

ANSYS ICEM CFD 11.0

Tutorial Manual

January 2007

Table of Contents

1: ANSYS ICEM CFD 11.0 Tutorial Manual	1
2: Introduction to ANSYS ICEM CFD	4
2.1: The Unified Geometry Concept	5
2.2: The ANSYS ICEM CFD Geometry Interface	6
2.3: Meshing Modules	7
2.4: Mesh Visualization and Optimization	9
3: ANSYS ICEM CFD GUI	11
3.1: Main Menu	12
3.2: Utilities	13
3.3: Function Tabs	13
3.3.1: The Geometry menu	13
3.3.2: The Mesh menu	14
3.3.3: The Blocking menu	14
3.3.4: The Edit Mesh menu	15
3.3.5: The Output menu	15
3.3.6: The Post Processing menu	16
3.4: The Display Control Tree	16
3.4.1: Geometry	16
3.4.2: Mesh	17
3.4.3: Parts	17
3.4.4: The Message window	17
3.5: The Histogram window	18
4: CFD Tutorials	19
4.1: Geometry Creation	19

4.1.1: 2D Pipe Junction	19
4.1.2: 3D Pipe Junction	32
4.1.3: Sphere Cube	46
4.1.4: Pipe Blade	58
4.1.5: Geometry Simplification using Shrinkwrap	72
4.2: Hexa Meshing	82
4.2.1: Introduction	83
4.2.2: 2D Pipe Junction	94
4.2.3: 2D Car	123
4.2.4: 3D Pipe Junction	149
4.2.5: Sphere Cube	174
4.2.6: Pipe Blade	189
4.2.7: Elbow Part	223
4.2.8: Wing Body	251
4.3: Hexa Meshing Appendix	277
4.3.1: The Most Important Features of Blocking	277
4.3.2: Automatic O-grid Generation	278
4.3.3: Important Features of an O-grid	278
4.3.4: Edge Meshing Parameters	279
4.3.5: Smoothing Techniques	280
4.3.6: Refinement and Coarsening	281
4.3.7: Replay Functionality	282
4.3.8: Periodicity	283
4.3.9: Mesh Quality	283
4.4: Tetra	286
4.4.1: Introduction	286

4.4.2: Sphere Cube	293
4.4.3: 3D Pipe Junction	307
4.4.4: Fin Configuration	328
4.4.5: Piston Valve	345
4.4.6: STL Configuration	359
4.5: Tetra Meshing Appendix	383
4.5.1: Mesh Editor - Before Creating the Tetra Mesh	384
4.5.2: Tetra	387
4.5.3: Editing the Tetra Mesh	388
4.6: Advanced Meshing Tutorials	398
4.6.1: Hexa Mesh in a Grid Fin	401
4.6.2: Hybrid tube	448
4.6.3: Tetra mesh for Submarine	474
4.6.4: STL Repair with Tetra meshing	487
4.6.5: Workbench Integration	502
4.7: Cart3D	556
4.7.1: Tutorial Three Plugs	557
4.7.2: Tutorial Onera M6 Wing with 0.54 M	568
4.7.3: Onera M6 Wing with 0.84 M	588
4.7.4: Supersonic Missile	608
4.7.5: Business Jet	645
4.7.6: Bomber	664
4.7.7: Advanced Pitot Intake Tutorial	686
4.7.8: Advanced Tutorial Converging-Diverging Nozzle flow	708
4.8: Output to Solvers	728
4.8.1: Unstructured Mesh	730

4.8.2: Structured Mesh	739
4.9: Post Processing Tutorials	743
4.9.1: Pipe Network	743
4.9.2: Pipe Network (Advanced)	759
4.9.3: Space Shuttle	774
4.9.4: Space Shuttle (Advanced)	785
5: ANSYS ICEMCFD - CFX Tutorial Manual	795
5.1: Static Mixer	795
5.1.1: Overview	795
5.1.2: Starting a New Project	796
5.1.3: Geometry Creation	798
5.1.4: Mesh Generation	816
5.1.5: Writing Output	819
5.1.6: Exiting ANSYS ICEMCFD - CFX	821
5.1.7: Continuing with the Static Mixer Tutorial	821
5.2: Static Mixer 2 (Refined Mesh)	823
5.2.1: Overview	823
5.2.2: Starting a New Project	823
5.2.3: Creating Parts for Prism layers	825
5.2.4: Mesh Generation	826
5.2.5: Writing Output	833
5.2.6: Exiting ANSYS ICEMCFD - CFX	835
5.2.7: Continuing with the Static Mixer (Refined Mesh) Tutorail	835
5.3: Blunt Body	836
5.3.1: Overview	836
5.3.2: Starting a New Project	837

5.3.3: Geometry	837
5.3.4: Mesh Generation	846
5.3.5: Spliting Prism Layer	851
5.3.6: Checking Mesh Quality	853
5.3.7: Saving the Project	855
5.3.8: Output	855
5.3.9: Continuing with the Blunt Body Tutorial	857
5.4: Heating Coil	858
5.4.1: Overview	858
5.4.2: Starting a New Project	859
5.4.3: Geometry	860
5.4.4: Mesh Generation	866
5.4.5: Spliting Prism layer	870
5.4.6: Checking Mesh Quality	872
5.4.7: Writing Output	874
5.4.8: Exiting ANSYS ICEMCFD - CFX	876
5.4.9: Continuing with Heating Coil Tutorial	876
6: FEA Tutorials	877
6.1: Structural Meshing Tutorials	877
6.1.1: T-Pipe	877
6.1.2: Bar	896
6.1.3: Frame	905
6.1.4: Connecting Rod	928
6.1.5: PCB-Thermal Analysis	937
6.1.6: Tube Frame	943
6.1.7: Tibia	970

6.2: Ansys Tutorial	988
6.2.1: T-Pipe(Nastran Modal): Modal Analysis	988
6.2.2: T-Pipe(Abaqus Modal): Modal Analysis	1004
6.2.3: Connecting Rod: Thermal Boundary Condition	1018
6.2.4: Contact Analysis	1050
6.2.5: PCB-Thermal Analysis	1081
6.3: LS-Dyna Tutorial	1099
6.3.1: Frame: Quasi-Static Analysis	1099
6.3.2: Front Door-Side Impact	1116
6.3.3: PDA Drop Impact	1132
6.4: Nastran Tutorial	1154
6.4.1: T-Pipe	1154
6.4.2: Bar	1170
6.4.3: Frame	1192
6.4.4: Connecting Rod	1210
6.4.5: Hood	1230
6.5: Abaqus Tutorial	1245
6.5.1: Taper Rod Problem: Linear Static Analysis	1245
6.5.2: Wing Problem: Modal Analysis	1256
6.5.3: PinHole: Contact Analysis	1267

1: ANSYS ICEM CFD 11.0 Tutorial Manual

1.1: The ANSYS ICEM CFD Projects

Each project is located within the/docu/Tutorials directory in the ANSYS Installation 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.

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	1
------------------------	--	---

1.2: 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 each tutorial should be copied over to the user's working directory. All of the input files for the tutorials can be found within the ANSYS ICEM CFD installation. For example:
~/Ansys_inc/v110/icemcf/doch/Tutorials. They can be downloaded from the ANSYS Customer Portal or from http://www-berkeley.ansys.com/icemcf_ftp/index.html#icemcf_ftp. The manuals in *.pdf format for hardcopy output are also available.

1.3: 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.

1.4: 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.

1.5: 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

2: Introduction to ANSYS ICEM CFD

Meeting the requirement for integrated mesh generation and post processing tools for today's sophisticated analysis, ANSYS ICEM CFD 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 ICEM CFD is used especially in engineering applications such as computational fluid dynamics and structural analysis.

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

Multiblock structured

Unstructured hexahedral

Unstructured tetrahedral

Cartesian with H-grid refinement

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

Quadrilateral and triangular surface meshes

ANSYS ICEM CFD provides a direct link between geometry and analysis. In ANSYS ICEM CFD, geometry can be input from just about any format, whether 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 ICEM CFD's open geometry database offers the flexibility to combine geometric information in various formats for mesh generation. The resulting structured or

unstructured meshes, topology, inter-domain connectivity and boundary conditions are then stored in a database where they can easily be translated to input files formatted for a particular solver.

2.1: The Unified Geometry Concept

The unified geometry input environment in ANSYS ICEM CFD 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 ICEM CFD's meshing modules.

Direct CAD Interfaces and Intelligent Geometry

The ANSYS ICEM CFD 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 ICEM CFD, allowing users to operate in their native CAD systems. ANSYS ICEM CFD 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 ICEM CFD's Direct CAD Interfaces is available in the ANSYS ICEM CFD 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.

2.2: The ANSYS ICEM CFD Geometry Interface

Geometry Tools

ANSYS ICEM CFD 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 ICEM CFD 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 software. ANSYS ICEM CFD provides tools to do both on either CAD or triangulated surfaces.

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	6
------------------------	--	---

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.

2.3: Meshing Modules

Tetra

ANSYS ICEM CFD 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 ICEM CFD Tetra works directly from the CAD surfaces and fills the volume with tetrahedral elements using the Octree approach. A powerful smoothing algorithm provides the element quality. Options are available to automatically refine and coarsen the mesh both on geometry and within the volume.

A Delaunay 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 ICEM CFD semi-automated meshing module presents rapid generation of multi-block structured or unstructured hexahedral volume meshes.

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

Blocks can be built and interactively adjusted to the underlying CAD geometry. This blocking can be used as a template for other similar

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	7
------------------------	--	---

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 ICEM CFD 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.

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

Shell Meshing

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

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

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	8
------------------------	--	---

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

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

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

2.4: 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 ICEM CFD 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 ICEM CFD 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.

3: ANSYS ICEM CFD GUI

ANSYS ICEM CFD'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

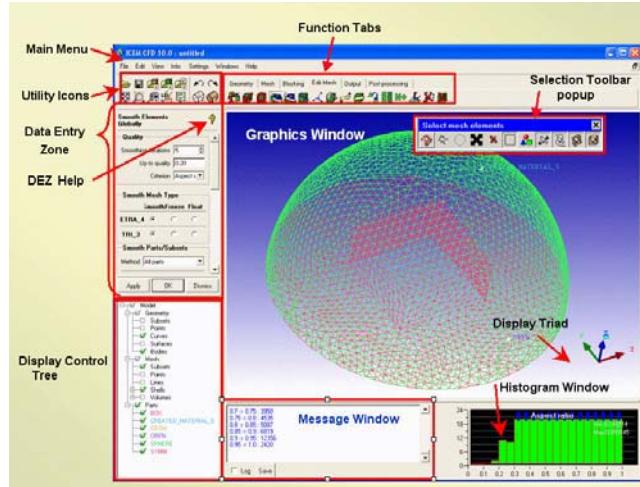
Graphics Window

Message Window

Histogram (Quality) Display Window

Selection Toolbar

Figure 3-1
Ai*Environment



3.1: 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

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	12
------------------------	--	----

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 tutorials, user's guide and version information.

3.2: 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.

3.3: 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** etc.

3.3.1: The Geometry menu

The Geometry menu includes functions for the creation, editing and repair of geometry.

For more information on ANSYS ICEM CFD'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.

3.3.2: The Mesh menu

These tools are the heart of ANSYS ICEM CFD. The Mesh menu contains the ANSYS ICEM CFD 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 ICEM CFD'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 ICEM CFD maintains and develops:

- Global Mesh Setup
- Part Mesh Setup
- Surface Mesh Setup
- Curve Mesh Setup
- Create Mesh Density
- Define Connectors
- Mesh Curve
- Compute Mesh

3.3.3: 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 Histogram
Pre-Mesh Smooth
Block Checks
Delete Block

3.3.4: The Edit Mesh menu

The Edit mesh menu contains tools necessary for mesh editing, both automated and manual. Operations include:

Create Elements
Extrude Mesh
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
Assign Mesh Thickness
ReOrient Mesh
Delete Nodes
Delete Elements

3.3.5: 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

3.3.6: 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 Step
- Variables
- Define Cut Plane
- Define Iso-surface
- Point Probe on Surfaces
- Import External Surface
- Streams
- Control All Animations
- Annotation
- XY or Polar

3.4: 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.

3.4.1: Geometry

Controls display of points, curves, surfaces and bodies (material volumes). Subsets can also be created, displayed and modified. A given subset can

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	16
------------------------	--	----

contain any number of different geometry types. A given entity can belong to more than one subset.

3.4.2: 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.

3.4.3: 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.

3.4.4: The Message window

The Message window contains all the messages that ANSYS ICEM CFD writes out to keep the user informed of internal processes. The Message window displays the 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 ICEM CFD was fired.

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	17
------------------------	--	----

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.

3.5: 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.

4: CFD Tutorials

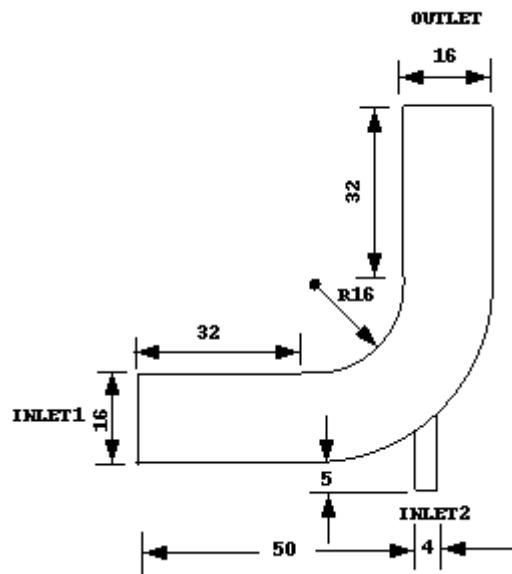
4.1: Geometry Creation

4.1.1: 2D Pipe Junction

Overview

We are going to create geometry for a two-dimensional pipe junction as shown in the figure below.

Figure4-1
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

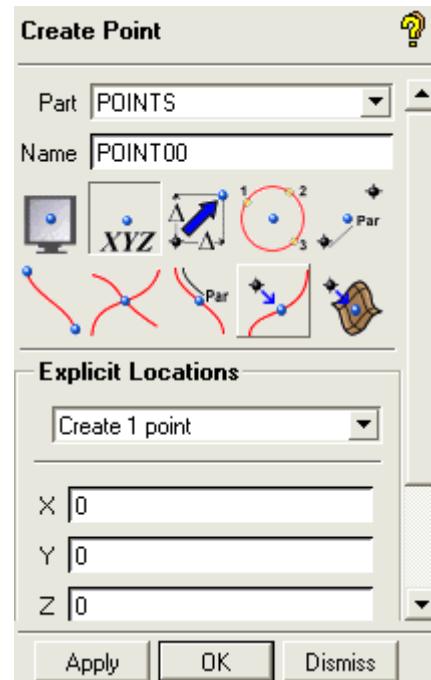
b) Point Creation

Note:

1. Settings > Selection >Auto pick mode should be turned OFF.
2. Settings>Geometry Options>Name new geometry must be turned ON.
3. Settings>Geometry Options>Inherit Part name>Create new must be toggled ON
4. In case UNDO is used after creation of any point, and then a new point is created, the new point will have the next name in series. For example, if Point05 is created and Undo is used, then the next point created will be named Point06.

Select Geometry Create Point  > Explicit Coordinates  >
Select Create 1 Point. Input the Part name POINTS and Name as POINT 00. Assign coordinates (0 0 0) shown below. Press Apply to create a point.

Figure 4-2
Point creation window



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 POINT00.

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:

POINT01 (32, 0, 0)

POINT02 (0, 16, 0)

POINT03 (32, 16, 0)

POINT04 (48, 32, 0)

Geometry Creation

POINT05	(48, 64, 0)
POINT06	(64, 32, 0)
POINT07	(64, 64, 0)
POINT08	(50, -5, 0)
POINT09	(54, -5, 0)
POINT10	(16, 32, 0)
POINT11	(0, 32, 0)
POINT12	(50, 16, 0)
POINT13	(54, 16, 0)

Figure 4-3
Points created thus far



Press **Dismiss** to close the window. Go to View > Front. The Display window should now show the points as seen in the figure above. The location of points can also be checked by following way –Go to Utility

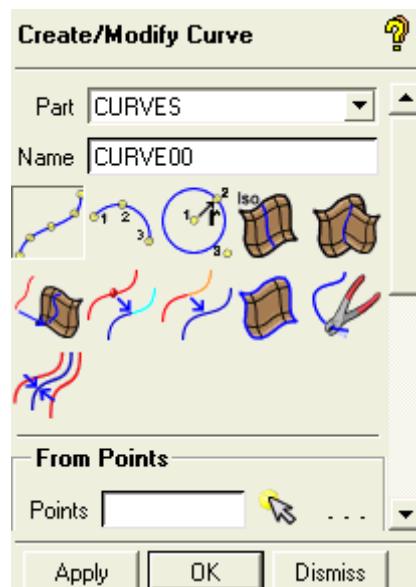
Icons > Click on the inverted arrow below Measure distance icon 

>Last option is Find Location  Select any point on screen. The Co-ordinates of the point will be shown on screen as well as will be visible in the Message Window.

c) Line Creation

Geometry > Create/Modify Curve  > From Points: Select the From Points option .

Figure 4-4 :
From points window



To select Points, click on  (select point icon) and then select POINT00 and POINT01 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 CURVE00. 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 CURVE00. Similarly, select the following points, pressing middle mouse button each time. Without changing the Name entry, by default the names of each new curve would appear as shown on the left:

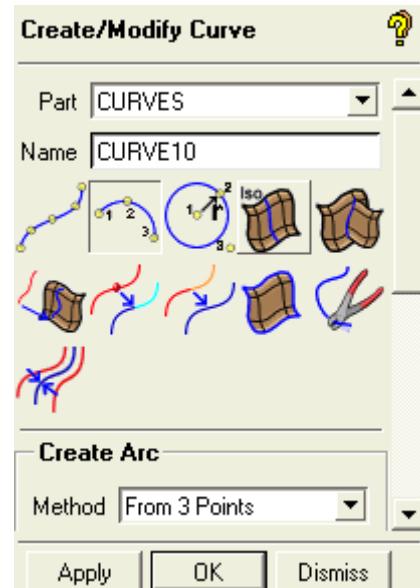
CURVE01 from POINT00 and POINT02
 CURVE02 from POINT02 and POINT03
 CURVE03 from POINT04 and POINT05
 CURVE04 from POINT05 and POINT07
 CURVE05 from POINT06 and POINT07
 CURVE06 from POINT08 and POINT09
 CURVE07 from POINT08 and POINT012
 CURVE08 from POINT09 and POINT013

Press **Dismiss** to close the window.

d) Arc Creation

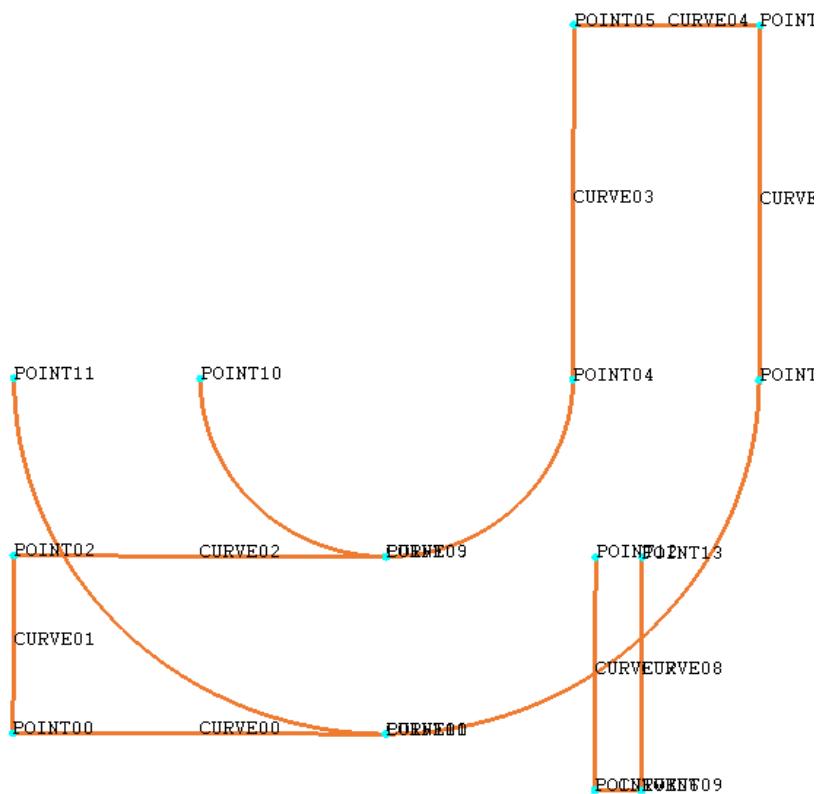
Geometry > Create/Modify Curves  > Select Arc Through 3 points
 to open the window here.

Figure 4-5
Arc from 3 points window



To select Points click on (select point icon), and select the points **POINT04**, **POINT03** and **POINT10** 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 **CURVE09** and press Apply to create the arc.
 Similarly, make another arc named **CURVE10** out of points **POINT06**, **POINT01**, and **POINT11**. Press Dismiss to close this window. The geometry after creating the two arcs is shown here.

Figure 4-6
Geometry after arc creation

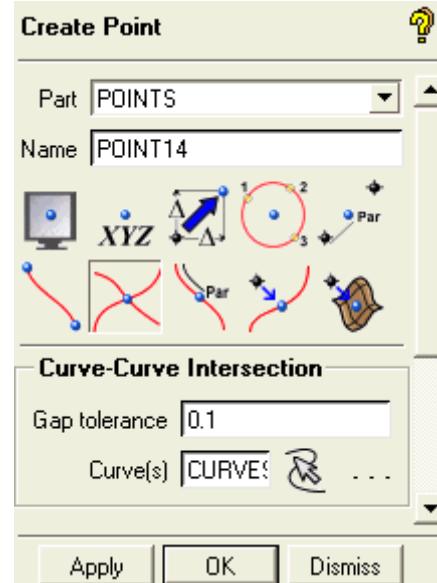


e) Curve-Curve Intersection

Geometry >Create Point > Select Curve-Curve Intersection the window opens as shown below. Select the Part name **POINTS**. Select **CURVE10** and **CURVE07** with the **left mouse button**. Press the **middle mouse button** to accept the selection. Give Gap a Tolerance of **0.1** and press **Apply**. This will create the intersection point called **POINT14**. Repeat the procedure for curves **CURVE10** and **CURVE08** and press

Apply without changing the name in the Name window to get the intersection point **POINT15**. Press **Dismiss** to close the Create Point window.

Figure 4-7
Selection window of Curve-Curve Intersection



f) Segmentation of Curves at existing points

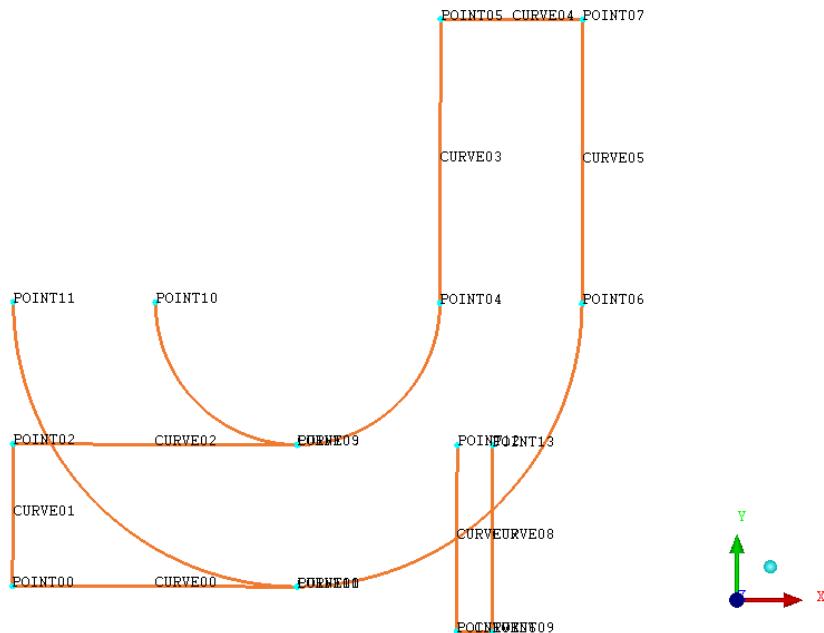
Geometry > Create/Modify Curve > Select Segment curve . In the dropdown, Segment by Point should be selected. Select the curve selection icon and select **CURVE10** with the **left mouse button**.

Now select the point selection icon and select **POINT01** with the left mouse button and then press the middle mouse button to accept the point. Select the Part **CURVES**. After pressing Apply, the **CURVE10** segments into two curves, **CURVE10** and **CURVE11**.

Similarly segment **CURVE09** at **POINT03** to get **CURVE09** and **CURVE12**. Segment **CURVE07** at **POINT14** to get **CURVE07** and **CURVE13**. Segment **CURVE08** at **POINT15** to get **CURVE08** and **CURVE14**. The geometry after segmenting the curve is shown below.

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

Figure 4-8
Geometry after curve segmentations



g) Deletion of unused entities

Geometry > Delete Curve  - This will open the Delete Curve window.

Select the curve selection icon  and select **CURVE11, CURVE12, CURVE13** and **CURVE14**. Press the **middle mouse** button to complete the selection. Press **Apply** to delete these curves.

Geometry >Delete Points  This will open the Delete Points window.

Select the point selection icon  and select **POINT10**, **POINT11**, **POINT12** and **POINT13**. Press the middle mouse button to complete selection, and press Apply to delete these points.

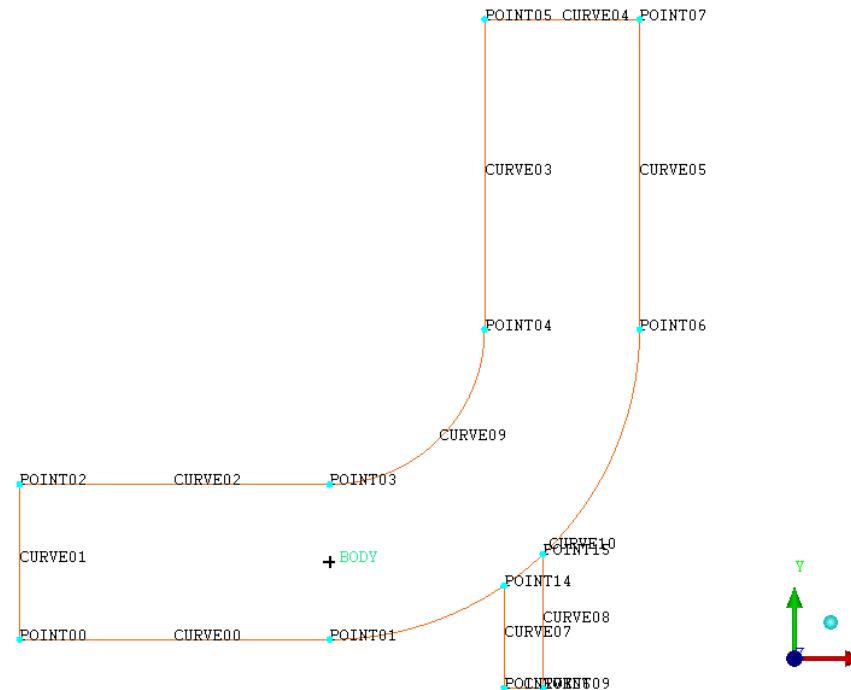
h) Creating the Material point

Geometry > Create Body  >  Material Point > Centroid of 2

points: Select the location selection icon  and click close to **POINT01** and **POINT03** 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 below.

Figure 4-9
Final Geometry

Geometry Creation



i) Saving Geometry

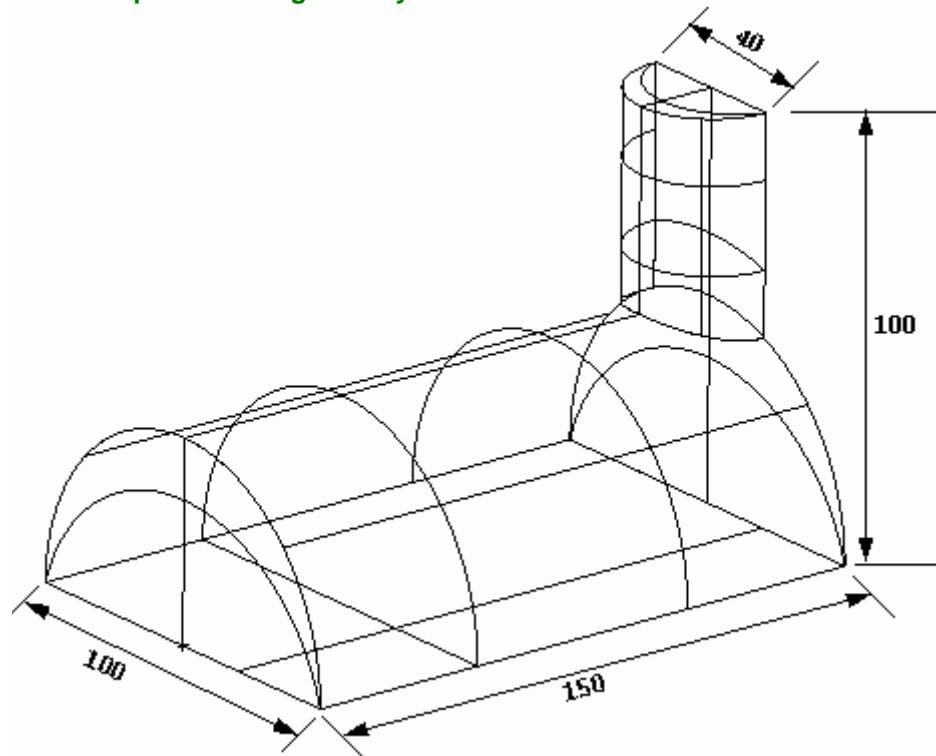
File > Geometry > Save Geometry As: Enter the file name as **Geo_2DPipe.tin** and press **Save** to save the geometry file

4.1.2: 3D Pipe Junction

Overview

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

Figure 4-10
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) Point Creation

Note:

1. Settings > Selection > Auto pick mode should be turned OFF.
2. Settings > Geometry Options > Name new geometry must be turned ON.
3. Settings > Geometry Options > Inherit Part name > Create new must be toggled ON
4. In case UNDO is used after creation of any point, and then a new point is created, the new point will have the next name in series. For example, if Point05 is created and Undo is used, then the next point created will be named Point06.

Geometry > Create Point  > Explicit Coordinates  - Select Create 1 Point. Input the Part name **POINTS**, and Name as **POINT00**. Enter the co-ordinates (0 0 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 will be shown as **POINT00**.

Now, create the rest of the points listed below by just entering the locations. The names continue on from POINT00, so they will automatically change as shown below:

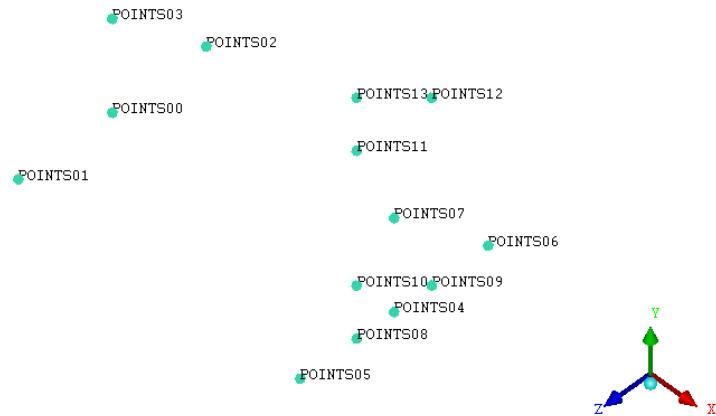
POINT01	(0, 0, 50)
POINT02	(0, 0, -50)
POINT03	(0, 50, 0)
POINT04	(150, 0, 0)
POINT05	(150, 0, 50)

Geometry Creation

POINT06	(150, 0, -50)
POINT07	(150, 50, 0)
POINT08	(150, 0, 20)
POINT09	(150, 0, -20)
POINT10	(130, 0, 0)
POINT11	(150, 100, 20)
POINT12	(150, 100, -20)
POINT13	(130, 100, 0)

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

Figure 4-11
Points created



c) Arc Creation

Geometry > Create/Modify Curve  > Select Arc through 3 points:

 to open command window. Type the Part name as **CURVES**. Enter the Name as **CURVE00**. Select the location selection icon  and select points **POINT01**, **POINT03** and **POINT02** with the **left mouse button**. 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 CURVE00.

Similarly, select POINT05, POINT07 and POINT06 and enter the name as **CURVE01**. 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:

CURVE02: POINT08, POINT10 and POINT09

CURVE03: POINT11, POINT13 and POINT12

Press Dismiss to close the window.

d) Line Creation

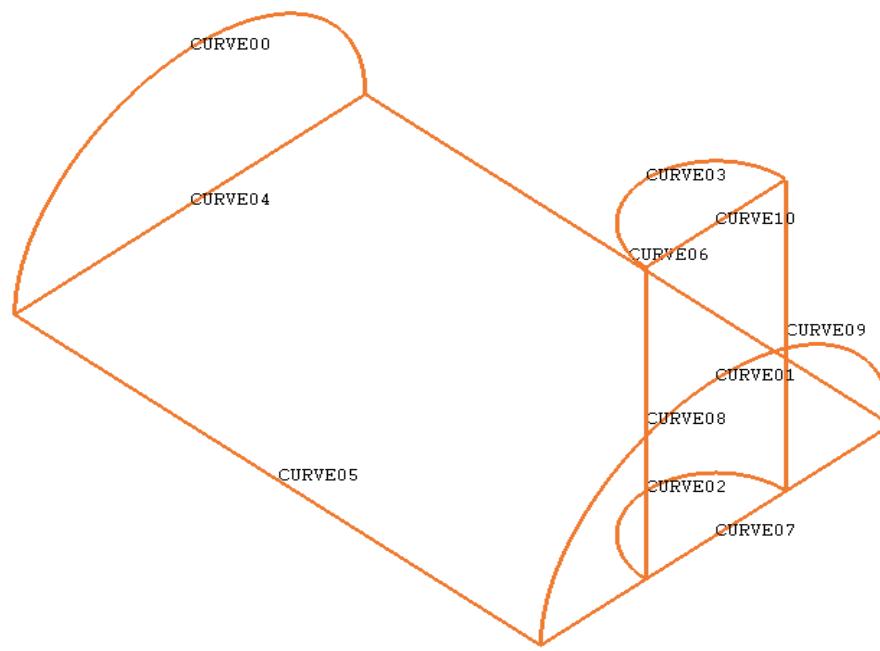
Geometry > Create/Modify Curves  > Select From Points:  Press the location selection icon  , select the Points **POINT01** and **POINT02** with the **left mouse button**, and press the **middle mouse button** to complete the selection. Enter the Part name **CURVES** and Name **CURVE04**.

Similarly, create six more lines using the points listed below. For each curve the curve names will adjust consecutively according to previous curve:

CURVE05:	POINT01 and POINT05
CURVE06:	POINT02 and POINT06
CURVE07:	POINT05 and POINT06
CURVE08:	POINT08 and POINT11
CURVE09:	POINT09 and POINT12
CURVE10:	POINT11 and POINT12

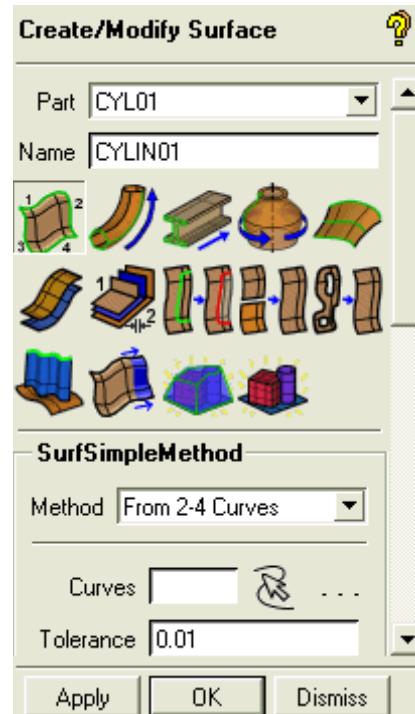
Press **Dismiss** to close the window. The Geometry after curve creation is shown below. Switch OFF the Points in the Display Tree window to avoid clutter on the screen.

Figure 4-12
Geometry after line creation

**e) Surface Creation**

Geometry >Create/Modify Surface > Select Simple Surface icon to open the window shown.

Figure 4-13
Surface creation from curve



Select the option From 2-4 curves. Press the curve selection icon  and select the curves CURVE00 and CURVE01 with the left mouse button. Press the middle mouse button to complete the selection. Enter a Tolerance as 0.01. Enter the Part name CYL1 and Name CYLIN01. Press Apply to create the surface.

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

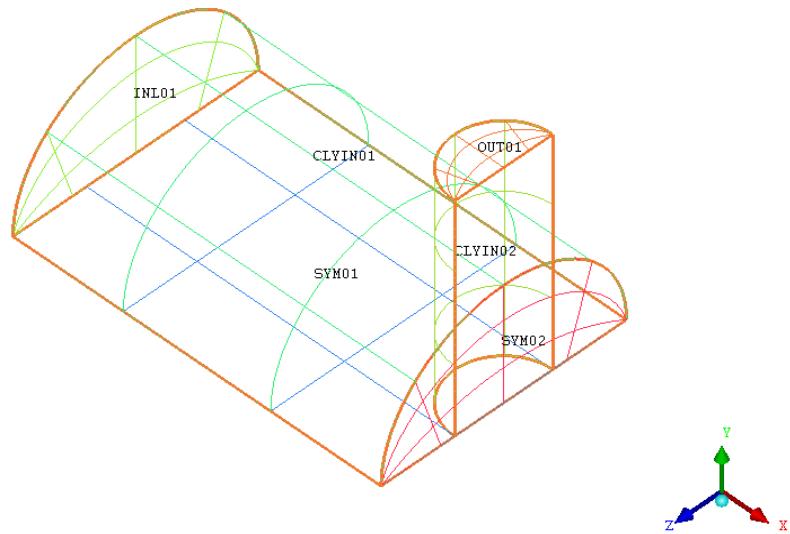
Switch **ON** the **surfaces** from the left side Display Tree window. To see the names of the surfaces, using the right mouse button, select Surface > Show Surface names in the Display Tree window. Use the right mouse

button to zoom out if needed. The newly created surface name would display as CYLIN01.

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

PART	NAME	SELECTED CURVES
INL	INL01	CURVE00, CURVE04
CYL2	CYLIN02	CURVE02, CURVE03
OUT	OUT01	CURVE03, CURVE10
SYM	SYM01	CURVE04, CURVE07
SYM	SYM02	CURVE01, CURVE07

Figure 4-14
Geometry after Surface creation



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

f) Surface-Surface Intersection

Geometry >Create/Modify Curves  > Surface-Surface
Intersection  >Select first option Surfaces. Select the surface

selection icon to select surfaces **CYLIN01** and **CYLIN02** with the **left mouse button**, pressing the **middle mouse button** to complete the selection each time. Check that the part name selected is CURVES. Press **Apply** to create the intersection curve.

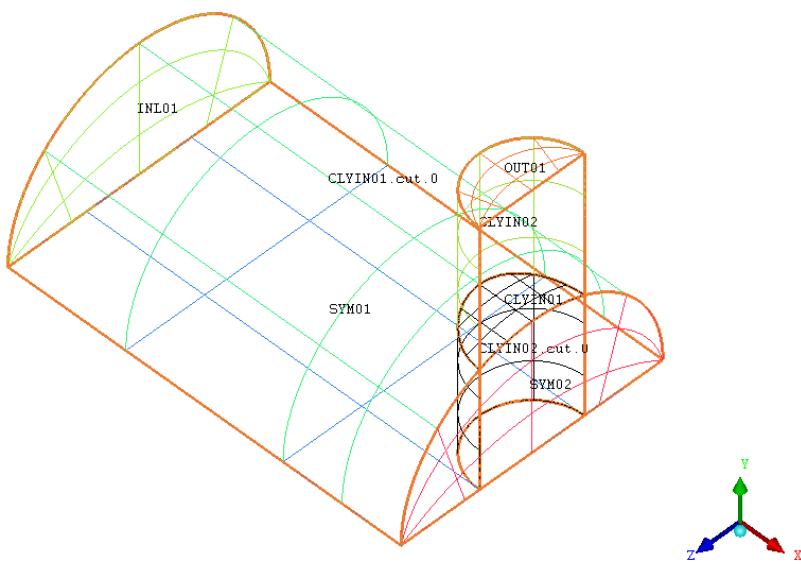
g) Segmentation of Surface

Geometry > Create/Modify Surface > Segment/Trim surface : Choose the Method **by Curves**, which is the default. Press the surface selection icon and select the surface **CYLIN01** 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. Check that the part name selected is **CYL01**. Press **Apply** to segment the surface **CYLIN01** into two parts. Similarly, segment the surface **CYLIN02** with the same intersection curve. If the two previous curves have been split into two, then select both curves. Check that the part name selected while segmenting is **CYL02**.

Deleting unused entities

Geometry > Delete Surface this opens 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. The surfaces are marked in the figure below.

Figure 4-15 Surfaces to be deleted



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

Geometry > 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 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.

Settings > Geometry Options > Inherit part name > Inherit should be Toggled ON. This will ensure that all the points and curves extracted during Build Topology operation will be moved to their respective parts.

h) Build topology

Geometry > Repair Geometry  > Build Diagnostic Topology 
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.

i) Surface creation

First, make sure **Curves** are ON in the Display Tree. Again we have to change Settings > Geometry Options > Inherit part name > to Create New.

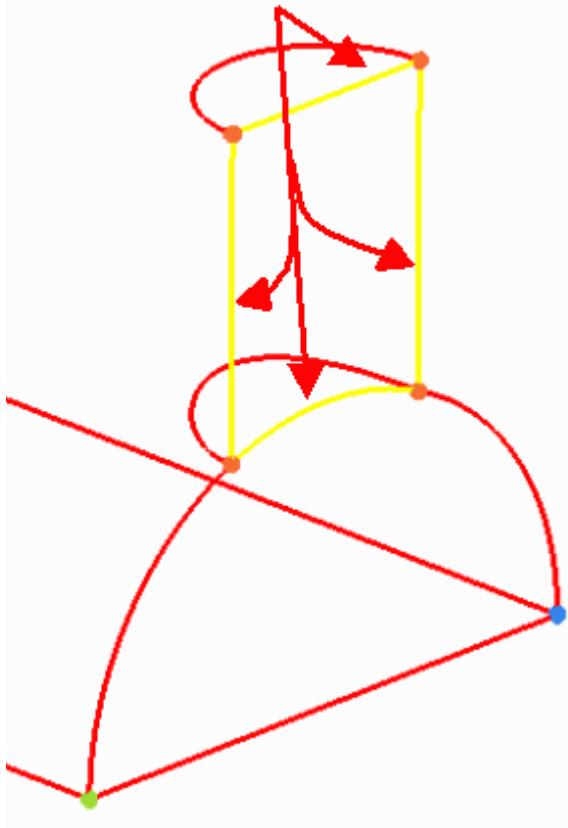
Now select Geometry > Create/Modify Surface : Select Simple surface

 to open the Create/Modify Surface window. Select Method-From Curves. Select the curves shown 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 SYM03. Press Apply to create the surface.

Press **Dismiss** to close the window.

Figure 4-16 Curves for surface

Select these curves



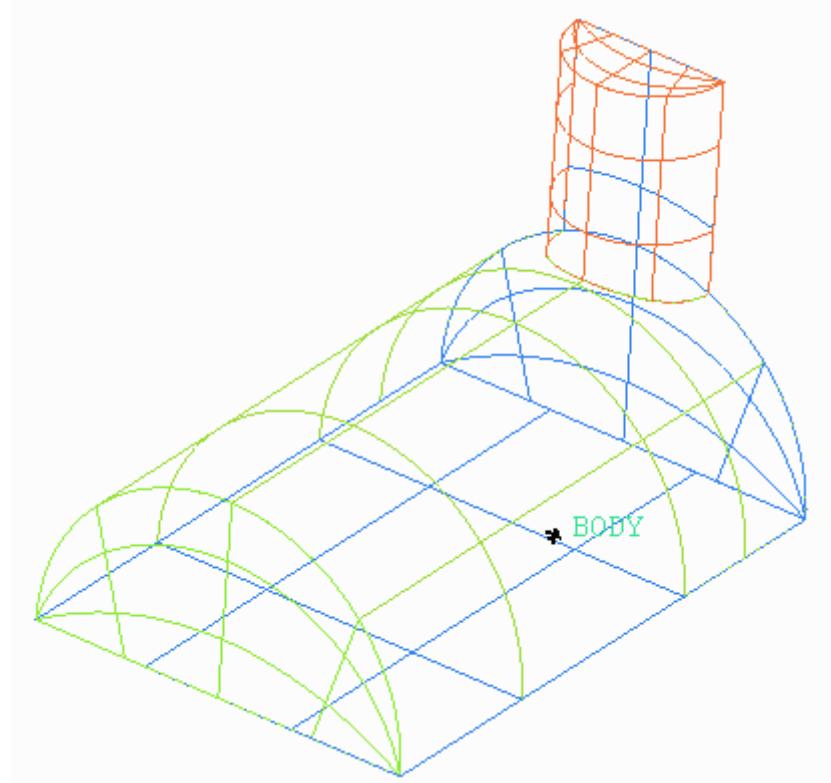
j) Creating the Material point

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. The final geometry is shown below.

Figure 4-17 Final Geometry



k) Saving Geometry

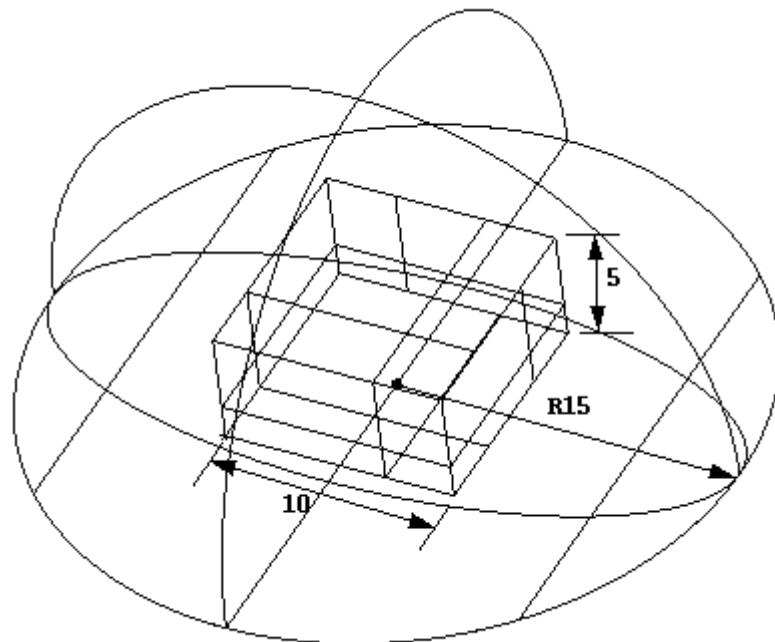
File > Geometry > Save Geometry As: Enter the file name as Geo_3DPipe.tin and press **Save** to save the geometry.

4.1.3: Sphere Cube

Overview

We will create geometry for a sphere cube as shown below.

Figure 4-18
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:

- 1 Settings > Selection > Auto pick mode should be turned OFF.
- 2 Settings > Geometry Options > Name new geometry must be turned ON.
- 3 In case UNDO is used after creation of any point, and then a new point is created, the new point will have the next name in series. For example, if Point05 is created and Undo is used, then the next point created will be named Point06.
- 4 Settings > Geometry Options > Inherit Part name > Create New must be toggled ON.

c) Point Creation

Go to Geometry > Create Point > Explicit Coordinates  . Give the Part name **POINTS**, and the Name **POINT00**. Enter the co-ordinates **(5, -10, 0)** and press Apply to create the point.

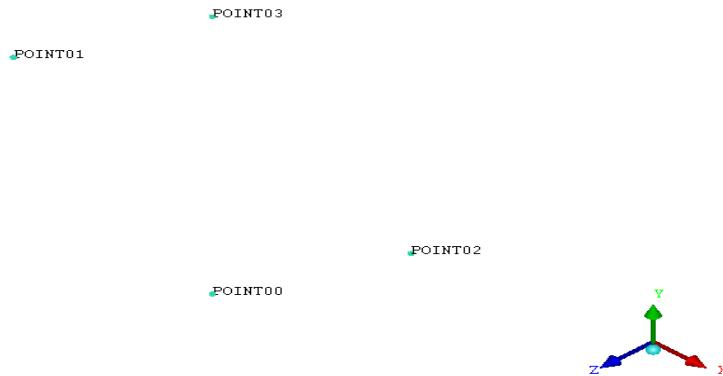
Switch on the **Points** in the Display Tree window. To see the names of the points, use the right mouse button to select **Points > Show Point names** in the Display Tree window. Select **Fit Window**  from the main menu. Use the right mouse button to zoom out if needed. The newly created point name would be displayed as POINT00.

Similarly, enter the coordinate as below and create 3 more additional points at the following locations. The names will automatically adjust as shown below:

POINT01	(-10,5, 0)
POINT02	(20, 5, 0)
POINT03	(5, 20, 0)

Press **Dismiss** to close the window. The Geometry after point creation is shown in below.

Figure 4-19 Points created so far



d) Arc Creation

Geometry > Create/Modify Curve > Arc through 3 points Select **POINT01**, **POINT00** and **POINT02** with the **left mouse button**, and press the **middle mouse button** to complete selection. Enter the Part as **CURVES** and Name as **CURVE00**. Press Apply to create the arc.

Similarly, create another arc called **CURVE01** from points **POINT01**, **POINT03**, and **POINT02**. Press Dismiss to close the window.

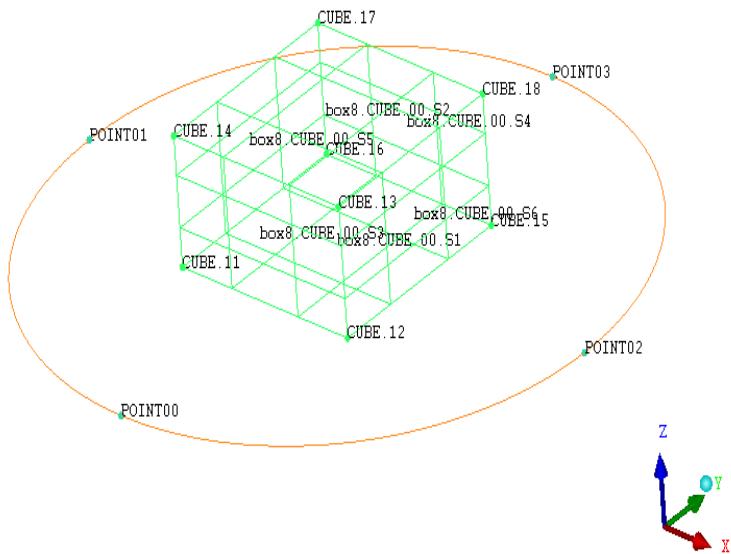
Note: Turn on **Curves** in the Display tree to see the curves.

e) Cube Creation

Go to Geometry > Create/Modify Surface > Standard Shapes  > Box

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 like the figure below.

**Figure
4-20:
Geometry
so far**

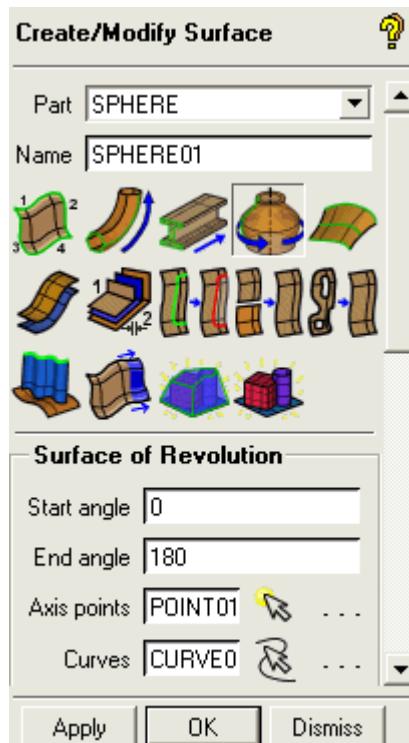


f) Hemisphere Creation

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

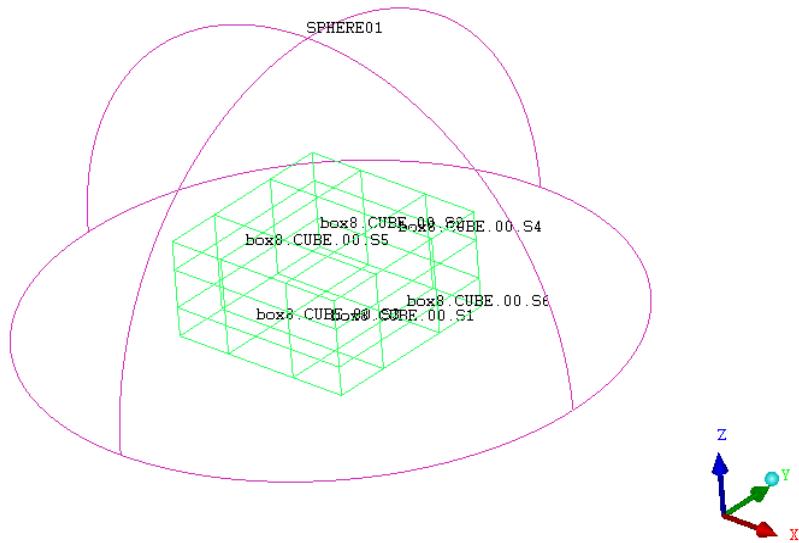
Geometry > Create/Modify Surface  > Surface of Revolution  . Change the Part to **SPHERE**, Name to **SPHERE01**. Enter the Start angle **0** and the End angle as **180**. Select Axis Points as **POINT01** and **POINT02**. Select curves as **CURVE00** and press Apply to create the hemisphere.

**Figure
4-21:
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 the figure below.

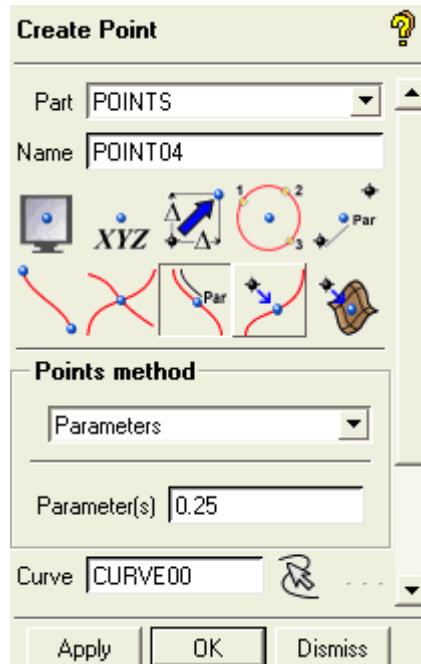
Figure 4-22
Geometry after revolution



g) Point Creation

Geometry >Create Point > Parameter along a Curve  to open the window as seen below. For a clearer view, the user can turn **OFF** Surfaces in the Display Tree, and make sure that Curves and Points are **ON**.

Figure 4-23: Point Parameter on curve window



Turn off the part **SPHERE** and Turn ON Points > Show Point Names in the Display Tree to be able to see the curve names better.

Select the Part as **POINTS**. And enter the Name as **POINT04**. Then select the curve, **CURVE00**. Enter Curve Parameter **0.25** and press Apply to create **POINT04**. Then change the parameter to **0.75**, and press Apply again to create **POINT05**.

Next, select the curve, **CURVE01**. With the parameter left at **0.75**, press Apply to create POINT06. Then change the parameter to **0.25**, and press Apply again to create POINT07.

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

Geometry > Delete Curve - Toggle ON **Delete permanently**. Select the curves, **CURVE00** and **CURVE01**, and press **Apply**.

h) Arc Creation

Geometry > Create/Modify Curve  **> Arc through 3 points:**

. Make sure Point Names are being displayed by right clicking in the Display Tree on Points > Show Point Names. Select the points, **POINT05**, **POINT02**, and **POINT06**. Enter the Part as **CURVES** and the Name as **CURVE00**. Press Apply to create the arc.



Similarly create three other arcs by using the following points:

CURVE01: POINT06, POINT03 and POINT07

CURVE02: POINT07, POINT01 and POINT04

CURVE03: POINT04, POINT00 and POINT05

Press **Dismiss** to close the window.

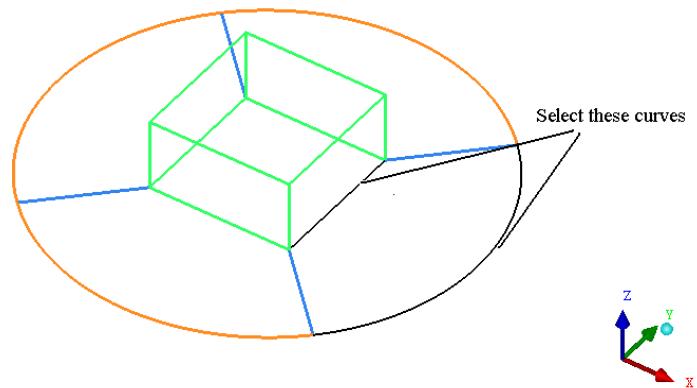
i) Surface Creation

Geometry >Create/Modify Surface  **> Simple Surface**  **>**

Select From Curves option to open the Select Curves window. Turn **OFF** the **Points** (Geometry) for a better view. Also turn OFF the curve names : Curves > Show Curve Names for a better view. Select the two curves shown in black below, and press the middle mouse button to complete the selection. Assign the Part as **SYM** and Name as **SYM.1**. Press Apply to create the surface.

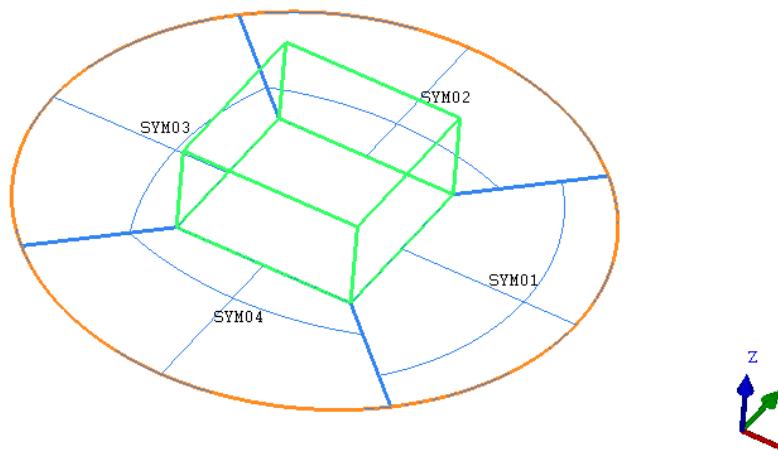
Press **Dismiss** to close the window.

Figure 4-24 Curves for Surface



Similarly, create the other three surfaces around the cube. The result is shown below.

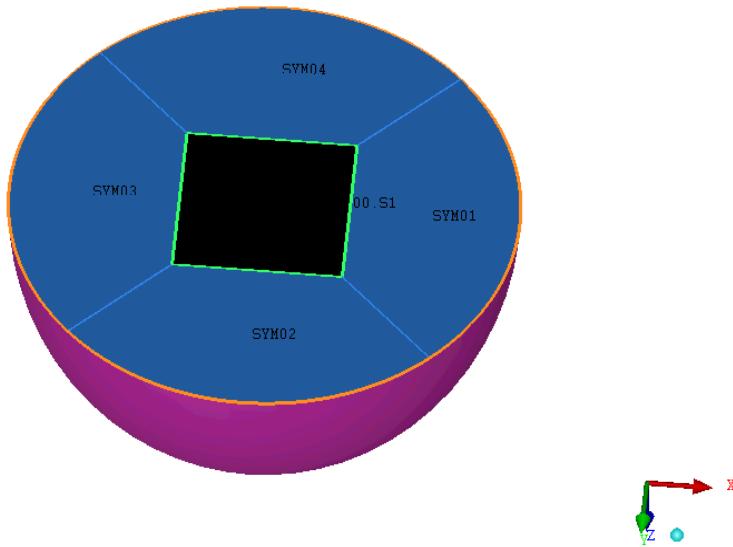
Figure 4-25 Symmetry Surfaces



j) **Deleting unused entities**

Geometry > Delete Surface Select the surface shown in black in the figure below 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.

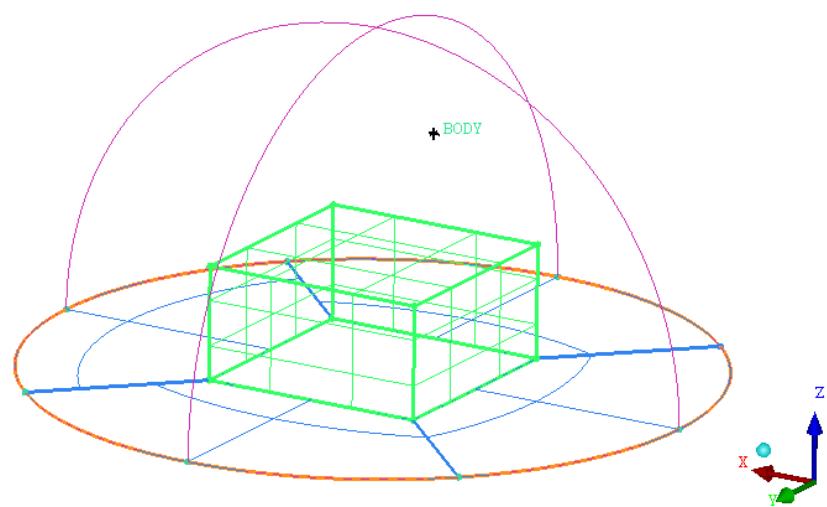
Figure 4-26 Surface to delete



k) Creating the material point

Geometry > Create Body >Material Point >Centroid of 2 Point -
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 below.

Figure 4-27 Final Geometry



I) Saving Geometry

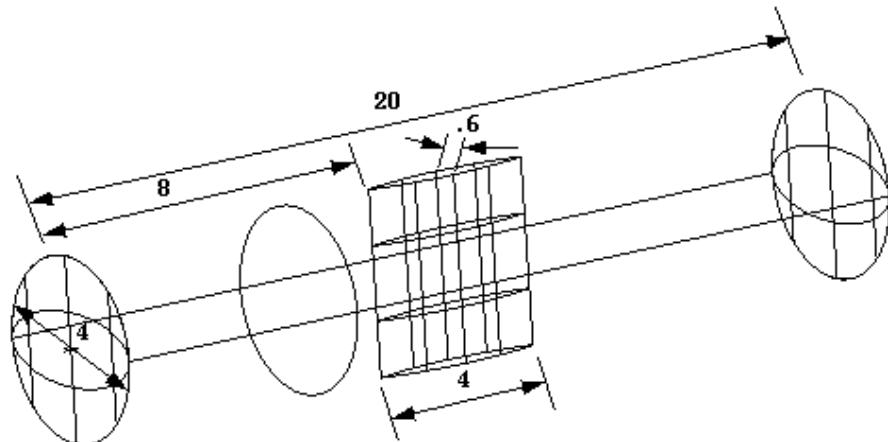
File > Geometry > Save Geometry As: Enter the file name **Geo_SphereCube.tin** and press **Save** to save the geometry.

4.1.4: Pipe Blade

Overview

We are going to create the geometry for a pipe blade.

Figure 4-28 : 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.

Note:

1 Settings > Selection >Auto pick mode should be turned OFF.

2 Settings>Geometry Options>Name new geometry must be turned ON.

3 In case UNDO is used after creation of any point, and then a new point is created, the new point will have the next name in series. For example, if Point05 is created and Undo is used, then the next point created will be named Point06.

4 Settings>Geometry Options>Inherit Part name>Inherited must be toggled ON.

b) Point Creation

Geometry >Create Point > Explicit Coordinates  . Assign the Part name POINTS, and the Name POINT00. Enter the co-ordinates **(0, 2, 8)**, and press **Apply** to create the point.

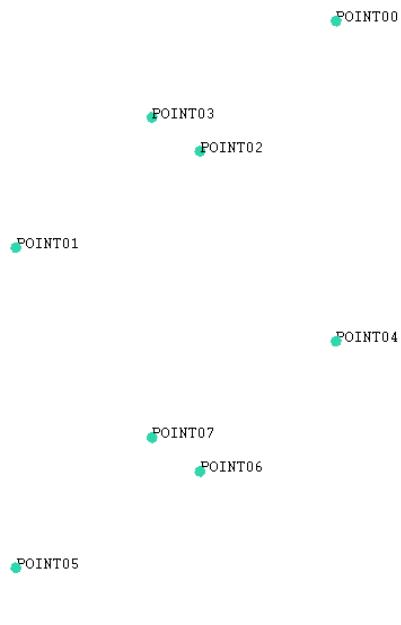
Switch on the **Points** in the Display Tree window. To see the names of the points, use the right mouse button and select **Points > Show Point names** in the Display Tree window. Select Fit Window  from main menu. Use the right mouse button to zoom out if needed. The newly created point name would be displayed as POINT00.

Similarly, create another point by entering the coordinate. The names will automatically adjust to the names shown below:

POINT01	(0,2, 12)
POINT02	(0.3, 2, 10)
POINT03	(-0.3, 2, 10)
POINT04	(0, -2, 8)
POINT05	(0, -2, 12)
POINT06	(0.3, -2, 10)
POINT07	(-0.3, -2, 10)

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

Figure 4-29-Points created so far



c) Arc Creation

Geometry > Create/Modify Curve > Arc through 3 points
 Enter the Part CURVES and the name as CURVE00. Select POINT00, POINT02 and POINT01. 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 CURVE00.

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:

CURVE01: POINT00, POINT03 and POINT01

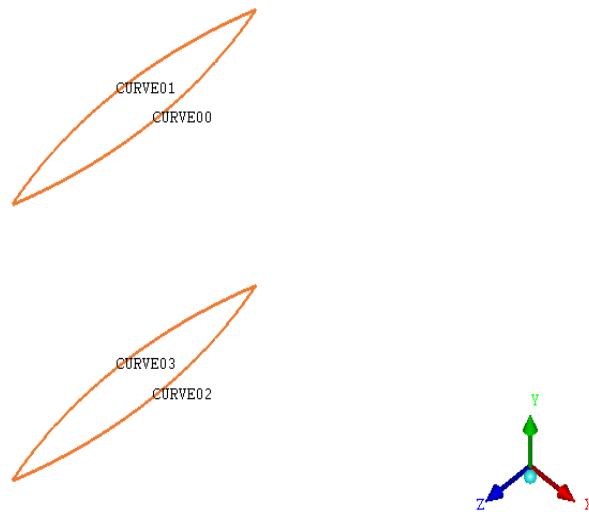
CURVE02: POINT04, POINT06 and POINT05

CURVE03: POINT04, POINT07 and POINT05

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 is shown below.

Figure 4-30 Geometry After Arc Creation



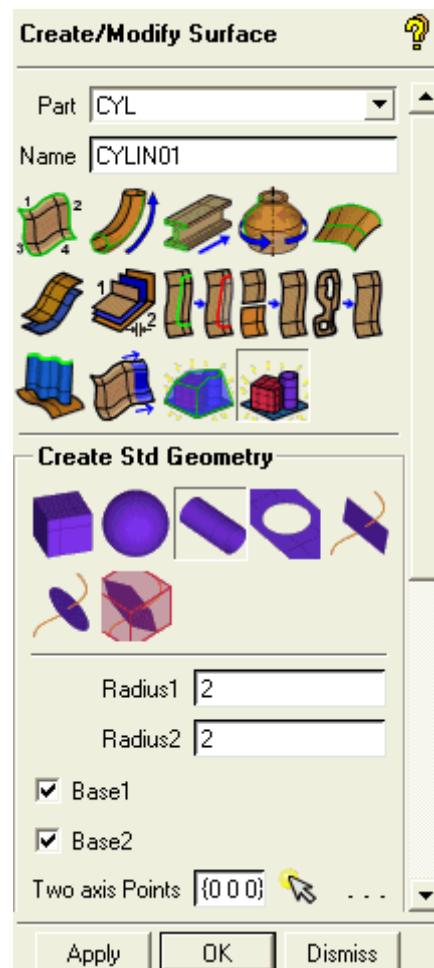
d) Cylinder Creation

Geometry > Create/Modify Surface > Standard Shapes



Cylinder: Select  (Cylinder) to open the Create Std Geometry window as shown below. Enter the Part name CYL and Name CYLIN01. Enter a Radius1 =2, Radius2 =2. For the Two axis Points, enter “{0 0 0} {0 0 20}”. Press Apply to create the cylinder. Press Dismiss to close the window.

Figure 4-31 Cylinder Creation



e) Surface Creation

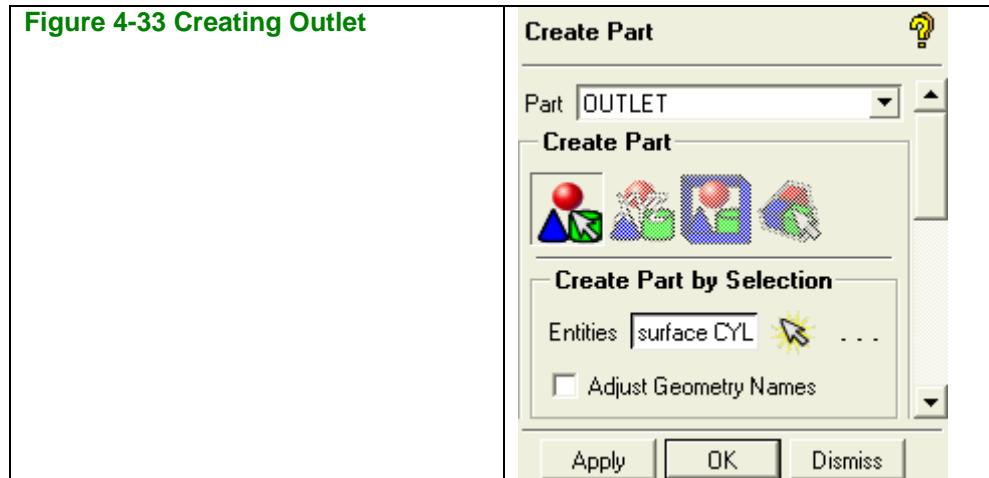
Geometry >Create/Modify Surface >Simple Surface
>Method- From 2-4 Curves> Enter the Part name as BLADE and the Name as BLADE00. Enter the **tolerance** as 0.1. Select CURVE00 and CURVE02 with the left mouse button. Press the middle mouse button to complete selection, and press **Apply** to create the surface.

Similarly, create the other blade surface BLADE01 by selecting CURVE01 and CURVE03.

To create the OUTLET surface, Go to the Display Tree and Right Click on Parts. It will open a window as shown below.

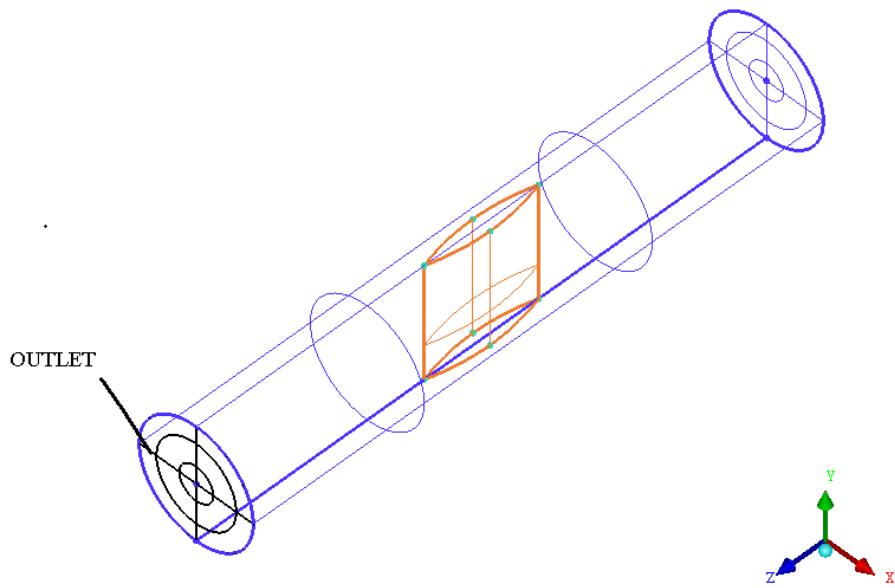


Select option Create Part .This will open another window as shown below.
Rename default part name PART.1 as OUTLET.



Select the Surfaces of Cylinder from Screen .Apply. Thus part OUTLET is created. Dismiss.

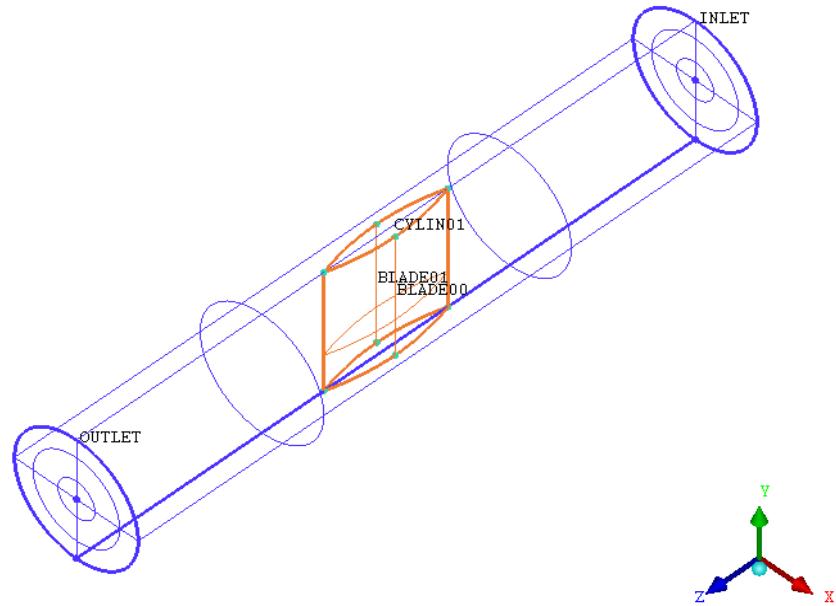
Figure 4-34-Creating Surface Outlet



Similarly select the surface opposite to OUTLET and rename it to INLET. **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 below.

Press **Dismiss** to close the window.

Figure 4-35 Geometry After Surface Creation



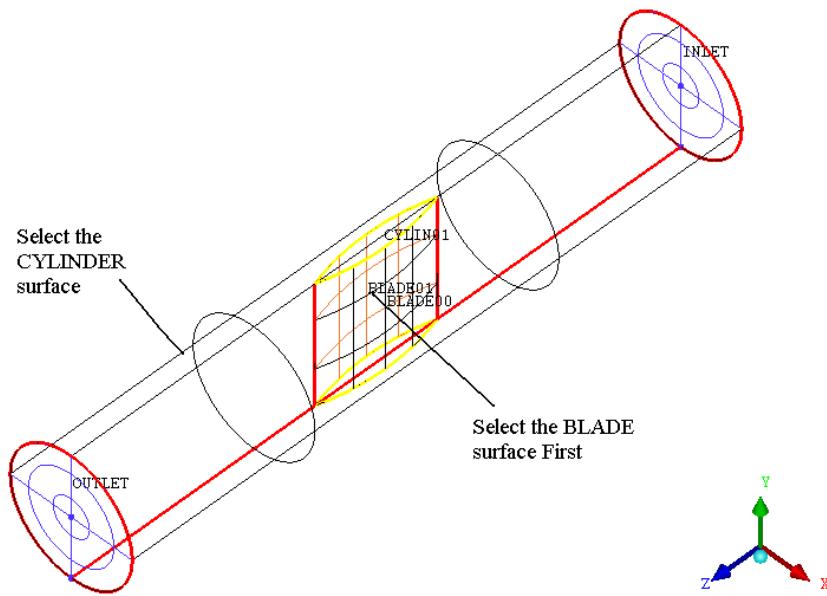
f) Surface-Surface Intersection

Geometry >Create/Modify Curve  > Surfaces-Surface Intersection

 Choose the B-Spline option. Select the two surfaces shown in the figure below. Select the BLADE00 surface for Set1 Surfaces and the CYLINDRICAL surface for Set2 Surfaces, pressing the middle mouse button each time. Select part name as CURVES. Press **Apply**. Repeat this for the other side of the blade.

Figure 4-36 First intersection curve

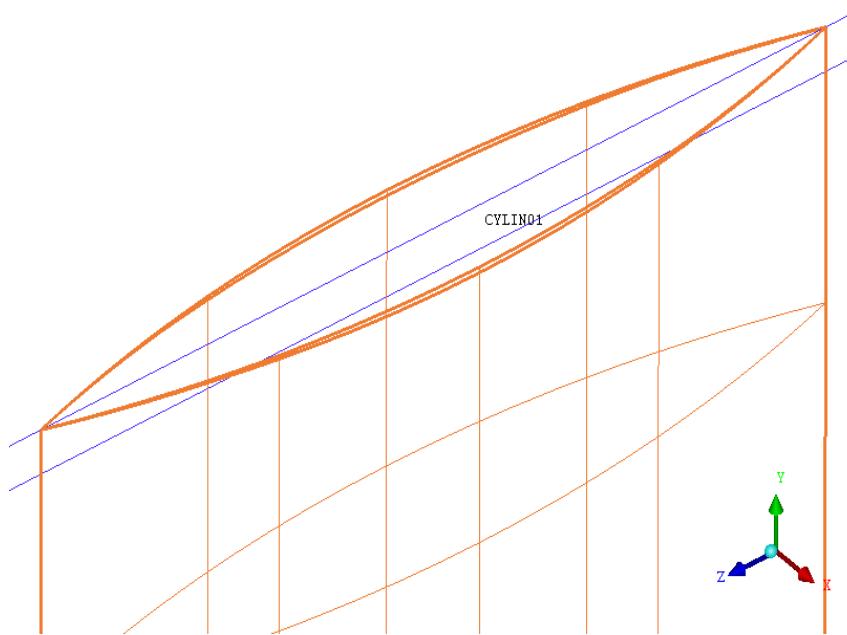
Geometry Creation



The detailed view is shown below.

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	68
------------------------	--	----

**Figure
4-37
Detailed
View**



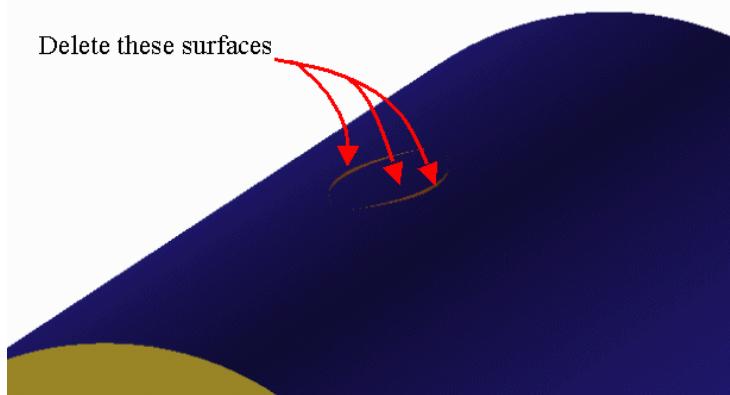
g) Build topology

Geometry > Repair Geometry **> Build Diagnostic Topology** .

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**.

h) Deleting unused entities

Geometry > Delete Surface . Delete the surfaces shown in the figure below. Repeat this for the other side of the tube.

Figure 4-38 Surfaces to delete**i) Build topology**

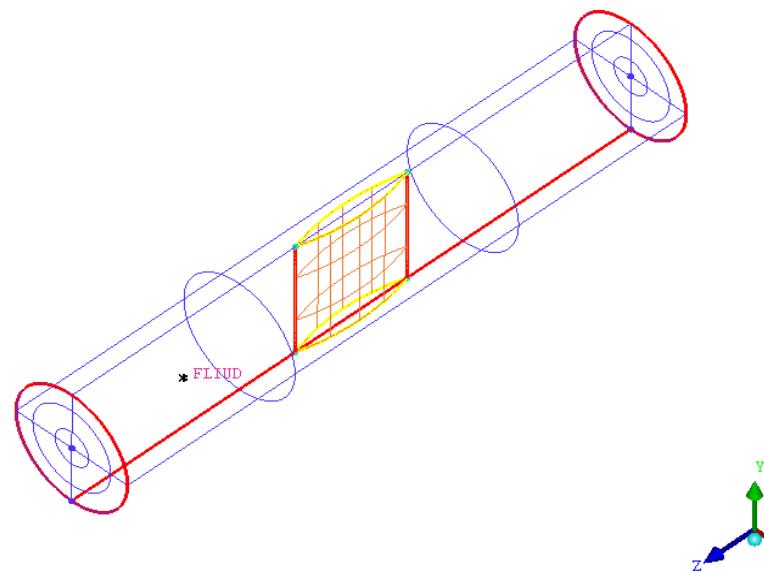
Geometry > Repair Geometry **> Build Diagnostic Topology**
Build topology once more, with same tolerance of 0.002.

j) 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 below .

Figure 4-39 Final Geometry



k) Saving geometry

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

4.1.5: Geometry Simplification using Shrinkwrap

a) Summary of steps

Importing the STL file
Creating a Shrinkwrap
Closing the Geometry
Creating Facets
Diagnostics
Creation of Geometry entities
Splitting of Sphere
Build topology

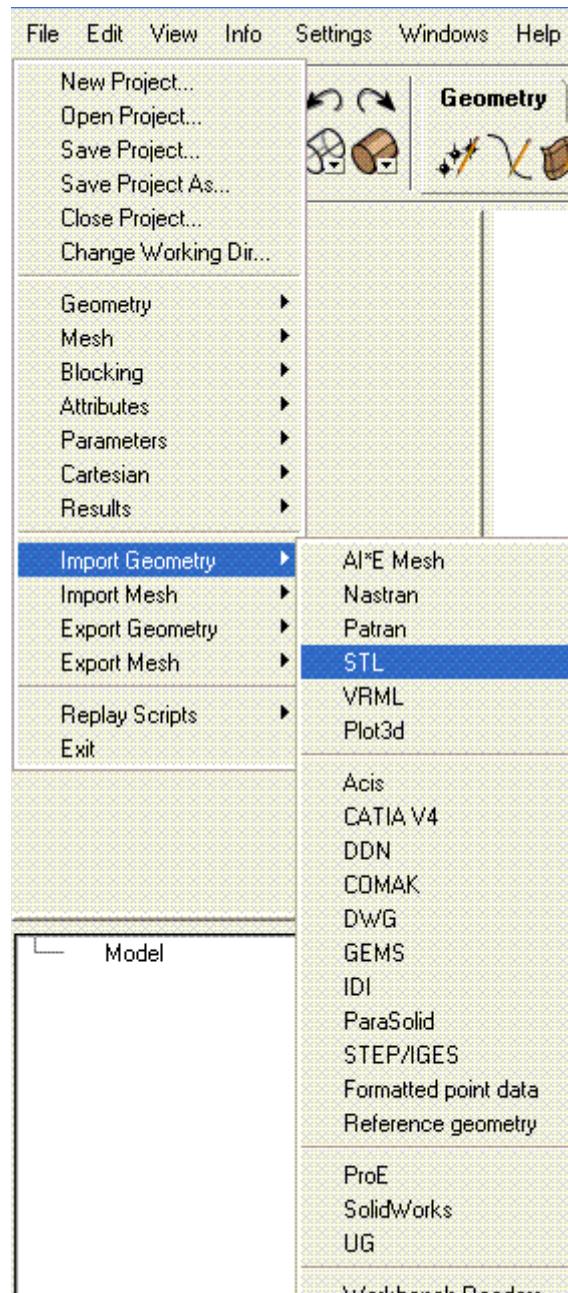
b) Starting the project

The input files for this tutorial can be found in the Ansys installation directory, under ..\v110\docu\Tutorials\CFD_Tutorial_Files\Geometry Shrinkwrap. Copy the files to your working directory and Select File > Import Geometry > STL > Select ‘eng_comp2.stl’ file as shown in the figure below and press Apply.

Figure 4-40 Import Geometry from STL

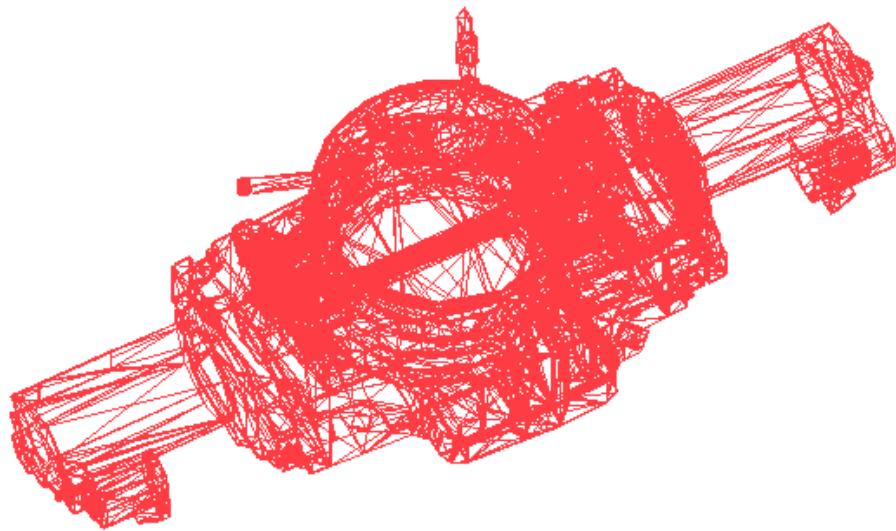
ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	72
------------------------	--	----

Geometry Creation



Geometry gets loaded on the screen. Right mouse select Geometry > Surfaces in the Display tree and select Show Full to see the full triangulation of the surfaces.

Figure 4-41 Geometry after Loading



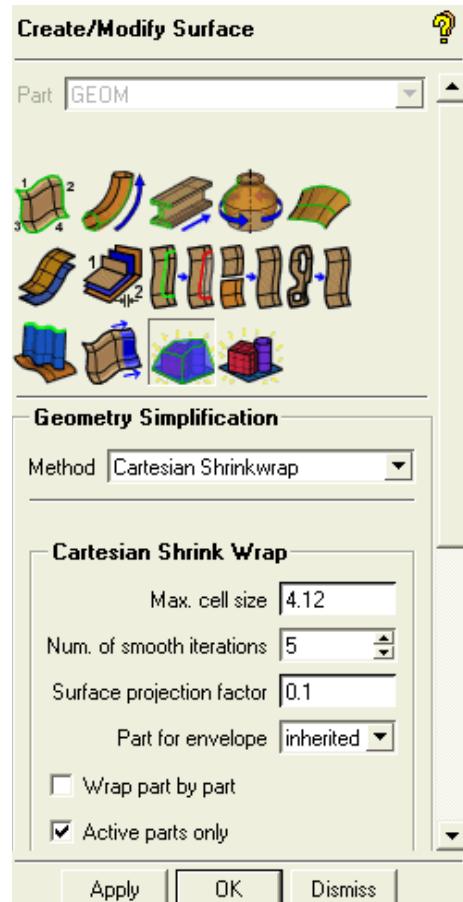
c) Geometry Simplification by Creating a Shrinkwrap

Go to Settings>Geometry Option > Inherit Part Name and toggle ON option Inherited.

Select Create/Modify Surfaces >Geometry Simplification > Cartesian Shrinkwrap This will open up the window shown below.

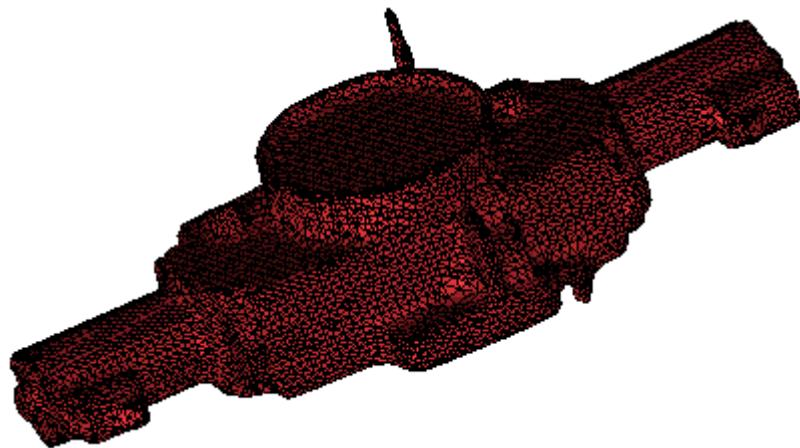
Figure 4-42 Cartesian Shrinkwrap

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	74
------------------------	--	----



Enter Max Cell Size as 4.12, Number of smooth iterations as 5, Surface projection factor as 0.1, Part for envelope as Inherited. Turn ON ‘Active Parts Only’ tab and Apply. A Shrinkwrap will be created as shown and will be listed in the Mesh Display Tree.

Figure 4-43 After creating Shrinkwrap



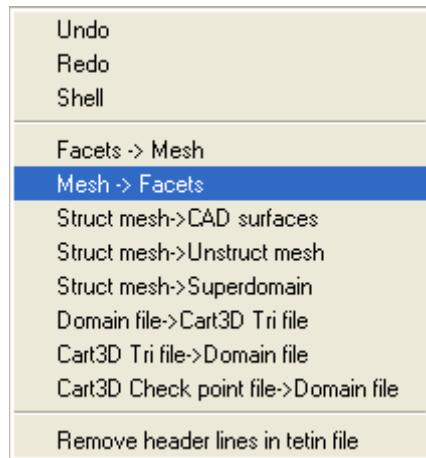
d) Closing the geometry-

Select File>Geometry>Close Geometry. This will close all the geometry entities and only Mesh will remain on the Screen.

e) Creating Facets

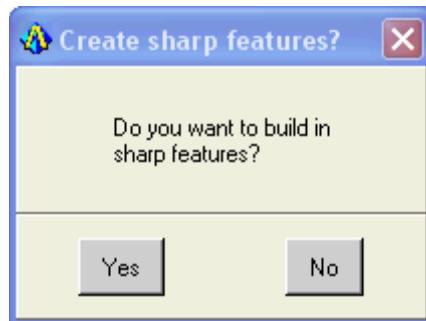
Select Edit in Main menu and then select Mesh ->Facets.

Figure 4-44 Converting Mesh to Facets



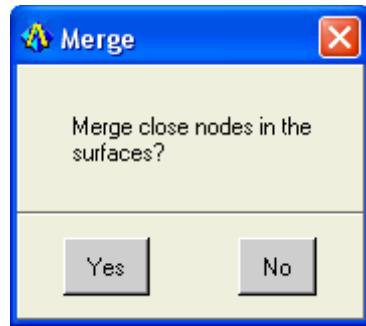
A window will pop up as shown below.

Figure 4-45 Sharp Features



Select 'NO' to build sharp features.

Another window will open as below

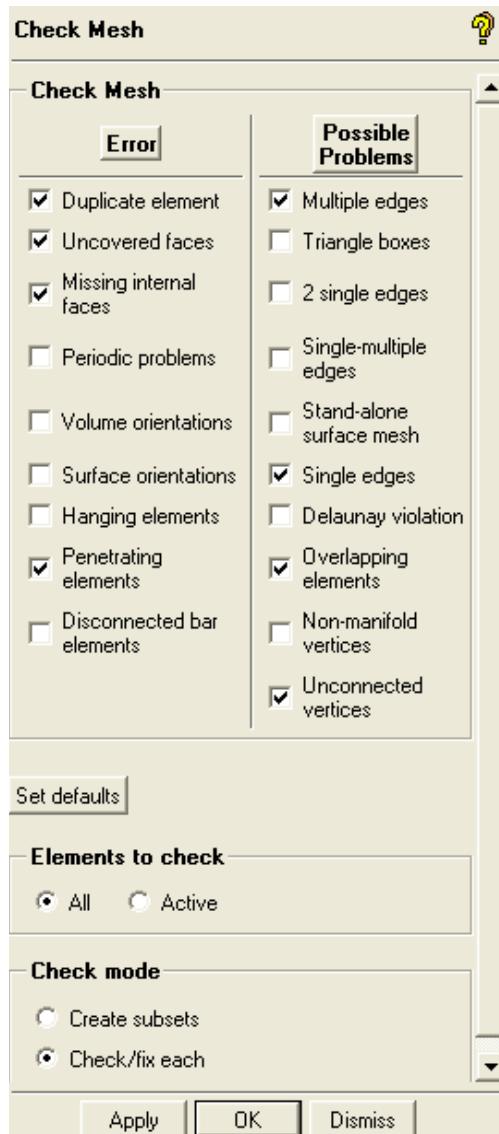


Select Yes.

The Shrinkwrap Mesh is created and listed in the Mesh Display Tree.

f) Checking the Mesh

Select Check Mesh . The Check Mesh window will open as shown below. Toggle ON ‘Penetrating Elements’ in Error List. Press Apply.

Figure 4-46 Check Mesh

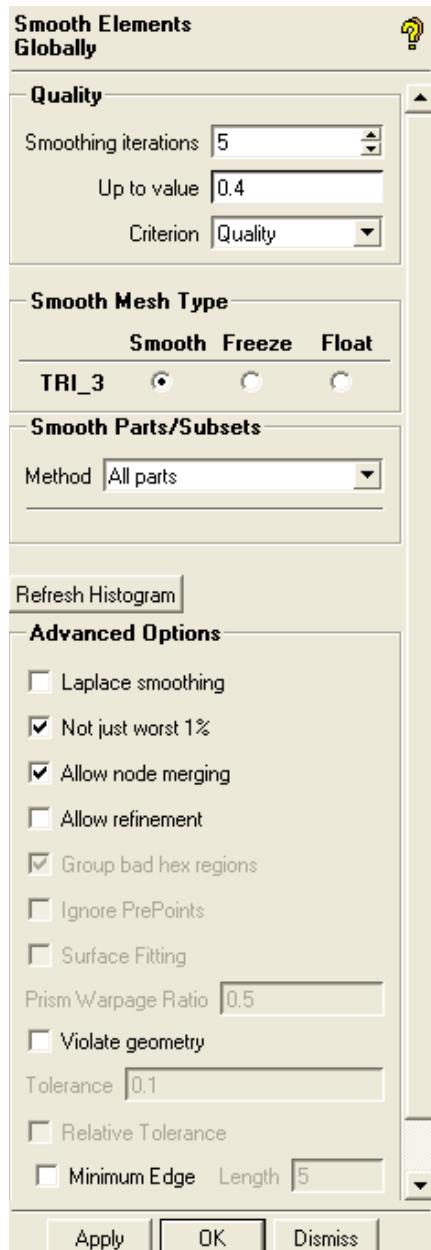
There should be no errors in the mesh.

g) Checking Quality of Mesh

We need to check the mesh for any errors or problems that may cause problems for analysis. Select Display Mesh Quality  . Keep the default settings and APPLY.

h) Smooth Elements Globally

Several elements have a lower quality so we need to smooth some elements so as to improve the Quality of mesh. Select Edit Mesh > Smooth Mesh Globally .

Figure 4-47 Smooth elements globally

Change the 'Up to value' to '0.4' and Toggle ON the options "Not just worst 1%" and "Allow node merging" and press Apply. The Quality reached can be seen in a Histogram that will appear on the lower right corner of the screen.

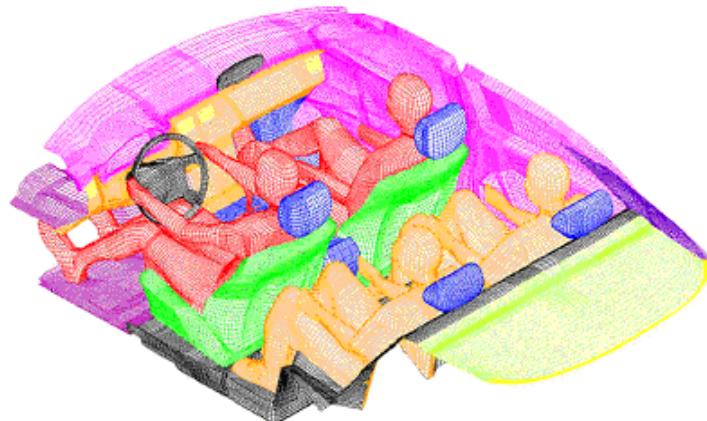
i) Saving geometry

File > Geometry >Save Project As: Enter the project name as **Shinkwrap** and press **Save** to save the geometry file

4.2: Hexa Meshing

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

Figure 4-48
This mesh for the
Mercedes SLK
(model courtesy
of Daimler-
Chrysler) was
generated with
ICEM CFD Hexa
combining the
exterior and
cabin flow



4.2.1: Introduction

ANSYS ICEM CFD 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 ICEM CFD 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 ICEM CFD: 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, scaling and translate and rotate 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 ICEM CFD 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 ICEM CFD Blocking

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

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	83
------------------------	--	----

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 faceted 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 whether the current blocking strategy is adequate or not.

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

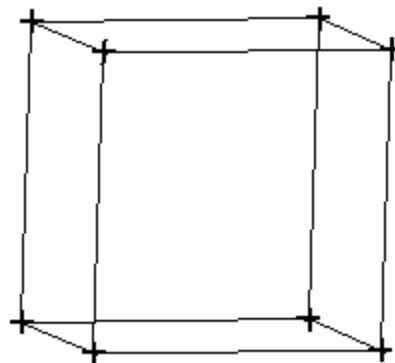
may be visually blanked/unblanked by using the Index control which toggles the blocks in I, J, K or radial (if o-grids exist) directions. Any new split will create a new I, J, K or radial (if splitting an o-grid) index.

O-grid Creation

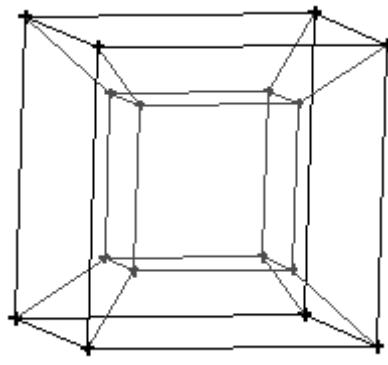
Subdivides selected blocks into a configuration of one central block surrounded by radial blocks. Accessed through Blocking > Split Block > O-grid Block. Recommended for cylindrical type geometries to avoid bad internal angles at block corners.

Figure 4-49
OGrid Block

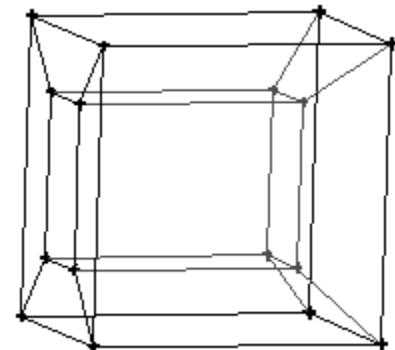
The initial block



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. The last diagram 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 surface 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.

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	88
------------------------	--	----

All vertices can also be constrained by fixing x, y 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. 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 3D 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, convert block type 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

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	90
------------------------	--	----

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

Pre-Mesh

Topology

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 option 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

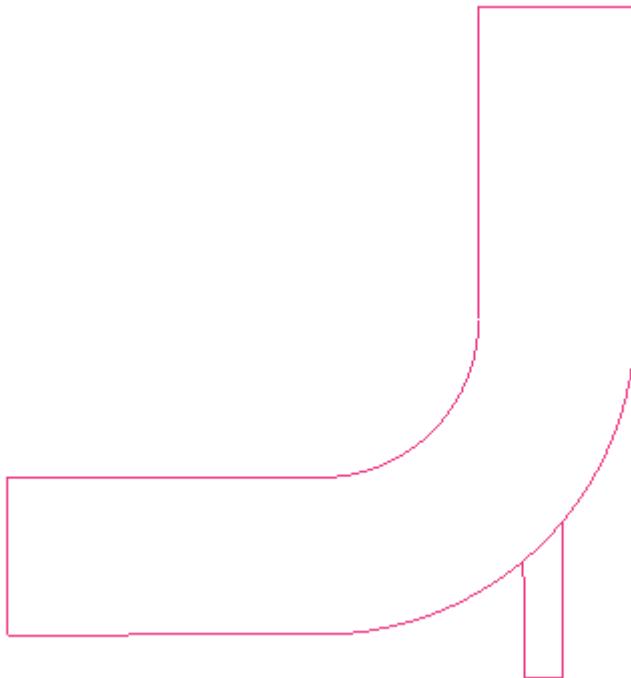
ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	92
------------------------	--	----

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.

4.2.2: 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

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	94
------------------------	--	----

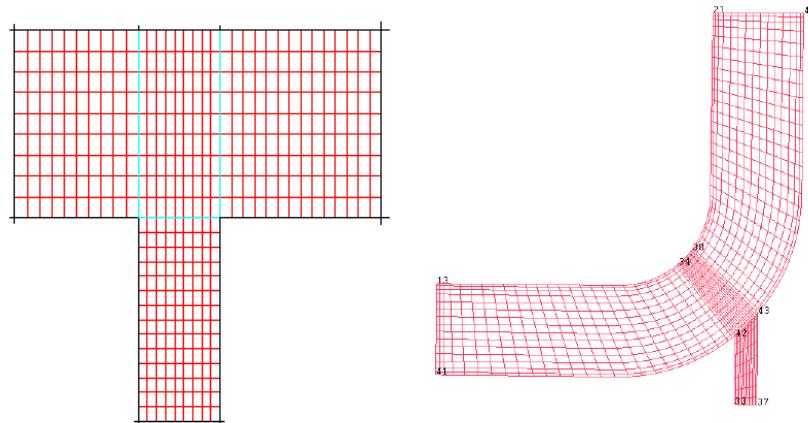
- Creating Composite Curves
- Projecting the Edges to curves
- Moving the Vertices
- Applying mesh parameters
- Refining the Mesh with Edge Meshing
- Saving the Blocking and Mesh before Quitting

b) The Blocking Strategy

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

Note: The geometry is equivalent to a “T” Shape. The right side of the blocking crossbar needs only to be bent upward to resemble the geometry.

Figure 4-50
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

The input files for this tutorial can be found in the Ansys installation directory, under

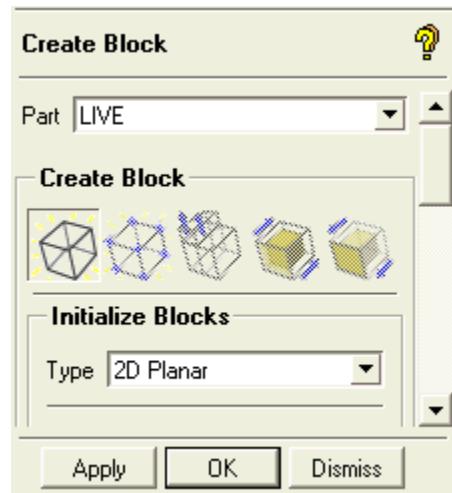
`../v110/docu/Tutorials/CFD_Tutorial_Files/2DPipeJunction` directory.

Copy and open the `geometry.tin` file in your working directory.

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

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

Figure 4-51
The Create Block Menu

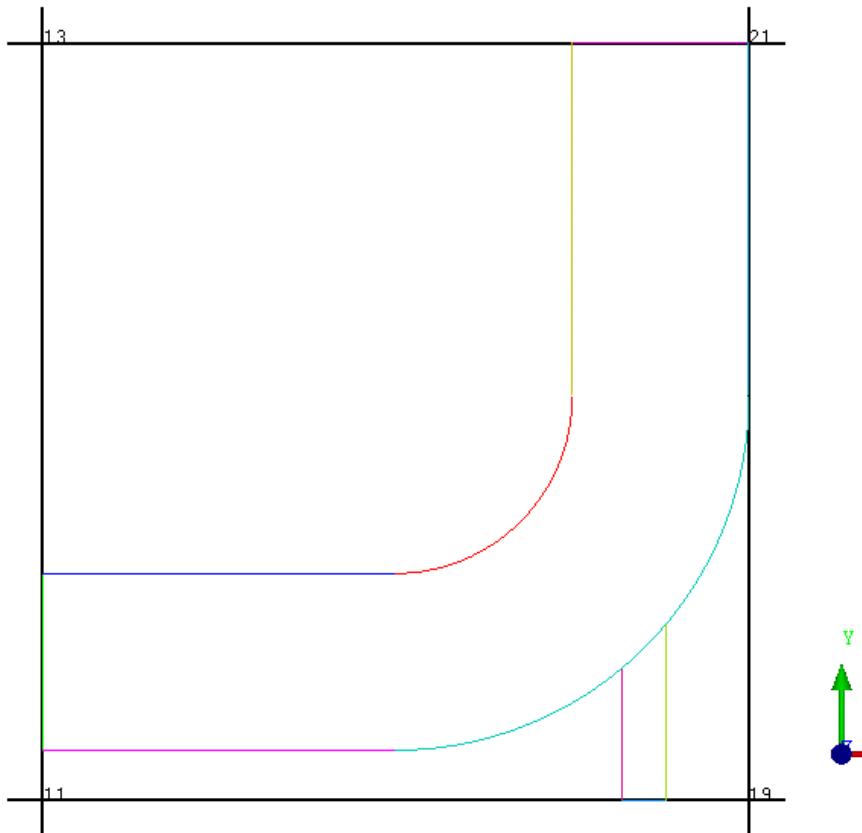


Turn on Vertices in the model tree. Then, right mouse select Vertices > Numbers. The following operations will refer to these numbers.

Note the white block that encloses the geometry, as shown. 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
4-52
Initial
LIVE
block**

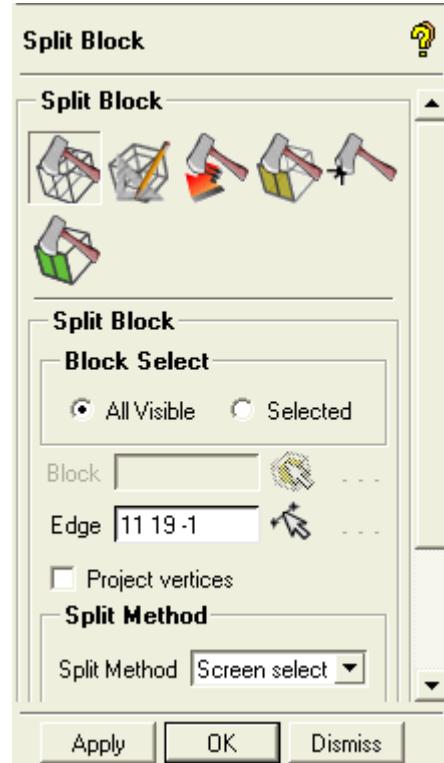


d) Block Splitting

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

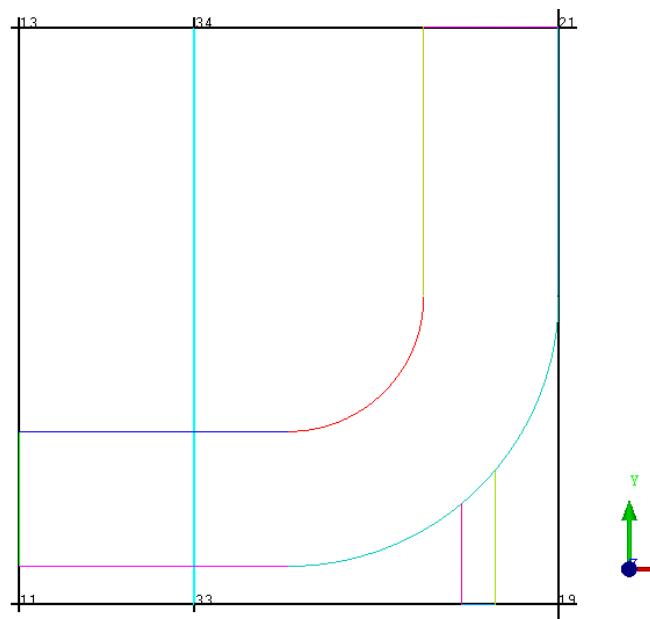
Select Blocking > Split Block > Split Block . Note the Split Method is set to Screen Select by default. In this case, the split may be done by approximation, as it is only the topology of the “T” shape that is essential, not the exact proportion.

Figure 4-53
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 the figure below. 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 below.

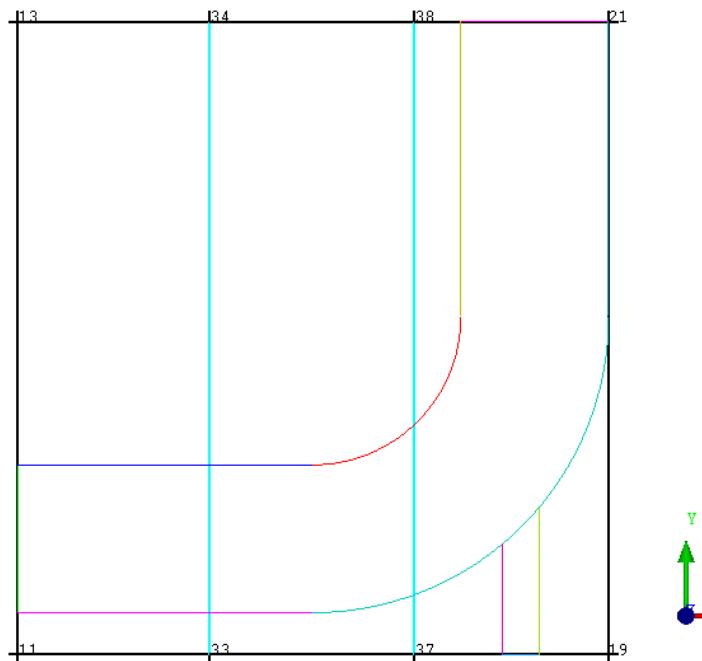
Figure 4-54
First Split Edge 11-19



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.

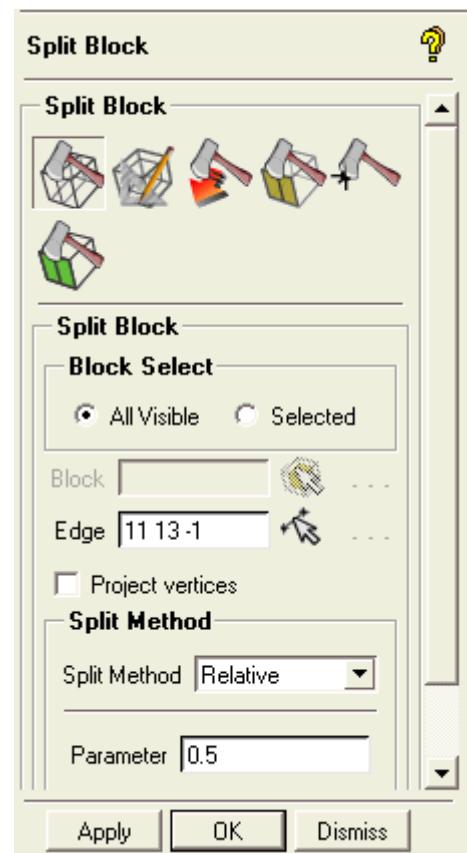
Repeat for edge 33 and 19 (or 34, 21). The results are shown below.

