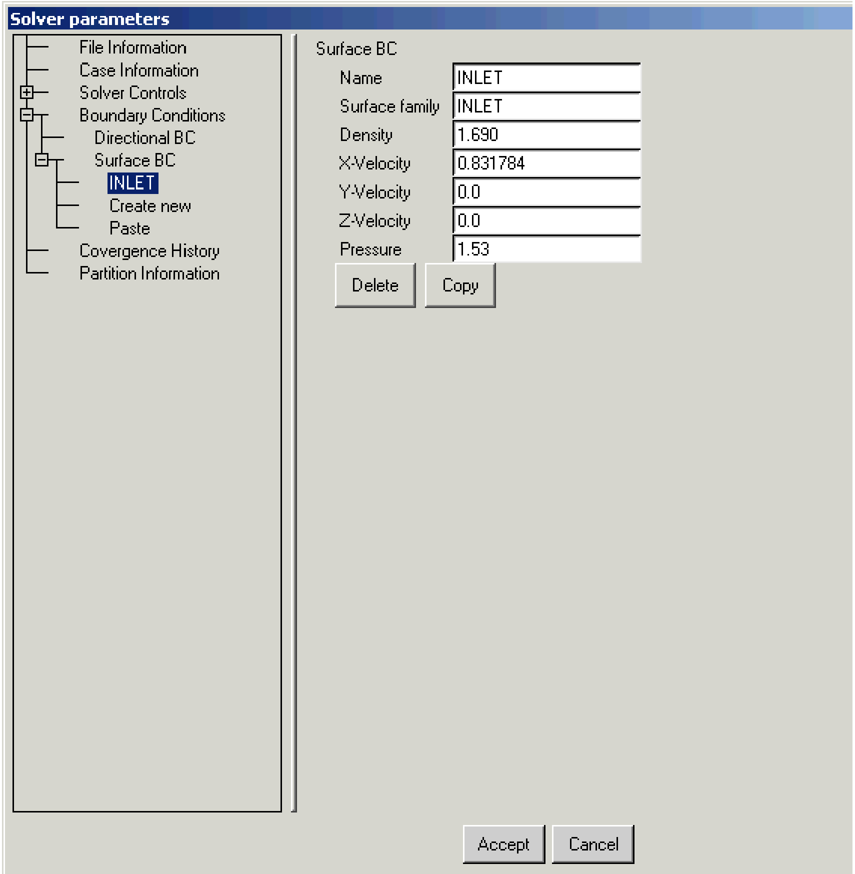
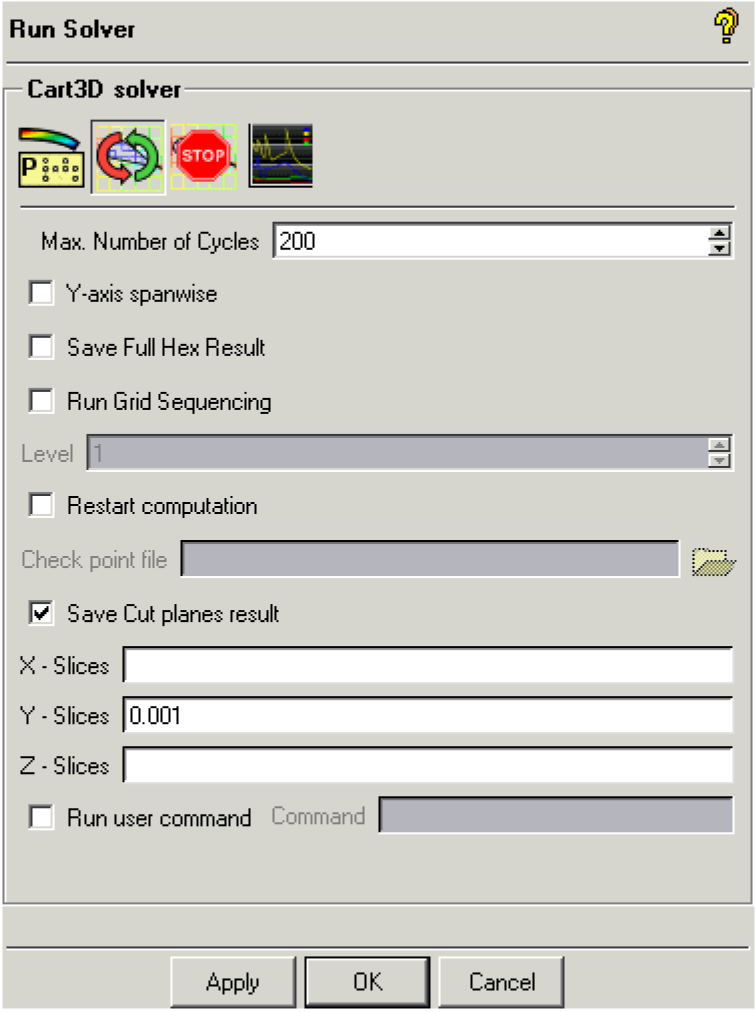


Figure 3.592 Surface Boundary Condition

e) Run solver

Click on Run solver  to get the Run Solver window as shown in Figure 3.593.
Specify Max. Number of Cycles as 200.
Enable Save Cut planes result and specify Y-Slices = 0.001 as shown in Figure 3.593 and press Apply.

Figure 3.593 Run Solver



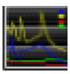
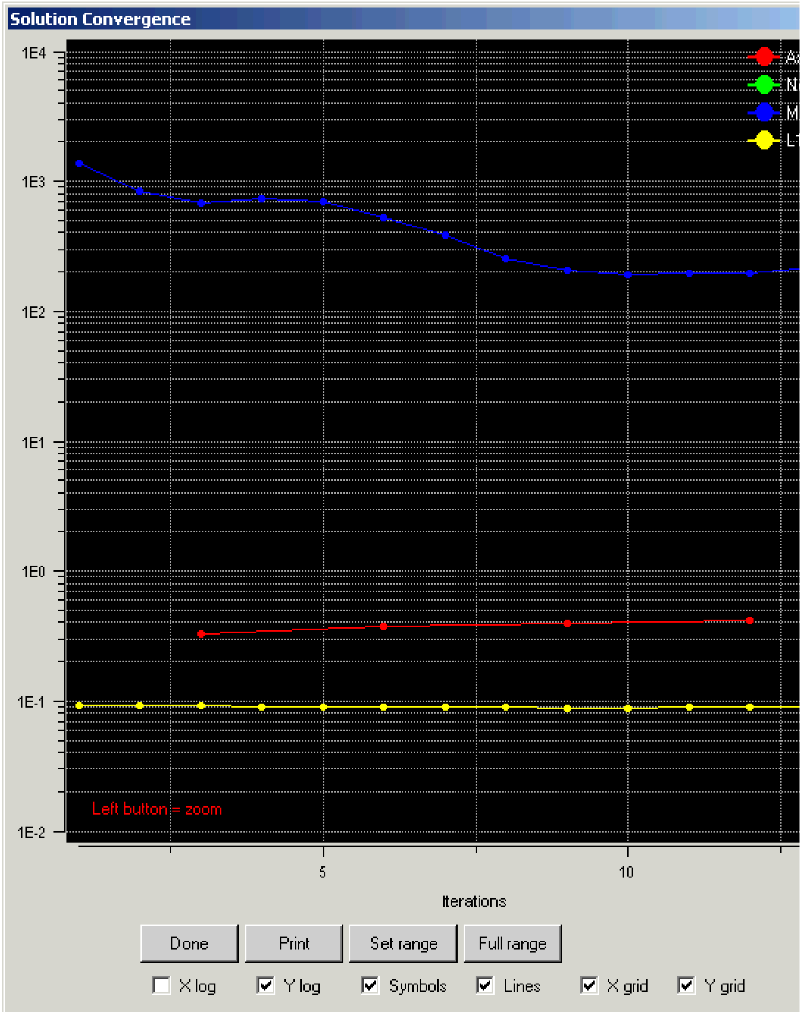
The user can view the convergence by clicking on the Convergence Monitor  to view the plot as shown in Figure 3.594. (The monitor may open automatically.)

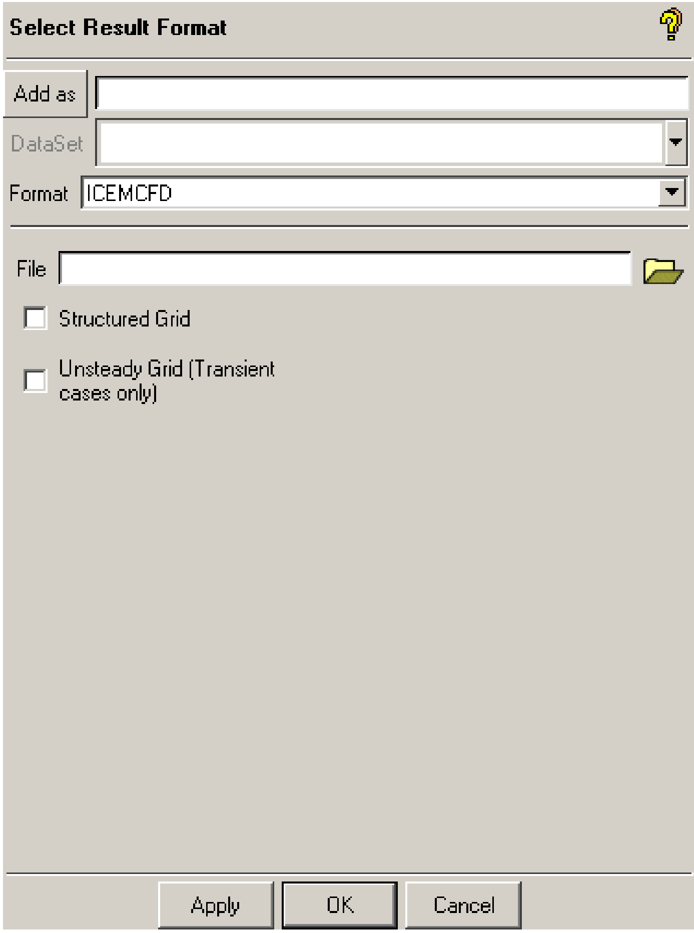
Figure 3.594 Solution Convergence Window



f) Visualization of Results

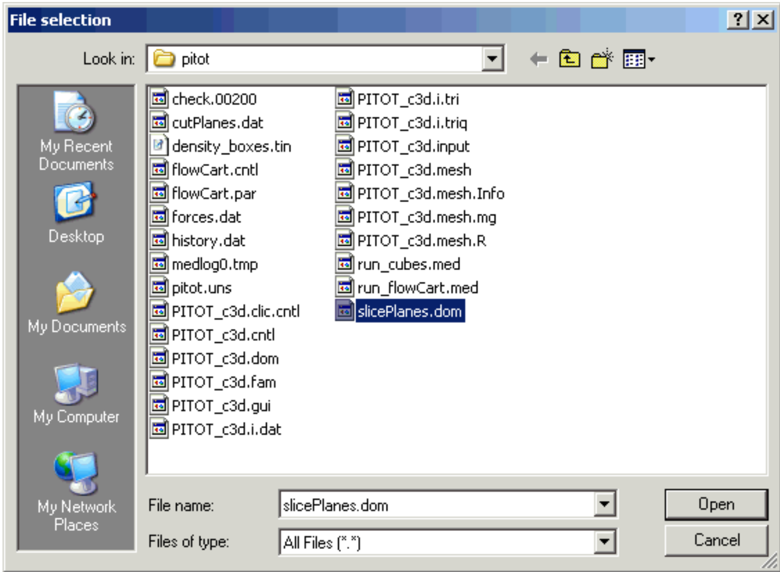
Go to File>Results>Open Results… The Select Result Format window opens as shown in Figure 3.595. Select ICEMCFD as the Format.

Figure 3.595 Select Result Window



Select the file slicePlanes.dom from the **Critical** run as shown in Figure 3.596 and press ‘Open’.

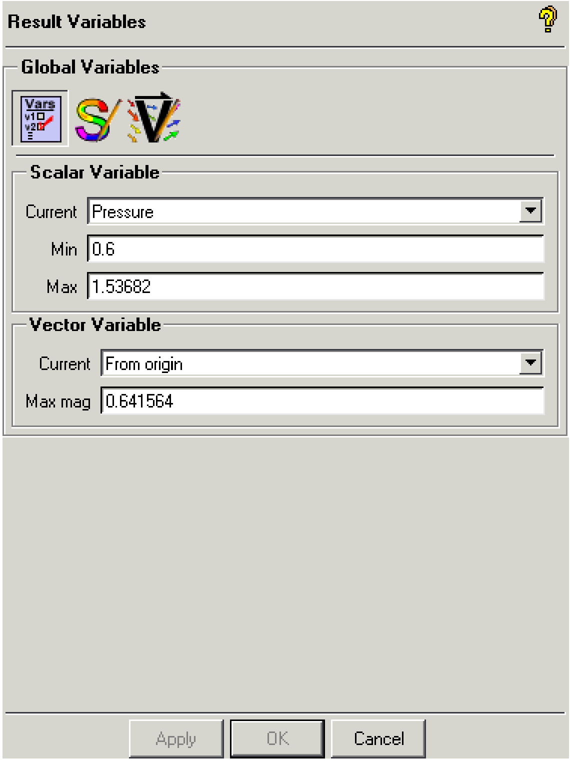
Figure 3.596 File Selection



Press Apply in the Select Result Format window.

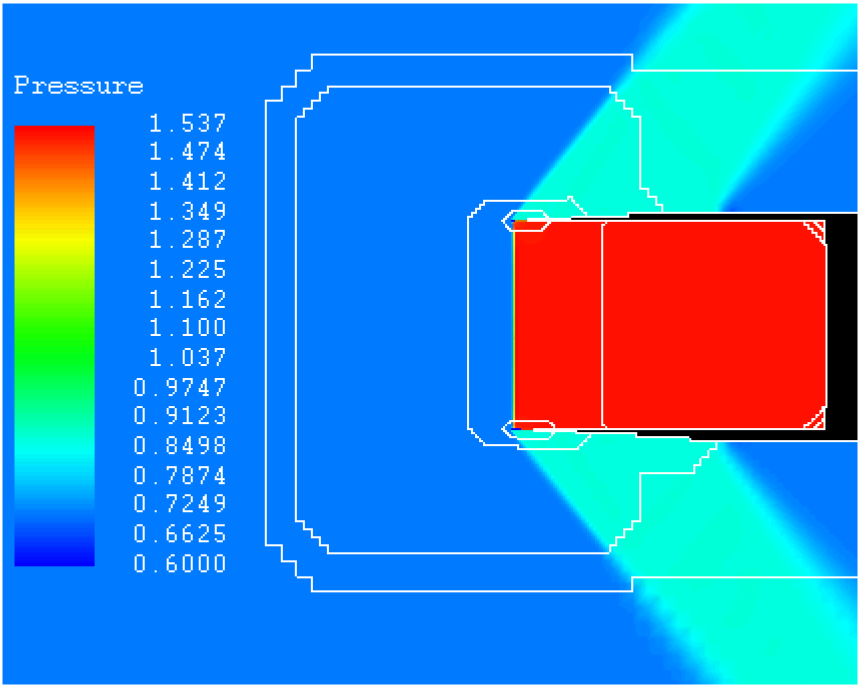
From the Post-processing tab select Variables  and Select Variables . In the Scalar Variable panel select Pressure and set Min = 0.6 as shown in Figure 3.597.

Figure 3.597 Result Variable Window



Press Apply in the Result Variables window to get the image shown in Figure 3.598.

Figure 3.598 Post Process Result



g) Case 2 Sub-Critical

Close the Post Processing session by File > Results > Close Result and confirm to close by pressing Yes.

Change the working directory via File > Change Working Dir… and set the location to the folder **Sub-Critical** into which the original files were copied.

From the Cart3D menu, select Solver > Define solver params. In Boundary Conditions > Surface BC > INLET set Pressure = 1.75. Click Accept to close.

Repeat steps 10.7.5 and 10.7.6 to run the solver and view the results. Be sure to set Max. Number of Cylcles = 200.

The Pressure result is shown in Figure 3.599. Note the Min and Max values for Pressure in the figure are 0.5406 and 1.600, respectively.

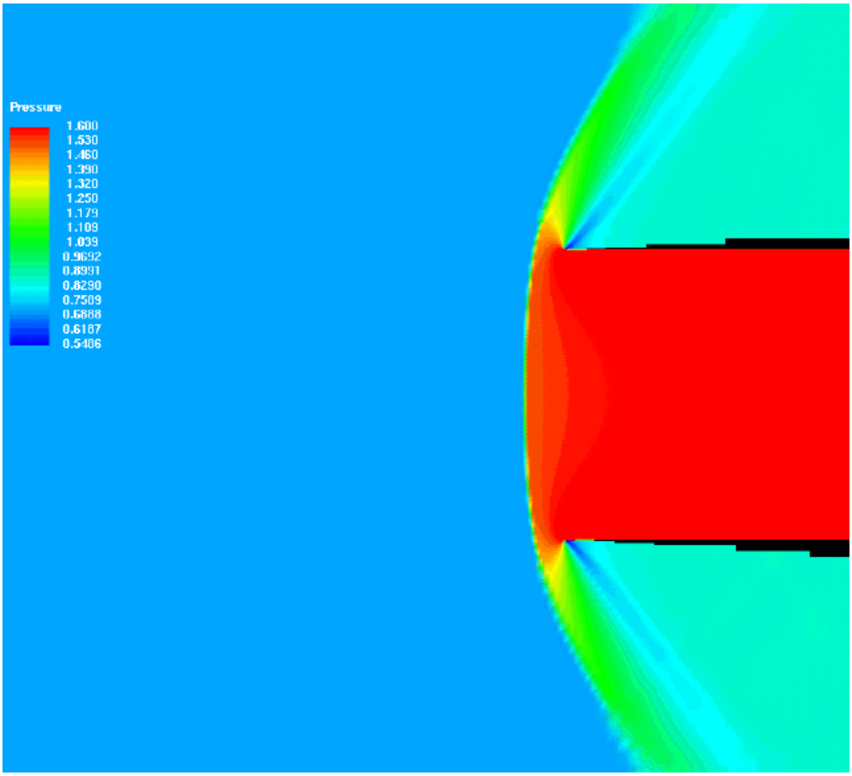


Figure 3.599 Sub Critical Post Processor Result

h) Case 3: Super-Critical

Close the Post Processing session with File > Results > Close Result and confirm to close by pressing Yes.

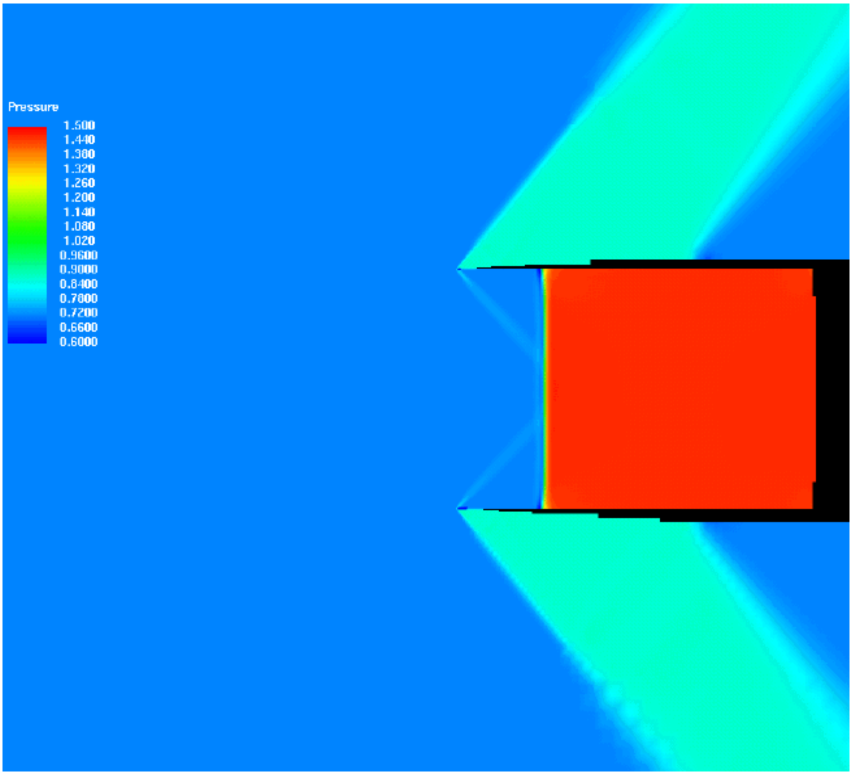
Change the working directory via File>Change Working Dir… and set the location to the folder **Super-Critical** into which the original files were copied.

From the Cart3D menu, select Solver>Define solver params. In Boundary Conditions>Surface BC>INLET set Pressure = 1.42. Click Accept to close.

Repeat steps 10.7.5 and 10.7.6 to run the solver and view the results. Be sure to set Max. Number of Cylcles = 200.

The Pressure result is shown in Figure 3.600. Note the Min and Max values for Pressure in the figure are 0.600 and 1.500, respectively.

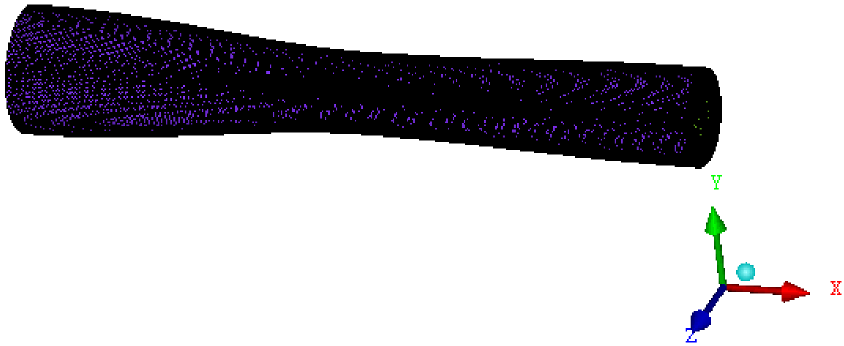
Figure 3.600 Super Critical Post Processor Result



3.7.8: Advanced Tutorial Converging-Diverging Nozzle flow

Overview

The main aim of this tutorial is to study compressible channel flow through a converging-diverging nozzle. This also verifies **INLET/EXIT** BCs with Cart3D.



Three types of flow are simulated.
Case A: Fully Subsonic, $\mathbf{p_{exit}/p_{total}} = 0.89$
Case B: Transonic, $\mathbf{p_{exit}/p_{total}} = 0.75$
Case C: Supersonic, $\mathbf{p_{exit}/p_{total}} = 0.16$

Display

The geometry is an axisymmetric converging-diverging duct. The figure above shows the general shape of the nozzle. It has an area of 2.5 in^2 at the inflow (x = 0 in), an area of 1.0 in^2 at the throat (x = 5 in), and an area of 1.5 in^2 at the exit (x = 10 in). The nozzle Area varies using a Cosine function and has the form:

If x < 5.0 then Area = 1.75 - 0.75 * Cos ((0.2 * x - 1.0) * π).
If x >= 5.0 then Area = 1.25 - 0.25 * Cos ((0.2 * x - 1.0) *π).
This nozzle comes from MS Liou's paper AIAA 87-0355.

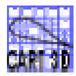
a) Starting the Project

Load **ANSYS ICEM CFD**. Change the working directory by File>Change Working Dir and set the location to the folder **nozzle** (with **nozzle.uns** in that folder).

Note: It is preferable to create a separate folder **nozzle** and put only **nozzle.uns** (domain file) in that folder before performing this tutorial.

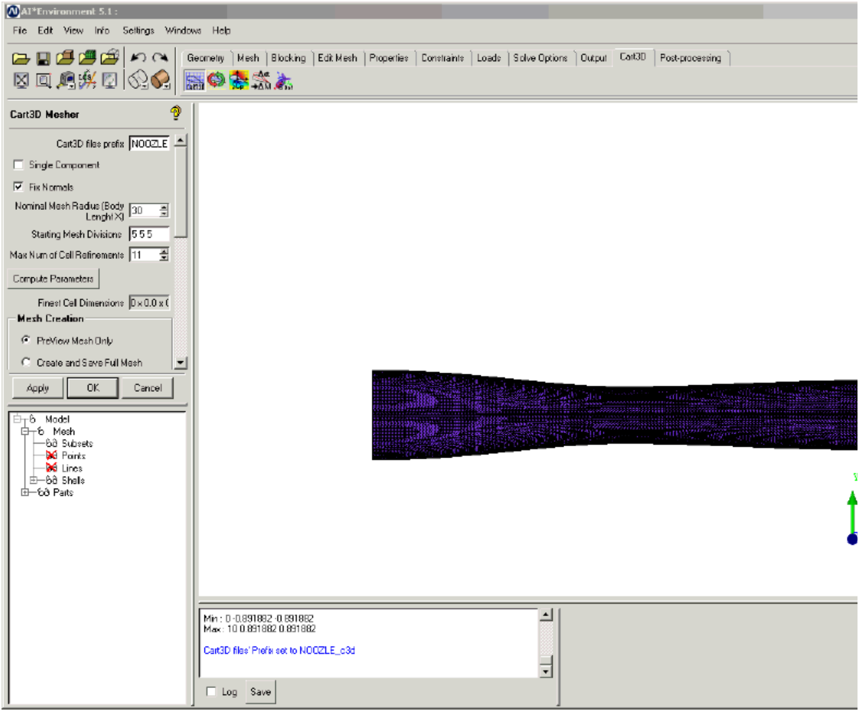
Select Open Mesh  from the main menu and select **nozzle.uns**.

b) Mesh Generation Preview Only

Click on Cart3D from the main menu. Select Volume mesher 

. We get the Cart3D Mesher GUI as shown in Figure 3.601.

Figure 3.601 Cart3D GUI window

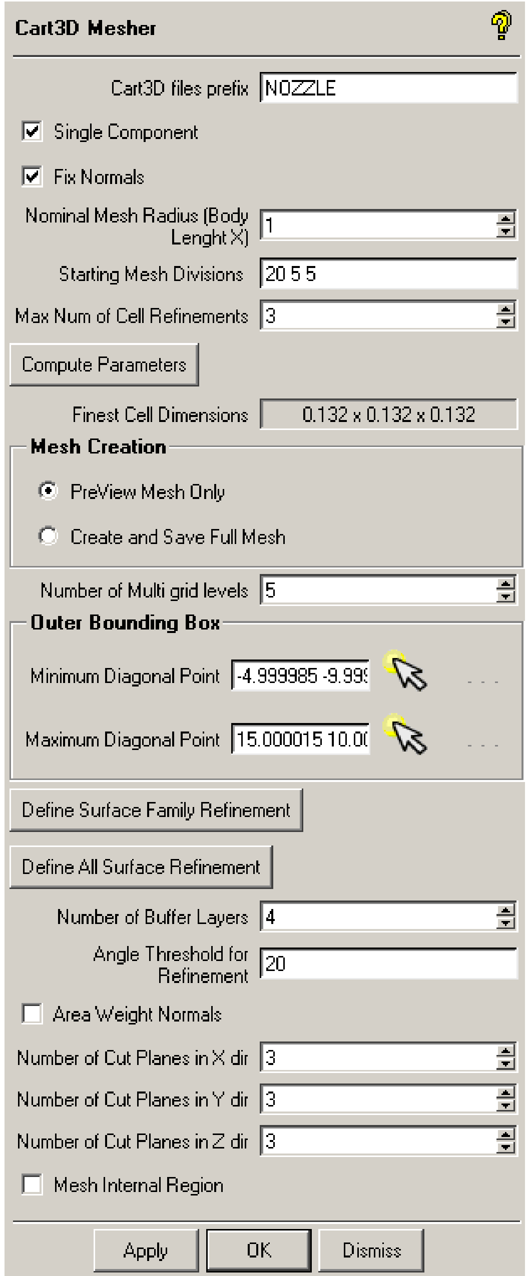


Leave the Fix Normals enabled to give the triangles outward normals.
Enable Single Component
Enter Nominal Mesh Radius (Body Length X) = 1, Starting Mesh Divisions = 20 5 5 and Max Num of Cell Refinements = 3

Leave the other values as default .

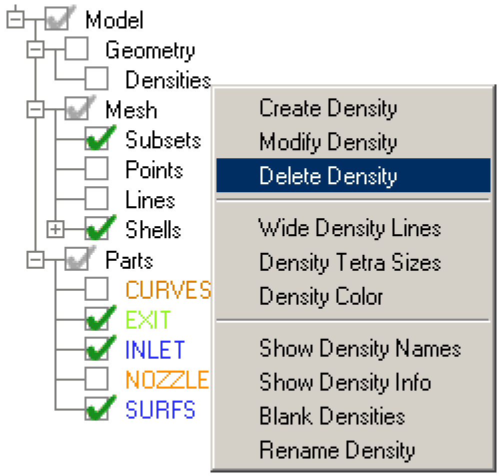
Click Compute Parameters. This saves the mesh in the local directory and converts it into Cart3D format. At the end, it displays the Finest Cell Dimensions as shown in Figure 3.602.

Figure 3.602 Cart3D Mesh Window



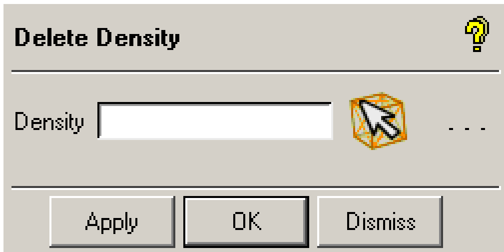
This will create 4 density polygons by default for mesh density control. These can be viewed by enabling Geometry >Densities in the Display Tree widget.
This also computes the finest cell size: 0.132 x 0.132 x 0.132. Varying the Starting Mesh Divisions and/or Max Num of Cell Refinements can vary this.
Right-click on Densities and select Delete Density in the Display Tree widget as shown in Figure 3.603.

Figure 3.603
Display Tree widget Delete



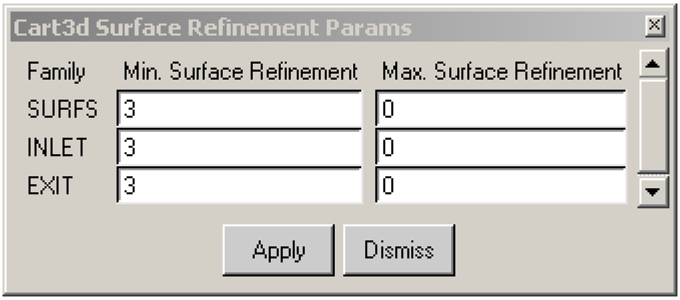
The Delete Density panel opens as shown in Figure 3.604. When in selection mode, select all the densities with the hotkey ‘a’ on the keyboard and press Apply.

Figure 3.604
Delete Density Window



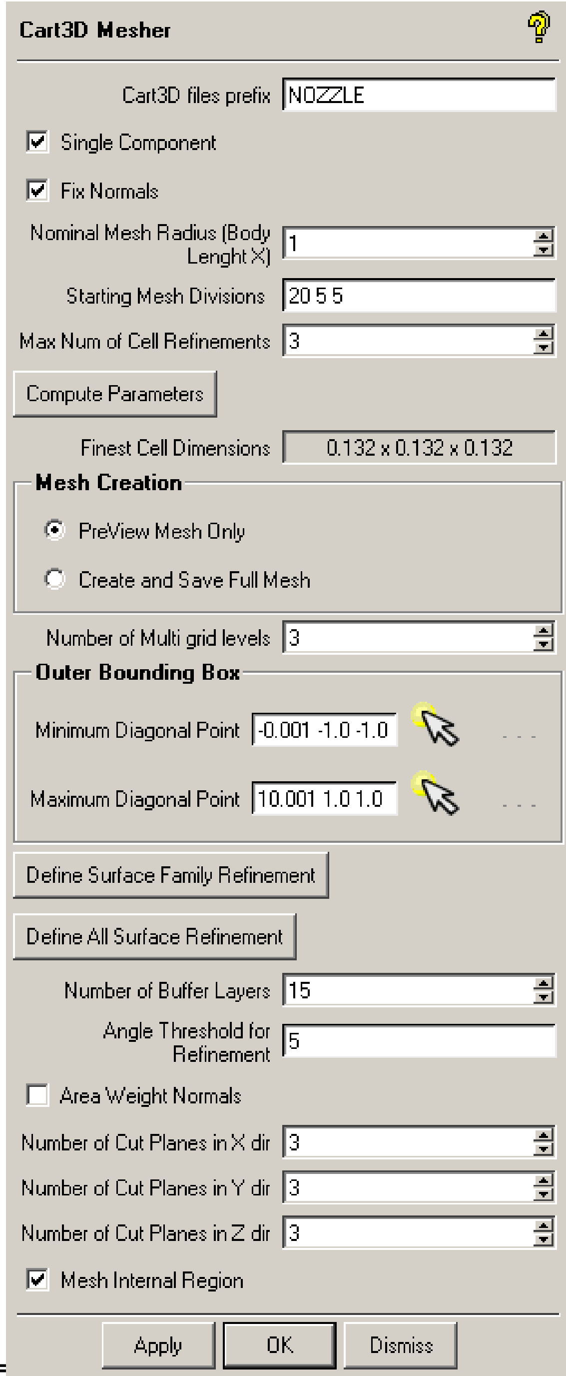
From the Cart3D menu select Volume Mesher . In the Cart3DMesher window select Define Surface Family Refinement In the Cart3d Surface Refinement Params window set Min. Surface Refinement for SURFS, INLET, and EXIT to 3 as shown in Figure 3.605. Press Apply and Dismiss.

Figure 3.605 Cart 3D Surface Refinement Params Window



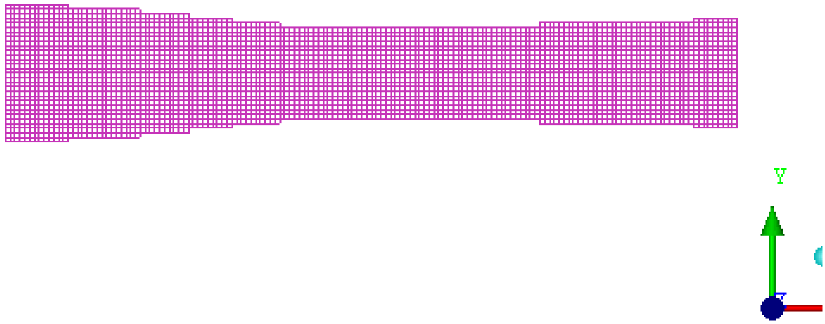
Change the Number of Multi grid levels to 3. Under **Outer Bounding Box** set Minimum Diagonal Point: -0.001 -1.0 -1.0 and Maximum Diagonal Point: 10.001 1.0 1.0 Set Number of Buffer Layers to 15 and Angle Threshold for Refinement to 5. Enable Mesh Internal Region. Make sure PreView Mesh Only is enabled as shown in Figure 3.606.

Figure 3.606 Preview Mesh Parameters



Click Apply to run the mesher. This will create a domain file with 3 Cut Planes (Quad Elements) in each coordinate direction and Cut Cells (Hex Elements) The PreView mesh will be loaded automatically.
Note: As in the case of previous tutorials the mesh can be viewed by switching on the Cut Plane that is to be viewed.
One such view (of CUTPLANE-Z2) is shown in Figure 3.607.

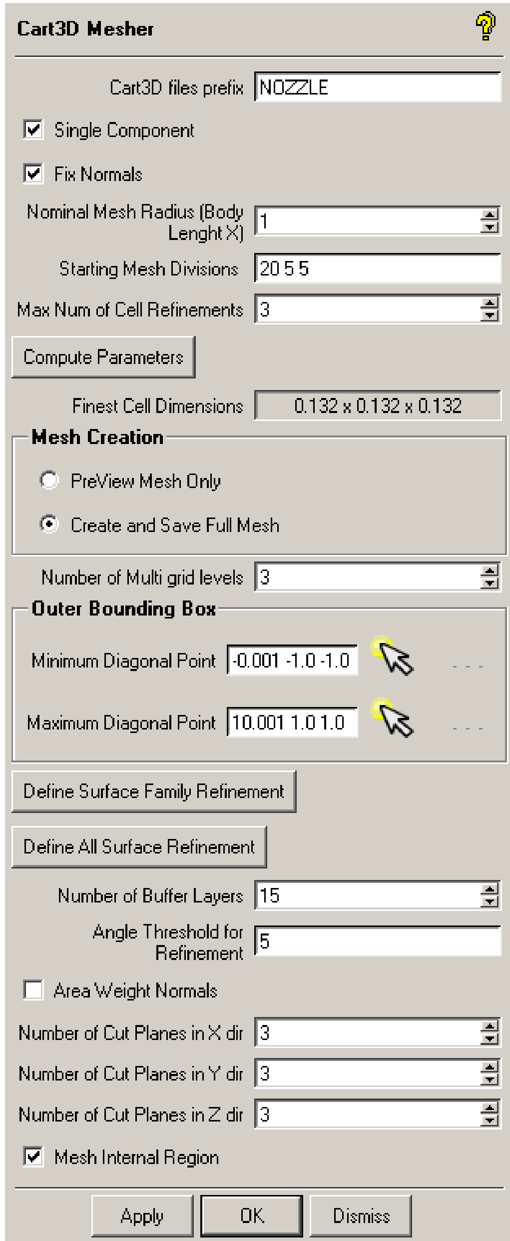
Figure 3.607 Cut Plane Z2 View



c) **Mesh Generation Full Mesh**

In the Cart3D Mesher window enable Create and Save Full Mesh as shown in Figure 3.608 and press Apply. This will create 3 levels of coarsened mesh which can be read by the solver.

Figure 3.608 Create and Save Full Mesh



The Cart3D Mesh window appears which asking about loading the cart3D Full Mesh. Press Yes.

Note: The final mesh generated can be examined through Mesh>Cut Plane as in the previous Tutorials.

d) Overview of Inlet/Exit Boundary Condition

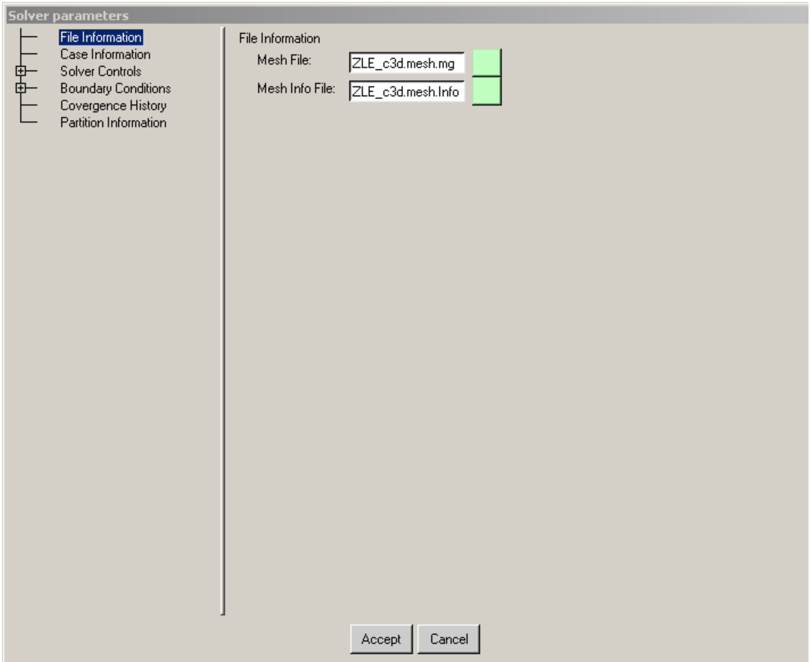
For all cases $\rho_{inlet} = 1.0$, $p_{inlet} = 1/\gamma$ with M_{inlet} taken from the exit solution. Velocities are normalized by the speed of sound at the inlet.
Exit conditions were found by using:
1) Conservation of mass
2) Isentropic Flow relation
3) 1-D normal shock relation
All cases are done using the INLET/EXIT surface Boundary condition for both inflow and outflow. Since there are no cells that get the far-field boundary condition, Mach number, Alpha, and Beta are used to define the initial condition. The subsonic and transonic cases use the subsonic initial condition. This is necessary since we need the solution to choose the fully supersonic solution downstream of the nozzle throat.

e) Case A: Fully Sub-Sonic Flow

Setup Flow Cart Parameters

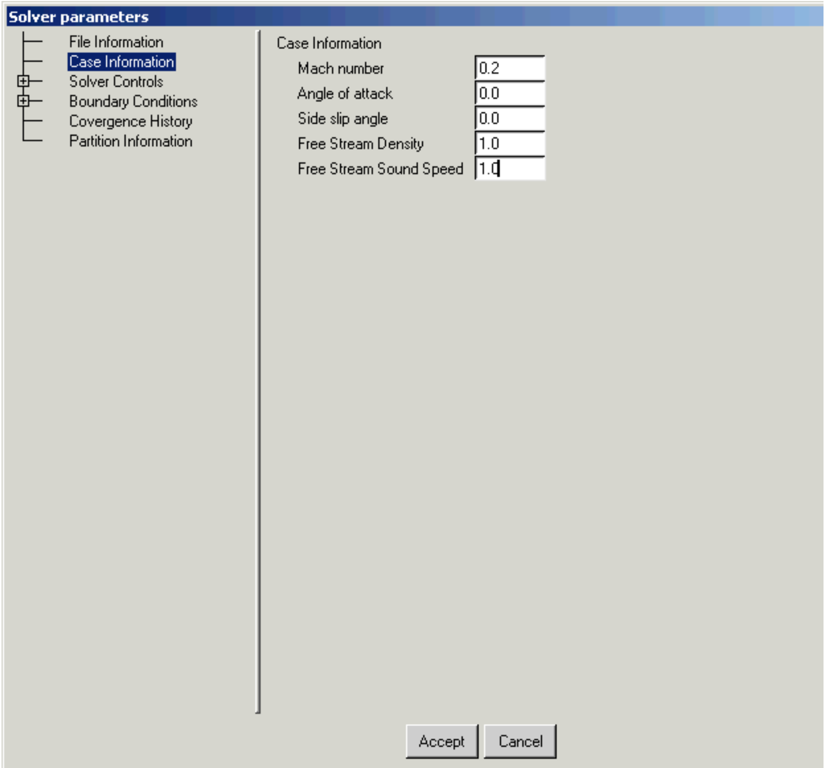
In the Cart3D Menu select Solver. Click on Define solver prams. The Solver parameters window appears as shown in Figure 3.609. (This window may open automatically.)

Figure 3.609 Solver Parameters Window



Choose File Information > Mesh File as **NOZZLE_c3d.mesh.mg** (this should be default).
Click on Case Information and enter Mach number = 0.2. Leave the other parameters as default as shown in Figure 3.610.

Figure 3.610 Case Information Window

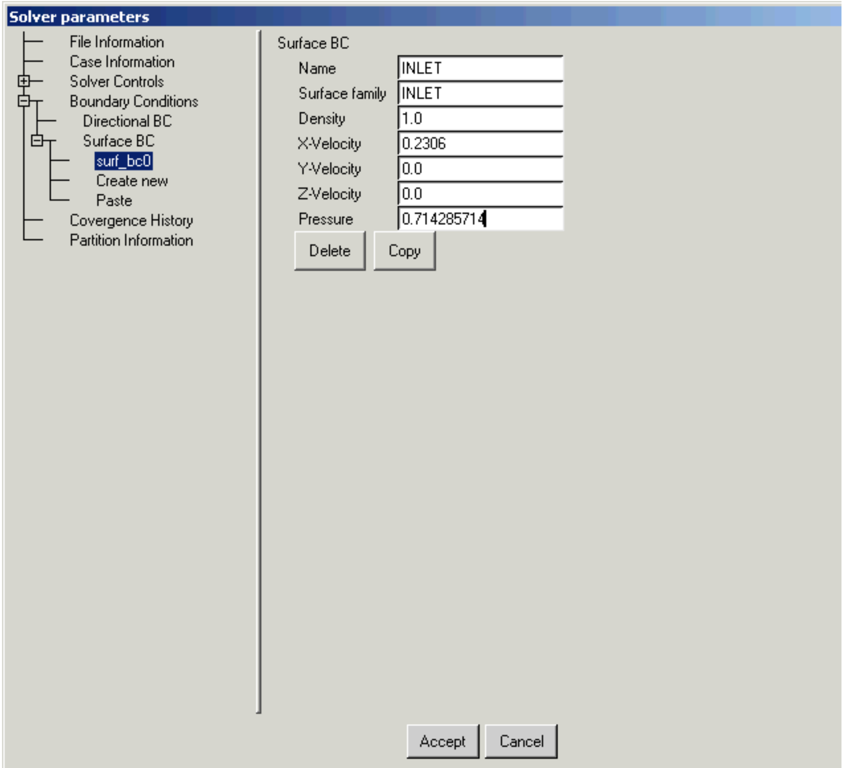


Under Solver Controls > Other controls set Number of Multi-Grid levels to 3.
Click on '+' for Boundary Conditions and '+' for Surface BC. Select Create New and enter the following:

Name	INLET
Surface family	INLET
Density	1.0
X-Velocity	0.2306 (from exact solution)
Pressure	0.714285714

The values are shown in Figure 3.611.

Figure 3.611 INLET Boundary Condition



Since the flow is fully subsonic there is no shock wave present in the nozzle and
Isentropic relations hold good.

$t_{total} = p * (1 + (\gamma -1)*M^2/2)\text{^}(\gamma /(\gamma -1)) = 0.7412294$

$p_{exit}/p_{total} = 0.89; => p_{exit} = .65969403$

$p_{inlet}/(\rho_{inlet})\text{^}\,\gamma = p_{exit}/(\rho_{exit})\text{^}\,\gamma$

$\rho_{exit} = .944801$

Conservation of mass:

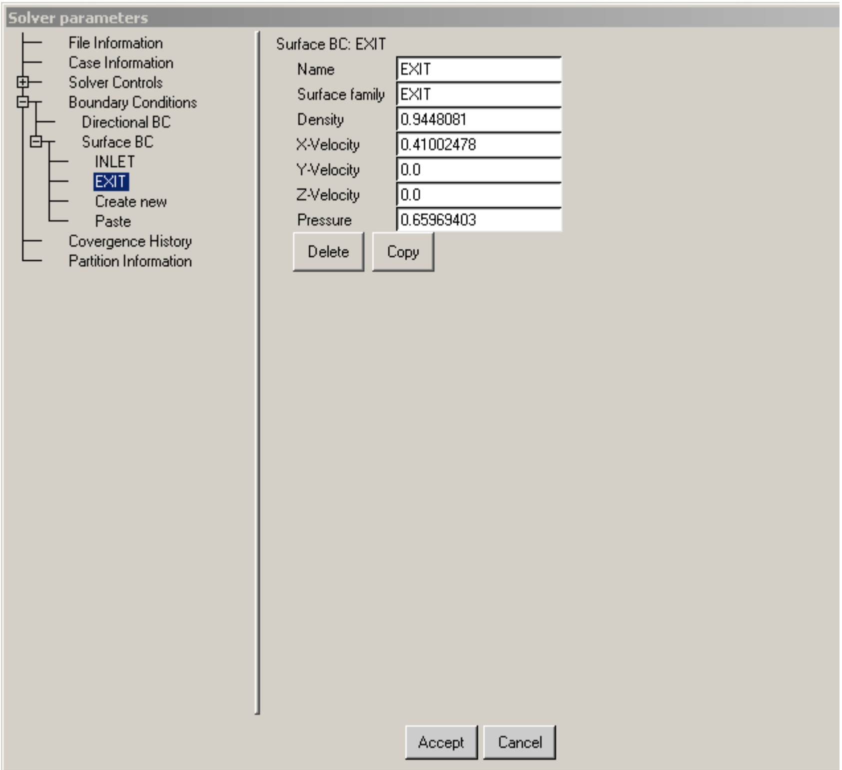
$\rho_{inlet} *Area_{inlet}*V_{inlet} = \rho_{exit} *Area_{exit}*V_{exit}$

$M_{exit} = .41002478$

Select Create new and specify the Surface BC for EXIT by entering the following values as shown in Figure 3.612.

Name	EXIT
Surface family	EXIT
Density	0.944801
X-Velocity	0.41002478
Pressure	0.65969403

Figure 3.612 Exit Boundary Condition



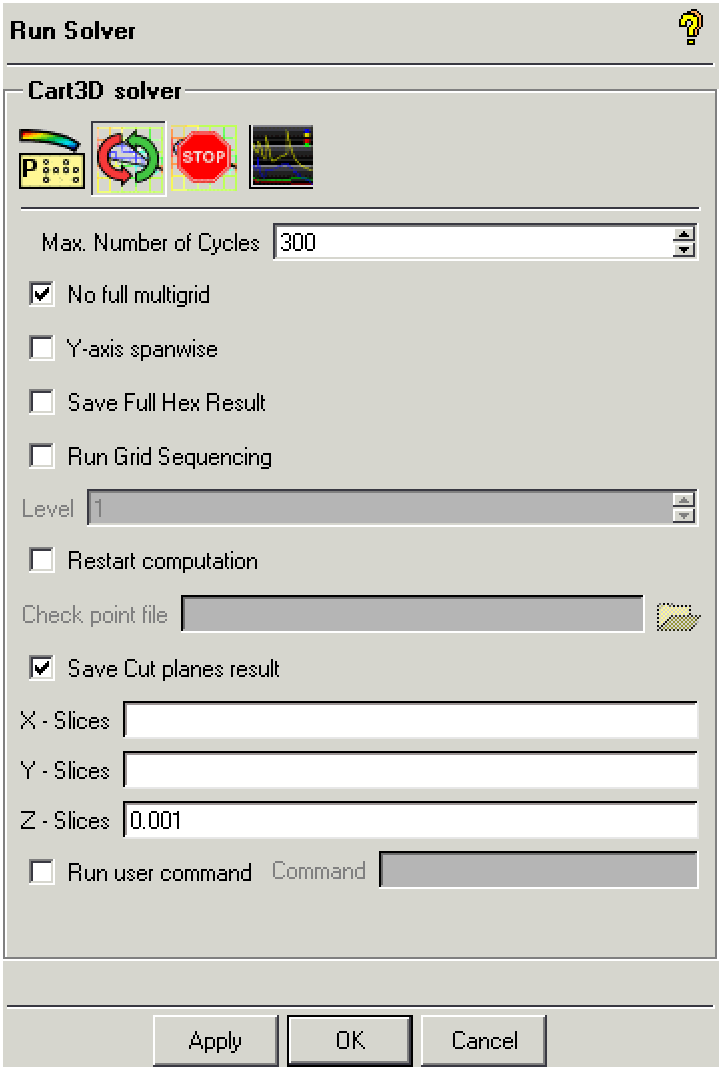
Use the other defaults and press Accept in the Solver parameters window.

f) Running the FlowCart Solver

Select Solver > Run solver to open the solver panel.
Specify Max. Number of Cycles = 300.
Enable No full multigrid.

Enable Save Cut planes result, and specify Z-Slices = 0.001 as shown in Figure 3.613. Press Apply.

Figure 3.613 Run Solver Window



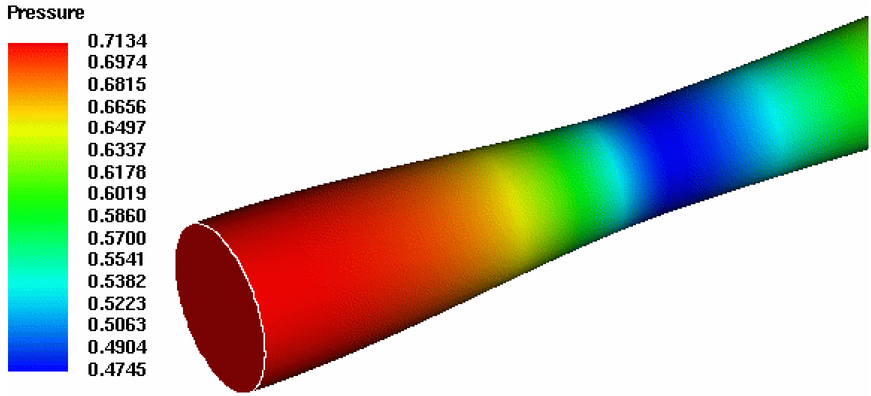
Note: Post processing is explained in previous tutorials. Follow the same procedure to view the results.

Mach number results in **slicePlanes.dom** for the Sub-Sonic Flow case are shown in Figure 3.614.

Figure 3.614 Sub Sonic Result Mach Number

Pressure results in **surface_results.dom** for the Sub-Sonic Flow case are shown in Figure 3.615.

Figure 3.615 Sub Sonic Result for Pressure



Select File > Results > Close Result to end the post processing session.

g) Case B: Transonic flow

Only the INLET and EXIT boundary conditions need to be changed
Surface BC:

Name	INLET
Surface family	INLET
Density	1.0
X-Velocity	0.23954 (for choked flow)
Pressure	$1/\gamma = 1/1.4 = .714285714$

Theory

In this case, a normal shock will occur downstream of the throat. So, isentropic relations are not valid and 1-D normal shock relations must be used to find exit conditions.

$t_{total} = p * (1 + (\gamma\text{-}1)*M^2/2)\hat{}(\gamma/(\gamma\text{-}1)) = 0.743390$
$p_{exit}/p_{total} = 0.75$
$p_{exit} = .55754252$
From 1-D normal shock relations:
$\rho_{exit} = 0.81060$

Conservation of mass
$\rho_{inlet} *Area_{inlet}*V_{inlet} = \rho_{exit}*Area_{exit}*V_{exit}$
$M_{exit} = .492519$

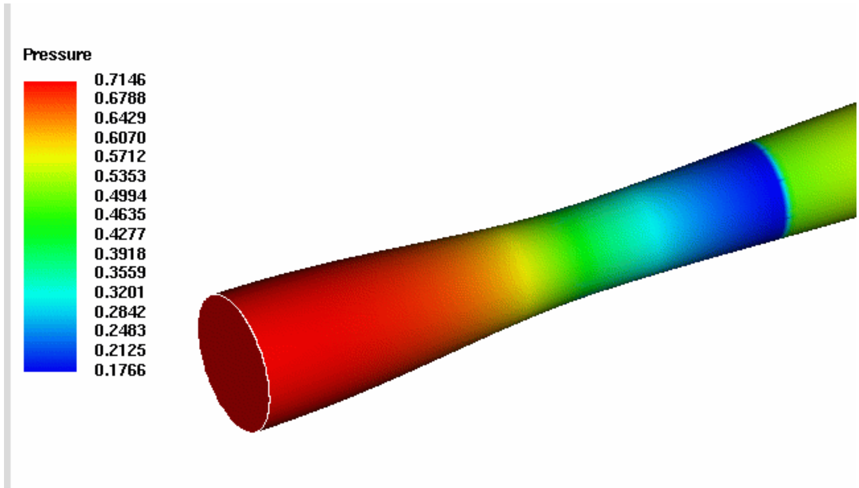
Name	EXIT
Surface family	EXIT
Density	0.81060
X-Velocity	0.492519
Pressure	0.55754252

Run the solver using the same procedure as for the Sub-Sonic case.
Mach number results from **sliceplanes.dom** for the Trans-Sonic Flow case are shown in Figure 3.616.

Figure 3.616 Trans sonic Result Mach Number

Pressure results in **surface_results.dom** for the Trans-Sonic Flow case are shown in Figure 3.617.

Figure 3.617 Trans Sonic Result for Pressure



h) Case C: Supersonic Flow

Note: For the Supersonic Flow case the Mach number needs to be changed to Mach number = 1.5 and:

From the Cart3D menu select Solver > Define solver params > Case Information and set Mach number = 1.5.

Theory

Since EXIT is a supersonic outlet it really doesn't matter what happens here as long as it allows a supersonic exit.

Since the flow is fully supersonic downstream after throat there is no shock wave present in the nozzle and isentropic relations still hold good.

$t_{total} = p * (1 + (\gamma -1)*M2/2)^{\wedge(\gamma / \gamma -1)} = 0.743390$

$p_{exit}/p_{total} = 0.16; \Rightarrow p_{exit} = .1189424$

$p_{inlet}/(\rho_{inlet})^{\wedge\gamma} = p_{exit} /(\rho_{exit})^{\wedge\gamma}$

$\rho_{exit} = .278127$

Conservation of mass:

$\rho_{inlet} *Area_{\_inlet}*V_{\_inlet} = \rho_{exit} *Area_{exit}*V_{exit}$

$\Rightarrow M_{\_exit} = 1.435448$

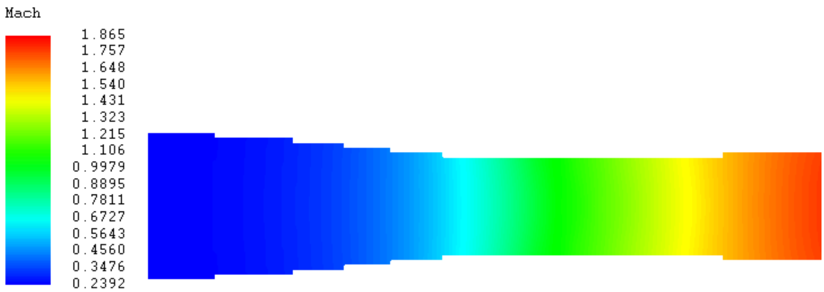
A change in the EXIT boundary condition needs to be incorporated. Expand Boundary Conditions>Surface BC

Name	EXIT
Surface family	EXIT
Density	0.278127
X-Velocity	1.435448
Pressure	0.1189424

Select Accept from the Solver parameters window and run the solver as before.

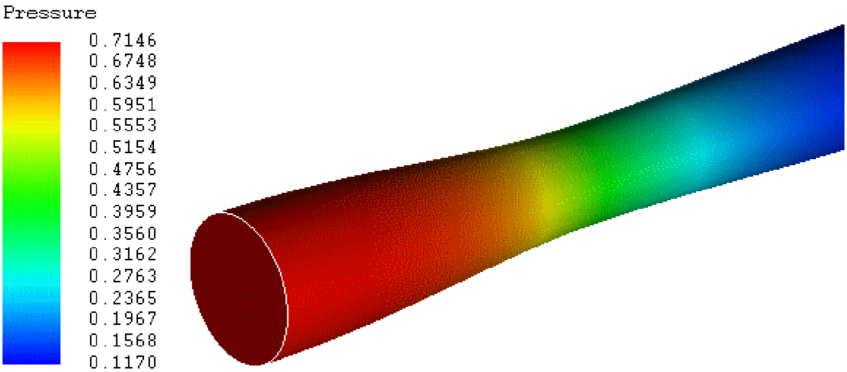
Mach number results from **sliceplanes.dom** for the Supersonic Flow case are shown in Figure 3.618.

Figure 3.618 Supersonic Flow Mach Number



Pressure results in **surface_results.dom** for the Supersonic Flow case are shown in Figure 3.619.

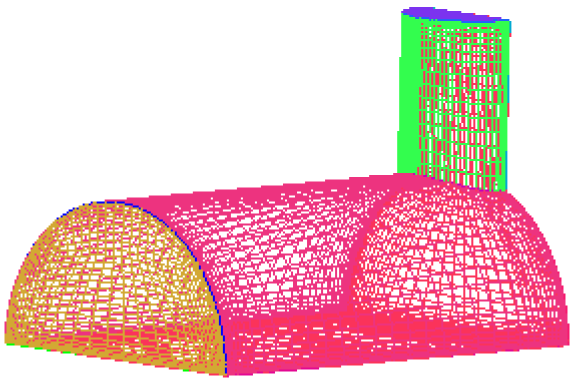
Figure 3.619 Super Sonic Flow Pressure



3.8: Output to Solvers

Now that the user has generated several meshes for the 3D Pipe Junction, he/she can write these meshes for input to any of the supported solvers. This section will provide two options for solvers -- one for unstructured domains (STAR-CD), and another for structured domains (CFX-TASCflow).

Figure 3.620
The hexa unstructured mesh for the 3D Pipe Junction. This mesh may be used for input to the STAR-CD solver.



a) Summary of Steps

Choosing the appropriate solver with Select Settings > Solver.

Adding boundary conditions with Boundary conds.

Writing output to the selected solver.

Note: Different solvers need different output structures. For example, STAR-CD requires an unstructured format, while CFX-TASCflow requires multi-block structured format. Input to the STAR-CD output interface then should be an unstructured Hexa, Tetra or Prism domain file, and input to the CFX-TASCflow output interface should be a set of structured Hexa domain files.

Note: For information on each of the output interfaces, consult the web page http://www-berkeley.ansys.com/interfaces/ToC.html where links are located, providing information to the details of each interface.

For users who wish to write output files for unstructured mesh, follow the instructions provided by the subsection Unstructured Mesh.

For users who wish to write output files for structured mesh, follow the instructions provided by subsection Structured Mesh.

Users may also successively go through both sections to write output files for different regions of the model.

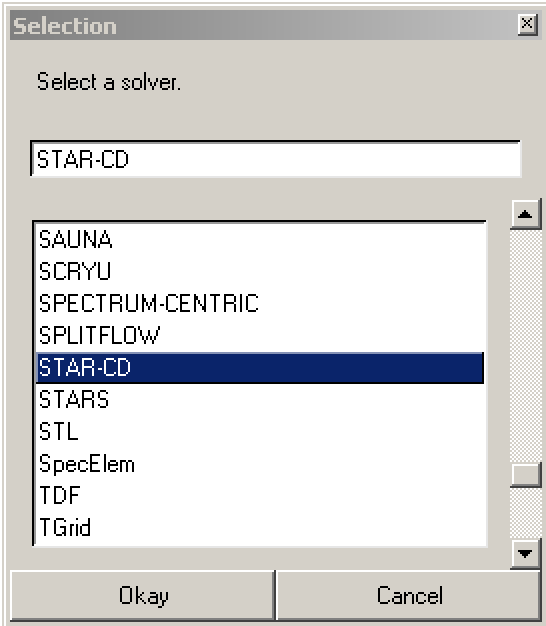
3.8.2: Unstructured Mesh

If the 3DPipeJunct is not the current project, choose File > Open Project and, from the File selection window, choose 3DpipeJunct and press Accept.
Load the Tetin file geometry.tin and the unstructured Hexa mesh hex.uns.

a) Setting your Solver

Select Output > Select solver  to open the Selection window shown in Figure 3.621.

Figure 3.621
Select STAR-CD



From the Selection window, select STAR-CD and then press Okay.

b) Editing Boundary Conditions

With the solver set, the user can browse and set solver-specific boundary conditions in the Mesh Editor.

Press Output > Boundary conds . This will bring up the Family boundary conditions window as shown in Figure 3.622

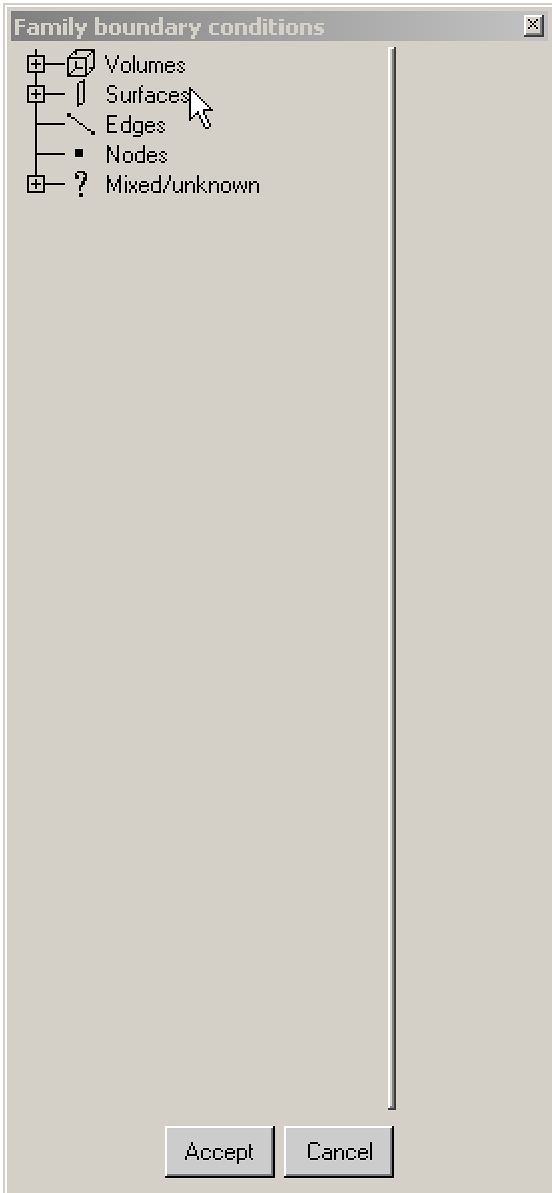
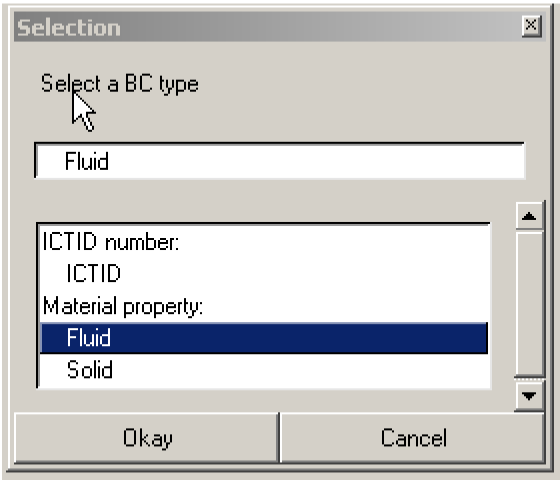


Figure 3.622
The family boundary condition window

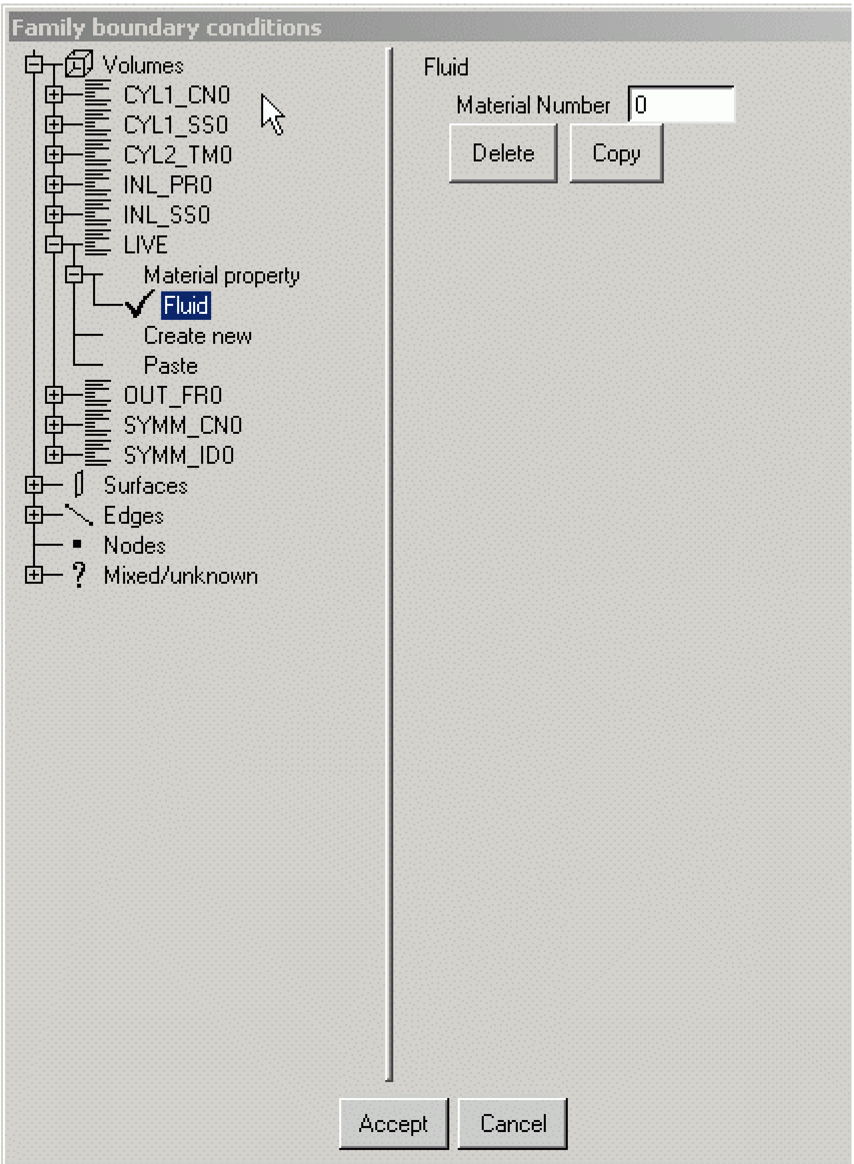
In the Family boundary conditions window (Figure 3.623), select Volumes > LIVE > Create new. This will open up a window to select the type of cells you have. Select Fluid BC type in this window as shown in Figure 3.623.

Figure 3.623
Select the FLUID BC to LIVE family



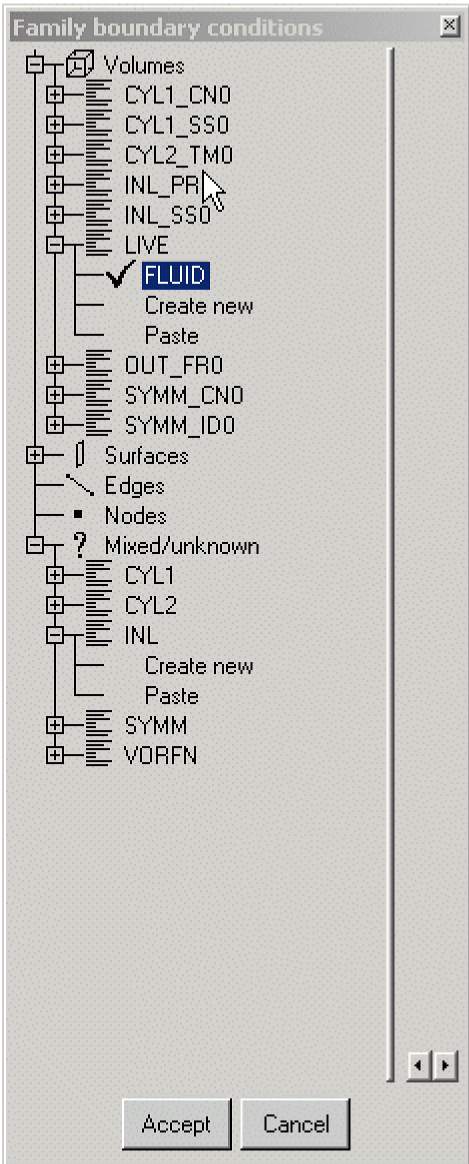
Press Okay and this should change the Family boundary conditions window as shown in Figure 3.624.

Figure 3.624 After defining cell type



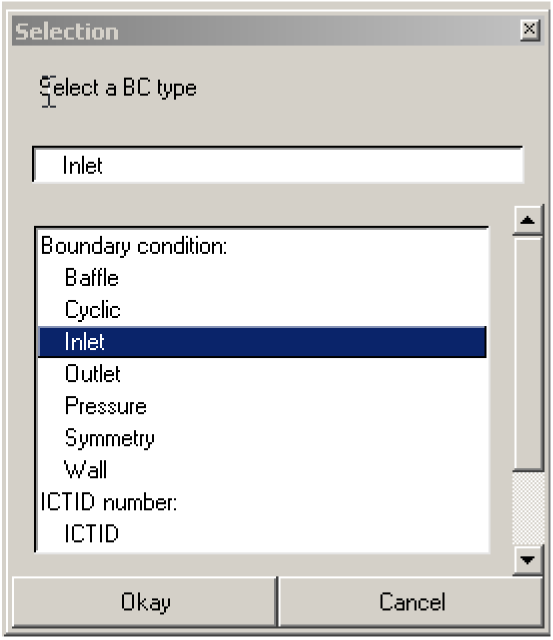
Now, from the Selection window, select Mixed/unknown > INL > Create new as shown in the following Figure 3.625.

Figure 3.625 Defining INL family type



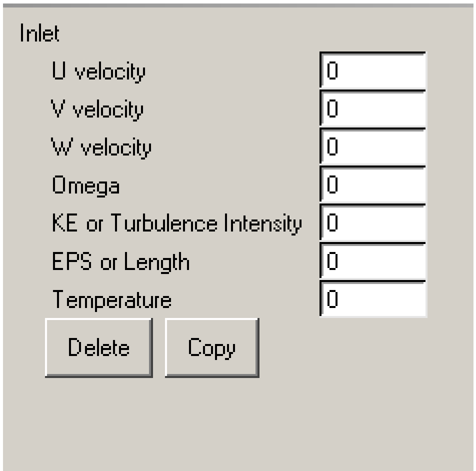
This will open up the bc selection window where select the BC type as Inlet as shown in Figure 3.626.

Figure 3.626 Choose Inlet as the BC type for INL family



Press Okay to close this window. Back in the Family boundary conditions window, set values to the boundary condition, as shown in Figure 3.627 and then press Accept.

Figure 3.627
Edit the boundary condition values



Before continuing, select File > Save Project to confirm that all modifications are stored in the family_boco.fbc file used for output.

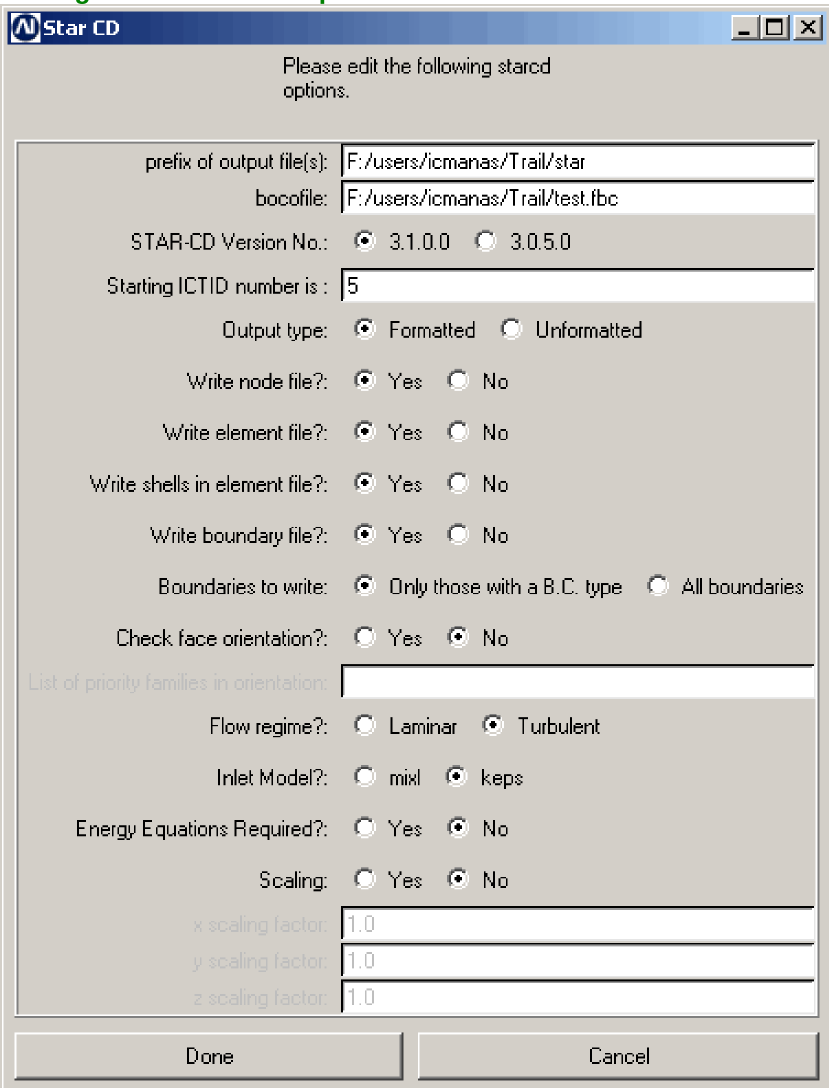
Some solvers accept input files that store information (solver execution, etc.) unrelated to ICEM CFD grid information. If the user's solver is one of those for which we support output of this additional file, you can set up the solver parameters in this file by selecting the params button adjacent to the Boundary conds button.

c) Writing the Solver Input File

Next, choose Output > STAR CD input to write STAR-CD input files, and choose the Hexa unstructured domain to write to STAR-CD format. It is opening the Star CD window shown in Figure 3.628.

As in Figure 3.628, assign the parameters for the STAR-CD input file set. Make sure that the boundary condition file is selected as family_boco.fbc.

Figure 3.628
Setting the STAR-CD file parameters



Select Done to close the Star-CD window , and the ICEM CFD messages window will indicate when the translation process is complete.

When the translation process is complete, the STAR-CD files will have been written to the 3DPipeJunct directory. The STAR-CD solver is now prepared to run with that file set.

Select File > Quit to close the ANSYS ICEMCFD window. The remainder of this section deals with writing output files for structured mesh.

3.8.3: Structured Mesh

Start ANSYS ICEMCFD and press File > Open > Project and, from the File selection window, choose 3DpipeJunct. Press Accept.

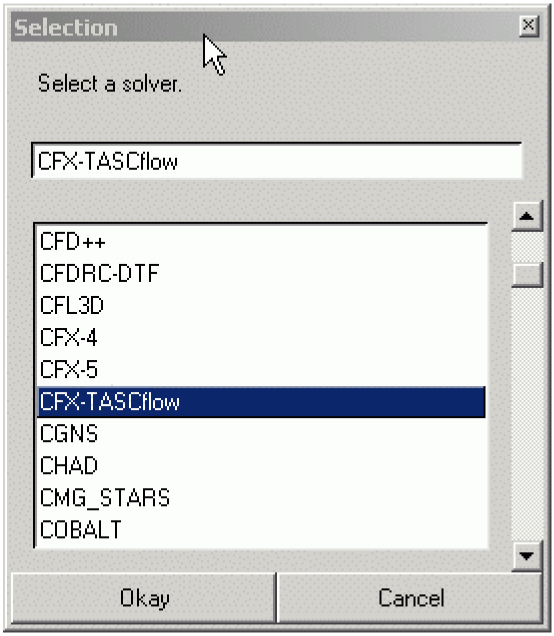
Load the Tetin file geometry.tin and the Multiblock structured Hexa mesh.

a) Writing Output to a Solver

The user can write output to CFX-TASCflow with the Multiblock version of the mesh.

Press Output > Select solver.

Figure 3.629 Select your second solver, CFX-TASCflow



From the Selection window, select CFX-TASCflow (refer to Figure 3.629). Press Okay.

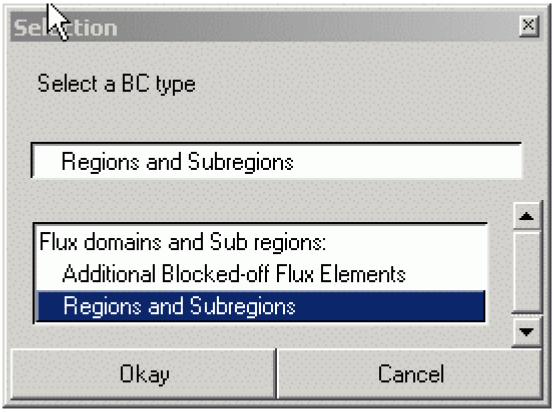
b) Solver-Specific Boundary Conditions

With the solver set to CFX-TASCflow, the user can define regions that CFX-TASCflow will recognize.

Press Output > Boundary conds . This will bring up a file selection window to select an existing boundary condition file. We should not pick the existing file since that is for STARCD. Press Cancel there. This will bring up the Family boundary conditions window.
From the Family boundary conditions window select Surfaces > Mixed/unknown > OUT > Create new. This will open up a window to

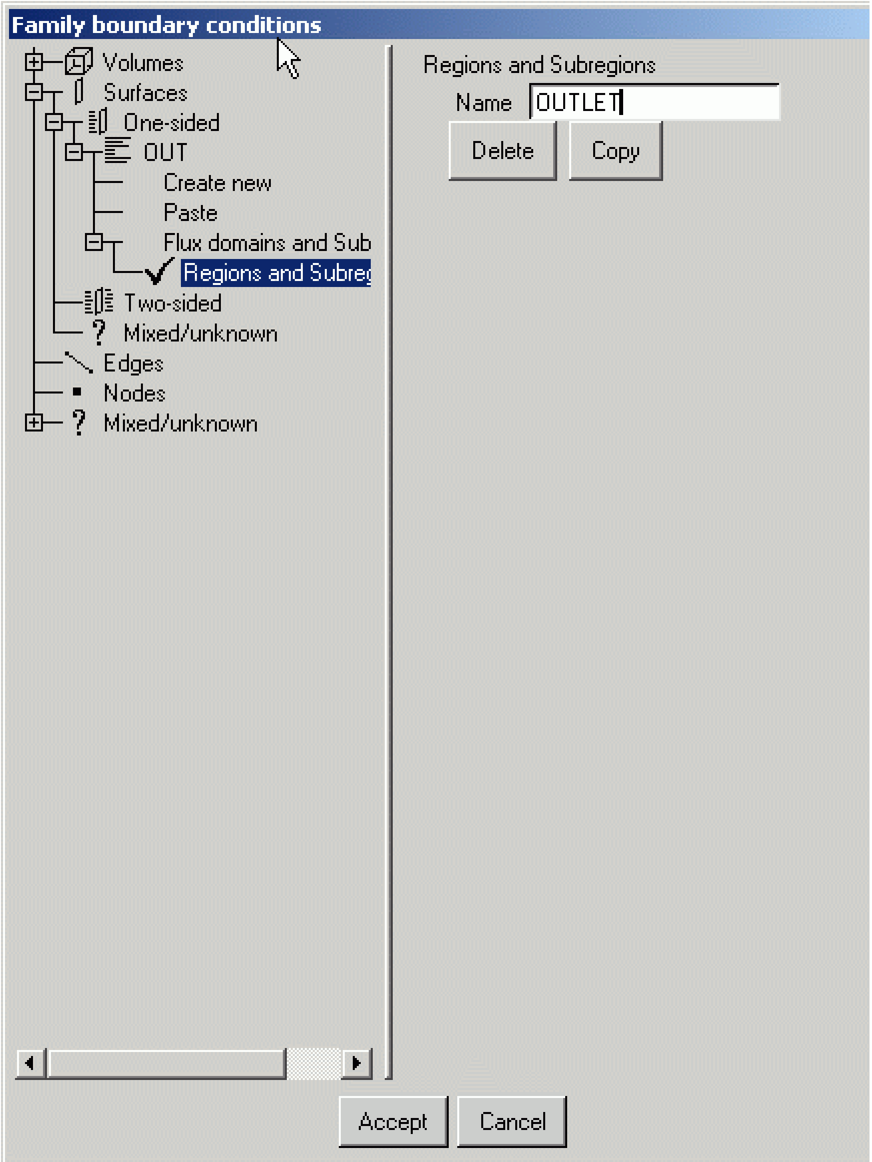
choose (Figure 3.630). Double-click Regions and Sub regions under Flux domains and Sub regions.

Figure 3.630 Creating a region on the OUT family



Name this region “OUTLET” as shown in Figure 3.631.

Figure 3.631 Name the region



The user may continue on to define other regions or flux-domains of the 3D pipe.

When the user has finished defining the desired regions, press Accept from the Family boundary conditions window.

Select File > Save Project.

Finally choose Output > CFX-TASCflow input.

Set the parameters for input to the solver, as shown in Figure 3.632.

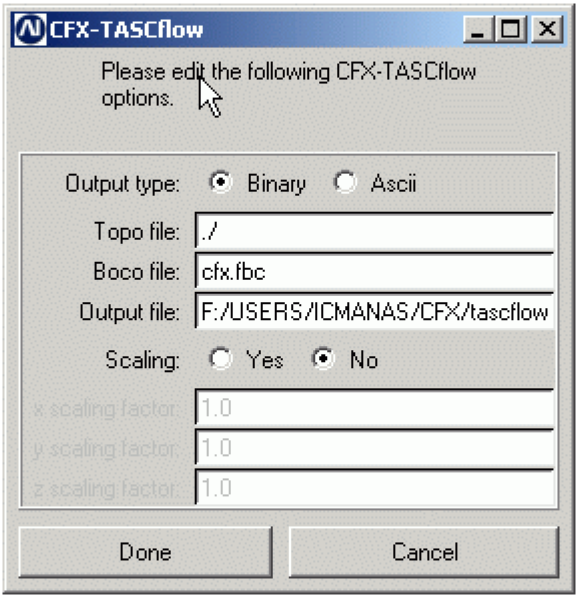


Figure 3.632 Set the translation parameters for CFX-TASCflow

Press Done to complete the translation to CFX-TASCflow database.

The user may now select File > Quit to exit.

3.9: Post Processing Tutorials

3.9.1: Pipe Network

In this tutorial, one will be referring to a Fluent file as an example. The input files are different from solver to solver but the post-processing is very similar. Normally, for post processing, ICEM CFD generated grid is not needed. In this case, only Fluent's case and data files are sufficient to show the results.

Operations introduced by this example

Starting up a new Visual3p project

Visualization of Surface Grid and Solid Contours

Plotting Solid Contours

Displaying Surfaces with Contour lines

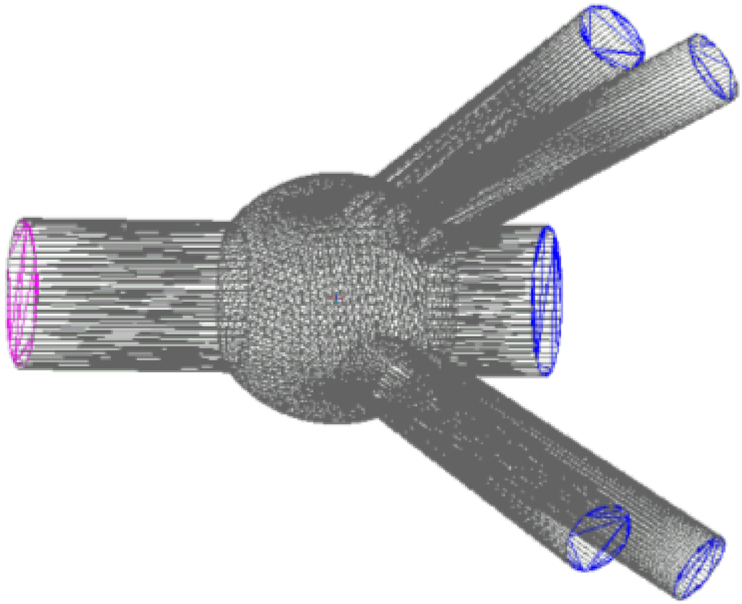
Visualization of Surface edges and Vector plots

Saving the Output

a) Case Description

The geometry of the Fluent file used for this tutorial is as shown in Figure 3.633. The configuration consists of an inlet pipe, which finally splits into four outlet pipes of similar area and another outlet with a larger area of cross section. The junctions of the inlet and outlet pipes are connected through a spherical region, which would be the most important section to analyze. The fluid material used for the analysis is Air.

Figure 3.633 Geometry for the Fluent file

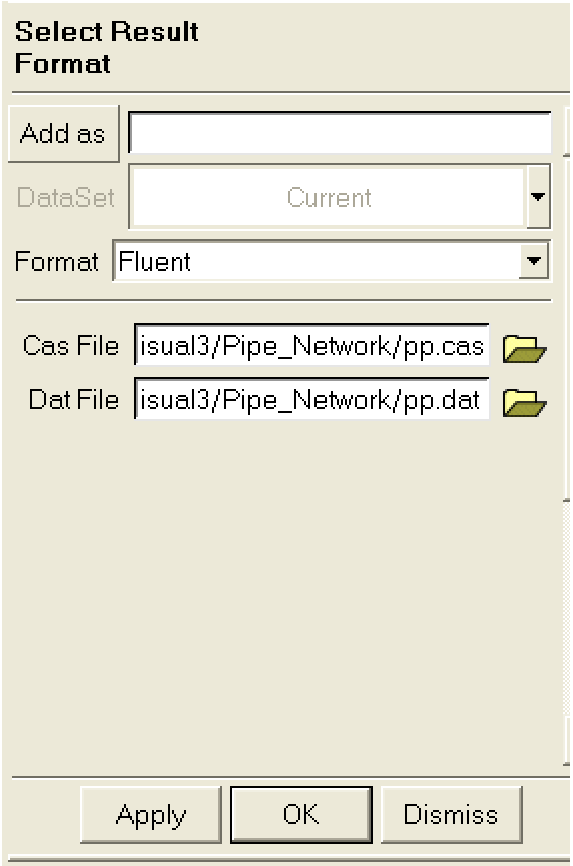


b) Starting up Post Processing

Go to the option, File > Results > Open Results to start the Post Processing of the results obtained from the different solvers.

This will open up Select Result Format window as shown in Figure 3.634. From this window, user can select different solver formats for which user wants to do post-processing of the results.

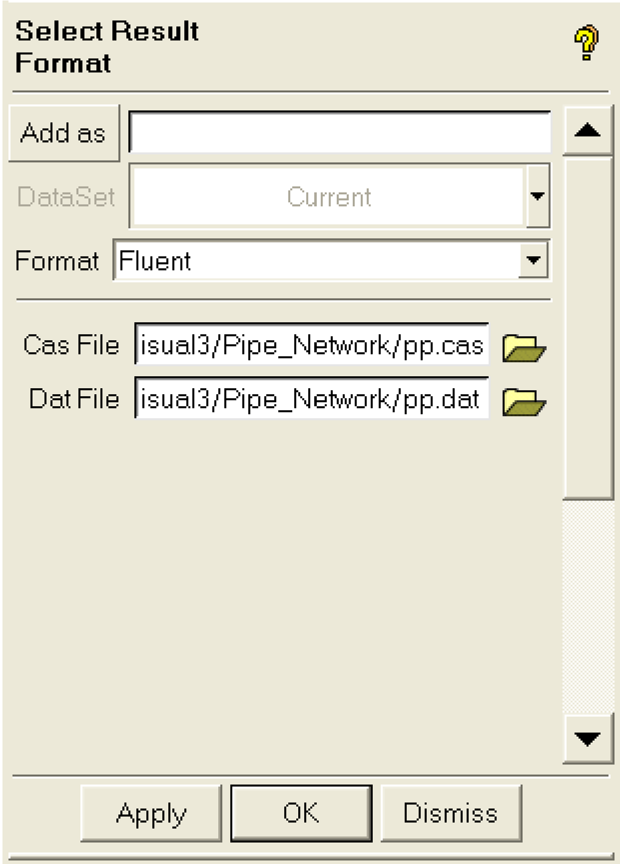
Figure 3.634 Select Result Format window



c) Reading Fluent Files

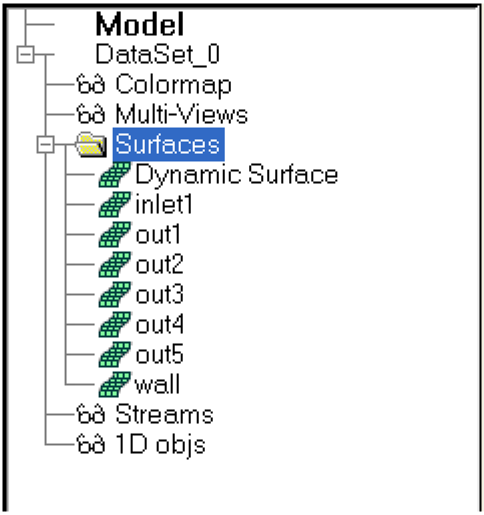
From the **Select Result Format** window, select Format as **Fluent** and press Apply. This will pop-up Fluent files selection window as shown in Figure 3.635. Through this window, browse the directories and select the **pp.cas** file with the square button towards right side of the Cas file option. Similarly, read **pp.dat** as the data file for Dat file. Finally, press **Apply** from Fluent files window. This will load the fluent case in the main window.

Figure 3.635 Reading Fluent Case and Data file



The boundary names (Family names) are read in by ANSYS ICEMCFD for easy post-processing. These names are organized in the model tree of the data set, Figure 3.636.

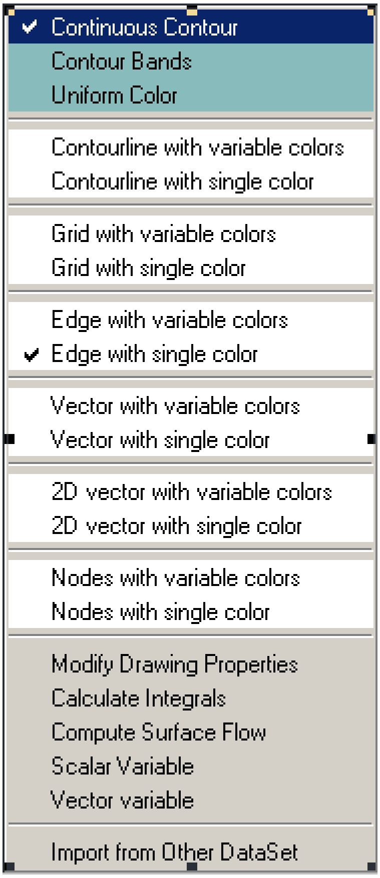
Figure 3.636
Surface Manager
under Display Tree



d) **Visualization of Surface Grid and Solid Contours**

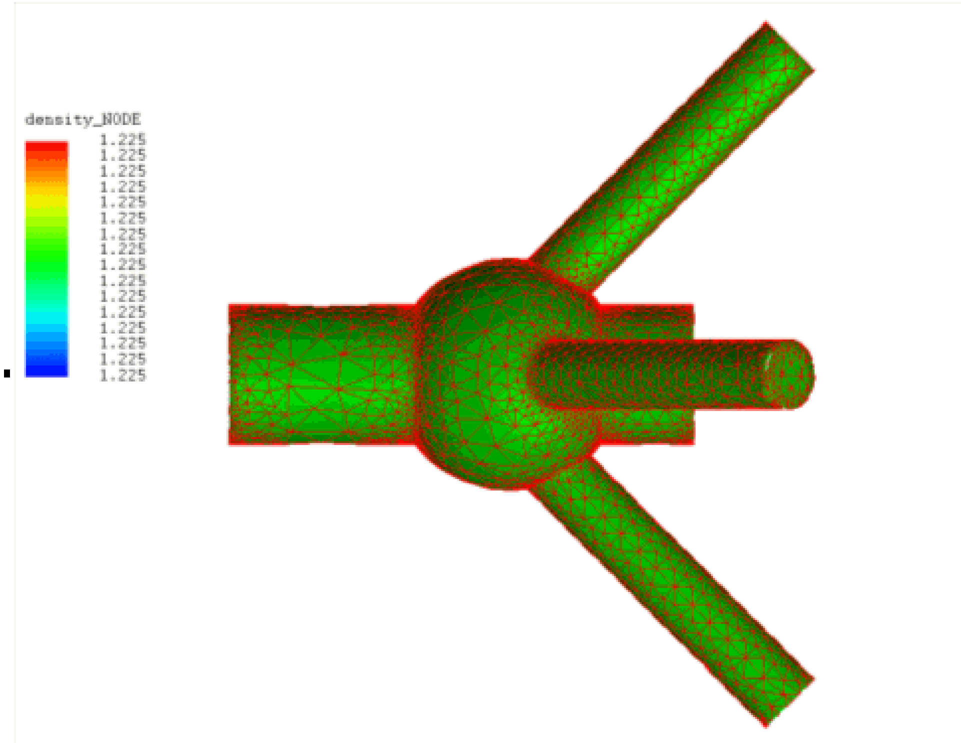
For post processing, the display controls are provided for individual surfaces. User can access those options from the entry to the surfaces in the Display Tree widget. To access these options, user has to click right hand mouse button after selecting the Surface from the tree. To do changes globally, user has to select “Surfaces” in the tree and click the right mouse button on it. The different options to control the display of the result variable are shown in Figure 3.637.

Figure 3.637
Right click options on Surfaces branch



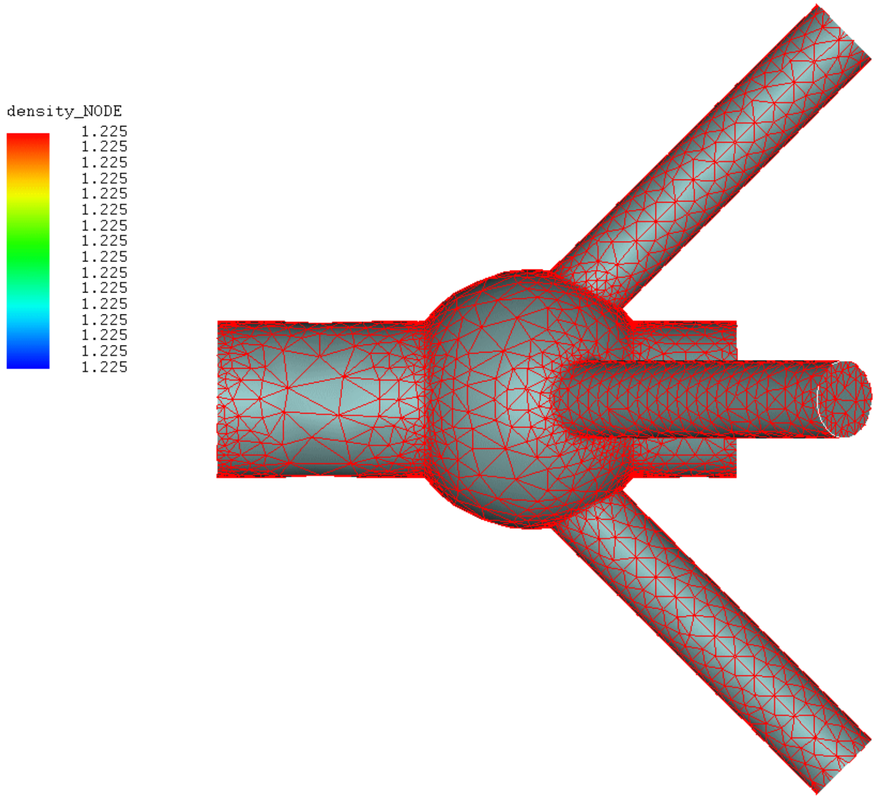
Select the option Grid with variable color from the pop-up menu list. The surfaces would now be displayed with the mesh in scalar variable color as shown in Figure 3.638. The user would find the mesh in one color since the default scalar variable density_NODE is not changing in this example.

Figure 3.638
Grid with Variable color:



To view the grid with the geometry, click the right mouse button on Surfaces and choose Uniform Color option. The surfaces would now be displayed with continuous contours as shown in Figure 3.639

Figure 3.639
Surface displayed with mesh and Solid color



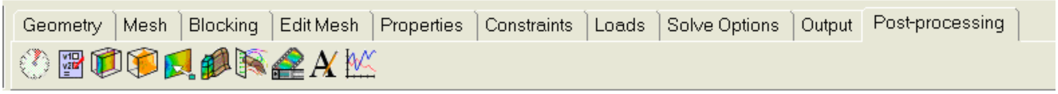
Switch OFF the Grid with variable colors and Uniform colors options by selecting them again.

e) Plotting Solid Contours

The mesh was displayed in uniform color since the variable density_NODE is constant in this example. One would now switch to another scalar variable i.e. pressure_NODE which is more appropriate in this situation.

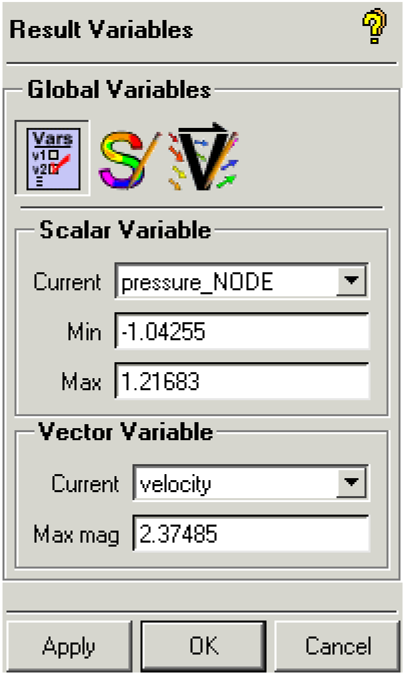
Select the Variables icon from the Post-processing tab Menu bar; Figure 3.640.

Figure 3.640
Post processing tab



This will pop out the Result Variables window shown in Figure 3.641.

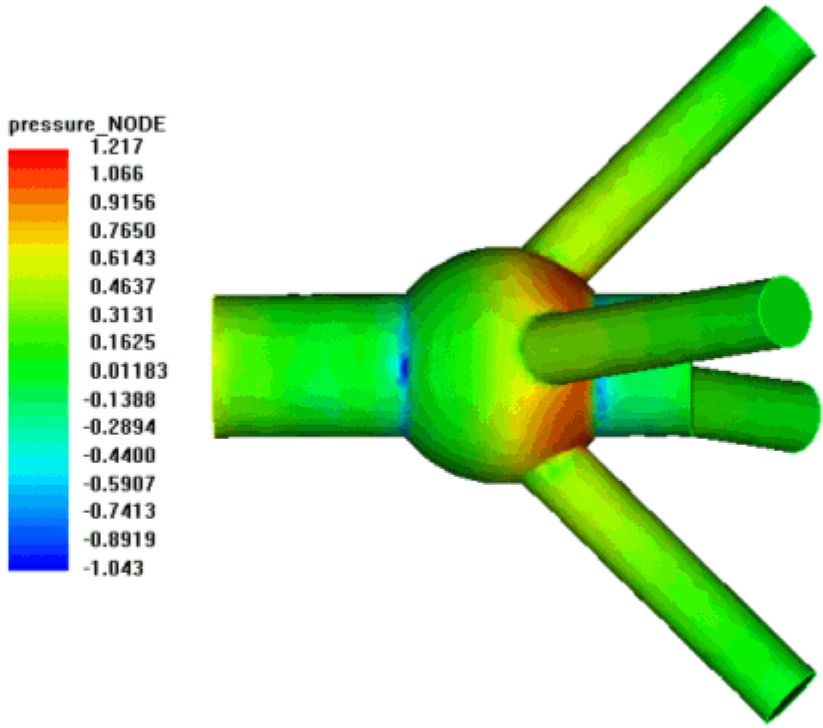
Figure 3.641
Variables window



Change the variable selection to pressure_Node from the list of Scalar Variables drop down menu. After changing the variable, close the Variables window by clicking on the OK button.

Toggle off Grid with Variable Colors and click the option Continuous Contour to display the pressure contours in solid shading as shown in Figure 3.642.

Figure 3.642
Solid Contours with Pressure Variation



In order to observe the solid contours in Flat shading, select Modify Drawing Properties option from Surfaces options. It will open up Selected Surfaces' Properties window, Figure 3.643 From this window, click on Use flat shading button to display the result in Figure 3.644.

Figure 3.643 Surface Properties panel

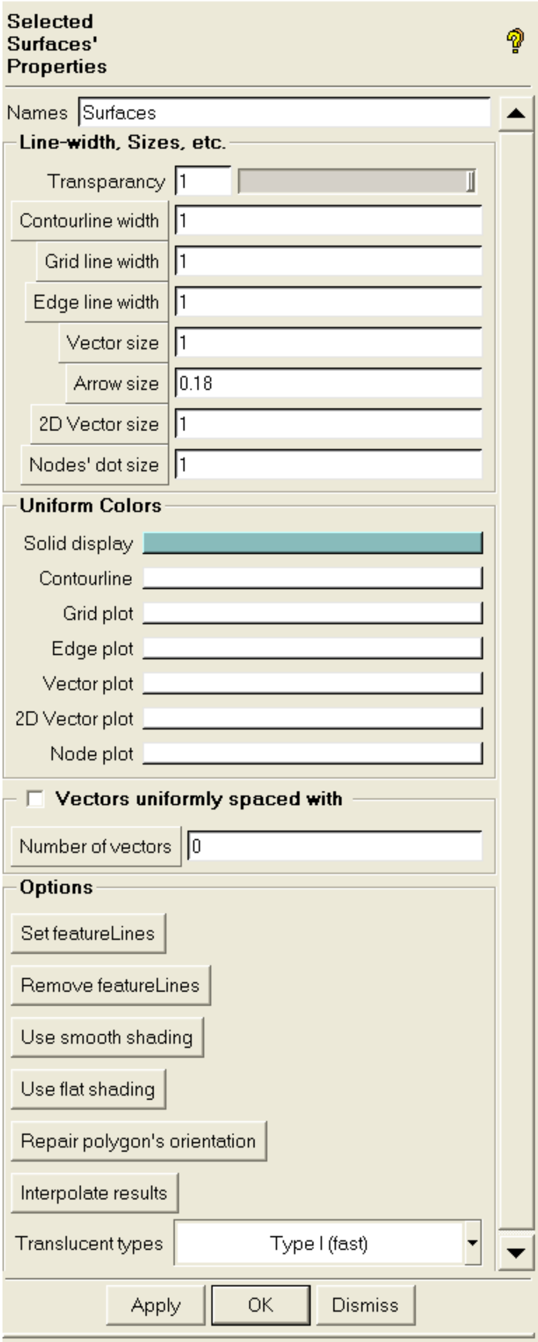
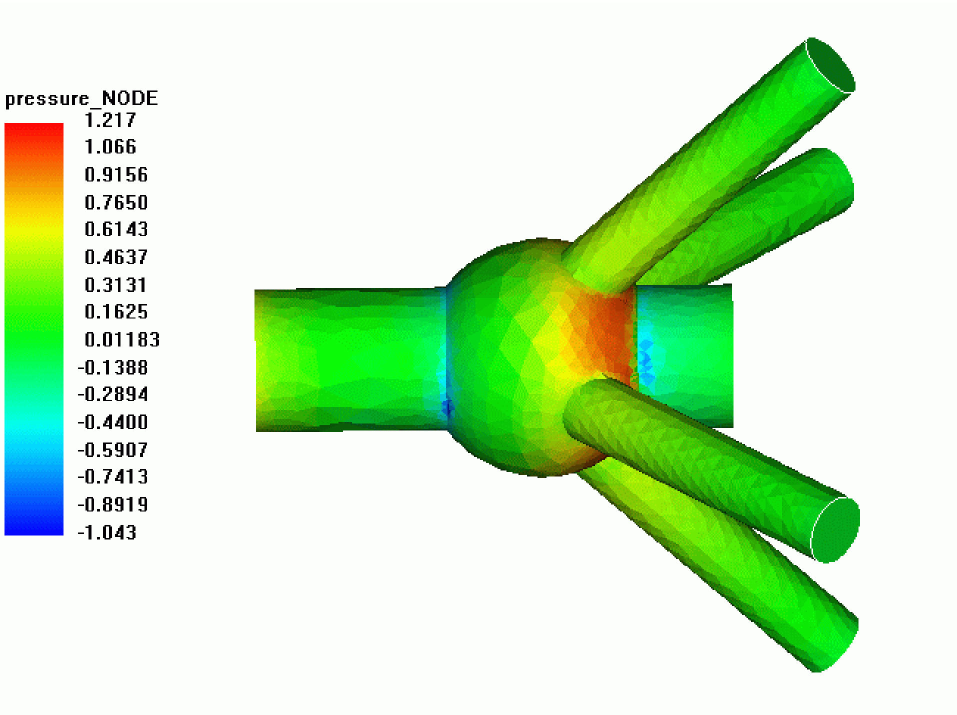


Figure 3.644
Solid contours with Flat shading



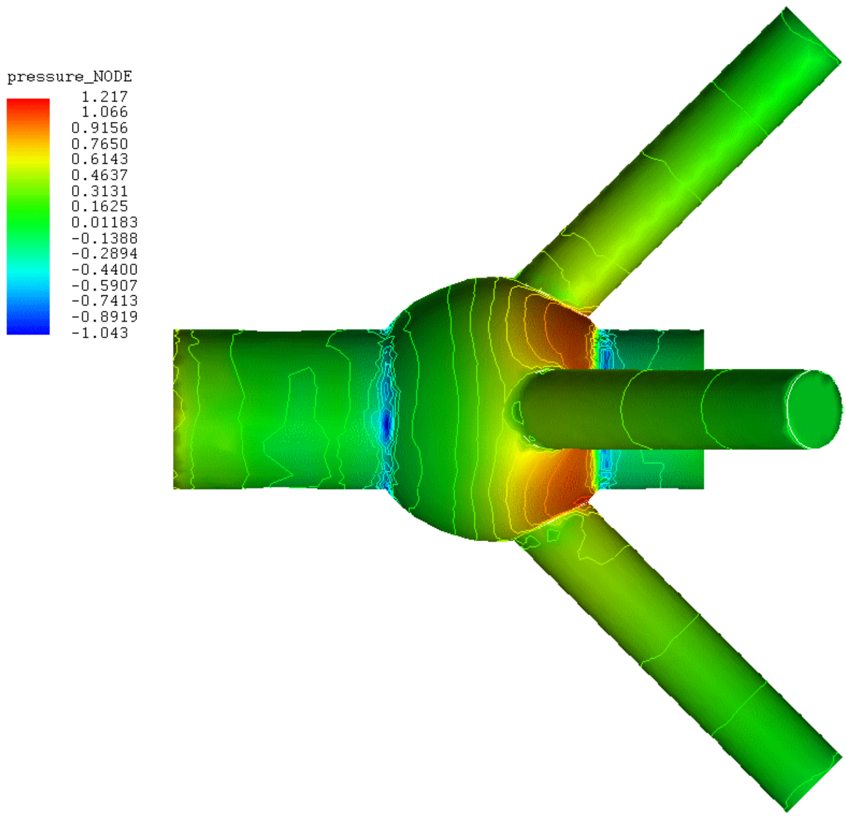
f) Displaying surfaces with contour lines

Contour lines are helpful in identifying the regions pertaining to the same values of the variable.