Figure 4-55
Second Split
Edge 33-19



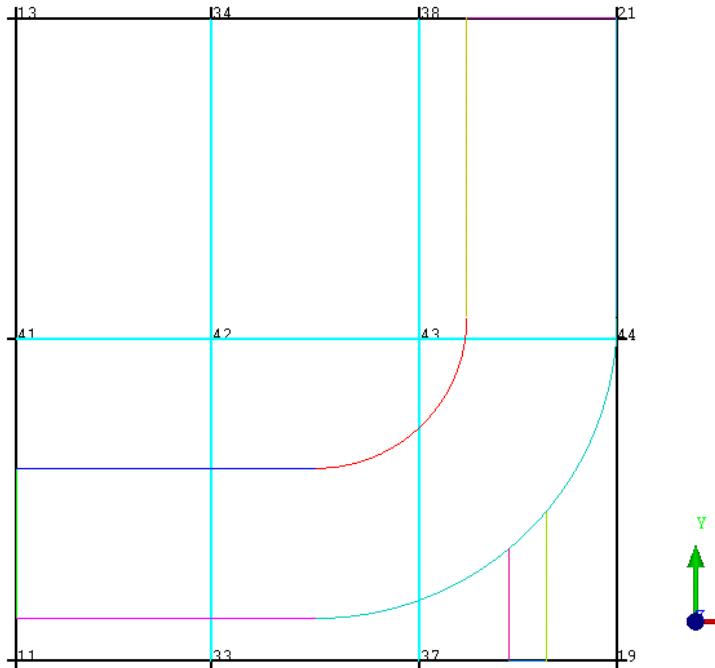
Create the horizontal split, this time changing Split Method to Relative. Enter 0.5 (mid-point of selected edge), select any one of the four vertical edges and press the middle mouse button or Apply.

Figure 4-56
Split Method Relative



This horizontal split is shown below.

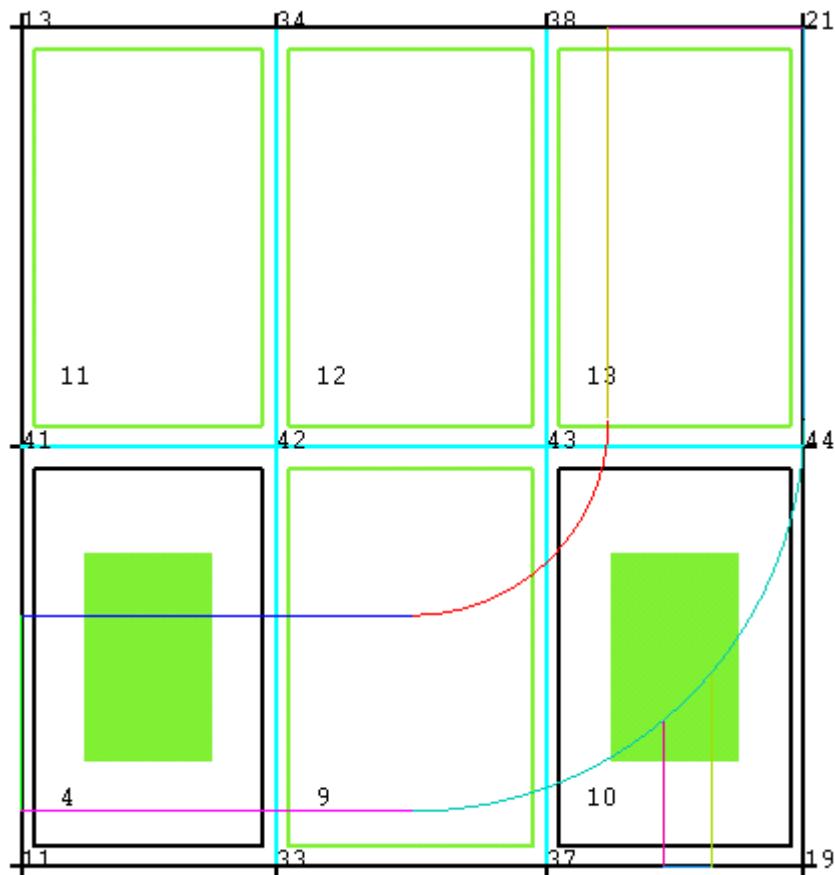
Figure 4-57
Display of the
curves and
LIVE block
after making
three splits



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 below and press the middle mouse button or Apply.

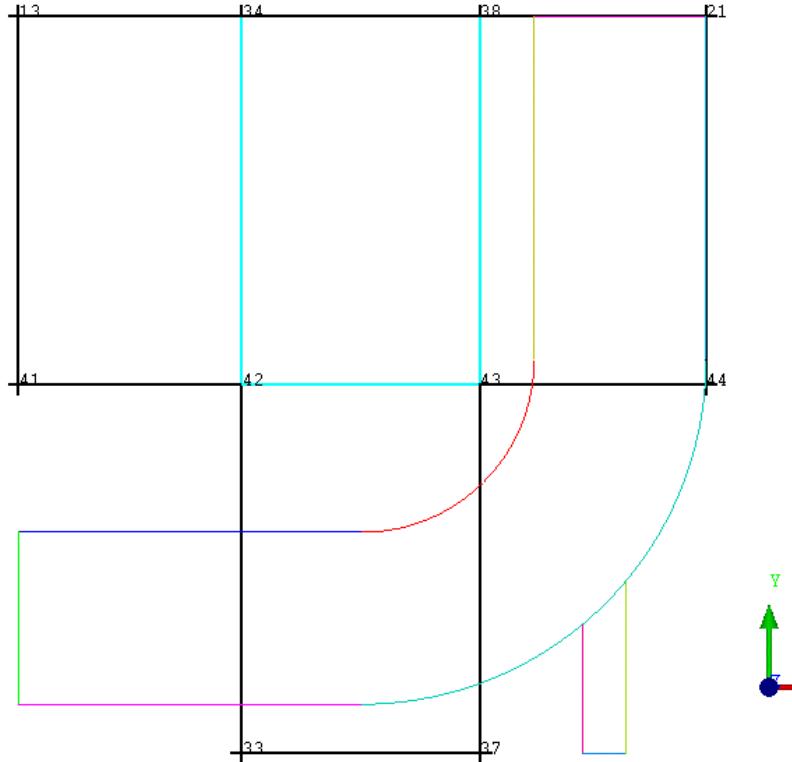
**Figure
4-58
Select
blocks
to
delete**



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 the figure below.

**Figure
4-59
Final T
Shape
Topology**

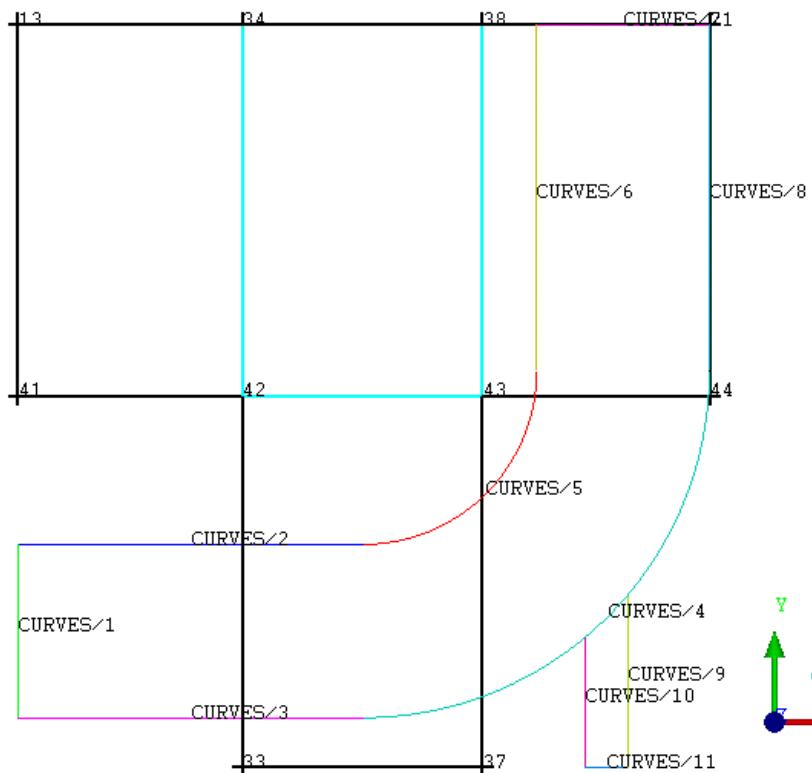


f) Associating to Geometry

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).

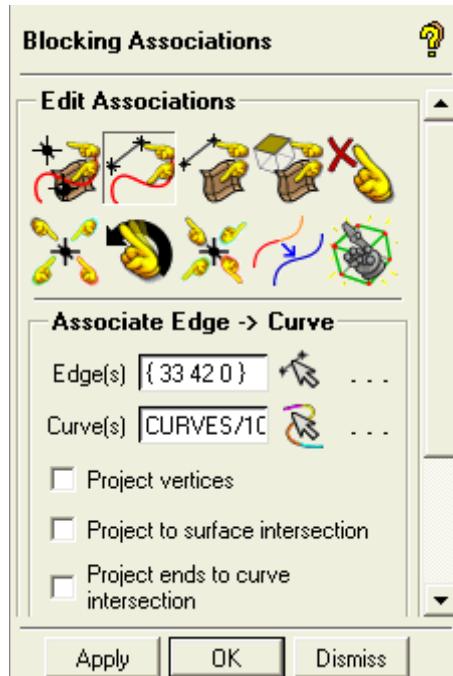
For reference turn on Curves > Show Curve Names in the Display tree.

**Figure
4-60
Vertex
numbers
and
Curve
names**



Select Blocking > Associate > Associate Edge to Curve as shown below.

Figure 4-61
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.

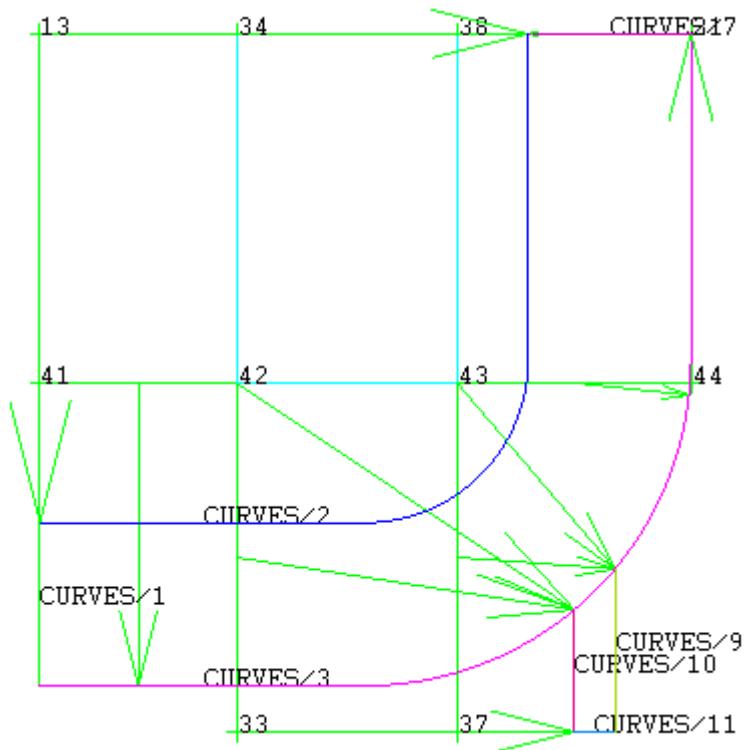
Note: It may help to toggle entity types off and back on to identify the right entity, if they overlap other entities. For example, turn off Vertices and Edges to verify the curve names. Turn Edges back on to proceed with the selection.

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. Similarly associate 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 shown, 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
4-62
Projection
of edges
to Curve**



Note: If, once completed, the associations do not appear correctly, 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.

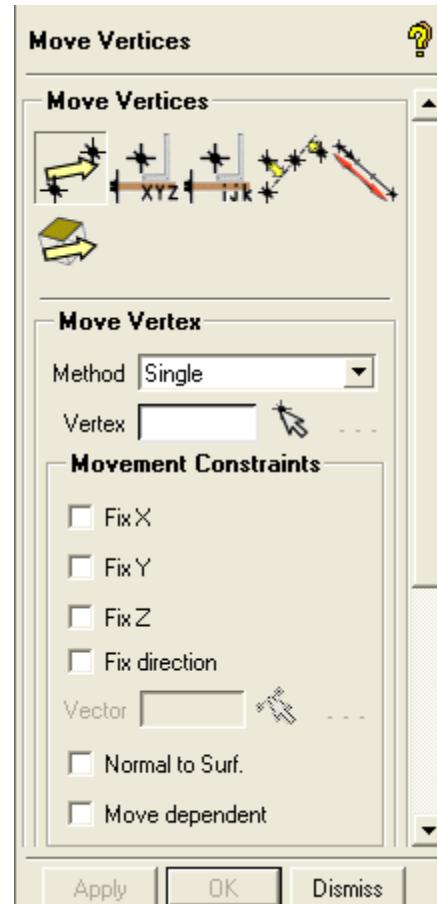
g) Moving the Vertices

Next, move vertices on to the geometry. Select Move Vertex  >

Move Vertex  (if not already selected).

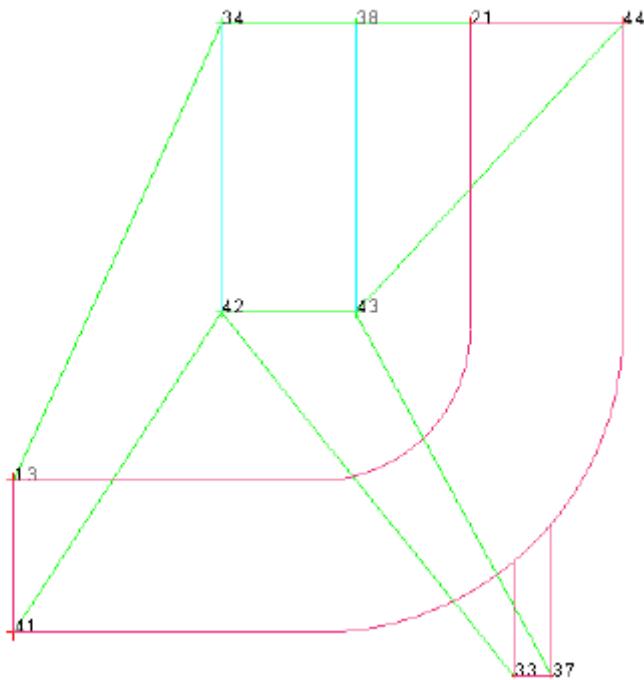
Note: Selecting Move Vertex from the Menu tab will immediately prompt you to select from the screen. It is usually not necessary to select Move Vertex from the **Data Entry Panel** unless another option was previously selected.

Figure 4-63
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 4-64
**Associate
Edges to
Curve**

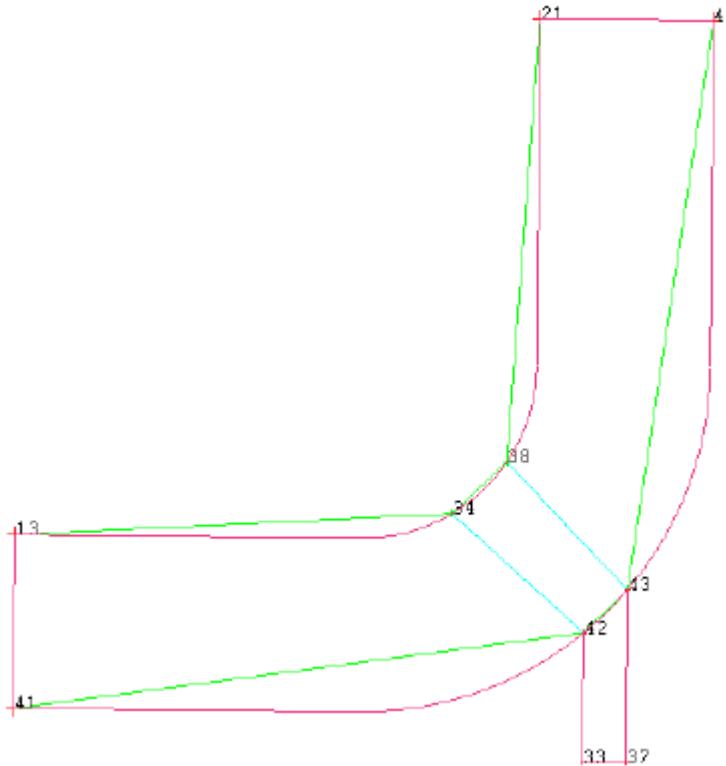


Note: Due to the associations made between the edges and curves, many of these vertices will “snap” to the correct position. Vertices may, however, be moved along the curve by dragging the mouse. To capture the ends of the curves, drag with the left mouse key depressed until the vertex can be moved no 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 the diagram below. Try to make the blocks as orthogonal (good internal angles) as possible. Toggle on Points > Show Point Names in the Display tree. Go to Blocking > Associate > Associate Vertex. By default the point option is toggle on. Select the vertex and select the point to which you want to associate the vertex. Associate the

vertices 13, 21, 41, 42, 33, 37, 43, 44 to points POINTS/2, POINTS/5, POINTS/1, POINTS/10, PONTS/9, POINTS/8, POINTS/11, POINTS/6. Toggle off Points > Show Point Names in the Display tree.

**Figure
4-65
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 > Open blocking.

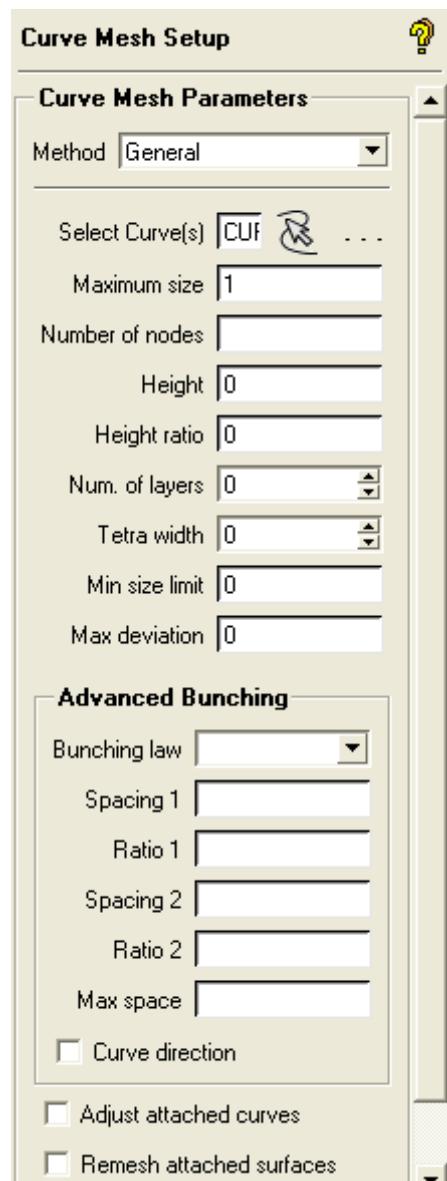
h) Applying mesh parameters

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

Select Mesh > Set Curve Mesh Setup  . Select the visible Curves (can select “v” for visible or “a” for all or select the appropriate icons from the selection tool bar).

Set Maximum Size to 1. Ignore all other parameters and press Apply.

Figure 4-66
Curve Mesh Parameter
Window



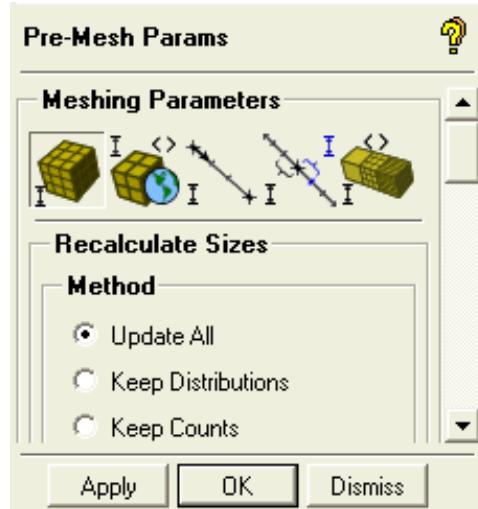
Note: Maximum Size determines the length of the edges on the curve (or surface for 3D). Height determines the length of the edge of the first layer normal to the

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.

i) Initial Mesh Generation:

Select Blocking > Pre-mesh Params  > Update Sizes .

Figure 4-67
Pre Mesh Param Window



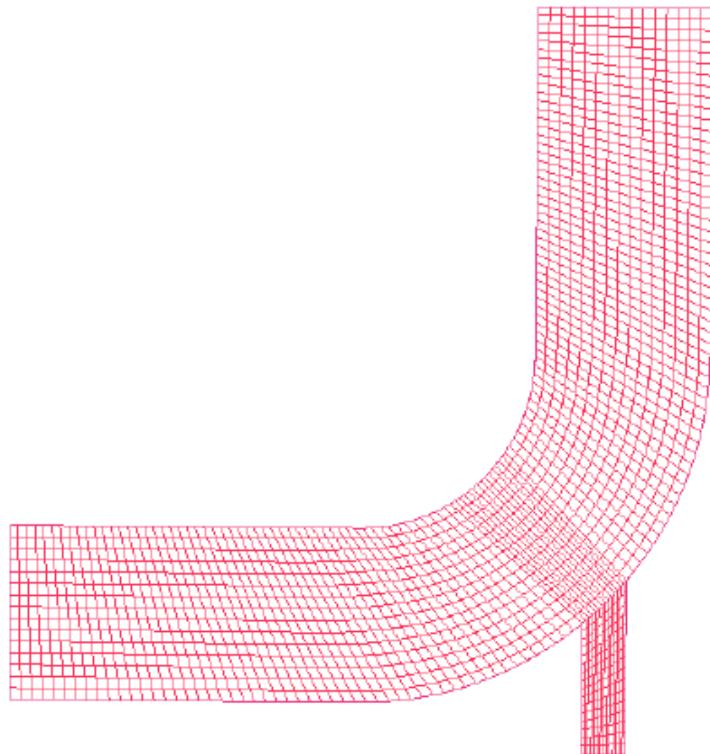
Toggle on Update All (default) and Press Apply.

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.

Figure 4-68
The initial
mesh



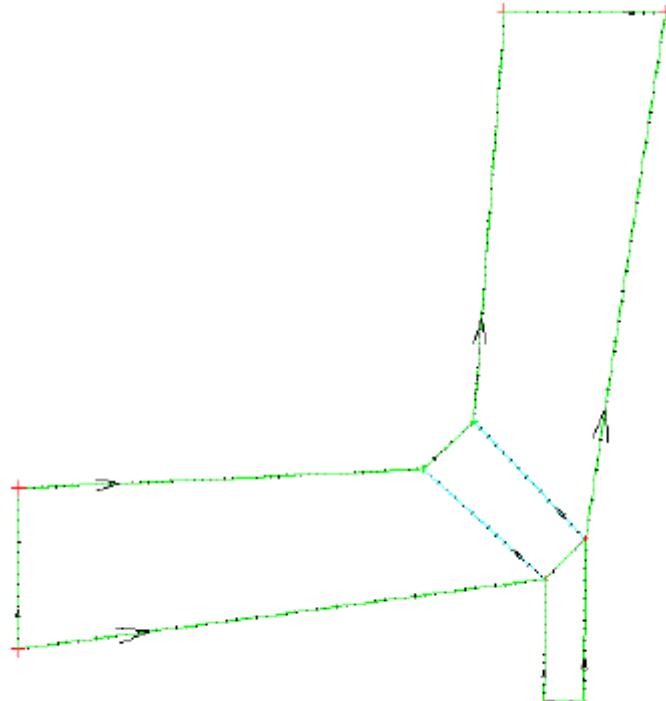
j) Refining the Mesh with Edge Meshing

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.

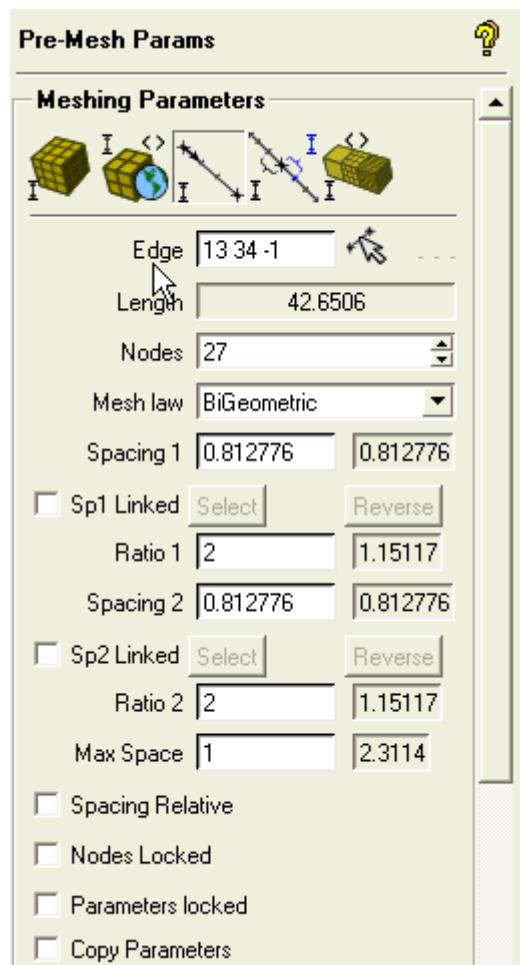
First, we'll reduce the number of nodes along the length of the large pipe.

Figure 4-69
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. Select again or and select edge 13-34 when prompted. In the panel, change the number of Nodes to 27 then Apply.

Figure 4-70
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 13-34 and 38-21 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



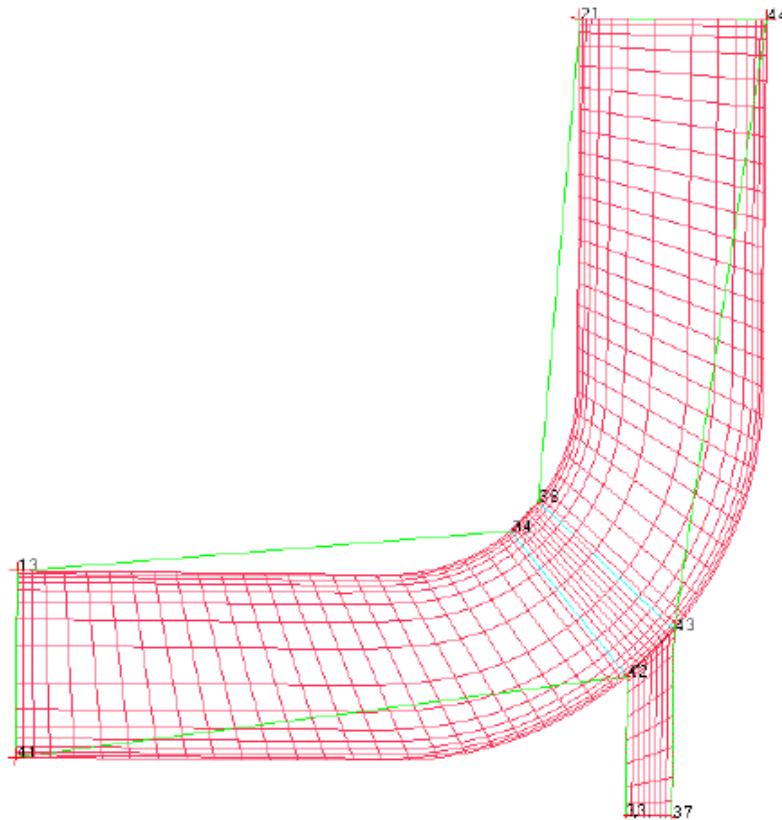
> Method > To Selected Edges Reversed. Select the Select edge(s) icon immediately underneath the Method field and select Edge 13-34. Press the middle mouse button or Apply.

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 below.

**Figure
4-71
The
Final
Refined
Mesh**



k) 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.

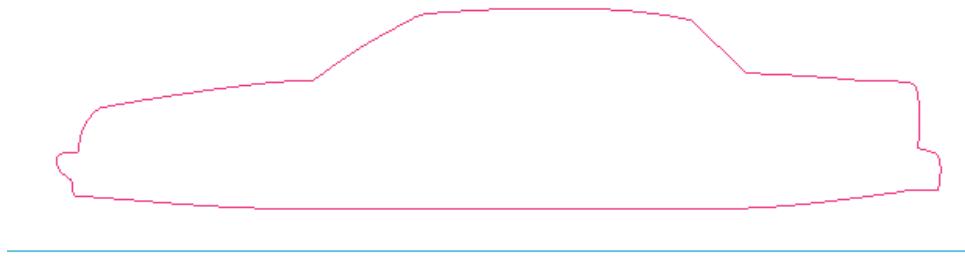
Select File > Save Project As... and type in a project name. All files: tetin, blocking and unstructured mesh will be saved.

File > Exit or continue with the next tutorial.

4.2.3: 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:

CAR: Vehicle geometry

GROUND: Ground surface of the wind tunnel

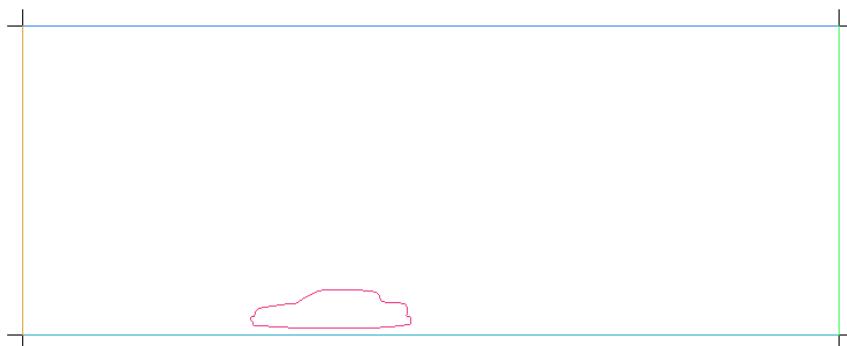
INLET: Inlet face of the wind tunnel

OUTLET: Outlet face of the wind tunnel

PNTS: Prescribed points associated with the Car.

TOP: Top surface of the wind tunnel.

**Figure
4-72
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.

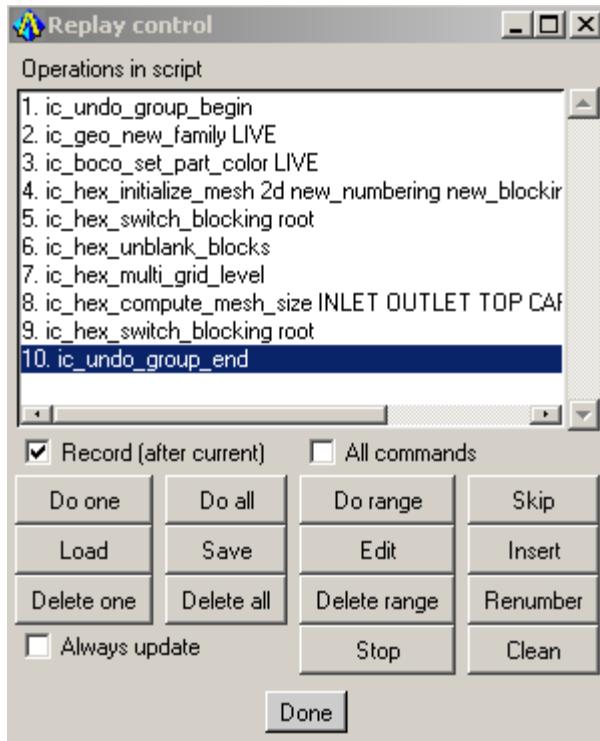
c) Starting the Project

The input files for this tutorial can be found in the Ansys installation directory, under ..\v110\docu\Tutorials\CFD_Tutorial_Files\2DCar. Copy and open the car_base.tin file in your working directory.

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
4-73
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.

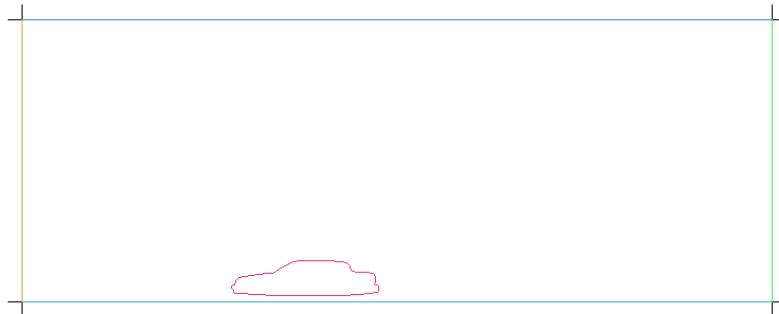
Select Blocking > Create Block  > Initialize Block  > Type 2D planar.

Name the Part as LIVE and press Apply.

Figure

4-74

**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 shown above.

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

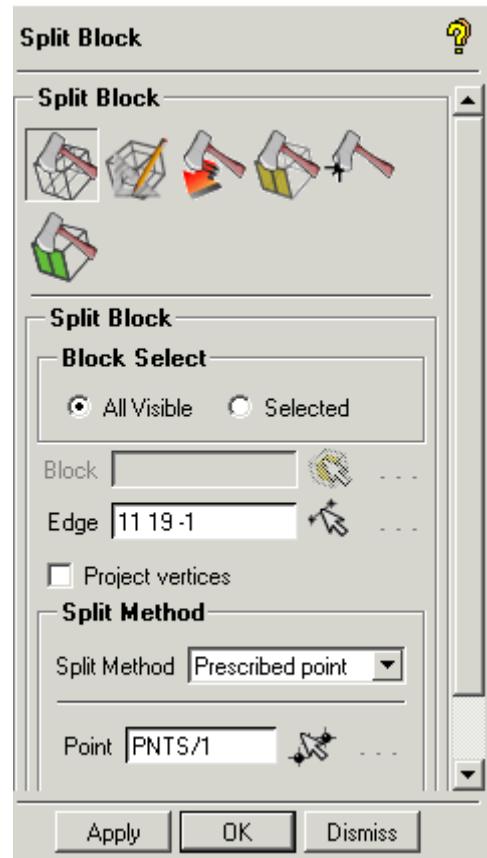
Select Blocking > Split Block  > Split Block  in the Data Entry Panel will be active by default. Don't select at this time.

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.

Figure 4-75
Split Block Window



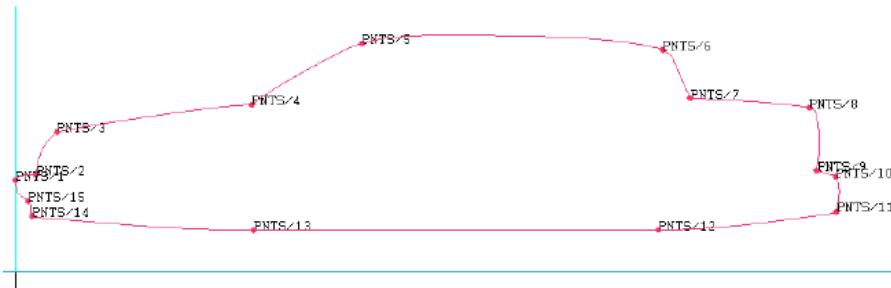


Now select either Split Block or Select Edge(s) and select any horizontal edge (top or bottom edge). Then select (PNTS/1) at the front of the bumper. The new edge will automatically be created as shown below.

Note how the new edge intersects the point.

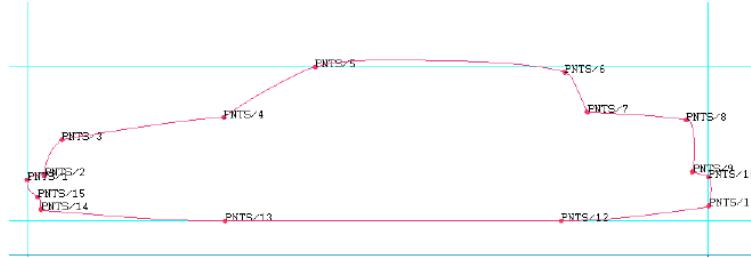
Note: At any point in time while in selection mode, you can toggle on dynamic mode by selecting F9. This may be necessary in order to zoom in to get a closer view of the points. Toggling F9 again will return to selection mode.

**Figure
4-76
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 shown below.

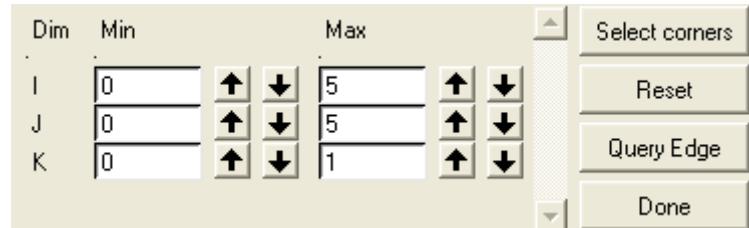
**Figure 4-77
Additional
Split**



e) Overview of the Index Control

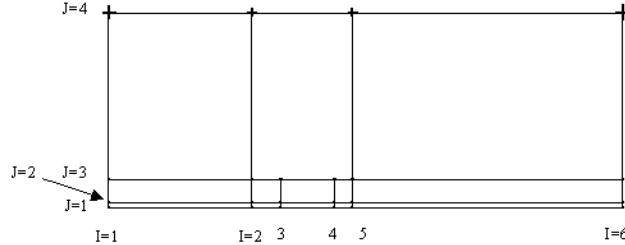
By default, splits only propagate through the displayed blocks. Blocks can be blanked by right mouse selecting Blocking (in the Display tree) > Index control which will appear in the lower right hand corner.

**Figure 4-78
Index Control**



All block edges and vertices are assigned an I, J, K value. For example, in the figure below, 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 4-79
Blocking Indices**



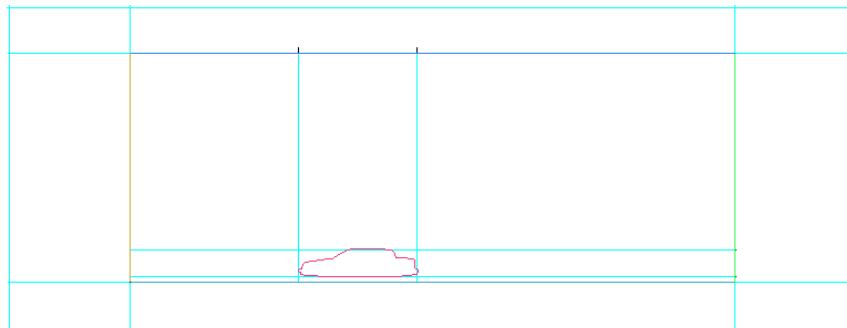
The Index control panel has two columns, Min and Max (left and right columns respectively). The range can be changed by toggling the arrows or entering an integer value in the appropriate field. Only the blocks within this range are displayed. Selecting Reset will turn all of the block indices back ‘on’.

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 (see figure below) 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.

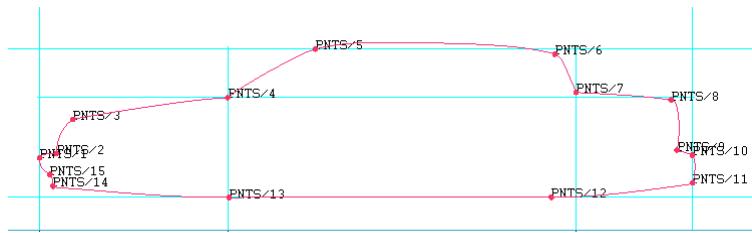
**Figure
4-80
VORFN
Blocks**



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 below. Note how these new splits don't propagate through all of the blocks.

Figure 4-81
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 shown below.

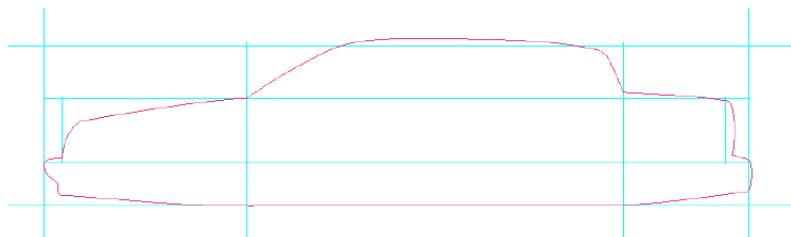
Figure
4-82
The
Mid
Block



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 shown below.

Figure
4-83
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.

Turn on Blocks under Blocking in the Display tree. This will show the blocks by their numbers.

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

Save blocking

**Figure
4-84
OGrid
Block**



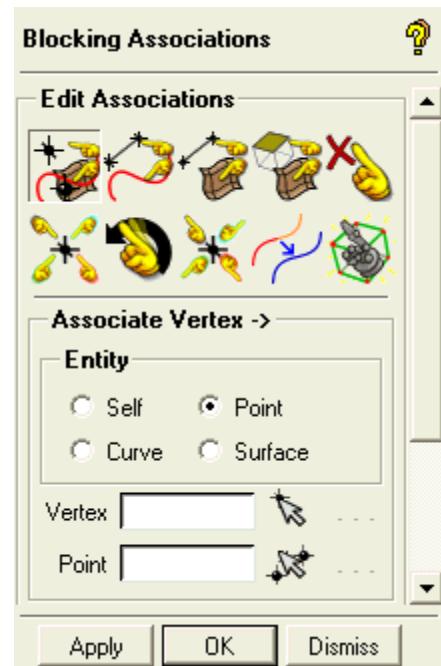
h) Body Fitting the Blocking

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. Also Turn on vertices under Blocking in the Display tree. Right click on the vertices in the Display tree and click on Numbers. This will show all the vertices by their numbers.

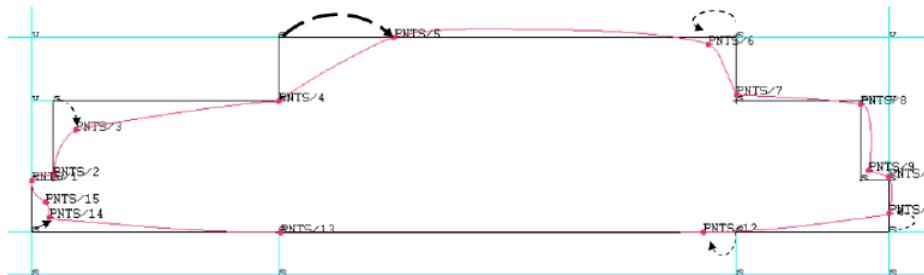
Select Blocking > Associate  > Associate Vertex  as shown below. The Entity type Point is toggled on by default.

Figure 4-85
Associate Vertex Panel



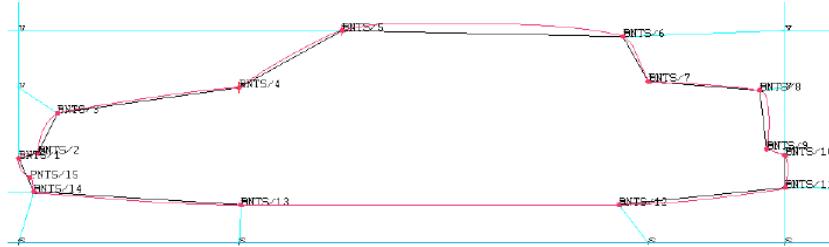
Select Associate Vertex again or the Select vert(s) icon and first select the vertex and then the appropriate point as shown below. The vertex will immediately jump to the selected point. Make sure you associate the vertices that are right on top of their respective points (e.g. PNTS/4).

Figure 4-86
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
4-87
The
Blocking
fit to the
Car**



i) Edge-Curve Association.

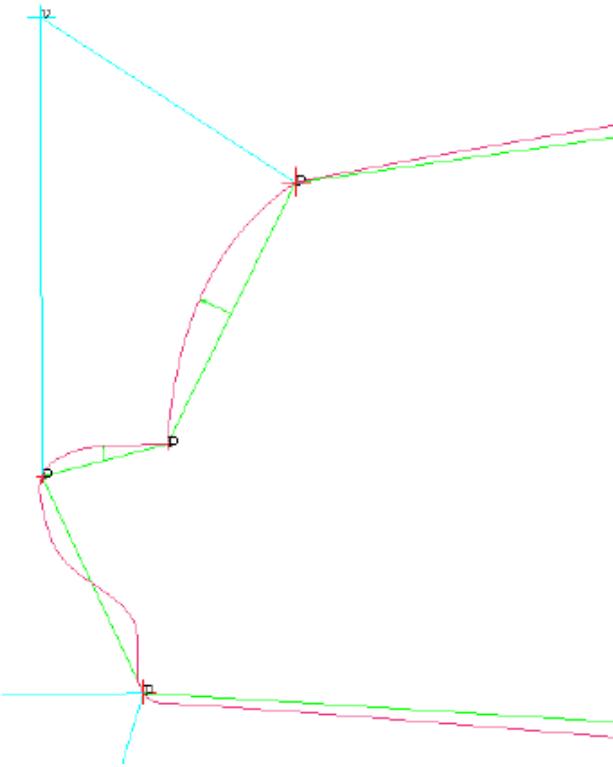
Turn Points off in the Display tree. Turn off the internal edges: right mouse select Edges and toggle off Internal Edges. Turn off all outer edges: Set Index control to I:2-6, J:2-5.

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 > Internal Edges 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 the figure below.

Figure 4-88
Display of Edge Projection



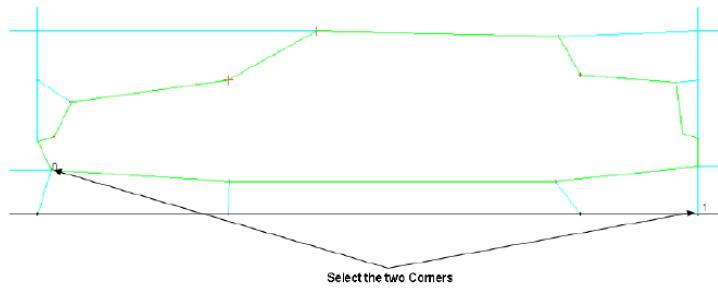
j) Aligning the Vertices

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 the figure below. Note the change in the I, J ranges within the Index control panel.

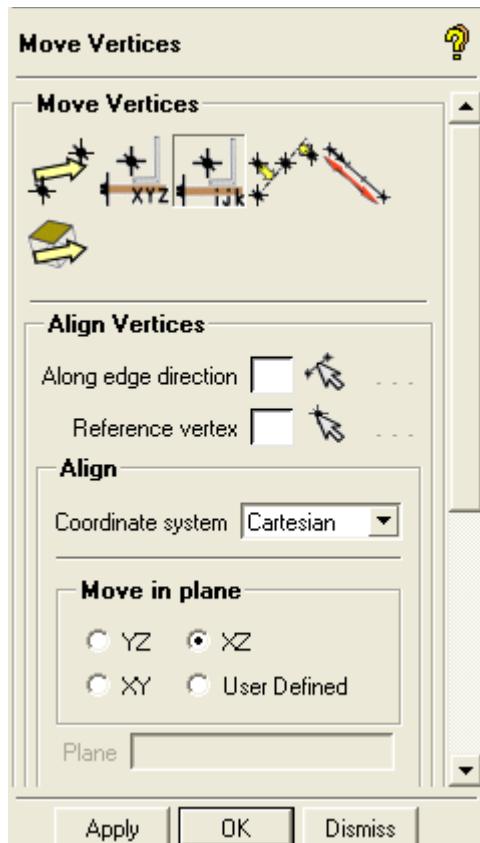
Figure 4-89
Adjusting the
index control
using From
Corners



Turn on Vertices > Indices in the Display tree for reference.

Select Blocking > Move Vertex > Align Vertices .

Figure 4-90
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 the figure below and Apply.

**Figure
4-91
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

 > Set location  and select a reference point, PNTS/3, as shown. 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 below and then Apply.

Figure 4-92
Using Set
location to align
Vertices

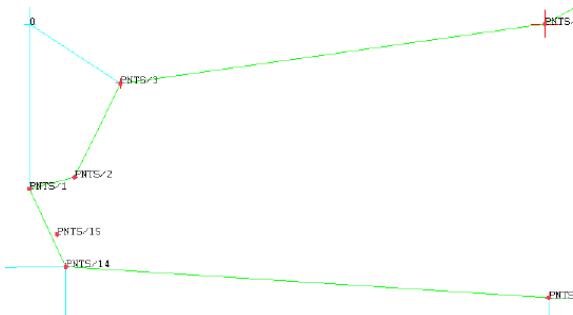
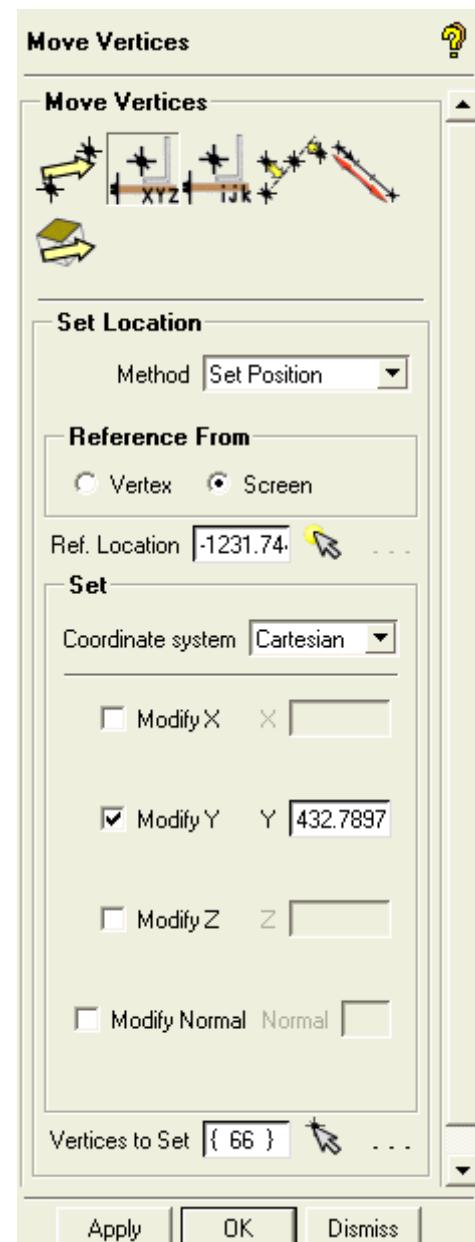
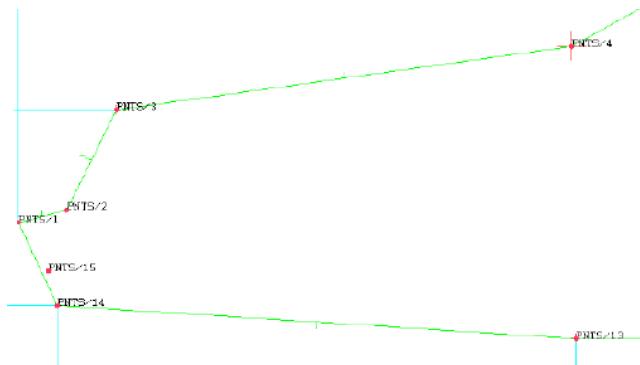


Figure 4-93
Setting the Vertex Location



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

Figure 4-94
After Performing
The Set Location
the Vertex will
line up

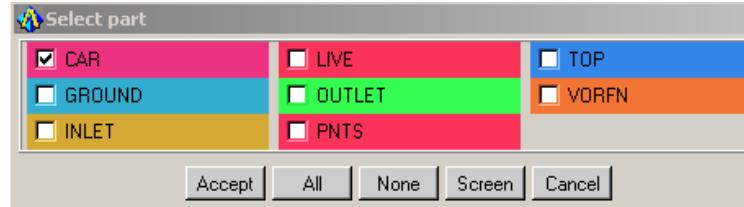


k) Meshing with Curve Parameters

As in the previous tutorial, appropriate node distributions for the edges must be made.

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

Figure 4-95
Select Part
'Car'



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

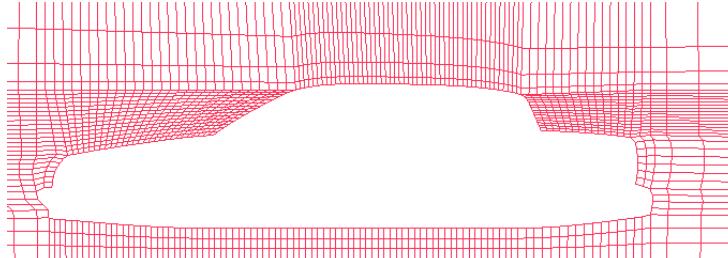
Repeat this procedure but toggle on INLET, OUTLET, TOP and GROUND, set Maximum Size to 500 and press Apply.

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

Turn on Pre-mesh in the Display tree and recompute as shown below.

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
4-96
The H
Grid
Blocking**



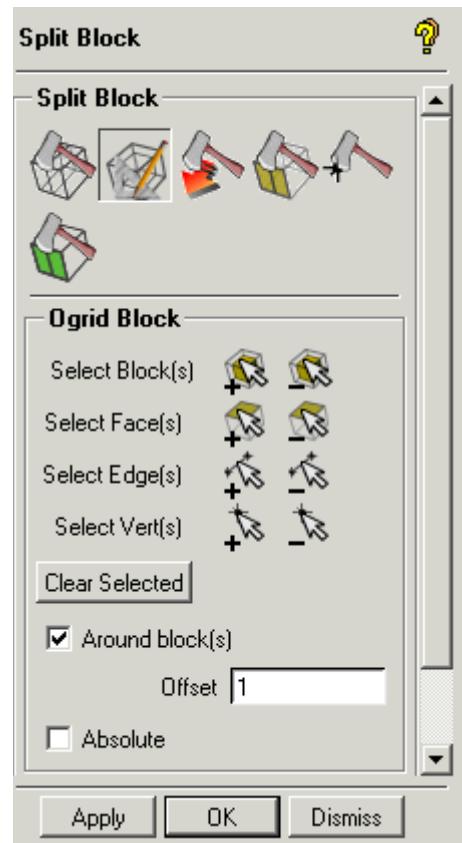
I) Creating an O-grid around the Car

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 (if it is off). Also turn on the VORFN part for we’re going to select the interior blocks

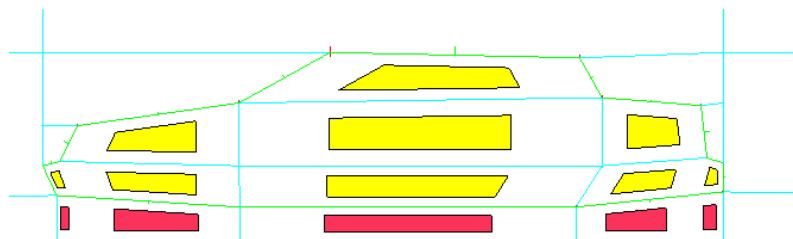
Select Blocking > Split Block  > O Grid  to obtain the panel shown below.

Figure 4-97
Creating an O grid in the
Blocking



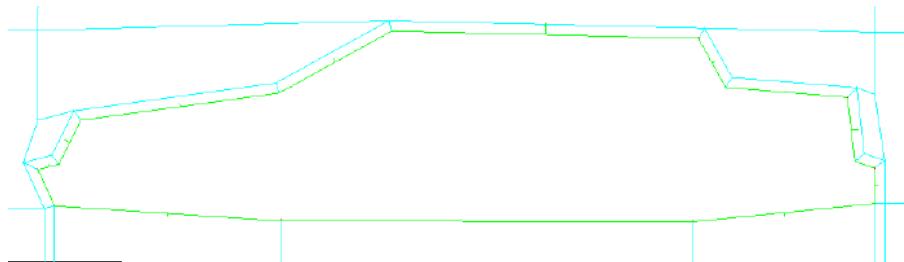
Using Select Blocks +, select the blocks as shown and press the middle mouse button to accept selection. Turn off VORFN. The selected blocks will disappear.

**Figure
4-98
Select the
blocks for
the O grid**



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

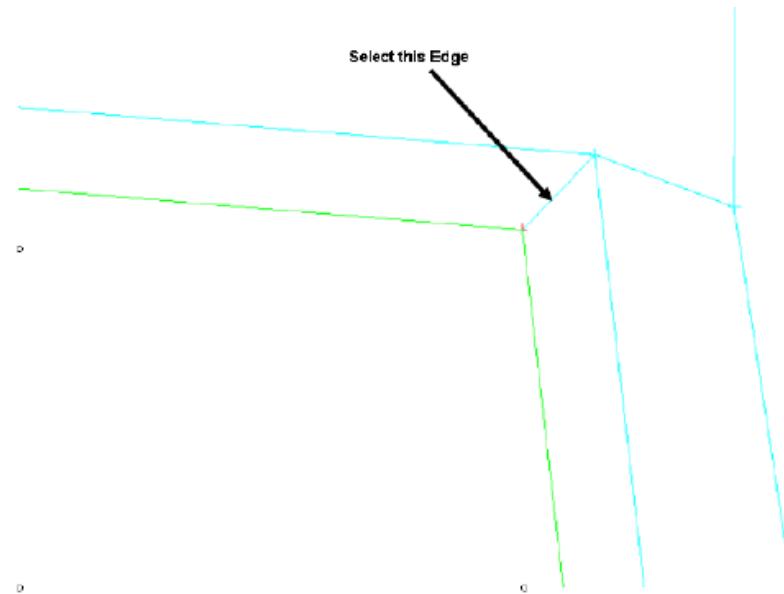
**Figure
4-99
Your
external
O grid
of the
car**



m) Meshing with Edge Parameters

Select Pre-mesh Params > Edge params and select one of the radial edges of the O-grid as shown below.

**Figure
4-100
Setting the
meshing
parameters
on the
edge**



Increase Nodes to 7. To bunch the elements close to the car, enter Spacing 2 to 1 and 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.

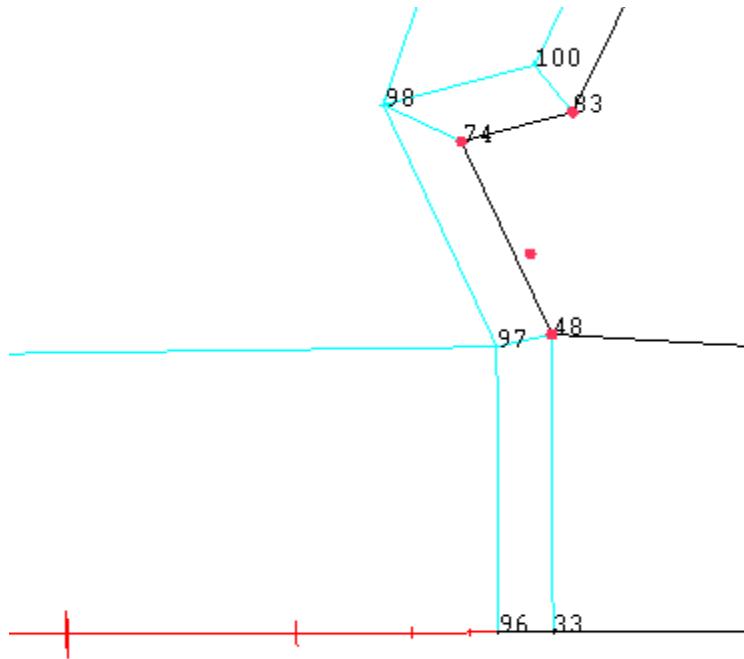
Select Edges > Bunching in the Display tree.

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). Press the middle mouse button to complete.

Figure 4-101
Display of the
bunching using
the
Edge>Bunching

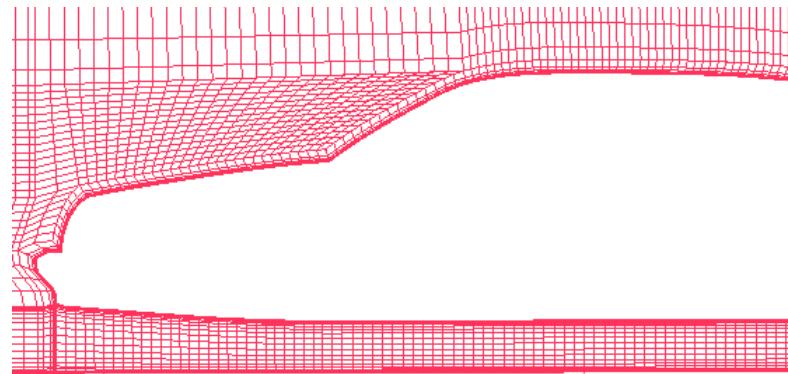


Next, back to Edge Params, select the previous target edge, and make sure Copy Parameters > To All Parallel Edges is on and Apply.

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

Turn on Pre-Mesh and recomputed.

Figure 4-102
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. Notice the differences in the geometry from the car_base subproject.

**Figure
4-103
The car
model
geometry**

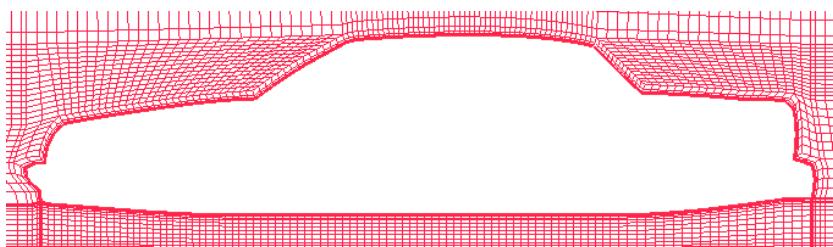


Select File > Replay Script > Replay Control. The Replay control window will show all the commands you previously saved in the subproject car_base. If a new session, you would have to select Load from the Replay control window and select the saved replay_file.rpl.

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
4-104
Final
Mesh**



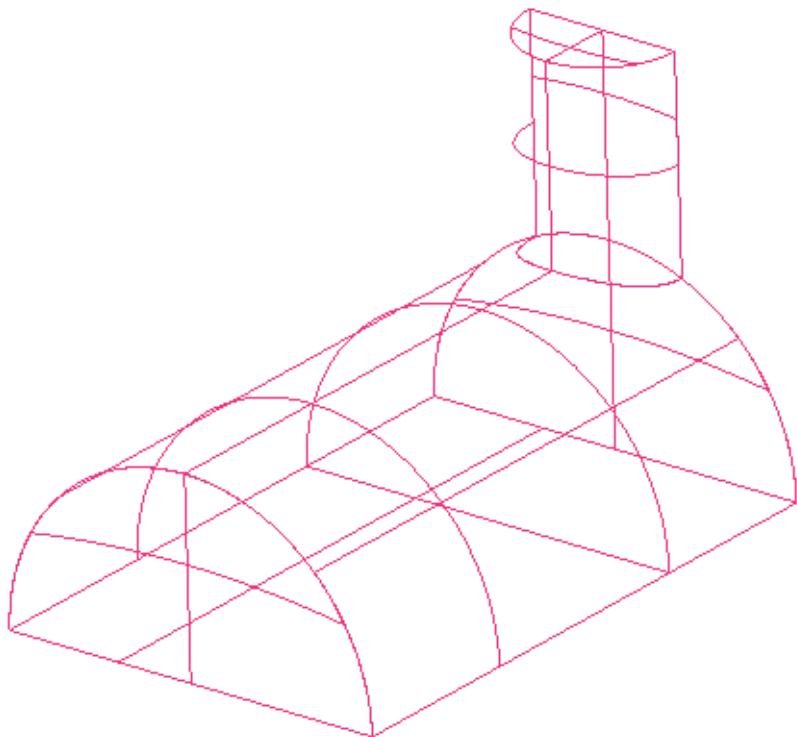
This mesh will have been generated using exactly the same parameters as the first, so the differences in solutions may be attributed to the changes in the geometry, rather than to any dissimilarity in the grids.

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.

4.2.4: 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

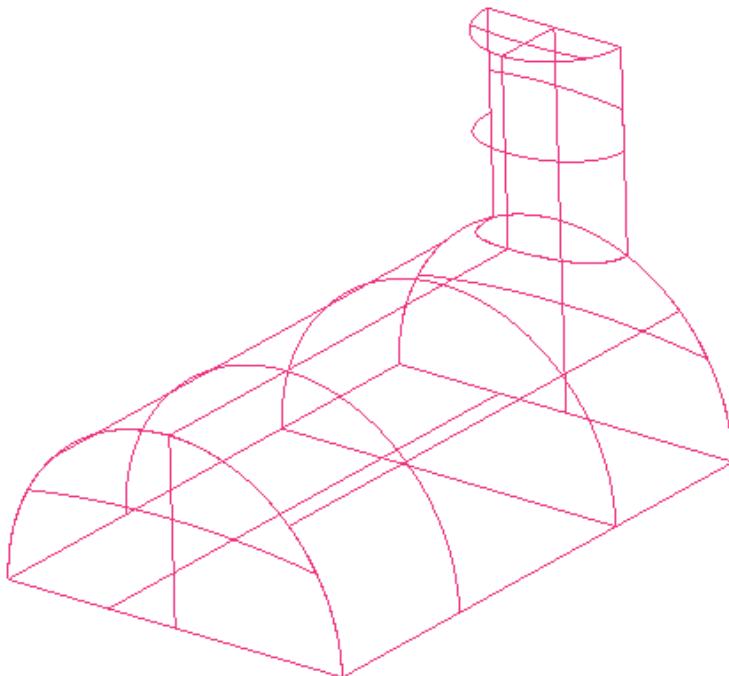
Starting the Project

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 two 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
4-105
3D Pipe
Geometry



c) Starting the Project

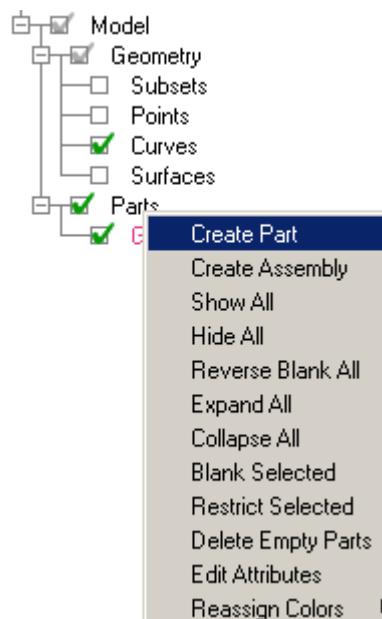
The input files for this tutorial can be found in the Ansys installation directory, under ..\v110\docu\Tutorials\CFD_Tutorial_Files\3DPipeJunct. Copy and open the tetin file, geometry.tin, in your working directory.

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 below.

Figure 4-106
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

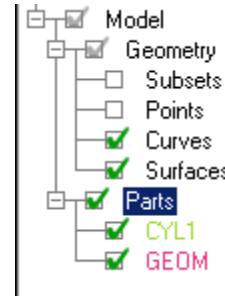
other than surfaces, turn off Toggle selection of points  , Toggle selection of curves  and Toggle selection of bodies (material region definition). Leave on Toggle selection of surfaces  as shown in the figure below. Entity types can also be deactivated (unselectable) by turning them off in the Display tree.

Figure 4-107
Select Geometry tool bar



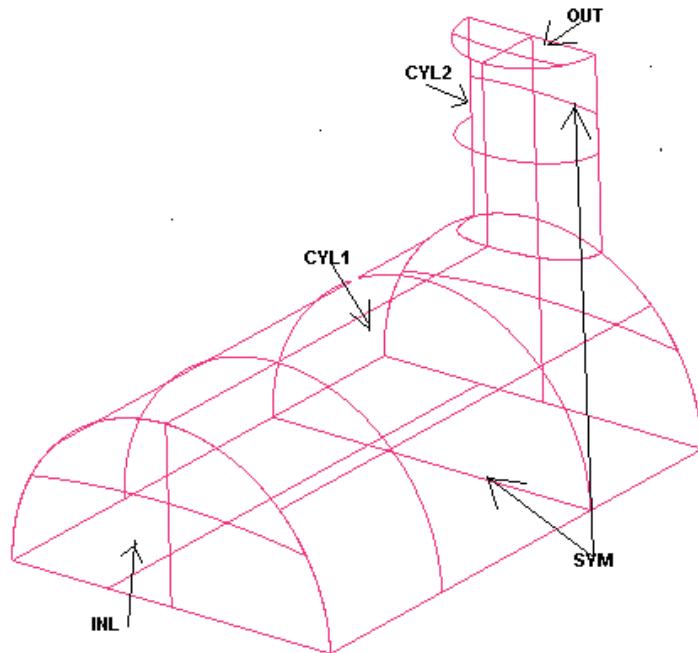
Select the largest semi cylinder with the left mouse button and press middle mouse button or Apply in the panel. Note the new part in the model tree as shown below.

Figure 4-108
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 below. 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 4-109
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.

Figure 4-110
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.

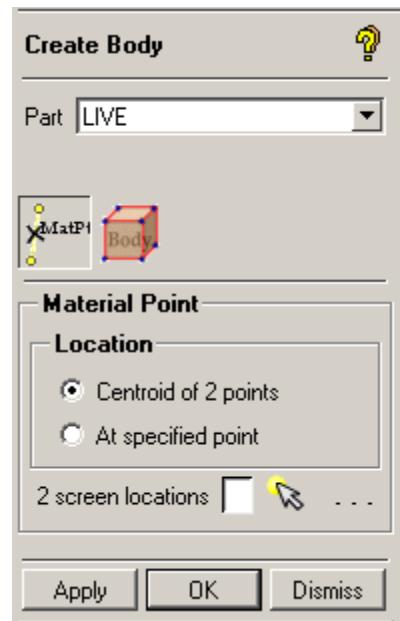
Figure 4-111
Geometry Part Point or Node Selection



e) Creating a Material point.

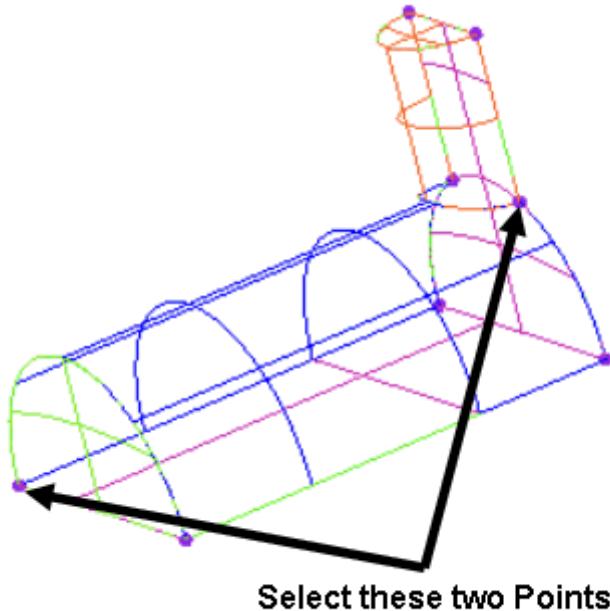
Select Geometry > Create Body from the geometry tab. Type in LIVE for the Part name.

Figure 4-112
Create Body panel



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

**Figure
4-113
Selection
of points
for
Material
point
creation**



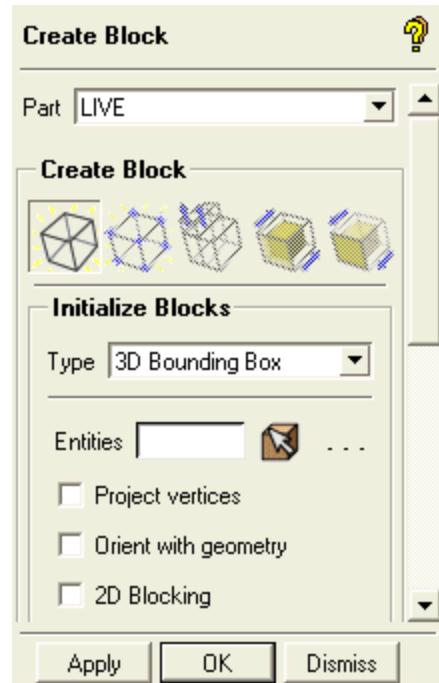
Press the middle mouse button or Apply.

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 the figure below. Select the LIVE Part, make sure Type > 3D Bounding Box is selected (default) and Apply.

Figure 4-114
Create Block Window



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

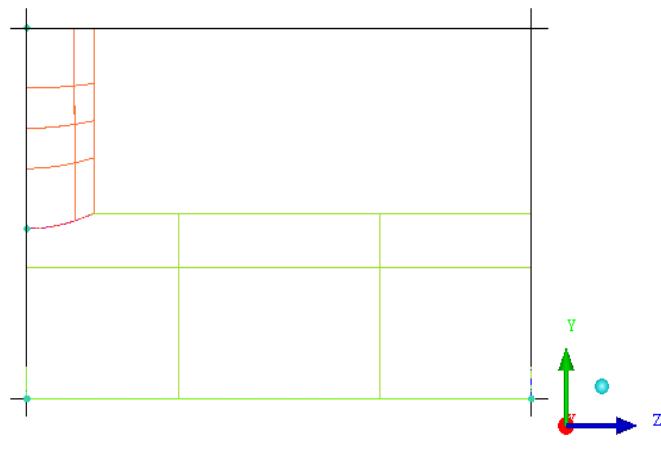
The next step is to split the block into four sub-blocks. Begin by turning on Curves and Surfaces from the Display tree.

The L-shaped topology is best seen in a side view. Select View > Left or



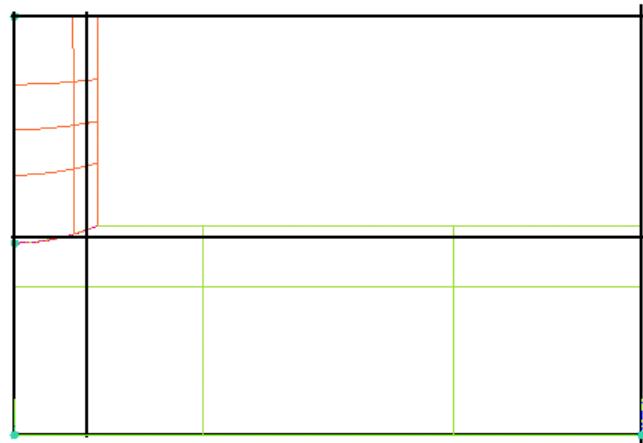
select the X axis in the Triad Display in the lower right hand corner to re-orient the model as it appears below.

Figure 4-115
Geometry showing
the split locations



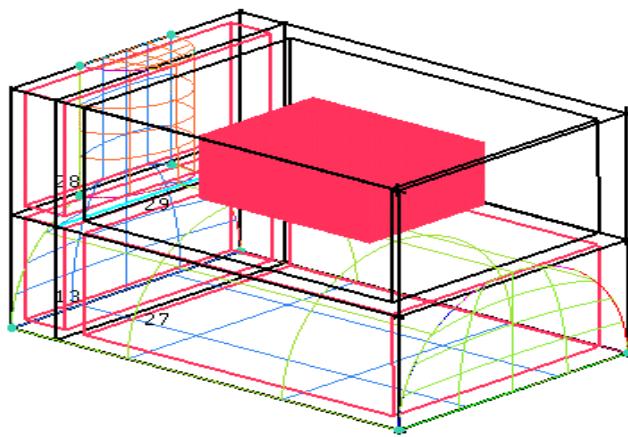
Select Blocking > Split Block > Split Block . Select any horizontal edge with the left mouse button; try to position the new edge near the front end of the small cylinder, and press the middle mouse button to accept. Next select any of the vertical edges and position the new edge near the top of the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

Figure 4-116
Block Splits



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

**Figure
4-117
Delete
Block**



g) Projecting the Edges to the Curves

Turn off the Surfaces, displaying Curves only.

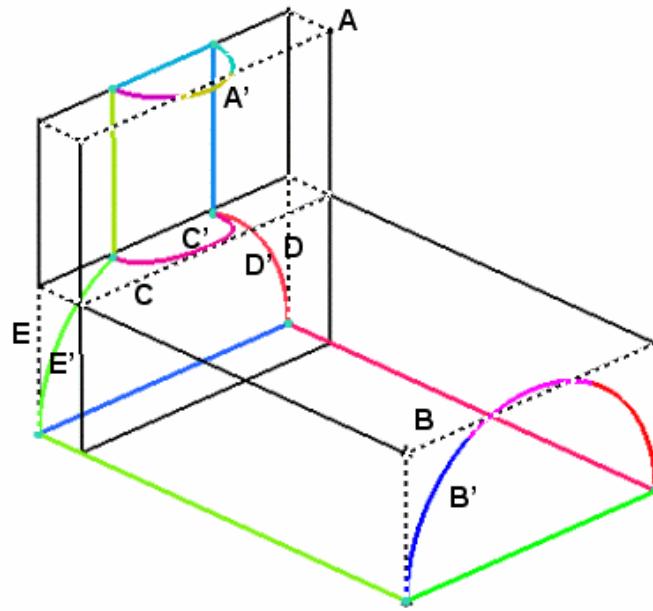
Select Associate > Associate Edge to Curve .Select the three edges at the top (A) indicated with dashed lines in the figure below. Press the middle mouse button, then select the three curves (A') making up the small semicircle. Press the middle mouse button to complete the operation.

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').

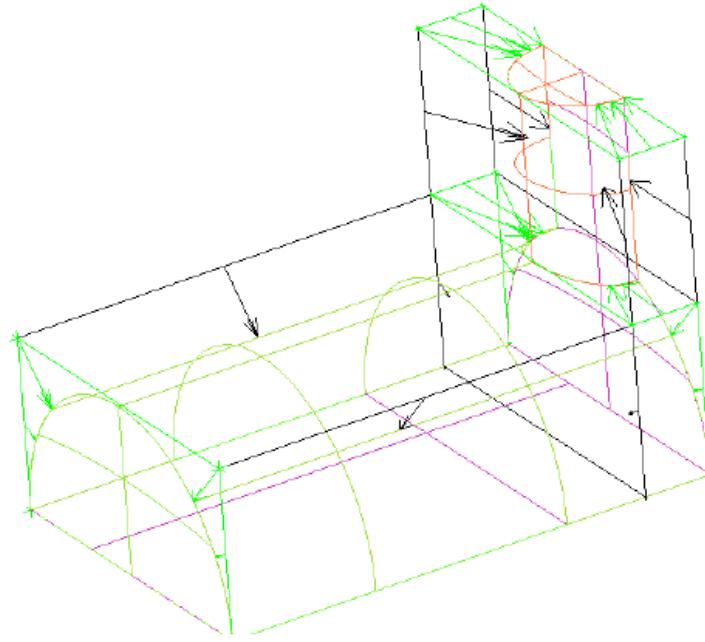
Finally, associate the side rear edges to the curves making up the backend of the large cylinder, D->D' and E -> E', as shown.

**Figure
4-118
Associating
edges to
curves**



Verify that the correct associations have been set: right mouse select Edges > and select Show Association in the Display tree. 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
4-119
Display of
the
projections
of the
edges to
the
associated
curve**



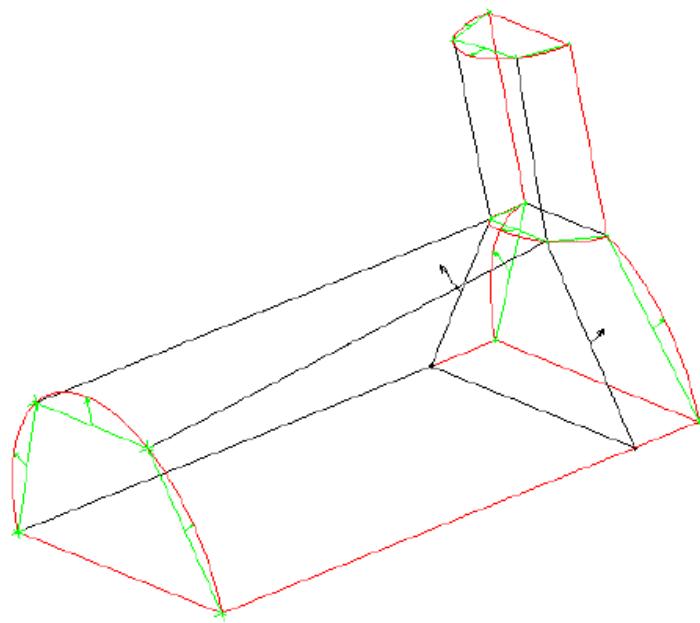
h) Moving the Vertices

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 shown below. 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 4-120
After moving
the vertices to
the
appropriate
locations on
the geometry



Turn off Edges > Association in the Display tree.
Save the Blocking!

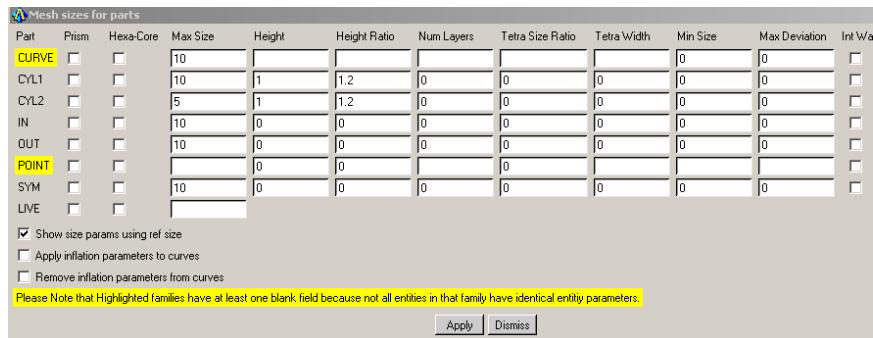
i) Generating the Mesh

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 > Part Mesh Setup.

Hexa Meshing

**Figure
4-121
Entering
new mesh
parameters**

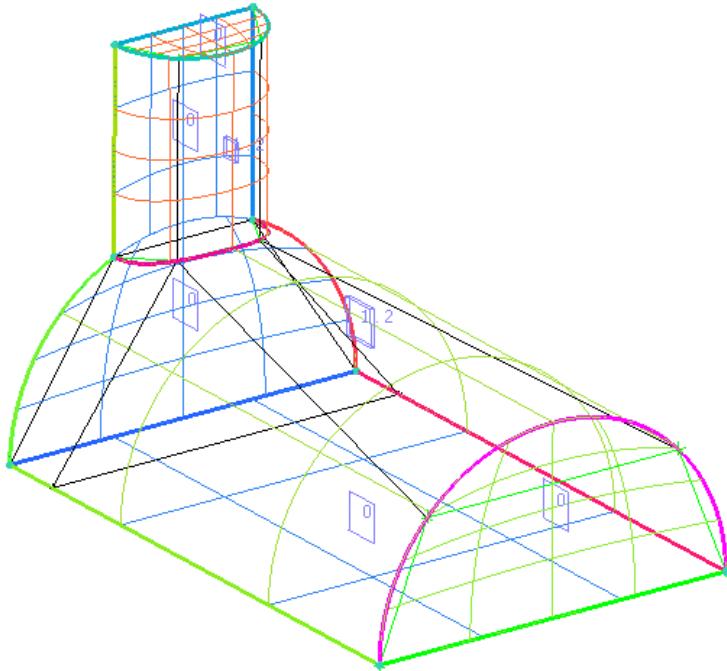


Set sizes as shown: Max Size of 10 on all surface parts except the small cylinder (CYL2) which can be 5, Height of 1 only on the wall boundaries (CYL1 and CYL2) and a Height Ratio of 1.2 on those same walls. Apply and Dismiss.

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

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

Figure 4-122
Hexa Mesh
sizes

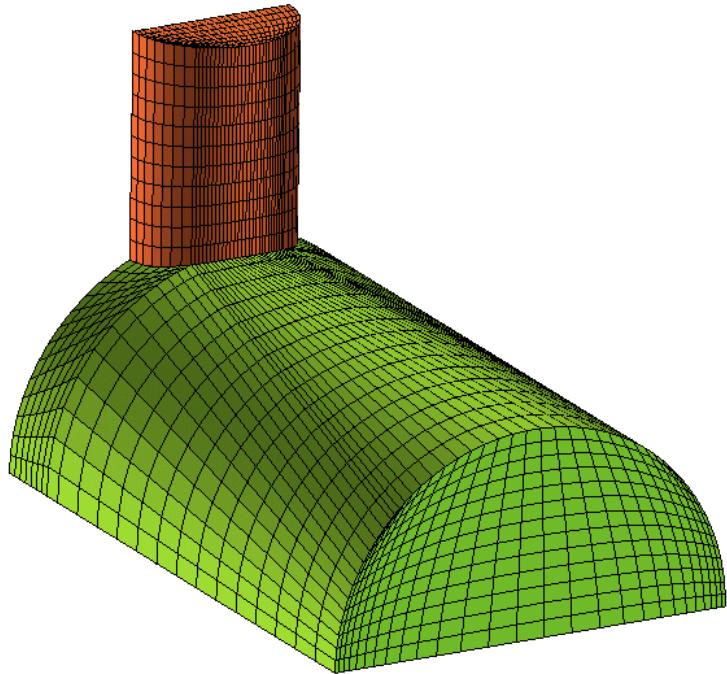


Select Blocking > Pre-Mesh Params > Update Size , make sure Method > Update all (default) is selected and press Apply.

Turn on Pre-Mesh and (re) compute.

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

Figure 4-123
**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:

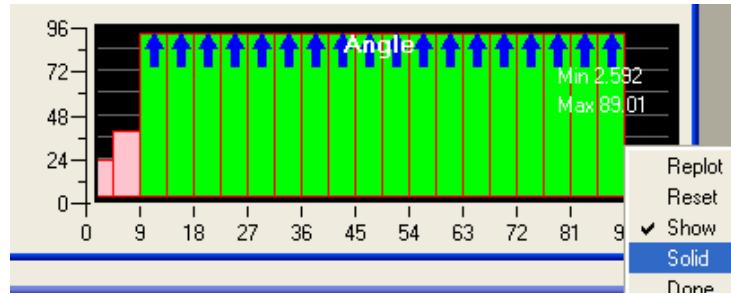
This calculates the determinant of all elements in the mesh, which is a volume measurement calculated from a Jacobian matrix.

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.

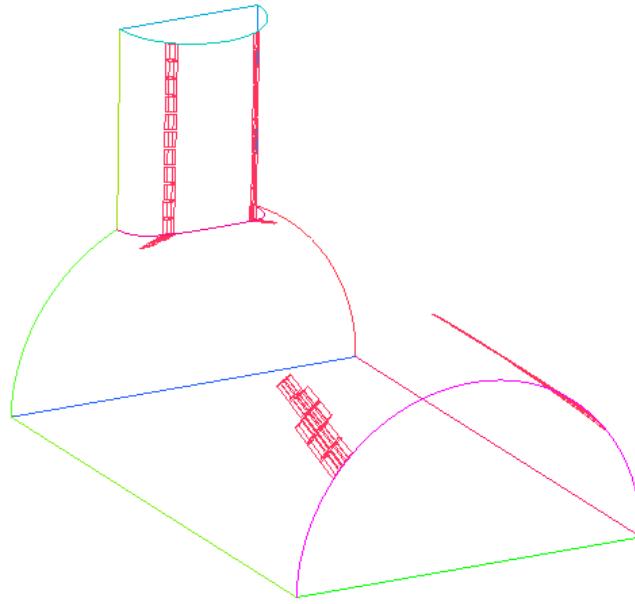
Select Blocking > Pre-mesh Quality  . For the Criterion, select Angle. A histogram (bar graph) of the values will be displayed. Select the two worst ranges (bars). They will be highlighted in pink. Select with the right mouse button anywhere within the histogram window. Make sure Show (default) is turned on in the pull down. You may wish to turn Solid off.

Figure 4-124
Histogram of Angle



View the highlighted elements as shown below. 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
4-125
The
highlighted
elements
in mesh**



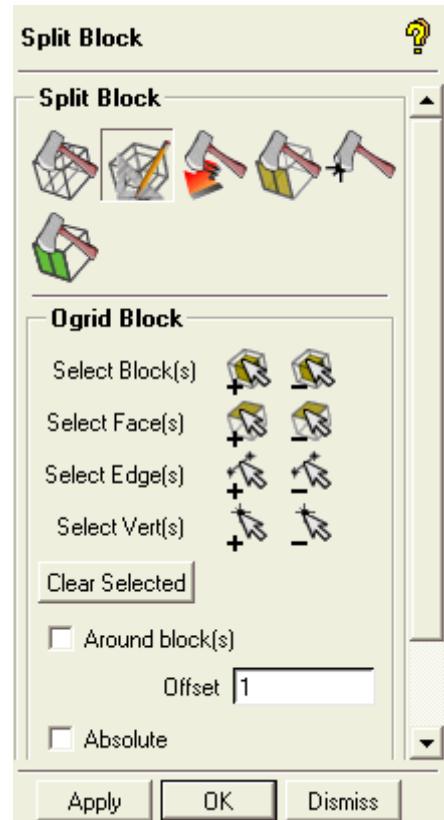
k) Creating an O-grid in the Blocking

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.

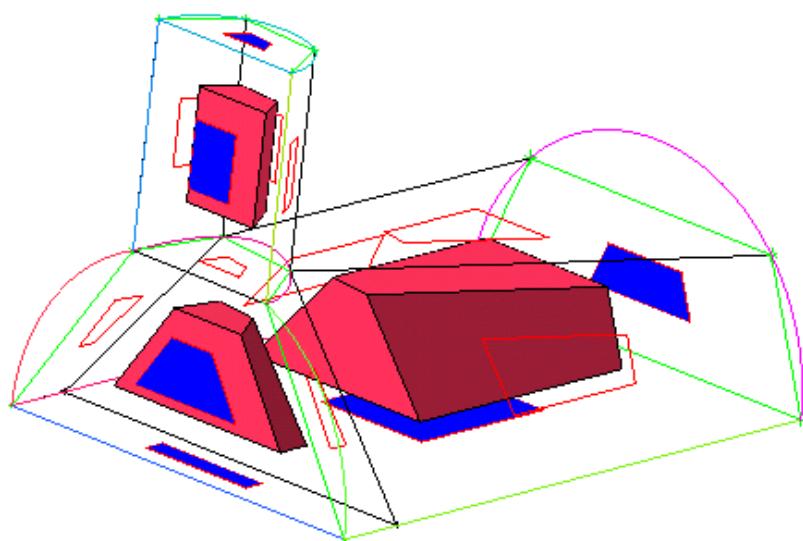
Figure 4-126
Creating an O-grid



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.

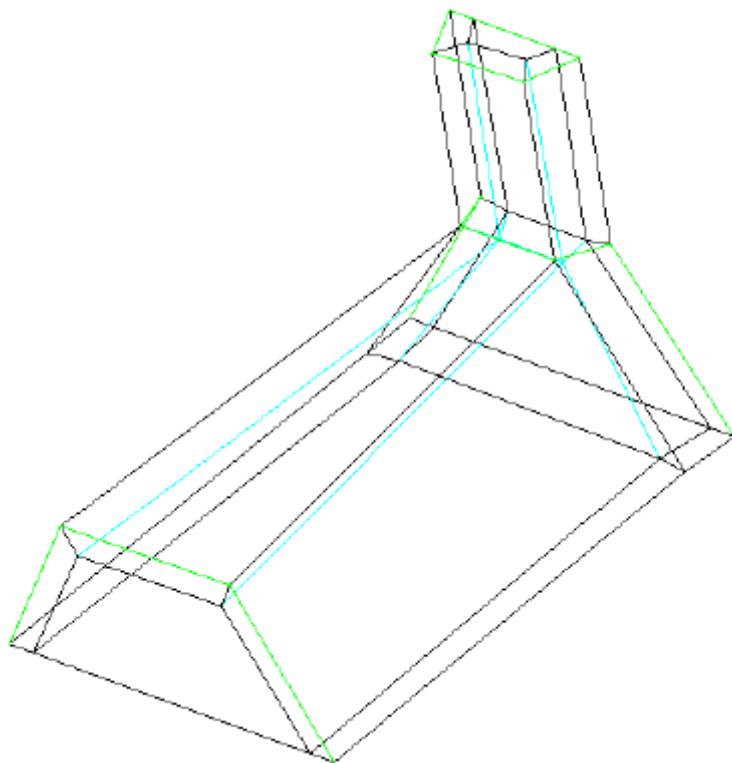
Select Face(s) and select the face icons as shown.
If there is difficulty in seeing the face icon, one can select a face (or block for that matter) selecting Select diagonal corner vertices from the Select blocks toolbar or typing Shft-D on the keyboard. This will allow you to select two diagonally opposing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

**Figure
4-127
Selected
Blocks
and
Faces**



Use the default Offset and Apply. An o-grid structure will be created as shown below. Note the o-grid “passing through” the selected faces. Radial blocks are only adjacent to the cylinder surfaces.

Figure 4-128
The blocking
with O grid
structure

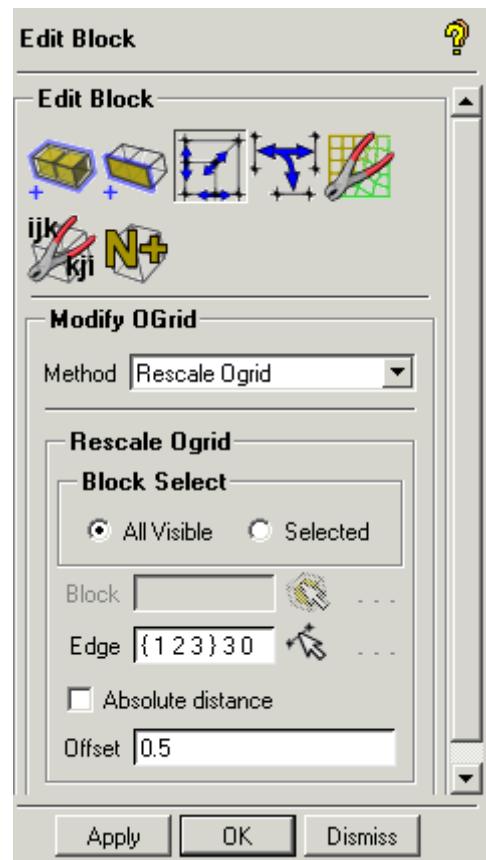


To re-size the o-grid after it's been created, select Edit Block >

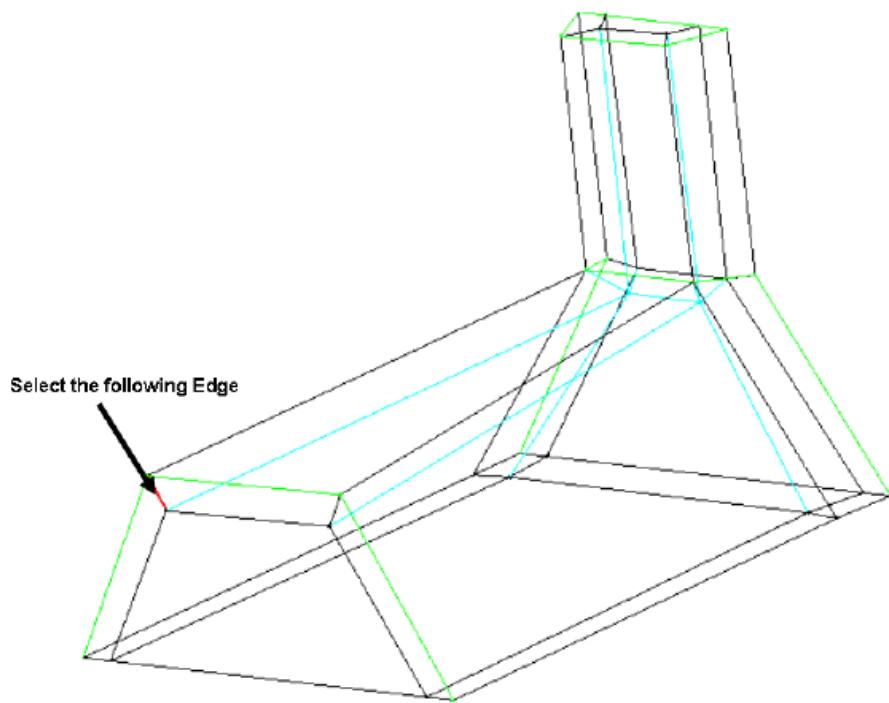


Modify Ogrid, this will open the Modify Ogrid panel as shown in the figure below. Select edge(s)  and select one of the radial edges as shown. Enter the Offset as 0.5, toggle off Absolute distance (default) and press Apply. The radial edge will be shrunk in half reducing the size of the radial blocks and increasing the size of the central block.

Figure 4-129
Modify OGrid panel



**Figure
4-130
Modify
Ogrid
edge**



Update surface mesh sizes on the blocking: Select Pre-mesh Params 

> Update Size  and Apply.

Turn on Pre-Mesh and recompute.

I) Further refinement with Edge Parameters

Again, turn off Pre-Mesh. Select Pre-Mesh Params  > Edge Params 

 > Select edge(s) and again select one of the radial edges.

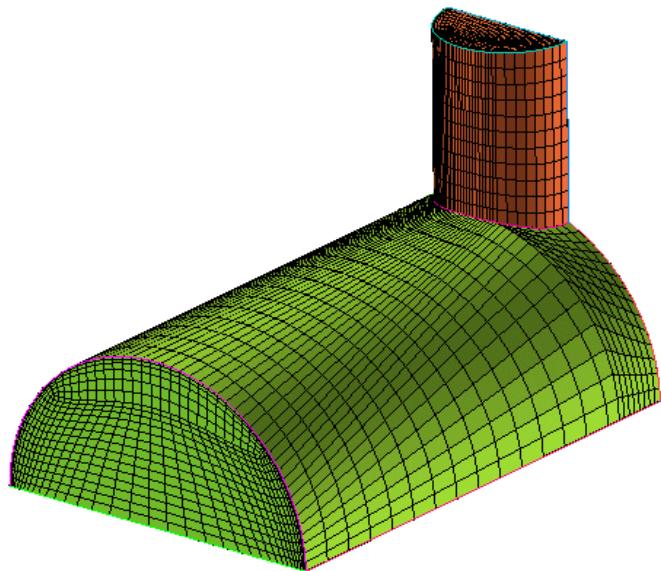
Increase the number of Nodes to 7. Change Spacing 1 (end near the wall) to 0.2. Turn on Copy Parameters and select Copy > Method > To All Parallel Edges (default). Turn on Copy Absolute and Apply. This will

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 in the Display tree and recompute.

Turn off Curves, Surfaces, and Edges and view the final mesh.

Figure 4-131
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.

4.2.5: 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.

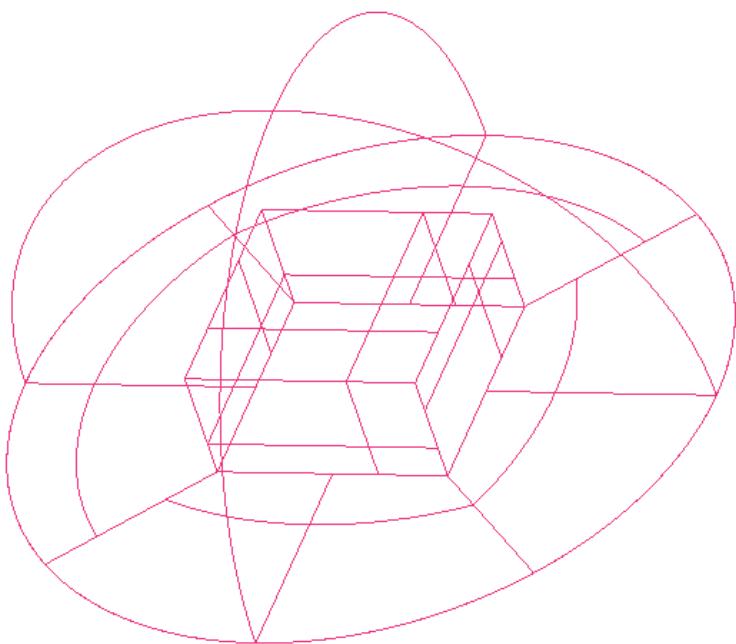
a) Summary of Steps

- 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 below

**Figure
4-132
The
Sphere
Cube
Geometry**



c) Starting the Project

The input files for this tutorial can be found in the Ansys installation directory, under ..\v110\docu\Tutorials\CFD_Tutorial_Files\SphereCube. Copy and open the tetin file, geometry.tin in your working directory.

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 in the Display tree.

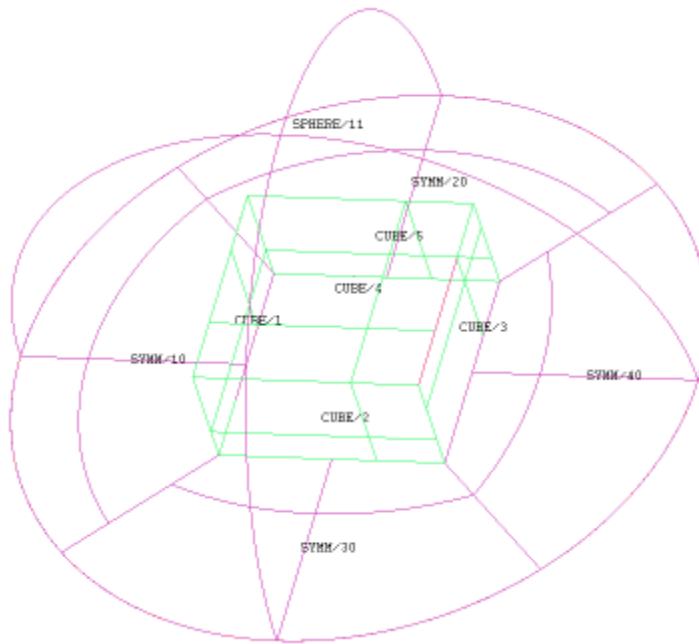


Type in SYMM in the Part field and select Create Part by Selection

or Select entities . Select the four surfaces on the bottom of the geometry Press the middle mouse button or Apply.

Similarly create new parts, SPHERE and CUBE referring to the figure below as a guide.

**Figure
4-133
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).

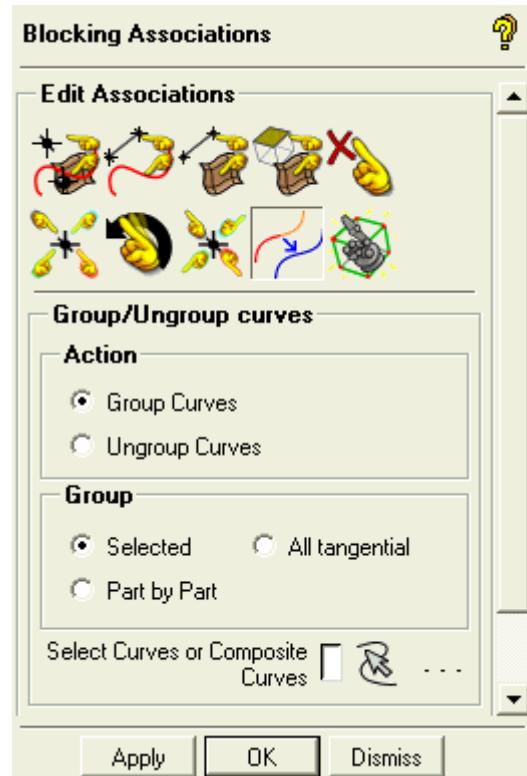
Select Part > LIVE (select the down arrow to get a pull down list of parts) and Apply.

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 the figure below.

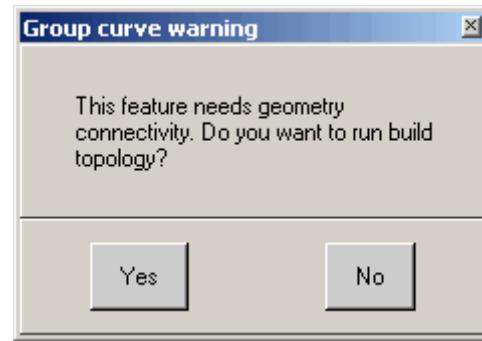
Figure 4-134
Group Ungroup curve window



This feature needs geometry connectivity so it will ask to run build topology. Select Yes.

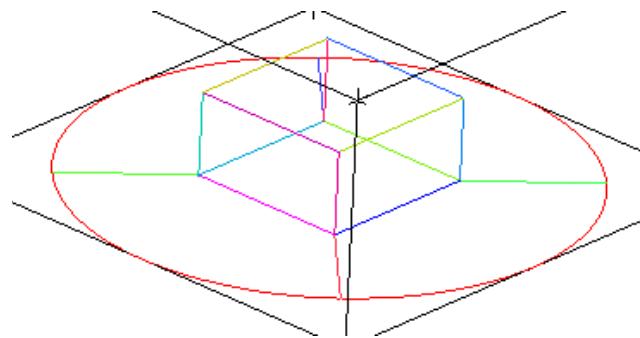
Note: Build Topology will generate a series of curves along all shared edges of surfaces. It is meant as a geometry diagnostic tool but is also used to determine logical connectivity between surfaces and to build curves and points to capture sharp features. To invoke independently, use Geometry > Repair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

Figure 4-135
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 shown in the figure below.

Figure 4-136
Grouping of all tangential Curves



h) Projecting the Edges to Curves

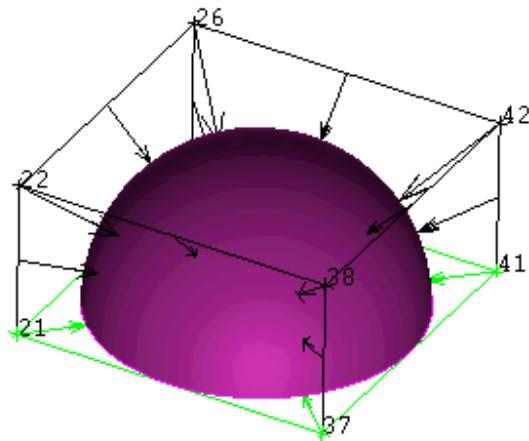
Select Associate > Associate Edge to Curve .

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.

Turn on vertices under Blocking in the Display Tree. Also Right click on the vertices in the Display tree and click on Numbers. This will show all the vertices by their numbers.

Verify association: In the Display tree turn on Surface > Solid and Edges > Show Association and view as in the figure below.

Figure 4-137
Projection to
the curve and
sphere
surface

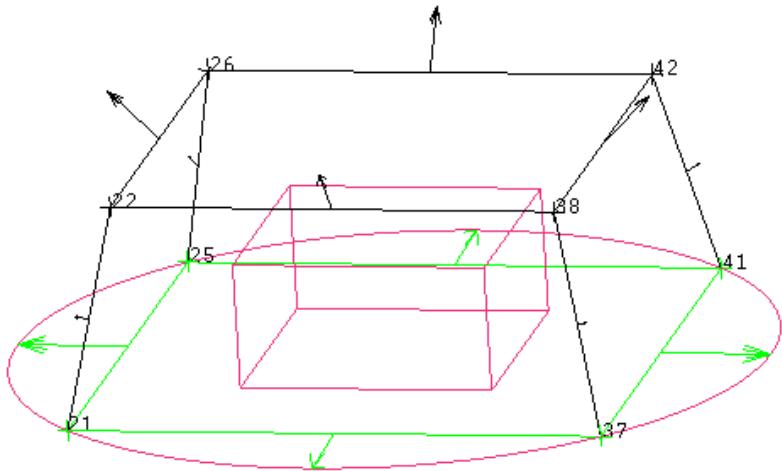


i) Moving Vertices

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

Turn off the surfaces from the Display tree to better view the new vertex positions.

**Figure
4-138
Vertices
moved
on the
geometry**



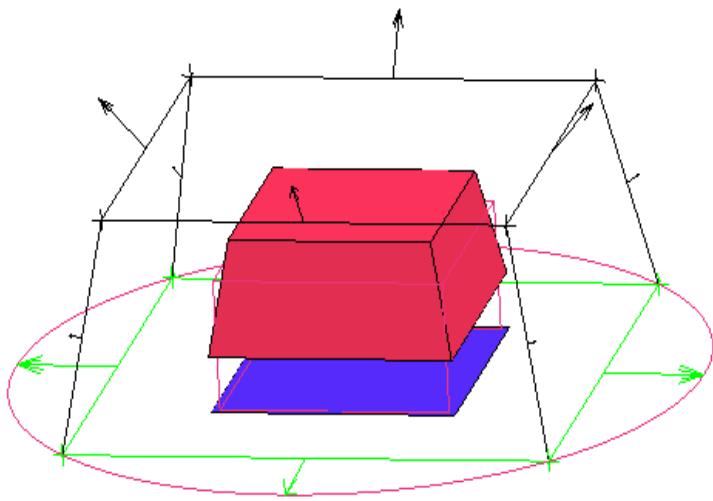
j) Creating the O-grid

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. 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
4-139
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 >

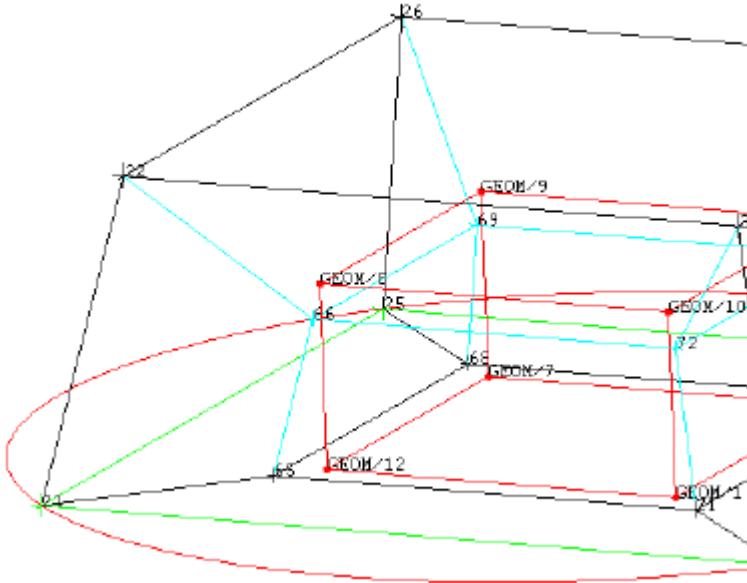
Associate > Associate Vertex .

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 below. Use F9 repeatedly to toggle between selection mode and dynamic mode to reorient the view (translate, rotate, zoom) whenever necessary.