After checking the flat shading, click on the button Use smooth shading to display geometry in smooth shading. Click on OK to close the Selected Surfaces' Properties window.

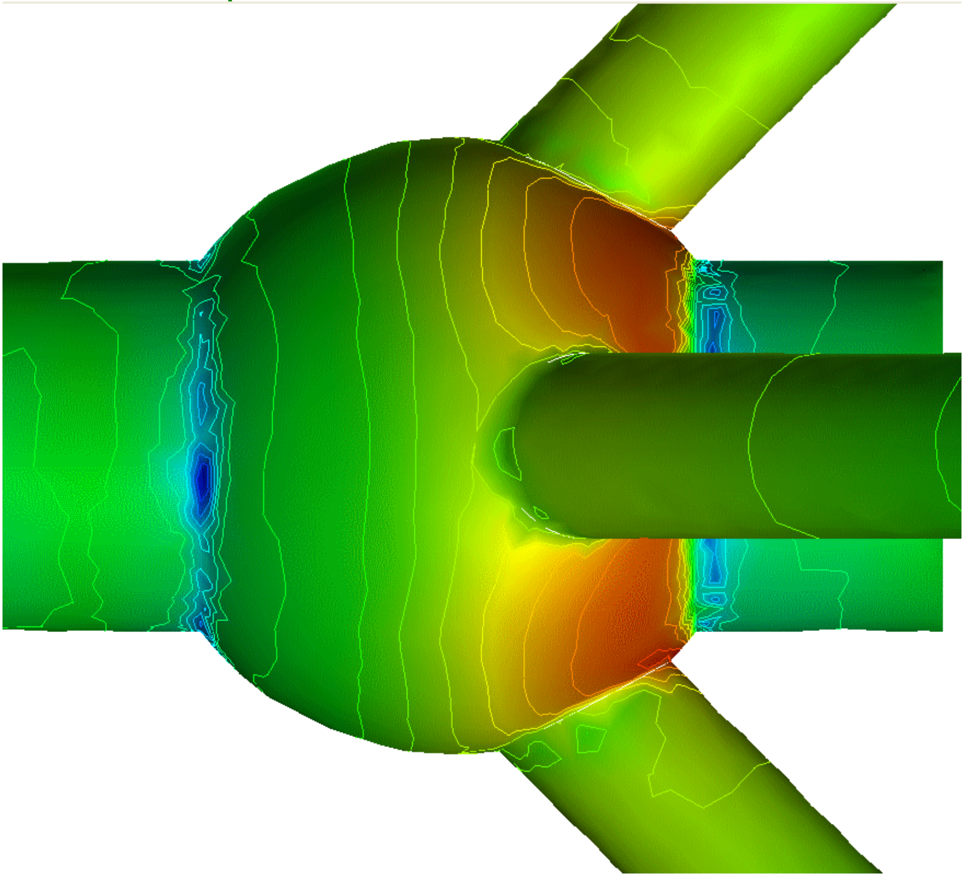
Select the option Contour line with variable colors for the Surfaces in the Display Tree widget. The contour bands on the surface will look like the Figure 3.645.

Figure 3.645
Surface display with Contours of pressure



The pressure values calculated inside Fluent are normally static pressure values with respect to the atmospheric pressure. Thus, user would see some negative values also (relative to the atmospheric pressure). Low pressure regions would be clearly visible in the region where flow expands into the spherical region. However, when the spherical region contracts, flow actually hits those boundaries and thus higher pressure is expected in those regions. The spherical region is zoomed and shown in the Figure 3.646. The user can use right mouse button to zoom into the region required.

Figure 3.646
Zoomed view of spherical location

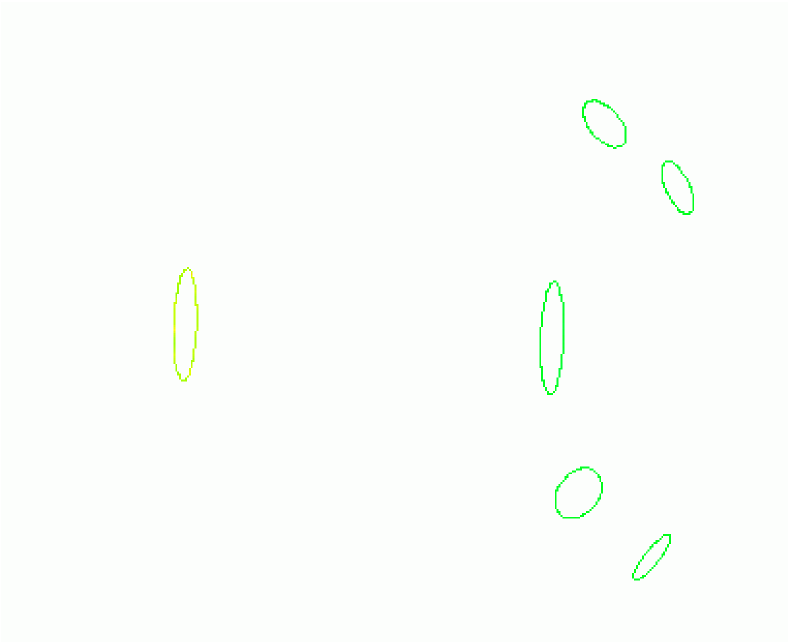


Toggle OFF the Contourlines with variable colors as well as Continuous Contour option.

g) Visualization of Surface edges and Vector plots

Select the display option Edge with variable color by right clicking on Surfaces. This will display the geometry boundaries shown in Figure 3.647. Boundaries are sorted out from the feature lines provided by the Fluent case and data files.

Figure 3.647 Surface displaying edges with variable color



Choose the option Vector with variable colors from the menu list. This will show the velocity vectors indicating the direction of flow as shown in Figure 3.648.

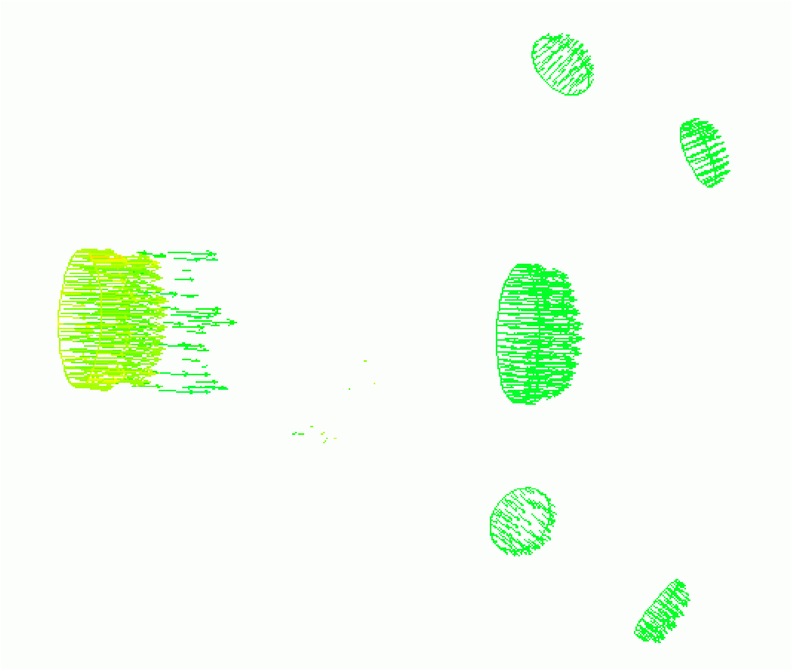


Figure 3.648 Surface displaying Vector with variable color

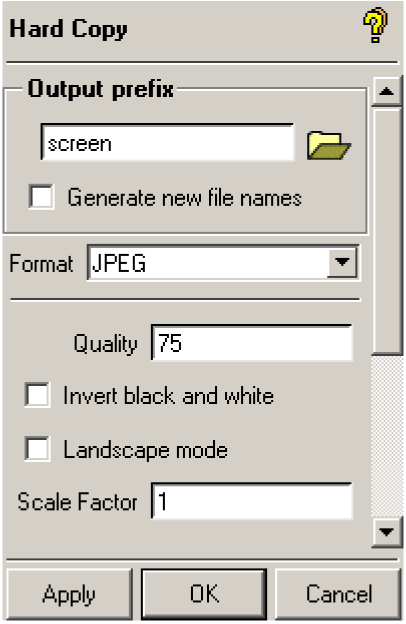
Note: The direction of the arrows corresponds to the currently selected vector variable and the colors of the vectors indicate the present scalar variable. The length of the vector arrows is relative to the magnitude of the vector variable.

Switch OFF the velocity vectors by selecting Vector with variable colors again.

h) Saving the Output

Go to View > Save Hardcopy option. This will bring out the Output window as shown in Figure 3.649.

Figure 3.649
Output Window



Select the Format as JPEG and click on Apply. This will save the image in the directory from where user has fired the ANSYS ICEMCFD.

User can view the image file in any image viewer software later.

3.9.2: Pipe Network (Advanced)

In this tutorial we will continue with the same Fluent example as picked up in last tutorial.

Operations introduced by this example:

Point Probing Technique

Dynamic Cut Plane

Movement of Cut Plane

Displaying Vectors in plane

a) Loading the Fluent file

Start Visual3p application.

From the Set Result Format window select the file format as Fluent.

Load the case file pp.cas and the data file pp.dat.

b) Point Probing Technique

The point probe returns the point's coordinates at the cursor position, the value of the active scalar, and vector functions. Here Point Probing Technique would be explained using the scalar variable pressure.

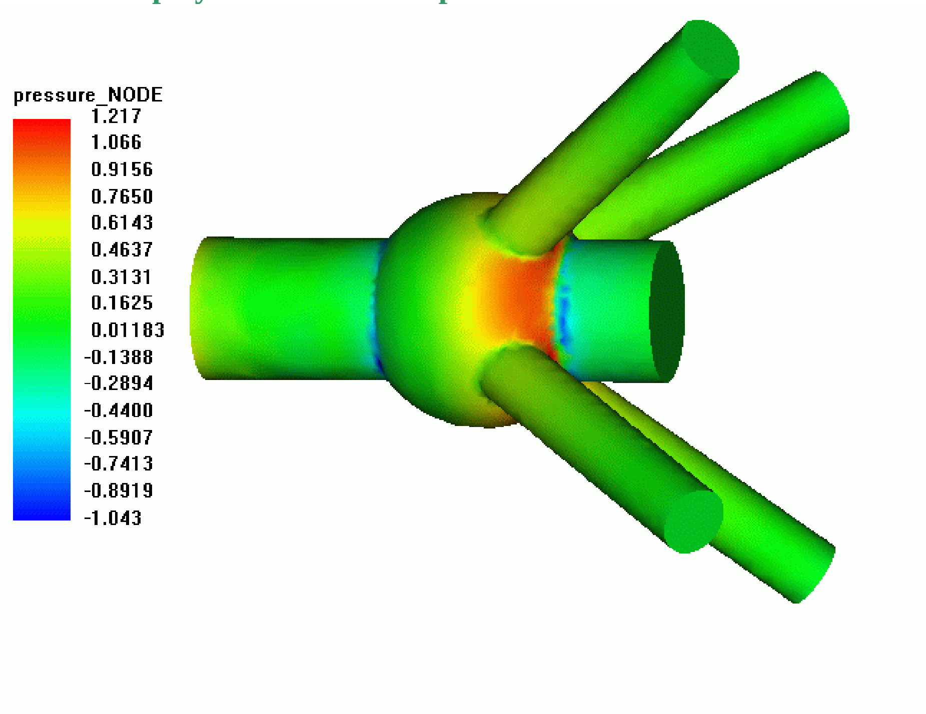
Select Variables from the Post-processing Tab menu. Choose the scalar variable pressure_NODE from the Scalar Variable list of Result Variables window.

From the Surfaces options, select the Continuous contour.

The surface display using pressure_NODE variable is presented in Figure 3.650.

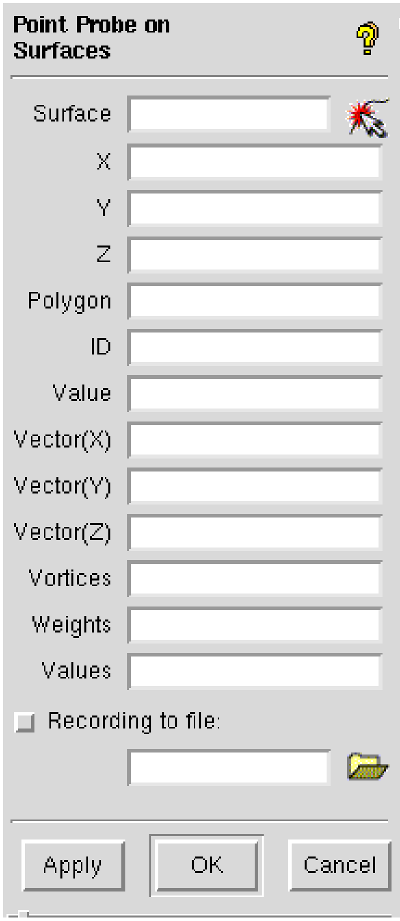
Figure 3.650

Surface display for the variable pressure



Go to Point Probe icon On the Post-processing tab menu. It will bring out the Point Probe on Surfaces window as given in Figure 3.651

Figure 3.651 Probe Window



Place the mouse pointer over any surface, which is to be examined and click with the left mouse button there. The coordinates of the cursor location will be displayed in the text windows displayed by the Probe tab as shown in Figure 3.652.

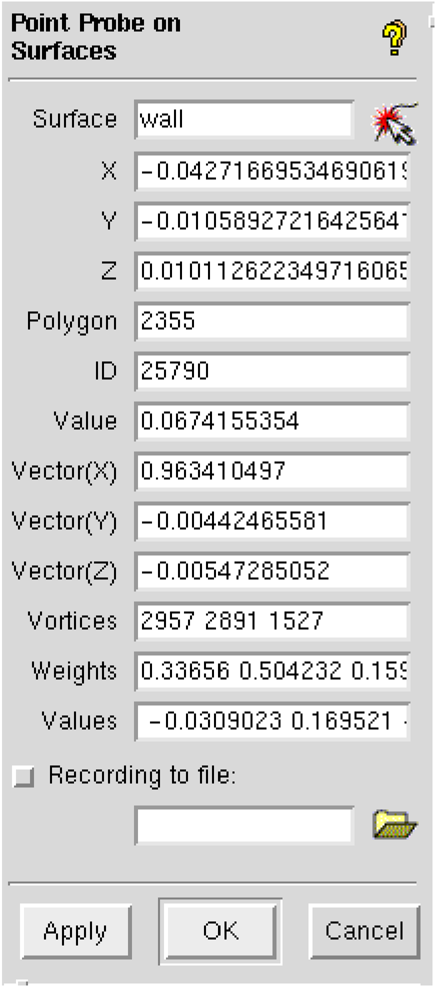


Figure 3.652 Probe window displaying the coordinates

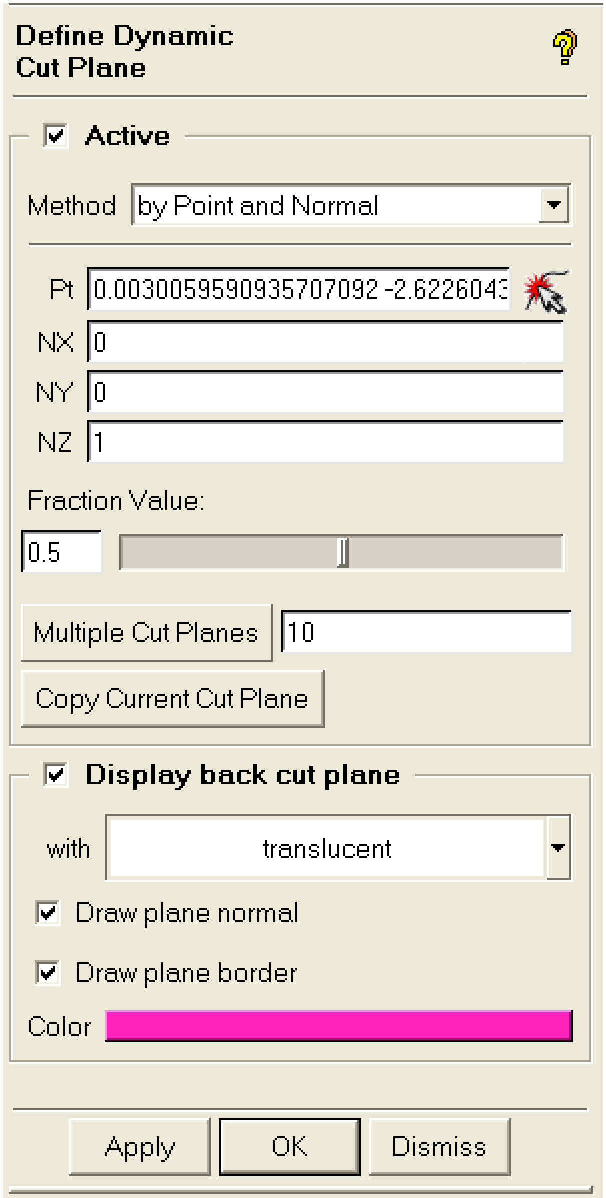
The parameter Value(s) specifically describes the value of the currently chosen scalar variable at the particular location of the surface under probe.

c) Dynamic Cut Plane

A cut plane is used to visualize results on a plane cut through the three dimensional model and the result obtained will be displayed in the Dynamic Surface window.

From Post-processing tab menu, select Define Cut Plane option. This will opens up Define Dynamic Cut Plane window. Select the method as by Point and Normal from the dropped down list. In Figure 3.653, Define Dynamice Cut Plane window is presented.

Figure 3.653
Define Dynamic Cut Plane window

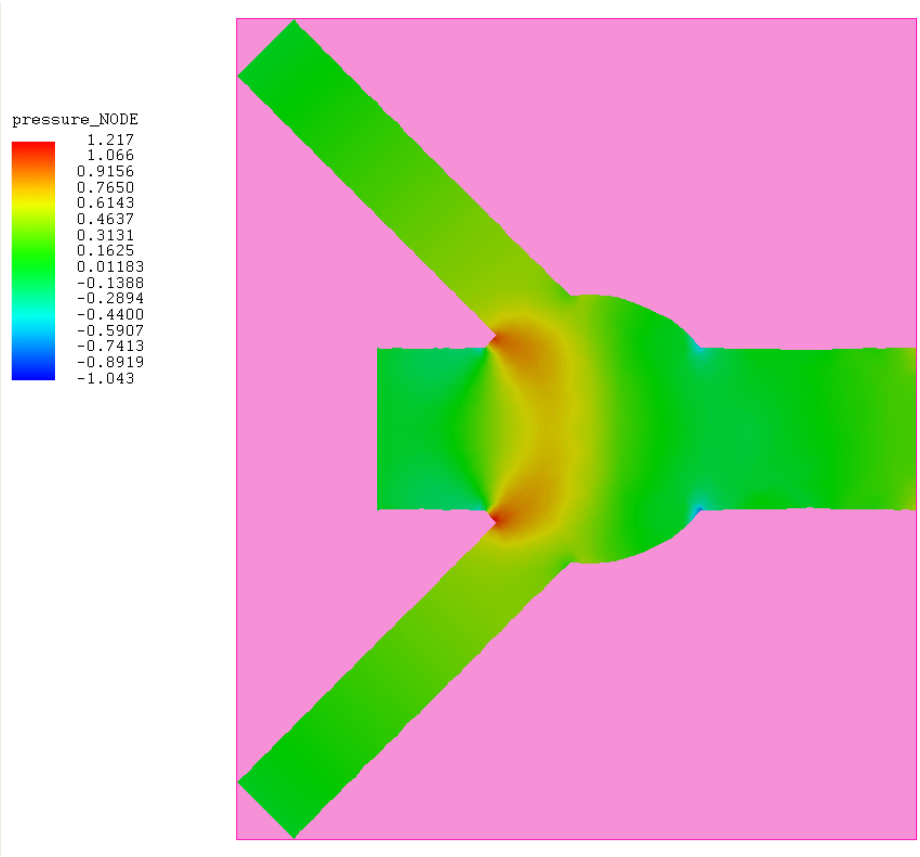


d) Contours on Dynamic Surface

Switch off the Continuous Contour option from the Surfaces, instead turn on Continuous Contour on the display options of Dynamic Surface (right click).

Continuous Contour for the Dynamic Surface will display the Cut Plane in the display window as shown in Figure 3.654.

Figure 3.654
Cut section of the geometry



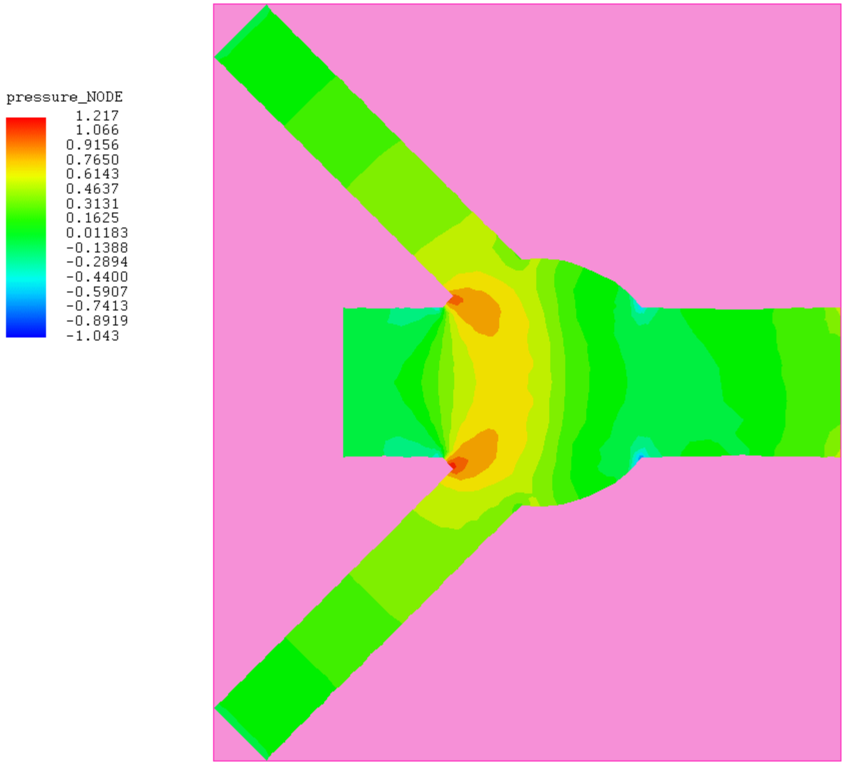
Before proceeding further, hide the background for the cut plane to visualize only the cut section of geometry. To do this, from Define Dynamic Cut Plane window check off Display Back Cut Plane.

In the cut plane window, ensure that the Fraction value for Cut Plane is set to a value of 0.5 and that the normal to the cut plane is in the Z-direction i.e., the variables NX, NY and NZ are set to values 0, 0, 1 respectively.

From the Dynamic Surface display options, select Contour Bands.

The Dynamic Surface will change as given in Figure 3.655.

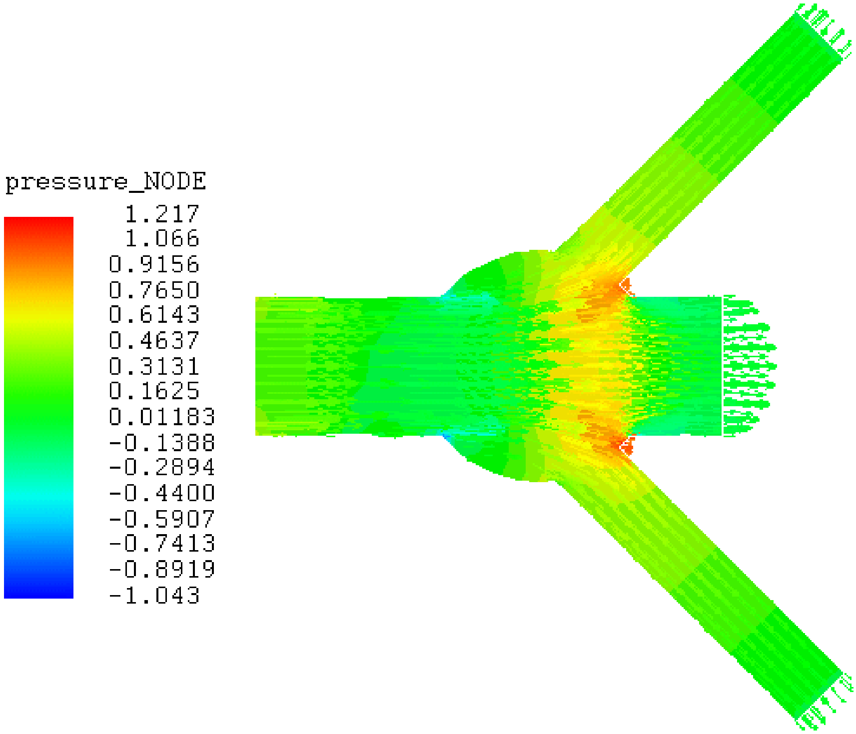
Figure 3.655
Cut plane surface displayed with Contour Bands

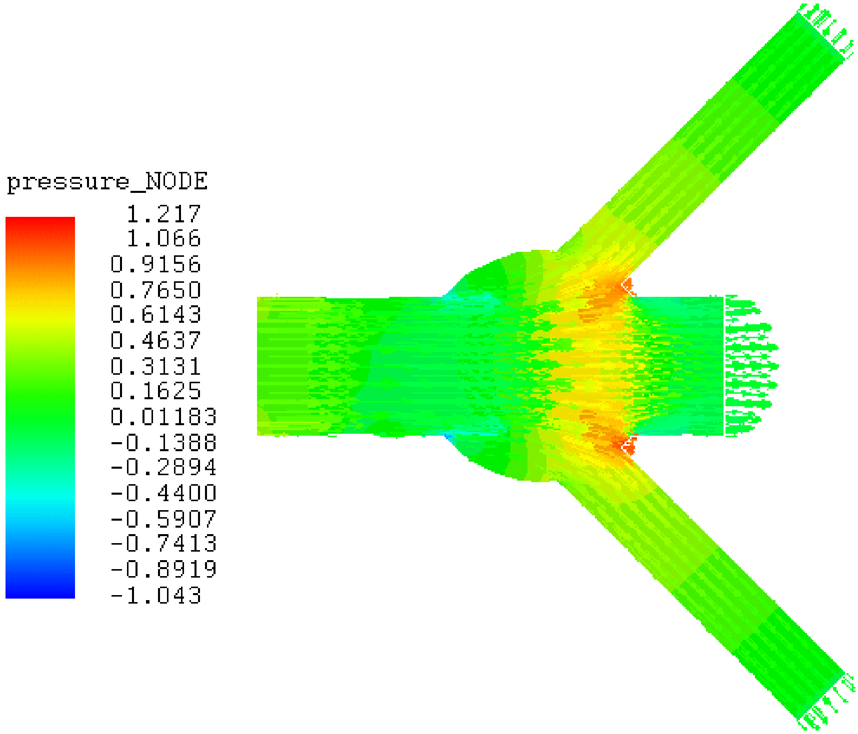


e) Velocity Vectors on Dynamic Surface

Choose the option Vector with variable colors from display control options. The display control options can be invoked by right clicking on Dynamic Surface. This will result in the display shown in Figure 3.656.

Figure 3.656
Dynamic Surface display with Vectors of Variable Colors


pressure_NODE
1.217
1.066
0.9156
0.7650
0.6143
0.4637
0.3131
0.1625
0.01183
-0.1388
-0.2894
-0.4400
-0.5907
-0.7413
-0.8919
-1.043



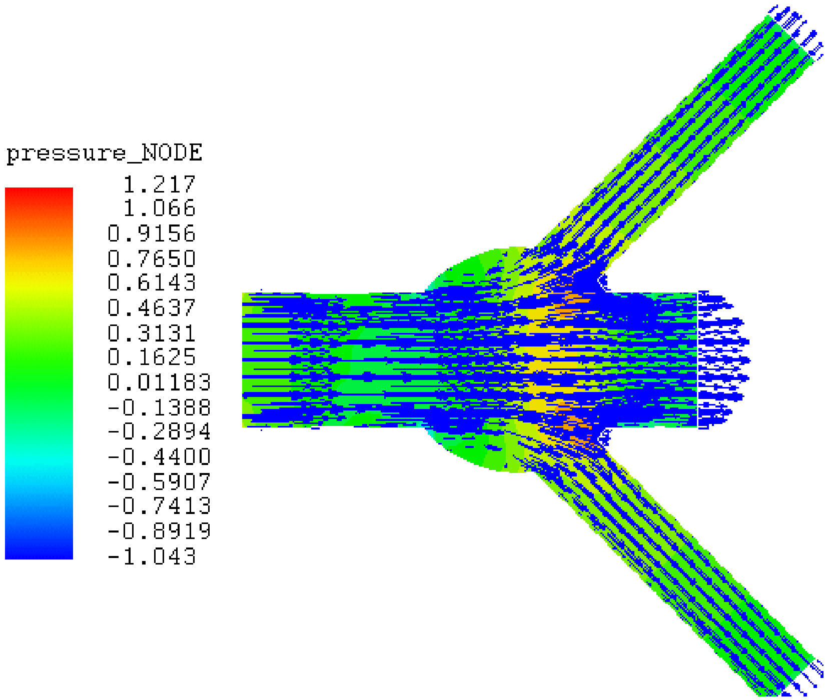
Here, contour bands and vectors display similar colors and hence it becomes difficult to distinguish them from each other. In order to observe the clear display, one can change the color of the vectors as follows. Shown in Figure 3.657.

On the display control options for Dynamic Surface, select 2D Vector with single color.

The uniform color can be set by first invoking Modify Drawing Properties on the display control properties mentioned above. On the ensuing panel, under Uniform Colors section, choose the 2D Vector Plot color.

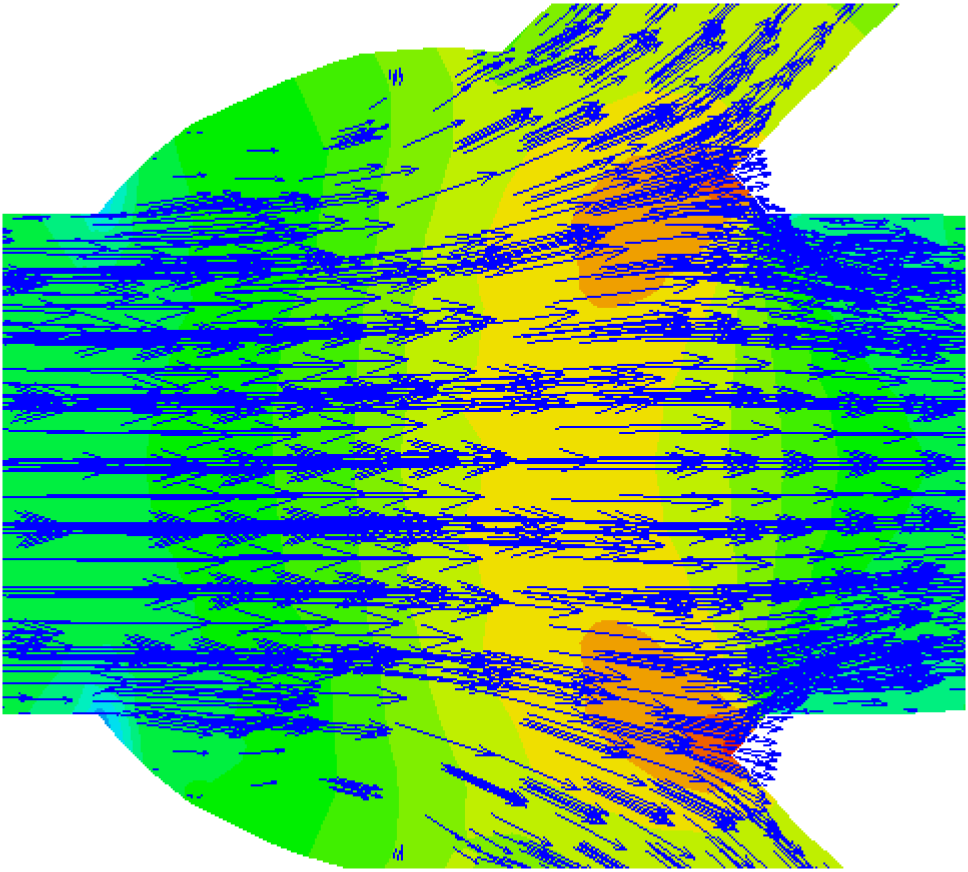
Note : One can control the Vector size and Arrow size by scaling the corresponding properties to desired value with the options Vector size and Arrow size from the Properties tab. A value of 0.4 for Vector size gives good results here.

Figure 3.657
Dynamic surface display with vectors of uniform color



The spherical region is zoomed and shown in the Figure 3.658. The user can use right mouse button to zoom into the region required.

Figure 3.658
Cut plane displaying the effects of pressure at different areas

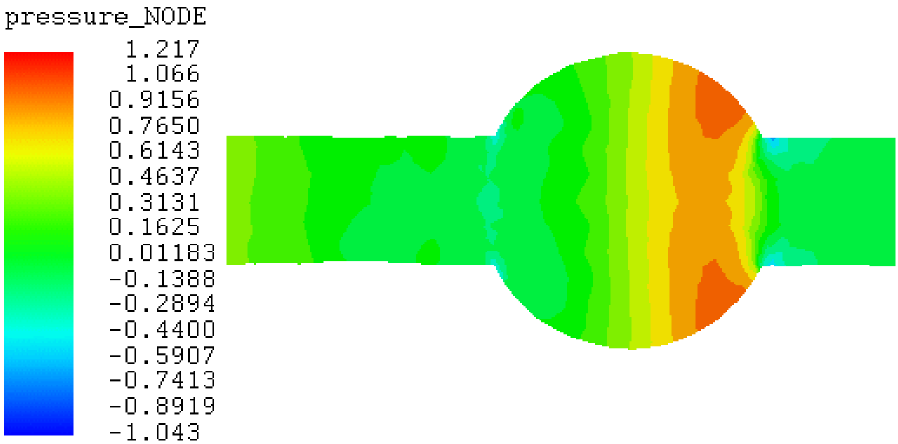


f) Movement of Cut plane

The user can do finer movements of cut plane by changing Fraction value manually. From Define Dynamic Surface panel, change the Fraction Value to let's say, to 0.572.

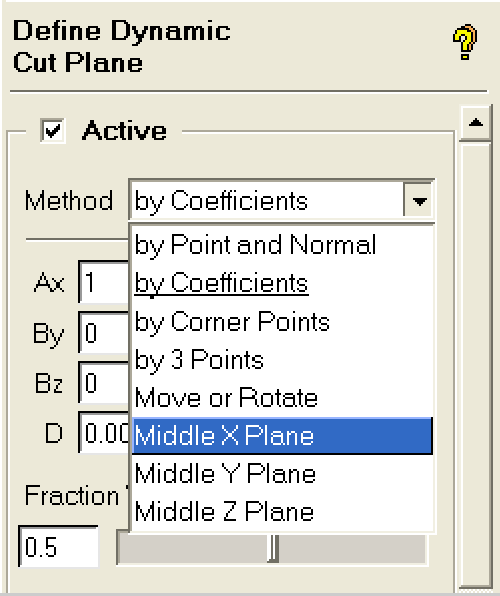
It's best to switch off the vectors and to switch On the cut plane from Tools to notice the cutplane movement. The output is as shown in Figure 3.659.

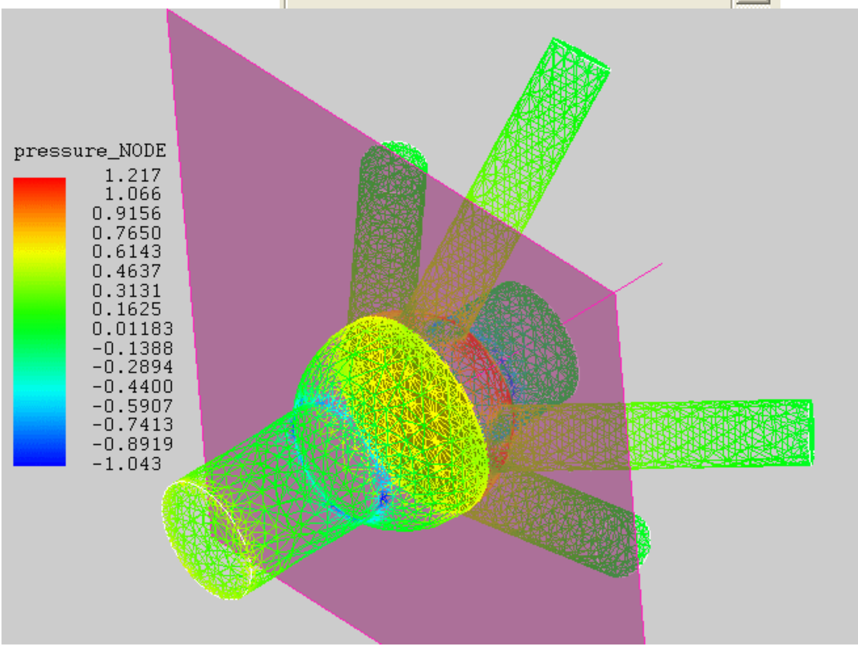
Figure 3.659
Cut Plane at a fraction value of 0.572



To switch the cut plane in the X direction, on Define Cut Plane panel, from Methods menu, select Middle X Plane as shown in Figure 3.660. To see the relative position of this cut plane, display the grid with variable color and show the back cut plane.

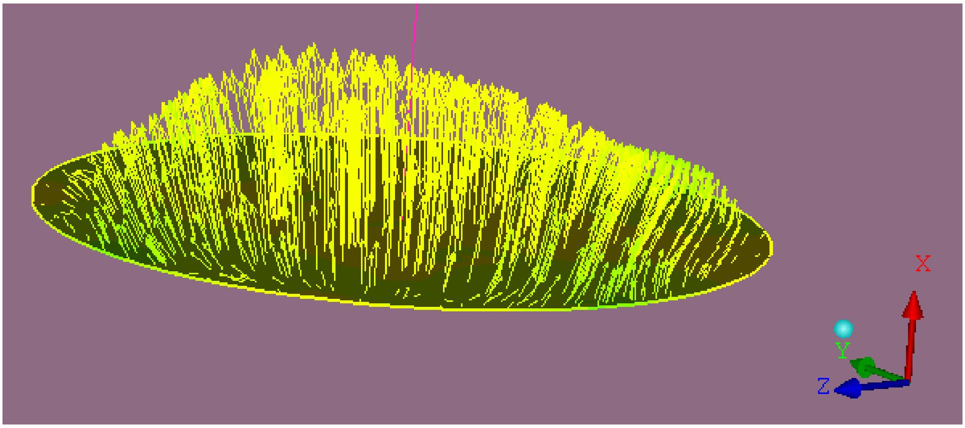
Figure 3.660 Cut Plane normal to X direction





On Surfaces display options, toggle off Grid with variables, and on Dynamic Surface display option, select Vector with variable colors. Figure 3.661 indicates that the vectors largely point in the X direction.

Figure 3.661
3D Vectors on Cut Plane



3.9.3: Space Shuttle

In this tutorial, one would be referring to a CFD++ result as an example. For the post-processing of this CFD++ simulation, the user will also provide ICEM CFD domain file.

Operations introduced by this example

Creating Streamlines along Surfaces

Creating Streamlines along 2D Containers

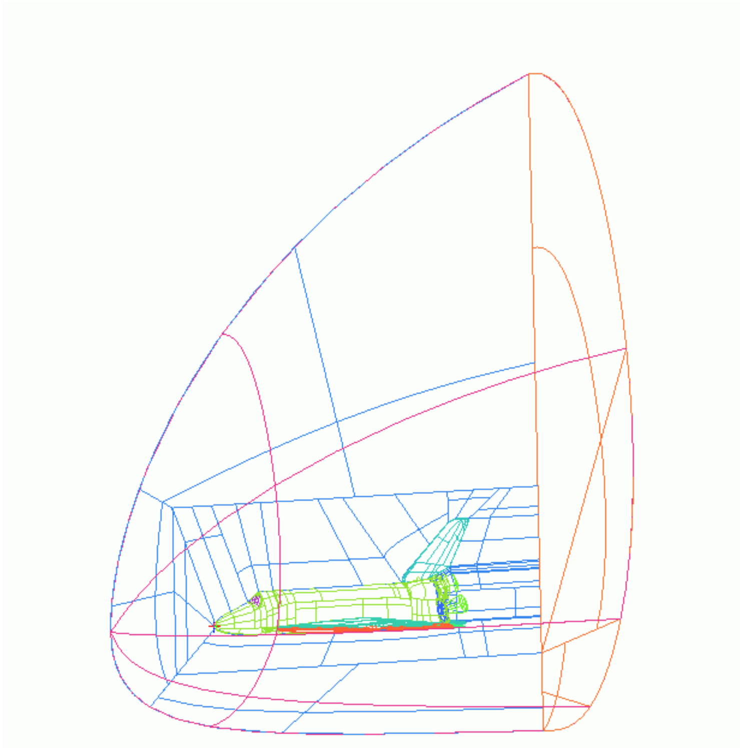
Animating Streamlines

Creating Movies

a) Case Description

The model consists of a space shuttle with the air flowing at a relative angle of attack of 10 degrees. As shown in Figure 3.662, Only half of the model is simulated because of the symmetry conditions prevailing.

Figure 3.662 Geometry



Following are the boundary conditions applied:

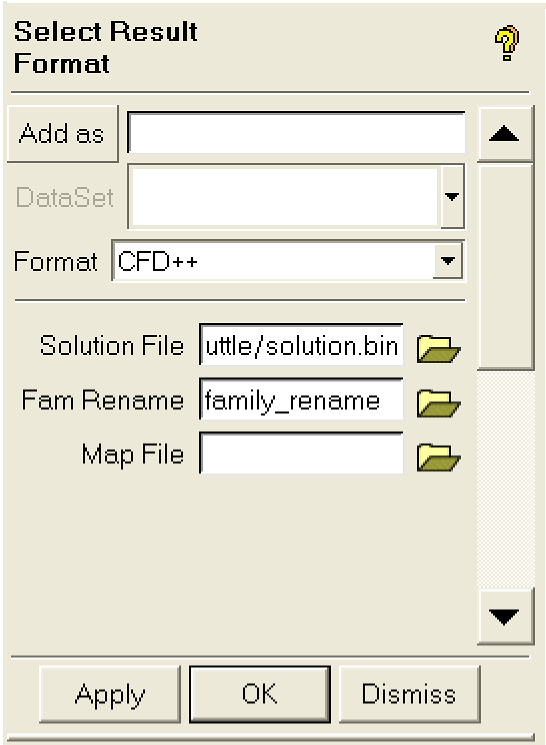
OUTER	Pressure far field,
SYM	Free slip
OUT	Outflow i.e. gradients=0

b) Loading CFD++ results

Start Visual3 application.

From the Set Result Format window select the file format as CFD++. It will pop-up the CFD++ file selection menu, which is shown in Figure 3.663.

Figure 3.663 CFD++ file selection window



For the CFD++ results, select the Project directory as Space_Shuttle. Also select solution.bin and family_rename as the Solution file and Family rename file respectively. Press Accept after verifying all these files to start analysis of the solution file.

c) Creating Streamlines along the Surfaces

In the Surfaces display options, Continuous contour and Edge with single color will be ON by default.

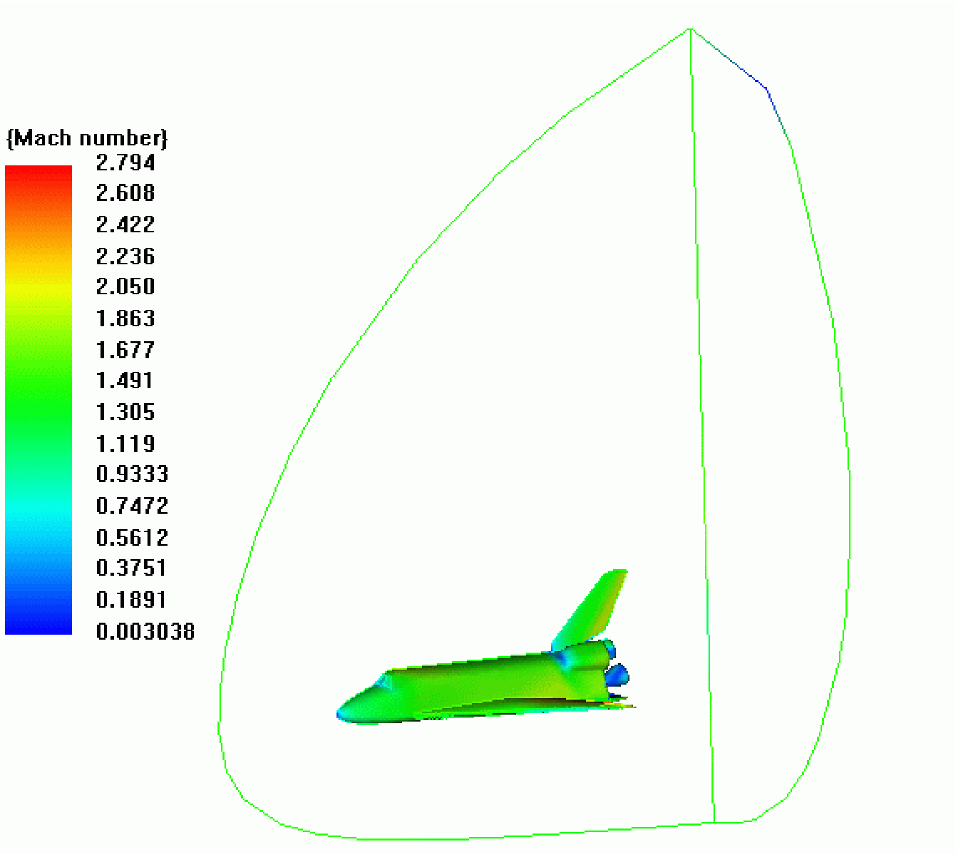
Switch OFF the Continuous contours for the surfaces beginning with SYM, OUTER, OUT and ORFN.

From the Post-processing Tab menu bar, select Variables icon.

Select Mach number from the Scalar Variable dropped down menu of the Result Variables window.

The display would be as shown in Figure 3.664.

Figure 3.664
Surfaces SYM, OUTER and OUT with no solid contours

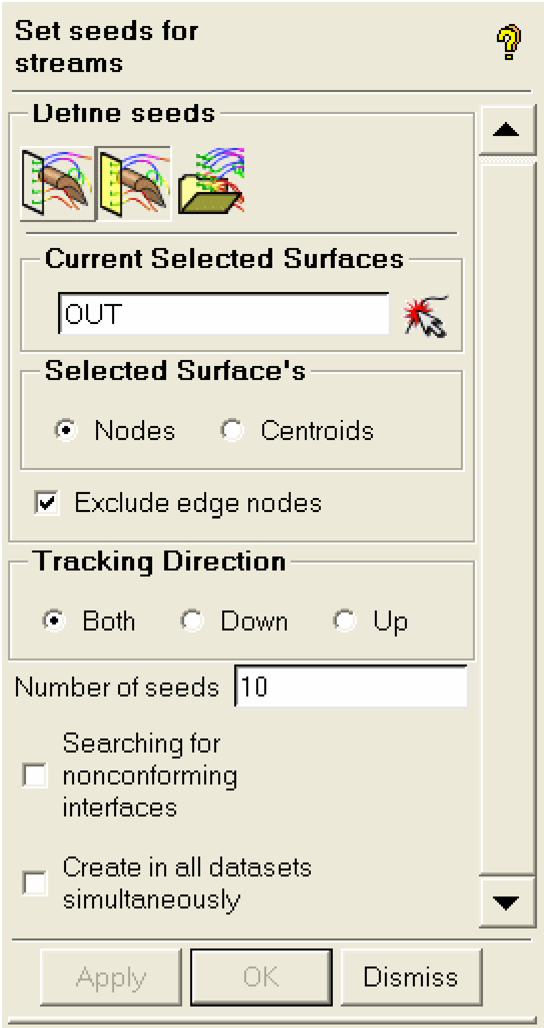


Select the Streams icon from the post processing tab menu. This would bring out the Set Seeds window, as shown in Figure 3.665.

In the Current Selected Surfaces box, the name of the currently highlighted surface (in the model tree) will appear.

This surface will be the source of new streamline seeds. If this is not desired name, click on the picker and select from the main display. You can also type the name you want in the box. Select OUT surface for this tutorial.

Figure 3.665 Streams Manager

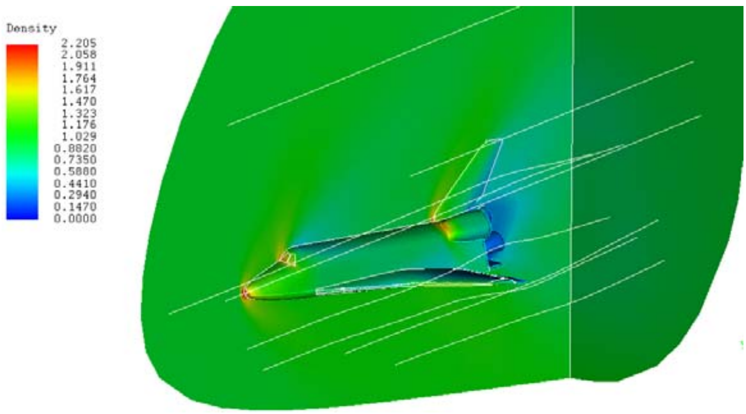


Make sure that the Nodes radio button is selected in order to ensure the seeds will be from node points.

Ensure that the Exclude edge nodes are checked to exclude edge nodes from the choice of seeds. Leave 10 as the desired value for the seeds into the Number of seeds text box.

After feeding this information, click on Apply to create the streamlines in the model, Figure 3.666.

Figure 3.666 Streams based on OUT surface



This will update the Streams branch of the Display Tree. Expand the Streams option, user will find the entry of Group 1. The different options for the stream lines are available on right mouse click on this Group 1 entry of Display tree.

Select Multiple colors to change the colors of the streams.

From the Group 1 options, select Remove permanently option. The use will be prompted to confirm about the delete Group 1 entry of Streams. After pressing Confirm, it will delete the streams.

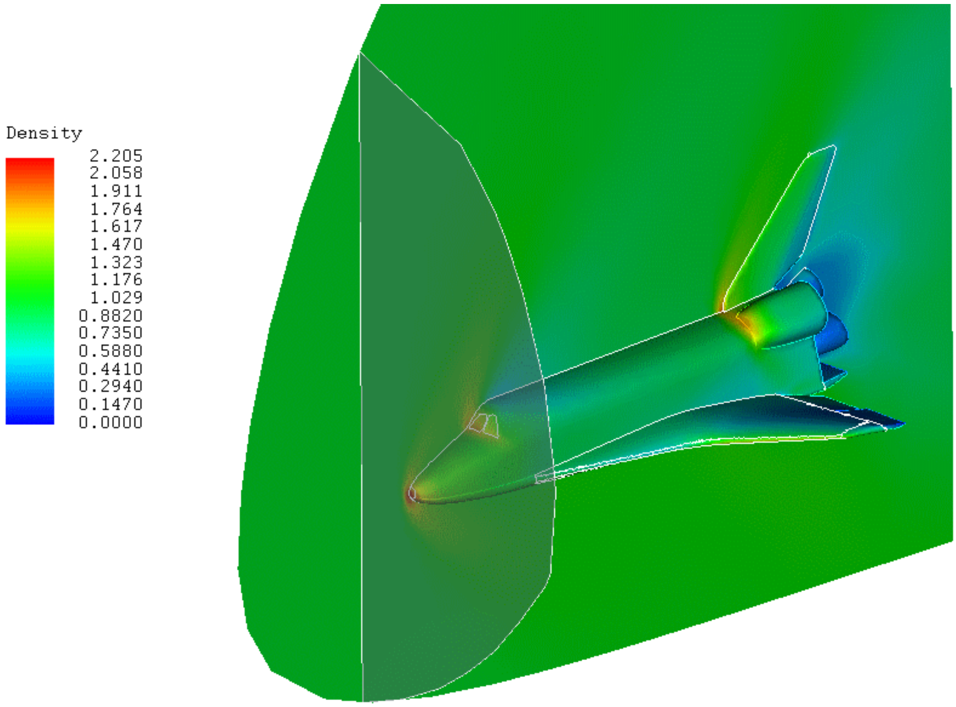
d) Creating Streamlines along 2D container:

Normally, a user would either define the seed through a surface or within a 2D container plotted on the 3D window. It's difficult to seed it through the 3D container.

Select Define Cut Plane option from the Post-processing Tab menu bar.

In **Define Dynamic Cut Plane** window, from Methods, select Middle X Plane and adjust the Fraction Value to **0.2**, so as to obtain the cut plane shown in Figure 3.667.

Figure 3.667
Cut Plane

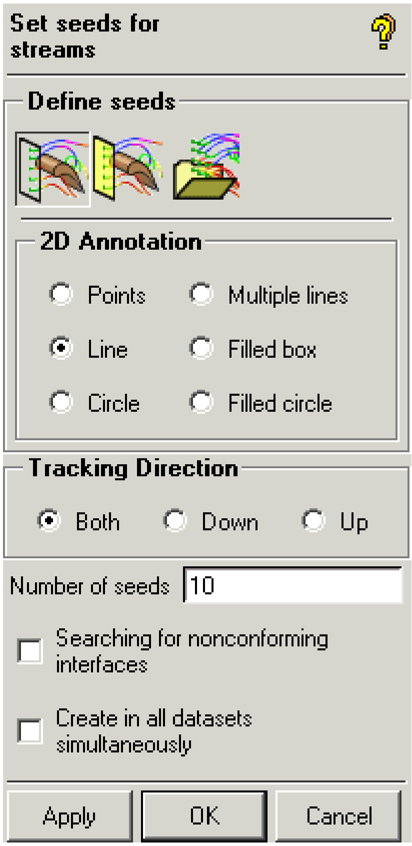


Now, this cut plane will be consider as the Dynamic Surface, under the Surfaces option of Display Tree.

Ensure that the Dynamic Surface is selected from the Display Tree and then go to Streams option of Post-processing Tab menu bar.

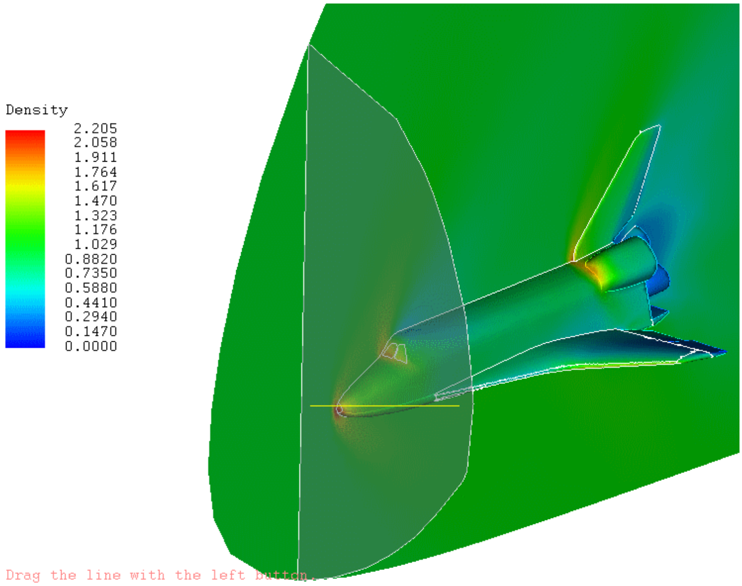
As shown in Figure 3.668, select the option Set seeds with 2D Annotation from Define Seeds panel for streams window. Choose the radio button Line to define the type of seeds.

Figure 3.668 Streams from 2D Annotation window



Press Apply. This would display a message on the main graphics window prompting the user to define the starting and ending points of the line. Click the left mouse button and drag it to define the length of the line as shown in Figure 3.669. The horizontal line is drawn from left side beginning till the back side outlet going through shuttle.

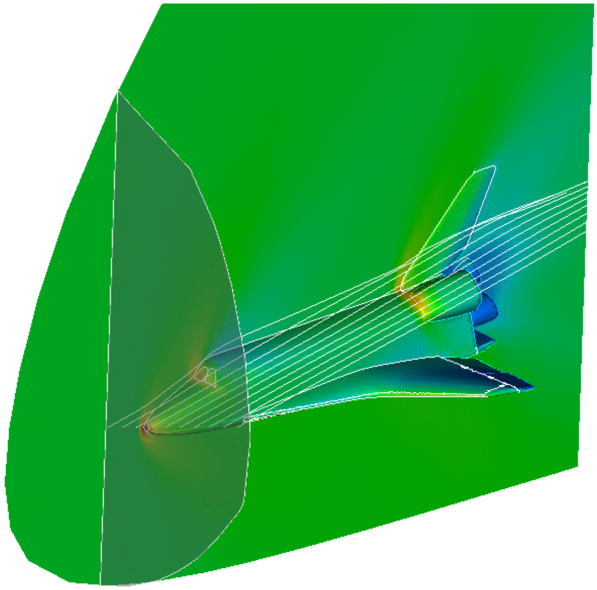
Figure 3.669 Dynamic Surface with the line defined



The Streamlines would appear in the graphics window as shown in Figure 3.670.

This will update the Streams branch of the Display Tree. If desired, from Group 2 in the model tree, switch to various display options.

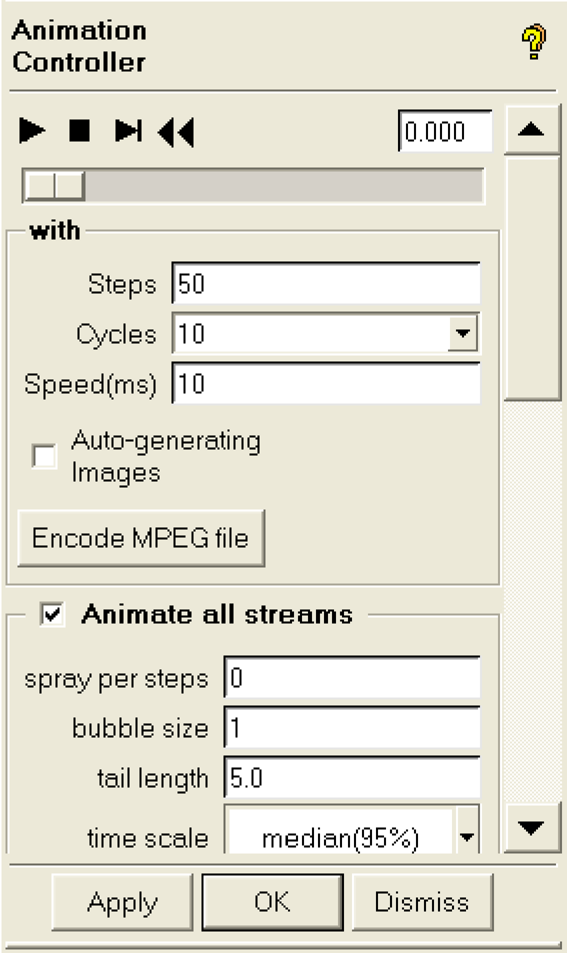
Figure 3.670 Streamlines based 2D option



e) Animating the Streamlines

Select the Animatation Controller icon from the post processing menu. This will bring out the following selection windows as shown in the figure below. Figure 3.671

Figure 3.671 All in one Animation Controller window



Select the option Animate All Streams on the controller panel.

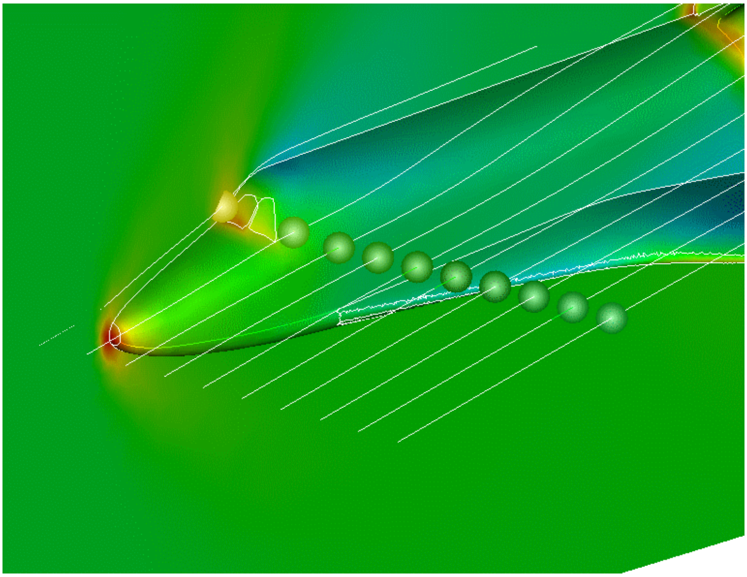
Set bubble size to 0.5.

Press Apply

On the top of Animation Controller, click on play button. Now bubbles begin moving along the streams, Figure 3.672.

The smoothness of the animation can be controlled by Steps parameter, and the speed per step can be adjusted by Speed parameter.

Figure 3.672 Bubbles on 2D



3.9.4: Space Shuttle (Advanced)

For this tutorial, the same example as in the previous tutorial is considered.

Operations introduced by this example

Set options

Displaying surface flow for the selected surface

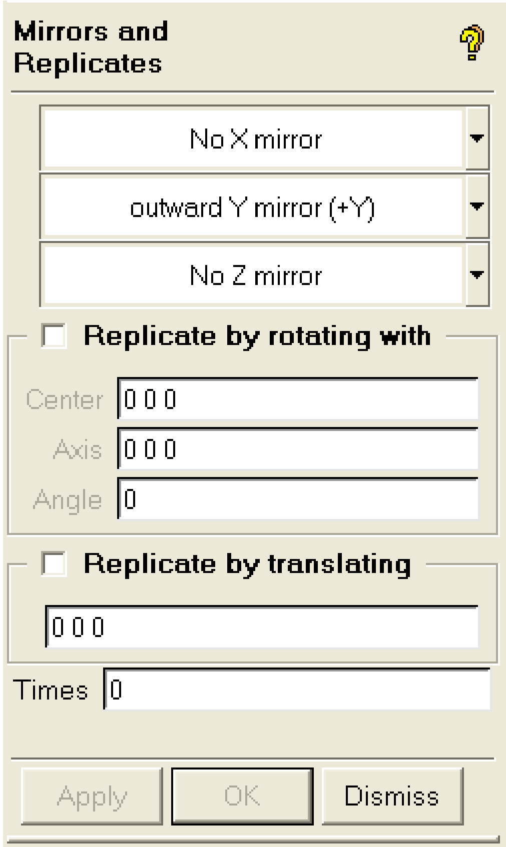
Movies

a) Loading the CFD++ file

Start Visual3 application and read the CFD++ results as in the previous tutorial.

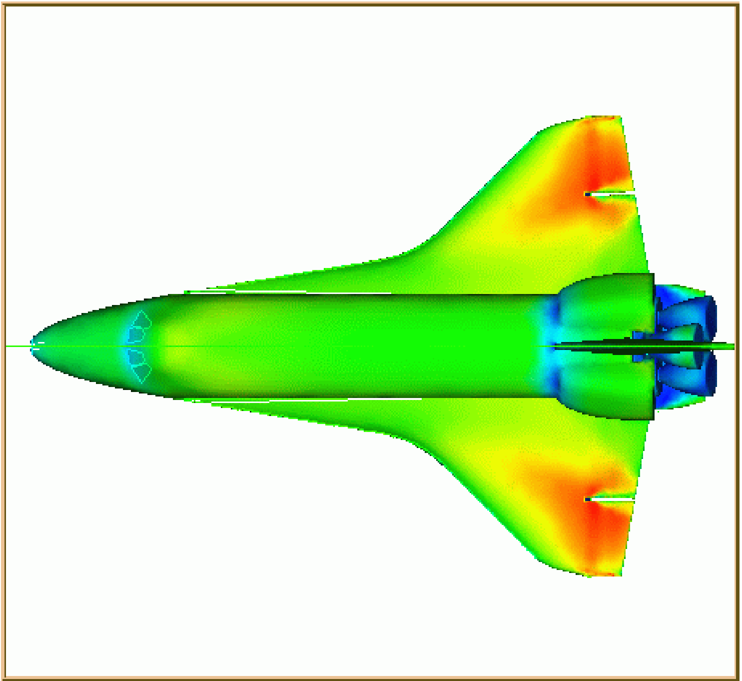
From View, select mirrors and replicates. This will bring out the Mirrors and Replicates window as shown in Figure 3.673.

Figure 3.673 Mirrors and Replicates window



Select the option outward Y mirror (+ Y) from the No Y mirror pull down menu list. Now the mirror image of the previous image on the Post 3D window can be seen as shown in Figure 3.674.

Figure 3.674
Mirror Image



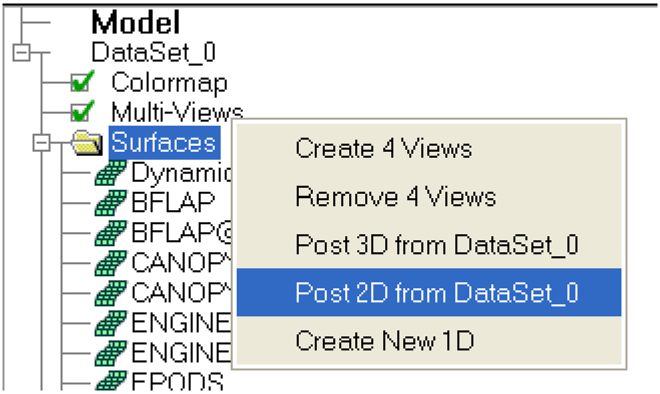
b) Post 2D Views

First invoke Define Dynamic Cut Plane from the post processing menu. From Methods, select middle Y plane. Set Fraction Value to 0.850 so the cut plane crosses the shuttle wing.

From model tree, right click on Muti-Views branch; Figure 3.675.

From the options, select Post 2D from Data_0. Upon prompt, drag a box on the graphics window.

Figure 3.675
Multi-Views options



From the options, select Post 2D from Data_0. Upon prompt, drag a box on the graphics window.

Now, the result on the cut plane will appear in the new 2D window. The main graphics window should appear as in Figure 3.676.

The 2D graphics window's name will appear in the model tree under Multi-Views. This will also has its own right click options which are shown in Figure 3.677.

Figure 3.676 Multi-Views > Post 2D window

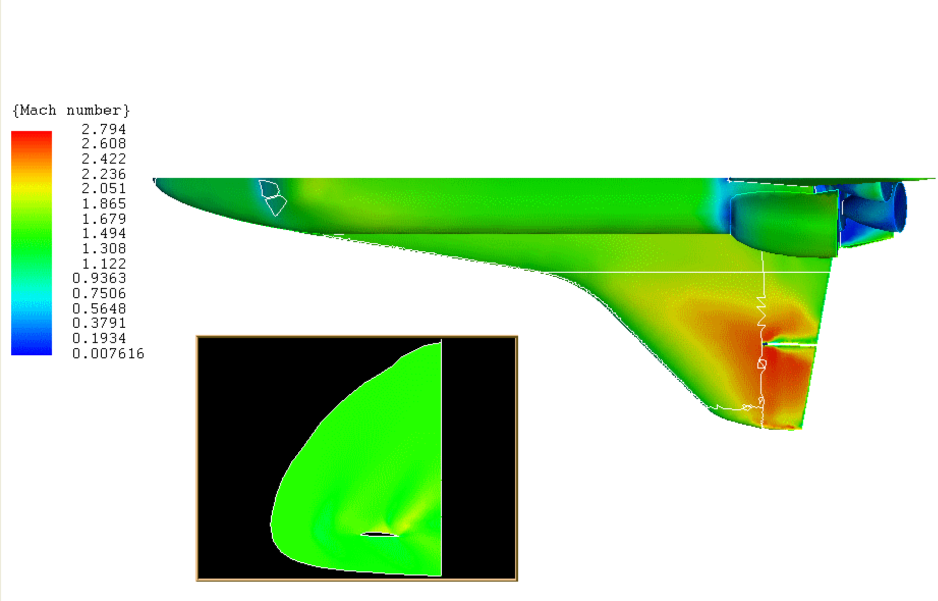
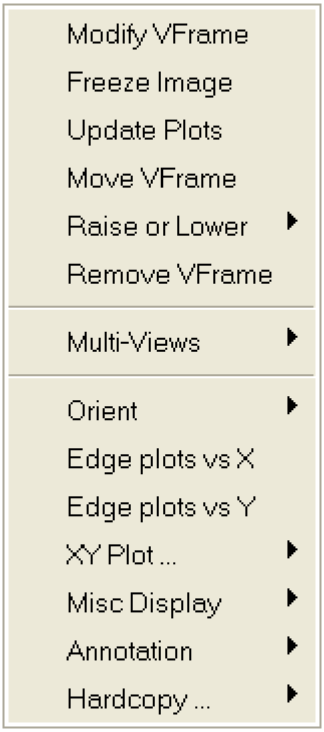


Figure 3.677 Post 2D window display options.



Select the option Remove Vframe to clear the newly created window. The user will be given a confirmation message window. Click OK to confirm..

Select the option Move Vframe to reposition the newly created window. The user will be given an instruction on the main window. Click anywhere on the main gui and move.

c) Displaying surface flow for the selected surface

The surface flow for the surface can be calculated and displayed in the following ways.

d) Display lines for the selected surface

From the model tree, select the surface FUSEL, by issuing a right click on the name.

From the display options, choose Compute Surface Flow which invokes the panel in Figure 3.678.

Click in the Display Lines check box.

Press Apply. This would render the selected surface with colored flow lines as shown in Figure 3.679.

Figure 3.678
Compute Surface Flow

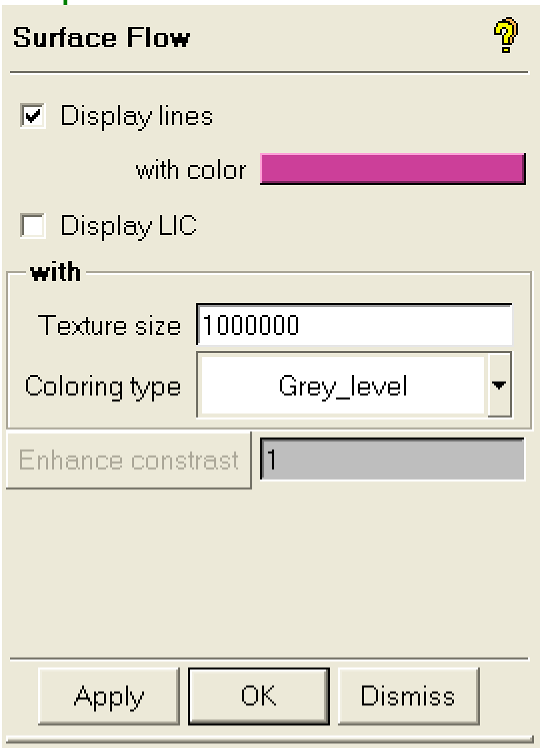
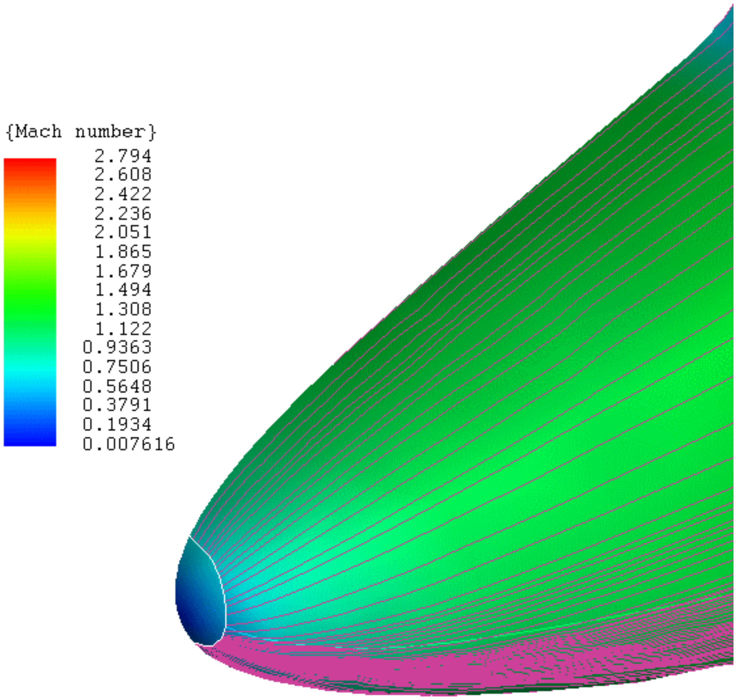


Figure 3.679 Compute Surface Flow Display Lines



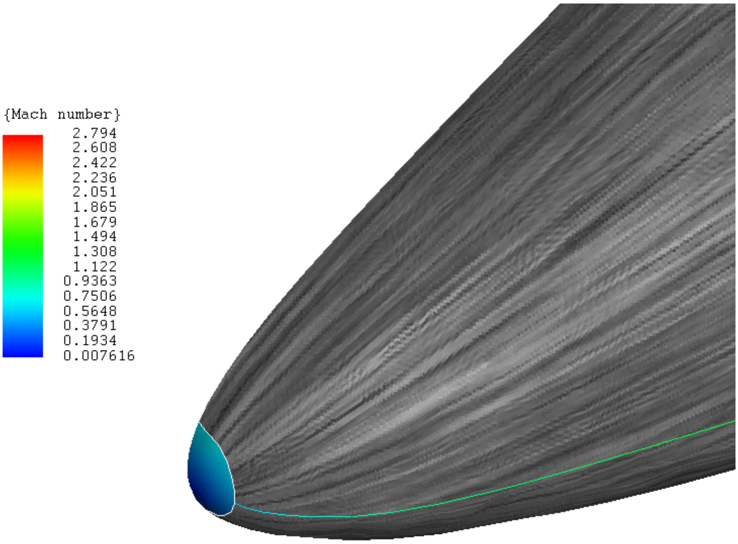
e) Line Intergral Convolution (LIC) of the selected surface

From the Compute Surface Flow panel (Figure 3.678), check the Display LIC check box.

Retain the default resolution available in the use texture size text box.

Choose with grey color to display the LIC in contrasting light and grey bands. The line integral convolution will be displayed as shown in Figure 3.680.

Figure 3.680 Compute Surface Flow Display LIC



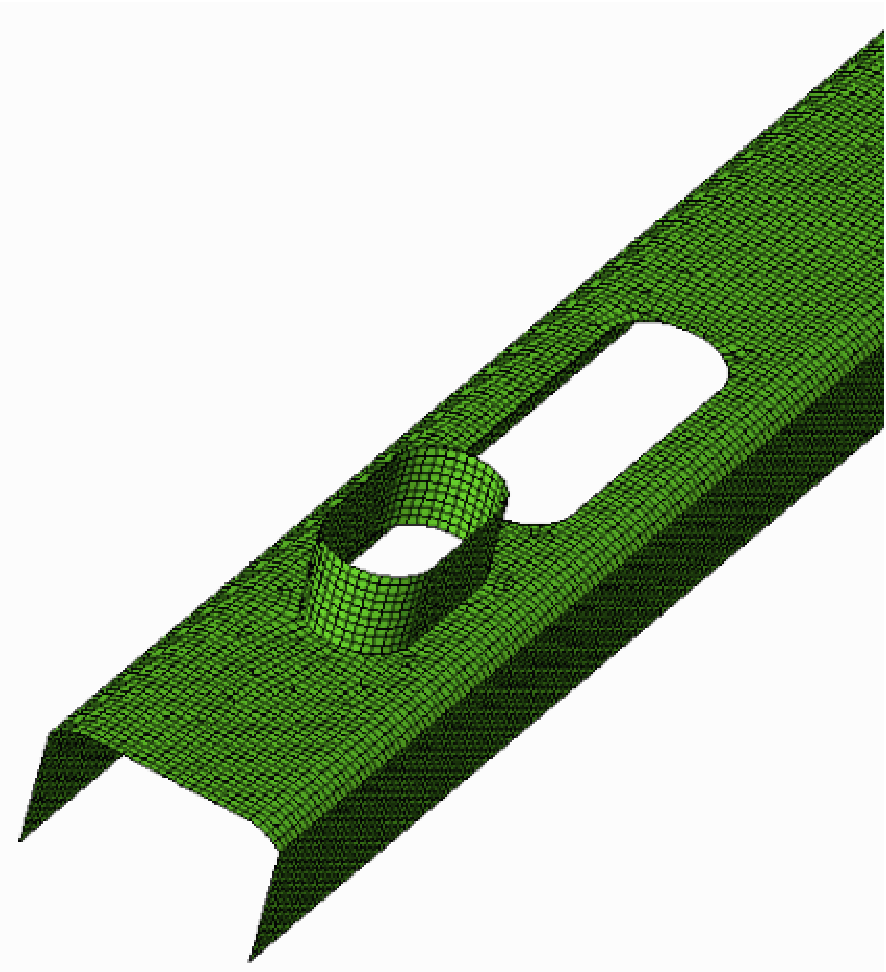
4: FEA Tutorials

4.1: Ansys Tutorial

4.1.1: T-Pipe: Modal Analysis

The main objective of this tutorial is to demonstrate legacy conversion from a Nastran model to an Ansys model. It also highlights the ease of use with **AI*Environment** in translating a model from one solver to another with little more than a flip of a switch. A Nastran modal analysis data file is provided as input. Once imported into **AI*Environment** and the solver is changed to Ansys, the shell element materials, which are defined for Nastran, are converted to the corresponding Ansys materials. The imported mesh is shown in Figure 4.1.

Figure 4.1 T-Pipe model



a) Summary of Steps

Data Editing

Launch AI*Environment and import an existing Nastran data file

Verification of imported data

Save project

Solver setup

Setup Ansys Run

Setting Solver Parameters

Write Ansys Input File

Solution and Results

Solving the problem

Visualization of Results

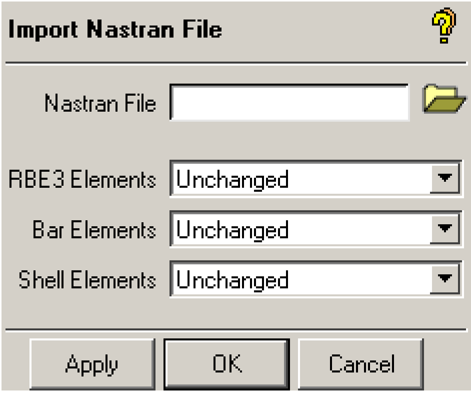
b) Data Editing

This tutorial continues on from the Nastran **Tpipe.dat** file created in the T-Pipe Tutorial. For those who have not done the Tpipe tutorial, the required Nastran file, **Tpipe.dat**, is provided in the **AI_Tutorial_Files** directory.

Launch AI*Environment.

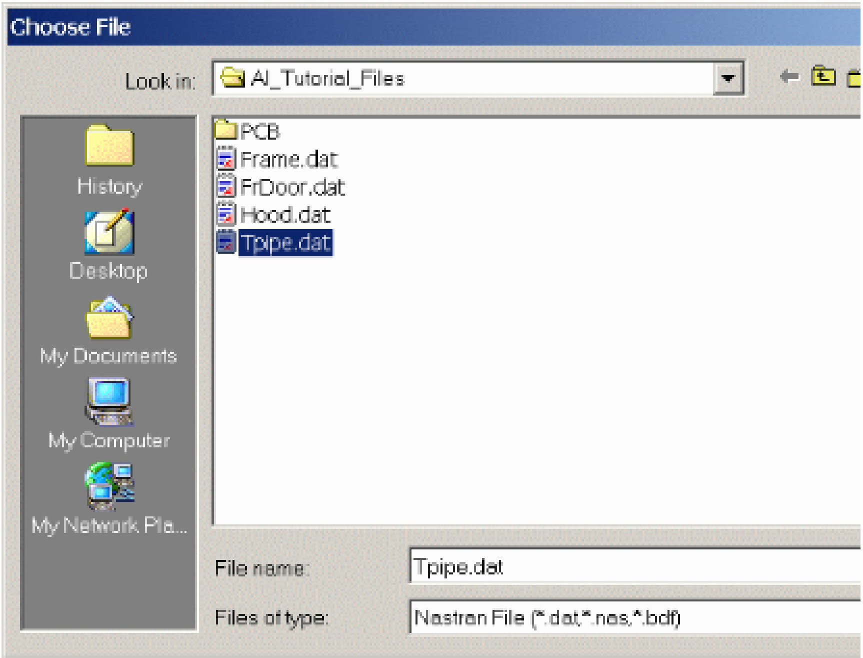
Select **File** > **Import Mesh** > **From Nastran** from the main menu, which will open the **Import Nastran File** window shown in Figure 4.2.

Figure 4.2 Import Nastran File window



Click on the open file icon for the file-browsing window. Select the file **Tpipe.dat** as shown in Figure 4.3 from the **AI_Tutorial_Files** directory.

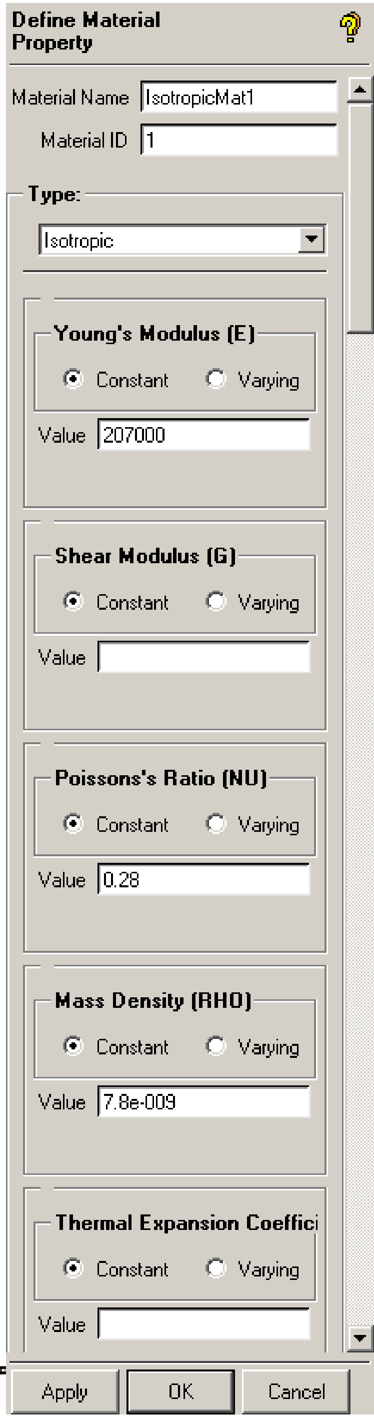
Figure 4.3 Selecting Nastran data file



c) Verification of imported data

Expand the **Material Properties** in the Model Tree by clicking on the +. To open the **Define Material Property** window for **IsotropicMat1** as shown in Figure 4.4, double click on the material, **IsotropicMat1**, with the left mouse button, or right click and select **Modify**.

Figure 4.4
Define Material Property window



d) Save Project

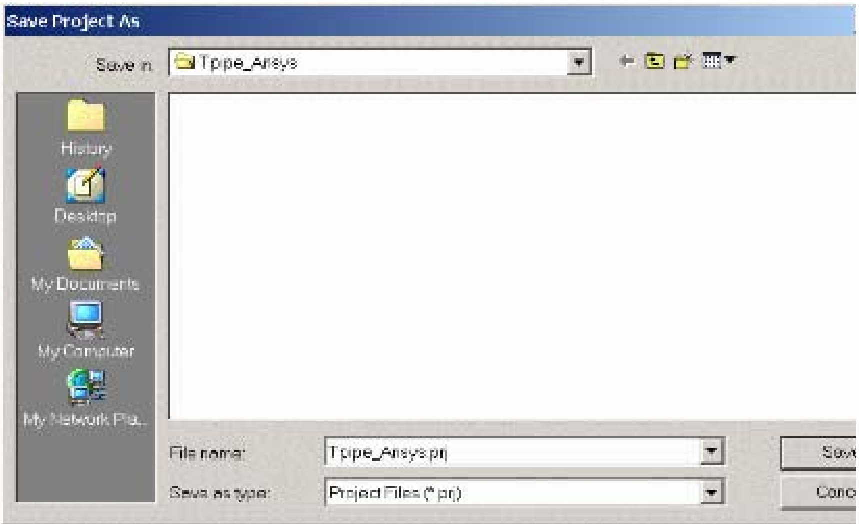
Select **File** > **Save Project As**, and press the new folder creation icon near the upper right. Create the new directory, **Tpipe_Ansys,** then enter that folder and enter **Tpipe_Ansys.prj** as the project name as shown in Figure 4.5.

Press the **Save** button.

Along with the Tpipe_Ansys.prj file, it will also store three other files: the mesh file (.uns), attribute file (.atr), and parameter file (.par).

This also sets the working directory as the project directory.

Figure 4.5 Save Project As window



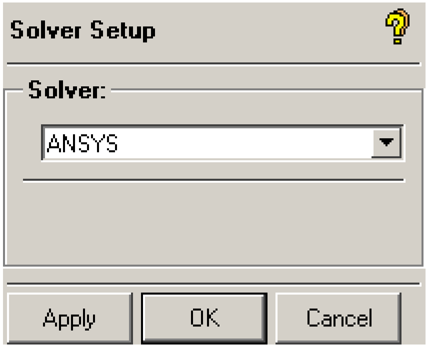
e) Solver Setup

First, the user should select the appropriate solver before proceeding further.

Setup Ansys Run

Select **Settings** > **Solver** from the main menu and select **Ansys** from the dropdown arrow. Then press Apply. Selecting a solver is shown in Figure 4.6.

Figure 4.6 Solver Setup window



Setting Solver Parameters

Click the **Solve Options** tab, then the **Setup Analysis Type** icon.  The window that appears is shown in Figure 4.7.

Enter the following:

Select the Solver as Ansys from the dropdown arrow if it is not already set.

Select the Analysis Category as Structural

Select Modal from the dropdown for Analysis Type and keep all the default options.

Press Apply to complete the setup.

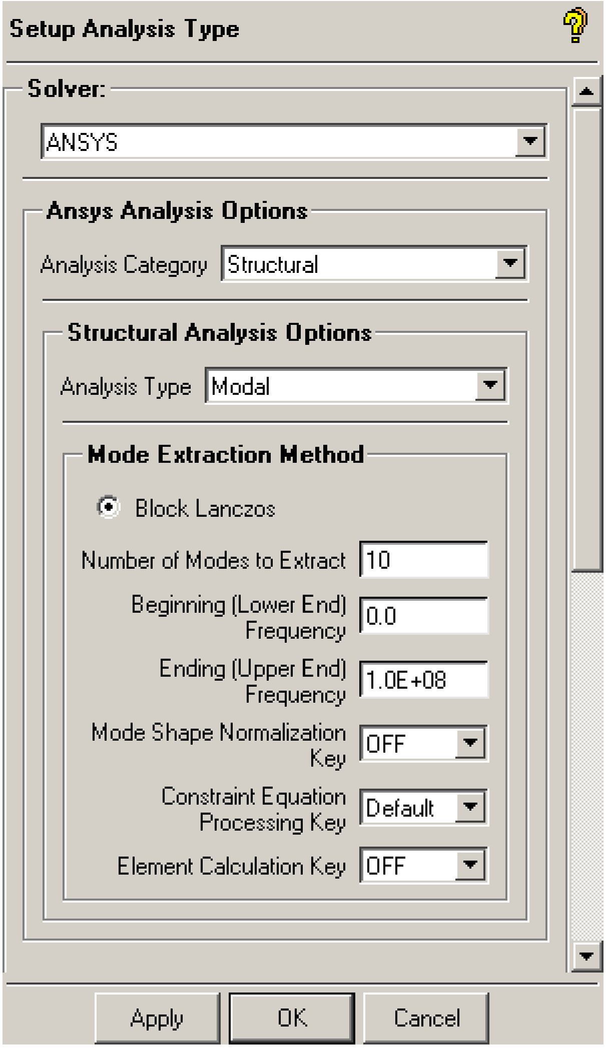


Figure 4.7 Setup Analysis Type window

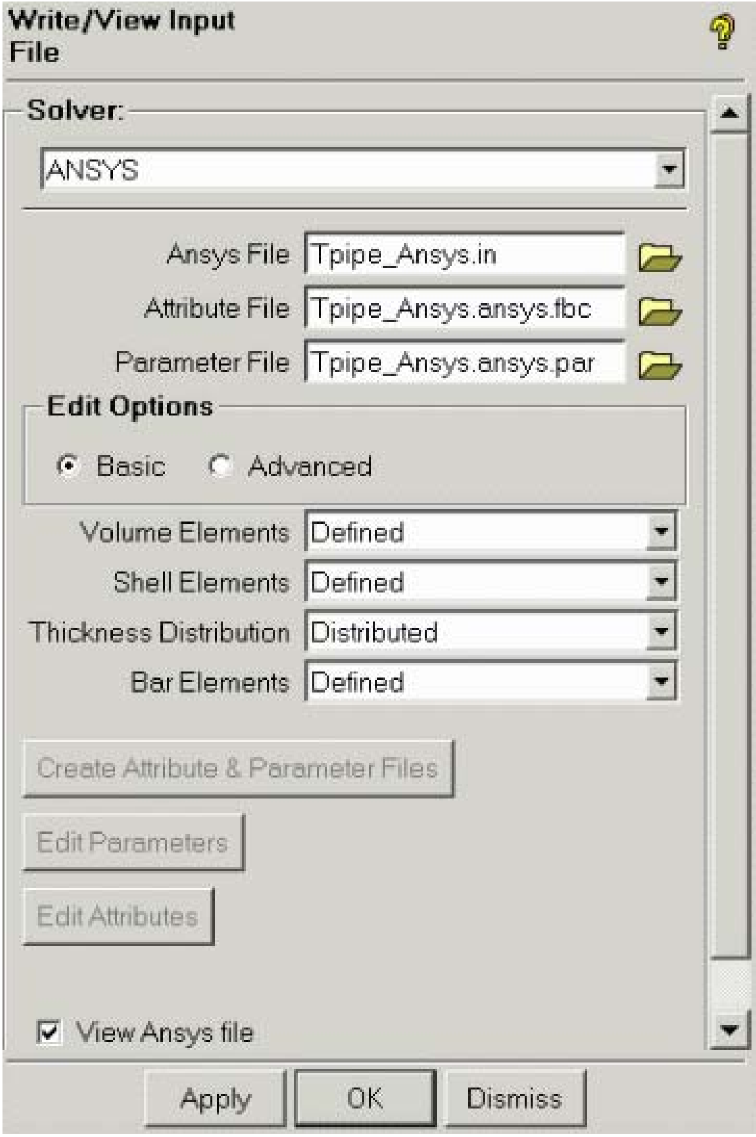
f) Write Ansys Input File

Press the **Write/View Input File** icon  from the **Solve Options** Tab Menu bar.

The Ansys File name should be Tpipe_Ansys.in.

Scroll to the bottom and switch **ON View Ansys file** as shown in Figure 4.8. Keep the other options as the default and press Apply.

Figure 4.8 Write/View Input File window



The Ansys input data file displays in the default text editor. This file can be directly edited and saved, if desired. Since there is no need to edit this example, just close the editor. This file will be saved to the project directory as Tpipe_Ansys.in.

g) Solution and Results

A modal analysis will be performed in Ansys on this model and the results will be visualized within ICEMCFD.

Solving the problem

Click on the **Submit Solver Run** icon  from the **Solve Options** Tab Menu bar to open the window shown in Figure 4.9.

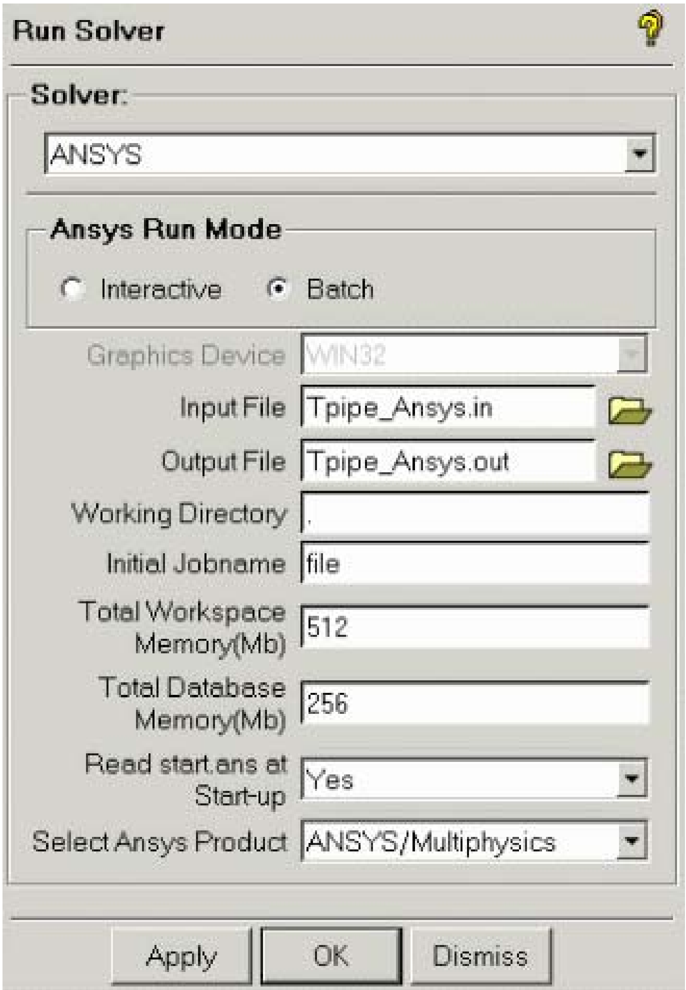
Select **Batch** under **Ansys Run Mode**. Next to **Input File**, the name of the previously written input file should appear, **Tpipe_Ansys.in.**

The **Output File** name can be anything you wish, as this is the Ansys messages file that will be written. Verify the **Working Directory** is correct. A dot means to use the current working directory. Also verify that the **Select Ansys Products** field is correct.

The ANSYS_EXEC_PATH environment variable may have to be set to the full path to the Ansys executable for ICEMCFD to be able to run Ansys.

Press Apply to run the Ansys solver in batch mode.

Figure 4.9 Run Solver window



Post Processing of Results

Click on the **Post Process Results** icon  from the **Solve Options** Tab Menu bar, which will open the **Post Process Results** window given in Figure 4.10.

Confirm that the **Solver** is set to Ansys.

The Ansys **Result file** should be set to **file.rst** where "file" is the **Initial Jobname** specified in the previous menu. Press Apply to launch the Post processor with the Ansys result file.

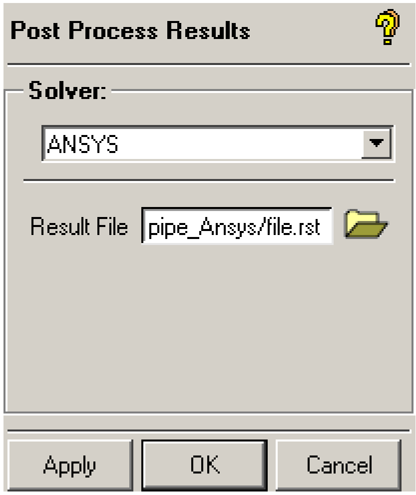
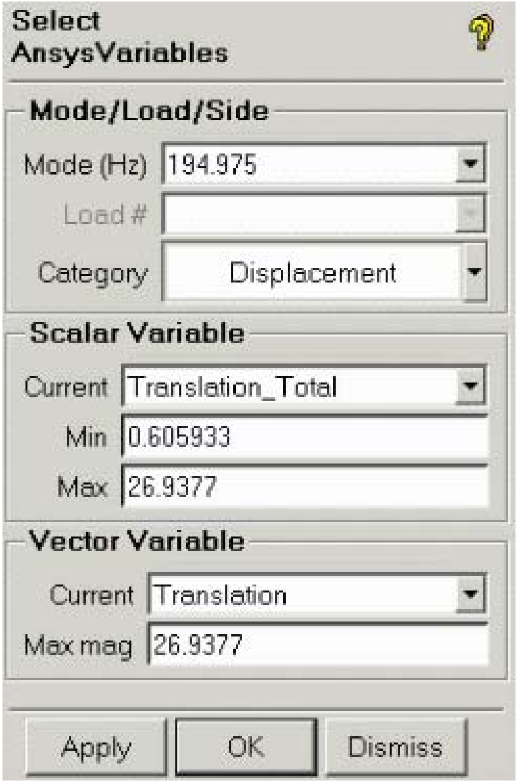


Figure 4.10 Post Process window

Click on the **variables**  icon from the **Post processing** tab to display the **Select AnsysVariables** window. Select the pull down next to **Mode(Hz)** to display the modal frequencies that resulted from the Ansys solution. The numbers near zero are the solutions to the homogeneous equation that result when a numerical method is used, so these are for solid body motion. The first valid number, then, is 194.975. The animation done in the next step will easily show which frequencies are for solid body motion and which are for deformations. Set the **Scalar Variable, Current** to **Total Translation** to see the total displacements as shown in Figure 4.11.

Figure 4.11 Select Ansys Variables window



Click on the **Control all Animations** icon  from the **Post-processing** Tab menu bar. Select ▶ **Animate** (play button).

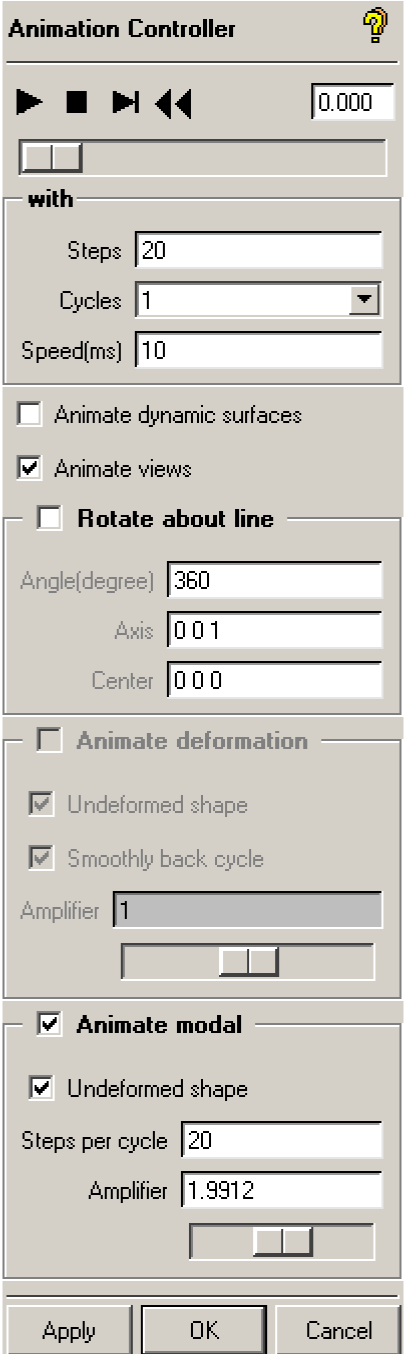
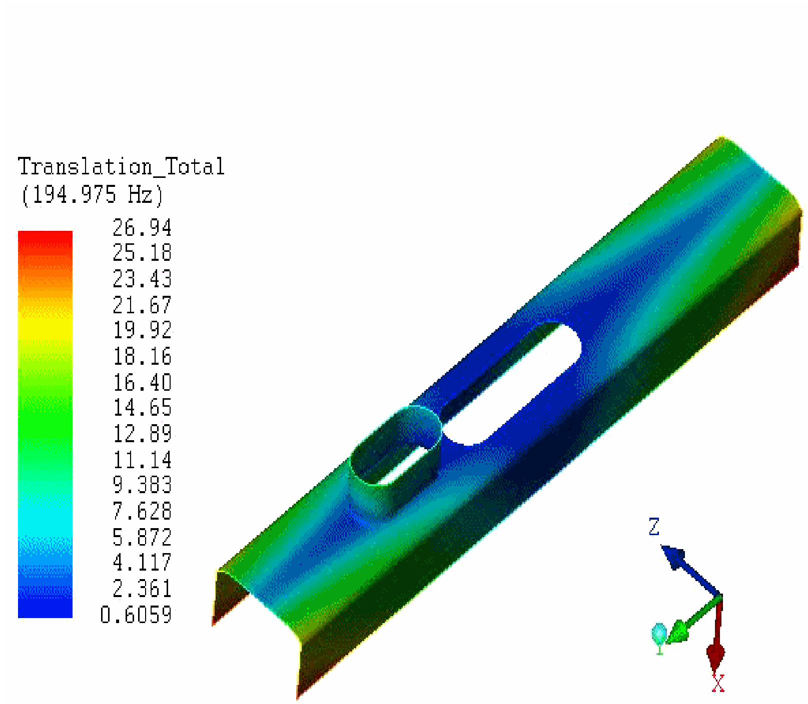


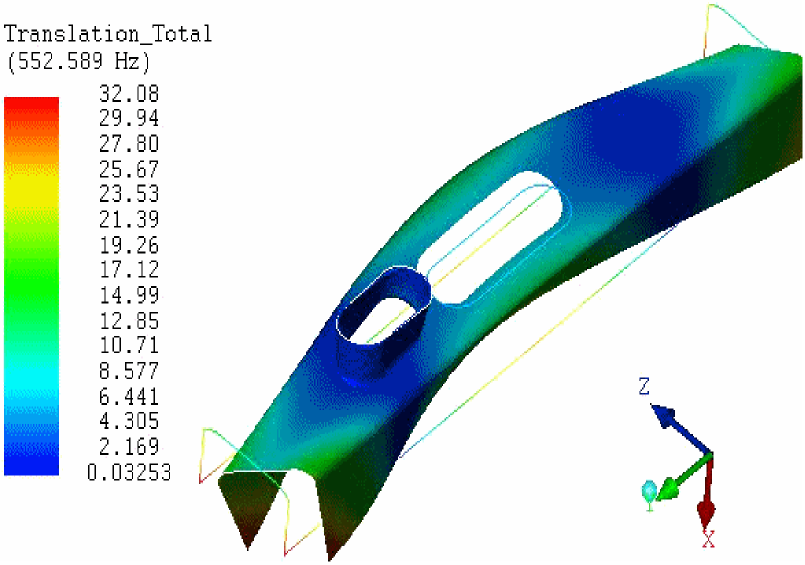
Figure 4.12 Animation Setup and Controller window

Figure 4.13 Mode shape at 194.975 Hz



Similarly to view another mode shape, select the next frequency 552.589 Hz from the **Select AnsysVariables** window and animate the mode shape as shown in the Figure 4.14.

Figure 4.14 Mode shape at 552.589 Hz

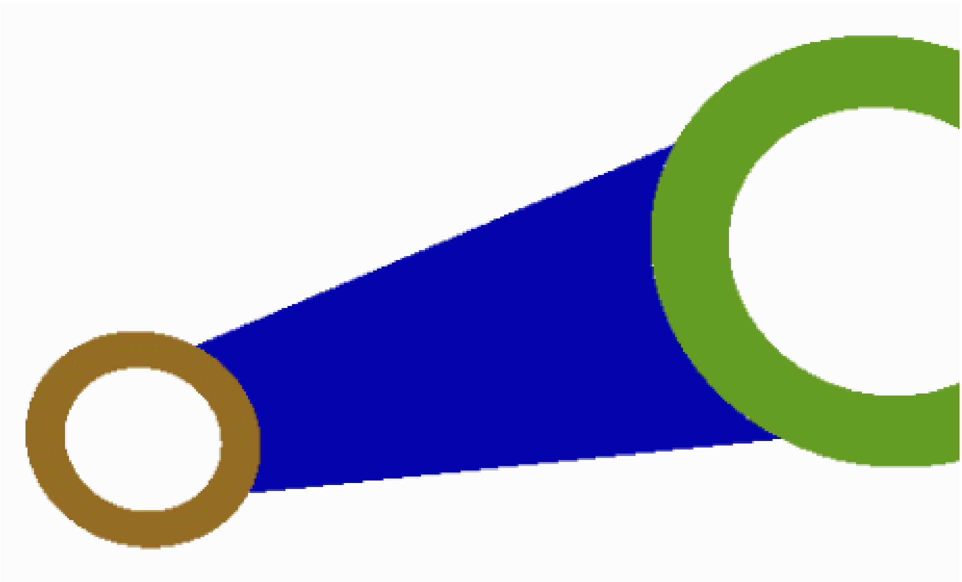


Finally, select **File > Results > Close Result** to quit the post-processor.

4.1.2: Connecting Rod: Thermal Boundary Condition

AI*Environment can be used to see the thermal effects too. Some examples of this category of problems include heat distribution in any automobile component and temperature distribution due to temperature difference. A simple connecting rod structure is used to demonstrate the process here. In the example, the crankshaft end (big end) is made fixed while a high temperature load is applied at the piston end (small end). The geometry is shown in Figure 4.15:

Figure 4.15 Connecting Rod Model



a) Summary of Steps

Geometry Editing

Launch AI*Environment and load geometry file

Extracting Curves and Points

Mesh parameters and Meshing

Mesh Sizing

Meshing

Extrusion of the surface mesh

Materials and Element Properties

Selection of Material

Element Properties

Subsets

Subset1

Subset2

Constraints and Loads

Constraints

Loads

Solver setup

Setup Ansys Run

Setting Solver Parameter

Save Project

Write Ansys Input File

Solution and Results

Solving the Problem

Post processing of results in Visual3p

b) Geometry Editing

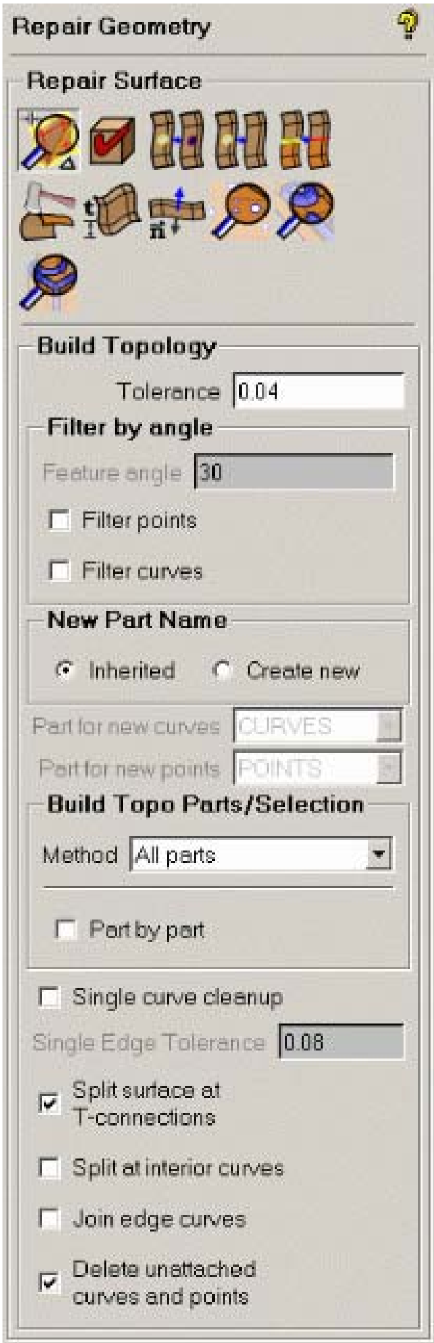
Launch AI*Environment.

Select the **Open Geometry** icon  from the main menu and select the file **Conrod.tin** from the **AI_Tutorial_Files** working directory.

Extracting Curves and Points

Click on **Geometry** > **Repair Geometry** . The window shown in Figure 4.16 will appear. The **Build Diagnostic Topology**  option is selected by default. The default **Tolerance** of 0.04 will work fine here. Make sure that **Inherited** is toggled **ON** for **New Part Name** and press Apply.