Figure 4-140
Fitting the
inner block to
the cube with
Prescribed
Points



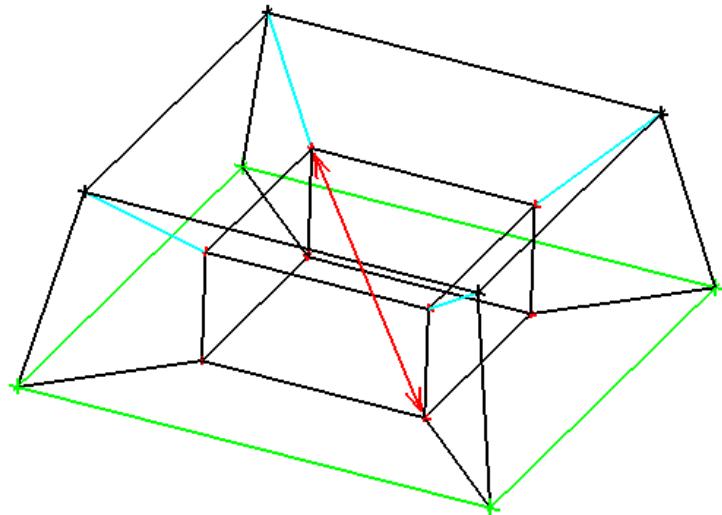
I) Discarding the Inner block

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 below. Press the middle mouse button or Apply.



Figure 4-141
Removing the central block



m) Generating the Mesh

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

Select Mesh > Part Mesh Setup . Type in the values as shown in the figure below. 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
4-142
Mesh Size
for Part**

Mesh sizes for parts						
Part	Prism	Hexa-Core	Max Size	Height	Height Ratio	Num Layers
CUBE	<input type="checkbox"/>	<input type="checkbox"/>	0.5	0.01	1.2	0
GEOM	<input type="checkbox"/>	<input type="checkbox"/>	0	0	0	
SPHERE	<input type="checkbox"/>	<input type="checkbox"/>	1	0.02	1.2	0
SYMM	<input type="checkbox"/>	<input type="checkbox"/>	1	0	0	0
LIVE	<input type="checkbox"/>	<input type="checkbox"/>	0			

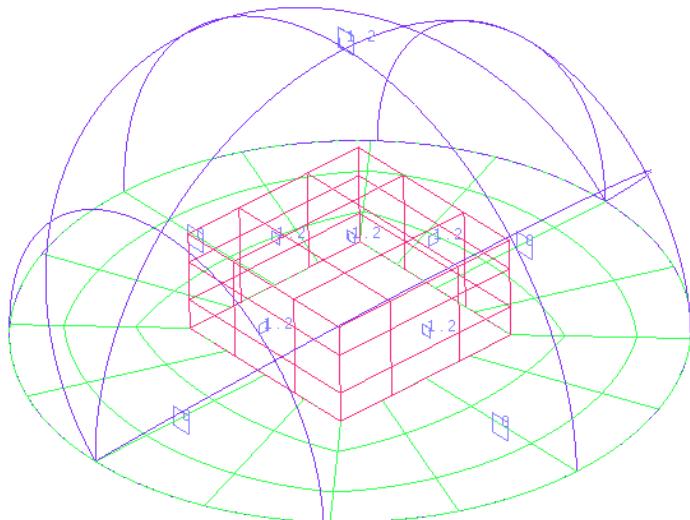
Show size params using ref size
 Apply inflation parameters to curves
 Remove inflation parameters from curves

Please Note that Highlighted families have at least one blank field because not all entities in that family ha

Apply

Verify the sizes by right mouse selecting Surfaces and turn on Hexa Sizes.

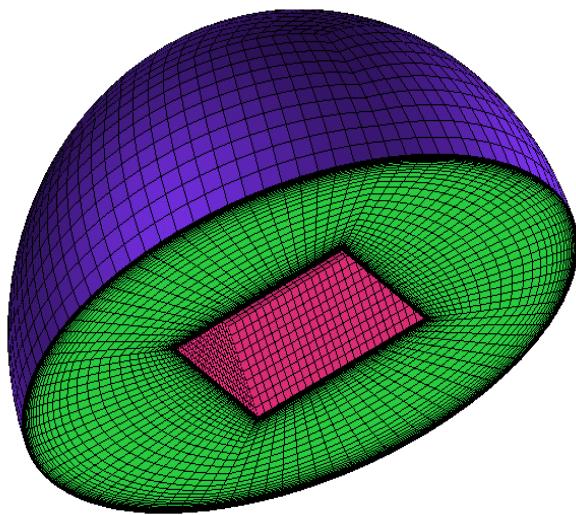
**Figure 4-143
Verifying
Hexa Sizes**



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 below.

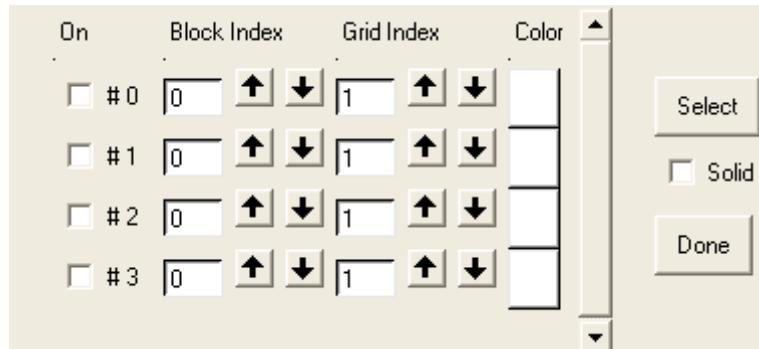
**Figure
4-144
Mesh after
Recompute
Operation**



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.

Figure 4-145
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.

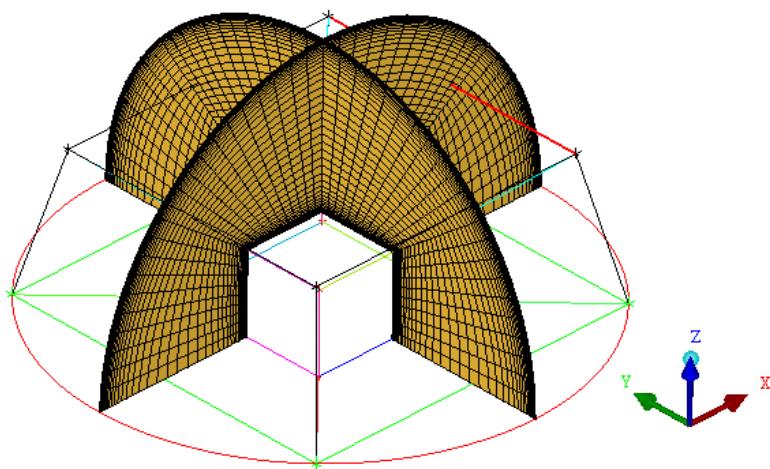
To move the scan plane toggle the up/down arrows underneath either Block Index or Grid Index. Block Index will increment one block at a time whereas Grid Index will increment one node at a time.

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. Note that the # 1 column is automatically turned on in the window.

**Figure
4-146
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.

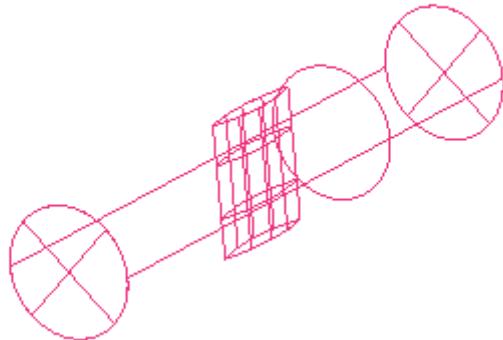
Check Pre-Mesh Quality  and once satisfied convert to unstructured mesh: Right mouse select Pre-Mesh and select Convert to Unstruct Mesh. Save the project within the File menu. This will save the tetin, unstructured mesh, blocking and project settings files all beginning with the project name.

Exit or continue on with the next tutorial

4.2.6: 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.

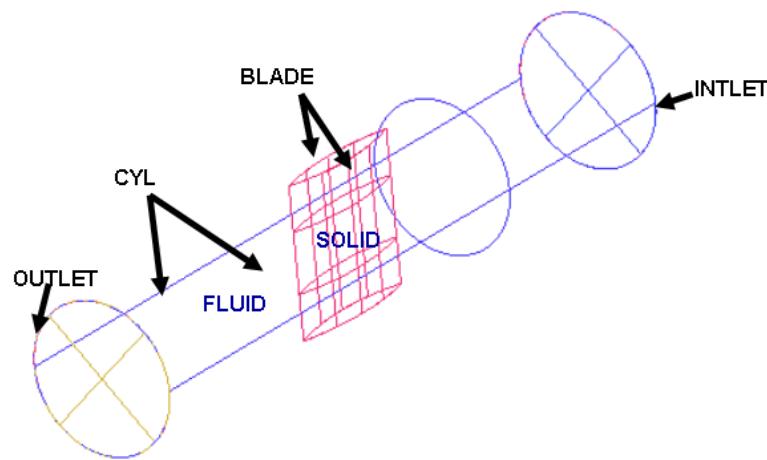
c) Starting the Project

The input files for this tutorial can be found in the Ansys installation directory, under ..\v110\docu\Tutorials\CFD_Tutorial_Files>PipeBlade. Copy and open the geometry.tin file in your working directory.

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 the figure below for the Surface part assignments.

**Figure
4-147
The
Pipe
Blade
configu
ration**



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 OUTLET.

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

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, CYL, 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. 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.

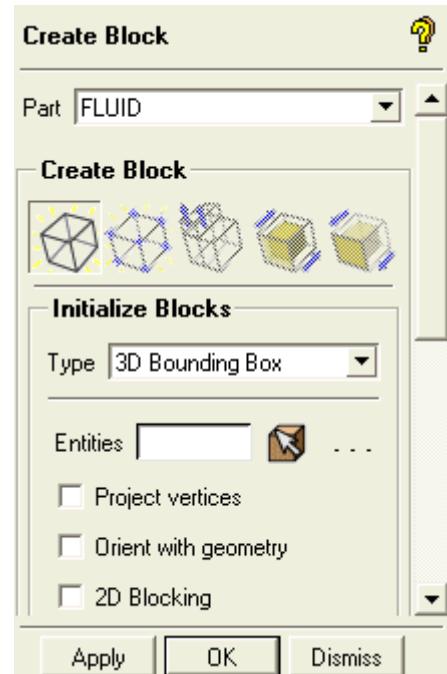
Delete any Empty Parts: From the Display Tree, right mouse select on Parts > Delete empty Parts.

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

h) Blocking

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

Figure 4-148
Create block window



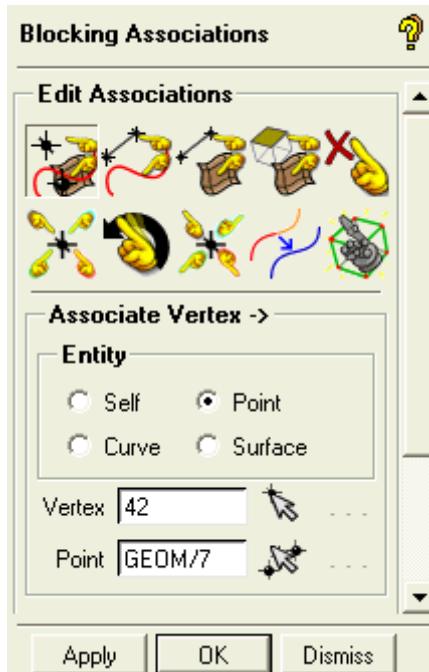
Select the block Type as 3D Bounding Box (default) from the pull down arrow. Name the Part as Fluid. Press Apply without selecting anything, and the initial block will be created around the whole model.

i) Association of vertex to point

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 will open. Toggle ON Blocking > Vertices and right mouse click on Vertices > Numbers under Blocking in Display Tree.

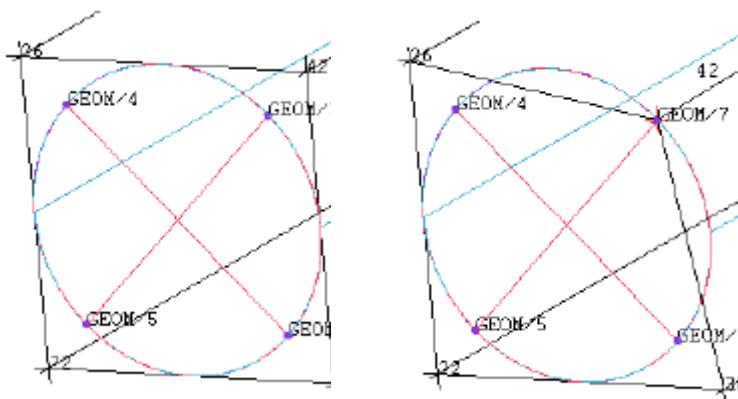
Figure 4-149
Associate vertex window



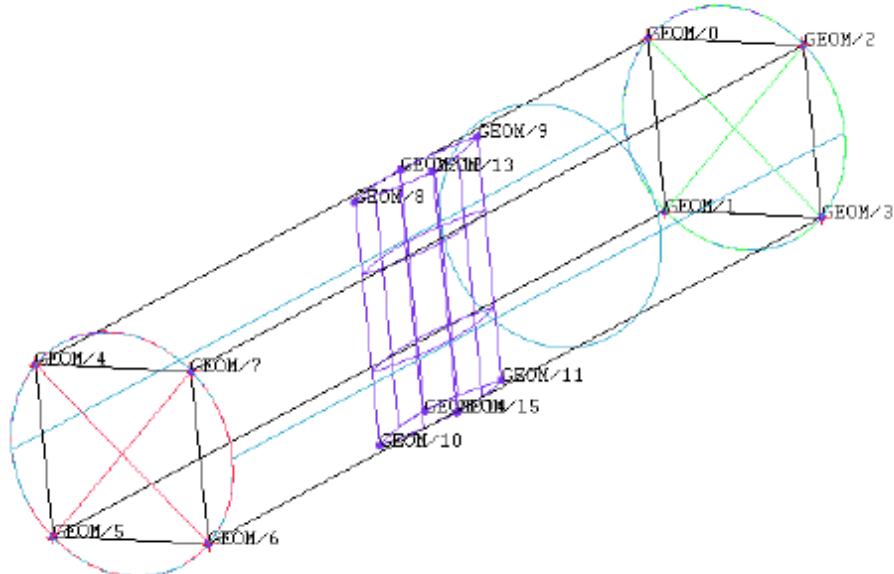
Select Point under Entity.

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 below. Similarly, associate the other vertices and points for the inlet and outlet so that after completion the geometry should look like below.

**Figure
4-150
Moving
the
vertices**



**Figure 4-151
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.

j) Associating edges to curves

Select Associate  > Associate Edge to Curve  . Press the edge selection icon  then select the four edges shown in the figure below and press the middle mouse button. Then press the curve selection icon  and select the four curves shown in the figure below and press the middle mouse button. 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 4-152
Association window

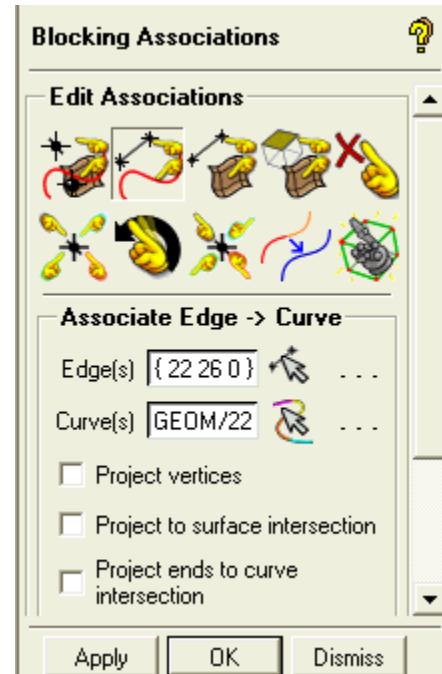
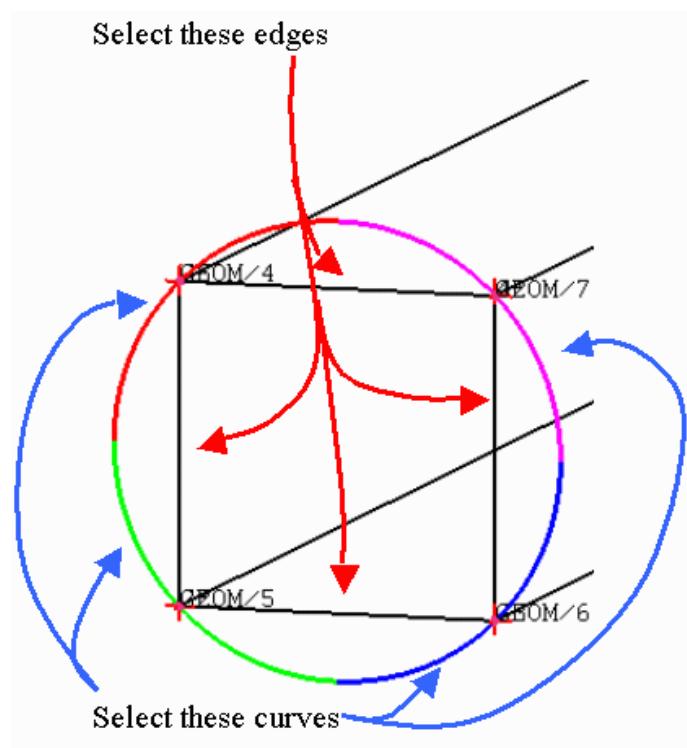
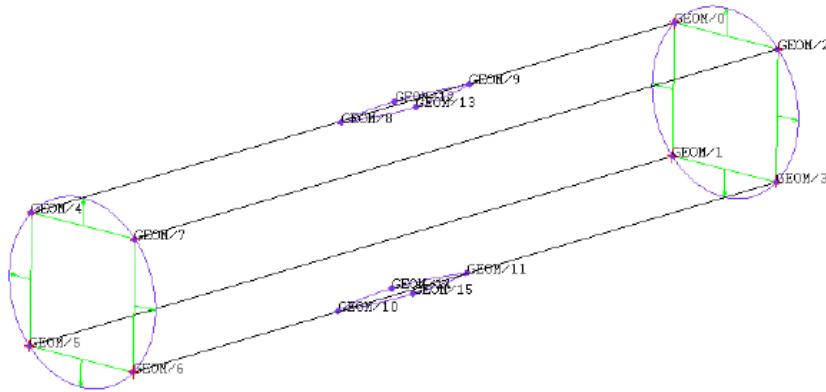


Figure 4-153
Edges and Curve
selection for
association



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.

**Figure
4-154
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

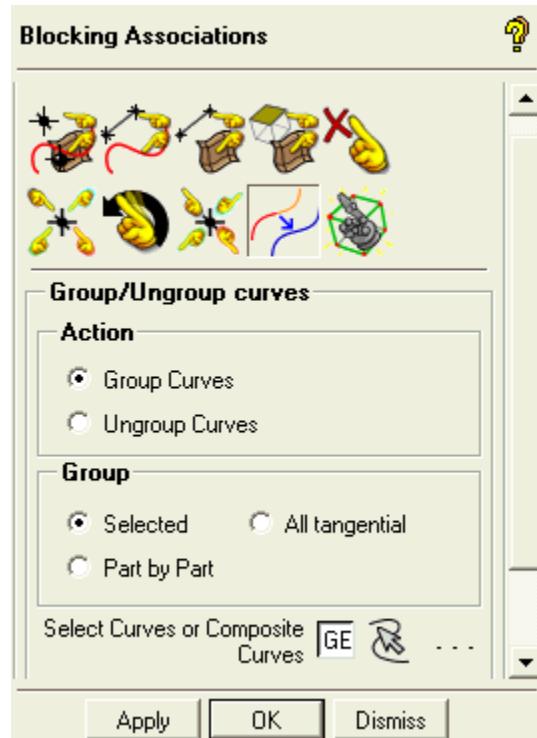
this association, which is really deleting an association, by pressing .

k) 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.

Figure 4-155
Group curve window



Select the four curves corresponding to OUTLET as shown in the figure and press Apply to group them.

I) 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 below. Choose All visible and Split method as Prescribed Point. Select the edge selection icon then select

one of the edge which is along z-direction. After selecting the edge it will prompt you to select the point Select the Prescribe point, GEOM/9 and press middle click to accept the selection.

Similarly, make another split using the same edge but through the 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 below.

Note: Make sure that the Edge that is selected lies within the range of the Prescribed Point that will be selected.

Figure 4-156
Split block window

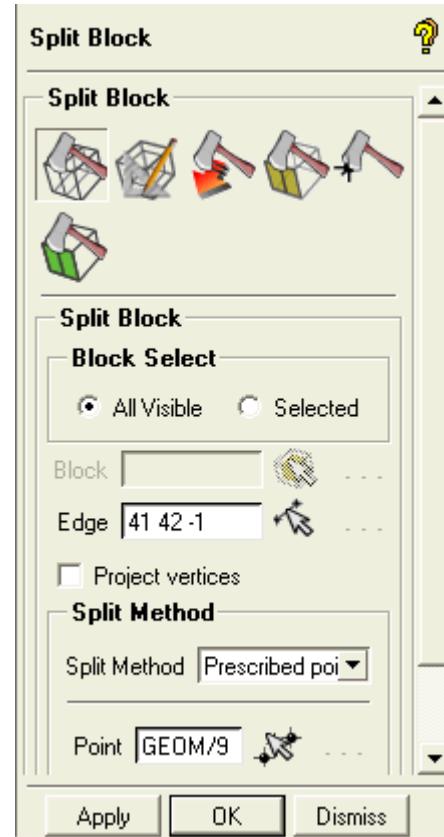
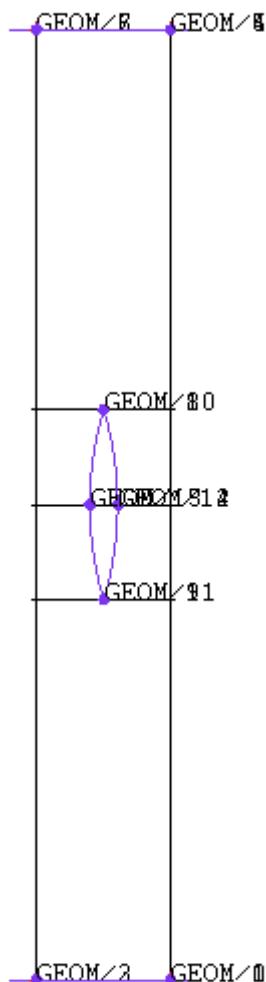


Figure 4-157
Make the
horizontal splits in
the block



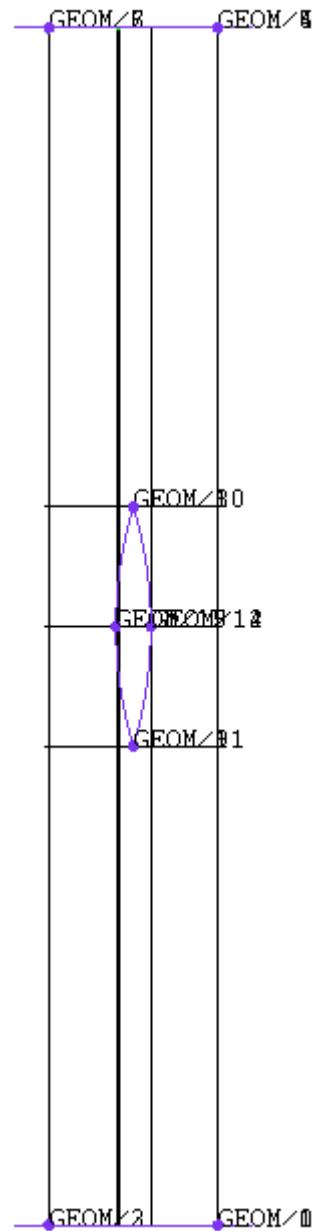
These are the splits in the 'k'-direction. The next set of splits will be in the 'i' direction.

Now select the Split method as Screen select. Press the edge selection

icon and select any of the horizontal edges (which is along x-

direction) 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.

Figure 4-158
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.

m) Collapsing Blocks to Display the Blade

In this section, the Collapse feature is introduced to create degenerate blocks for the blade.

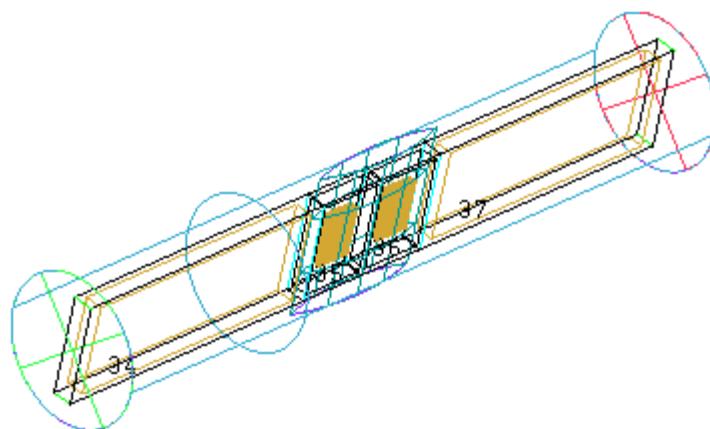
For clarity in these operations, right mouse click in the Display tree on Blocking>Index Control. Change the Index control for the 'I' dimension so that the Min is 2 and the Max is 3. Turn OFF the Points from the Display window. The restricted topology consists of four blocks, where the two center blocks belong to the blade.

Before collapsing the blocks, change the Part family of the two center blocks to SOLID, the material representing the blade.

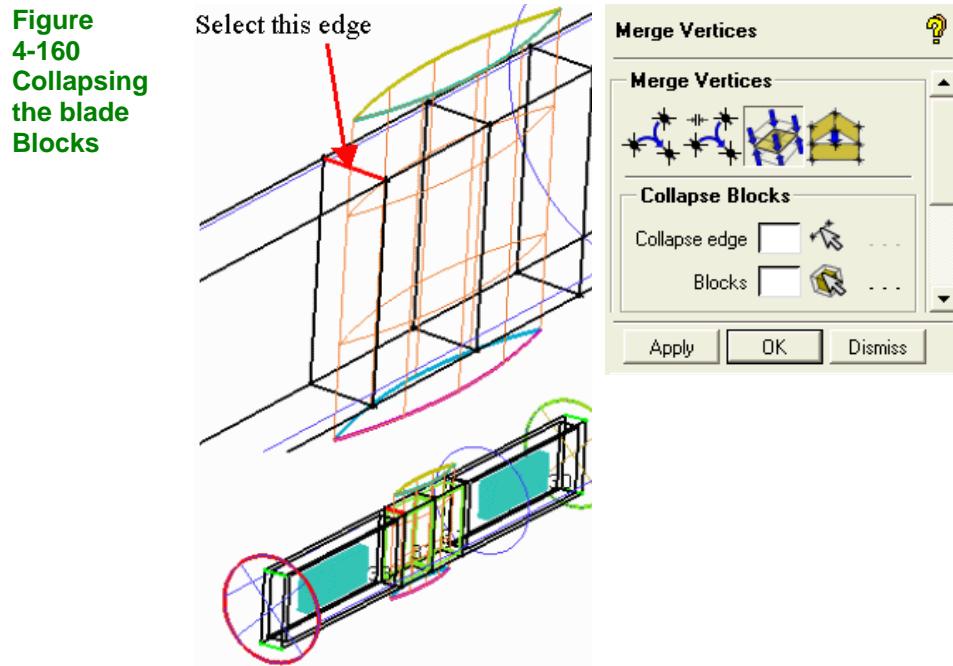
Right mouse click on SOLID>Add to part underneath Parts in the Display Tree, and it will open the Add to Part window. Select Blocking Material,

Add Blocks to Part  , and select the blocks of the blade as shown below, then press the middle mouse button to complete the operation.

**Figure
4-159
Assigning
the blade
blocks**

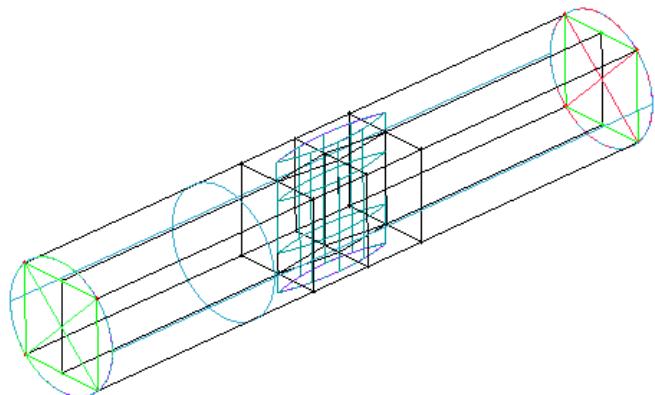


Now select Blocking > Merge Vertices  > Collapse Block  . Choose the edge that should be collapsed. In this case it is the shortest edge of the selected blocks. Select the two blocks shown in the figure below. Press Apply to Collapse the blocks.



After collapsing we get the model as shown below.

Figure 4-161
The Collapsed
Blocking

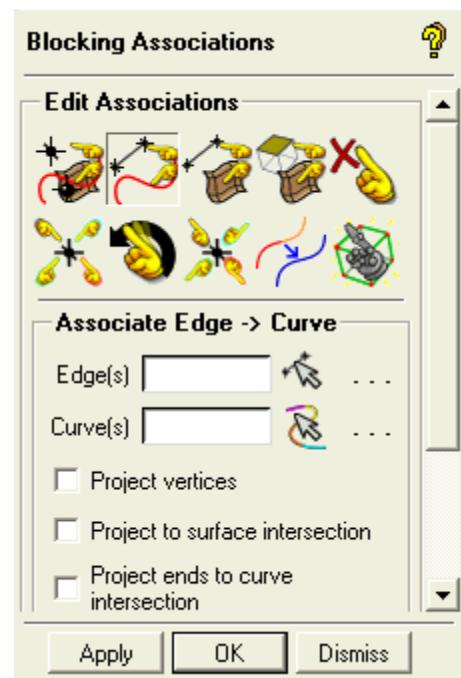


n) Edge to Curve Association on the Blade

Choose Blocking> Associate >Associate Edge to Curve . The Associate edge to curve window will open as shown below.

Note: Make sure Project Vertices is disabled.

Figure 4-162
Association Edge to Curve Window



You should associate the Edges and corresponding blade curves as shown below. 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.

Figure 4-163
**Blade edges to be
association to curves**

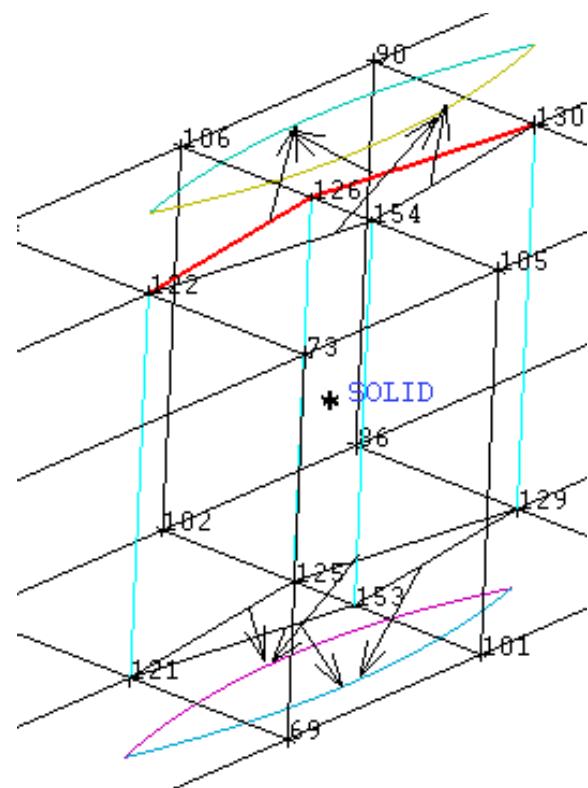
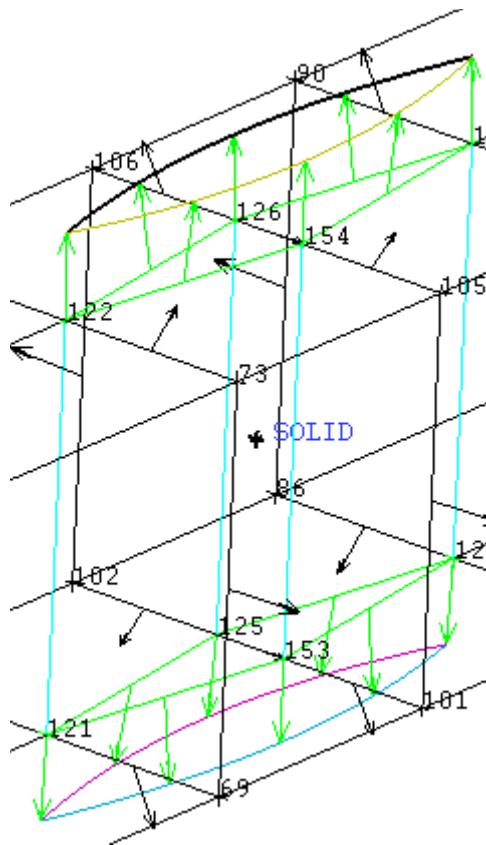


Figure 4-164
Blade edges
Associated to curves



o) Moving the Vertices

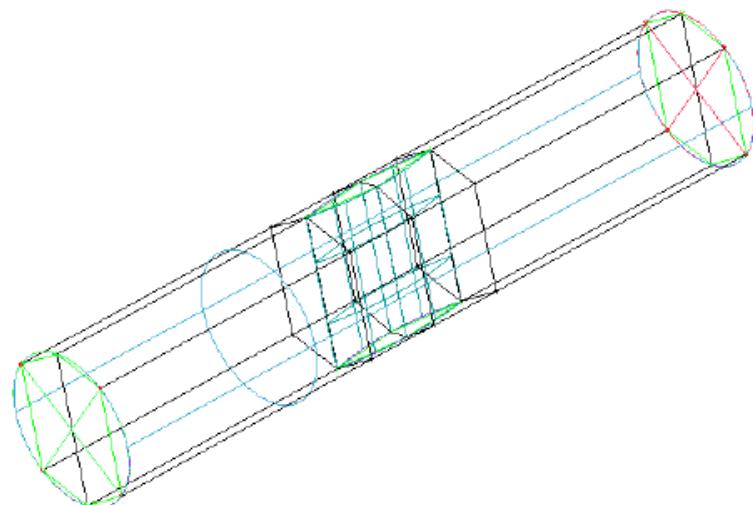
This section shows the user how to move all the associated vertices onto the geometry in one step.

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 below.

**Figure
4-165
The final
positions
of the
vertices
before
the O
grid**



p) 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.

q) Generating the O-Grid

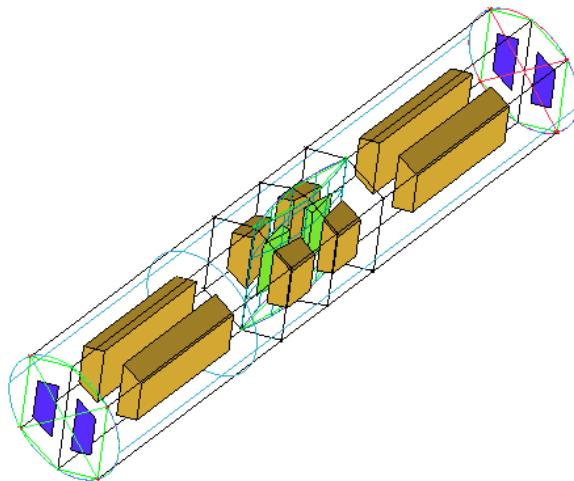
If the pre-mesh is generated at this point, the existing blocking would result in skewed cells on the four 'corners' of the pipe. Converting the existing H-Grid type topology to an O-grid type topology inside the pipe will produce a mesh that is low in skewness, with orthogonal grid on the pipe walls. 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 below. Press the middle mouse button to accept.

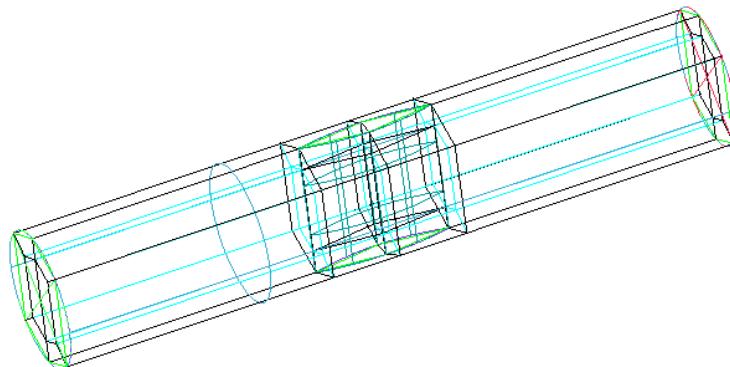
Similarly, press  and select the two INLET faces and two OUTLET faces as shown. Press the middle mouse button to accept, and Press Apply to create the O-grid.

**Figure
4-166
Add the
faces of
the outlet
and inlet
to O-grid**



After creating the O-Grid, the blocking will appear as shown.

**Figure
4-167
The O-grid**

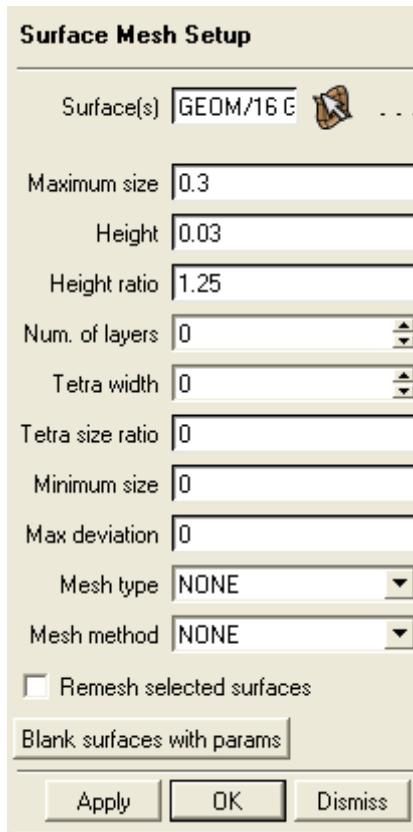


r) Defining Surface Parameters for the Mesh

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.

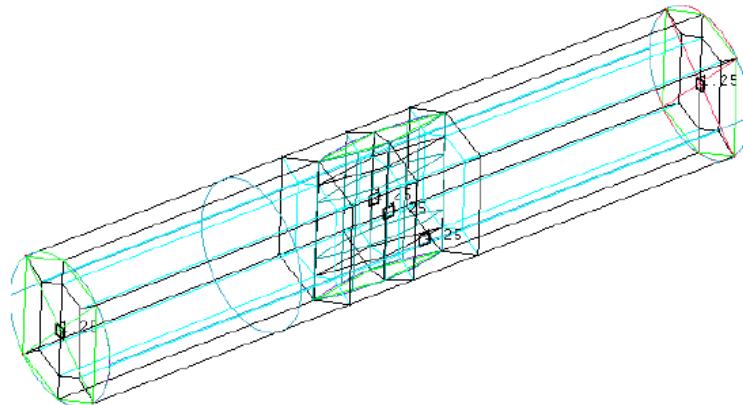
Select Mesh > Surface Mesh Setup and select the surface selection icon . Then select all the surfaces by box selecting the entire model or pressing “a.” Enter the Maximum Element size as 0.3, Height as 0.03 and Ratio as 1.25, as shown.

Figure 4-168
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.

**Figure
4-169
The
surface
parameters**



Switch OFF Surface > Hexa Sizes.

s) 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.

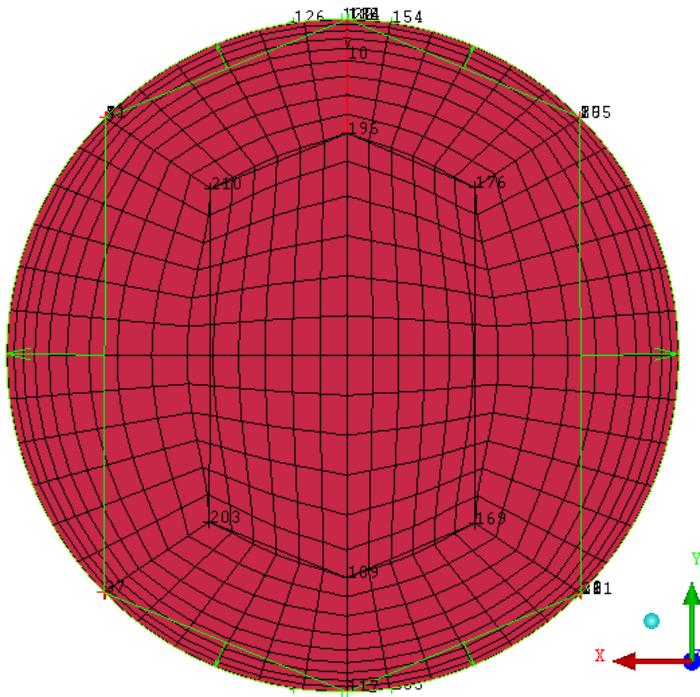
For the convenience of selecting the edges, right mouse click in the Display Tree to turn ON Vertices > Numbers and Edges > Bunching. Then make sure Vertices in ON. Zoom-in on the OUTLET area of the blocking.

Select Blocking >Pre-mesh Params >Update Sizes . Make sure Update All is toggled on (default), 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 and Wire in the Display Tree to display the mesh in Solid/Wire for better Visualization. The mesh will look like as shown below when viewing the OUTLET.

Figure 4-170
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 . 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 the figure below, 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 4-171
Setting edge meshing parameters

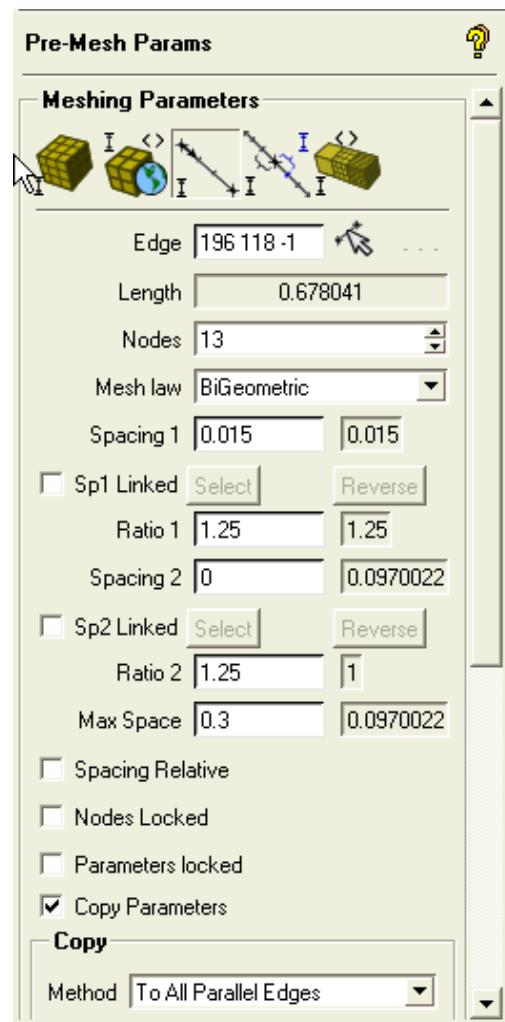
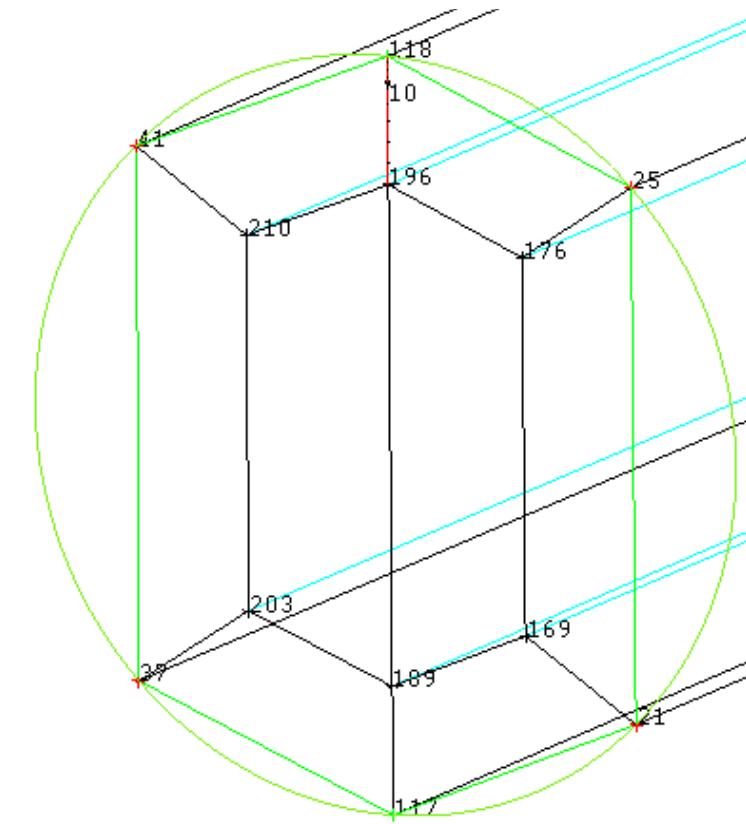


Figure 4-172
Selection of
edge for
changing
Parameters



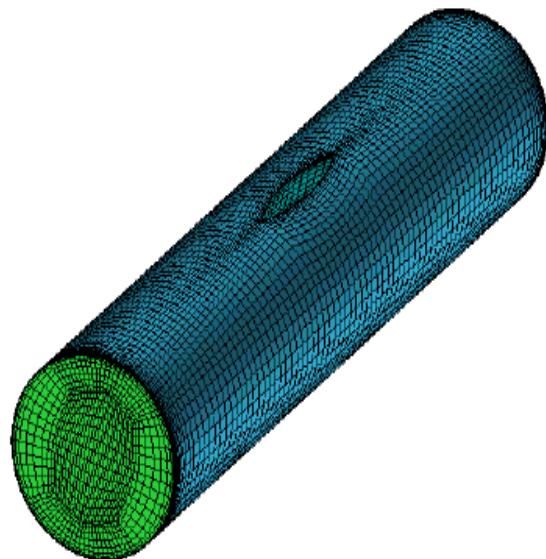
Note: Spacing1 is the first element size at vertex 118 while spacing2 is the first element size at vertex 196. Side 1 and Side 2 are indicated by the direction arrow that displays on the edge after it is selected.

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 and Wire if it is not already on.

**Figure
4-173
The final
mesh
displayed
in Solid
and Wire**



t) Checking mesh quality for determinants and angle

To check the mesh quality, select Blocking >Pre-mesh Quality Histogram



. Select the criterion as Determinant (2x2x2) and enter the Min-X value 0, Max-X value 1, Max-y height 12 and Num of bars 20. Press Apply. The histogram will be displayed in the lower right.

A value of determinant greater than 0.2 is acceptable for most commercial solvers.

Figure 4-174
Pre-mesh quality
window while selecting
Determinant 2x2x2

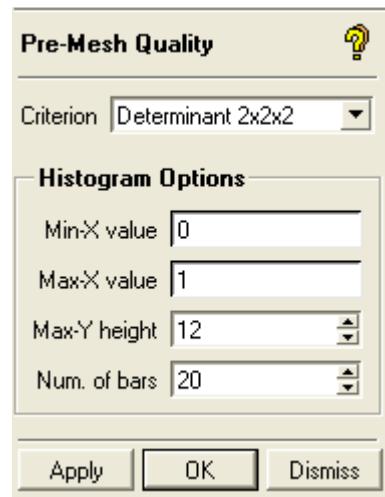
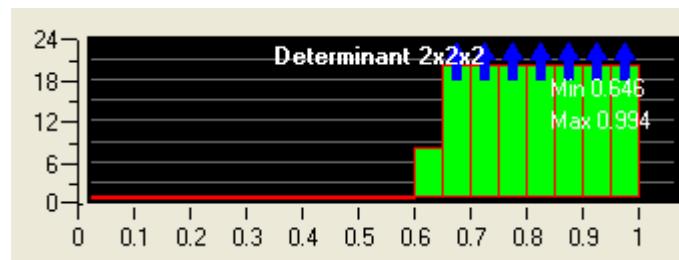
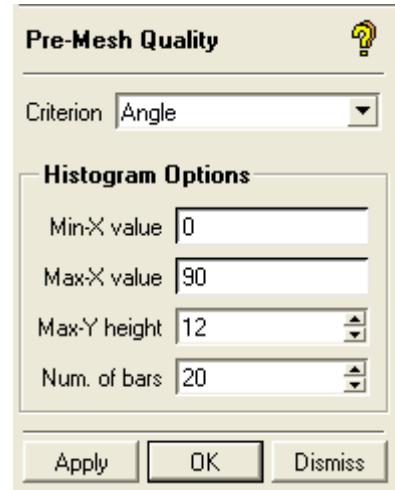


Figure 4-175
Histogram showing
Determinant 2x2x2



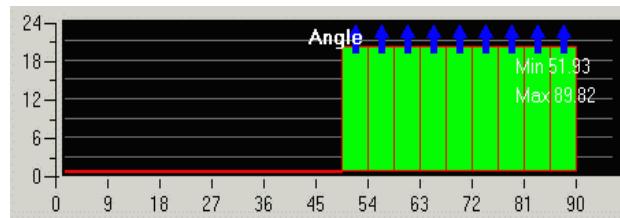
Then, in the Pre-Mesh Quality window at the upper left, select Angle from the Criterion pull down. Enter the values as shown below and press Apply. A new histogram will appear for the internal angles of elements as shown.

Figure 4-176
Pre-mesh quality
Window while
selecting Angle



An angle greater than 18 degrees is acceptable for most commercial solvers.

Figure
4-177
Histogram
showing
Angle



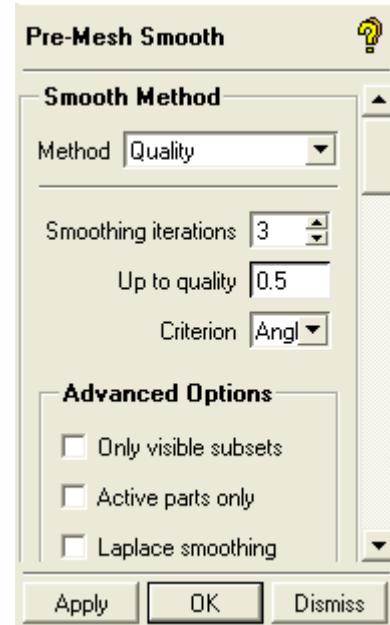
Note: As taught in the 3DPipeJunct example, to display cells of a particular determinant or angle value, select a histogram bar and then select **Show**. Cells within that range will be highlighted. The user should then inspect the elements and decide on a solution. In most of the cases, block vertices can be moved or edge parameters can be changed to improve the area.

u) Running Pre-mesh smoother

Before converting the Pre-mesh to an unstructured or structured mesh, the user may choose first to smooth the mesh.

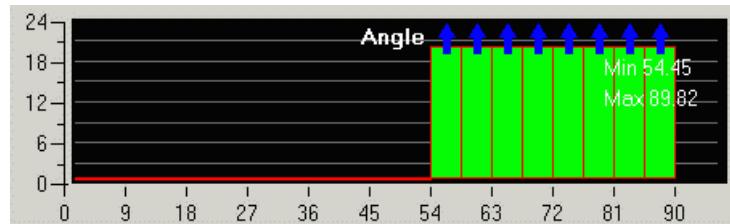
Select Blocking > Pre-mesh Smooth . The Pre-mesh smooth window will then appear. Select the Method as Quality. Select the Criterion as Angle and enter Smoothing iterations 3 and Up to quality 0.5 as shown.

Figure 4-178
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. 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 produce the mesh before smoothing. So at this point, you should not recompute the mesh.

**Figure 4-179
Histogram after
running
smoother**



v) 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.

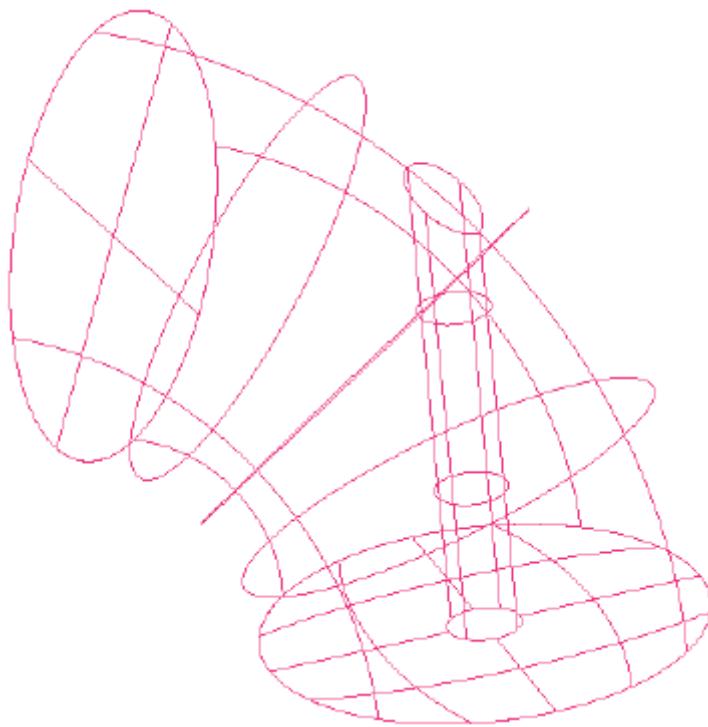
To write the mesh in a structured format, right mouse click in the Display Tree on Blocking > Pre-mesh > Convert to MultiBlock Mesh.

Finally, save the project.

4.2.7: 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

The input files for this tutorial can be found in the Ansys installation directory, under ..\v110\docu\Tutorials\CFD_Tutorial_Files>ElbowPart. Copy and open geometry.tin in your working directory.

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, and then press the middle mouse button to accept. Press Apply to create new part.

Refer to the figure below 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. Press Apply to accept that 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.

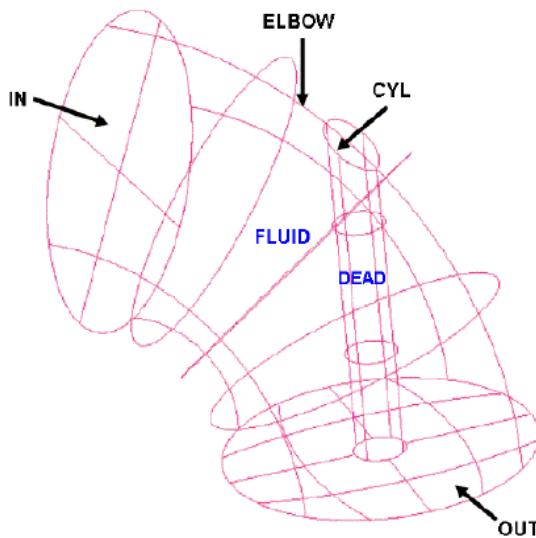
Enter FLUID for the Part name.

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 below. 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.

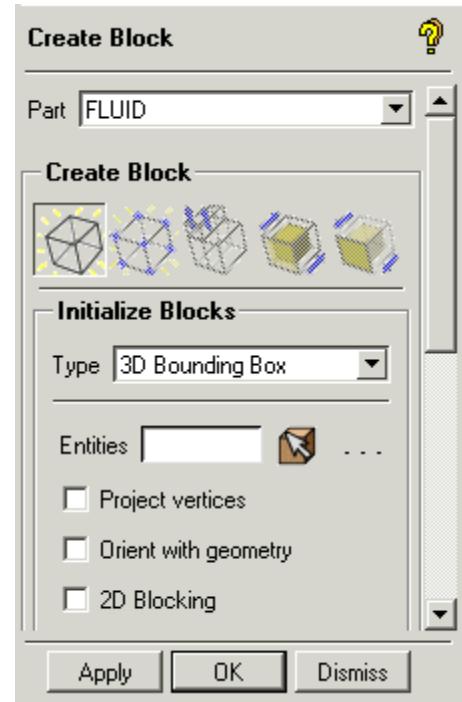
Figure 4-180
The geometry of
the Elbow Part
with the labeled
Surfaces and
Material



f) Blocking

Select Blocking > Create Block  > Initialize Block  . Choose 3D Bounding 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 4-181
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 below. The Split method should be set as Screen select by default. Create splits as shown below 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 the figure below.

Figure 4-182
Split blocking window

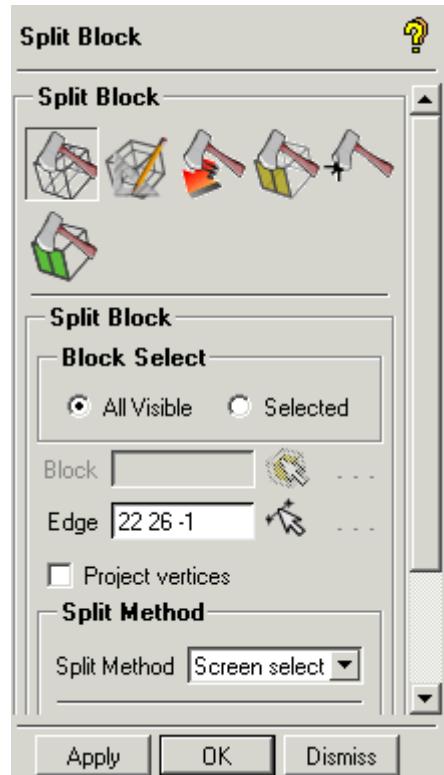
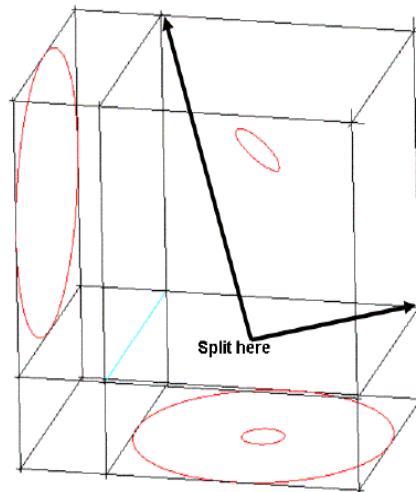
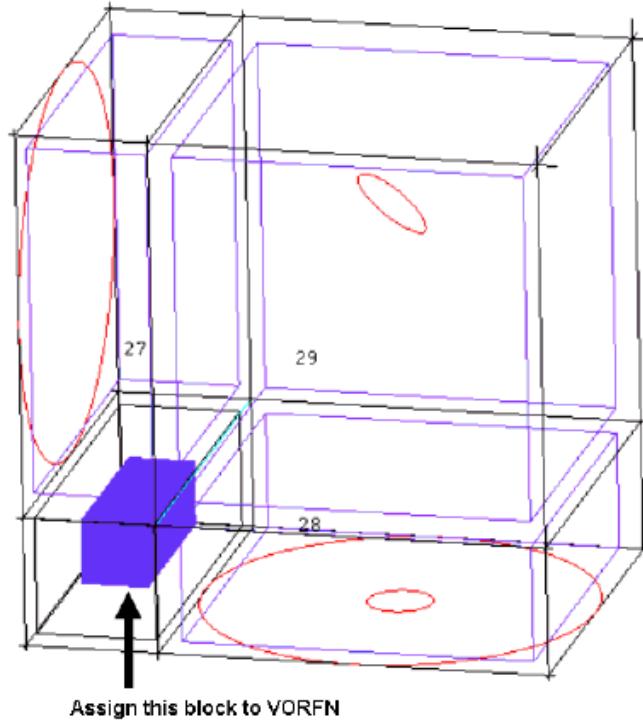


Figure 4-183
The two Block Splits



Next, select Blocking > Delete Block , and select the block shown highlighted below. Delete permanently should be turned OFF, then press Apply.

Figure 4-184
**Deleting a
block**

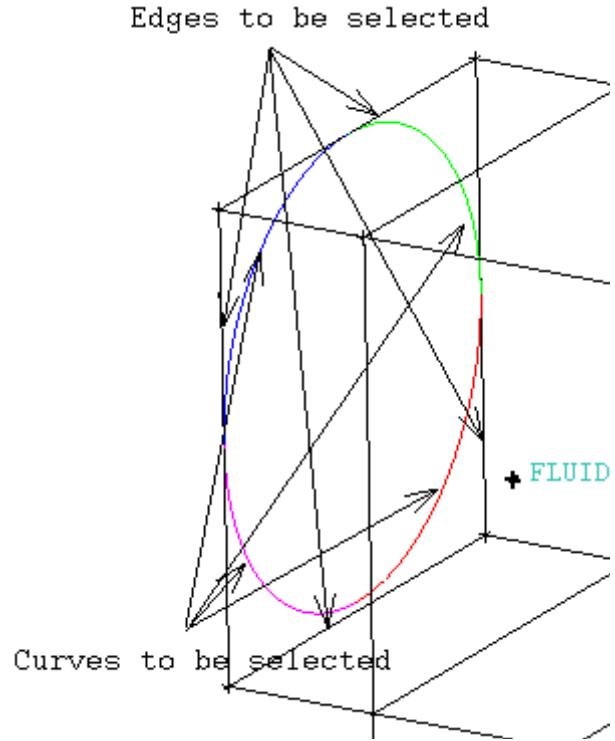


g) Fitting the Blocking to the Geometry

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, and press middle mouse button to complete selection. Next, select the four curves shown below, and press the middle mouse button to complete selection. Press Apply to associate the edges to the curves.

Figure 4-185
Selection of edges and curves for association

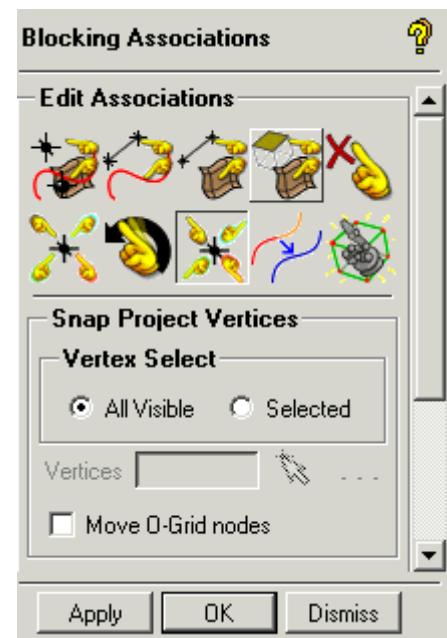


In the same way, select the surrounding four edges and curves of the OUT surface for association.

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.

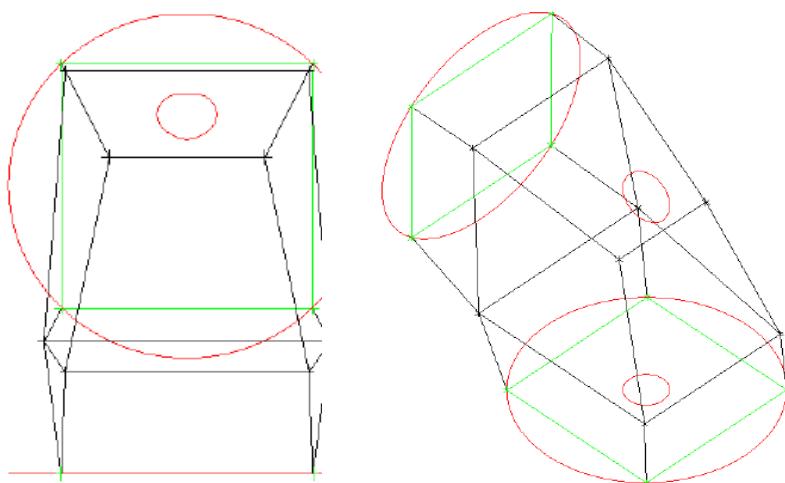
Select Blocking > Associate > Snap Project Vertices to open the window shown below. All Visible should be toggled on by default. Then Press Apply.

Figure 4-186
Snap Project vertices window



Note: View > Right can be used to orient the model as seen on the left side of the figure below. View > Isometric can be used to orient the model as shown on the right.

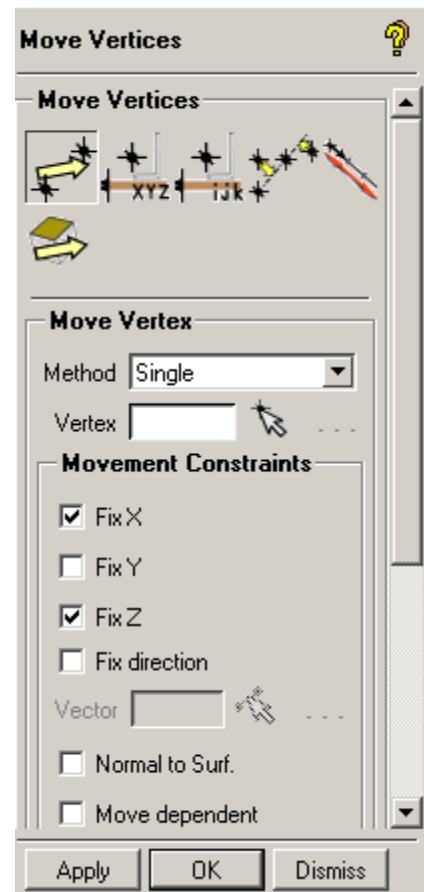
**Figure
4-187
Project
the
display
ed
edges**



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. Orient the model as shown below, and move the vertices to their new position as indicated. You'll need to left mouse click on the vertex and hold the button while you slide the vertex on the surface.

Figure 4-188
Movement constraints window



**Figure
4-189
Vertices to be moved**

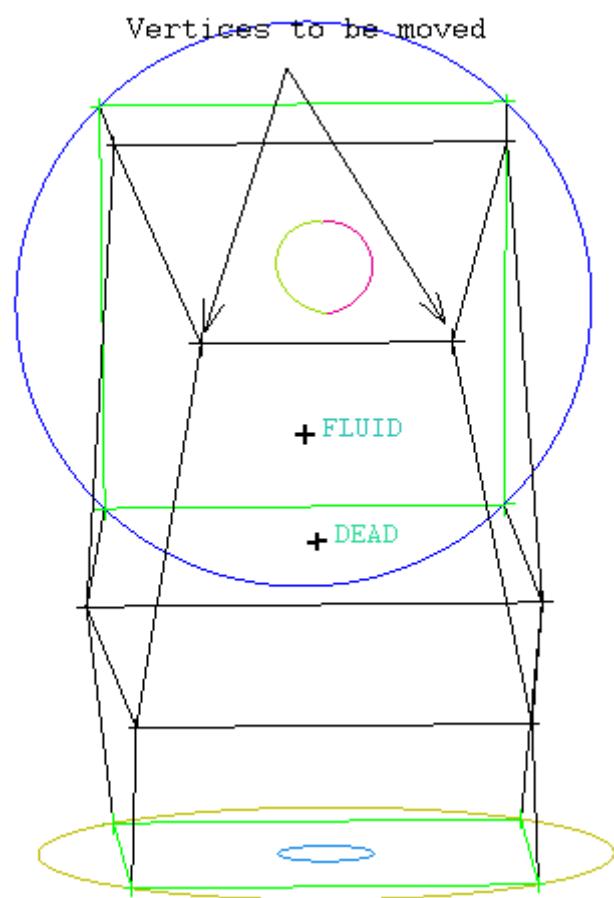


Figure 4-190
Vertex positions after
moving

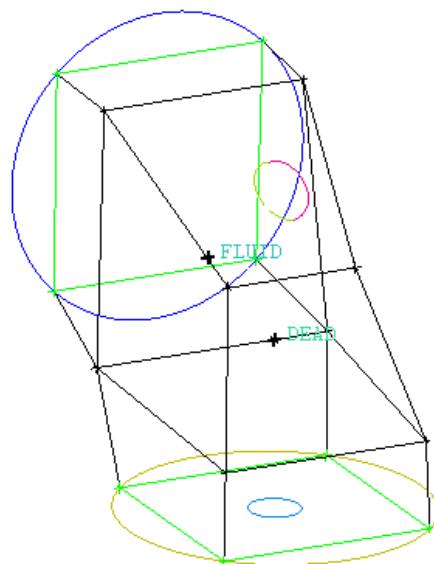
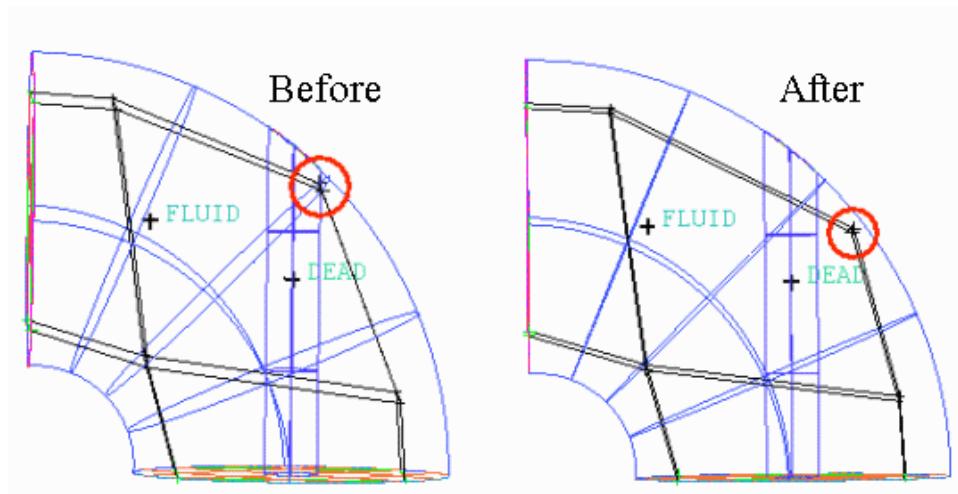


Figure 4-191 Vertex positions after moving

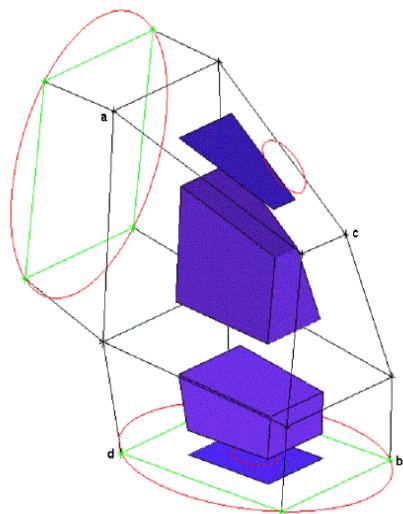


h) Creating the First O-grid

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.

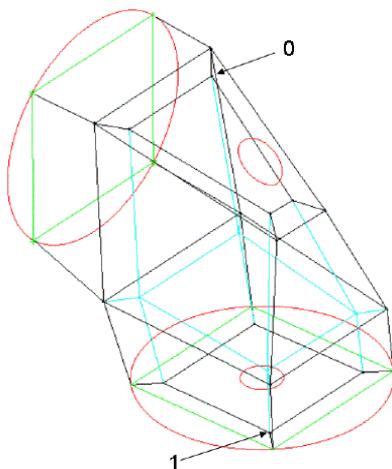
Select Blocking >Split Block  > Ogrid Block  . Within the O-grid Block window, press the Select Block(s)  button. A long, horizontal selection window will appear at the upper right. Press the last  button, called “Select diagonal corner vertices”  , and select two corners diagonally spanning the blocks in which we want the O-grid. Selecting the vertices ‘a’ and ‘b’, as specified in the figure below 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 the figure. Press the middle mouse button to finish selection, and press Apply to create the first O-grid.

**Figure
4-192
Creating
the first O
grid**



After creating the first O-grid, the geometry will appear as shown.

**Figure 4-193
Assigning the block to DEAD**



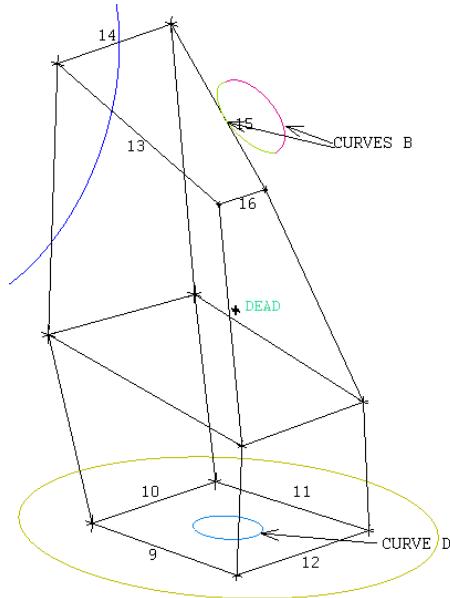
Next, the user will assign the material inside the cylinder to the DEAD part. This will remove this region from the mesh if it is computed with the DEAD part turned off.

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.

Right mouse click in the Display Tree on Part > Dead > Add to part.

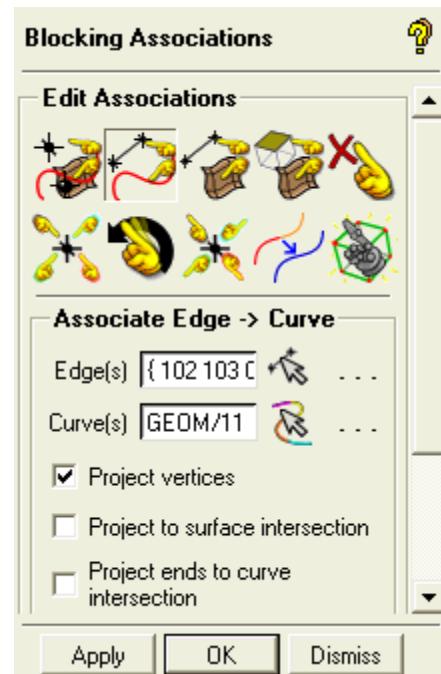
Select the Blocking Material, Add blocks to Part  button. Press the Select Blocks  button, and then box select the entire model or press 'v' on the keyboard. Press the middle mouse button, and then press Apply. The selected blocks will then be assigned to the DEAD part.

Figure 4-194
Projecting the inner block to the small pipe curves



Press Associate  > Associate Edge to Curve . Make sure that Project vertices are ON.

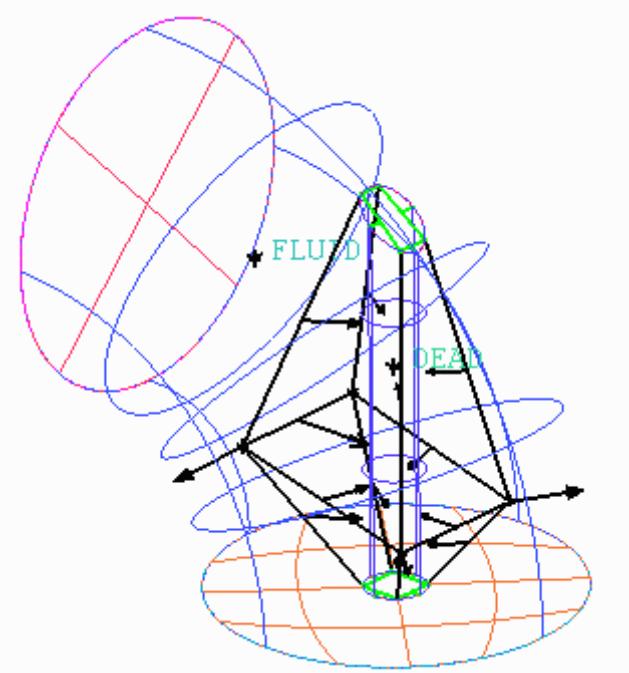
Figure 4-195
Associate edge to curve window



Now Associate Edges 9, 10, 11, and 12, to CURVE D using the figure below as a guide.

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 the figure below.

Figure 4-196
The edges to Curve
projection



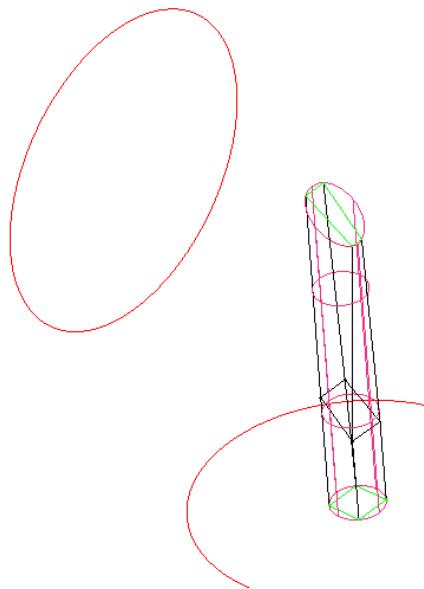
From the Display Tree, turn on Surfaces (if it is off).

i) **Moving the remaining vertices.**

Notice the association arrows pointing to the outside surfaces of the elbow part as shown in the figure below. 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.

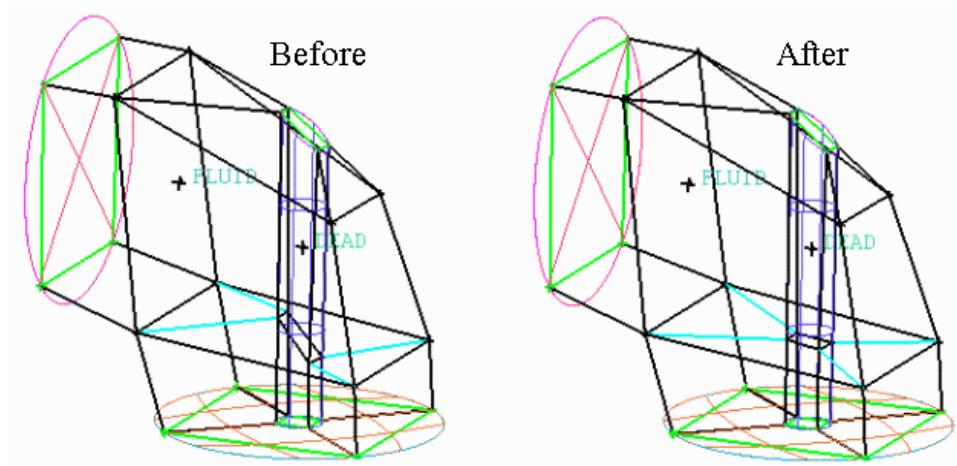
Go to Association > Snap Project Vertices .All Visible should be toggle on by default. Press Apply. The model should look like the figure below. Then press Reset at the lower right where the Index Control window is located.

Figure 4-197
After the projection



Use Blocking > Move Vertex > Move Vertex to improve the placement of the vertices on the cylinder. See below.
Turn the ELBOW part back on.

Figure 4-198
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. 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 below.

Figure 4-199
Select the FLUID material and
add faces for the O grid

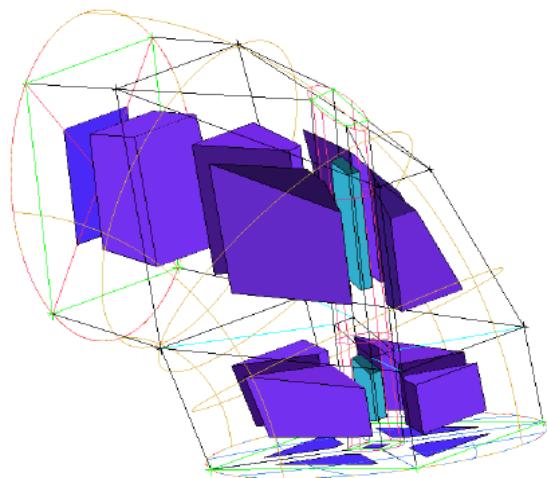
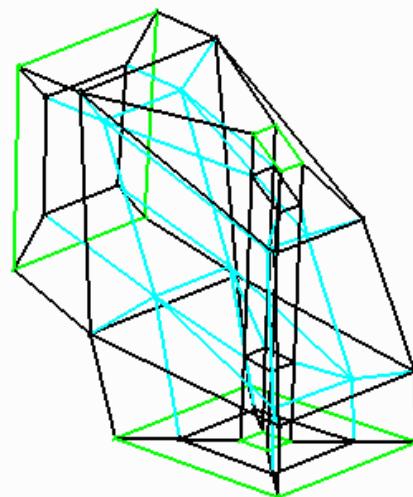
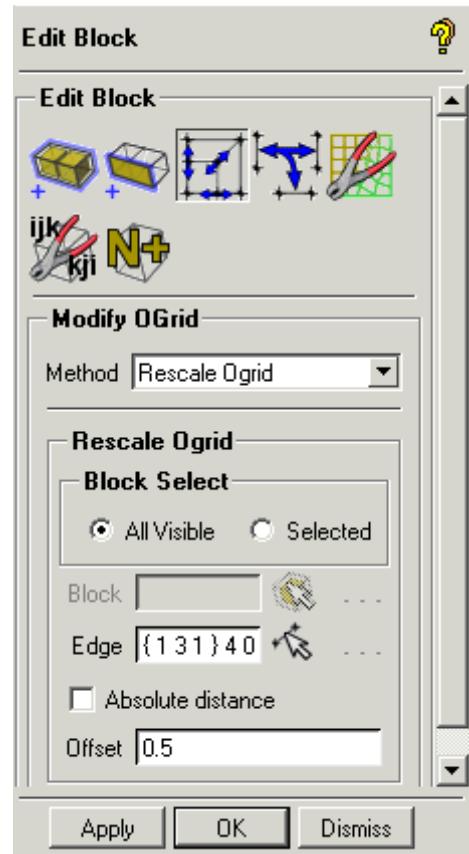


Figure 4-200
The second O-grid



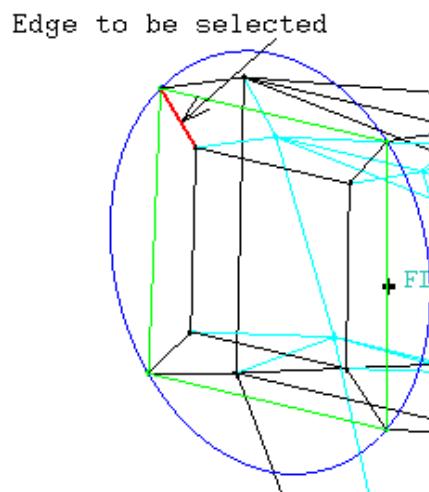
To resize the O-grid, select Blocking > Edit Block  > Modify O-Grid . Choose Rescale O grid from the dropdown.

Figure 4-201
Rescale O-grid window



Select any of the small radial edges of the second O-grid. The figure below 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.

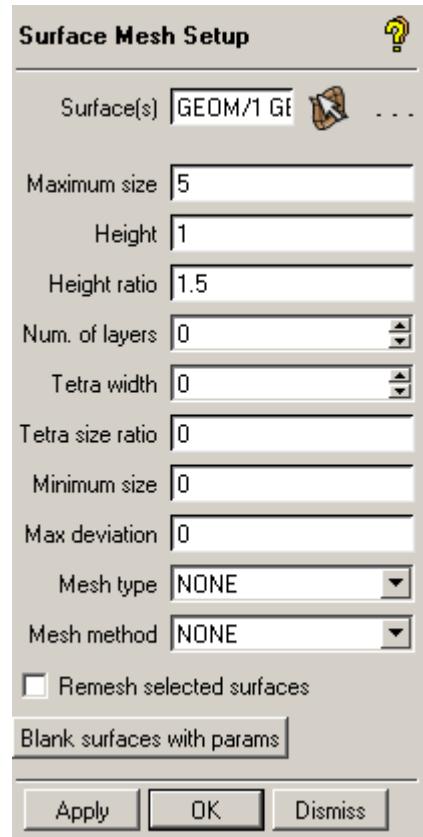
Figure 4-202
Edge to be selected for rescaling



k) Generating the Mesh

Select Mesh > Surface Mesh Setup  and box select all surfaces followed by clicking the middle mouse button or press “v” on the keyboard. Enter the following parameters as shown. Max Element size 5, Height 1, and Ratio 1.5. Then press Apply.

Figure 4-203
Surface mesh size window

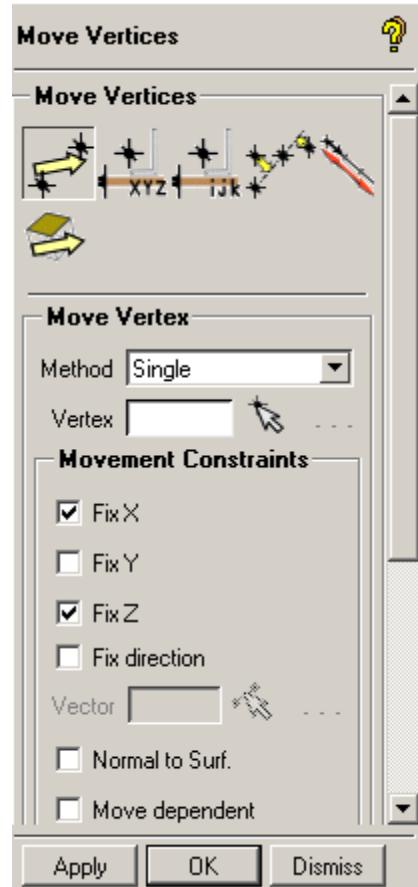


Before generating the mesh, there is an additional step that will improve the quality of the mesh.

Select Blocking > Move Vertex > Move Vertex and reposition the vertices indicated below to improve the denoted angle. Under Movement constraints, toggle on Fix X and Fix Z as shown. Then press the vertex selection button and left mouse click and hold to move the vertex down the CYL tube. Refer to the figure below for reference. The before and after pictures of the vertex positions are shown. 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 below, 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 4-204
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 4-205
Moving the vertices

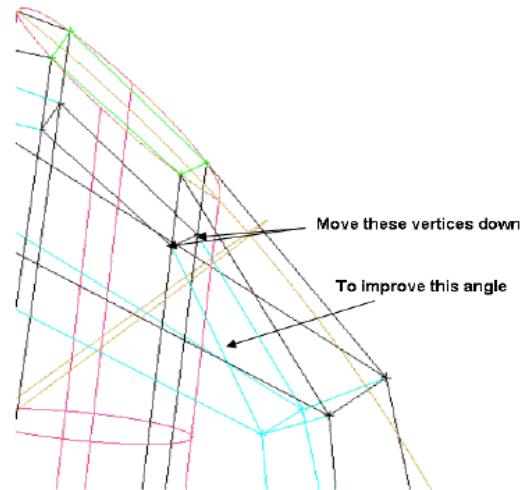
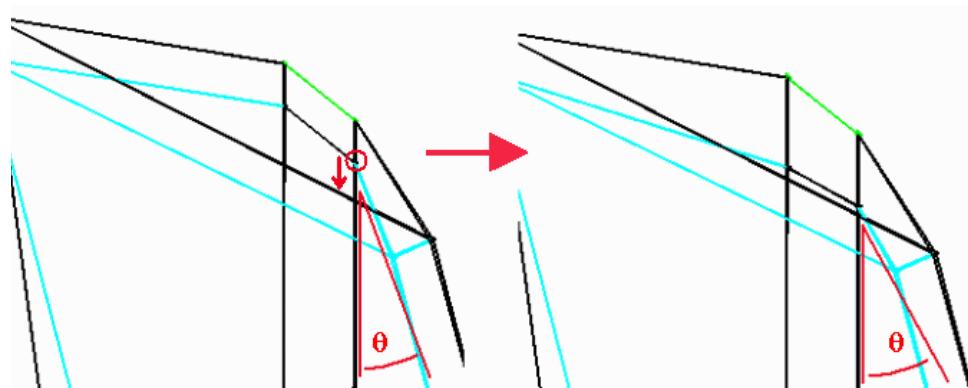
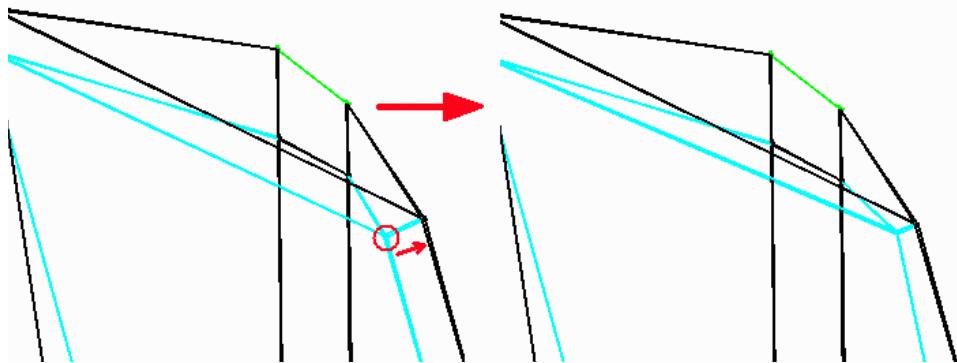


Figure 4-206
Vertex positions after moving, which shows the improved angle





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.

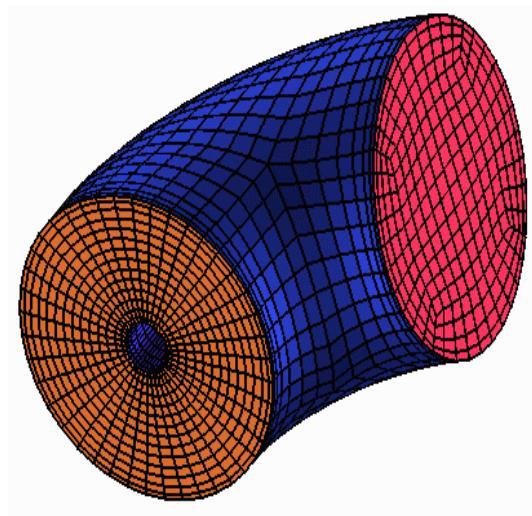
Before computing the Pre-mesh, turn the DEAD part off so that the mesh is not computed for that part.

In the Display Tree, right click on Blocking > Pre-mesh and ensure that Project Faces is checked (default). 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.

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 4-207
The final mesh

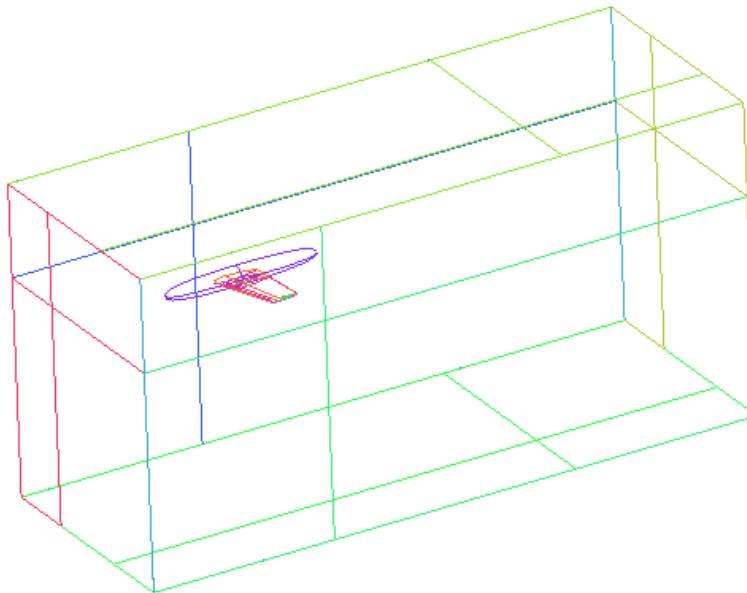


Save the blocking. File > Blocking > Save Blocking As.
Right click in the Display Tree on Blocking > Pre-mesh > Convert to Unstructured Mesh. This will write out the unstructured mesh to the default name hex.uns to the working directory. Then it will automatically load the mesh. You can resave (File > Mesh > Save Mesh As) to a different name if you'd like after that.

4.2.8: 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

The input files for this tutorial can be found in the Ansys installation directory, under ..\v110\docu\Tutorials\CFD_Tutorial_Files\WingBody. Copy and open geometry.tin in your working directory.

Figure 4-208
The Wing Body Far field Surface parts

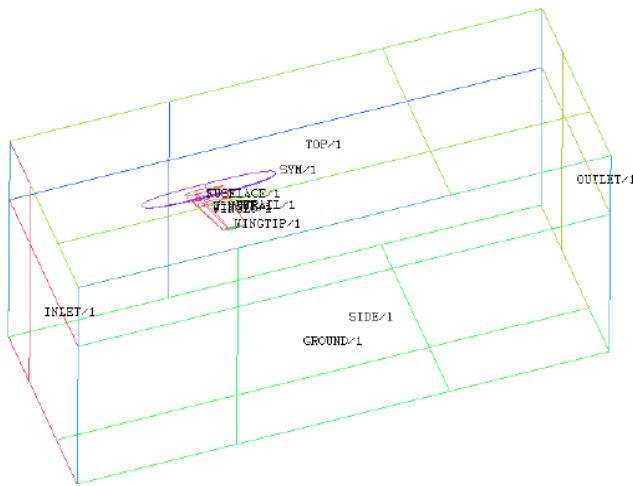
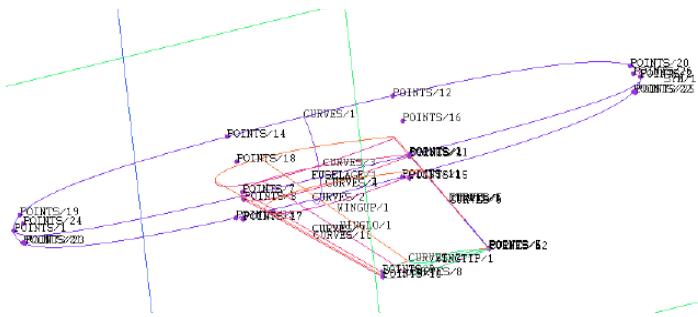


Figure 4-209
Curves and
points on the
fuselage and
wing



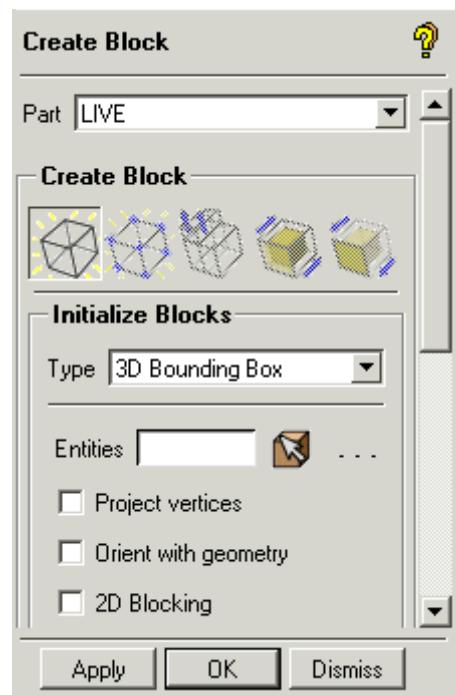
In this geometry, the points, curves, and surfaces have already been placed into separate part names. Thus, the user can go directly to the blocking process.

d) Blocking

Select File > Replay Scripts > Replay Control to start recording all the commands executed while blocking.

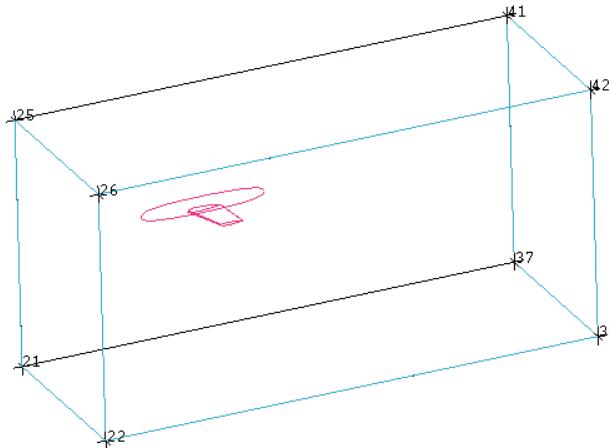
Press Blocking > Create Block > Initialize Block to open the Create Block window. The default Type is 3D Bounding Box. Verify that this is shown by the Type. Enter the Part name as LIVE, and press Apply without selecting anything. This will create the initial block around everything.

Figure 4-210
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.

Figure 4-211
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. 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 4-212
Split points

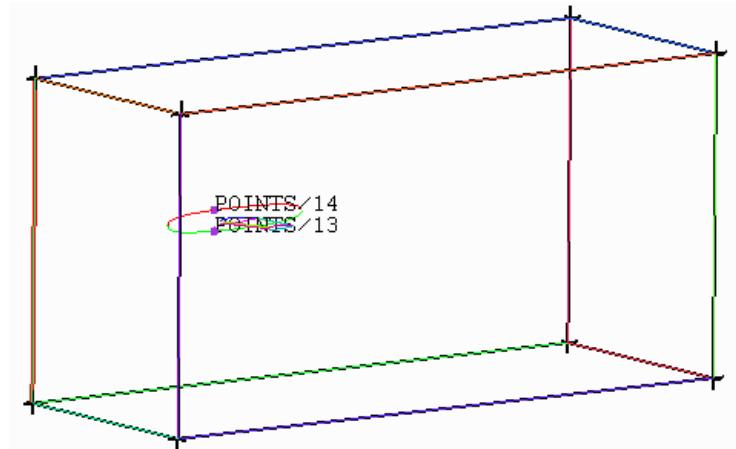
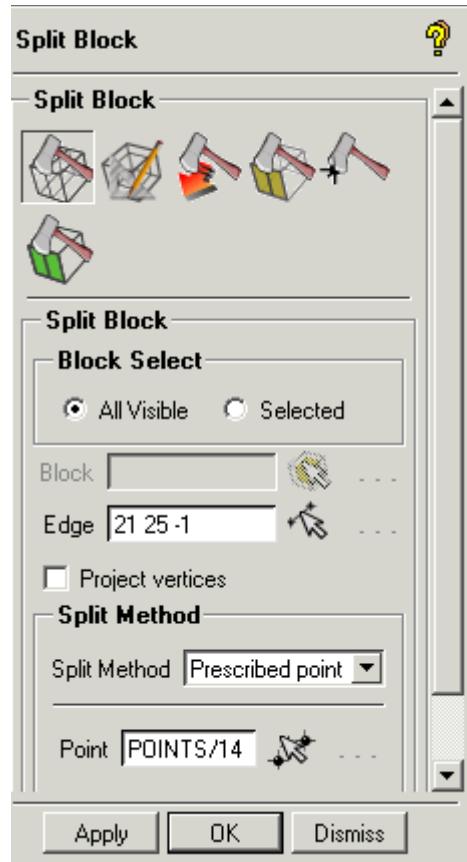
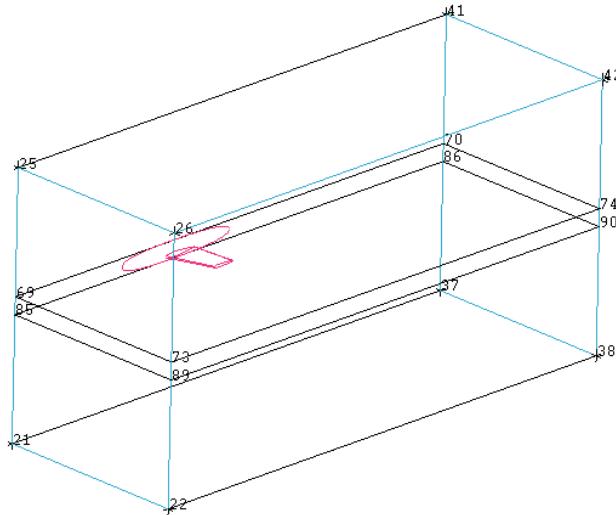


Figure 4-213
The Split block window



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 here. Switch off Points to have a better view. The blocking should now look like the figure below.

Figure 4-214
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.

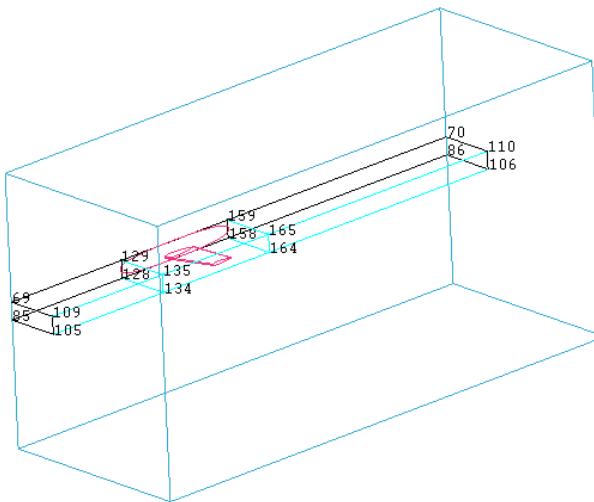
In the Index control, use Select corners to further restrict the blocking by selecting vertices 105 and 70.

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 the figure shown below.

**Figure
4-215
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. Turn on Points from the Display tree when required.

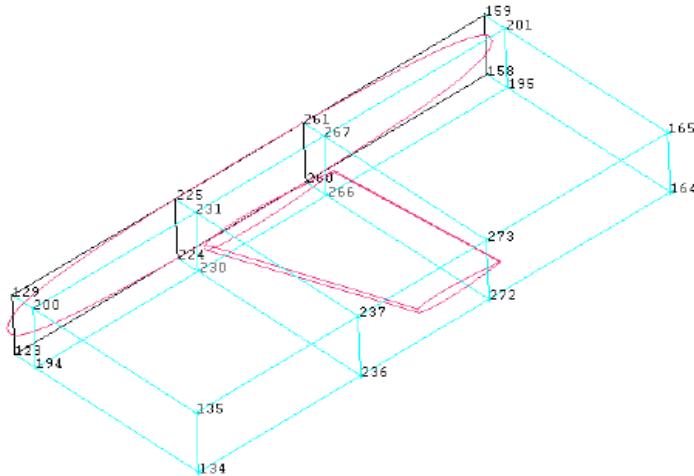
Blocking > Split Block > Split Block : Select the edge connecting vertices 129 and 135 and split this edge by the prescribed point, POINTS/18, which is near the base of the wing.

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 below.

**Figure
4-216
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.

**Figure
4-217
Splits
around
the wing**

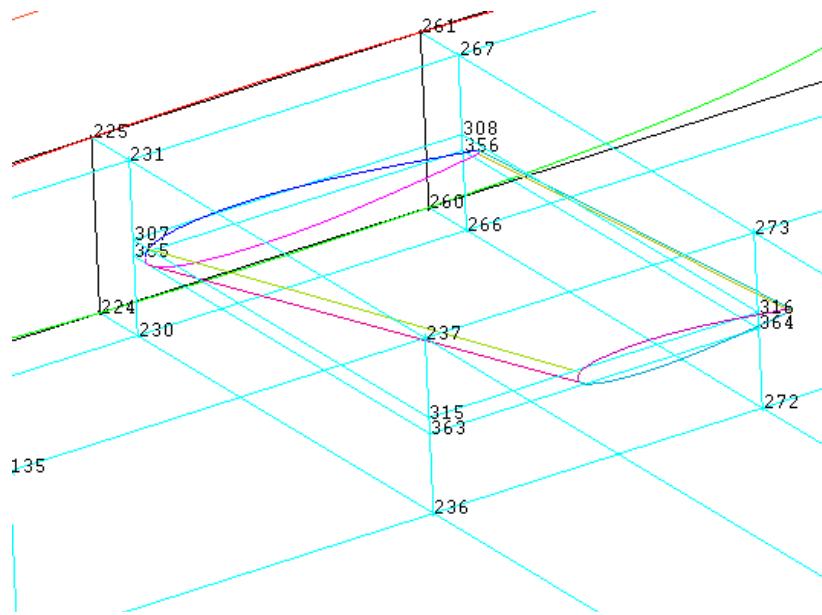


Figure 4-218
Create part window

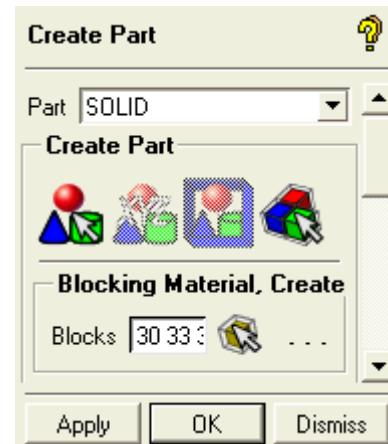
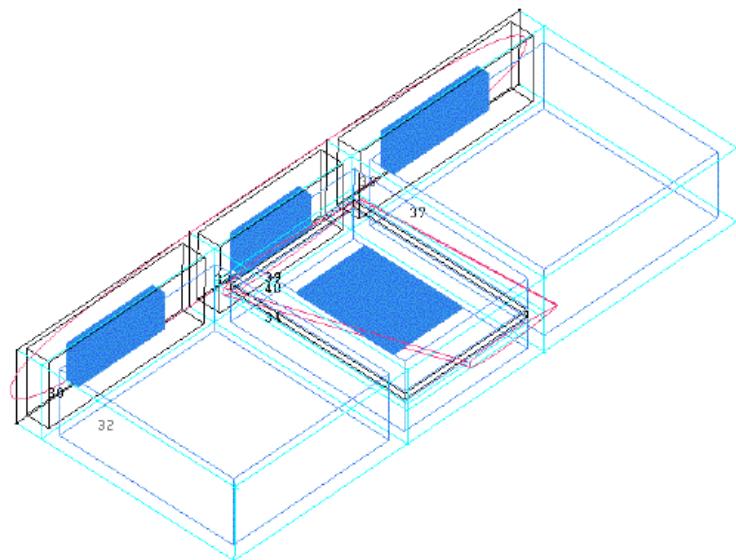


Figure 4-219
Assign these
blocks to
SOLID



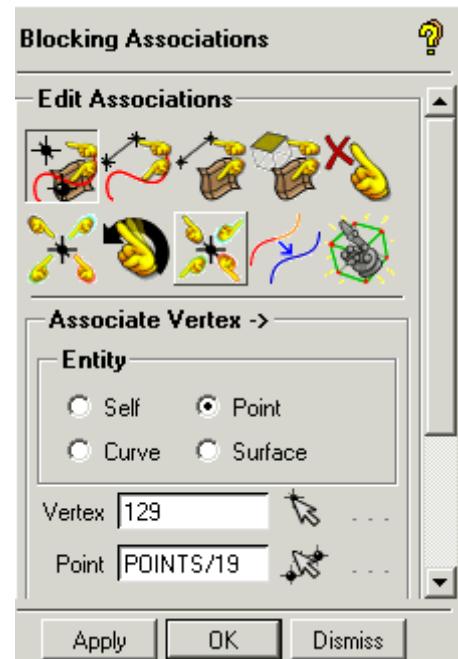
g) Fitting the blocking to the fuselage and wing

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.

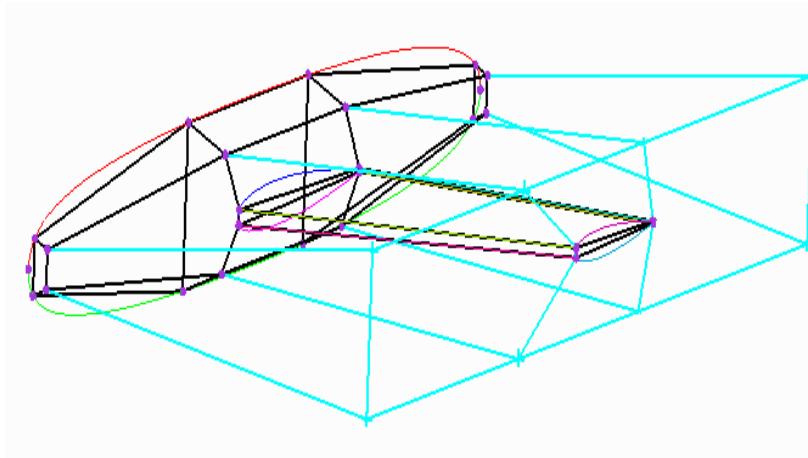
Select Blocking > Associate  > Associate Vertex  and you should see the window as shown. Make sure the Entity type to associate to its Point (default). Select the vertex 129. Then select the point POINTS/19. Press Apply. This will assign the association and move the vertex to the point all in the same step.

Figure 4-220
Associate vertex to entity window



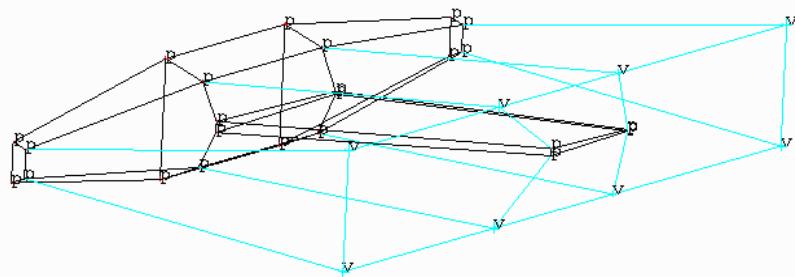
Similarly place other vertices to the corresponding points shown below.

**Figure
4-221
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 > Proj type in the Display Tree. Then turn ON Vertices. You should see a “p” next to each point-associated vertex as shown in the figure below. A “v” stands for a volume vertex while a “c” means a curve vertex and an “s” stands for a surface-associated vertex.

**Figure
4-222
Display
Proj
Type**

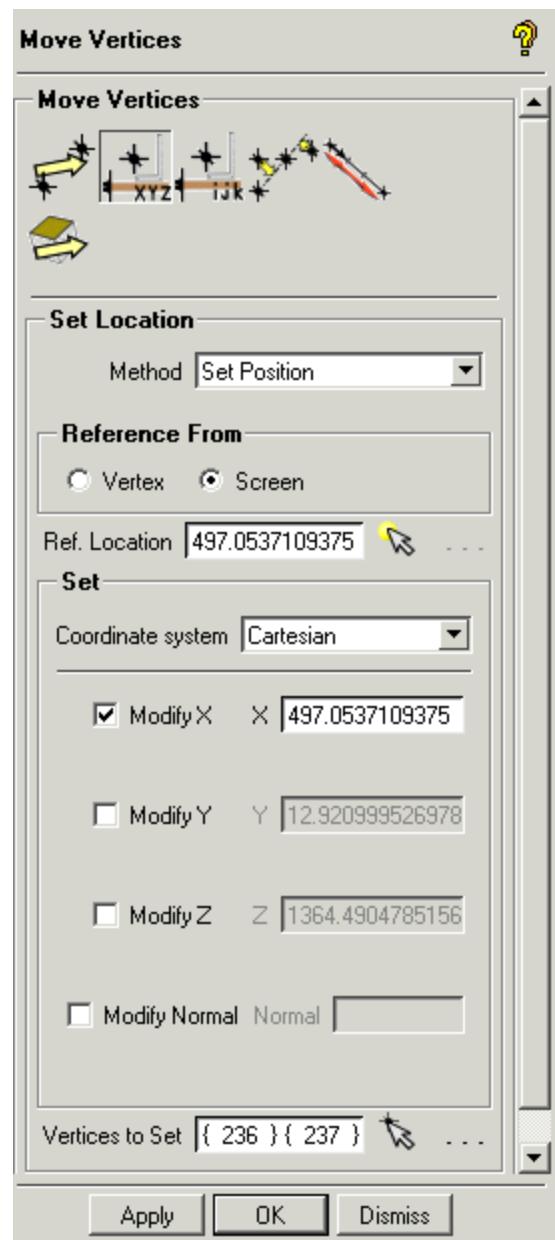


To align the volume vertices near the wing tip, select Blocking > Move

Vertex > Set Location .