Figure 4.16 Repair Geometry window



You will now see the points and curves as shown in Figure 4.17.

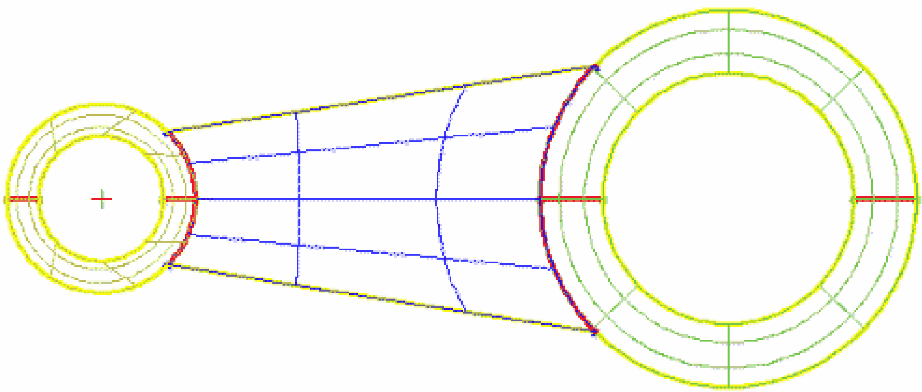


Figure 4.17 Geometry after Build Topology

c) Mesh Parameters and Meshing

We will create a 3D mesh, but right now the geometry is only 2D, so we will first mesh the 2D section, then extrudes it into volume mesh. The patch-based mesher will be used to mesh the surfaces, and this mesher only uses element sizes prescribed on curves, not surfaces sizes, so we will next set curve parameters.

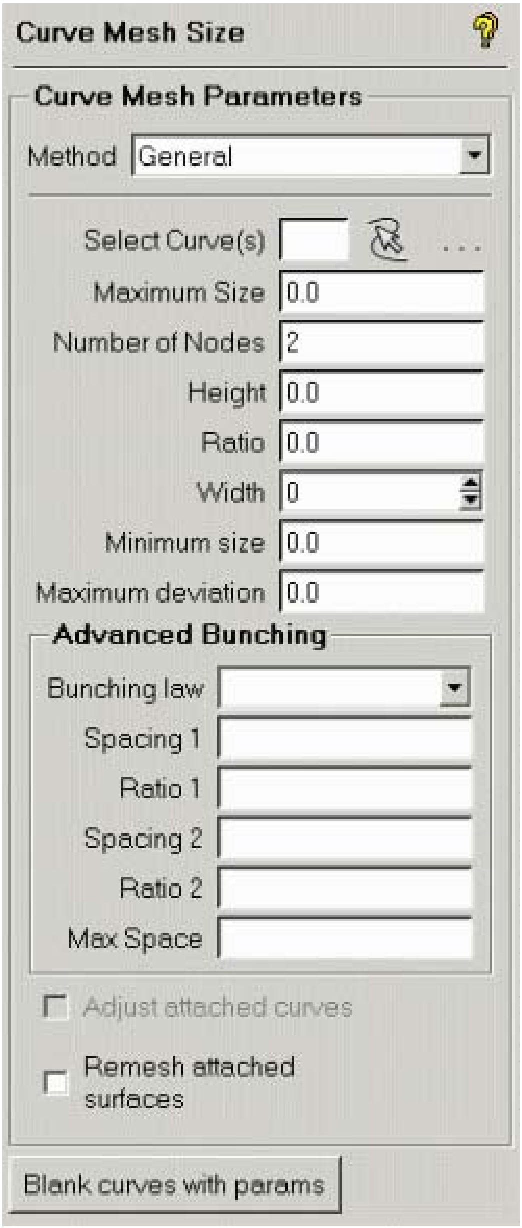
Mesh sizing

Select **Mesh>Set Curve Mesh Size** , and the window shown in Figure 4.18 will display.

The **Method** should be set to **General**. Then select the curve selection icon and select all the curves by pressing '**a**' (ensure that the mouse cursor is in display window). Now enter a **Maximum Size** of **1**, and press Apply.

You can see the node positions by right clicking in the Model Tree on **Geometry>Curves>Curve Node Spacing**.

Figure 4.18 Curve Mesh Size window



Meshing

Select Mesh > Surface Meshing > Patch based  .

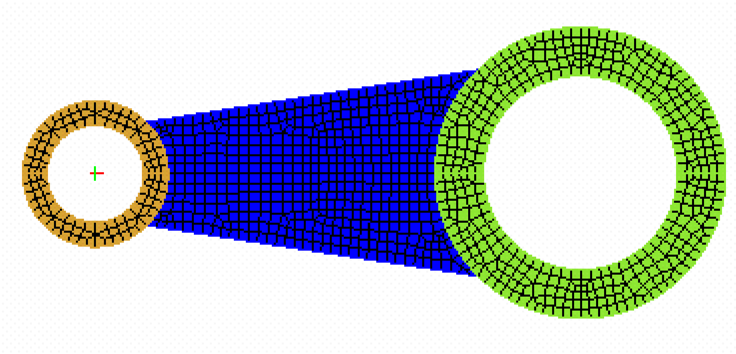
Change the Mesh Type from **Quad dominant** to **All Quad**.

Click on the surface selection icon and select all the surfaces by pressing '**a**' (ensure that the mouse cursor is in display window). Then press Apply in the **Mesh Surface** window as shown in Figure 4.19 to begin the meshing.

Figure 4.19 Mesh Surface window

Click on the Solid display icon  in the main menu. In Model Tree, make sure that **Surfaces** are OFF so you are not looking at surfaces and mesh in solid. The mesh should appear as shown in Figure 4.20 with **Curves** and **Points** 'OFF' also.

Figure 4.20 Mesh in Solid & Wire mode



Extrusion of surface mesh

In the Model Tree, expand under **Mesh**, and make sure that **Points** and **Lines** are turned OFF, and **Shells** are turned ON. This is so we only select and extrude shell elements.

Click on the **Mesh>Extrude Mesh** icon, and select **Extrude by vector** next to **Method**. You should see the window shown in Figure 4.21:

Click on the element selection  icon, and select all the surface mesh elements by drawing a rectangular box around the model with the left mouse button or pressing the hotkey "**v**" for visible. Click the middle mouse button click to accept.

Set the New volume part name to STEEL_ELEMENTS.

Set the New side part name to SIDE.

Set the New top part name to TOP.

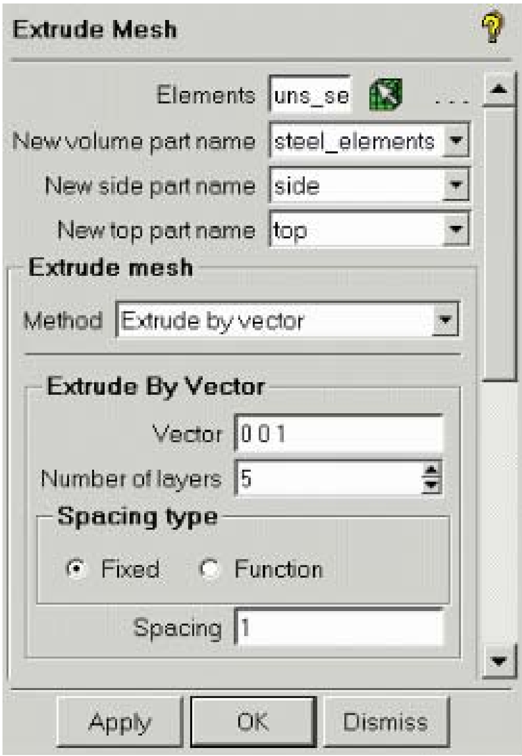
Enter **Vector** as **0 0 1**.

Set the Number of layers to 5,

The **Spacing type** should be set to **Fixed** and the **Spacing** at **1.**

Press Apply.

Figure 4.21 Extrude Mesh window



The mesh should appear as shown in Figure 4.22.

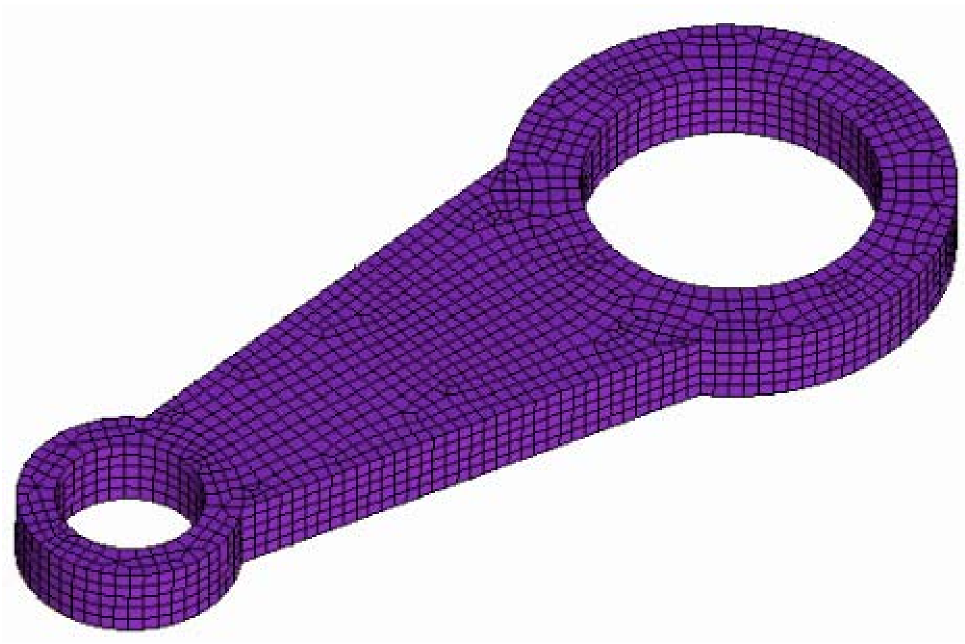


Figure 4.22 Extruded mesh

d) Material and Element Properties

Before applying Constraints and Loads on the elements, define the type of material and assign properties to the elements.

Material Definition

Select the **Properties> Create Material Property**  icon.

Enter the name **STEEL** for the **Material Name**.

The **Material ID** can be left as **1**,

The **Type** can be left as **Isotropic**,

Define Young's Modulus as a **Constant** value of **207000**,

Define Poisson's ratio as a **Constant 0.28**,

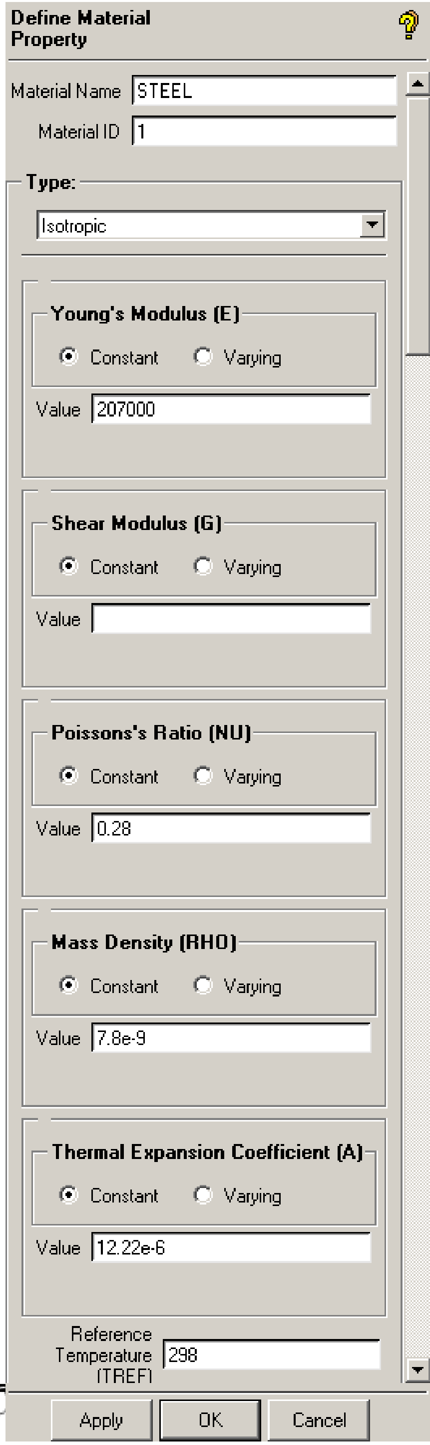
Define the Mass Density as a **Constant 7.8e-9**.

Define the Thermal Expansion Coefficient as a **Constant 12.22e-6**.

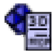
Enter the Reference Temperature (TREF) as 298.

Press Apply.

Figure 4.23 Define Material Property window



Element Properties

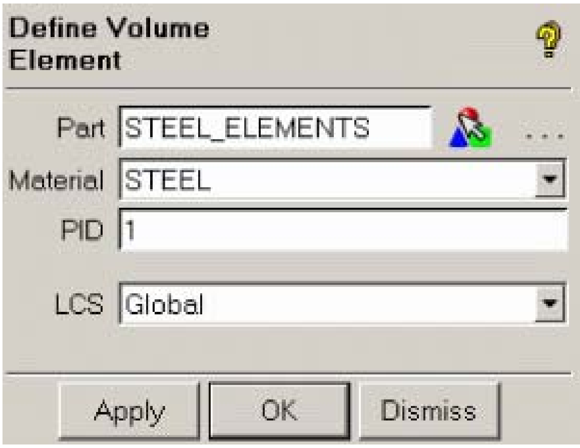
Select the **Properties> Define 3D Element Properties**  icon, and the **Define Volume Element** window as shown in Figure 4.24 will appear.

Press the part selection icon , and select the part, **STEEL_ELEMENTS**.

Select the **Material** as **STEEL**, which was previously defined.

Set the **PID** (Property ID) as **1** in the Press Apply**.**

Figure 4.24
Define Volume
Element window



e) Subsets

Constraints and Loads can be applied to geometry and mesh, and either can be 0D (points), 1D (lines), 2D (surfaces or surface elements), or 3D (bodies or volume elements). It is also possible to set up sub-groups of any combination of these, and then apply the constraints and loads on these groups. These are called **Subsets**. So we will use subsets to assign constraints and load.

Subset0

In the Model Tree, right mouse click on **Subsets>create** under **Mesh**. This will pop up the **Create Subset** window as shown in Figure 4.25.

The **Subset** name should read **Subset0** if none have been created yet. Then press the first icon for the screen selection method, (**Create Subset by Selection**) if it is not already selected.

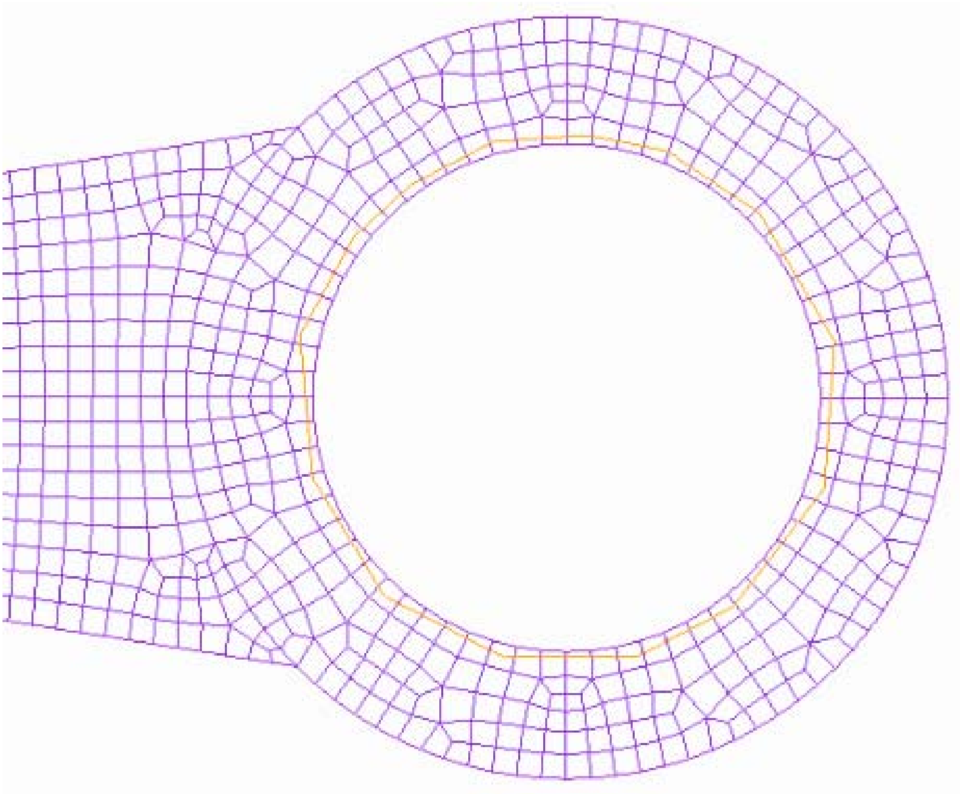
Make sure that the only **Mesh** elements that are ON in the Model Tree are **Shells**.

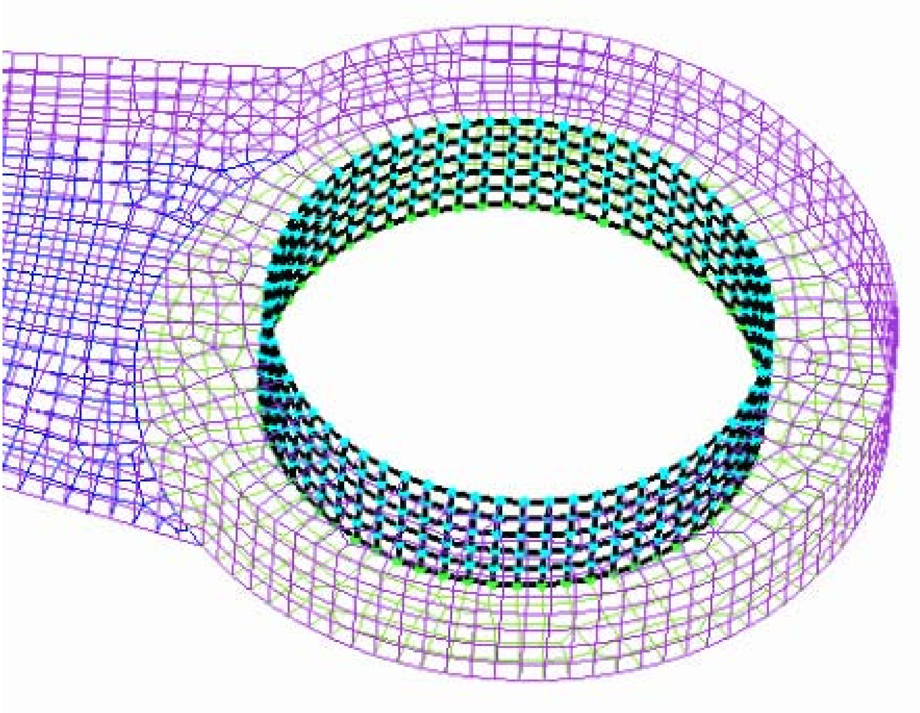
Use **View>Front** to orient the model for easy selection. Right mouse click and move the mouse up or down to zoom in or out in order to see the large hole side.

To select the elements on the Crankshaft end for this subset, click on the **Select elements** button, then press "**p**" from key board (ensure that the mouse cursor is in the display window) or click the polygon selection icon , then keep left mouse clicking on the screen to draw a polygon as shown in Figure 4.26. Click the middle mouse button click to finish the polygon and then middle mouse click again to finish all selection. Then press Apply to create the subset. You'll see the name **Subset0** appear under **Mesh>Subsets** in the Model Tree.

Figure 4.25 Create subset window

Figure 4.26 Elements selection by polygon and elements selected for Subset0





Subset1

In the Model Tree, right mouse click again on **Subsets > Create**. Enter the name **Subset1** this time. Use polygon selection again, but this time select around the small hole, which is the Piston end as shown in Figure 4.27. Press Apply to create Subset1.

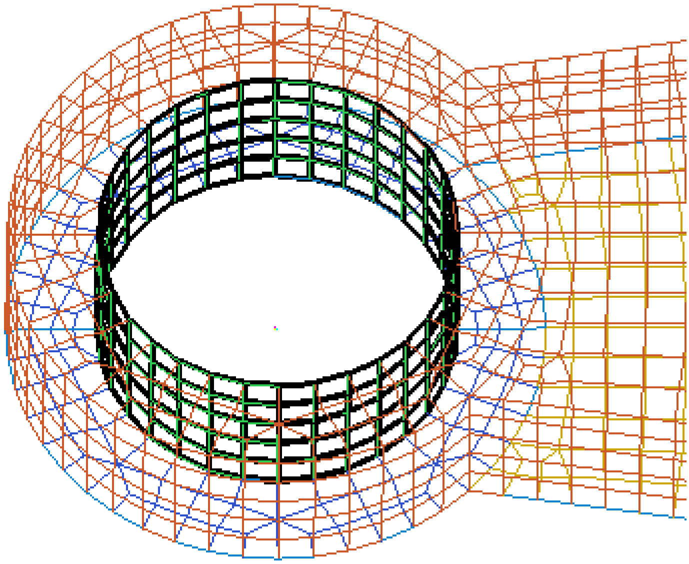


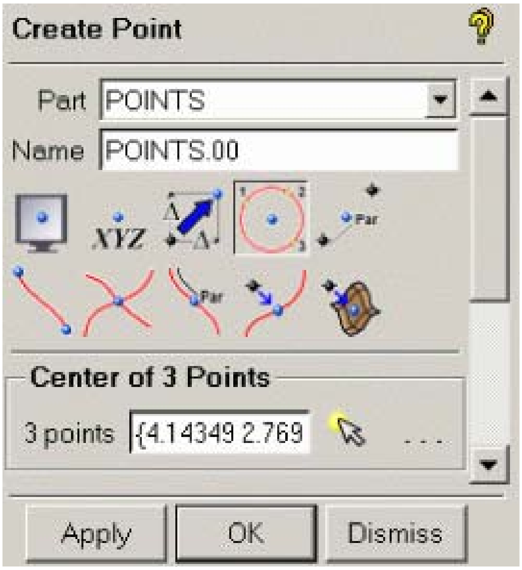
Figure 4.27 Elements selected for Subset1

f) Cylindrical Coordinate System

In order to fix translations radially around the crankshaft hole, we'll need to create a local cylindrical coordinate system. But we'll first need to create points at the center of the large hole.

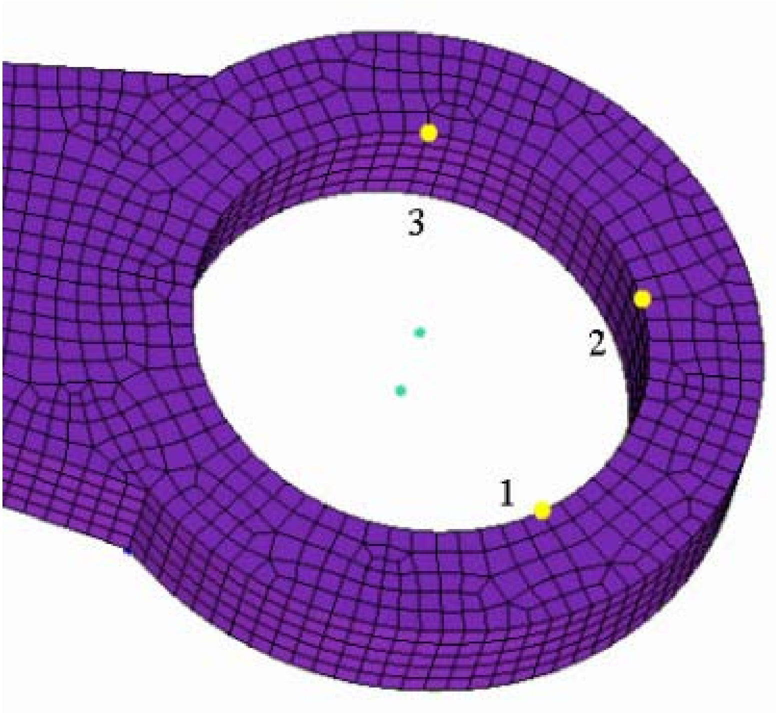
Press **Geometry>Create point>**  **Center of 3 Points** . You should see the window shown in Figure 4.28.

Figure 4.28 Center point menu



Select 3 points on the nodes at one side of the large hole as shown in Figure 4.29. Then press Apply to create the center point. Do this for the other side of the hole as well, so that there are two center points.

Figure 4.29
Three points



Press the **Local Coordinate Systems** button  from the main menu. You should see the window shown in Figure 4.30.

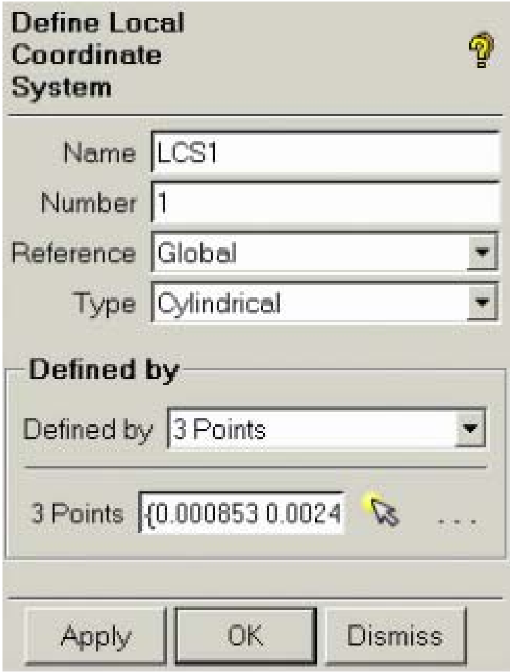
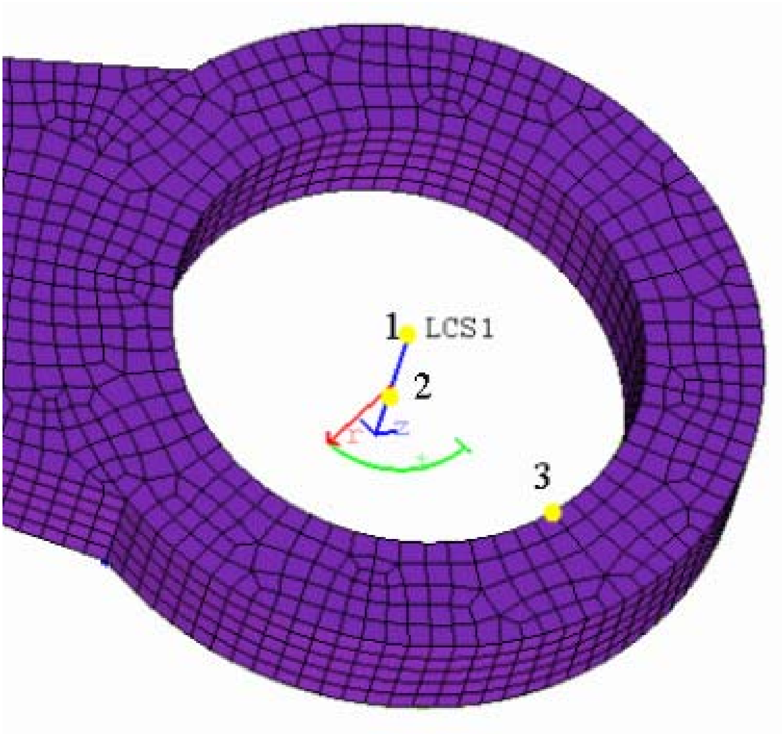


Figure 4.30 Define Local Coordinate System Window

The **Name** should read **LCS1**.
Select **Cylindrical** from the pull down next to **Type**.
When selecting the three points, the first point is the origin. The second point defines the direction for the z-axis, which is the cylindrical axis. And the third point defines the starting point of the angle, theta. Select the three points in the order shown in Figure 4.31. Press the middle mouse button to accept, and then Apply.

Figure 4.31 Cylindrical coordinate system



After creating this, turn OFF Local Coord Systems > LCS1 **in the Display Tree.**

g) Constraints and Loads

Constraints

Click on the **Constraints> Displacement on Subset** icon. This will bring the **Create Displacement on Subset** window given in Figure 4.32.

In this window, enter the **Name** as **CNST1**.

From the pull down arrow next to **LCS**, select **LCS1**. The UX, UY, and UZ translations will change to UR, Utheta, and UZ translations.

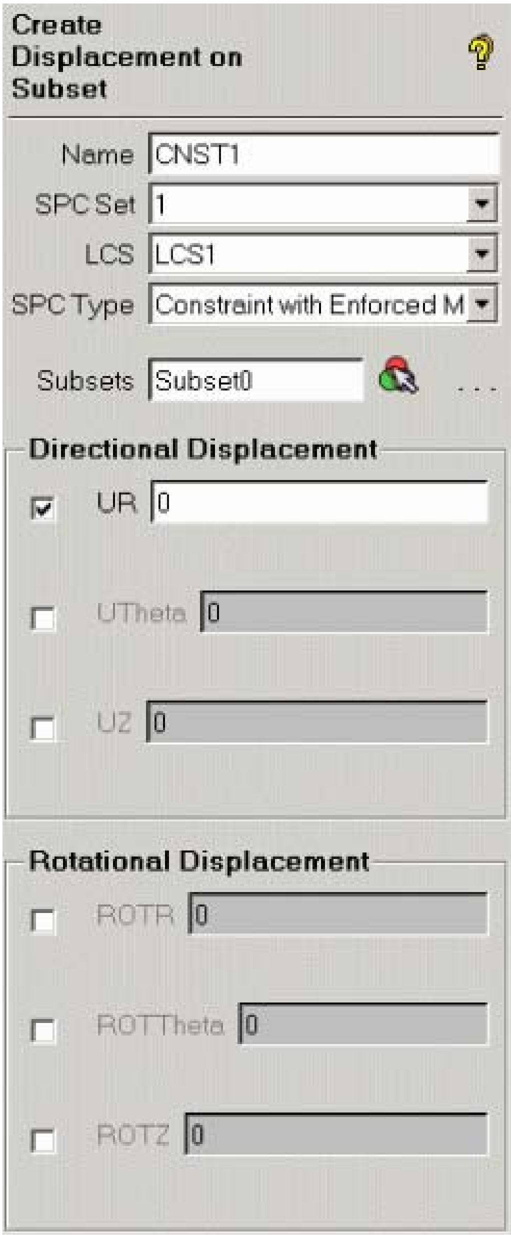
Click on the subset selection button and select **Subset0** as shown in Figure 4.32.

Toggle **ON** the option **UR**. Leave the default of “0” for this field to fix translations in the radial direction. Then press Apply.

Do not be concerned about the direction that the displacement arrows point. They do not indicate the displacement direction. They always point to the left.

Turn **OFF Displacements** from the Model Tree to simplify the display.

Figure 4.32 Create Displacement on Subset window



Constraint of Solid-Body Motion

The radial constraints will constrain the model from solid body motion radially from the shaft hole, but it also needs to be constrained in the Z direction, and rotation around the shaft hole. You should choose a node on the surface of the large hole, preferably on the symmetry plane of the model, since we know that will not rotate around the shaft.

Press the **Constraints>Displacement on Point** button. You should see the window shown in Figure 4.33.

Enter the Name, FIXED_MOTION.

The **LCS** should be set as **LCS1**.

Check ON the **Directional Displacement**, **UTheta,** and **UZ**, and leave the numbers at “0.”

For the **Points**, select one node on the symmetry plane of the model that is on the large hole. See Figure 4.34 as a reference. Then press Apply.

Turn **OFF Displacements** from the Model Tree to simplify the display.

Figure 4.33 Create Displacement on Subset window

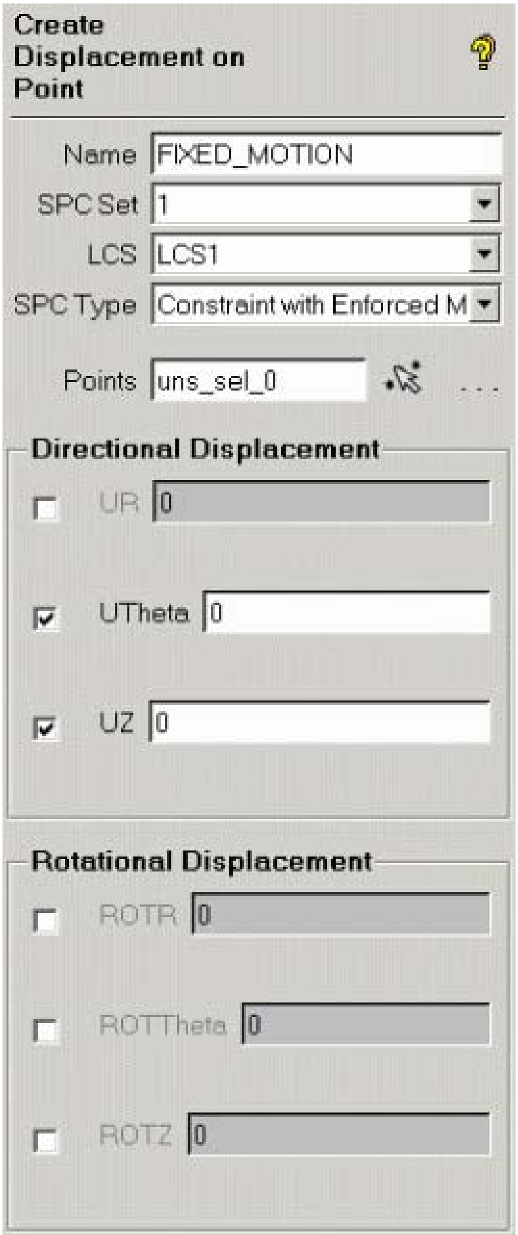
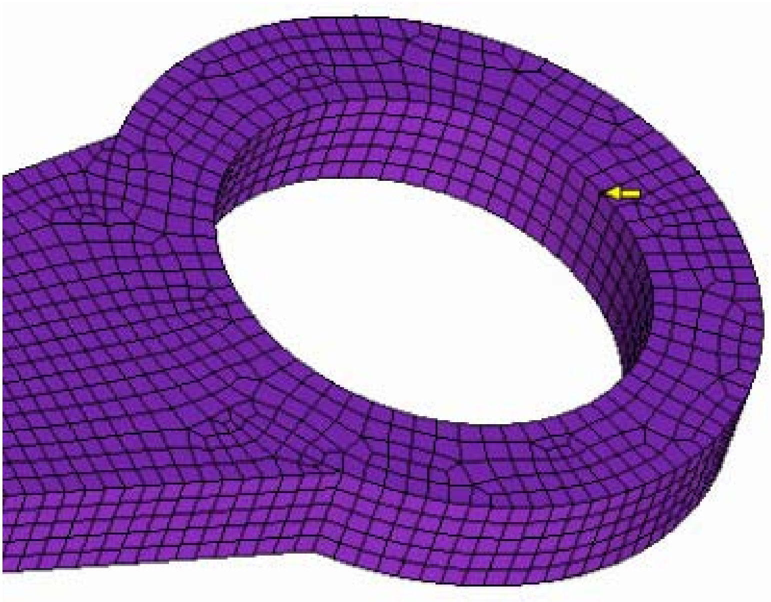


Figure 4.34
Node to constrain solid body motion



Loads

Click on **Loads>Temperature on Subset**, which will bring up the **Define Temperature Boundary Condition on Subsets** window shown in Figure 4.35.

Enter the Name as **TEMPERATURE**.

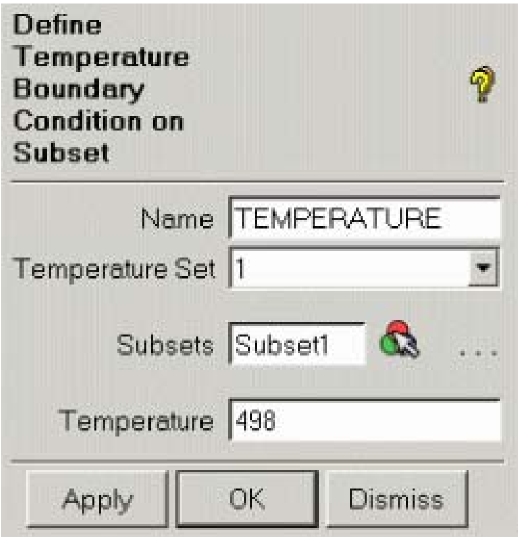
Click on the subset selection button and select **Subset1**.

Enter a value of **498** for the **Temperature** and press Apply**.**

Temperature loads display as red dots.

Turn **OFF Temperatures** from the Model Tree to simplify the display.

Figure 4.35 Define Temperature Boundary Condition on Subsets window



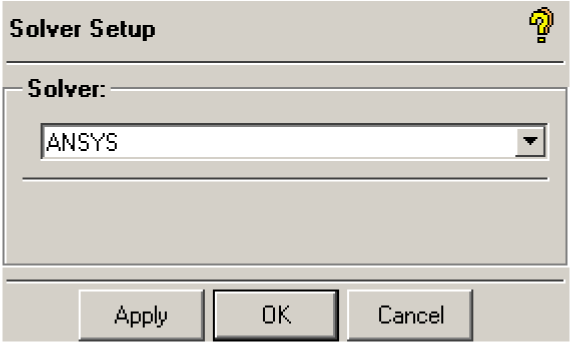
h) Solver Setup

First, the user should select the appropriate solver before proceeding further.

Setup Ansys Run

Select **Settings** > **Solver** from the main menu. Select **Ansys** as the solver and press Apply. Selecting a solver is shown in Figure 4.36.

Figure 4.36 Solver Setup window



Setting Analysis Type

Click on the **Solve Options > Setup Analysis Type**  button to setup an Ansys run to do Linear Static Analysis. This will bring up the **Setup Analysis Type** window, as shown Figure 4.37.

The solver should be set as **Ansys**.

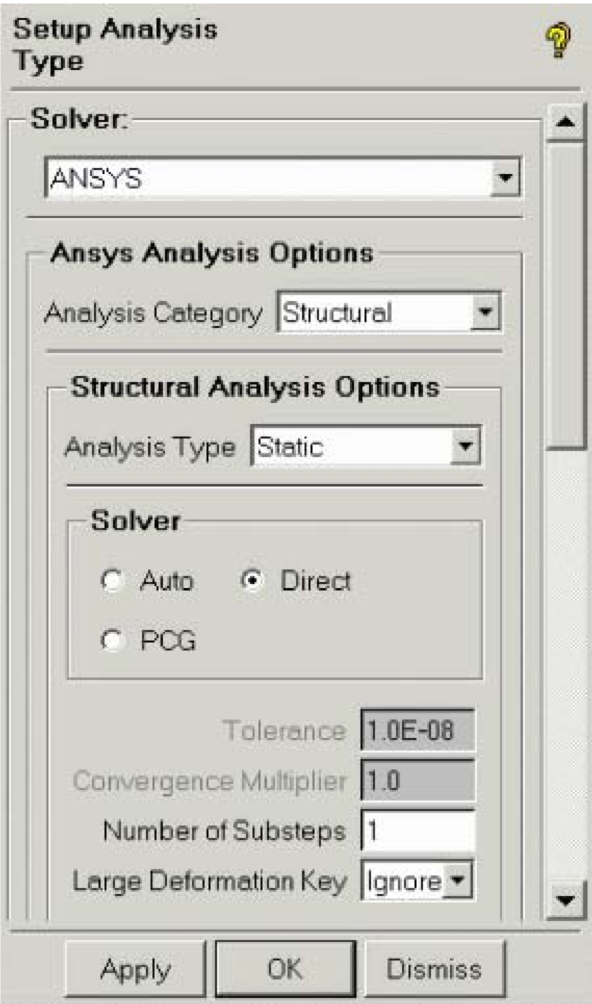
Set the **Analysis Type** to **Static** from the pull down.

Select the **Direct** option under the **Solver**.

Leave all other options as default,

Press Apply to complete the setup.

Figure 4.37 Setup Analysis Type window



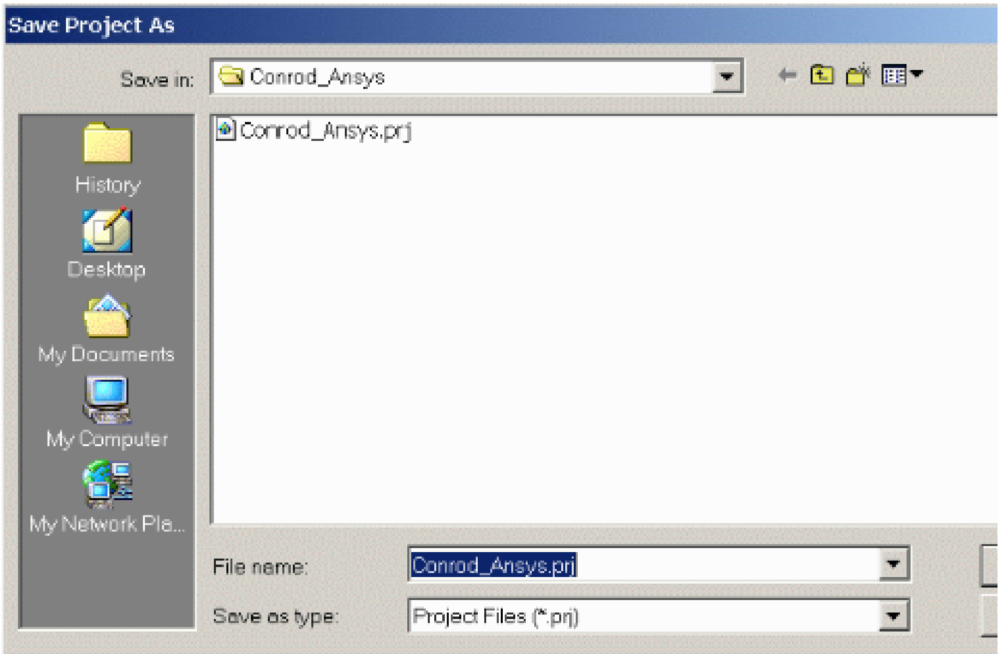
Save Project

Through **File** > **Save Project As**, create a new directory called **Conrod_Ansys** and enter into it.

Enter **Conrod_Ansys** as the project name and press **Save** to save the geometry, mesh, constraints, and loads in this directory as shown in Figure 4.38.

It will save six files: Geometry file (.tin), Mesh file (.uns), Attribute file (.atr), Parameter file (.par), boundary conditions file (.fbc), and the project settings file (.prj).

Figure 4.38 Save Project As window

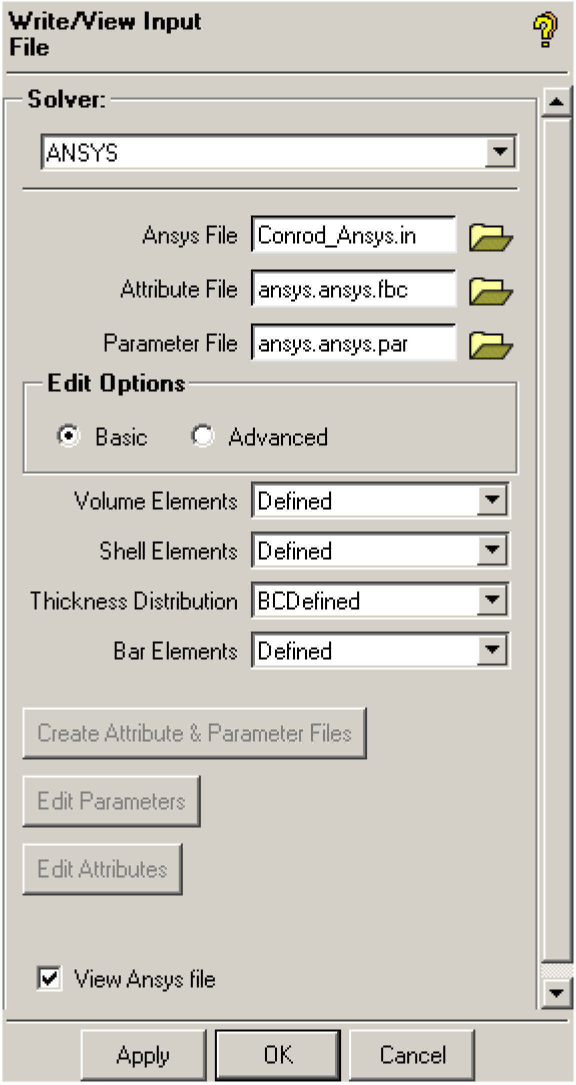


Write Ansys Input File

Click the **Solve Options> Write/View Input File**  button.

Enter the **Ansys file** name as **Conrod_Ansys.in** and switch **ON View Ansys file** at the bottom as shown in Figure 4.39. Press Apply.

Figure 4.39 Write/View Input File window



You will see that the Ansys input data file comes up in the default text editor. This file can be edited and saved, if desired. Since there is no need to do any editing for this example, just close the editor.

i) Solution and Results

A Linear Static analysis will be performed on this model and the results will be visualized within ANSYS ICEMCFD.

Solving the problem

Click on the **Solve Options> Submit Solver Run**  button to display the window shown in Figure 4.40.

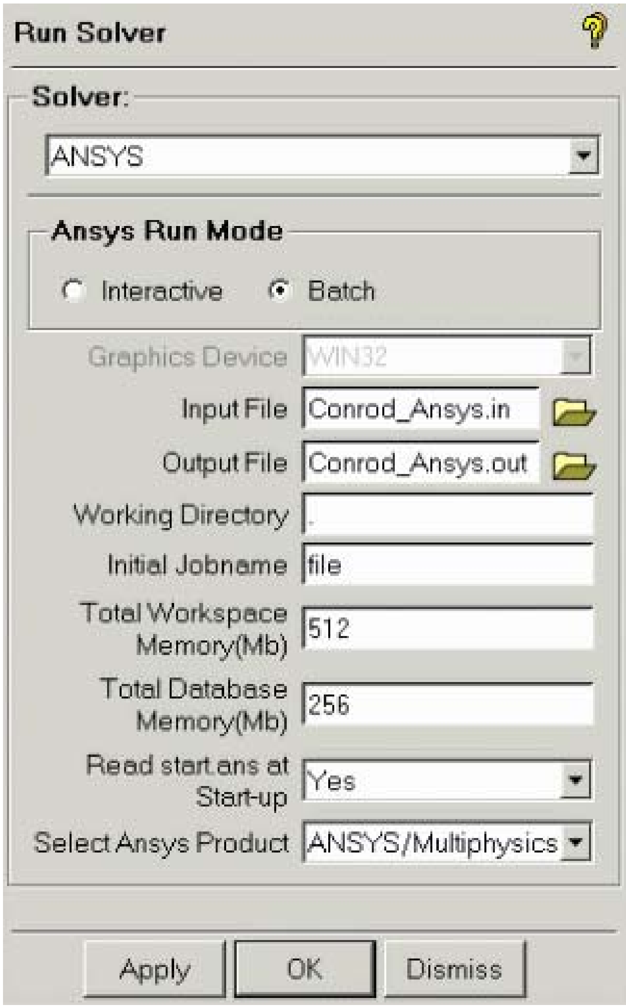
Select the **Batch** option and ensure that the **Input File** is set to the **Conrod_Ansys.in** file created in the above step.

The **Output File** can be any name you give it, but the default will be **Conrod_Ansys.out**. Verify the **Working Directory** is correct. A dot means to use the current working directory. Also verify that the **Ansys Products** field is set correctly.

The ANSYS_EXEC_PATH environment variable may have to be set to the full path to the Ansys executable for ICEMCFD to be able to run Ansys.

Press Apply to run the Ansys solver in batch mode.

Figure 4.40
Run Solver window

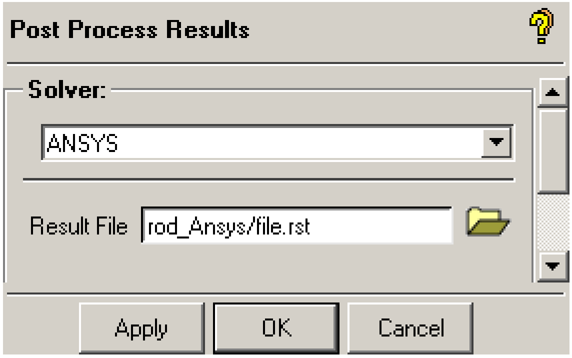


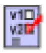
Post Processing of Results

Click on the **Solve Options>Post Process Results** button, which opens the **Post Process Results** window given in Figure 4.41.

Press the yellow folder button next to **Result file**, and select the file, **file.rst.** This file name comes from the **Initial Jobname** used in the previous window. Press Apply to launch the post processor with the Ansys result file.

Figure 4.41 Post Process Results window



Click on  **Variables** from the **Post-processing** Tab menu bar.

To display the **Total Translation** Displacement, select **Load#** as **1** and **Category** as **Displacement** in the **Select AnsysVariables** window as shown in Figure 4.42.This is the default.

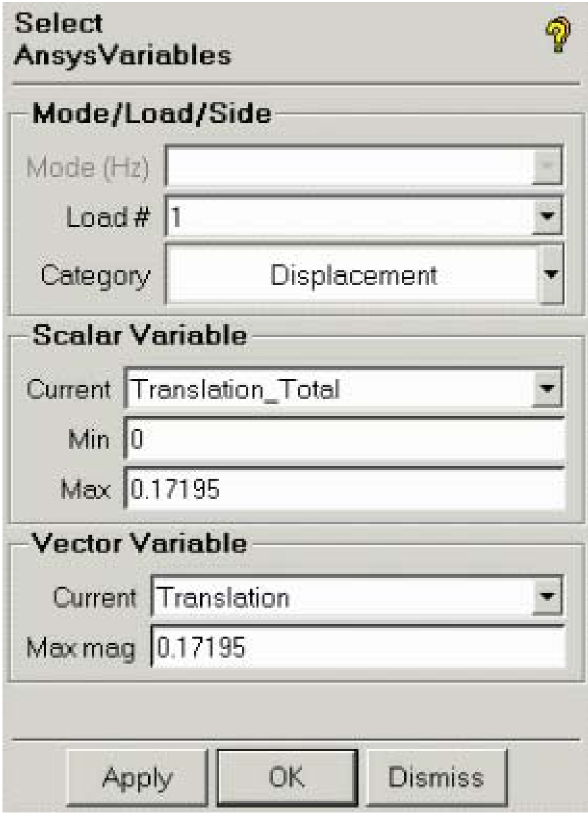


Figure 4.42 Ansys Variables window

Click on **Control All Animations** from the **Post-processing** Tab menu bar. The window shown in Figure 4.43 will appear.

Select **Animate**. The deformation is shown in Figure 4.44.

Figure 4.43 Animation Setup and Controller window

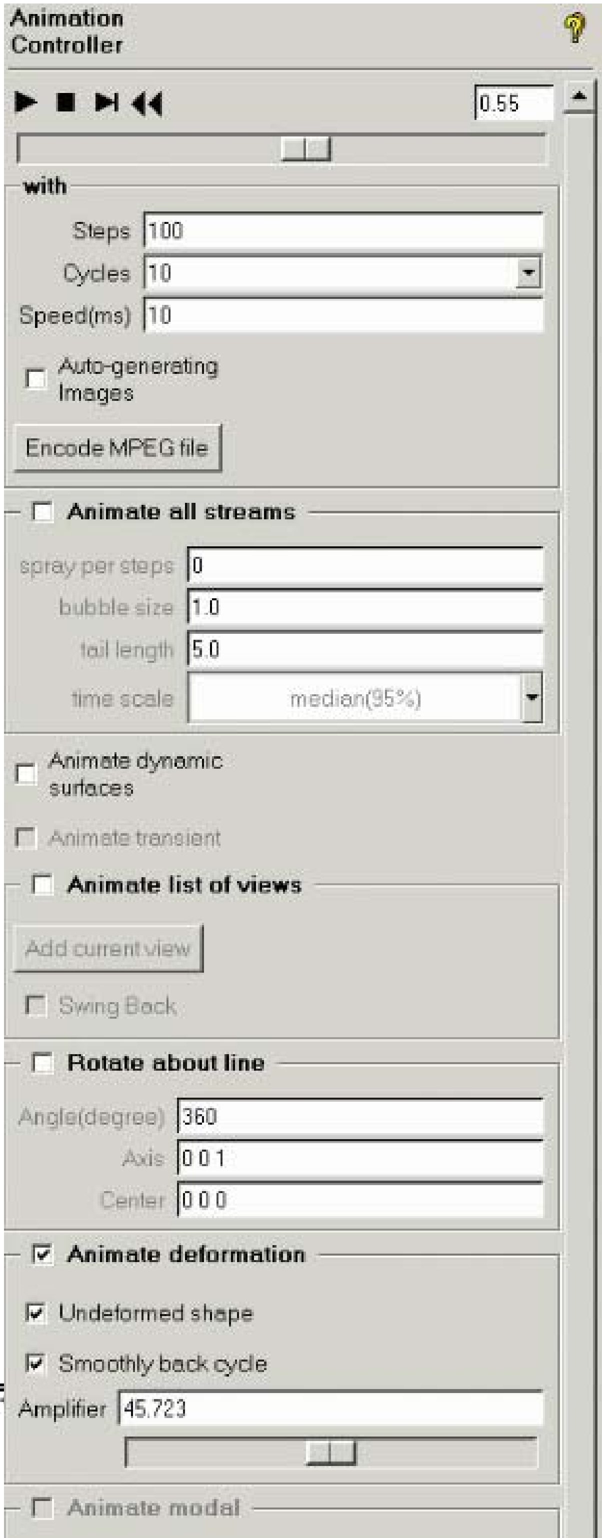
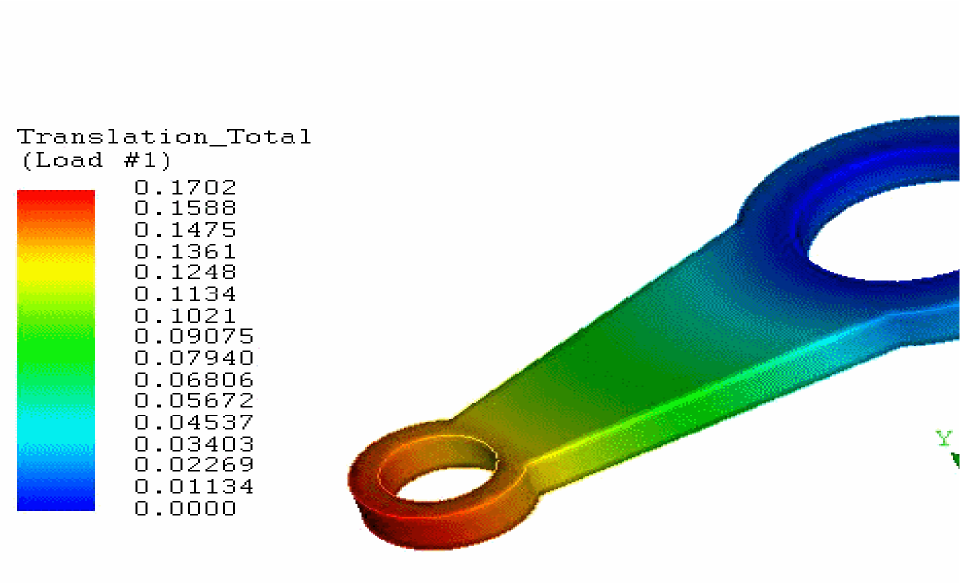


Figure 4.44 Animated model of Total Translation

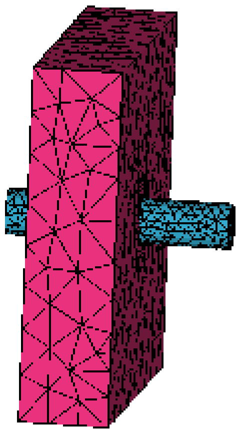


Finally, select **File** > **Results** > **Close Result** to quit the post processor.

4.1.3: Contact Analysis

The main objective of this tutorial is to demonstrate the ease of use in generating a tetra mesh in AI*Environment and then defining contacts. After defining the contact between the pin and block, contact analysis will be done in Ansys. The mesh for this tutorial is shown in Figure 4.45.

Figure 4.45
Pin Block
Geometry



a) Summary of Steps

Geometry Editing

Launch AI*Environment

Repair

Mesh Sizing

Meshing and Internal wall

Tetra Meshing

Define Internal wall

Material and Element Properties

Selection of Material

Element Properties

Constraints and Displacements

Constraints

Displacement

Contact

Solver setup

Setup Ansys Run

Save Project

Write Ansys Input File

Solution and Results

Solving the Problem

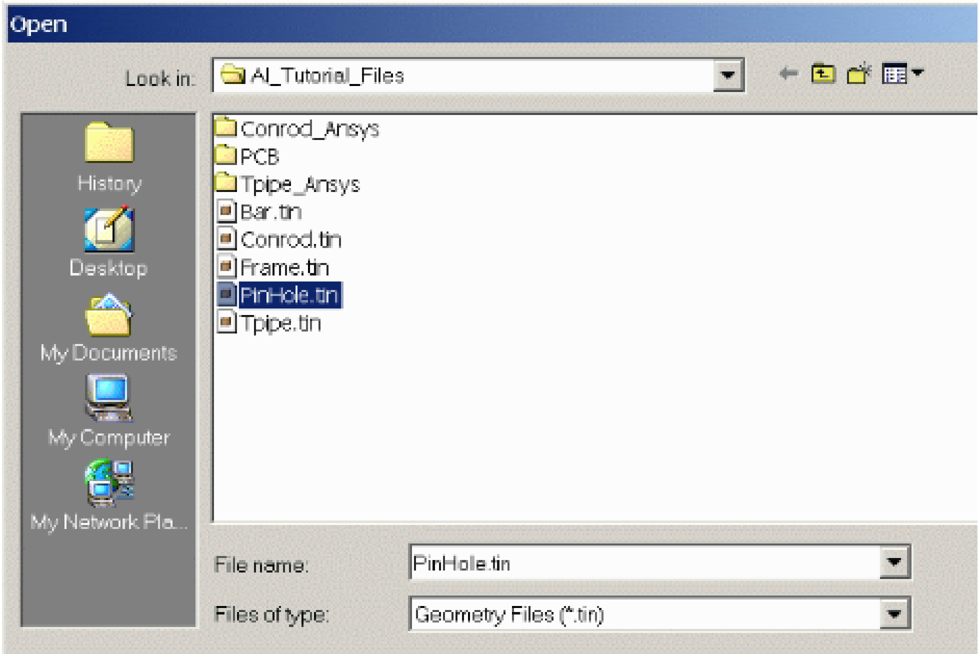
Post processing of Results

b) Geometry Editing

Launch AI*Environment

Launch AI*Environment.
Then use **File > Change working directory**, and set the current directory to **$ICEM_ACN/../docu/FEAHelp/AI_Tutorial_Files**. Then use **File>Geometry>Open Geometry** and load **PinHole.tin.**

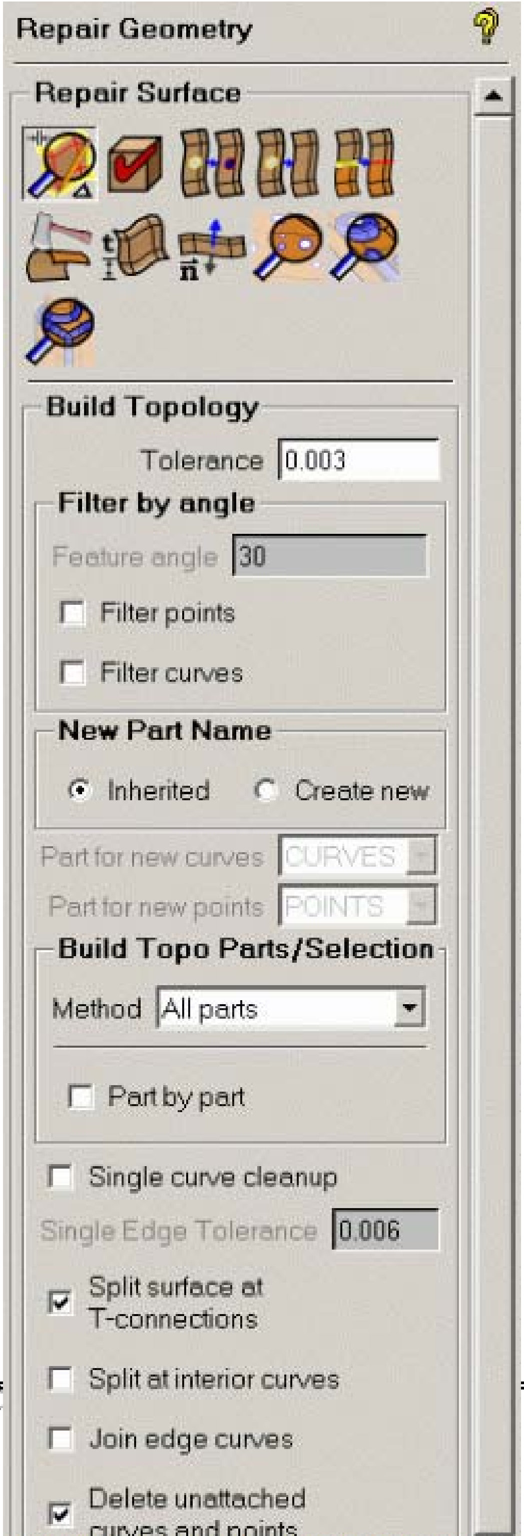
Figure 4.46 Open Geometry File window



Repair

Click on the **Geometry>Repair Geometry**  button, which will bring up the **Repair Geometry** window as shown in Figure 4.47. The default **Tolerance** of 0.003 should work fine here. Make sure that **New Part Name** is set to **Inherited**. Then press Apply.

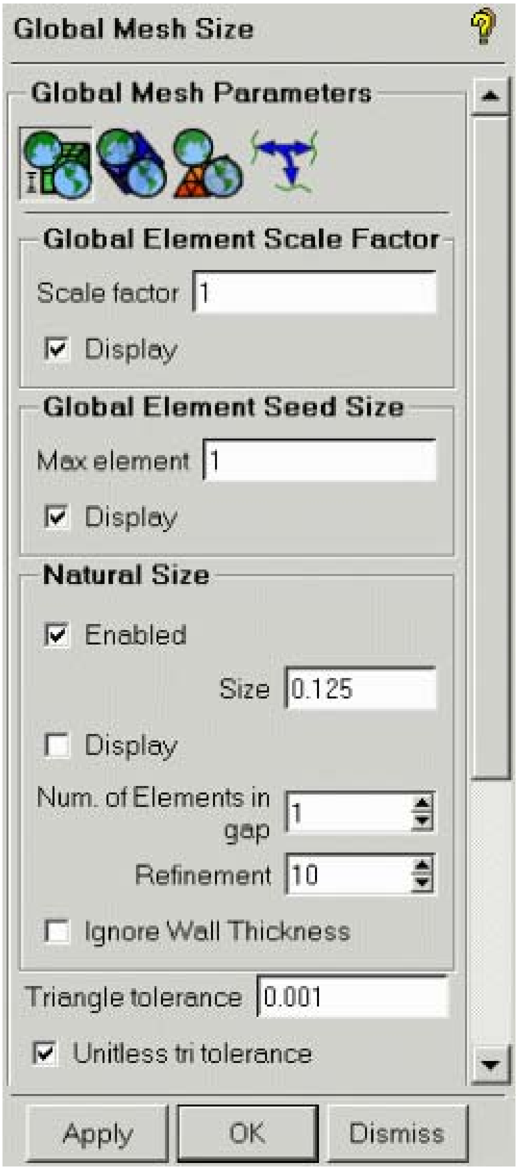
Figure 4.47 Repair Geometry window



Mesh sizing

Select the **Mesh > Set Global Mesh Size** >**General Parameters** button. In the **Global Mesh Size** window, enter a **Scale Factor** of **1.0** and **Max Element** of **1.0.** Under **Natural Size**, toggle ON **Enabled.** Next to Enabled, enter a **Size** of **0.125**. Enabling Natural size turns on an algorithm that automatically refines the mesh size where there is small curvature and small gaps in order to accurately resolve the geometry. Leave all other fields as default in the **Global Mesh Size** window as shown in Figure 4.48 and press Apply.

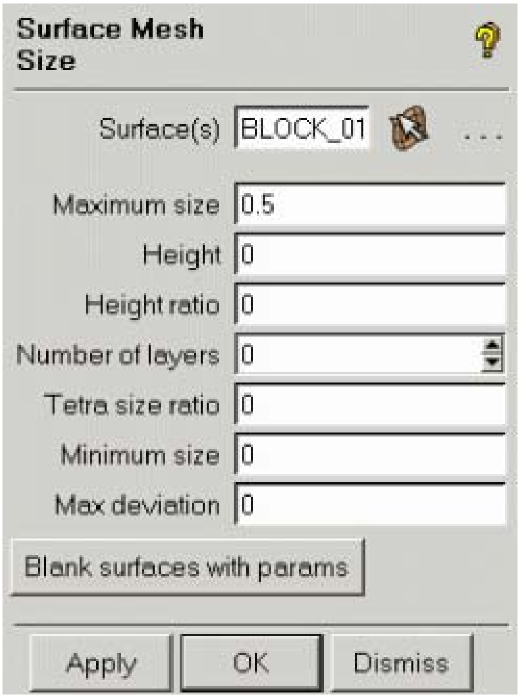
Figure 4.48
Global Mesh Size window



Select the **Mesh** > **Set Surface Mesh Size** button, which brings up the **Surface Mesh Size** window as shown in Figure 4.49.

Click on the surface selection button (Choose an item) and select all the surfaces by pressing "**a**" (ensure that the mouse cursor is in display window). Enter a **Maximum size** of **0.5** as shown in Figure 4.49 and press Apply.

Figure 4.49
Surface Mesh Size window



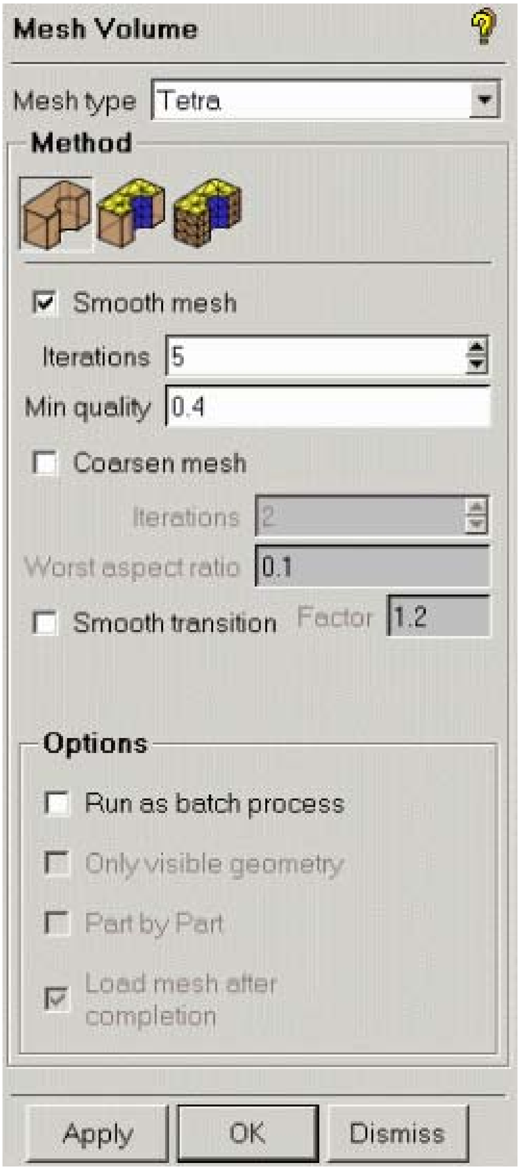
c) Meshing & Internal Wall

Tetra Meshing

Select the **Mesh >Volume Meshing** button. It opens the **Mesh Volume** window shown in Figure 4.50. Ensure that the **Mesh type** is set to **Tetra**.

Select the **From geometry**  button.

Figure 4.50
Mesh Volume

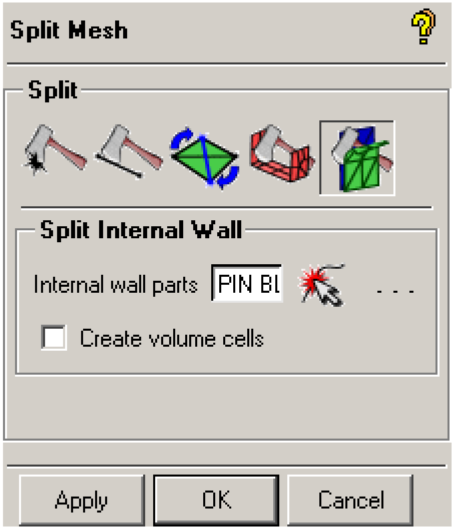


For this tutorial, don't change anything here. Leave the default parameters as they are and press Apply to start meshing.

Creating Internal wall

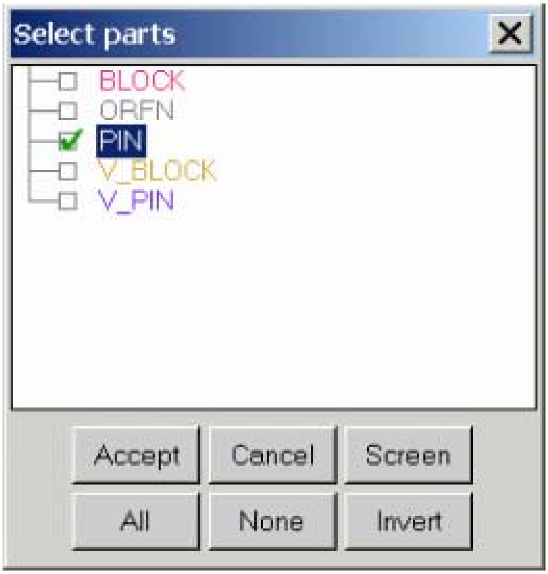
Click on the **Edit Mesh** > **Split Mesh**  > **Split Internal Wall**  button. It opens the **Split Mesh** window shown in Figure 4.51.

Figure 4.51
Split Mesh window



Click on the part selection  button. A window with the current parts in the model will appear. Select the part, **PIN** as shown in Figure 4.52 and press Accept to close the **Select parts** window.

Figure 4.52
Internal Wall Parts Selection window

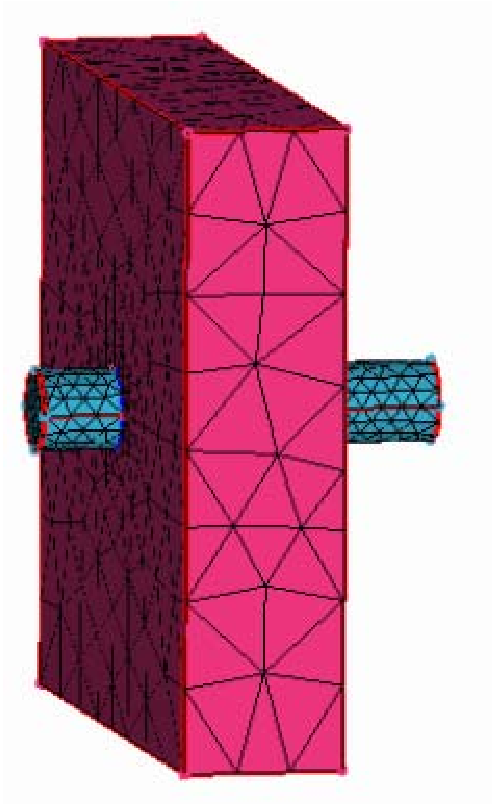


Now, Click Apply to split the internal wall.

A new part will appear in the **Model Tree** called **PIN_BACK**. Turn OFF all the parts except for **PIN_BACK** to see these new surface elements. Make sure to turn all parts back on. This will disconnect the mesh of the BLOCK and the PIN at the internal wall. In most cases, this will be the result when geometries are meshed separately and the meshes loaded together, but here it is easy to just split the internal wall. A contact is then defined where the meshes meet.

To see the mesh in **Solid & Wire**, press the **Solid Simple Display**  from the main menu. Make sure **Surfaces** are off in the Model Tree so you are not looking at surfaces on top of mesh. Now, the mesh should look as shown in Figure 4.53.

Figure 4.53
Mesh in Solid & Wire mode



d) Material and Element Properties

Definition of the Material

Select **Properties**> **Create Material Property** .

Define the **Material Name** as **MAT1**.

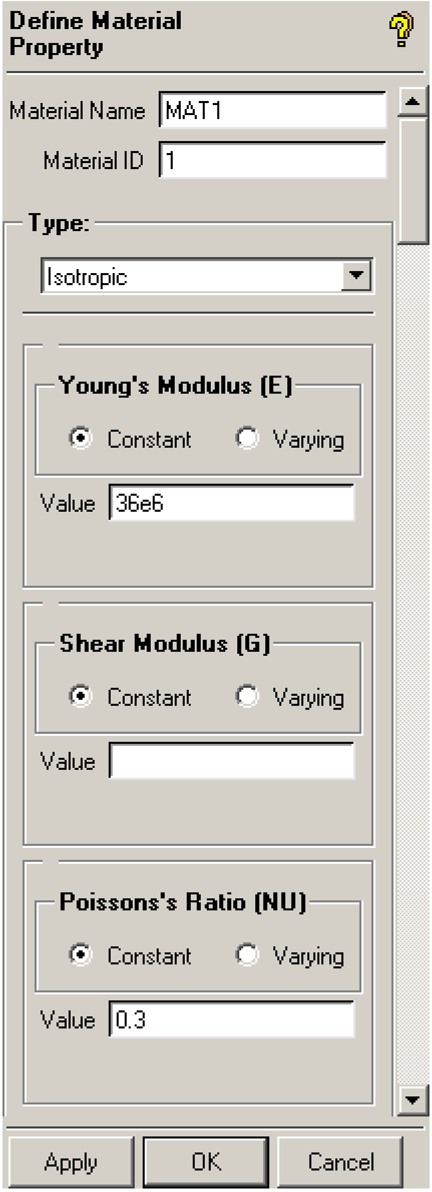
The **Material ID** can be left as **1**,

Select the material **Type** as **Isotropic** (which is the default),

Define **Young's Modulus** as a **Constant 36e6,**

Define **Poisson's Ratio** as a **Constant 0.3**, Leave all other fields as they are. The window should look like Figure 4.54. Then press Apply.

Figure 4.54
Define Material Property window



Element Properties

Select **Properties> Define 3D Element Properties** .

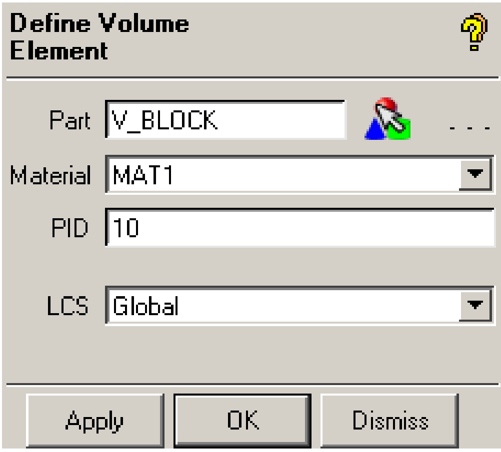
Select the **Part** as **V_BLOCK**.

Select the **Material** as **MAT1.**

Set the PID to **10.**

The **Define Volume Element** window should look like Figure 4.55 when you are finished. Then press Apply.

Figure 4.55 Define Volume Element window



We also need to define the volume elements of the PIN region. These are in the V_PIN part. So select the **Part** as **V_PIN.** Leave the **Material** as **MAT1**, and specify the **PID** as **11**. Then press Apply again.

e) Constraints and Displacement

Relevant Constraints and Displacements still need to be applied on the model. There will be no applied force for the model. The non-zero initial displacement will serve as the applied load. This can be done as follows:

Constraints

Click on the **Constraints>Displacement on Surface** button, which opens the **Create Displacement on Surface** window as presented in Figure 4.56.

Now, use the hot key “**h**” to display geometry in the front view. We will fix all displacements and rotations on the bottom surface of this model. Turn **OFF** all Geometry entities in the Model Tree. This will allow you to select mesh.

Next to **Name**, enter **FIX**.

Select the surface selection icon

for mesh or geometry. Then make sure you are using the “entire” selection method by using the “**p**” hotkey or changing to in the selection window. Box select the surface elements at the bottom as shown in Figure 4.57.

Toggle **ON** all options of X, Y and Z for the Directional displacement. Press Apply. The constraint applied is shown in Figure 4.57.

Switched **OFF** the **Displacements** from the Model Tree after the constraint has been applied.

Figure 4.56 Create Displacement on Surface window

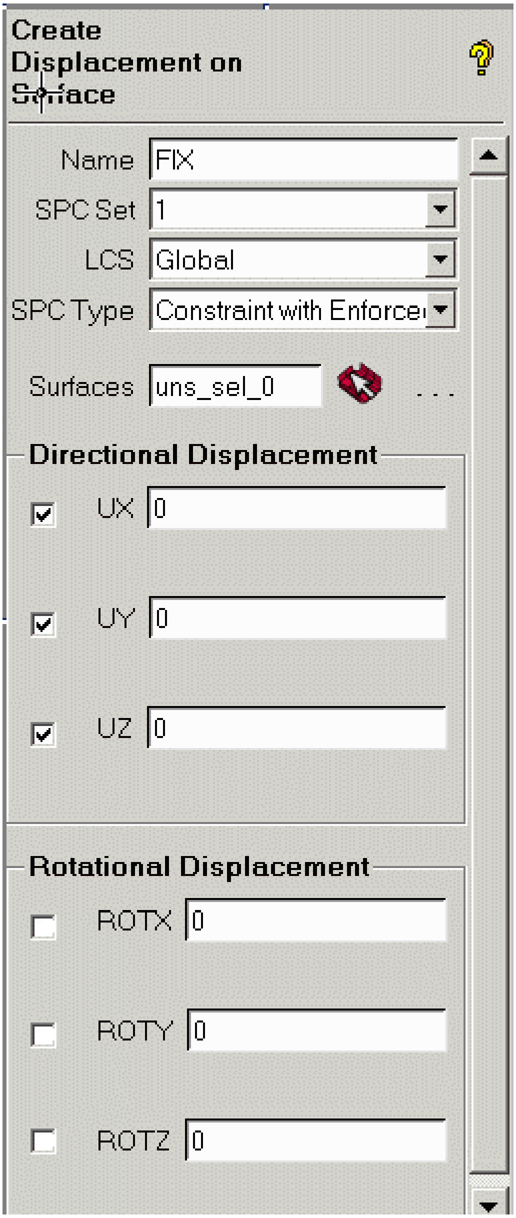
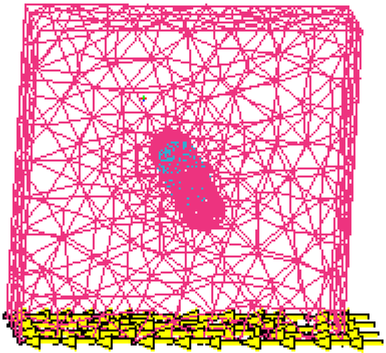


Figure 4.57 Constraint Display



Displacement

Now, in the same window, enter the **Name** as **DISPLACEMENT.**

Set the **SPC Set** to **2**.

Toggle OFF all Displacements and Rotations except **UY**.Enter **–0.2** for **UY**.

Select the surface selection icon  for mesh or geometry, and box select all the surface elements at the top as shown in Figure 4.58.

The window should now look like Figure 4.59.

Press Apply.

Switched **OFF Displacements** in the Model Tree.

Figure 4.59 Create Displacement on Surface window

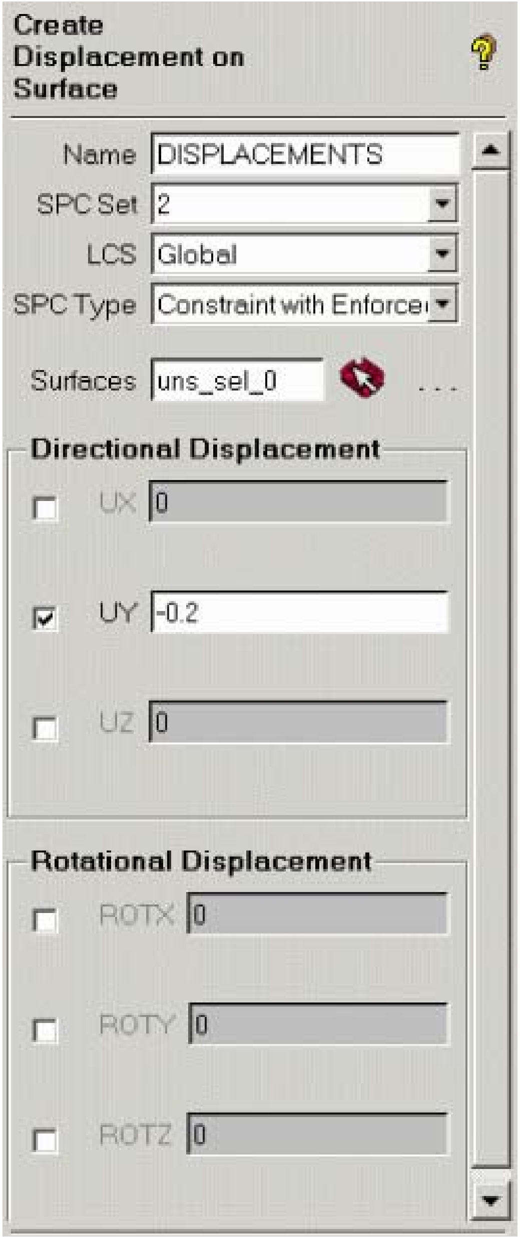
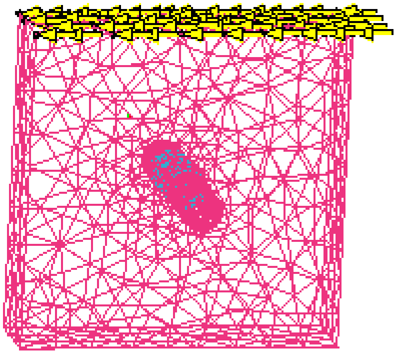


Figure 4.60 Displacement Display



Contact

Click on **Constraints>Define Contact**  **>Manual Definition**  (the second one). The window as presented in Figure 4.61 will display.

Enter the **Name** as **CONTACT_ALL**.

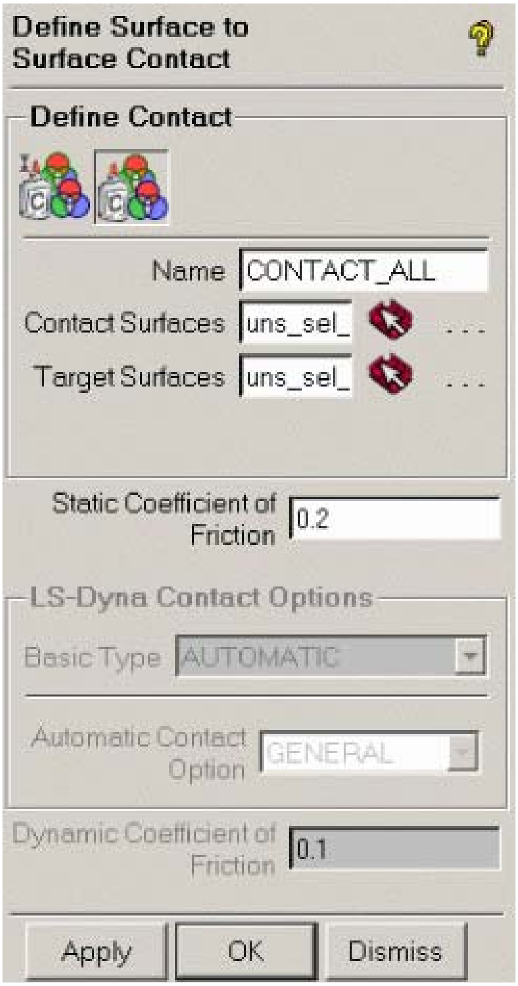


Figure 4.61 Define Surface to Surface Contact window

Select the surface mesh selection icon for **Contact surfaces**, and then select the part selection icon from the popup menu. Select the part, **PIN_BACK** from the list of parts.

Then select the surface mesh selection icon for **Target surfaces**, and select the part selection icon from the popup menu. Select the part, **PIN** from the list of parts

Enter a value of 0.2 for the **Static Coefficient of Friction**, and press Apply.

f) Solver Setup

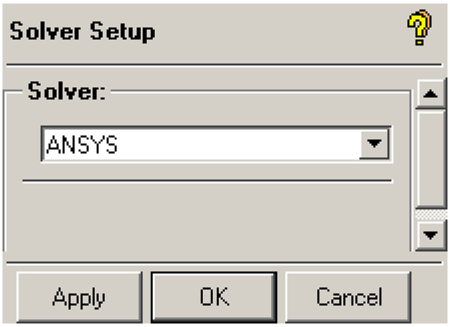
On this model, analysis is to be performed in Ansys, so parameters and variables should be defined accordingly. This can be done as follows:

Setup Ansys Run

First, the user should select the appropriate solver before proceeding further.

Select **Settings** > **Solver** from the main menu and select **Ansys** from the dropdown arrow. Press Apply. Selecting the solver is shown in Figure 4.62.

Figure 4.62
Solver selection



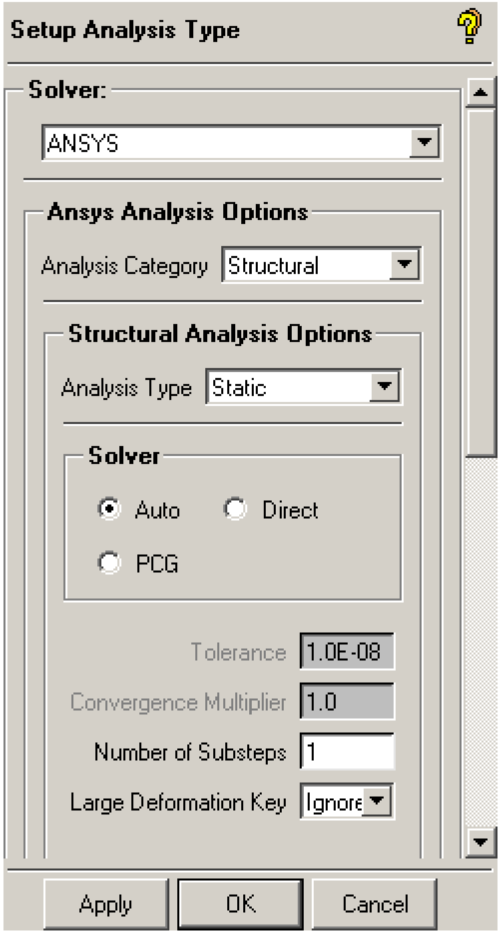
Click on the **Solve Options> Setup Analysis type**  button. This will bring up the **Setup Analysis Type** window as shown Figure 4.63.

The solver should read as ANSYS.

Select the **Analysis Type** as **Static.**

Keep all other options as default, and press Apply to complete the setup.

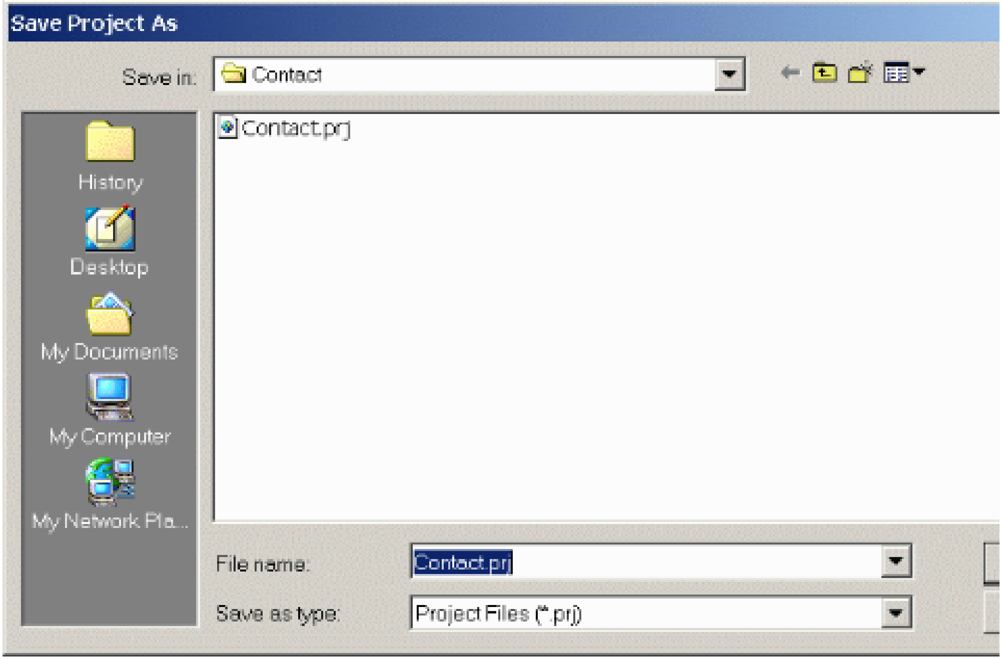
Figure 4.63
Setup Analysis Type window



Save Project

Select **File > Save Project As**, and in the new window press the icon to create a new folder. Name this folder **Contact** and enter into it. Then enter the file name, **Contact,** as shown in Figure 4.64.

Figure 4.64 Save Project As window

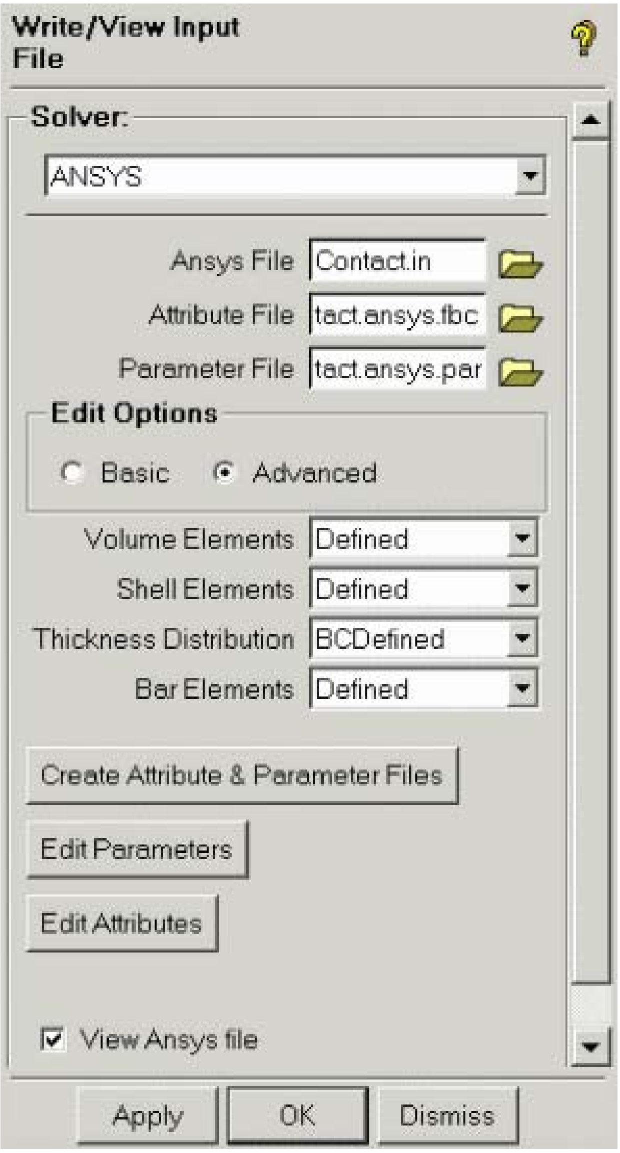


Write Ansys Input File

Click on the **Solve Options> Write/View Input File**  button.

Toggle ON the **Advanced** option under **Edit Options**, and click on **Create Attribute & Parameter Files**.

Figure 4.65 Ansys Input File window



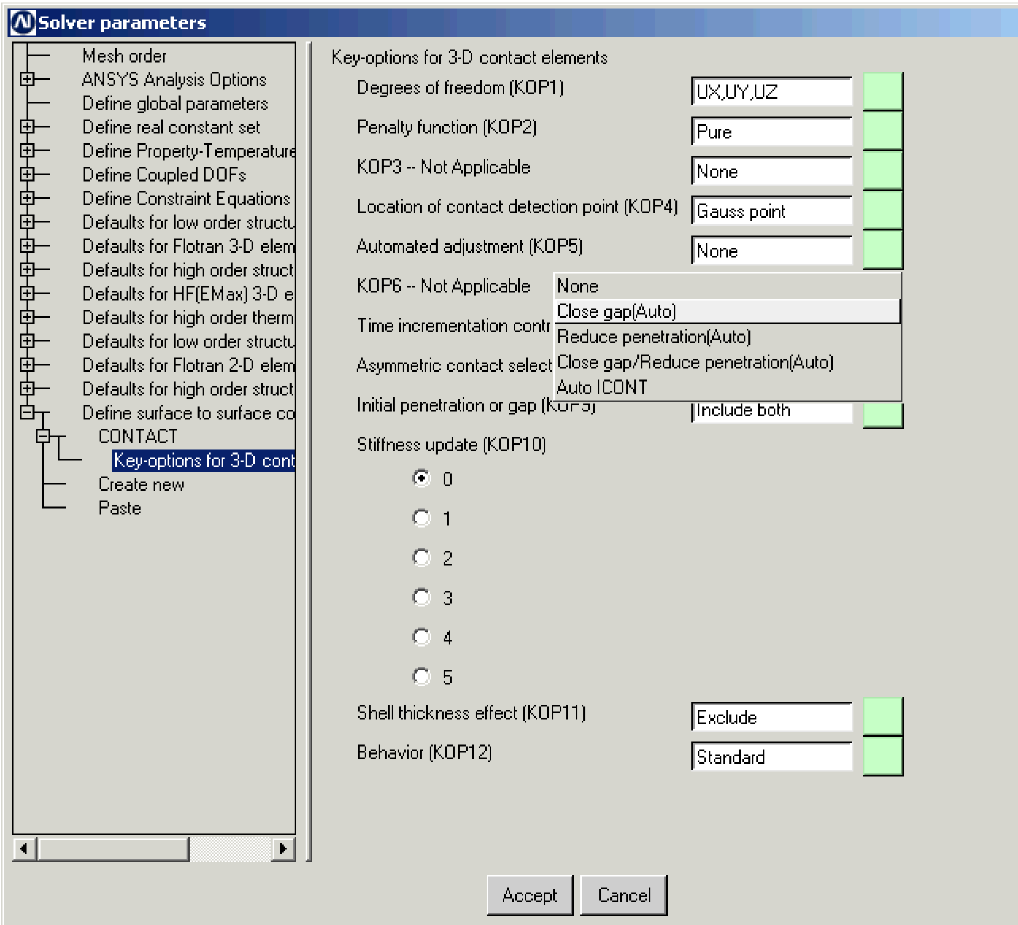
Click on the Edit Parameters button, which will open the Solver Parameters window.

Expand under Define surface-to-surface contact configuration, then expand under the part name CONTACT, and select on the words, Key-options for 3-D contact elements.

Change the option for Automated Adjustment (KOP5) to Close gap (Auto) as shown in Figure 4.66.

Press Accept to save these changes in the parameter file (.par file) and close the Solver Parameters window.

Figure 4.66 Solver Parameters window



Also, switch ON the View Ansys file option in the Write/View Input File window as shown in Figure 4.65 and press Apply.

The Ansys input data file will come up in the default text editor. This can be edited and saved to the same file, if desired. Since there is no need to do any editing for this example, just close the editor.

g) Solution and Results

Linear Static analysis is to be performed on this model and the results will be visualized in ICEMCFD's post processor.

Solving the problem

Click on the **Solve Options> Submit Solver Run** button, which should display the **Run Solver** window as shown in Figure 4.67.

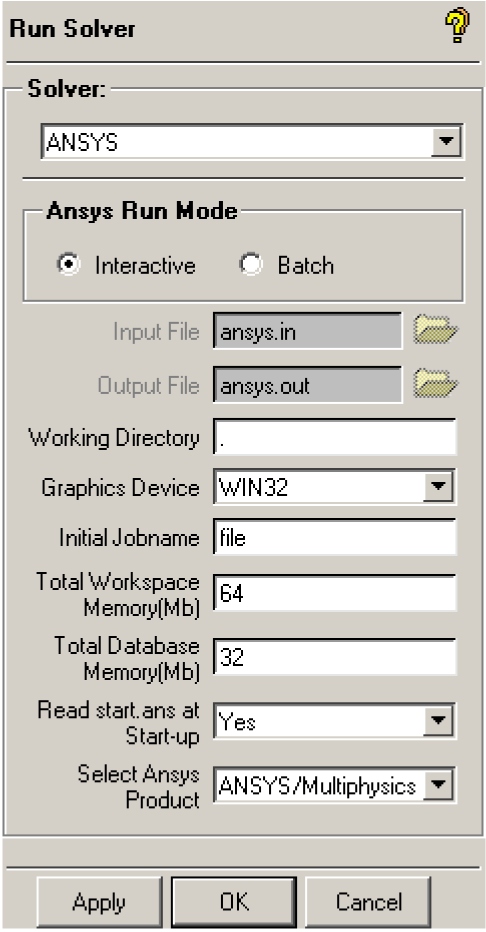
Leave the **Ansys Run Mode** set to **Interactive**.

Under **Select Ansys Product**, select your Ansys product.

The ANSYS_EXEC_PATH environment variable may have to be set to the full path to the Ansys executable for ICEMCFD to be able to run Ansys.

Press Apply to start the Ansys solver in Interactive mode.

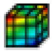
Figure 4.67
Run Solver window



After the Ansys Interactive window has come up, load the input file by going to **File** > **Read Input From**. Select **file.in** where "file" is the **Initial Jobname** specified in the previous window. Check that the solution converges in the convergence graph. This happens when the two lines cross.

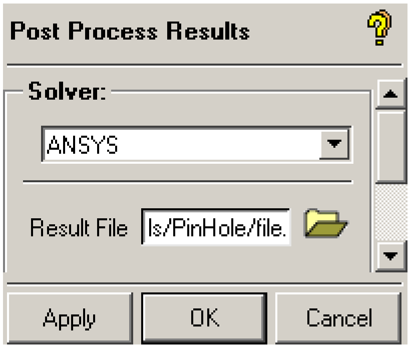
The user can work in Ansys after this, if desired, or exit out of Ansys and Post Process the results in ICEMCFD.

Post-Processing of Results

Click on the **Solve Options > Post Process Results**  button, which opens the **Post Process Results** window given in Figure 4.68.

Select the folder button to browse for the file, **file.rst,** where "file" is the **Initial Jobname** specified earlier. Press Apply to launch the Visual3p Post processor with the Ansys result file.

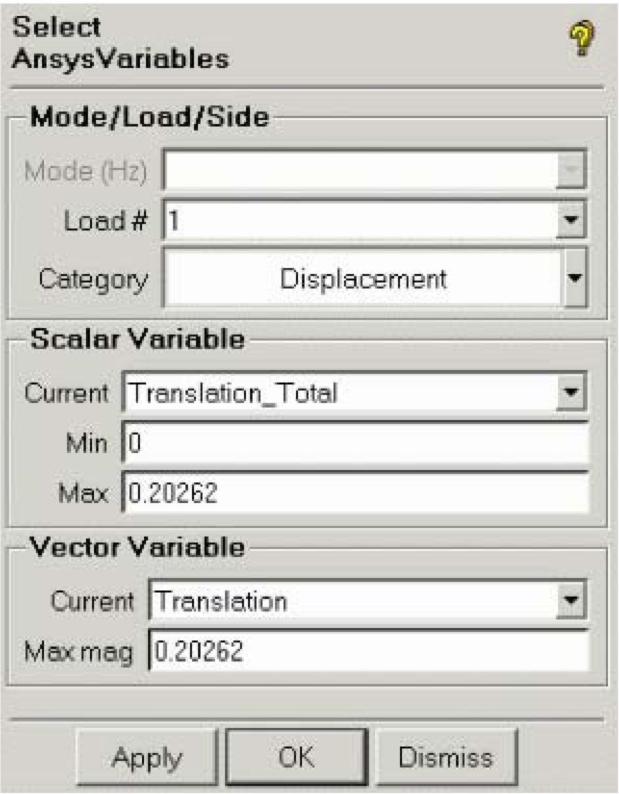
Figure 4.68
Post Process window



Click on  **Variables** from the **Post-processing** Tab menu bar.

To display the Total Translation Displacement, select the **Load#** as **1** and **Category** as **Displacement** in the **Select AnsysVariables** window as shown in Figure 4.69. This should be the default.

Figure 4.69 Ansys Variables window



Click on **Control All Animations** from the **Post-processing** Tab menu bar. The window shown in Figure 4.71 will appear.

Select **Animate** to see the deformation. The deformed shape is shown in Figure 4.70.

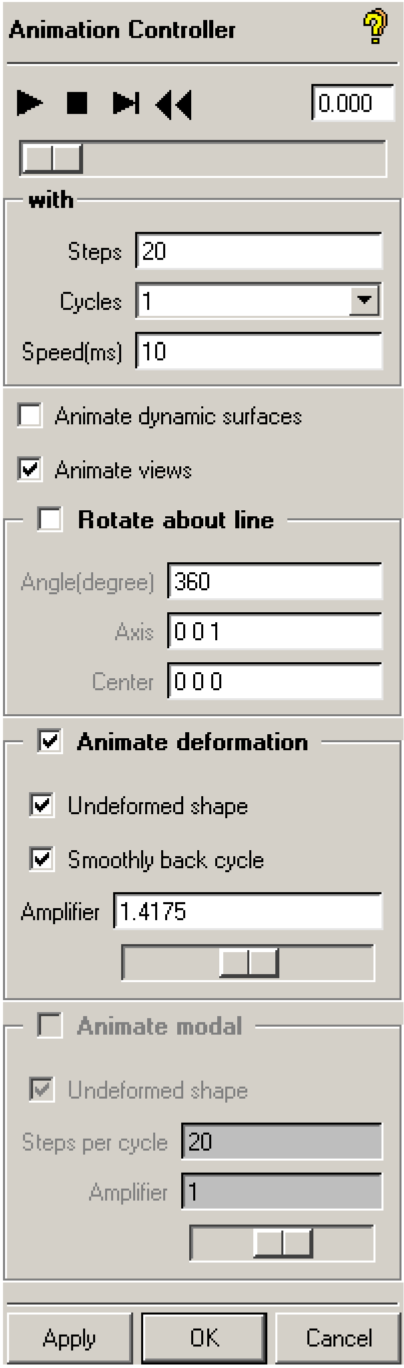
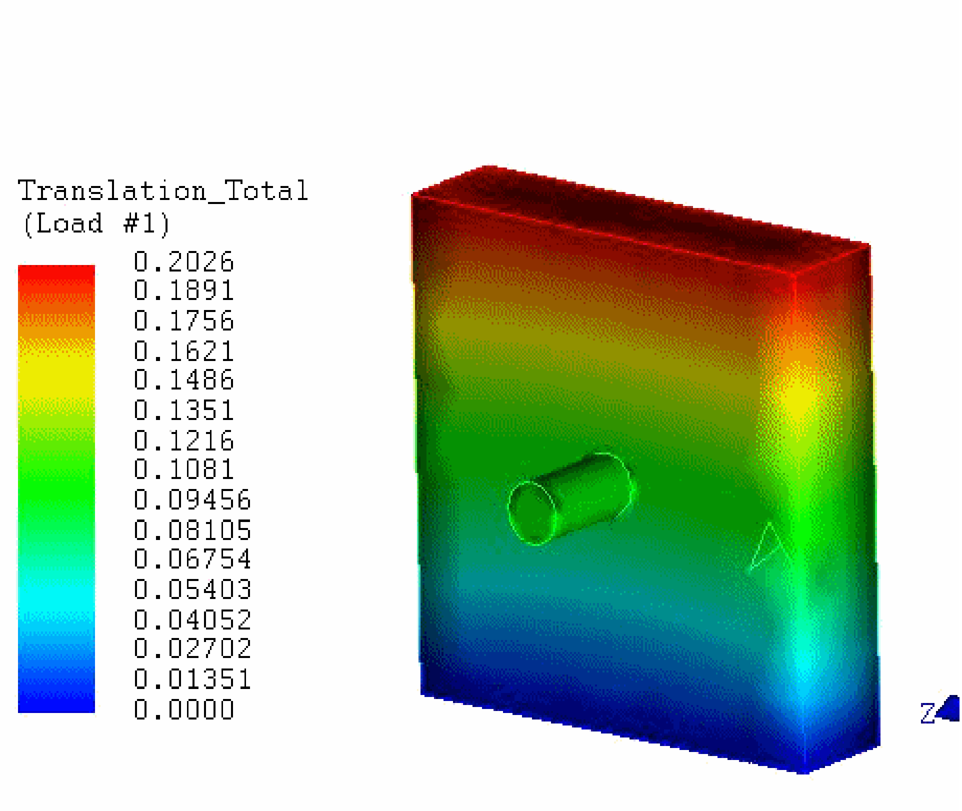


Figure 4.71 Animation Setup and Controller window

Figure 4.72 Animated model of Total Translation

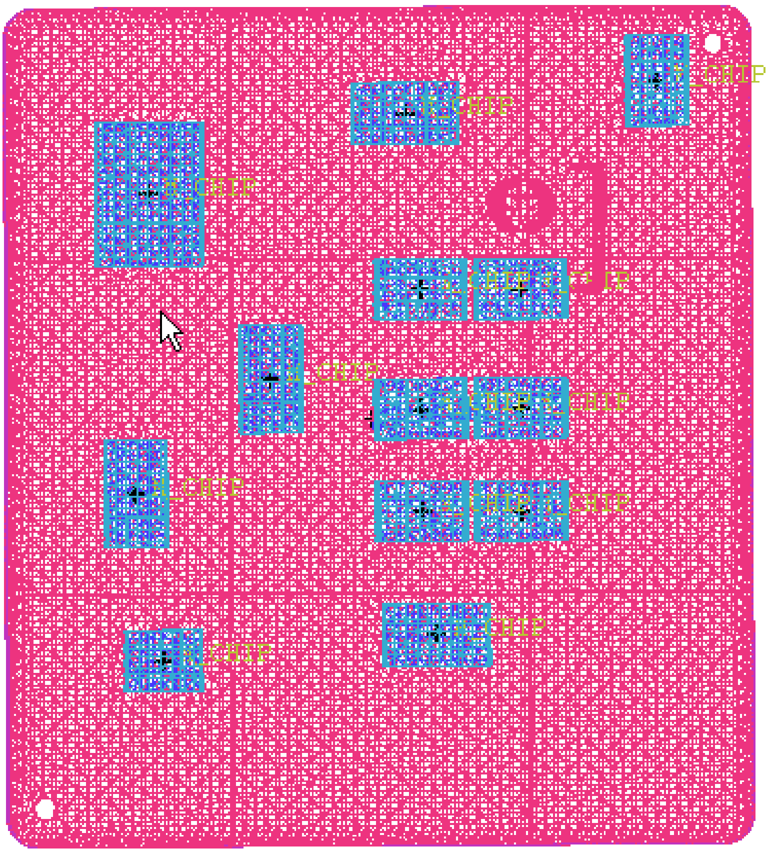


Finally, select **File** > **Results** > **Close Result** to quit the post processor

4.1.4: PCB-Thermal Analysis

Overview

In this tutorial, it is shown that how easy to create a mesh in the PCB model and then do thermal analysis in Ansys using AI*Environment.



a) Summary of steps

Starting the project

Repairing the geometry

Assigning the mesh sizes

Generating the tetrahedral mesh

Smoothing and checking the mesh

Defining the material properties

Setting the solver parameters

Writing the input file

Solution and results

Saving the project

b) Starting the project

Launch the AI*Environment from UNIX or DOS window. Then File > Change working directory, $ICEM_CAN/../docu/FEAHelp/AI_Tutorial_Files > PCB: Thermal analysis project. Load its tetin file geometry.tin.

c) Repairing the geometry

For repairing geometry, select Geometry > Repair geometry  > Build topology. 