Switch on Vertices > Numbers and Switch on Geometry > Points > Show Point Names, and turn ON Points. Toggle on Screen and 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 4-223
Set location window



Similarly, set the location for vertices 272 and 273 using the Reference Point as POINTS/5.

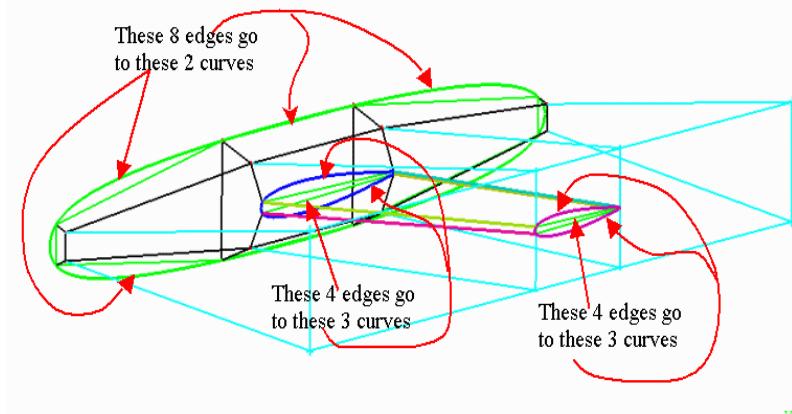
Switch On Geometry > Curves and switch OFF Geometry > Points from the Display Tree.

Select Blocking > Association  > Associate Edge to Curve .

Associate the edges to the curves as shown. The green colors of the edges indicate that they are associated to a curve.

Figure 4-224

**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.

Then press Apply to create the O-grid. The O-grid should appear as shown in the second figure below.

Figure 4-225
O-grid
selection

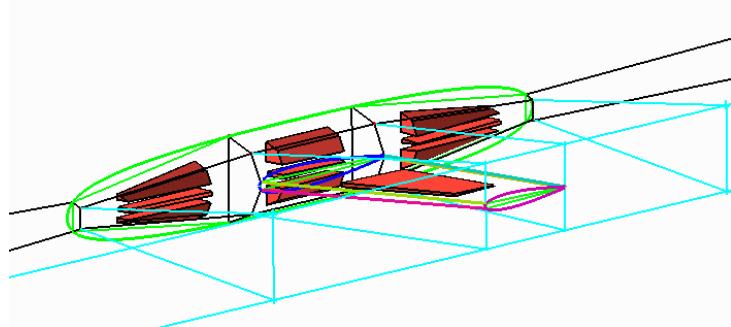
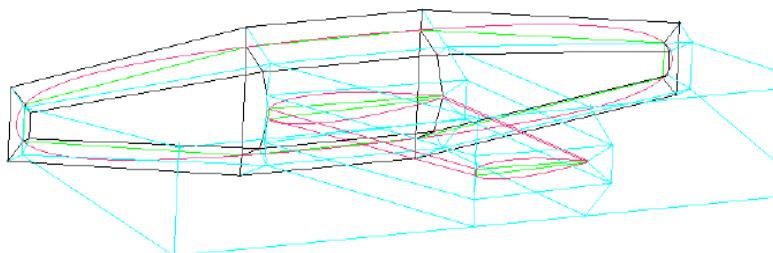


Figure
4-226
Blocking
after
creating O-
grid

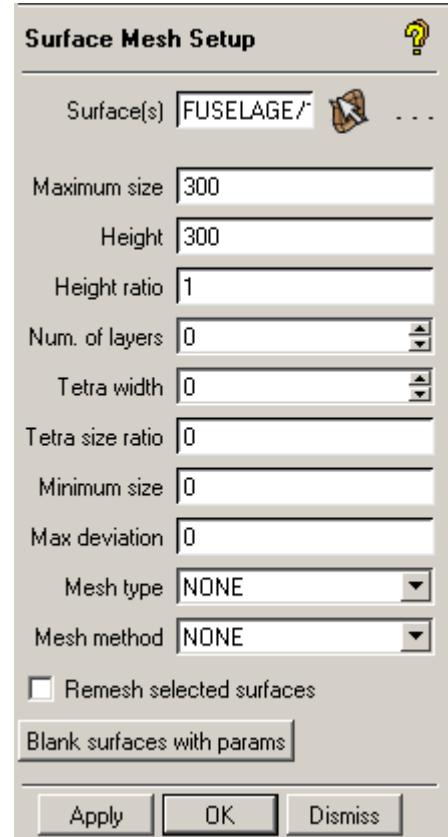


i) Setting Mesh Parameters on Surfaces for an Initial Mesh

- Press **Mesh > Surface Mesh Setup** . 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.

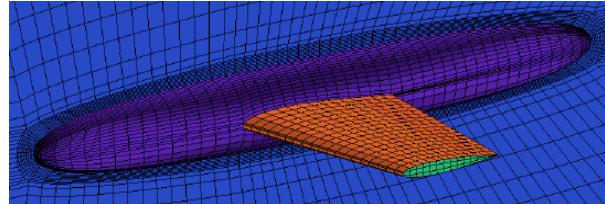
Figure 4-227
Setting mesh sizes on surfaces



- 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 below.

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 4-228
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.

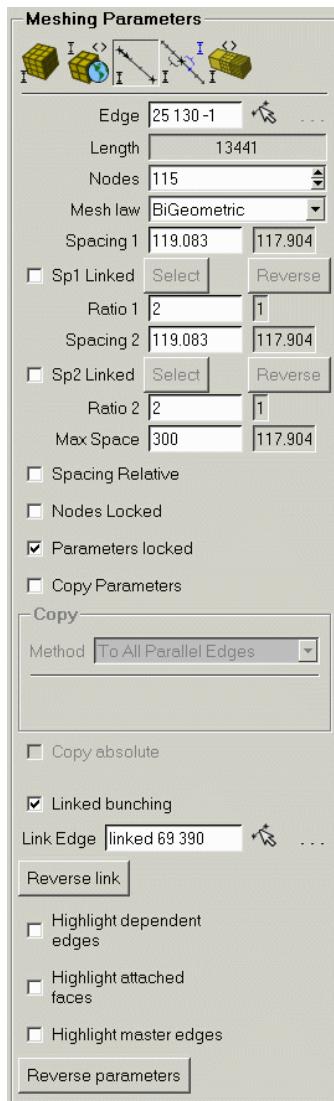
The mesh is distorted in the farfield region. The Linked Bunching function will be utilized to link the mesh distribution.

Switch off Pre-mesh in the Display Tree.

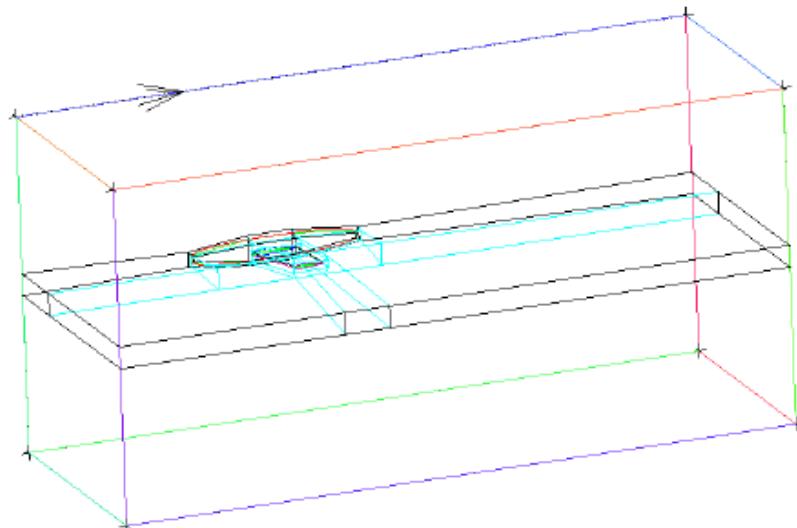
Select Blocking > Pre-Mesh params > Edge Params . The Meshing Parameters window will open. Select the edge to be modified indicated in the figure below. The selected edge has an arrow displayed on it, which indicates side 1 and side 2 of the edge. Side 1 is the back of the arrow, while side 2 is the front on the arrow.

Hexa Meshing

Figure 4-229
Edge meshing parameters
window

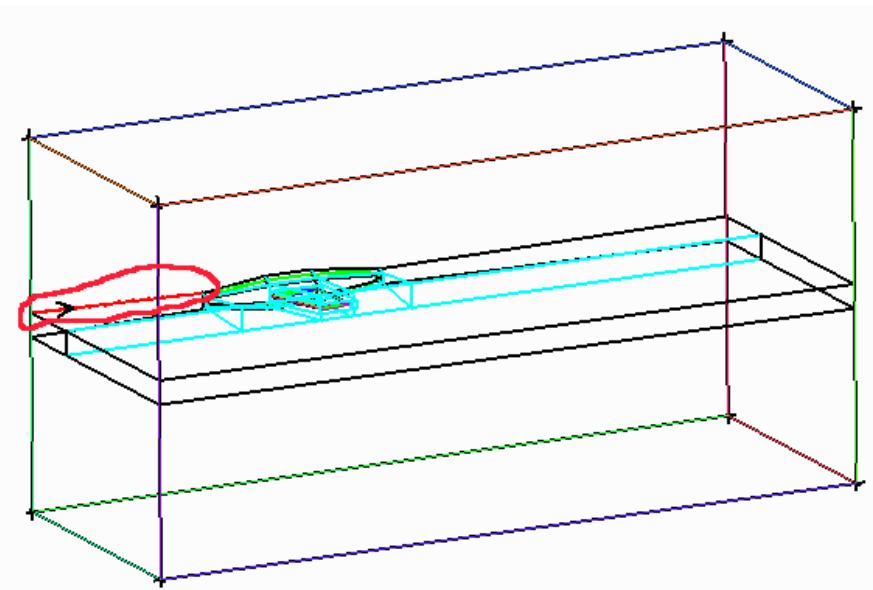


**Figure
4-230
Select this
edge for
setting
edge
parameters**



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 below. Remember that the beginning of the larger edge is shown by a white arrow.

**Figure
4-231
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 below. 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 4-232
Select the
edges to link

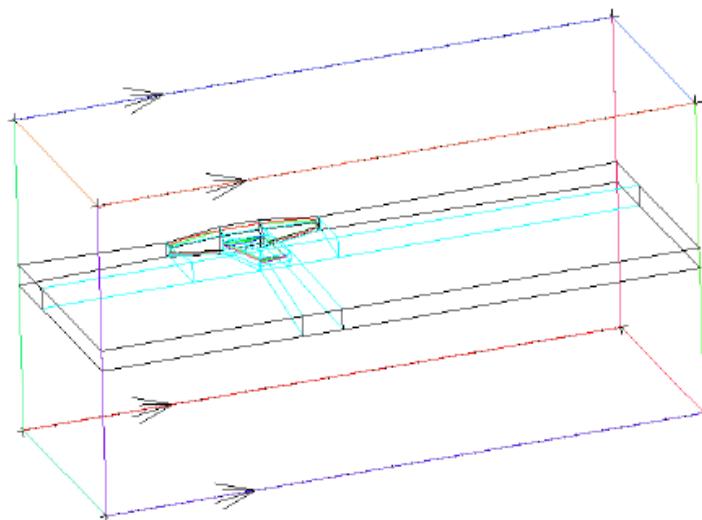
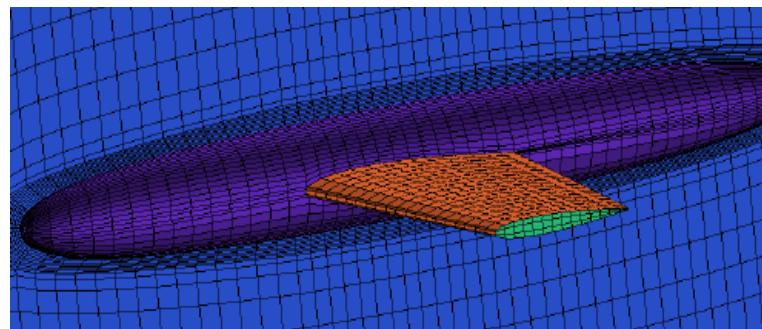


Figure 4-233
New mesh
distribution
after Linked
bunching



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.

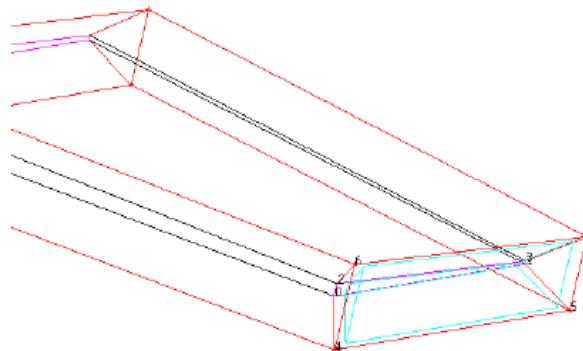
Compute the Pre-mesh with Project faces turned ON (default) from now on, so that everything projects as in the final mesh.

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 4-234
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.

4.3: 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.

4.3.1: 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

4.3.2: 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.

4.3.3: 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 an O-grid is generated, the size of the O-grid is scaled based upon the Factor (Offset) in the Blocking > Edit Block > Modify O-grid parameter window. The user may modify the length of the O-grid using the 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.

4.3.4: 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 Blocking > Pre-Mesh Params > Edge Params

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

Default (Bi-Geometric Law)

Uniform

Hyperbolic

Poisson

Curvature

Geometric1

Geometric2

Exponential1

Exponential2

Bi-Exponential

Linear

Spline

The user may modify these existing laws by Applying pre-defined edge meshing functions, accessible by selecting Mesh law as From-Graphs in Blocking > Pre-Mesh Params > Edge Params.

This option yields these possible functions:

Constant

Ramp

S curve

Parabola Middle

Parabola Ends

Exponential

Gaussian

Linear

Spline

Note: By selecting Mesh law as From-Graphs, 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.

4.3.5: 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 skewness 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.

4.3.6: Refinement and Coarsening

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

Refinement

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

Coarsening

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	281
------------------------	--	-----

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.

4.3.7: Replay Functionality

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

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 Scripts > Replay Control. 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

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	282
------------------------	--	-----

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.

4.3.8: Periodicity

Periodic definition may be applied to the model in ICEM CFD Hexa. The Periodic nodes function, which is found under Blocking > Edit Block > Periodic Vertices, 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.

4.3.9: Mesh Quality

The mesh quality functions are accessible through Blocking > Pre-Mesh Quality Histograms. 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 to 1. 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

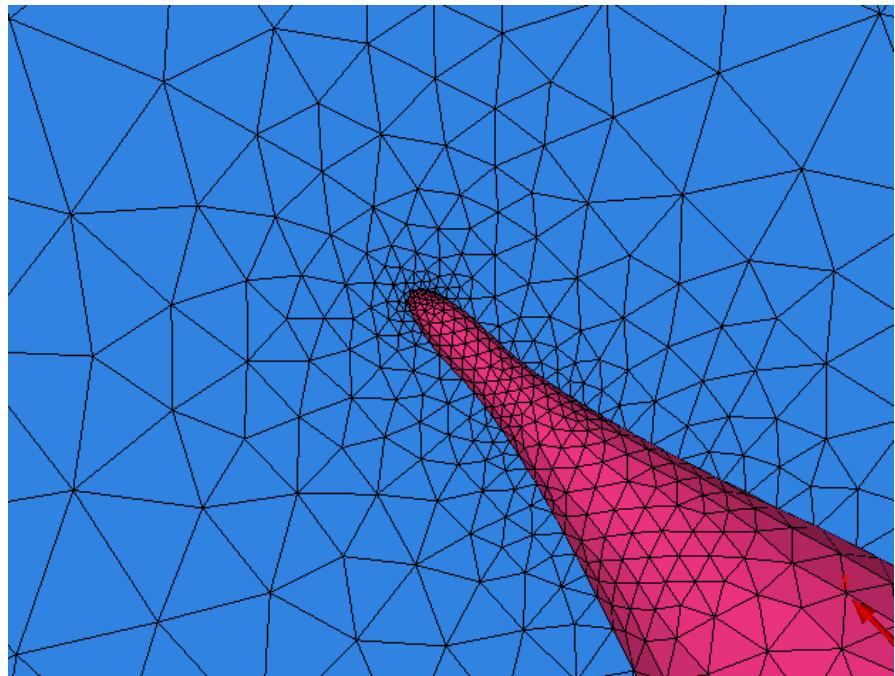
distorted will increase an elements distortion, giving a high degree of warpage. The y-axis is the scale for the number of elements represented in the histogram -- a value determined by the assigned Height. The x-axis, which ranges from a Min of 0 to a Max of 90, is the degree of warpage that an element experiences.

4.4: Tetra

Tetra Meshing

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

Figure 4-235
This mesh
was
generated
using ANSYS
ICEM CFD
Tetra. The
model has
approximately
550,000
tetrahedral
elements.



4.4.1: 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 ICEM CFD 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

Curvature/Proximity Based Refinement 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 Delaunay 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 ICEM CFD Tetra

The following are possible inputs to ANSYS ICEM CFD Tetra:

B-Spline Curves and Surfaces

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	287
------------------------	--	-----

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 ICEM CFD 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 ICEM CFD Tetra

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, 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.

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	288
------------------------	--	-----

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, and 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 ICEM CFD Tetra

Curvature /Proximity Based Refinement

If the maximum tetrahedral size defined on surface parts is larger than a geometric entity in the specified part, the user must employ the Curvature/Proximity Based Refinement limit. The user can specify a 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 Curvature/Proximity Based Refinement 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 Curvature/Proximity Based Refinement 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 ICEM CFD 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 ICEM CFD 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

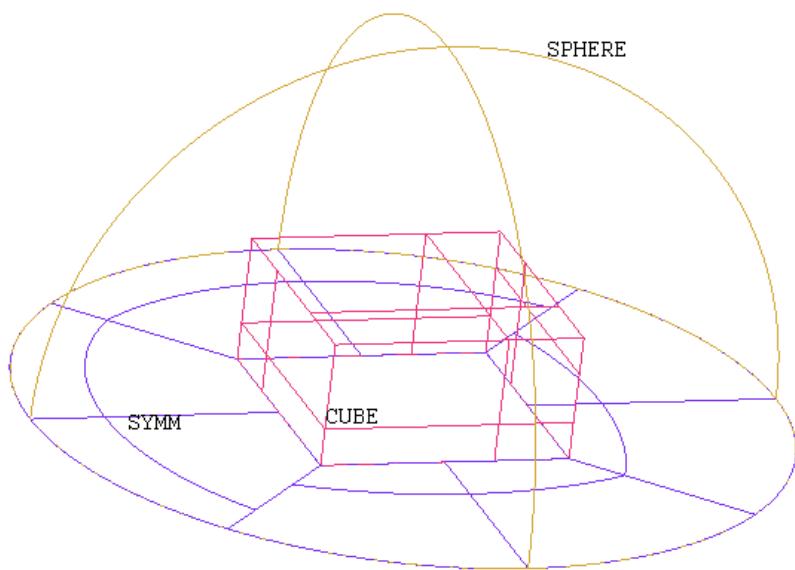
The mesh Density definition for ANSYS ICEM CFD Tetra allows users to control the tetra size locally where no geometry is present. Densities can be of different shapes: point, sphere, line, arbitrary volume.

4.4.2: 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
4-236
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) Starting the Project

From UNIX or DOS window, start ANSYS ICEMCFD. The input files for this tutorial can be found in the Ansys installation directory, under
`..v110/docu/Tutorials/CFD_Tutorial_Files>SphereCube`. Copy and open the tetin file `geometry.tin` in your working directory.

c) Parts Creation

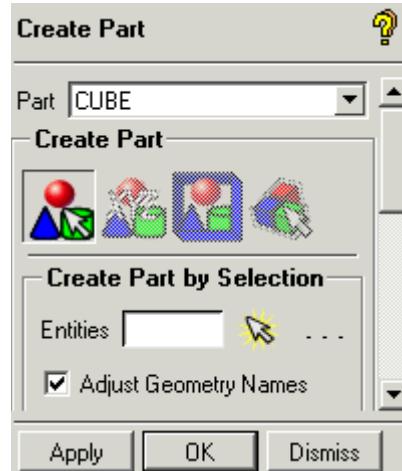
All points, curves and surfaces are initially assigned to one part, GEOM. The user needs to create and assign separate parts for surfaces, curves and bodies. The parts for the surfaces (SPHERE, CUBE, and SYMM) are labeled in the figure above.

To change the part names of surfaces, in the Display Tree right-click on **Parts > CreatePart > CreatePart by Selection**.



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 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 . 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 4-237
Create part window

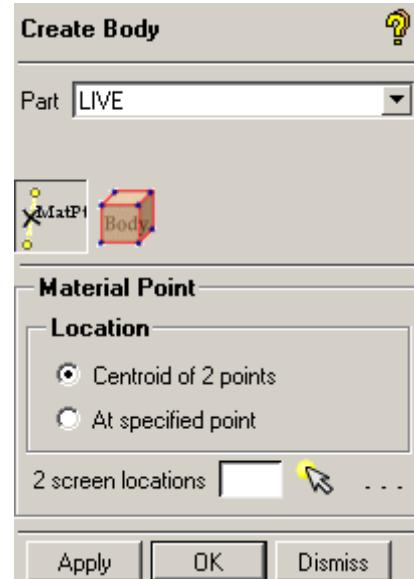


d) Creating Body

The body of the model – which will hold the tetrahedral elements - will be placed into the part, LIVE. Select

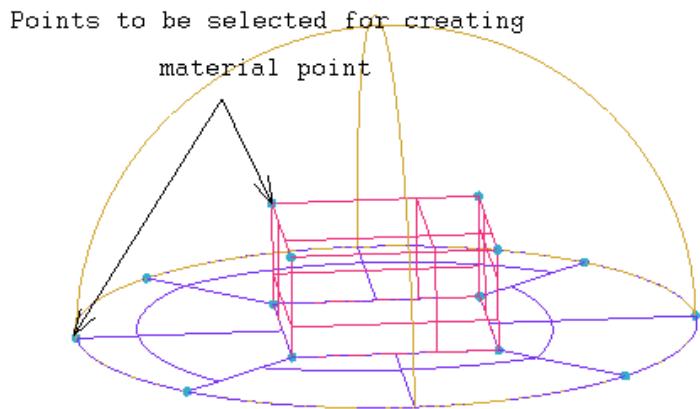
Geometry > Create Body.  In the Create Body window, use the Material Point option. Enter Part as LIVE in the window and then in Location, enable Centroid of 2 points with the left mouse button.

Figure 4-238
Creating body window



Select two points as shown below 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 4-239
Points to be selected for creating material point



e) Set global mesh size

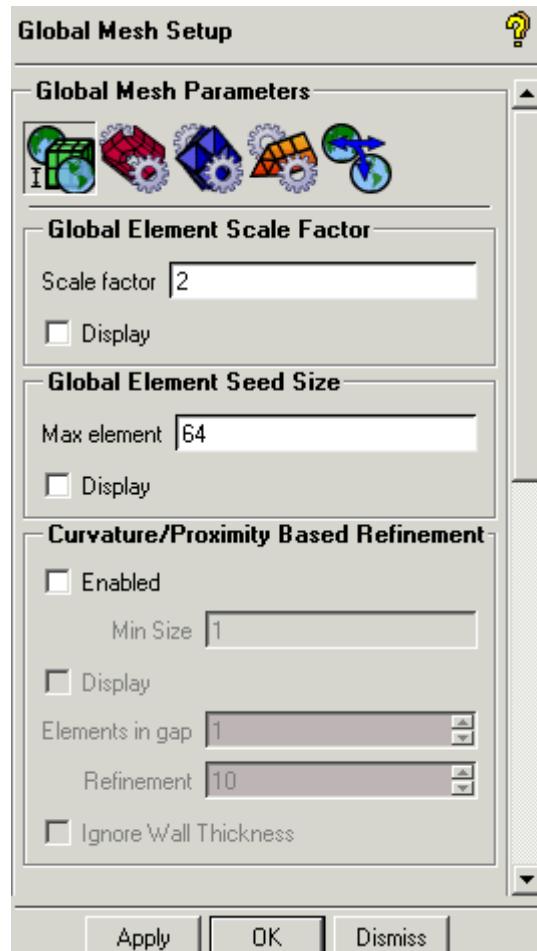
The user must define mesh sizes before mesh generation.

Select Mesh > Global Mesh Setup > Global Mesh

Size to obtain the Global Mesh Size window as shown in below. Enter **2** for **Scale factor** and **64** for **Max element** (see below). 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 4-240
Setting the Global
mesh sizes for the
model



From the Display Tree, right click on Surfaces > Tetra Sizes and Curves > Curve Tetra sizes. This displays icons representing the maximum element sizes specified on the entities.

f) Set surface mesh size

The meshing can be adjusted on the different parts of the model

via Mesh > Surface Mesh Setup.  Make only the SYMM part visible from the Display Tree. Select  and click 'v' on the keyboard to select the visible surfaces. Set Maximum size = 2 as shown below. Press Apply. Make only CUBE and SPHERE visible from the Display Tree .

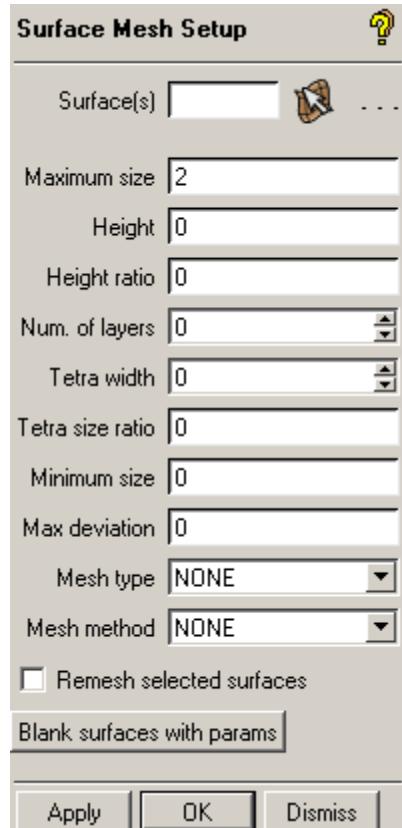
Select  and click 'v' on the keyboard to select the visible surfaces. Set Maximum size = 1.

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.

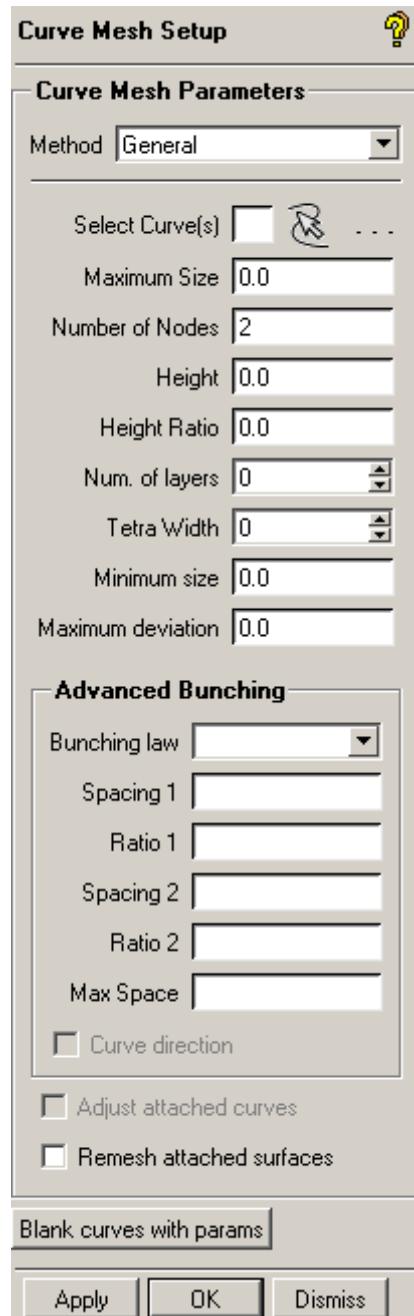
Figure 4-241
Setting the mesh sizes for the selected surface parts



g) Set Curve Mesh Size

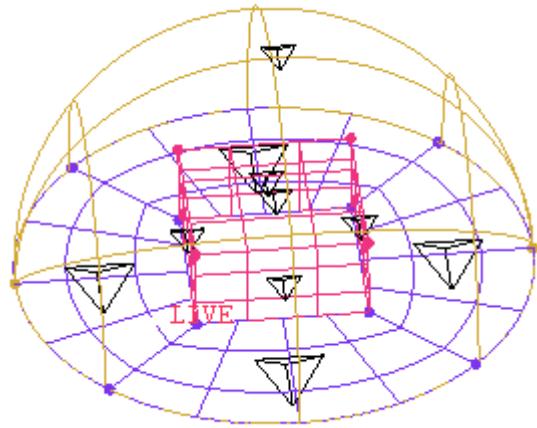
Similarly, select Mesh > Curve Mesh Setup  to open the window as seen below. Select  and choose all the curves by clicking ‘a’ on the keyboard. All the curves will be highlighted in the display. Set all the parameters in the Curve Mesh Size window to 0 and then press Apply followed by Dismiss.

Figure 4-242
Curve mesh size
window



The assigned Tetra sizes are represented on the geometry as shown below.

Figure 4-243
Tetra sizes on the
geometry



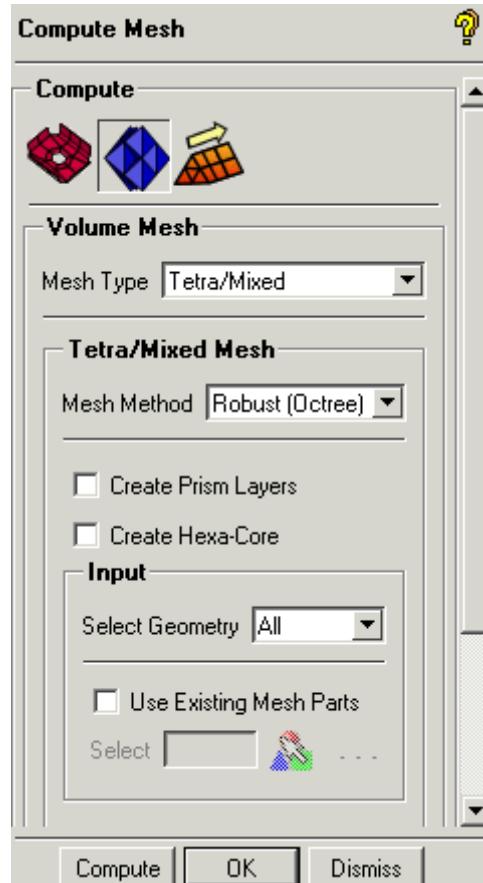
Make these displayed tetra invisible by right clicking on Surfaces > Tetra Sizes and Curves >Curve Tetra Sizes from the Display Tree .

When satisfied with the results, press File > Save Project to save the tetin file. Use the default project name.

h) Generating the tetrahedral mesh

Choose Mesh > Compute Mesh > Volume Mesh > Tetra/Mixed > Robust (Octree) The Mesh Volume window will appear as shown below.

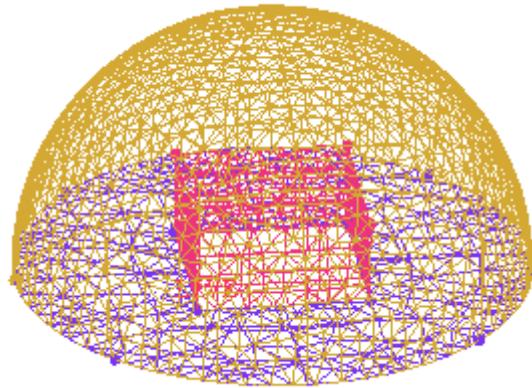
Figure 4-244
Mesh with Tetrahedral
window



Press Compute to start the meshing process. The mesh opens automatically once the meshing process is complete.

Once the meshing process is completed, the mesh should appear as shown below.

Figure 4-245
The smoothed mesh



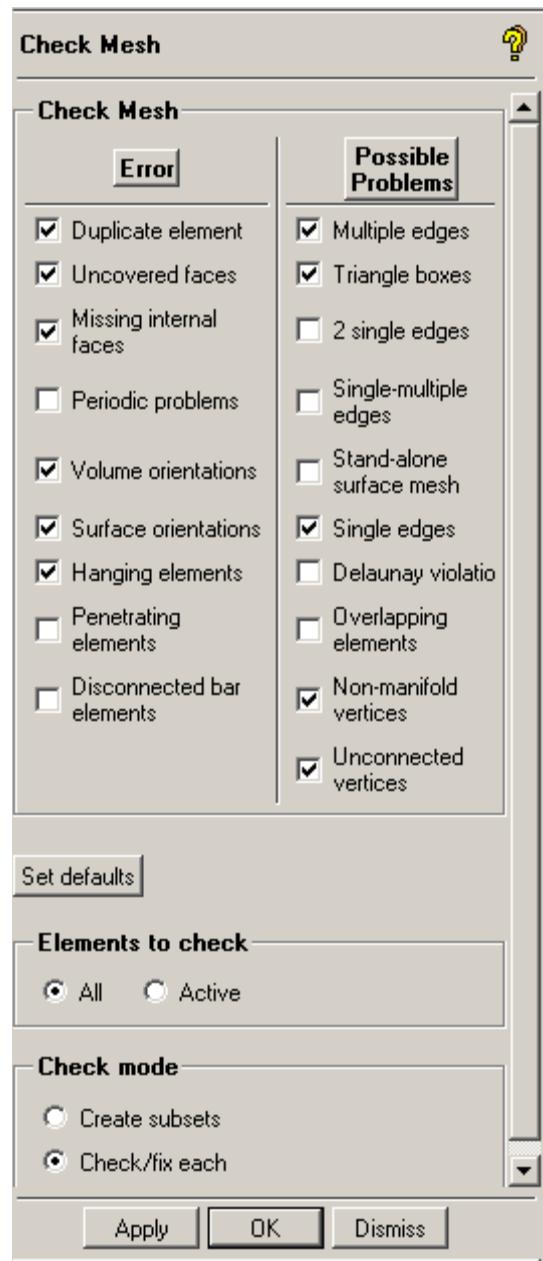
i) **Diagnostics**

The user should check the mesh for any errors or problems that may cause problems for analysis. The Check Mesh window



is accessible from the Edit Mesh menu.

Figure 4-246
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, ANSYS 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.

j) Saving the Project

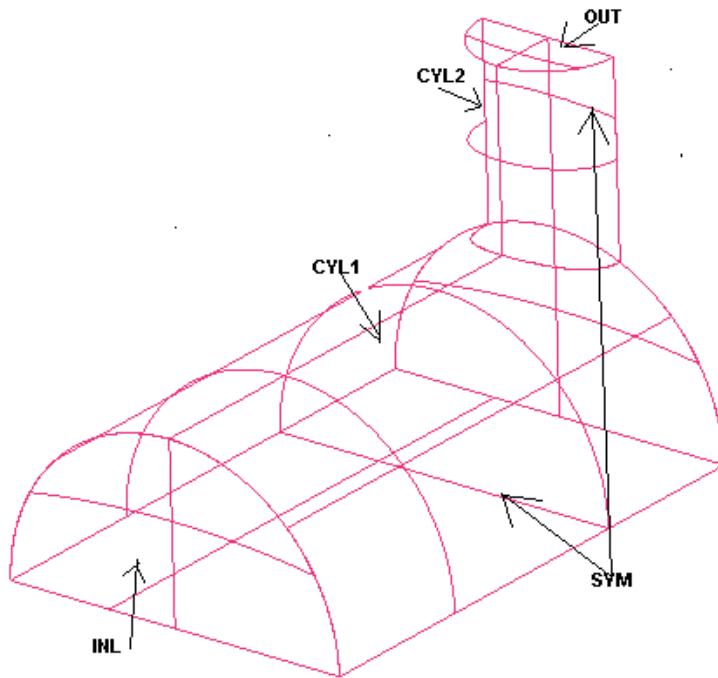
Save the mesh and geometry by selecting File > Save Project
Close the project by selecting File > Close Project.

4.4.3: 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 below.

**Figure
4-247**
**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

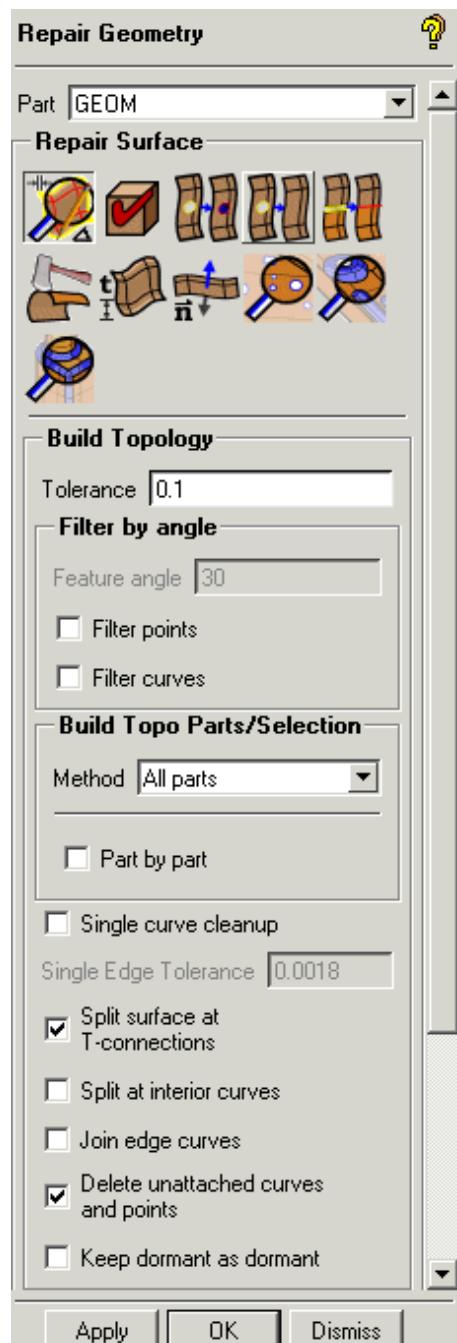
From UNIX or DOS window, start ANSYS ICEM CFD. The input files for this tutorial can be found in the Ansys installation directory, under ..\v110\docu\Tutorials\CFD_Tutorial_Files>3DpipeJunct. Copy and open the tetin file geometry.tin in your working directory.

c) Repairing the Geometry:

Select Geometry > Repair Geometry  > Build Diagnostic Topology .

Enter Tolerance as 1 Use other default values and click Apply.

Figure 4-248
Repair Geometry Window

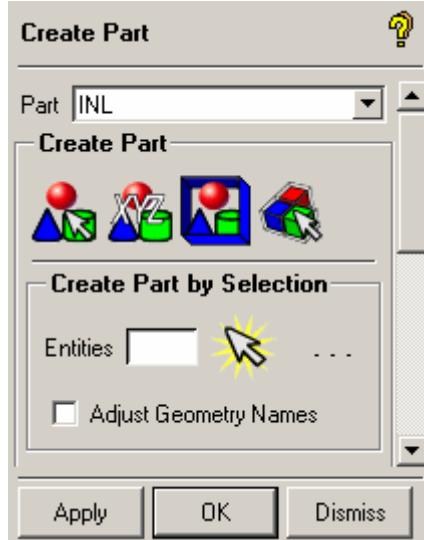


Note: Build Topology creates the curves and points necessary for Mesh generation

d) Parts Creation

Create new Parts and add the appropriate surfaces to the parts. Initially, all the geometry is grouped into the GEOM part. Referring to the figure below, create and add the appropriate surfaces to the Parts INL, OUT, CYL1, CYL2, and SYM.

Figure 4-249
Create part window



To create Parts for the surfaces, right-click on Parts > Create

Part by Selection . 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 put 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. 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. 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

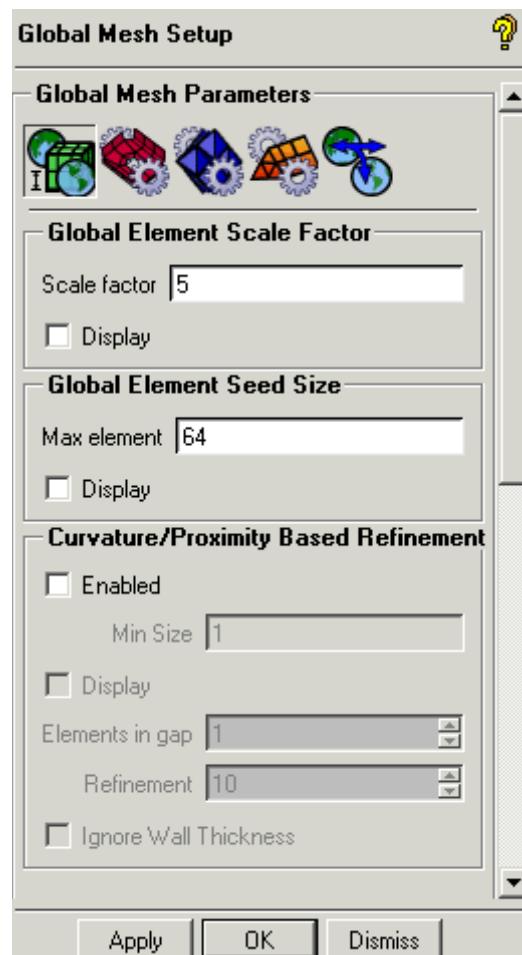
 to choose 2 screen locations in the geometry between which the material point will be created. LIVE should then appear inside the model. Dynamically rotate the model to ensure that LIVE is located within the interior of the volume, and not outside.

f) Reassigning Mesh Parameters

The user will now specify the mesh size on the entire model with Mesh >

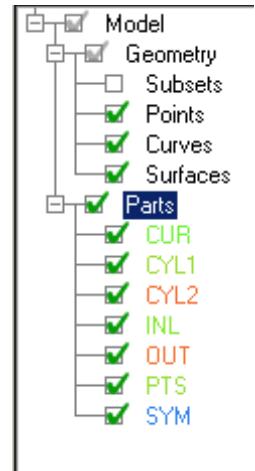
 > Global Mesh Size . Change the Scale factor to 5 and Max element to 64. Click Apply. The scale factor is a multiplier for all size specifications applied to parts or individual curves and surfaces. The local element size will be equivalent to the local size applied to that entity, multiplied by the Scale factor.

Figure 4-250
Assigning Global mesh sizes to the entire model



The user can make the parts visible from the Display Tree that appears below.

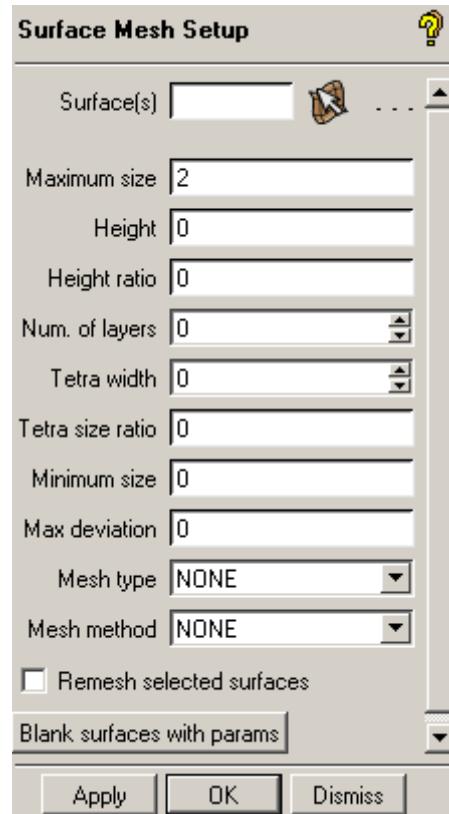
Figure 4-251
Select parts to modify



To change the mesh size on specific surfaces, select Mesh > Surface Mesh

Setup 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, assign Maximum element size of 2. Press Apply and Dismiss.

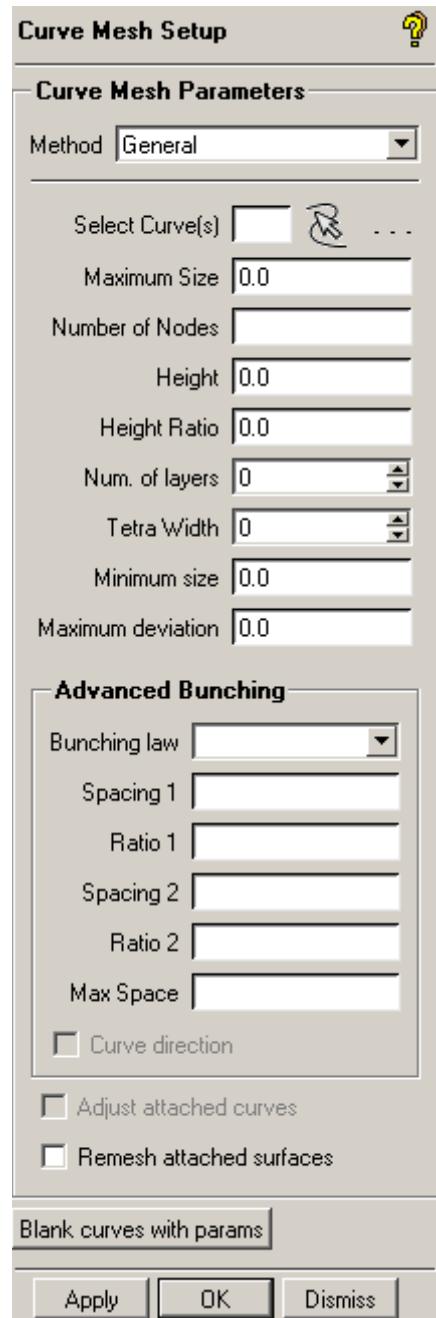
Figure 4-252
Adjusting the surface mesh sizes Associated to the selected surfaces



g) Setting curve mesh size

Select Mesh > Curve Mesh Setup. Select and click 'a' on the keyboard to select all curves. Set Maximum Size = 0 to all the curves as shown. Select Apply and Dismiss.

Figure 4-253
Adjusting the curves Mesh sizes

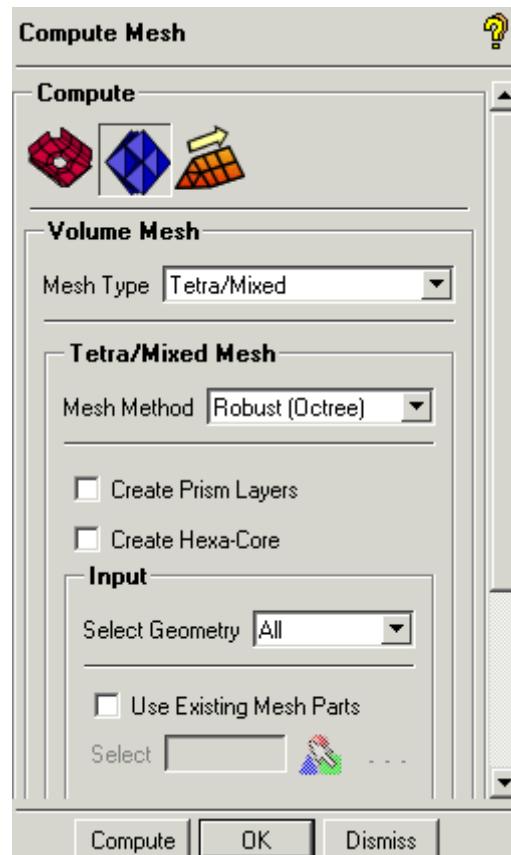


When satisfied with the meshing parameters, press File > Save Project to save the changes made to the model before proceeding further.

h) Generating tetrahedral Mesh

Choose Mesh > Compute Mesh > Volume Meshing > Tetra/Mixed > Robust (Octree) . A Mesh Volume window will open as shown below.

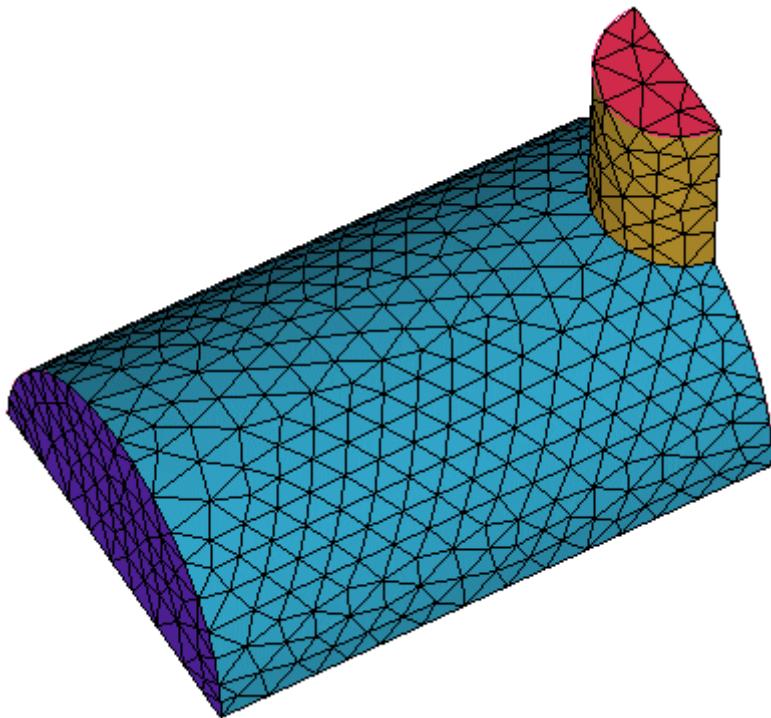
Figure 4-254
Mesh with
tetrahedral



Press Compute.

The mesh will appear in the display when the meshing process is finished. Make sure that the Mesh type Shells in the Display Tree is active so that the mesh, represented by its triangular surface elements, should appear as shown below.

**Figure
4-255
The
tetrahedral
mesh**

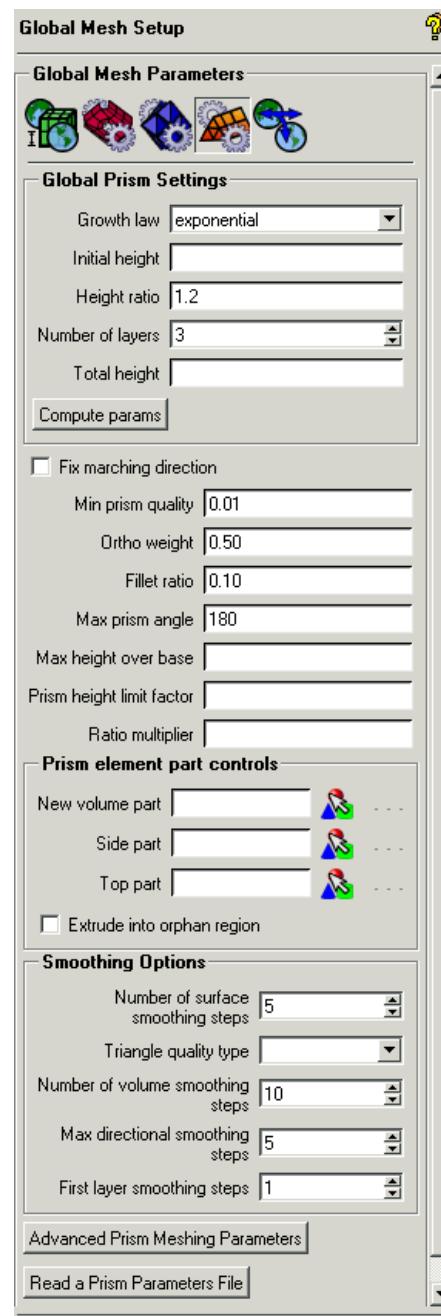


i) Assigning Prism Parameters

Select Mesh > Global Mesh Setup > Prism Meshing Parameters

to open the Global Prism Settings panel. 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.

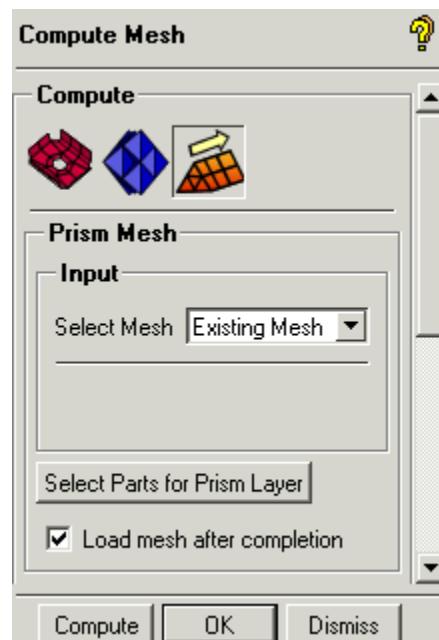
Figure 4-256
Global mesh size window



j) Generating the Prism Mesh

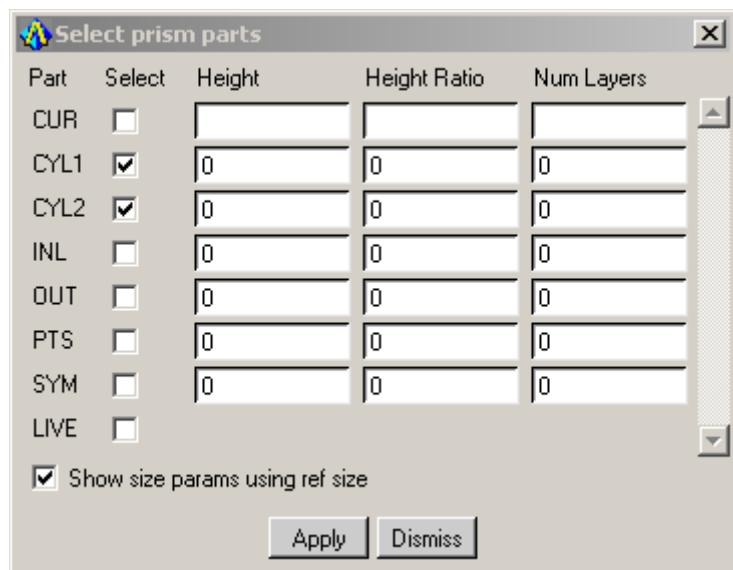
Select Mesh > Compute Mesh  > Prism Mesh.  (Save and overwrite the project as prompted.)

Figure 4-257
Mesh with prism window



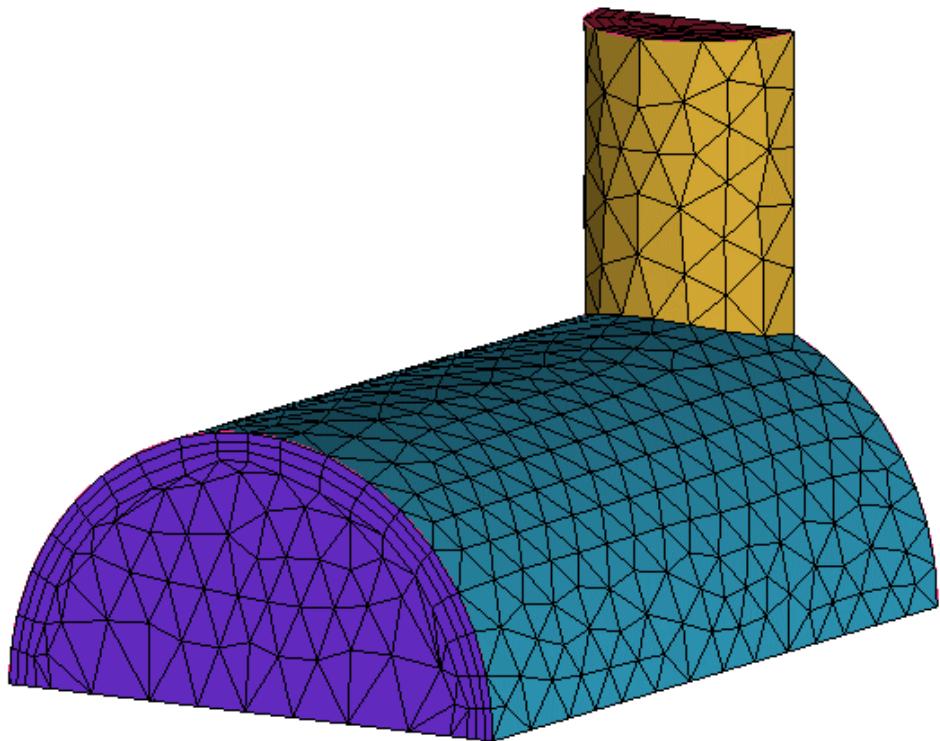
Click on Select Parts for Prism Layer. In the Mesh sizes for parts window, enable Prism for the CYL1 and CYL2 Parts. Click Apply and Dismiss.

Figure 4-258
Selecting parts for prism mesh generation



In the Prism Mesh window, enable Load mesh after completion and select Compute to start the prism mesher.
The resulting tetra/prism mesh is shown below.

**Figure
4-259
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.

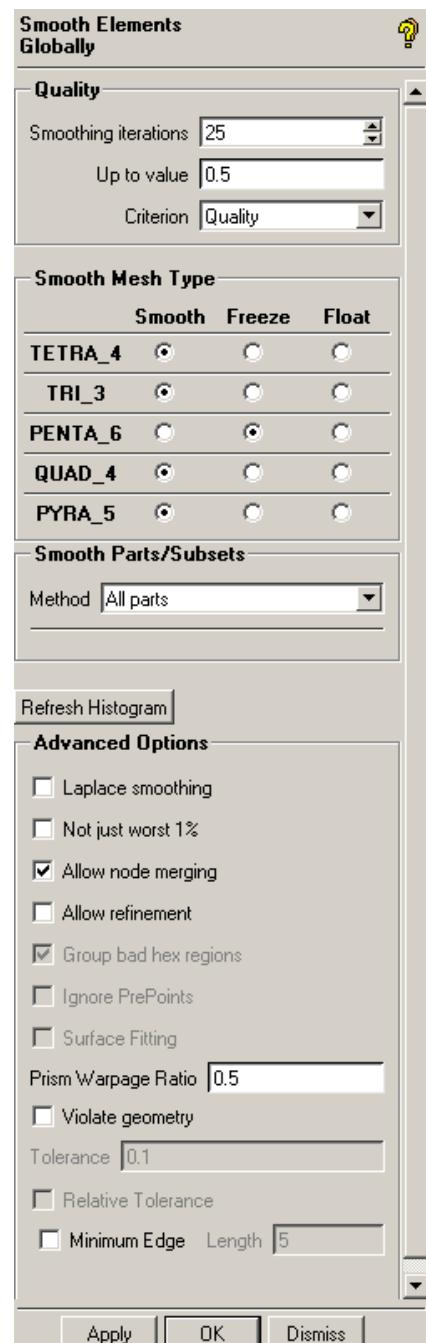
Smoothing the Mesh

Once the ANSYS ICEM CFD Mesh Editor has reported no Errors or Possible problems, the user may continue by smoothing the generated Tetra/Prism mesh.



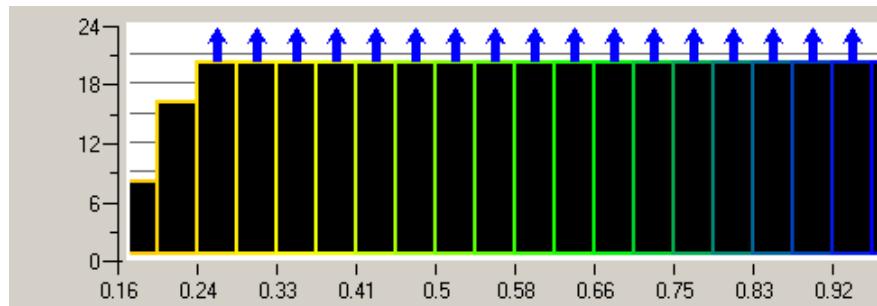
Press Edit mesh > Smooth Mesh Globally to open the Smooth Elements Globally window.

Figure 4-260
Smooth elements globally



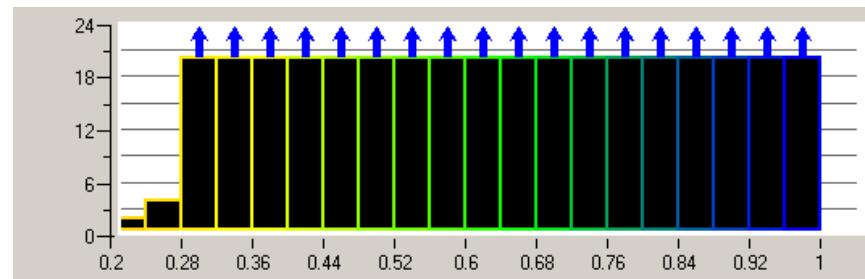
Set Smoothing iterations to 25, Up to quality to 0.5, and Criterion to Quality. With a Tetra/Prism mesh, first smooth the interior elements without adjusting the prisms. Under Smooth Mesh Type set PENTA_6 (the prisms) to Freeze. The mesh quality histogram is shown below. Press Apply to start the smoother.

Figure 4-261
The Quality histogram before smoothing



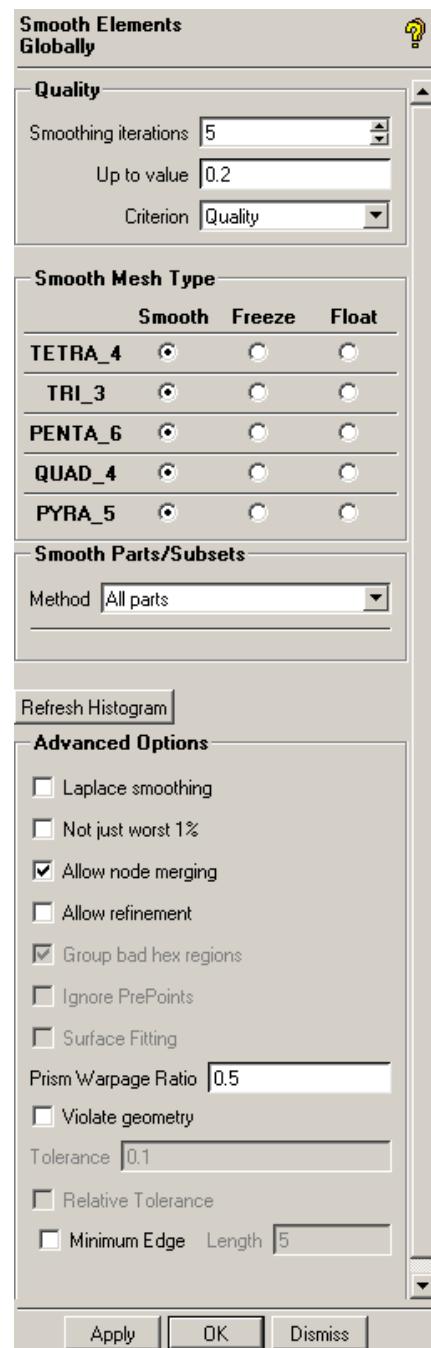
The improvements are noticeable in the histogram seen below. There is no element below the quality of 0.2. Note that only element types set to Smooth are included in the histogram.

Figure 4-262
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. Select Apply to do the final smoothing.

Figure 4-263
Smooth elements globally window



I) Saving the project

Save the mesh by selecting File > Save Project.

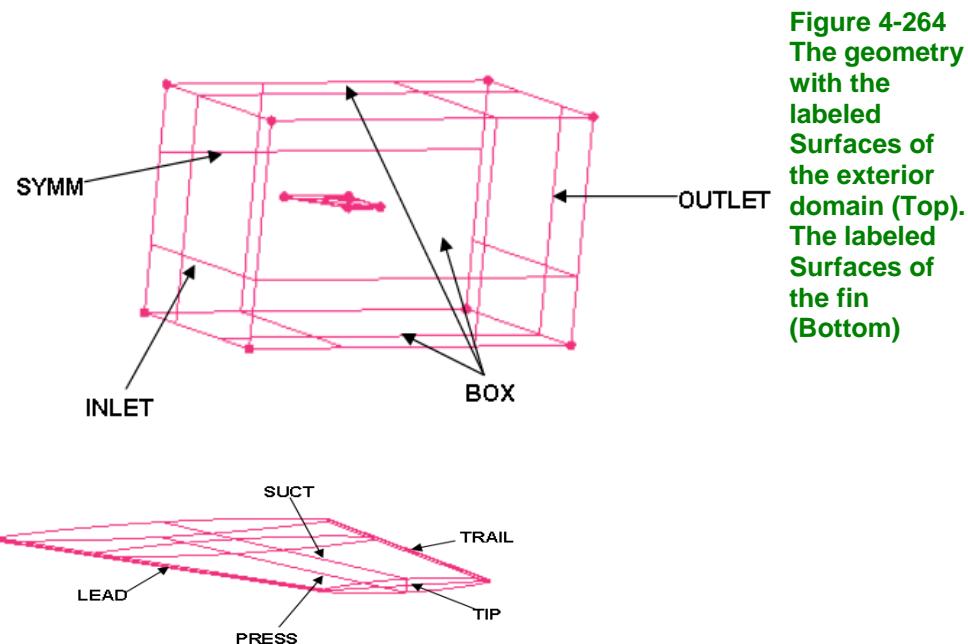
Close the project by selecting File > Close Project.

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	327
------------------------	--	-----

4.4.4: 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.



a) Summary of steps

- Starting the project
- Repairing the Geometry
- Assigning Mesh sizes
- Generating Tetrahedral/Prism mesh
- Diagnostics

Smoothing the mesh
 Generating Hex-Core mesh
 Smoothing
 Saving the project

b) Starting Project

From UNIX or DOS window, start ANSYS ICEM CFD. The input files for this tutorial can be found in the Ansys installation directory, under ..\v110\docu\Tutorials\CFD_Tutorial_Files>FinConfig. Copy and open the geometry file geometry.tin in your working directory.

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

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 the figure above.

Enable the display of Surfaces from the Display Tree. 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  > Global Mesh Size  to open the Global Mesh Size window.

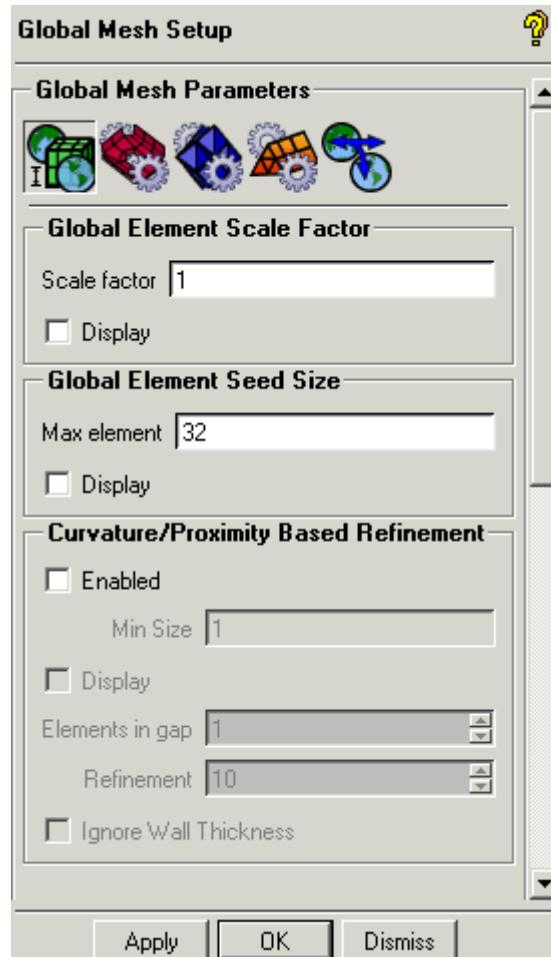
Enter the value of 32 for Max element

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 4-265
Editing the Global Mesh sizes



g) Setting Surface mesh size

Select Mesh > Surface Mesh Setup  > 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 > Curve Mesh Setup  > 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.

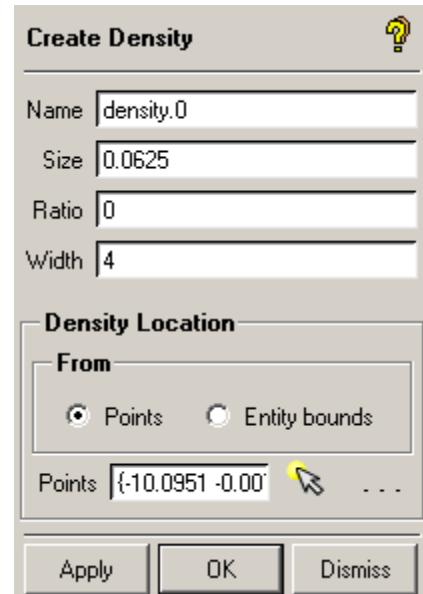
The mesh density will be a region in which one can prescribe a certain maximum element size. This is useful for refining the mesh in a volumetric region.

i) Creating the density regions

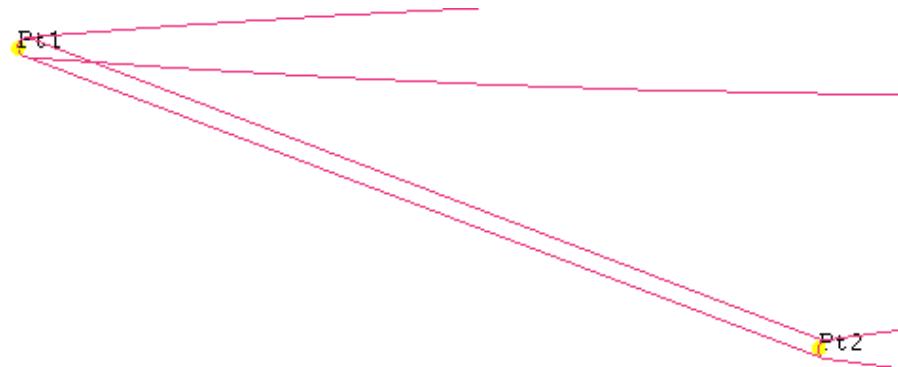
Select Mesh > Create Mesh Density .

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 the figure below. 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.

Figure4-266
Create density window



**Figure
4-267**
Density
Creation
at the
Leading
Edge

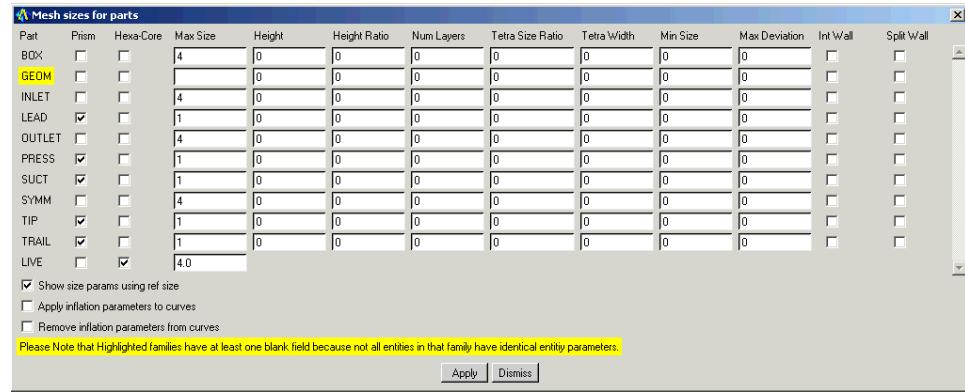


j) Defining Parts for prisms and hexa-core

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. Enable Hexa-Core for LIVE and set Max Size to 4.0. Select Apply and Dismiss.

Figure 4-268
Selecting parts for Prism and Hexa-Core mesh



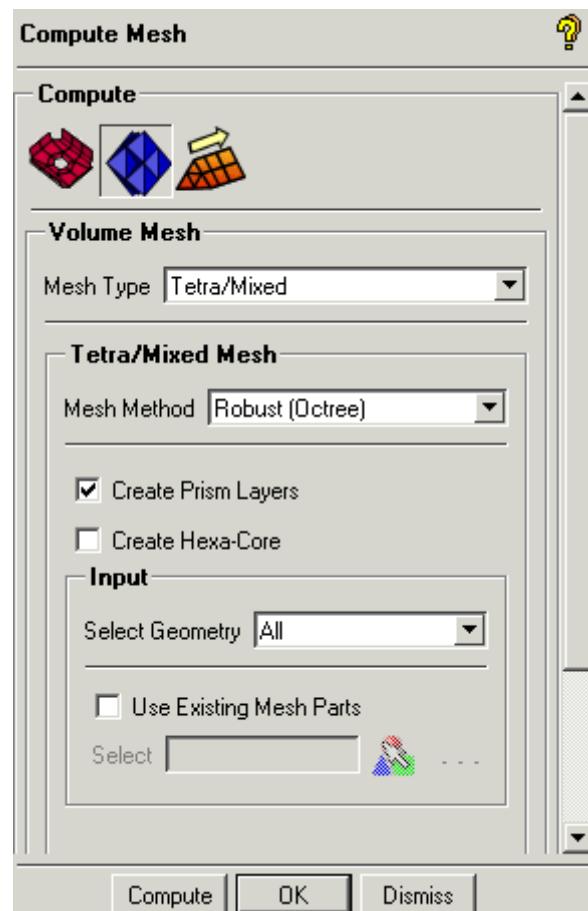
Choose File > Save Project to save the changes in the Tetin file. Accept the default project name.

k) Generating the Tet/Prism Mesh

Press Mesh > Compute Mesh > Volume Mesh > Tetra/Mixed > Robust (Octree). Toggle On create Prism Layers. Press Compute to generate the mesh using the default parameters.

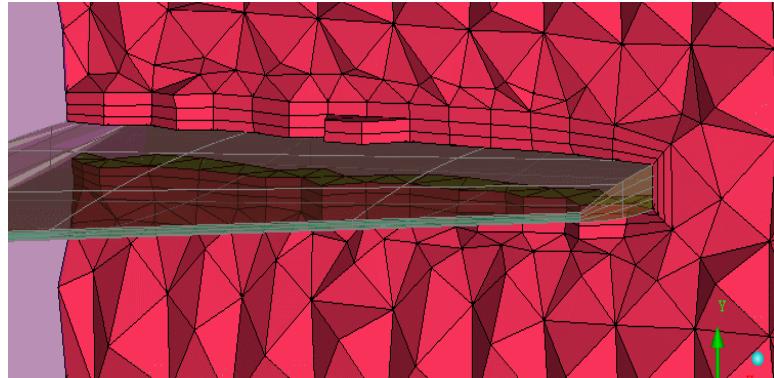
Note: The default prism parameters (Mesh > Global Mesh Setup > Prism Meshing Params) indicate 3 prism layers using the local prism thickness based on the surface triangle size (as no initial height is specified).

Figure 4-269
Mesh with Tetrahedra
parameters



A cut plane through the complete tetra/prism mesh is shown below.

**Figure
4-270
Tetra/Prism
mesh**



1) 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

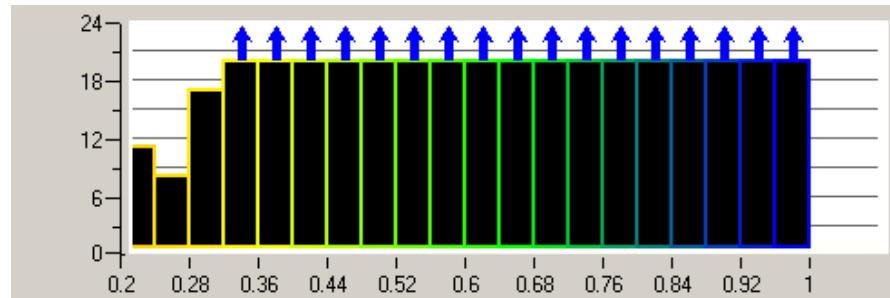
After confirming that no serious problems or errors exist, continue by smoothing the generated tetra/prism mesh.



Select Edit Mesh > Smooth Mesh Globally.

Several elements have a lower quality than the "acceptable" value of 0.3. Set the Smoothing iterations to 5 and the Up to quality to 0.4. Make sure Criterion is set to Quality.

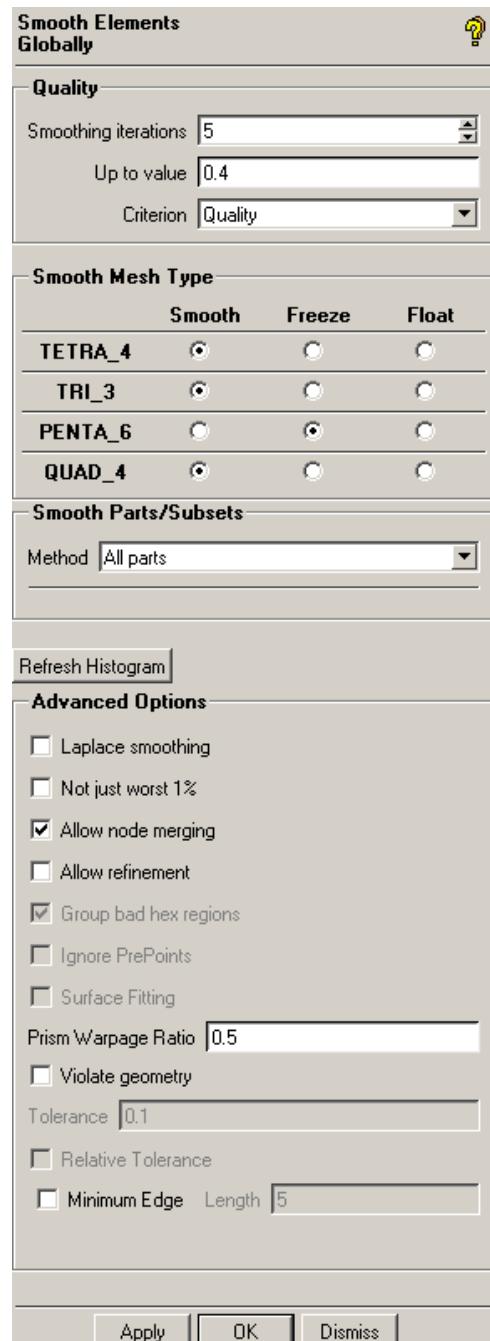
**Figure
4-271
Quality of
the mesh
before
smoothing**



Select the Smooth option for TETRA_4 and TRI_3 and QUAD_4. Select the Freeze option for PENTA_6 (the prisms).

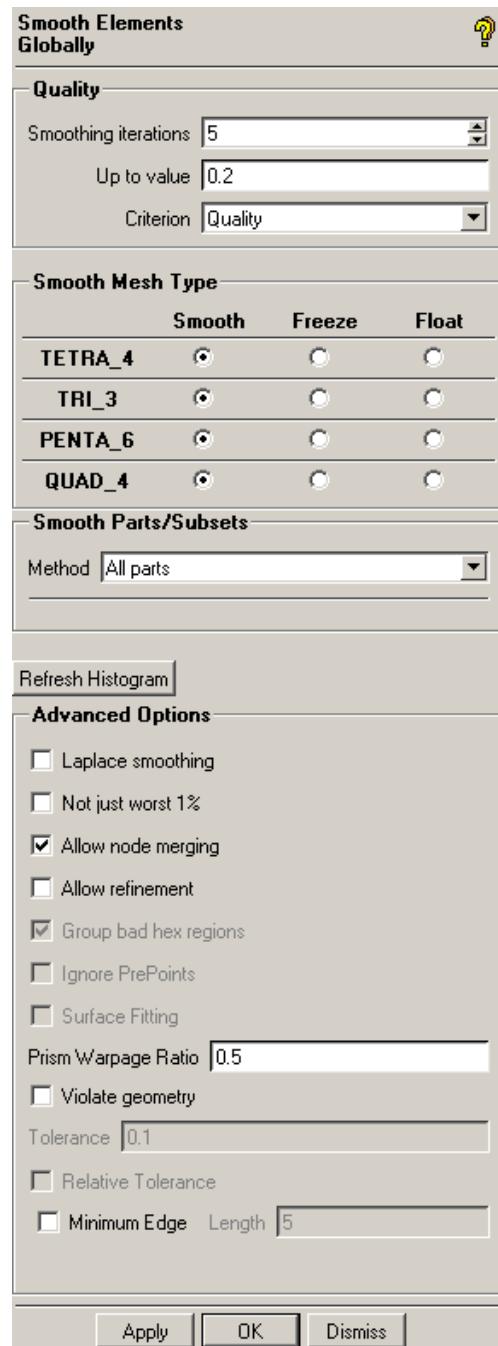
Press Apply when the operation is complete, a new histogram will be displayed.

Figure 4-272
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 to prevent dramatic warpage of the prism layers. Press Apply.

Figure 4-273
Smooth Elements Globally



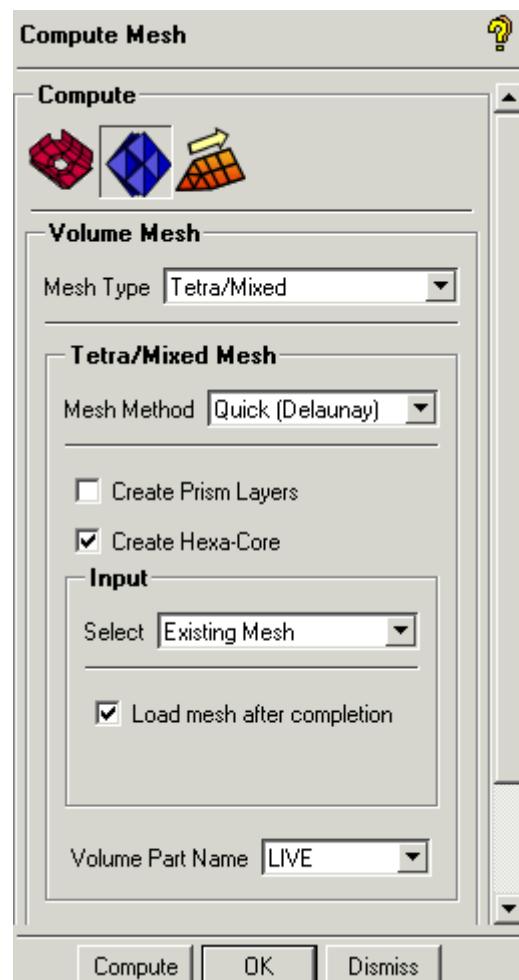
Select Info > Mesh Info. Scan the messages area to find the number of elements in the LIVE part (the volume elements belong to this part – tets/prisms). The information indicates there are roughly 800,000 elements in the LIVE part.

Save the project.

n) Building the Hex-Core mesh

Select Mesh > Compute Mesh  > Volume Mesh  > Tetra/Mixed > Quick (Delaunay) Toggle ON Create Hexa-Core. Select Existing Mesh option for Input. Toggle ON Load Mesh After Completion. 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.

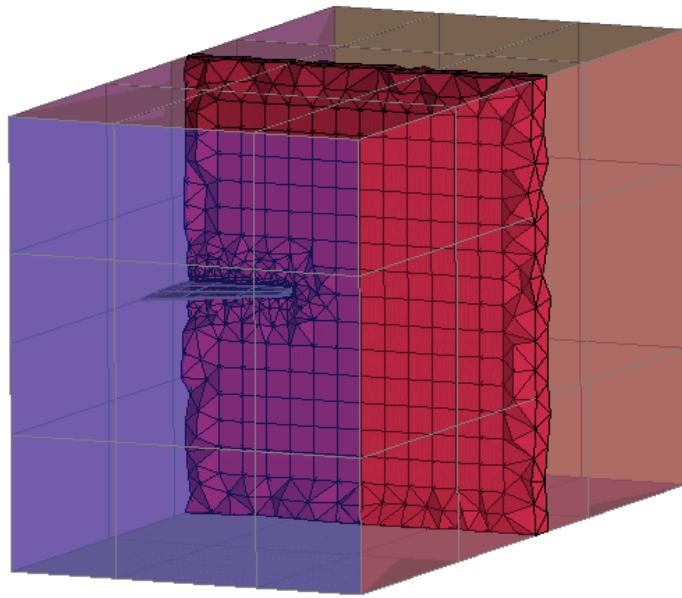
Figure
4-274
Hexa Core
Mesh
Setup



Click Compute to start the mesher.

A cut plane through the mesh is shown below.

Figure 4-275: Cut-plane showing volume mesh

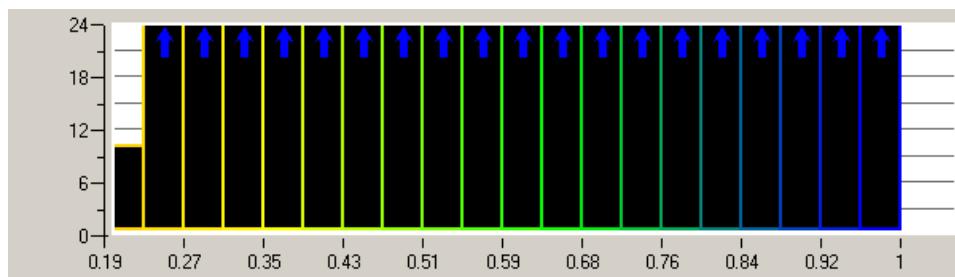


Select File > Save Project As... and give the project a new name. This preserves the existing tet/prism mesh in one project, and the hex-core mesh in another.

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 below.

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
4-276
Final
quality
histogram**



o) Saving the project

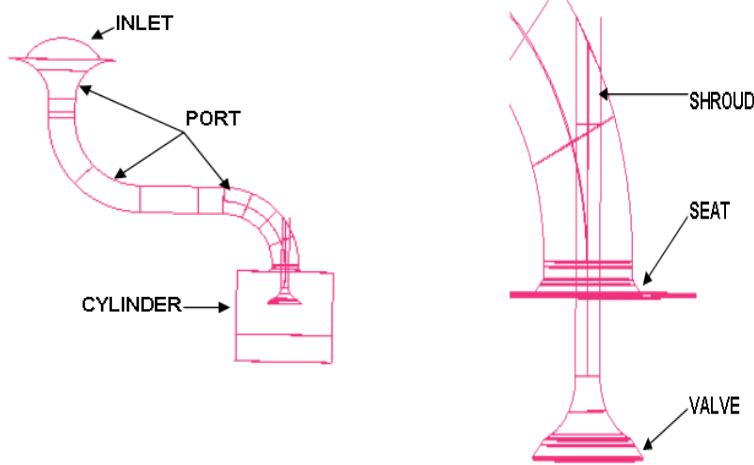
Save the mesh by selecting File > Save Project. Then close the project by selecting File > Close Project.

4.4.5: Piston Valve

Overview

In this tutorial example, the user will define a thin cut in the Geometry to mark a region where ANSYS ICEM CFD Tetra will generate a thin layer of elements. The user will then generate and smooth a tetrahedral mesh for a piston valve configuration

**Figure
4-277
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

The input files for this tutorial can be found in the Ansys installation directory, under `../v110/docu/Tutorials/CFD_Tutorial_Files > Piston`

Valve project. Copy and open the Tetin file geometry.tin in your working directory.

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 the figure at the beginning of the tutorial.

e) Defining the Material Point

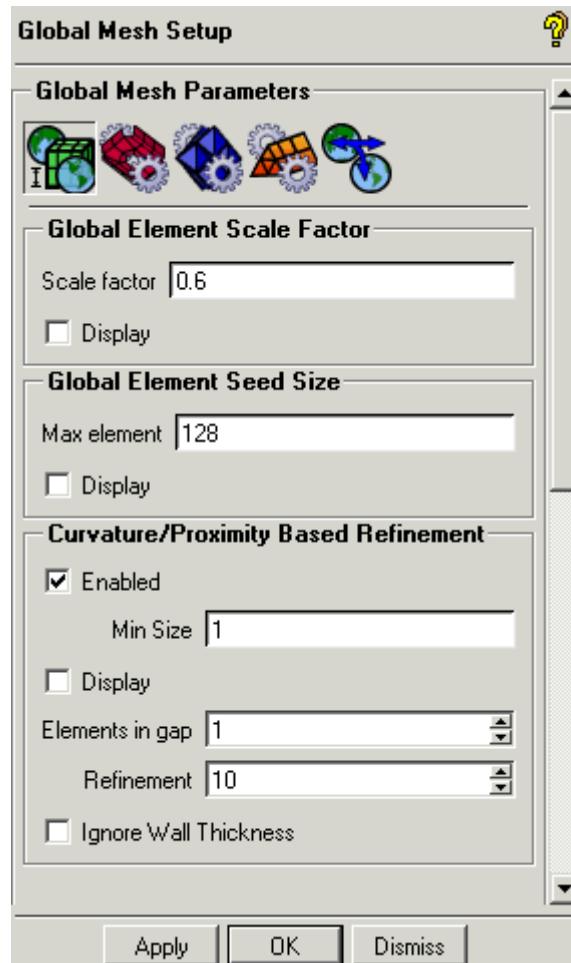
The material of the model will be assigned to the material point, LIVE.

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 > Global Mesh setup  > Global Mesh Size  to bring up the global mesh size as seen below. Enter 0.6 as the Scale factor and 128 for Max Element. Switch ON Curvature/Proximity Based Refinement by providing the value of 1as shown below.

Figure 4-278
Global Mesh Size
Window



Curvature/Proximity Based Refinement allows ANSYS 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, Curvature/Proximity Based Refinement is a multiplier of the scale factor.

The value given by Curvature/Proximity Based Refinement multiplied with the scale factor represents a minimum element size.

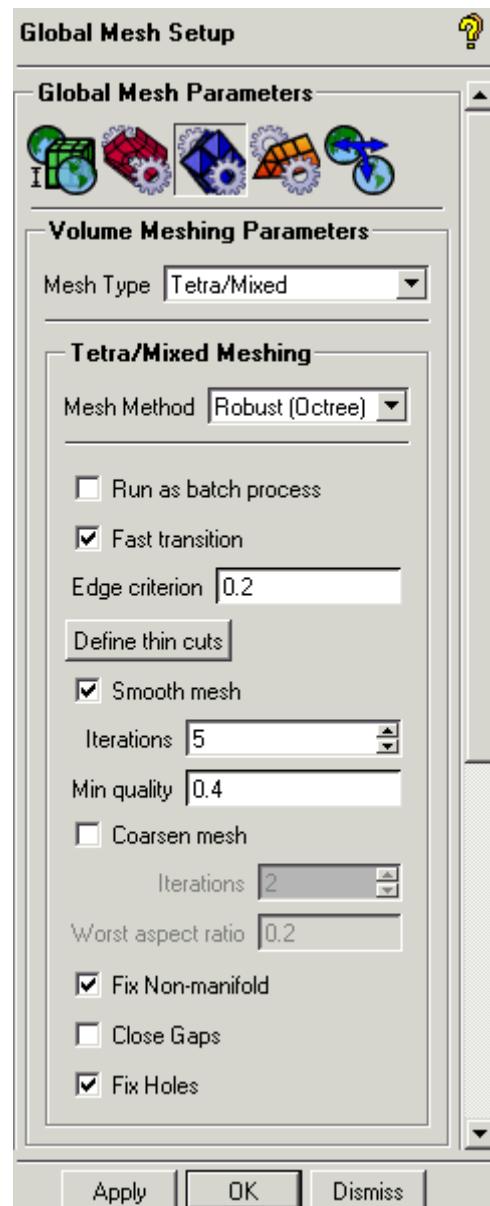
The Curvature/Proximity Based Refinement > Refinement parameter defines the number of edges along a radius of curvature. Refinement parameter is used to compute the Curvature/Proximity Based Refinement, consequently, the larger this parameter, the smaller will be the computed the Curvature/Proximity Based Refinement size. Refinement should always be a positive integer value.

Note: For more information on Curvature/Proximity Based Refinement, see the ANSYS ICEM CFD online help.

The value entered for Curvature/Proximity Based Refinement limit is a factor multiplied by the **scale factor**. The Curvature/Proximity Based Refinement 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 Setup  > Volume Meshing Parameters
 > Tetra/Mixed > Robust (Octree).

Figure 4-279
Tetra Meshing Parameters
Window

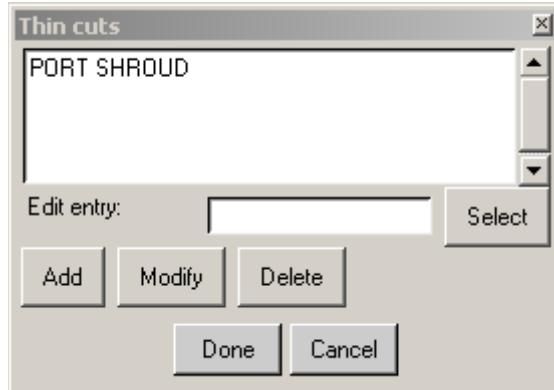


To add a thin cut to the model, consider a region between any two parts that may be thinner than the tetrahedral size defined on them, in which a fine layer(s) of tetrahedral elements may be created.

Note: For more information on Thin cuts, refer to the ANSYS ICEM CFD online Help.

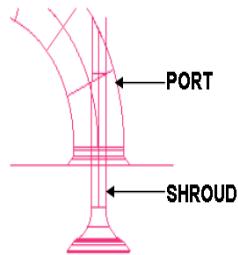
Select Define thin cuts, the Thin cuts window will appear, press Select. The first part is PORT and the second part is SHROUD. Press Add, and the two part names will appear in the ‘Thin cuts’ window, as shown below.

Figure 4-280
The Thin cuts window



Using parts in the Display Tree, the user is able to browse the parts of the model. The close-up view of the PORT and SHROUD part is shown, between which the thin cut will be defined.

Figure 4-281
The PORT and SHROUD parts



When finished, press Done.

Select Apply in the Global Mesh Setup window to activate the modifications.

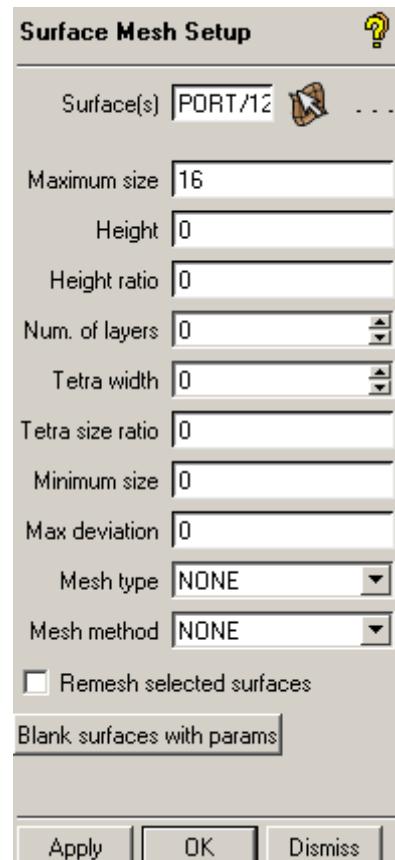
Press Dismiss to Close the Global Mesh Size window.

g) Setting the surface mesh size

Select Mesh > Surface Mesh Setup 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, enter Maximum Element size of 16 and press Apply.

Figure 4-282
Edit the surface meshing sizes

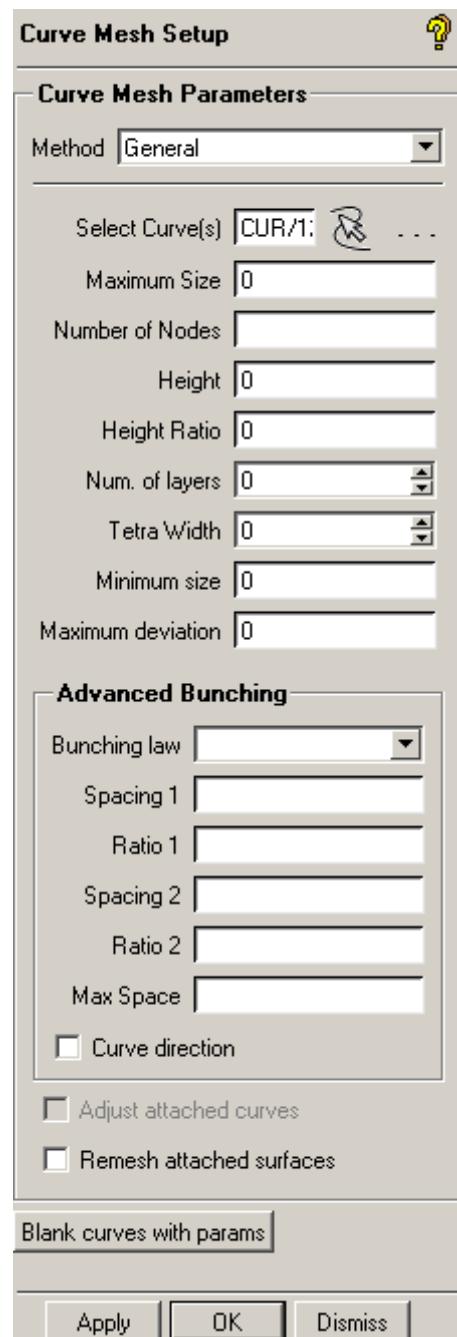


h) Setting curve mesh size

Next, select Mesh > Curve Mesh Setup 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, enter all the parameters 0. Press Apply followed by Dismiss to close the window.

Figure 4-283
Curve mesh size window

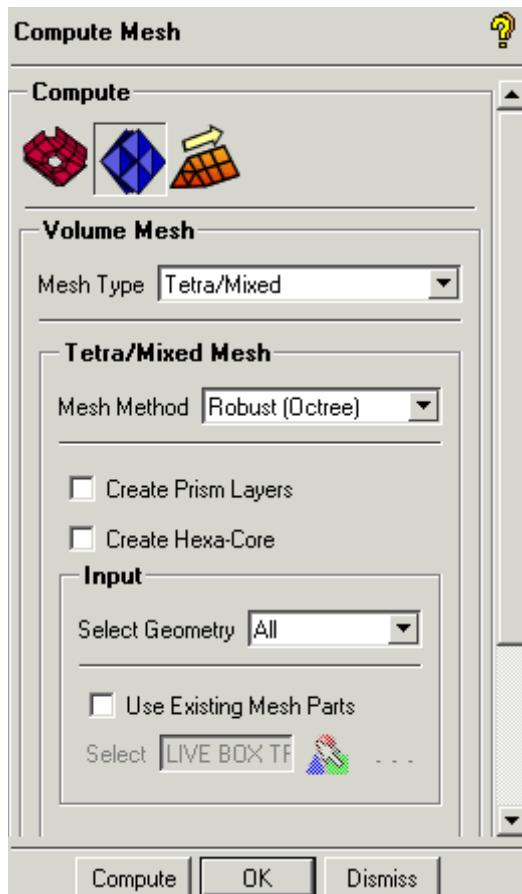


Make sure that Surfaces and Curves are visible in Display Tree. Right click on Surfaces > Tetra sizes and Curves >Tetra sizes. Check that all the surfaces have the Maximum element size of 16, and curves have size of 0. Choose File > Save project to save the additions to the tetin file.

i) Generating the Tetrahedral Mesh

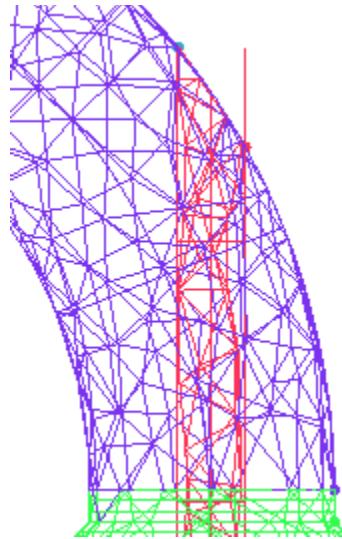
Click on Mesh > Compute Mesh  > Volume Meshing  > Tetra/Mixed > Robust (Octree). Press Compute to create the mesh

Figure 4-284
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. Zoom in on the region between PORT and SHROUD where the thin cut was defined, the mesh should resemble the figure below.

Figure 4-285
The mesh in the Thin cut region



j) Checking the mesh

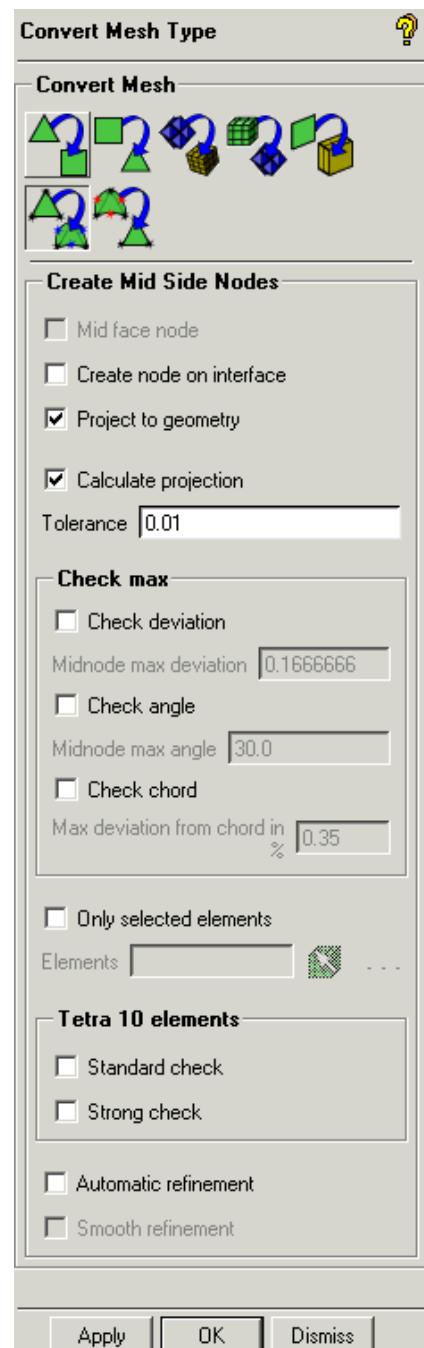
Check the mesh for different errors and possible problems with Edit Mesh
> Check Mesh.

k) Conversion of Elements from Linear to Quadratic

Choose Edit mesh >Convert Mesh Type > Create Mid Side

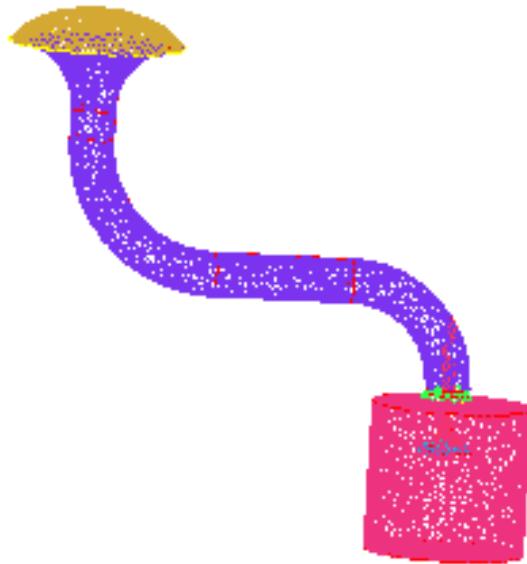
Node . A new window will appear as seen below. choose all elements to be converted to quadratic before selecting Apply.

Figure 4-286
Linear to quadratic window



The TRI_3 elements get converted to TRI_6 and TETRA_4 get converted to TETRA_10 as shown.

Figure 4-287
The mesh
after
conversion



I) 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

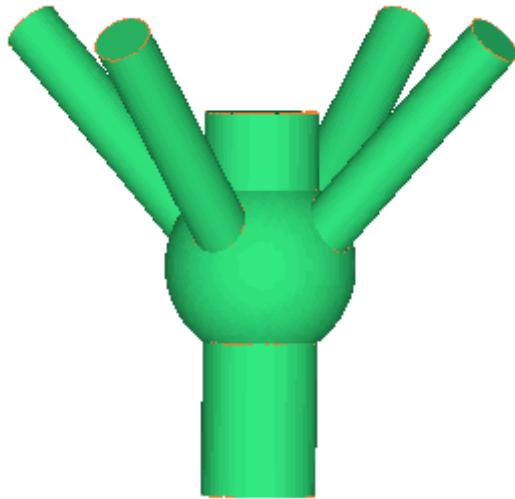
Save the mesh by selecting File > Save Project.

Close the project by selecting File > Close Project.

4.4.6: STL Configuration

Overview

In this tutorial example, the user will import STL data to the ANSYS ICEM CFD 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

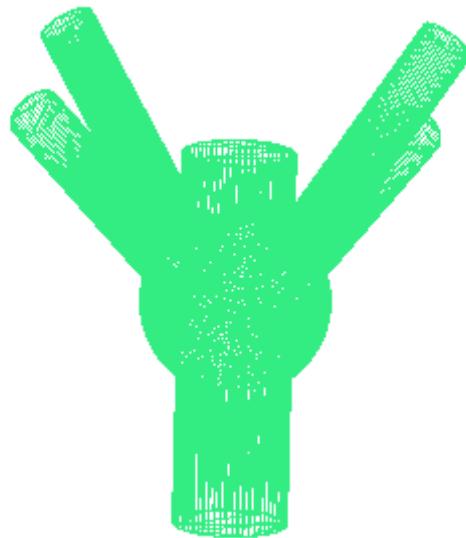
b) Converting the STL File to a Geometry File

Start the ANSYS ICEM CFD 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 4-288
Detailed display of the
surface



In the Display Tree, Select Surfaces >Show Full to see the surface. Reset the display to Simple before preceding to speed-up the display.

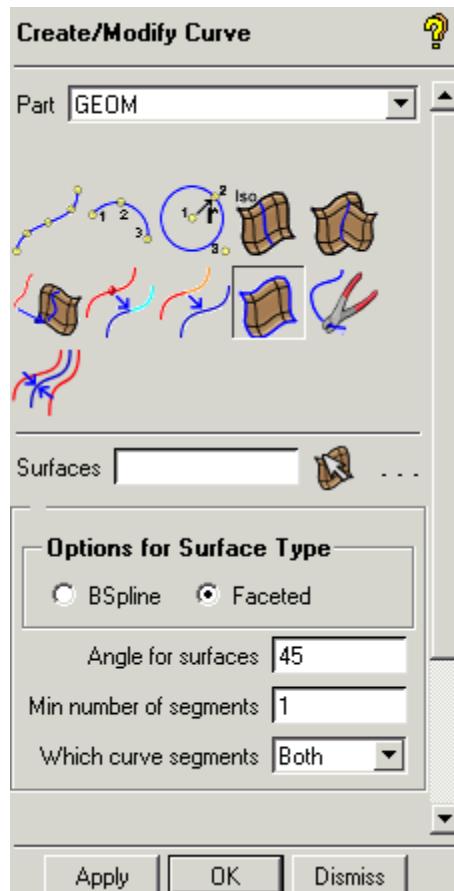
c) Extracting the Curve

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 below.

Figure 4-289
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.

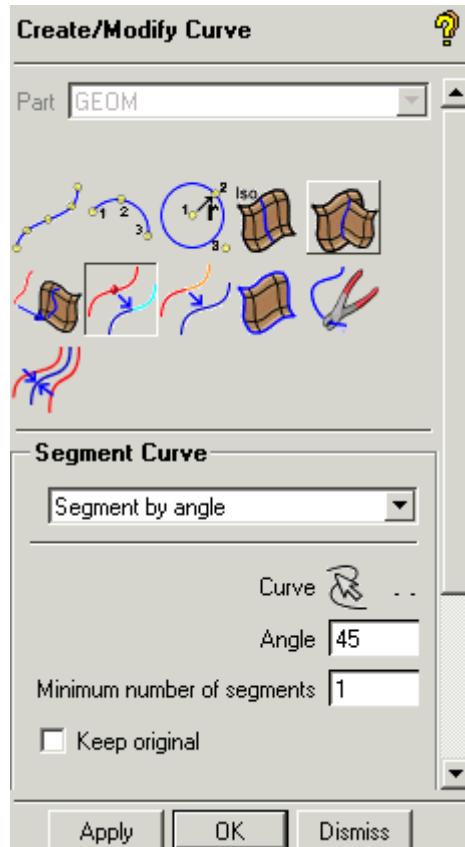
d) Segmenting the Curves

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. Notice that the curve is named GEOMETRY/0.0 is the first curve in GEOMETRY.

Select Geometry > Create/ Modify Curve  > Segment Curve .

Figure 4-290
Segmenting curves



Select the GEOMETRY/0.0 curve and complete the selection.

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 (GEOMETRY0.0.1 to GEOMETRY/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

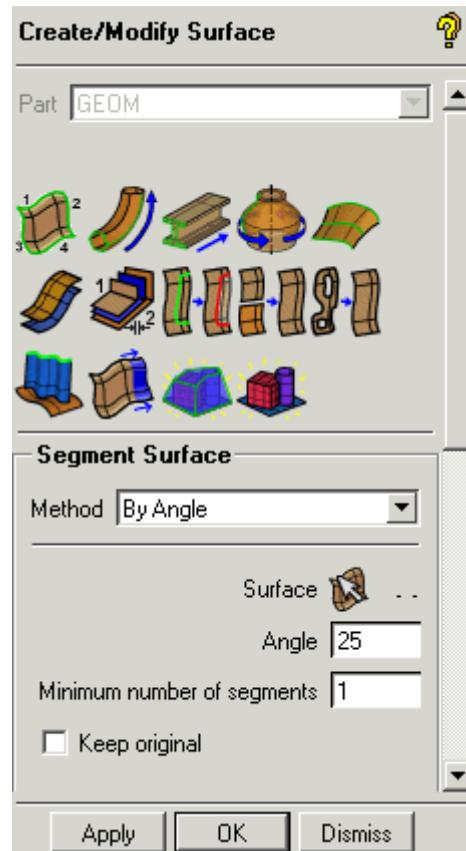
Now, based on these new curves, the user can segment the surface into regions on which to generate mesh.

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 below. In the Method select Faceted Surfaces by Angle.

Figure 4-291
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

Turn OFF the Curves names by right clicking on Curves > Show curve names and turn ON Surfaces > Show surface names in the Display Tree to see the new surfaces labeled.

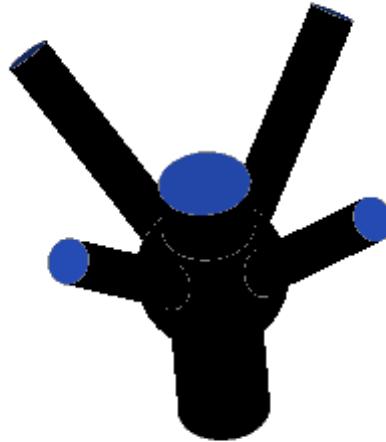
f) Parts creation

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 and complete the selection.