Run the build topology with the default parameters

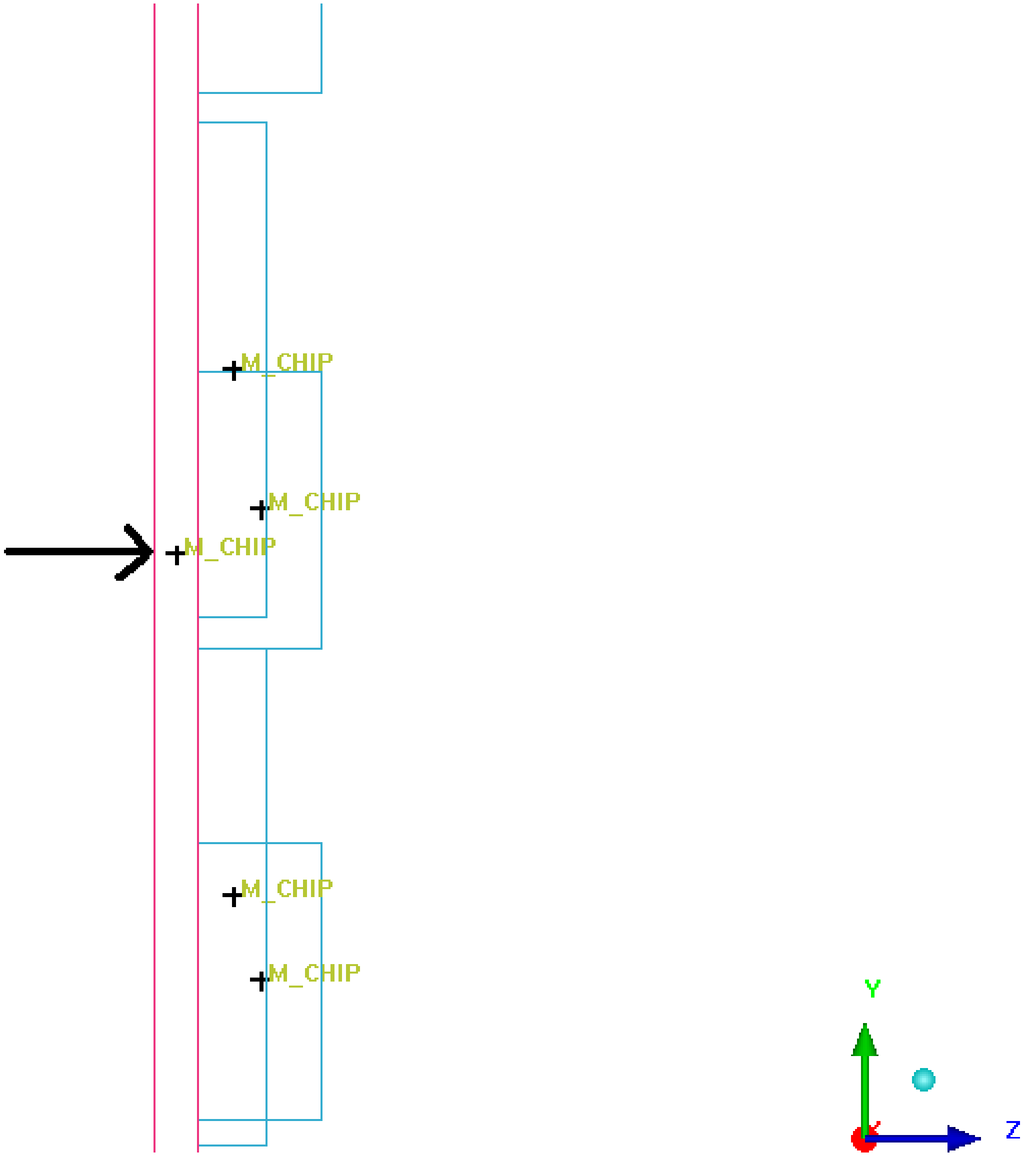
d) Assigning the mesh sizes

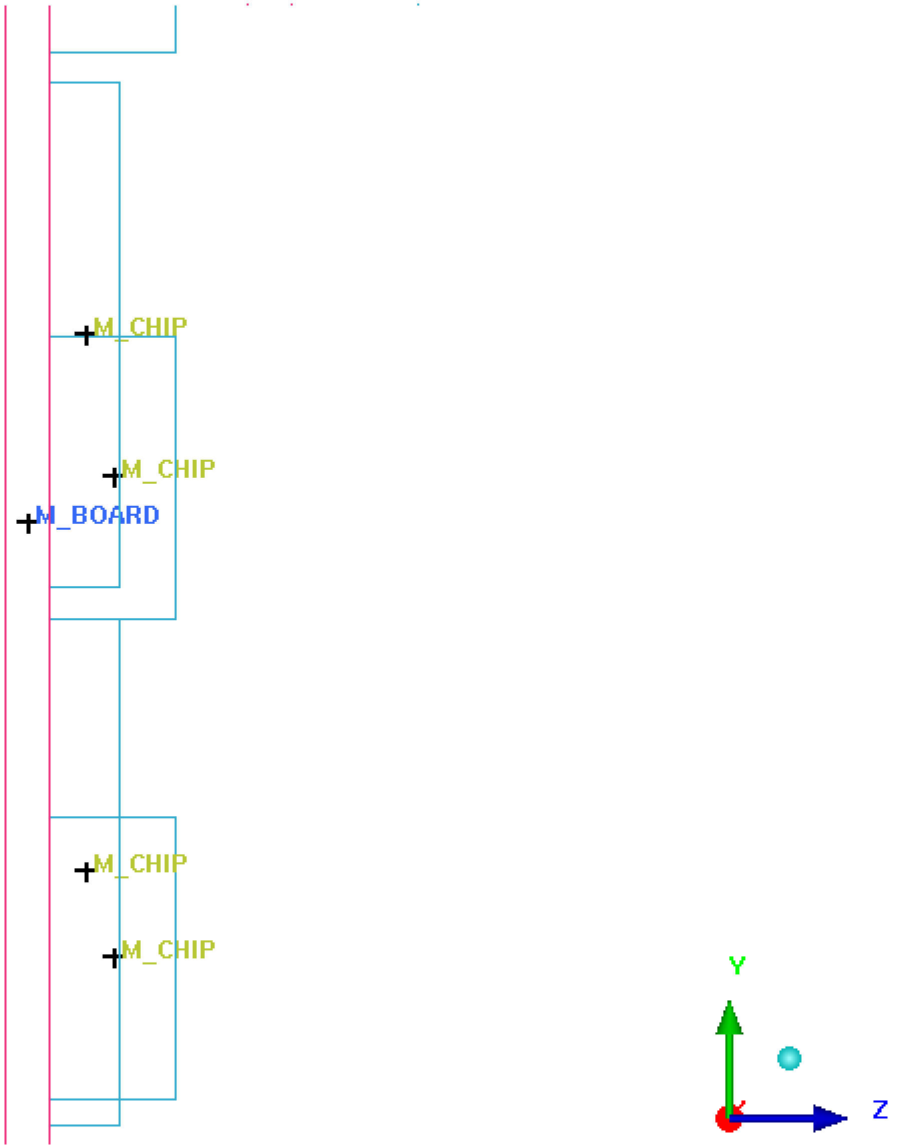
Creating bodies

Before defining the mesh sizes, we have to define the material point. For defining the material point, select Geometry > Create body.  This will bring a create body window. In create body window, Assign part as **M_CHIP, select** by topology  and method as **Entire model**. Press Apply. To see the bodies in the geometry on screen, please make bodies visible from the model tree.

Now, Move the material, **M_CHIP** at the center of the surface.
To move material "M_CHIP" to Material "M_BOARD", right click on Parts > Create Part >Create part by selection from the Display Tree widget.
This will bring the create part window, in create part window, enter part as **M_BOARD**, Click on Create Part by Selection. Click on select entities option in Entities to select **M_CHIP** and press Apply.
Please refer Figure 4.73 for details.

Figure 4.73
Modify BODY


M_CHIP
M_CHIP
M_CHIP
M_CHIP
M_CHIP
Y
Z


M_CHIP
M_CHIP
M_BOARD
M_CHIP
M_CHIP
Y
Z

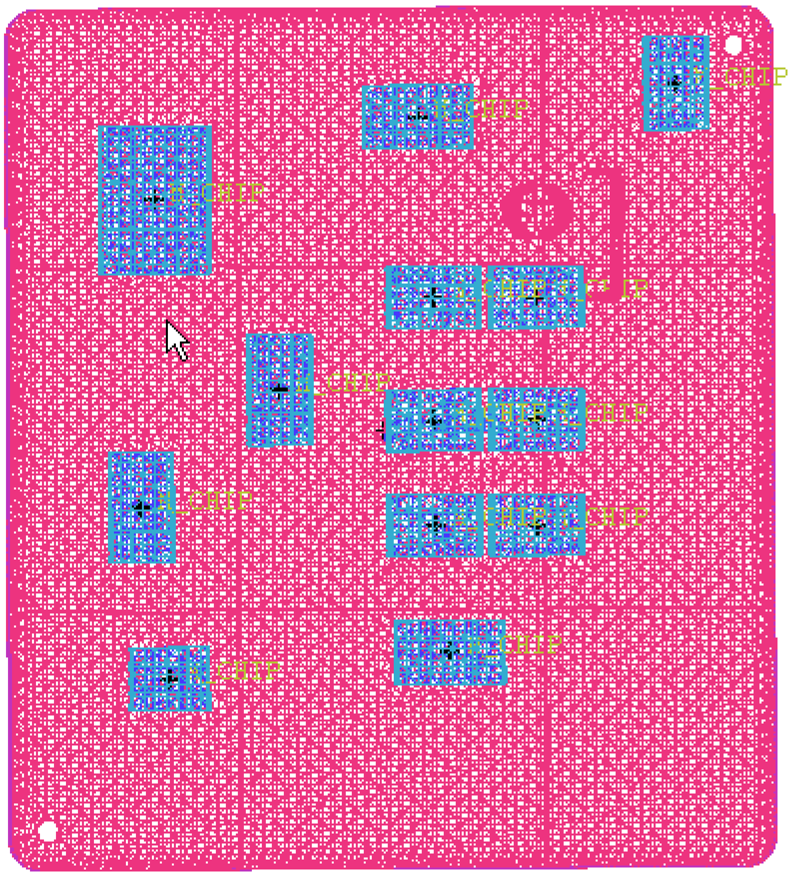
e) Assigning the mesh sizes

To define the surface mesh size, select Mesh > Set surface mesh size  this will bring the surface mesh size window. Enter the **Maximum element size** as **2** for the all-surface parts
Turn on only CHIP_SURF and INTERFACE and turn off all the other parts. Set surface mesh size  this will bring the surface mesh size window. Enter the **Maximum element size** of **1** and select only visible parts by using option ‘v’
After assigning mesh sizes save the changes with **File > Save project.**

f) Generating the tetrahedral mesh

To generate the tetrahedral mesh, Select Mesh > Mesh Tet  > From Geometry.  In Mesh with tetrahedral window, press Apply with the default parameters. This will generate the tetrahedral mesh on the geometry as shown in Figure 4.74.

Figure 4.74 Completed tetra mesh



g) Smoothing and checking the mesh

To smoothen the mesh, Select Edit mesh > Smooth mesh globally. This will invoke a smooth mesh globally window. Press Apply with the default parameters. After smoothing the mesh, just check the mesh for any errors and possible problems with Edit mesh > Check mesh. Pressing Apply in the check mesh window will check for the error and possible problems in the generated mesh. If there are no any errors then we can proceed for defining the material properties.

h) Defining the material properties

Now after generating the tetrahedral mesh, we have to define the material properties.

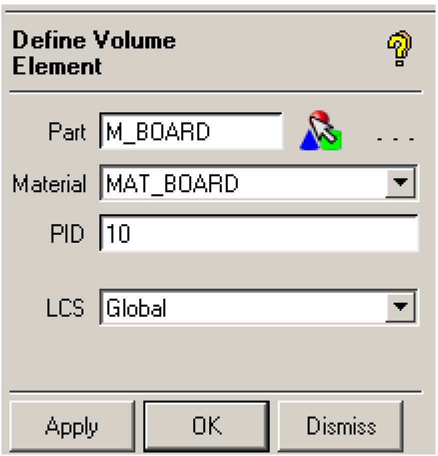
Select Properties > Create material property. This will bring up a define material property window. Enter **Material name** as **MAT_BOARD**, material ID as **1**, Type as **Isotropic**, Value for **Young modulus (E)** as **15000**, **Poisson's ratio (nu) 0.28**, **Density (RHO)** as **1.4e-9**, **Thermal Expansion coefficient** as **19e-6** and **Ref. Temperature** as **298**. Press Apply

After defining material property for board material, **MAT_BOARD**, we have to define material property for **MAT_CHIP**.

Enter Material name as **MAT_CHIP**, Material ID as 1, Type as **Isotropic**, **Young modulus** as **70000, Poission's ratio** as **0.17**, **Density as 2.2e-9**, **Thermal Expansion Coefficient** as **10e-6** and **Reference Temperature** as **298**. Press Apply.

Now we will define the 3D element properties. Select Properties>Define 3d element properties . This will bring Define volume element window. Enter parameter as shown in Figure.

Figure 4.75
Define volume element window



Now after defining volume element properties for **M_BOARD**, we have to define properties to the **M_CHIP** also. So in the same window, enter Part as **M_CHIP**, Material as **MAT_BOARD**, PID as 11 and with default option press Apply**.**

i) Setting the solver parameters

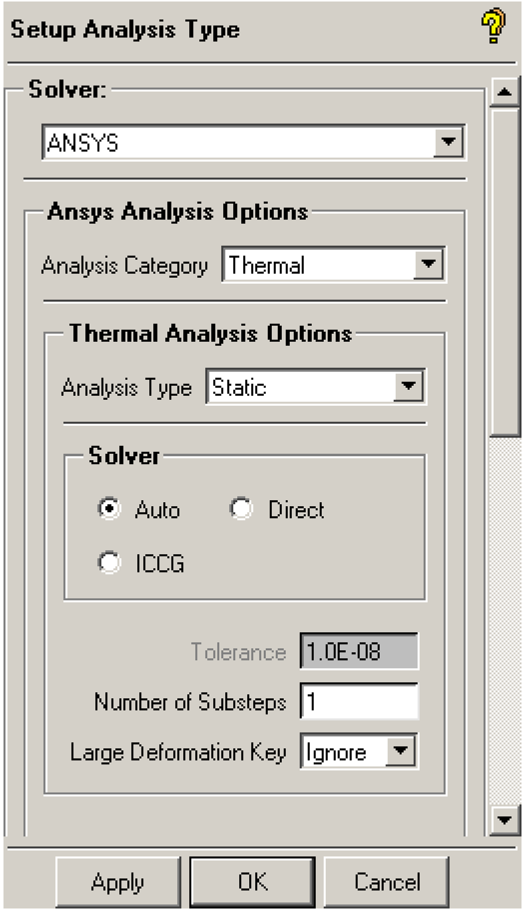
For solver settings, select Settings > Solver, then solver set up window will pop up. Select solver as ANSYS and press Apply.

Select Solve options > Setup Solver Parameters, select ANSYS as solver and press Apply.

Then select Solve options > Setup Analysis Type. In Setup Analysis Type window,

Enter parameters as shown in Figure 4.76.

Figure 4.76
Setup Analysis Type window



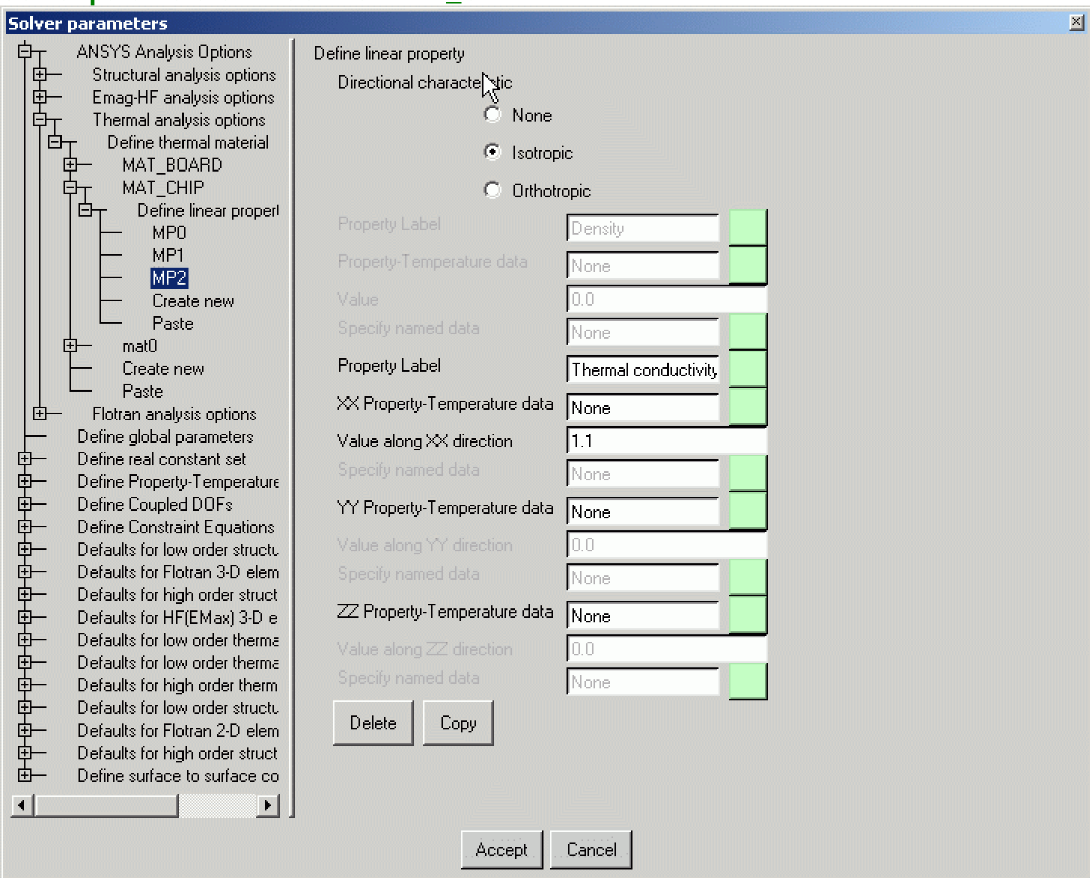
After entering the parameters, press Apply.

j) Writing the input file

Now we have to write the input file for ANSYS solver. Select Solve options >  write/View input file. In **Edit options**, press **Advanced.** Now click on **Create Attribute and Parameter Files.**

Now click on **Edit parameters**, which will invoke a solver parameters window.
Go to ANSYS Analysis options > Thermal analysis options > Define thermal material option.
Select **MAT_CHIP**> Define linear property > Create new. This will invoke Define linear property window. In this window, enter directional characteristic as **Isotropic**, value along xx direction as **1.1**, then press copy.

Figure 4.77
Solver parameters window for MAT_CHIP



Do the same operation to define the thermal conductivity to the **MAT_BOARD** material.
Then click on Accept in solver parameters window.
Now press **Edit attributes** which will invoke a Boundary Conditions window.
Please follow the images to define the boundary condition for **M_CHIP** and **BOARD_SURF** part.
Click on Volumes > M_CHIP > Create New option to select the boundary condition type. Select the **Nodal Body Loads** as shown in Figure 4.78.

Figure 4.78
Boundary condition selection window for MAT_CHIP

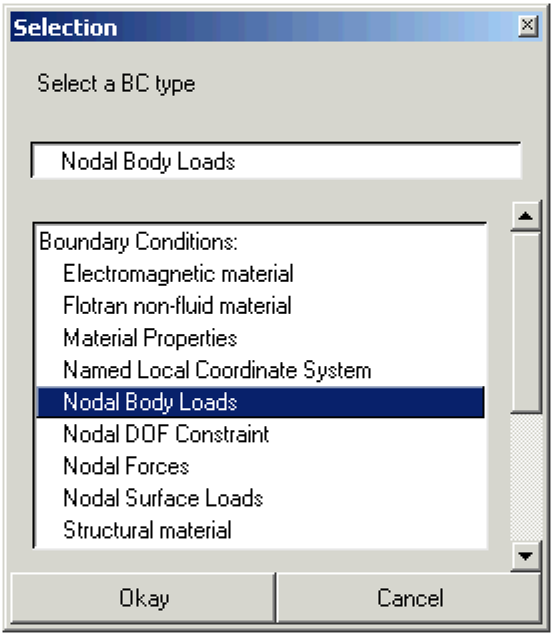
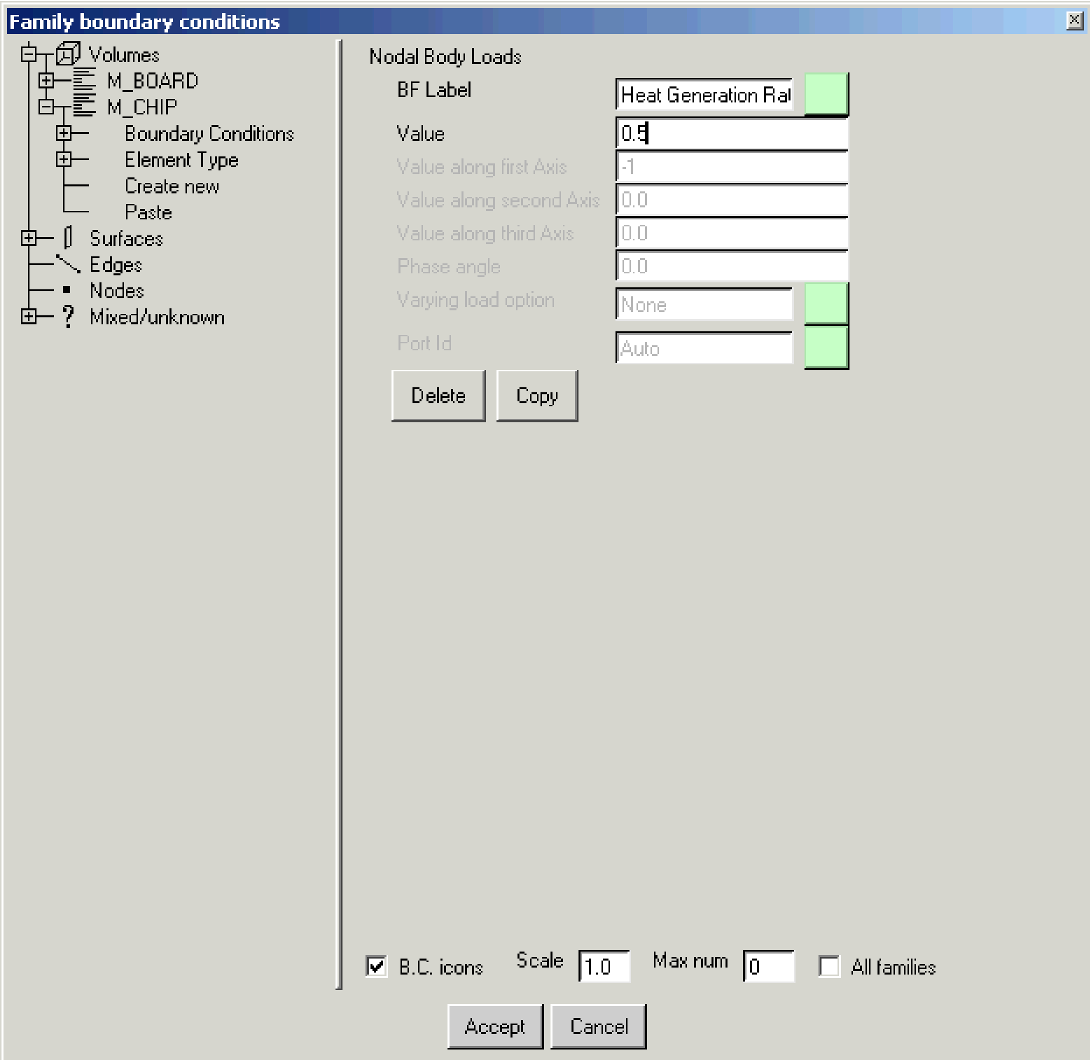
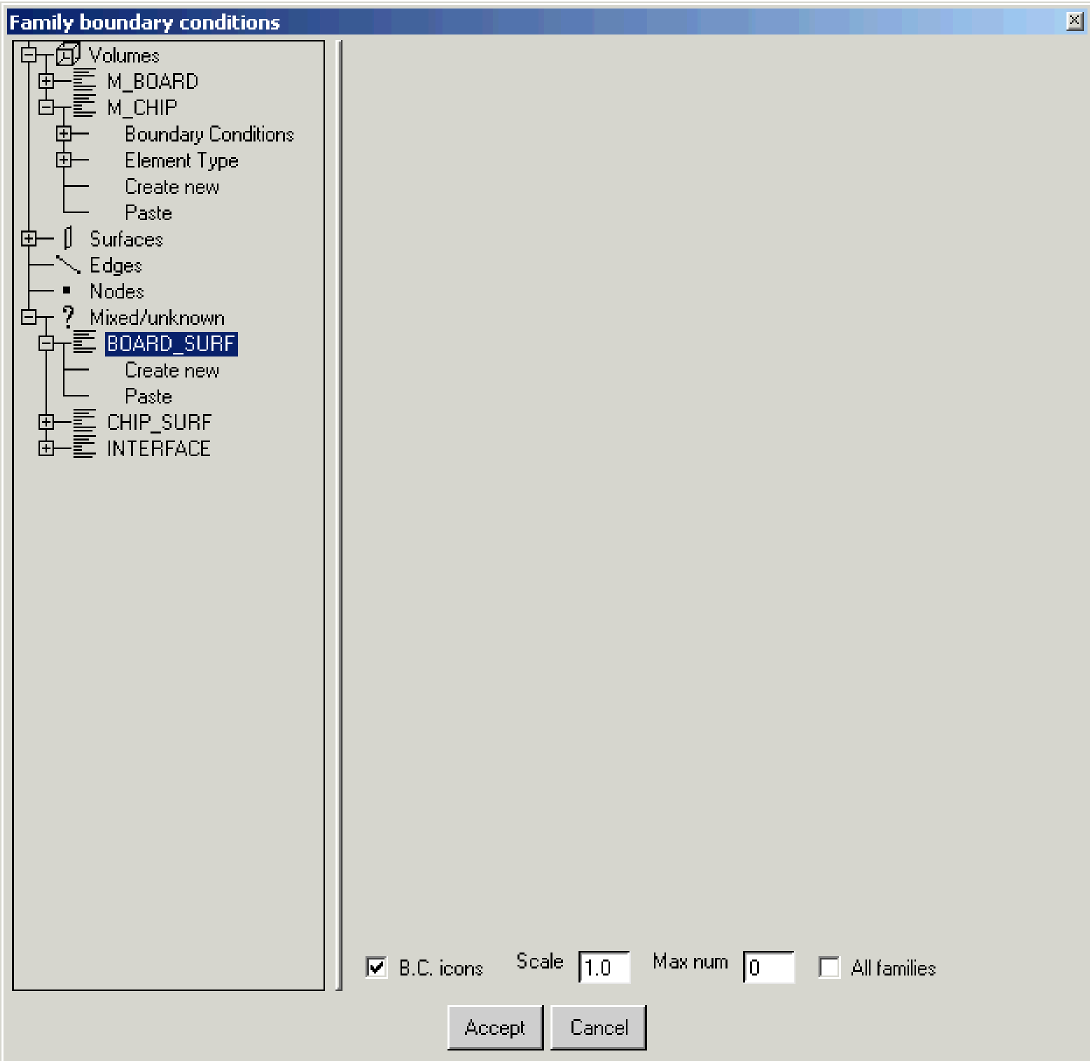


Figure 4.79
Family Boundary Condition window for M_CHIP



Click Copy.

Figure 4.80
Family Boundary Condition window for BOARD_SURF



In the same window, now got to **Mixed/Unknown** > **BOARD_SURF** > **Create new** and select **Nodal Surface Loads** type from the list.

Figure 4.81
BC selection window for BOARD_SURF

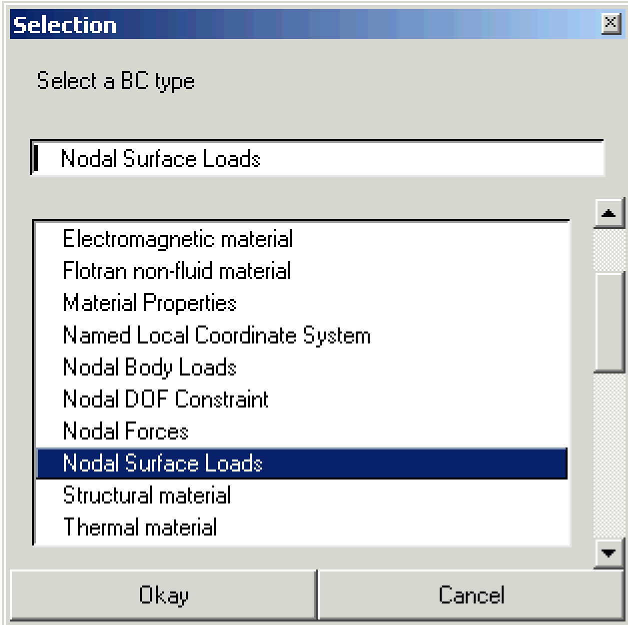
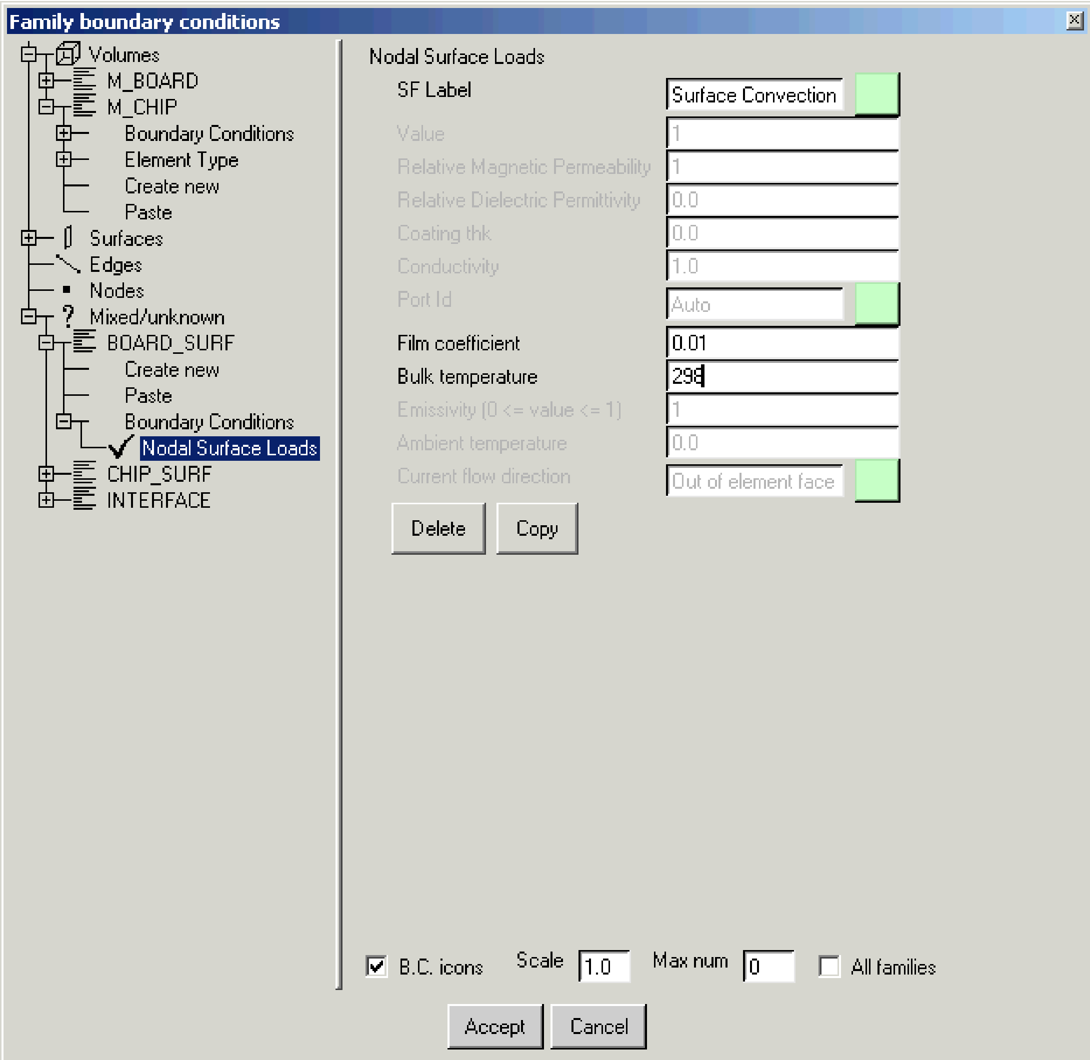


Figure 4.82
Bulk Temperature defining for BOARD_SURF



Press **Accept** in Edit attributes window.
Now in the Write/view input file window, select Yes for Ignore BAR elements, keep View Ansys file ON and other option as default. Press Apply**.**

k) Solution and Results

Thermal analysis is to be performed on this model and the results should be visualized in a post processor.

Solving the problem

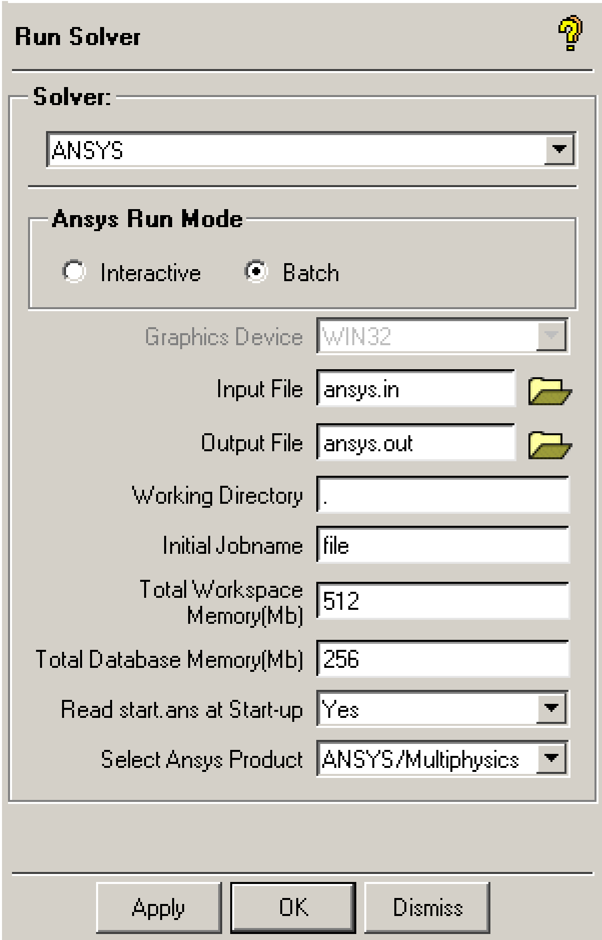
Click on  (Submit Solver Run) icon from the Solve Options Tab Menubar to start the Ansys as shown in Figure 4.83.

Press Apply in Run Solver window.

Verify Working Directory as well as Select Ansys Products field. Ansys/Multiphysics User can change both of these fields, if he/she is interested.

Press Apply to start Ansys solver in Interactive mode.

Figure 4.83
Run Solver window



As it launches the Ansys Interactive window, load the ansys.in file in Ansys for the analysis through File > Read Input From option. User can check out the convergence graph in the Ansys window during the process.

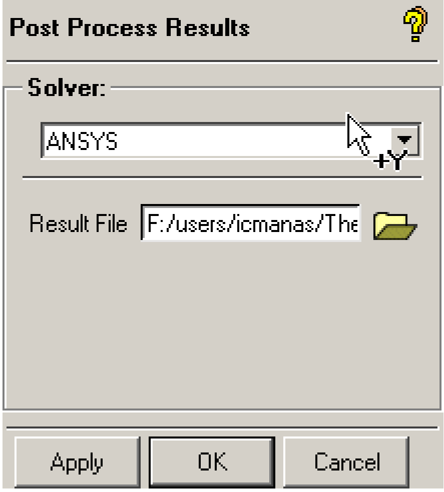
After solution is done, user can do post processing in Visual 3p.

Post-Processing of Results

Click on  (Post Process Result) icon from the Solve Options Tab Menubar, which opens Post Process Results window given in Figure 4.84.

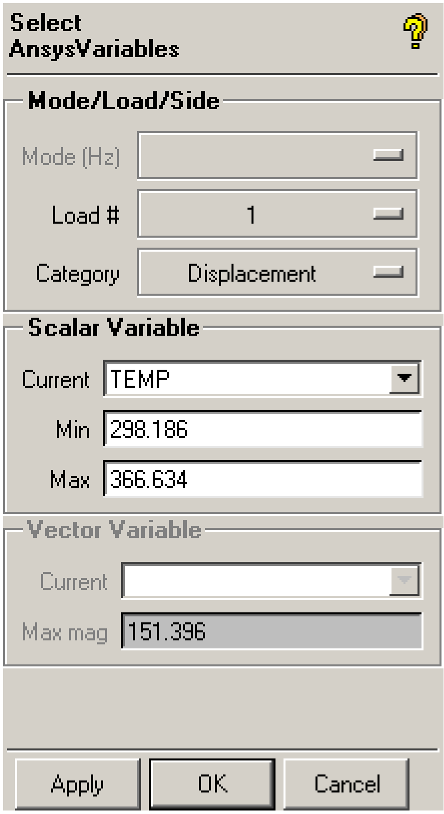
Supply Ansys Result file **file.rth** in this window and press Apply to launch Visual3p Post processor with Ansys result file.

Figure 4.84 Post Process Results window



Select **variables**  from **Post processing** tab menu bar, to display the Total Translation Displacement, select Load as **1** and Category as **Displacement** in Ansys Variables window as shown in Figure 4.85

Figure 4.85 Select Ansys Variables window



Select Animate, which pops up the Setup Animations window as shown in Figure 4.86.

Now press **Animate** to view the deformation. The deformation is shown in Figure 4.87

Figure 4.86 Animation Setup and Controller window

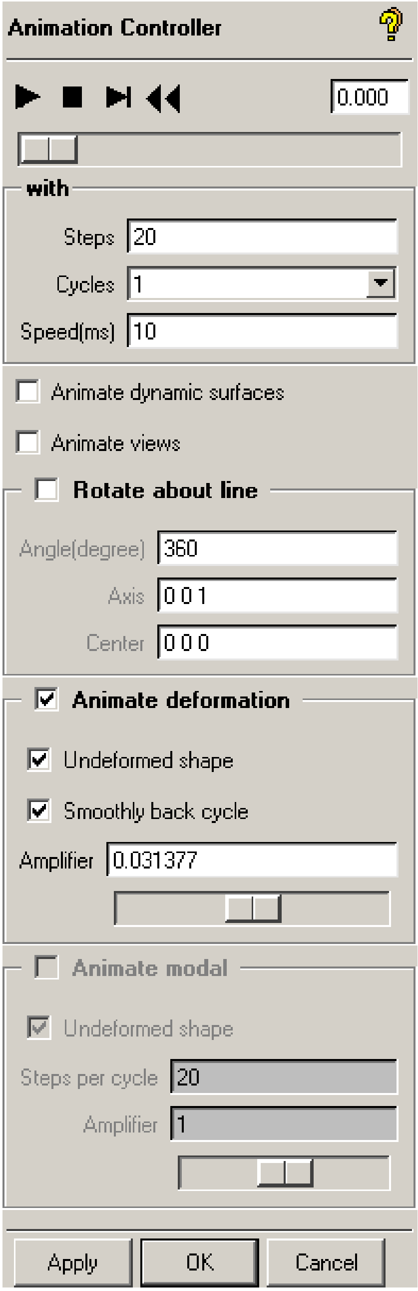
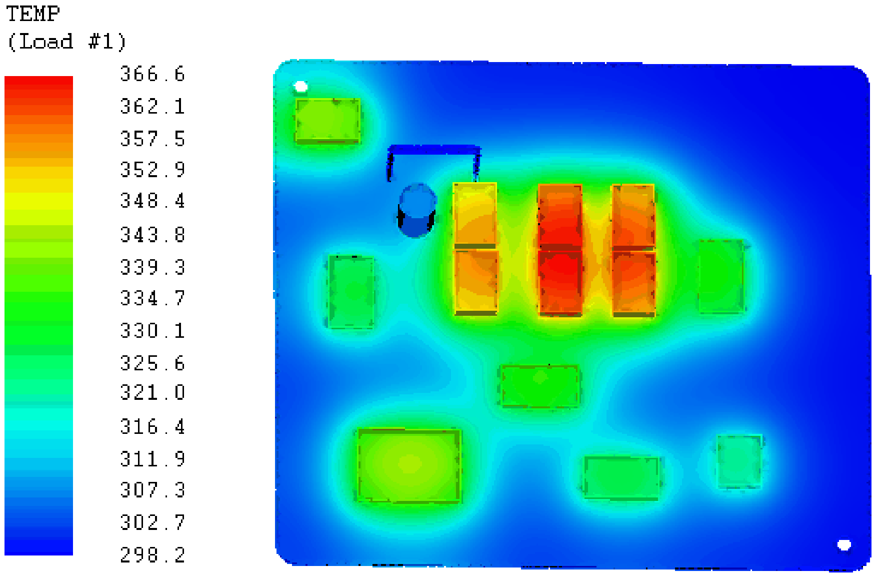


Figure 4.87 Animated model of Total Translation



Finally select **Exit** to quit the post processor.

l) Saving the project

In the save the project with File > Save Project and close with File > Close Project.

4.2: LS-Dyna Tutorial

4.2.1: Frame: Quasi-Static Analysis

The main objective of this tutorial is to demonstrate legacy conversion from a Nastran model to an LS-Dyna model. It highlights the ease of use of **AI*Environment** in translating a model from one solver to another by one simple command. A Nastran linear static analysis data file is provided as input and converted to LS-Dyna. Material properties for the shell elements are converted to nonlinear by using LS-Dyna material type **24** ***MAT_PIECEWISE_LINEAR_PLASTICITY.** The stress-strain curve for steel (mild steel **1010** grade) is used for this purpose. The frame is constrained at both ends and a quasi-static load is applied to the middle bracket. The Frame model used is shown in Figure 4.88.

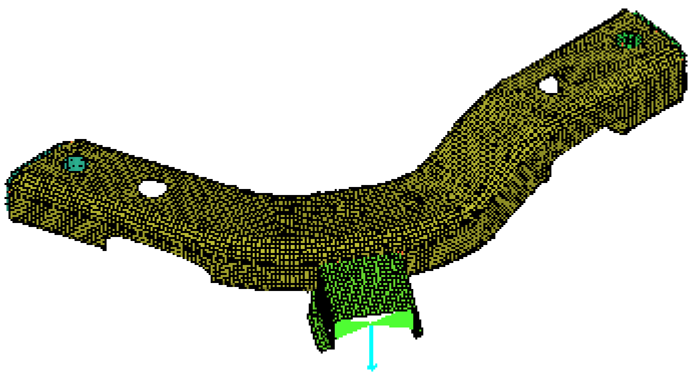


Figure 4.88
Frame model

a) Summary of Steps

Data Editing

Launch AI*Environment and import an existing Nastran data file

Define Contact

Save Project

Solver Setup

Setup LS-Dyna Run

Write LS-Dyna Input File

Solution and Results

Solving the problem

Visualization of Results

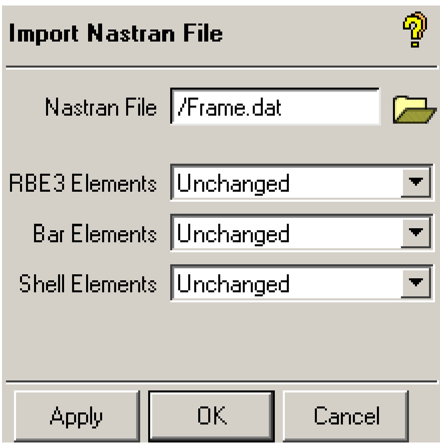
b) Data Editing

Launch AI*Environment

Start ANSYS ICEMCFD and File > Change Working Dir… to $ICEM_ACN/../docu/FEAHelp/AI_Tutorial_Files.

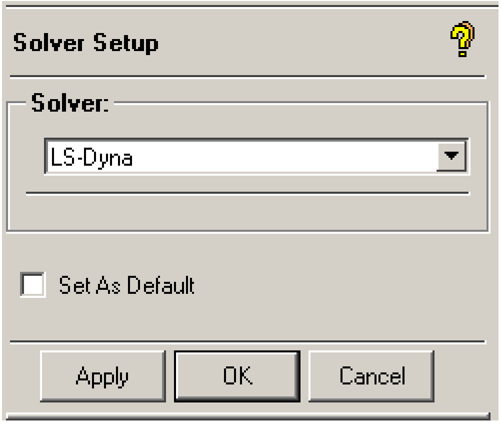
Select File > Import Mesh > From Nastran browse and select the file Frame.dat as shown in Figure 4.89, and Apply.

Figure 4.89 Import Nastran file window



Select Settings > Solver, select LS-Dyna from the pull down list and Apply at the bottom of the Solver Setup panel as in Figure 4.90. All defaults and options applicable for LS-Dyna will be made active.

Figure 4.90
Set Up Solver



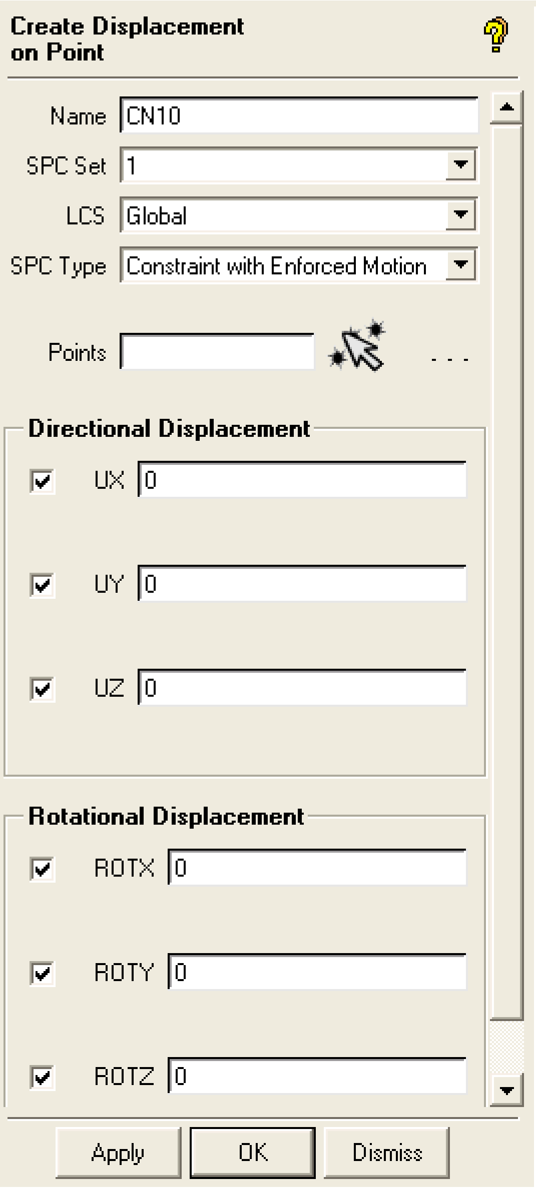
Define Constraints

First a displacement constraint will be applied to one bolt hole. Another bolt hole was already defined in the original Nastran deck.

Turn off all the Parts except ET2D3. Hit Fit Window in the Utilities panel in the upper left hand corner to make the part fill the screen. Turn on ET1D5 part. Turn on Mesh > Points and Lines. We will define a constraint on the center point of this bolt hole definition.

Select Constraints > Displacement on Point as presented in Figure 4.91. Enter CN10 for the Name. Toggle on options UX, UY, UZ, ROTX, ROTY, ROTZ. Leave all settings and values as default.

Figure 4.91
Create Displacement on Point window



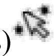
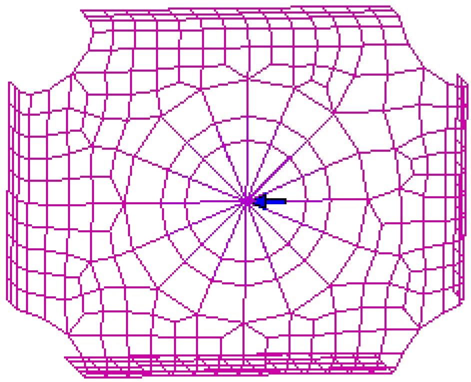
Click on Points > Select node(s)  ; select the node as shown in Figure 4.92 and Apply.

Figure 4.92
Point selection window

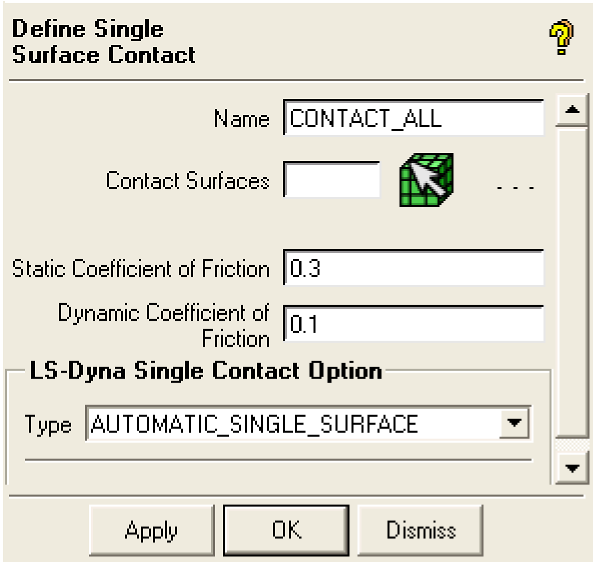


Turn on all Displacements in the Model tree. Turn on Parts > ET2D4 in the Model tree. Note the displacement icon (arrow) applied to this other bolt hole center point. This was read in from the original Nastran deck.

Defining Contact

Select Constraints > Define Single SurfaceContact  as presented in Figure 4.93.

Figure 4.93 Define-Single Surface- Contact window



Enter CONTACT_ALL for the Name.

Turn off Points and Lines under Mesh in the Model tree. Turn on all Parts in the Model tree. Note that Mesh > Shells is already turned on.

Select Contact Surfaces > Select element(s)  option and type “v” on the keyboard or Select all appropriate visible objects  from the Select mesh elements toolbar.

Select the AUTOMATIC SINGLE_SURFACE (default) from the drop down menu.

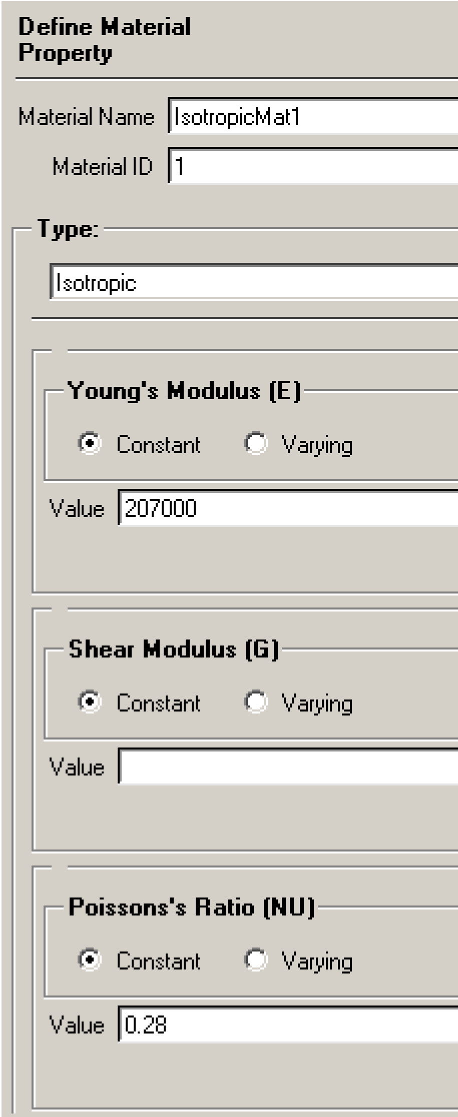
Keep other options as default and press Apply.

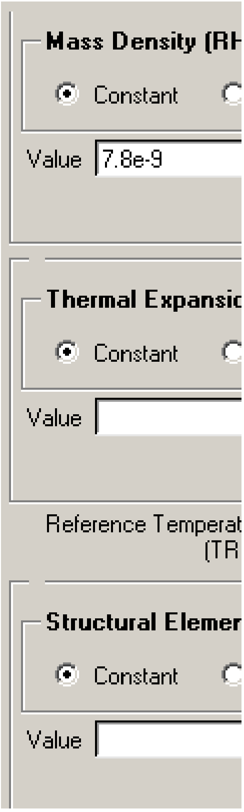
Defining Material Property

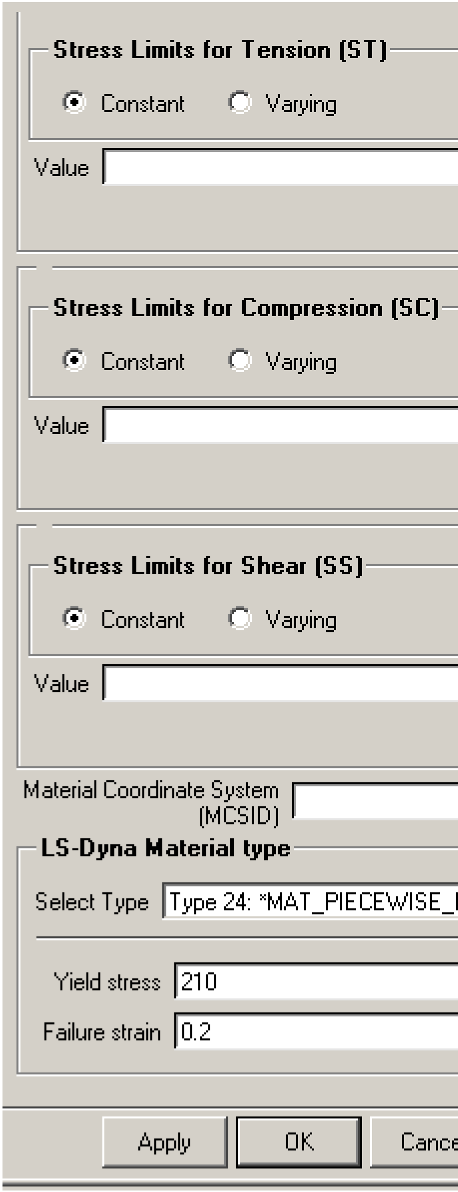
Modify material properties read in from the Nastran deck to reflect the appropriate LS-Dyna material type.

From the Model tree, expand Material Properties, right mouse select IsotropicMat1 and select Modify to get the menu shown in Figure 4.94.

Figure 4.94
Define Material Property window




Mass Density (RH
Constant
Value
7.8e-9
Thermal Expansic
Constant
Value
Reference Temperat
(TR
Structural Elemer
Constant
Value

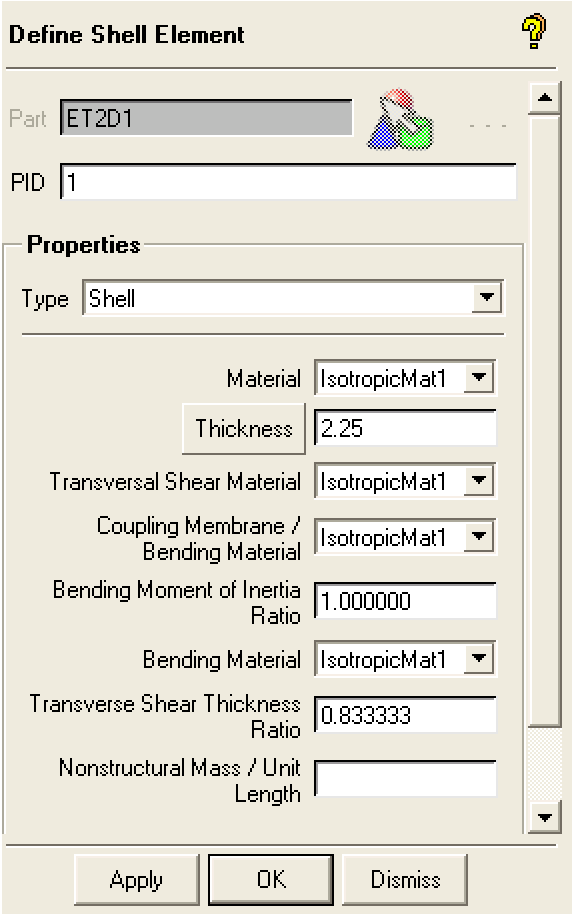

Stress Limits for Tension (ST)
Constant
Varying
Value
Stress Limits for Compression (SC)
Constant
Varying
Value
Stress Limits for Shear (SS)
Constant
Varying
Value
Material Coordinate System
(MCSID)
LS-Dyna Material type
Select Type
Type 24: *MAT_PIECEWISE_I
Yield stress
210
Failure strain
0.2
Apply
OK
Cance

Scroll down to the bottom of this panel and change LsDyna Material type > Select Type to Type: 24: *MAT_PIECEWISE_LINEAR_PLASTICITY. Specify the Yield Stress as 210 and Failure strain as 0.2 and press Apply.

Similarly, modify the material property for IsotropicMat2. Set the Mass Density to 7.84e-06.

Review properties of the different Parts. Expand the Parts tree and the individual part, for example part ET2D1, and right mouse select Surface/Line Properties and select Modify to reveal the Define Shell Element panel, as shown in Figure 4.95. Note the Material assignment and Thickness. Review all other shell parts (ET2D*).

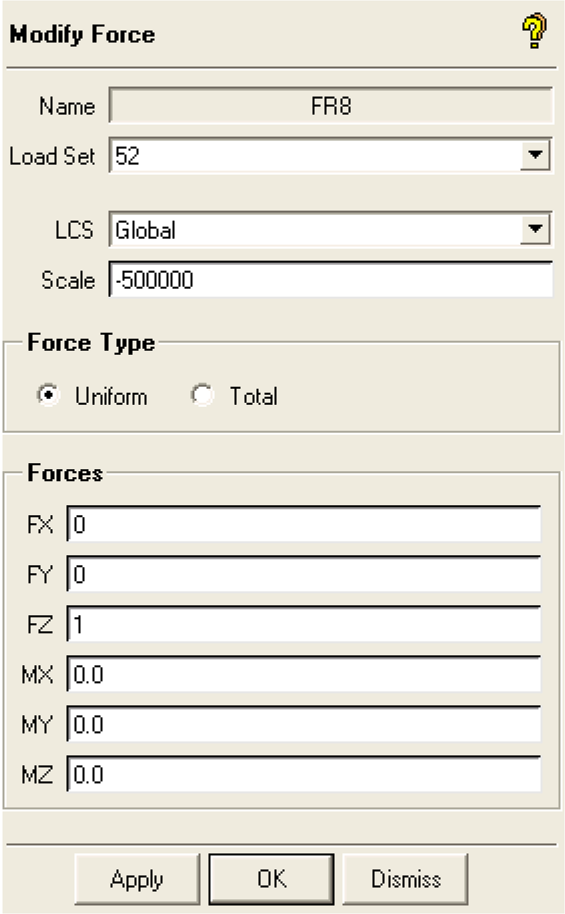
Figure 4.95
Save Project As window



Also review the line properties for ET1D5 (bars representing the bolt holes) and RBE2 > ET1D16 (rigid bodies connecting the main shell parts). Turn on Line Properties within these parts and note the icons representing the different line element types.

Review the Load. Expand Loads > Set 52 > FR8 in the Model tree. Right mouse select FR8 and Modify. Review the panel as shown in Figure 4.96.

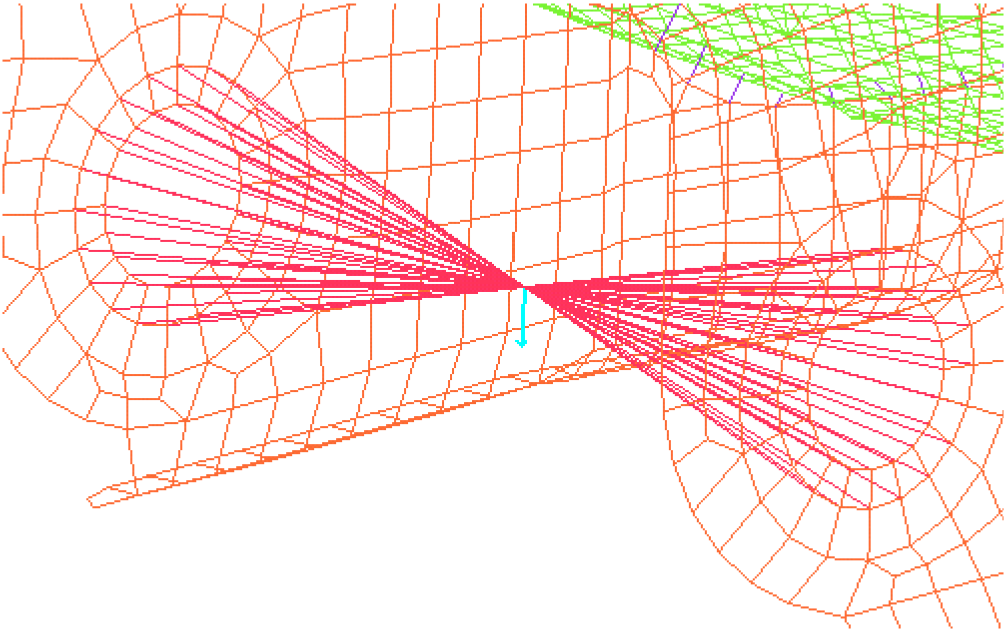
Figure 4.96 Modify Force panel



Turn on Mesh > Lines. Also, turn on the Load in the Model tree and view as in Figure 4.97.

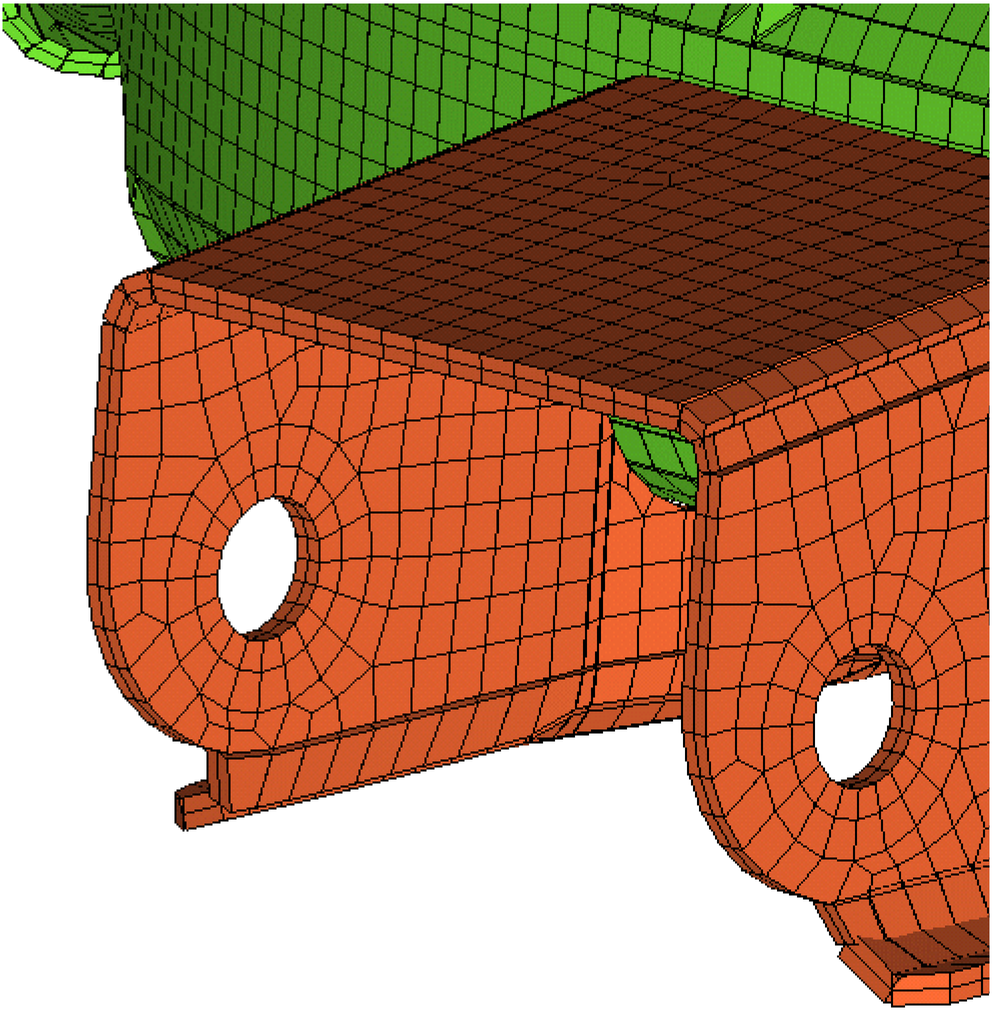
Note the downward force applied to the center of the bars representing the bolt across the flange, ET2D2.

Figure 4.97 View of Force



View the shell thickness. Turn off Lines, Loads and leave on Shells. Right mouse select Mesh > Shells and select Shell Thickness to visualize the shell thickness as in Figure 4.98.

Figure 4.98 Modified Force

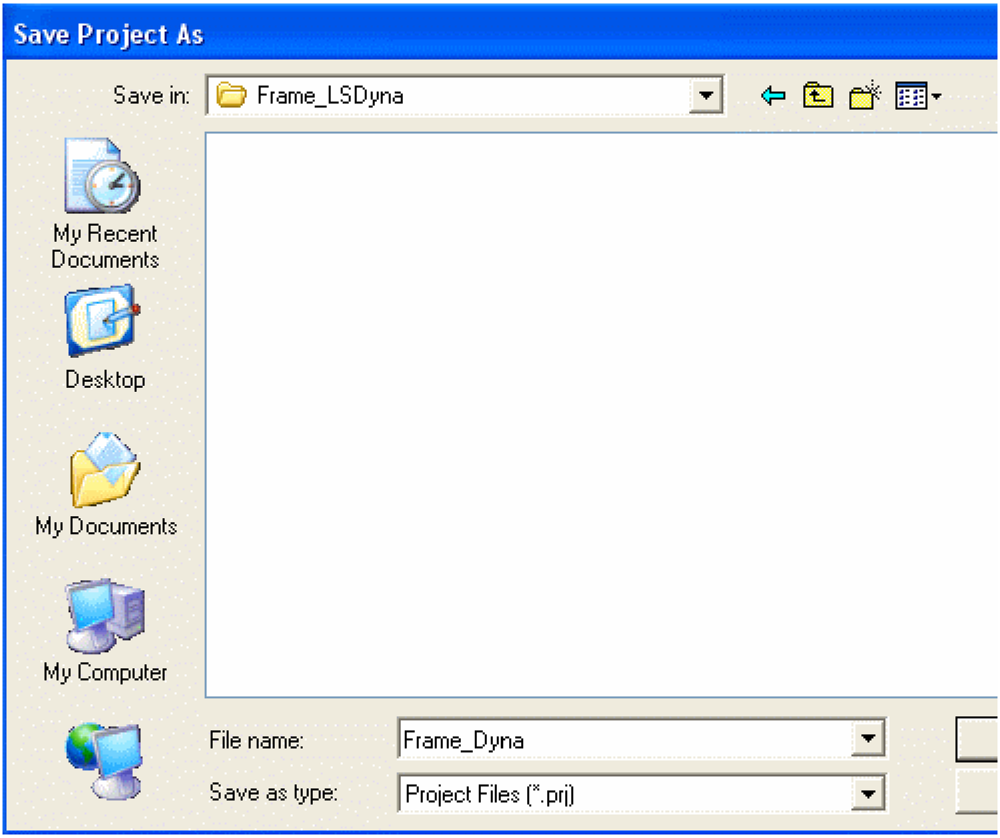


c) Save Project

Select File > Save Project As…, Create New Folder, rename it FRAME_LSDyna and enter a project name, e.g. Frame_Dyna and Save from the window as shown in Figure 4.99.

Along with the Frame_Dyna.prj (project) file, it will also write out the Mesh file, Attribute file and Parameter files: Frame_Dyna.uns, Frame_Dyna.fbc and Frame_Dyna.par respectively.

Figure 4.99 Save Project As window

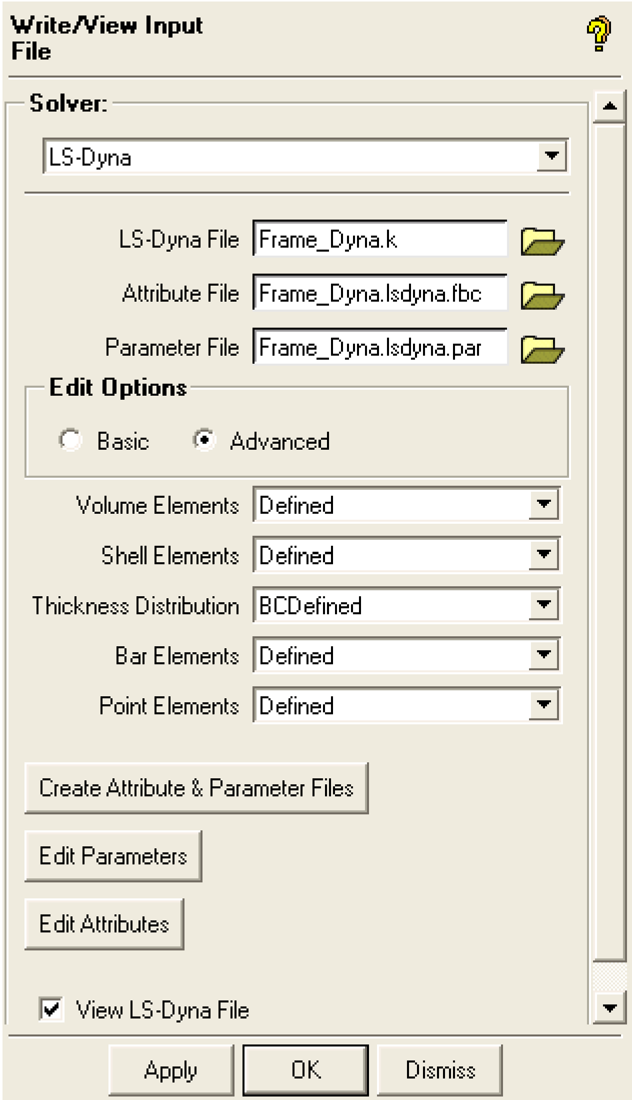


d) Solver Setup

Write LS-Dyna Input File

Select Solve Options > Write/View Input File  to get the panel shown in Figure 4.100.

Figure 4.100 Write/View Input File window



Change Thickness Distribution to BCDefined. Turn on Edit Options > Advanced and select Create Attribute & Parameter files. Parameters (global settings) and Attributes (local part settings) can be edited within this panel to modify various properties, element type, and constraint and load definitions. For this tutorial, however, all of these have been set either in the original Nastran deck or as a result of all of the above modifications.

Note: Two sets of Attributes (*.fbc) and Parameters (*.par) files are usually created. One set, [project name].fbc/par is for general internal settings and the other, [project name].lsdyna.fbc/par is for solver specific settings.

Turn on View LS-Dyna File and Apply. The LS_Dyna deck (Frame_Dyna.k) can be edited in the text editor if necessary (not for this tutorial).

Solving the problem

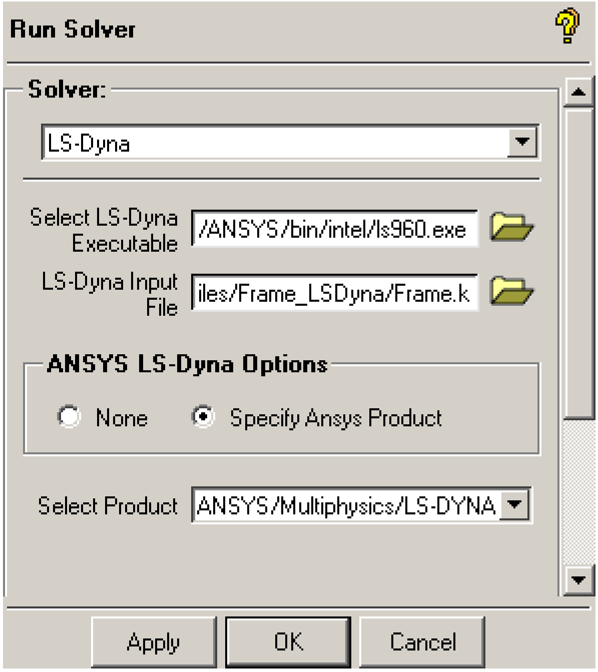
Select Solve Options > Submit Solver Run to start LS-Dyna as in Figure 4.101. Make sure the LS-Dyna Input File is selected as Frame_Dyna.k. Check and verify the location of the LS-Dyna executable.

If using LS-Dyna within Ansys, turn on Specify Ansys Product and select the appropriate product description from the pull down list.

Press Apply.

LS-Dyna will generate the binary result file d3plot in the project directory Frame_LSDyna. For the purposes of this tutorial, it is not necessary to run through the entire transient solution. Kill the LS-Dyna run after, say, 20 time steps.

Figure 4.101
RunSolver window



Select File > Close Project and save if prompted.

Post Processing of Results

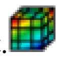
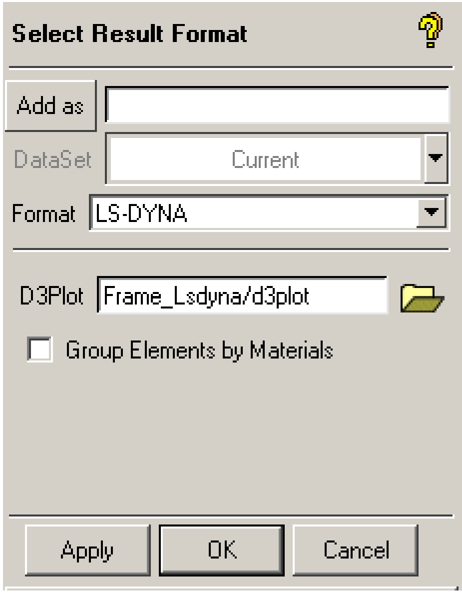
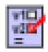
Select File > Results > Open Result. Choose LS-DYNA as the Format. Select the d3plot file from the browser and press Apply as in Figure 4.102.

Figure 4.102
Select Result Format window

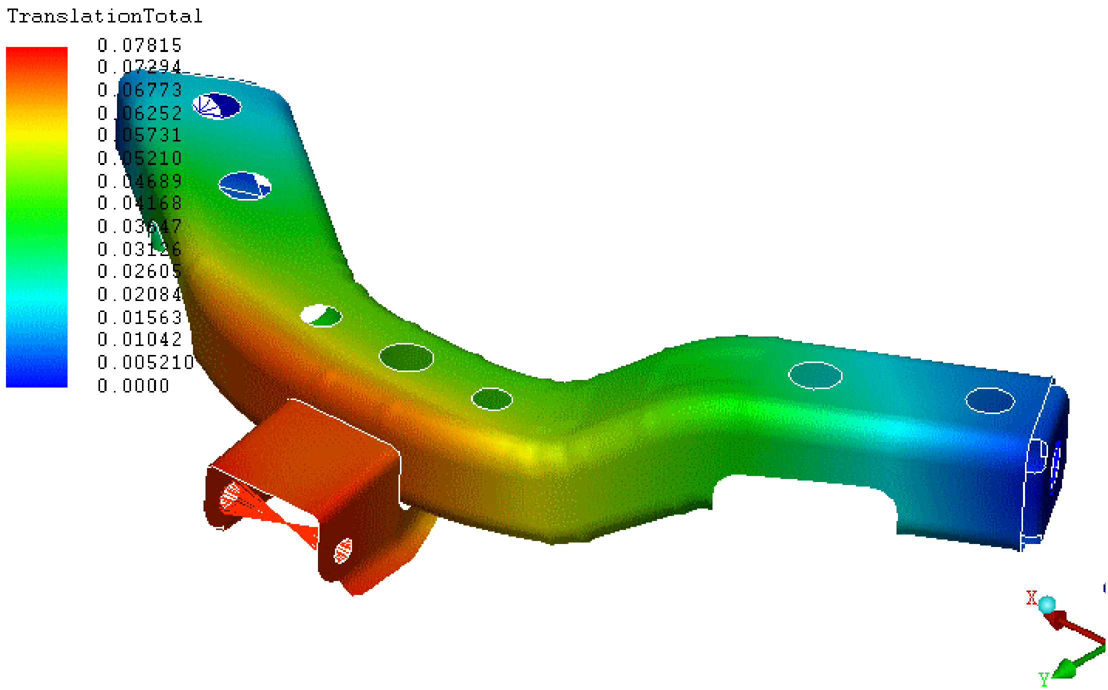


After loading, the default panel will be Select Transient Steps. The default displayed variable will be total translation. To change the displayed variable, select Variables . Change Category, Scalar and Vector variables as desired.

For a quick animation of the time step results, select Control All Animations and hit the Animate arrow.

View the results as shown in Figure 4.103. For a more complete tutorial of post-processing functionality, please refer to the CFD > Post Processing or the FEA > Ansys tutorials.

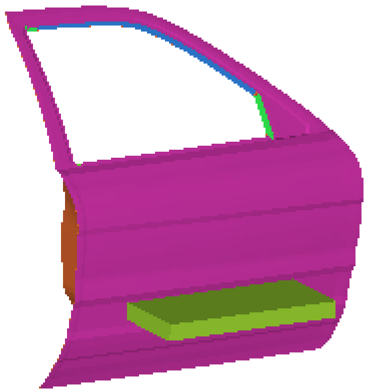
Figure 4.103
Results Displayed in the Graphics window


TranslationTotal
0.07815
0.07294
0.06773
0.06252
0.05731
0.05210
0.04689
0.04168
0.03647
0.03126
0.02605
0.02084
0.01563
0.01042
0.005210
0.0000
X
Y

4.2.2: Front Door-Side Impact

AI*Environment can be used to carry out various types of dynamic impact analysis. Some examples of this category of problems include automotive frontal impact, side impact, bird-strike, high velocity projectiles, etc. A simple door structure is used to demonstrate the process. A rigid impactor strikes the door structure in the lateral direction. **Yield Stress** and **Failure Strain Criteria** are provided for the door structure so that failed nodes and hence failed elements are deleted from the analysis. The geometry is shown in Figure 4.104:

Figure 4.104
Front Door Model



a) Summary of Steps

Data Editing

Launch AI*Environment and import an existing Nastran data file

Verification of imported data

Modify Density

Contacts and Velocities

Define Contact

Define Initial Velocity

Save Project

Solver Setup

Setup LS-Dyna Run

Write LS-Dyna Input File

Solution and Results

Solving the problem

Visualization of Results

b) Data Editing

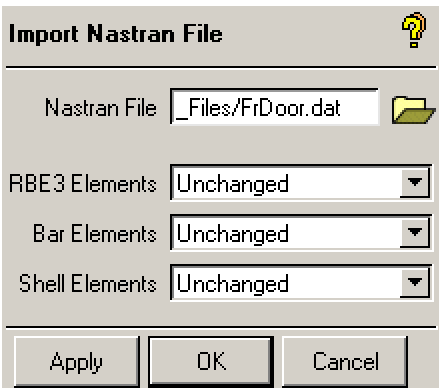
For this Tutorial, use the **FrDoor.dat** file from the **AI_Tutorial_Files** directory.

Launch AI*Environment

Launch AI*Environment user interface.

Select File > Import Mesh > From Nastran from the Main menu, which will open the Import Nastran File window shown in Figure 4.105. Make sure LS-Dyna is selected as the solver via Settings > Solver, and hit Apply.

Figure 4.105 Import Nastran File window



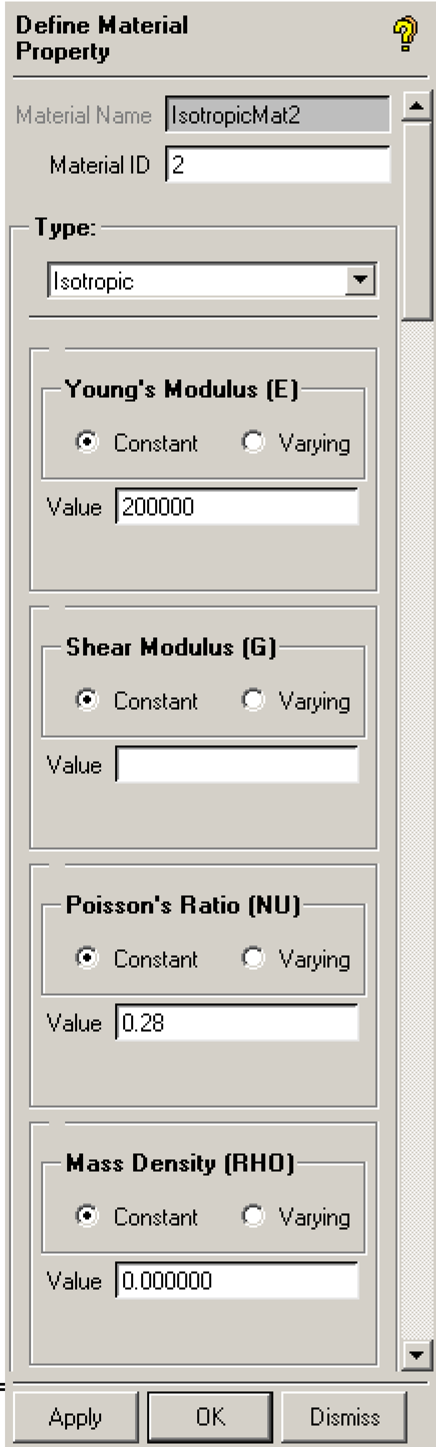
Press Apply in the **Import Nastran File** window.

Change Solver to LS-Dyna

Verification of imported data

Expand the Material Properties branch of the Display Tree by clicking on the + button. Double-click on IsotropicMat2 or right-click on it and select Modify to open the Define Material Property window as shown in Figure 4.106.

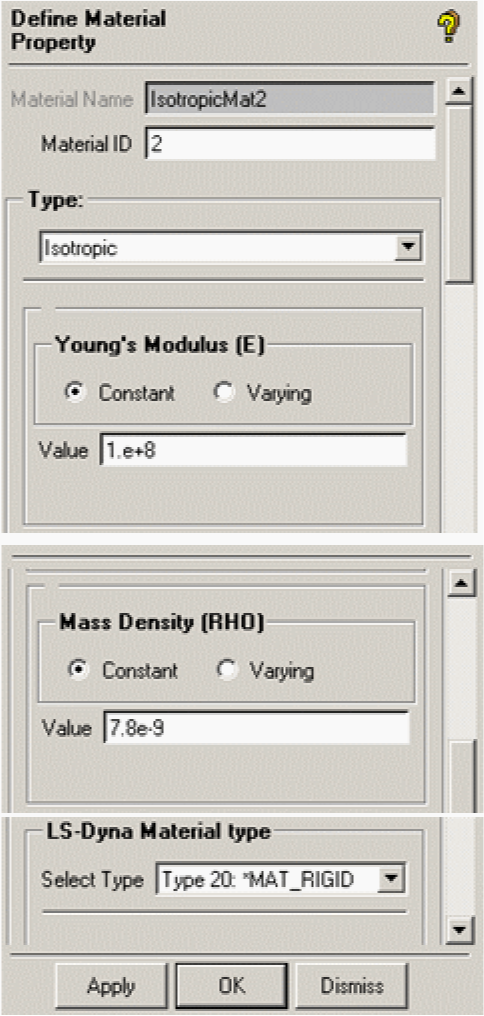
Figure 4.106
Define Material Property window



Modifying Density and LS Dyna Material Type for Materials

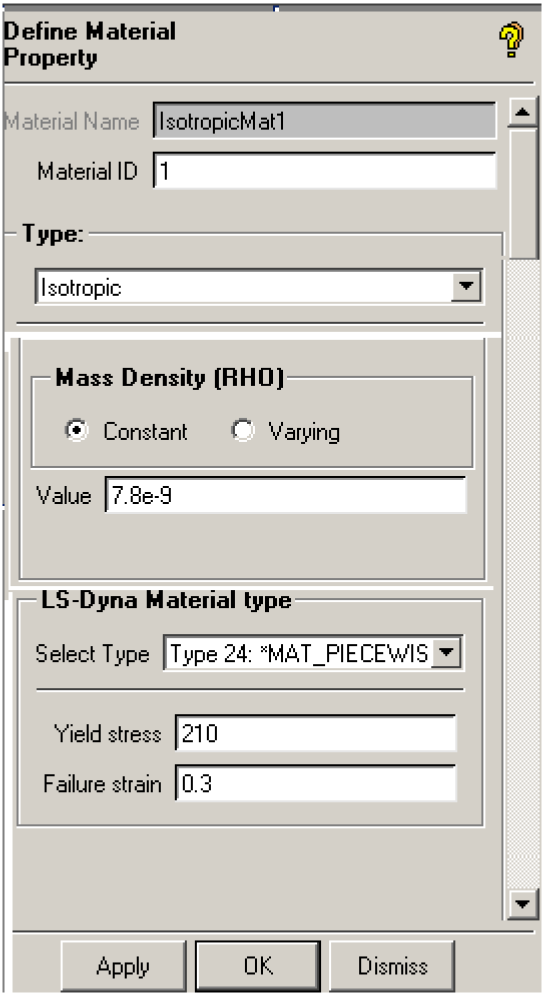
Change the constant value of Young’s Modulus from **200000** to **1.0e+8** and the constant value of Mass Density from **0.0** to **7.8e-9** change **LS-Dyna Material type to Type 20: *MAT_RIGID** as shown in Figure 4.107 and press Apply.

Figure 4.107
Modification Isotropic Mat 2



Similarly modify the properties for IsotropicMat1. In the **Define Material Property** window under **LS-Dyna Material type** change the type to **Type 24: *MAT_PIECEWISE_ LINEAR_ PLASTICITY**. Change Density to 7.8e-9, set Yield stress to 210.0 and Failure strain to 0.3 as shown in Figure 4.108 and Press Apply.

Figure 4.108
Modify Isotropic Material 1



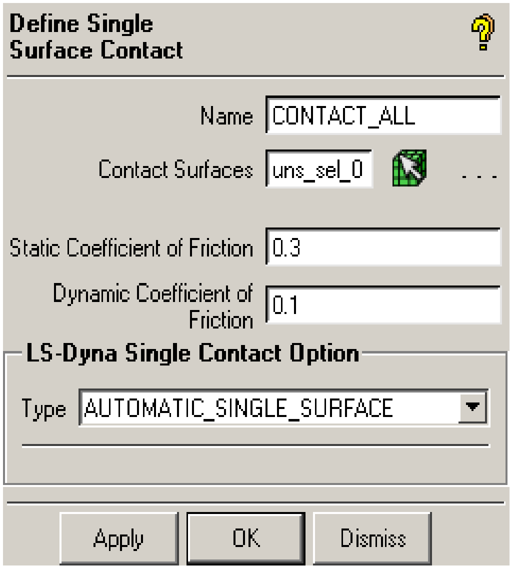
c) Contacts and Velocities

To map the real crash situation the necessary contact and velocity must be applied. This is explained in this section.

Contact

From the Constraints tab, click on Define Single Surface Contact  to open the **Define Single Surface Contact** window as shown in Figure 4.109.

Figure 4.109 Define Single Surface Contact window



As shown in Figure 4.109, supply the following information for the contact.

Name: CONTACT_ALL

Contact surfaces: Select all elements using hotkey “a” from the keyboard. (Make sure that Points and Lines are switched **Off** in the Mesh branch of the Display Tree widget.) The messages area should indicate “11081 elements.”

Select **AUTOMATIC SINGLE_SURFACE** option for Automatic contact option.

Press Apply to generate Contact Surface.

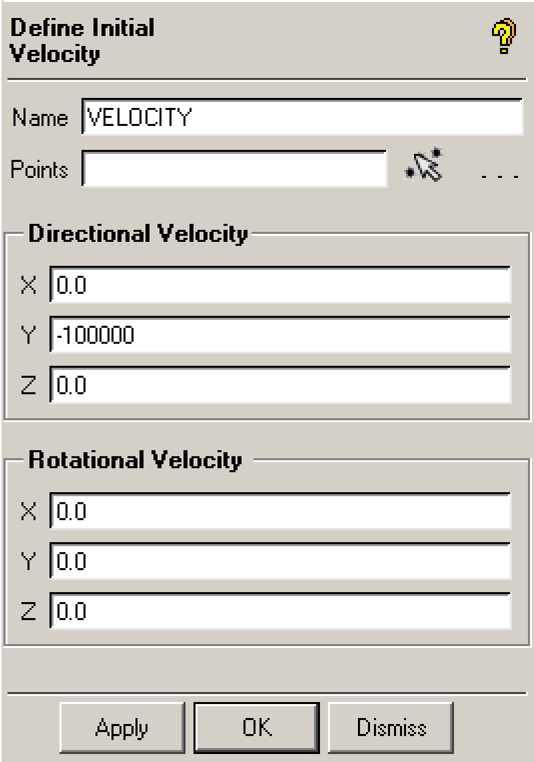
Turn **OFF** Single Surface Contacts display from Display Tree widget.

Velocity

From the Constraints tab click on Define Initial Velocity  to open the **Define Initial Velocity** window as presented in Figure 4.110.

Expand Parts in Display Tree by clicking on +, and turn **OFF** all the parts except ET2D22.

Figure 4.110 Define Initial Velocity window

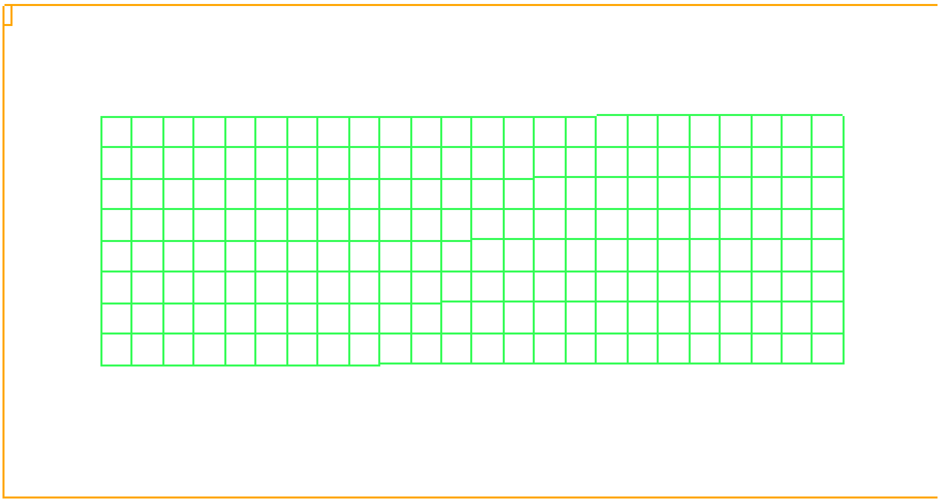


As shown in Figure 4.110, supply the following information in the **Define Initial Velocity** window.

Enter Name as **VELOCITY**

For **Points** click on Select node(s),  and select all the nodes by clicking on the Left Mouse button and dragging the selection window as shown in Figure 4.111. (Make sure that Points and Line are switched ‘**Off’** in the Display Tree widget). The message area should indicate “514 nodes.”

Figure 4.111 Selection of Region where to Apply Velocity



Enter a value of **-10000** for the Directional Y-Velocity.

Press Apply to define Initial velocity.

Turn **OFF** Velocities display from Display Tree widget.

Switch **On** all the Parts by Parts > Show All in the Display Tree widget.

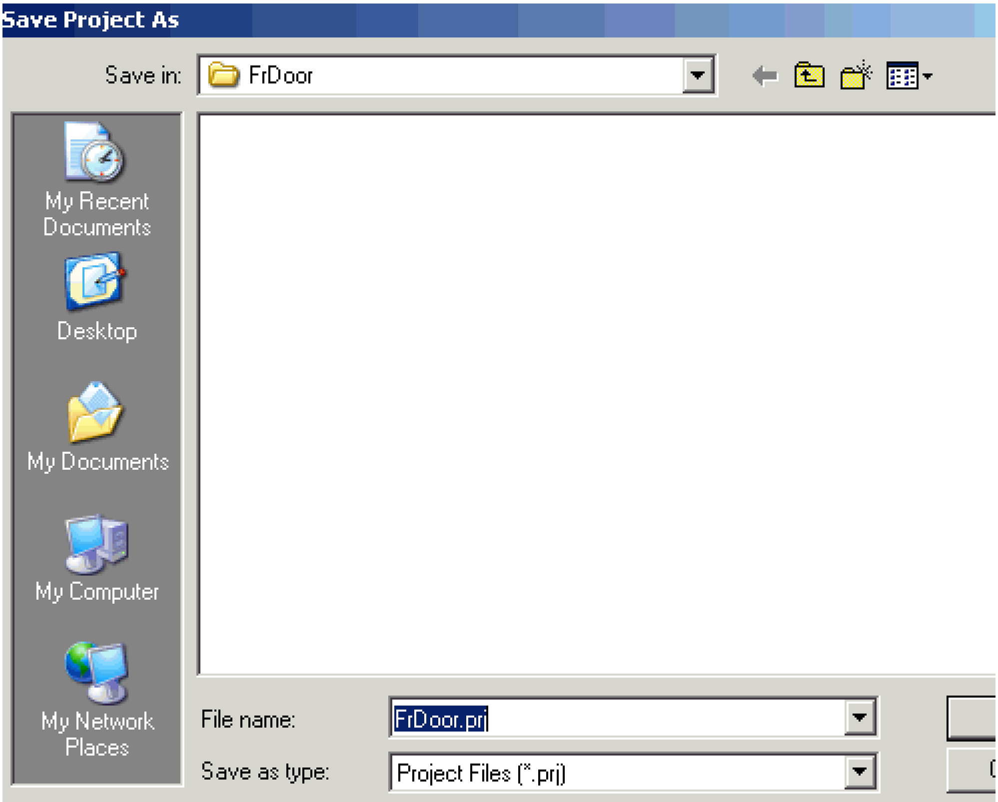
Save Project

Through File > Save Project As option, create new directory **FrDoor** as said in earlier tutorials.

Enter **FrDoor** as project name and press '**Save'** to save the project in this directory as shown in Figure 4.112.

Along with the FrDoor.prj file, it will also store three other files, Mesh file, Attribute file and Parameter files as FrDoor.uns, FrDoor.fbc and FrDoor.par respectively.

Figure 4.112 Save Project As window



d) Solver Setup

Setup LS-Dyna Run

First, user should select the appropriate solver before proceeding further.

Select Settings > Solver from the Main menu and select LS-Dyna and press ‘Apply’ as shown in Figure 4.113.

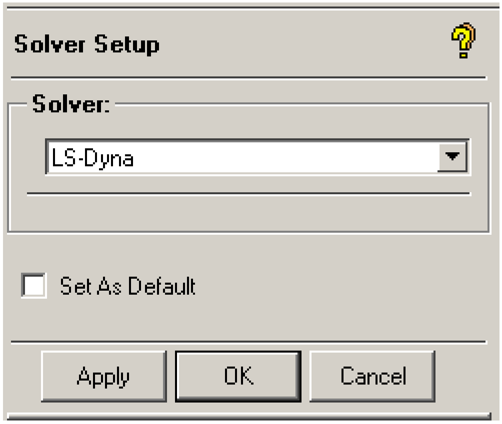
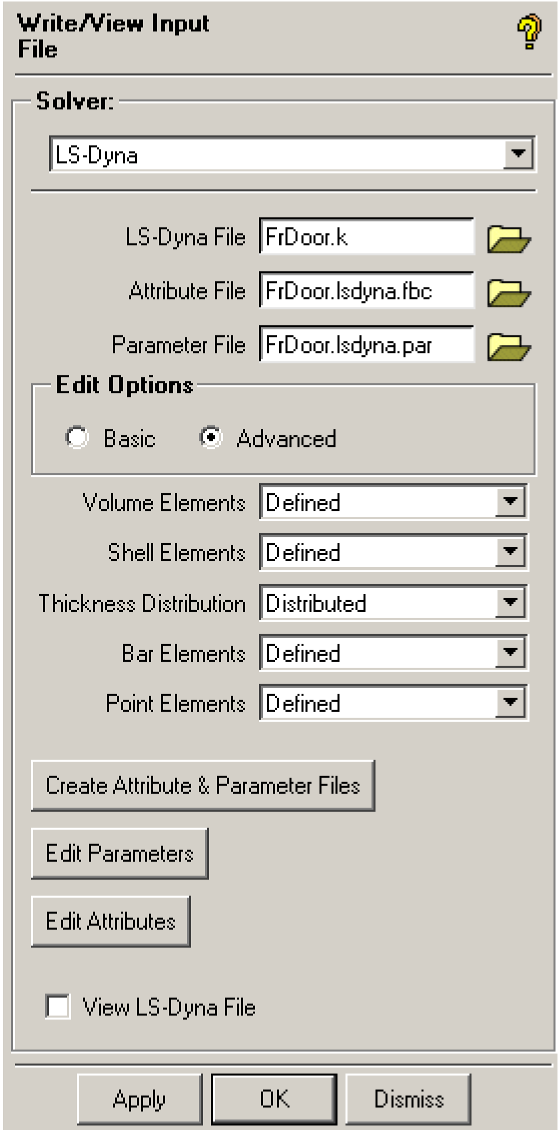


Figure 4.113 Solver Setup Window

Write LS-Dyna Input File

From the Solve Options tab click on Write/View Input File to open the Write/View Input File window as shown in Figure 4.114.

Figure 4.114 Write/View Input File window



In Edit Options, Enable **Advanced**, and click on **Create Attribute & Parameter Files,** which will create the attribute and parameter files. This file will be used for translating the information into the LS-Dyna .k file.

Note: User can switch **ON** View LS-Dyna File option of the **Write/View Input file** window to verify the modification done through Solver Parameter window.

Press Apply to generate LS-Dyna input file.

User will see that the LS-Dyna input data file comes up in the default text editor. If necessary, the user can edit and save the file through this text editor. Since there is no need to do any editing for this example, just close the editor.

e) Solution and Results

Modal analysis is to be performed on this model and the results visualized in the post processor.

Solving the problem

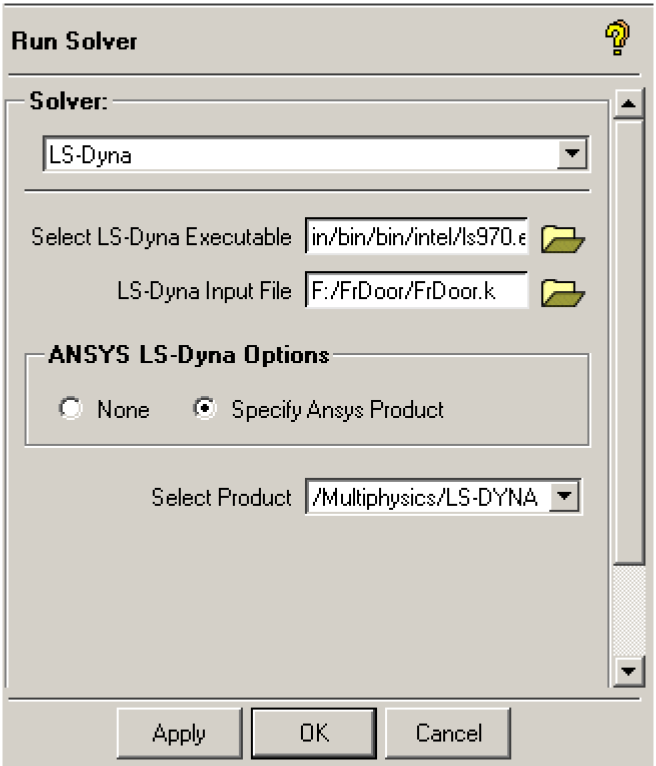
From the Solve Options tab click on Submit Solver Run to start LS-Dyna with the **Run Solver** window given in Figure 4.115.

Specify the LS-Dyna Input File as **FrDoor.k** and the LS-Dyna executable path.

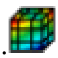
User can specify ANSYS LS-Dyna license product to launch LS-Dyna through AI*Environment.

Press Apply in **Run Solver** window to begin the solution process.

Figure 4.115
LS-Dyna Run window

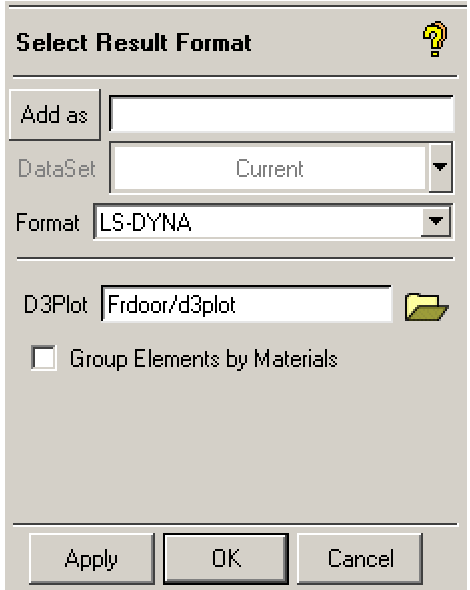


Post Processing of Results

From the main menu select File > Results > Open Results.  The Select Result Format window is displayed in Figure 4.116. For the Format choose LS-DYNA from the drop down box. Select the d3plot file (the LS-Dyna results file) from the FrDoor directory and press Apply.

In the Select Transient Steps panel set Single-step to step#1 0.005 and enable the Display Transient Time option. (Note that Run-time instead of Single-step will process through all the time steps updating the results display at each step.)

Figure 4.116
Select Result Format



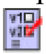
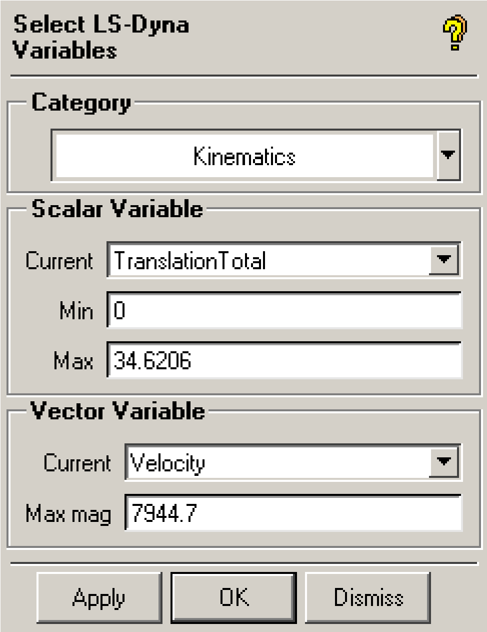
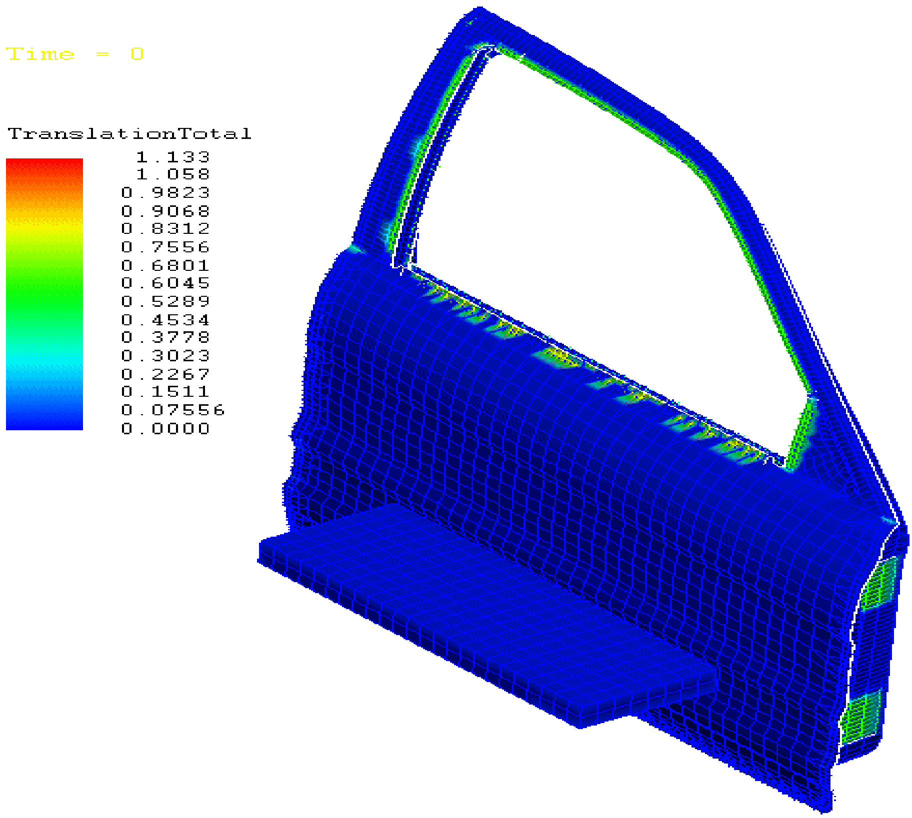
The model will be displayed in the graphics window. From the Post-processing tab click on Variables  and the Select LS-Dyna Variables window will be displayed. Set the Category to Kinematics and the Current Scalar Variable to Translation Total as shown in Figure 4.117.

Figure 4.117
Result Variables window



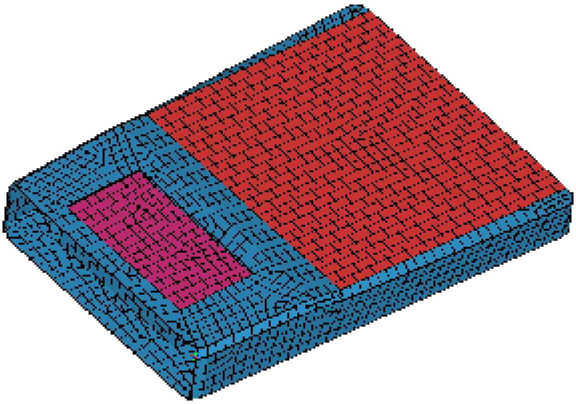
The following results can be seen in the graphics window shown in Figure 4.118.

Figure 4.118
Display Results in the Graphics window


Time = 0
TranslationTotal
1.133
1.058
0.9823
0.9068
0.8312
0.7556
0.6801
0.6045
0.5289
0.4534
0.3778
0.3023
0.2267
0.1511
0.07556
0.0000

4.2.3: PDA Drop Impact

This tutorial demonstrates a drop impact simulation using AI*Environment. It is customary to test electronic consumer products like PDAs, mobile phones and laptops for drop impact survivability. An AI*Environment user can setup a drop impact analysis by simply defining a rigid wall (floor), initial velocity and gravity loading boundary conditions.



a) Summary of Steps

- Data Editing
    - Launch AI*Environment and Load the Mesh
    - Define Properties through Table Editor

  Material and Element Properties
    - Define Material Properties
    - Define Elements Properties

  Contacts and Velocities

  Define Contact
  Define Initial Velocity
  Define Planar Rigid Wall
  Define Gravity
  Save Project

- Solver Setup
    - Setup LS-Dyna Run
    - Write LS-Dyna Input File
- Solution and Results
    - Solving the problem
    - Visualization of Results

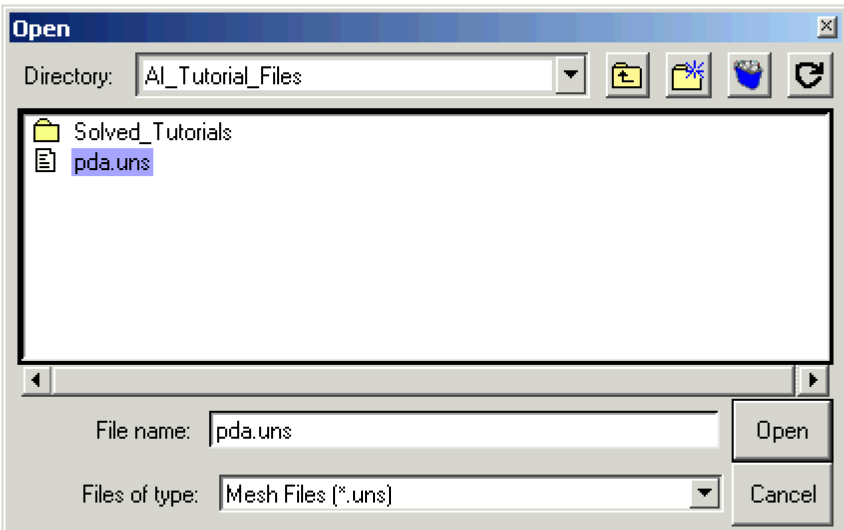
b) Data Editing

Launch AI*Environment

Launch AI*Environment from a UNIX or DOS window. Then File > Change Working Dir… to $ICEM_ACN/../docu/FEAHelp/AI_Tutorial_Files/. For this tutorial, use the pda.uns file from the AI_Tutorial_Files directory.