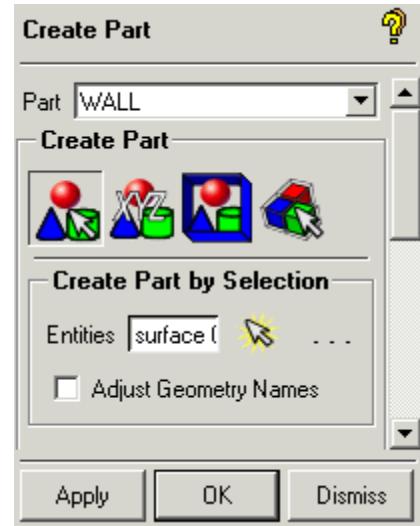
Figure 4-292
Selecting WALL entities



Select point curve surface material body densities with the left
deselect, 

In the Create Part window, enter the part as WALL and press Apply.

Figure 4-293
Creating the WALL part



Make the WALL part invisible in the Display Tree 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 below.

Figure 4-294
Part definition of inlet and outlet surfaces

OUT4/0.5 OUT2/0.6

OUT1/0.1
 OUT5/0.4 OUT3/0.7

INLET/0.8

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

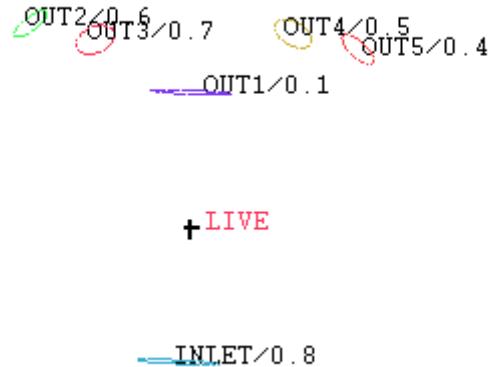
Display only curves in the Display Tree and select the remaining curves by drawing a box around them using the left mouse button Continue by adding them to the WALL part.

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 4-295
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. 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.

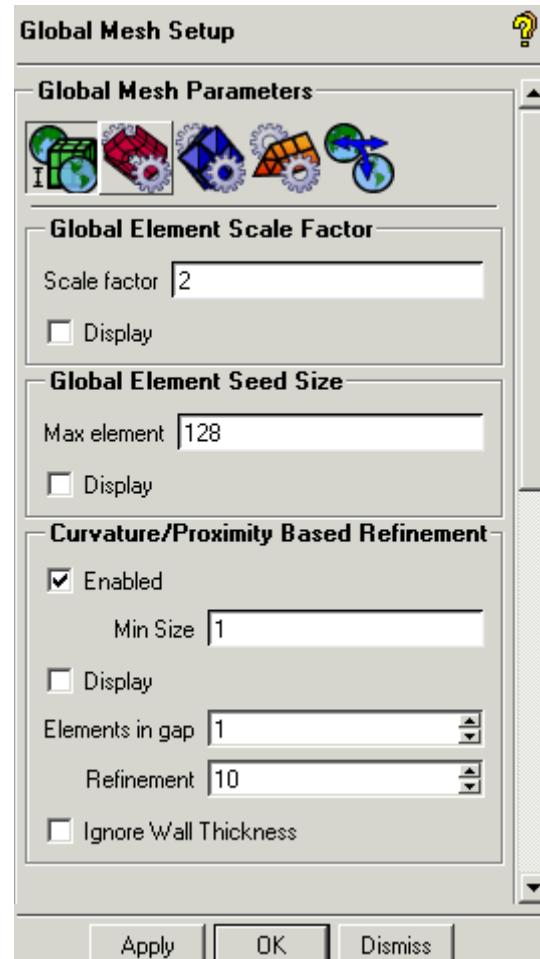
h) Assigning the Mesh Sizes

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 > Global Mesh Setup .> Global Mesh Size , it will open the Global Mesh Size window, enter a Scale factor of 2.0, a Max Element of 128, Curvature/Proximity Based Refinement of 1, Curvature/Proximity Based Refinement > Refinement of 10. Leave the other parameters at their default settings. Press Apply and then Dismiss.

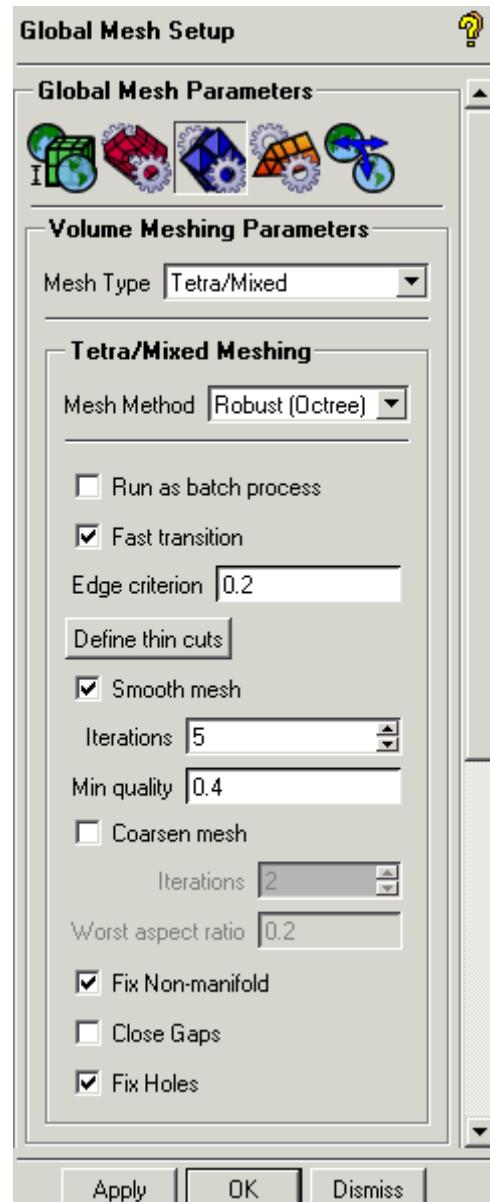
Figure 4-296
Edit the Global mesh sizes



Select Mesh > Global Mesh Setup  > Volume Meshing Parameters

 > Tetra/Mixed > Robust (Octree). Turn ON Fast transition and press Apply.

Figure 4-297
Tetra Meshing Parameters
Window

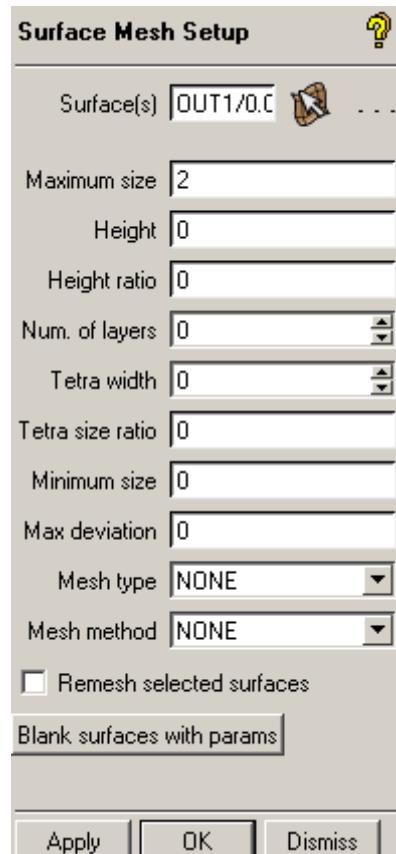


j) Setting surface mesh size

Next, Select Mesh > Surface Mesh Setup  to set the meshing parameters on the surfaces of the model.

Select  and Press the "a" keyboard key to select all surfaces.

Figure 4-298
Edit the surface mesh sizes



In this Surface Mesh Size window, enter a Maximum element size of 2 and press Apply followed by Dismiss.

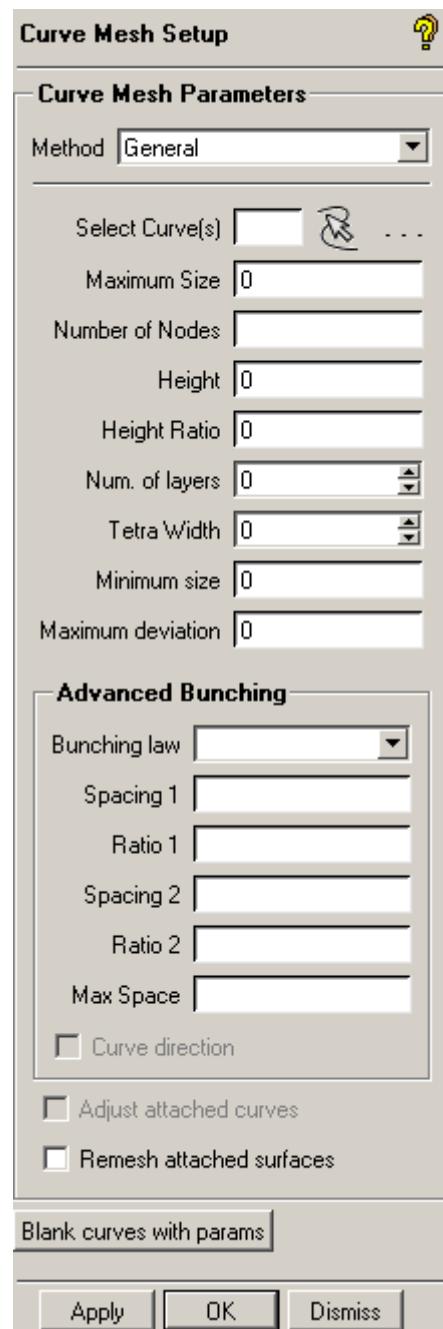
k) Setting the curve mesh size

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> Curve Mesh Setup  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, enter a value for Max Size parameters. Press Apply followed by Dismiss to close the window.

Figure 4-299
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.

I) Generating the Tetrahedral Mesh

Select Mesh > Compute Mesh  > Volume Mesh  > Tetra/Mixed > Robust(Octree) press Compute. The complete mesh is shown below.

Figure 4-300
Mesh with Tetrahedral window

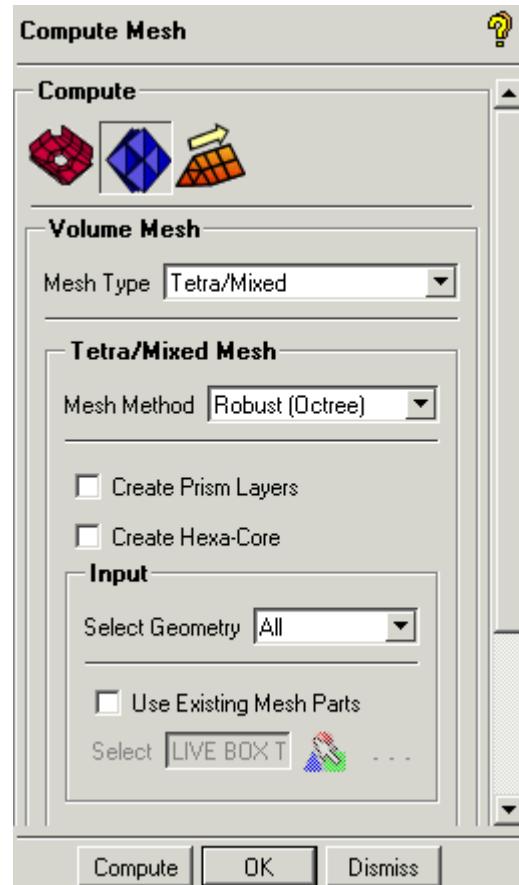
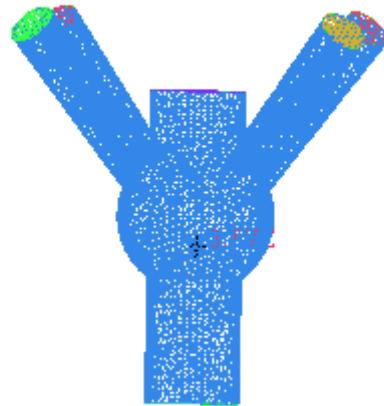


Figure 4-301
The complete mesh



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. Close the project by selecting File > Close Project.

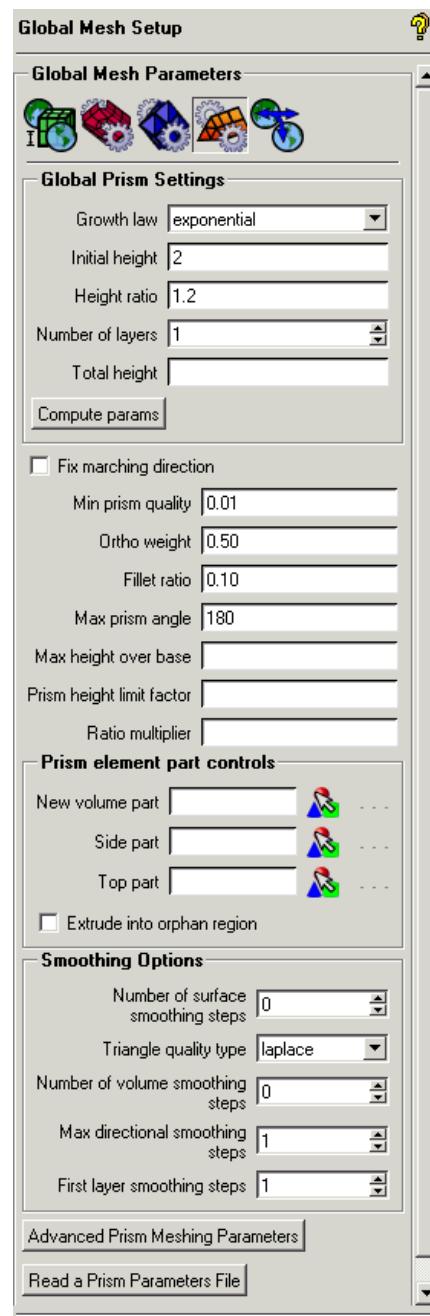
n) Adding Prism Layers

In this example, adding prism layers as a separate 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 initial 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.



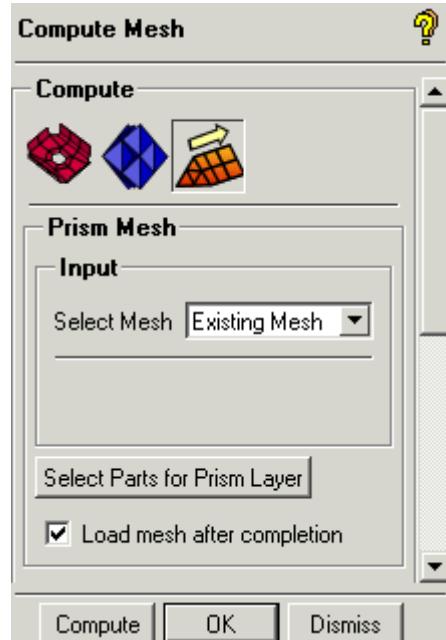
In Mesh > Global Mesh Setup > Prism Meshing Parameters , Set Initial height to 2 and the Number of layers to 1.

Figure 4-302
**Global Prism
 Parameters**



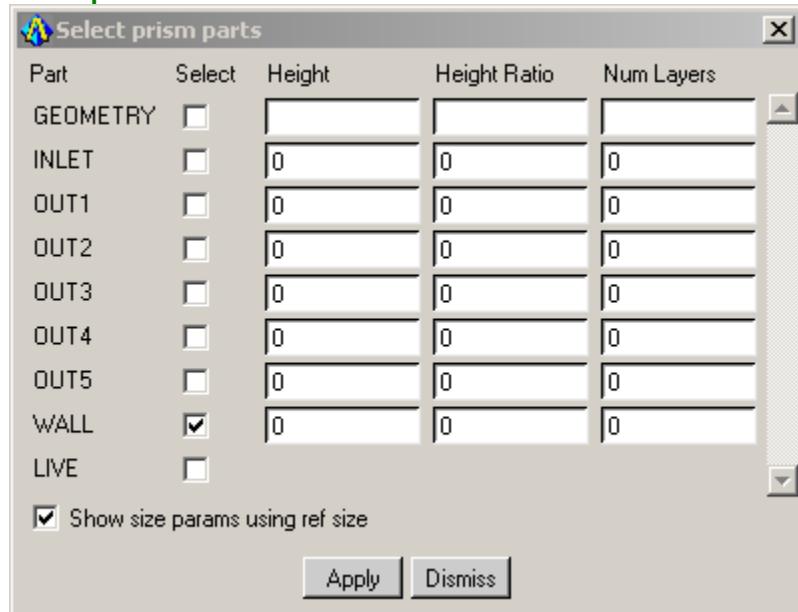
select Mesh > Compute Mesh  > Prism Mesh  to set parts to grow prism from as shown below. Click on Select Parts for Prism Layer.

**Figure 4-303
Prism mesh
parameters**



In the ensuing table, 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.

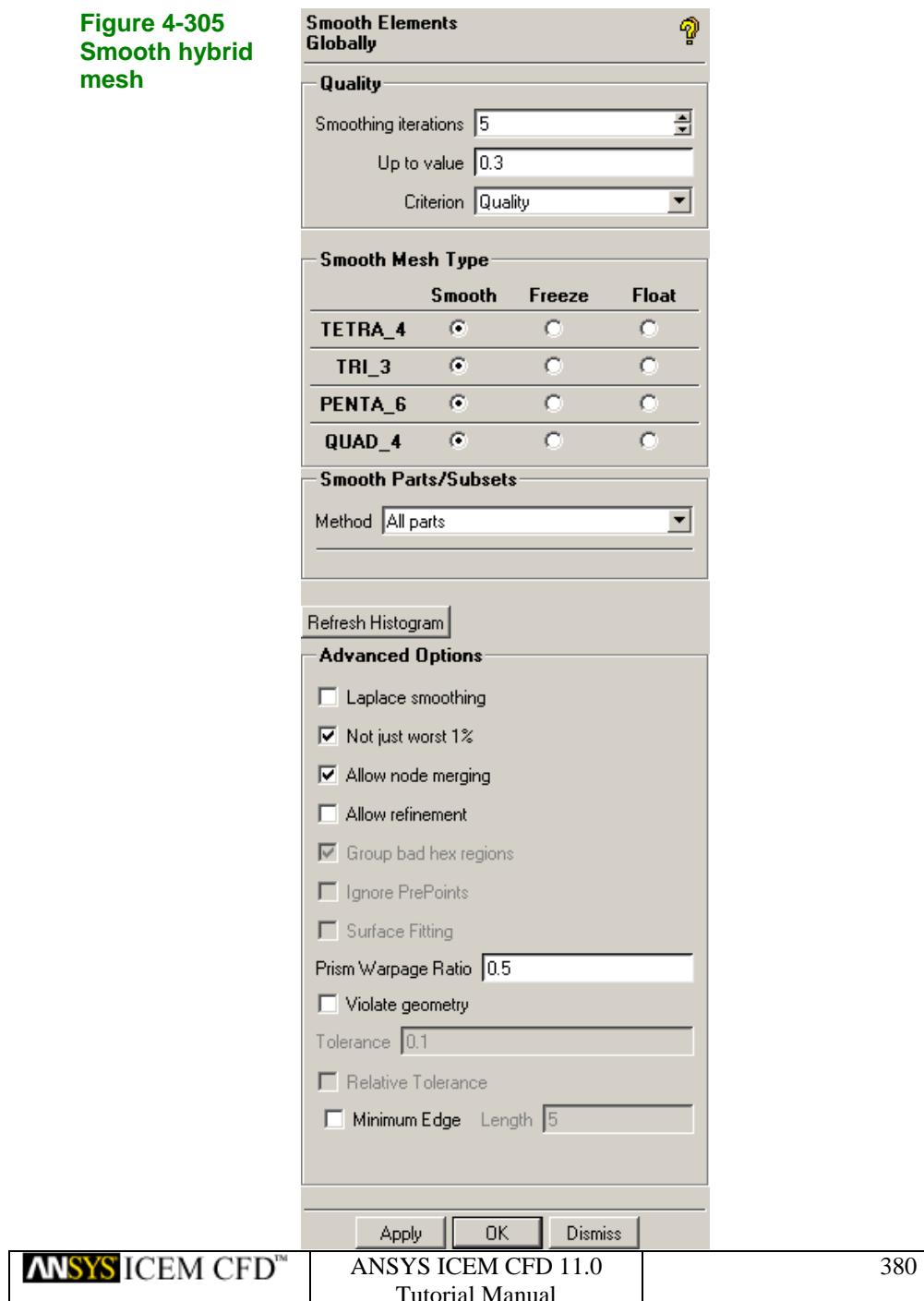
Figure 4-304
Prism parts table



Part	Select	Height	Height Ratio	Num Layers
GEOMETRY	<input type="checkbox"/>			
INLET	<input type="checkbox"/>	0	0	0
OUT1	<input type="checkbox"/>	0	0	0
OUT2	<input type="checkbox"/>	0	0	0
OUT3	<input type="checkbox"/>	0	0	0
OUT4	<input type="checkbox"/>	0	0	0
OUT5	<input type="checkbox"/>	0	0	0
WALL	<input checked="" type="checkbox"/>	0	0	0
LIVE	<input type="checkbox"/>			

On the prism form, press Apply to start prism mesh computation. Generally it is a good idea to check the quality of the hybrid mesh (tetra/prism). Select Edit Mesh > Smooth Mesh. 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 4-305
Smooth hybrid
mesh



o) Subdividing the prism layer

On the Mesh Display Trees, select Cut Plane and observe the single layer of prism, shown in the figure below.

Select Edit Mesh Tab > Split Mesh  > Split Prism The single prism layer will be split into 5 layers.

Figure 4-306 Cut plane showing a layer of prism

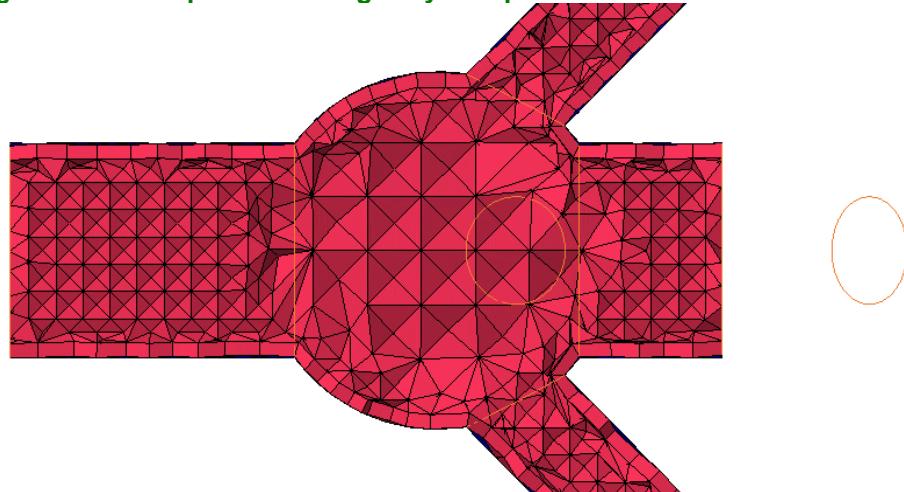


Figure 4-307
Split prism window

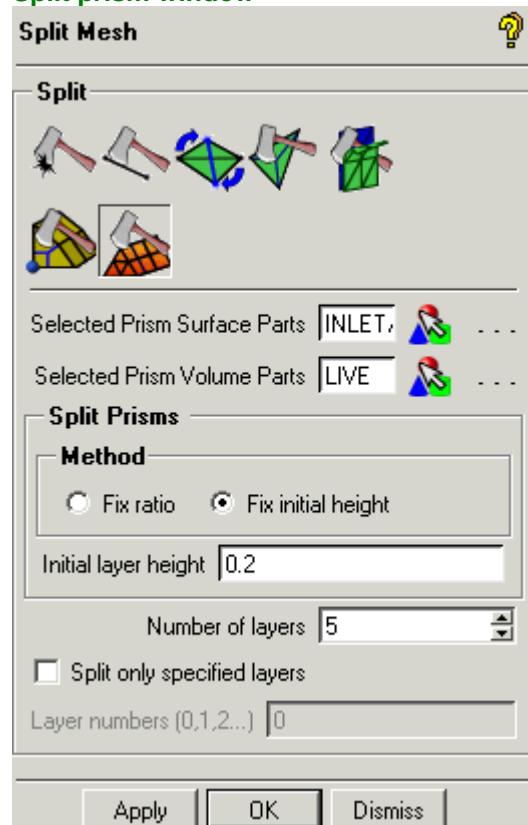
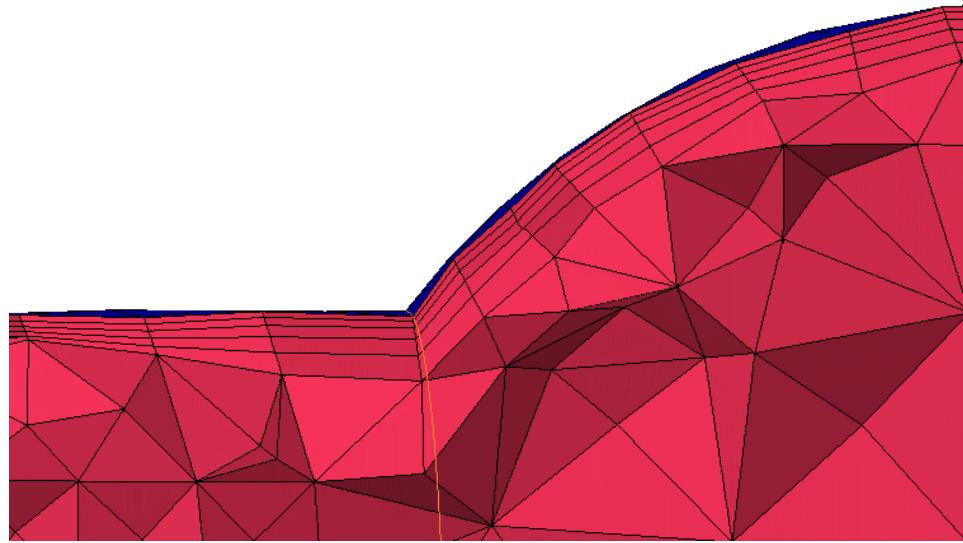


Figure 4-308
A single was subdivided into 5 layers



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 Edge

Set Initial 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.

4.5: Tetra Meshing Appendix

4.5.1: 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 ICEM CFD 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 Geometry > 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. 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 online Help.

Curves and points on sharp edges

ANSYS ICEM CFD 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 shown below.

Figure 4-309 Example-1 illustrates two flat surfaces, with a fillet surface going between the two. In Example 2, the two flat surfaces meet.



Example 1

Example 2

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... 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 > Global Mesh Setup**. 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. Then, select **Mesh > Surface Mesh Setup**. 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, 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.

4.5.2: Tetra

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.

Additional Options:

Visible

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

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	387
------------------------	--	-----

process is complete. If this option is not selected, then tetra will not load the mesh in the screen.

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.

4.5.3: 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

Go to **Edit mesh> Check mesh**, and 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. 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 Mesh > Repair Mesh> 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.

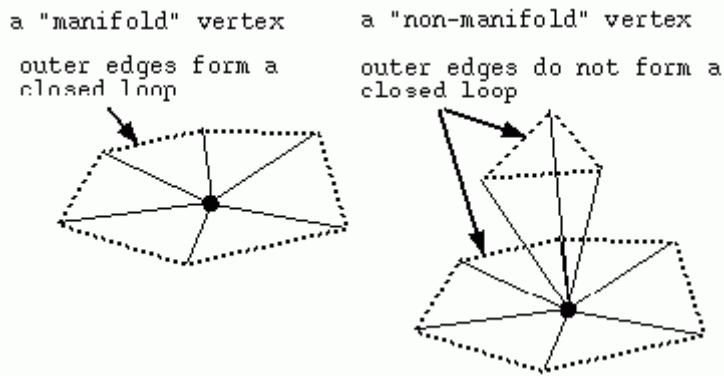
Possible problems > Overlapping Elements

It depicts elements 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 below.

Figure 4-310
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.

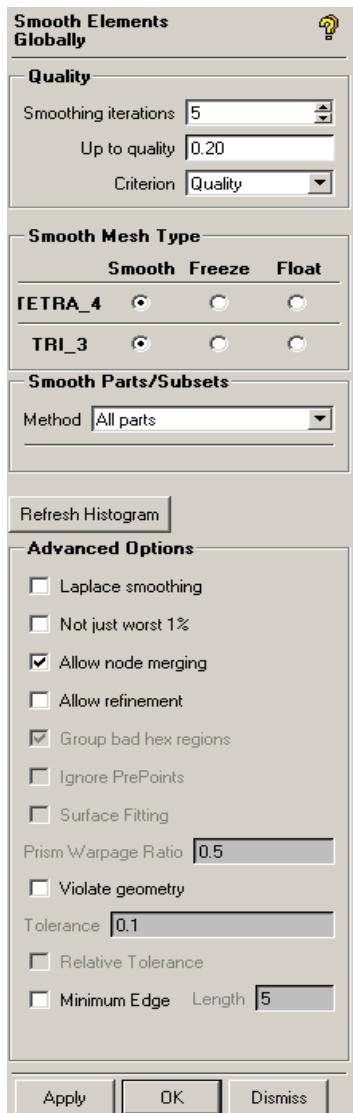
Possible problems > Un-connected vertices

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 4-311
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.

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	394
------------------------	--	-----

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.

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.

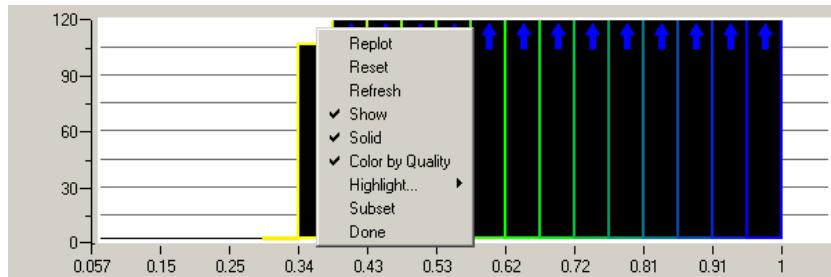
Max othogls

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.

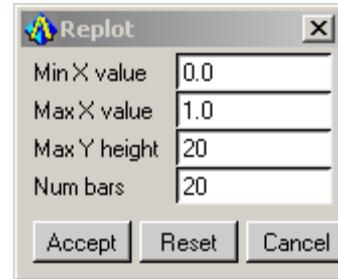
**Figure
4-312
Histogram
window**



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 and Num 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 4-313
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.

Colour By Quality : If this option is selected then it displays the elements as per their quality.

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 histogram 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.

4.6: Advanced Meshing Tutorials

ANSYS ICEM CFD 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

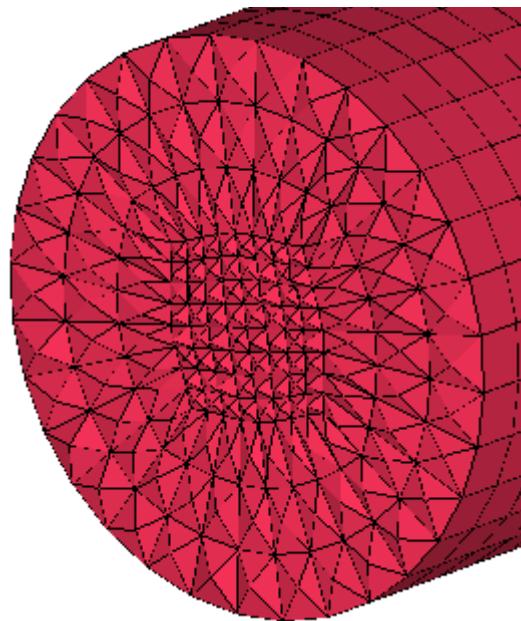
Getting a tetra mesh out of a case where the surface mesh data is available additionally for some of the regions

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
4-314
Hybrid
Mesh**

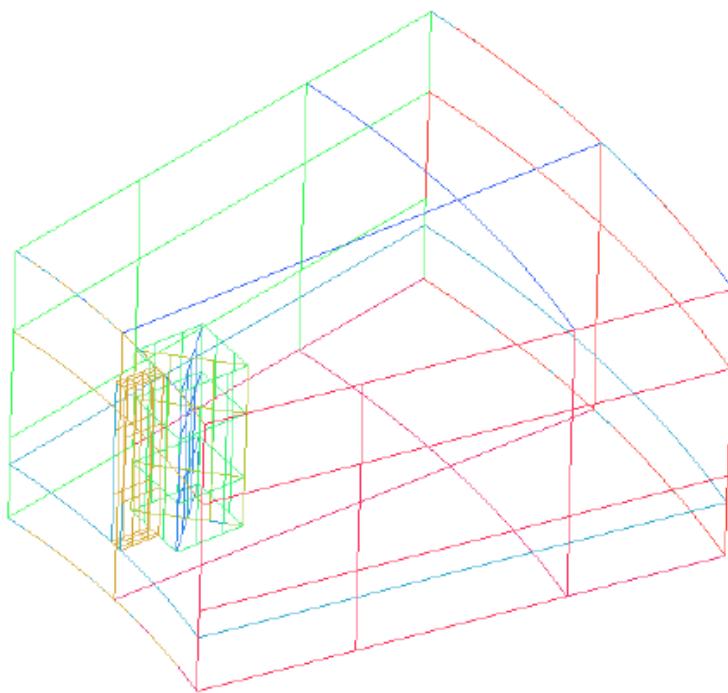


The tutorials in this section are related to these advanced steps only and will try to take the user to deal with real world applications.

4.6.1: 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 ICEM CFD

The input files for this tutorial can be found in the Ansys installation directory, under/v110/docu/Tutorials/CFD_Tutorial_Files > Gridfin.

Copy and open the geometry.tin file to your working directory.

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



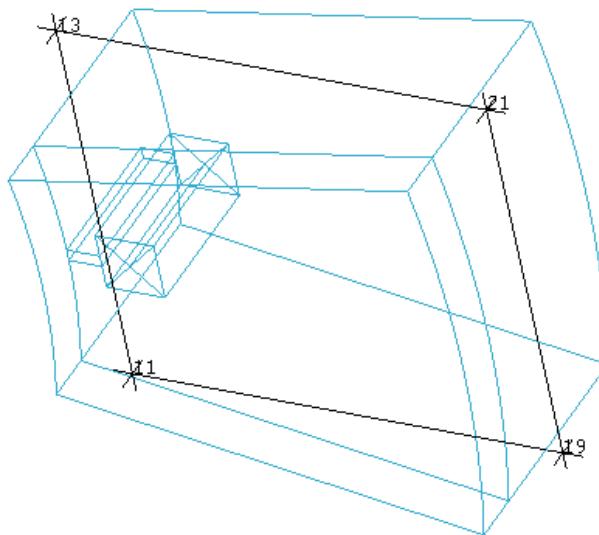
Enter LIVE for the Part.

Next to the Type, choose 2D Planar from the pull down menu.

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 the figure below.

**Figure
4-315
Initial 2D
Blocking**

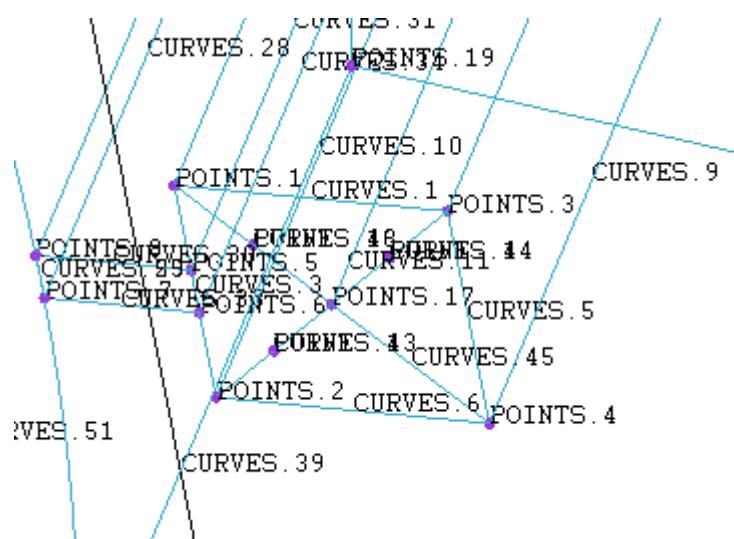


Turn on the Vertices and their numbers from the Display Tree by right mouse clicking on Blocking > Vertices > Numbers. The vertex numbers are shown above.

Right click in the Display Tree to turn on Geometry > Curves > Show Curve Names.

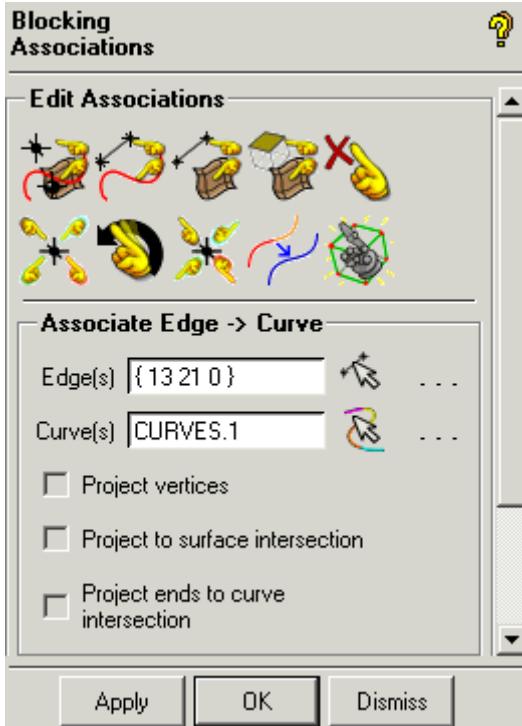
Zoom in toward the bottom of the inner blocked-shaped geometry, and it should look like the figure below.

Figure.4-316
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 below.

Figure.4-317
Associate Edge to Curve
Window

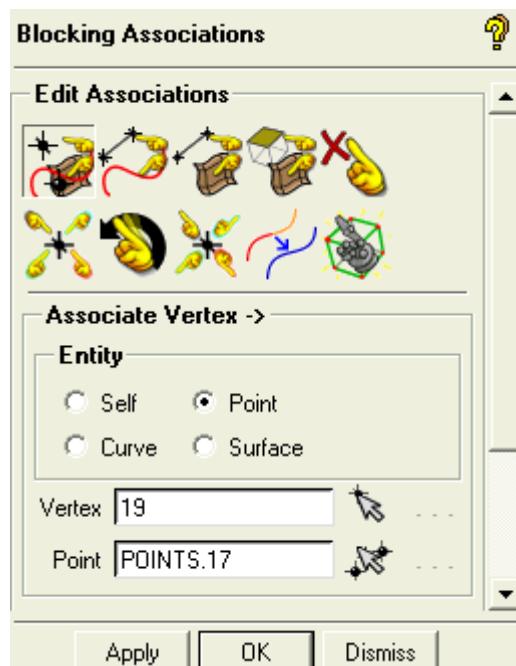


Press Apply. Similarly, associate the edge 11-13 to the curve, CURVES.3. Note that the color of Edges after association to Curve changes to Green.

Note: The user can toggle Off and ON the **Curves** and **Points** to better see what needs to be selected.

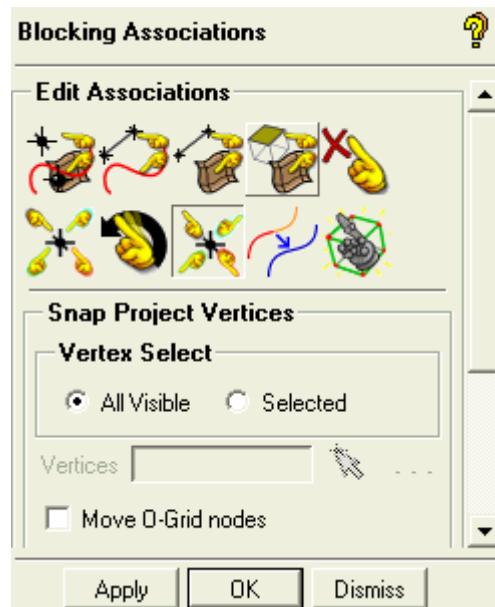
Select Associate > Associate Vertex. The Entity to associate to should be set as Point. Select the vertex 19. Then select the point, POINTS.17. Press Apply to associate the vertex to the point.

Figure.4-318
Associate Vertex to Point



Select Associate > Snap Project Vertices Toggle on ‘All visible’ as shown. Press Apply.

Figure.4-319 Snap Project Vertices Window

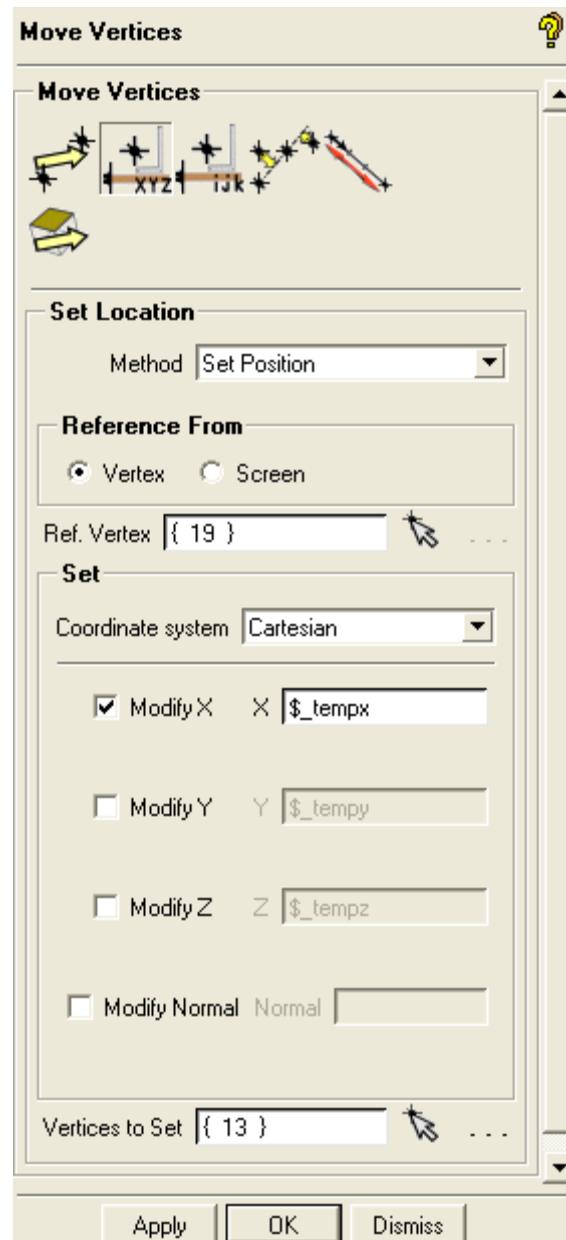


Switch OFF Curves.

Select Move Vertex > Set Location. The Set Location window will appear as shown. 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.

Select 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.

Figure.4-320
Set Position of Vertex 13

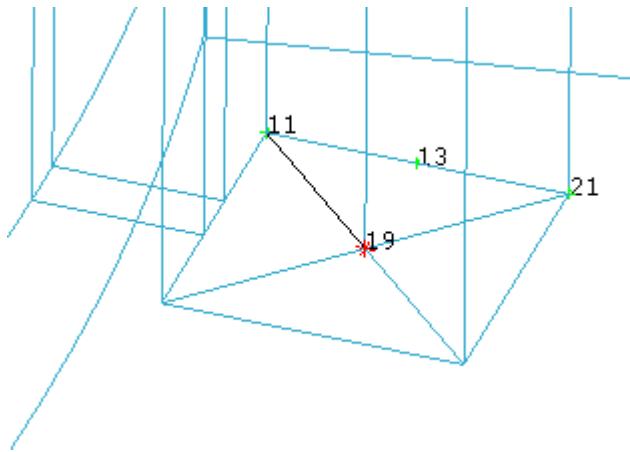


Associate > Associate Edge to Curve . Turn OFF Points. Select the edge 11-13. Turn ON Curves, and select the curve, CURVES.1. Press Apply.

Select Associate > Snap Project Vertices Toggle on ‘All visible’. Press Apply. The final image is shown below.

Note: The user should switch off Curves > Show Curves Names and Points > Show Points Name from the Display Tree 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.

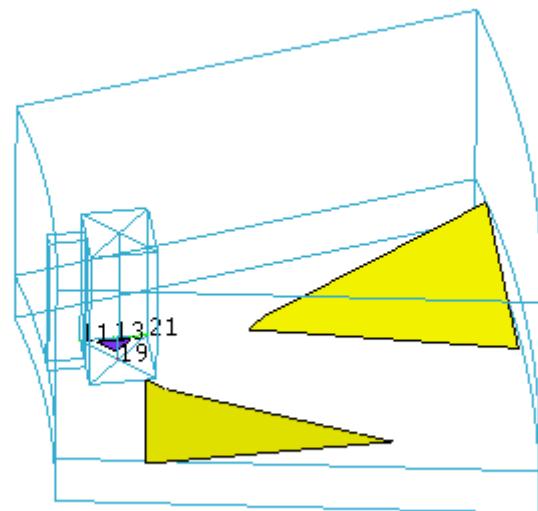
Figure.4-321
Initial Blocking after
vertex placement



e) Creating the O-grid

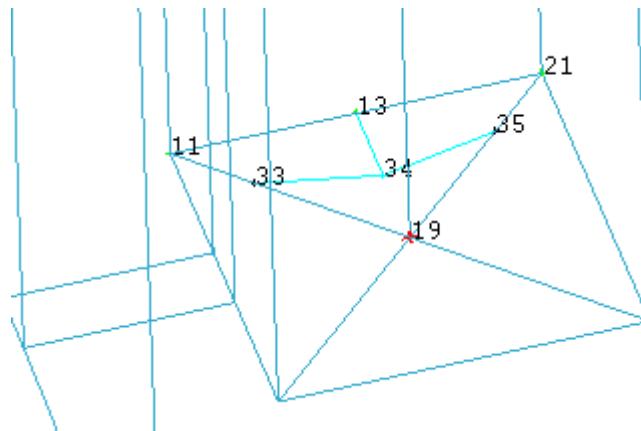
Select Blocking > Split Block > O Grid Block . Select the Face and then select the Edges 11-19 and 19-21 as shown in the figure below and press Apply.

Figure 4-322
**Selection of edges
and Faces for the O-
Grid**



The Blocking after O-grid creation is shown.

Figure 4-323
**Blocking
after O-grid
creation**



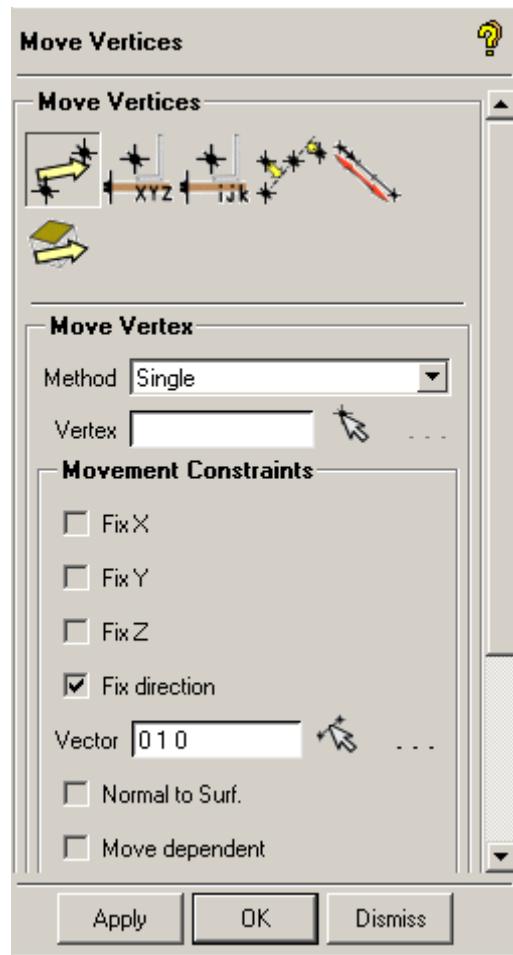
Switch on Points > Show Points Name, select, Association  > Associate

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 >

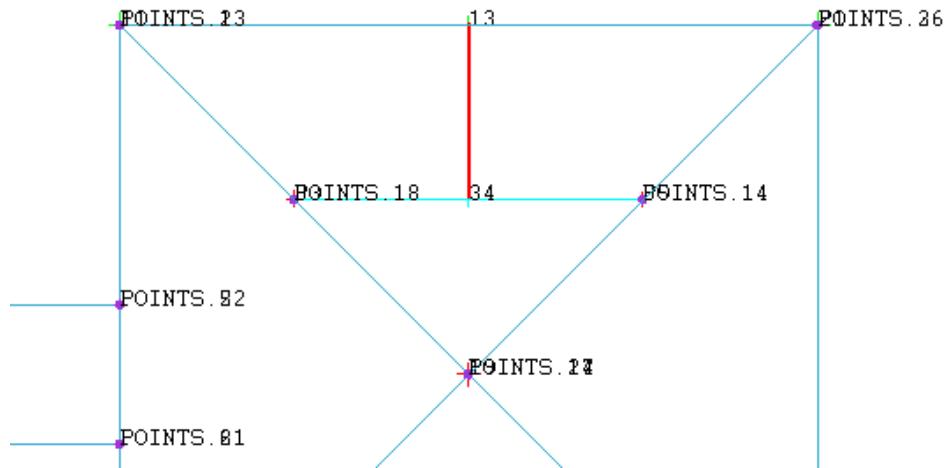
Move Vertices  > Move Vertex 
Enable Fix Direction as shown.

Figure 4-324
Fix Direction Window



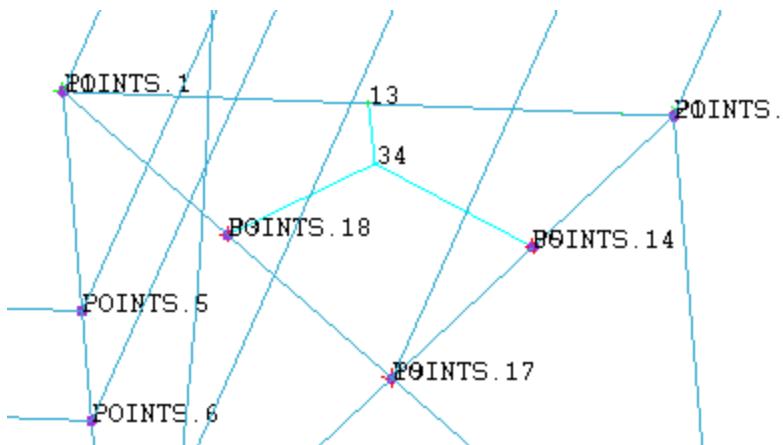
Select edge 13-34. This will select the Direction highlighted as shown.

**Figure
4-325
Move
Vertex
Fix
Direction
Option**



Select vertex 34 and drag it towards vertex 13. Place the vertex closer to vertex 13 as shown so that all the Blocks are of good quality.

**Figure
4-326
Blocking
after
vertices
placements**

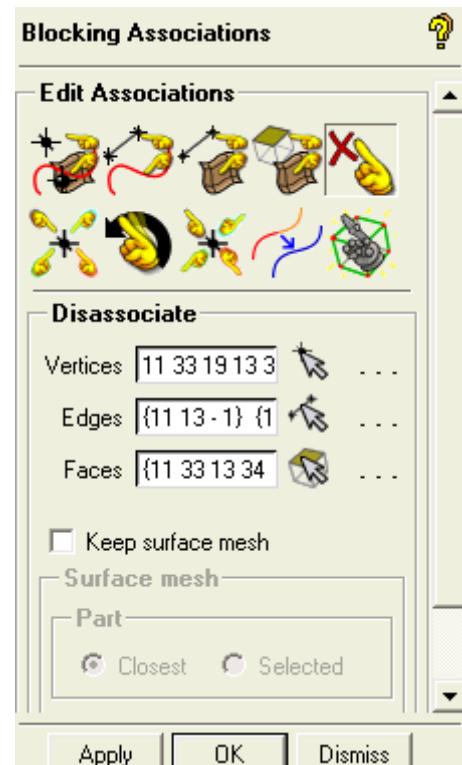


f) Resolving Other Grids

Copy/Rotate can be used to resolve other triangular portions 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 Disassociate window appears as shown in the figure below.

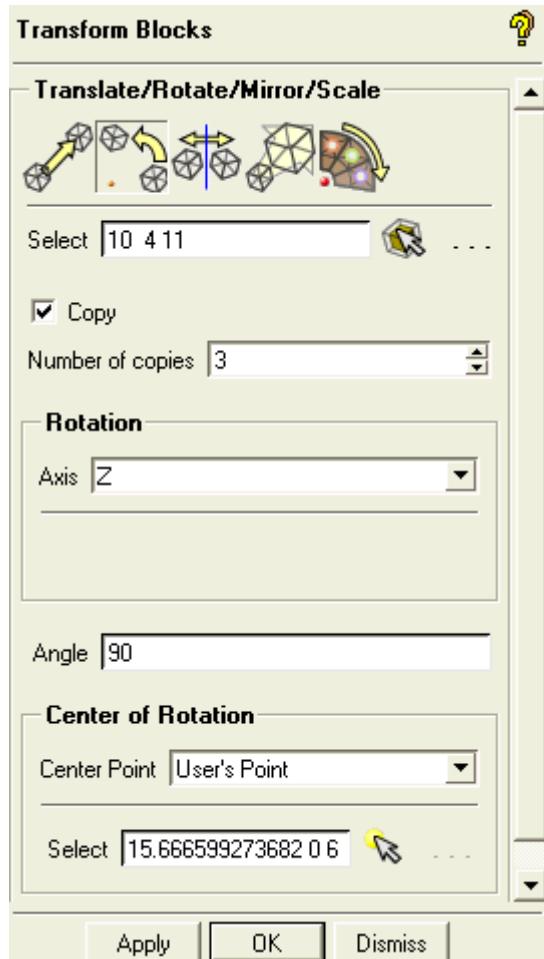
Figure 4-327
Disassociation Window



Select all the Edges, Faces and Vertex and Press Apply. Note that the color of edges and vertices changes to blue.

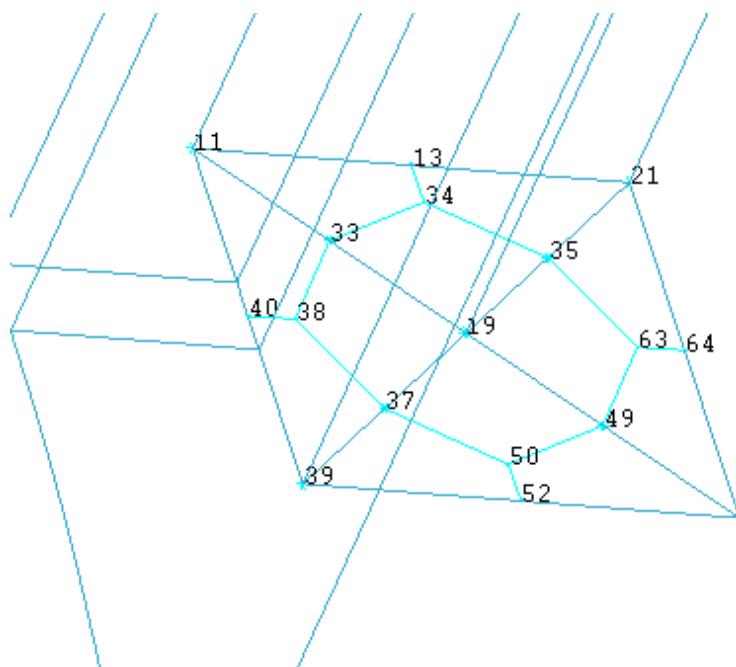
Go To Blocking > Transform Block > Rotate Block .

Figure.4-328
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 below.

Figure.4-329
Blocking after
Transformatio
n

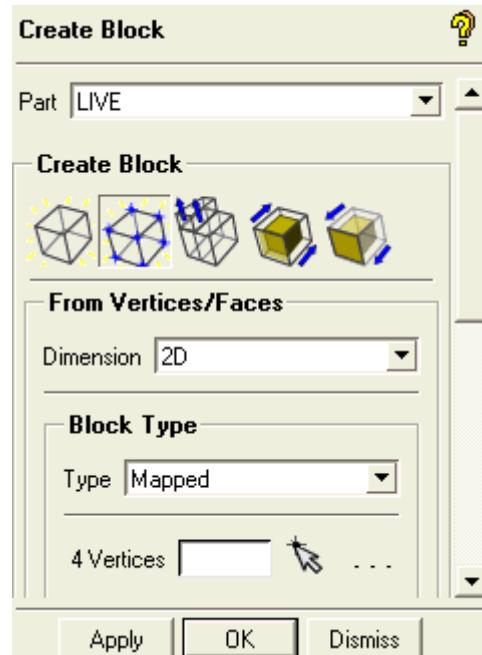


g) Creating remaining Blocks

Blocking > Create Block > From Vertices/Faces , in the dimension select 2D and in the Block Type select Mapped as shown.

Note: Part Name will be LIVE by default.

Figure.4-330
Create Vertices/Faces

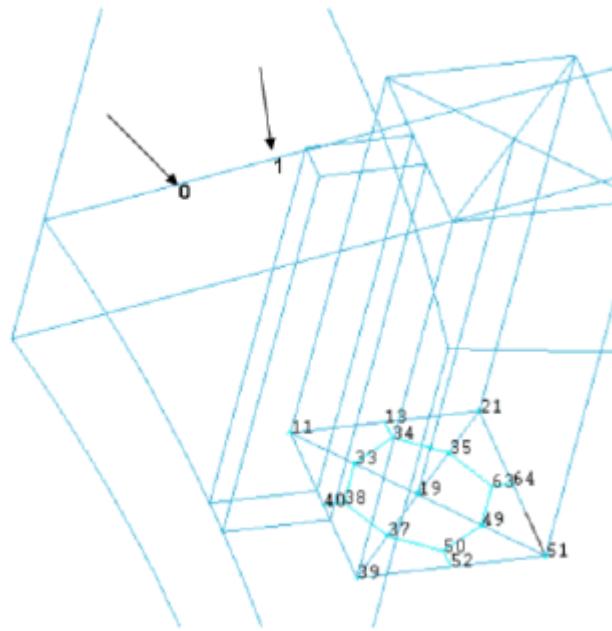


Select vertices 11 and 13 with the left mouse button in that order and press middle mouse button.

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 below. Press middle mouse button to accept the selection and press Apply.

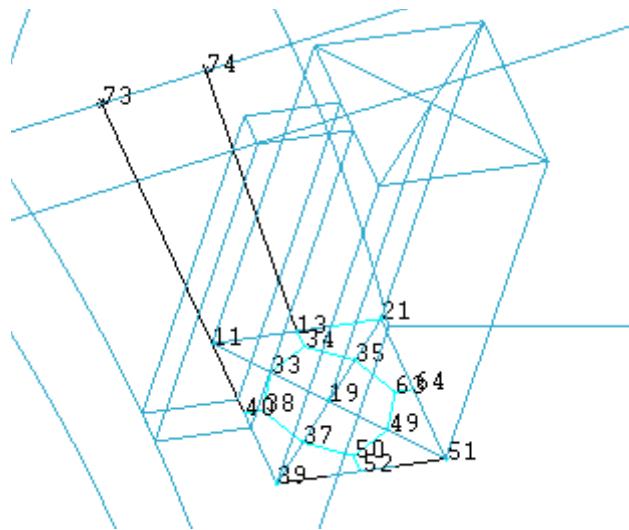
Figure.4-331
Vertex location for block and Material Selection Window



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.

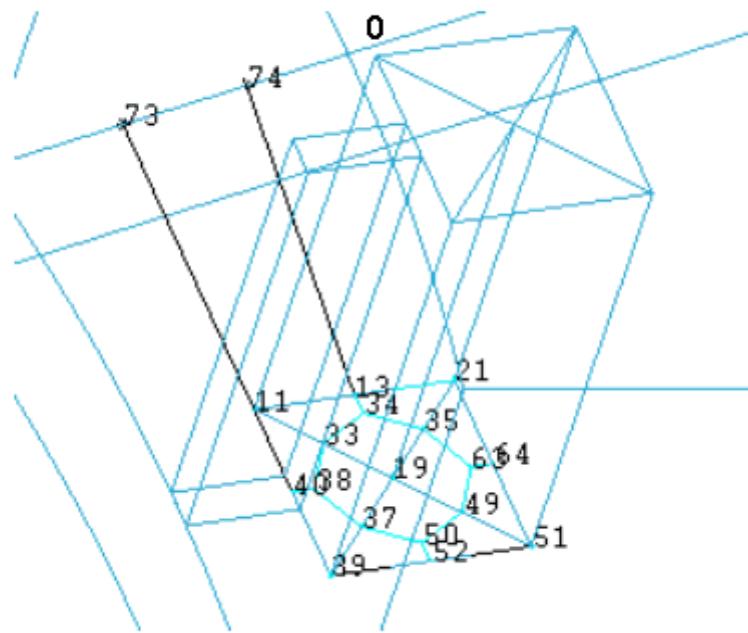
The blocking after this operation should like as shown below.

Figure.4-332
Blocking after creation of block



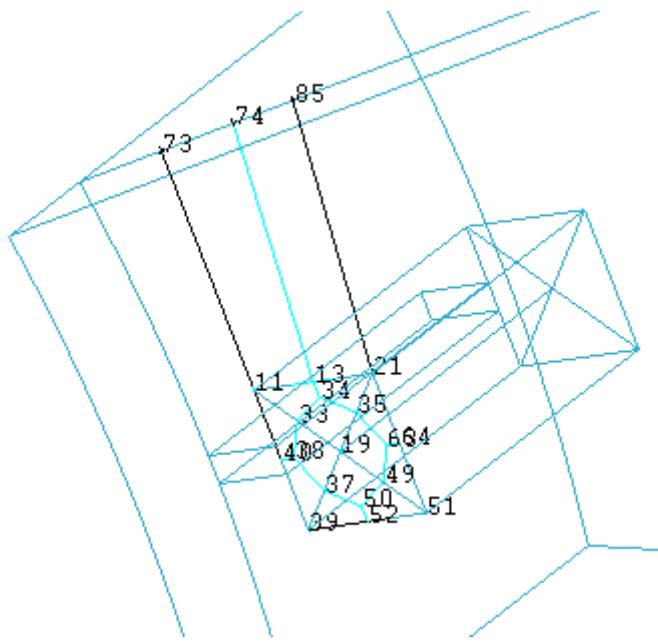
Similarly create the block by selecting the vertices 13, 21 and 74 (in that order) and press the middle mouse button. Screen select for vertex 0 as shown.

Figure 4-333
Selecting vertices for another block creation



The blocking after creation of the second block is shown below.

Figure.4-334
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 above, 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.

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.

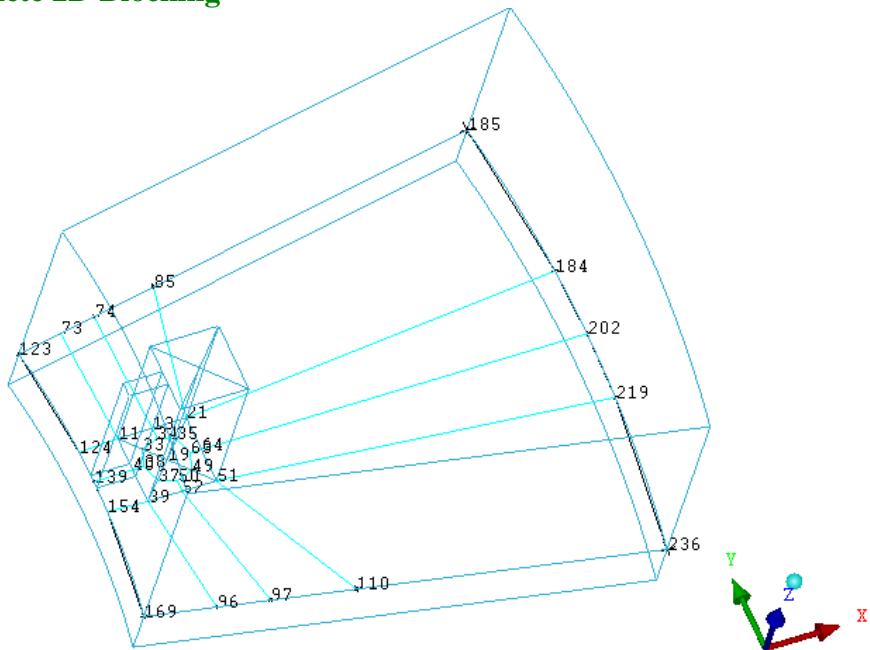
Select vertex 21, 64 and 184 and then select the point corresponding to the position of vertex 202.

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 below.

Figure 4-335 Complete 2D Blocking



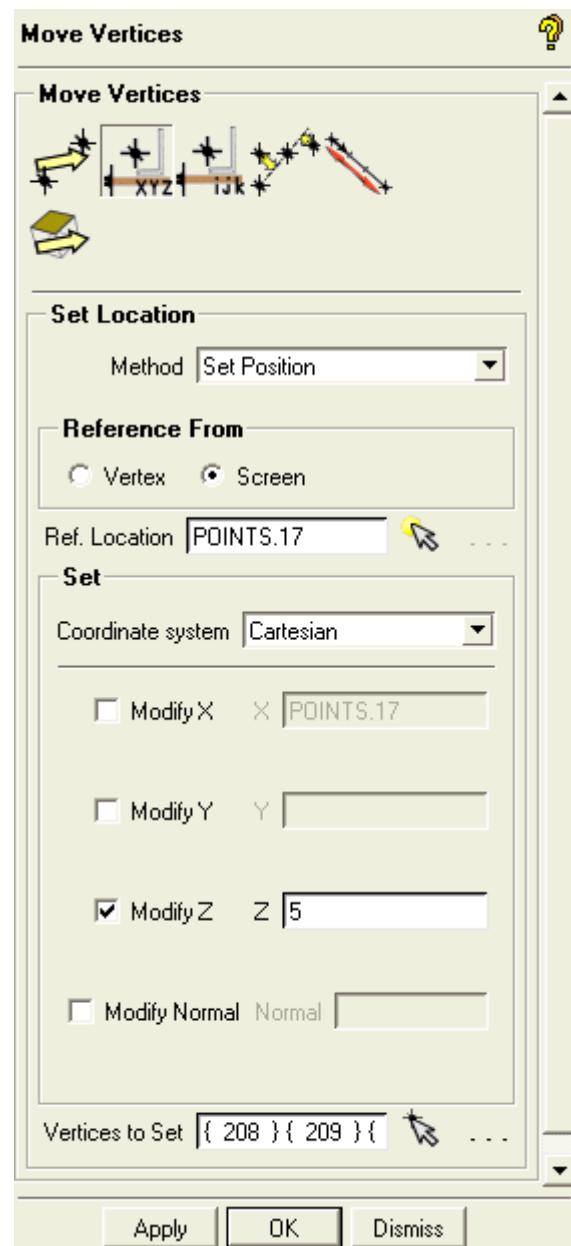
Note: It is advisable to switch off the Points when position of vertex is to be selected.

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.

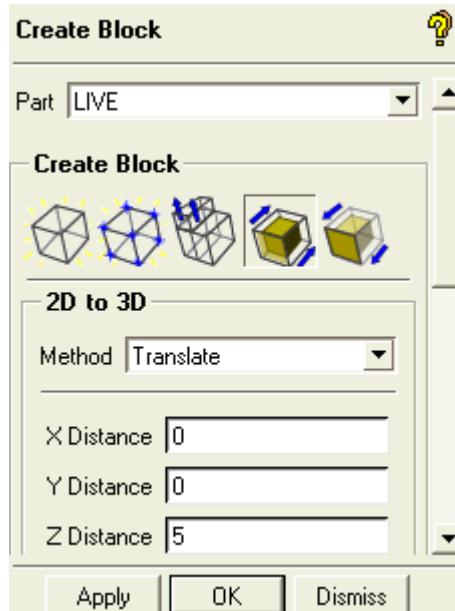
Blocking > Move Vertex  > Set location.  A new window will appear as shown below. Switch On Points > Show Point name. Toggle on Modify Z, select all the vertices. Enter 5 and press Apply to move complete topology to Z=5 plane. Click on Dismiss to close the panel.

Figure.4-336
Vertex Positions
Window



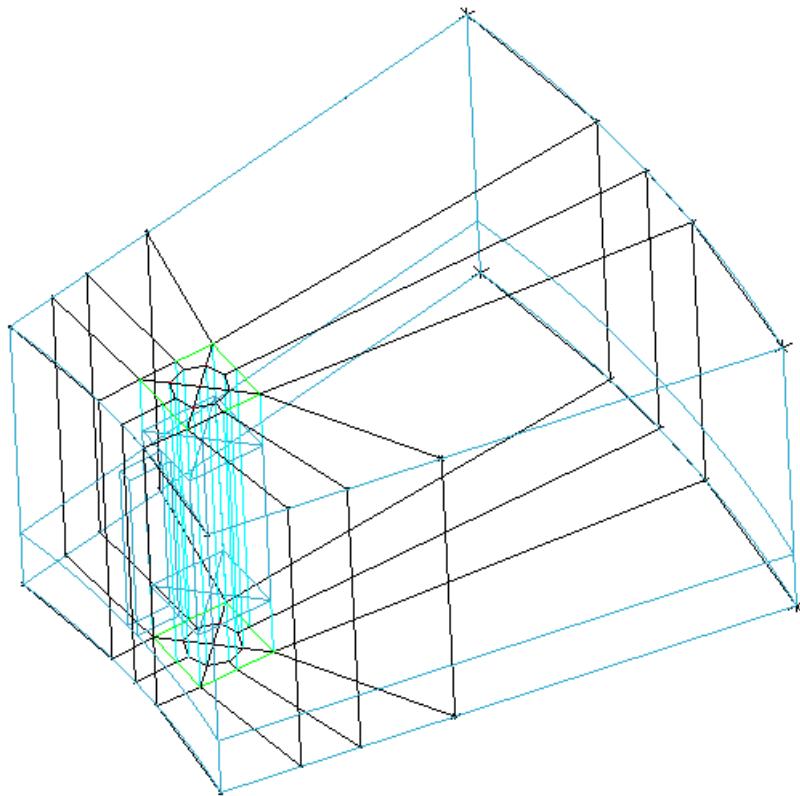
j) 3-D Blocking**Extruding 2D blocking**Blocking > Create Block  > 2D to 3D 

For the method, select Translate. Enter the value 5 and press Apply.

**Figure. 4-337
Extrusion Window**

Switch off Vertices and Points. The extruded 3-D blocking is shown.

**Figure.4-338
3D Blocking after Extrusion**



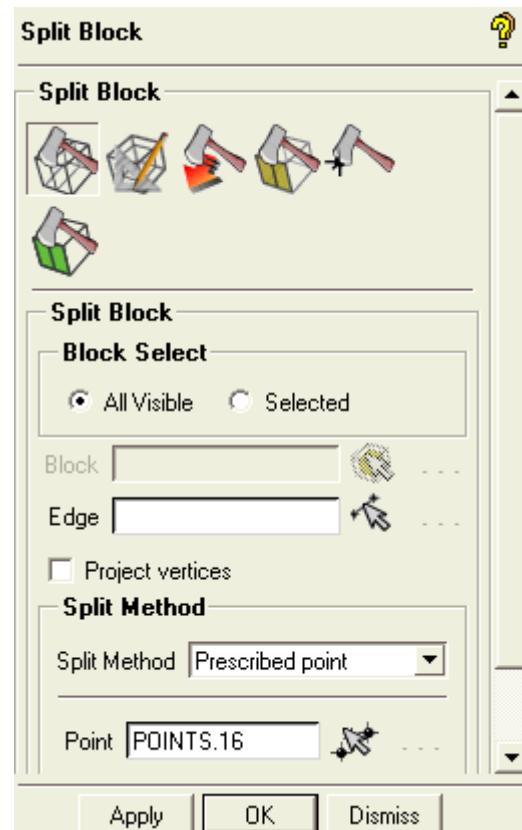
k) Getting other regions Resolved

Go to Blocking > Split Block  > Split Block .

Switch on Point > Show Point Names.

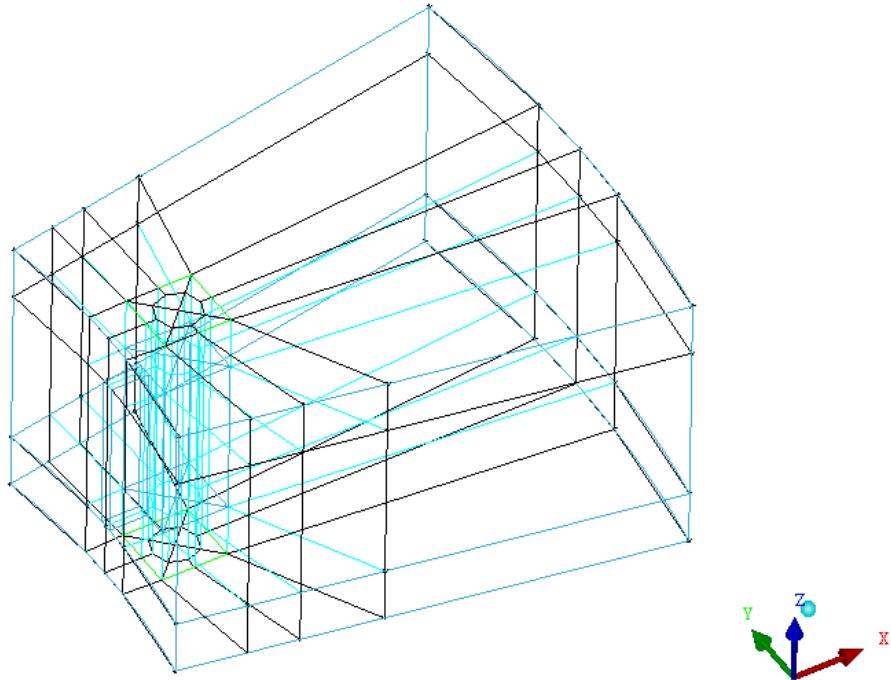
Now select prescribed point. Click on Screen select and accept POINTS.16 using the left mouse button, and press the middle mouse button to accept the selection as shown. Select an edge representing the Z-direction with the left mouse button and Press Apply.

Figure.4-339
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.

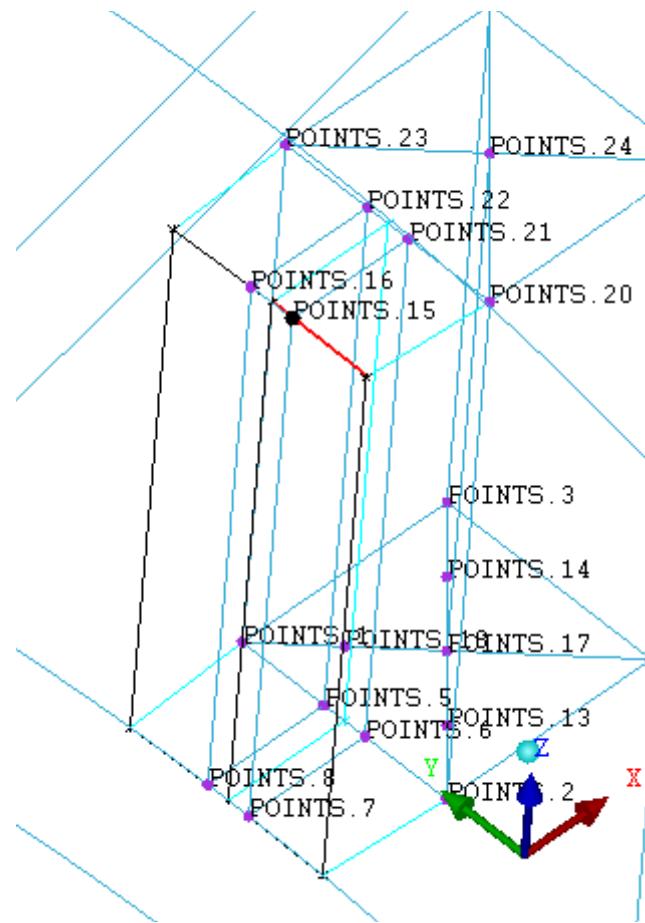
Figure.4-340
Blocking after splitting



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 You will need to readjust the index control so that the ranges are I:0-1, J:0-1, 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.

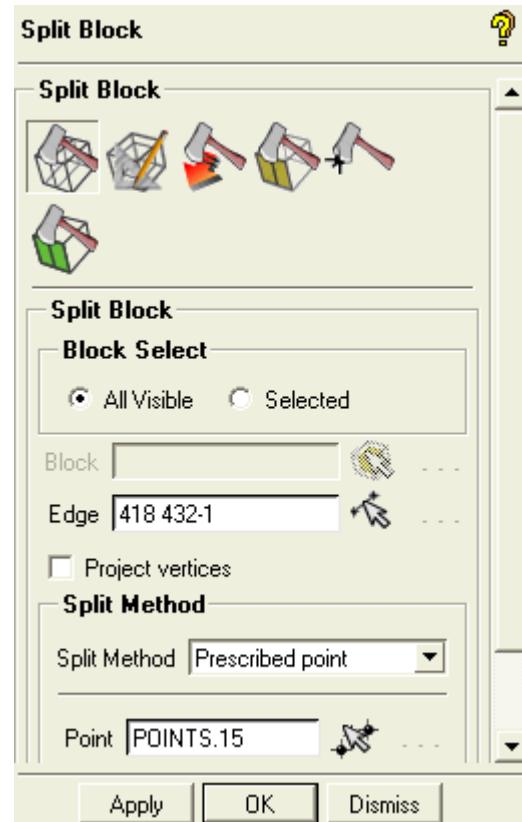
Blocking > Split Block > Split Block . Switch on Points > Show Point Names. Select one of the circumferential edges on the right block as shown below.

Figure.4-341
Edge Selected



Select POINTS.15 as shown.

**Figure.4-342
POINTS.15 selected**



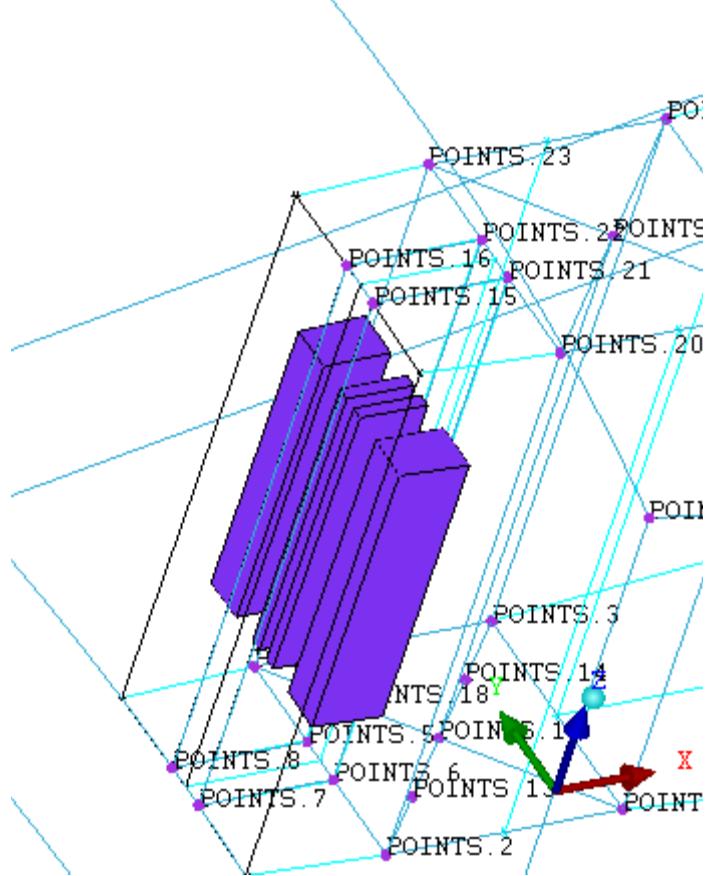
Select the method ‘Prescribed Point’ and select POINTS.15. It is common that upon split, one may start seeing extra blocks/edges. In that case, blocking should be restricted as explained in the previous step.

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 below.

Note: Don’t use the Whole Block option in Blocking > Blocks (Display Tree).

Figure.4-343
Block after Split in Selection of blocks for Splitting

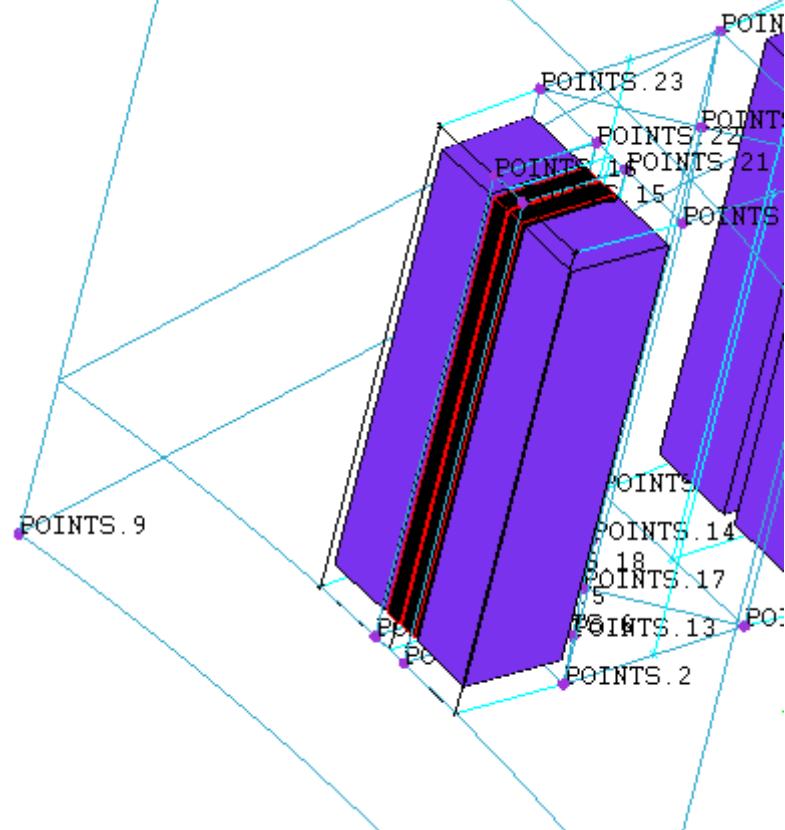


Switch OFF Blocking > Blocks. All blocks to the VORFN part by going to Parts >VORFN (keep it in Off Mode) > Add to Part > Blocking

Material, Add to Part by Selection  with Blocks, and selecting the Block as shown. Press the middle mouse button and then Apply.

Figure.4-344

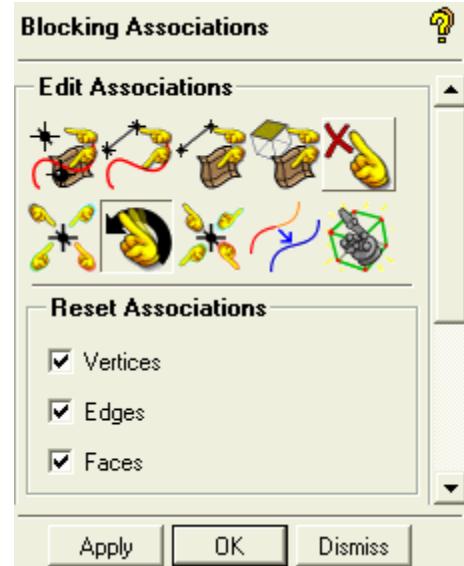
ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	431
------------------------	--	-----

Blocking with VORFN block selection

Select Blocking > Association > Reset Association
Enable Vertices, Edges and Curves and Faces as shown.



Figure.4-345
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 .

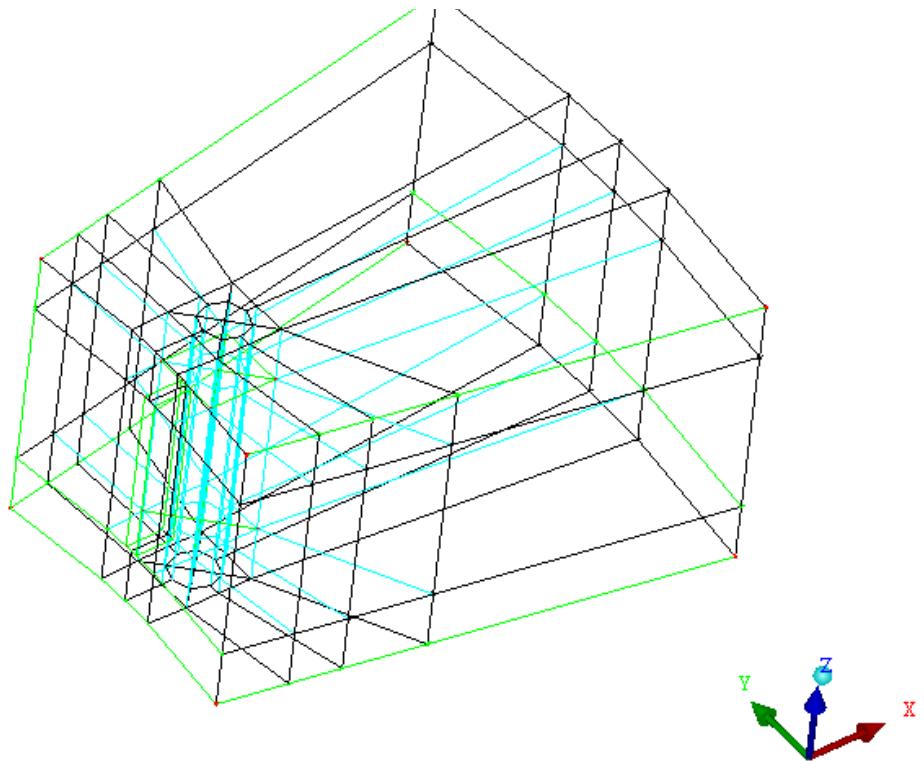
Note: Associate the edge to their respective Curves as much possible.

Associate all the circumferential edges to their respective curves wherever the curve exists.

Similarly, associate the vertex to its nearest point wherever possible.
Switch ‘On’ Points.

The Blocking should look like the figure below.

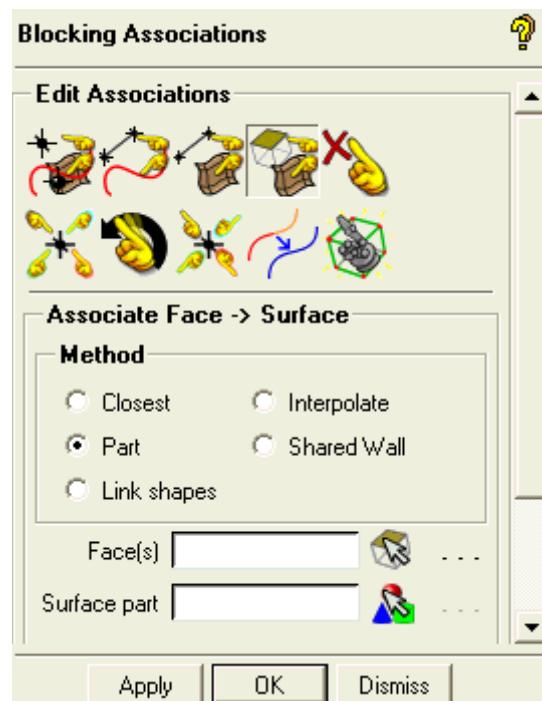
Figure.4-346
Blocking after associations and vertices placement



I) Resolving zero thickness walls

Select Associate >Associate Face to Surface . For Method, select Part.

Figure.4-347
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.

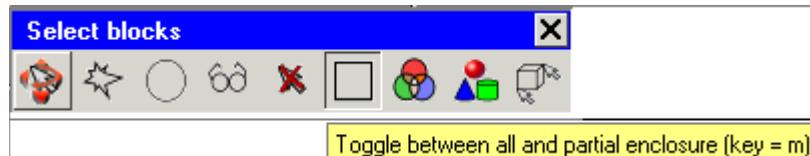
In the Display Tree switch ‘Off’ All Parts except PLATE 1, PLATE 2, SHELL and LIVE.

Press hotkey ‘h’.

Turn On Faces. Select the FACES and its corresponding Part.

Note: Make sure that “**Toggle between all and partial enclosure**” is enabled as shown in the toolbar.

Figure.4-348
Toggle
Between All
and Partial

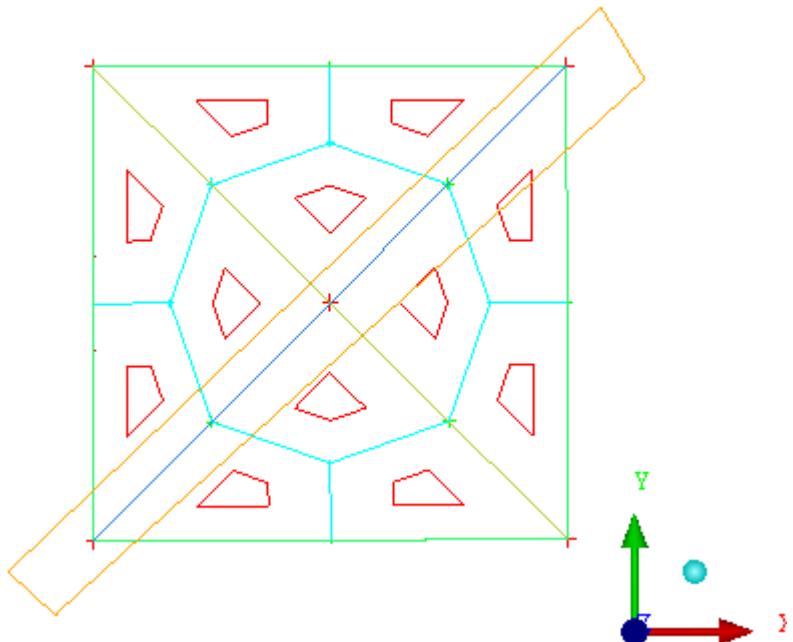


option

Use Polygon  selection to select the Faces.
 As shown below, we can easily select the Face to be associated to PLATE 1.
 Now in the Surface Part window select Plate 1.

Figure.4-349

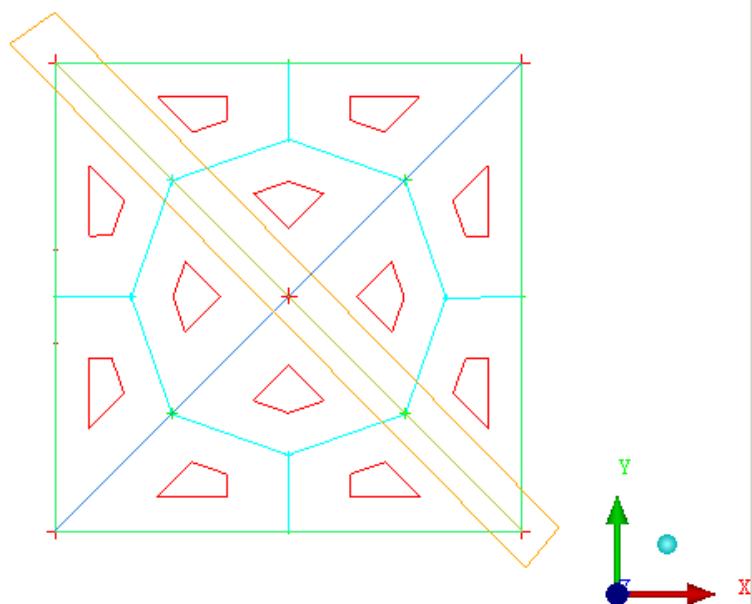
**Faces
selected to
be
Associated
to PLATE 1**



Similarly select the Face to be associated to PLATE2 as shown below.

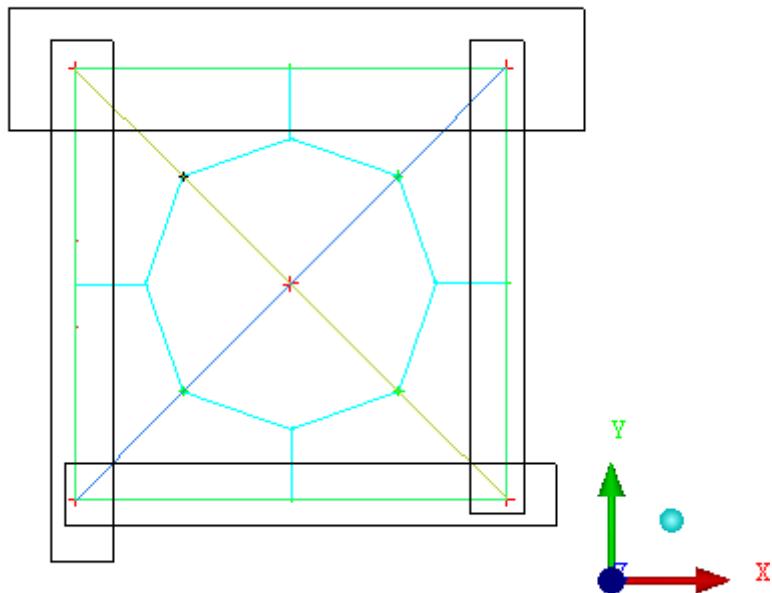
Figure.4-350

**Faces
selected to
be
Associated
to PLATE 2**



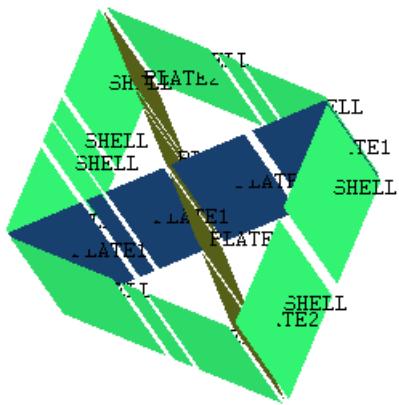
Note: Select the following region using Box Selection. Select one by one the four regions as shown. The Part must be **Shell**.

Figure4-351
Faces
selected to
be
Associated
to SHELL



To see the face projection toggle on the Faces > Face Projection. The Face projection is shown below.

Figure4-352
Blocking with face projection on family PLATE1, SHELL and
PLATE2



Switch on all Parts and switch Off Faces in the Display Tree.

Go To Blocking > Index Control and Reset.

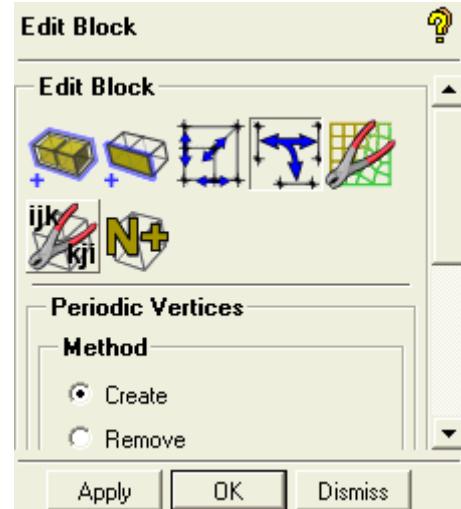
Note: The two faces corresponding to the HUB location remain unprojected.

m) Defining Periodicity

Note: Defining periodicity in ANSYS 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.

Figure.4-353
Periodic Vertices Panel

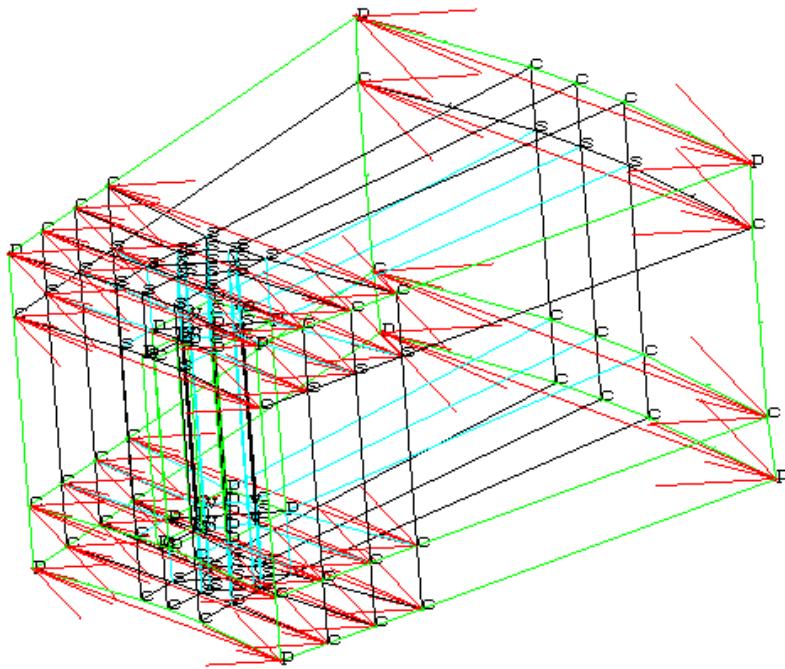


Toggle on Create, select a vertex lying on the periodic plane PERIODIC1 and the corresponding vertex on periodic plane PERIODIC2. This will define the periodicity between two vertices. One can see the periodicity by switching on, Vertices > Periodic from the Display Tree.

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 Vertices > Periodic we get the display shown below.

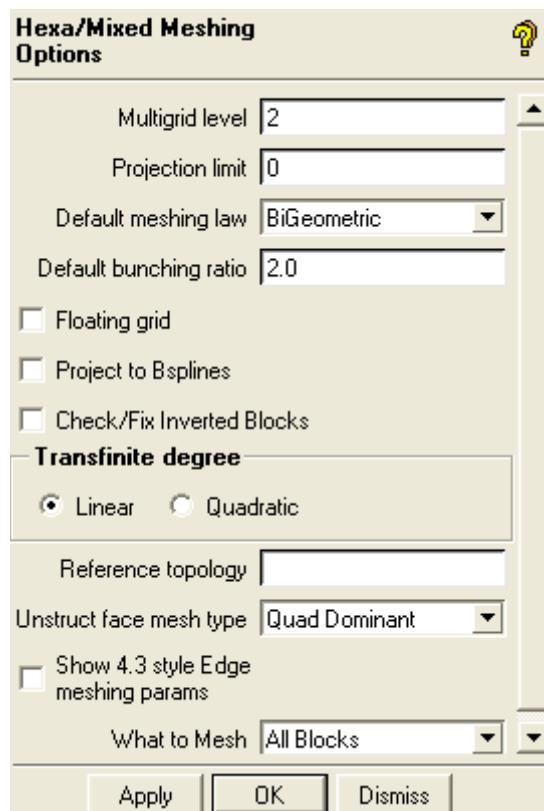
Figure.4-354
Periodicity in the blocking



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, and press Apply.

Figure.4-355
Meshing option window

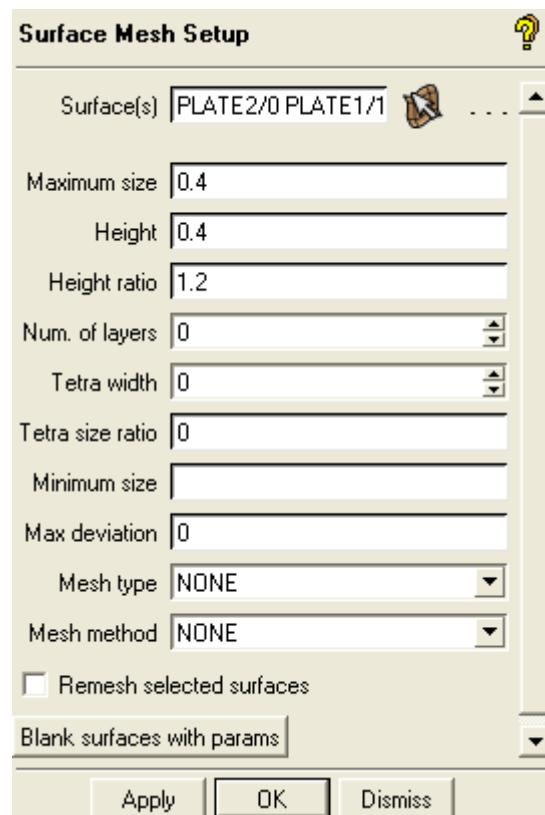


Press Mesh > Set Surface Mesh Size to open the Mesh parameters window.

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.

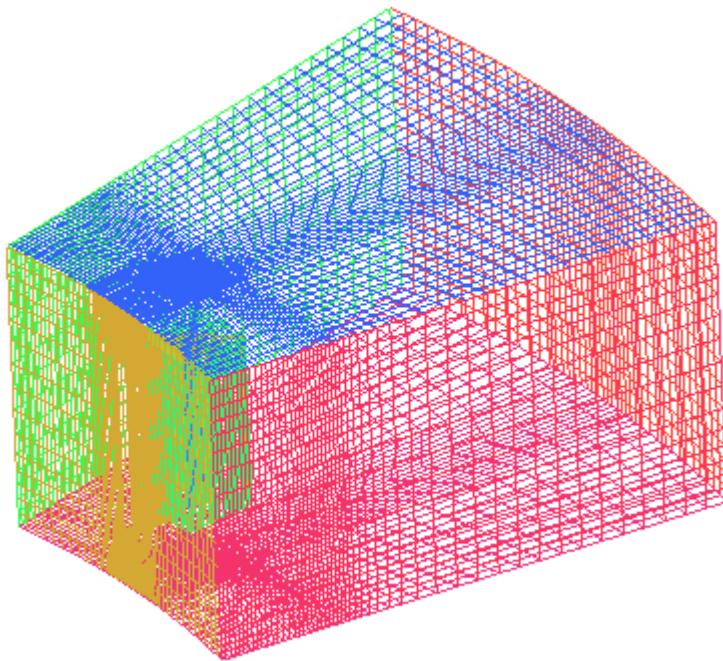
Press Dismiss to close the window.

Figure.4-356
Meshing Parameter
window



Blocking > Pre-mesh Params > Update Size toggle on ‘Update All’ and press Apply.

Figure.4-357
Mesh in geometry

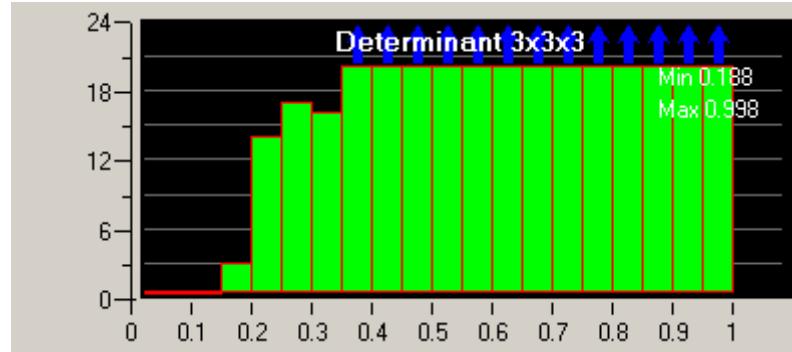


In the Display Tree turn on PreMesh>Right Click -Project faces and answer ‘Yes’ when asked whether to recompute the mesh. Turn on the Mesh in the Display Tree to see the mesh as shown.

o) Checking the Mesh Quality

Select Blocking > Pre-mesh Quality. For the Criterion, select Determinant (3x3x3 stencil) to view the histogram as shown.

Figure.4-358
Determinants histogram

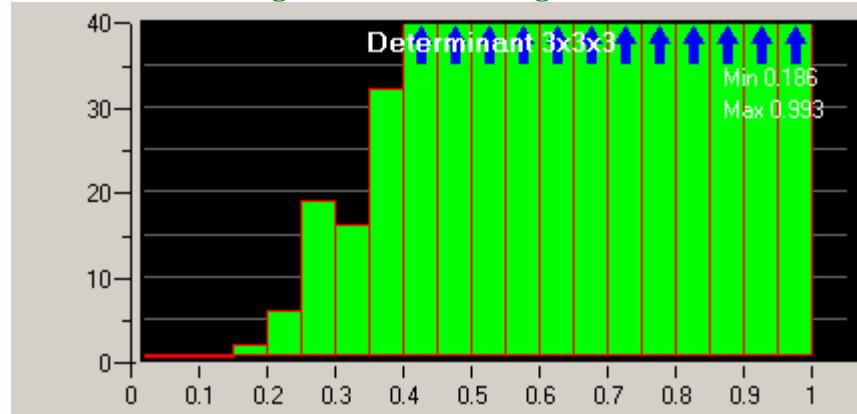


As is shown, there are many bad determinants in first bar from 0-0.05. This happens because inverted blocks were created while creating the block.

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 (3x3x3 stencils). In the Mesh window select 'Yes' to recompute the mesh. Now the histogram appears as shown, without bad determinants.

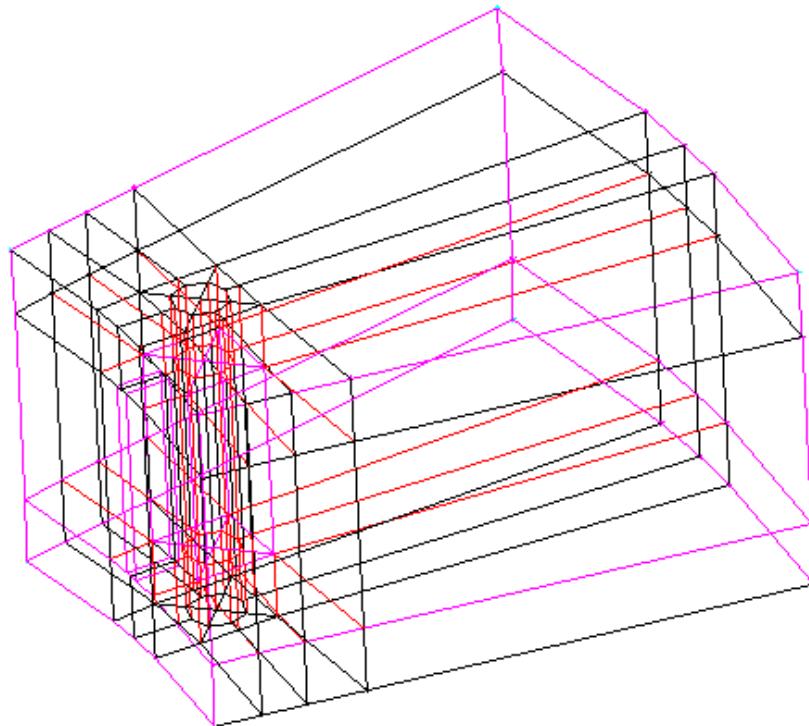
Figure.4-359
Determinants histogram after inverting bad blocks



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.

Figure.4-360
Blocking before reduction of number of blocks

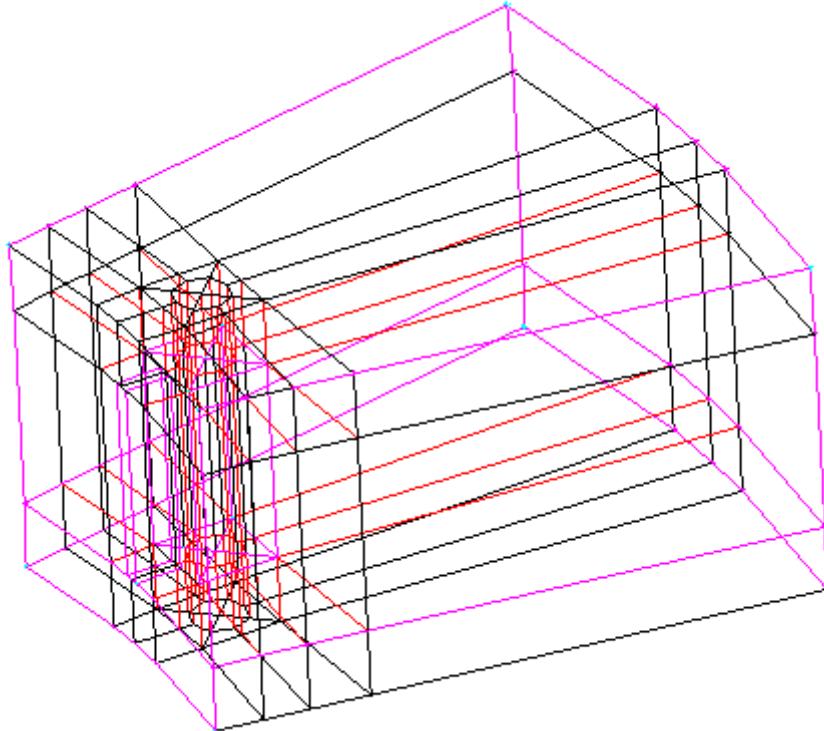


Select Blocking (from the Display Tree) > Init Output blocks. This will initialize the output topology for Multiblock mesh.

Toggle on Pre-mesh > Output blocks in the Display Tree.

Select Blocking > Edit Block > Merge blocks > Select option- Toggle on Automatic. This will merge the unnecessary blocks.

Figure.4-361
Blocking after Auto merge



q) Saving the files

Save the blocking, using File > Blocking > Save blocking.

Save the Multiblock mesh with File > Blocking > Save Multiblock Mesh and select Volume when asked to select the type of domain.

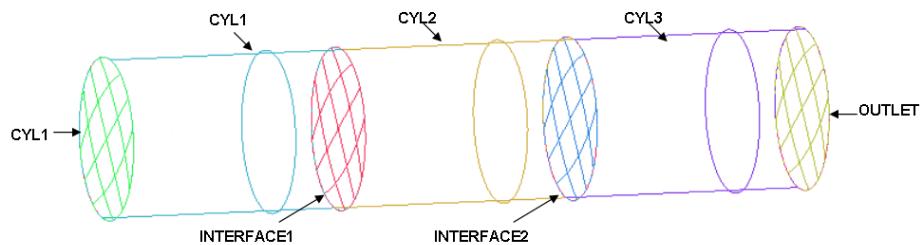
Finally File > Exit to quit ANSYS ICEM CFD

4.6.2: Hybrid tube

Overview

In this tutorial, the user will generate a hybrid mesh for the Hybrid Tube geometry shown below. 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 4-362
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

The input files for this tutorial can be found in the Ansys installation directory, under ..\v110\docu\Tutorials\CFD_Tutorial_Files > Hybrid tube project. Copy these files to your working directory and open geometry.tin.

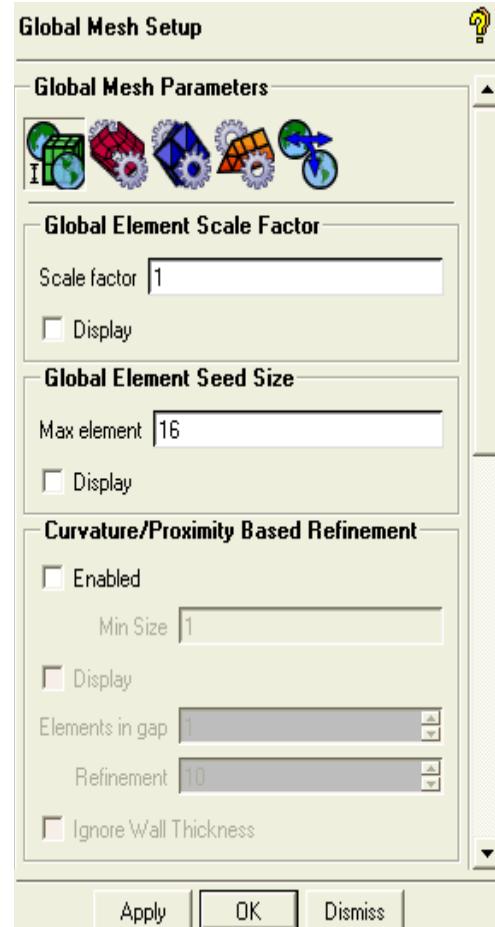
c) Generating the Hybrid Mesh

Generating the Tetra Mesh in Left Section

Select Mesh > Set Global Mesh Size  > General Parameters 

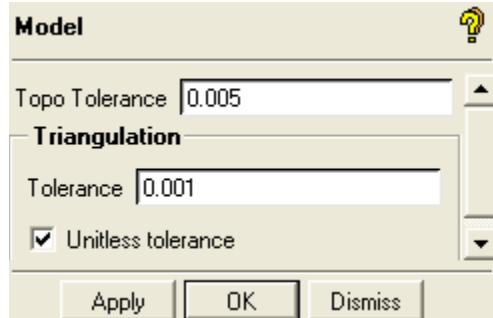
Enter 16 as Maximum size in the Global mesh size window and Scale factor as 1. Press Apply followed by Dismiss to close the window.

Figure 4-363
Global Mesh Size window



Then to set the tolerance select Settings > Model > Triangulation tolerance as 0.001 and Toggle ON the Unitless tri tolerance option. Press Apply followed by Dismiss to close the window.

Figure 4-364 Setting Model Tolerance



Select Mesh > Set Surface Mesh Size . A Surface mesh size window will appear.

Press Select Surf(s) .

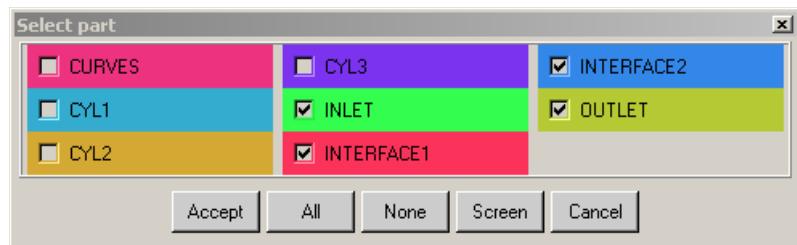
Press Select Item in Part .

Select INLET, INTERFACE1, INTERFACE2 and OUTLET. Press Accept.

Figure.4-365

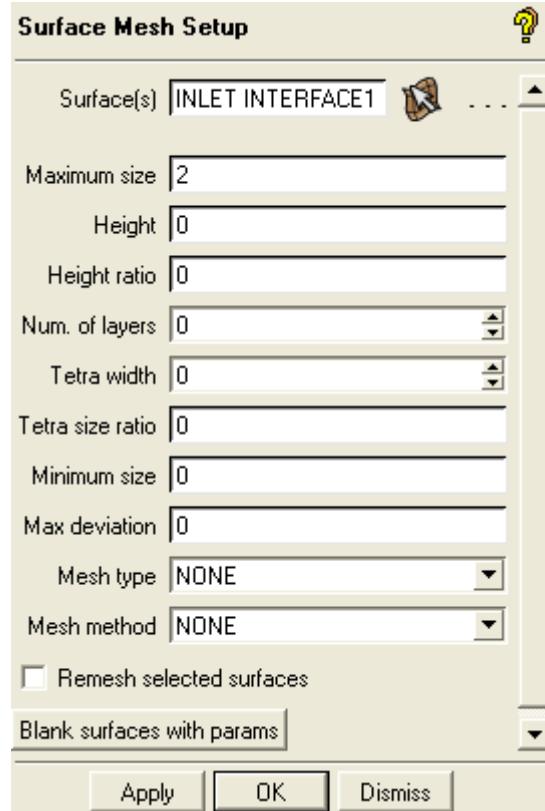
Select Part

Window



Enter Maximum size as '2' as shown.

Figure.4-366
Surface Mesh Size
window

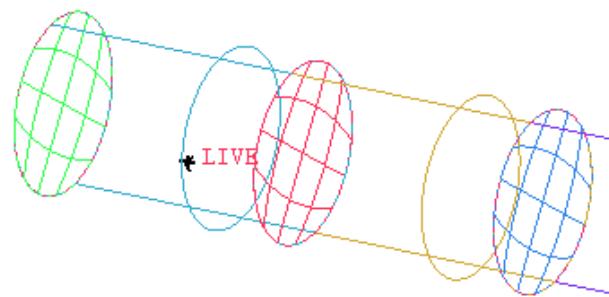


Similarly select surfaces CYL1, CYL2 and CYL3. Enter Maximum element size 4. Press Apply followed by Dismiss to close the window.

Select Mesh >Curve Mesh Set-Up . 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 parameters window and press Apply followed by Dismiss to close the window.

Create Body >Material Point . A window will appear. Select the Part 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.

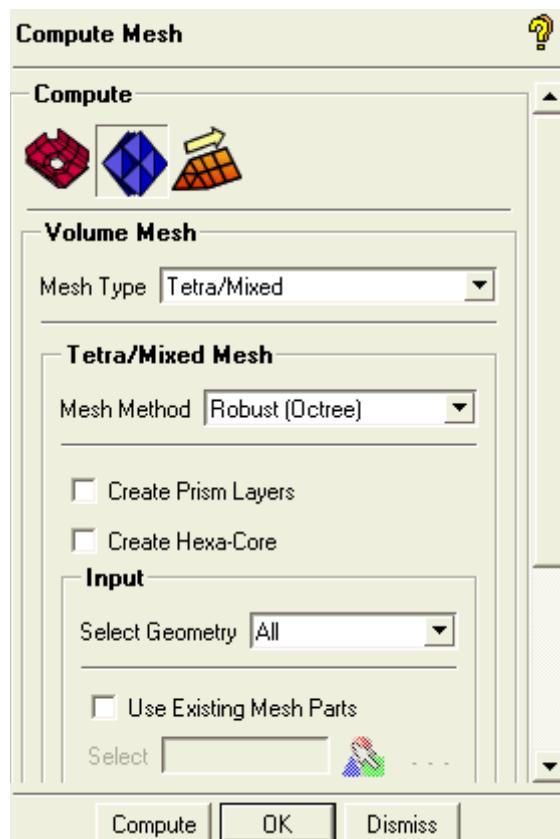
Figure.4-367
LIVE Body
Created



Select File > Save project Enter any Name.

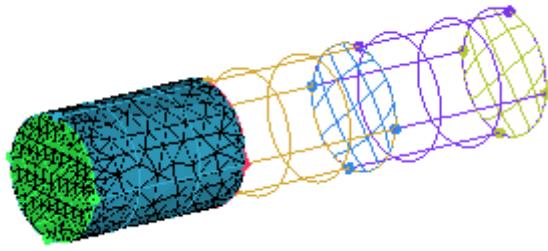
Select Compute Mesh > Volume Meshing Select Mesh Type > Tetra/Mixed, Mesh Method>Robust (Octree), input Select geometry > All as shown. Press Compute to generate the tetra mesh.

Figure.4-368
Mesh with Tetrahedral
window



Tetra mesh will be generated as shown below.

Figure.4-369
Tetra Mesh in Left Section with Solid/wire mode



Note: Even though Tetra Mesh is created we again save by another name.

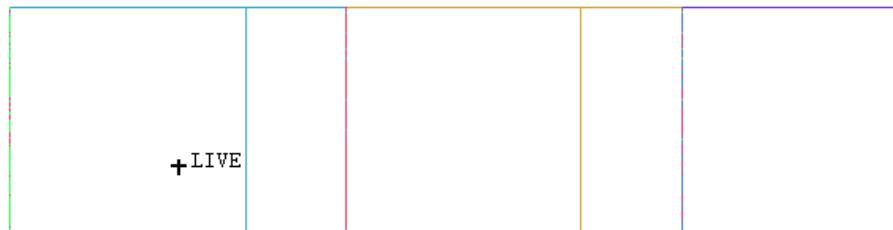
From Main menu, select File > Mesh > Save mesh as tetra_mesh1.uns followed by File > Mesh > Close Mesh.

d) Generating the Tetra Mesh in Middle Section

Select View >Front.

Select Geometry >Transform Geometry > Translate Geometry
Select the LIVE (Body) with the left mouse button. Press middle mouse button to accept.

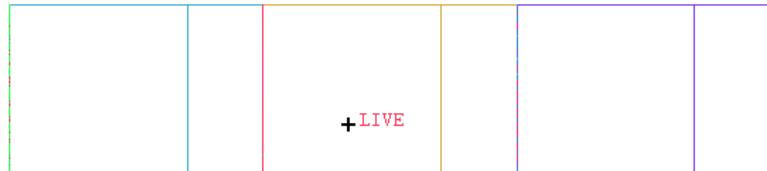
Figure 4-370
Live region selected



Select option Explicit and select X Offset as 25. Apply. Dismiss. Checks that now LIVE is located in CYL2. It should be repositioned as shown.

Figure.4-371

LIVE
region
repositioned

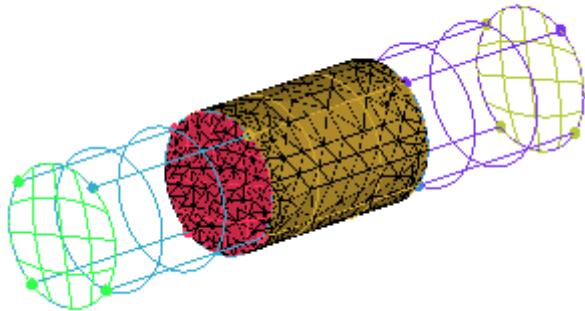


Go to Part > LIVE>Right Click and Rename it as LIVE1.
From Main menu, select File > Save project.

Select Compute Mesh > Volume Meshing .Select Mesh > Tetra/Mixed, Mesh method > Robust Octree, Input Select geometry > All. Press Apply .The Tetra mesh will be generated in the middle region as shown in the figure below.Then Dismiss the Window.

Figure.4-372

Tetra Mesh in the middle section with Solid/Wire model



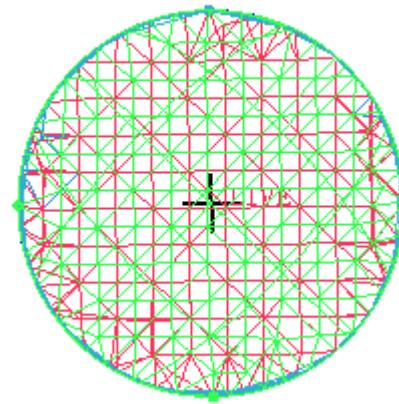
e) Merging the Tetra Mesh between Left and Middle Section

From Main menu, select File > Mesh > Save Mesh as tetra_mesh2.uns followed by File > Mesh > Open mesh A window will appear as shown. Select Merge button. A selection window will appear as shown.

Figure.4-373
Window with Merge Option



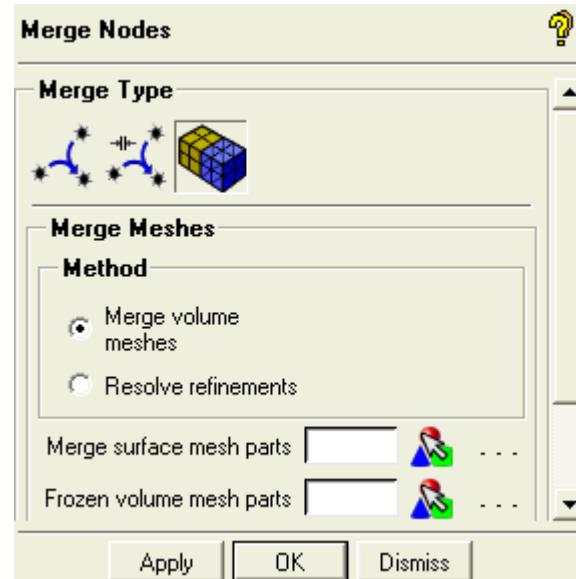
Figure.4-374
Tetra Mesh before Merging



Before merging, turn on CYL1, CYL2 and INTERFACE1 and LIVE. The surface mesh at the INTERFACE1 will look like the figure above.

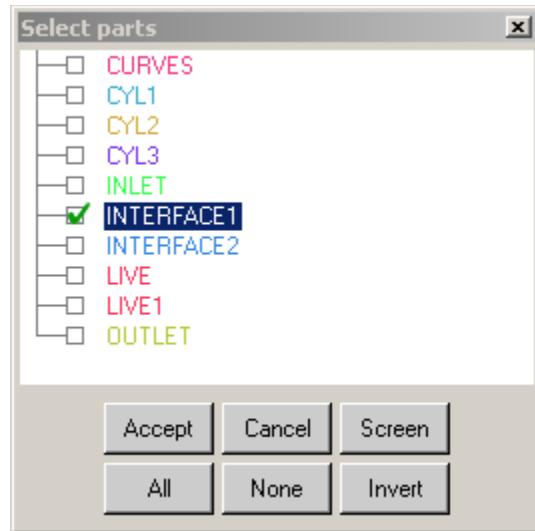
Select Edit Mesh > Merge Node  > Merge Meshes  . Select merge volume meshes and select Merge surface mesh parts.

Figure.4-375
Merge meshes Window



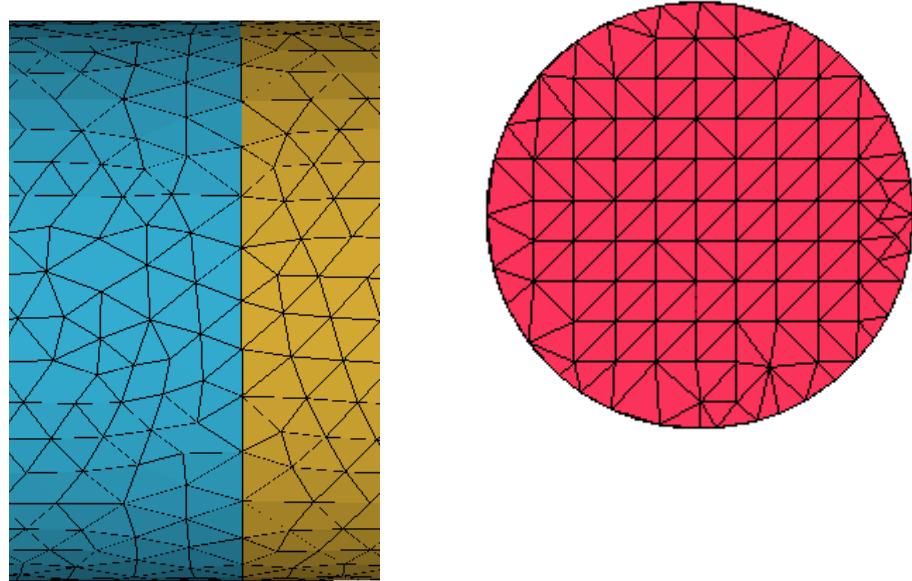
Select INTERFACE1. Press Accept and then press Apply.

Figure.4-376
Select parts to Merge
Meshes window



After merging, the surface mesh at the INTERFACE1 will look like the figure below.

Figure.4-377
Tetra Mesh after Merging



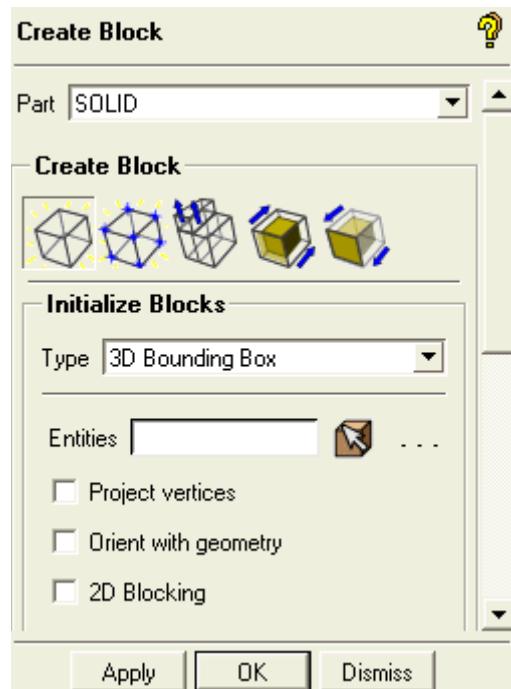
Switch OFF lines and triangles elements to see just the geometry. Switch on all the parts if they are turned OFF. From Main menu, select File > Save project.

f) Generating the Hexa Mesh in Right Section

Switch Off the Mesh in The Display Tree.

Select Blocking > Create Block > Initialize Block as shown and Press Apply. By default all the entities will be contained in the created block.

Figure.4-378
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 > Associate Edge to Curve .

Enable Project Vertices.

Select CURVES.4 and corresponding Edges 37-38, 38-42, 42-41 and 41-37 as shown below by using the left mouse button. Click the middle mouse button to accept the selection and then as shown in press 'Apply' as shown below.

Figure.4-379
Blocking Association window

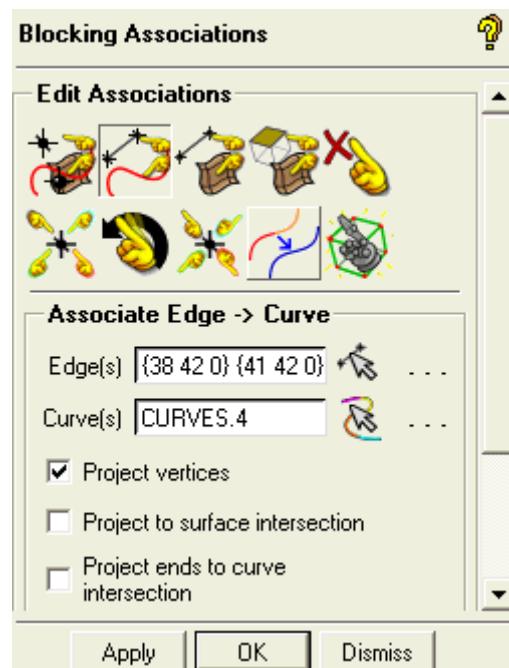
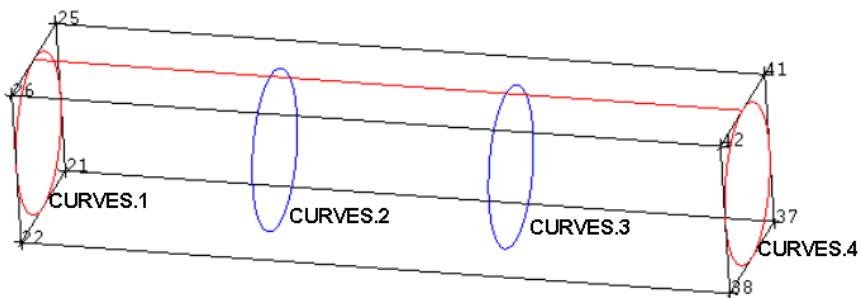


Figure.4-380
Projecting the edges on curves

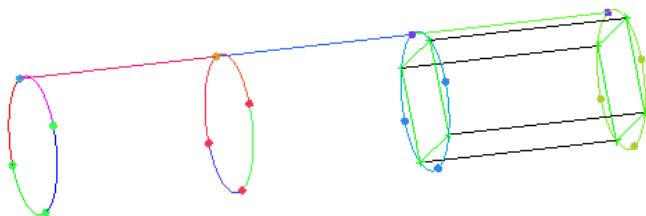


Note: Make sure that Project Vertices is enabled.

Repeat the same procedure for CURVES.3 and corresponding Edges 21-22, 22-26, 26-25 and 25-21.

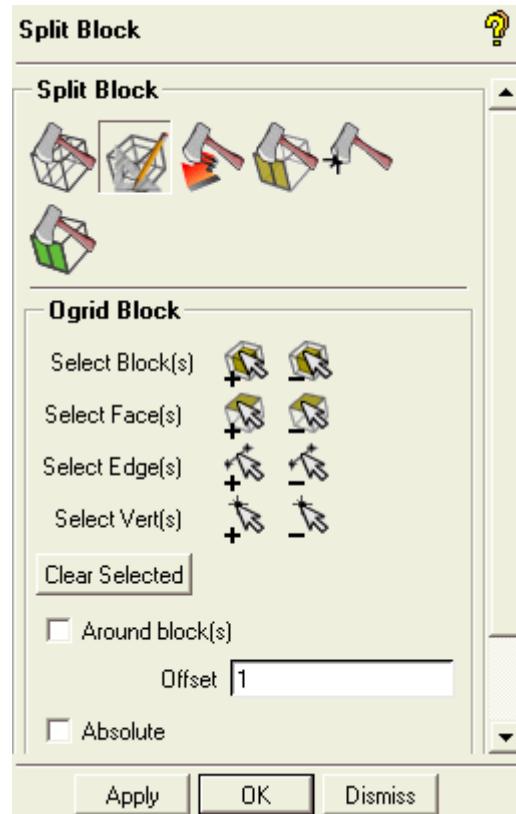
Figure.4-38

**1
Blocking
After
projecting
vertices**



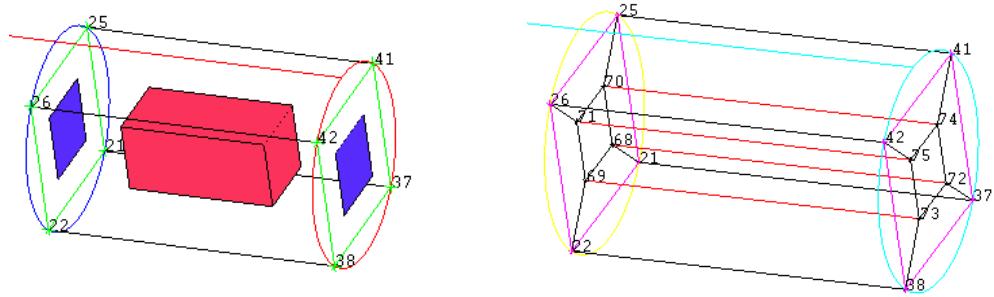
Select Blocking >Split Block > O-Grid . Press Add in front of Select Block and select existing block by using the left mouse button and then middle mouse button to accept the block.

Figure.4-382
Inner O-grid creation window



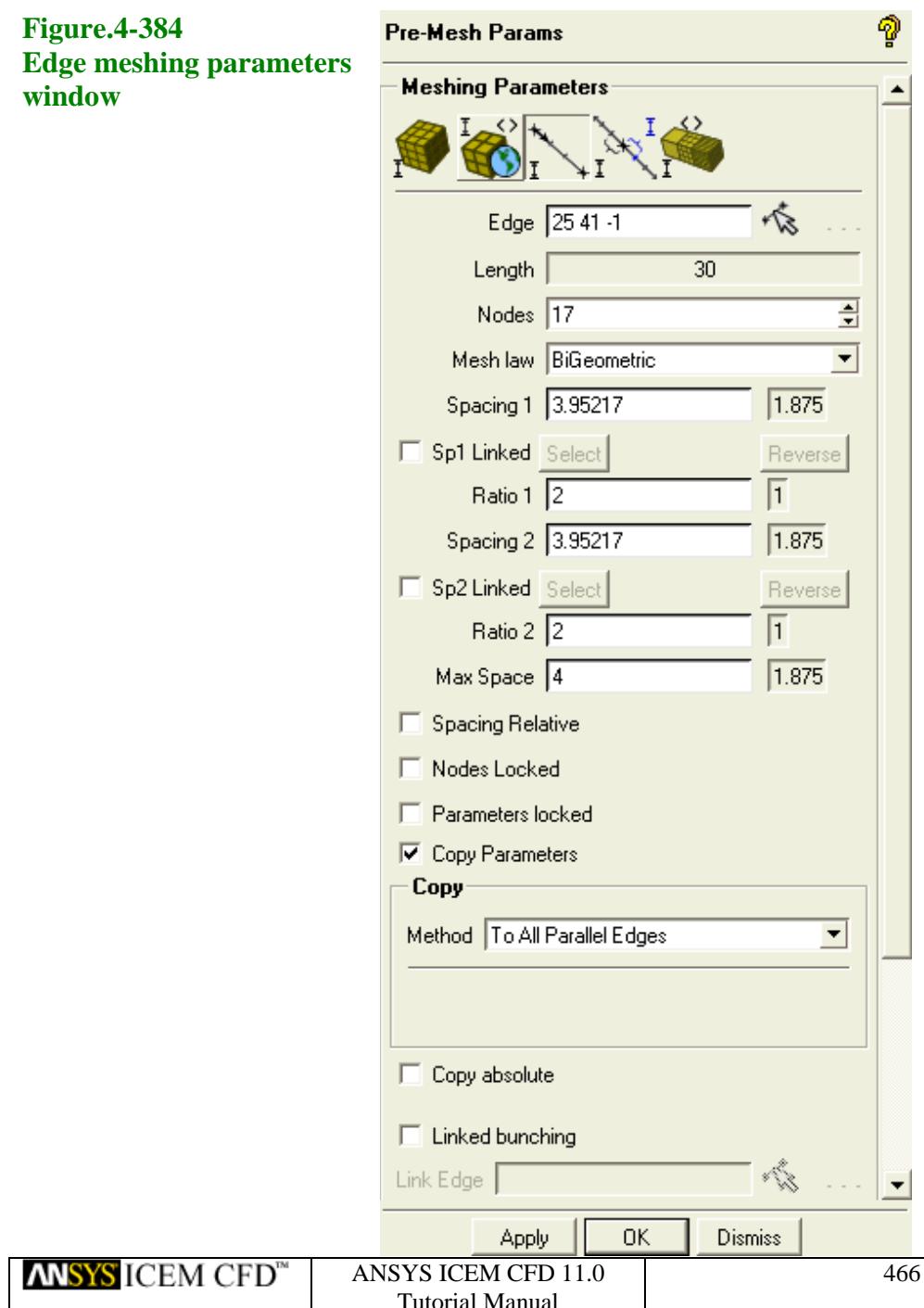
Similarly select two faces by FACES as shown, press Apply and the O-Grid shown in the right below will appear.

Figure.4-383
Before creation of O-grid (Left) and 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 and press Apply.

Figure.4-384
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.

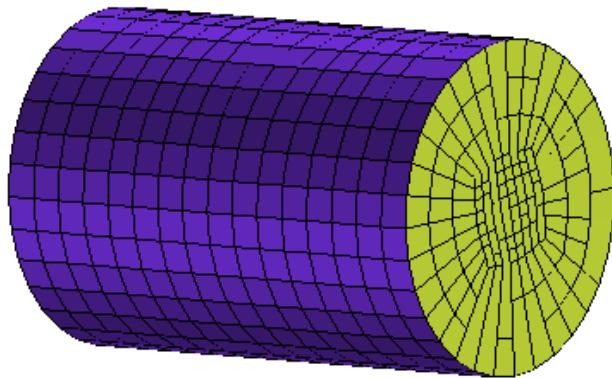
Repeat the procedure for Edge 37-41, give Nodes as 9

For 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 the figure below. The user might have to switch off the Vertices, Edges and Curves to reduce clutter on the screen.

**Figure.4-385
Hexa Mesh
in Cylinder3**

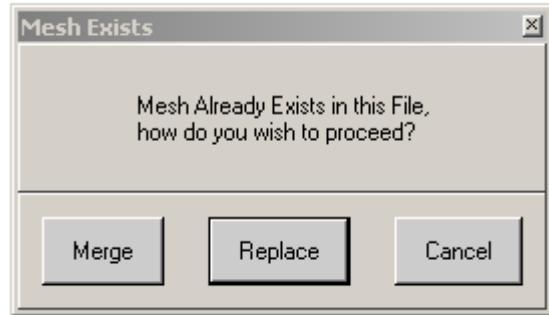


File > Save blocking will save the Blocking File

File > Mesh > Load from Blocking.

In the Mesh Exist window press Merge.

Figure.4-386
Window Asking for
Merging New Mesh

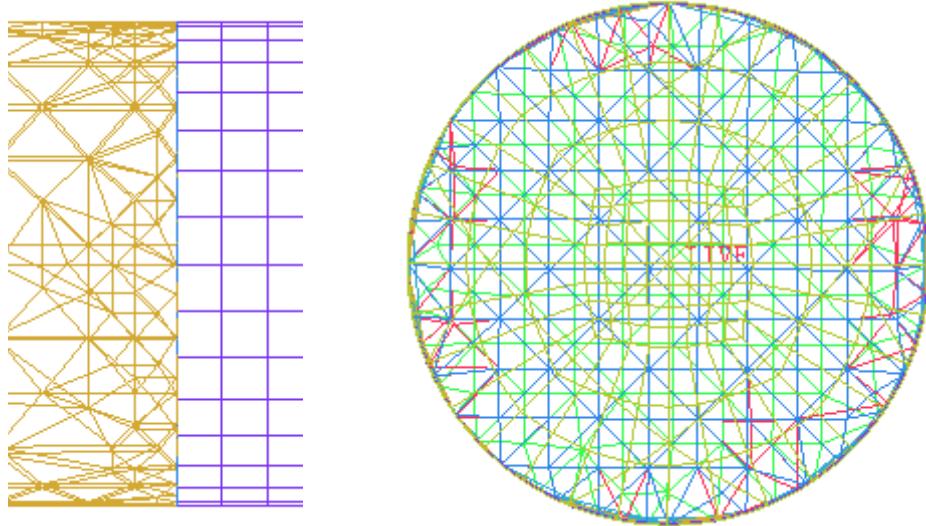


Switch Off Blocking in the Display Tree.

g) Merging the Resultant Mesh with Hexa Mesh at Interface2

Before merging the surface mesh at the INTERFACE2 will look as shown below. The user might have to switch off all the families except INTERFACE2, CYL2 and CYL3.

Figure.4-387
Hexa Mesh before Merging



Select Edit mesh > Merge Node  >Merge meshes.  A selection window will appear.
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 below. And pyramid at INTERFACE2 will be as shown below as well.
You can see the pyramids by switching on pyramids with LIVE 1 family switched ON.

Figure.4-388
Hexa Mesh after Merging

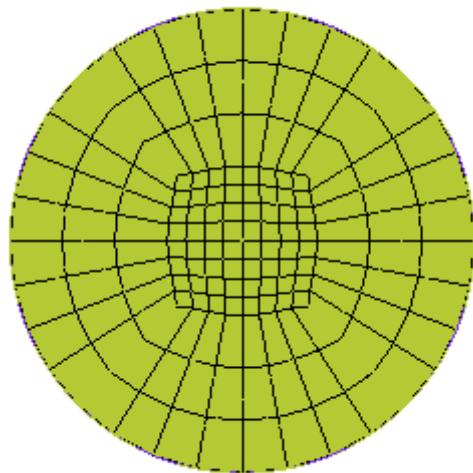
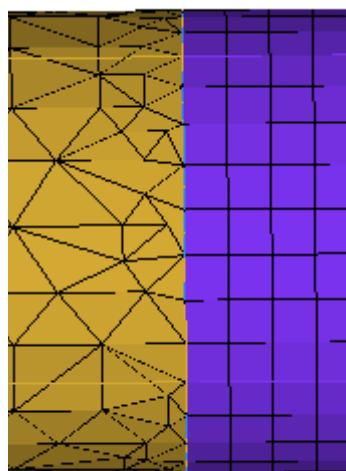
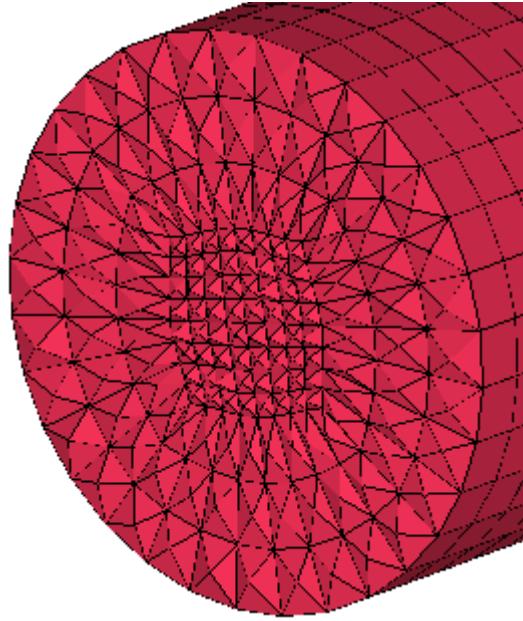


Figure.4-389
Pyramid at INTERFACE2



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 below.

Modify the display of the histogram to have a Height of 24 elements.

Click on Replot to replot the Histogram. To improve the quality of hybrid mesh, change the Number of smoothing iterations to 12. Assign Up to quality value to 0.4. Press Apply.

Figure.4-390
Quality before Smoothing

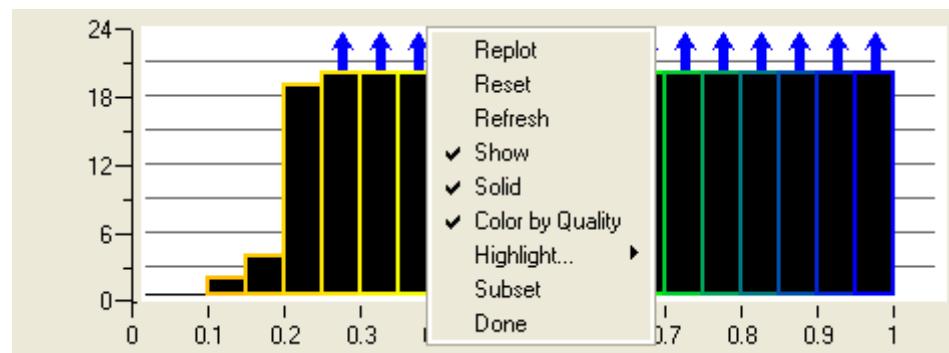


Figure.4-391
Smooth globally
window

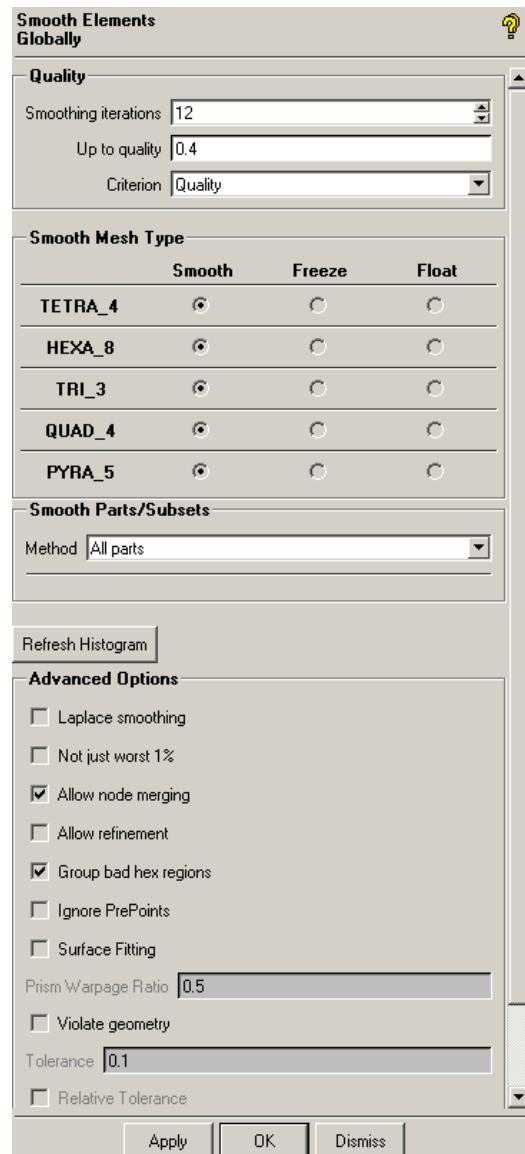
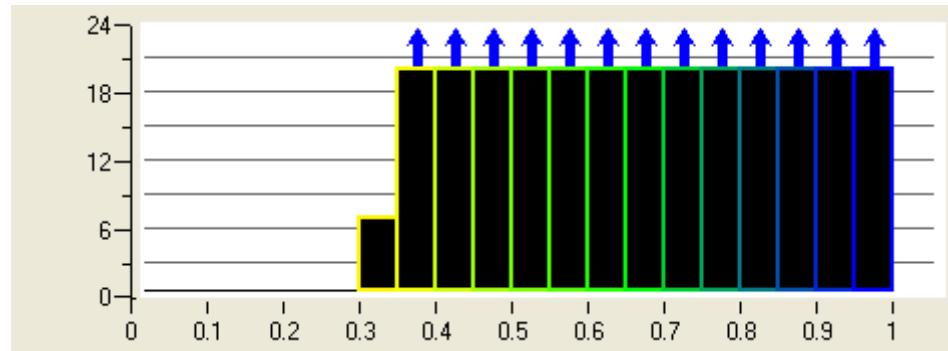


Figure.4-392
Quality after Smoothing



The quality of the hybrid mesh after smoothing is shown. 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 > Close Project.

4.6.3: 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 the figure below. A cylindrical water channel, that extends a few body lengths upstream and downstream, contains the entire geometry.

Figure.4-393

Surface parts and surface mesh of the regions composing the submarine

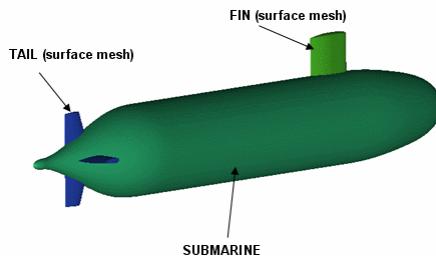
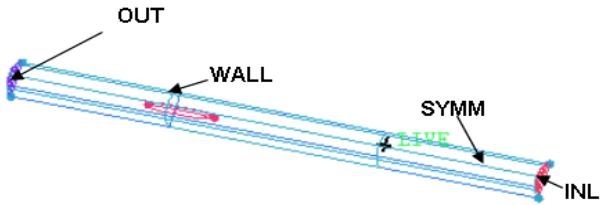


Figure.4-394

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

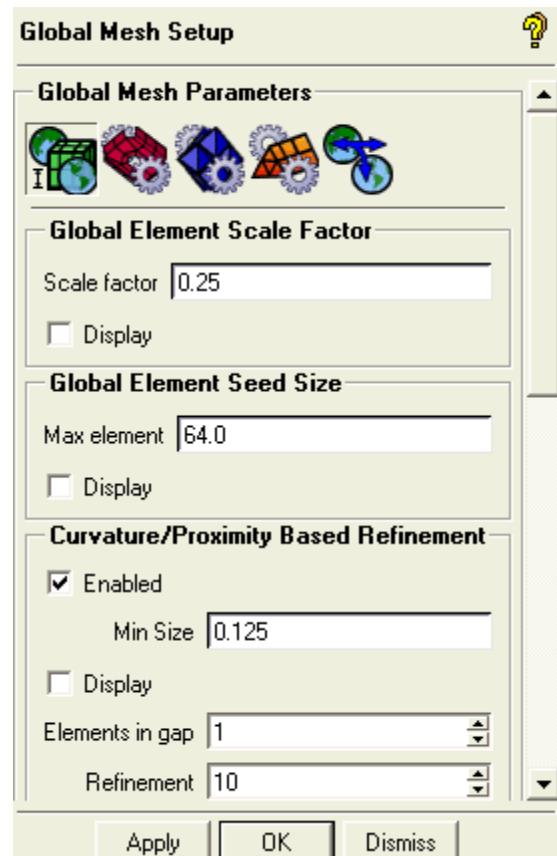
b) Starting the Projects

The input files for this tutorial can be found in the Ansys installation directory, under ..\v110\docu\Tutorials\CFD_Tutorial_Files > submarine project. Copy these files to your working directory and open the geometry file (geometry.tin) and domain surface_mesh.uns.

c) Setting Global mesh size

Choose Mesh > Global Mesh set-Up > Set Global Mesh Size .

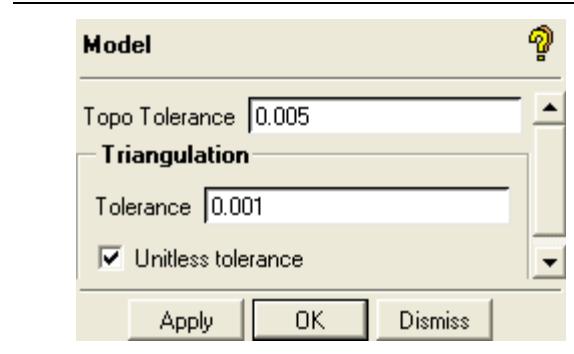
Figure 4-395
Global Mesh
Parameters window



In the Global mesh size window, enter a scale factor of 0.25, a Maximum size of 64. Enable Natural size and enter Min Size as 0.125, Elements in gap as 1 and Refinement of 10. Leave the other parameters at their default settings.

Go to Settings > Model > Triangulation > Tolerance 0.001 and toggle ON Unitless tolerance. Press Apply with default Topo Tolerance value as shown in followed by Dismiss.

Figure 4-396-Setting Model tolerance

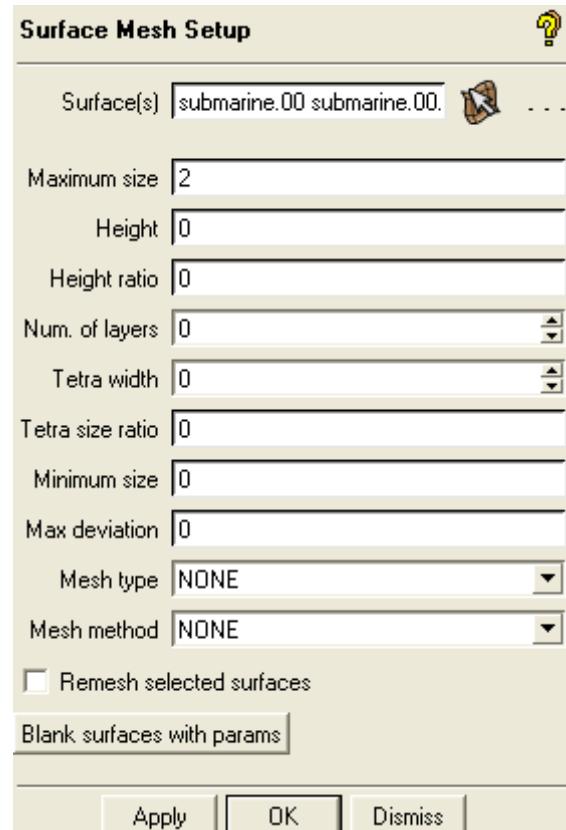


d) Setting surface mesh size

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 pressing “a” on the key board, and enter the Maximum element Size of 8.

Next, repeat the step and from the selection filter, click on Select items in a part 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 and press Apply.

Figure.4-397
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 . Enter Size as 1 (a scale factor multiplier), ratio as 1.2 and width as 1.5. Choose option

Points, and Press  to pick back point BODY.161 of the submarine on the axis of symmetry as the first point and POINTS.01 as the second point. Refer to the figure below. Press Apply.

Figure.4-398
Create density box

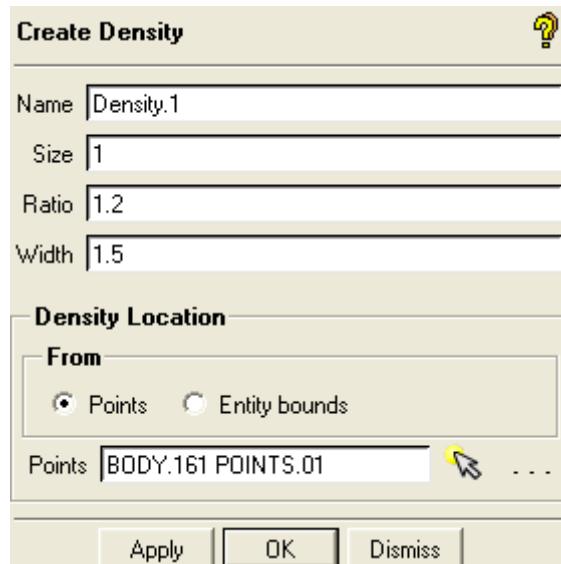
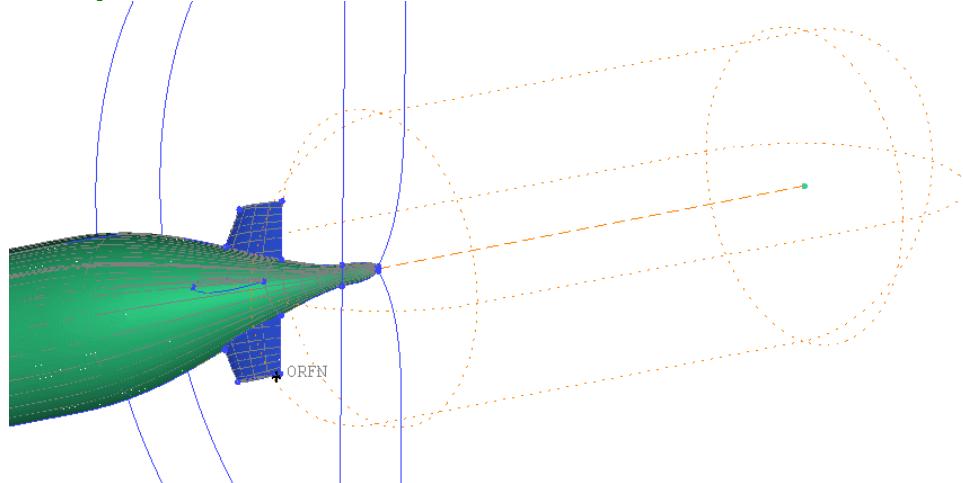


Figure 4-399 Points to select for density creation



The figure below shows the density box after creation.

Figure.4-400
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 Select Compute Mesh > Volume Meshing . Select Mesh > Tetra/Mixed, Mesh method > Robust Octree, Input > Select geometry > All. Turn On the option Use Existing Mesh Parts, a new window select subpart will open. Select the parts FIN and TAIL for existing surface mesh as shown. Press Apply to generate the tetrahedral mesh.

Figure.4-401
Mesh with tetrahedral
window

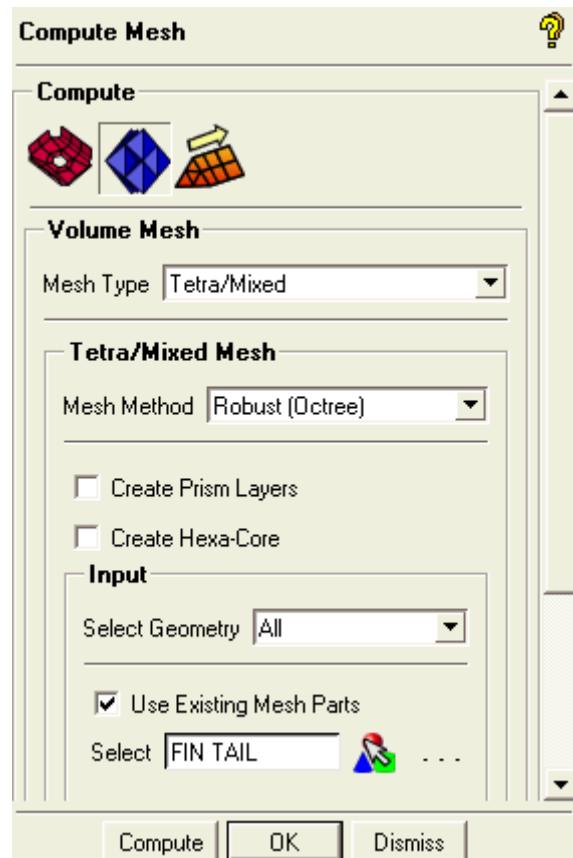
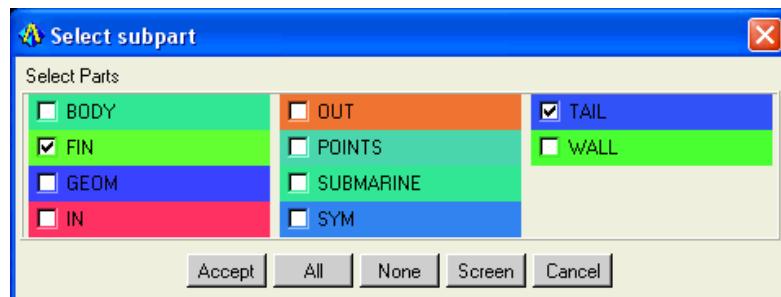


Figure.4-402
Select
subpart for
existing
surface
mesh



When the tetra process has finished, the complete tetra mesh should look like the figure below.

Figure.4-403
Complete Tetra Mesh on symmetry plane

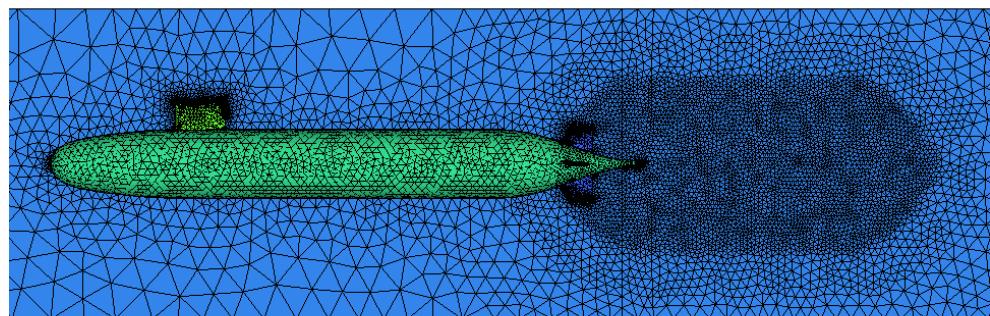
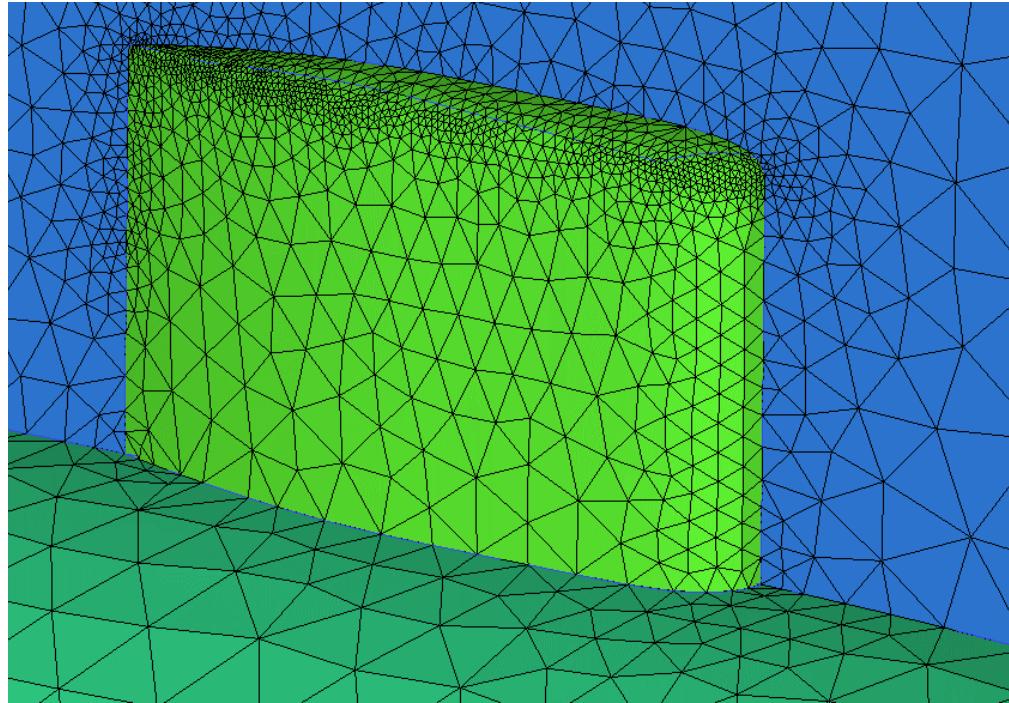


Figure.4-404
Mesh in the Fin area



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.

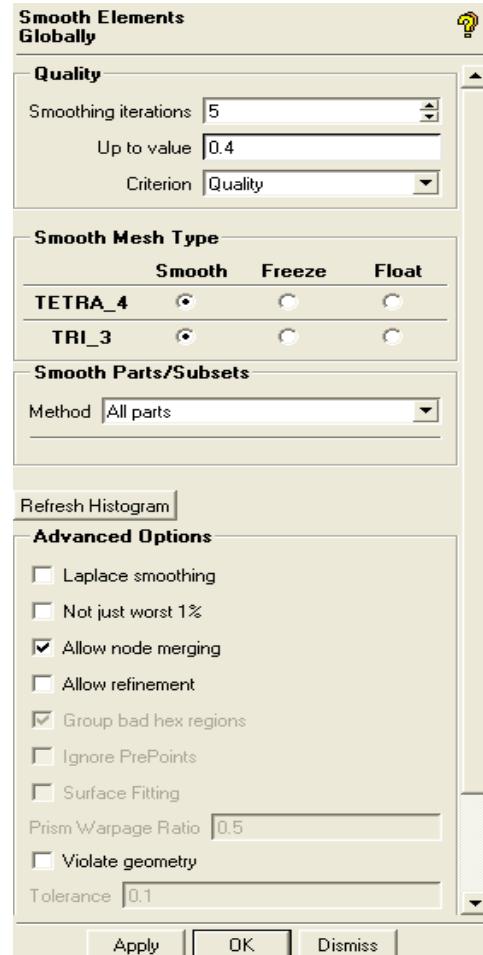
i) Smoothing

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 the figure below, select Apply. The smoother histogram is also displayed below.

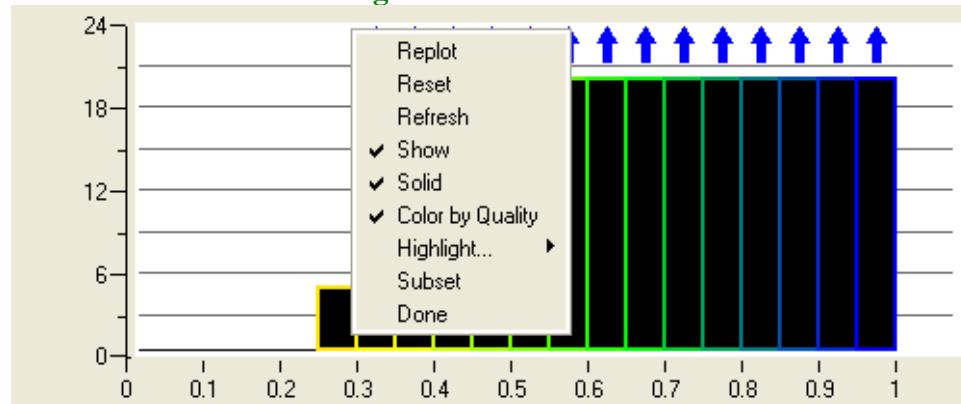
Figure.4-405
Smooth Mesh Globally window



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.4-406

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	485
------------------------	--	-----

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

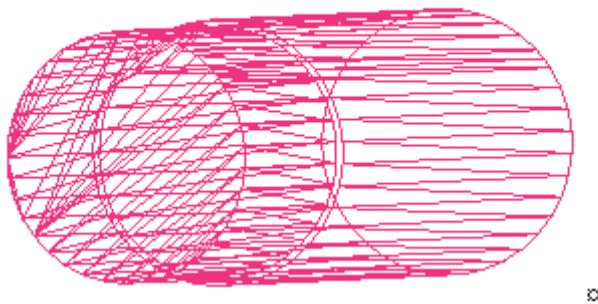
Save the mesh by selecting File > Save project. If a question box pops up to delete disconnected vertices, respond by saying Yes. Then close the project by selecting File > Close Project.

4.6.4: 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 4-407 After Opening the file



a) Summary of steps

- Starting the project
- Repairing the geometry
- Saving the project

b) Starting the Project

The input files for this tutorial can be found in the Ansys installation directory, under `..v110/docu/Tutorials/CFD_Tutorial_Files/STL_Repair`. Copy these files to your working directory and load the tetin file, `geometry.tin`.

c) **Repair Geometry**

Right click mouse button and select Geometry > Surfaces in the Display tree and select Show Full to see the full triangulation of the surfaces.

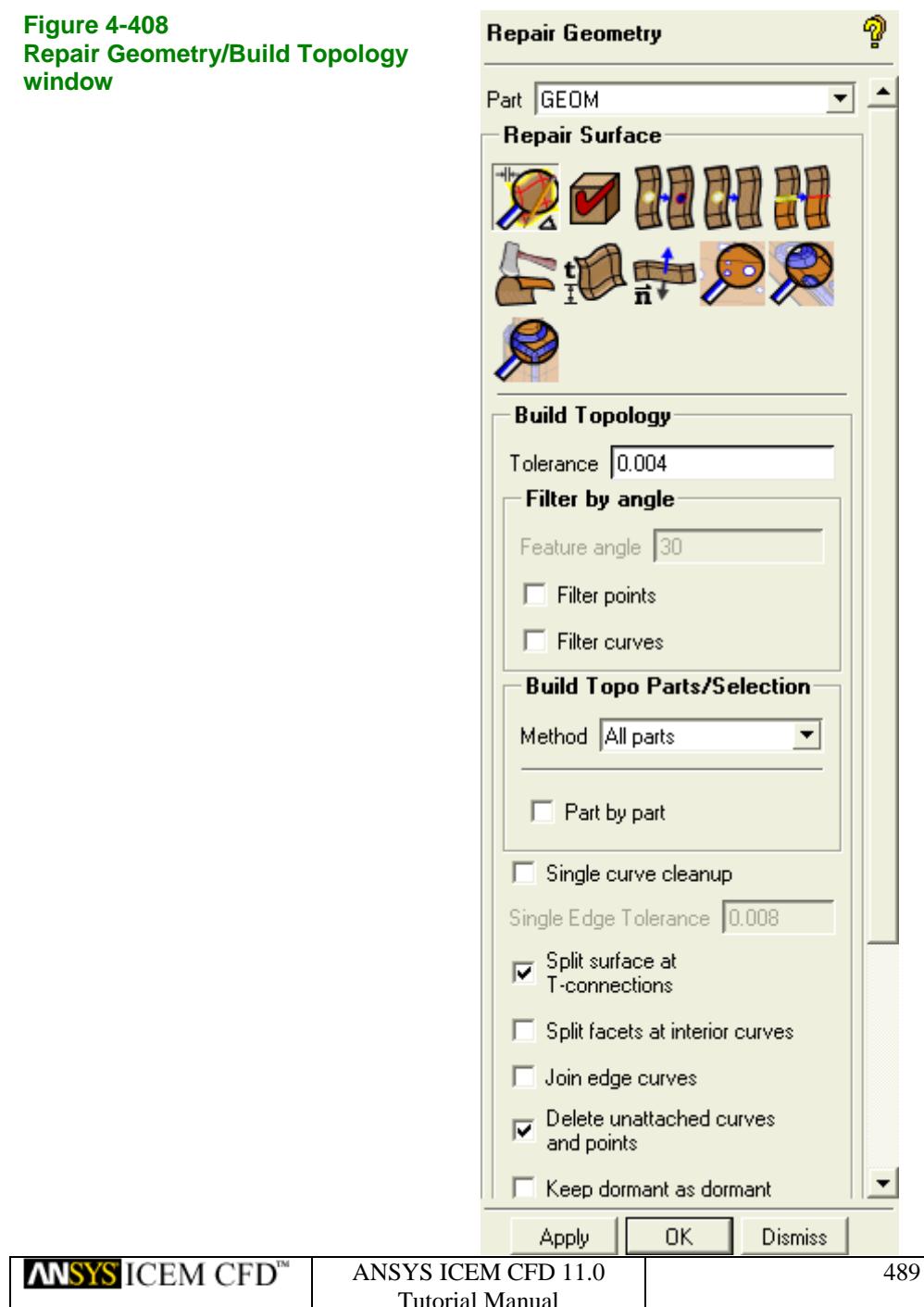
Now run Build Topology to find any possible problems with the geometry.

Select Geometry > Repair Geometry  > Build Diagnostic Topology



. This will open up the window shown below.

Figure 4-408
Repair Geometry/Build Topology
window

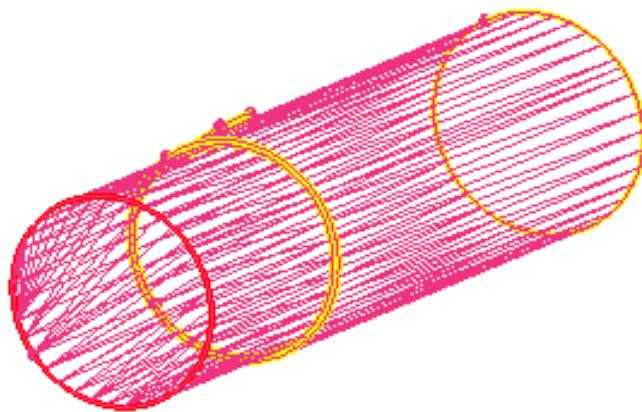


The more important settings are:

Tolerance – maximum gap distance between surface edges is 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.

Settings>Geometry Options>Inherit part name>Select Inherit option. Use all default settings, including Tolerance, and Apply. Note the curves as shown below.

Figure 4-409
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. This usually needs to be fixed.

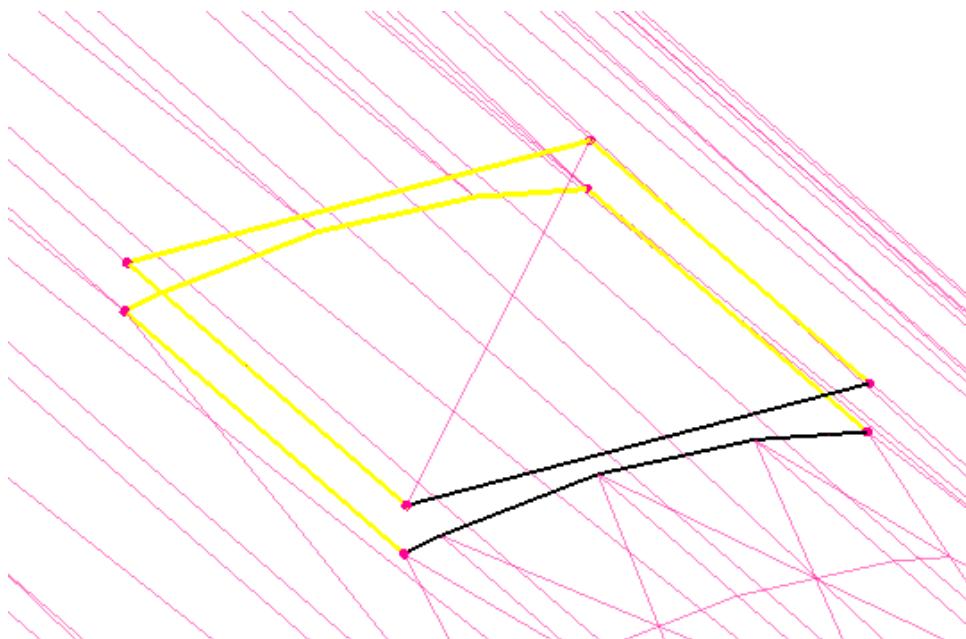
Blue – Curve is shared by three or more surfaces. Usually indicates a t-junction or a sliver surface that's thinner than the tolerance. This is most likely okay but in some cases may cause potential problems.

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. These curves can also be removed manually.

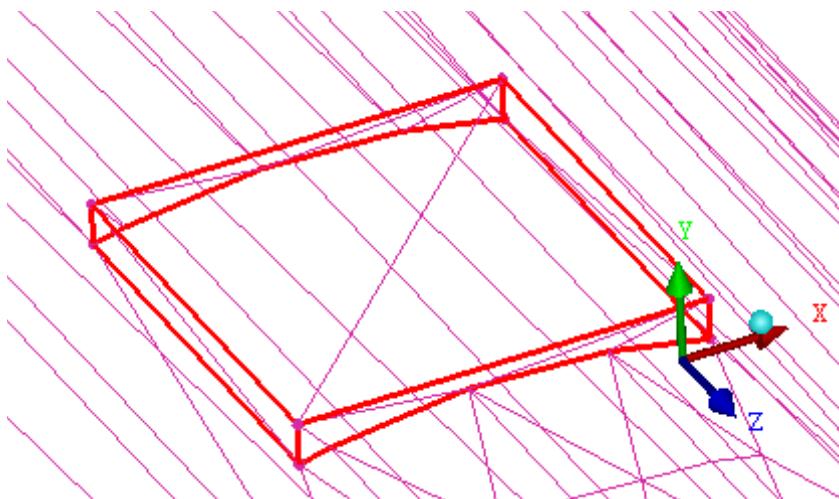
First close the hole for the portion that juts out as shown below.

**Figure
4-410
Square
portion
before
repair**



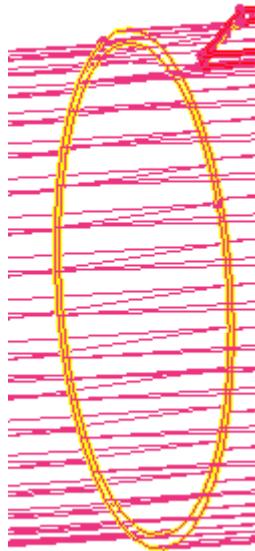
Select Repair Geometry > Close Holes . Select the curves, one pair at a time and press the middle mouse button. Repeat for all four pairs. Triangles will be created to fill in the gaps and the curves will be automatically updated to red as seen here.

**Figure
4-411
Square
portion
after
repair**



The user will now focus on the two concentric circles in the center (see figure below). Perhaps this feature is small enough to ignore, so rather than fill in the gap, we'll match or stitch the edges.

Figure 4-412
Circular portion
before repair



Select Repair Geometry > Stitch/Match Edges . Select Method > User Select. Enter a Max. gap distance =0.1. Toggle ON Single curves only option. Select the two concentric curves and press the middle mouse button .Then Apply. Note that the edges of the second curve will be moved to match the edges of the first selected curve.
A single Green unattached curve will be generated. So in order to delete it
Select Delete Curve > Toggle On option –Delete Unattached-Apply. The unattached curve gets deleted. So the geometry will appear as shown below.

Figure 4-413
Circular portion after repair

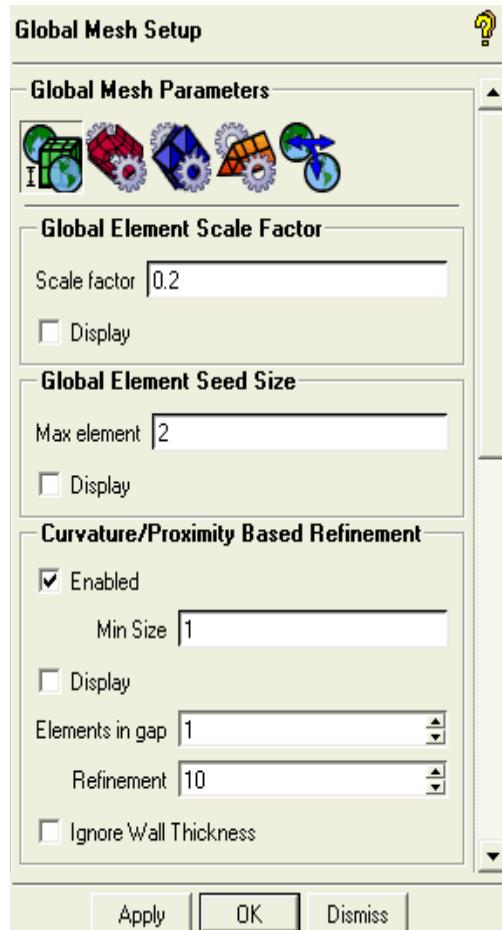


The large hole at the end of the pipe will be fixed on the mesh level to follow.

d) Assigning Mesh Sizes.

Select Mesh > Set Global Mesh Size > General Parameters . This will open up the window as shown below.

**Figure 4-414
Global Mesh Sizes
window**

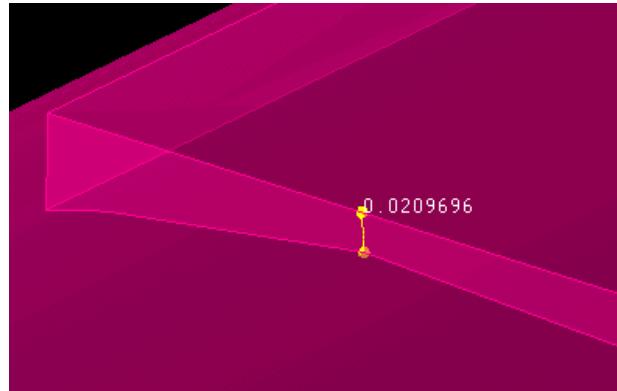


Set Scale factor to 0.2, Global Element Seed Size (Max element) to 2. All surfaces and curves will take on the Global Element Seed Size (Max element) of 2×0.2 (scale factor) = 0.4.
Zoom in to the square portion that sticks out, as shown in the figure below.

Select Measure distance  by using the upper left hand Utility Menu and then select two locations along the lower and upper curves of the

square portion that sticks out. Note the prescribed elements size is too large to capture this feature.

Figure 4-415
Tetra sizes on surfaces



Go back to Mesh > Set Global Mesh Size > General Parameters . Turn on Curvature/Proximity Based Refinement (check Enabled) as shown in and change the Min size Limit to 0.1. This value is multiplied by the Scale factor whose product is the global minimum size. Thus $0.1 \times 0.2 = 0.02$ is the lower limit of subdivision. With Natural Size enabled, the tetra mesher will automatically subdivide to smaller elements in this area.

Then Settings > Model > Triangulation, and set the Triangulation tolerance as 0.001. Toggle ON Unitless Tolerance Option.

e) Defining Material point

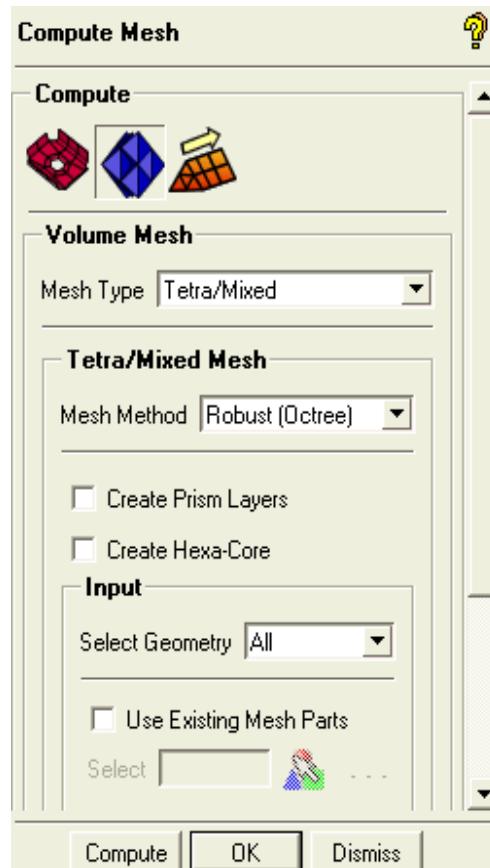
Select Geometry > Create body > Material point >Centroid of 2 Points Enter the Part name as LIVE. Select again or Select

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

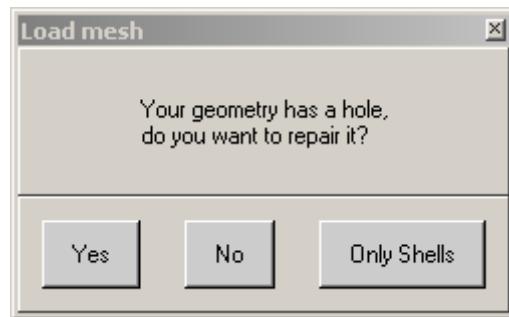
Select Compute Mesh  > Volume Meshing  . Select Mesh Type > Tetra/Mixed, Mesh Method >Robust Octree, Input-Select geometry > All.

Figure 4-416
Mesh with
Tetrahedral window



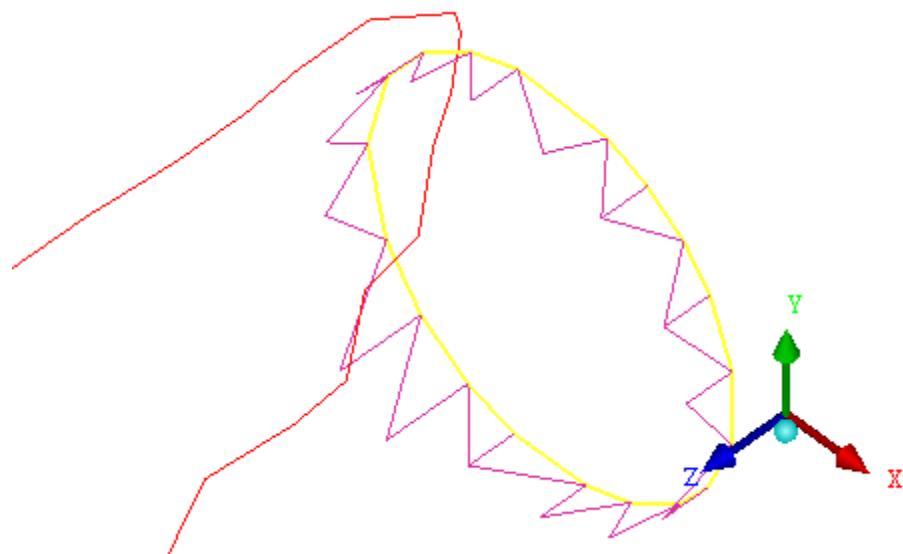
Due to the open end, a window will warn you of leakage (hole).

Figure 4-417
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.

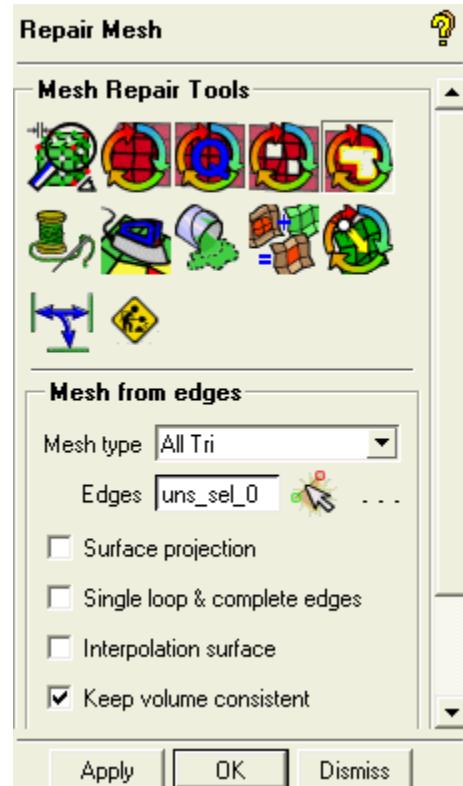
Figure
4-418
Leakage
in
display



Selecting Yes to repair will also bring up the mesh repair window. 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 4-419
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

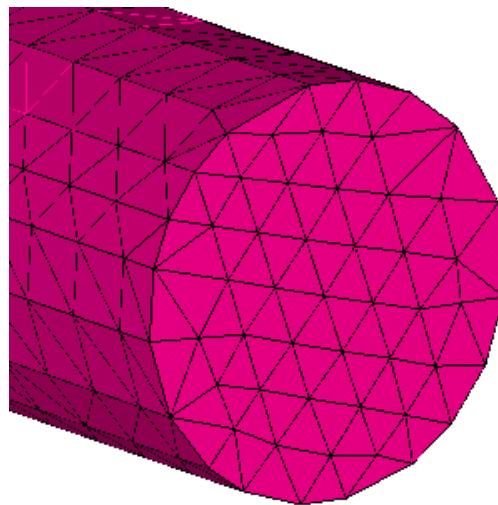
edges will eventually disappear indicating the closing of the hole was successful.

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).

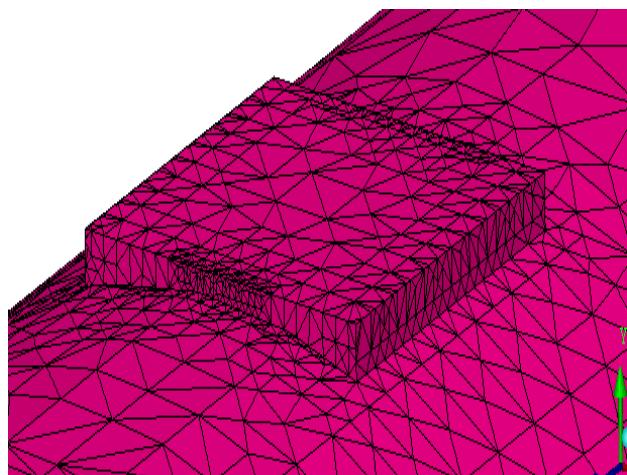
Turn on Mesh > Shells. Right mouse select Shells and select Solid & Wire. Turn On all the Parts if they are OFF. View the corrected surface mesh as shown here.

Figure 4-420
Mesh in circular
region after repair



Also note the refined mesh in the square portion that jutted out as a result of using Natural Size.

Figure 4-421
Final mesh detail



g) Final Steps

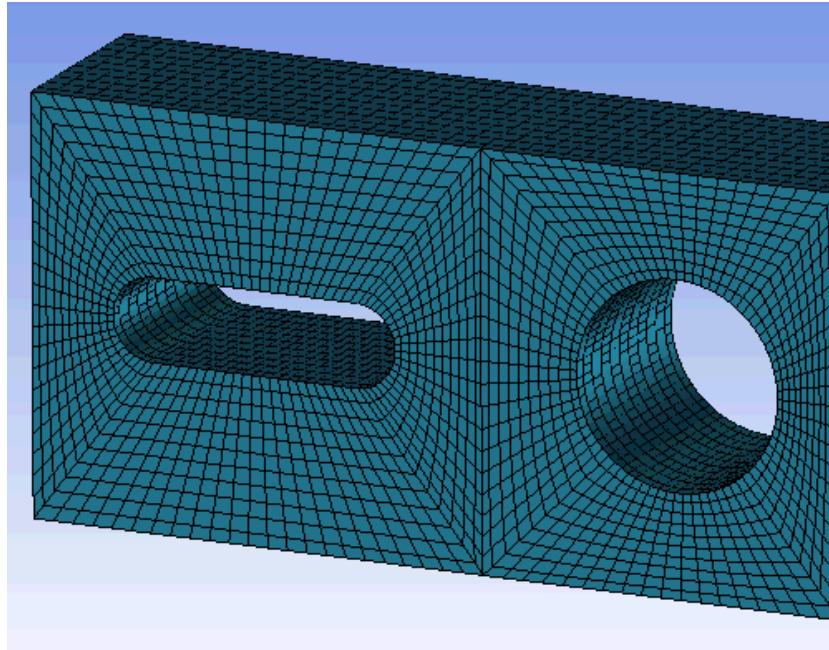
Smooth the mesh using Select Edit Mesh > Smooth Mesh Globally  Note the current bad quality in the Histogram Window. Use the defaults in the Smooth Elements Globally panel and Apply. Note the improvement in quality in the histogram.

Select File > Save project and Exit

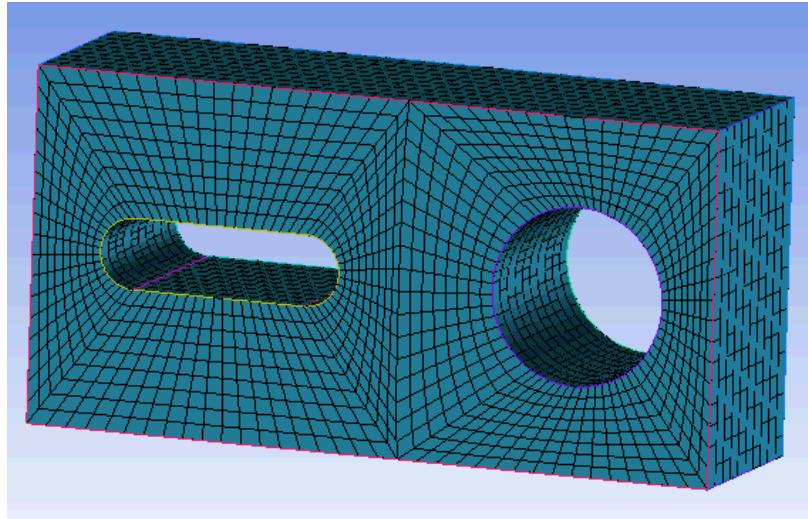
4.6.5: Workbench Integration

This tutorial will demonstrate parametric changes in the blocking with the geometry.

**Figure
4-422
Blocking
geometry**



**Figure
4-423
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

Start Ansys workbench integration. Select the **Geometry module**. This will open a design modeler (DM) graphics user interface (GUI) as shown below. Press **OK** in the desired length unit window. This will keep units to the default SI unit system.

Figure4-424
Workbench
main
window

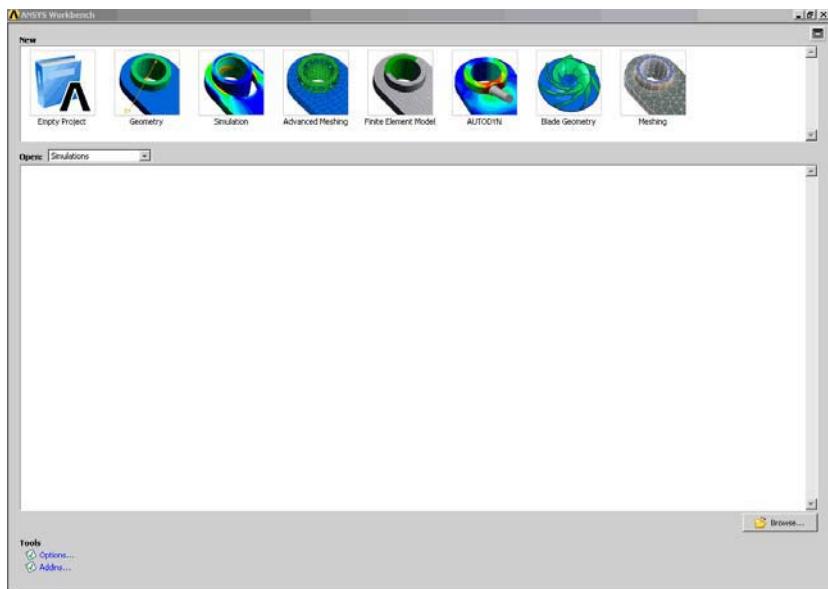
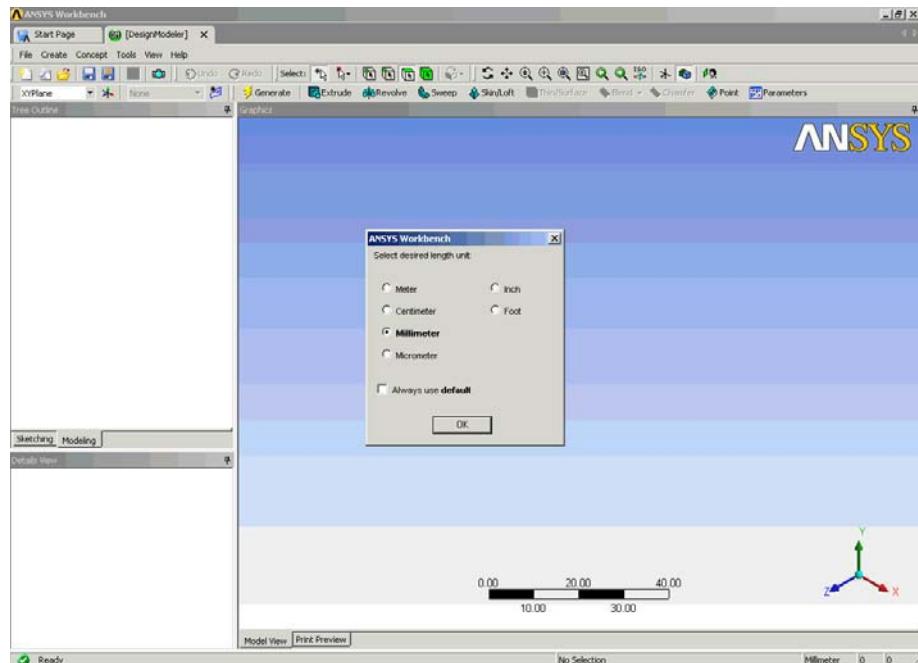


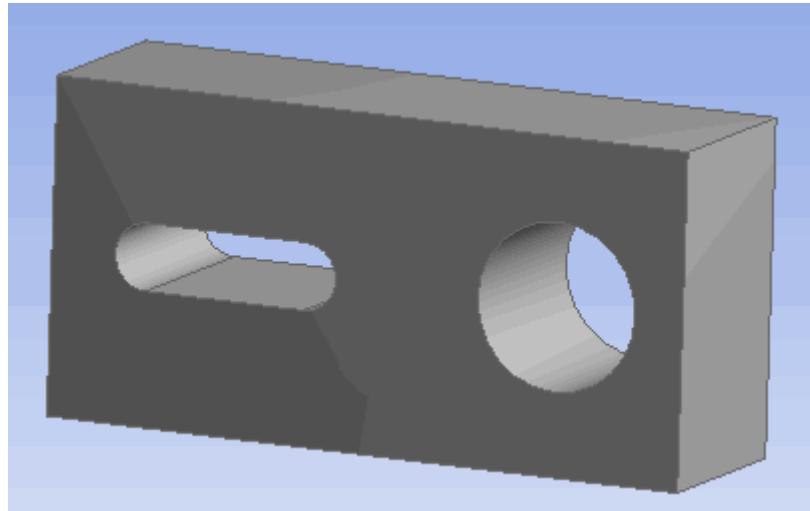
Figure
4-425
Design
modeler
interface



c) Loading Geometry in DM

The input files for this tutorial can be found in the Ansys installation directory, under ..\v110\docu\Tutorials CFD_Tutorial_Files\WB_Int. Copy the files to your own working directory. For loading geometry in the DM, go to **File > Open**. Select the parametric.agdb file. This will show the geometry in the GUI as shown here.

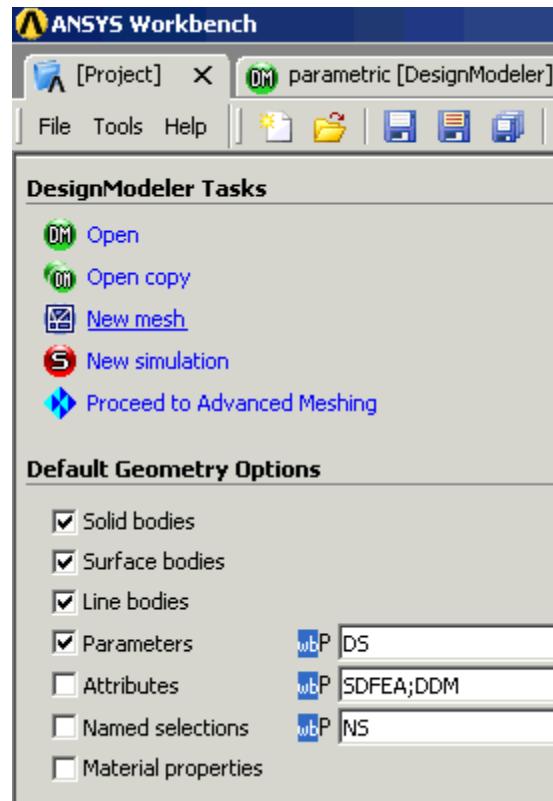
Figure 4-426
Loaded
geometry in
the
workbench
environment



d) Proceeding to the Advanced Meshing

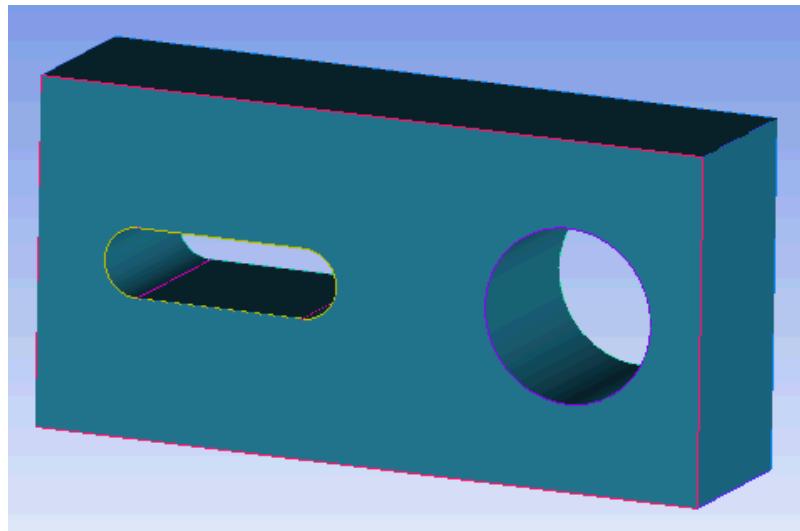
For creating the blocking, user has to go to the advanced meshing tab. Go to the Project window, click on the link **Proceed to Advanced meshing**.

Figure 4-427
**Proceeding to advance
meshing**



Clicking on the link **Proceed to Advanced Meshing** will invoke the Advanced Meshing GUI, Select File > Geometry > Update geometry > Replace geometry..., it will open the geometry in Advanced Meshing as shown here.

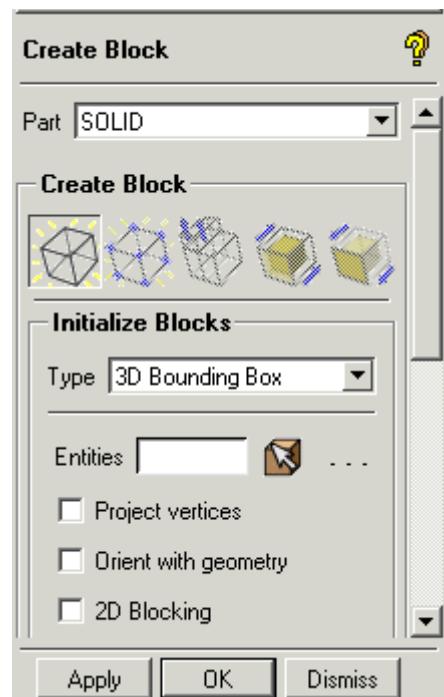
**Figure
4-428
Geometry
in the
Advance
meshing**



e) Blocking

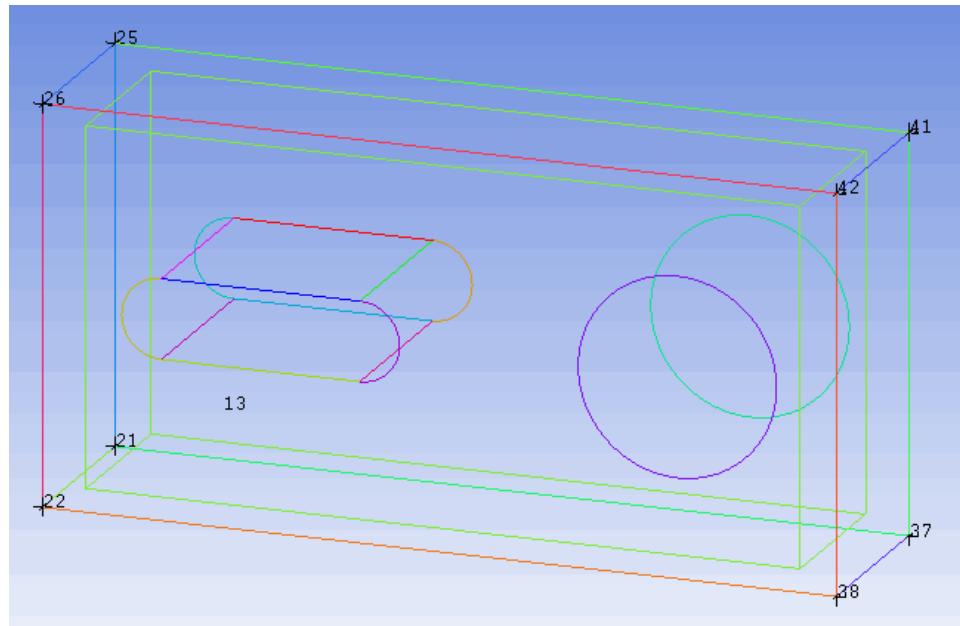
Select Blocking > Create Block , it will open the window. Select Initialize block (default) and 3D Bounding Box (default) Type. Select all entities and press Apply to create blocking.

Figure 4-429
Create block window



For vertices number, turn ON Blocking > Vertices > Number from Display Tree. Also turn on Blocks from the Display Tree. After creating the block, geometry will look like as shown here.

**Figure
4-430
Geometry
after
creating
the block**

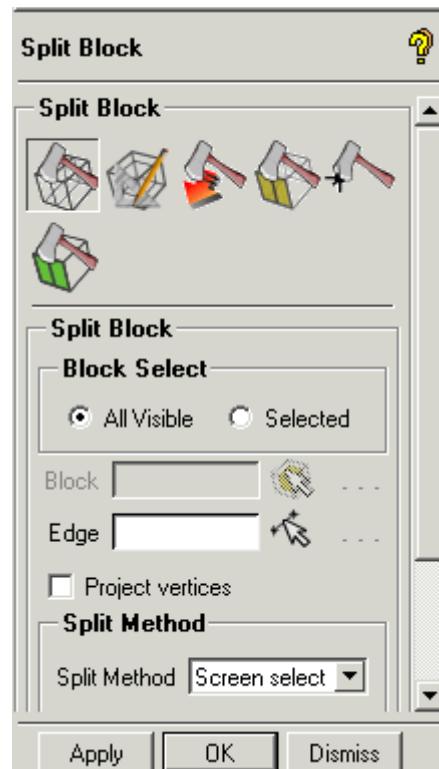


f) Split block

Now user can split the block in I, J and K direction in order to capture the shape of the geometry.

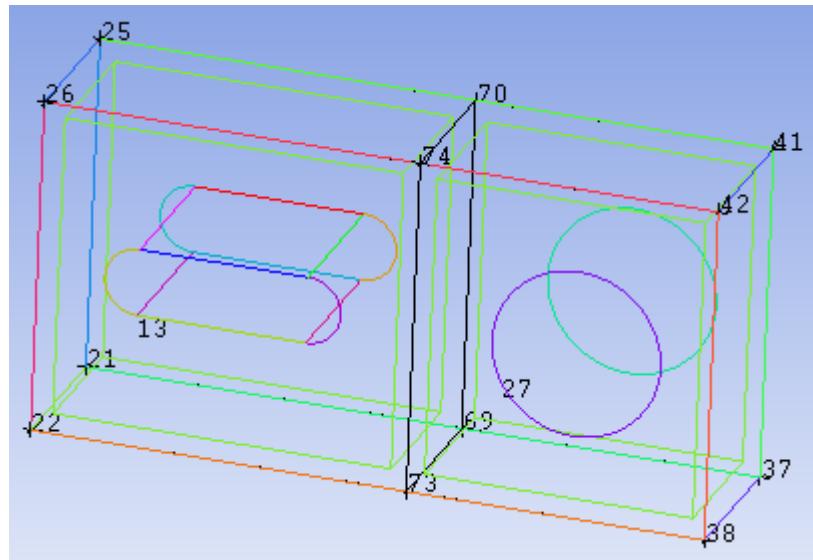
Select Blocking > Split Block , it will open the window as shown here.

Figure 4-431
Split block window



Select Split method as Screen select and select edge passing through the vertices 26-42 and make a split at the center of the edge.
After splitting the edge, the geometry will look like the figure below.

**Figure
4-432:
Geometry
after
splitting
the 26-42
edges.**

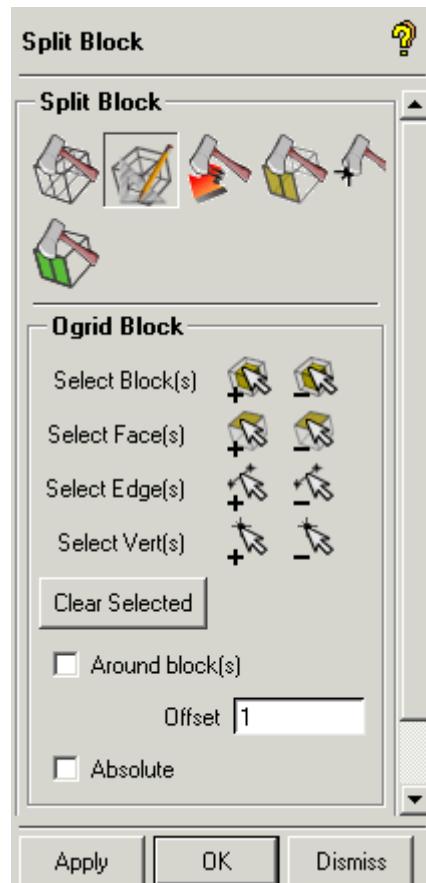


g) Creation of first O-grid

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 below.

Figure 4-433
O-grid block window



Now select block 13 and its two corresponding faces as shown below, after selection press Apply to create first O-grid. After creation of first O-grid blocking will look the second figure below.

**Figure
4-434
Block
and faces
selection
for first
O-grid
selection.**

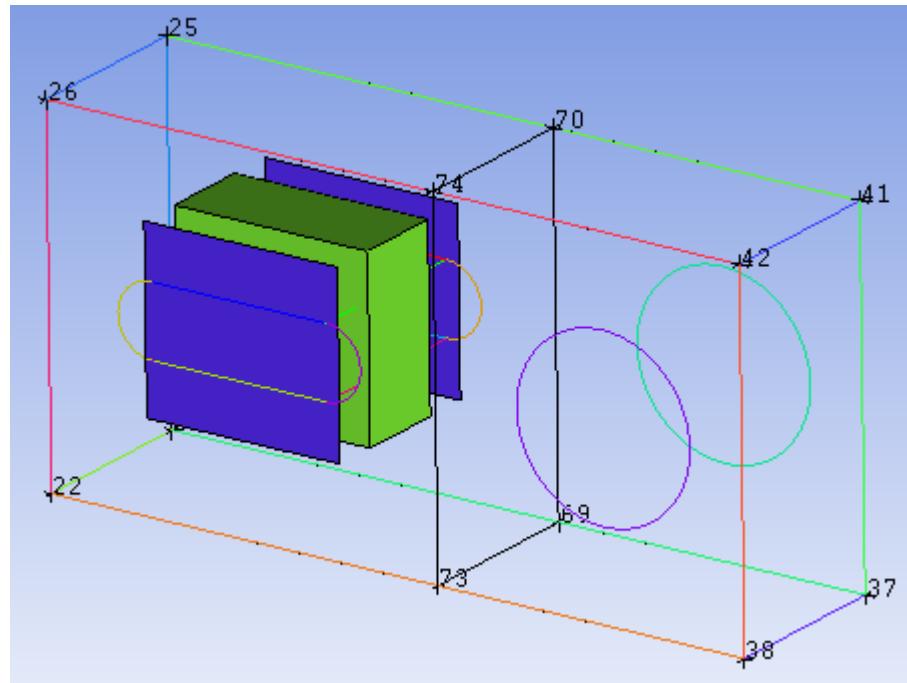
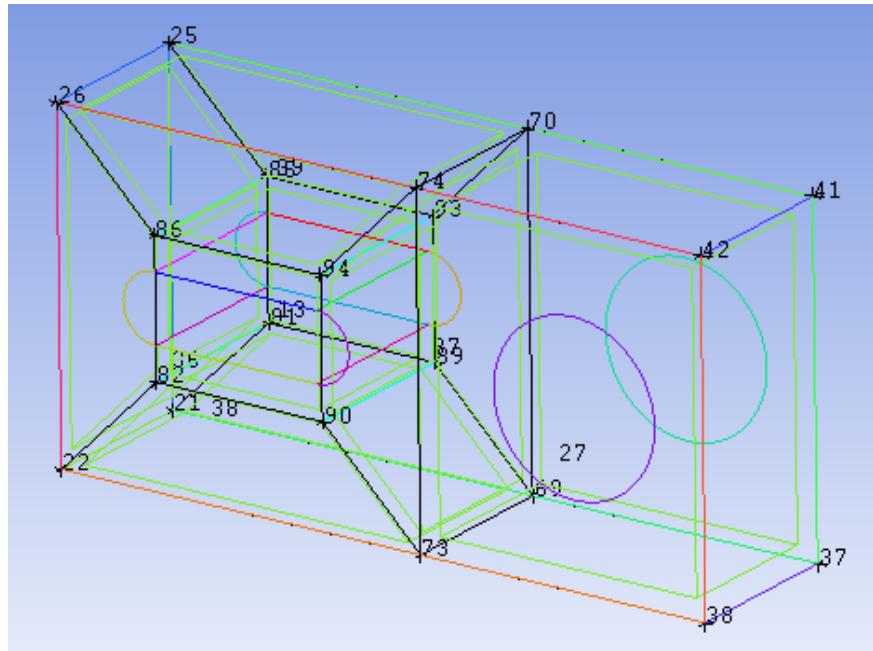


Figure 4-435
Blocking after first O-grid creation.



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.

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 below and press Apply.

Figure 4-436
Selection of block and faces selection for second O-grid creation.

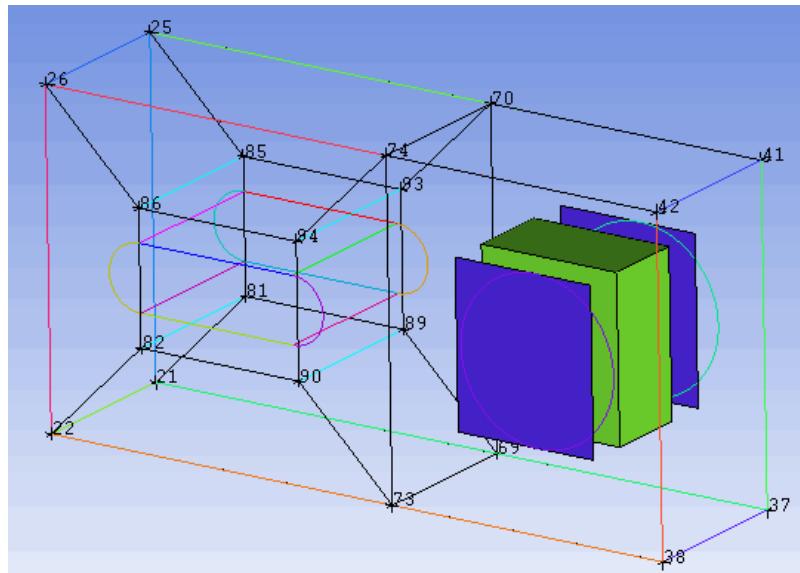
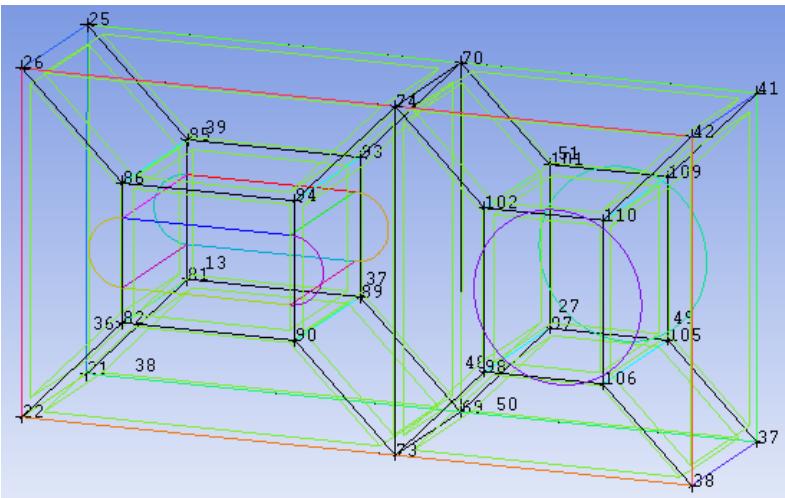


Figure 4-437
Blocking after second O-grid creation.



i) Association of Edges

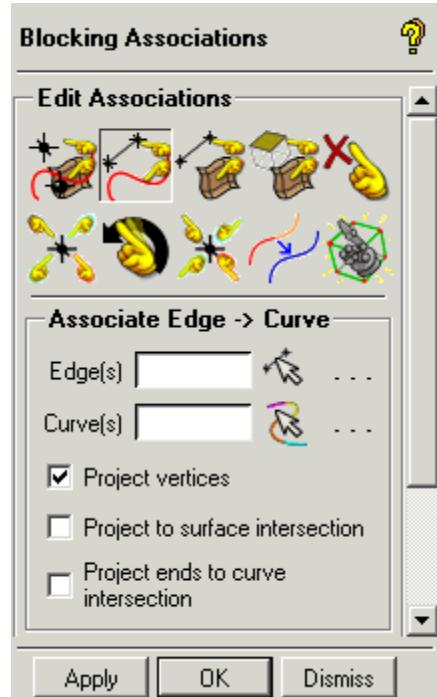
Now user will associate edges to corresponding curves to capture the geometry.

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	515
------------------------	--	-----

Select Blocking > Association  > Associate edge to curve , it will open the window as shown below. Toggle on Project Vertices.

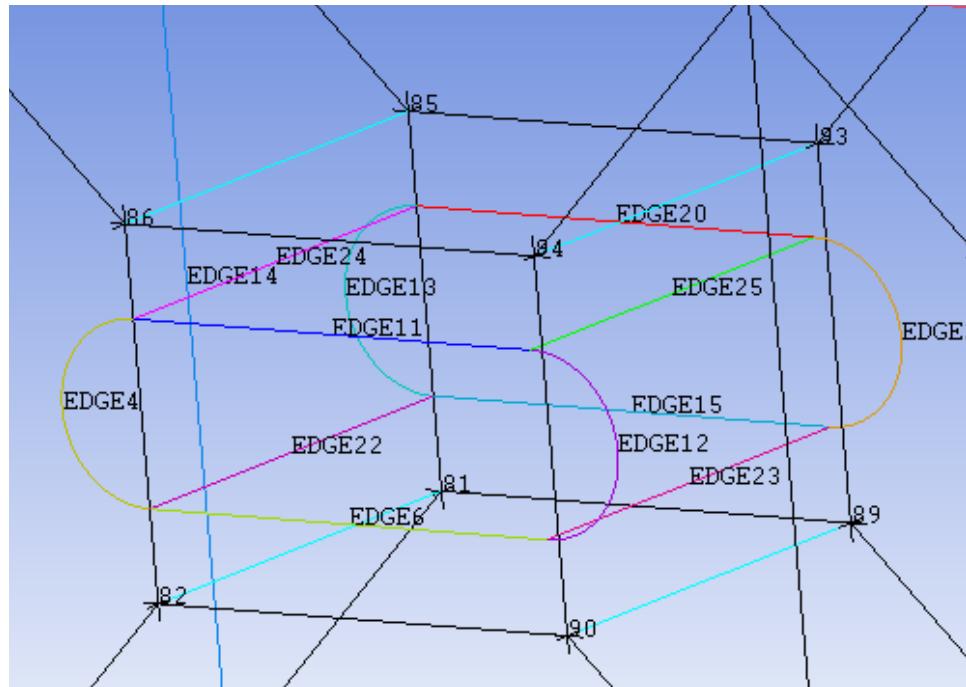
Turn on Curves (default) from the Display tree. Right Click on Curves > Show Curve Names in the Display tree. Also turn off Blocks from the Display Tree.

Figure4-438
Associate edges to curve
window



Select edges 82-86, 86-94, 94-90, 90-82, and associate it to curves named EDGE4, EDGE11, EDGE12, EDGE6 as shown below.

Figure 4-439
Association of edges to curves



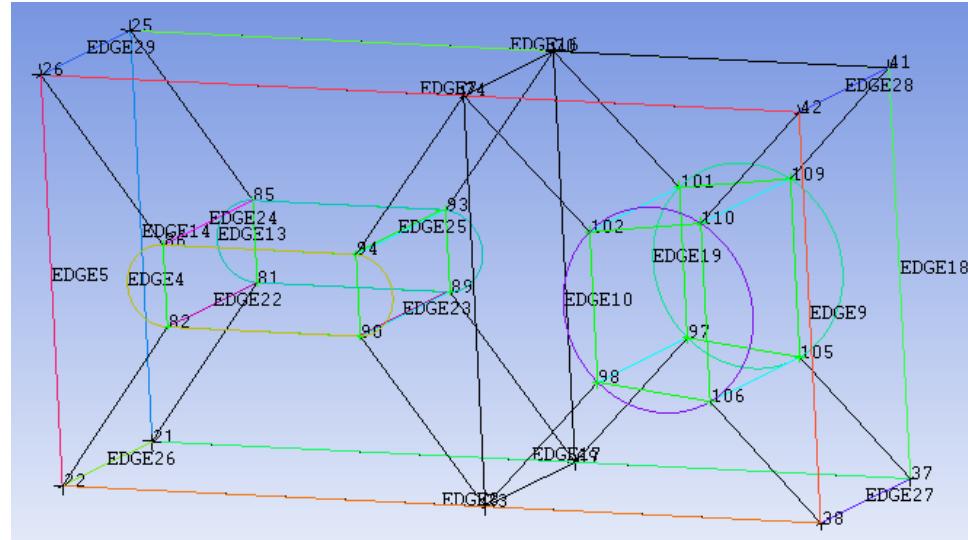
Select edges 81-85, 85-93, 93-89, 89-81 and associate it to the curves named EDGE13, EDGE20, EDGE21, EDGE15.

Select edges 82-86, 86-94, 94-90, 90-82 and associate it to the curves named EDGE4, EDGE11, EDGE12, EDGE6.

Similarly select edges 98-102, 102-110, 110-106, 106-98 and associate it to the curve named EDGE10.

Select edges 97-101, 101-109, 109-105, 105-97 and associate it to the curve named EDGE19. After association of edges it will look like as shown here.

Figure 4-440
Association of edges to curves.