Click on Open Mesh  from the main menu to open the window as shown in Figure 4.119.
Select the pda.uns file and click on Open to load it into AI*Environment.

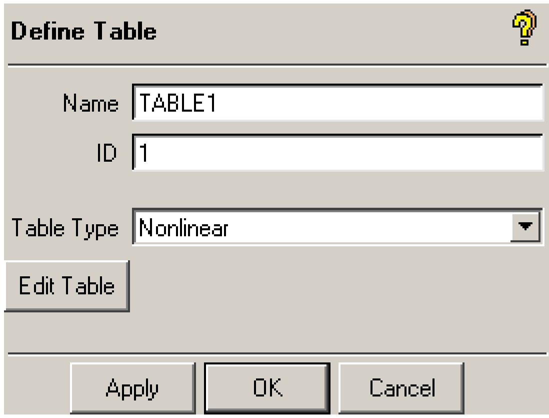
Figure 4.119 Load Mesh Window



Go to Define Properties in Table

- From the Properties tab click on Create Table to open the **Define Table** window as presented in Figure 4.120.

Figure 4.120 Define Table window



- Enter the **Name** as TABLE1 and **ID** as 1. (Both will come by default)
- In the **Table Type** window select Nonlinear.
- Click on Edit Table and enter the data shown in Figure 4.121. Delete the extra rows using Delete Row(s).
- Press Accept in the Table Editor window to close it and press Apply in the Define Table window to save the table editor information.

Note: It will make entry of Table1 in the display tree under Tables tree. User can expand Tables tree and verify it.

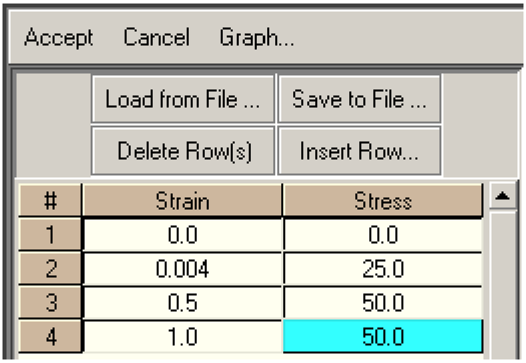


Figure 4.121 Table Editor window

c) Material and Element Properties

After creating Table, the material and element properties should be defined for the model. The table will be used for creating Non-linear properties.

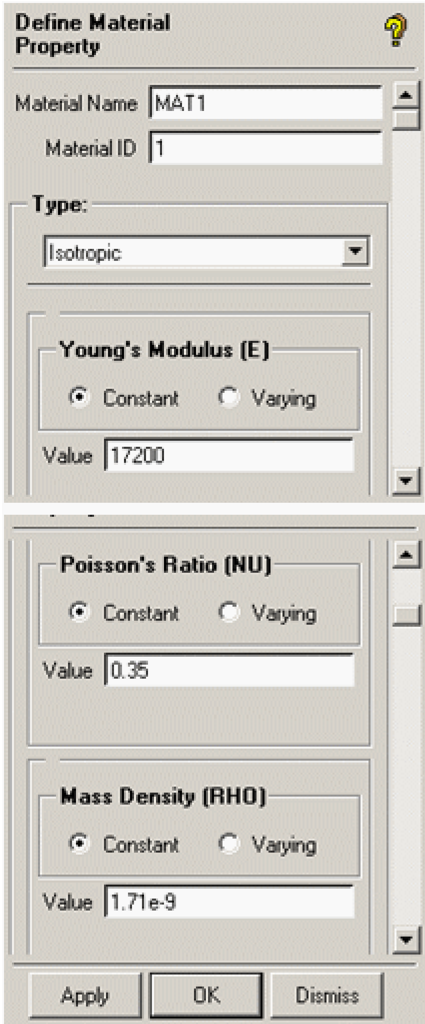
Selection of Material

From the Properties tab select Create Material Property.

- Define the Material Name as **MAT1** in the **Define Material Property** window shown in Figure 4.122.
- Material ID can be left as **1**,
- Select the type as **Isotropic** from the drop down menu.
- Define the Constant **Young's modulus** as **17200**
- Define the Constant **Poisson's ratio** as **0.35**,
- Define the Constant Mass Density as **1.71e-9**,
- Leave other fields as they are and press Apply.

Note: Material Properties branch becomes active in the Display Tree.

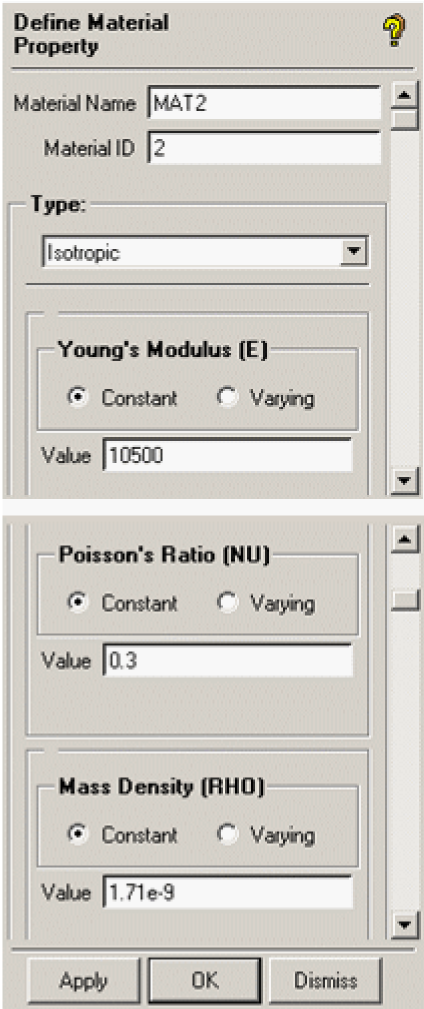
Figure 4.122 Define Material Property window



- Create another material Named as **MAT2** in **Define Material Property** window shown in Figure 4.123.
- Material ID can be left as **2**,
- Select the type as **Isotropic** from the drop down menu.
- Define the Constant Young's modulus as **10500** ,

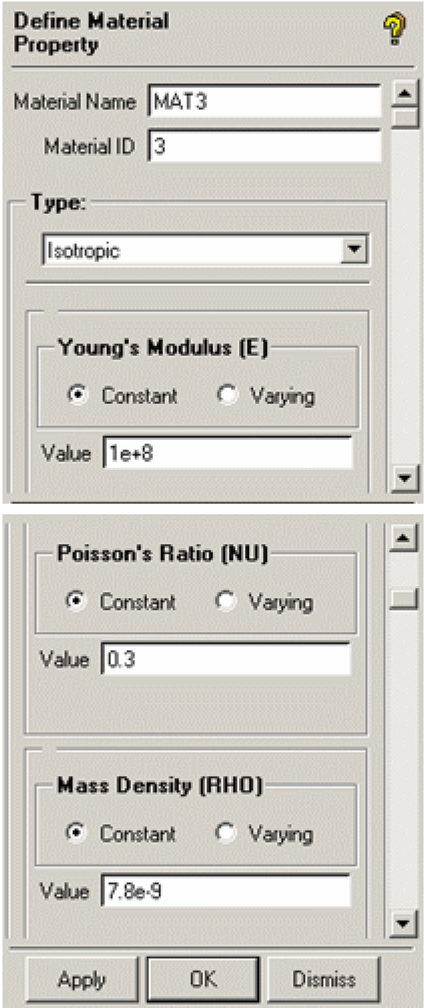
- Define the Constant Poisson's ratio as **0.3**,
- Define the Constant Mass Density as **1.71e-9**,
- Leave other fields as they are and Press Apply.

Figure 4.123 Define Material Property window

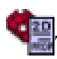


- Create another material Named as **MAT3** in **Define Material Property** window shown in. Figure
- Material ID can be left as **3**,
- Select the type as **Isotropic** from the drop down menu.
- Define the Constant **Young's Modulus** as **1e+8**,
- Define the Constant Poisson's ratio as **0.3**,
- Define the Constant Mass Density as **7.8e-9**,
- Leave other fields as they are and Press Apply.

Figure 4.124 Define Material Property window



Element Properties

- From the Properties tab select Define 2D Element Properties. The Define Shell Element window appears as shown in Figure 4.125.
- Select Part as **PDA.**
- Set PID as **10**.

- Select Type as **Shell**
- Select material as **MAT1.**
- Supply Thickness as **0.5**,
- Keep all other parameters as default and press Apply.

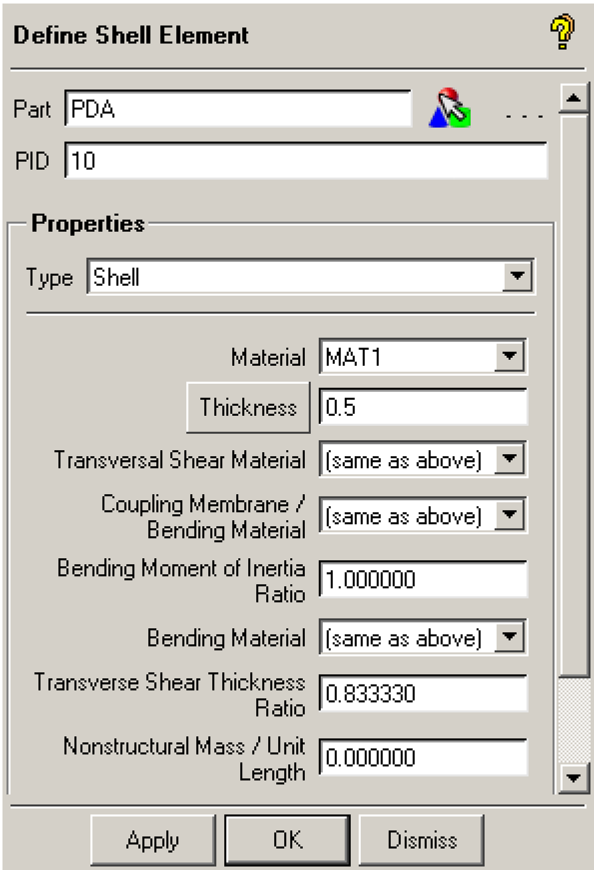
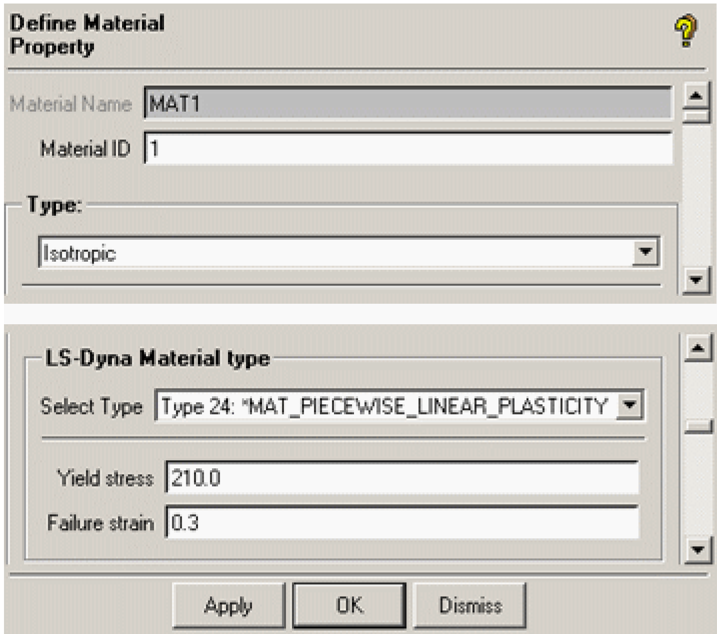


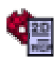
Figure 4.125 Define Shell Element window

- Similarly, define the shell properties on the BAT_COVER part also. All the properties are going to be the same for BAT_COVER except PID. Set the PID to **11** and supply the same thickness of **0.5**.

In the main menu select Settings > Solver and select LS-Dyna from the pull-down. Press Apply. Expand Material Properties in the Display Tree widget. Right-click on **MAT1** and select Modify to open the **Define Material Property** window as shown in Figure 4.126, change the **LS-Dyna Material Type** to **Type24: *MAT_PIECEWISE_LINEAR_PLASTICITY**, Input **Yield Stress** as 210.0 and **Failure Strain** as 0.3 Press Apply.

Figure 4.126 Define Material Property



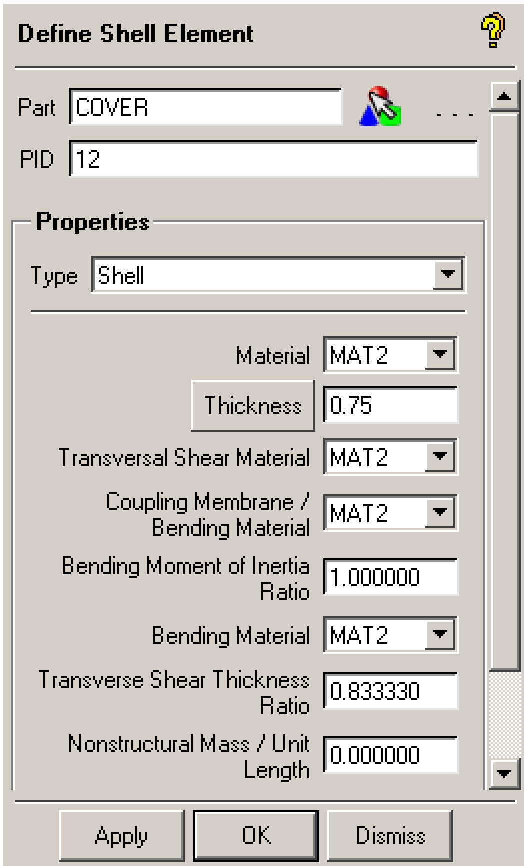
From the Properties tab select Define 2D Element Properties  to open the Define Shell Element window as shown in Figure 4.127.

For the Part **COVER**, MAT2 material has to be used. For that, do the following steps.

- Select Part as **COVER**.
- Set PID to **12**.

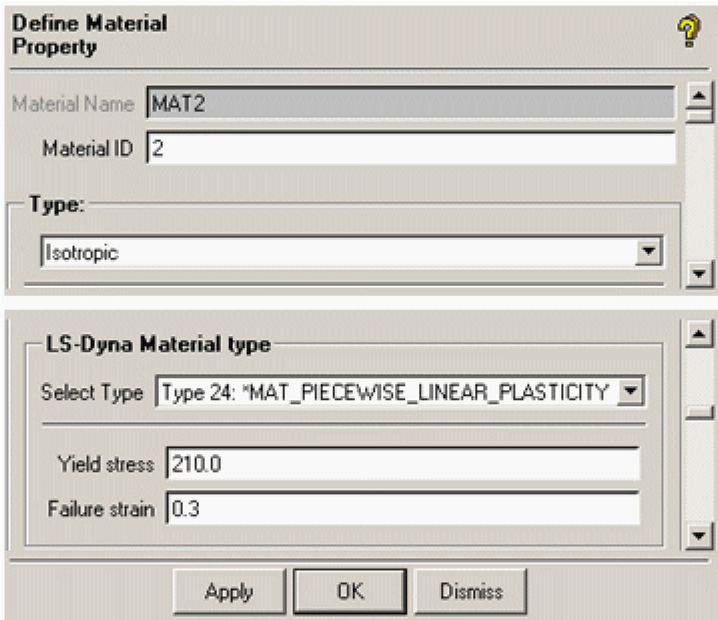
- Set Type to **Shell**.
- Select material as **MAT2,**
- Supply thickness as **0.75**,
- Select **MAT2** for the Transversal Shear Material, Coupling Membrane/Bending Material and Bending Material option.
- Press Apply to complete the operation.

Figure 4.127 Define Shell element window



Expand Material Properties in the Display Tree widget. Right-click on **MAT2** and select Modify to open the **Define Material Property** window shown in Figure. Change the LS-Dyna Material Type to **Type24: *MAT_PIECEWISE_LINEAR_PLASTICITY**, Input **Yield Stress** as 210.0 and **Failure Strain** as 0.3 Press Apply.

Figure 4.128 Define Material Property Window



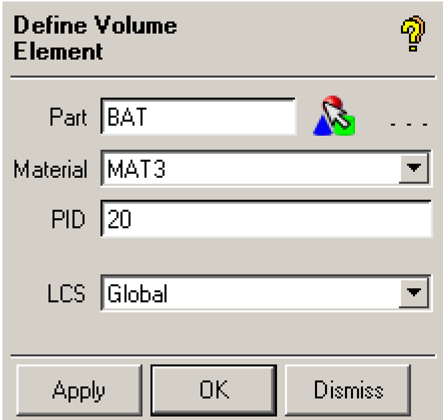
From the Properties tab select Define 3D Element Properties  to open the **Define Volume Element** window shown in Figure 4.129

As BAT part is made of Solid elements, 3D elements property has to be assigned for that. MAT3 material has to be used for this part

- Select Part as **BAT**
- Select material as **MAT3**.
- Set PID to **20**.

- Press Apply to complete the operation.

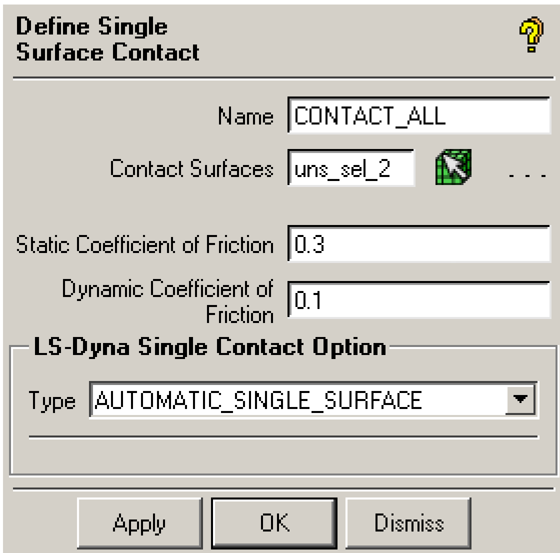
Figure 4.129 Define Solid Element window



d) Contact

- From the Constraints tab click on Define Single Surface Contact to open the **Define Single Surface Contact** window shown in Figure 4.130.

Figure 4.130 Define Single Surface Contact window



- Supply the following information.

Enter Name as **CONTACT_ALL**

For Contact surfaces select all elements using hotkey “**a**”

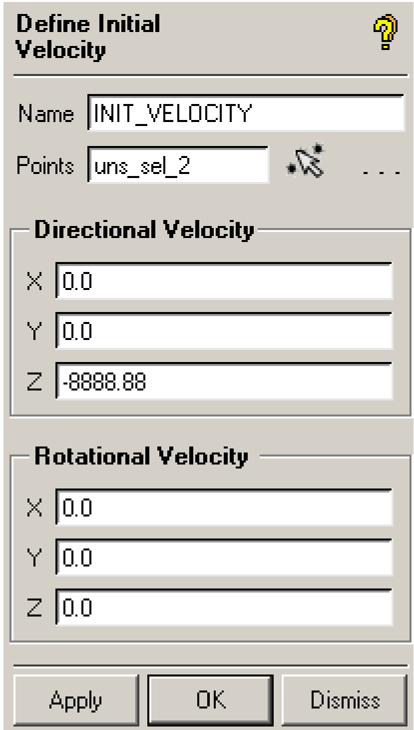
Under the **LS-Dyna Single Contact Option** select AUTOMATIC_SINGLE_SURFACE

- Press Apply to generate Contact information.
- Turn **OFF** Single Surface Contacts display from the Display Tree widget.

e) Velocity

- From the Constraints tab click on Define Initial Velocity to open the **Define Initial Velocity** window as shown in Figure 4.131.

Figure 4.131 Define Initial Velocity window



Enter name as **INIT_VELOCITY**.

For Points click on Select node(s) and select all nodes using hotkey "**a**".

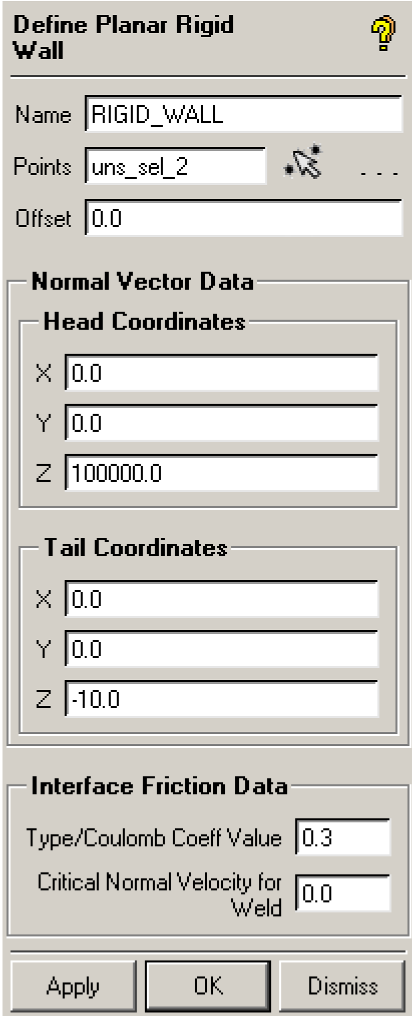
Enter a value of **-8888.88** for the Z - Directional Velocity.

- Press Apply to define Initial velocity.
- Turn **OFF** Velocities display from the Display Tree widget.

f) Rigid Wall

- From the Constraints tab click on Define Planer Rigid Wall to open the **Planar Rigid Wall** window as presented in Figure 4.132.

Figure 4.132 Define Planar Rigid Wall window



Enter name as **RIGID_WALL**

For Points click on Select node(s)  and select all the nodes (0d elements) using hotkey "**a**".

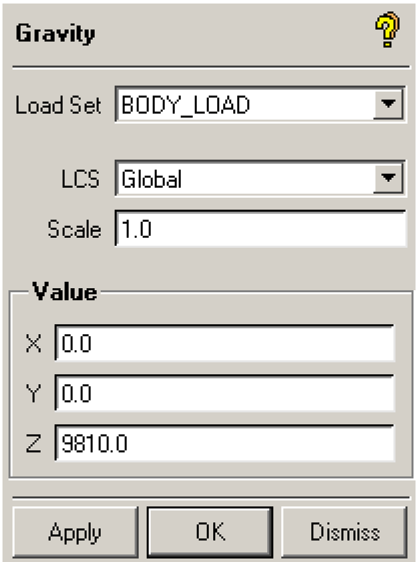
Enter **100000.0** for the Z component of Head Coordinates and **-10.0** as the Z component of the Tail coordinates, Supply **0.3** as the Type/Coulomb Coeff Value under **Interface Friction Data**.

- Press Apply to define the Planar Rigid Wall.
- Turn **OFF** Rigid Wall display from Display Tree widget.

g) Gravity Loading

- From the Loads tab click Set Gravity  to open the **Gravity window** as presented in Figure 4.133.

Figure 4.133 Gravity window



Enter Load Set as **BODY_LOAD**.

Supply the Value of **9810.0** as the Z component for the Gravity.

- Press Apply to define gravity.

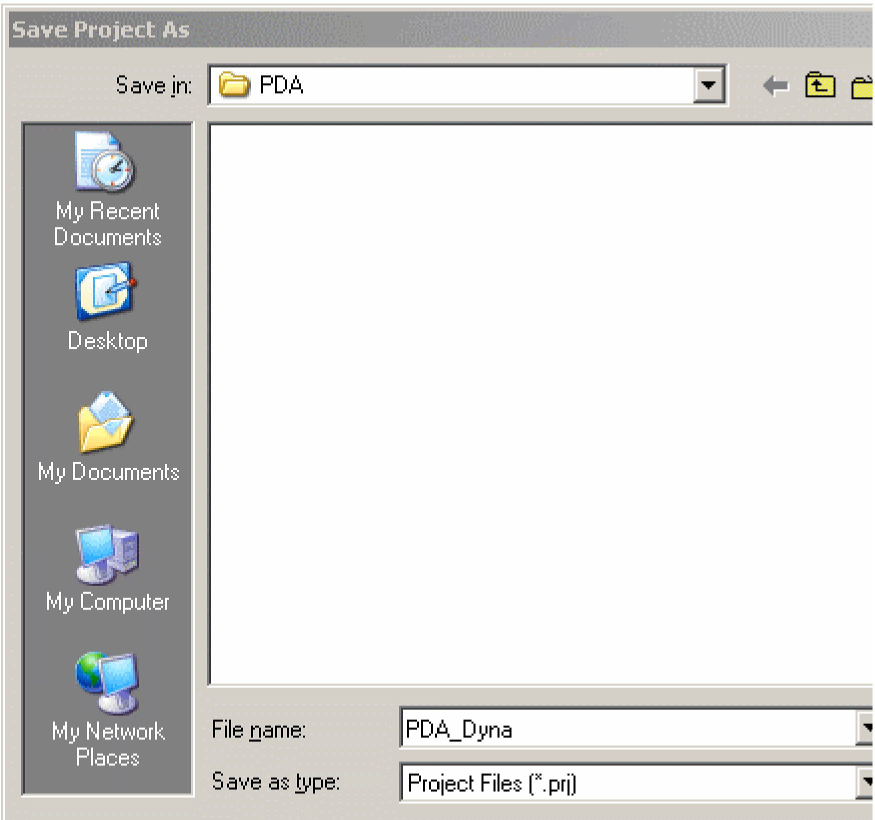
h) Save Project

- From the main menu select File > Save Project As…, create a new directory **PDA** as said in earlier tutorials.

- Enter **PDA_Dyna** as project name and press '**Save'** to save the files in this directory as shown in Figure 4.134.

Along with the PDA_Dyna.prj file, it will also store three other files, Mesh file, Attribute file and Parameter files as PDA_Dyna.uns, PDA_Dyna.fbc and PDA_Dyna par respectively.

Figure 4.134 Save Project As window



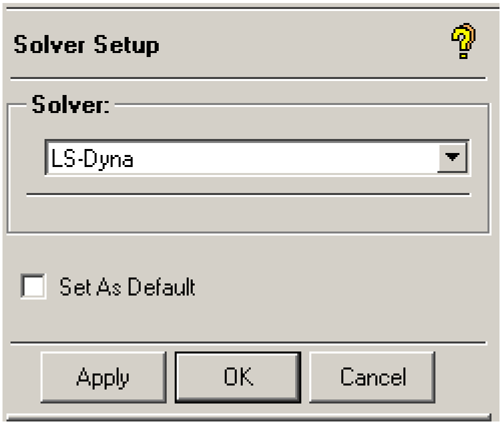
i) Solver Setup

Setup LS-Dyna Run

First, select the appropriate solver before proceeding further.

- Select Settings > Solver from the Main menu and select LS-Dyna as solver and press Apply. Selection of a solver is shown in Figure.

Figure 4.135 Solver selection



From the Solve Options tab click Write/View Input File to open the Write/View Input File window as shown in Figure 4.136.

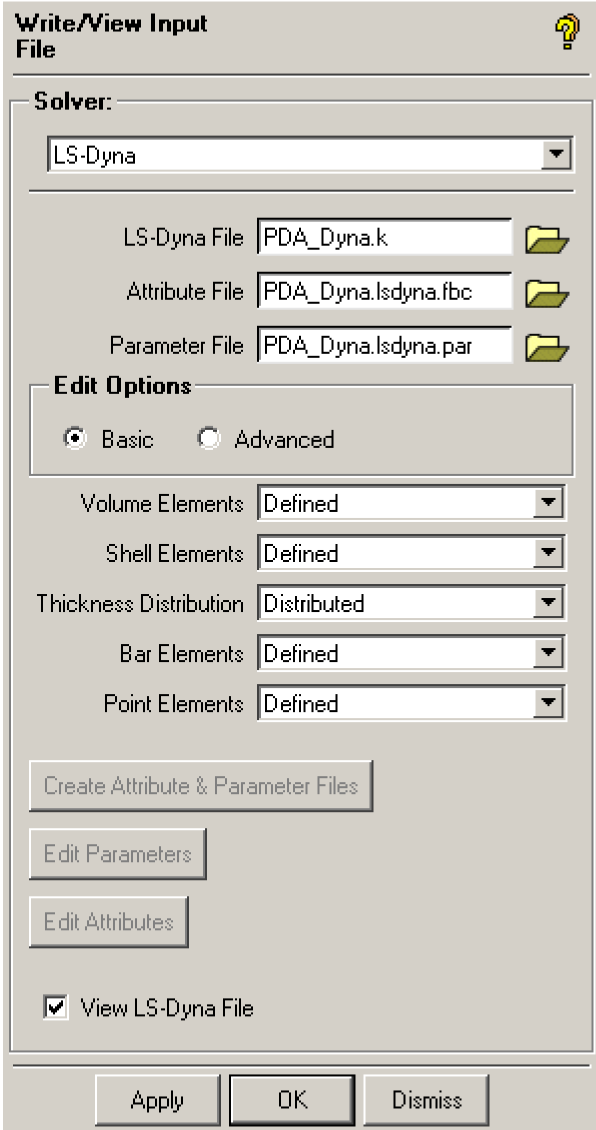


Figure 4.136 Write/View Input File window

- Enable View LS-Dyna File and keep the other options as default,
- Press Apply.

The LS-Dyna input data file comes up in the default text editor. If necessary the input file can be edited and saved through this text editor. Since there is no need to do any editing for this example, just close the editor.

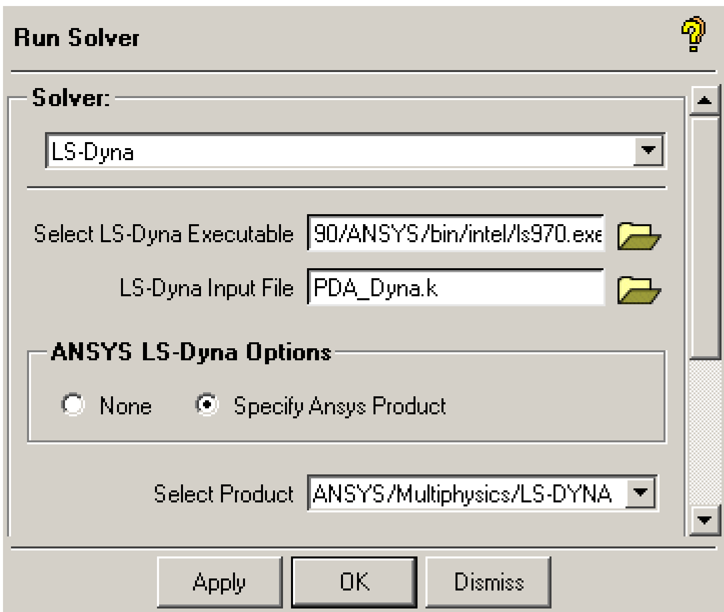
j) Solution and Results

Modal analysis is to be performed on this model and the results should be visualized in a post processor.

Solving the problem

- From the Solve Options tab click on Submit Solver Run  to start LS-Dyna with the **Run Solver** window given in Figure 4.137.
- Supply LS-Dyna file as **PDA_Dyna.k** and the LS-Dyna executable path.
- User can specify ANSYS LS-Dyna license product to launch LS-Dyna through **AI*Environment**.
- Press Apply in the **Run Solver** window.

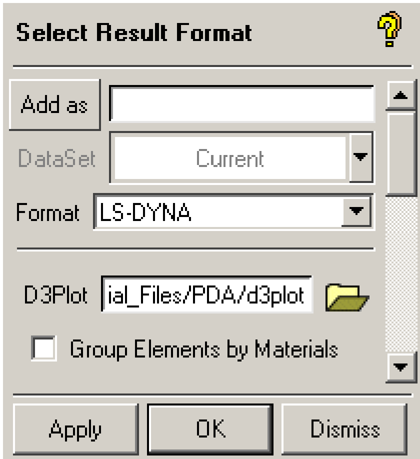
Figure 4.137 Run Solver Window



Post Processing of Results

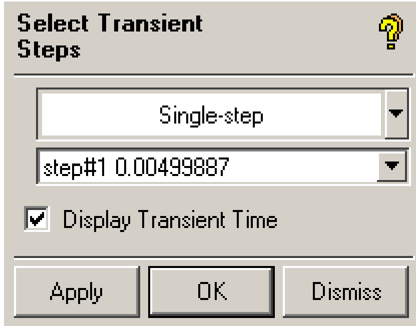
From the main menu select File > Results > Open Results. The Select Result Format window is displayed as shown in Figure 4.138. For the Format choose LS-DYNA from the drop down box. Select the d3plot file (the LS-Dyna results file) from the PDA directory and press Apply.

Figure 4.138 Select Result Format



As soon as Apply button is pressed Select Transient Steps window will be displayed. as shown in Figure 4.139. A specific time step can be selected from the second pull-down area, or Run-time can be selected from the first pull-down area to start moving forward in time with the results display updated at each time step. Select Single-step and step#1. Enable Display Transient Time so the time will appear in the display window.

Figure 4.139
Select Transient Steps window



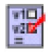
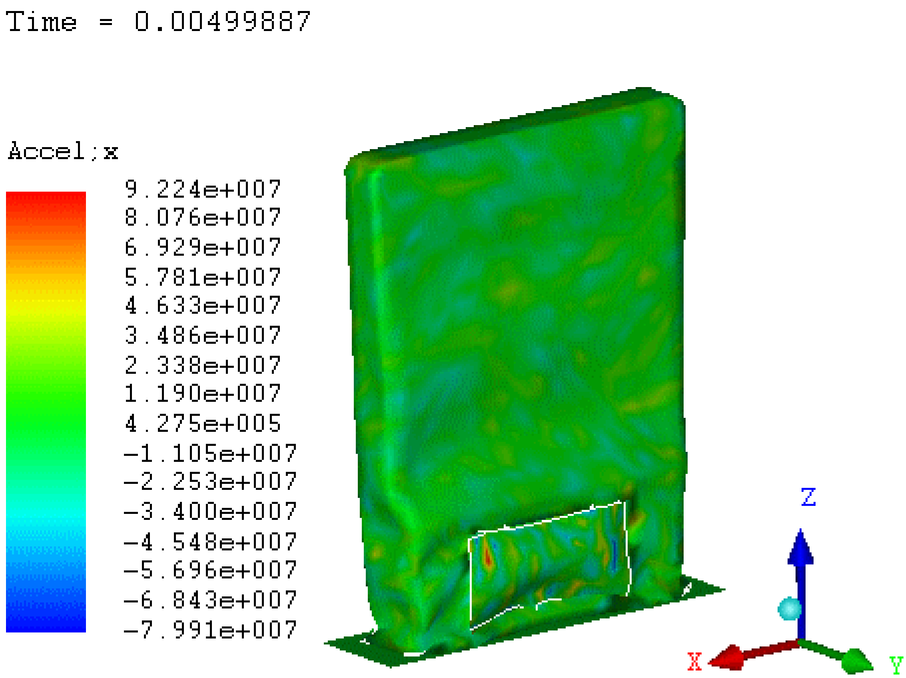
From the Post-processing tab click on Variables.  For Category select Kinematics, and set the Current Scalar Variable to Accel;x. The following results can be seen in the graphics window shown in Figure 4.140.

Figure 4.140
Results Displayed in the Graphics window

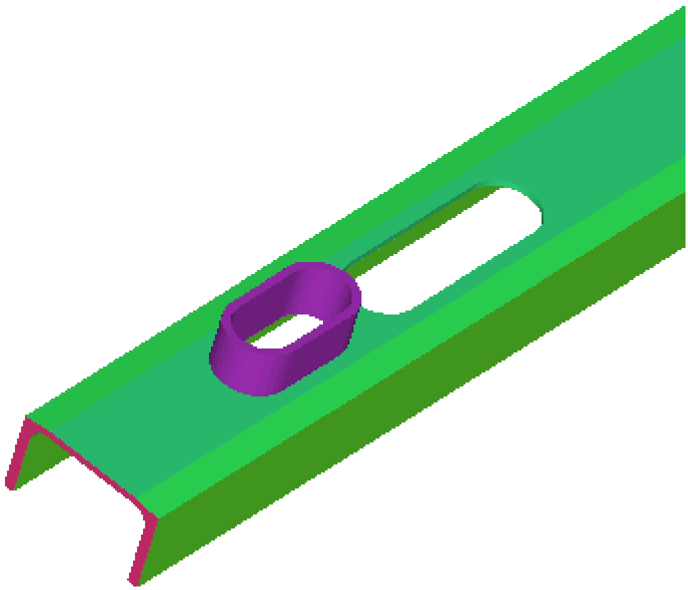

Time = 0.00499887
Accel;x
9.224e+007
8.076e+007
6.929e+007
5.781e+007
4.633e+007
3.486e+007
2.338e+007
1.190e+007
4.275e+005
-1.105e+007
-2.253e+007
-3.400e+007
-4.548e+007
-5.696e+007
-6.843e+007
-7.991e+007
Z
X
Y

4.3: Nastran Tutorial

4.3.1: T-Pipe

This exercise includes meshing of **T-Pipe** geometry by simplifying the thickness using Mid-Surface technique and writing the input file (***.dat**) to perform Modal Analysis in **NASTRAN**. The visualization of results in Post Processor Visual3p is also explained. The geometry of the model is shown in Figure 4.141.

Figure 4.141
T-Pipe Geometry



Note: Before proceeding to tutorials, user is advised to go through the Appendix of this tutorial manual for some important information, which will help to understand tutorials better.

The tutorial input files to do these tutorials are available at the following location of your AI*Environment installation:

For windows: %ICEM_ACN%\docu\FEAHelp\AI_Tutorial_Files

For Unix: $ICEM_ACN/../docu/FEAHelp/AI_Tutorial_Files

User can copy the directory AI_Tutorial_Files to their local area before starting the tutorials or he can browse to this location while doing the tutorial.

Also fully solved tutorials with results (obtained from AI*Nastran) are available in **Solved_Tutorials** directory under **AI_Tutorial_Files** directory.

a) Summary of Steps

Launch AI*Environment and load geometry file

Geometry Editing

Midsurface model

Delete Geometry

Remove Holes

Re-Intersection by Build Topology

Mesh Parameters and Meshing

Mesh Sizing

Meshing

Material and Element Properties

Selection of Material

Element Properties

Solver setup

Setup a Nastran Run

Save Project

Write Nastran Input File

Solution and Results

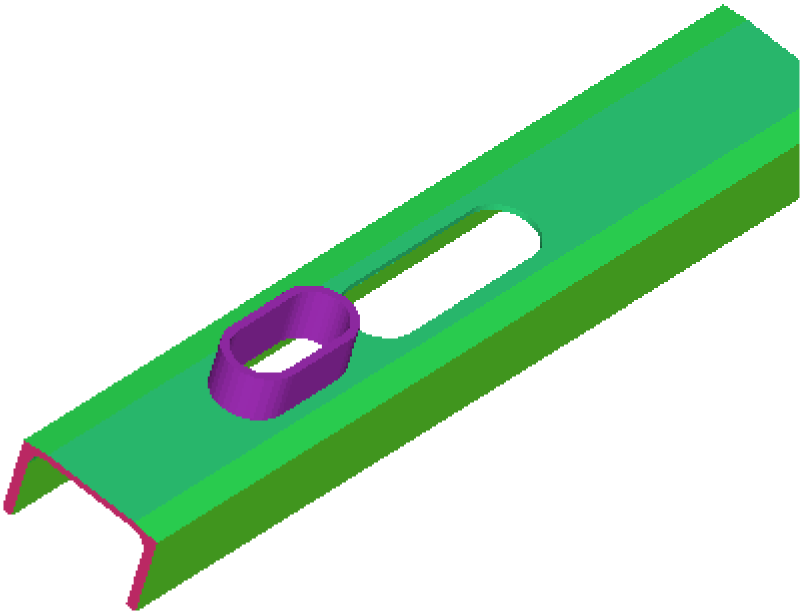
Solving the Problem

Post processing of Results

b) Launch AI*Environment

Launch the AI*Environment from UNIX or DOS window. Then File > Change working directory, $ICEM_ACN/../docu/FEAHelp/AI_Tutorial_Files. Load the tetin file ‘Tpipe.tin’. The geometry is shown in Figure 4.142.

Figure 4.142 Open Geometry File window



c) Geometry Editing

For this tutorial, use the **Tpipe.tin** (tetin file) geometry file from the input files supplied as mentioned above.

Mid Surface Model

The model currently has thickness and will eventually be modeled using thin shells. To do this, the model needs to be collapsed to a Mid-Surface representation.

Expand **Geometry** menu of the Model Tree by clicking on + sign besides Geometry menu. Turn **ON** Surfaces in display by clicking on button for Surfaces in Model Tree.

Click on (Create/Modify Surface) icon from **Geometry Tab Menubar**.

Enter Part as **MID** as shown in Figure 4.143 in **Create/Modify Surface** window and leave the Name blank.

Click on (Mid surface) icon. In the **Method** window select **By Surface**.

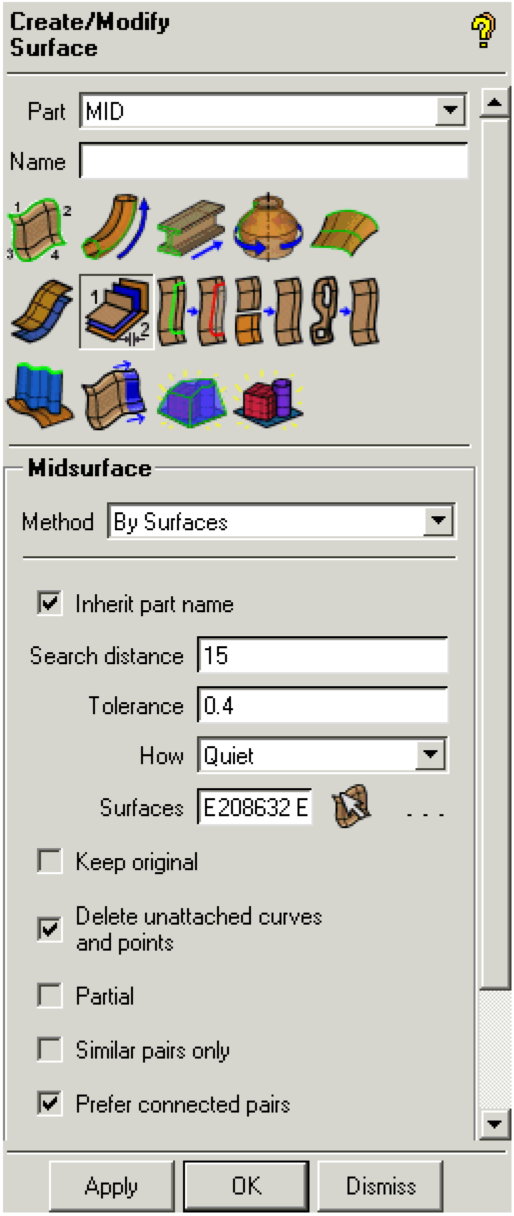
Enable **Inherit part name** enter **15** as the **Search distance,** in the **How** window select **Quiet**. Present object thickness is less than **15** units, hence this value is supplied.

Enter Tolerance of 0.4. Press 'Surf button' , select all the Surfaces using hotkey **'a'**

Enable Delete unattached Curves and Point and Prefer Connected Pairs press Apply.Rest of the setting is default as shown in Figure 4.143.

Note: The thickness can be measured using (Measure Distance) icon .

Figure 4.143 Create/Modify Surface Window



Note:

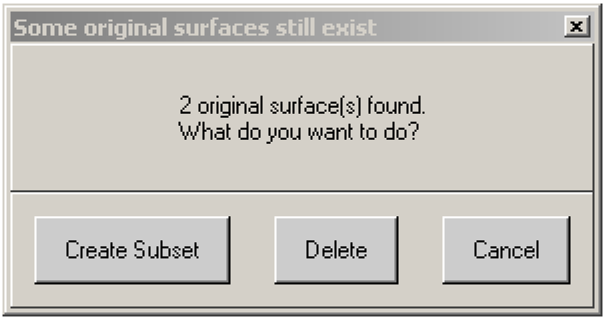
a) From the supplied Surfaces, **AI*Environment** will automatically try to find the pair of **Surface** within the supplied distance of **15** and will quietly create the Mid-Surface without asking any questions.

b) There are some hot keys defined for easy selection. For example “**a**” key selects all the entities of all parts and “**v**” selects only those displayed. Press “?” from keyboard to see all the available hot keys while in selection mode.

c) Because we have used Inherit Part name it will keep the Part name as T4 and will not change it.

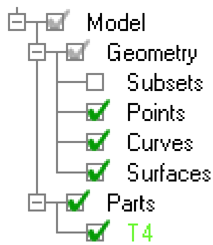
When it asks to ‘Delete’ some original Surface Press Delete as shown in Figure 4.144.

Figure 4.144
Some original Surface Exist



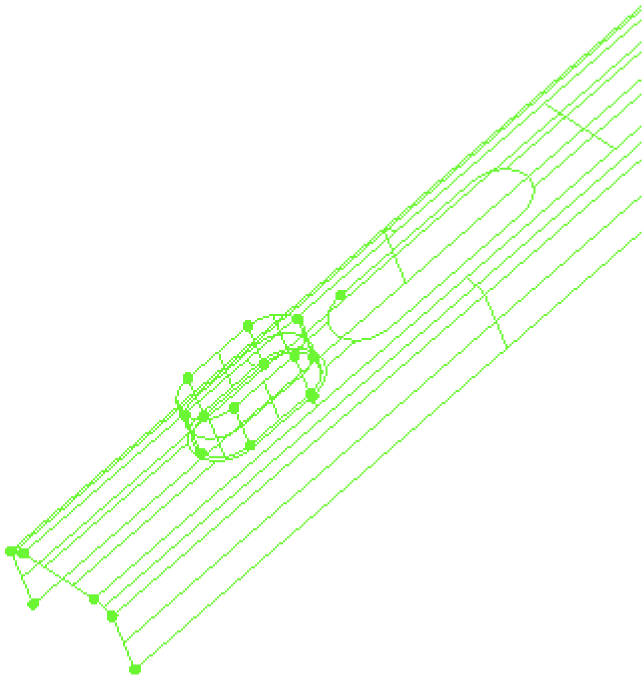
Try to make the Model Tree look similar to the one shown in Figure 4.145.

Figure 4.145
Model Tree display



The image after setting the display of Model Tree as above is shown in Figure 4.146

Figure 4.146
Geometry Display
Part

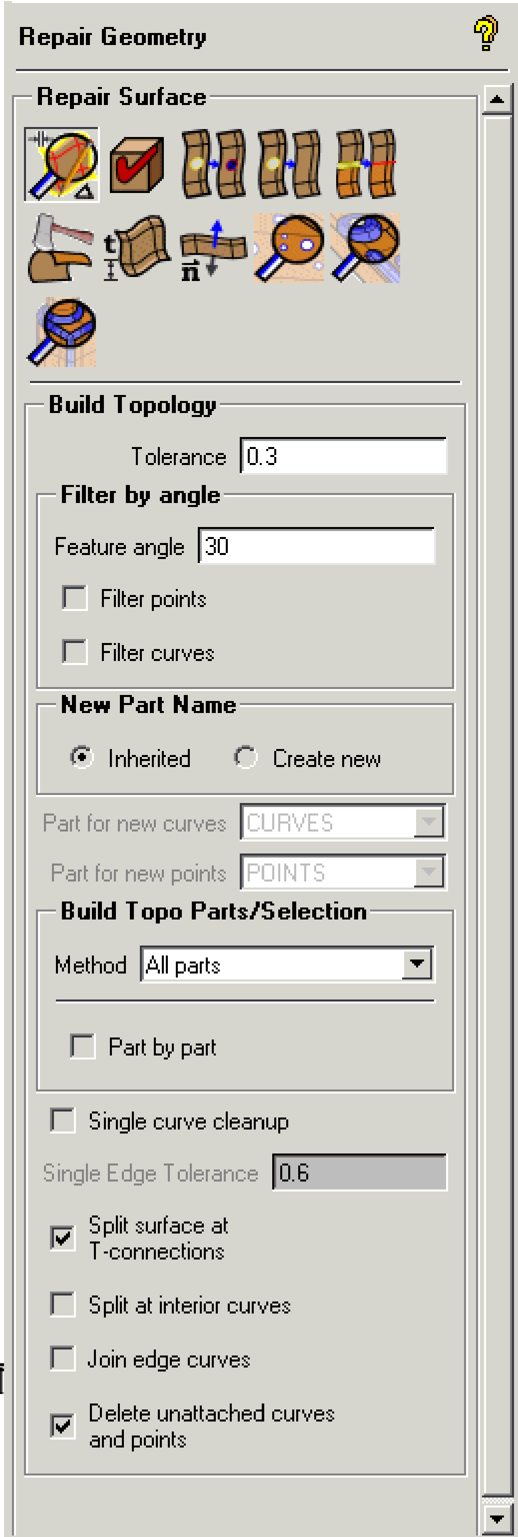


Remove Holes

The geometry should be studied to find whether it needs any repairing i.e. if it has any cracks or holes. The **Build Topology** function, located in the **Repair Geometry** window, extracts Curves and Points from the existing Surfaces and Deletes the un-necessary un-attached Curves and Points. The newly created Curves are Color-coded based on how many surfaces they are attached to and can be used for the purpose of model diagnosis and repair.

Click on (Repair Geometry)  icon from **Geometry Tab Menubar**. By default the Build Topology function  is highlighted. Make sure that **Inherited** (By default **Inherited** is **ON**) is toggled **ON** for **New Part Name** and in the Method select **All Parts** as shown in Figure 4.147 and press Apply to extract Curves and Points from the current Surface model.

Figure 4.147 Repair Geometry Window



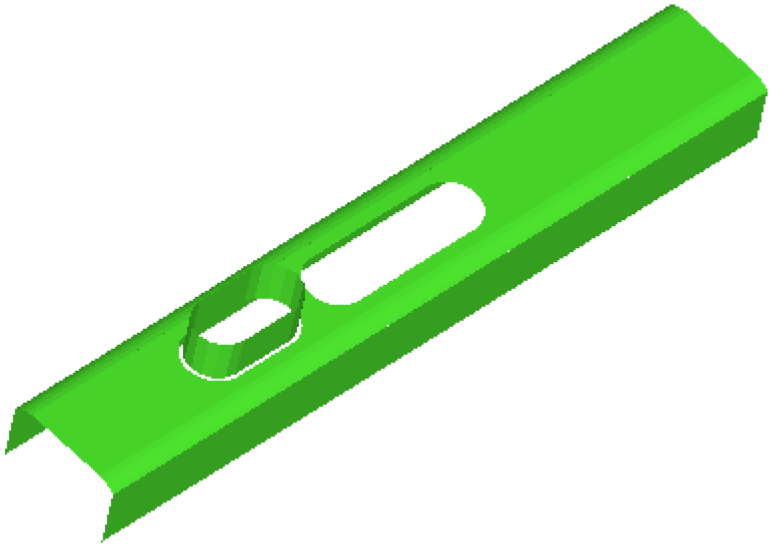
Note: Curves can be colored, and displayed by connectivity

Green = Unattached
Yellow = Single
Red = Double

Blue = Multiple

In the Model Tree, turn off **Points** and **Curves** and click the right mouse button on Geometry > Surfaces> Solid to display geometry modified so far as shown in Figure 4.148.

Figure 4.148 Geometry modified so far



Note: There is a minor Gap between the junctions of the two pipes. This can be filled up by two ways. Either

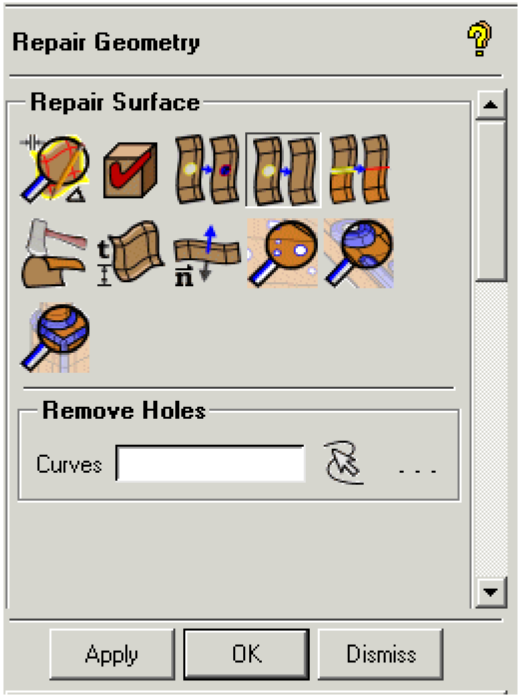
a) Fills the gap straight away

b) Removes the hole in the main Surface and then do the trimming based on the second pipe.

Choice (b) is selected for this Tutorial.

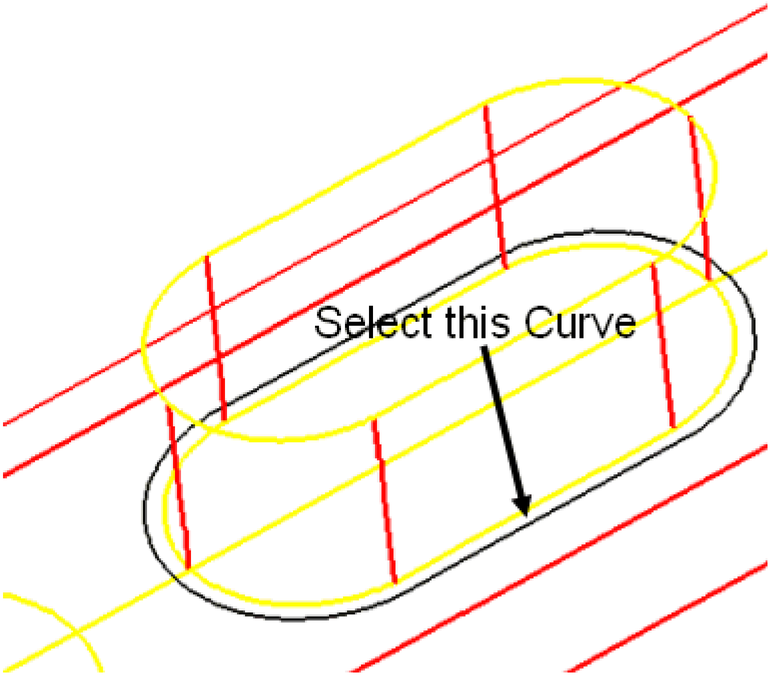
Click on  (Remove Holes) icon in the Repair Geometry window as shown in Figure 4.149.

Figure 4.149 Repair Geometry Window



This option needs curves at the boundary of a hole, so turn on **Curves** and turn off **Surface** from the Model Tree. Click on  (Select Curve(s)) button to select Curve for removing hole. Select the outer curve as shown in Figure 4.150 and press Apply to remove the hole.

Figure 4.150 Curve selection to Remove Hole



Switch On **Surfaces** in the Model Tree.

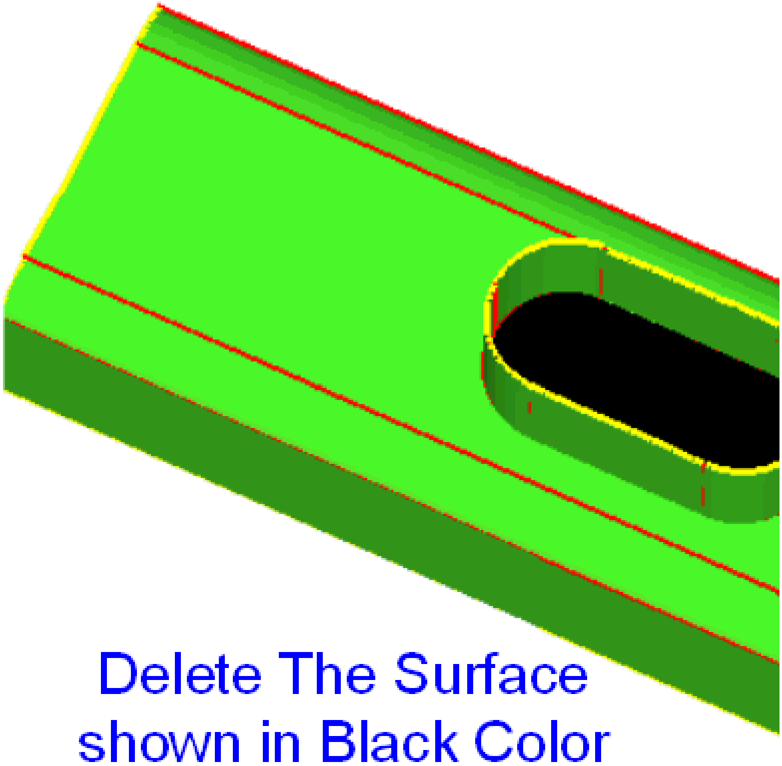
Re-intersection by Build Topology

Now, the main Surface needs to be segmented at the intersection of the two pipes. **Build Topology** automatically does that so run Repair Geometry >Build Diagnostic Topology with default values and press Apply.

Note: User can notice that the yellow curve there has turned blue now since it's attached to surfaces from more than 2 sides.

Click on (Delete Surface) icon from **Geometry Tab Menubar.** Click on (Select Surface(s) button to select surfaces to 'Delete'. Select the surface highlighted in Figure 4.151 and press Apply.This would remove the internal piece of the Surface, which is not required. The user can notice the changes in Color for the Curves around this surface from blue to red after deleting it.

Figure 4.151 Surface to be Deleted



Change the display of surfaces from solid to wire frame mode by clicking right mouse button on Geometry > Surfaces and select **Wire Frame** option from the Display Tree. The geometry after deleting the surface is shown in Figure 4.152.

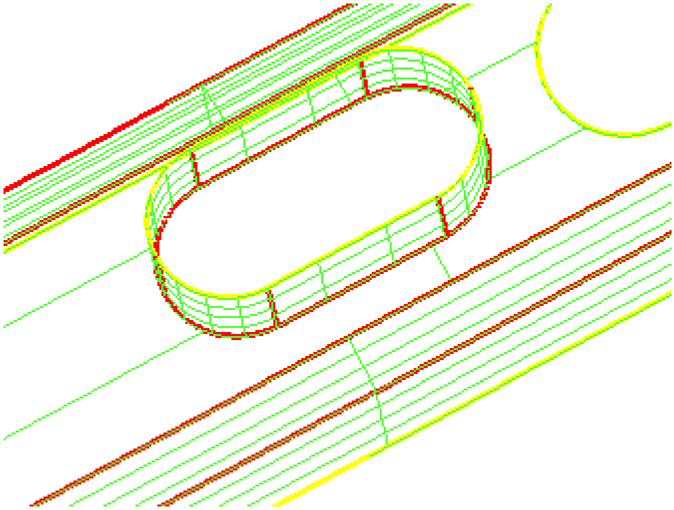


Figure 4.152 Geometry after deleting surface

d) Mesh Parameters and Meshing

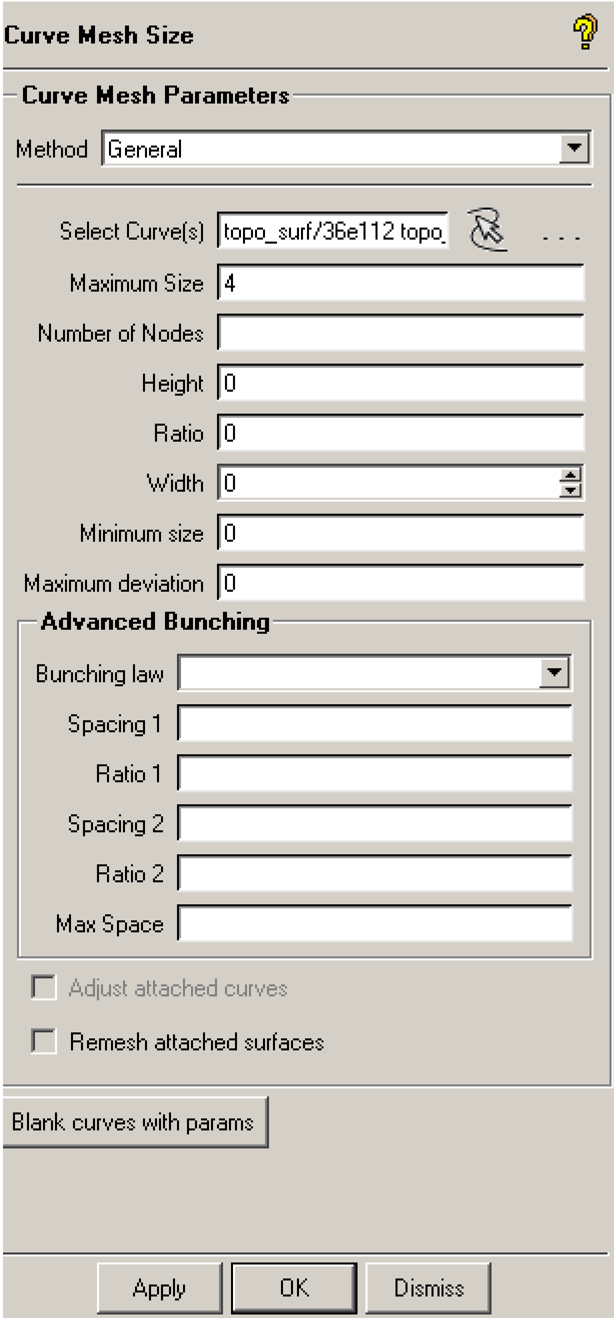
Since this is a shell model, Mesh Size information needs to be assigned to the curves. The mesh generated will be **Quad Dominant** i.e. it will have more number of QUAD elements than TRI elements. The mesh generated is associated with the geometry.

Mesh sizing

Select  (Set Curve Mesh Size) icon from **Mesh Tab Menu bar**, which pops up **Curve Mesh Size** window shown in Figure 4.153.

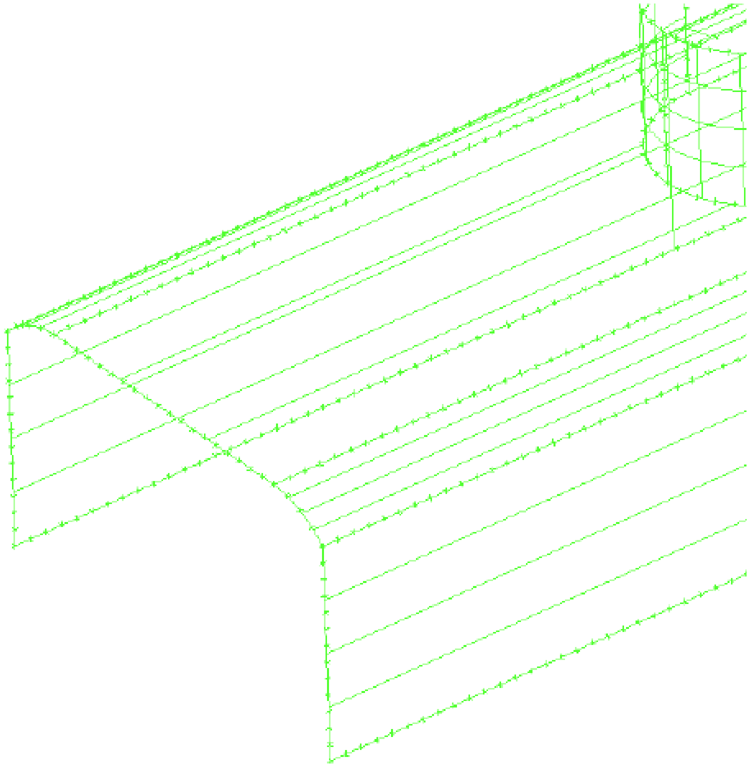
In the Method select 'General', click on  (Select Curve(s)) button to select Curves. Place the mouse cursor in display window and press "**a**" from keyboard to select all curves. Enter **Maximum Size** of **4** for this case and press Apply.

Figure 4.153 Curve Mesh Size window



Now, in the Model Tree**,** place the mouse cursor on **Curves**, press right button and select **Curve Node Spacing.** Also de-select **Color by count** and **Show wide** which will show the Curve Node Spacing in Figure 4.154.

Figure 4.154 Curves with Node Spacing ON



Now de-select Curves > Curves Node Spacing under the the Model Tree.

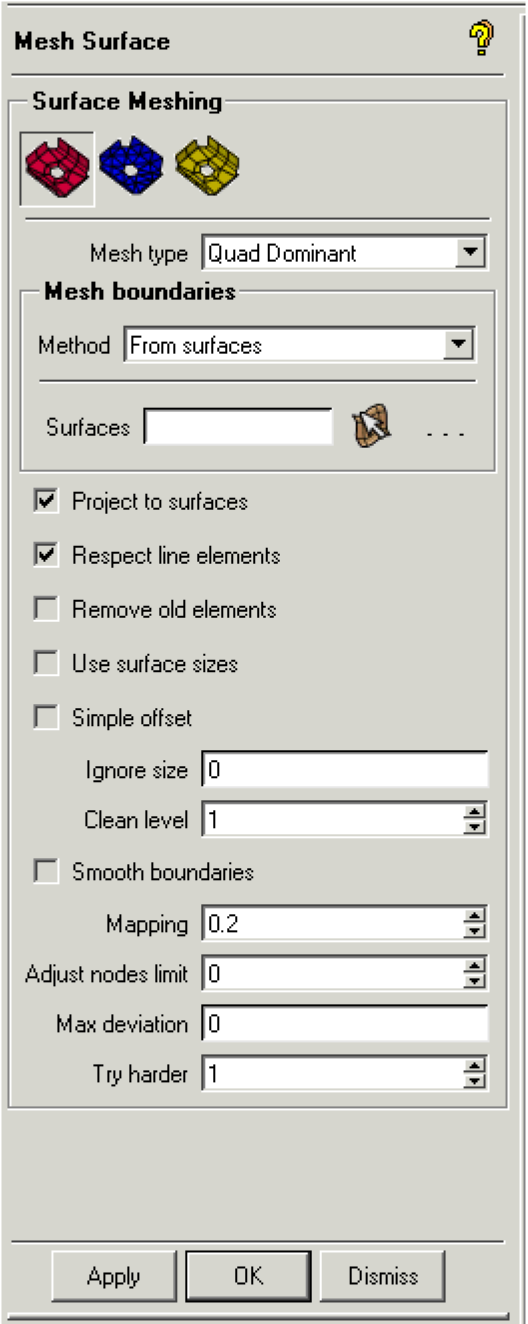
Meshing

Select the (Mesh Shell)  > Patched Based  icon from **Mesh Tab Menu bar**. By default, patch based surface meshing is in Quad dominant mode. Select the option From Surfaces.

Enable Respect Line element and press Apply with default settings as shown in Figure 4.155.

Note: If Surfaces are not selected then it considers all the Surfaces.

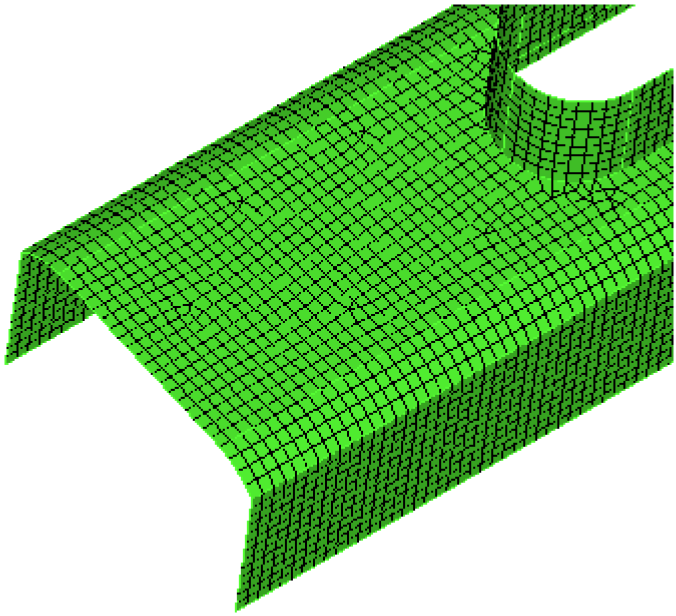
Figure 4.155 Mesh Surface window



Turn off Geometry branch in the Model Tree

In the Model Tree, expand the Mesh branch of the tree by clicking on the +. Click the right mouse button on **Shells** and select **Solid & Wire,** the mesh appears as shown in Figure 4.156.

Figure 4.156 Mesh in Solid & Wire Frame mode

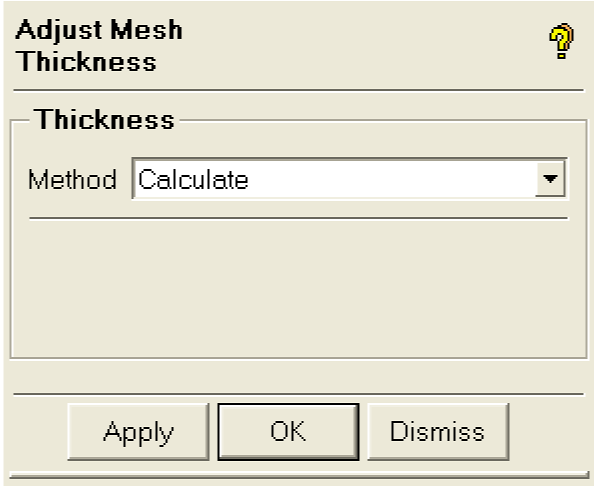


Assigning Shell Thickness

For analysis purpose it is better to assign the thickness to the shell.

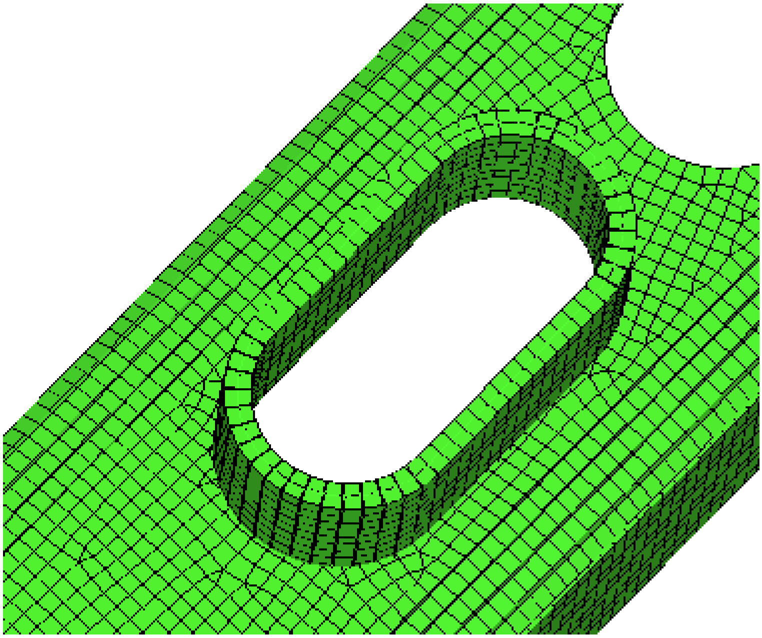
Select (Assign Mesh Thickness) icon from **Edit Mesh Tab Menubar**, which pops up **Adjust Mesh Thickness** window shown in Figure 4.157. From **Method**, select Calculate and press the Apply. It will automatically calculate the original thickness of the geometry and assigns it to the mesh

Figure 4.157
Adjust Mesh Thickness Window



Note: If the usres wants to see the assigned mesh thickness Click the right mouse button on Mesh > Shell and select **Shell Thickness.** The Mesh will appear as seen in Figure 4.158.

Figure 4.158 Geometry Showing Mesh Thickness



Now to turn off Mesh>Shell>Shell Thickness from Model Tree.

e) Material and Element Properties

Material for this model is **STEEL**. So the properties like Young's modulus, Poisson's ratio and Density should be defined. Since the original geometry had a thickness while we have resolved only the mid-surface, the shell thickness also needs to be defined.

Selection of Material

Select  (Create Material Property) icon from **Properties Tab Menubar**.

Define the Material Name as **STEEL** and supply the required parameters for it in the **Define Material Property** window as shown in Figure 4.159.

Material ID can be left as **1**.

Select Type as **Isotropic** material,

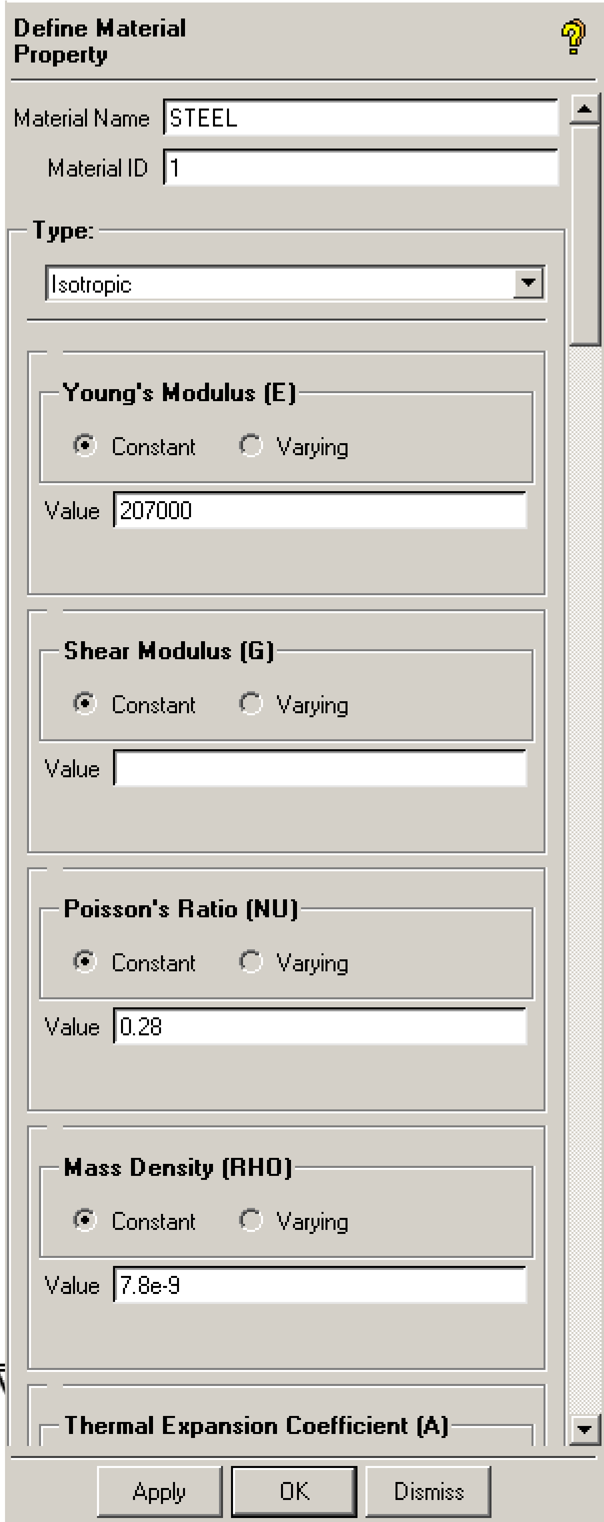
Define Young’s modulus as **207000**,

Define Poisson's ratio as **0.28**,

Define Mass Density as **7.8e-9**, and leave other fields as default.

Press Apply.

Figure 4.159 Define Material Property window



Element Properties

Select (Define 2D Element Properties) icon from the **Properties Tab Menu bar**.

Select Part as T4 for applying property.

Set PID as **1** in the **Define Shell Elements** window as shown in Figure 4.160.

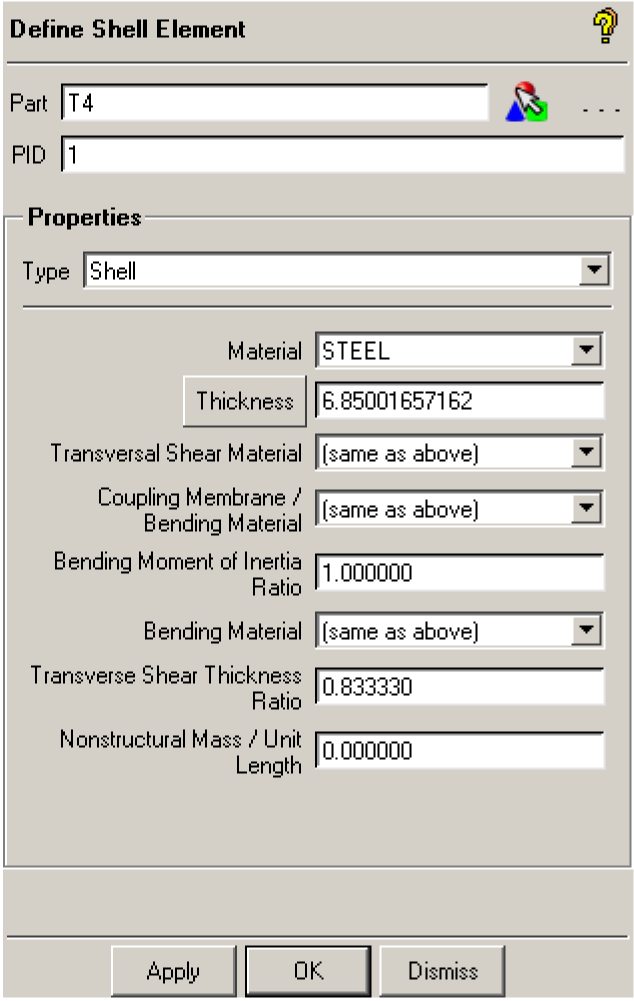
In the **Type** window select Shell

Thickness comes by default

Select Material as STEEL.

Press Apply.

Figure 4.160 Define Shell Element window



f) Solver Setup

Modal analysis is to be carried out on this model, so this has to be setup for Nastran and write an input file for NASTRAN.

Setup Nastran Run

First, user should select the appropriate solver before proceeding further.

Select Settings > Solver from Top Menubar and select appropriate solver viz. Nastran and press Apply as shown in Figure 4.161.

Figure 4.161 Solver selection

Setup Solver Parameters

Click on  (Setup Solver Parameters) from **Solve Options Tab Menu bar** that will open Setup Solver Parameters window.

Select Solver Parameter as Eigen Value Extraction (EIGR/EIGRL).

Select Type as **EIGRL**.

Note: EIGRL is Real Eigenvalue Extraction Data, Lanczos Method.

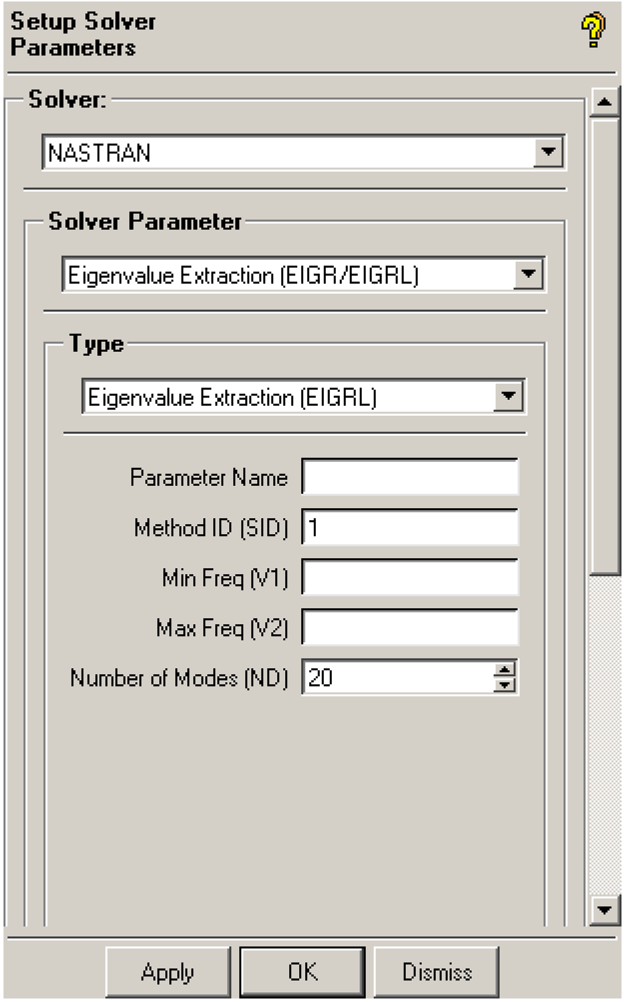
Set **Number of Modes** to 20. Min and Max could be defined to limit this, however in general it is easier to let Nastran just return first 20 frequencies.

Leave the other parameters as Default as shown in Figure 4.162.

Press Apply.

A default Subset by the name of EIGRL1 is created under Parameters in the Model Tree.

Figure 4.162 Setup Solver Parameters window



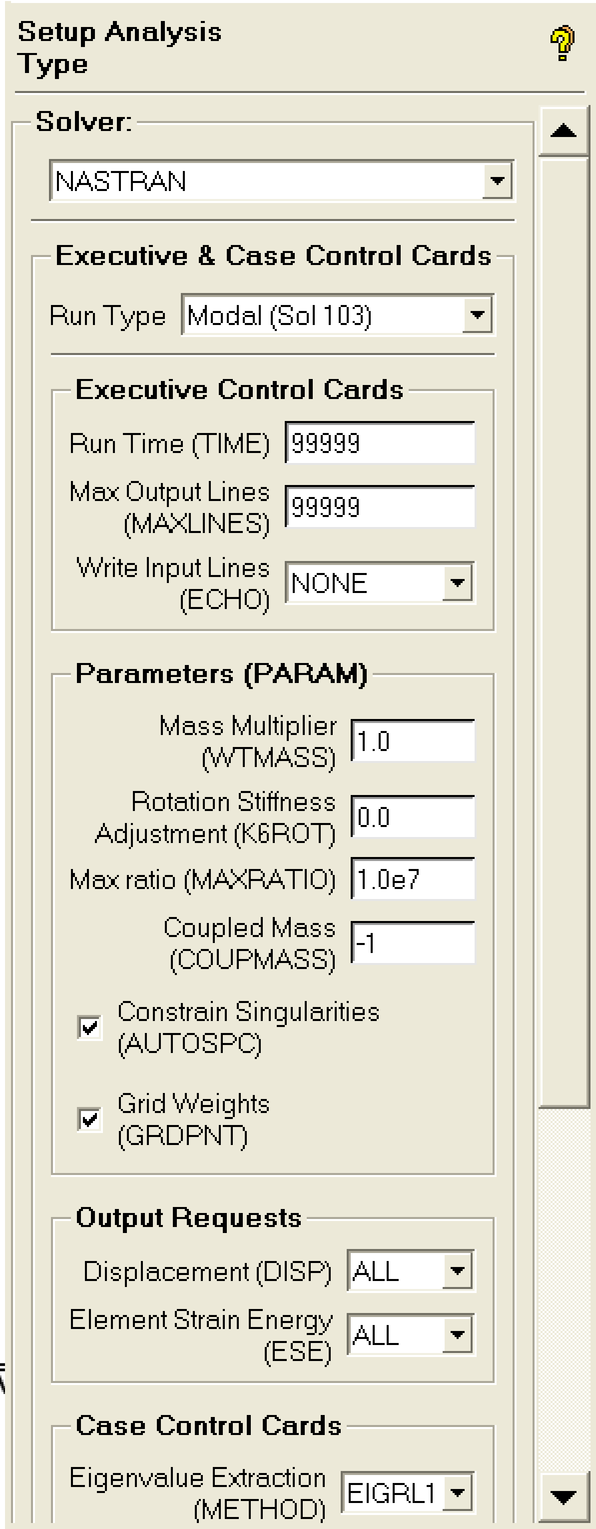
Click on  (Setup Analysis Type) icon from **Solve Options Tab Menu bar** to setup Nastran run to do Modal Analysis that pops up **Setup Analysis Type** window shown in Figure 4.163.

Select Run Type as **Modal (Sol 103),**

Select **ALL** in Output Requests section for Displacement (DISP) and Element Strain Energy (ESE). Also select the Case Control Cards as **EIGRL1**.

Press Apply.

Figure 4.163 Setup Analysis Type window

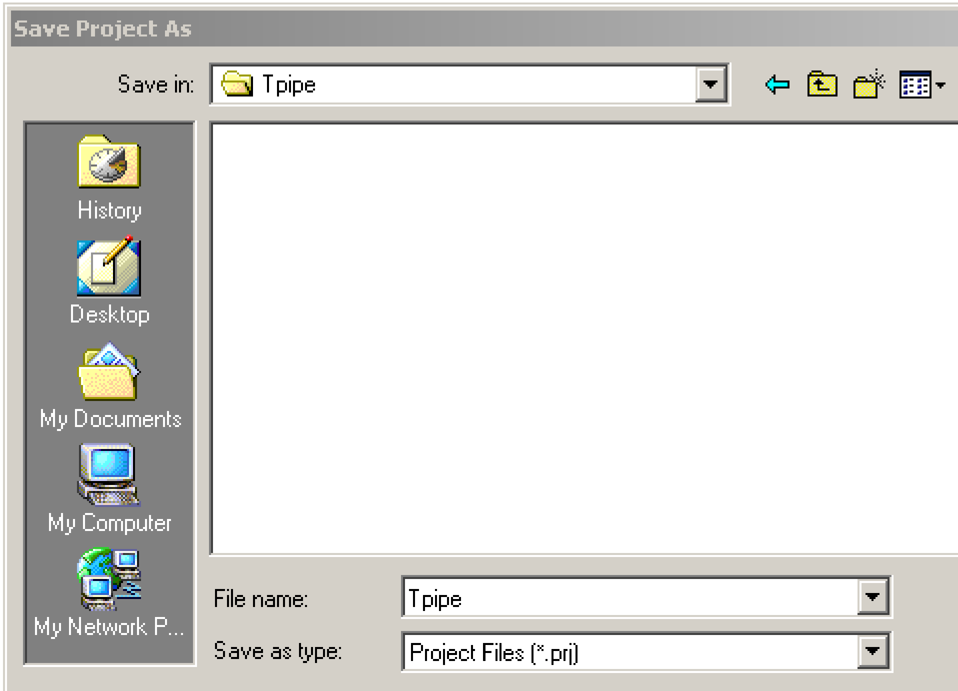


Save Project

Select File > Save Project As option from the Main Menu and click on 'Create New Directory' icon and enter folder name as Tpipe as shown in Figure 4.164.

Now enter the project name as **Tpipe** as shown in Figure 4.164 to save all the information in Tpipe directory.

Figure 4.164 Save Project in a New Directory window



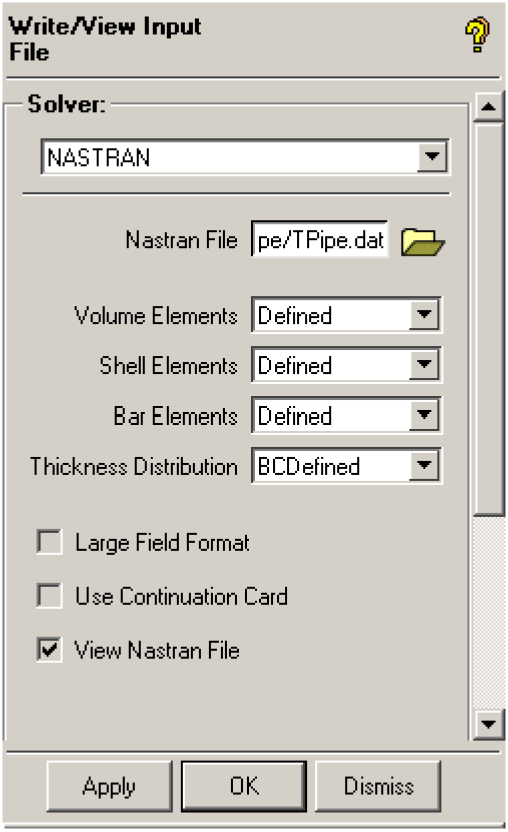
It saves additional fours files Geometry file, Mesh file, Attribute file and Parameter files as Tpipe.tin, Tpipe.uns, Tpipe.fbc and Tpipe.par respectively along with the project file, Tpipe.prj.

g) Write Nastran Input File

Click on (Write/View Input File) icon from **Solve Options Tab Menu bar**.

Feed the Nastran file name as Tpipe.dat and switch **'On'** View Nastran file as shown in Figure 4.165 and press Apply in **Write/View Input File** window.

Figure 4.165 Write /View Input File Window



User will see that the Nastran input data file comes up in the default text editor. If user likes to edit this file directly then this can be done and can save the edited file through this text editor. Since this example needs no editing, just close the editor.

h) Solution and Results

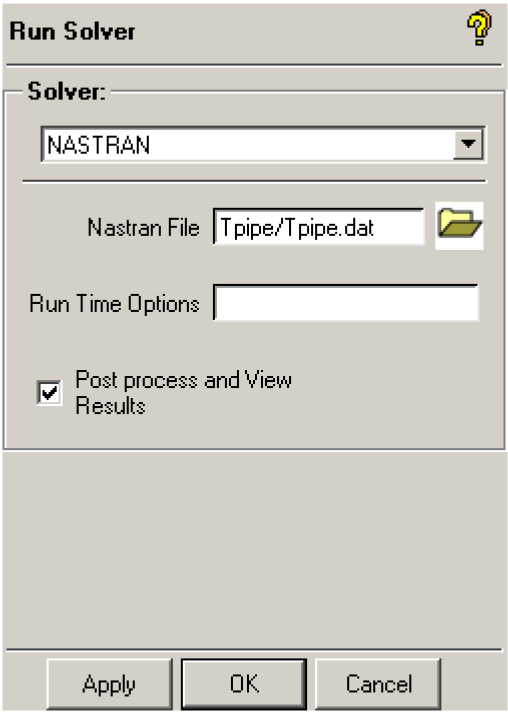
Modal analysis is to be performed on this model and the results should be visualized in a post processor.

Solving the problem

Click on (Submit Solver Run) icon from the **Solve Options Tab Menubar** to start the Nastran as shown in Figure 4.166. The Nastran file will be selected by default as **Tpipe.dat**.

Toggle **ON** Post process and View Results and press Apply in **Run Solver** window.

Figure 4.166 Run Solver window



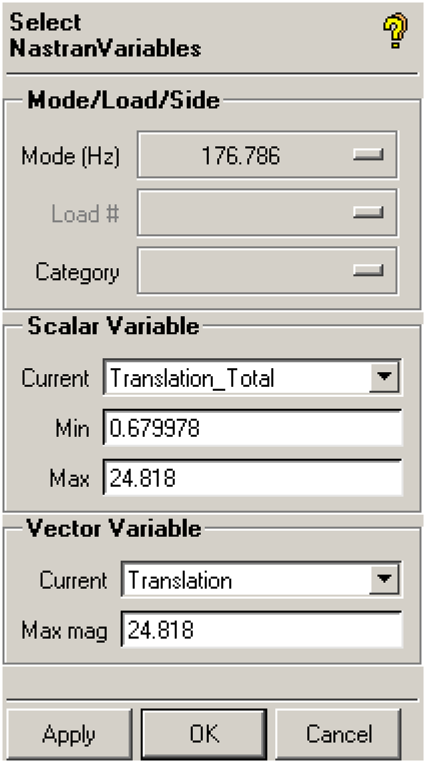
Note: If Nastran doesn't go through, Please refer the **FATAL** error in the file *.f06 and fix the issues accordingly in the dat file (*.dat).

Post Processing of Results

After completion of Nastran run, the results will be automatically loaded into the Post Processor tab.

Click on  **Variables** option in **Post-processing** Tab menu bar. In Select Nastran Variables window, select **Category** as **Solid** and Current scalar variable as **Translation_Total** as shown in Figure 4.167.

Figure 4.167 Select Nastran Variables window



Note: MSC Nastran run obtains Results shown here. Results may differ with those of AI*Nastran run depending on the version.

To display mode shape at Total Translation Frequency, select **Category** as **Displacement** and Current Scalar Variable as **Translation_Total** in Select NastranVariables window as shown in.

Select  (Control All Animation) option from **Post-processing** tab menu bar which will open Animation Controller window as shown in Figure 4.168.

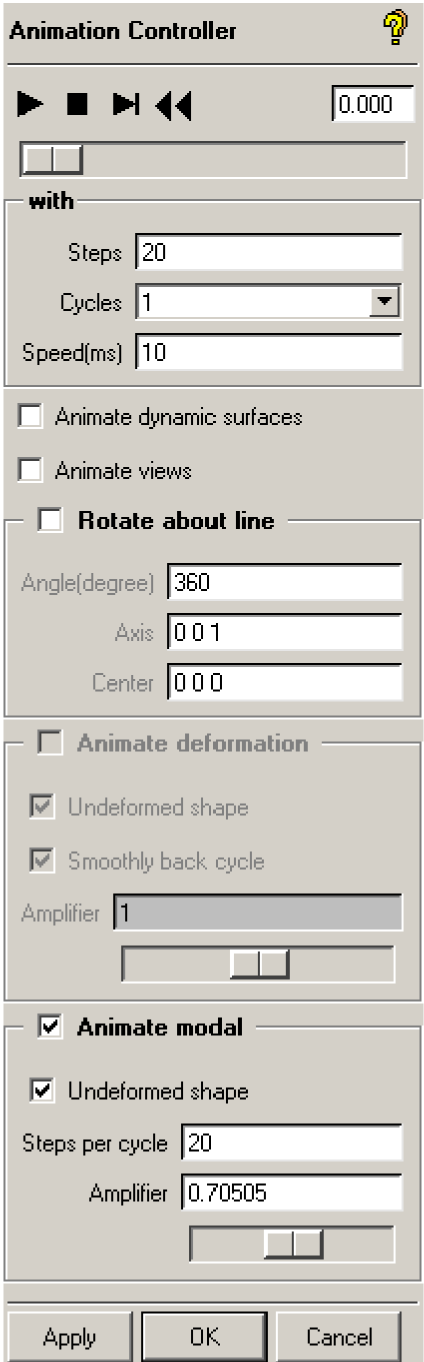
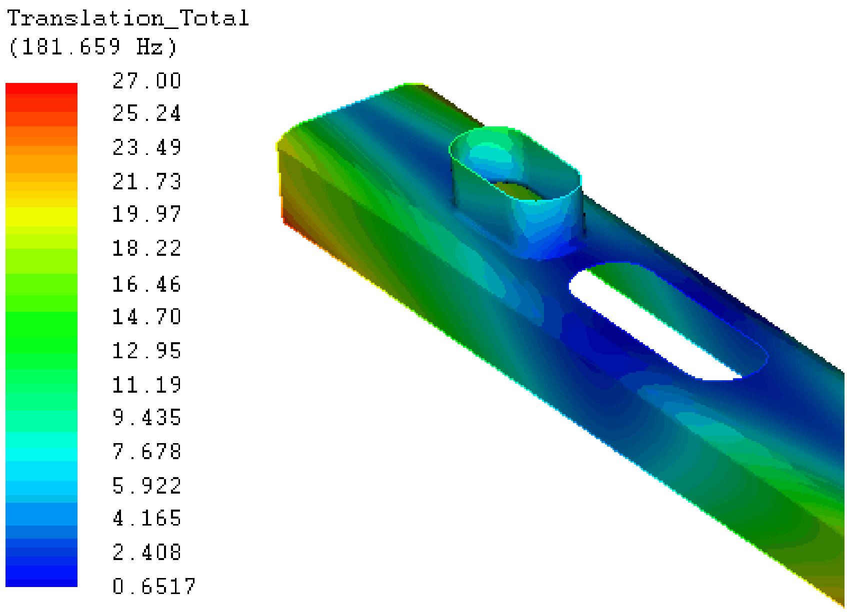


Figure 4.168 Animation Setup and Controller window

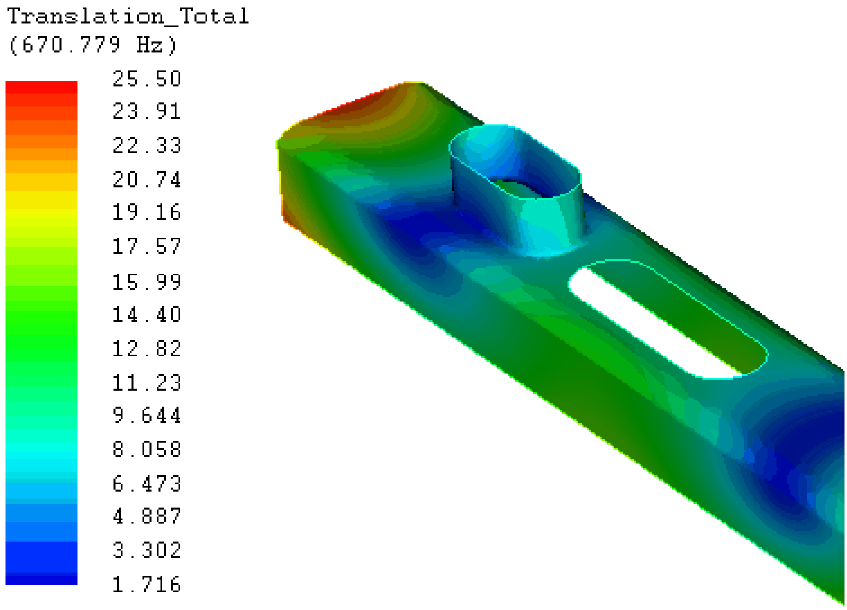
Set the values as shown in Figure 4.168 and press (Animate) to view the mode shape as shown in. Figure 4.169 &Figure 4.170

Figure 4.169 Animated model at 181.659 Hz



Finally select Exit to quit the post processor.

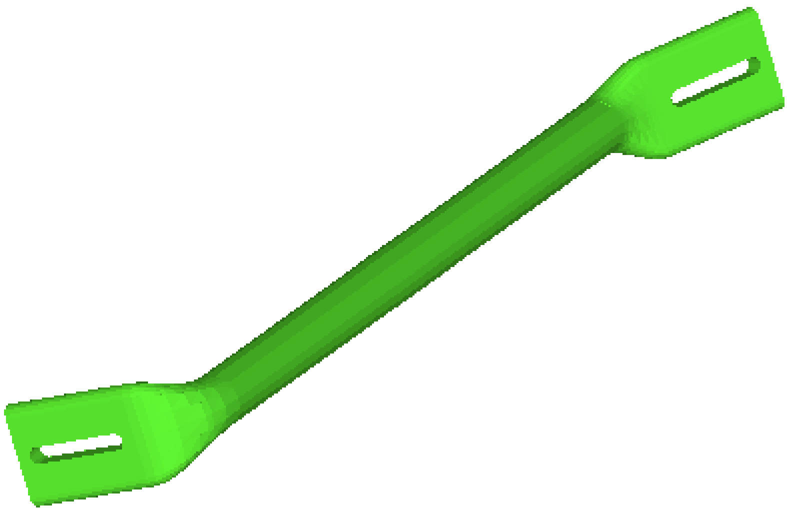
Figure 4.170 Animated model at 670.779 Hz



4.3.2: Bar

This exercise explains Tetrahedral meshing of bar geometry, writing the input file to solve this Linear Static problem in Nastran and post processing the results the geometry is shown in Figure 4.171.

Figure 4.171 Bar Geometry



a) Summary of Steps

Launch AI*Environment and load geometry file

Geometry Editing

Repair

Mesh Parameters and Meshing

Mesh Sizing

Meshing

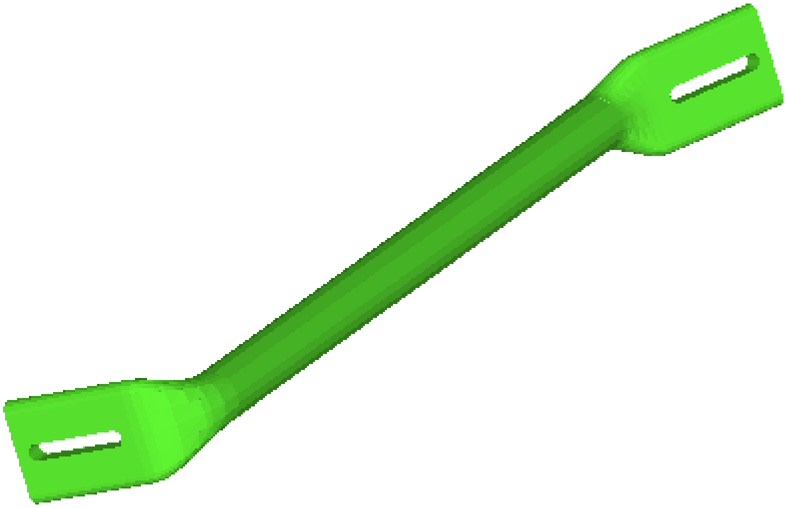
Material and Element Properties

Selection of Material

b) Launch AI*Environment

Launch the AI*Environment from UNIX or DOS window. Then File > Change working directory, $ICEM_ACN/../docu/FEAHelp/AI_Tutorial_Files. Load the tetin file 'Bar.tin'.

Figure 4.172 The Bar geometry



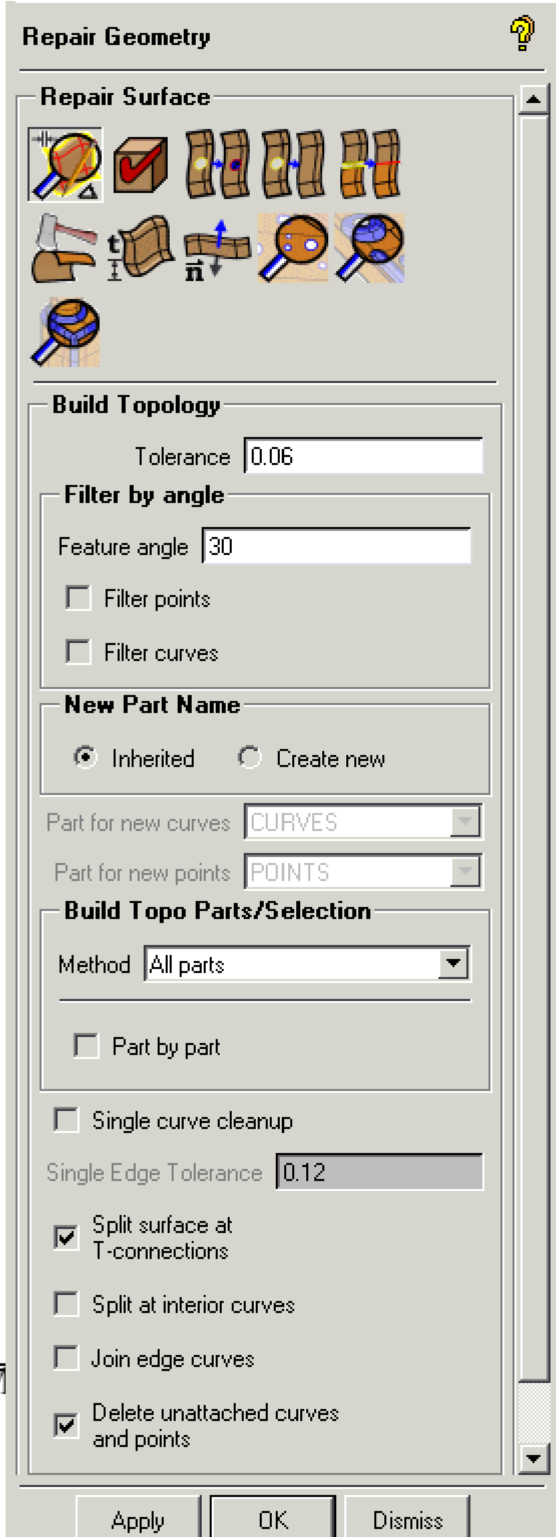
c) Geometry Editing

Repair

Expand the Geometry branch of Model Tree and turn Surfaces on.

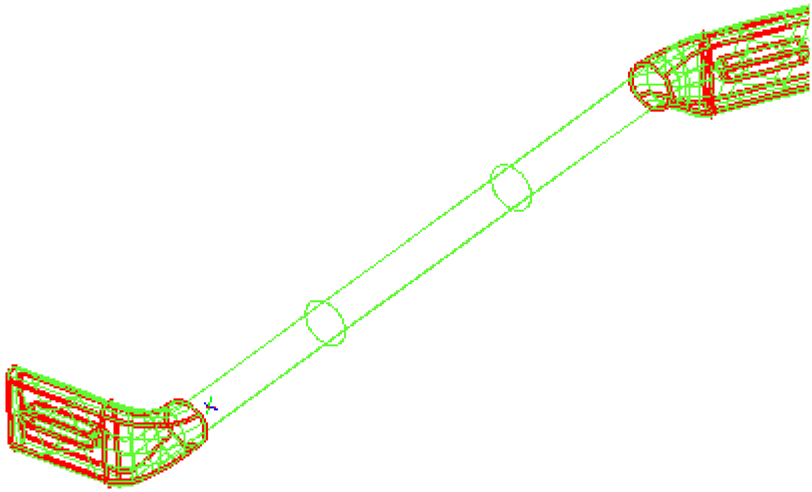
Click on  (Repair Geometry) icon from Geometry Tab Menu bar, which will pop up **Repair Geometry** window as shown in Figure 4.173. By default Build Topology  is highlighted. Make sure that Inherited is toggled **ON** for New Part Name and press Apply to extract curves and points.

Figure 4.173 Repair Geometry window



The geometry will be displayed in the Main Display window as shown in Figure 4.174.

Figure 4.174 Geometry after Build Topology



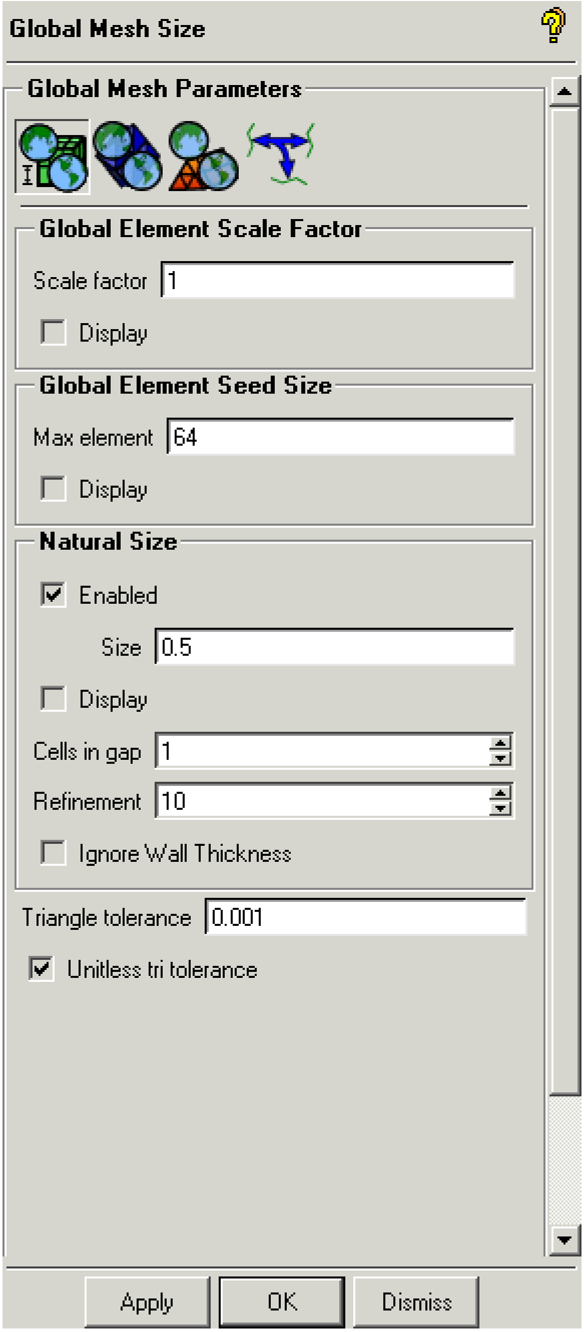
d) Mesh Parameters and Meshing

Since this is a 3D model, the mesh will be a volumetric one. So the mesh parameters should be given on Surfaces.

Mesh Sizing

Select  (Set Global Mesh Size) icon from **Mesh Tab Menubar** and toggle **ON** Enabbled under the Natural size window and enter a value of **0.5** for **Size** window and leave all other fields as default in **Global Mesh Size** window as shown in Figure 4.175 and press Apply.

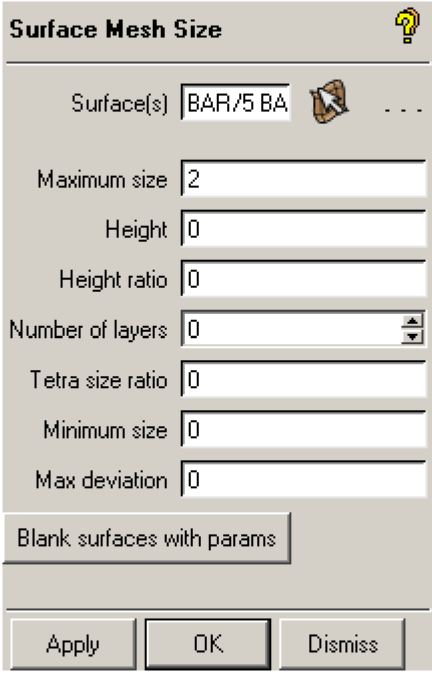
Figure 4.175 Global MeshSize window



Select (Set Surface Mesh Size) icon from Mesh Tab Menubar, which will pop **Surface Mesh Size** window as shown in Figure 4.176.

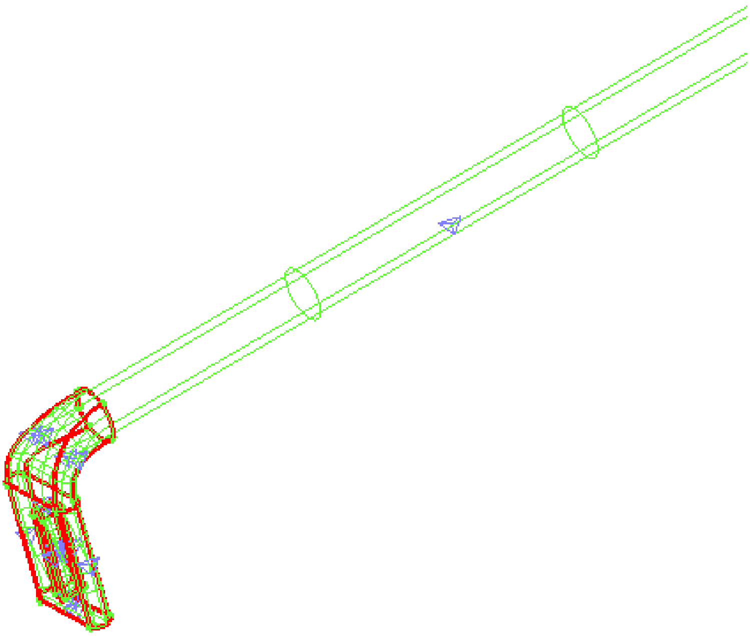
Click on (Choose an item) and select all the surfaces by pressing “**a**” (ensure that the mouse cursor is in display window). Enter **Maximum Size** of **2** as shown in Figure 4.176 and press Apply.

Figure 4.176 Surface Mesh Size window



Now in the Model Tree, place the mouse cursor on Geometry > Surfaces and press right button and select **Tetra Sizes** which will show tetra sizes of the surfaces as shown in Figure 4.177 and then turn ‘off, the **Surface Tetra Sizes** by deselecting Surface > Tetra Sizes in Model Tree.

Figure 4.177 Surfaces with Tetra Sizes ON



Meshing

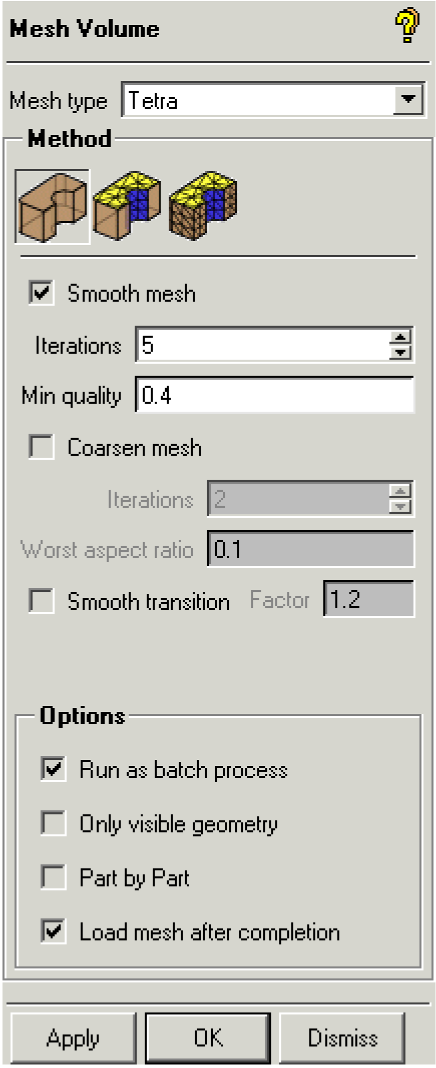
Select  (Mesh Tet) icon from Mesh Tab Menubar.

Before launching the options for the tetrahedral meshing, it will ask to save the changes done so far to the project and invokes the Save Project As window. Click on ‘Create New folder’ icon and enter directory name as **‘Bar’.** Supply **Bar** as the project name and press ‘**Save’**

Along with the Bar.prj -project file, it will save other files in Bar directory for geometry and boundary condition as Bar.tin and Bar.fbc respectively. Once the project file is saved, the options for the tetrahedral meshing can be reached through **Mesh with Tetrahedra** window shown in Figure 4.178.

Enable Run as Batch Process and Load mesh after completion.

Figure 4.178 Mesh with Tetrahedral window

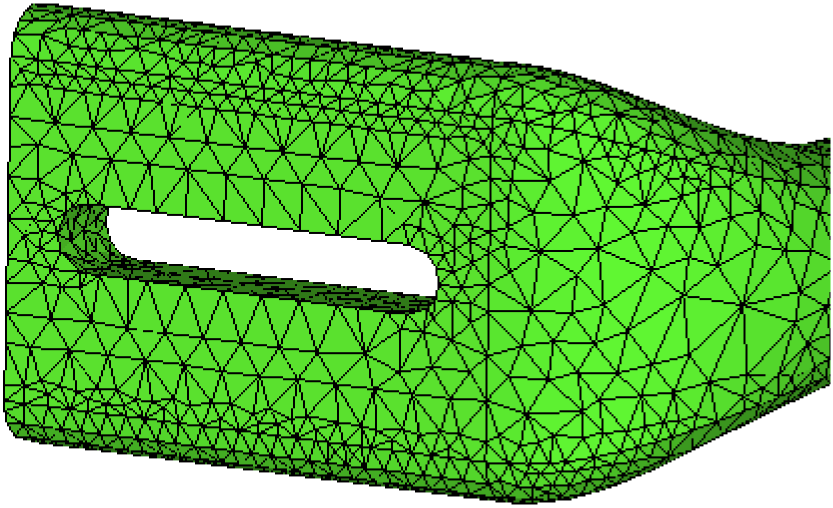


Press Apply to start meshing.

Switch off Geometry from the Model Tree.

In Display Tree, click on + to expand the Mesh menu. Click the right mouse button on **Shells** and select **Solid & Wire**. Now, the mesh should look as shown in Figure 4.179.

Figure 4.179 Mesh in Solid & Wire mode



e) Material and Element Properties

After getting the mesh, the material and element properties should be defined for the model as follows:

Selection of Material

Select  (**Create Material Property**) icon from Properties Tab Menubar.

Define the Material Name as STEEL in **Define Material Property** window shown in Figure 4.180.

Material ID can be left as **1**,

Select **Isotropic** as the type of the Material,

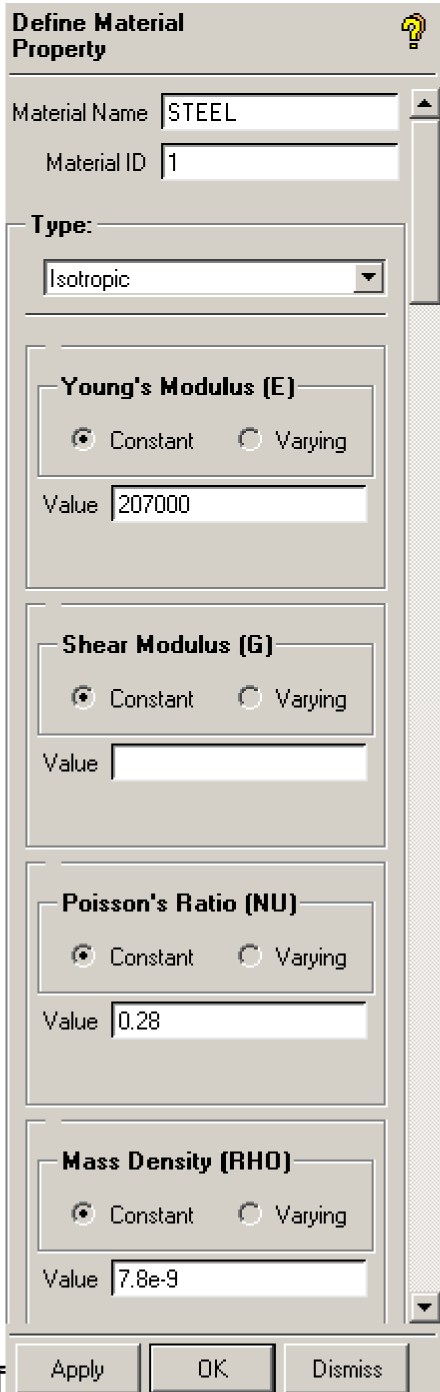
Define Young's modulus as **207000,**

Define Poisson's ratio as **0.28**,

Define Density as **7.8e-9**,

Leave other fields as it is and Press Apply.

Figure 4.180 Define Material Property window



f) Element Properties

Select  (Define 3D Element Properties) from the Properties Tab Menubar.

Set PID as **1** in the **Define Volume Element** window as shown in Figure 4.181.

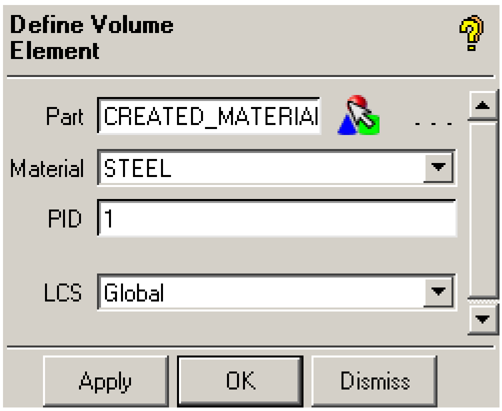
Select Part as CREATED_MATERIAL_2,

Note: The part CREATED_MATERIAL_2 contains the volume mesh, which is automatically generated during Tetra meshing. The number 2 in the part name CREATED_MATERIAL_2 is a random number, which might vary with each run of Tetra mesher. Select the appropriate part based on the present run.

Select material as **STEEL,**

Press Apply.

Figure 4.181 Define Volume Element window



g) Constraints and Loads

To map the real system of geometric model, relevant constraints and loads should be applied on model. This can be done as follows:

Constraints

Click on (Displacement on Surface) icon from the Constraints Tab Menu bar, which will pop up Create **Displacement on Surface** window as presented in Figure 4.182.

As the Constraints has to apply on the Surface, switch off Shells option of Mesh menu in the Display Model Tree and Switch on Surfaces.

In **Create Displacement on Surface** window enter Name as **CNST1**.

Toggle **ON** all options UX, UY, UZ, ROTX, ROTY and ROTZ and select the surfaces shown in Figure 4.183 and Figure 4.184, and press Apply. From Displacement branch, right mouse click and select show all. This will show the constraint symbols placed on the target surfaces, Figure 4.184,

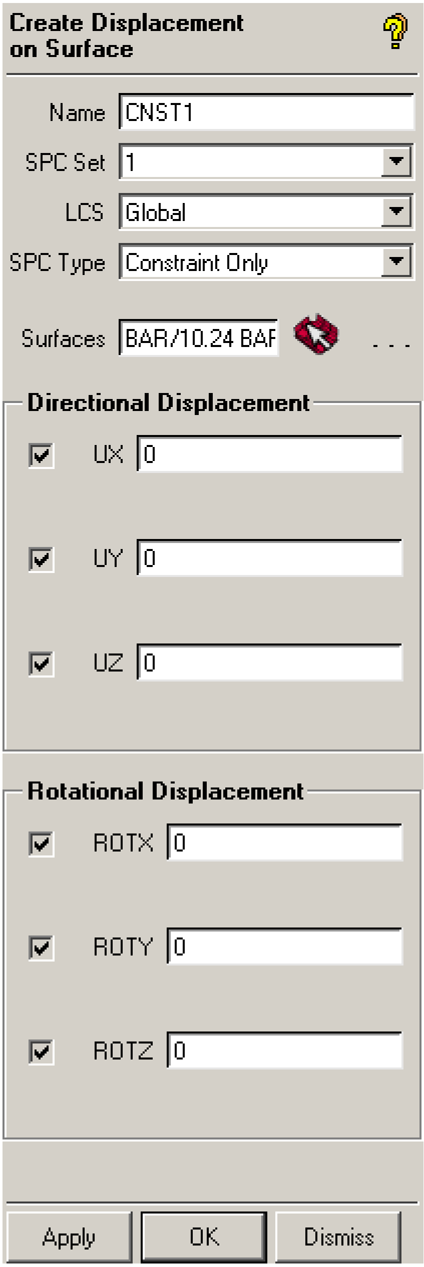


Figure 4.182 Create Displacement on Surface window

Figure 4.183 Surfaces for Displacement and Loads

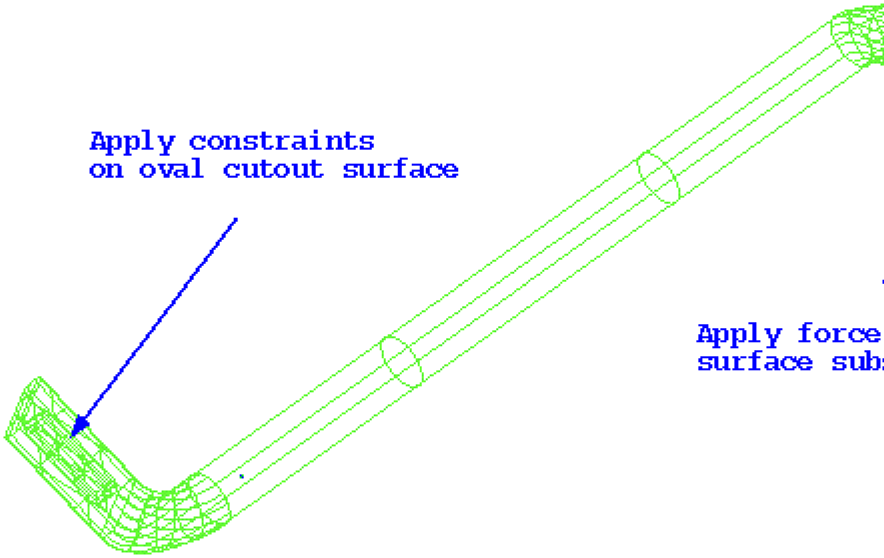
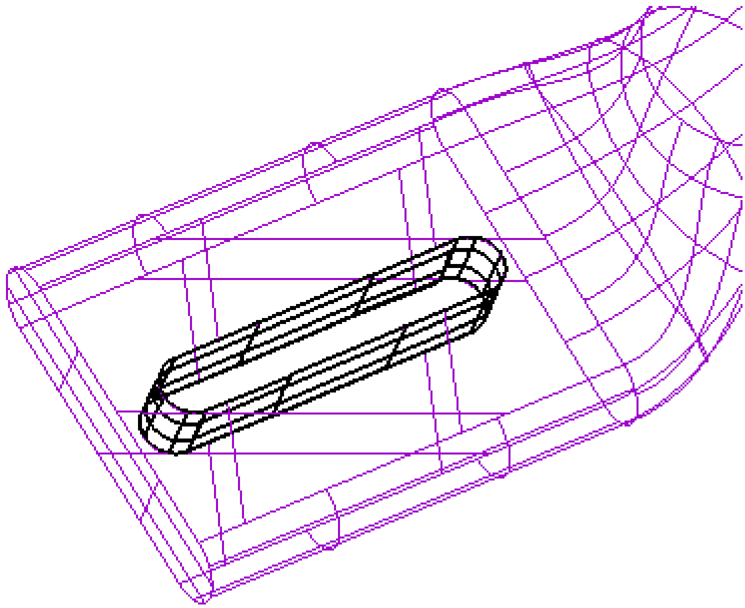
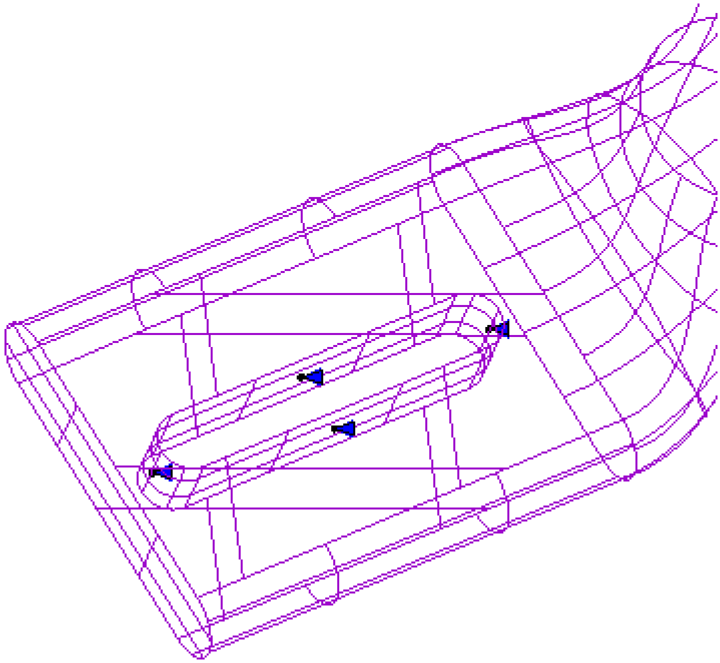


Figure 4.184 Surfaces for Displacement and after applying the Displacement





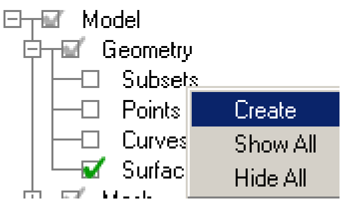
h) Loads

1) Create Subset

To apply the load on surface/surfaces, a subset should be created which contains these surfaces.

In Model Tree, right mouse button on Geometry > Subsets to select Create as shown in Figure 4.185.

Figure 4.185
Create Subset

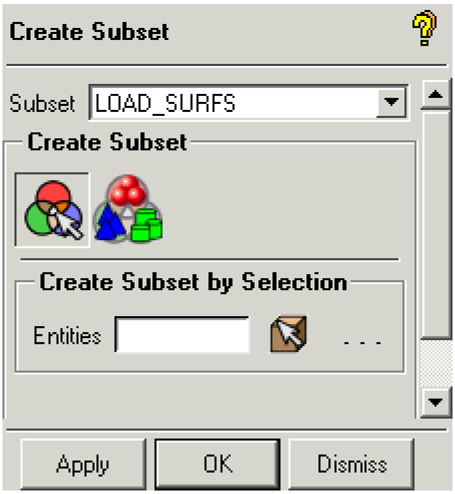


Note: Even the Subset is switched ‘**Off’** in the Display Tree it does’nt alter the appearance of the Create Subset window.

This **Create Subset** window pops up as shown in Figure 4.186.

In this window, enter Subset as **LOAD_SURF** and click on  (Create Subset by Selection) icon. Click on  (Select Geometry) button and select the surface as shown in Figure 4.187 and press Apply. This creates the subset **LOAD_SURF**.

Figure 4.186 Create Subset window



Switch ‘**Off’** Geometry>Subset in the Model Tree.

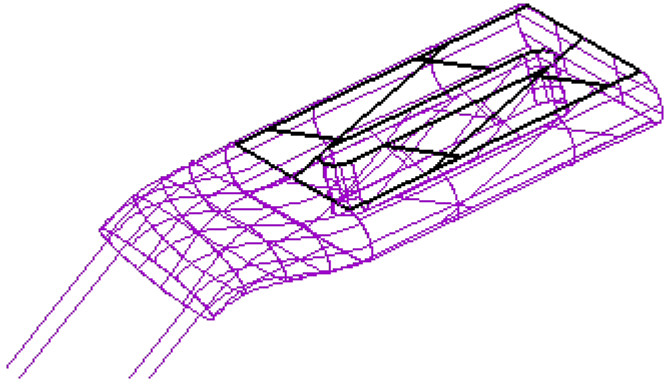


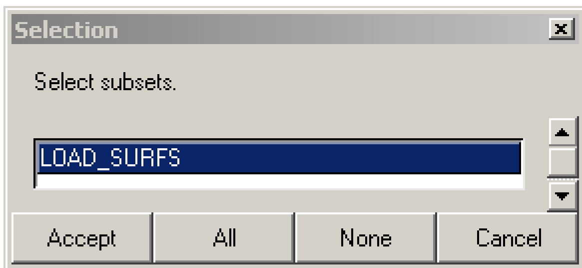
Figure 4.187 Surface for subset

2) Force on subset

Click on  (Force on Subset) icon from **Loads Tab Menubar**, which pops up **Create Force on Subset** window. In this window enter Name as **FORCE.** Enter values of **FX** as 0.467, **FY** as 0.2 and **FZ** as-0.862, it is shown in Figure 4.189.

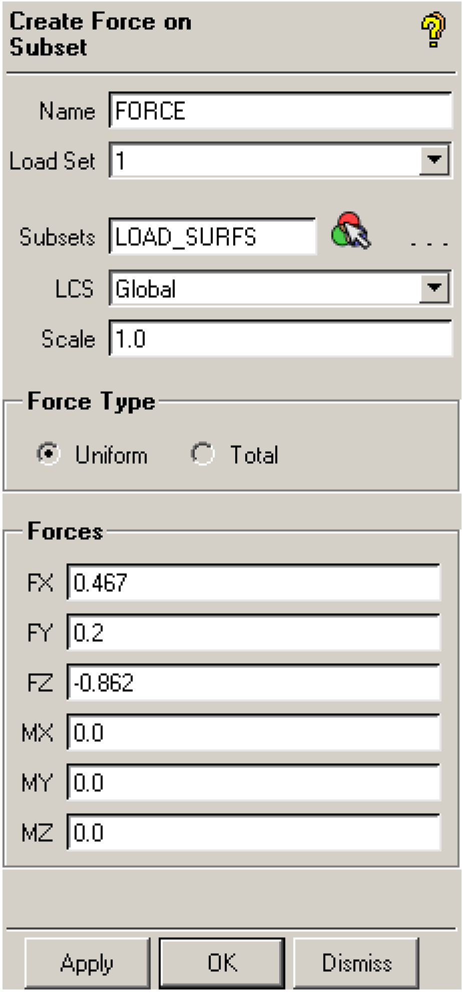
Press Select Subset  In the **Selection** window select LOAD_SURFS as shown in Figure 4.188 press Accept in the Selection window.

Figure 4.188 Selection window for Subset



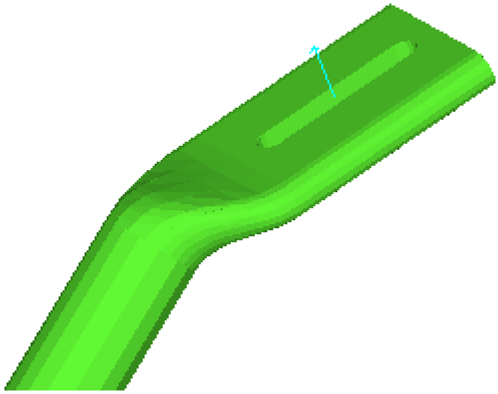
Press Apply in the Create Force on Subset window.

Figure 4.189 Create Force on Subset window



Note: If the user wants to view the applied Force as shown in Figure 4.190, he can do so by Swiching Loads **'On'** in the Model Tree.

Figure 4.190 Force applied



i) Solver Setup

On this model, linear static analysis is to be performed in Nastran, so parameters and variables should be defined accordingly. This can be done as follows:

Setup Nastran Run

First, user should select the appropriate solver before proceeding further.

Select Settings > Solver from Main menu (Top left and select appropriate solver and select Nastran press Apply. Selecting a solver is shown in Figure 4.191.

Figure 4.191 Solver Setup window

Click on  (Setup Analysis Type) icon from Solve Options Tab Menubar to setup Nastran run which pops up **Setup Analysis Type** window as shown Figure 4.192.

In the **Setup Analysis Type** window, do the following:

Select **Run Type** as Linear Static (Sol 101),

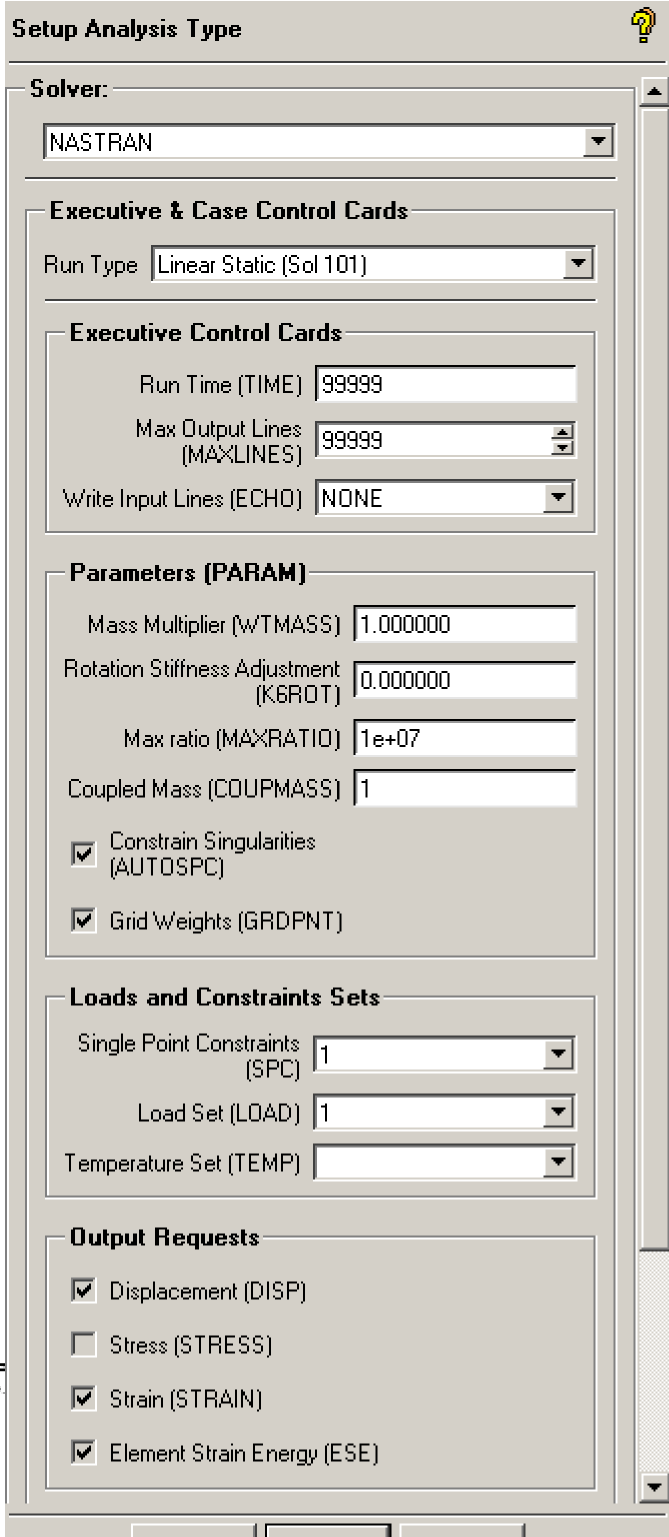
Make sure that Constrain Singularities (AUTOSPC) and Grid Weights (GRDPNT) are turned ON

For the Default Sets, select Single Point Constraints (SPC) and Load Set (LOAD) as 1,

In the Output Requests toggle ON Displacement (DISP), Stress (STRESS) and Element Strain Energy (ESE).

In the end, press Apply to complete the setup.

Figure 4.192 Setup Analysis Type window



Save Project

Select Save Project icon from Main Tab Menubar.

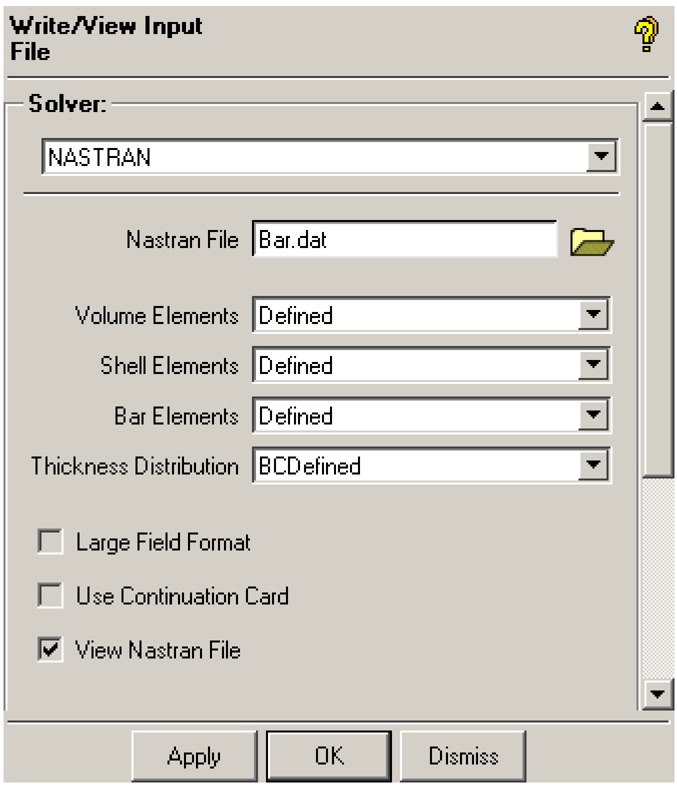
It will save four files; Geometry file, Mesh file, Attribute file and Parameter files as Bar.tin, Bar.uns, Bar.fbc and Bar.par respectively.

Write Nastran Input File

Click on  (Write/View Input File) icon from the Solve Options Tab Menubar.

Enter the Nastran file name as **Bar.dat** and switch **ON** View Nastran file option in **Write/View Input File** window as shown in Figure 4.193 and press Apply.

Figure 4.193 Write/View Input File window



User will see that the Nastran input data file comes up in the default text editor. User can edit this file manually and can save the changes with the same file name. Since no need to do any editing for this example, just close the editor.

j) Solution and Results

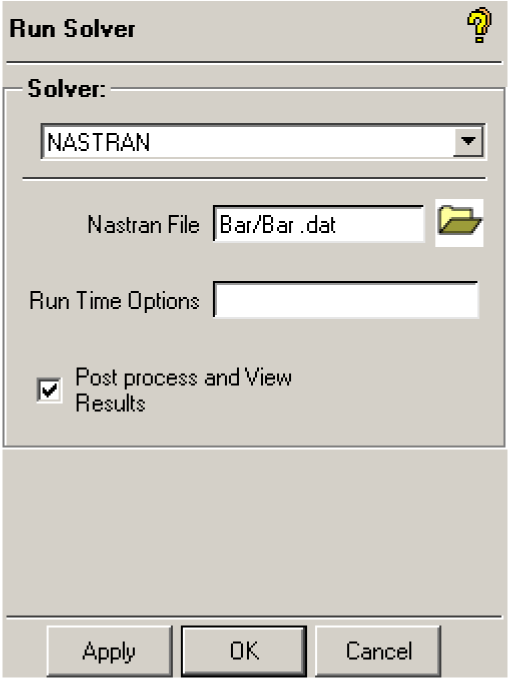
Linear Static analysis is to be performed on this model and the results should be visualized in a post processor.

Solving the problem

Click on  (Submit Solver Run) icon from the Solve Options Tab Menubar to start the Nastran as shown in Figure 4.194. The Nastran file will be selected by default as **Bar.dat**.

Toggle 'O**n' Post process and View Results** and press Apply in Run Solver window.

Figure 4.194 Run Solver window



k) Post Processing of Results

After completion of Nastran run, the results will be automatically loaded into the post processor tab.

Click on  **Variables** option in **Post-processing** Tab menu bar. In Select Nastran Variables window, select **Category** as **Solid** and Current Scalar Variable as **VonMises_Stress** as shown in Figure 4.195. The VonMises Stress distribution is shown in Figure 4.196.

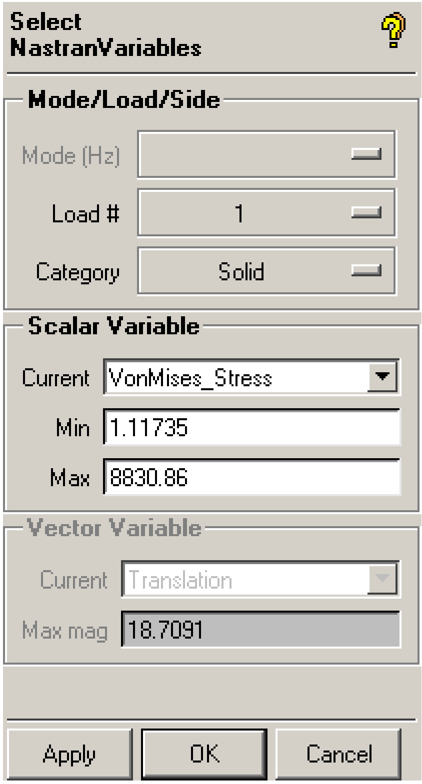
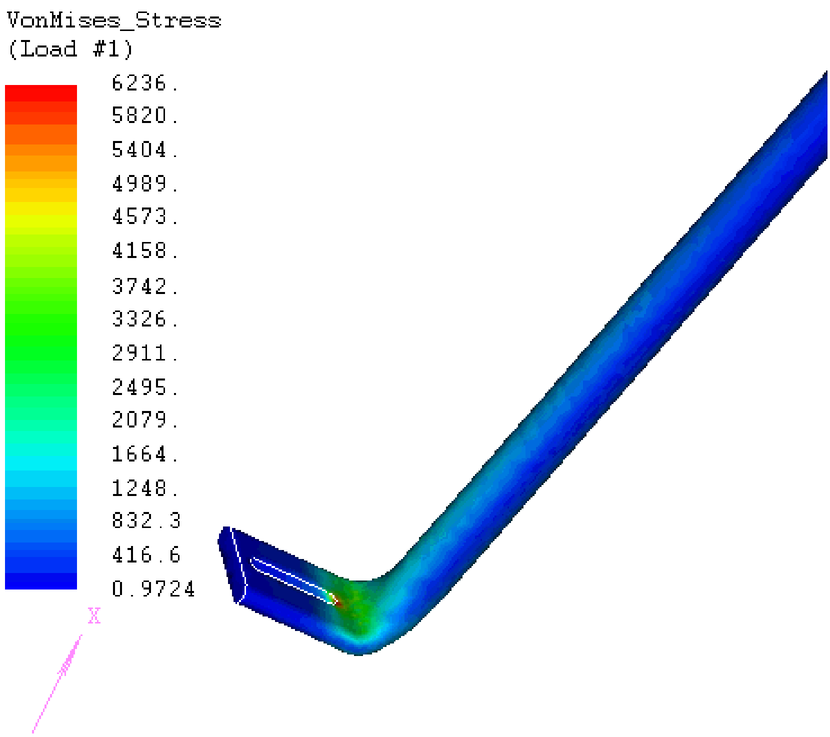


Figure 4.195 Nastran Variables window

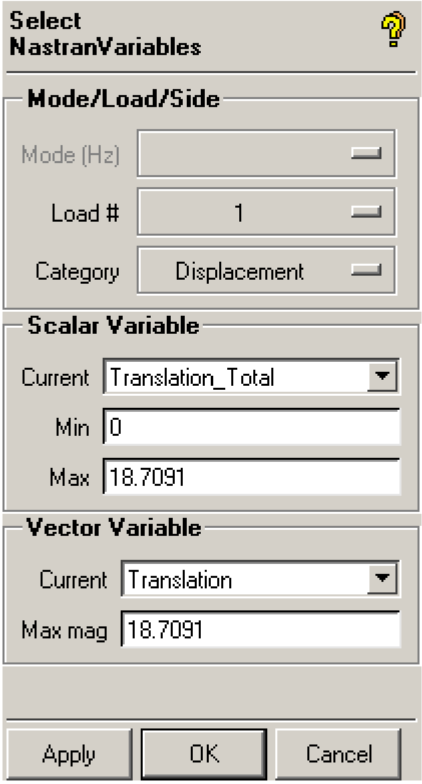
Note: Results shown here are obtained by MSC Nastran run. Results may differ with those of AI*Nastran run depending on the version.

Figure 4.196 VonMises Stress distribution



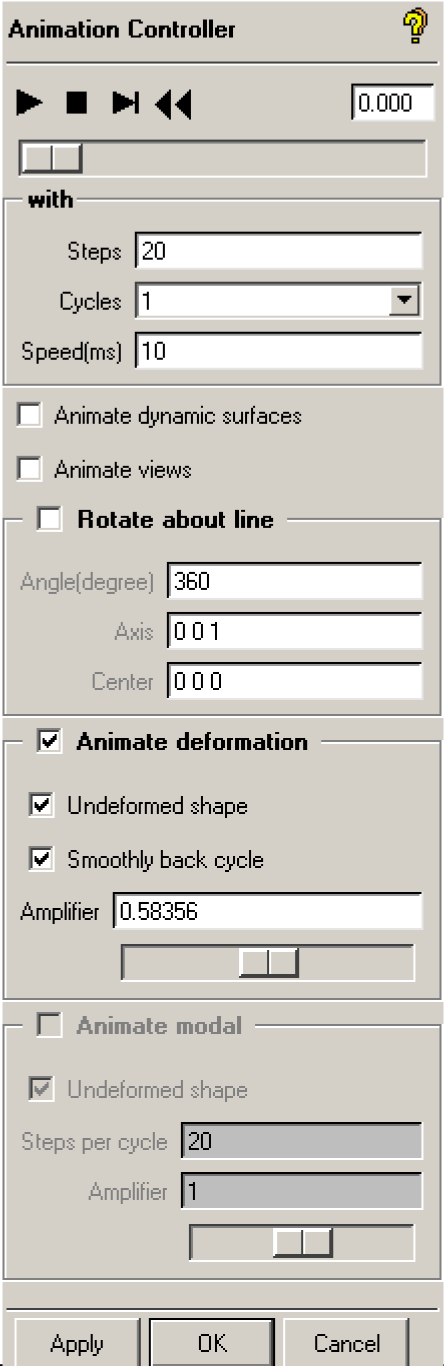
To display mode shape at Total Translation Frequency, select **Category** as **Displacement** and Current Scalar Variable as **Translation_Total** in Select NastranVariables window as shown in Figure 4.197.

Figure 4.197 Nastran Variables window



Select (Control All Animation) option from **Post-processing** tab menu bar which will open Animation Controller window as shown in Figure 4.198.

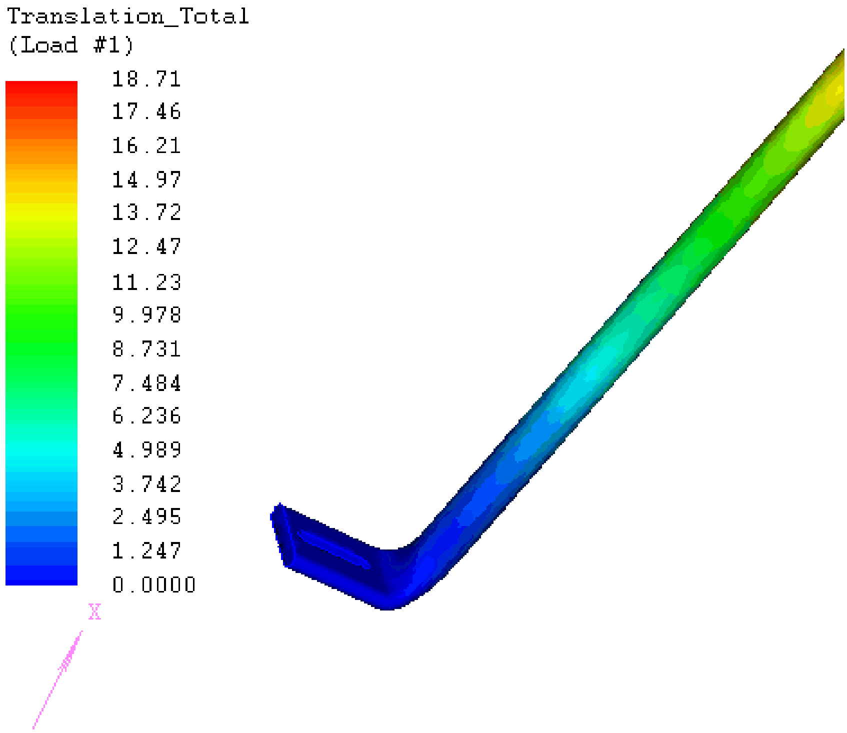
Figure 4.198 Animation Cotroller window



Set the values as shown in Figure 4.198 and press (Animate) to view the mode shape as shown in Figure 4.199.

Finally select **Exit** to quit the post processor.

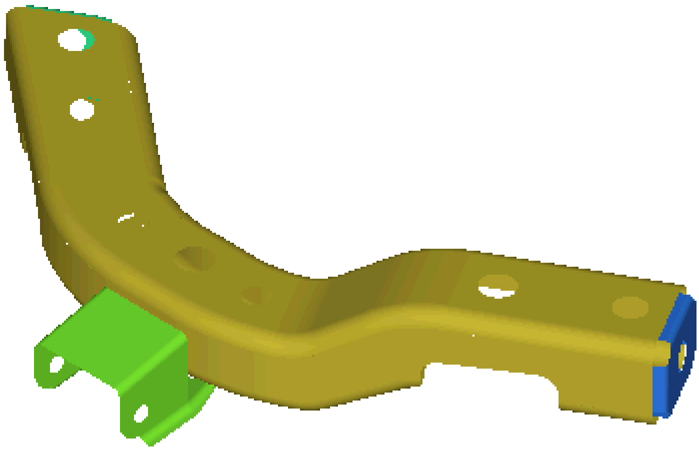
Figure 4.199 Animated Model



4.3.3: Frame

This exercise explains meshing of Frame geometry including the Seam and Spot welds and writing the input file to solve this Linear Static problem in NASTRAN. The geometry model is shown in Figure 4.200:

Figure 4.200 Frame Geometry



a) Summary of Steps

Launch AI*Environment and load geometry file

Geometry Editing

Geometry Repair

Connectors

Create Seam Weld

Create Spot welds

Create bolt connectors

Mesh Parameters

Mesh Sizing

Meshing

Surface Meshing

Checking mesh quality

Improving mesh quality

Material and Element Properties

Selection of Material

Element Properties

Constraints and Loads

Constraints

Loads

Solver setup

Setup Nastran Run

Save Project

Write Nastran Input File

Solution and Results

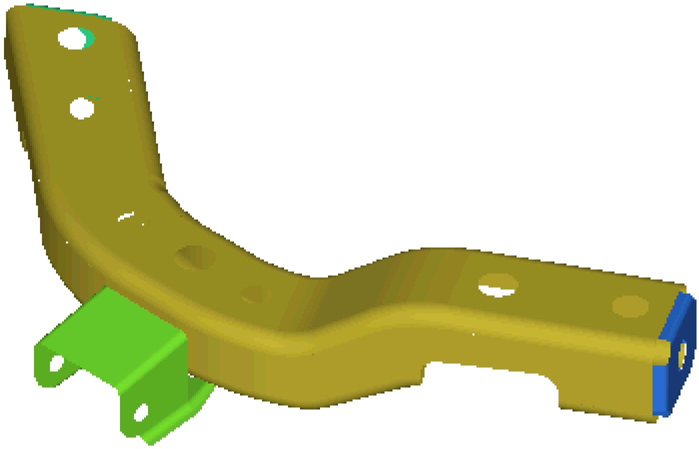
Solving the Problem

Post Processing of Results

b) Launch AI*Environment

Launch the AI*Environment from UNIX or DOS window. Then File > Change working directory, $ICEM_ACN/../docu/FEAHelp/AI_Tutorial_Files. Load the tetin file 'Frame.tin', and examine the parts, Figure 4.201.

Figure 4.201 Frame Geometry

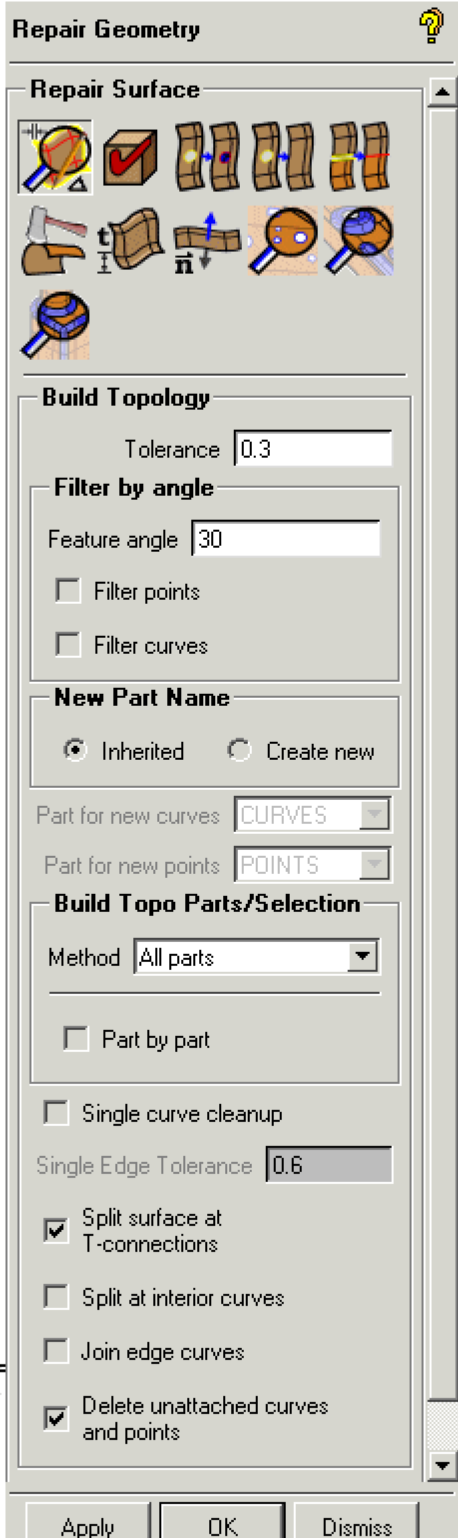


c) Geometry Editing

Repair

Click on  (Repair Geometry) icon from **Geometry Tab Menubar**, which pops up **Repair Geometry** window, by default Build Topology option  is highlighted. Now, make sure that **Inherited** is toggled '**on**' for **New Part Name** and the **Tolerance** is set to 0.3, as shown in Figure 4.202 and press Apply.

Figure 4.202 Repair Geometry window



d) Connectors

Since this is a surface meshing model, mesh parameters should be defined on curves. Furthermore, since the geometry is made of several separte parts, it is desireable to have individual mesh parts connected. Connectors are premesh definitions by which individual surfaces mesh parts are welded together.

Seam Weld

To create Seam weld between **PART_1003** and **PART_1004**, select  (Define Connectors) icon from **Mesh Tab Menu bar**. Click on  (Seam Weld) icon from **Define Connectors** window as shown in Figure 4.203.

Turn the Surface display '**Off**' from Display Model Tree by clicking on Geometry > Surfaces and also turn '**Off' Points** from Model Tree.

Turn '**Off**' Curves > Show Wide and Color by Count**,** and in Parts turn '**On**' only PART_1003.

Click on (Select Curve(s)  button, and then select the curves as shown in Figure 4.204.as Source Curves and notice that the New Part Name for Curves comes default as **SEAM_CURVES0**.

Figure 4.203 Define Connectors window for Seam Weld

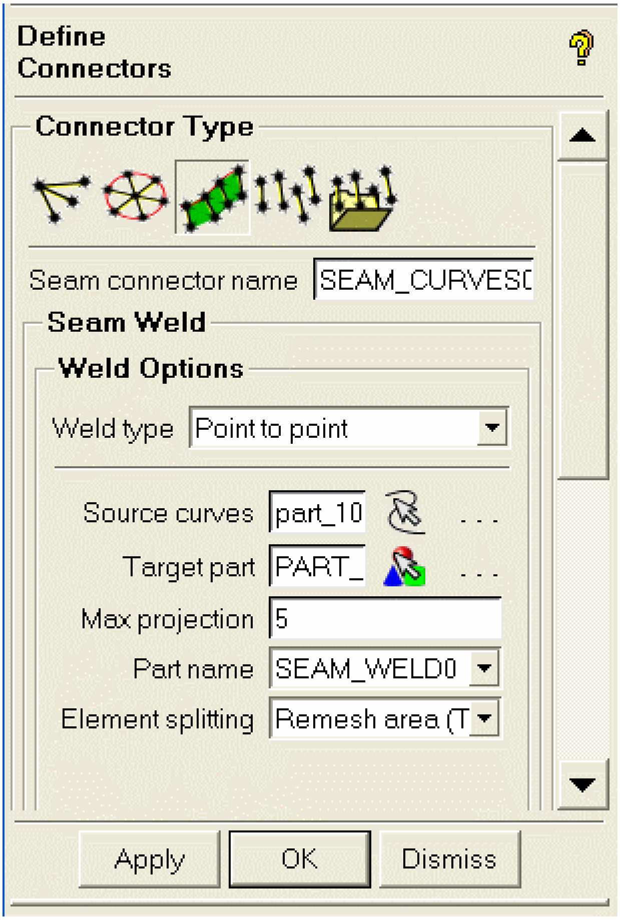
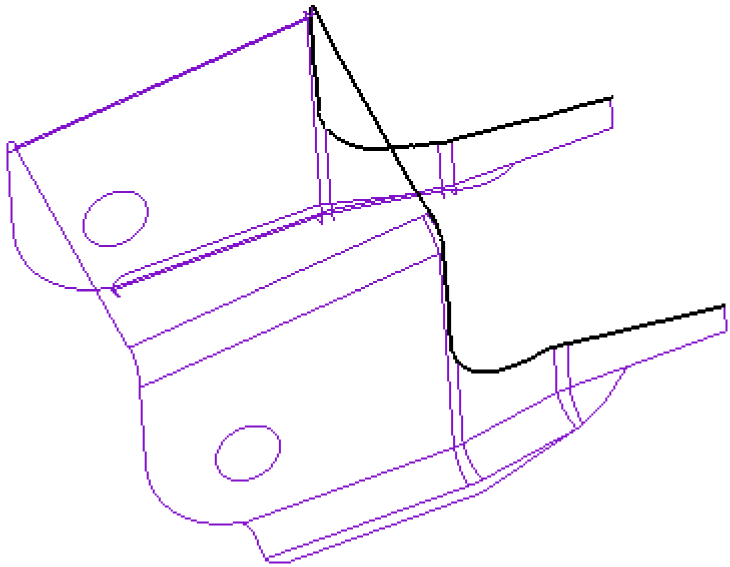


Figure 4.204 Curves for Seam weld



Select the Target part name as **PART_1004.** For Connector, Connector Part Name comes by default as **SEAM_WELD0** and enters Max Projection as **5.0.** Leave Element Splitting as **Re mesh area (Tri/Quad)** and then press Apply.

Switch '**Off'** Connectors from the Model Tree.

Switch '**On**'all the Parts in the Model Tree

Note: Only directives will be saved at this stage. Actual Seam Weld or any other connector will appear only when surfaces are meshed.

Spot Welds

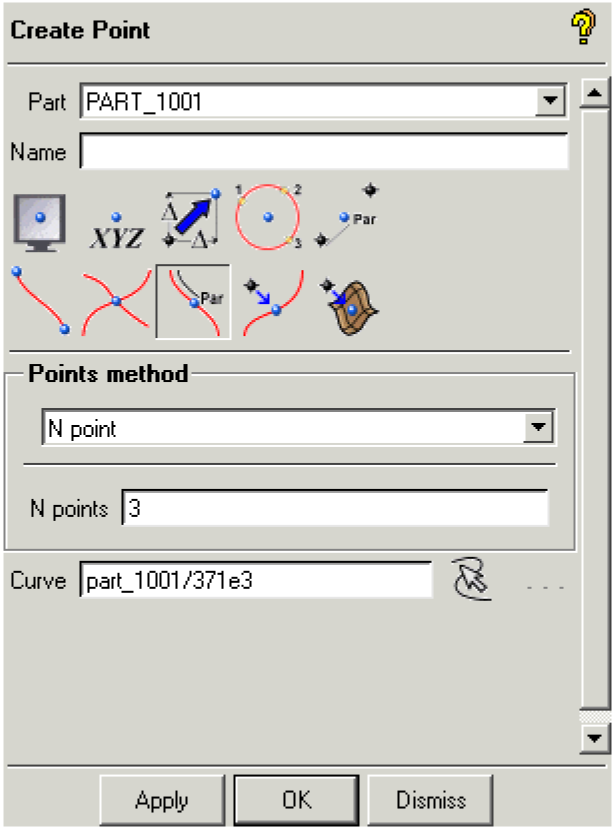
PART_1001 and PART_1004

Points representing the **Spots** should be created on the Geometry before defining spot-weld.

Now turn **ON** points from Display Tree. Also from Parts turn **OFF** all parts including Connectors except PART_1001.

From the **Geometry Tab Menu bar**, select  (Create Point) icon. Select **Part** as PART_1001 and leave the **Name** Blank as shown in Figure 4.205. Select  (Parameter along a Curve) icon select '**N Point Method**' enters N point as '3'.

Figure 4.205
Create Point Window



Select the Curves as shown in Figure 4.206 one by one and Press Apply. Thus Three Points are created.

Make sure the Part Name is same for all the Points.

After all the points are created the geometry should look as shown in Figure 4.207.

Figure 4.206 Curves (Dark) on which parametric points to be created

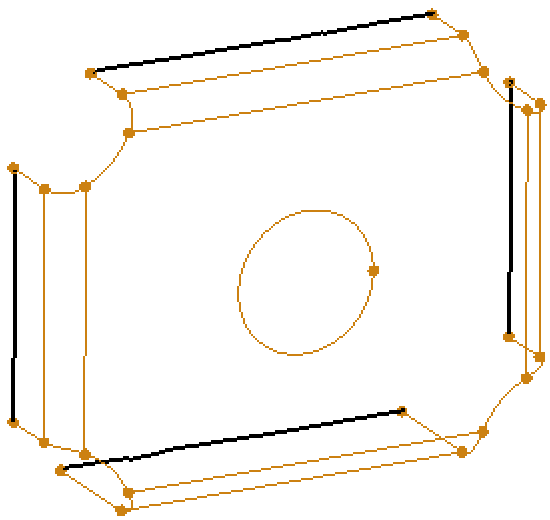
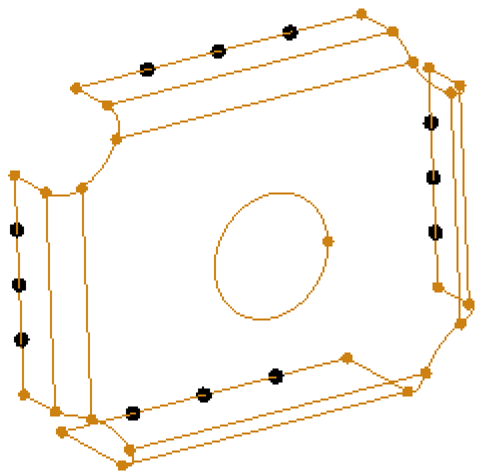


Figure 4.207 The Dark Point indicate sthe newly Point created



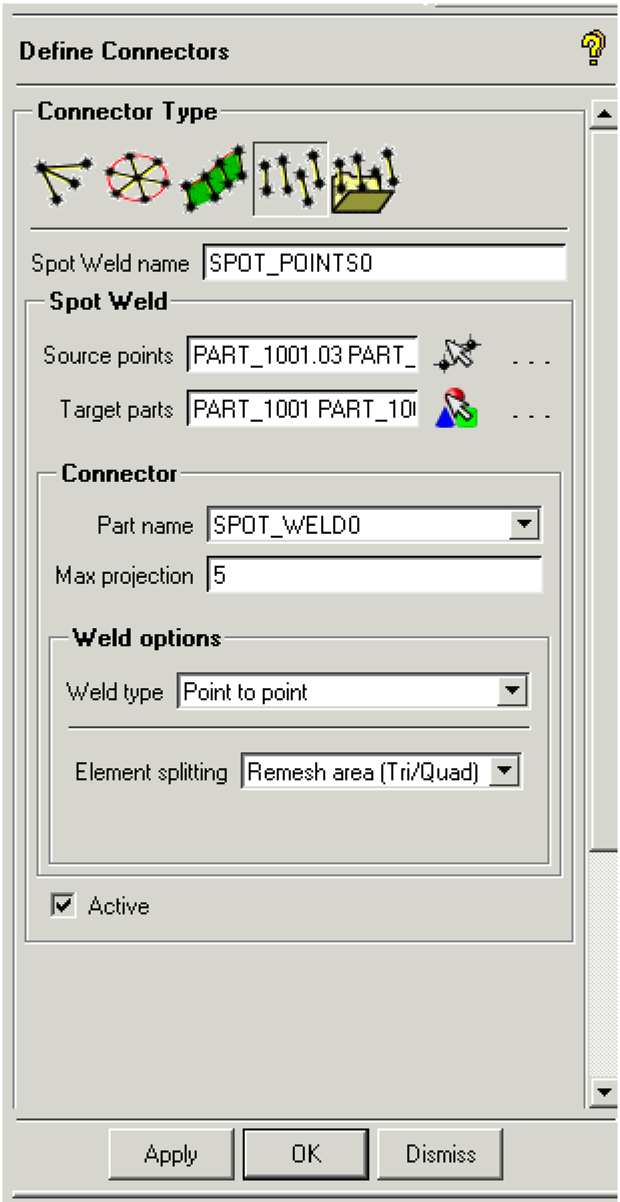
Select  (Define Connectors) icon from **Mesh Tab Menu bar**. Now click on  (Spot Weld) icon from **Define Connectors** window as shown in Figure 4.208.

Spot Weld Name: SPOT_POINTS0 (It will appear by default)
Source Points: Select the 12 points created as shown in Figure 4.207.

Target Parts: Select the parts PART_1001 and PART_1004

Connector Part Name: SPOT_WELD0 (It will appear by default)
Max Projection: 5
Weld Type: Point to Point (It will appear by default)
Element Splitting: Remesh area (Tri/Quad) (It will appear by default) and Press Apply.
Switch '**Off'** Connectors in the Display Tree.
All the values are shown in Figure 4.208.

Figure 4.208 Define Connectors window



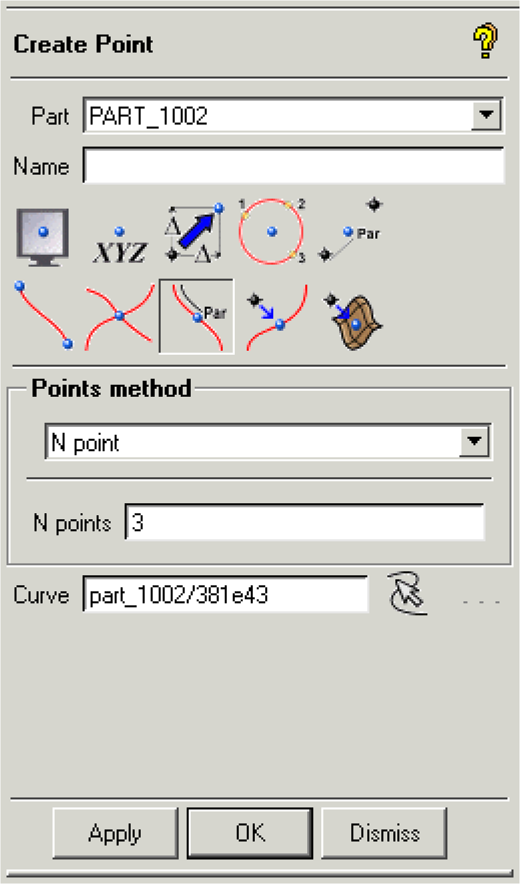
Turn '**OFF'** all Parts except PART_1002 in the Model Tree.

Note: Make sure that Points and Curves are switched '**On**' in the Model Tree**.**

PART_1002 and PART_1004

As explained for Spot-weld between PART_1001 and PART_1004, repeat all the steps for Spot-weld between PART_1002 and PART_1004. While creating Points for PART_1002, please make sure that **PART_1002** is selected for Part in **Create Point** window and the Name should start from **PART_1002.0** as shown in Figure 4.209.

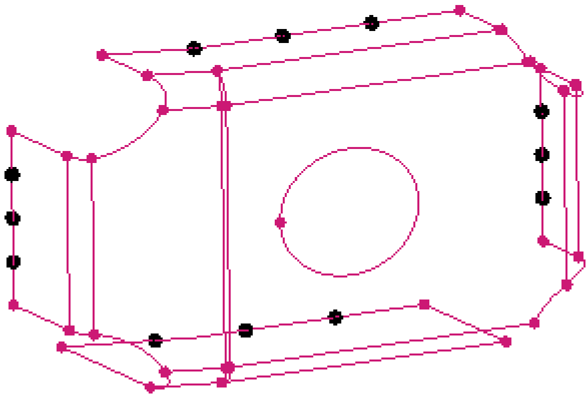
Figure 4.209
PART_1002 Point Creation Window



Select each of the four curves as explained for Part_1001.

The ‘12’ Points corresponding to Part_1002 as shown in Figure 4.210.

Figure 4.210 12 points created on all four curves of PART_1002

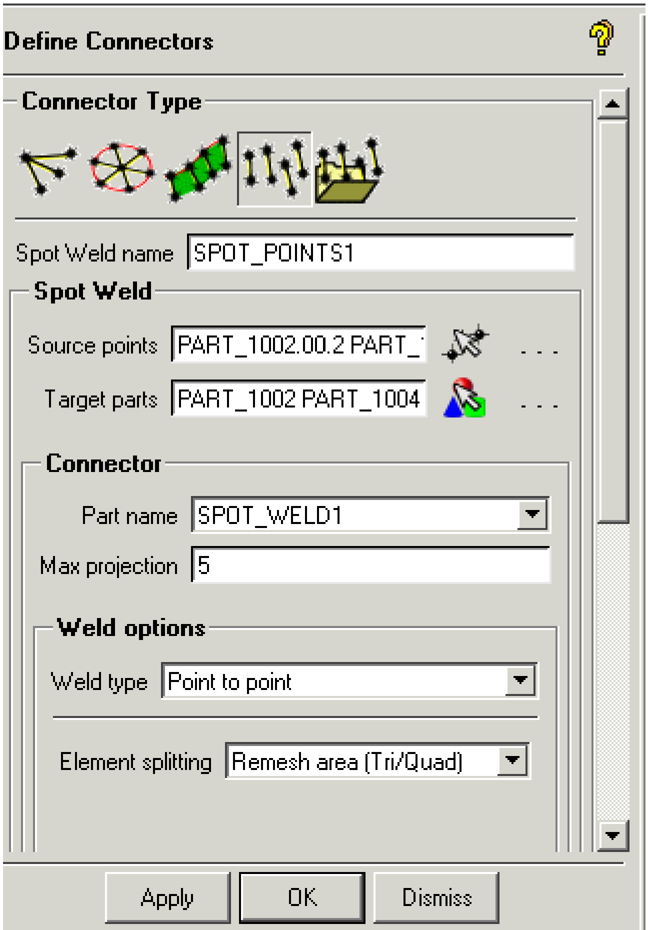


The target weld parts here would be **PART_1002** and **PART_1004**.and the Points selected are shown in Figure 4.210,
Spot Weld Name: SPOT_POINTS1 (It will appear by default)
Source Points: Select all the 12 points created as shown in Figure 4.210.
Target Parts: Select PART_1002 and PART_1004
Connector Part Name: SPOT_WELD1 (It will appear by default)

Max Projection: 5

Weld Type: Point to Point (It will appear by default)
Element Splitting: Remesh area (Tri/Quad) (It will appear by default) and Press Apply.
Switch **‘Off’** Connectors in the Display Tree.
All the values are shown in Figure 4.211.

Figure 4.211
Define Connector
Window for Spot Weld
Part 1002 and
Part1004

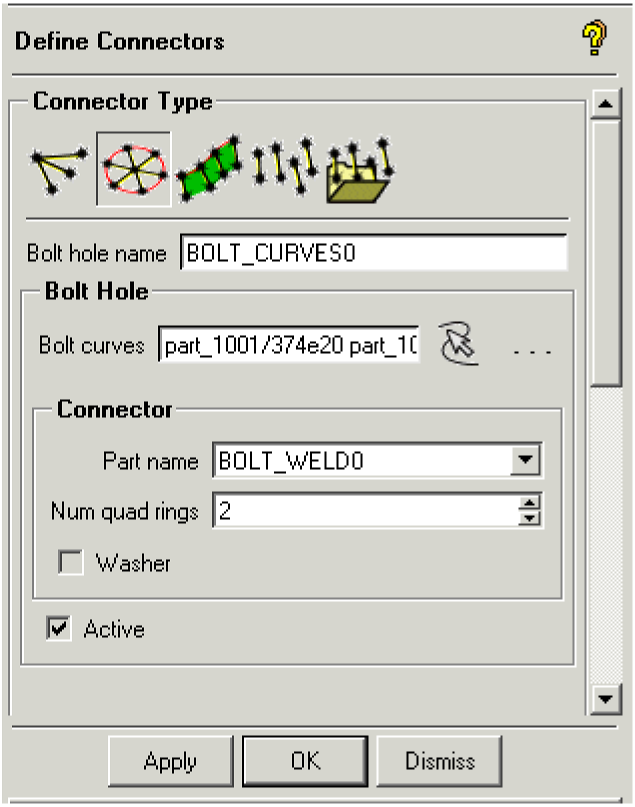


Create Bolt Connectors

Now from the Display Tree turn **OFF** Points and turn **ON** all parts. The two flanges (**PART_1001 & PART_1002**) and the hanger (**PART_1003**) have got Bolt Hole connections. Here Bolt connections should be defined.

Select  (Bolt Hole) icon from **Define Connectors** window as shown in Figure 4.212.

Figure 4.212 Define Connectors window



Enter the following parameters in this window. Click on  button and select each of four curves, as shown in Figure 4.213 and press Apply.
New Part Name for Curve: **BOLT_CURVES0** (this comes by default).
Connectors Part Name: **BOLT_WELD0** and
Enter No of Quad Layers as **2** and press Apply.
This will create Bolt Hole connector at all the four curves.

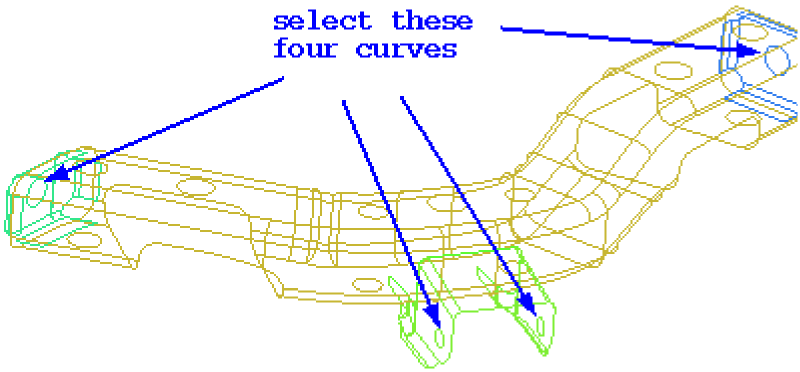
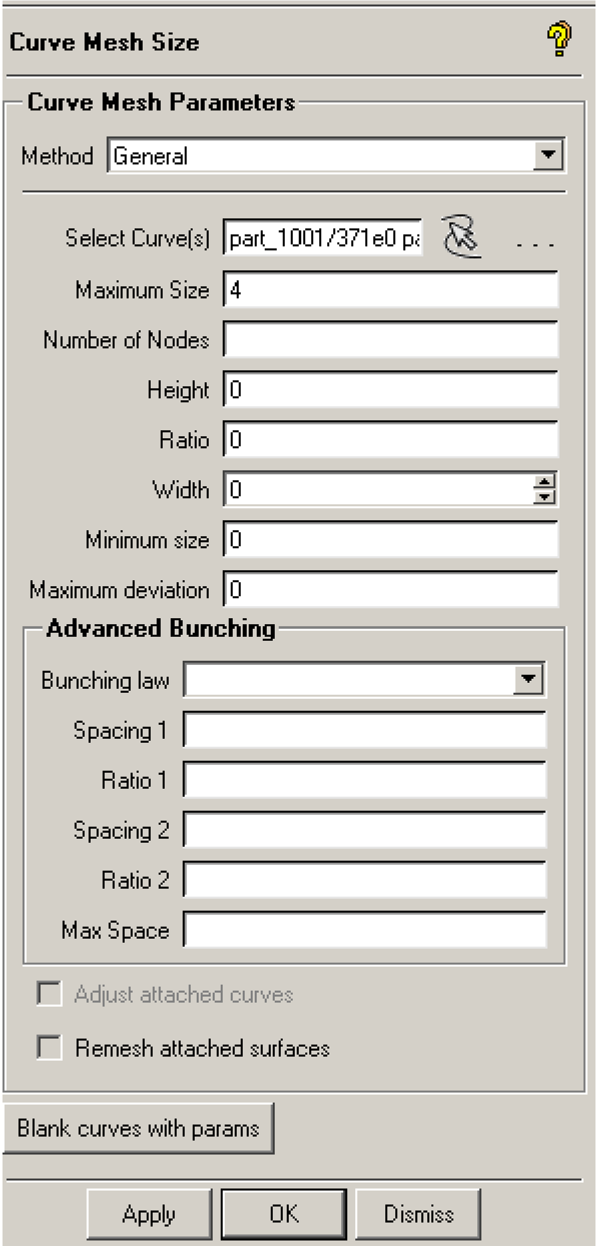


Figure 4.213 Four Curves for Bolt Connections

Mesh Parameters

Select  (Set Curve Mesh Size) icon from **Mesh Tab Menubar**, which pops up **Curve Mesh Size** window as shown in Figure 4.214. Click on  (Select Curve(s)) button and press **'a'** (ensure that the mouse cursor is in display window) from keyboard to select all curves and enter **Maximum Size** of **4** for this case and press Apply and then Dismiss.

Figure 4.214 Curve MeshSize window



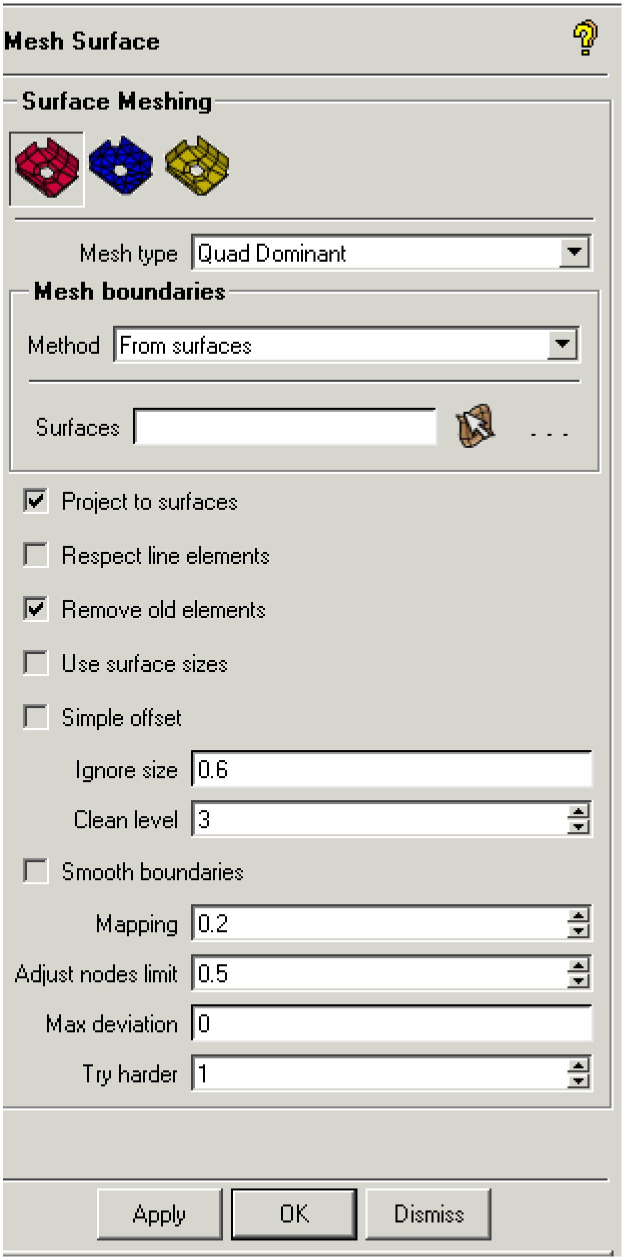
e) Meshing

From the Model Tree turn on display of Surfaces.

Select the  (Mesh Shell) icon from **Mesh Tab Menubar**. Select 'Patched Based'  icon and in the **Surface Meshing** window, Change **Ignore Size** to **0.6** and **Clean level** to **3**. Now, press Apply in the **Mesh Surface** window as shown in Figure 4.215.

Note: If Surface Window is left Blank it selects all the surfaces.

Figure 4.215 Mesh Surface window



In Model Tree, click on '+' to expand the Mesh options. Click the right mouse button on **Shells** and select **Solid & Wire**. Turn 'Off' Surface The mesh is shown in Figure 4.216.

Figure 4.216 Mesh in Solid & Wire mode

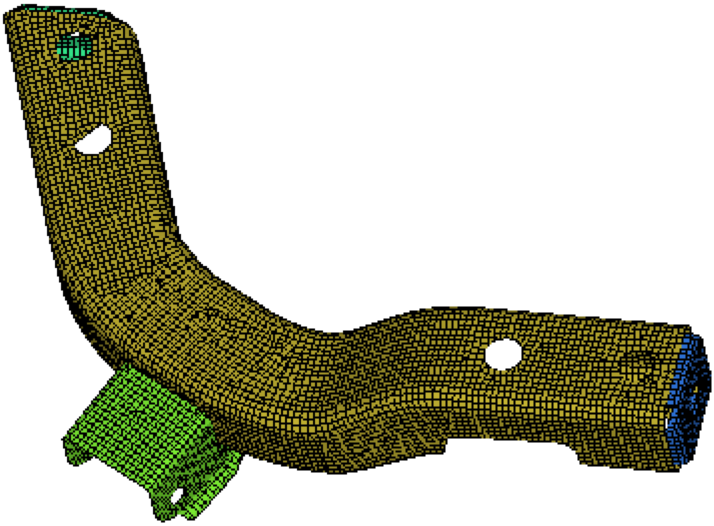


Figure 4.217 Bolt hole

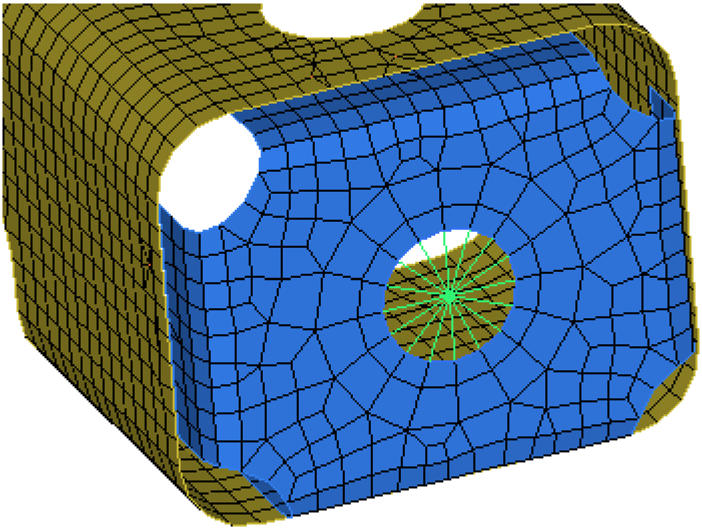


Figure 4.218 Seam Weld

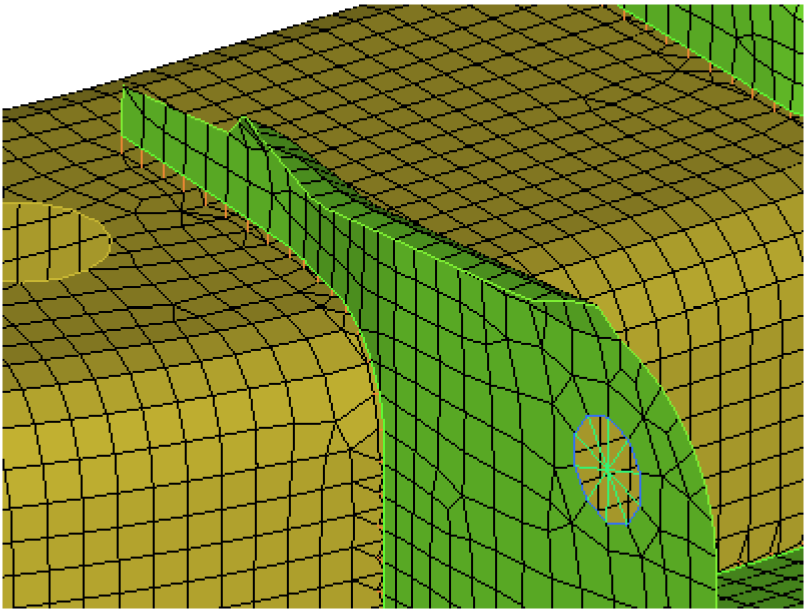
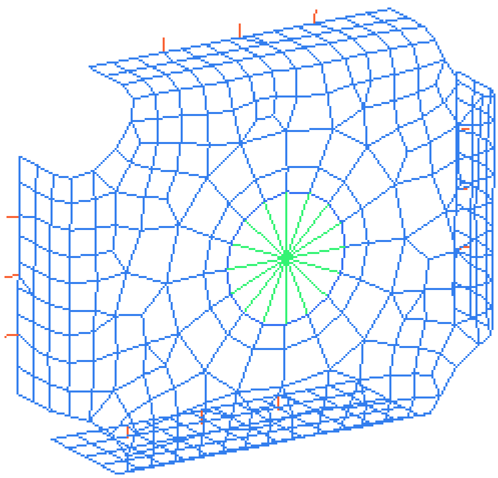


Figure 4.219 Spot Weld

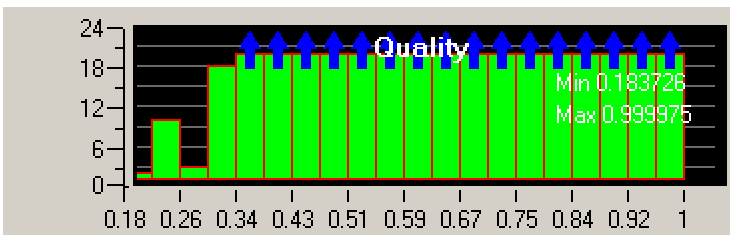


f) Mesh Quality

It should be ensured that quality of the elements does not go below certain value before applying constraint. For this case, let us say a quality of 0.2 should be good enough.

To check the quality of the elements, select  (Smooth mesh globally) icon of the Edit Mesh Tab Menu bar. In the Criterian Type select Quality and press Apply. Right Click Mouse button on any Quality Bar situated at Right hand corner of the GUI and select Reset by Clicking Right Mouse button, the Quality histogram will appear as shown in Figure 4.220.

Figure 4.220 Quality Histogram before Smoothing



There are six options when we Click the right mouse button on any Histogram Bar as shown in Figure 4.221.

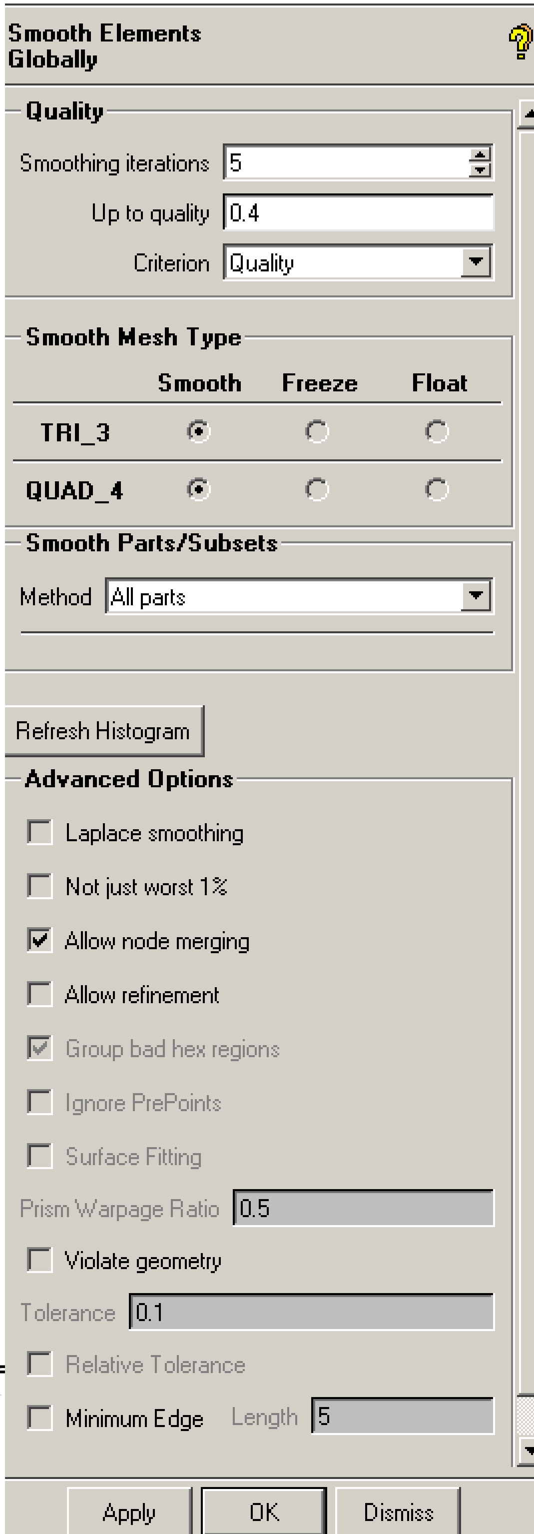
Figure 4.221
Histogram Option

To differentiate the display of element quality, click the right mouse on Mesh > Shell icon in the Display Model Tree, and select **Color by Quality** option).

Since there are some elements with quality less than 0.2, quality should be improved.

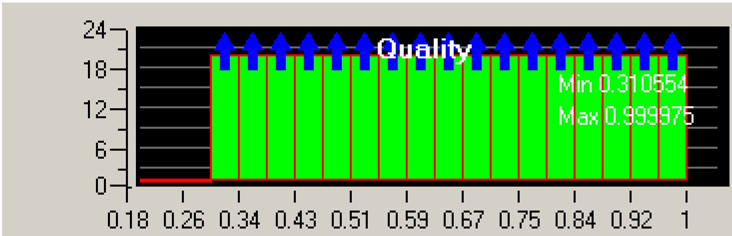
In the **Smooth Elements** window, shown in Figure 4.222, enter Quality value of 0.4 (target quality should always be above the required value so that smoother can select more element i.e. more freedom to improve quality). Accept the default setting and press Apply. This will start the smoother, which automatically tries to improve the quality to the targeted quality of 0.4.

Figure 4.222 Smooth Elements window



After it completes the smoothing it will replot the histogram again as shown in Figure 4.223.

Figure 4.223 Histogram After Smoothing



Now if there is no elements below 0.2 Quality then proceed further to define Material properties, otherwise again smooth for some more iteration till no element lies below 0.2 'Quality'. Finally Right Mouse Click within Histogram window and select 'Done' to close the Histogram window.

g) Material and Element Properties

Material for this model is STEEL. So the properties like Young's modulus, Poisson's ratio and Density should be defined. This problem is modeled by shell elements so properties like thickness of the shell elements needs to be defined.

Selection of Material

Select 

(Create Material Property) icon from **Properties Tab Menu bar**.

Define the Material Name as **STEEL**,

Material ID can be left as **1**.

Define Young's modulus as **207000**,

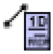
Define Poisson's ratio as **0.28**,

Define Density as **7.8e-9**; leave other fields as it is in **Define Material Property** window.

Press Apply.

Element Properties

1D element properties

Select  (Define 1D Element Properties) icon from the Properties Tab Menu bar.

It will open **Define Line Elements** window as shown in Figure 4.224

Select the **Part** as SEAM_WELD0,

Select Type as **Rigids**,

Select rigid type as **Rigid Body** (**RBE2).**

Press **Ok.** So that this window will be closed and again click **Define 1D element properties.**

Note: The Material and PID turns grey scale after selecting Rigid Beam

Repeat these steps to define properties for other Line elements of Spot welds and BOLT HOLES.

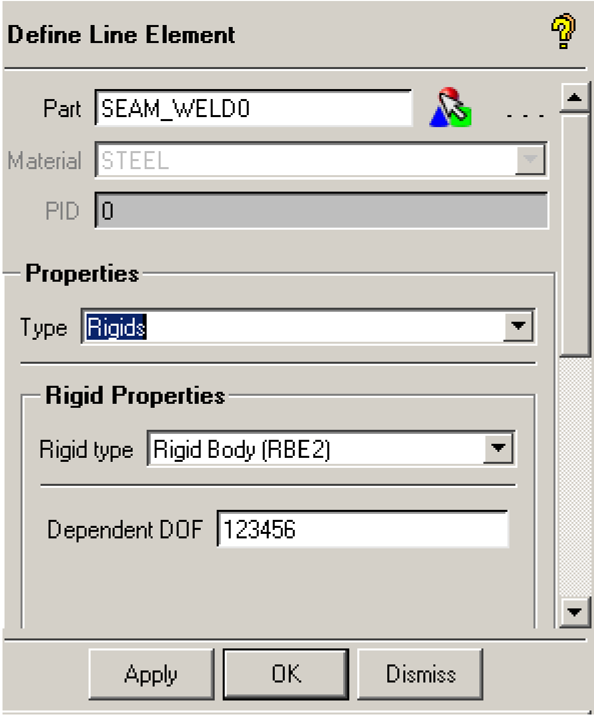


Figure 4.224 Define Line Element window

Shell Element Properties

Select  (Define 2D Element Properties) icon from the Properties Tab Menubar.

Set PID as **1001**, in **Define Shell Elements** window.

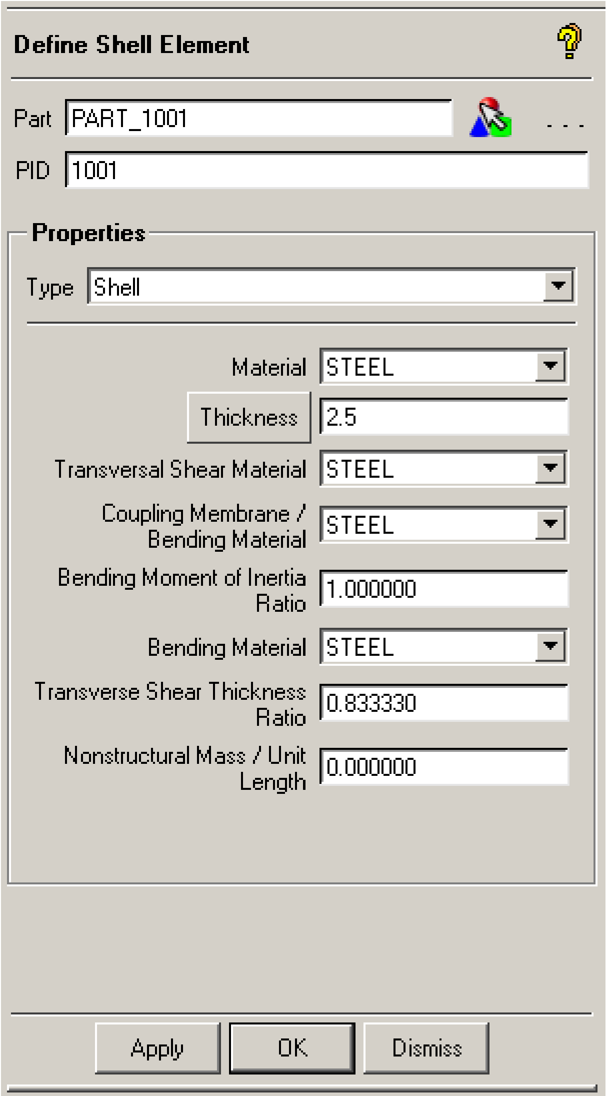
Select the Part as **PART_1001** to apply the property to.

Select Material as **STEEL** and leave other things are as default.

Enter Thickness as **2.5**.

Press Apply.

Figure 4.225 Define Shell Element window



Repeat these steps to define properties for other Shell elements for PART_1002, PART_1003 and PART_1004 except the PID and Thickness as follows:

Part Name	PID	Thickness
PART_1002	1002	2.5
PART_1003	1003	3.0
PART_1004	1004	2.0

h) Constraints and Loads

Constraints

Click on (Displacement on Point) icon from the Constraints Tab Menu bar, which pops up the window shown in Figure 4.226.

In this window toggle on all options UX, UY and UZ, for both Directional and Rotational Displacement, and select the center points of the bolt connections on PART_1001and PART_1002 as shown in Figure 4.227 and press Apply.

Also, while selecting the nodes, if the Points are ‘**On**’ in the display, user may not be able to select the center points of the bolt connections. So it would be better to make sure that Geometry > Points are toggled ‘**Off**’ before selecting these points.

Figure 4.226 reate displacement on Point window

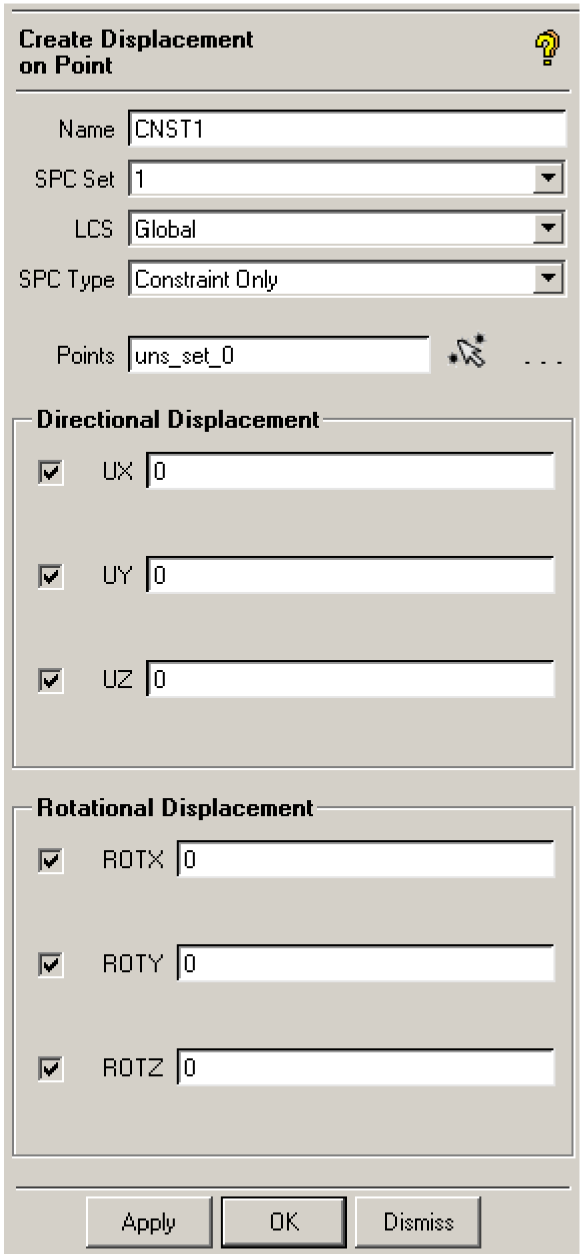
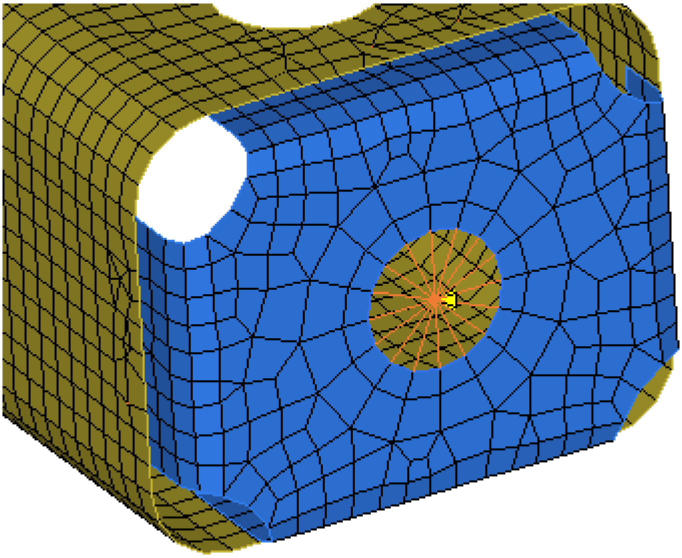


Figure 4.227 Constraint on Point



Loads

Click on  (Force on Point) icon from Loads Tab Menubar, which will pop up **Place Force on Point** window as shown in Figure 4.228.

In this window enter Name as **FORCE** and select the two center points of the bolt connections on **PART_1003** as shown in Figure 4.229 and enter a value of **100** for FZ and press Apply.

Figure 4.228 Force on Point window

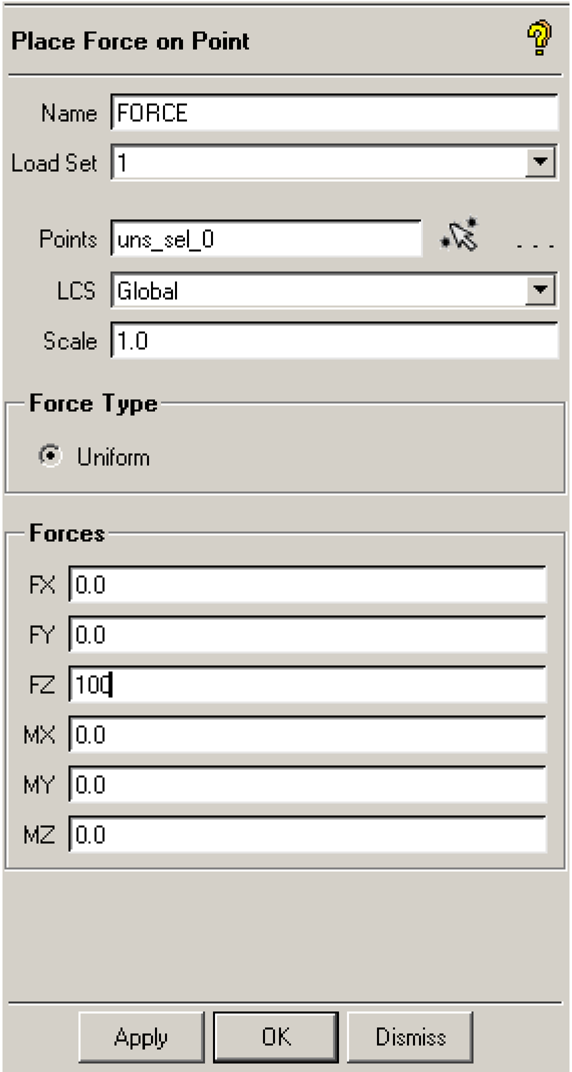
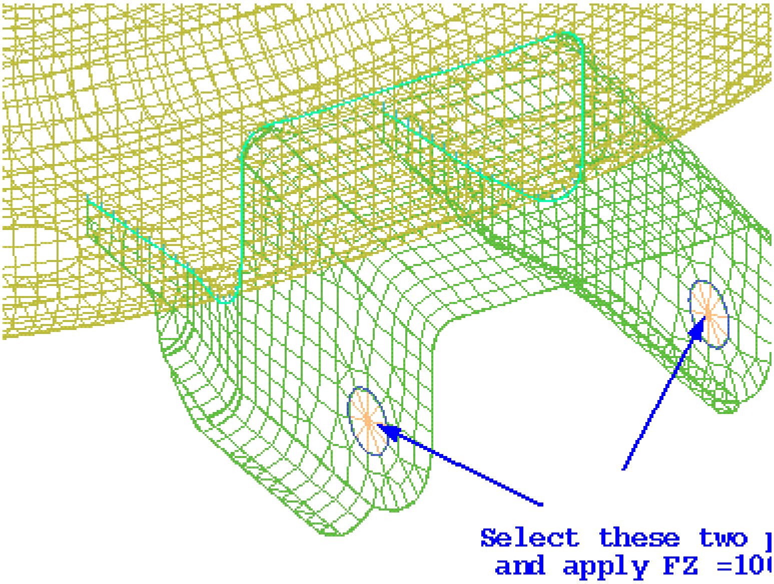


Figure 4.229 Points for Load application



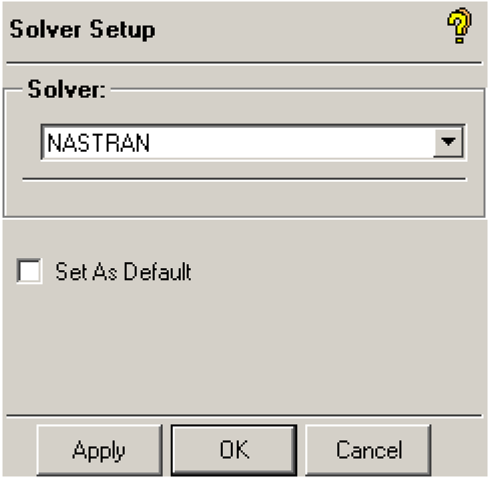
i) Solver Setup

Setup Nastran Run

First, user should select the appropriate solver before proceeding further.

Select Settings > Solver from Main menu and press Apply as shown in Figure 4.230.

Figure 4.230 Solver Setup window



Click on (**Setup Analysis Type**) icon from Solve Options Tab Menubar to setup Nastran run to do Linear Static that pops up Setup Analysis Type window as shown Figure 4.231.

In the **Setup Analysis Type** window, enter the following:

Select Run Type as Linear Static (Sol 101)

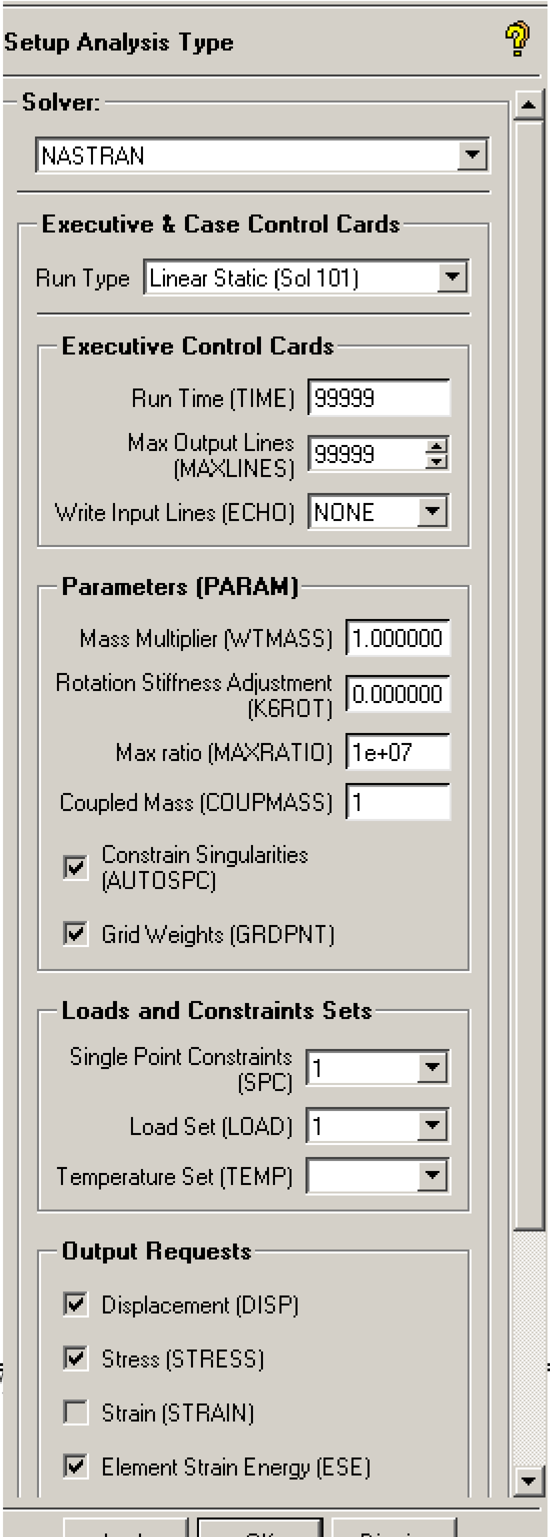
Make sure that **Constrain Singularities** (AUTOSPC) and Grid Weights (GRDPNT) are turned **'On'.**

For the Default Sets, select Single Point Constraints (SPC) and Load Set (LOAD) as **1**,

In the **Output Requests** toggle '**On**' Displacement (DISP), Stress (STRESS) and Element Strain Energy (ESE).

Press Apply to complete the setup.

Figure 4.231 Setup Analysis Type window



Save Project

Through File > Save Project As option, create new directory Frame as said in earlier tutorials.

Enter **Frame** as project name and press **Save** to save all these information in this directory

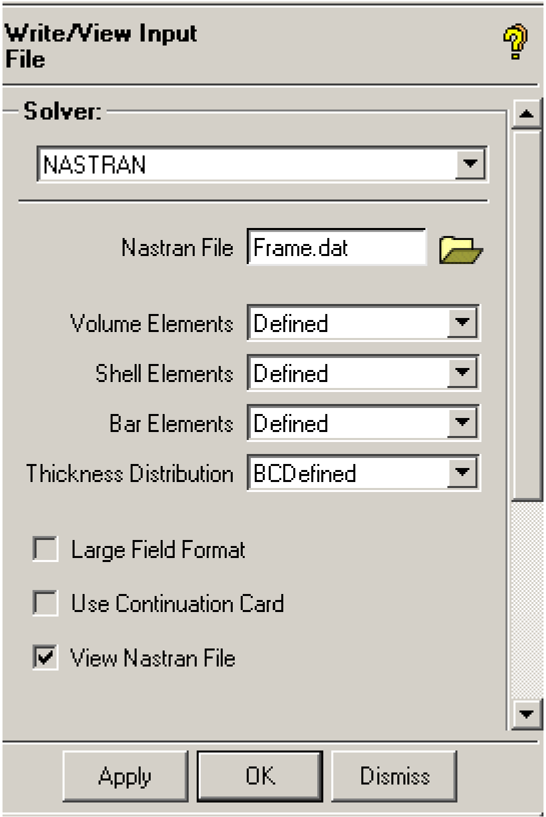
It will save four files; Geometry file, Mesh file, Attribute file and Parameter files as Frame.uns, Frame.fbc and Frame.par respectively along with the project file Frame.prj.

Write Nastran Input File

Click  (Write/View Input File) icon from Solve Options Tab Menu bar, which will open **Write/View Input File** window presented in Figure 4.232.

Give the Nastran file name as **Frame.dat** and switch ‘**On**’ View Nastran file as shown in Figure 4.232 and press Apply.

Figure 4.232 Write/View Input File window



User will see that the Nastran input data file comes up in the default text editor. If user likes to edit this file directly then this can be done and can save the edited file through this text editor. Since no need to do any editing for this example, just close the editor.

j) Solution and Results

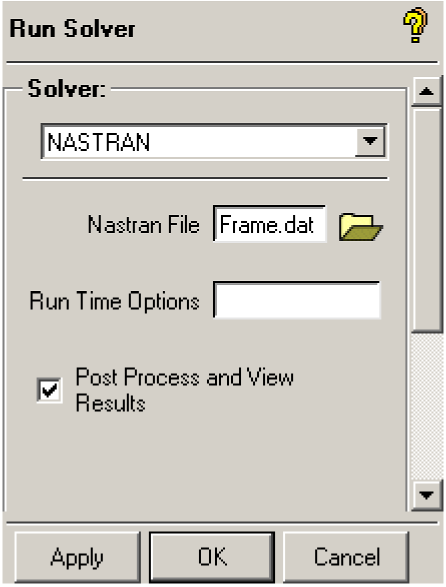
Linear Static analysis is to be performed on this model and the results should be visualized in a post processor.

Solving the problem

Click on  (Submit Solver Run) icon from the Solve Options Tab Menubar to start Nastran as shown in Figure 4.233. The Nastran file will be selected by default as **Frame.dat**.

Toggle **ON** Post process and View Results and press Apply in **Run Solver** window.

Figure 4.233 Nastran Run +start window



Post Processing of Results

After completion of Nastran run, the results will be automatically loaded into the post processor tab.

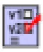
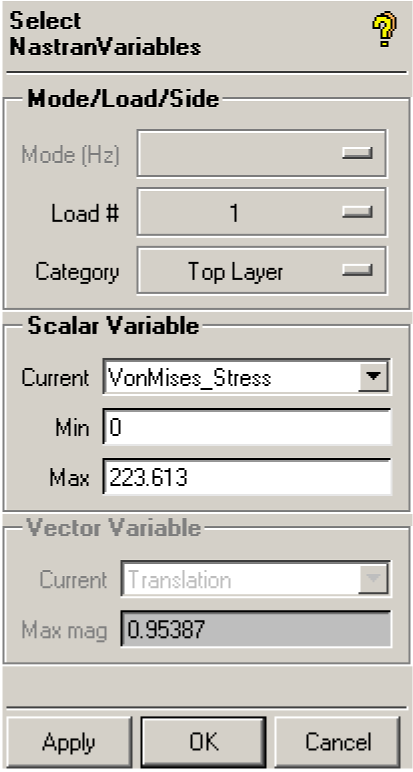
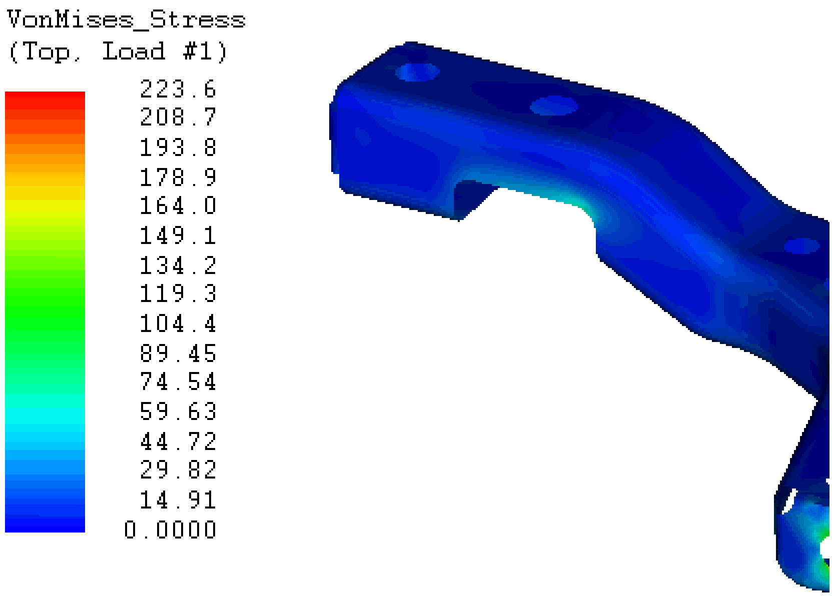
Click on  **Variables** option in **Post-processing** Tab menu bar. In Select Nastran Variables window, select **Category** as **Solid** and Current Scalar Variable as **VonMises_Stress** as shown inFigure 4.234. The VonMises Stress distribution is shown in.Figure 4.235

Figure 4.234 Vonmises_Stress selection



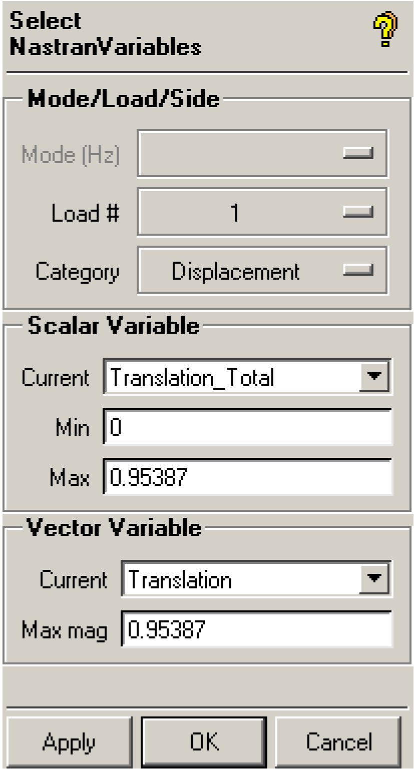
Note: Results shown here are obtained by MSC Nastran run. Results may differ with those of AI*Nastran run depending on the version.

Figure 4.235 VonMises Stress distribution



To display mode shape at Total Translation Frequency, from the Nastran Variables window as shown in Figure 4.236 select Category as **Displacement** and variable as **Translational_Total**.