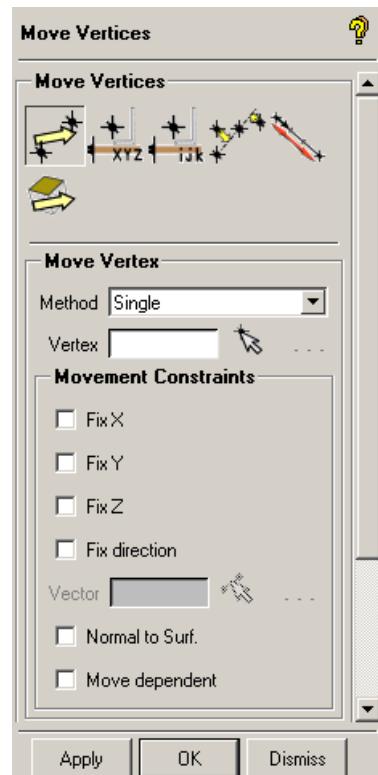


j) Moving vertices

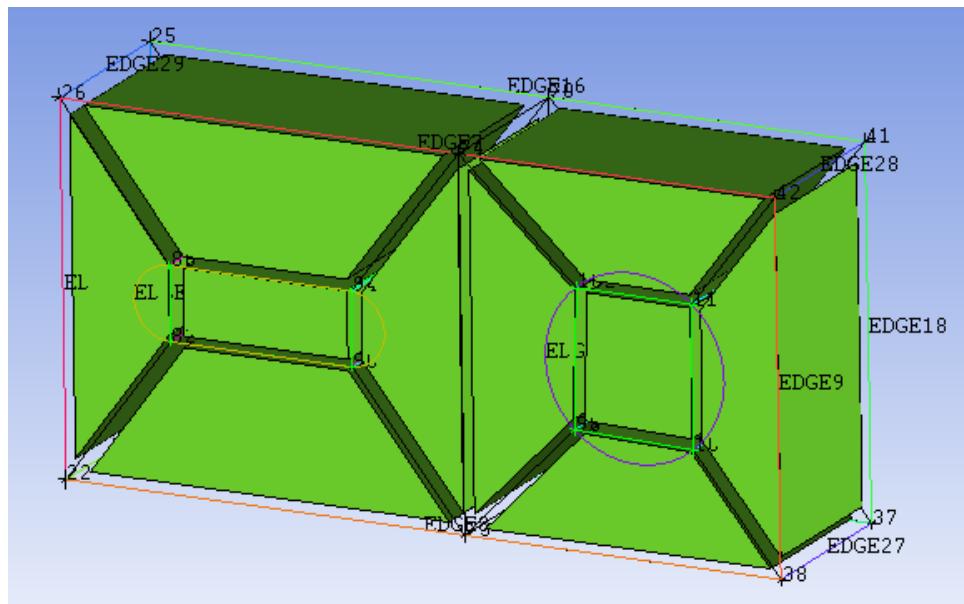
Now user has to move the vertices to improve the quality of blocks.

Select Blocking > Move vertices > Move vertex , and move the vertices from curves named EDGE10, and EDGE19. After turning the display of blocks ON and turning it to SOLID, the blocking will look like the figure below.

Figure 4-441
Move vertex window



**Figure
4-442
Blocking
after
moving vertices**



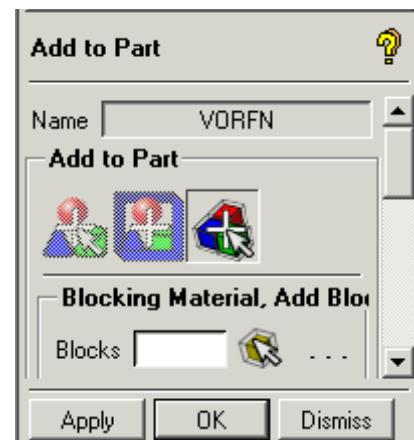
k) Adding blocks to VORFN

Now the user has to assign unrequired blocks to VORFN family. First Turn off the solid mode from the Display Tree and then Select Parts > VORFN > Add to part, it will open Add to part window. Select Blocking



material, Add blocks to Part  and add unrequired blocks (No. 13 & 27) as shown in the figure below to VORFN. Turn ON Blocking > Blocks > Solid, so that after adding parts to VORFN geometry should look like the last figure below.

Figure 4-443
Add to part window



**Figure
4-444:**
Blocks
to be
selected
in vorfn
family

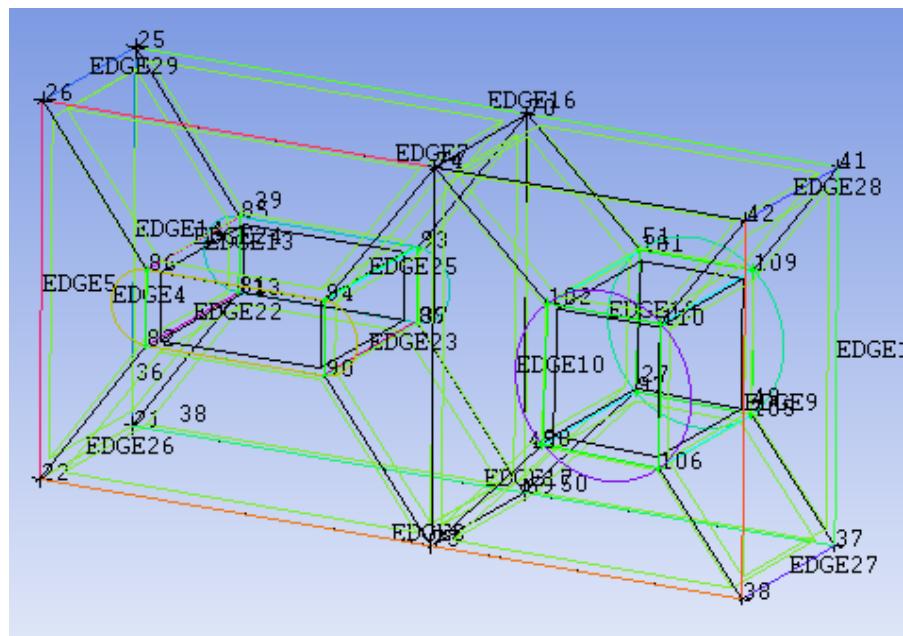


Figure4-445
Geometry
after
adding
blocks to
VORFN

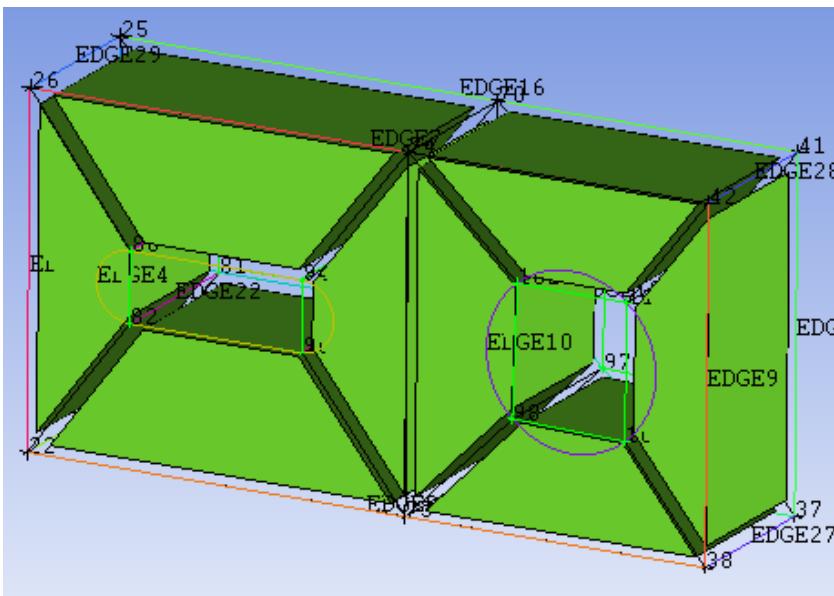


Figure4-446
Associate edge to curve
window

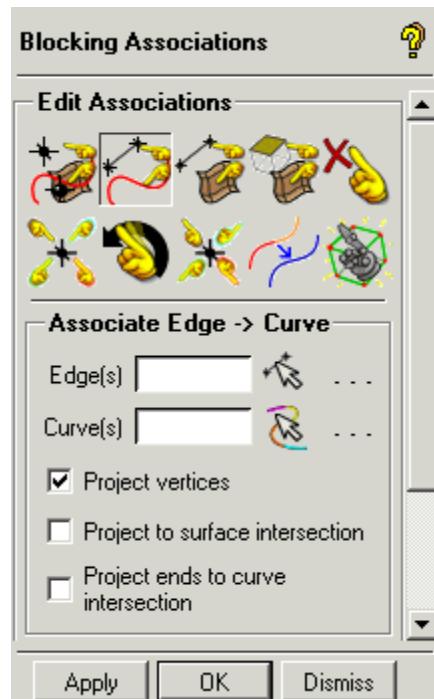
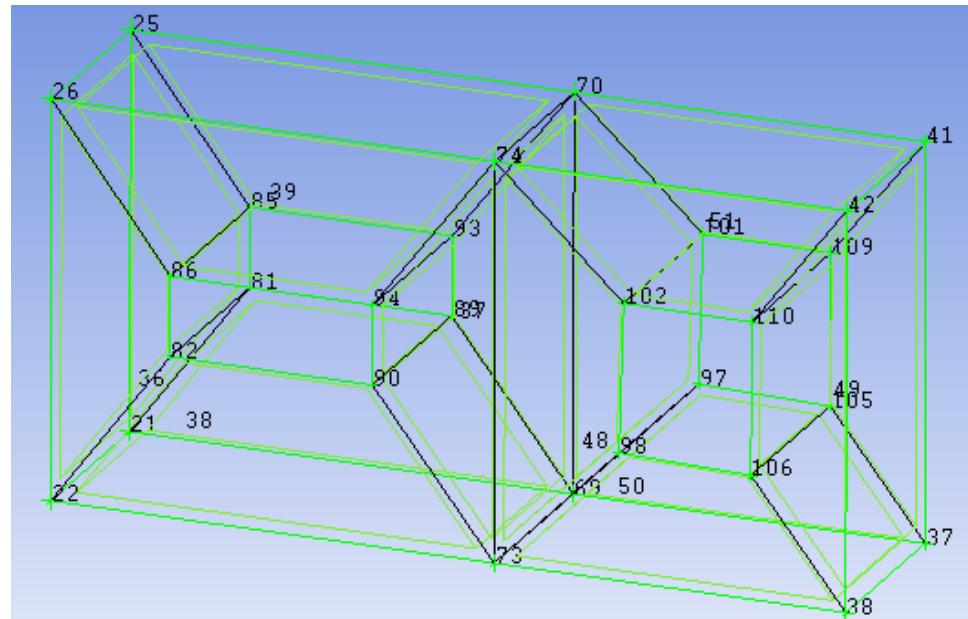


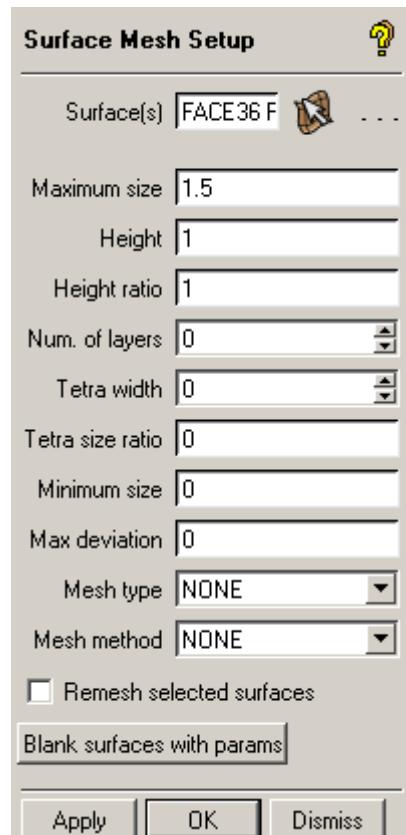
Figure 4-447
Association of edges to curve



m) Surface mesh size

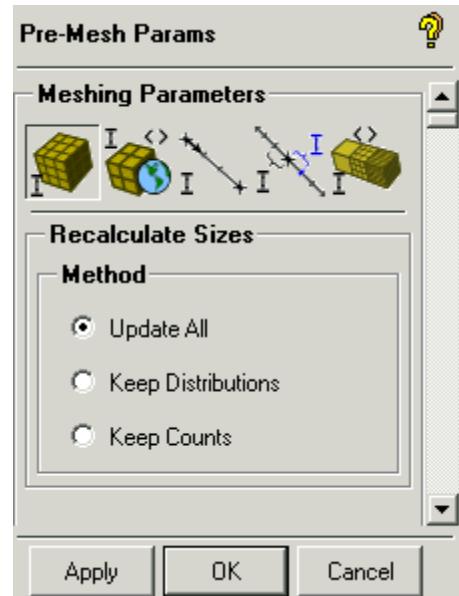
Mesh > surface mesh size , it will open surface mesh size window, Select  and select all the surfaces. Enter Maximum size as 1.5, Height 1 and Height ratio 1. Press Apply.

Figure 4-448
surface mesh size window



Now select Blocking > Pre-mesh params > Update Sizes. Select Update All and press Apply.

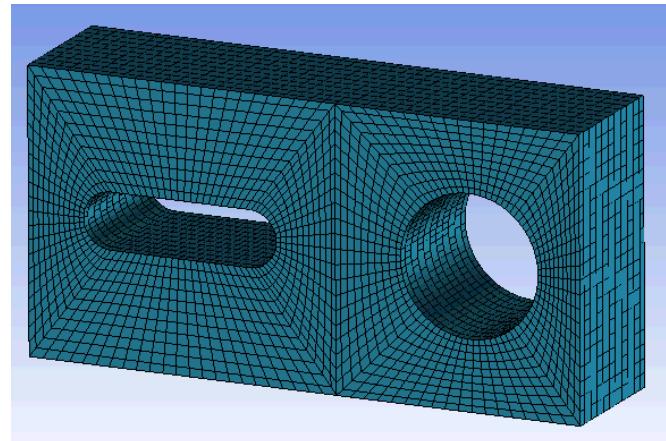
Figure4-449
Recalculate sizes window



Now turn ON Blocking > Pre-mesh from Display Tree, it will ask for Recompute. Press Yes to recompute.

Turn off Blocks from the Display Tree and turn on the display of Pre-mesh to Solid from Display Tree Pre-mesh > Solid. After turning it to solid blocking will look like as shown in

Figure 4-450
Blocking after
completing meshing



n) Checking Quality and running Pre-mesh smoother

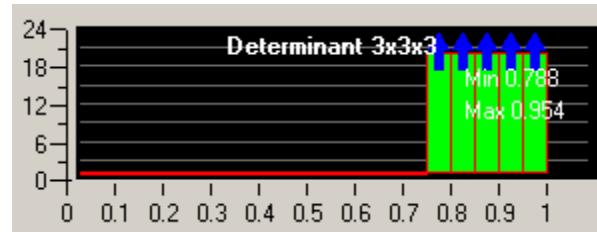
Now user will check the quality of mesh which will be created from blocking and run Pre-mesh smoother to improve its quality.

Choose Blocking > Pre-mesh quality , and criteria as Determinant 3x3x3 and enter the parameters as shown below . It will show the quality of mesh in histogram similar to quality shown here.

**Figure 4-451
Pre-Mesh quality window**



**Figure 4-452
Histogram showing
Determinant 3*3*3
quality**



a) Modifying geometry in DM

Now user will modify the geometry in DM. Go back to DM and Select Tools >Parameters from main menu as shown here. It will open the parameter window.

Figure 4-453
Selection of Parameters from Main Menu

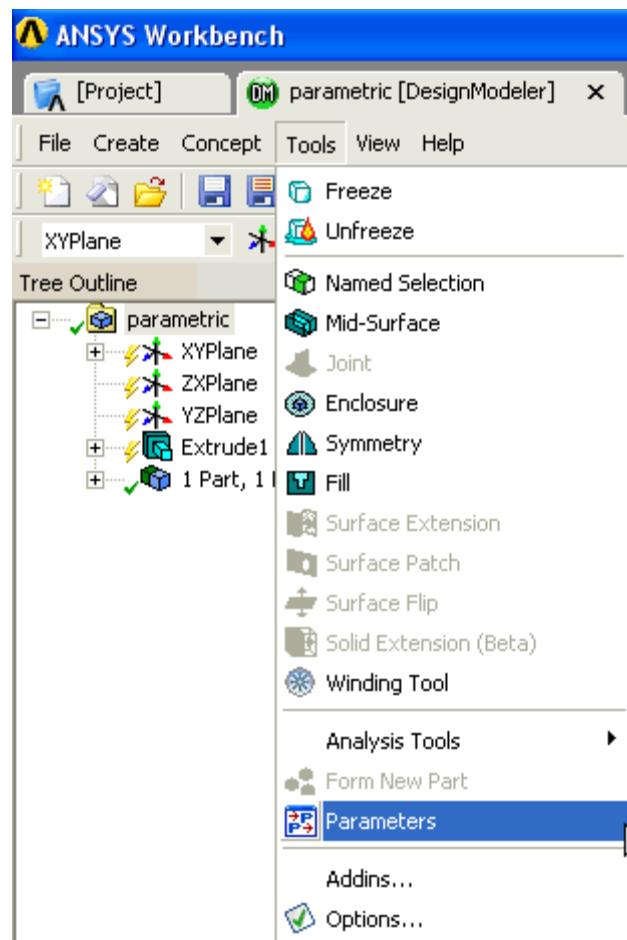
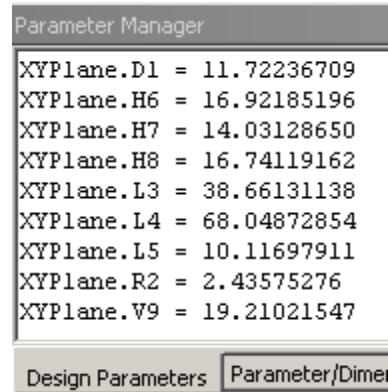
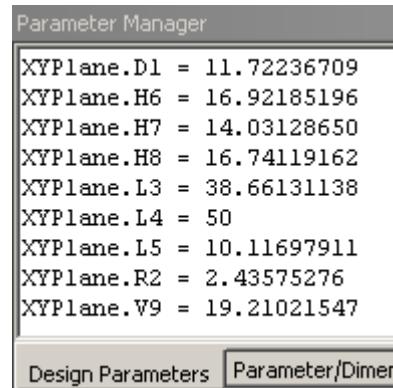


Figure 4-454
Parameter window



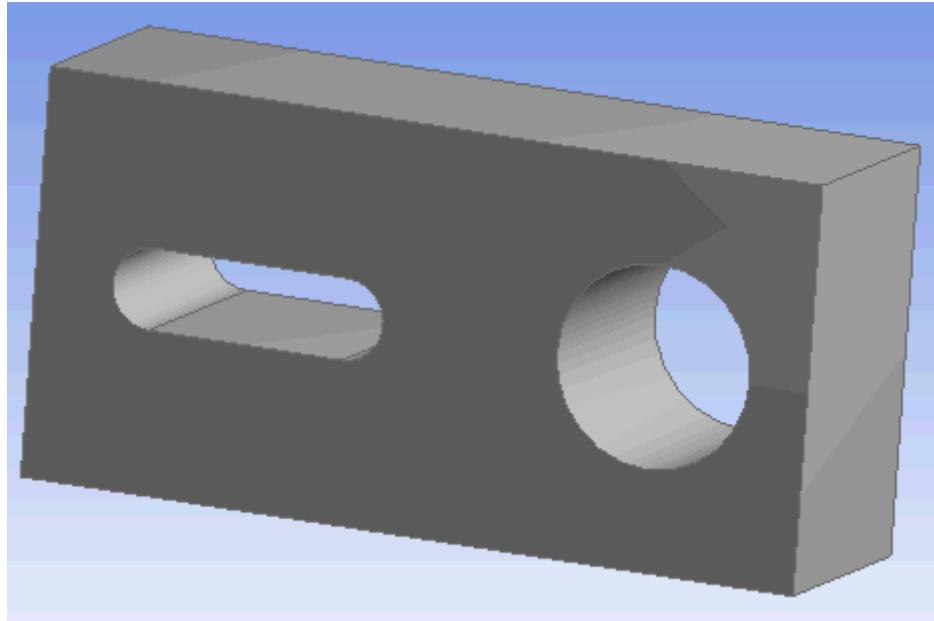
Now user will change the width of block change the dimension of XYPlane.L4 = 50. Parameter window after changing length is shown here. Similarly do the similar changer for “Parameter/Dimension Assignments tab” and for “Check” tab in the same window. Now close the parameter window.

Figure 4-455
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 below.

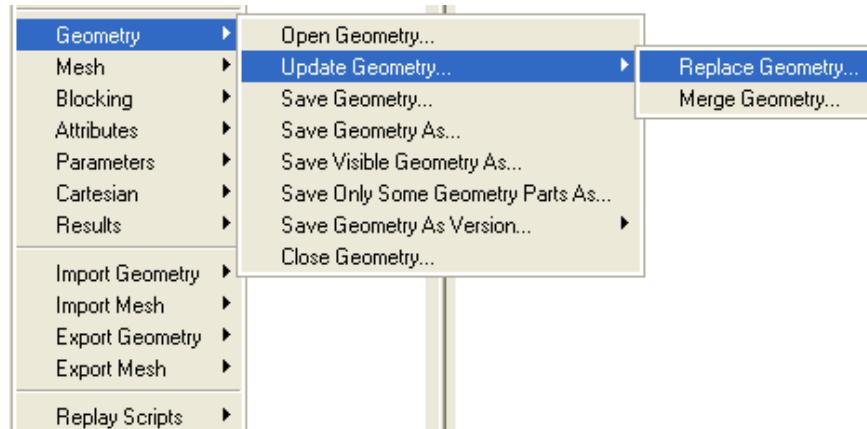
**Figure
4-456
Geometry
after
modifying
dimensions**



p) Updating blocking in Advance meshing

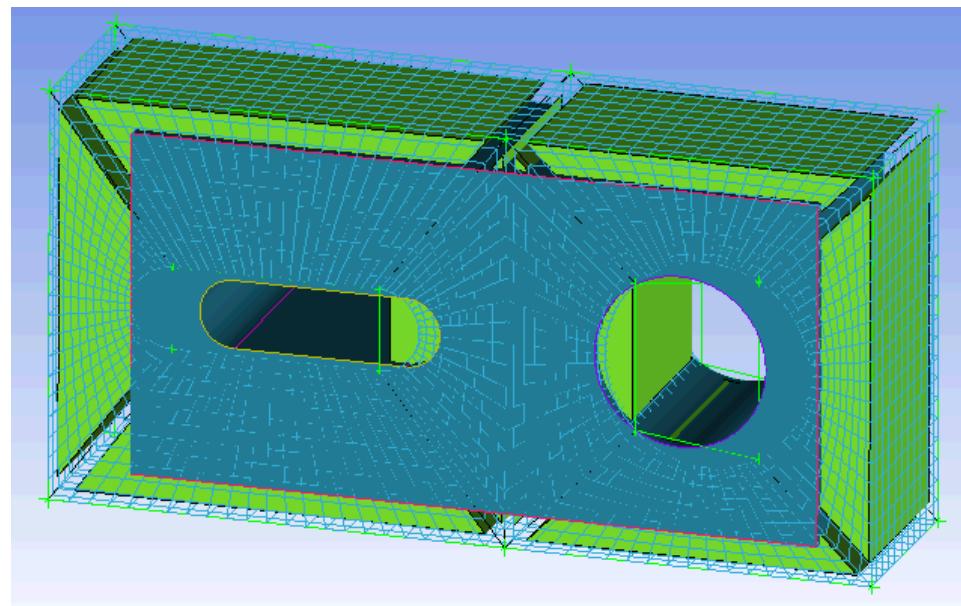
Now user will update blocking for changes made in original geometry. To open modified geometry in Advanced Meshing select File > Geometry > Update Geometry > Replace Geometry as shown. It will ask for saving the changes in geometry, select Yes to make changes. Toggle on Surfaces under Geometry in the Display Tree. In the Display Tree, right click on Surfaces > Solid.

**Figure
4-457:
Updating
geometry
in
Advance
Meshing**



It will open the modified geometry file and merge with the original blocking file as shown.

**Figure4-458
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. Turn on Update Blocking and keep rest of the things as default. Press Apply. After updating the blocking, new blocking will look like the figure below.

Figure 4-459
Update association window

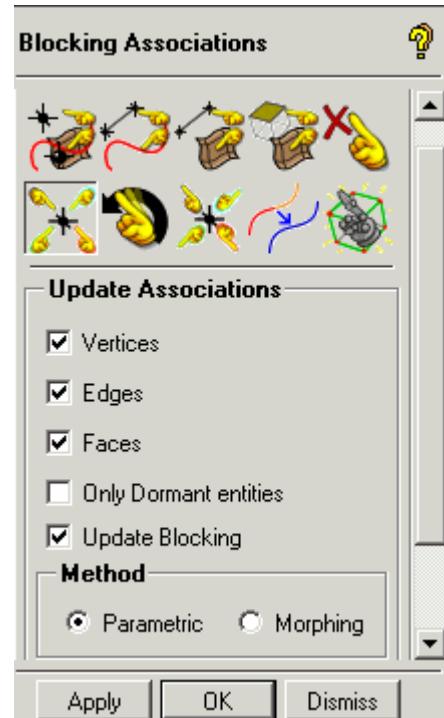
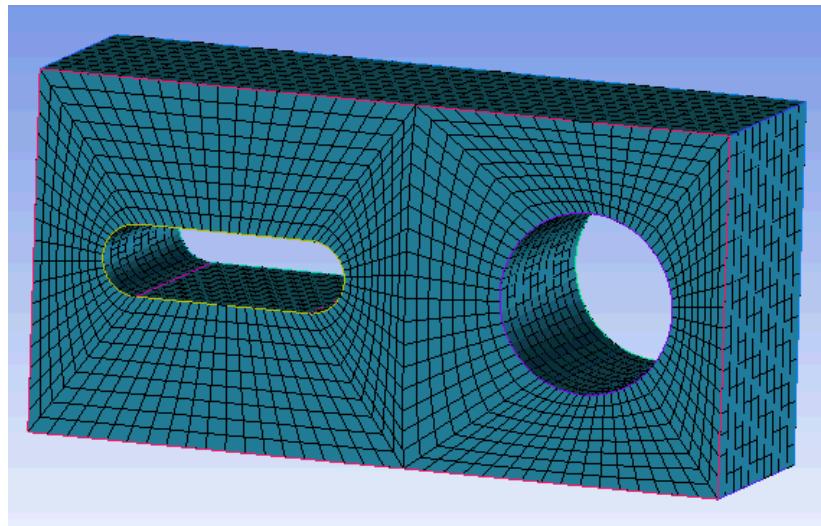


Figure 4-460
Blocking
after
updating
associations



q) Saving the Project

Now user will save project, Select File > Save project as and enter name as Parametric.

4.6.6: Hexa-Core Mesh from Shrinkwrap mesh

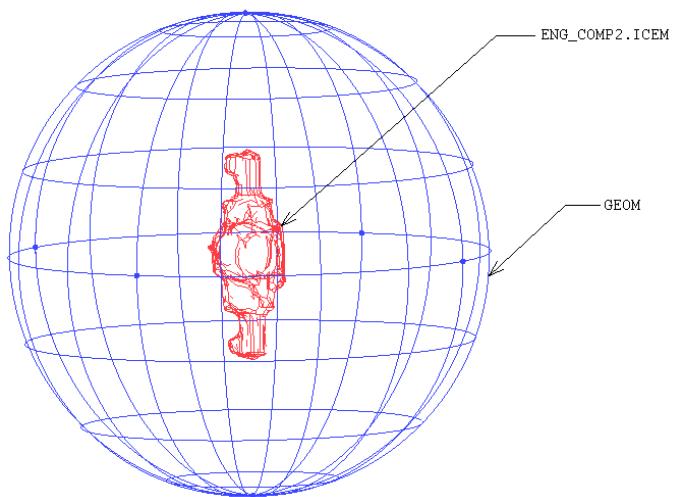
Overview

In this tutorial example, the user will generate and smooth a combined Tetra/HexaCore mesh. The mesh will be for the region between the geometry and the surrounding sphere. Finally after meshing, the user will perform smoothing which generally provides more uniformly spaced mesh.

Figure 4-461

Geometry

Showing Parts



a) Summary of steps

- Starting the project
- Generating Sphere Geometry
- Setting Global Mesh Sizes
- Assigning Mesh sizes
- Generating Surface mesh
- Smoothing Surface Mesh
- Generating Volume Mesh

Smoothing Mesh
Saving the project

b) Starting the Project

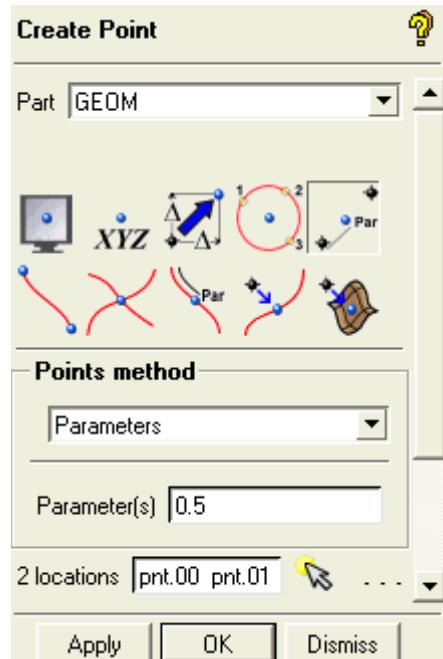
From UNIX or DOS window, start ANSYS ICEM CFD. The input files for this tutorial can be found in the Ansys Installation directory, under/docu/Tutorials/CFD_Tutorial_Files > HexCore_Shinkwrap. Copy and the tetin (*.tin) and mesh (*.uns) file in your working directory.

c) Generating Sphere Geometry

Select Geometry > Create Point  > Screen Select  > Select 2 locations at two opposite ends and Apply. Two points will be created at two ends as shown below.

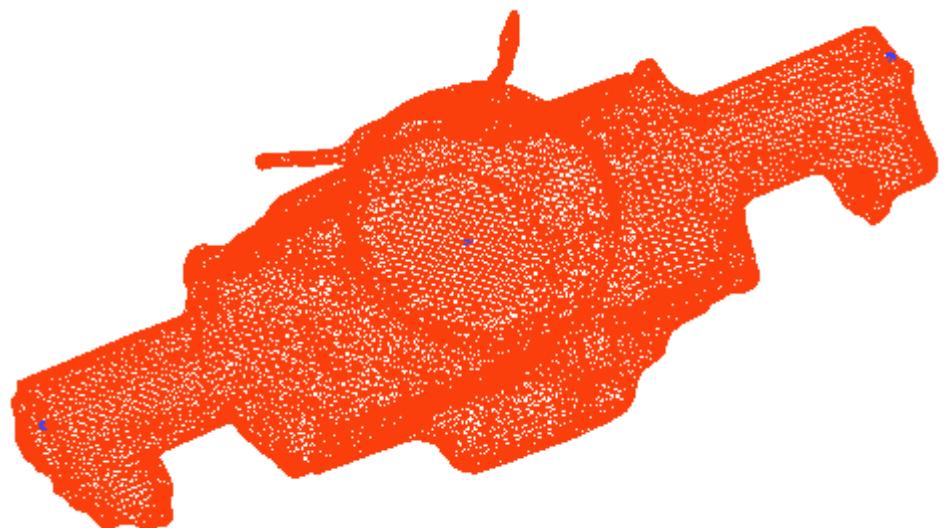
Select Create Point  > Based on 2 Locations  > Points method > Parameter. Enter the parameter value as 0.5 as shown.

Figure 4-462 Point Creation Window



Then select two end points created earlier. A new point will be created at the center of two points as shown.

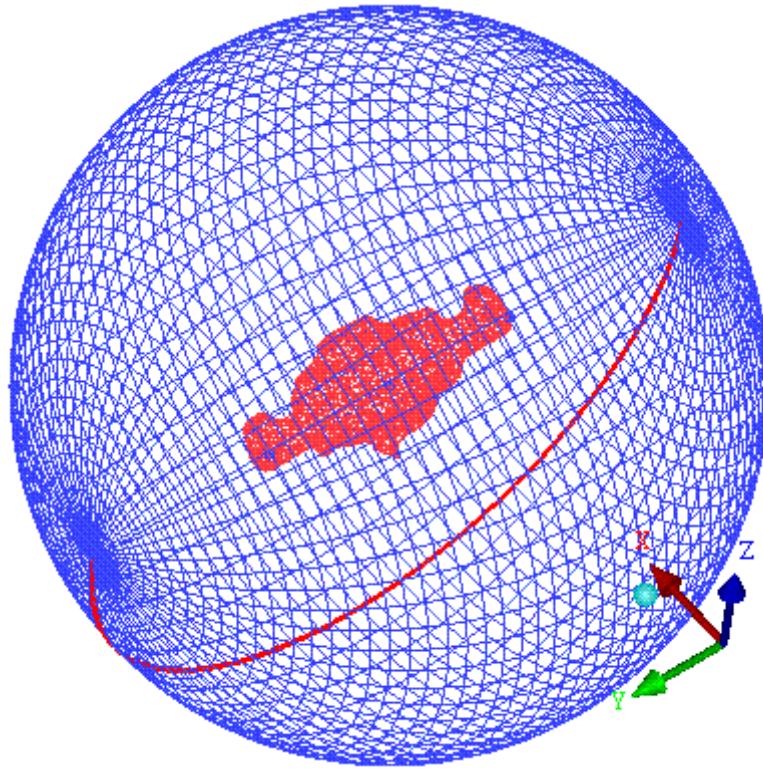
Figure 4-463 Geometry after Points created



To create the sphere, go to

Create/Modify Surface > Standard Shapes > Sphere .
Toggle ON the Radius option. Enter the Radius Value as 400, Start Angle as 0, and End Angle as 180. For locations select the center point as the first point and select one of the end points as the second point. Apply.

Figure 4-464 Sphere Creation



d) Creating a Body

We have to create a Body in the Sphere. Select Create Body  >

 Material Point  > Centroid of 2 Points. For the 2 Screen locations , select the first point as one of the end point of the given model and thesecond point as any point on the Sphere. Apply. A Body gets created and is added as ‘BODY’ in the part list. Note that the body gets created between the sphere and the given model not inside the model.

To split the sphere, go to

Select Repair Geometry  > Split Folded Surfaces .

Figure 4-465 Splitting of Sphere



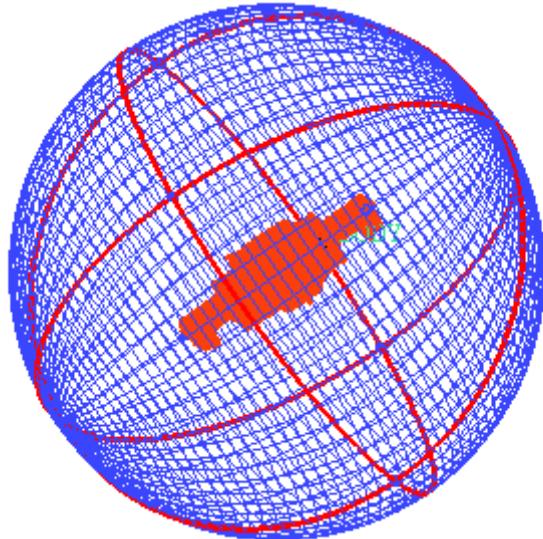
Set the Max angle to 90. Select  and either select “P” from the keyboard or Select items in a part (key =shift-P)  from the selection toolbar. A window will open with a list of parts.

Figure 4-466 Selecting Part



Select GEOM and press Accept. After splitting, the Sphere geometry should appear as shown below.

Figure 4-467 Sphere after splitting

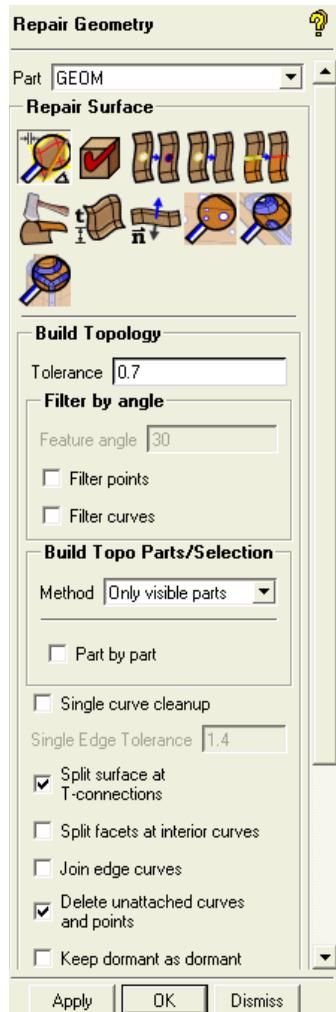


Go to Geometry > Repair Geometry > Build Topology.

Select Settings > Geometry Option s> Inherited is to be Toggled ON.
Choose the Only Visible Parts Method.In the Display Tree, only Part

'GEOM' should be Toggled 'ON'.Keep all the default values as shown below. Press Apply.

Figure 4-468 Building Topology

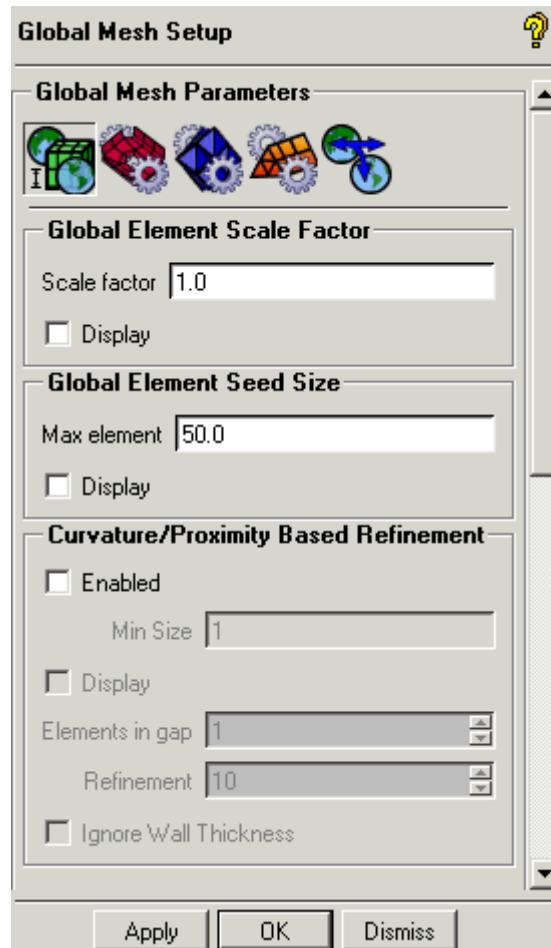


e) Setting Global Mesh Sizes

Choose Mesh > Global Mesh Setup > Global Mesh Size to open the Global Mesh Size window.

Enter Max Element size as 50, accept all other defaults and press Apply.

Figure
4-469
Global
Mesh Size
Window



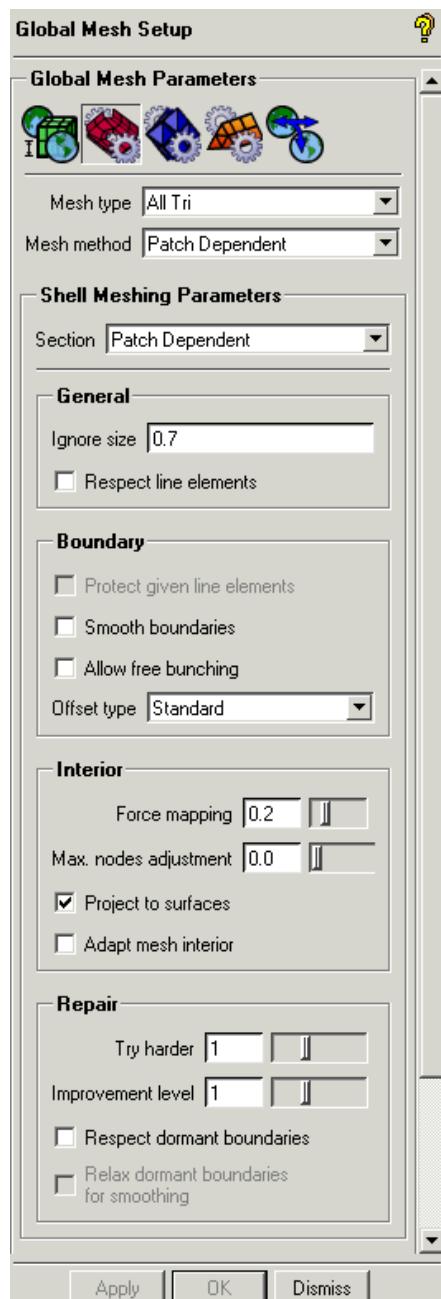


Select Mesh > Global Mesh Setup > Shell Meshing



Parameters . Select Mesh Type All Tri and Mesh Method Patch Dependent, accept all other defaults and press Apply.

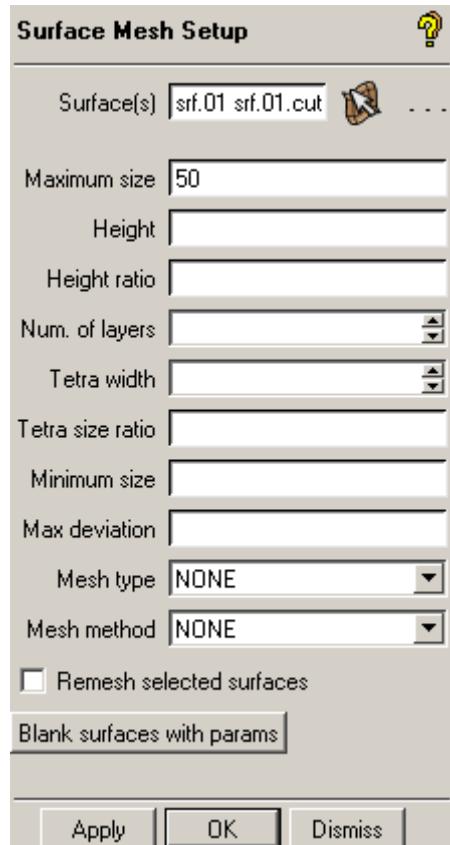
Figure 4-470
Global Shell
Meshing Setup



f) Assigning Surface mesh size

Select Mesh > Surface Mesh Setup  > Select surface(s)  . Select Items in Part option  , and select part GEOM. Assign a Maximum size of 50.

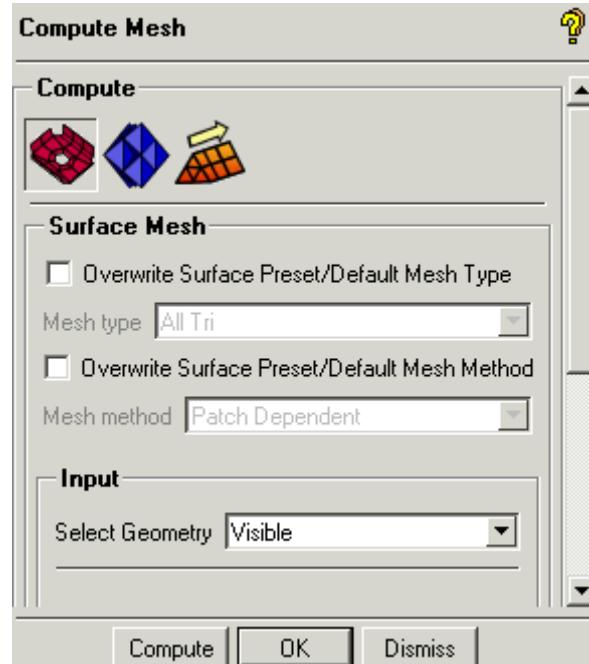
Figure 4-471
Surface Size



g) Generating Surface Mesh

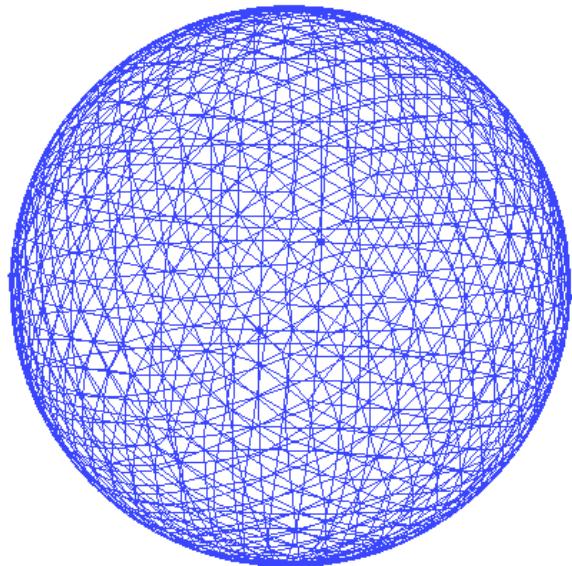
Select Mesh > Compute Mesh  > Shell Mesh Only  . Select option Visible in the Select Geometry. Make sure only GEOM part is active. Press Compute.

Figure
4-472
Shell
Mesh
Creation
for
Sphere
Geometry



Tri mesh created on the sphere is shown below.

Figure
4-473
Tri
Mesh
Created



h) Surface Mesh Smoothing

Select Edit Mesh > Display Mesh Quality . Select Quality Type as Quality and press Apply. It will show the corresponding Quality Histogram below.

Figure 4-474
Mesh Quality

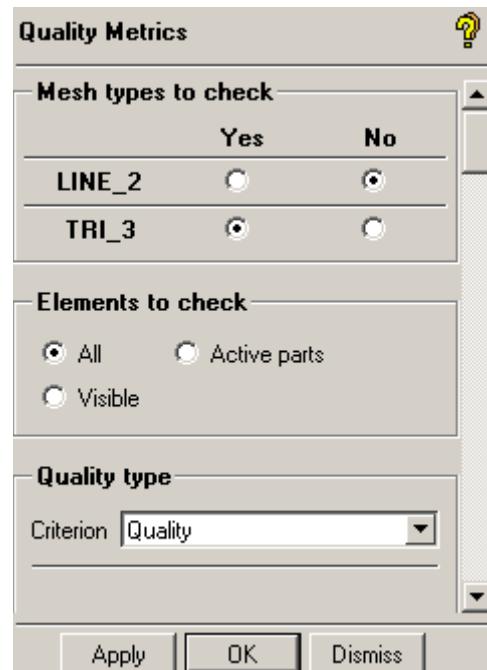
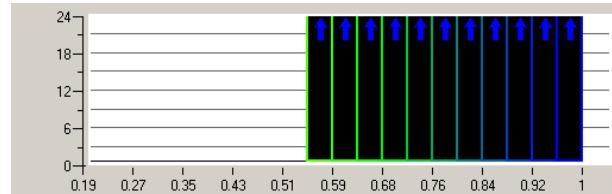
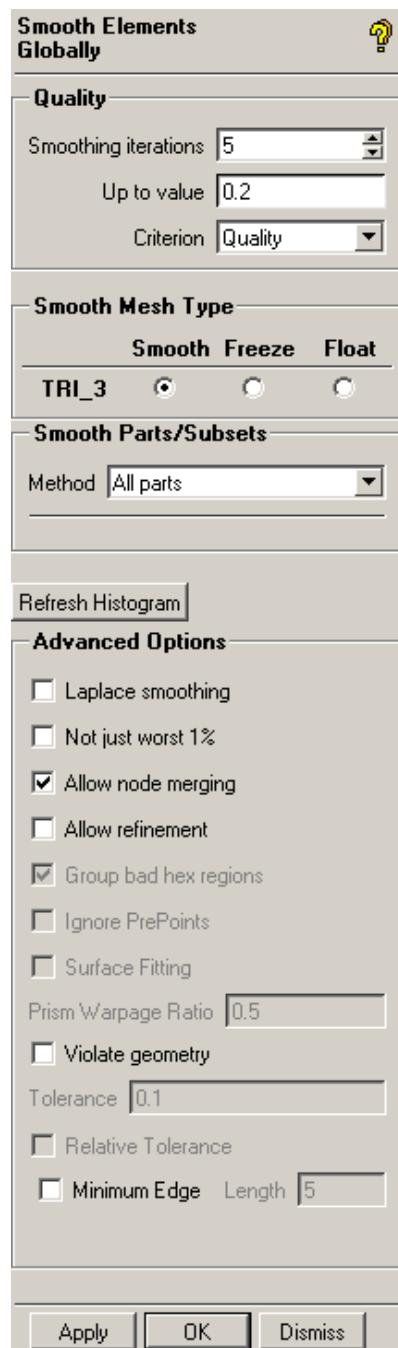


Figure
4-475
Histogram
Before
Smoothing



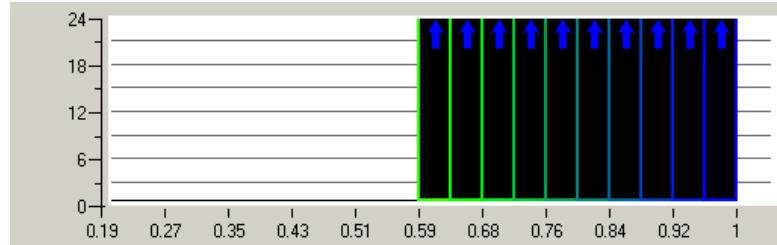
Select Edit Mesh > Smooth Mesh Globally . If quality is below 0.2 then Enter Smoothing Iterations as 5, Up to Value 0.2 and Criterion as Quality and press Apply. Repeat smoothing by increasing the value to 0.5 in steps of 0.1 units.

Figure 4-476
Smooth Shell Mesh



The Histogram displays the improved quality.

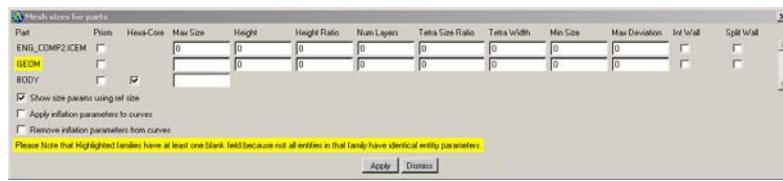
Figure
4-477
Histogram
After
Smoothing



i) Setting Meshing Parameters

Select Mesh > Part Mesh Setup . Toggle ON Hexa Core for BODY, press Apply followed by Dismiss.

Figure
4-478
Part
Mesh
Setup



j) Generating Volume Mesh

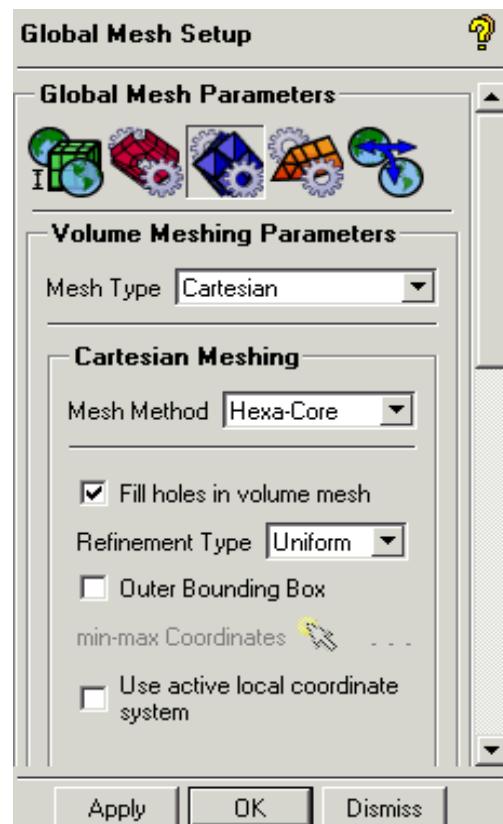
Select Mesh > Global Mesh Setup > Volume Meshing Parameters



> Cartesian > Hexa-Core Mesh Method.

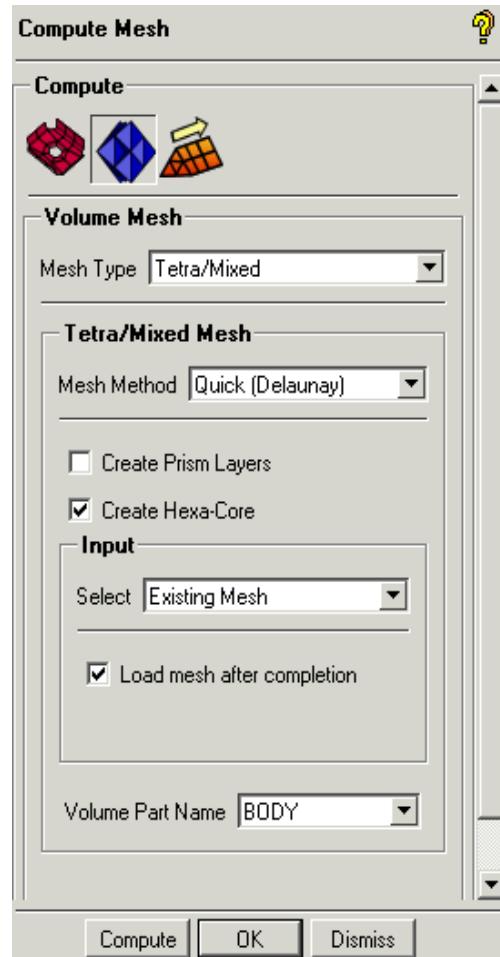
Toggle ON the option **Fill Holes In Volume Mesh**, press Apply.

Figure 4.479
Global Mesh
Setup Window



Select Mesh > Compute Mesh > Volume Mesh > Tetra/Mixed > Quick(Delaunay). Select Existing Mesh option as Input, Toggle ON Create Hexa Core and Load Mesh After Completion. Enter Volume Part name as BODY. Press Compute.

Figure 4-480
Compute Mesh
Window



Volume Mesh gets generated.

Turn ON shell elements, select Mesh Tree > Volume Mesh > Solid and Wire. Select Mesh Tree > Cut Plane and adjust the fraction value to scan this image. Hexa elements gets created at the middle region between the parts ENG_COMP2.ICEM and GEOM and in the remaining portion tetra elements gets created. Pyramid elements gets created between Hex and Tetra elements.

k) Mesh Smoothing

Select Edit Mesh > Smooth Mesh Globally , enter Smoothing Iterations to 5, Up to Value 0.3, Select Criterion Quality, and press Apply.

Figure 4-481
Smooth Volume
Mesh



The user can repeat these steps by adjusting the parameters to get a better quality mesh.

I) Saving the Project

Save the project by selecting File > Save Project. Then close the project by selecting File > Close Project.

4.7: Cart3D

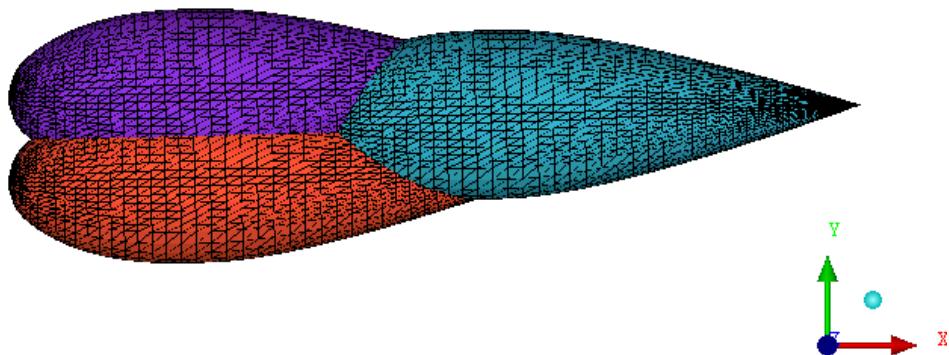
The Cart3D menu has the following options:

- Volume Mesher 
- Solve 
- Integrate Cp 
- Run Trials 
- Run 6 DOF 

4.7.1: 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:

1. Use of the Cart3D mesher for mesh generation.
2. Multigrid preparation – running mgPrep.

a) Starting the Project

The input files for this tutorial can be found in the Ansys Installation directory, under/docu/Tutorials/CFD_Tutorial_Files/Cart3D_Examples. Copy and open the **plugs.uns** file in your working directory.

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.

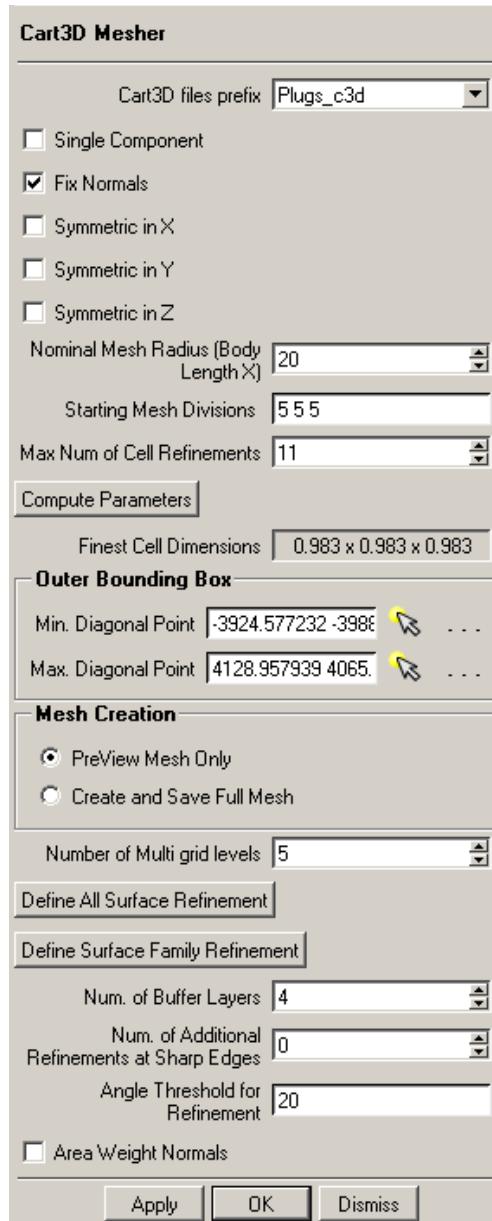
b) Mesh Generation Preview only

Click on the Cart 3D tab. Select the Volume Mesher icon . The following window will open.

Toggle ‘ON’ Fix Normals-it 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. Press ‘Compute Parameters’ button. This saves the mesh in the local directory, converts it 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 below.

**Figure
4-482
Cart3D
Mesher
window**



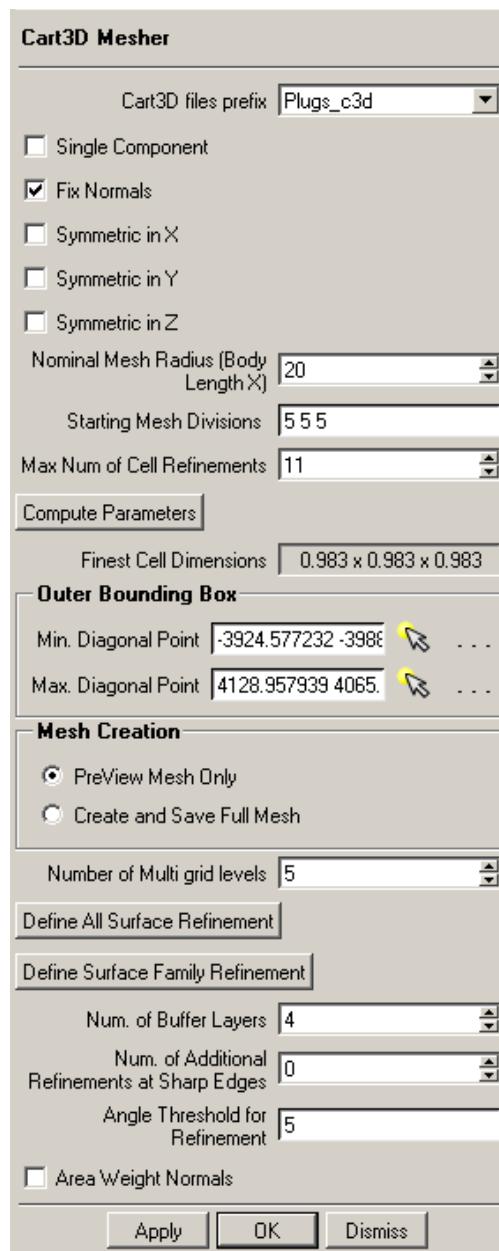
This will create 4 density polygons for mesh density control, which can be viewed in the Display Tree by switching on the Geometry > Densities.

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. They can be changed if desired.

Set the Angle Threshold for Refinement to 5 as shown in below.

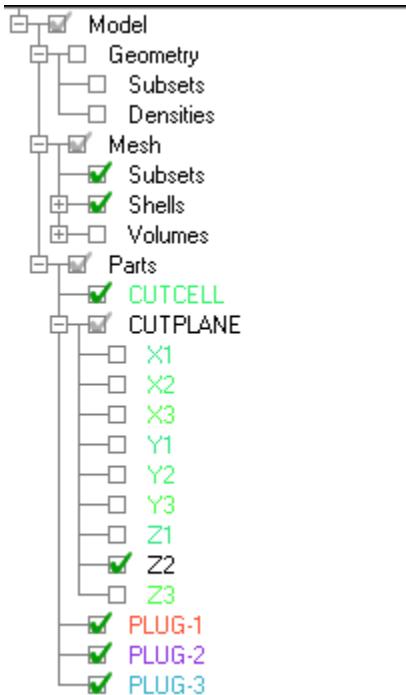
Figure 4-483
Change
Angle of
Refinement



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.

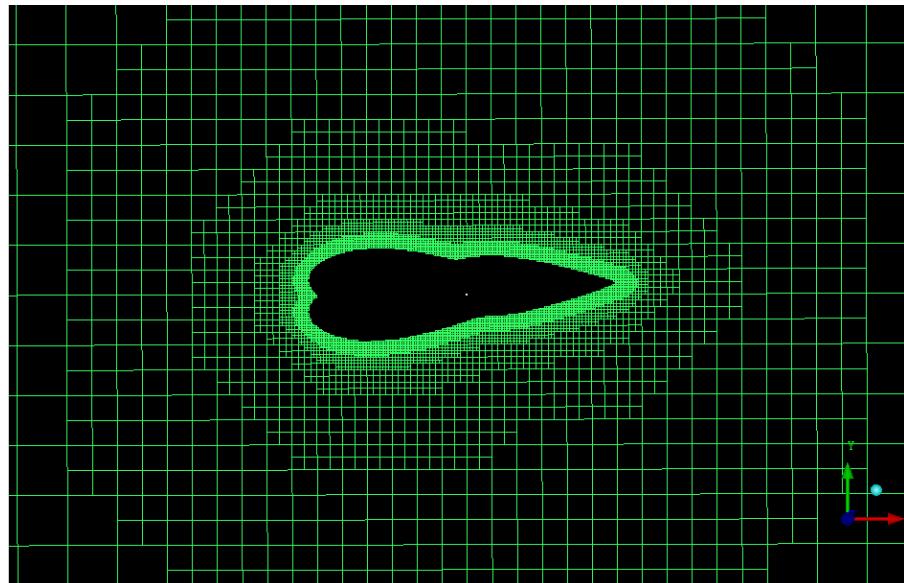
In the Part Menu under the Display Tree right-click on Parts and select Hide All. Then turn on only the Part **CUTPLANE-Z2**.

**Figure
4-484
Display
Tree**



The mesh is shown here. This is the projected mesh on the middle plane in the Z-direction **CUTPLANE-Z2**.

**Figure
4-485
Cut
Plane
Z2
Mesh**

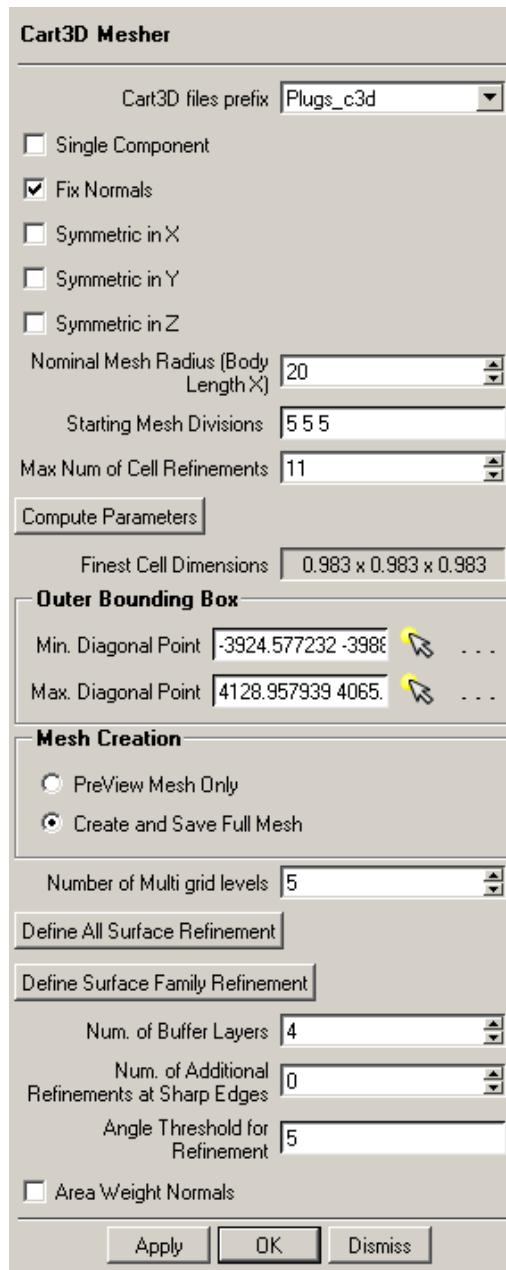


Right-click in the Display Tree and select Parts > Show All after viewing the mesh.

c) Mesh Generation Full Mesh

Now in the Cart3D Mesher window enable Create and Save Full Mesh as shown.

**Figure
4-486
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, press Yes.

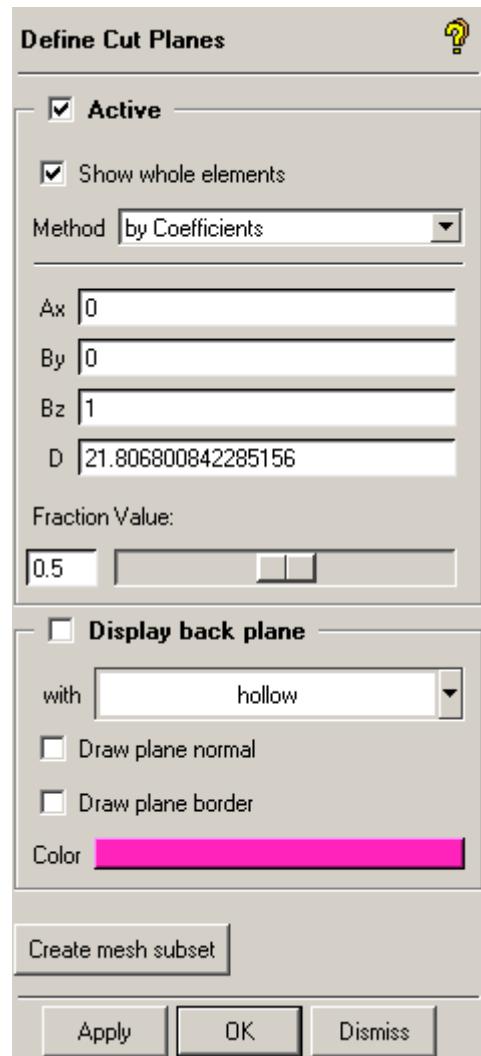
Figure 4-487
Cart3D Mesh window



Switch on Mesh > Volumes in the Display Tree.

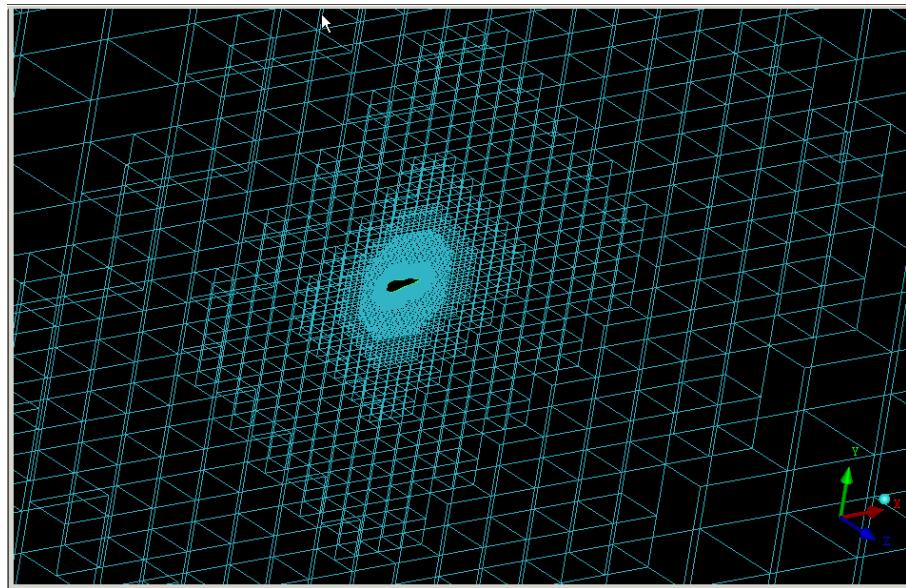
The final mesh generated can be examined through Mesh > Cut plane. The Define Cut Planes window appears as shown. Accept the default settings.

Figure 4-488
Cut Plane Display



The mesh viewed using the above parameters is shown below.

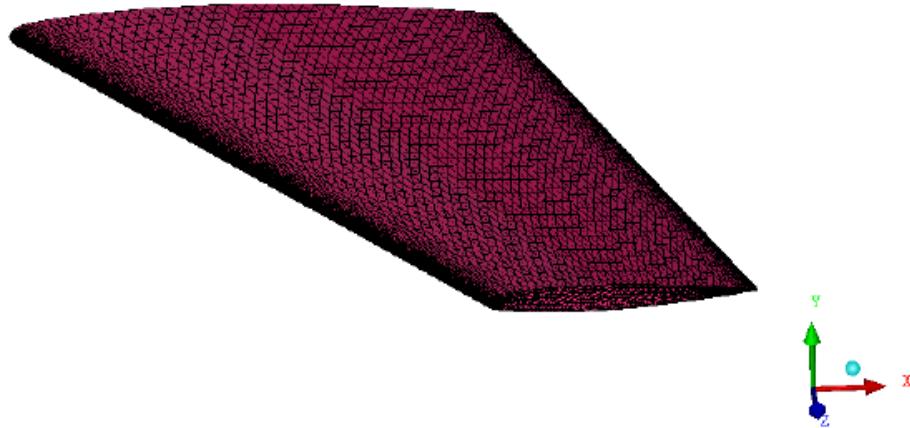
**Figure
4-489
Cut
Plane
mesh**



4.7.2: Tutorial Onera 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:

1. Use of the Cart3D mesher for mesh generation
2. Multi grid preparation with mgPrep
3. Running the solver for AOA=3.06 and Mach=0.54
4. Computing Forces and Moments using Clic.
5. Visualizing the result in Post Processing

a) Starting the Project

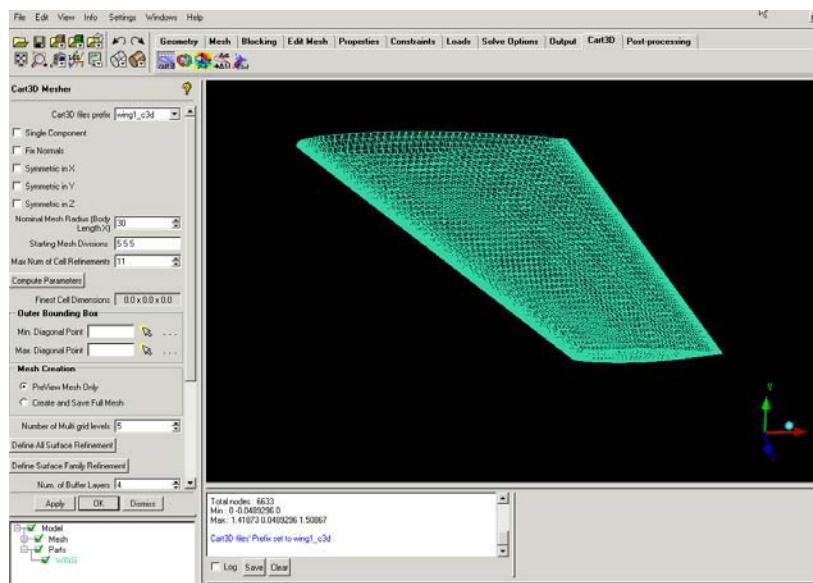
The input files for this tutorial can be found in the Ansys Installation directory, under `../docu/Tutorials/CFD_Tutorial_Files/Cart3D_Examples`. Note: It is preferable to create a separate folder `wing1` and put the `oneraM6.uns` (domain) file in that folder before performing this tutorial.

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 below.

**Figure
4-490
Cart3D
GUI
window**



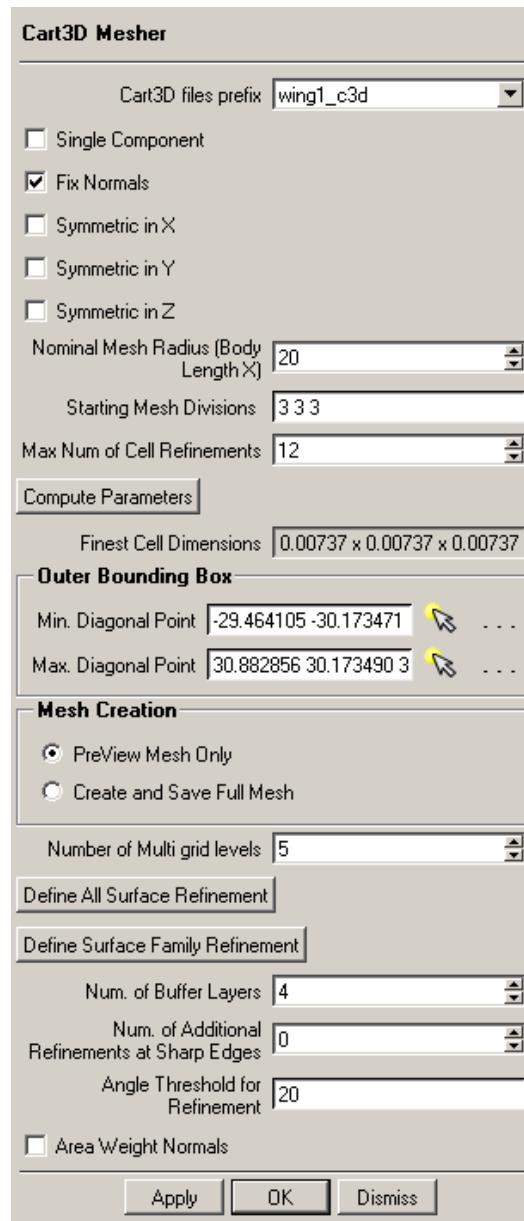
Toggle ON- Fix Normals. This will fix the orientation of the triangles such that their normals are pointing outward.

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 it 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.

Cart3D

Figure 4-491
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.

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.

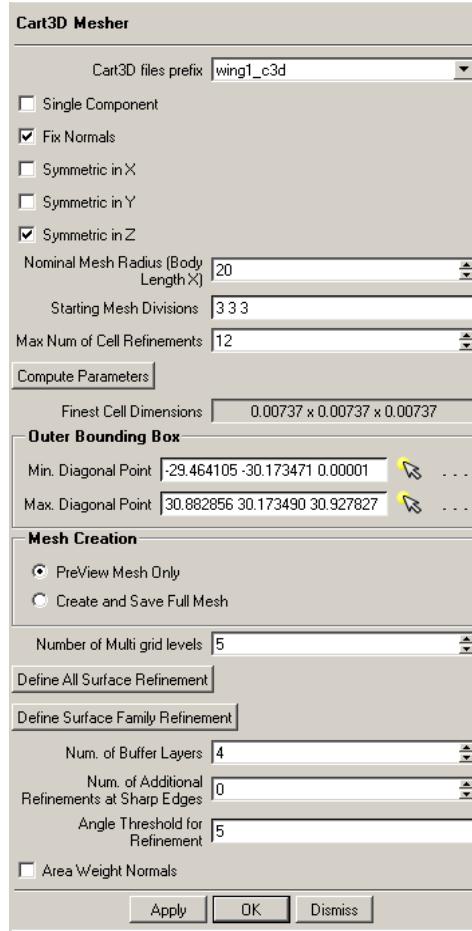
Set the Angle Threshold for Refinement to 5

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). If the model itself is symmetric, turn on Half-Body Mesh (Symmetric in Z).

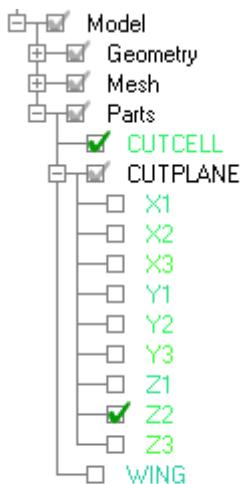
Click Apply (after specifying minimum Z coordinates as 0.00001) as shown below to run the mesher. This will create a domain file with 3 Cut Planes (Quad Elements) in each coordinate direction and Cut Cells (Hex Elements). The Preview Mesh will be loaded automatically.

Figure 4-492
Change Angle of
Refinement



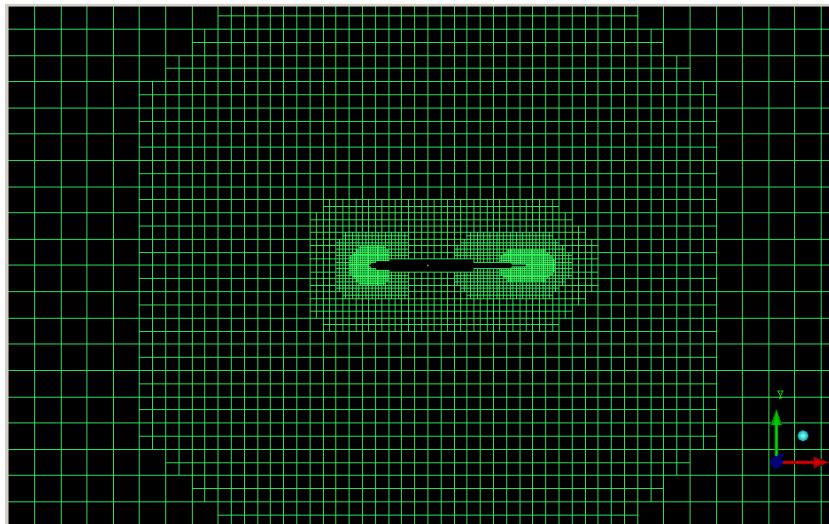
In the Parts menu under the Display Tree perform the operation
 Parts>Hide All (right-click on Parts to access) and then turn on only the
 Part **CUTPLANE-Z2**.

**Figure 4-493
Display Tree**



The mesh projected onto the middle z-direction plane (in Part **CUTPLANE-Z2**) is shown below.

**Figure
4-494
CUTPLANE-
Z2 Mesh**

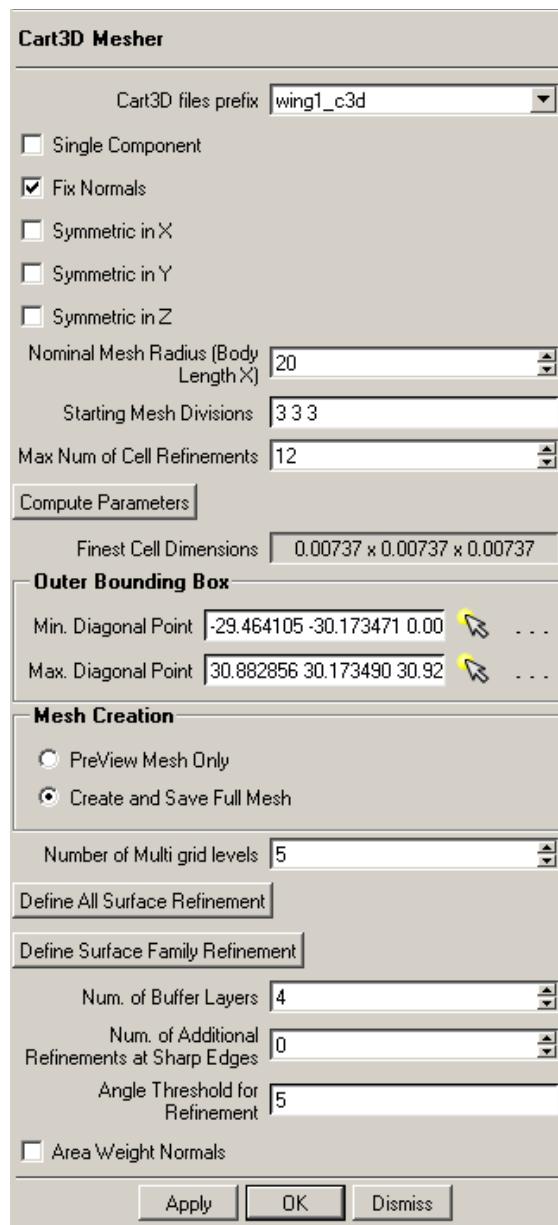


Perform the operation Parts > Show All by a right-click on Parts in the Display Tree after viewing the mesh.

c) Mesh Generation-Full Mesh

Now in the Cart3D mesher window enable 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.

Figure 4-495
Create and Save Full Mesh



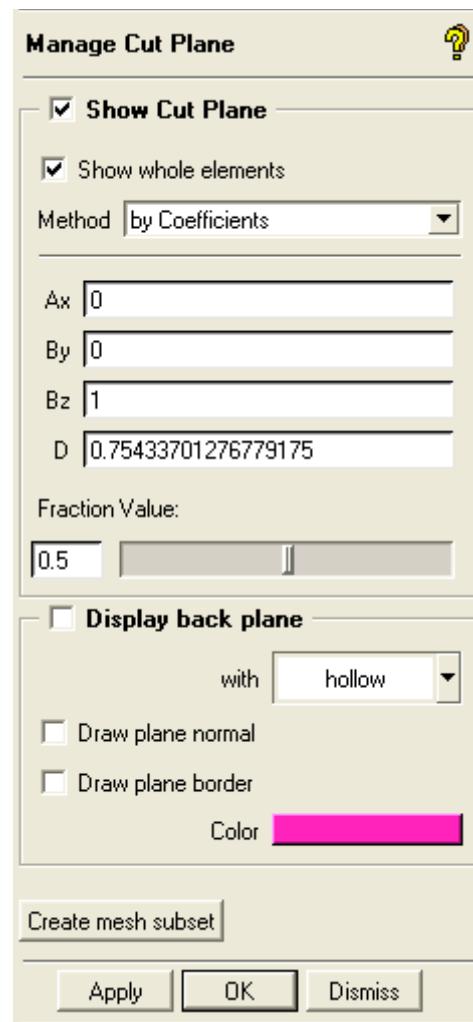
Press Apply. The Cart3D Mesh window appears which asks about loading the Cart3D Full Mesh as shown. Press Yes.

Figure 4-496
Cart3D Mesh
window



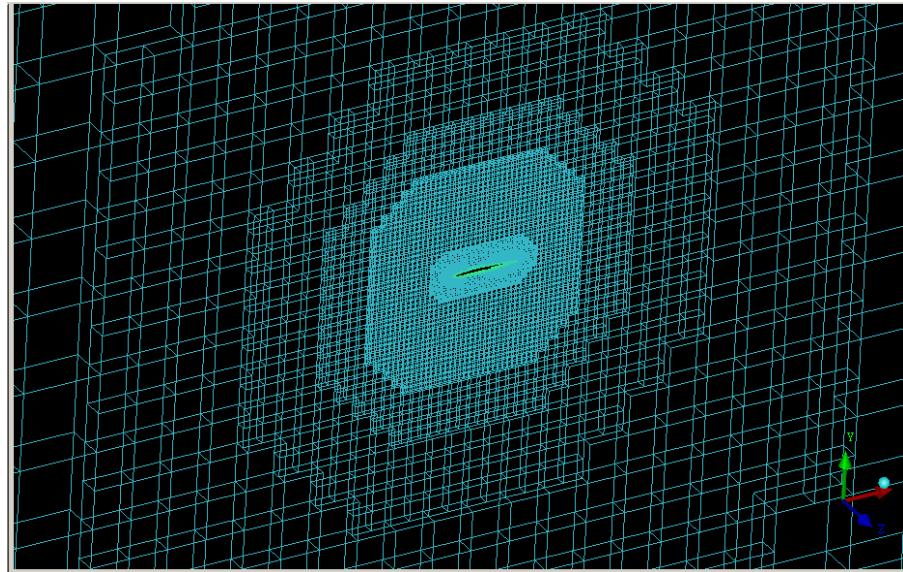
The final mesh generated can be examined through Mesh>Cutplane. The **Define Cut Planes** window appears as shown. Accept the default settings.

Figure 4-497
Define Cut Planes Window



The mesh cut plane using the above parameters is shown below.

**Figure
4-498
Cut
Plane
Mesh**

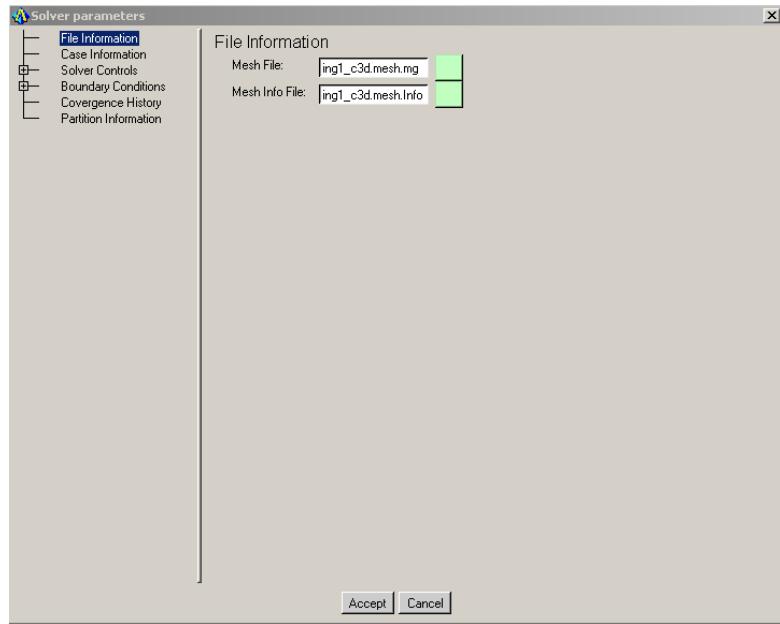


d) Setup Flow Cart Parameters

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.

**Figure
4-499
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

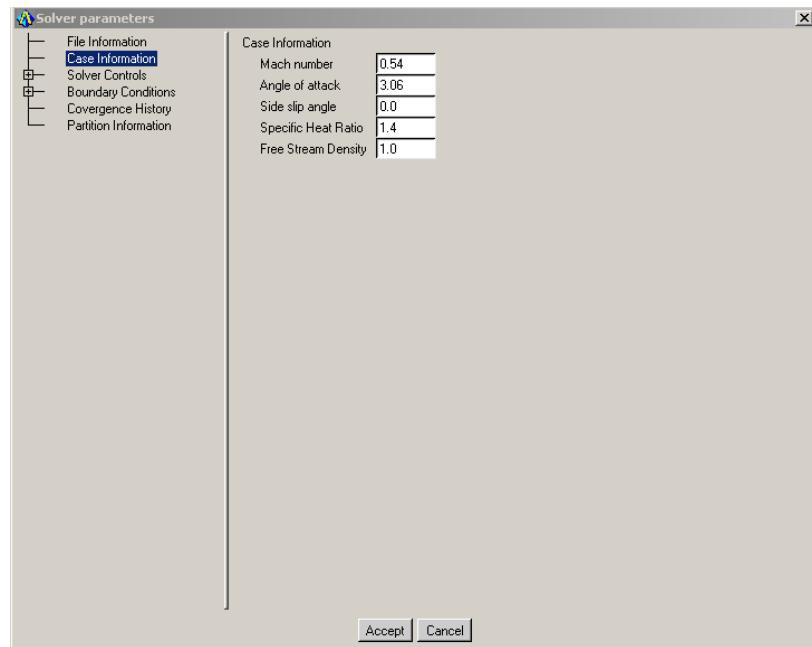
Angle of Attack = 3.06

Side Slip angle = 0.0

Specific Heat Ratio = 1.4

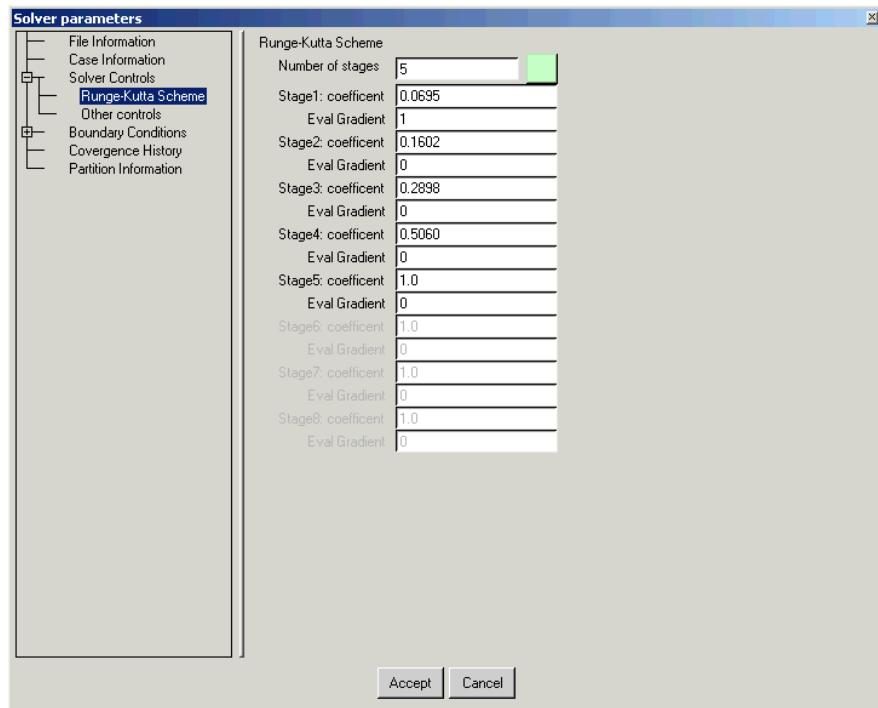
Free Stream Density = 1.0

**Figure
4-500
Case
Information
window**



Expand Solver Controls > Runge-Kutta Scheme in the Display Tree as shown and accept the default settings.

**Figure
4-501
Runge-
Kutta
Scheme
window**



In **Other controls** specify the following parameter values:

CFL number: 1.4

Limiter Type: van leer

Flux function: van Leer

Cut-Cell BCtype: Agglomerated Normals

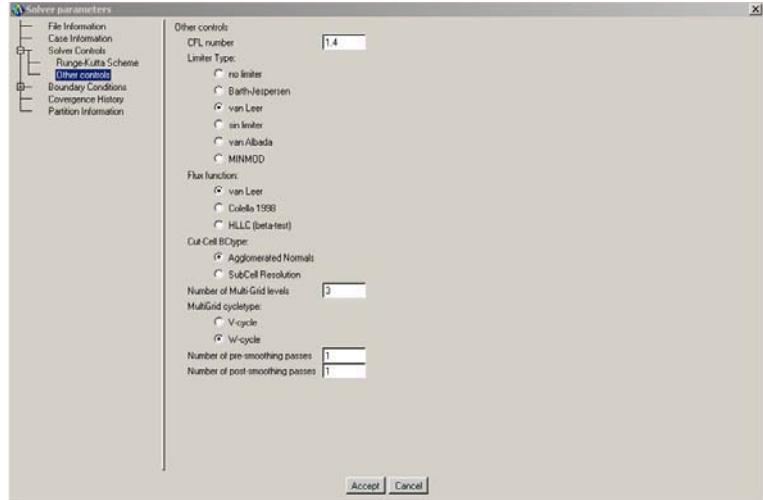
Number of Multi-Grid levels: 3

MultiGrid cycle type: W-cycle

Number of pre-smoothing passes: 1

Number of post-smoothing passes: 1

**Figure
4-502
Other
controls
window**

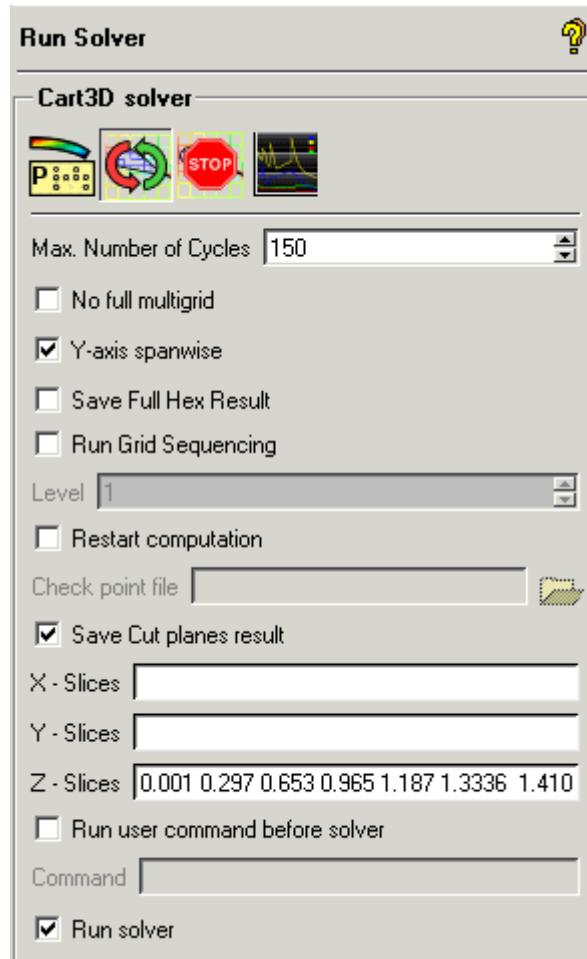


Keep defaults for Boundary Conditions, Convergence History, and Partition Information and press Accept.

e) Running the FlowCart Solver

Select Solver >Run Solver to open the flow chart solver panel. 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 4-503
Run Solver
window

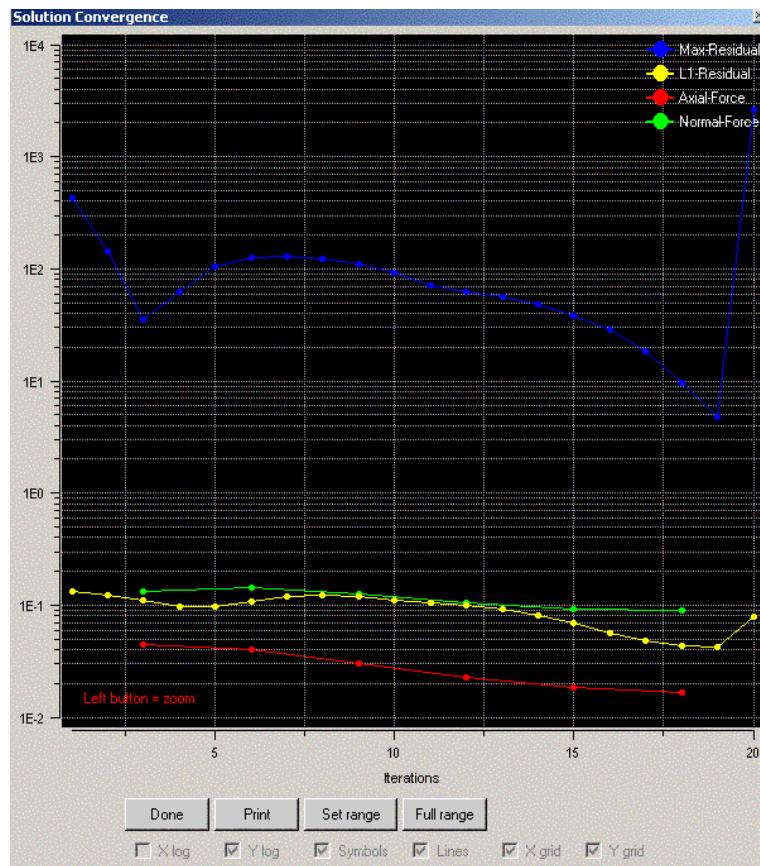


The user can view the convergence by selecting the Convergence monitor



icon as shown. (The monitor may open automatically.)

Figure 4-504
Solution
Convergence
Window

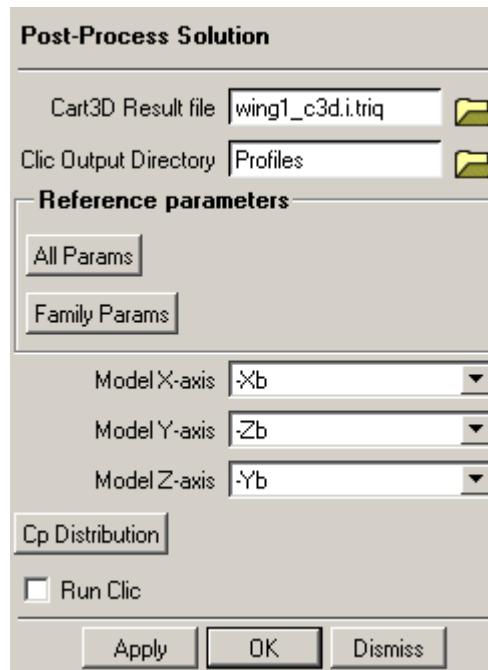


f) Computing Force and Moments



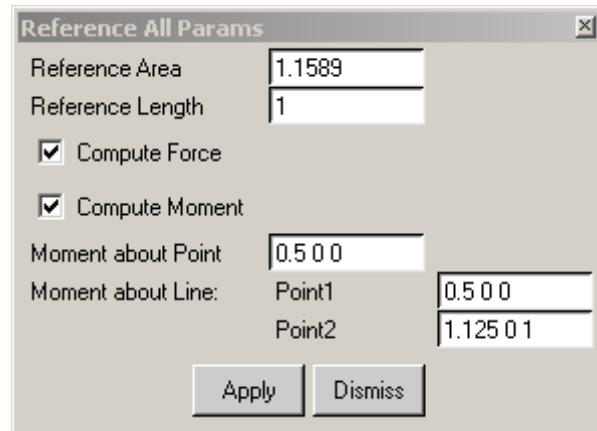
In the Cart3D main menu select Integrate Cp. The Post-Process Solution window will appear.

Figure 4-505
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** = 0.5 0 0, and **Point2** = 1.125
0 1.
Click **Apply** in the Reference All Params window and then **Dismiss** to close.

Figure 4-506
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:-

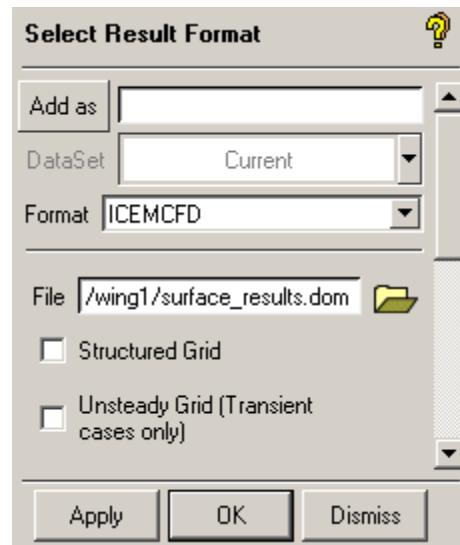
- i) **WING1_c3d.i.triq** - Contains Pressure, Velocity and Density extrapolated to the Surface triangles. This can be converted to a domain file by Edit>Cart3D Tri File->Domain file. The default resultant domain file will be WING1_c3d.uns.
- ii) **slicePlanes.dom** - Cut Plane results.
- iii) **results.dom** - Full mesh results.

Go to File > Results > Open Results.

Select Format as **ICEM CFD**.

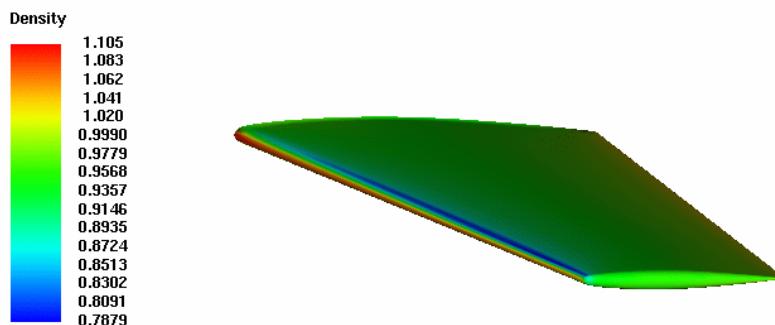
Specify **surface_results.dom** as the File.

Figure 4-507
Select Result File Window

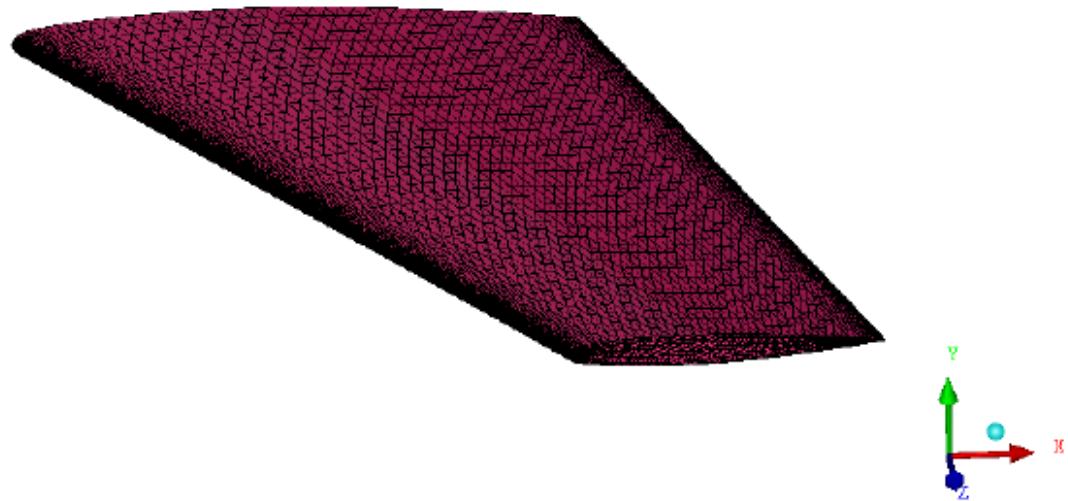


Select **Apply** from the panel to get the default result as shown. Right click on Colormap from the Display Tree and select Modify Entries to adjust the Min and Max values for the displayed variable.

Figure 4-508
**Visualization
of Results**



4.7.3: 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:-

1. Use of the Cart3D mesher for mesh generation.
2. Multi grid preparation - running mgPrep.
3. Running the solver for AOA=3.06 and Mach=0.84.
4. Computing force and moment information.
5. Visualizing the result in the post processor.

a) Starting the Project

The input files for this tutorial can be found in the Ansys Installation directory, under `../docu/Tutorials/CFD_Tutorial_Files/Cart3D_Examples`. Note: It is preferable to create a separate folder **wing1** and put the **oneraM6.uns** (domain) file in that folder before performing this tutorial.

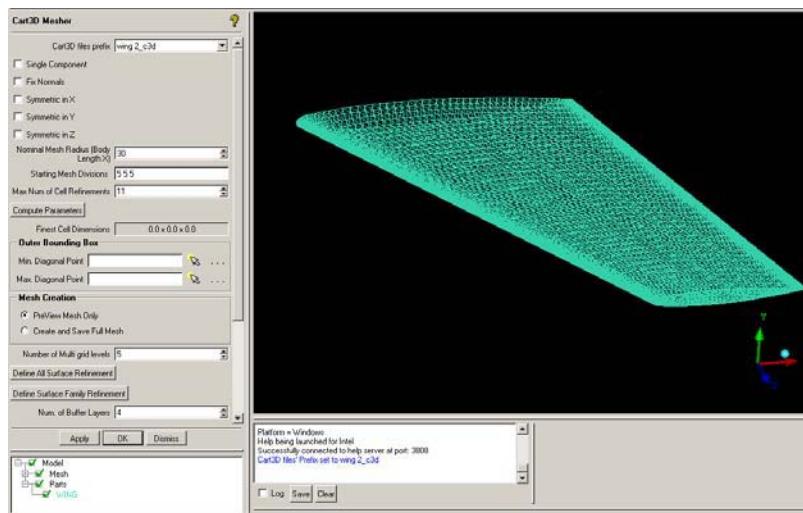
ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	588
------------------------	--	-----

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 below.

**Figure
4-509
Cart 3D
Mesher
window**



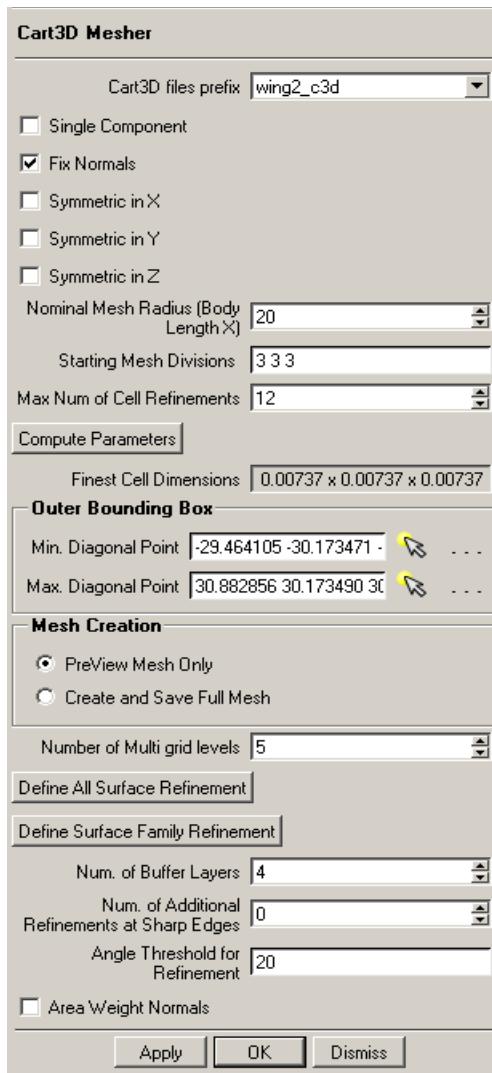
Toggle ON Fix Normals. This will fix the orientation of the triangles such that their normals are pointing outward.

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 it 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.

Cart3D

Figure 4-510
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.

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.

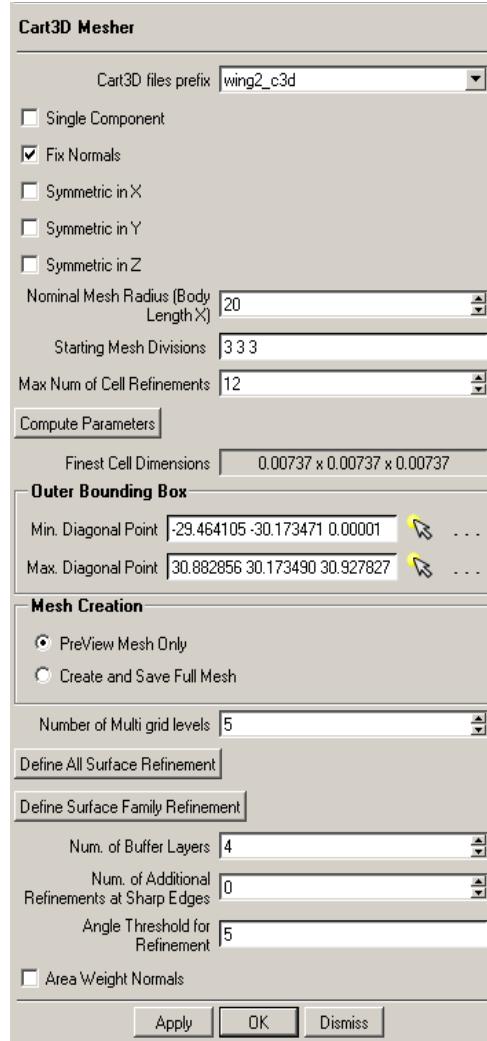
The diagonal points displayed under the ‘Outer Bonding Box’ are the maximum and minimum points of the bounding box of the Mesh region. They can be changed if desired.

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). If the model itself is symmetric, turn on Half-Body Mesh (Symmetric in Z).

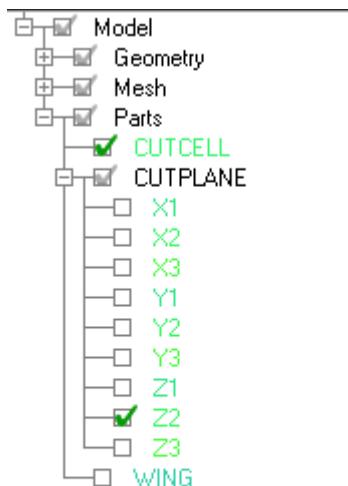
Click Apply (after specifying minimum Z coordinates as 0.00001) as shown to run the mesher. This will create a domain file with 3 Cut Planes (Quad Elements) in each coordinate direction and Cut Cells (Hex Elements). The Preview Mesh will be loaded automatically.

Figure 4-511
Change Angle of
Refinement



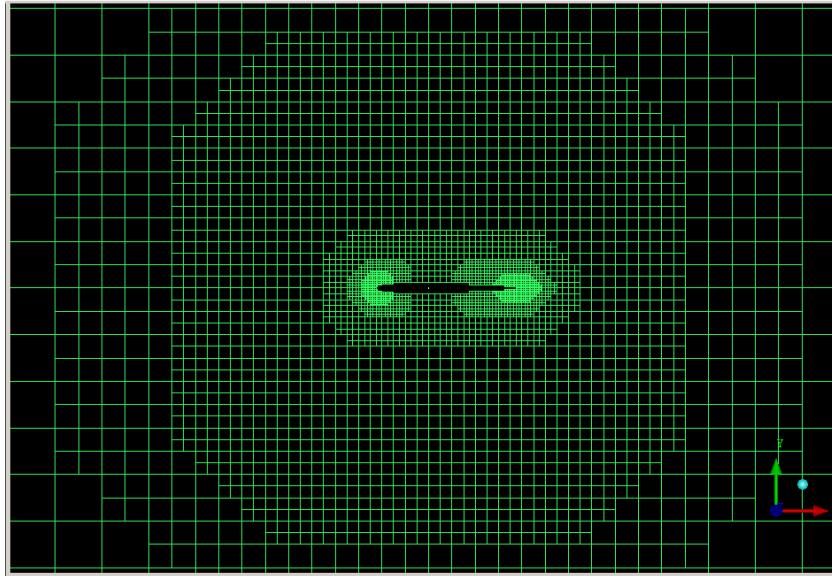
In the Parts menu under the Display Tree perform the operation Parts > Hide All (right-click on Parts to access) and then turn on only the Part **CUTPLANE-Z2**.