Figure 4.236 Nastran variables window



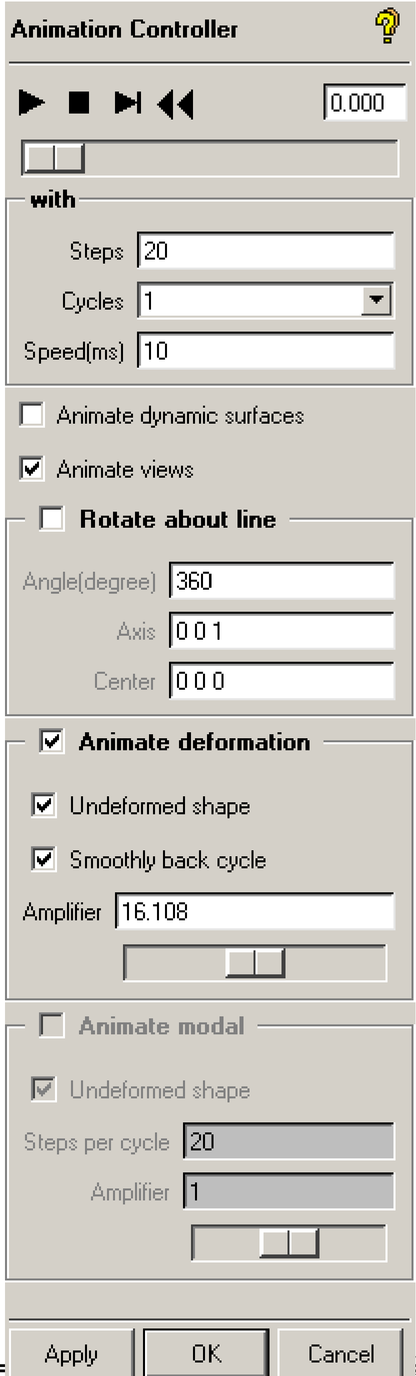


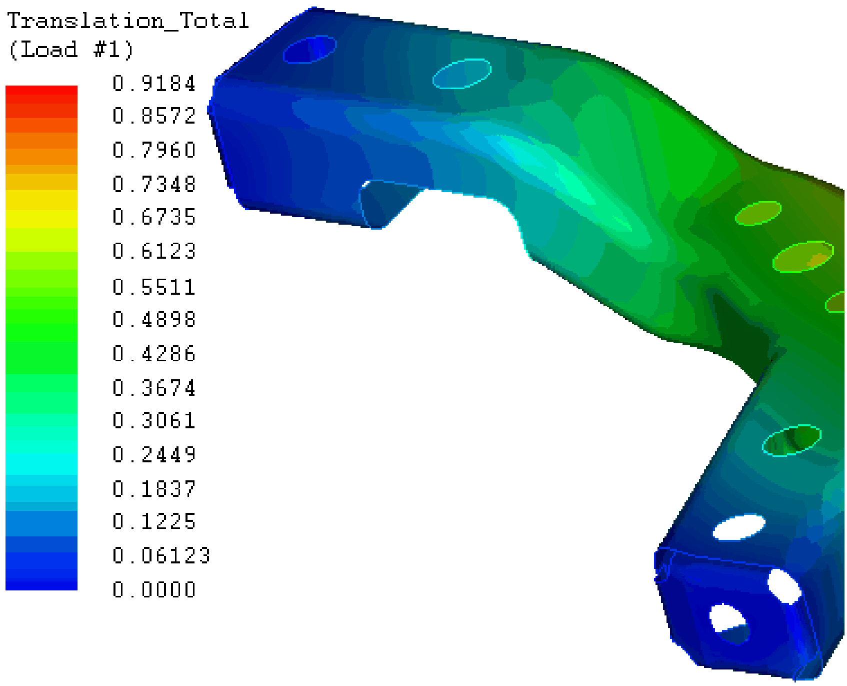
Figure 4.237 Animation Setup and Controller window

Select (Control All Animation) option from **Post-processing** tab menu bar which will open Animation Controller window as shown in.Figure 4.237

Set the values as shown in Figure 4.237 and press (Animate) to view the mode shape as shown in Figure 4.238

Finally select **Exit** to quit the post processor.

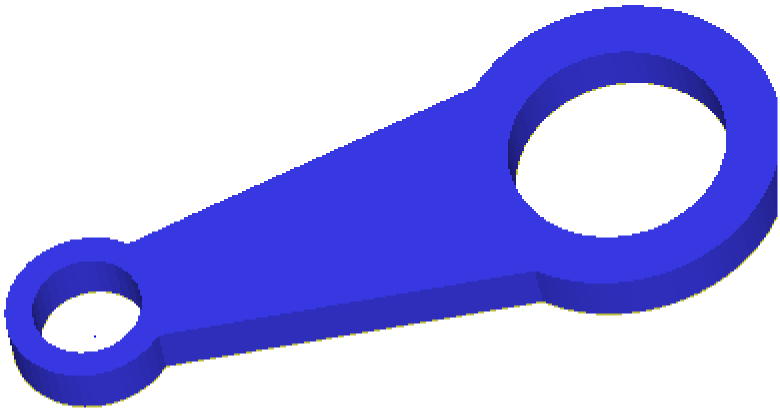
Figure 4.238 Animated model



4.3.4: Connecting Rod

This exercise explains Hexahedral meshing of Connecting Rod geometry by extruding the shell elements, writing the input file to solve this Linear Static problem in Nastran and Post Processing the results. The geometry is shown in Figure 4.239:

Figure 4.239 Connecting Rod Model



a) Summary of Steps

Launch AI*Environment and load geometry file

Geometry Editing

Repair

Mesh parameters and Meshing

Mesh Sizing

Meshing

Extrusion of the surface mesh

Materials and Element Properties

b) Launch AI*Environment

Launch the AI*Environment from UNIX or DOS window. Then File > Change working directory, $ICEM_ACN/../docu/FEAHelp/AI_Tutorial_Files. Load the tetin file 'Conrod tin'.

c) Geometry Editing

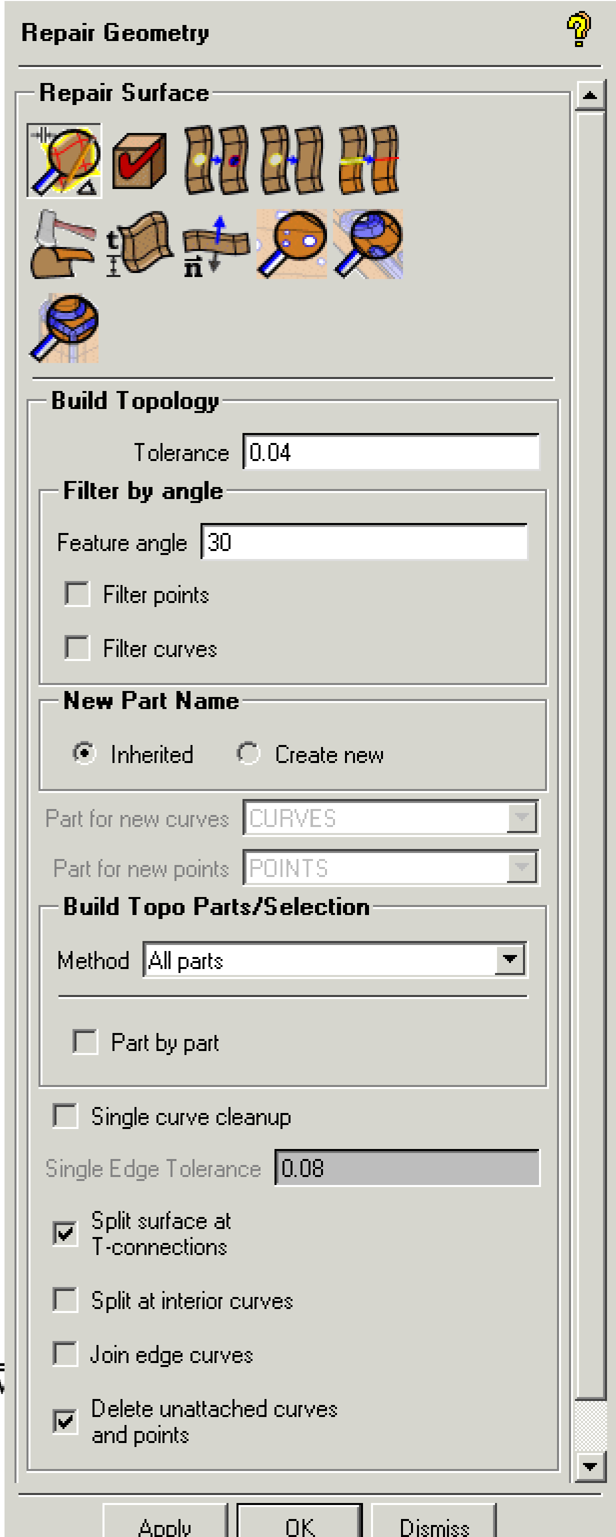
For this tutorial, user is requested to use **Conrod.tin** file lying in the **AI_Tutorial_Files** directory as mentioned in Tpipe tutorial.

Repair

Click on (Repair Geometry) icon from Geometry Tab Menubar, which pops up **Repair Geometry** window as shown Figure 4.240. By default **Build**

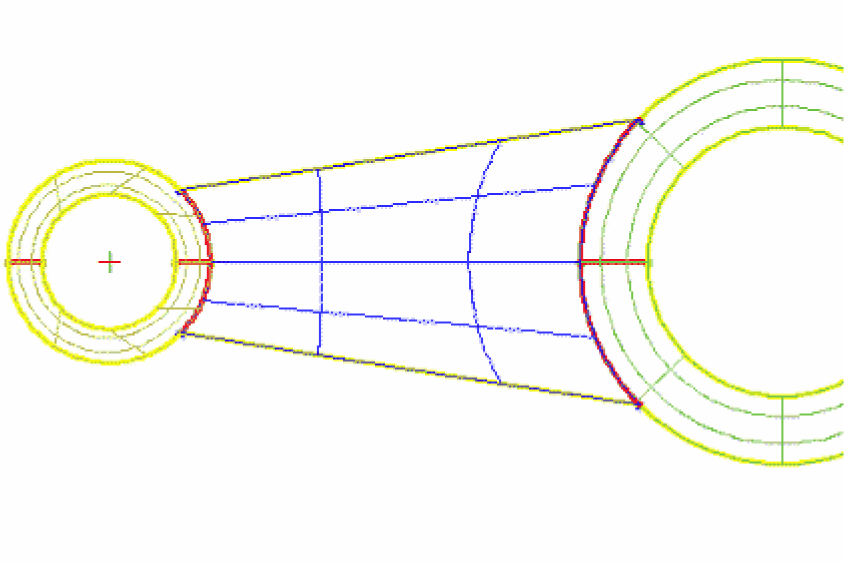
Topology  option is selected. Make sure that **Inherited** is toggled **ON** for **New Part Name** and press Apply.

Figure 4.240 Repair Geometry window



Now the geometry appears as shown in Figure 4.241.

Figure 4.241 Geometry after Build Topology



d) Mesh Parameters and Meshing

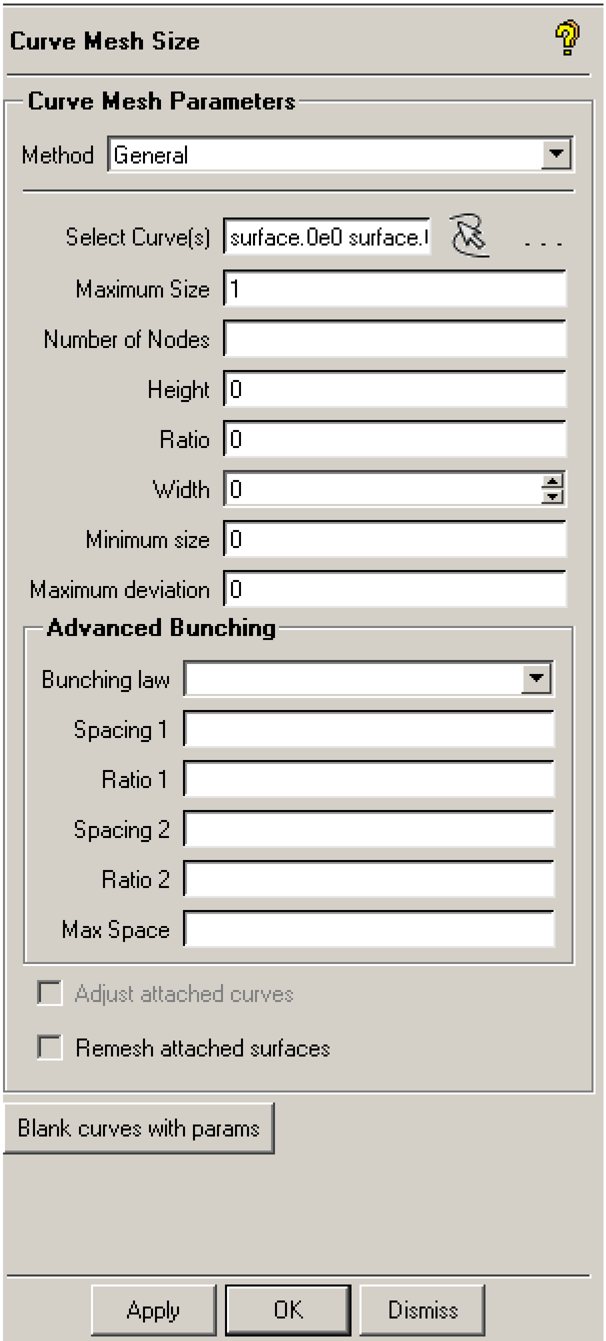
Even though this will be a 3D model, right now it is only 2D geometry, so mesh size should be given on curves. Once the surface mesh is ready then it should be extruded to get the volume mesh. This can be done as follows:

Mesh Sizing

Select  (Set Curve Mesh Size) icon from **Mesh Tab Menubar**, which pops up **Curve Mesh Size** window as shown in Figure 4.242.

Click on  (Selecr Curve(s)) button and select all the curves by pressing **'a'** (ensure that the mouse cursor is in display window) and enter Maximum element Size of **1** for this case and press Apply.

Figure 4.242 Curve MeshSize window



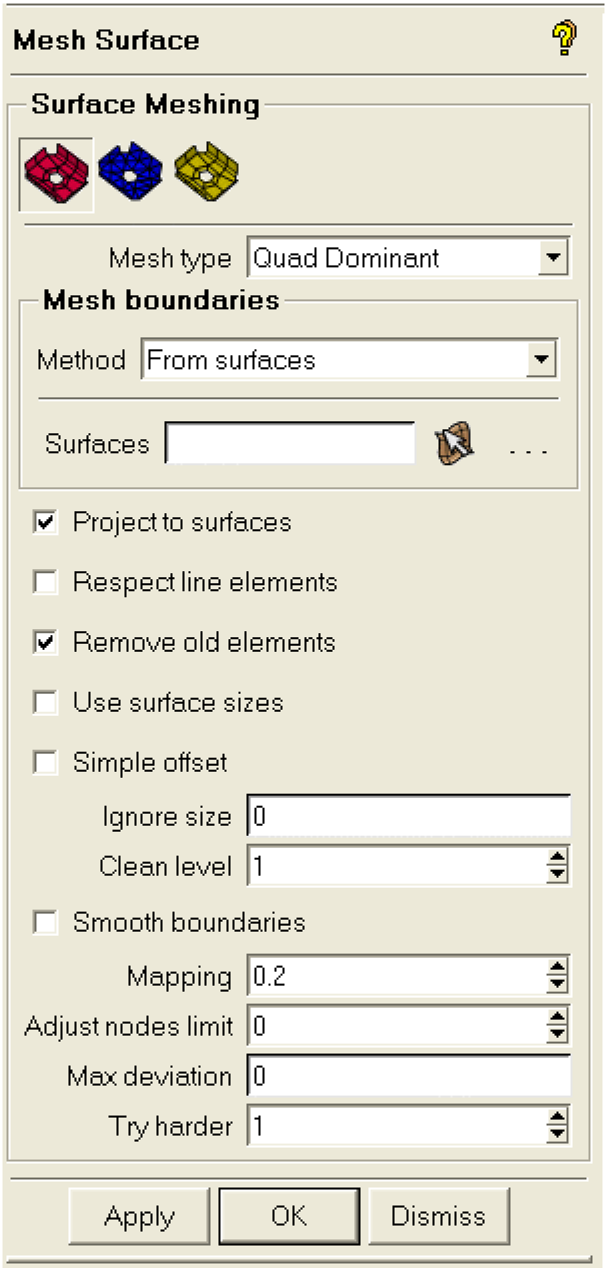
Meshing

Select the (Mesh Shell) icon from **Mesh Tab Menubar**.

Select (Patched Based) Change the Mesh Type from **Quad Dominant** to **All Quad**.

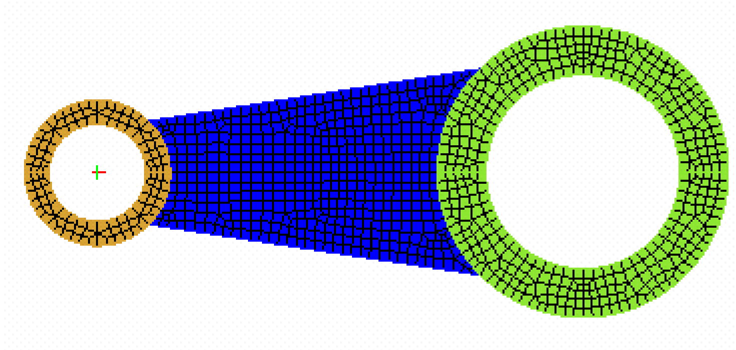
Click on (Select surface(s)) button and select all the surfaces by pressing **'a'** (ensure that the mouse cursor is in display window) and press Apply in the **Mesh Surface** window as shown in Figure 4.243.

Figure 4.243 Mesh Surface window



In Display Tree, click on a branch of Mesh. Click the right mouse button on Mesh > Shells and select **Solid & Wire**. The mesh looks as shown in Figure 4.244.

Figure 4.244 Mesh in Solid & Wire mode



Extrusion of Surface mesh

In Display Tree make sure that under Mesh tree only Shells is **ON** and all others are turned '**OFF**'.

Click on  (Extrude Mesh) icon from Mesh Tab Menubar which pops up **Extrude Mesh** window as shown in Figure 4.246:

Note: Before proceeding for extrusion make sure all the line, point under mesh in the tree wideget are turned **Off**.and similarly points and lines under Geometry are Turned **Off.**

Click  (Select Element) and in the **Select Mesh Element** window press

Select all Surface Element  select button and to select the entire Surface mesh elements as shown in Figure 4.245.

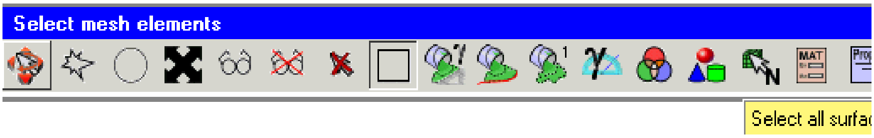


Figure 4.245 Select Mesh Element Window

Enter the **New volume Part Name** as EXTRUDED

Select the **Method of extrusion** as Extrude by vector,

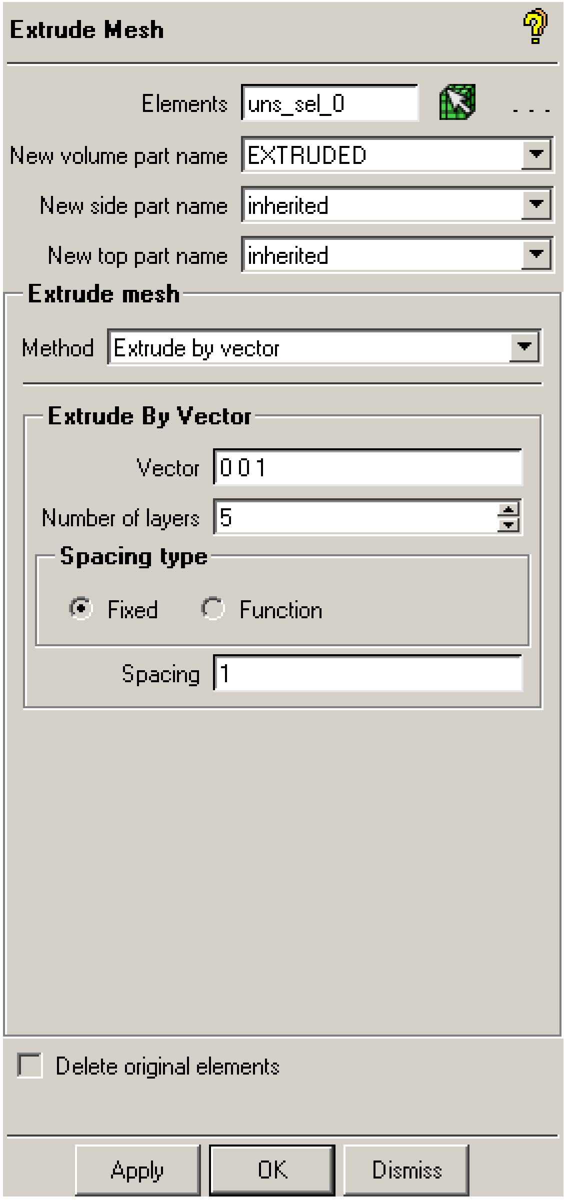
Enter Vector as **0 0 1**.

Number of Layers as **5**.

Spacing as **1** and rest of the option as default.

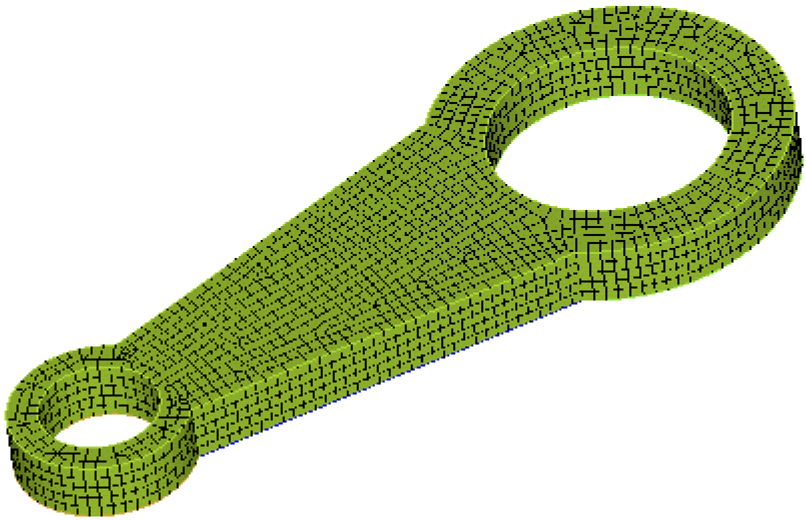
Press Apply.

Figure 4.246 Extrude Mesh window



Switch **Off** Shell and Geometry under the Display Tree and Switch '**On'** Volume > Solid and Wire, the mesh looks as shown in Figure 4.247.

Figure 4.247 Extruded mesh



Switch **Off** Mesh and then Switch **On** Mesh.

Note: User can view the default setting of the Mesh in the Displat Tree by switching **Off** the Mesh and then switching **On** the Mesh, by default only Shell and Line are switched '**On'.**

e) **Material and Element Properties**

Before applying Constraints and Loads on elements, define the type of material and assign properties to the elements.

Selection of Material

Select  (Create Material Property) icon from **Properties Tab Menubar**.

Define the Material Name as **STEEL**.

Material ID can be left as **1**.

Select **Isotropic** type from the drop down list.

Define Young's modulus as **207000**.

Define Poisson's ratio as **0.28**.

Define Density as **7.8e-9** and leave other fields as it is.

Press Apply.

Element Properties

Select  (Define 3D Element Properties) icon from the **Properties Tab Menu bar**.

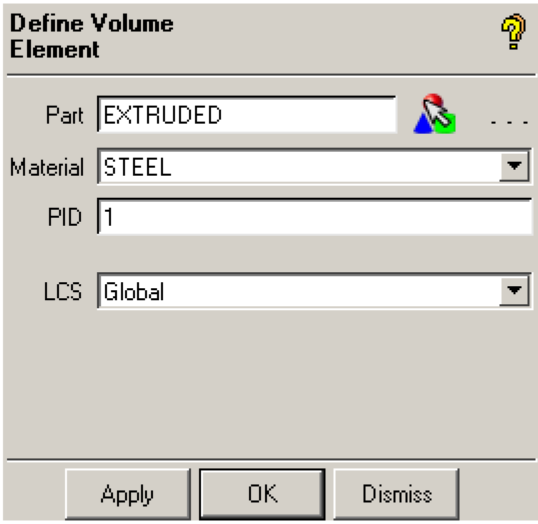
Set PID as **1** in the **Define Volume Element** window as shown in Figure 4.248

Select the Part as **EXTRUDED**.

Select Material as **STEEL**.

Press Apply**.**

Figure 4.248 Define Volume Element window



f) Subsets

Constraints and Loads can be applied on Points, Lines, Surfaces and elements. But here there is an extra option called subset, which may contain a group of surfaces or elements.

Subset1

In Display Tree, click the right mouse button on **Subset** under Mesh and select Create option. This will pop up **Create Subset** window as shown in Figure 4.249.

Ensure that all the geometry entities are turned '**Off**' from Display Model Tree.

Enter Subset as **Subset1** and click on  (Create Subset by Selection) icon in **Create Subset** window.

Toggle '**Off**' Points, Lines and Volume under Mesh in the Display Tree.

To select the elements on Crank end for this subset click on  (Selct Element(s)) button and press "**p**" from key board (ensure that the mouse cursor is in display window) which allows to select the shell elements as shown in Figure 4.250 by drawing a polygon shown in Figure 4.250 and press Appl**y**.

Figure 4.249 Create Subset window

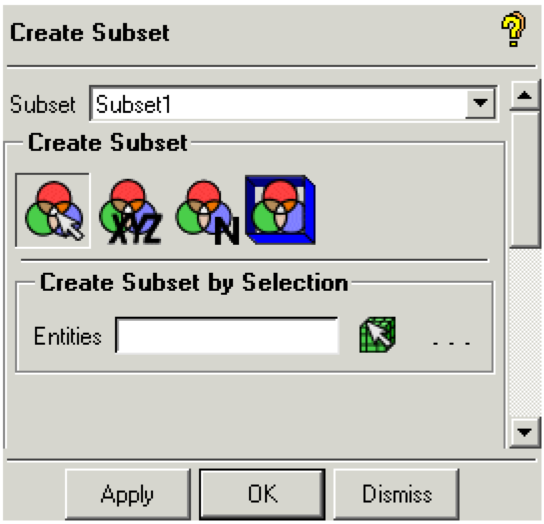
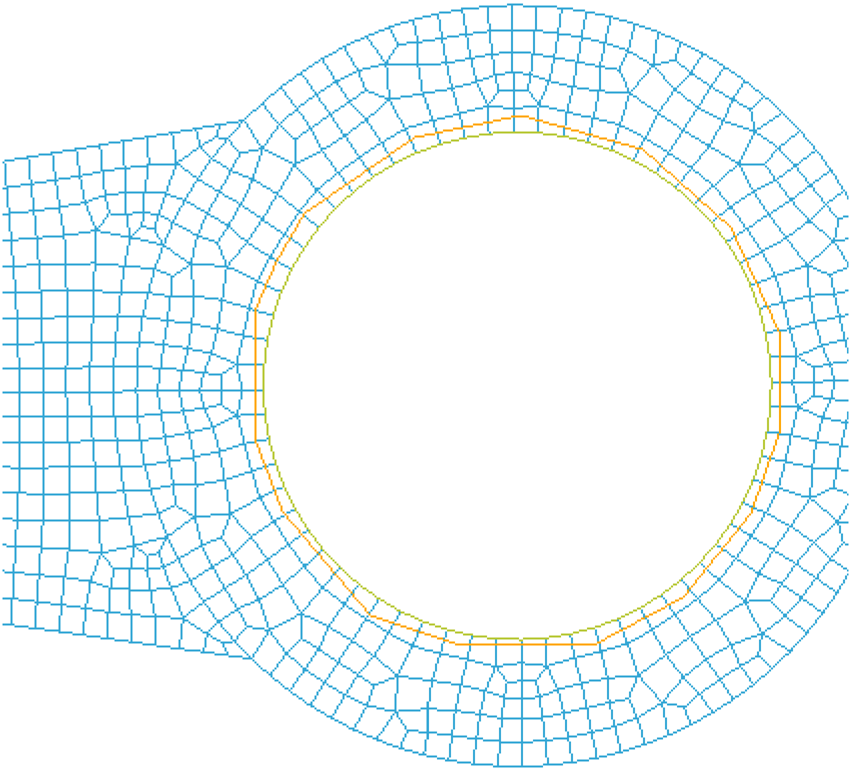
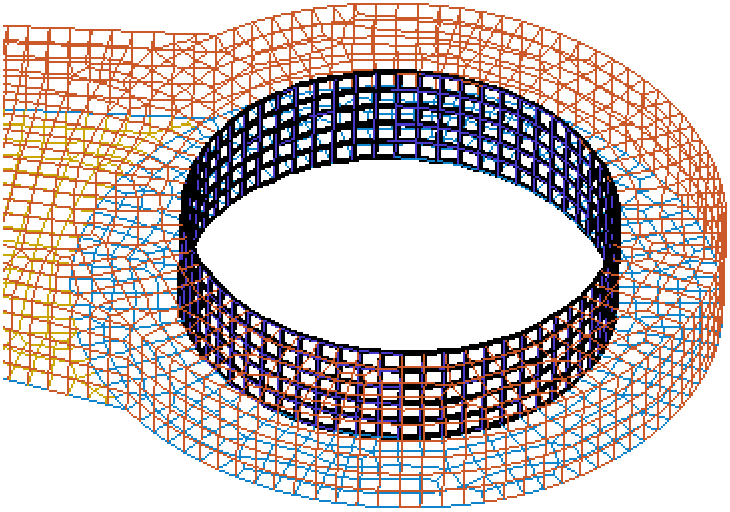


Figure 4.250 Elements selection by polygon and elements selected for Subset 1





Subset2

In Display Tree, click right mouse button on **Subset** under Mesh and select Create option. As shown in the **Create Subset** window in Figure 4.251, to select the elements on Piston end for this Subset.

Ensure that all the geometry entities are turned '**Off**' from Display Model Tree.

Enter Subset as **Subset2** and click on (Create Subset by Selection) icon in Create subset window.

Toggle '**Off**' Points/Nodes, Line/2D & Volumes/3Delements.

To select the elements on Piston end for this subset click on (Select Element (s))button and press "**p**" from keyboard (ensure that the mouse cursor is in display window), which allows selecting the Shell elements by drawing a polygon, explain as selection 4.4.1 and press Apply.

Figure 4.251 Create Subset window

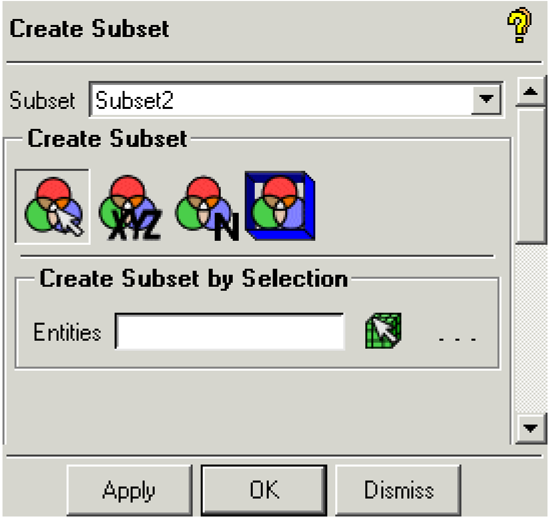
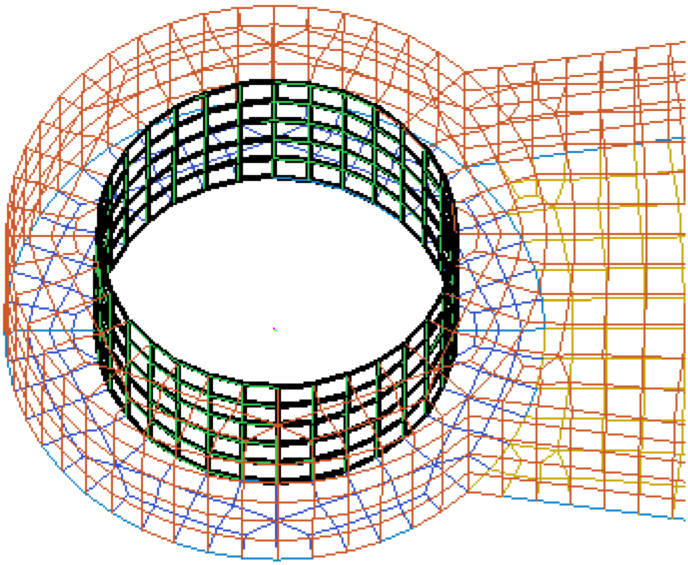


Figure 4.252 Elements selected for Subset2



g) Constraints and Loads

Constraints

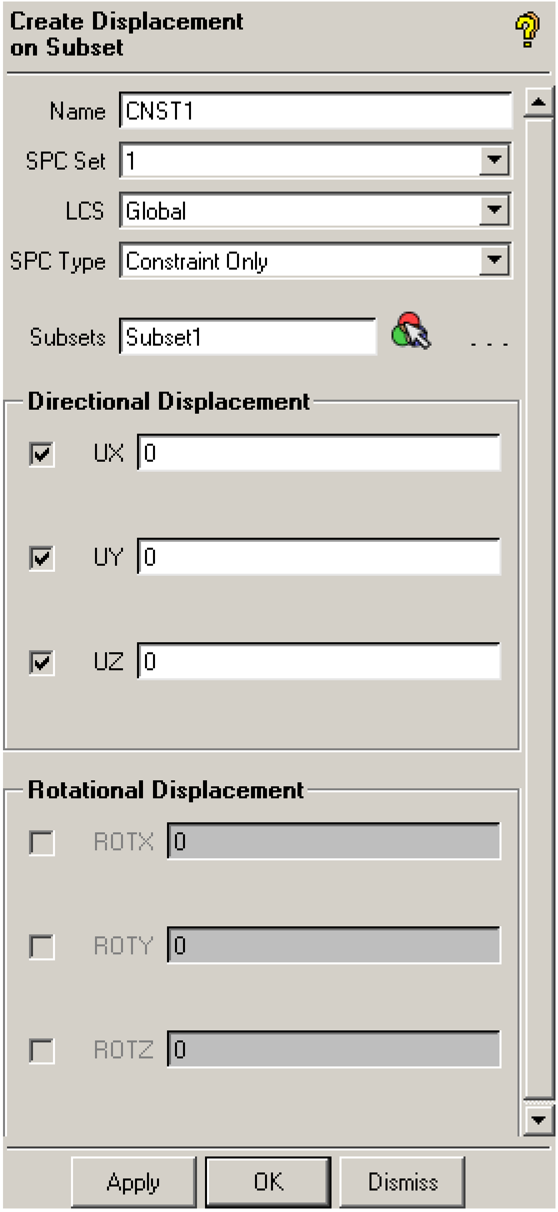
Click on  (Displacement on Subset) icon from **the Constraints Tab Menubar**, which pops up **Create Displacement on Subset** window given in Figure 4.253.

In this window, enter Name as **CNST1** and toggle **ON** options UX, UY and UZ of Directional Displacement.

Click on  (Select Subset) button and select **Subset1** for subsets as shown in Figure 4.253 and press Apply.

Turn '**Off**' **Displacement** display from Model Tree.

Figure 4.253 Create Displacement on Subset window



Loads

Click on (Pressure on Subset) from **Loads Tab Menubar**, which will pop up **Place Pressure on Subsets** window shown in Figure 4.254.

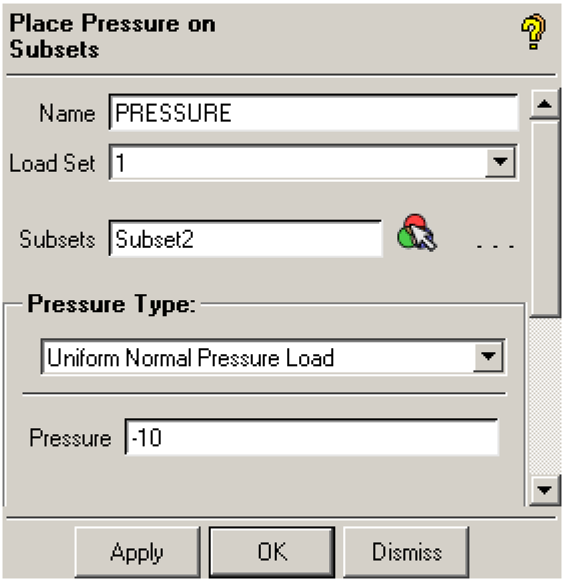
Enter Name as **PRESSURE.**

Click on (Select Subset) button and select **Subset2** for subsets as shown in Figure 4.254.

Enter a value of **"–10"** (negative value) for Pressure and press Apply**.**

Turn '**Off**' Loads display from Display Model Tree.

Figure 4.254 Place Pressure on Subsets window



h) Solver Setup

Setup Nastran Run

First, user should select the appropriate solver before proceeding further.

Select Setting > Solver from Main menu and select Solver as **NASTRAN** as shown in Figure 4.255 and press 'Apply'.

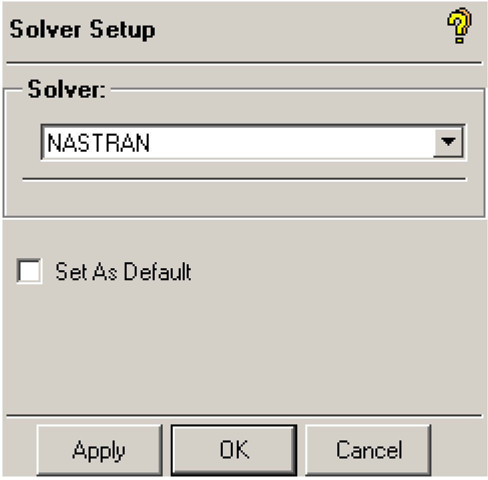


Figure 4.255 Solver Setup window

Click on  (Setup Analysis Type) icon from **Solve Options Tab Menu bar** to setup Nastran run to do Linear Static Analysis that will pop up **Setup Analysis Type** window as shown Figure 4.256.

In the **Setup Analysis Typ**e window, do the following:

Select **Run Type** as Linear Static (Sol 101),

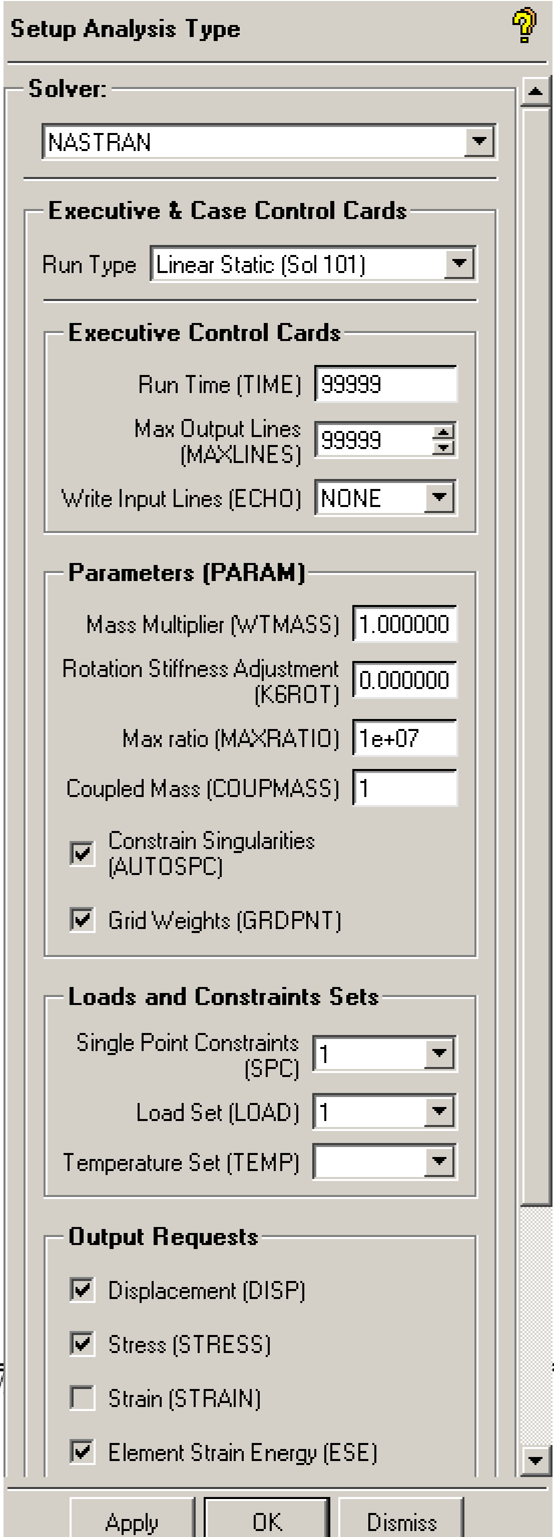
Make sure that **Constraint Singularities** (AUTOSPC) and **Grid Weights** (GRDPNT) is turned **ON.**

For the Default Sets, select Single Point Constraints (SPC) and Load Set (LOAD) as 1,

In the Output Requests toggle ‘**ON’** Displacement (DISP), Stress (STRESS) and Element Strain Energy (ESE).

Press Apply to complete the setup.

Figure 4.256 Setup Analysis Type window



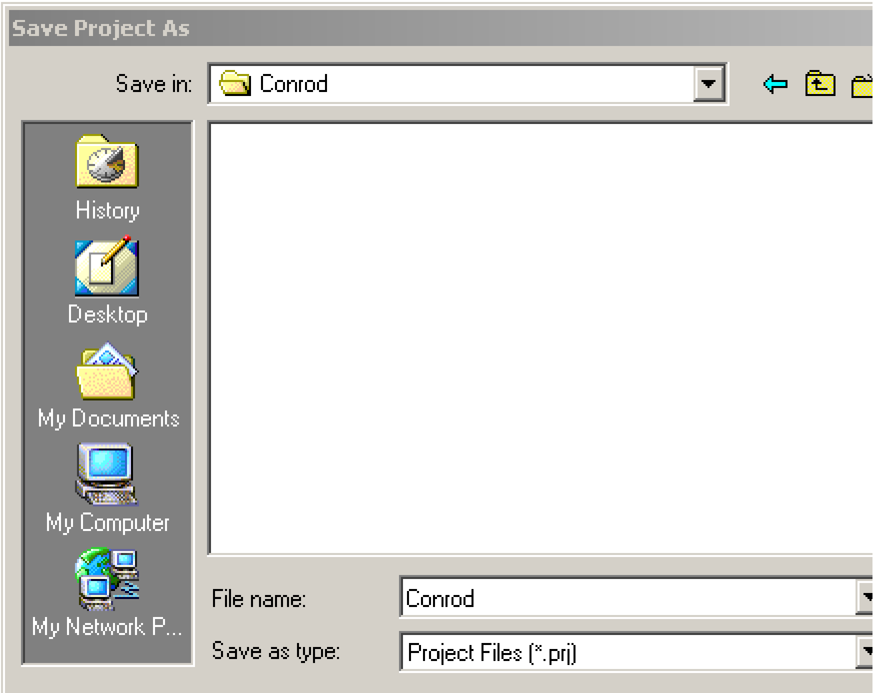
Save Project

Through File > Save Project As option, create new directory **Conrod** as said in earlier tutorials.

Enter **Conrod** as project name and press '**Save**' to save all these information in this directory as shown in Figure 4.257.

It will save four files, geometry file, mesh file, attribute file and parameter files as Conrod.uns, Conrod.fbc and Conrod.par respectively along with the supplied project file - Conrod.prj.

Figure 4.257 Save Project As window

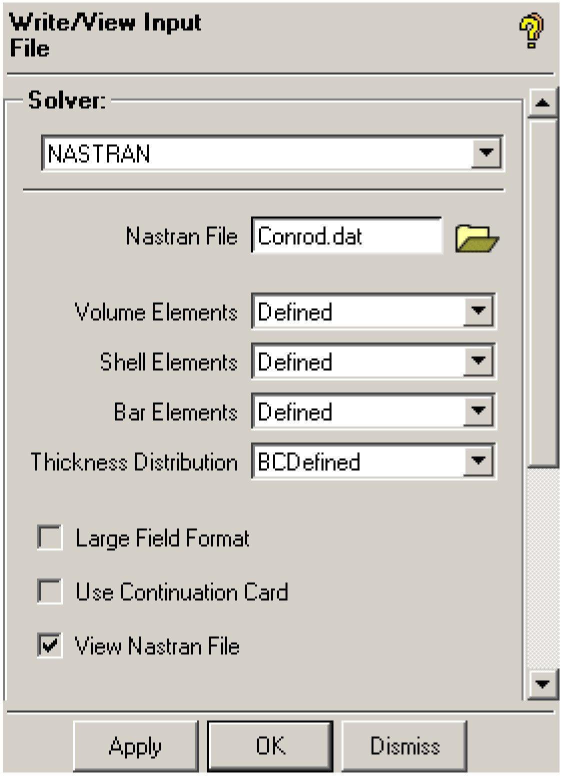


Write Nastran Input File

Click  (Write/View Input File) icon from **Solve Options Tab Menubar**.

Enter the Nastran file name as **Conrod.dat** and switch **ON** View Nastran file as shown in **Write/View Input File** window presented in Figure 4.258 and press Apply.

Figure 4.258 Write/View Input File window



User will see that the Nastran input data file comes up in the default text editor. If user likes to edit this file directly then this can be done and can save the edited file through this text editor. Since no need to do any editing for this example, just close the editor.

i) Solution and Results

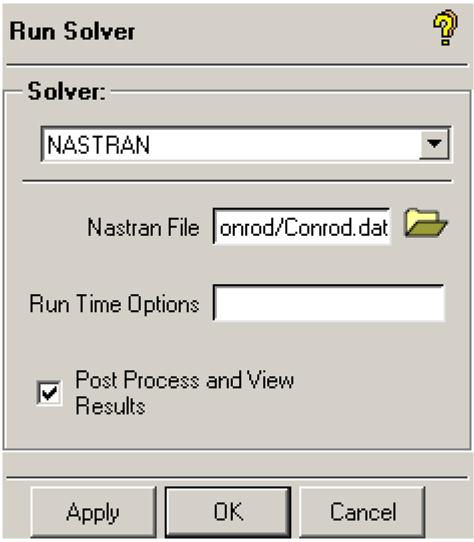
Linear Static analysis is to be performed on this model and the results should be visualized in a post processor.

Solving the problem

Click on (Submit Solver Run) icon from the **Solve Options Tab Menubar** to start Nastran as shown in Figure 4.259. The Nastran file will be selected by default as **Conrod.dat**.

Toggle **ON** Post process and View Results and press Apply in **Run Solver** window.

Figure 4.259 Run Solver window

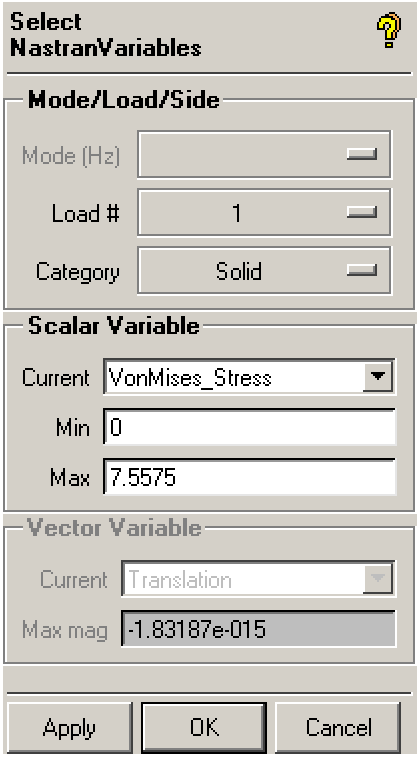


Post Processing of Results

After completion of Nastran run, the results will be automatically loaded into the post processor Visual3p.

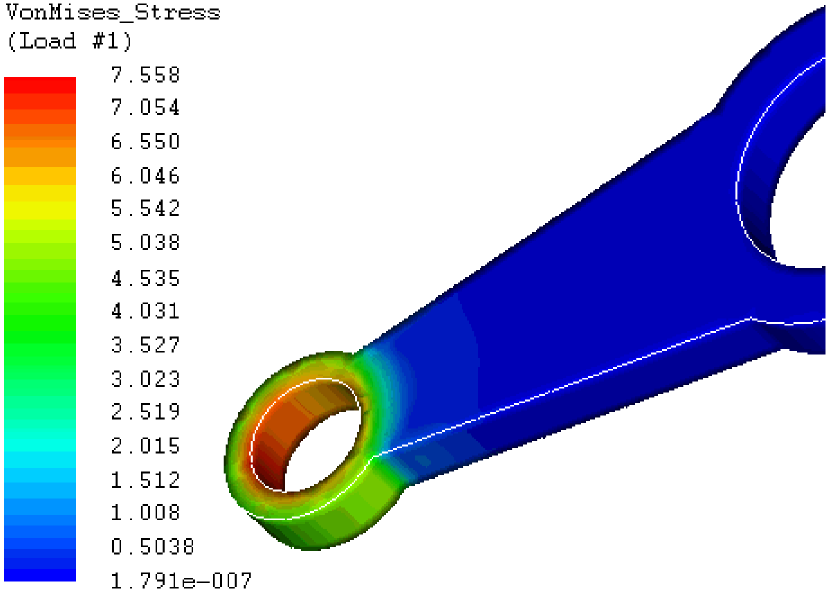
Click on **Variables** option in **Post-processing** Tab menu bar. In Select Nastran Variables window, select **Category** as **Solid** and Current Scalar Variable as **VonMises_Stress** as shown in Figure 4.260. The VonMises Stress distribution is shown in.Figure 4.261

Figure 4.260 Nastran Variables window



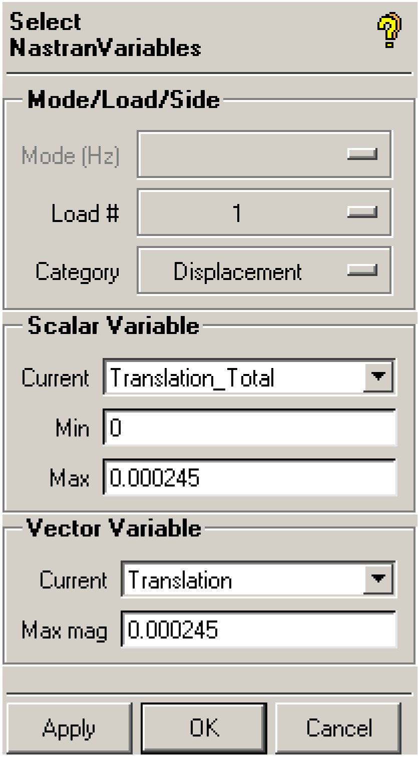
Note: The results shown here are obtained by '**MSC Nastran**' Solver.

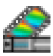
Figure 4.261 VonMises Stress Distribution



To display mode shape at Total Translational Frequency, select side as **Single** and variable as **Translational_Total** from the **Nastran Variables** window as shown in Figure 4.262.

Figure 4.262 Nastran Variables window



Select  (Control All Animation) option from **Post-processing** tab menu bar which will open Animation Controller window as shown in.Figure 4.263

Set the values as shown in Figure 4.263 and press (Animate) to view the mode shape as shown in Figure 4.264

Finally select **Exit** to quit the post processor.

Figure 4.263 Animation Setup and Controller window

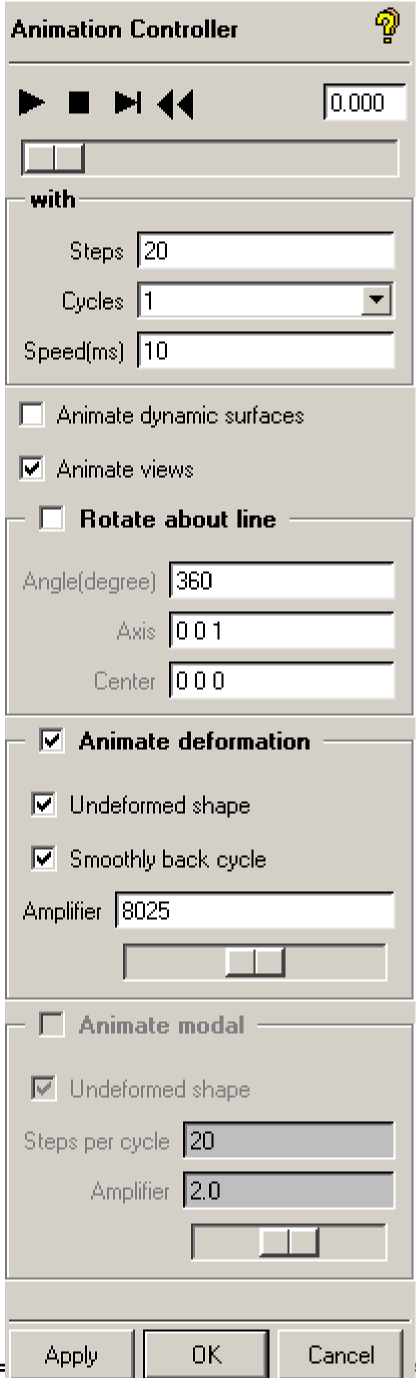
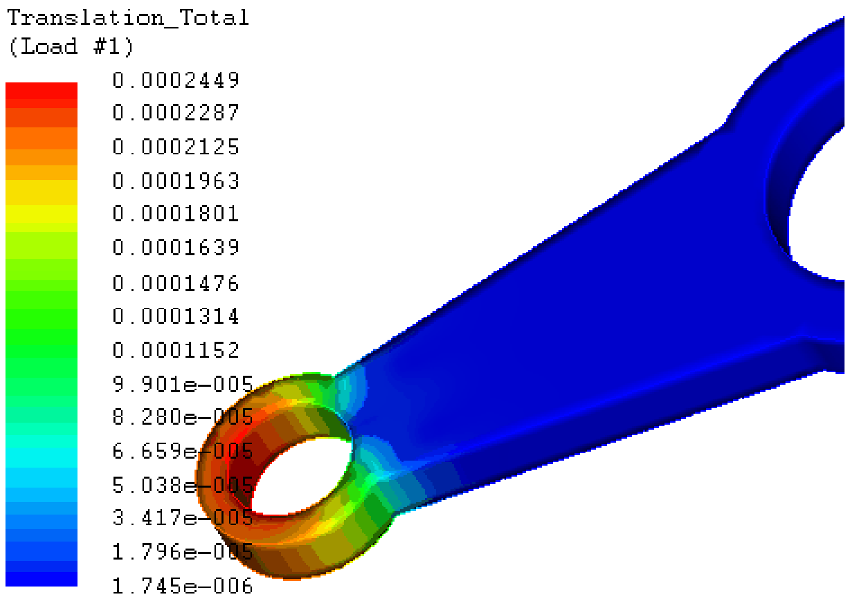


Figure 4.264 Animated model



4.3.5: Hood

This exercise explains import of existing Nastran data, modifying that data and rewriting the new Nastran data. It also explains solving the problem and visualization of results. The imported Hood model is shown in Figure 4.266.

a) Summary of Steps

Launch AI*Environment and import an existing Nastran data file

Data Editing

Verification of imported data

Modify some element properties

Save Changes of imported data

Write Nastran Input File

Solution and Results

Solving the problem

Visualization of Results

b) Launch AI*Environment and Import Data

Launch the AI*Environment from UNIX or DOS window. Then, File > Change working directory, and set $ICEM_ACN/../docu/FEAHelp/AI_Tutorial_Files. Next, File > Import Mesh > Nastran which pops up the window shown in Figure 4.265. Select the file 'Hood.dat', leave other option as default and press Apply to import the data. Figure 4.266 shows the imported data in AI*Environment.

Figure 4.265 Import Nastran File window

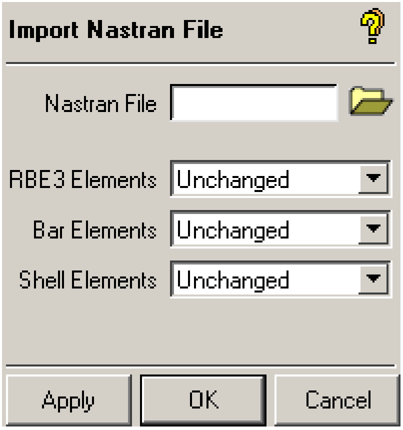
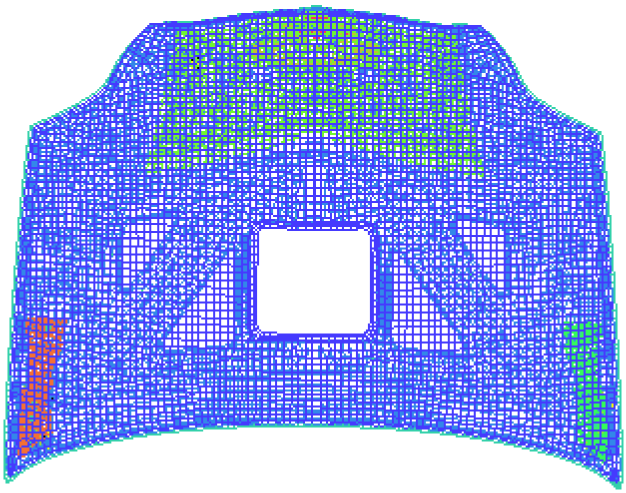


Figure 4.266 Hood model

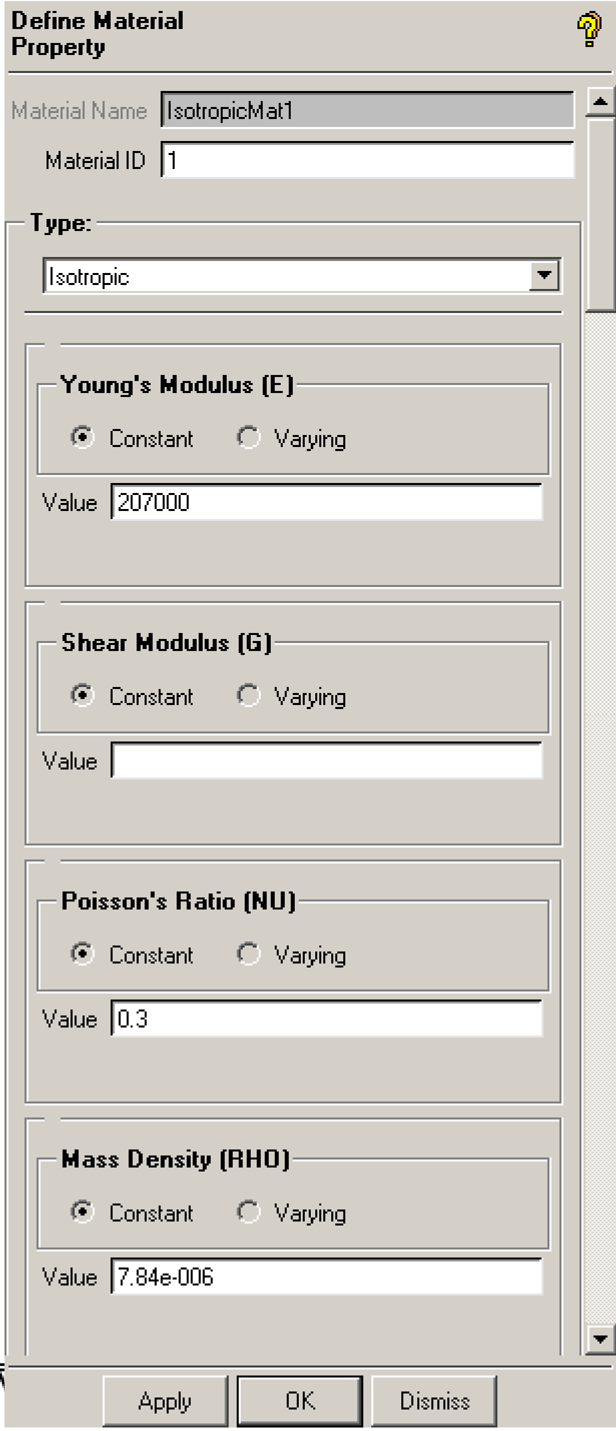


c) Data Editing

Verification of Imported data

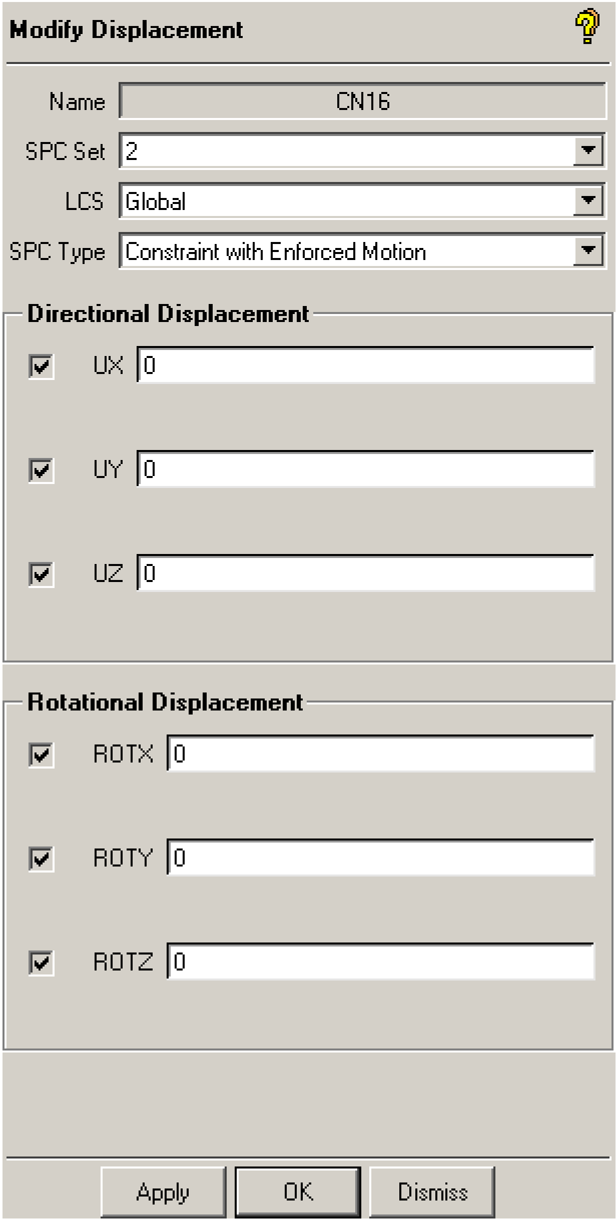
Expand M**aterial Properties** in Model Tree by clicking on +, select **IsotropicMat1**. To open **Define Material Property** window as shown in Figure 4.267, double click on the selected Material (IsotropicMat1) with left mouse button.

Figure 4.267 Define Material Property



Expand **Displacement** in Model Tree by clicking on + and also expand **Set 2** and double click on **CN16** which pops up **Modify Displacement** window as shown in Figure 4.268.

Figure 4.268 Modify Displacement window

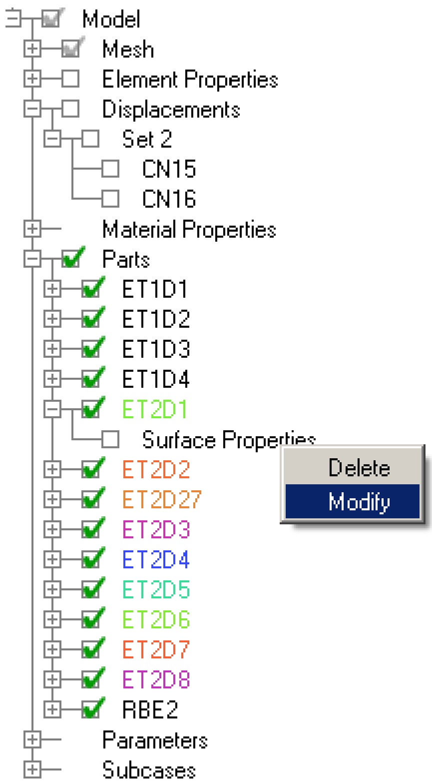


Modifying thickness

Expand the **Part** option in Model Tree by clicking on + and also expand **ET2D1** as shown in Figure 4.269.

Right Click on **ET2D1>Surface Properties>Modify**.

Figure 4.269
Display Tree

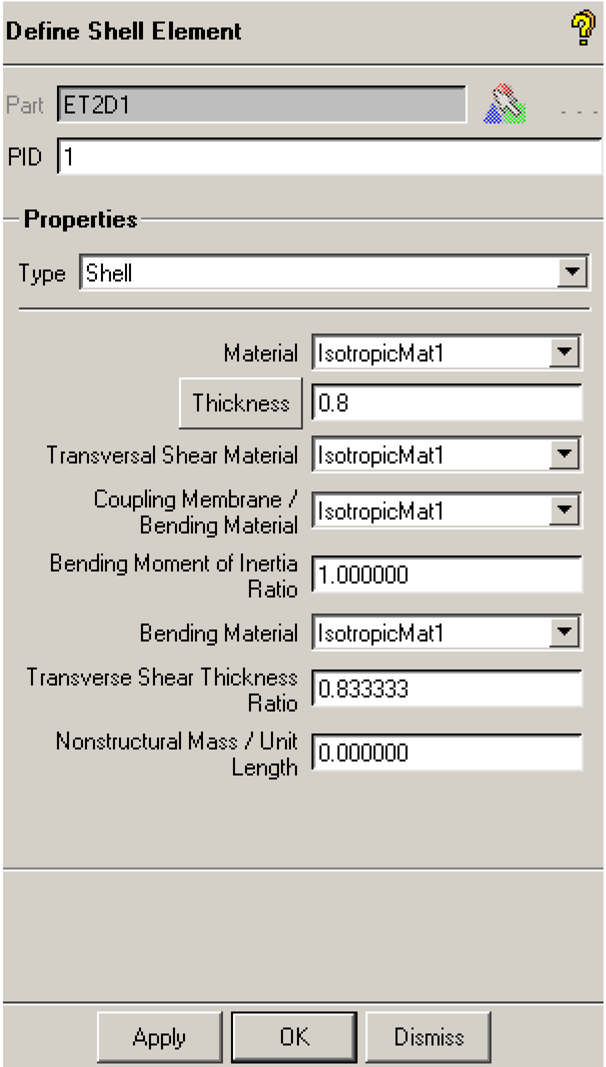


The window that pops up is shown in Figure 4.270.

Change the value of Thickness from **0.75** to **0.8** and press Apply.

Similarly select **ET2D2** and change the value of Thickness from **0.7** to **0.8** and press Apply

Figure 4.270 Define Shell Element window



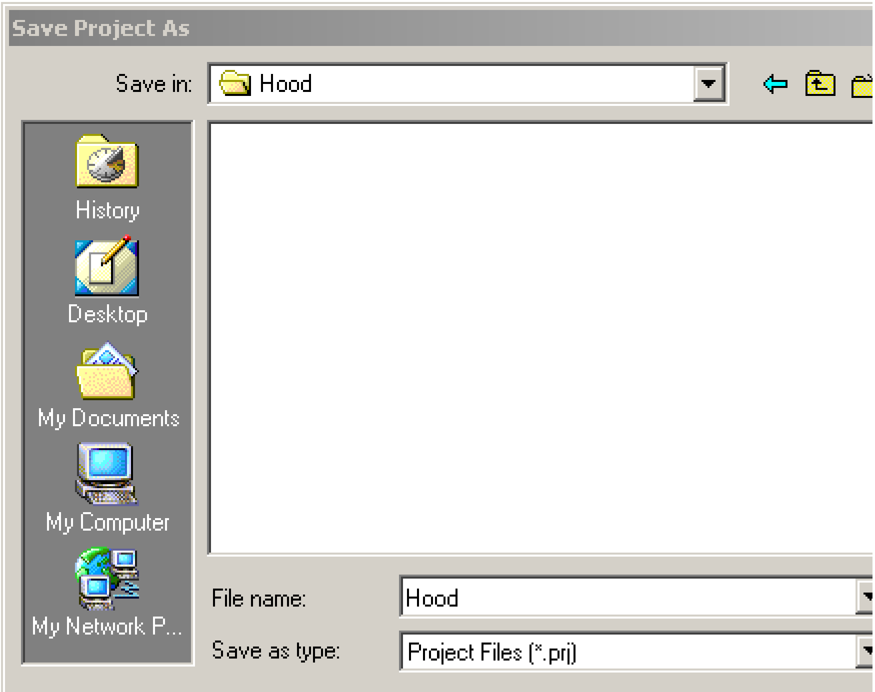
Save Project

Through File > Save Project As option, create new directory **Hood** as said in earlier tutorials.

Enter **Hood** as project name and press '**Save'** to save all these information in this directory as shown in Figure 4.271.

Along with the Hood.prj file, it will also store three other files, Mesh file, Attribute file and Parameter files as Hood.uns, Hood.fbc and Hood.par respectively.

Figure 4.271 Save Project As window



d) Write Nastran Input

First, user should select the appropriate solver before proceeding further.

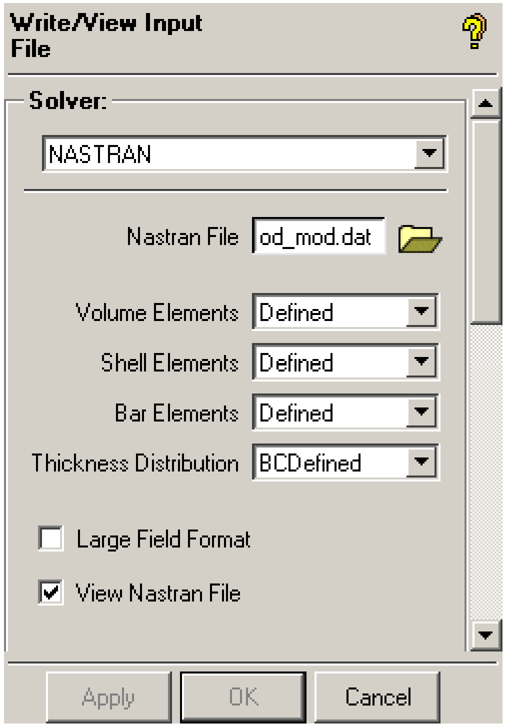
Select Settings > Solver from Main menu and press Apply as shown in Figure 4.272.

Figure 4.272 Solver Setup window

Click (Write/View Input File) icon from the Solve Options Tab Menubar.

Enter the Nastran file name as **Hood_mod.dat** and switch **ON** View Nastran file in **Write/View Input File** window as shown in Figure 4.273 and press Apply.

Figure 4.273 Write/View Input File window



User will see that the Nastran input data file comes up in the default text editor. If the user likes to edit this file directly, then he can do the editions and can save the edited file through this text editor. Since no need to do any editing for this example, just close the editor.

e) Solution and Results

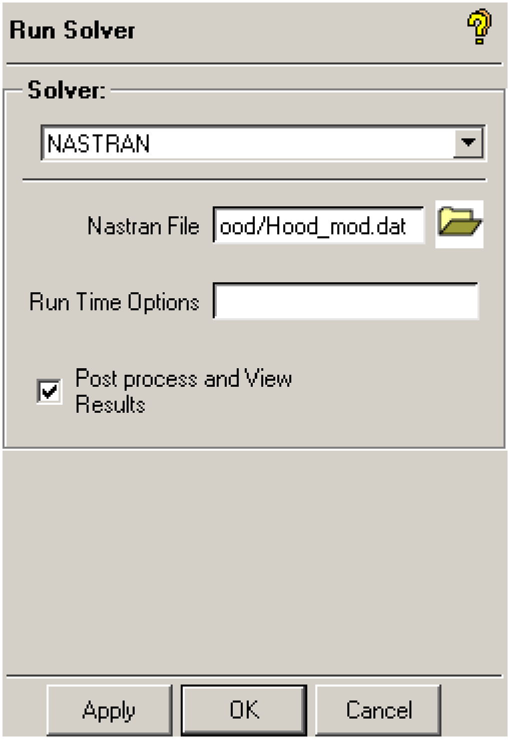
Modal analysis is to be performed on this model and the results should be visualized in a post processor.

Solving the problem

Click on  (Submit Solver Run) icon from the Solve Options Tab Menubar to start Nastran with Nastran Input File window given in Figure 4.274. Supply Nastran file as **Hood_mod.dat**.

Toggle **ON** Post process and View Results and press Apply in Run Solver window.

Figure 4.274 Run Solver window



Post Processing of Results

After completion of Nastran run, the results will be automatically loaded into the post processor Visual3p.

Click on  **Variables** option in **Post-processing** Tab menu bar. In Select Nastran Variables window, set Scalar Variable as **Translation_Total,** as shown in Figure 4.275, and press Apply. The Translation _Total distribution is shown in Figure 4.277.

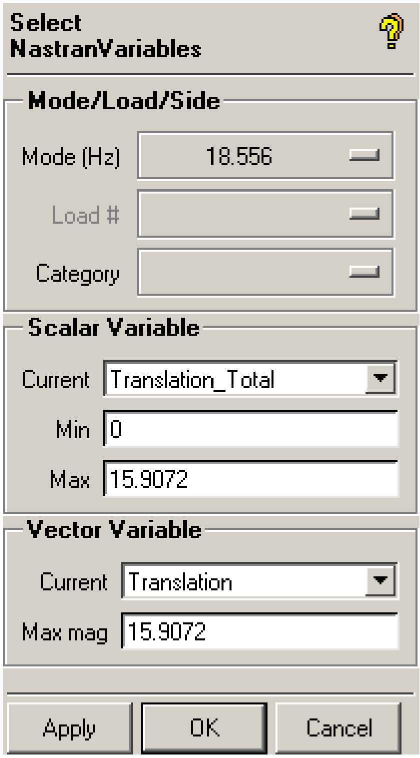
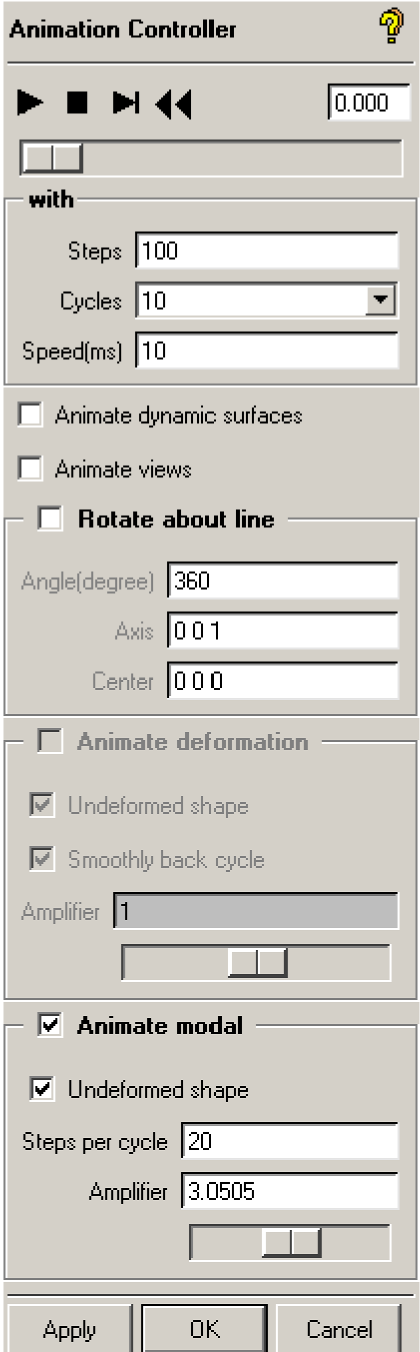


Figure 4.275 Select Nastran Variables window

Note: Results shown here are obtained by MSC Nastran run. Results may differ with those of AI*Nastran run depending on the version.

Figure 4.276 Animation Setup and Controller window

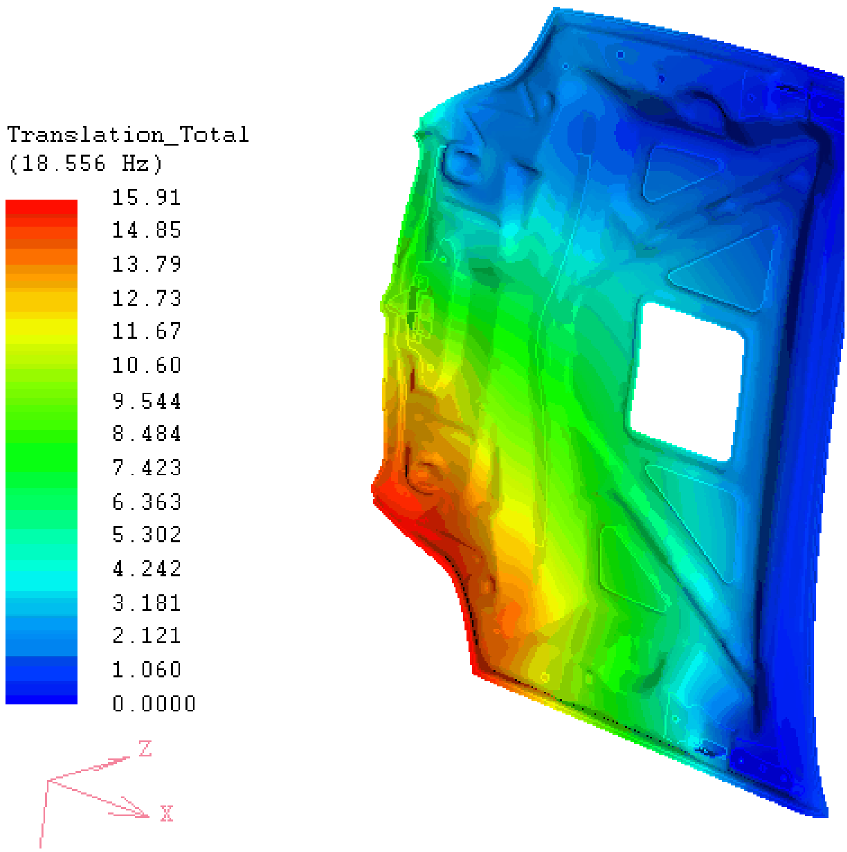


Select (Control All Animation) option from **Post-processing** tab menu bar which will open Animation Controller window as shown in Figure 4.276.

Set the values as shown in Figure 4.276 and press (Animate) to view the mode shape as shown in Figure 4.277

Finally select **Exit** to quit the post processor.

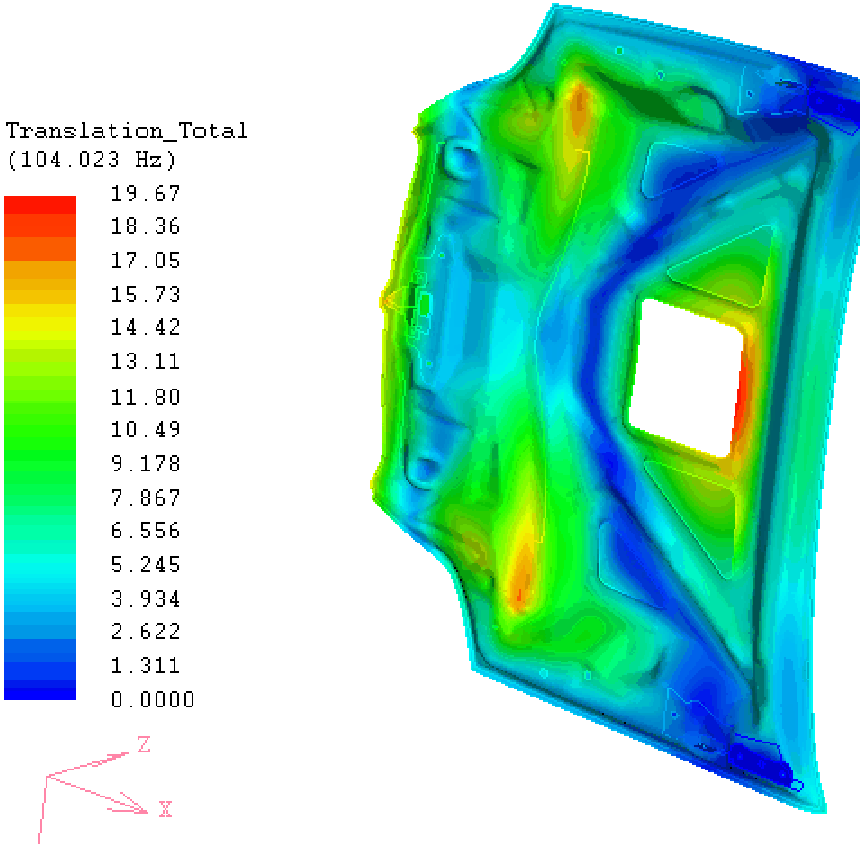
Figure 4.277 Animated model at 18.556 Hz



Similarly, for the frequency **104.023** HZ also, the result can be animated as shown in Figure 4.278

Finally select **Exit** to quit the post processor.

Figure 4.278 Mode shape at 104.023 Hz



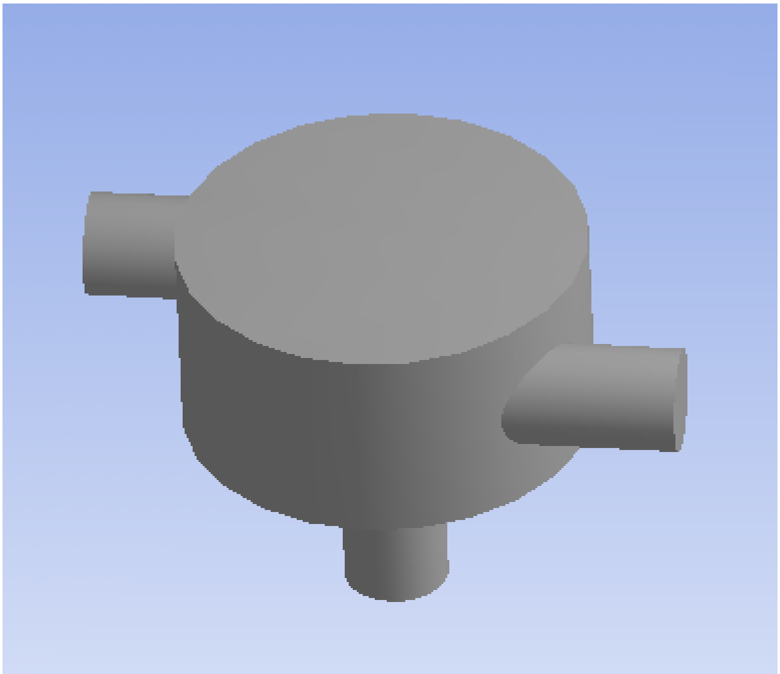
5: ANSYS ICEMCFD - CFX Tutorials

5.1: Static Mixer

5.1.1: Overview

This tutorial covers geometry creation and meshing for a simple static mixer using **ANSYS Workbench DesignModeler and Advance meshing - CFX**. It is intended to be compatible with CFX-5 Tutorial 1, Flow in a Static Mixer. This tutorial would effectively replace the section entitled Creating the Model in CFX-Build. After completing this tutorial, the user could complete the remaining as sections of the CFX-5 Static Mixer tutorial, picking up with Defining the Simulation in CFX-Pre.

Figure 5.1
Static Mixer Geometry



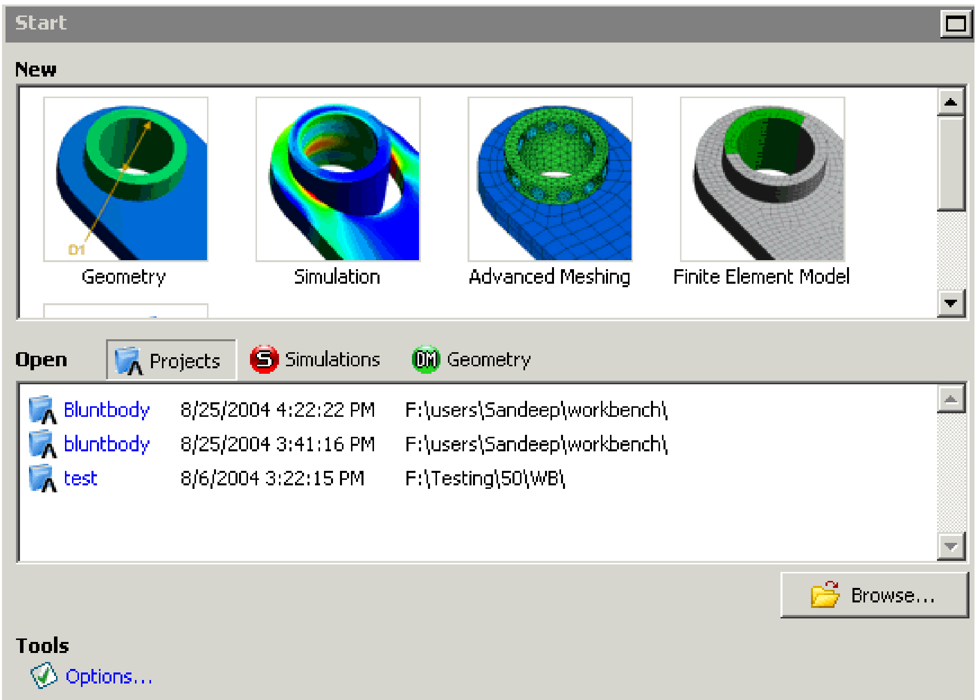
a) Steps Involved in this Example

- Creating Geometry in DesignModeler.
- Automatically generating a tetrahedral mesh in Advance meshing.
- Writing input mesh file for CFX-5.

b) Starting a New Project

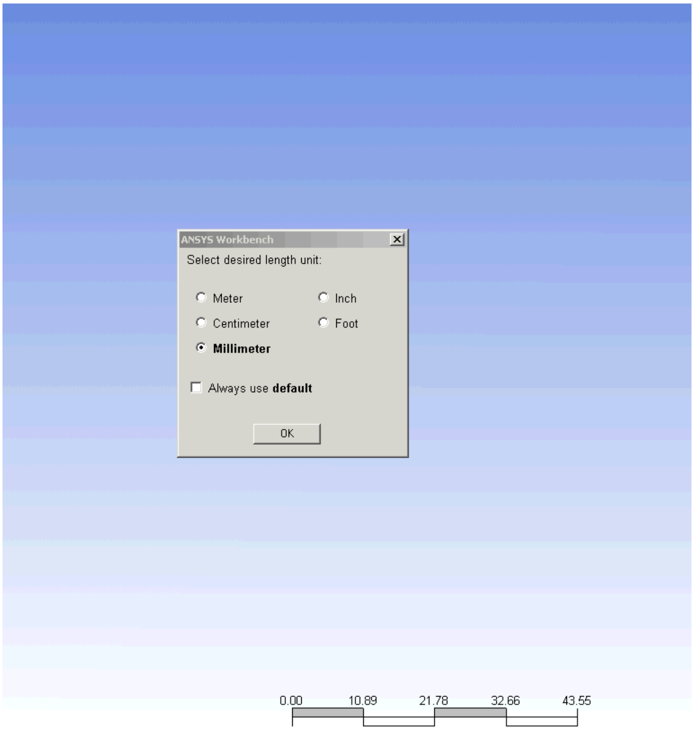
Launch the ANSYS Workbench, ANSYS Workbench window will appear then select Geometry tab.

Figure 5.2
Selection window



This will open DM [DesignModeler] window. Another ANSYS Workbench window will pop up for selection of desired length unit, select Millimeter and press OK. The DesignModeler and desired unit window is shown in Figure 5.3

Figure 5.3 Workbench window



c) Geometry Creation

This software is designed to allow a maximum flexibility to the user about how and where geometry models are created. This tutorial covers the creation of a model "Simple Static Mixer" geometry using DesignModeler, the geometry creation tools contained within the **ANSYS Workbench** itself. To create the model numerous alternatives exist.

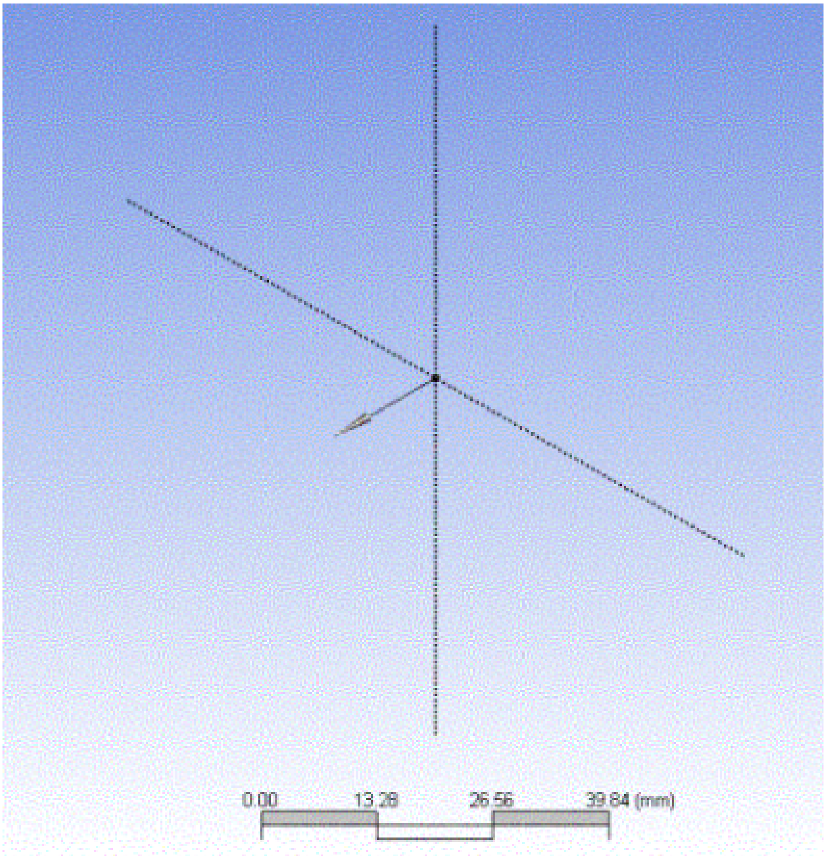
The geometry for this tutorial is divided in two sub steps. Revolving the profile curve about the vertical axis of the mixer will generate the main body of the mixer. Then the inlet pipe will be generated. The detailed description is as follows.

d) Creating Main Mixer Body

Creation of Profile Curves

Select the XY Plane from the project tree which is located at the left upper side of the main window. It will display the XY plane in the graphics window as shown in Figure 5.4

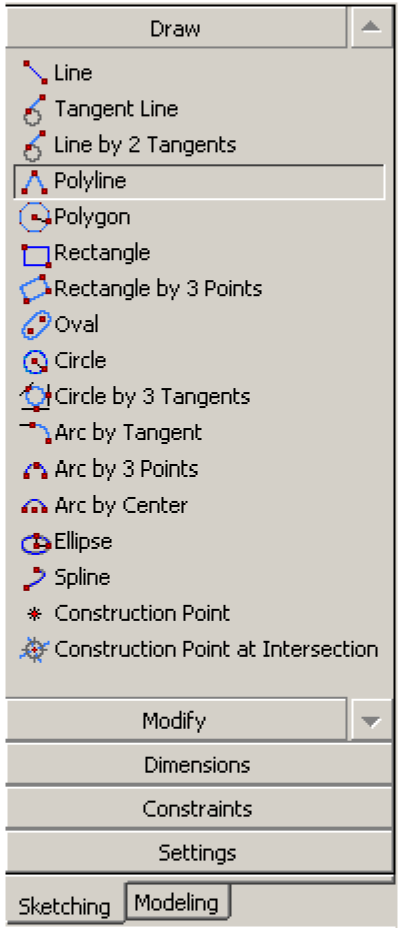
Figure 5.4 Workbench graphics window



Select (Look at Face/Plane/Sketch) the icon from main tool bar.

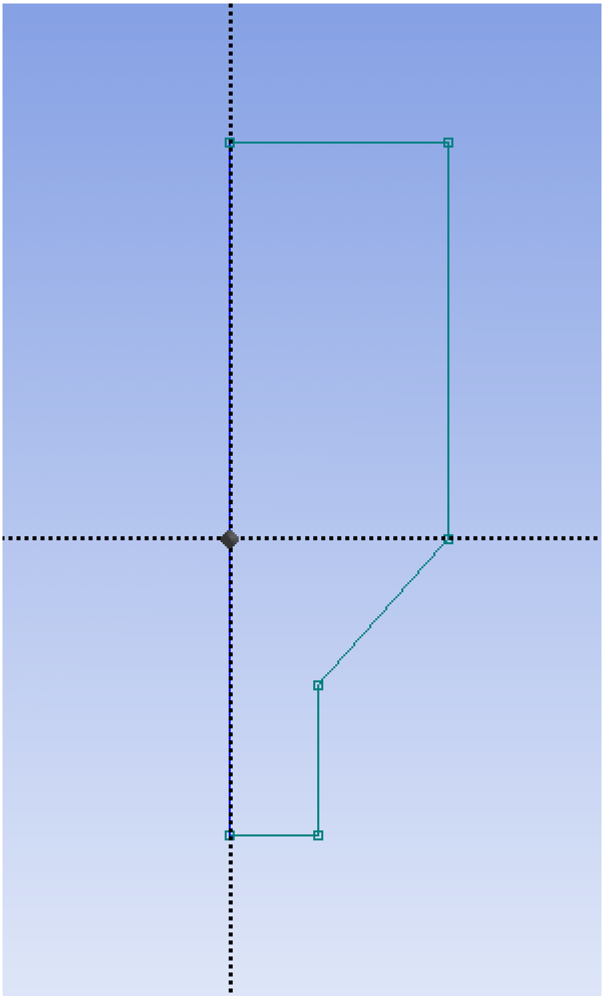
Select **sketching** from project tree; it will open the draw tool bar. Now select polyline from the draw tool bar as shown in Figure 5.5.

Figure 5.5
Dimensions window



Now Draw approximate shape with the help of cursor as shown in Figure 5.6.

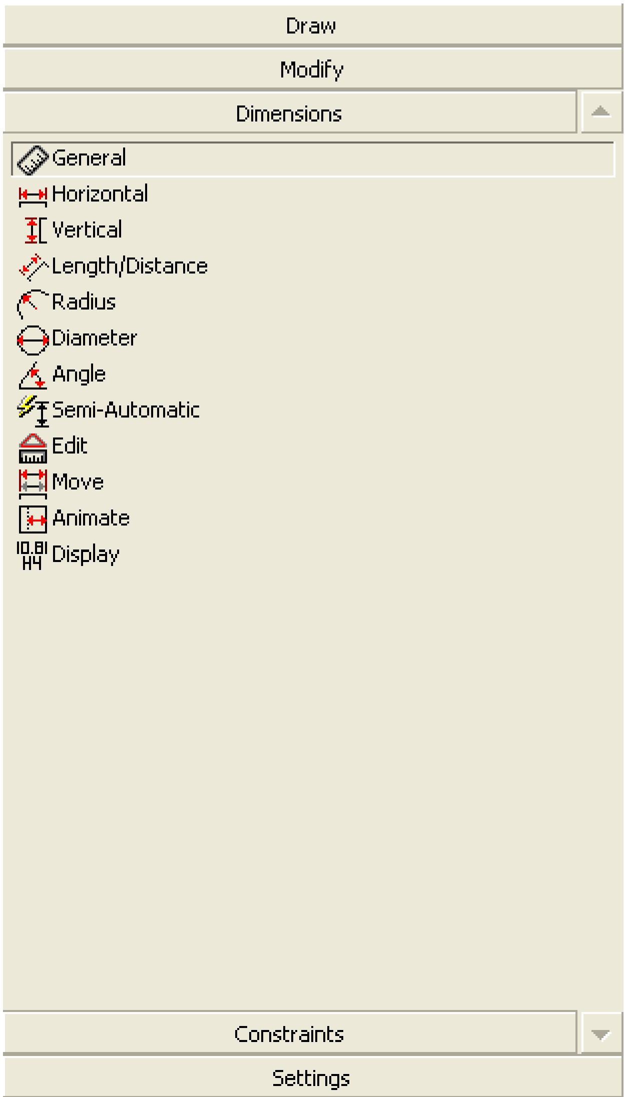
Figure 5.6 Approximate diagram of static mixture body.



After drawing approximate shape to revolve, user has to define the exact dimension to the curves so that shape of the revolved component will match to the geometry. Click on dimensions in the sketching tab. One dimension window will pop up as shown in Figure 5.7.

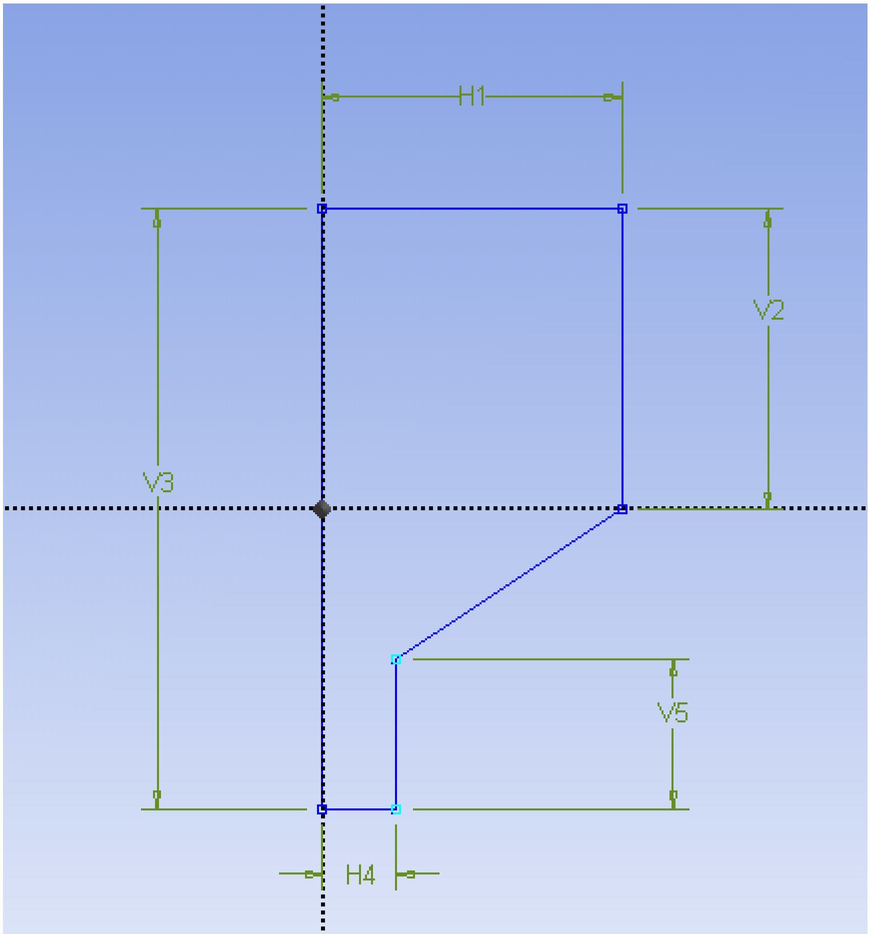
•

Figure 5.7
Dimensions Window



Select general as default option. Take the cursor on to the screen; move on to the edge on which you want to apply the dimensions. Apply the dimensions according to the figures shown Figure 5.8

Figure 5.8 General Deimesions graphics window



After giving the general dimensions,In the left lower corner there is a window called detailed view, enter the values as given in Figure 5.9

Figure 5.9
Exact Dimensions window

Details of Sketch1	
Sketch	Sketch1
Show Constraints?	No
Dimensions: 5	
H1	20 mm
H4	5 mm
V2	20 mm
V3	40 mm
V5	10 mm
Edges: 6	
Line	Line7
Line	Line8
Line	Line9
Line	Line10
Line	Line11
Line	Line12

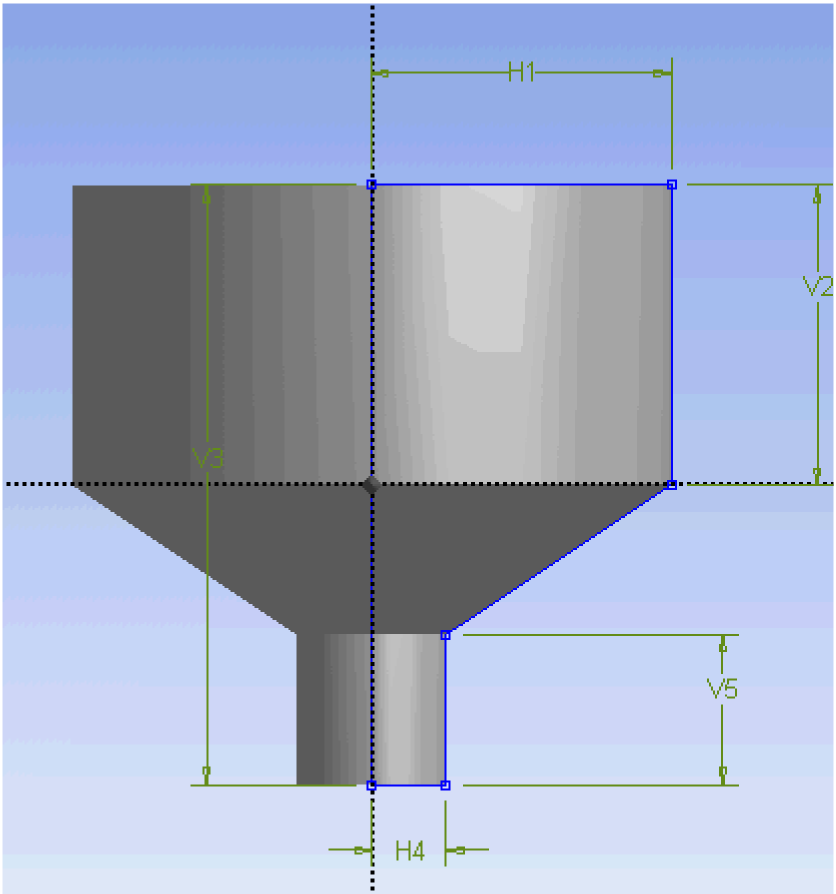
After giving the exact dimensions, Click on **Revolve** in the **3D features toolbar**.This will ask for details in the detail views window at the left bottom side of the screen.Enter the details as per theFigure 5.10. Click on the Axis and select the axis as XY plane from the screen and press **Apply**.

Figure 5.10
Revolve detail window

Details of Revolve2	
Revolve	Revolve2
Base Object	Sketch1
Axis	Selected
Operation	Add Material
Direction	Normal
FD1, Angle (>0)	360 °
As Thin/Surface?	No
Merge Topology?	Yes

Press **Generate** so that it will generate the mixture body as shown in Figure 5.11.

Figure 5.11 Geoemtry after revolution

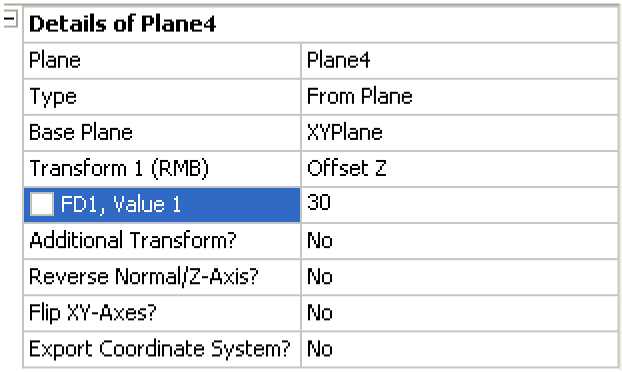


This is the generated mixture body.

Generating the inlet pipes

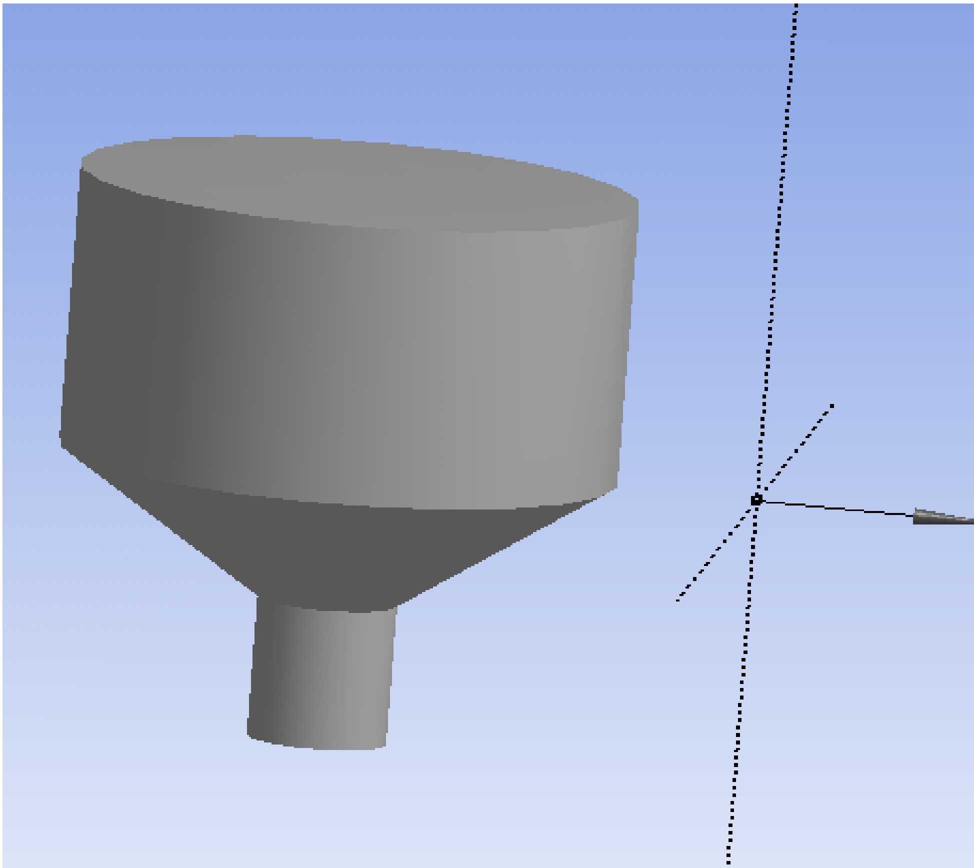
Now we have to generate the two inlet pipes of equal sizes and opposite in direction to each other. For this we have to offset the plane and create the circles on that plane and extrude them. Click on the **XY plane** in the project tree and then click on new plane in the 3D features toolbar. This will come up with the new plane. Offset this plane from original XY plane accordingly by entering the values in the details view as shown in Figure 5.12

Figure 5.12 Offsetting parameter for plane



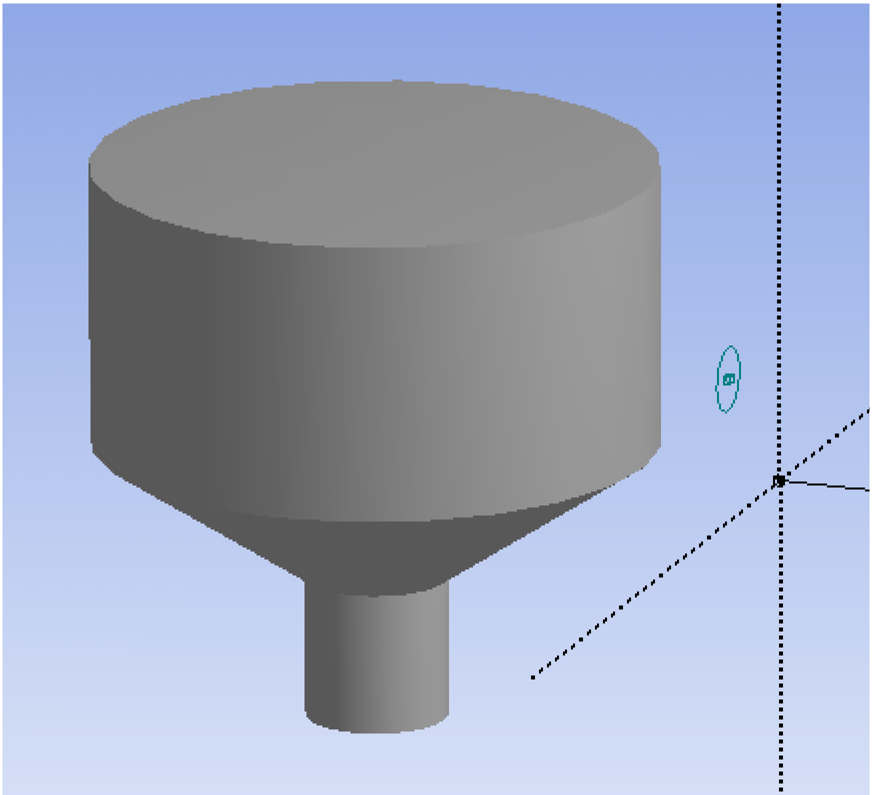
Details of Plane4	
Plane	Plane4
Type	From Plane
Base Plane	XYPlane
Transform 1 (RMB)	Offset Z
FD1, Value 1	30
Additional Transform?	No
Reverse Normal/Z-Axis?	No
Flip XY-Axes?	No
Export Coordinate System?	No

Figure 5.13 Created planes



After generating the offset plane go to the sketching window and select circle. Create a circle of any size in the new plane as shown in figure.

Figure 5.14 Created Circle in random manner



Press dimensions select general and select the horizontal and vertical two dimensions. Then after select the sketch and apply dimensions according to the Figure 5.16

Figure 5.15
General dimension s To circle

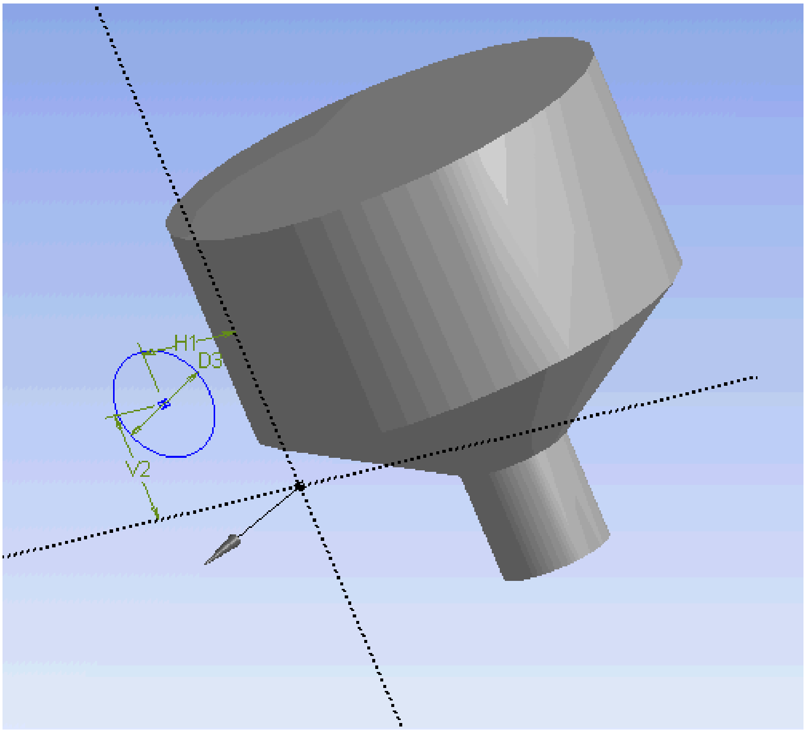


Figure 5.16
Detail view for creating circle

Details of Sketch2	
Sketch	Sketch2
Show Constraints?	No
Dimensions: 3	
D3	10 mm
H1	10 mm
V2	10 mm
Edges: 1	
Full Circle	Cr15

Figure 5.17
Extrude details for first curves

Details of Extrude1	
Extrude	Extrude1
Base Object	Sketch3
Operation	Add Material
Direction Vector	None (Normal)
Direction	Reversed
Type	To Faces
Target Faces	1
As Thin/Surface?	No
Merge Topology?	Yes

After extruding the geometry will appear as shown in the figure below.

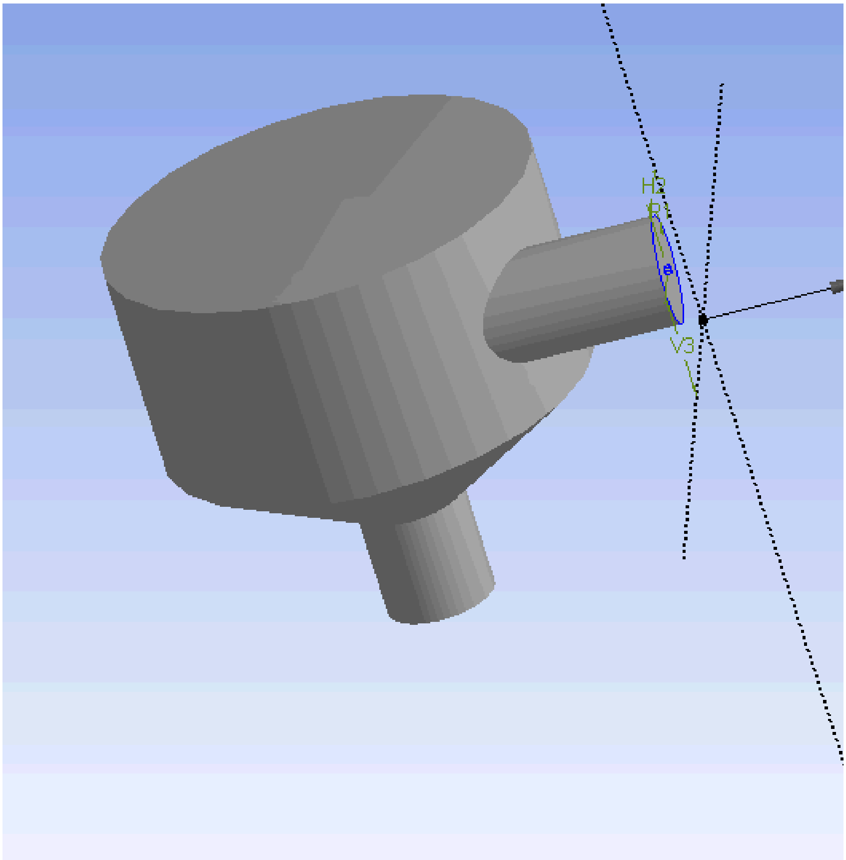


Figure 5.18 Geometry after extrusion of the circle

Now we have to create the same type of extruded pipe on the other side. Select the XY plane in the project tree and select new plane from the main toolbar. This will give new plane on which we will create a circle. Enter the details as shown in Figure 5.19.

Figure 5.19
Create Second Offset plane Details

Details of Plane5	
Plane	Plane5
Type	From Plane
Base Plane	XYPlane
Transform 1 (RMB)	Offset Z
FD1, Value 1	-30 mm
Additional Transform?	No
Reverse Normal/Z-Axis?	No
Flip XY-Axes?	No
Export Coordinate System?	No

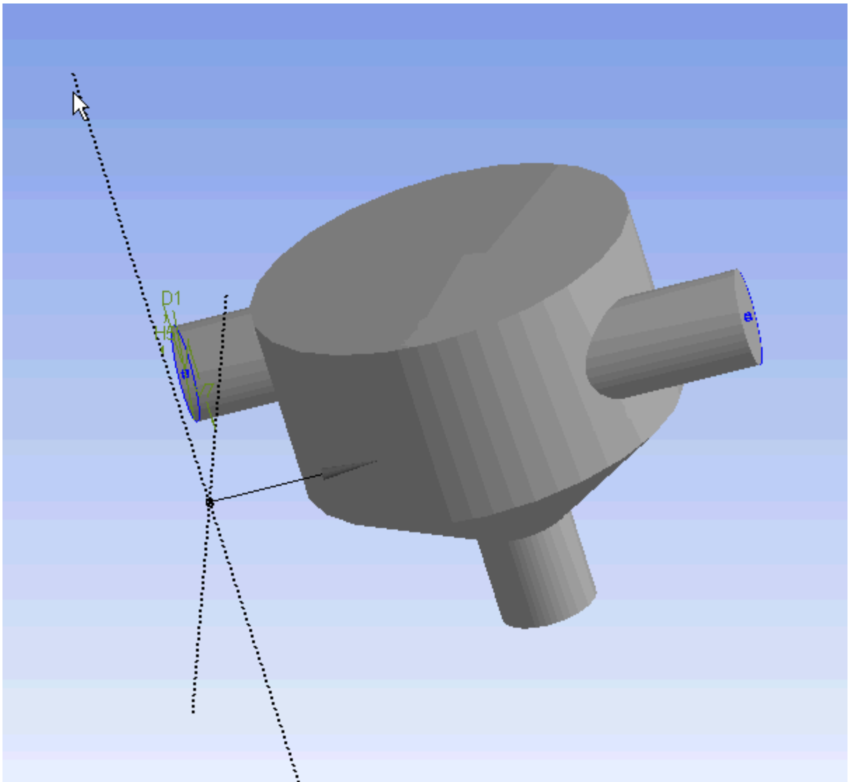
This will create the offset plane in the opposite direction. Now open the sketching window and select the circle to create the approximate circle. Create the approximate circle as done previously. After generating the circle apply the dimensions as the same dimensions shown in Figure 5.16. After applying the dimensions, press extrude from the main toolbar and enter the details in detail view as shown in Figure 5.20.

Figure 5.20
Extrude details

Details of Extrude2	
Extrude	Extrude2
Base Object	Sketch4
Operation	Add Material
Direction Vector	None (Normal)
Direction	Normal
Type	To Faces
Target Faces	1
As Thin/Surface?	No
Merge Topology?	Yes

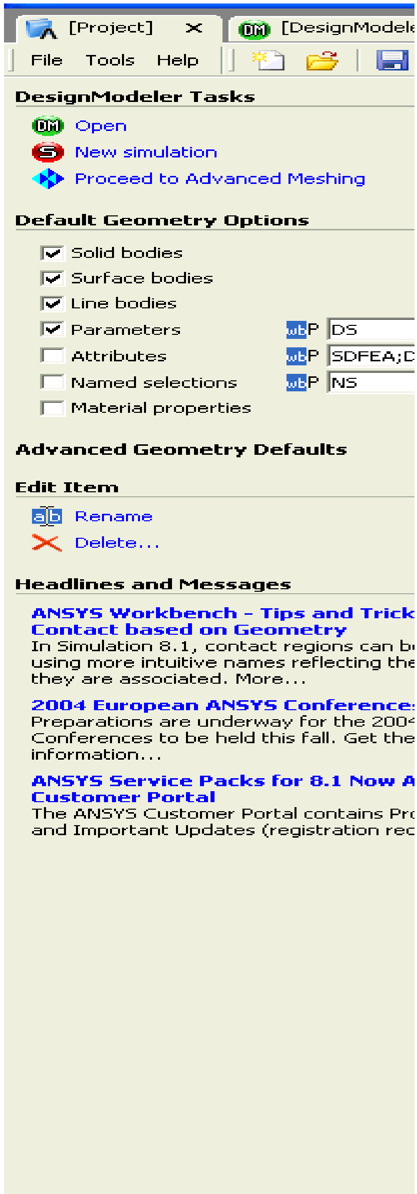
After extrusion the geometry will look like the figure shown below.

Figure 5.21 Final geometry after complete extrusion



Now we are done with the geometry creation. We now proceed to the Advanced Meshing tab for meshing. Click on **Project** in the main menu. Select P**roceed to Advanced Meshing**. This will open the Advanced Meshing interface where the user can repair the geometry, mesh it, and write the output file for CFX.

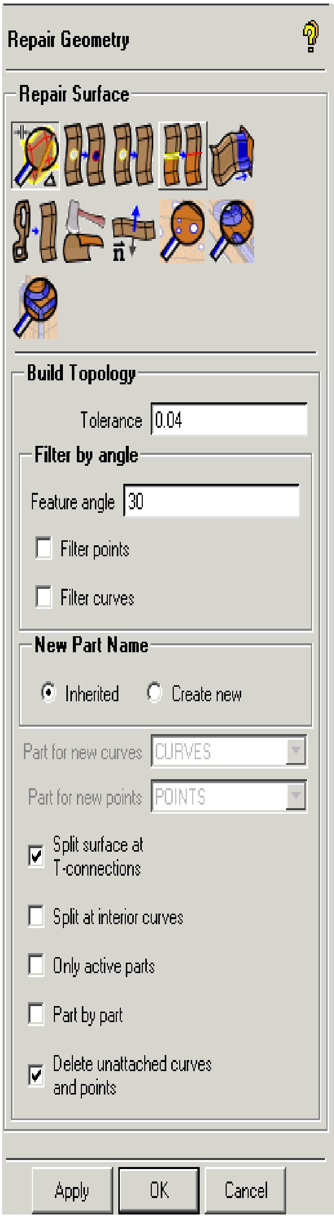
Figure 5.22
Project options window



Now the Graphics User Interface for Advanced Meshing opens. The user has to run Build Topology to get the necessary curves and points.

Select Geometry > Repair Geometry  > Build Diagnostic Topology.  Turn on Filter points and Filter curves. Select Create new for New Part Name. Press Apply.

Figure 5.23
Repair geometry window



e) Creating Body

Select **Geometry** > Create Body.

Give STATICMIXER as the Part, click on Material Point and toggle on the **Centroid of 2 points** option as shown in Figure 5.24.

Turn off all Surfaces and Points and display only curves from the Display Tree.

Click on Select location(s) and select two opposite locations on the screen as suggested in Figure 5.25 and press the middle mouse button.

Press **Apply**.

Figure 5.24: Create Body Window

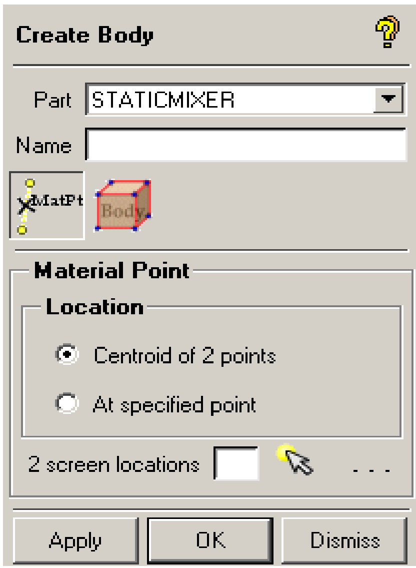
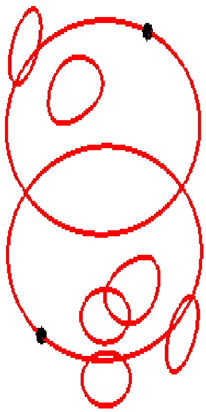


Figure 5.25: Two Opposite points for Material point



f) Mesh Generation

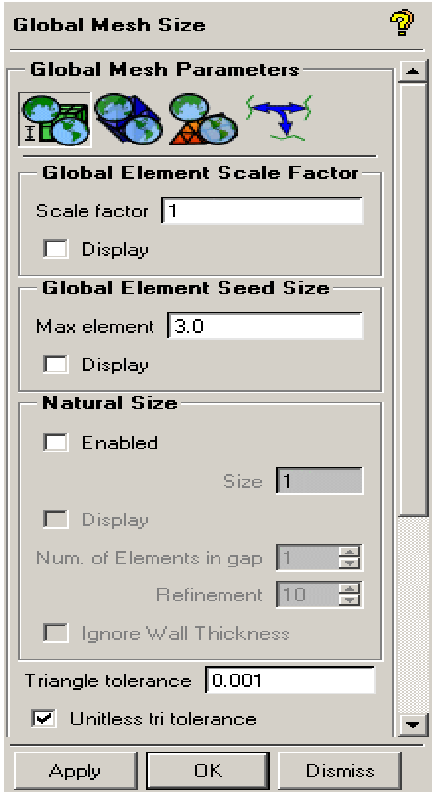
Assigning Mesh Parameters

Select **Mesh** > Set Global Mesh Size 

It defaults into General Parameters. In that window, change **Max Element** to 3.0 and leave the other fields as the default as shown in Figure 5.26 and press **Apply**.

The **Scale Factor** is used to scale the mesh size up and down by changing this number. Please note that all the sizes in **ANSYS ICEMCFD - CFX** get multiplied by the Scale Factor. Thus, it's important to keep a note of the Scale Factor all the time.

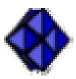
Figure 5.26
Global Mesh Size window



Saving the Project

Select Save Project  from the main menubar.

g) Meshing

Select Mesh > Volume Meshing  from the menubar.

Make sure Mesh type is set to Tetra, and use From geometry as the method. Leave the other fields as default as shown in Figure 5.27 and press **Apply** to start the tetra run.

The tetra mesh generated is shown in Figure 5.28.

Select File > Save Project As… Specify StaticMixer as the File name and press Save.

Figure 5.27: Mesh Tetrahedral Window

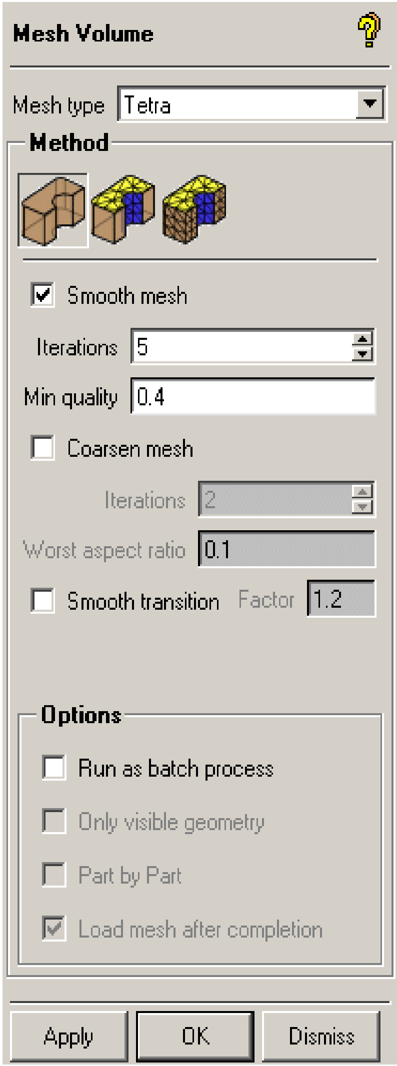
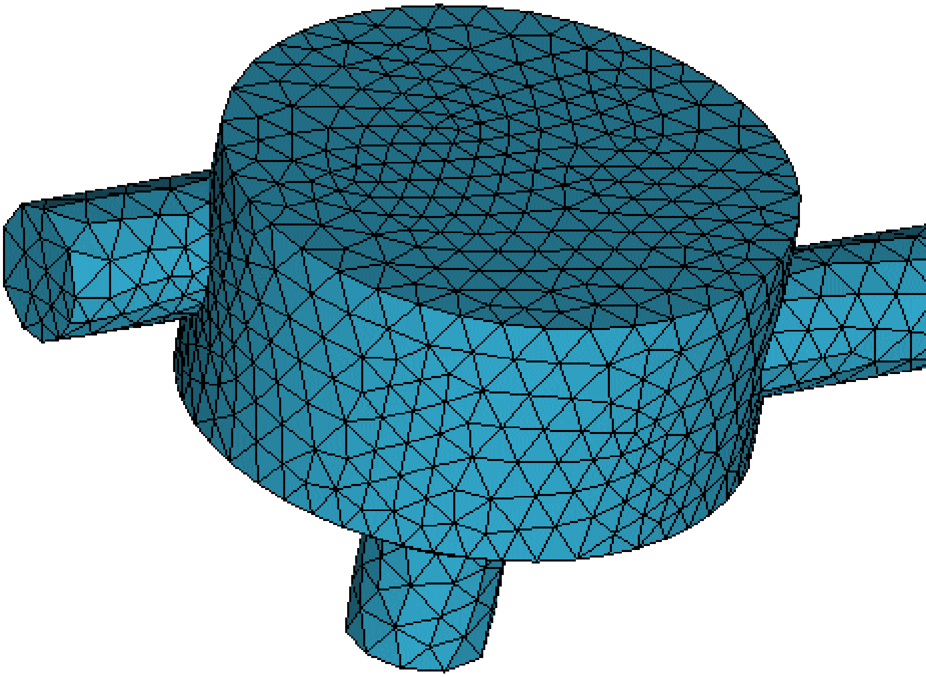


Figure 5.28 The Generated Tetra Mesh

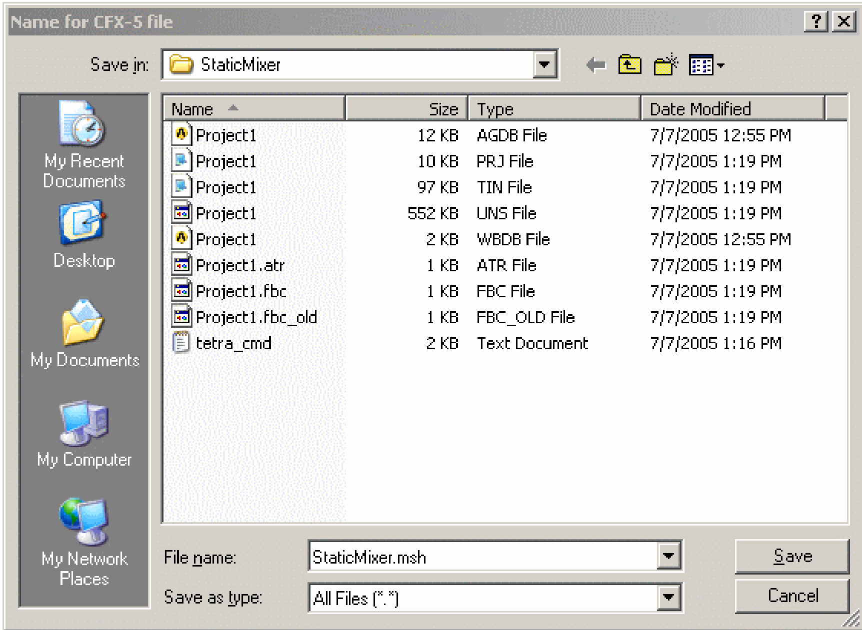


h) Writing Output

Select Output > Output to CFX from the main menu.

Enter StaticMixer.msh as the File name as shown in Figure 5.29 and press **Save**.

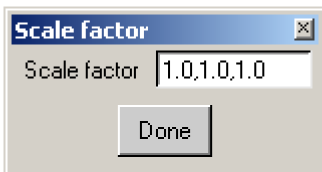
Figure 5.29: Name for CFX-5 file Window



Accept ASCII as the Output type and select **Done**.

While writing the CFX file, user can scale the output through **Scale factor** window shown in Figure 5.30. Press **Done** as no scaling is required for this tutorial.

Figure 5.30: Scale Factor for CFX-5 output



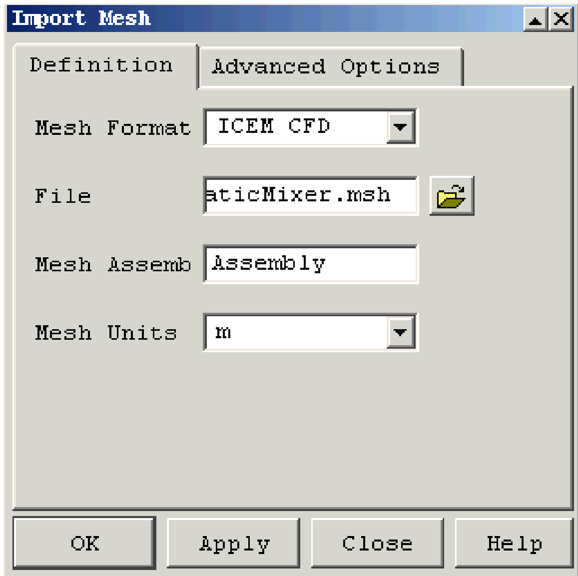
i) Exiting ANSYS ICEMCFD - CFX

Select **File** > **Exit** from the main menu to quit out of **ANSYS ICEMCFD - CFX**.

j) Continuing with the Static Mixer Tutorial

From this point, the user can continue the CFX-5 Static Mixer tutorial from the section entitled Defining the Simulation in CFX-Pre. The only required change to those instructions would be in the subsection entitled Importing the Mesh. When importing the mesh, set **Mesh Format** to ICEM CFD, and **File** select the mesh file, StaticMixer.msh, output from **ANSYS ICEMCFD - CFX** as shown in Figure 5.31.

Figure 5.31 : CFX Mesh Import window



The only other minor change to the remaining tutorial is in the section entitled Define Physics. In the **Define Physics** panel, **Select Mesh** should be set to staticmixer (the part name assigned to the volume elements). Since this is the only volume region, this name should be selected automatically.

5.2: Static Mixer 2 (Refined Mesh)

5.2.1: Overview

This tutorial covers the creation of a refined mesh for the static mixer using **ANSYS ICEMCFD - CFX**. It is assumed that the user has already completed tutorial number 1. This tutorial is intended to be compatible with CFX-5 Tutorial 2, Flow in a Static Mixer (Refined Mesh). This tutorial would effectively replace the section entitled Modifying the Model in CFX-Build. After completing this tutorial, the user could complete the remaining sections of the CFX-5 tutorial, picking up with Defining the Simulation in CFX-Pre.

a) Steps Involved in this Example

Modifying the meshing parameters,

Creating the refined tetra mesh,

Checking for quality,

Creating prism layers inflated from the walls,

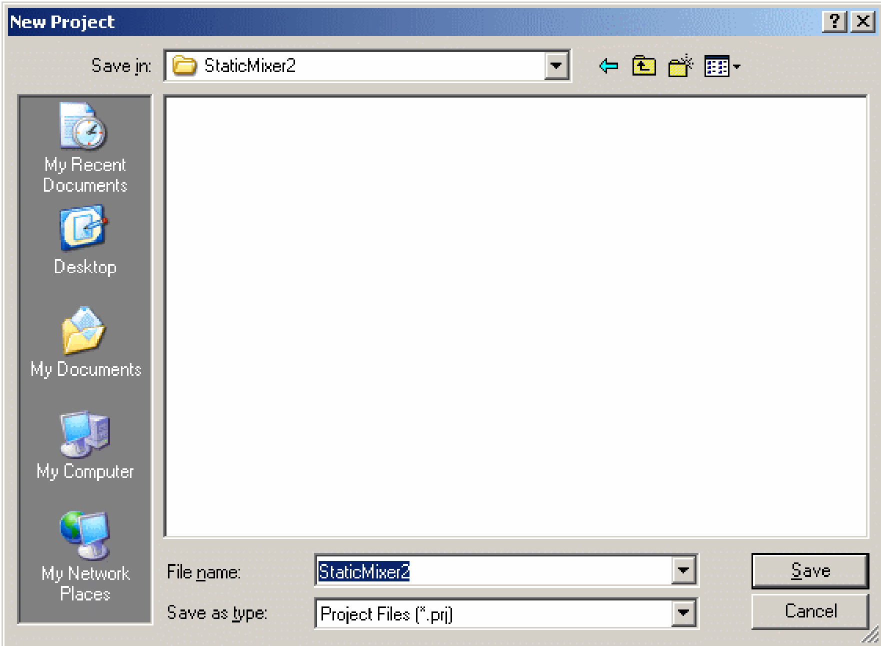
b) Starting a New Project

Creating a New Project

Launch ANSYS ICEMCFD - CFX.

Select **File** > **New Project** from the Main menu and click on  (Create New Directory) and enter StaticMixer2 as the Directory name – and also as the File name as in Figure 5.32. Press Save.

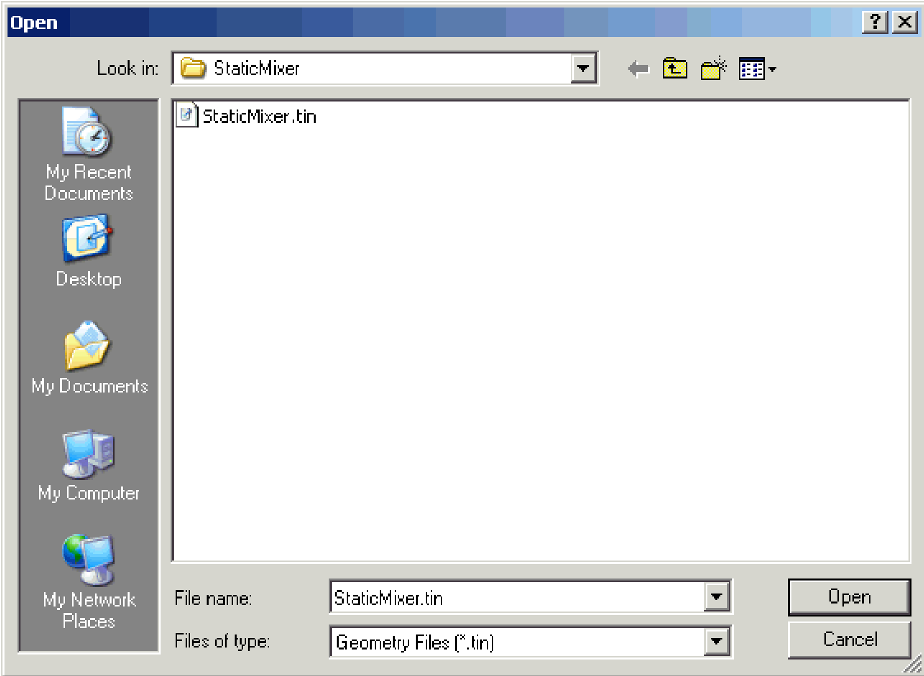
Figure 5.32 New project window



Loading a Geometry File

From the Main Menu, click on  (Open Geometry) and select the geometry file `StaticMixer.tin` created in the previous tutorial by browsing as shown in.Figure 5.33.

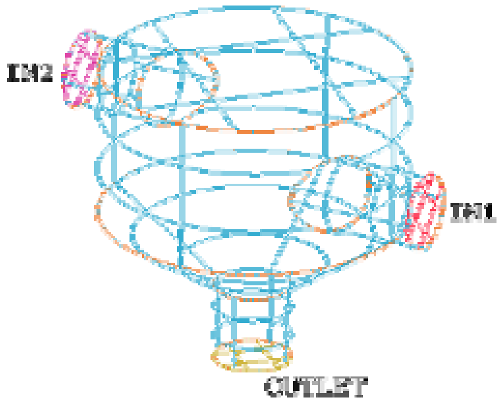
Figure 5.33: Loading the previous Tutorial Geometry File



Creating Parts for Surfaces (see Figure 5.34)

From the Main Menu, click on  (Open Geometry) and select the geometry file `StaticMixer.tin` created in the previous tutorial by browsing as shown in Figure 5.33.

Figure 5.34: Geometry Parts



Right-click on Parts from the Display Tree widget and choose Create Part. It goes by default into the choice of Create Part by Selection.

Enter IN1 as the Part name.

Click on Select entities, select the surface at the end of one of the small side pipes, and middle-click to accept. (Note that selection mode remains active.)

Enter IN2 as the Part name.

Select the surface on the end of the other small side pipe and middle-click to accept.

Enter OUTLET as the Part name.

Select the small surface at the end of the extension of the funnel and middle-click to accept.

Middle-click again to cancel out.

c) Mesh Generation

Reassigning Mesh Parameters

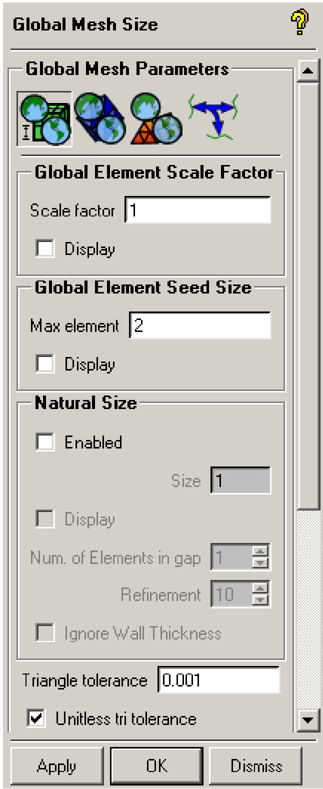
From the **Mesh** tab menubar click on Set Global Mesh Size.

Leave Scale Factor as 1 for Global Element Scale Factor.

Change **Max Element** to 2 from 3 for **Global Element Seed Size** as shown in Figure 5.35.

Click **Apply** to save this setting.

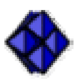
Figure 5.35:
Global Mesh Size window



Saving the Project

Save the project by clicking on Save Project  from the Main Menu. This saves the geometry file as **StaticMixer2.tin** in the **StaticMixer2** directory.

Meshing

Select Volume Meshing  from the **Mesh** tab menu bar to create the refined tetrahedral mesh on this geometry.

Make sure Mesh type is Tetra and the Method is From geometry. 

Keep the defaults for the meshing. Notice that by default there will be 5 iterations of smoothing after the tetra meshing to improve the elements of low quality.

Click **Apply** to create the tetrahedral mesh.

Once the mesh is created, it gets loaded on the screen.

Verifying Mesh Quality

Click on Smooth Mesh Globally  from the **Edit Mesh** tab menubar to check the quality of the mesh.

Set Up to quality to 0.5 and Criterion to Quality.

Right-click In the histogram and select **Replot** which pops up the Replot window.

Change **Min X** value to 0.

Change **Max X** value to 1.

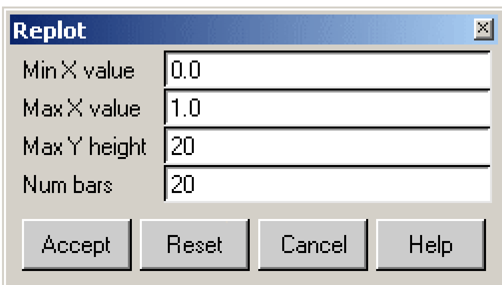
Change **Max Y** height to 20 as shown in Figure 5.36.

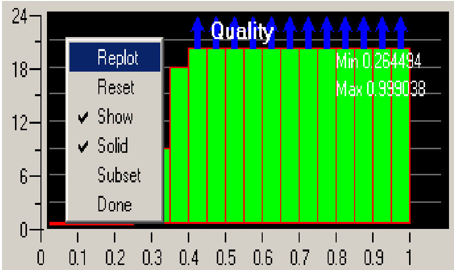
Click **Accept** to replot the Histogram as shown in Figure 5.36.

Press Apply in the Smooth Elements Globally panel to smooth.

From the messages and the smoothing histogram, it can be seen that the mesh has all elements above a target minimum quality of 0.3.

Figure 5.36: Replot window and Quality Histogram





Saving the Project

Save the project by clicking on Save Project from the Main Menu.

d) Inflated Boundary Generation

Prism Mesh Generation

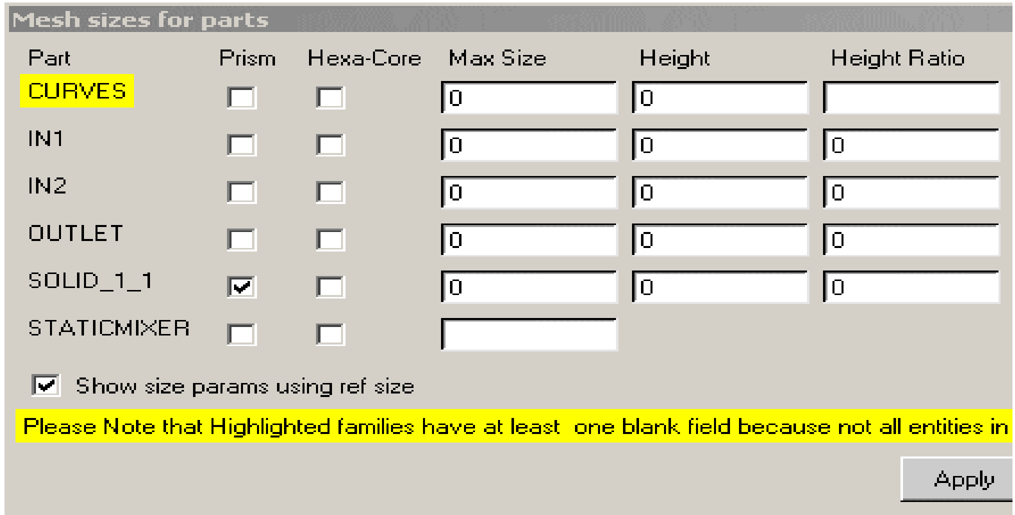
Prism meshing gives layers of flat prismatic (wedge-shaped) elements which provide a smaller mesh length scale in the direction perpendicular to the wall. This provides better resolution of the velocity field in the boundary layer near the wall, where it changes rapidly.

Prism meshing can greatly improve accuracy, particularly in a model with a high aspect ratio, such as a long narrow pipe, or in a model where turbulence is significant. Prism meshing should be used when lift, drag, or pressure drop in the model is of interest.

Click on Mesh Prism from the **Mesh** tab menubar to create inflated prism layers from the walls.

Click on **Select Parts for Prism Layer**. Toggle on **SOLID_1_1** from the Part list as shown in Figure 5.37. Click on **Apply** and **Dismiss** to accept and close the panel.

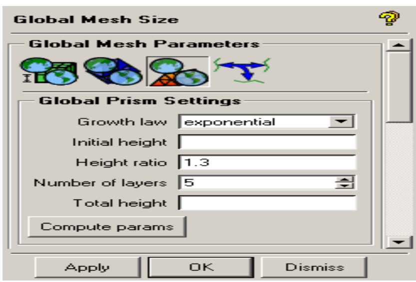
Figure 5.37: Parts for Prism layer window



Click on Set Global Mesh Size  from the Mesh tab. Click on Prism Meshing Parameters .

Change the **Height ratio** to 1.3, and the **Number of layers** to 5. Click on Apply.

Figure 5.38: Mesh Prism window



Other parameters should be left as it is as shown in Figure 5.38.
Leaving the **Initial Height** blank attempts to make the volume of the last prism element approximately equal to the volume of the attached tetrahedral element. This

is based on the size of the base triangle for each prism column, so the total prism thickness will vary through the mesh.

Click on Mesh Prism  from the **Mesh** tab menubar to create inflated prism layers from the walls.

Set **Number of volume smoothing steps** to 5.

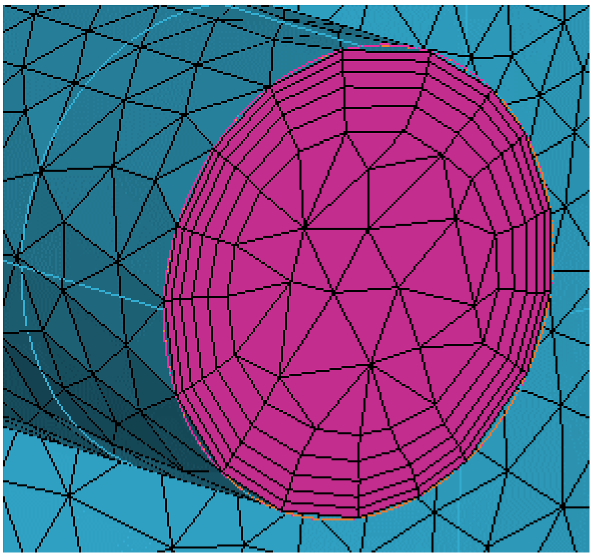
Click **Apply** to begin the prism meshing process.

After the prism mesh generation, a panel appears asking if the new mesh file should be loaded. Click on "**Yes**" and then on "**Replace**" (if prompted) to replace the existing tetra mesh with this new prism mesh.

After prism mesh generation, the mesh at one of the inlets will be similar to that shown in Figure 5.39.

Note: To view the mesh as solid/wire mode, right-click on **Mesh** > **Shells** in the Display Tree and select **Solid & Wire** option.

Figure 5.39: Prism Layers at inlet

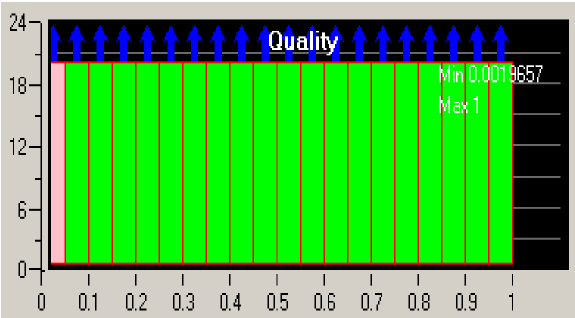


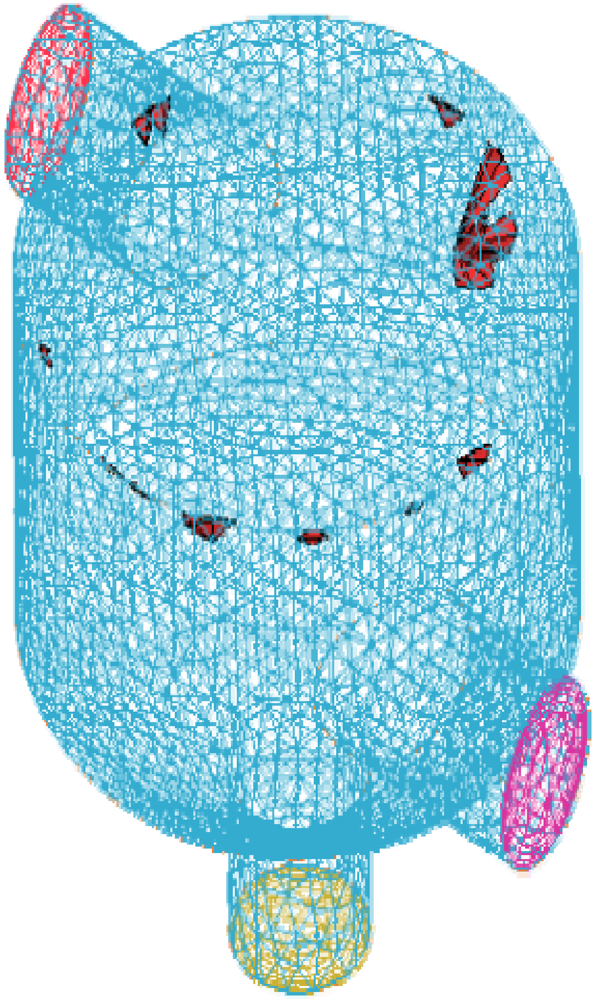
Editing the Prism Mesh

Go to the **Edit Mesh** tab menubar and click on Smooth Mesh Globally  to check the mesh quality.

In the quality histogram, select the first bar with the left mouse button to display the bad elements on the screen as shown in Figure 5.40.

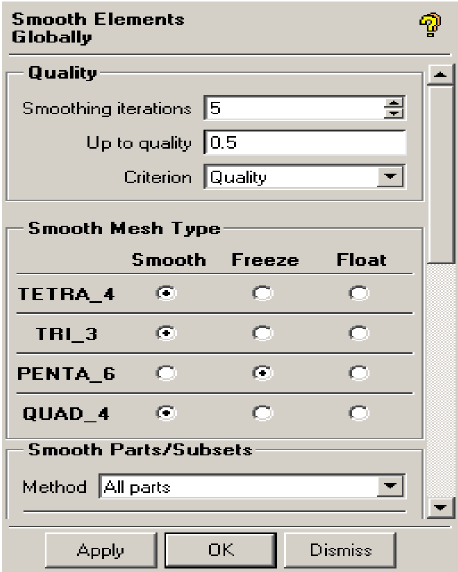
Figure 5.40: Low quality Elements displayed in first bar of histogram





These are the low quality tetrahedral elements getting stuck due to prism meshing. Smoothing both prism and tetrahedral elements will improve the quality of these elements.

Figure 5.41: Smooth Elements Window



Change the value of **Up to Quality** to 0.50.

Set the **Criterion** to **Quality**

First set PENTA_6 to **Freeze** as shown in Figure 5.41.

Click on **Apply** to smooth the mesh.

Note that the histogram doesn't change much, as the quality problems are likely due to the prism/tetra interface.

Now set PENTA_6 back to **Smooth**

Set **Up to quality** to 0.2 – so as not to warp the prisms too much.

Click on **Apply** to smooth the mesh.

The low quality elements will be smoothed out as shown in the Figure 5.42.

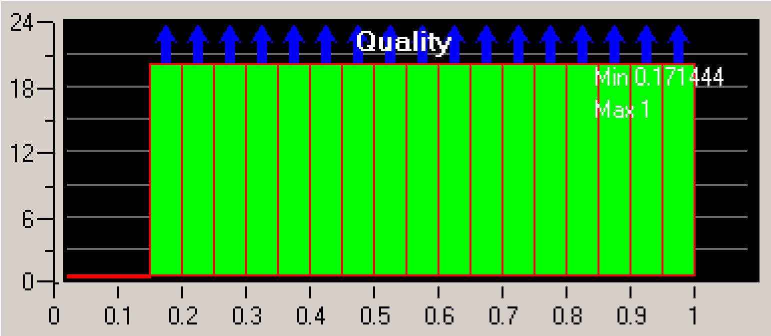


Figure 5.42: Histogram after Smoothing

Saving the Project

Save the project by clicking on Save Project from the Main Menu.

e) Writing Output

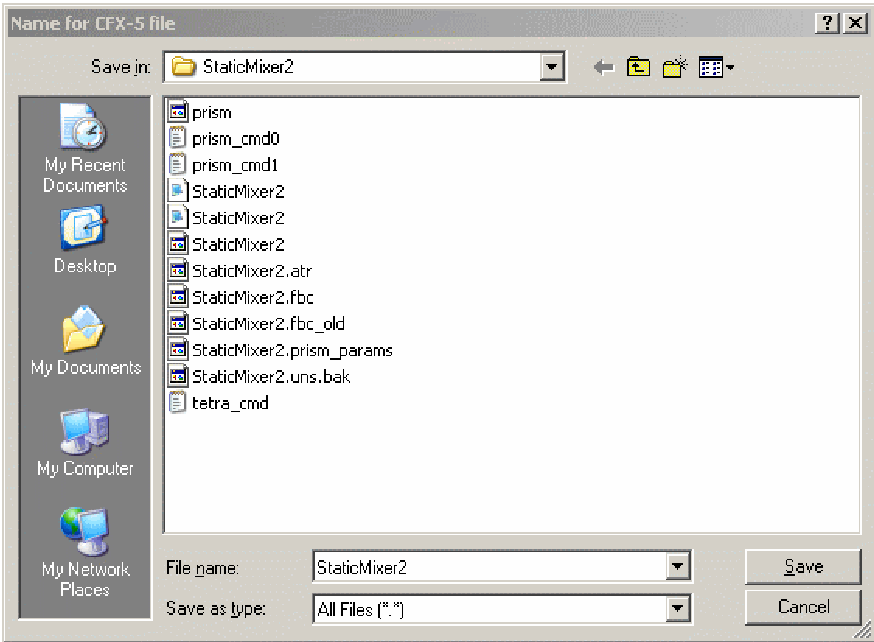
From the **Output** tab menubar; click on Output to CFX.

Accept the default name for File name as shown in Figure 5.43 and press **Save**.

Accept ASCII as the Output type and select **Done.**

For this tutorial, there is no need to scale the mesh. Press **Done** to convert the mesh into CFX format – StaticMixer2.msh.

Figure 5.43: Output to CFX window



f) Exiting ANSYS ICEMCFD - CFX

Select **File** > **Exit** from the main menu to quit out of **ANSYS ICEMCFD - CFX**.

g) Continuing with the Static Mixer (Refined Mesh) Tutorail

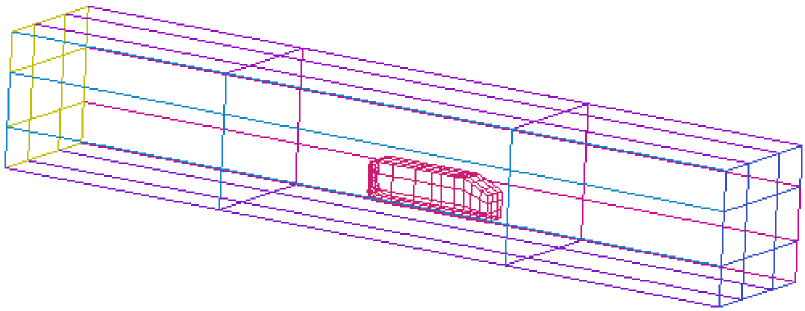
As described in previous tutorial, the user can continue CFX-5 Static Mixer (Refined Mesh) Tutorial from the section Defining the Simulation in CFX-Pre.

5.3: Blunt Body

5.3.1: Overview

This tutorial covers parasolid geometry import, geometry clean up and meshing for an automotive-style blunt body using **ANSYS ICEMCFD - CFX**. It is intended to be compatible with CFX-5 Tutorial 5, Flow around a Blunt Body. This tutorial would effectively replace the section entitled Creating the Model in CFX-Build. After completing this tutorial, the user could complete the remaining as sections of the CFX-5 Blunt Body tutorial, picking up with Defining the Simulation in CFX-Pre.

Figure 5.44: Geometry Model



a) Steps Involved in this Example

Importing the Geometry,

Geometry modification,

Modifying the meshing parameters,

Creating the refined tetra mesh,

Checking the mesh quality,

Creating inflated prism layers from the walls,

Writing output file to the CFX-5

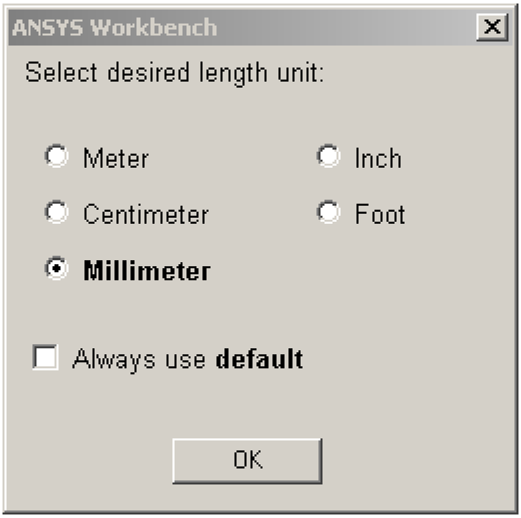
b) Starting a New Project

Creating a New Project

Launch ANSYS ICEMCFD - CFX.

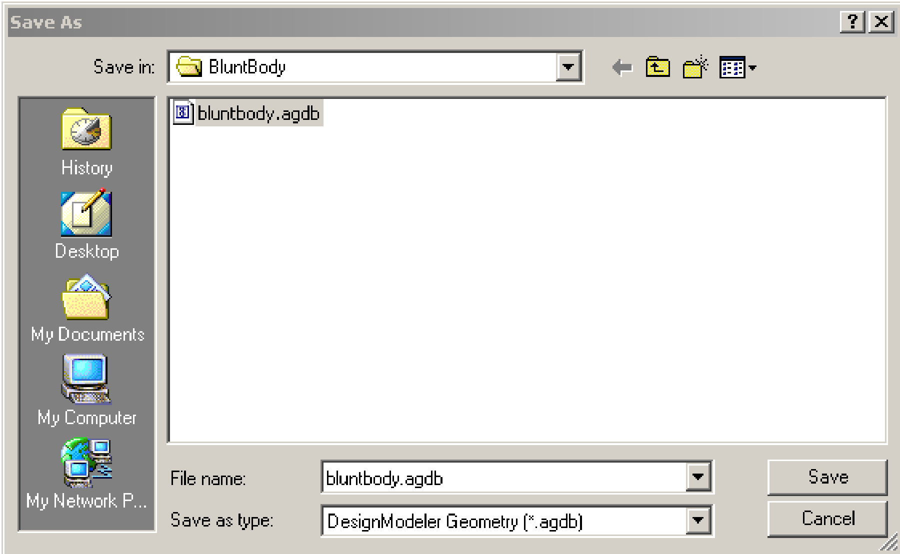
Select **Empty project** from the Start window and click on New geometry in left side menu. This will open the Ansys Workbench interface. As soon as Workbench interface open up a new window Ansys Workbench pops up to select the desired length unit. Default desired length unit is millimeter. User can select any type of unit and press ok. For this tutorial, keep default length unit i.e. millimeter and press Ok.

Figure 5.45
Ansys workbench desired unit length window



Enter BluntBody as the Directory name as shown in Figure 5.46 and press **Save.**

Figure 5.46: Save New Project window



c) Importing a Geometry File

From the Main Menu, select **File** > **Import External Geometry file** . Select the `BluntBody.x_t` file supplied by browsing as shown in Figure 5.47.

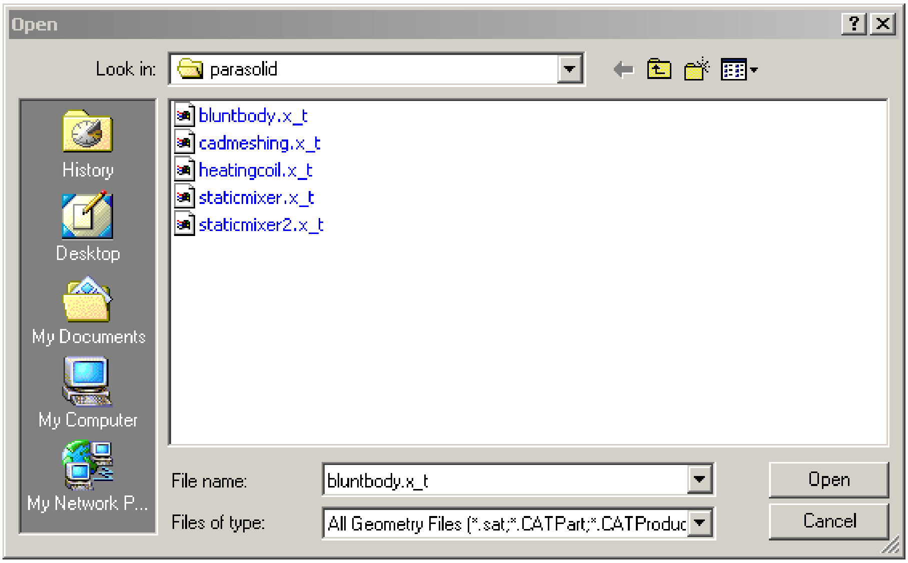
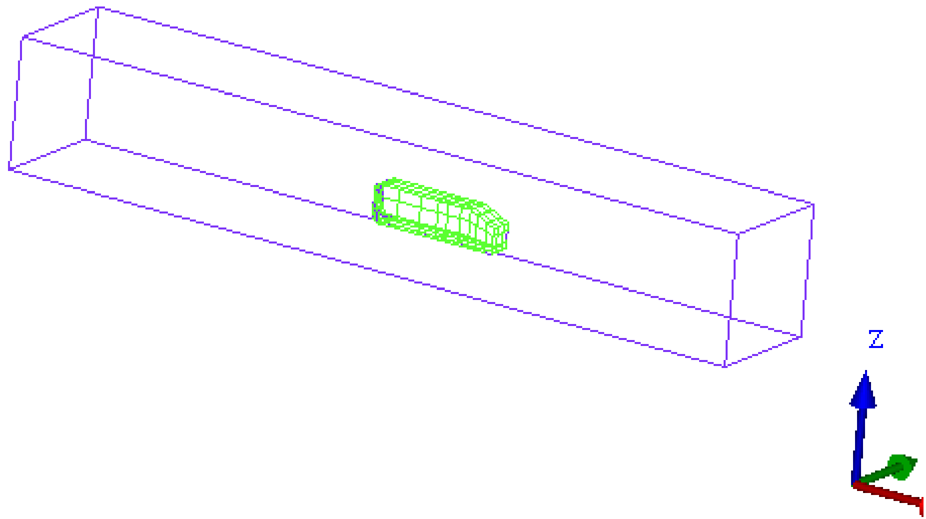


Figure 5.47: Select parasolid File window

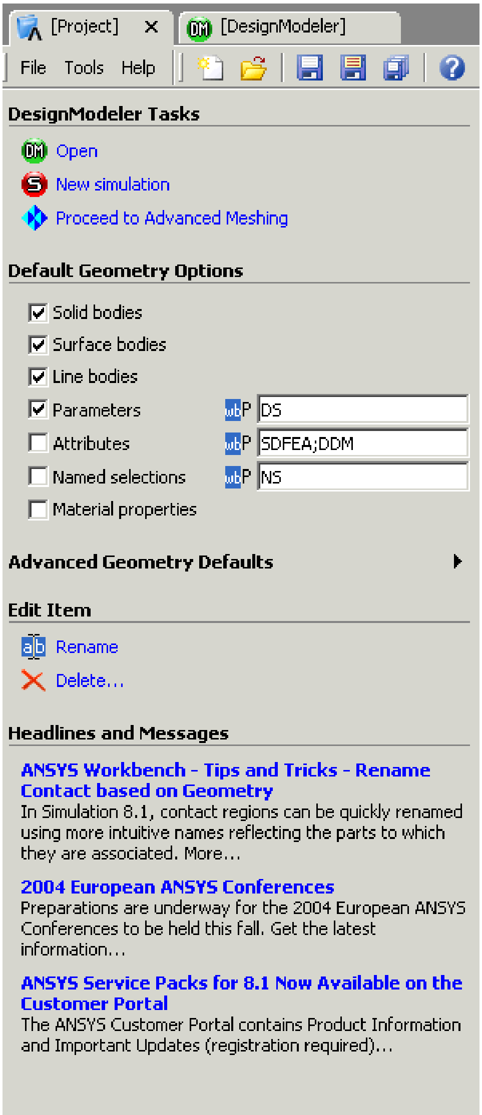
After importing press **Generate** button from the top menu so that geometry can be visualized on the screen. This loads the geometry file in the DesignModeler space as shown in Figure 5.48

Figure 5.48: Imported Geometry



Now we have to proceed for geometry repairing and meshing in advance meshing. To go into the Advance meshing, Click on **Project** in the upper left corner. After opening project window, click on **Proceed to Advance meshing**. This will open up the advance-meshing interface.
The project window is shown in the Figure 5.49

Figure 5.49 Design_Modeler_tasks_window



d) Geometry Cleanup

Because of the difficulties in maintaining a common standard for graphical entities across all CAD systems, imported parasolid models usually require some cleanup before they can be used to create a continuous enclosed region for CFD analysis. The imported geometry consists of a body made of surfaces, surrounded by a bounding rectangular box.

e) Parts Creation

Right-click on **Parts** from the Display Tree widget and select **Create Part**. It defaults into Create Part by Selection.

Enter INLET in the **Part** field.

Click on Select entities, select the surface at the min-X end, and middle -click to accept. (Note that selection mode is still active.)

Enter OUTLET in the **Part** field.

Click on Select entities, select the surface at the max-X end, and middle-click to accept.

Enter SYMP in the **Part** field.

Click on Select entities, select the max-Y surface at the base of the body, and middle-click to accept.

Enter BODY in the **Part** field.

Click on Select entities, select the surfaces of the body (either one by one or with the drag box), and middle-click to accept and again to cancel out.

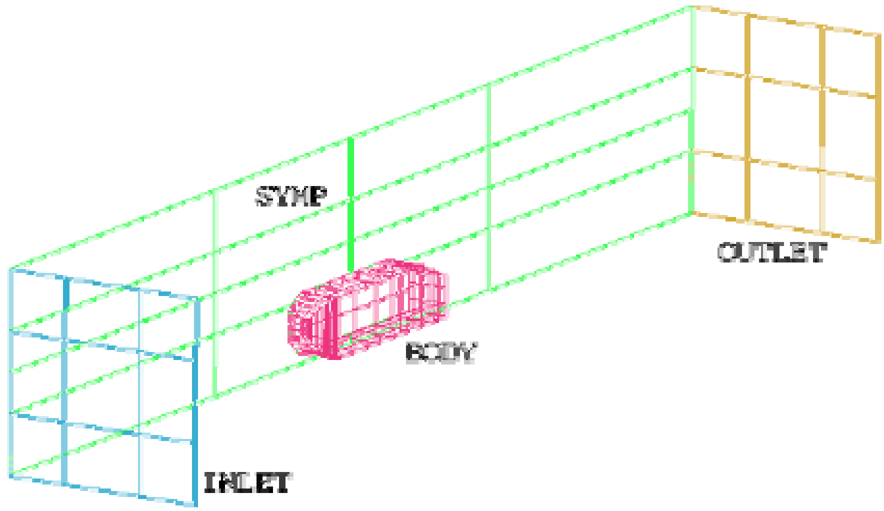
Right-click on Parts in the Display Tree widget, select Reassign Colors and "Good" Colors.

The surfaces should be grouped as in Figure 5.50. Note the three surfaces of the outer box are not displayed.

Click on (Choose an Item) and select the min-X top and bottom curves shown in and press the middle mouse button.

Click on **Apply** to create the surface.

Figure 5.50 Surface Parts



Build Diagnostic Topology.

Go to the Geometry tab menu bar and select Delete Curve. 

Press the hotkey "**a**" on the keyboard to select and delete all the curves in the model.

Similarly, select Delete Point,  and press the hotkey "**a**" on the keyboard to select and delete all the points.

Creating Curves and Points

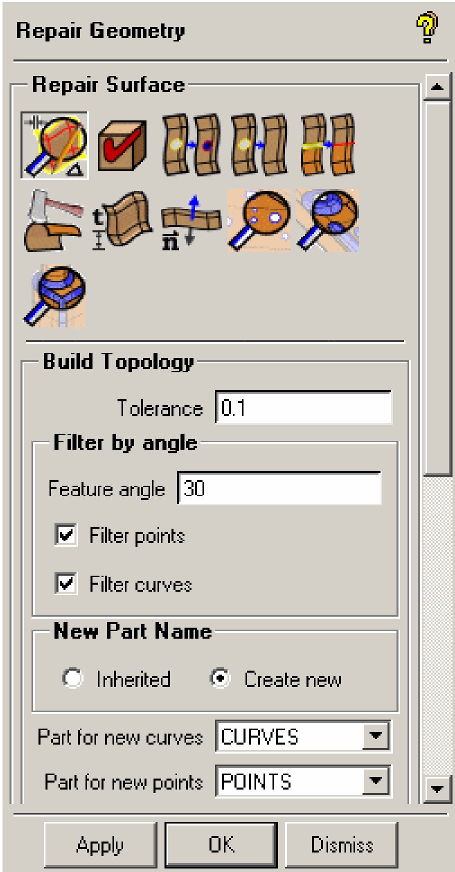
To create the necessary curves and points,

Click on Repair Geometry  from the **Geometry** tab menubar.

Make sure that by default Build Diagnostic Topology  is selected.

Set **Tolerance** to 0.1. Enable Filter points and Filter curves. Set New Part Name to Create new and use others as default as shown in Figure 5.51.

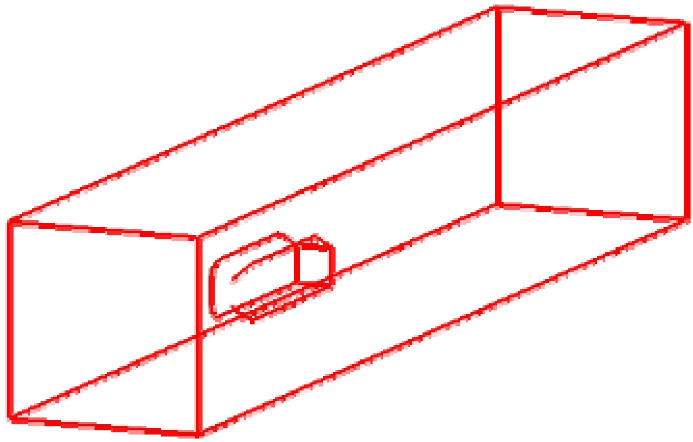
Figure 5.51
Build Topology



Click Apply.

Turn OFF the display of Surfaces and Points in the tree to make sure that all the curves is in red as shown in Figure 5.52.

Figure 5.52 All curves in Red color



Body Creation

Go to the **Geometry** tab menubar and select Create Body. 

Switch on Surface in the Display Tree.

Give the Part name as BLUNTBODY

Click on Material Point and toggle on the **Centroid of 2 points** option as shown in Figure 5.53.

Click on Select locations,  select two opposite corners on the screen as suggested in Figure 5.54 and middle-click; and again to cancel out.

Save the Project.

Figure 5.53 Create Body Window

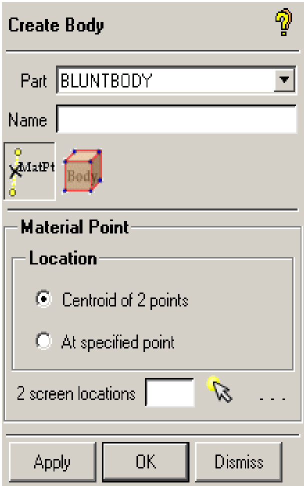
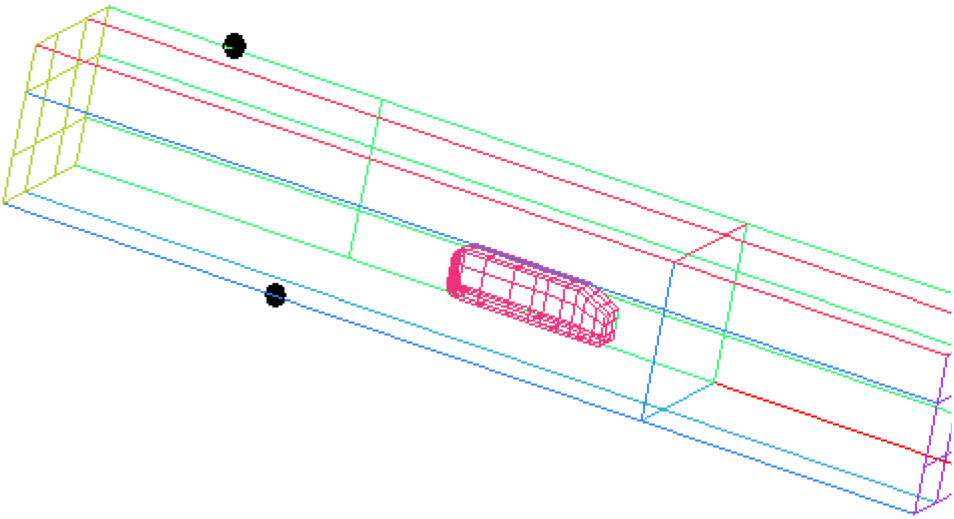


Figure 5.54 Two Opposite points for Material point



f) Mesh Generation

Global Mesh Parameters

From the **Mesh** tab menubar click on Set Global Mesh Size  .

In General Parameters  leave Scale factor as 1 for Global Element Scale Factor.

Change Max element to 20 for Global Element Seed Size.

Enable **Natural Size** and set **Size** to 2 and Refinement to 16 as shown in Figure 5.55. Natural size is the minimum element size that will be generated using automatic refinement methods to capture curvature and proximity in the meshing process.

Click **Apply** to save these settings.

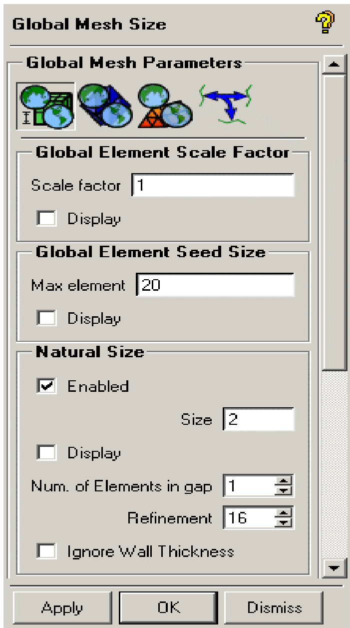
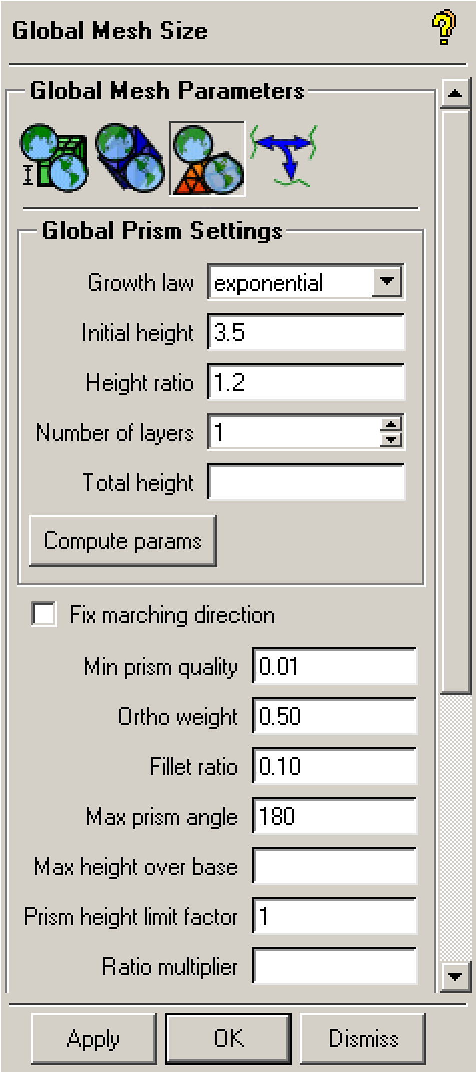


Figure 5.55
Global Mesh Size window

Click on Prism Meshing Parameters. 

Set **Initial height** to 3.5, **Number of layers** to 1, and **Prism height limit factor** to 1 as shown in Figure 5.56. This ensures that the height of a prism is not larger than its base triangle size. Click **Apply**.

Figure 5.56
Global Prism Settings



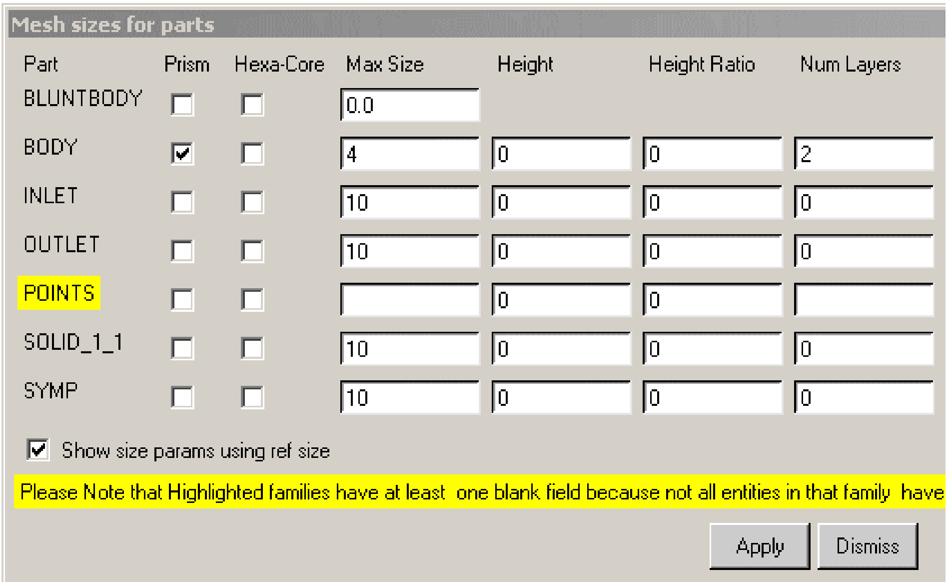
Mesh Parameters by Parts

Click on Set meshing Params by Parts from the **Mesh** tab menubar.

For the BODY part, enable **Prism**. Also set **Max Size** to 4 and **Num Layers** to 2 as shown in Figure 5.57. Num Layers of 2 ensures 2 layers of similar size tetrahedral elements around the BODY. For the INLET, OUTLET, SOLID_1_1, and SYMP parts enter **Max Size** of 10.

Click **Apply** and **Dismiss** to save this setting.

Figure 5.57 Mesh Params by Parts

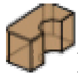


Saving the Project

Save the project by clicking on (Save Project) from the Main Menu. This saves the geometry file as **BluntBody.tin** in the **BluntBody** directory.

g) Meshing

Select Volume Meshing from the **Mesh** tab menubar to create the mesh on this geometry.

Set **Mesh type** to **Tetra** and **Method** to **From Geometry**.  Keep the defaults for Tetra Meshing. Note that by default there will be 5 iterations of smoothing after the tetra meshing to improve the elements of low quality.

Click **Apply** to create the tetrahedral mesh.

Once the mesh is created, it gets loaded on the screen.

h) Editing the Mesh

Smoothing

Click on Smooth Mesh Globally  from the **Edit Mesh** tab menubar to check the quality of the mesh.

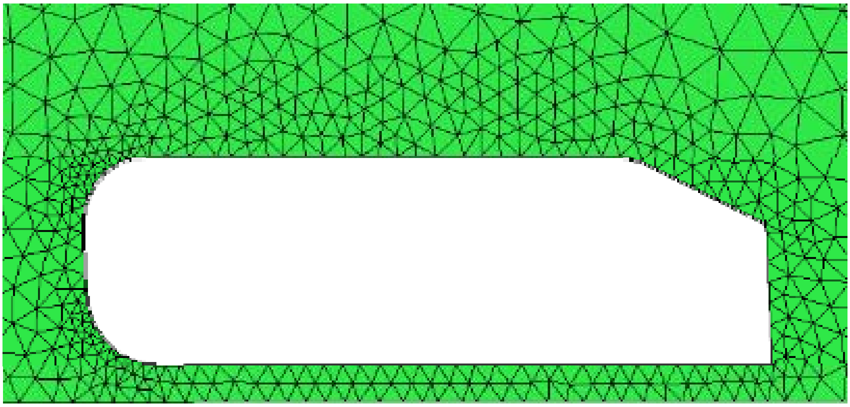
From the messages and the smoothing histogram, it can be seen that the mesh has quality more than 0.2. Still, additional smoothing can be performed if desired.

Set **Criterion** to **Quality** and set **Up to Quality** as 0.5 as shown in Figure 5.58 and press **Apply**.

Figure 5.58 Smooth Elements Window

Switch OFF all the parts in the Tree except SYMP to see the smooth mesh as shown in Figure 5.59.

Figure 5.59 Mesh after Smoothing



Saving the Project

Save the project by clicking on (Save Project) from the Main Menu.

i) Inflated Boundary Generation

Prism Mesh Generation

Click on Set meshing Params by Parts from the **Mesh** tab menubar.

For the BODY part, set **Num Layers** to 1, as we only want 1 prism layer. **Apply** and **Dismiss**.

Click on Mesh Prism from the **Mesh** tab menubar to create inflated prism layers from the walls.

This combination of parameters attempts to make the prism layer of the same height as the attached tetrahedral elements.
Press **Apply** to generate the prism layer.

After the prism mesh is generated, a panel appears asking if the new mesh should be loaded. Click on **Yes** and then on **Replace** (if prompted) to replace the existing tetra mesh with this new prism mesh. The mesh on SYMP and BODY would look like as shown in Figure 5.60.

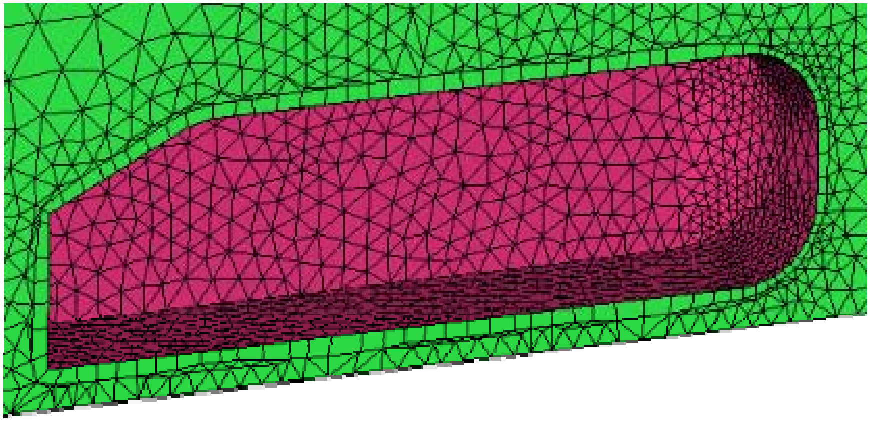


Figure 5.60 Prism layer on Symmetry surface

Editing the Prism Mesh

Now, we will split this prism mesh into several pieces. This is normally much faster to do rather than creating several prism layers. From the **Edit Mesh** tab menubar; click on Split Mesh. 

Select Split Prisms,  set **Number of Layers** as 5 and **Prism Ratio** as 1.3 as shown in Figure 5.61. Click on **Apply** to get the final mesh.

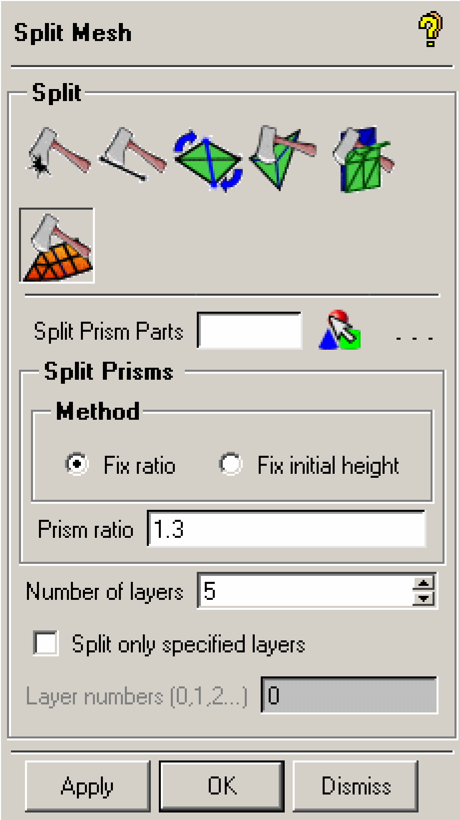
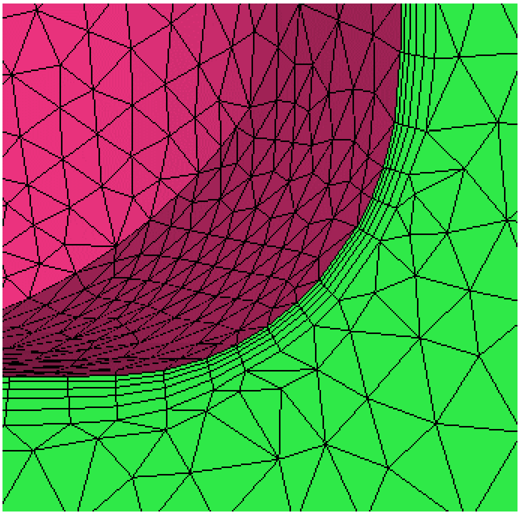


Figure 5.61
Split Mesh window

Figure 5.62 shows a portion of the mesh on SYMP and BODY after splitting the prism layers.

Figure 5.62
5 Layers after Splitting



The prism layers may not have the desired first cell height. Thus we would redistribute them to achieve a constant first layer thickness of 0.1.

Go to the **Edit Mesh** tab menubar and click on Move Nodes.

Click on Redistribute Prism Edge, set **Initial Height** as 0.1 as in Figure 5.63 and click on **Apply**.

Figure 5.63
Redistribute Prism Edge window

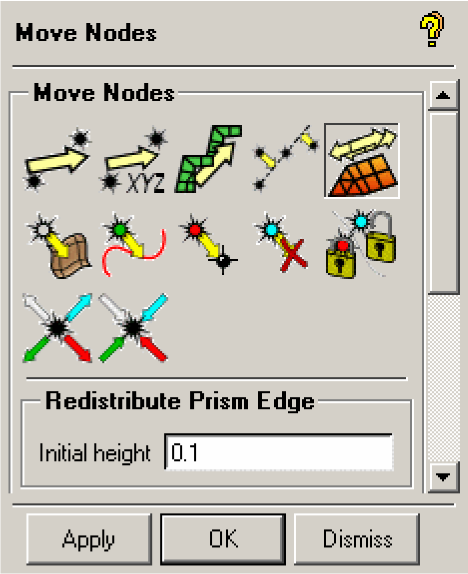
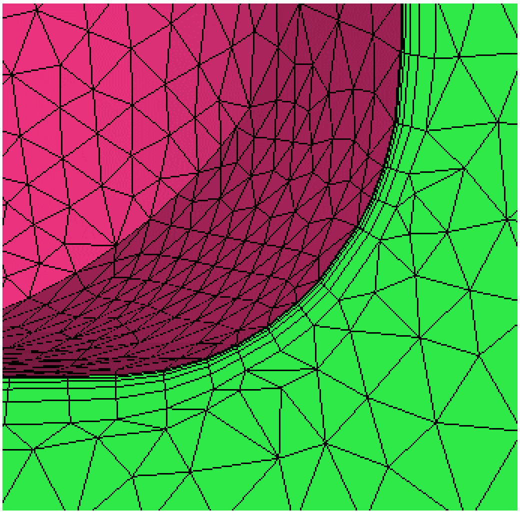


Figure 5.64
Layers after redistribution



This redistributes the prism layers such that the initial height is uniformly 0.1 as shown in Figure 5.64. However, the growth ratio now varies based on the total thickness of each prism column.

Smoothing

Click on Smooth Mesh Globally from the **Edit Mesh** tab menubar to check the quality of the mesh.

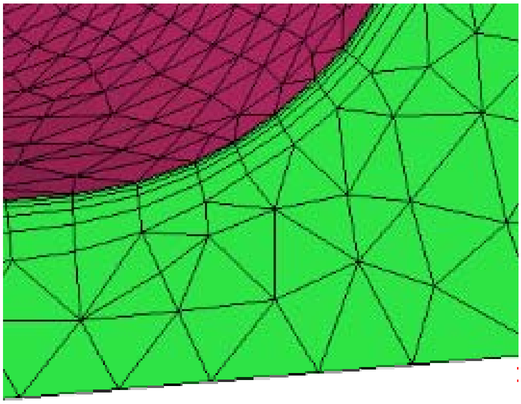
Set **Criterion** to **Quality** and **Up to quality** to 0.5.

Set **PENTA_6** to **Freeze** to prevent prisms from smoothing initially and **Apply.**

Now set **PENTA_6** to **Smooth** and **Up to quality** to 0.2 and Apply.

The final smoothed mesh should look something like in Figure 5.68 .

Figure 5.65
Layers after redistribution

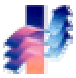


Saving the Project

Save the project by clicking on  (Save Project) from Main Menu.

If Overwrite window occurs press Yes.

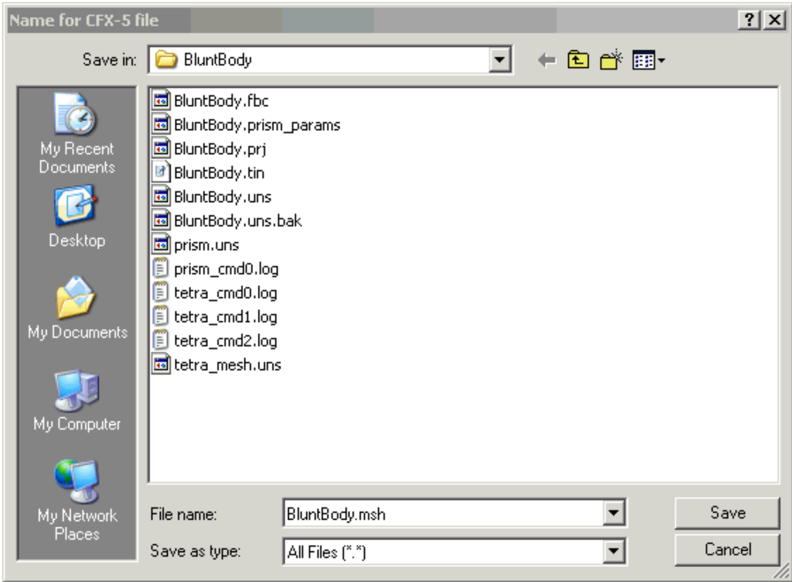
j) Output

From the **Output** tab menubar; click on Output to CFX. 

Accept the default File name for CFX file as shown in Figure 5.66 and press **Save**.

Select ASCII and when asked for the scaling factor, use the default (1.0, 1.0 ,1.0) to complete the translation.

Figure 5.66 Output to CFX window



k) Continuing with the Blunt Body Tutorial

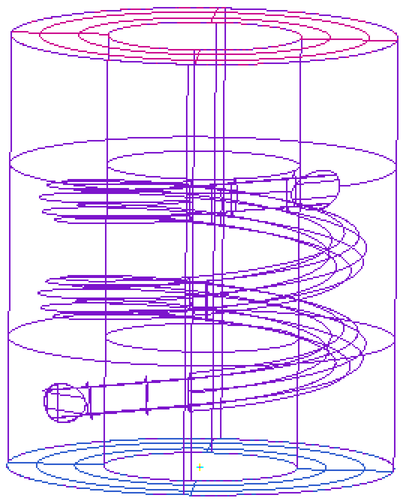
As described in Tutorial 1, the user can continue CFX-5 Flow Around a Blunt Body Tutorial from the section Defining the Simulation in CFX-Pre.

5.4: Heating Coil

5.4.1: Overview

This tutorial covers geometry import and meshing for Heating Coil geometry using **ANSYS ICEMCFD - CFX**. It is intended to be compatible with CFX-5 Tutorial 14, Conjugate Heat Transfer in a Heating Coil. This tutorial would effectively replace the section entitled Creating the Model in CFX-Build. After completing this tutorial, the user could complete the remaining as sections of the CFX-5 Heating Coil tutorial, picking up with Defining the Simulation in CFX-Pre. In this example, part of a simple heat exchanger is used to model the transfer of heat from a solid to a fluid. The model consists of a fluid domain and a solid domain. The fluid domain is an annular region through which water flows at a constant rate. The heater is a solid copper coil modeled as a constant heat source. The surfaces of the geometry are shown in Figure 5.67.

Figure 5.67
Geometry Model



a) Steps Involved in this Example

Importing the geometry in **Design modelar.**

Proceeding to the Advance meshing.

Creating the Tetra mesh,

Checking for quality,

Creating inflated prism layers from the walls.

b) Starting a New Project

Creating a New Project

Launch the Ansys Workbench and select New > Advance Meshing

Select **File** > **New Project** from the Main menu and click on (Create New Directory). Enter HeatingCoil as the Directory name and press **Done**.

Enter HeatingCoil as the project name and press **OK**.

c) Geometry

Importing a Geometry File

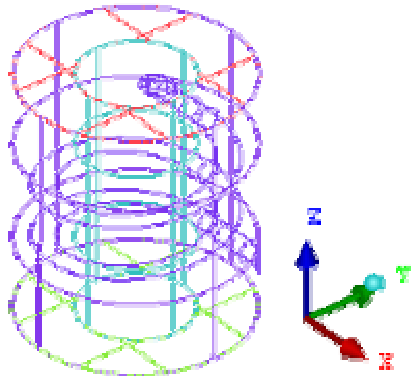
From the Main Menu, select **File > Import external Geometry file**.

Select the `HeatingCoil.x_t` file supplied by browsing.

Press **Generate** from top menu.

This loads the geometry file HeatingCoil.agdb automatically after the conversion.
The imported geometry appears as shown in Figure 5.68.

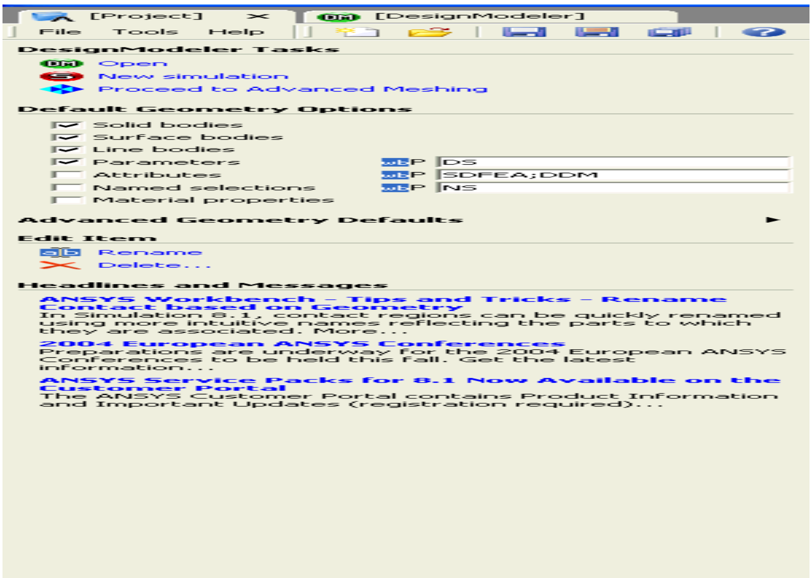
Figure 5.68 Imported Geometry



d) Geometry Manipulation

From the main project window as shown in Figure 5.69.

Figure 5.69 Proceed to advance meshing window



This will shift geometry in to the ICEMCFD_CFX environment for geometry clean up. Because of difficulties in maintaining a common standard for graphical entities across all CAD systems, imported parasolid models usually require some cleanup before they can be used to create a continuous enclosed region for CFD analysis.

INFLOW:

Right click on **Parts** in the Display tree and select **Create Part**.
Give the Part name as INFLOW as shown in Figure 5.70 and click on Create Part by Selection. 
Toggle off Curves in the Display Tree widget since only surfaces need to be put into INFLOW.

Click on Select entities  and then select the bottom (min-Z) surface as shown in Figure 5.71 and press the middle mouse button.
Press **Apply** to move all the surfaces into the part INFLOW.

Figure 5.70
Create Part window

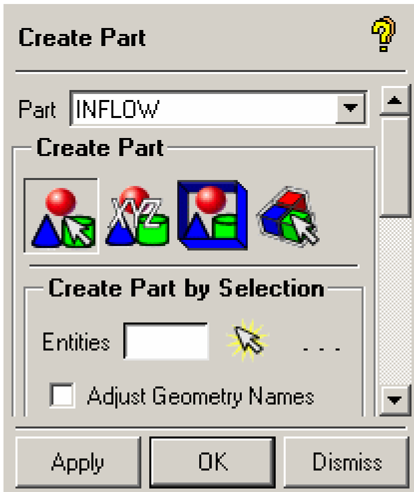
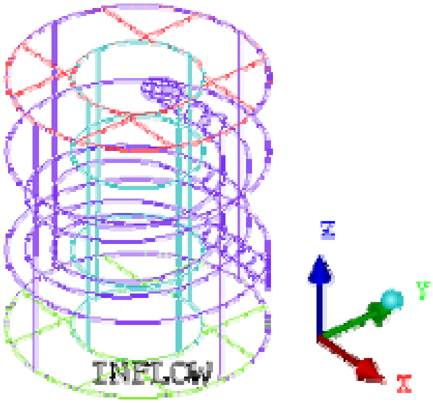


Figure 5.71
Surface for INFLOW



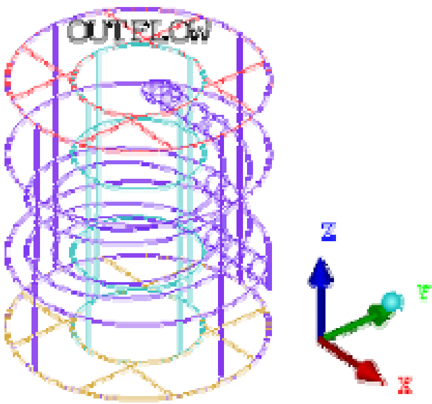
OUTFLOW:

Change the part name to OUTFLOW.

Click on Select entities,  select the top (max-Z) surface as shown in Figure 5.72 and click the middle mouse button.

Press **Apply** to move the surface into the part OUTFLOW.

Figure 5.72
Surface for OUTFLOW



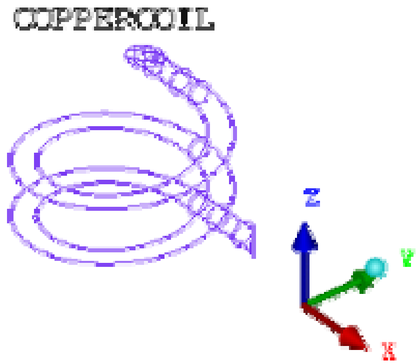
COPPERCOIL

Change the part name to COPPERCOIL.

Click on Select entities  and then select the surfaces representing the circular copper coil as shown in Figure 5.73. There are total of five surfaces, including two closing surfaces on the circular cylinder. After selecting the surfaces, click the middle mouse button to complete the selection.

Press **Apply** to move the surface into the part COPPERCOIL.

Figure 5.73
Surface for COPPERCOIL

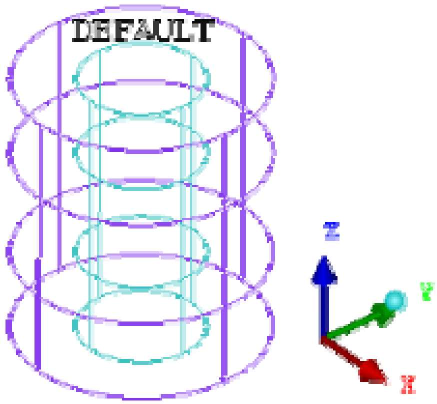


DEFAULT

Change the part name to DEFAULT.

Click on Select entities  and select the two cylindrical surfaces as shown in Figure 5.74. After selecting the surfaces, click the middle mouse button to complete the selection.

Figure 5.74
Surface for DEFAULT



Delete all the curves and then run Build Topology to create the curves and points in their respective parts.

Thus, go to the **Geometry** tab menubar and select Delete Curve. 

Click on Select curves  and press the hotkey "**a**" from the keyboard to select all the curves in the model.

Now, click on **Apply** to delete all the curves.

The original parts (PART_1 through PART_5) are empty of useful geometry. To delete empty Parts,

Right click on Parts and select **Delete Empty Parts** to delete the empty and un-necessary parts.

Some of these parts still have geometry in the “dormant” state and are not considered empty. To delete these parts right-click on the part name and select **Delete** and the **Delete** from the pop-up. Do this for PART_1 through PART_5.

The Display Tree before and after deleting and renaming parts is shown in Figure 5.75.

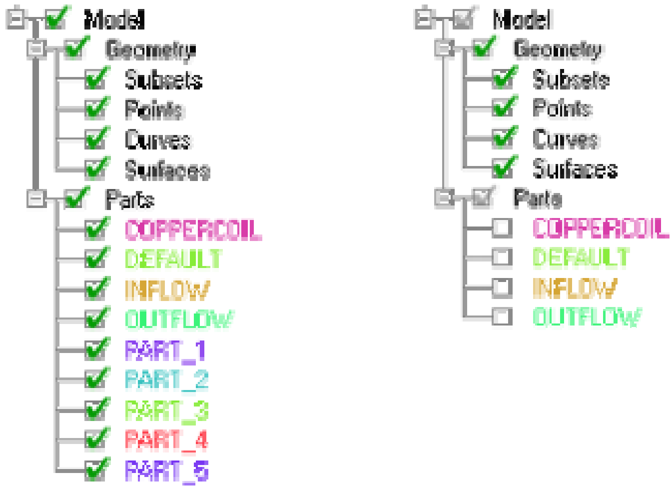


Figure 5.75 Display Tree before and after Deleting and renaming Parts

Creating Curves and Points

To get only the necessary curves and points:

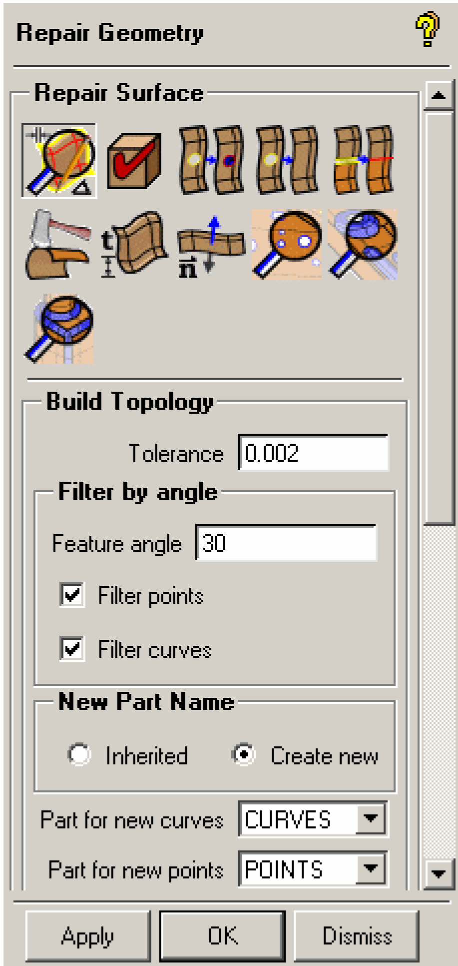
Click on Repair Geometry  from the Geometry tab menubar.

Click on Build Diagnostic Topology  and the window as shown in Figure 5.76 will appear.

Enable **Filter points** and **Filter curves**.

Set **New Part Name** to **Create new**; and click **Apply**.

Figure 5.76
Repair Geometry window



Note: Build Topology will turn ON the **Color by Count** and **Show Wide** option of the Curves Display. Right-click on **Curves** in the display tree to change the display options. User can turn OFF these options for the normal display of Curves.

Body Creation

Go to the **Geometry** tab menubar and select Create Body.

Give HEATINGCOIL as the Part Name and click on Material Point and toggle on the **Centroid of 2 points** option as shown in Figure 5.77.

Turn OFF all curves and points and display only surfaces from the Display Tree.

Click on Select location(s), select two opposite corners on the screen to place the Material Point within the tube as suggested in the Figure 5.78 and press the middle mouse button

Press **Apply**.

Figure 5.77
Create Body window

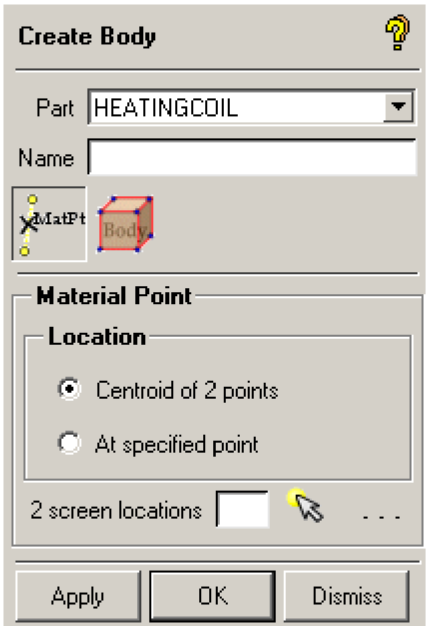
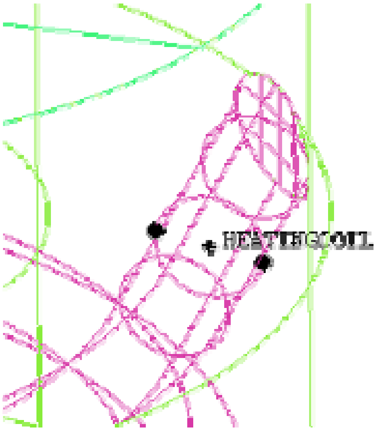


Figure 5.78 Two Opposite points for Material point

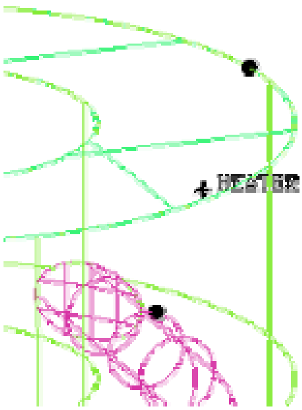


Give fluid Zone Part name as HEATER.

Click on Select location(s) and select two opposite corners on the screen to place the Material Point within the larger volume outside the tube as suggested in the Figure 5.79 and press the middle mouse button

Press **Apply**.

Figure 5.79 Two Opposite points for Material point



e) Mesh Generation

Global Mesh Parameters

From the **Mesh** tab menubar, click on Set Global Mesh Size. This will default into the General Parameters  section.

Leave Scale Factor as 1 for Global Element Scale Factor.

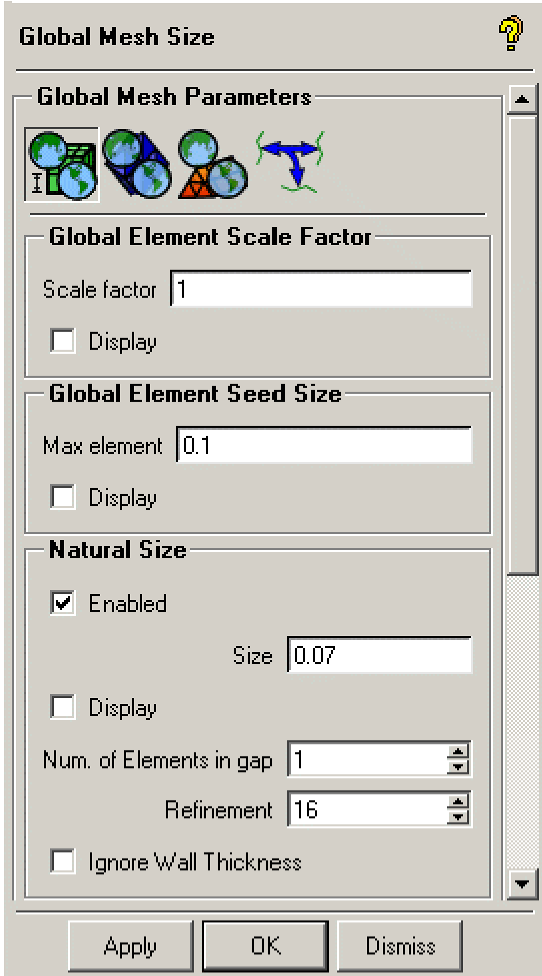
Change Max Element to 0.1 for Global Element Seed Size.

Set Refinement to 16.

Switch on **Natural Size** and give **Size** of 0.07 as shown in Figure 5.80. Natural size is the minimum elements size that will be achieved through automatic element subdivision based on local curvature and feature proximity. (A size smaller than natural size can be prescribed on a surface or curve and can still be reached.)

Click **Apply** to save this setting.

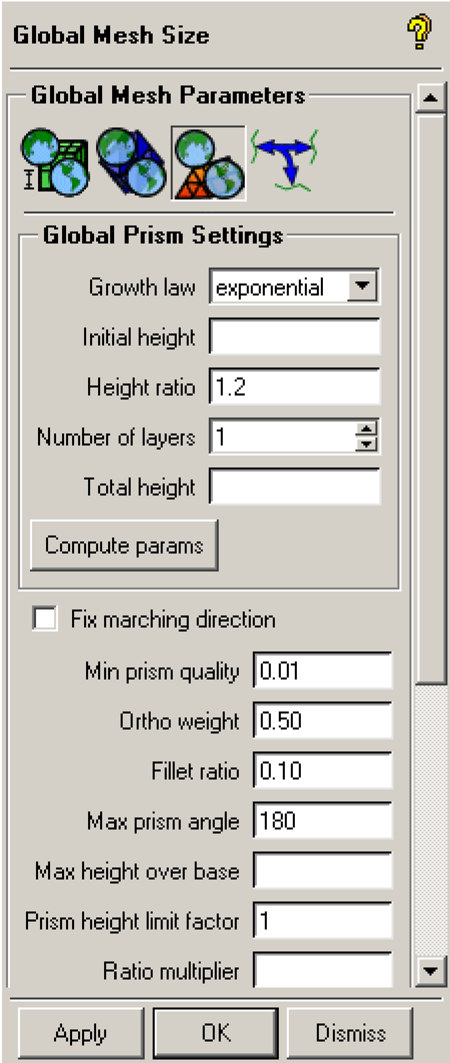
Figure 5.80
Global Mesh Size window



Global Prism Parameters

Now click on Prism Meshing Parameters. 

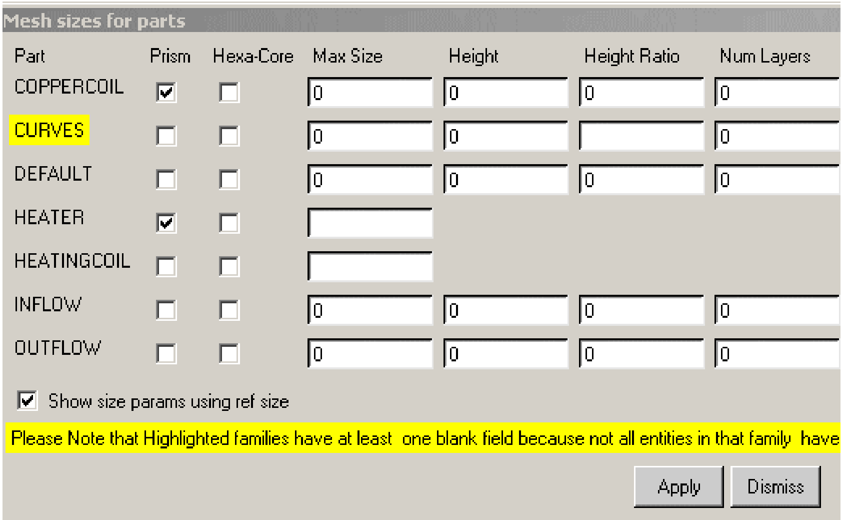
Leave **Initial height** blank and set **Number of layers** to 1. Leave the others default as shown below and press **Apply**.



Parts for Prisms

From the **Mesh** tab menubar select Set meshing Params by Parts. 

Enable **Prism** for the HEATER and COPPERCOIL parts as below and press **Apply** and **Dismiss**. This will enable prisms off the COPPERCOIL surfaces into the HEATER volume (the fluid side).



Saving the Project

Save the project by clicking on Save Project  from the Main Menu. This saves the geometry file as **HeatingCoil.tin** in the **HeatingCoil** directory.

Meshing

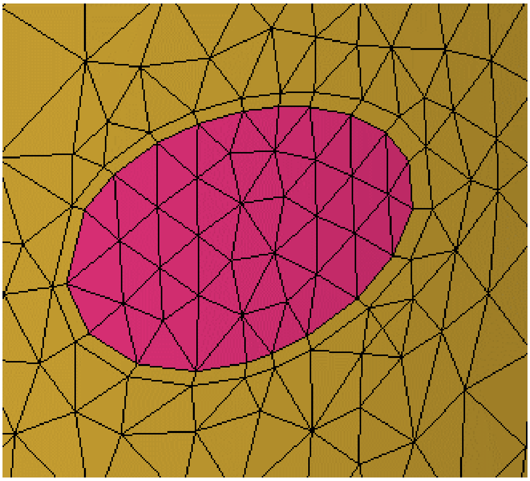
Select Volume Meshing  from the **Mesh** tab menubar to create the tet/prism mesh on this geometry.

Set the **Mesh type** to **Tetra + Prism Layers**. Click on From geometry  for the **Method**. Keep the defaults for the meshing. Notice that by default there would be 5 iterations of smoothing after the tetra meshing to take care of the bad elements.

Click **Apply** to create the tet/prism mesh.

Once the mesh is created, it gets loaded on the screen. The figure shows the mesh near the end of the copper coil where it passes through the outer wall – with one prism layer.

Figure 5.81
Tetra/Prism Mesh



f) Editing the Mesh

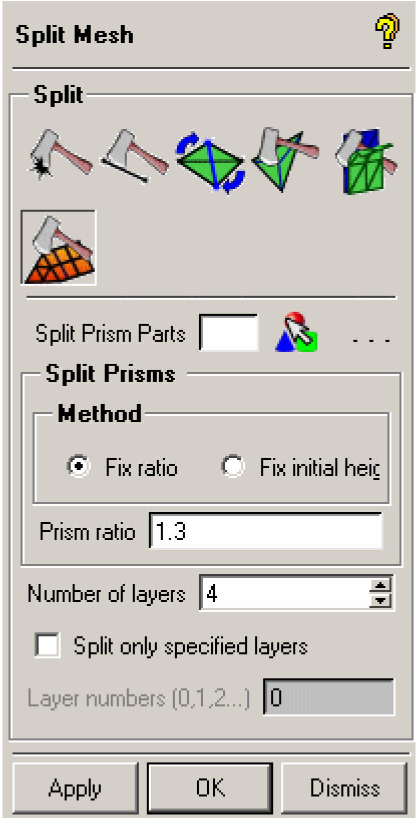
Splitting Prisms

From the **Edit Mesh** tab menubar select Split Mesh  then Split Prisms. 

Set **Prism ratio** to 1.3 and **Number of layers** to 4 as in Figure 5.82

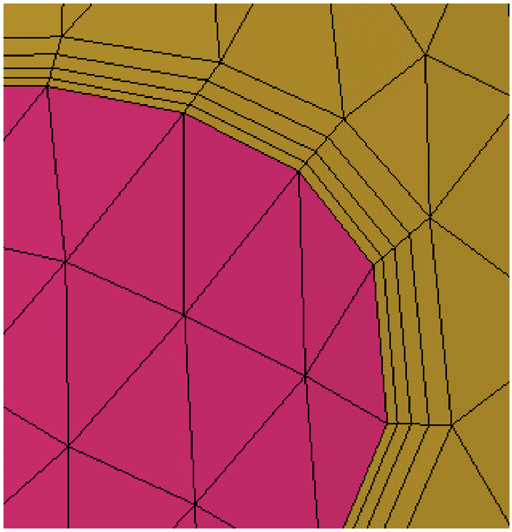
Press **Apply**.

Figure 5.82
Tetra/Prism Mesh



The resultant mesh will appear as in Figure 5.83.

Figure 5.83
Prism Layers



Redistributing Prisms

If the first prism height is required to be 0.001 everywhere, the created prism layers would need to be redistributed

From the **Edit Mesh** tab menubar select Move Nodes  then Redistribute Prism Edge. 

Set **Initial height** to 0.001.

Press **Apply**.