Figure 4-512
Display Tree



The mesh projected onto the middle z-direction plane (in Part **CUTPLANE-Z2**) is shown below.

**Figure
4-513
CUTPLANE-
Z2 Mesh**

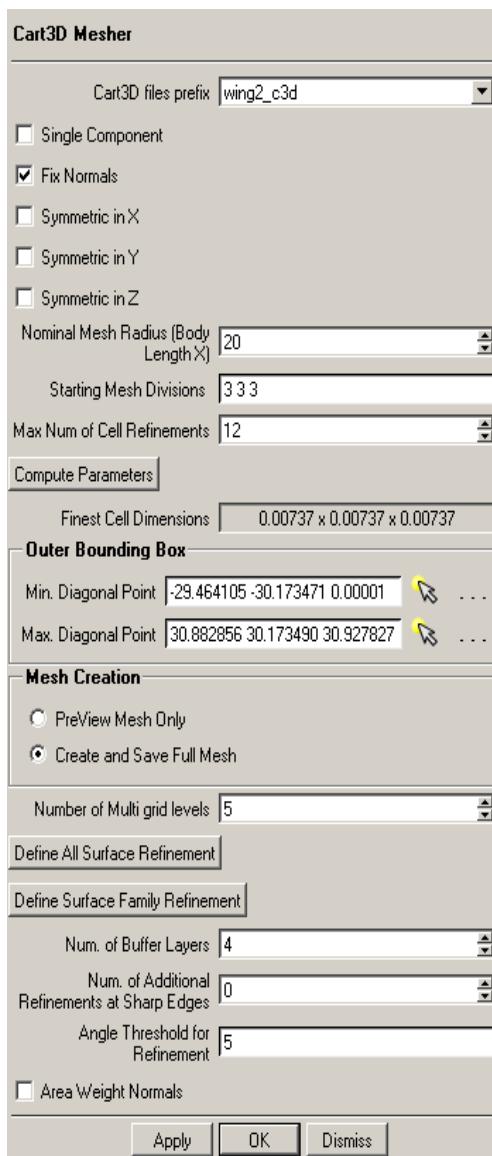


Perform the operation Parts > Show All by a right-click on Parts in the Display Tree after viewing the mesh.

c) Mesh Generation-Full Mesh

Now in the Cart3D mesher window enable 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.

Figure 4-514
Create and Save Full Mesh



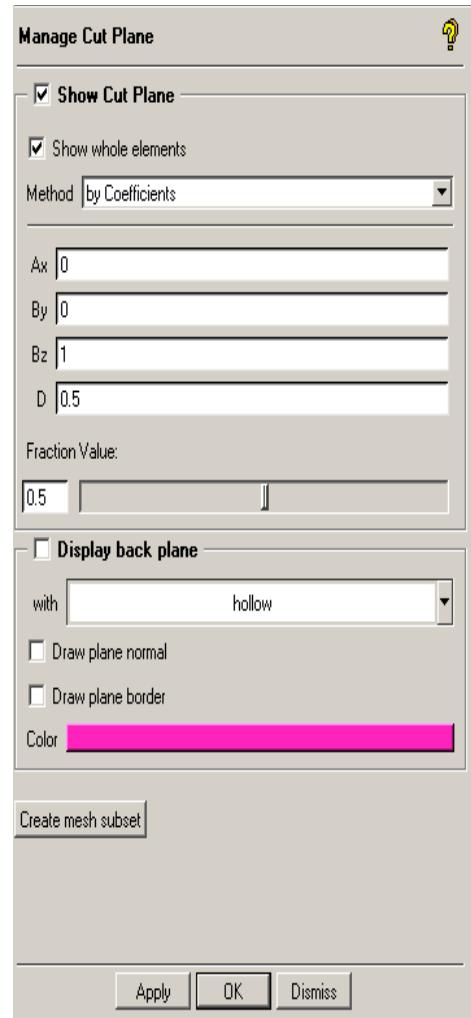
Press Apply. The Cart3D Mesh window appears which asks about loading the Cart3D Full Mesh as shown. Press Yes.

Figure 4-515
Cart3D Mesh window



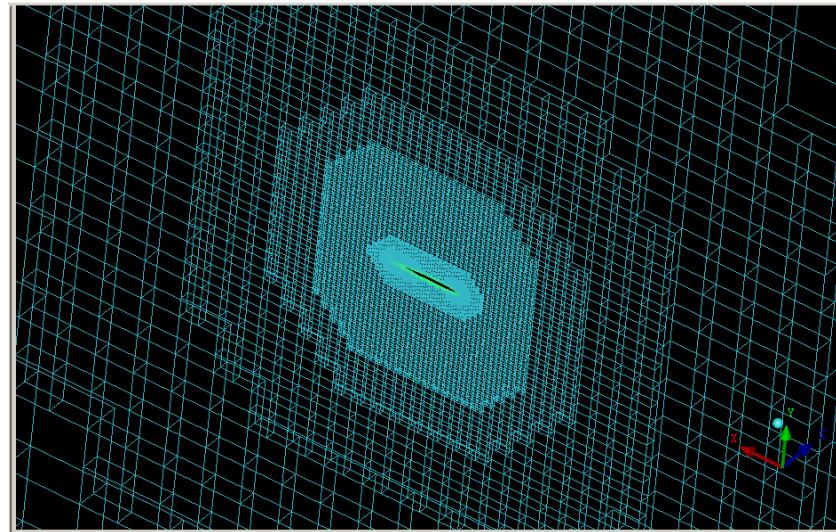
The final mesh generated can be examined through Mesh > Cutplane. The **Define Cut Planes** window appears as shown. Accept the default settings.

Figure 4-516
Define Cut Planes Window



The mesh cut plane using the above parameters is shown below.

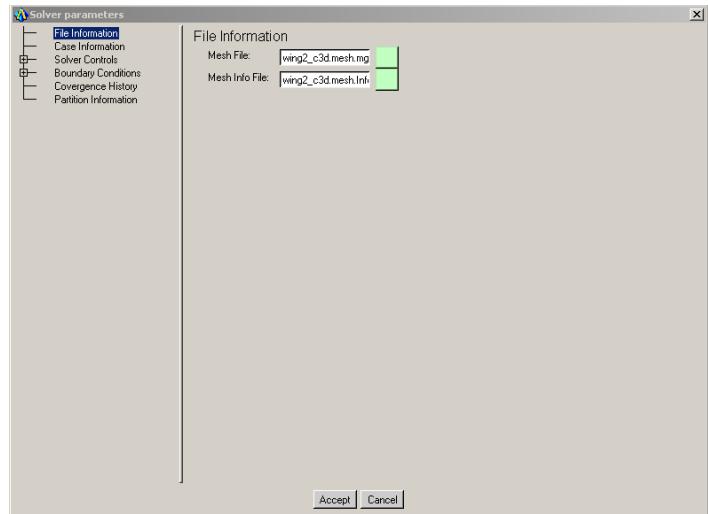
**Figure
4-517
Cut
Plane
Mesh**



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.

**Figure 4-518
Solver
parameters
window**

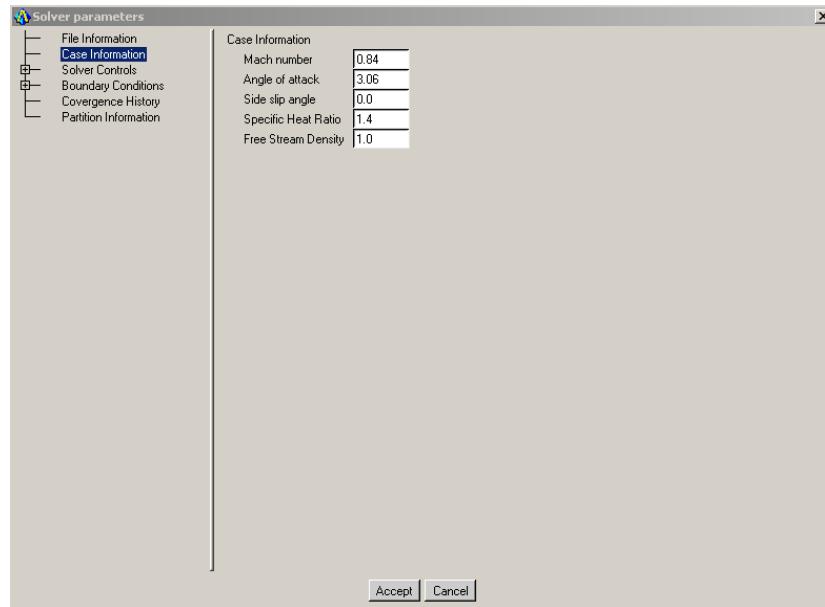


Set File Information > Mesh File as **WING2_c3d.mesh.mg** (should be default).

Click on Case Information window and enter the following parameters

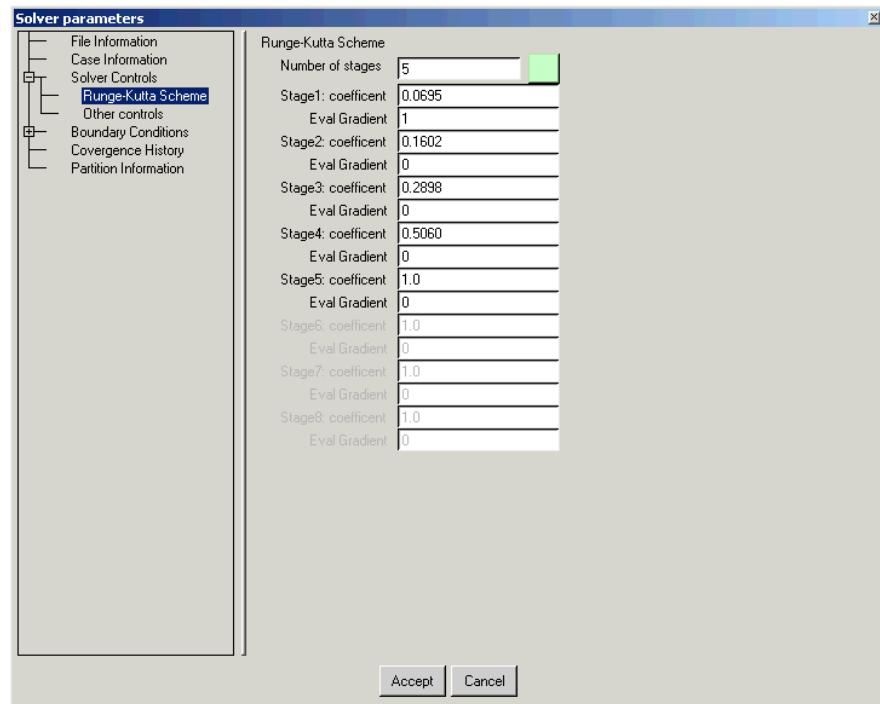
Mach Number = 0.84
Angle of Attack = 3.06
Side Slip angle = 0.0
Specific Heat Ratio = 1.0
Free Stream Density = 1.0

**Figure
4-519
Case
Information
window**



Expand Solver Controls > Runge-Kutta Scheme in the Display Tree as shown and accept the default settings.

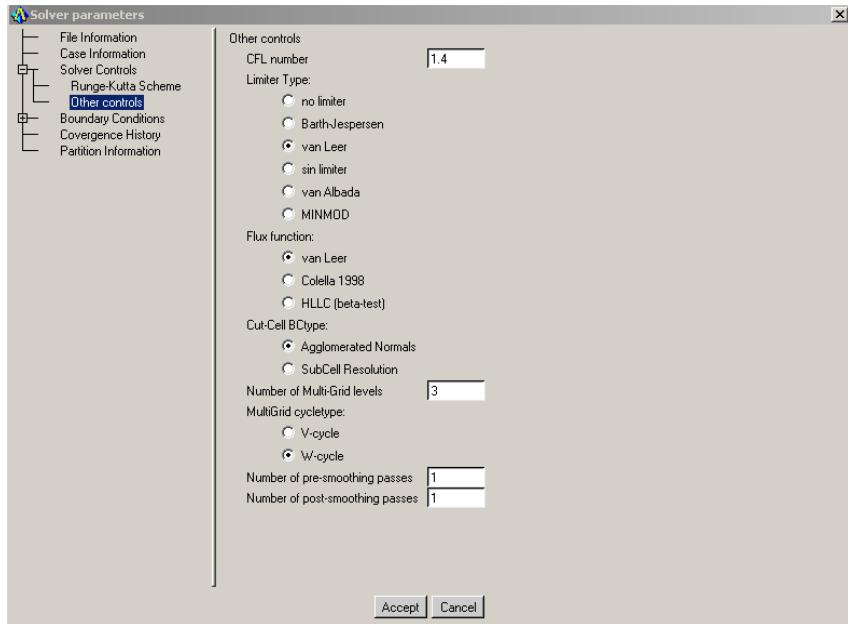
**Figure
4-520
Runge-
Kutta
Scheme
window**



5. In **Other controls** specify the following parameter values:

CFL number:	1.4
Limiter Type:	van Leer
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
4-521
Other
Control
window**

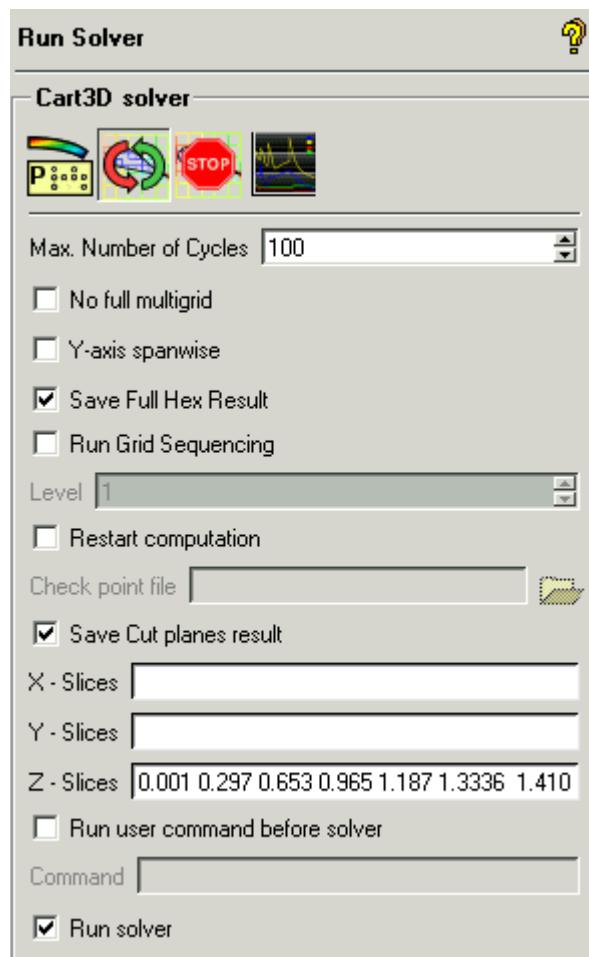


Keep defaults for Boundary Conditions, Convergence History, and Partition Information and press Accept.

e) Running the FlowCart Solver

Select Solver >Run Solver to open the flow chart solver panel. Specify Max. Number of Cycle=150. Turn on Save Full Hexa 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 and run the solver.

Figure 4-522
Run Solver
window

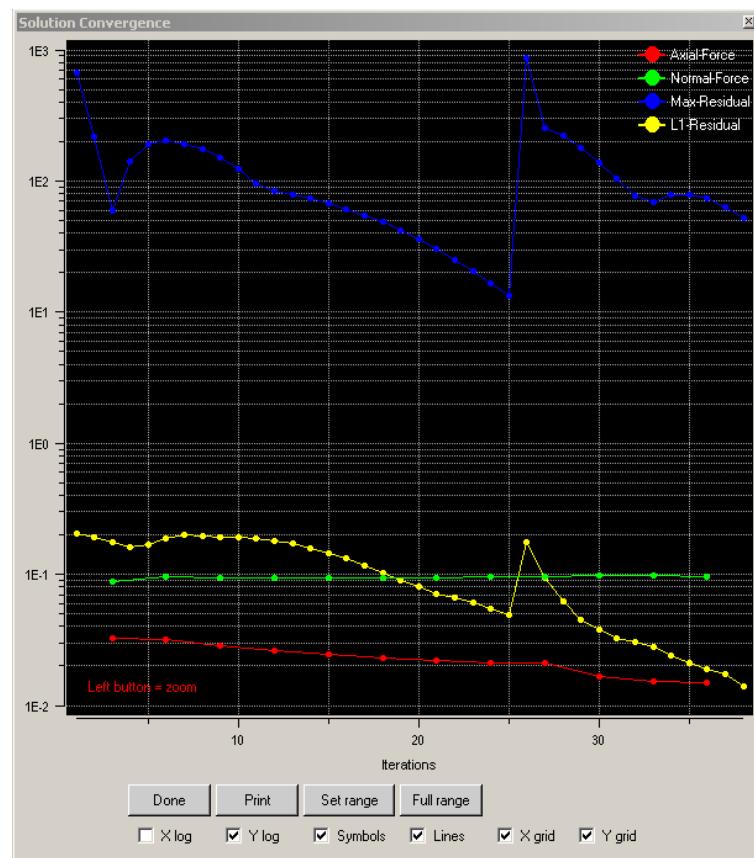


6. The user can view the convergence via the Convergence Monitor icon



(The monitor may open automatically.)

Figure 4-523
Solution
Convergence
Window

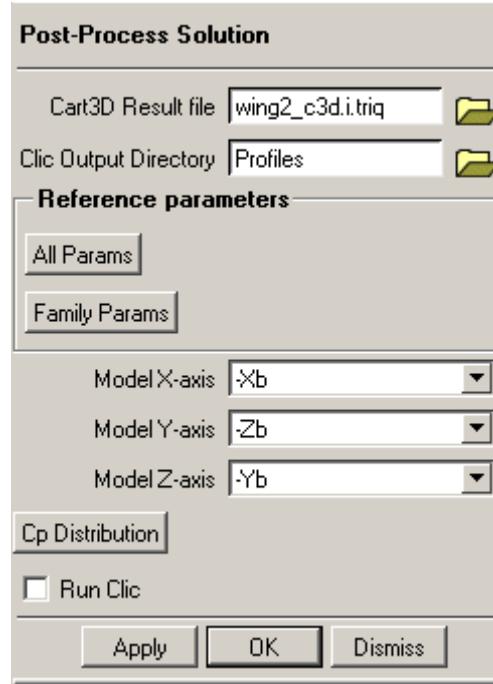


f) Computing Force and Moments

In the Cart3D main menu select Integrate Cp. The Post-Process Solution window appears as shown.

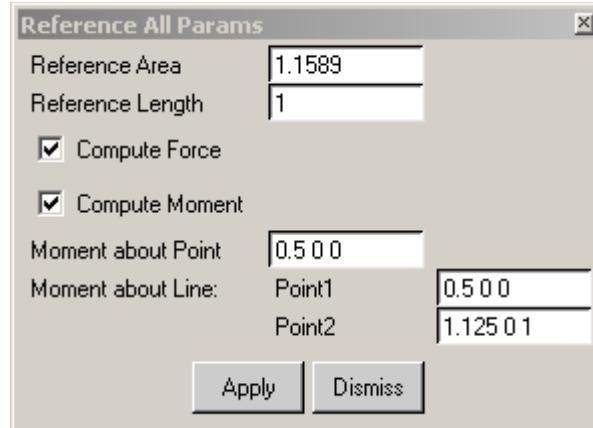


Figure 4-524
Post-Process
Solution window



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** = 0.5 0 0, and **Point2** = 1.125 0 1. Click **Apply** in the Reference All Params window and then **Dismiss** to close.

Figure 4-525
Reference All Params window



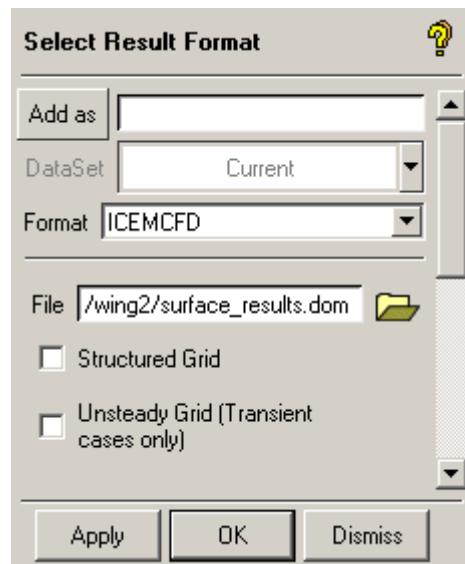
Press Apply in the Post-Process Solution window. The results appear in the GUI messages area.

g) Visualizing the results

Flow charts write three output files:

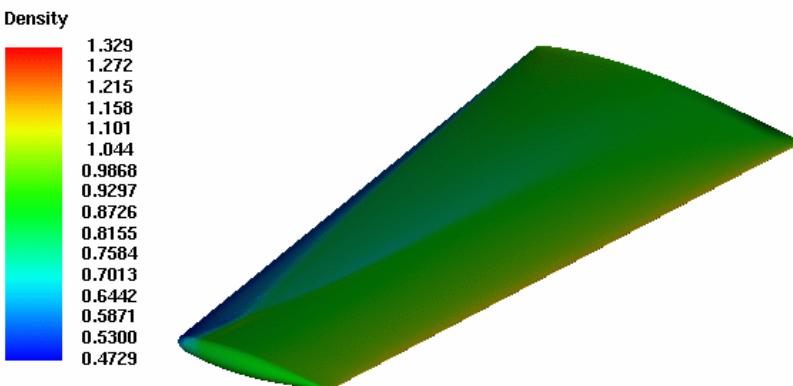
- i) **WING2_c3d.i.triq** - Contains Pressure, Velocity and Density extrapolated to the Surface triangles. This can be converted to a domain file by Edit>Cart3D Tri File->Domain file. The default resultant domain file will be WING2_c3d.uns
 - ii) **slicePlanes.dom** - Cut Plane results
 - iii) **results.dom** - Full mesh result
- Go to File > Results > Open Results. Select Format as **ICEM CFD** and specify **surface_results.dom** as the File.

Figure 4-526
Result File Format Window



Select Apply from the panel to get the default result as shown below.
Right click on Color map from the Display Tree and select Modify Entries to adjust the Min and Max values for the displayed variable.

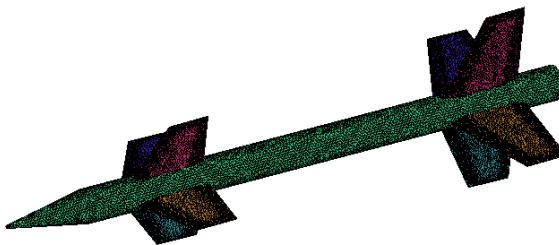
**Figure
4-527
Post
Processing
Result**



4.7.4: 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

The input files for this tutorial can be found in the Ansys Installation directory, under `../docu/Tutorials/CFD_Tutorial_Files/Cart3D_Examples`. 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.

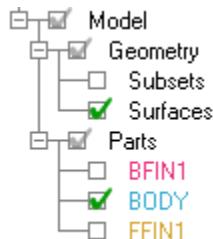
Note: To skip the surface triangle generation process, proceed to the Mesh Generation Preview Only section. Also put the `missile.uns` (domain) file into the **missile** directory.

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.

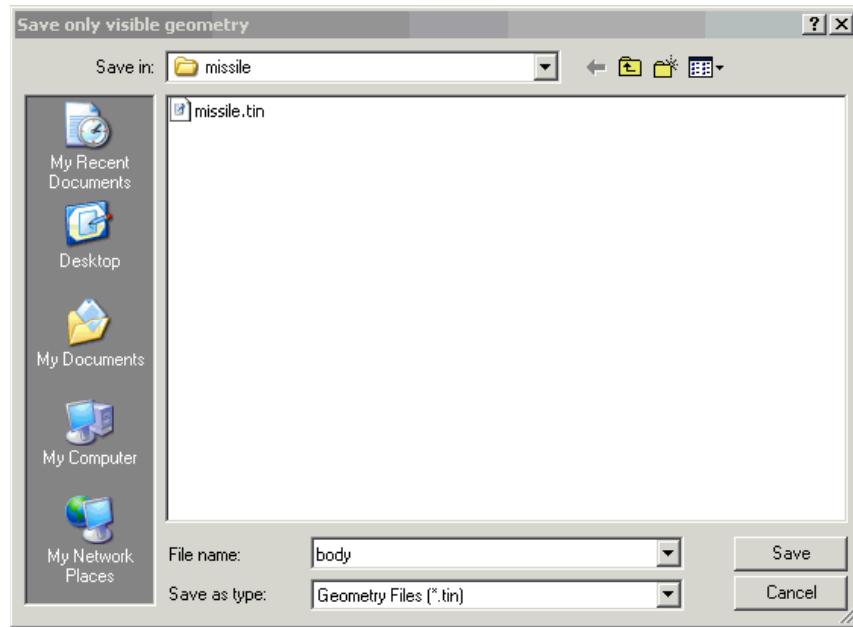
Figure 4-528
Display Tree



Save only the visible geometry to a new tetin file using File > Geometry > Save Visible Geometry As... Specify the file name as **body.tin** and save the file.

Note: Don't save it as `missile.tin` as we will lose the rest of the geometry data.

**Figure
4-529
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.

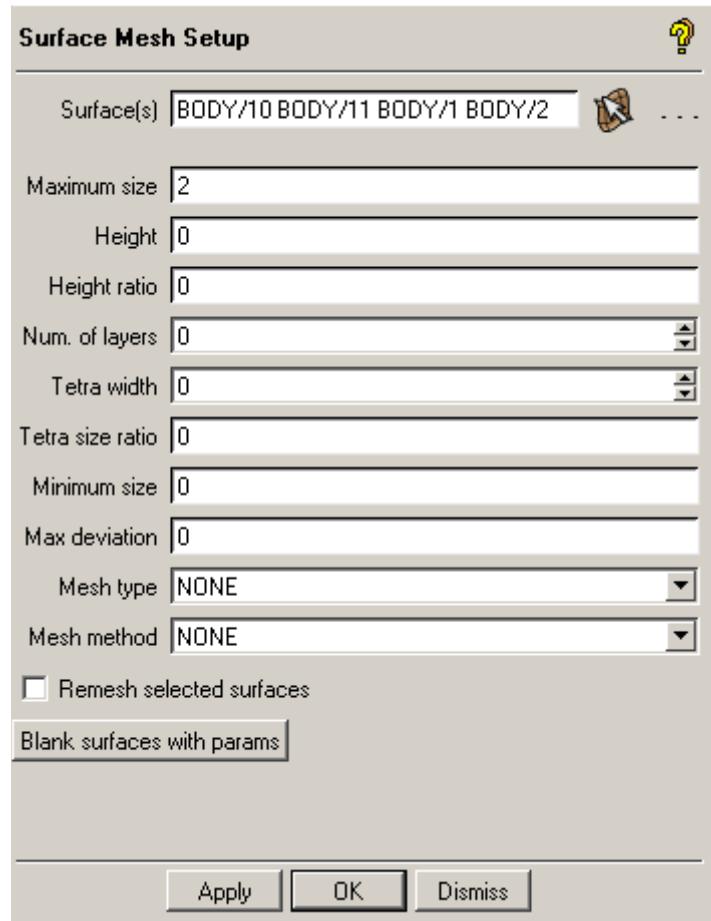
6. Now close the current tetin file using File > Geometry > Close Geometry.

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.

Figure 4-530
Surface Window



Click the Select surface(s) icon and select the hemispherical surface at the tip (see figure) with the left mouse button. Middle-click to accept the selection. Specify a **Maximum size** of 0.25.

Figure 4-531
Surface
Selected

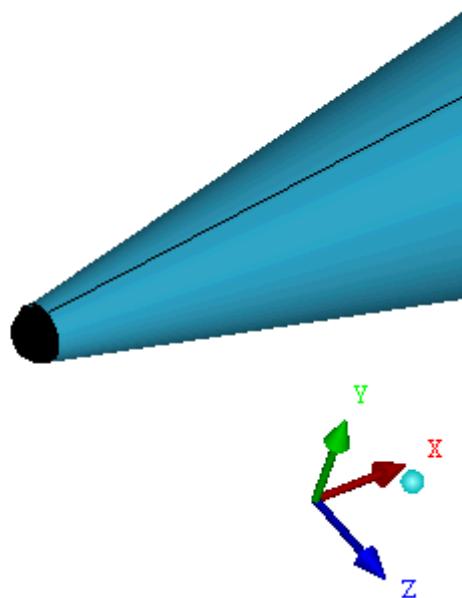
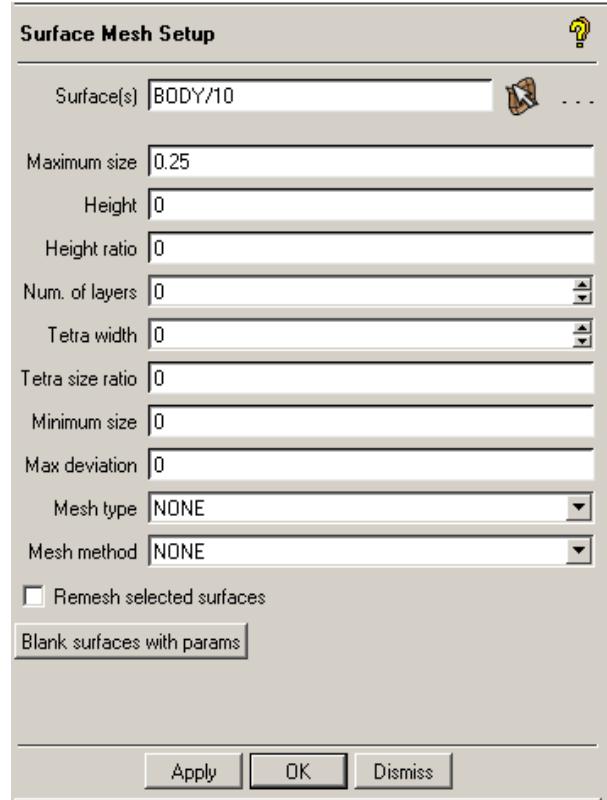
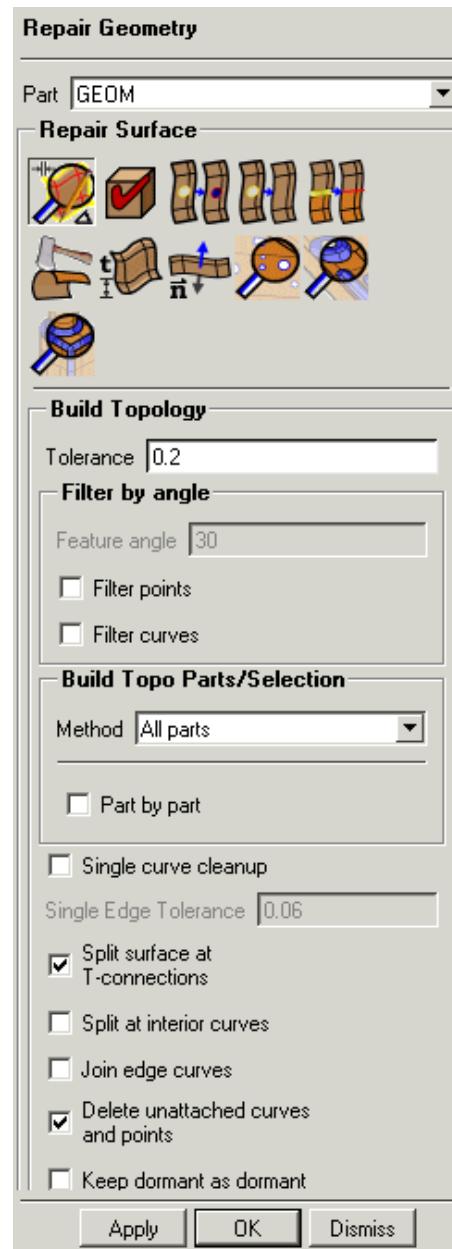


Figure 4-532
Surface Mesh
Size window



Extract the hard curves and points on the geometry using Build Diagnostic Topology. Select Geometry > Repair Geometry  > Build Topology. Use the defaults and press Apply.

Figure 4-533
Build Topology Window

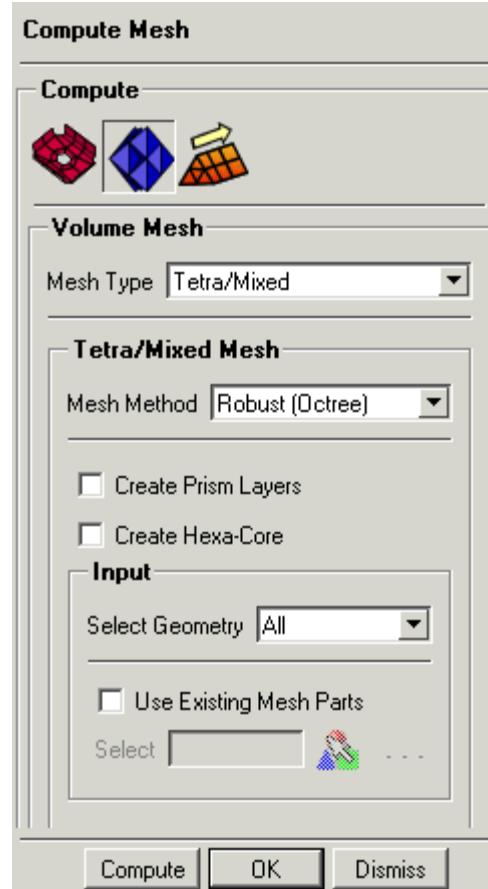


Save the tetin file using File > Geometry > Save Geometry.

Note: By default it saves the geometry file to body.tin.

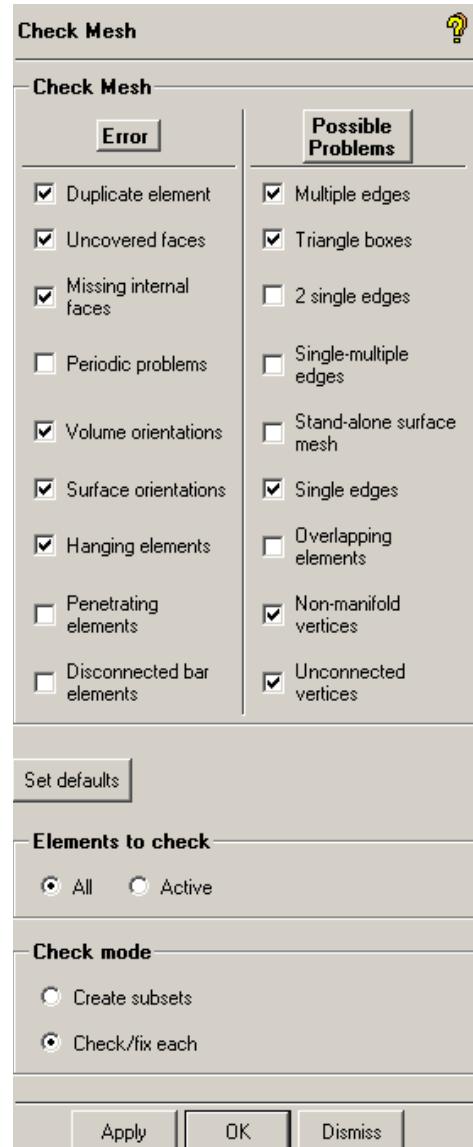
Run Tetra from Mesh > Compute Mesh > Volume Mesh . Select Mesh Type as **Tetra/Mixed** and Mesh Method as **Robust(Octree)**. Accept the default settings and press **Compute**.

Figure 4-534
Mesh Volume window



From the Edit Mesh tab select Check Mesh.  Accept the default settings as shown below and press Apply.

Figure 4-535
Check Mesh window



In the Diagnostic window it asks to Delete the unconnected vertices. Press Yes.

Expand the Mesh branch in the Display Tree. 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, 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.

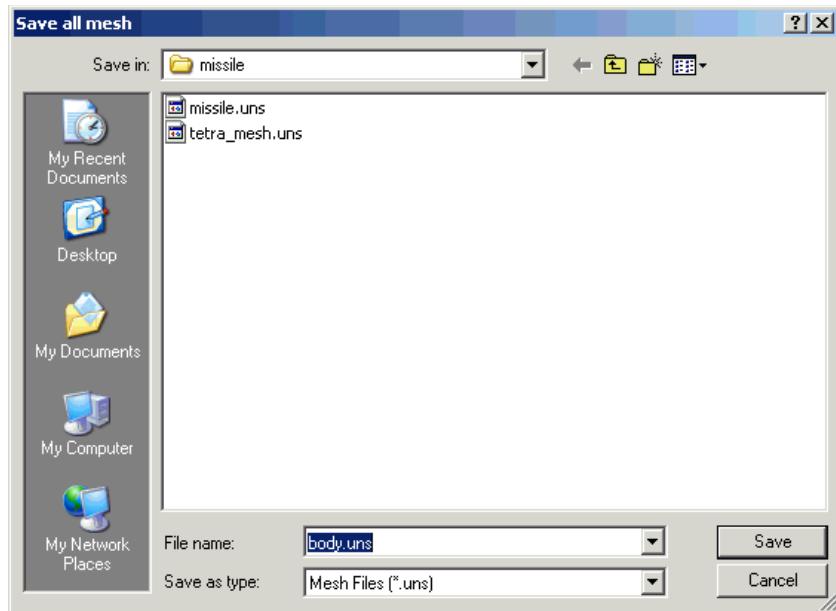
Activate Shells from the Display Tree, and de-select Shells>Face Normals.

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 and press Save.

Note: User should only use the Save Mesh As option

**Figure
4-536
Save
All
Mesh
window**



Select File > Geometry > Close Geometry and File > Mesh > Close Mesh.

d) Generating Surface Mesh on Front and Back Fins

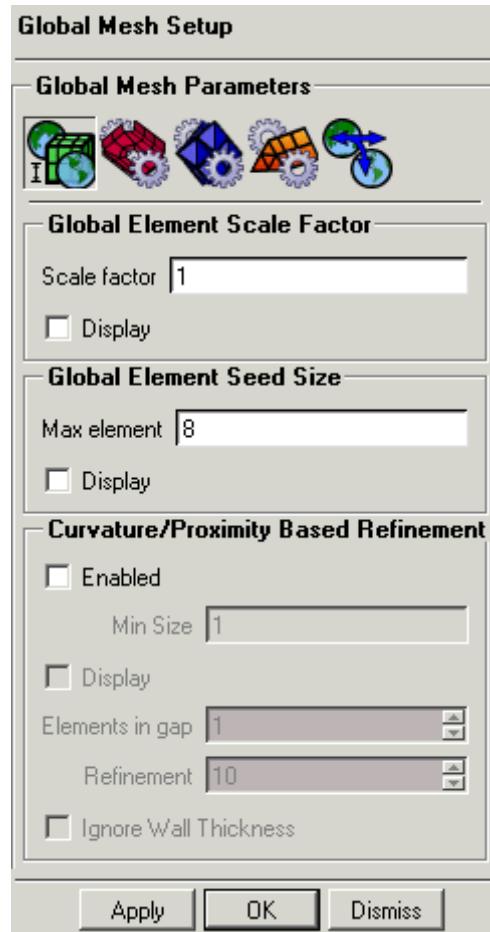
Load the geometry file ffin.tin via File > Geometry > Open Geometry, and select ffin.tin.

Repair the Geometry (Build Diagnostic Topology) in the same way as for body.tin. 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 Global Mesh Setup and Global

Mesh Size. Enter Max element as 8 and press Apply.

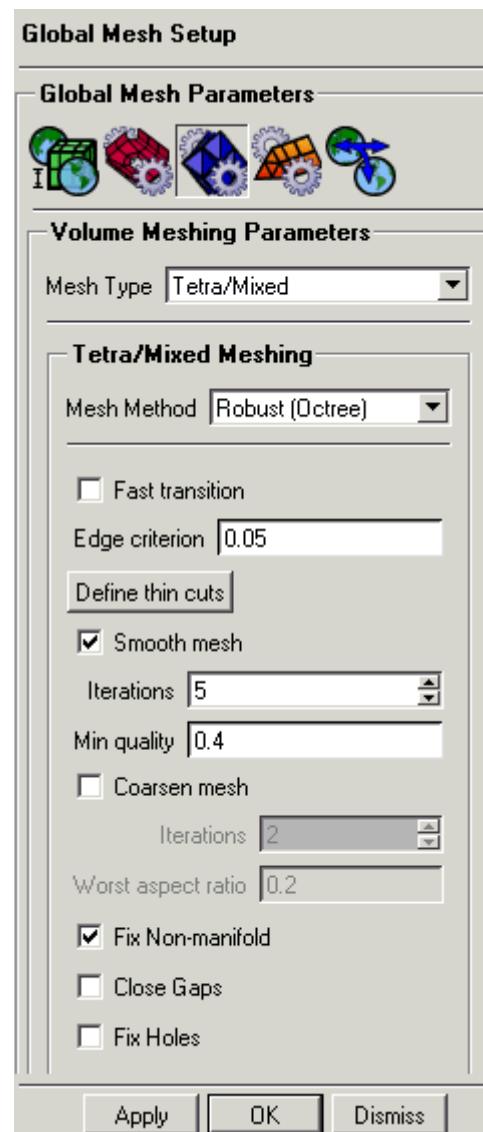
Figure 4-537: Global mesh size window



Select Global Mesh Setup>Volume Meshing Parameters> Tetra Meshing

Parameters  and set Edge criterion to **0.05** as shown below. Press Apply.

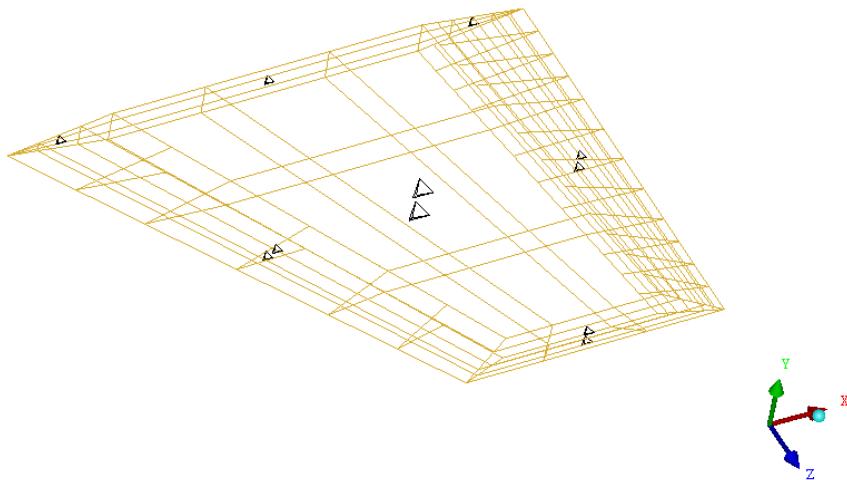
Figure 4-538
Global Mesh Size:Tetra
meshing parameters
window



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 and select Tetra Sizes.

**Figure
4-539
Tetra
Size**



Select Compute Mesh > Compute Volume Mesh > Mesh Type Tetra/Mixed, Rest all Parameters Default, Press ‘Compute’ to generate the mesh. 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 save the surface triangular mesh as **bfin.uns**. Close the geometry and mesh files.

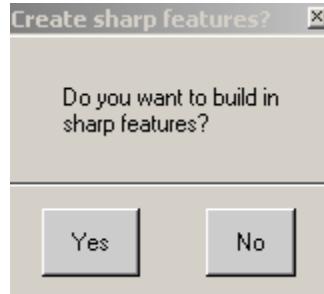
e) Working on the entire display

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.

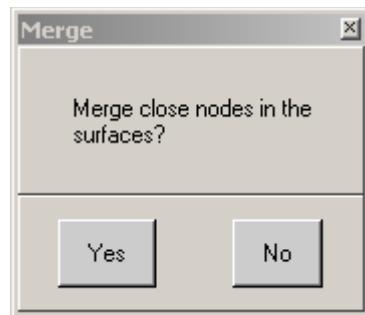
Select Edit > Mesh > Facets in the main menu. The Create sharp features window appears as shown below. Press No.

Figure 4-540
Create Sharp Features
window



Then the **Merge** window appears asking whether to Merge close nodes in the surfaces. Press No.

Figure 4-541
Merge window



In the Display Tree switch off Mesh and switch on Surfaces > Solid. 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 .

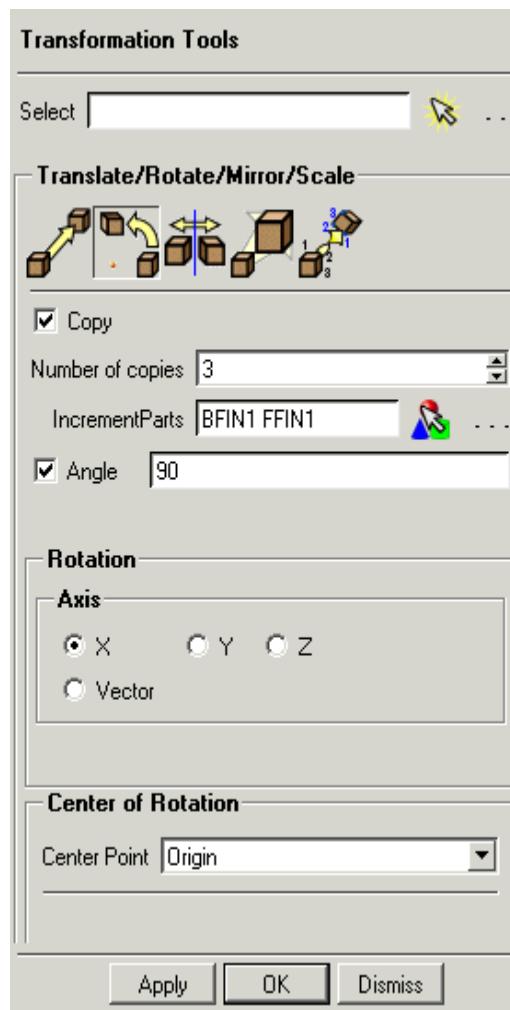
In the window press Select entities. 

In the Select geometry toolbar press Select items in a part . The Select part window appears as shown below. Select BFIN1 and FFIN1 and press Accept.



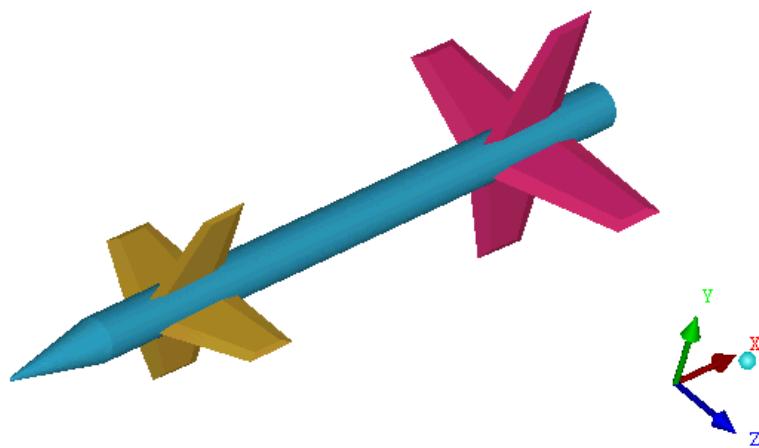
In the **Transformation Tools** window enable **Copy**; enter Number of copies as 3, select X for Axis, Angle = 90 and Center Point as Origin as shown in the figure below. Then press Apply.

Figure 4-543
Transformation Tool window



The geometry of BFIN1 and FFIN1 gets rotated as displayed in the figure below.

**Figure
4-544
After
Rotation.**



Switch off BODY and BFIN1 from the Display Tree 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. Select Create Part by Selection,  and select the region as shown below. Middle-click to accept.

**Figure
4-545
Region
selected**

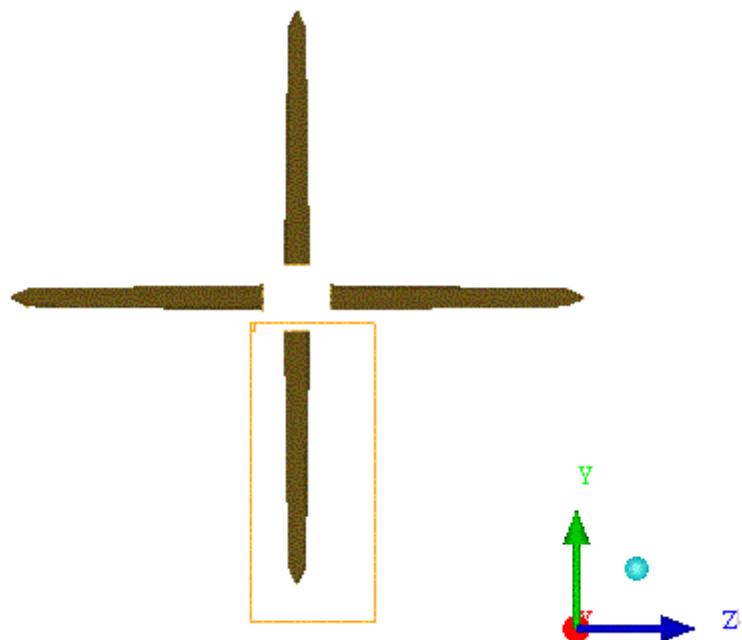


Figure 4-546
Create Part window

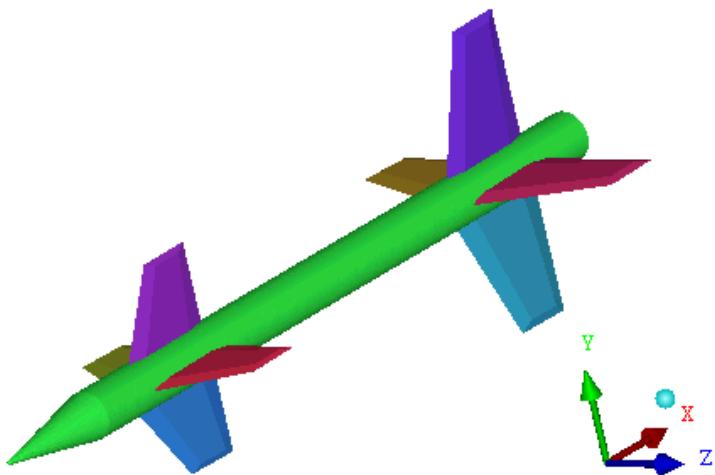


Similarly move each fin into a different Part with the front fins in FFIN1, FFIN2, FFIN3, and FFIN4; and the back fins in BFIN1, BFIN2, BFIN3, and BFIN4.

Note: It is better to keep each component in separate Parts.

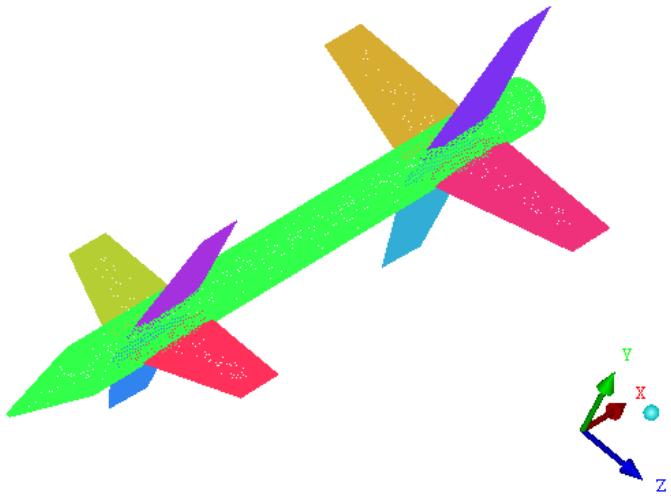
The final geometry image, which we get after the Part assignments, is shown here.

**Figure
4-547
After Part
Assignment**



From the main menu select Edit > Facets > Mesh that would give us the desired mesh displayed here.

**Figure
4-548
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.

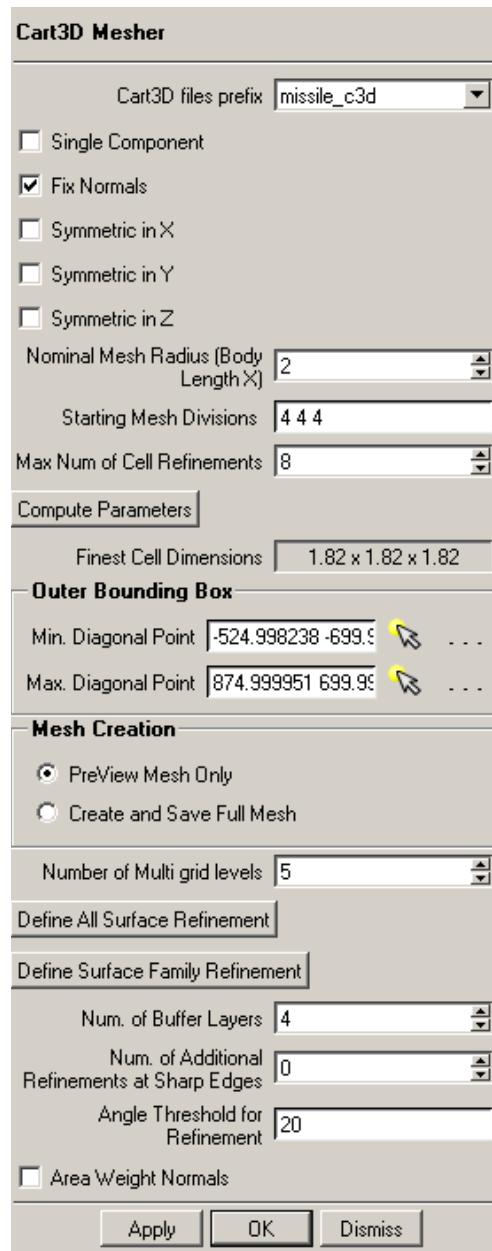


Click on Cart3D from the main menu. Select the Volume Mesher Icon.

Toggle ‘ON’ Fix Normals 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.

Figure 4-549
Cart3D Mesher
window



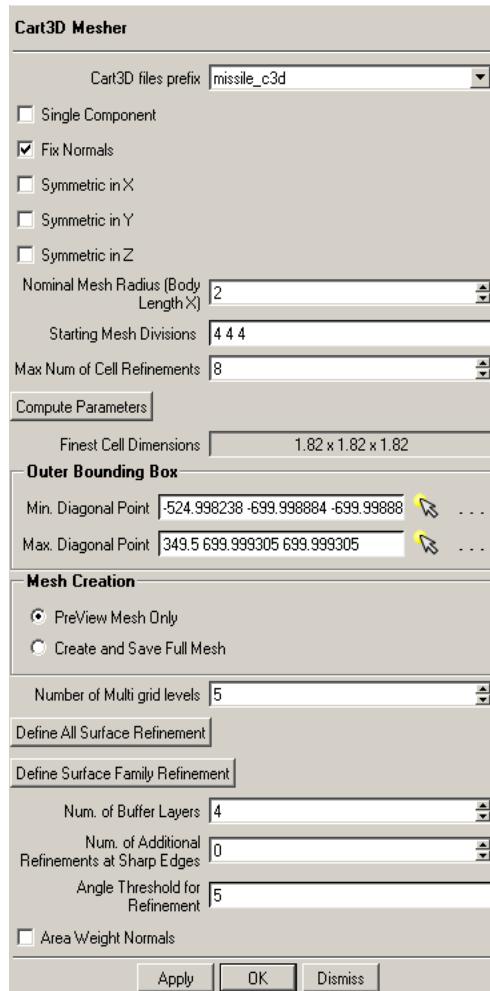
This will create 10 density polygons for mesh density control, which can be viewed in the Display Tree by activating Geometry >Densities.

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. 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 below.

Figure 4-550
Angle of refinement changed



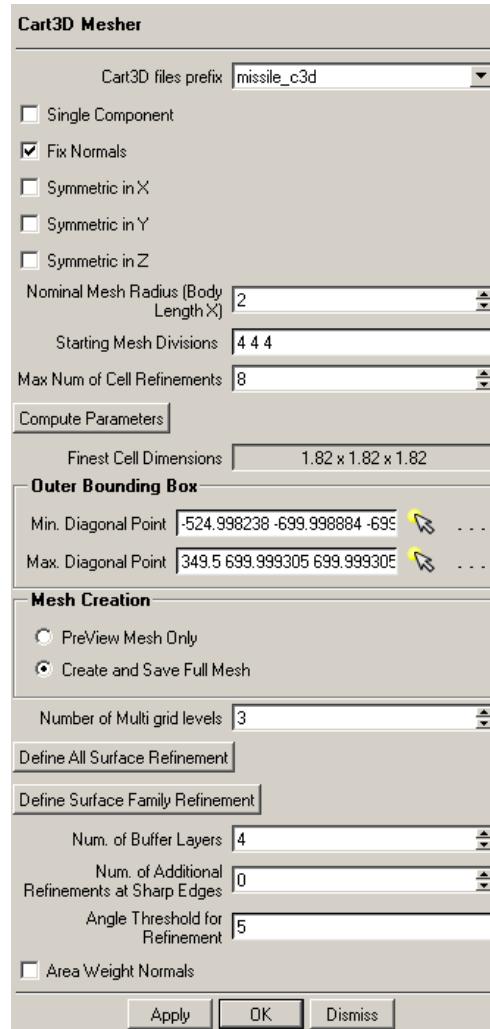
Click **Apply** to run the mesher. This will create a domain file with 3 Cut Planes (Quad Elements) in each coordinate direction and Cut Cells (Hex Elements). The Preview mesh will be loaded automatically.

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 below. This will create 3 levels of coarsened mesh, which can be read by the solver.

Figure 4-551
Create and Save Full
View.



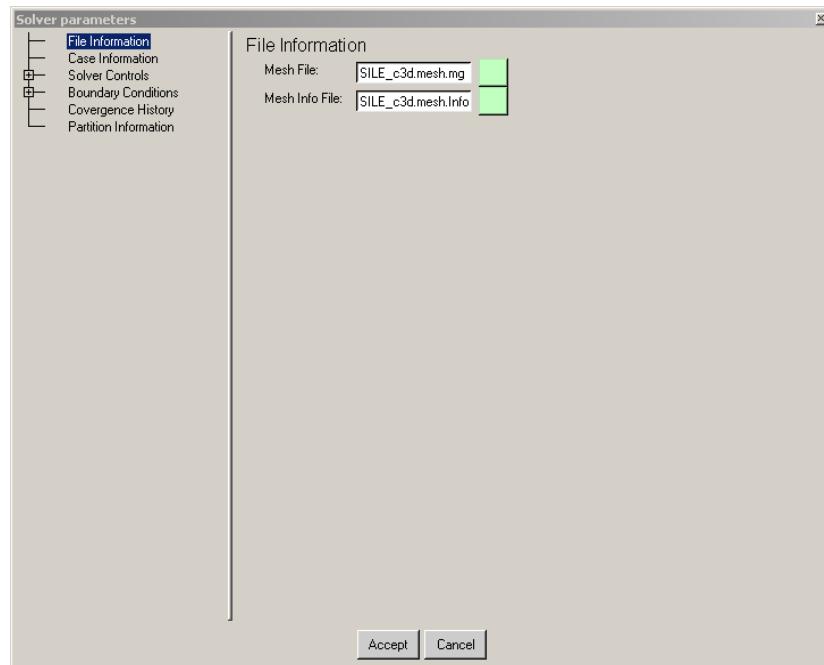
Press Apply. The Cart3D Mesh window appears which asks us to load the cart3D Full Mesh. Press **Yes**.

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 below.

**Figure
4-552
Solver
Parameter
window**

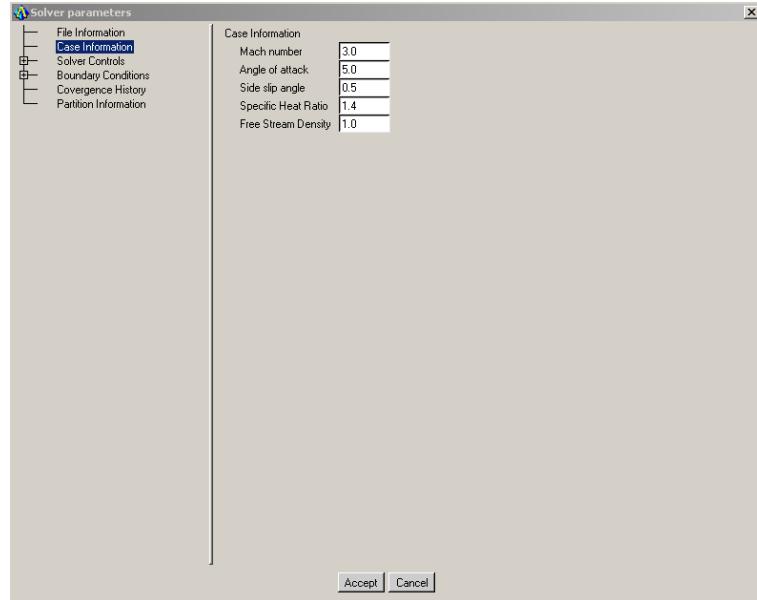


Choose File Information > Mesh File as **MISSILE_c3d.mesh.mg** (this should be the default).

Click on Case Information and enter the parameter values:
Mach number = 3

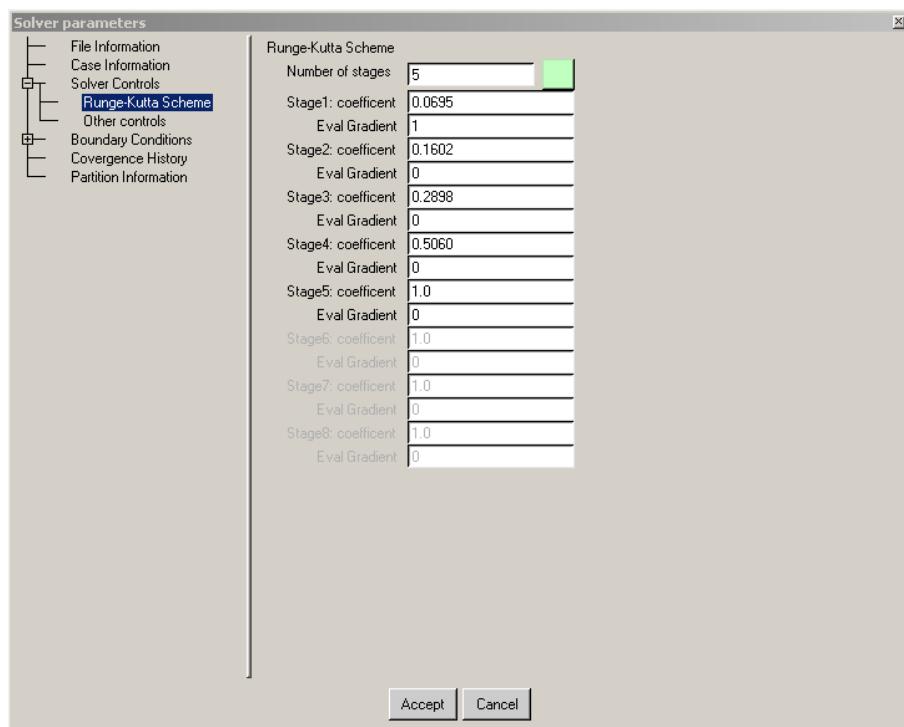
Angle of attack = 5
Side slip angle = 0.5
Specific Heat Ratio=1.4
Free Stream Density = 1.0

Figure 4-553
Case
Information
window



Expand Solver Controls > Runge-Kutta Scheme and evaluate the coefficient only at the first stage.

**Figure
4-554
Runge
Kutta
Scheme
window**



In other controls specify the following parameter values:

CFL number: 1.2

Limiter Type: MINMOD

Flux function: van Leer

Cut-Cell BCtype: Agglomerated Normals

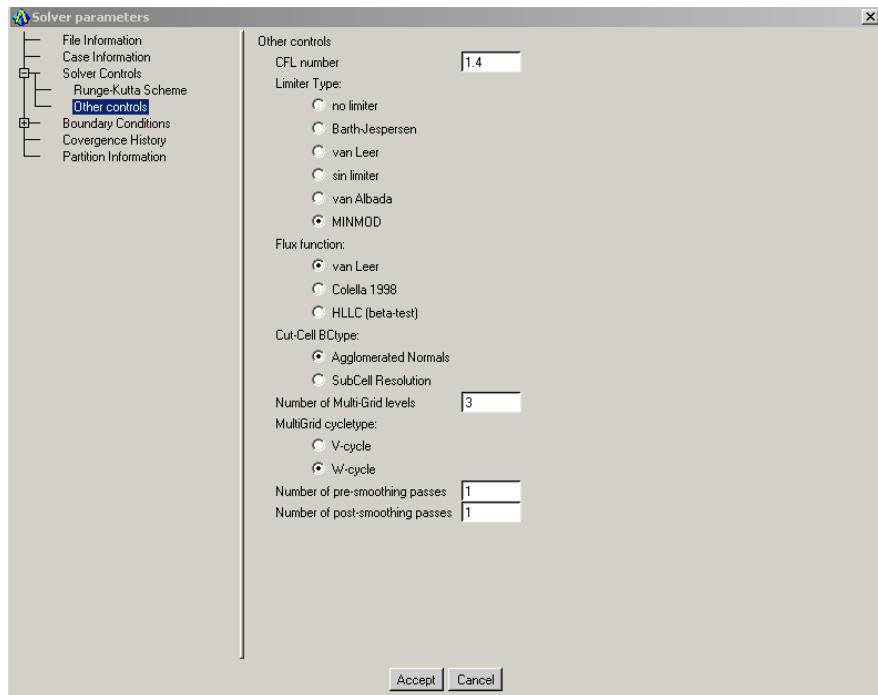
Number of Multi-Grid levels: 3

Multi Grid cycle type: W-cycle

Number of pre-smoothing passes: 1

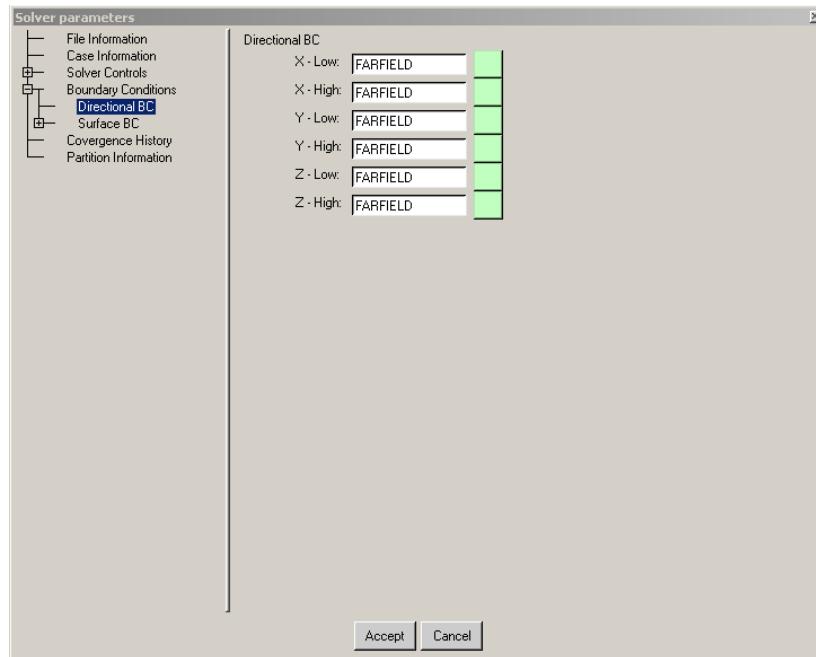
Number of post-smoothing passes: 1

**Figure
4-555
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.

**Figure
4-556
Directional
BC
window**

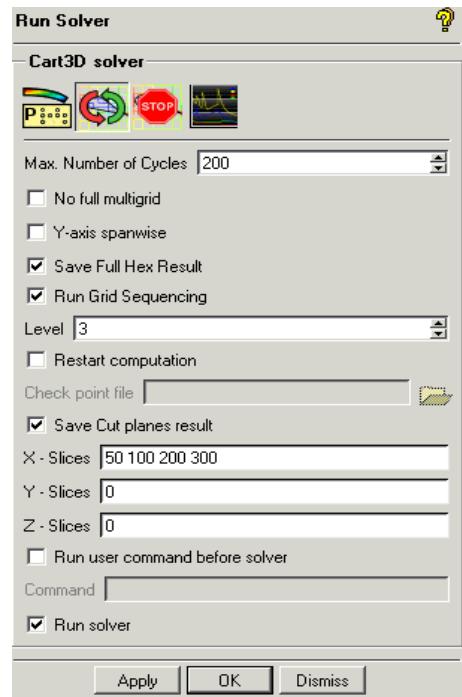


Keep the defaults for Convergence History and Partition Information
Click Accept to save the parameters.

i) Running the FlowCart Solver

Select Solver  > Run Solver  to open the FlowCart solver panel.
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 4-557
Run Solver Window



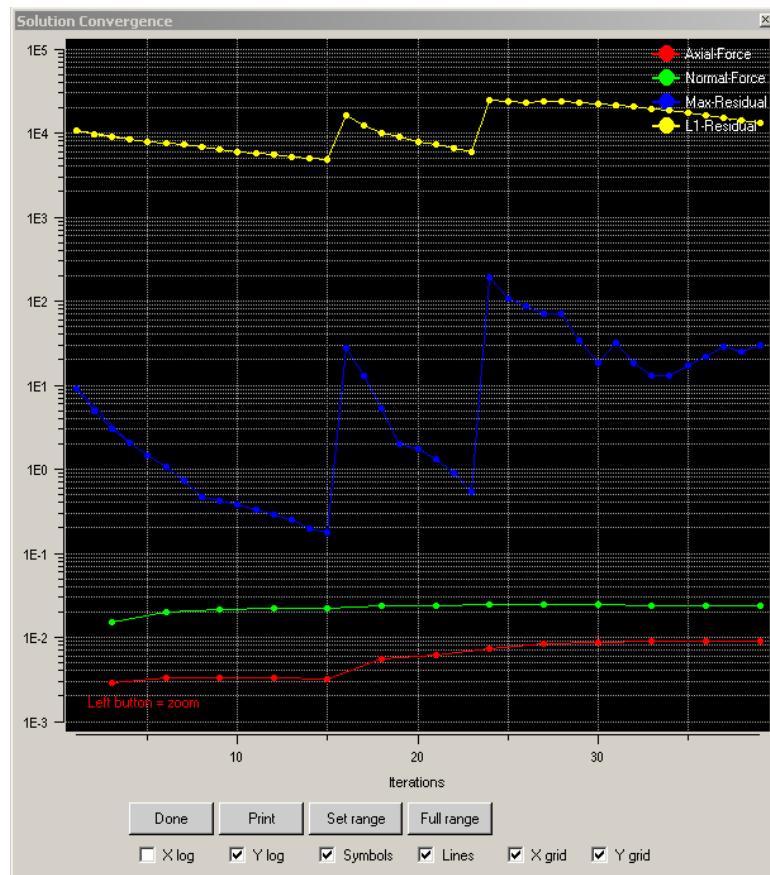
Click Apply and run the solver

The user can view the convergence by clicking on the Convergence



Monitor icon. (The monitor may open automatically.)

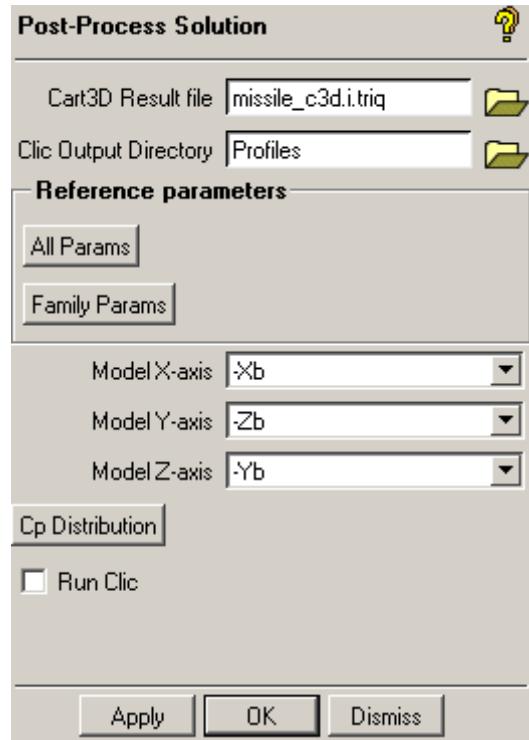
Figure 4-558
Solution
Convergence
Window



j) Computing Force and Moments

1. In the Cart3D main menu select Integrate Cp  . The Post-Process Solution window appears as shown here.

Figure 4-559
Post Process Solution Window



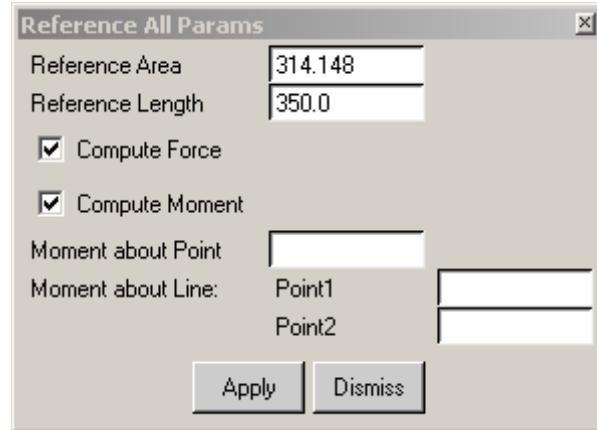
Click All Params icon in the Post-Process Solution window.

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.

Figure 4-560
Reference All Param
window



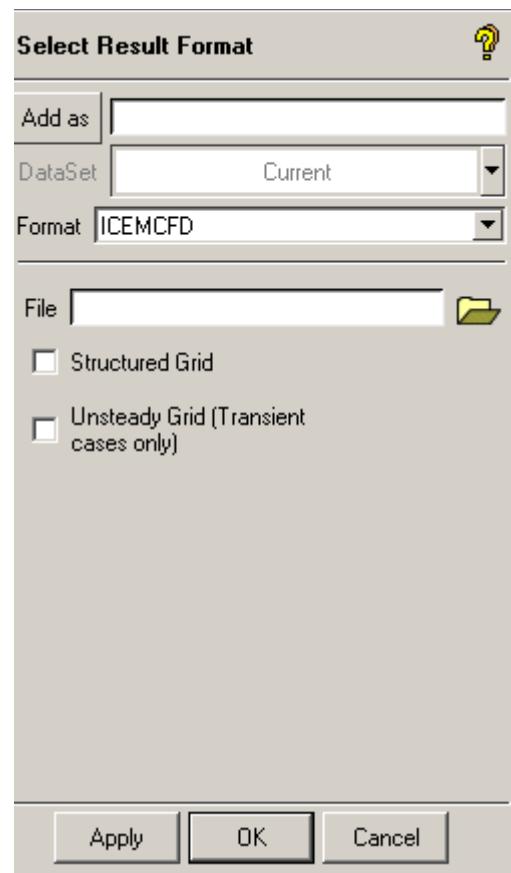
Press Apply in the Post-Process Solution window
The computed force and moment coefficients will be displayed in the
Messages area.

k) Visualizing the results

FlowCart writes three output files:

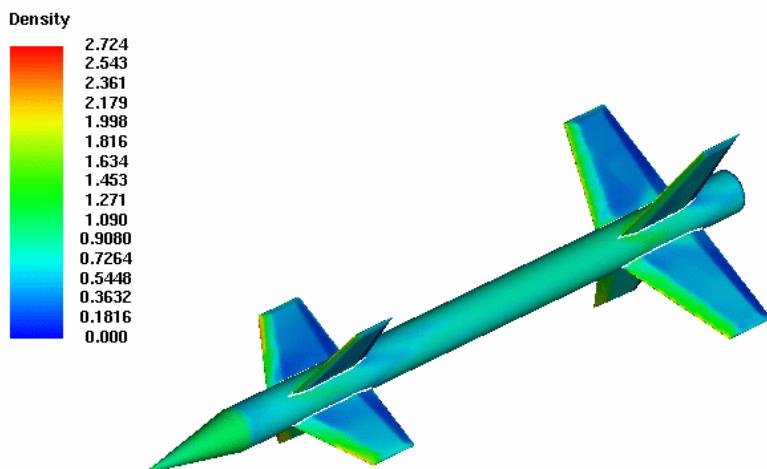
- i) **MISSILE_c3d.i.triq** - Contains Pressure, Velocity and Density extrapolated to the surface triangles. This can be converted to the domain file format via Edit> Cart3D Tri File->Domain file. The default resultant domain file will be **MISSILE_c3d.uns**.
- ii) **slicePlanes.dom** - Cut Plane results
- 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 below. Select **ICEM CFD** as the Format.

Figure 4-561
Select Result Window



Select the result file **surface_results.dom** to get the default result as shown here.

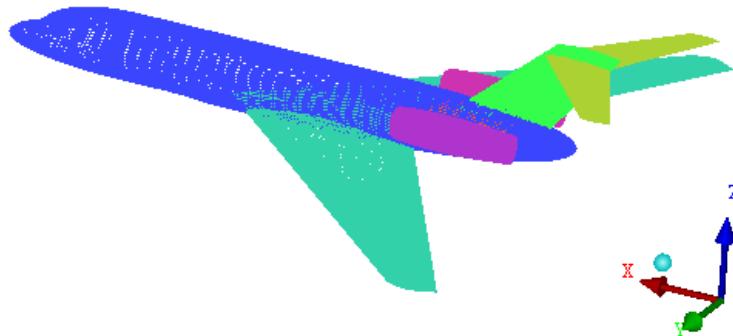
**Figure
4-562
Post
Processor
Display**



4.7.5: 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

1. Compute force and moment information using Clic.
2. Visualize the results.

a) Starting the Project

Load **ANSYS ICEM CFD**. The input files for this tutorial can be found in the Ansys Installation directory, under
..../docu/Tutorials/CFD_Tutorial_Files/Cart3D_Examples. Note: It is preferable to create a separate folder **bjet** and put only **bjet.uns** (domain file) in that folder before performing this tutorial.

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	645
------------------------	--	-----

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.

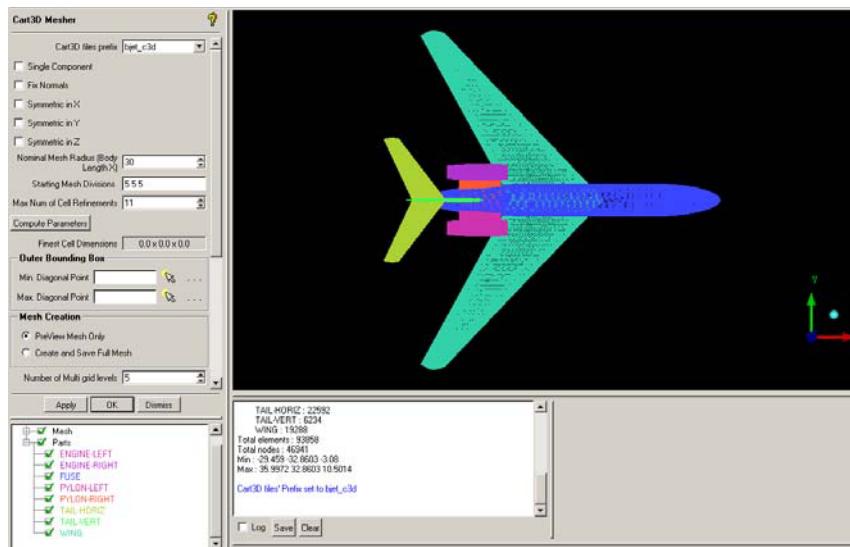
b) Mesh Generation Preview only

Click on Cart3D from the main menu. Select the Volume Mesher button.



to open the window shown below.

**Figure
4-563
Cart3D
mesher
window**

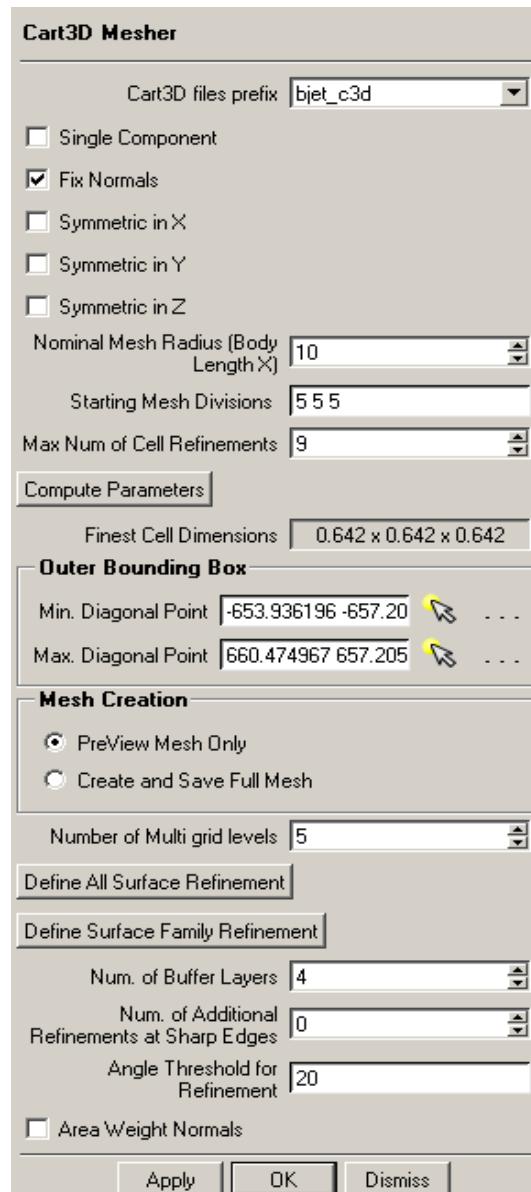


Toggle ON Fix Normals to fix the orientation of the triangles such that their normals point outwards.

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 it to Cart3D format and determines the intersections. At the end, it displays the Finest Cell Dimensions as shown here.

Figure 4-564
Cart3D mesher window



This will create 9 density polygons by default for mesh density control, which can be viewed in the Display Tree 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.

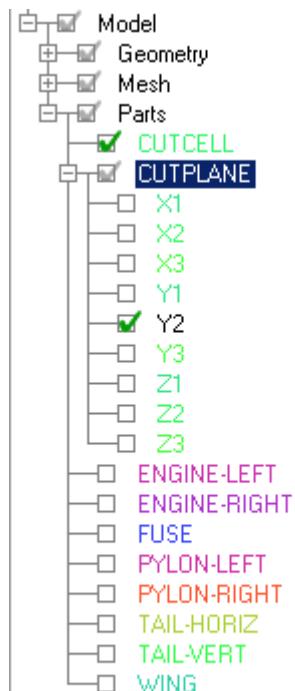
Leave the Angle Threshold for Refinement as 20.

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 turn on only CUTPLANE-Y2 as shown in the figure below.

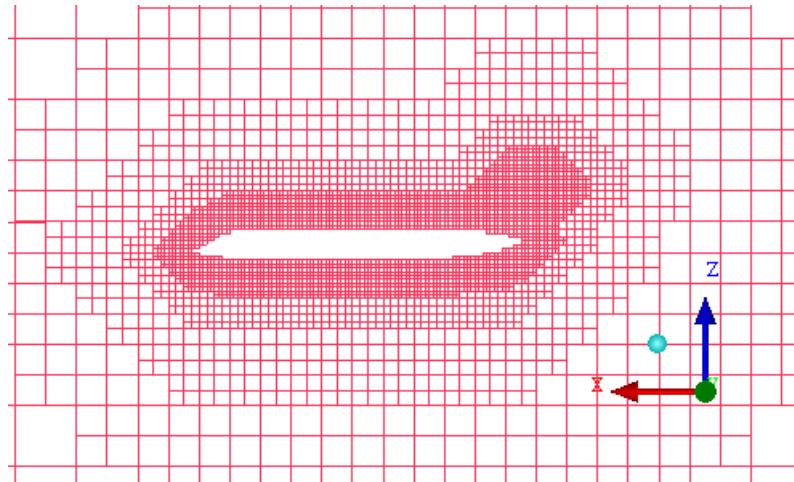
Note: It is advisable to use Parts > Reassign Colors > "Good Colors" to see the results.

Figure 4-565
Display Tree



The mesh is shown below. Go to View > Top. This is the projected mesh on the middle plane in the Z direction, **CUTPLANE-Y2**.

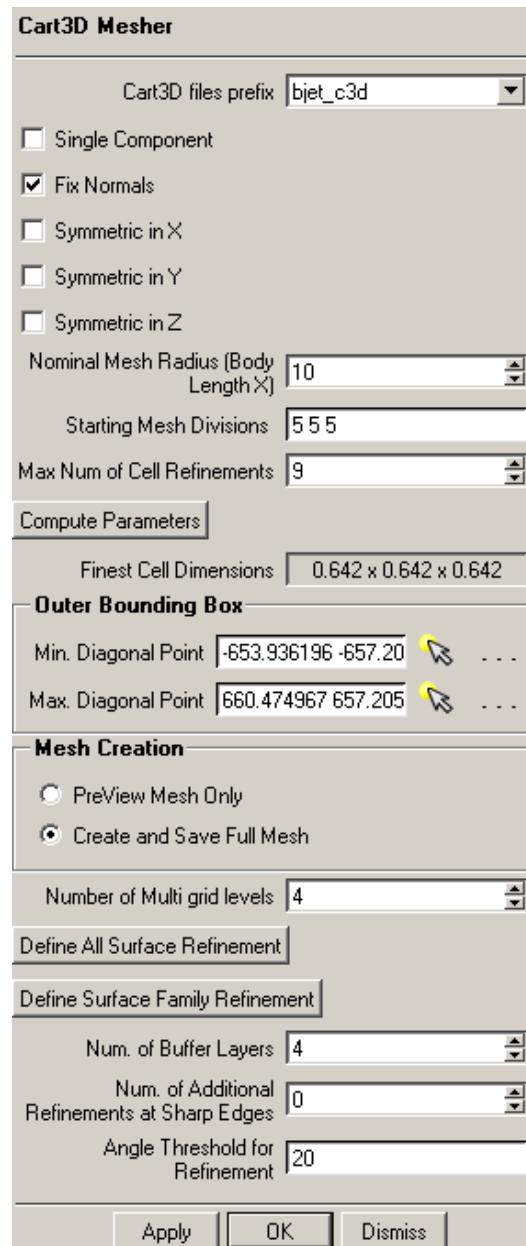
**Figure
4-566
Projected
mesh
CUTPLANE-
Y2**



c) Mesh Generation Full Mesh

In the Cart3D Mesher window enable 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.

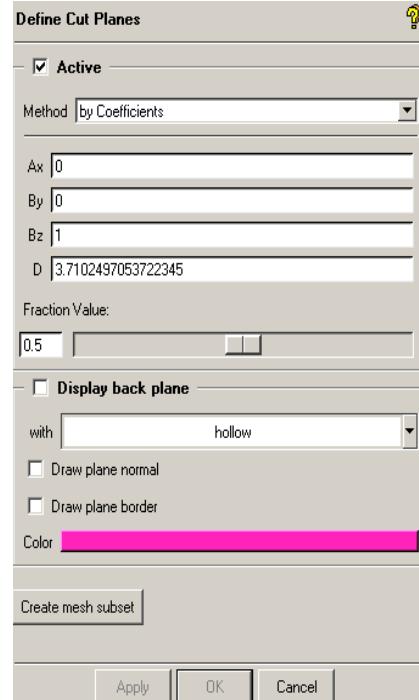
Figure 4-567
Create and Save Full Mesh



Press **Apply**. The Cart3D Mesh window appears which asks us to load the cart3D Full Mesh. Press Yes.

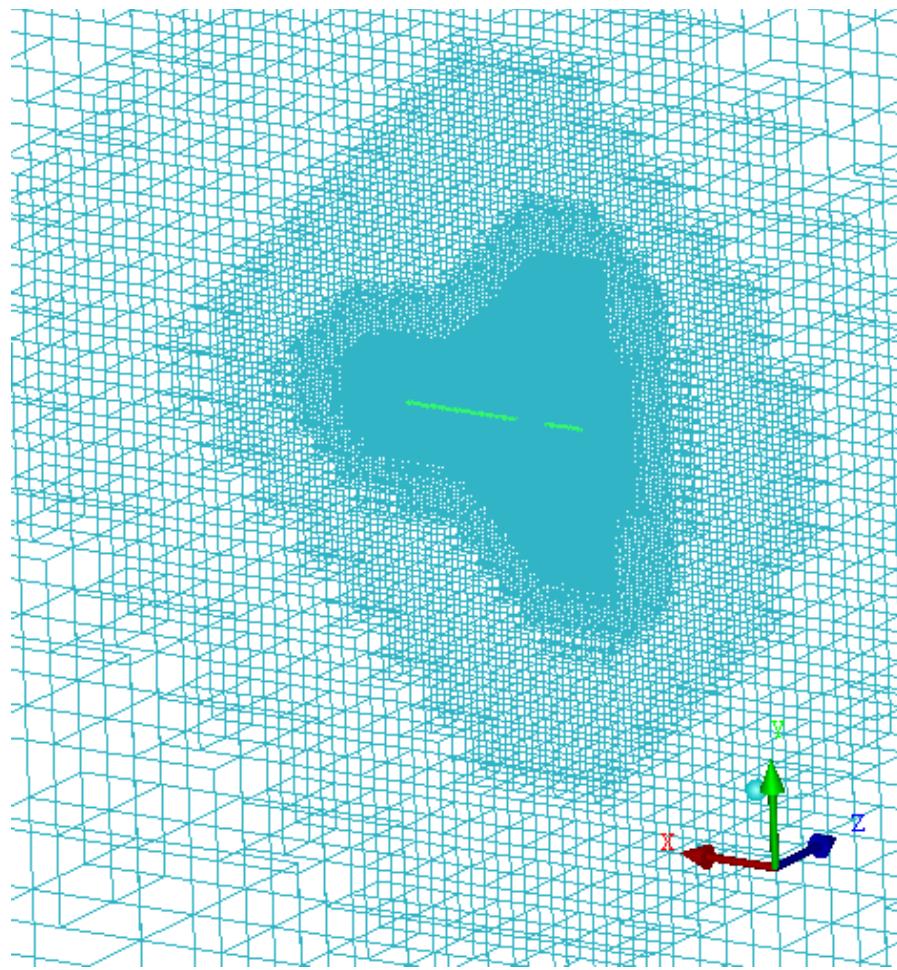
The final mesh generated can be examined through Mesh > Cutplane in the Display Tree. Accept the defaults in the Define Cut Planes window.

Figure 4-568
Cutplane



Enable Volumes from the Mesh branch in the Display Tree. The mesh viewed using the above parameters is shown here.

**Figure
4-569
View
Mesh**

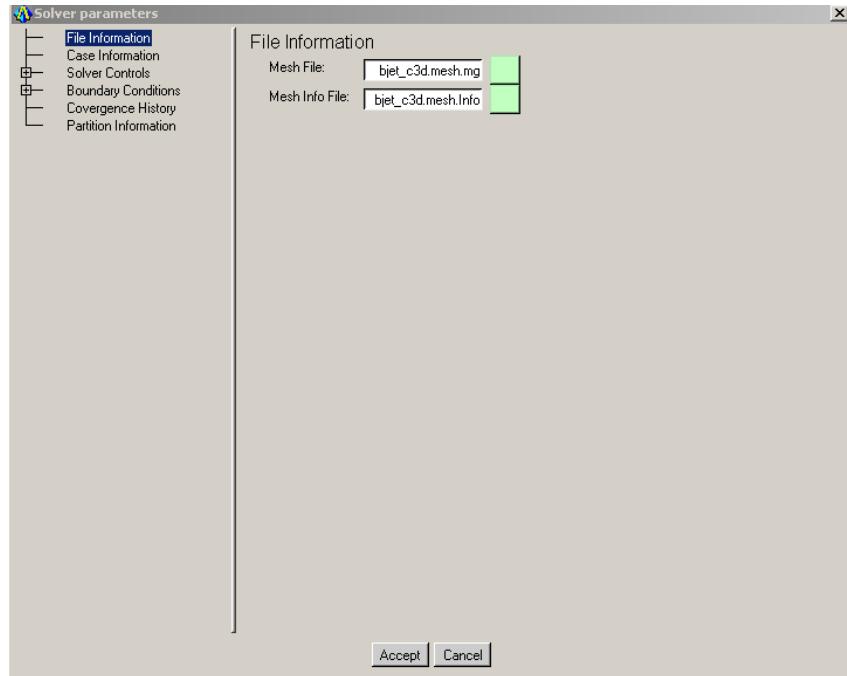


d) Setup Flow Cart Parameters

In the Cart3D Menu select Solver.  Click on Define solver params

 if the panel doesn't open automatically. A Solver parameters window will open.

**Figure
4-570
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 parameter values:

Mach number = -0.8

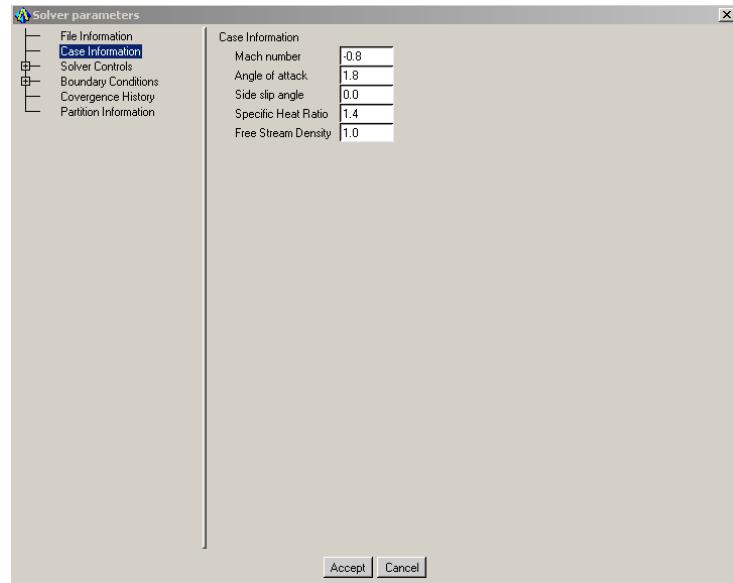
Angle of attack = 1.8

Side slip angle = 0.0

Specific Heat Ratio = 1.4

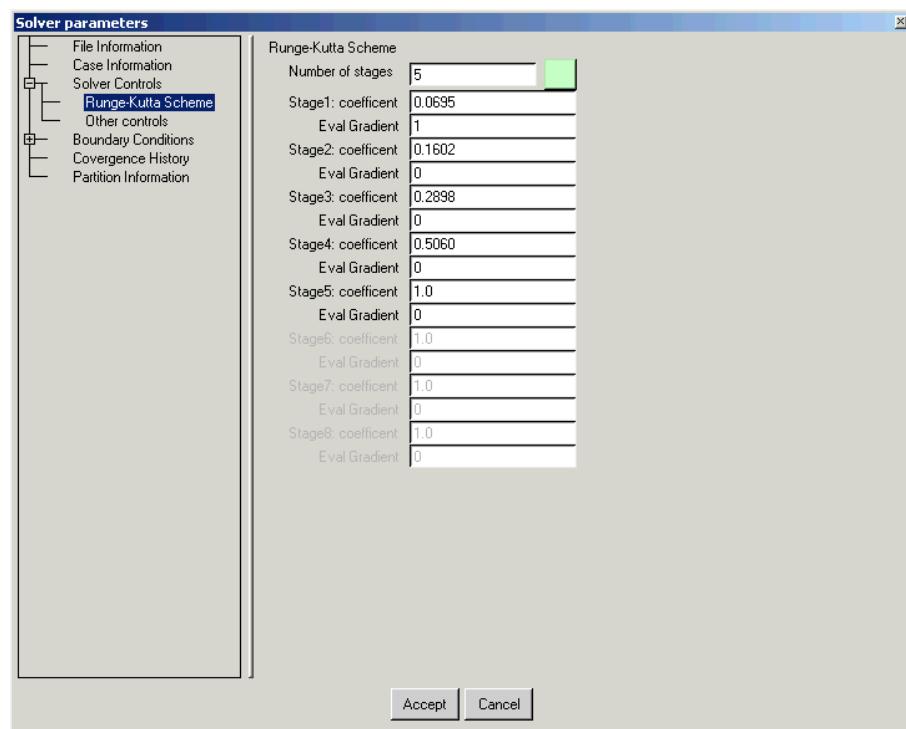
Free Stream Density = 1.0

Figure 4-571
Case
Information
Window



Expand Solver Controls > Runge-Kutta Scheme in the Display Tree and accept the default settings.

**Figure
4-572
Runge-
Kutta
Window**



In Other controls specify the following parameter values:

CFL number: 1.4

Limiter Type: van Leer

Flux function: van Leer

Cut-Cell BCtype: Agglomerated Normals

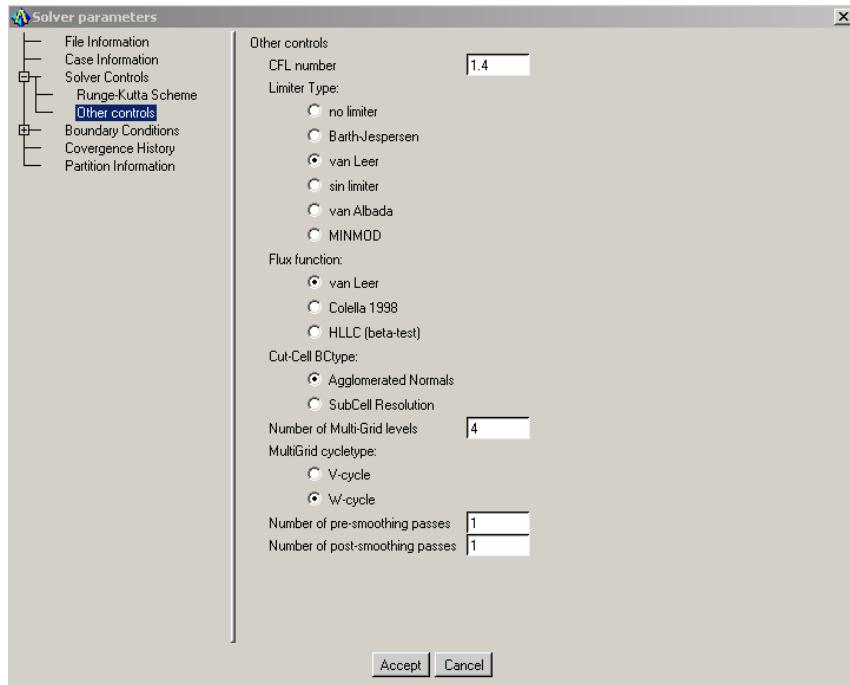
Number of Multi-Grid levels: 4

MultiGrid cycletype: W-cycle

Number of pre-smoothing passes: 1

Number of post-smoothing passes: 1

**Figure
4-573
Other
Control
Window**

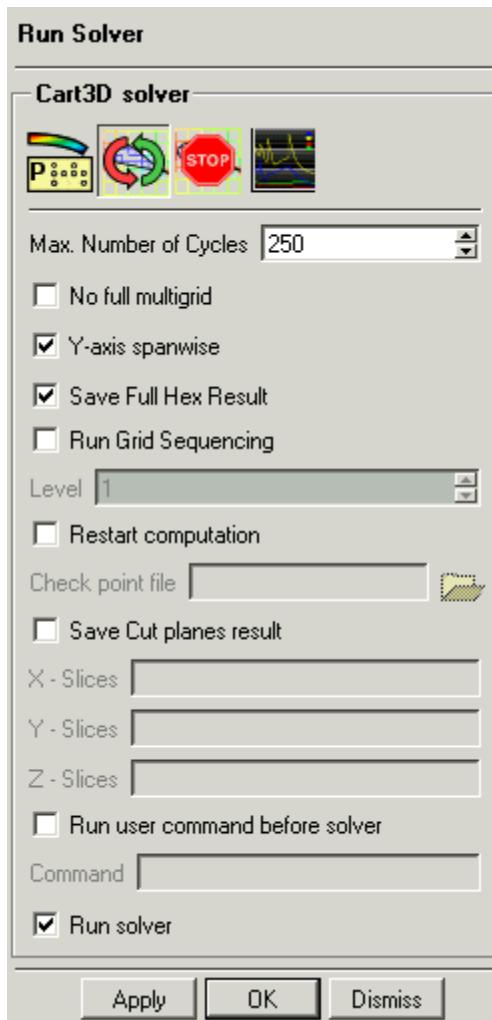


Keep the defaults for Convergence History and Partition Information and press Accept.

e) **FlowCart Solver**

Select Solver  > Run Solver  to open the Run Solver panel.

Figure 4-574
Run Solver window



Specify Max. Number of Cycles = 250.

Enable Y-axis spanwise.

Enable Save Full Hex Result.

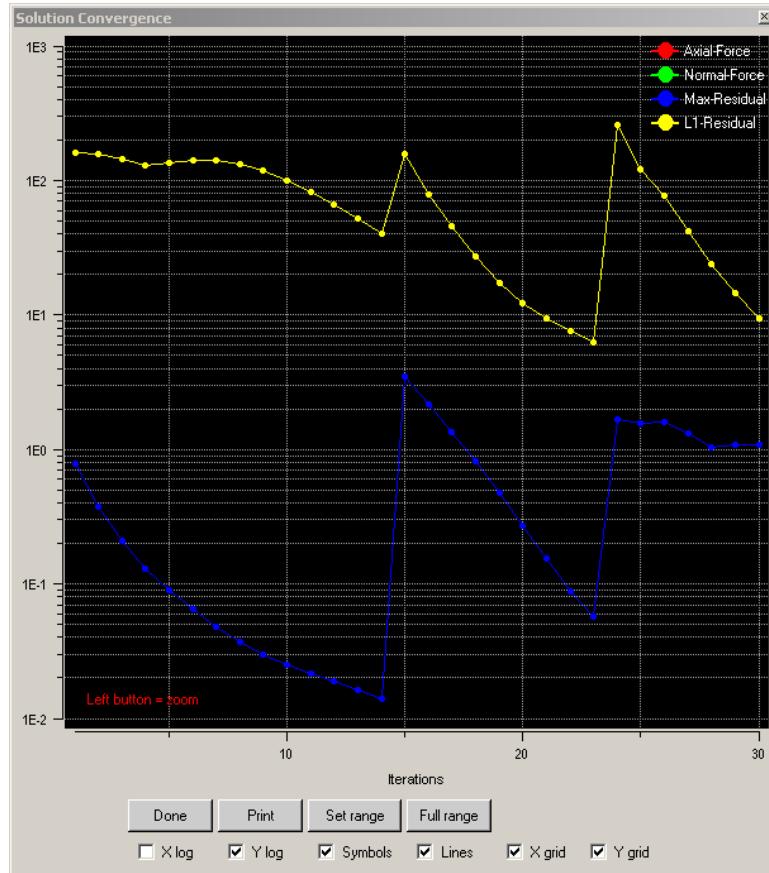
Click Apply and run the solver.

The user can view the convergence by clicking on the Convergence



Monitor icon and the window pops up as shown below. (This may open automatically.)

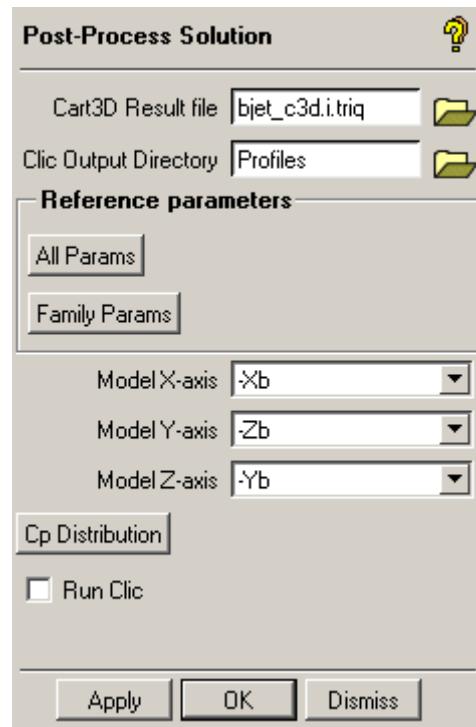
Figure 4-575
Solution
Convergence
window



f) Computing Force and Moments

In the Cart3D main menu select Integrate Cp The Post-Process Solution window will appear.

Figure 4-576
Post Process Solution window



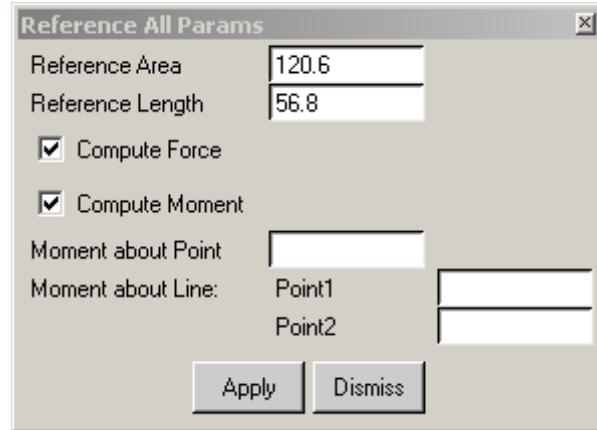
Click **All Params** in the window.

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.

Figure 4-577
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

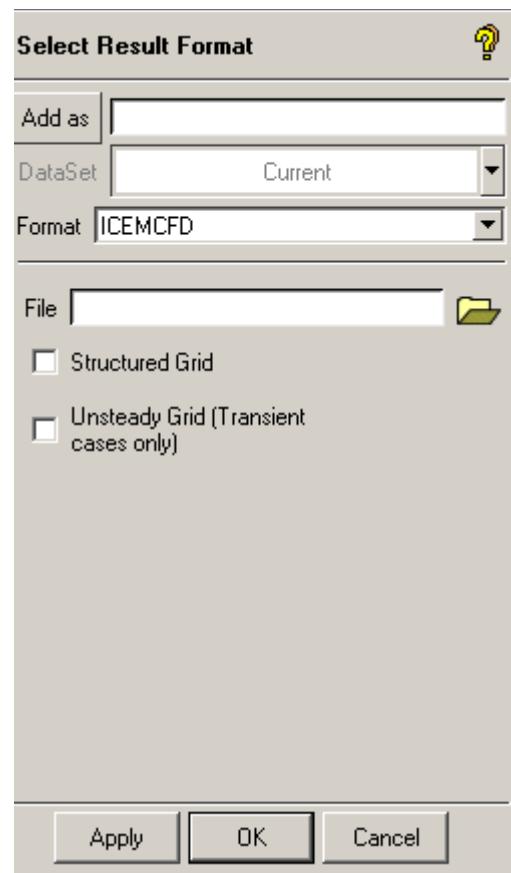
FlowCart writes two output files.

- 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.

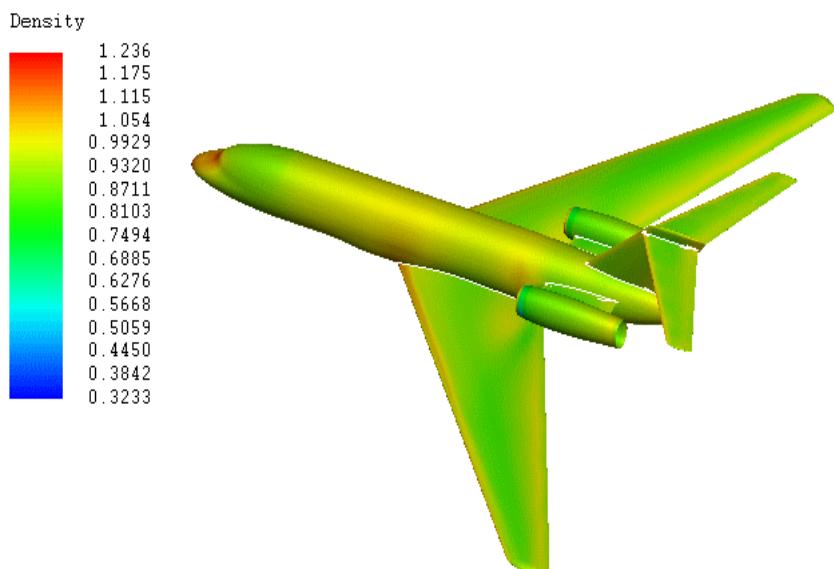
The Select Result Format window will open. Select **ICEM CFD** as the Format.

Figure 4-578
Result File Window



Select the result file **surface_results.dom** and press Apply to get the default result as shown here.

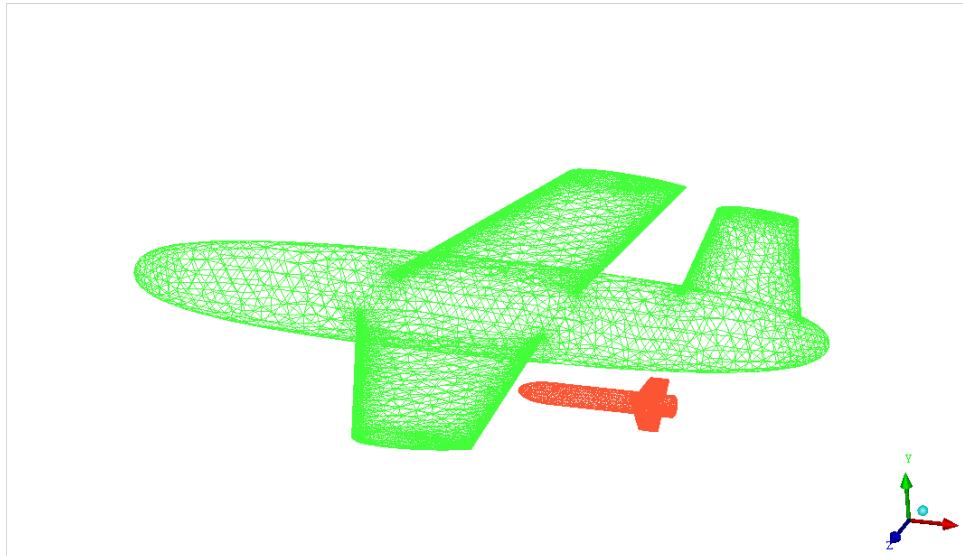
**Figure
4-579
Post
Processing
Result**



4.7.6: 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.



1. This tutorials introduces the following operations:
2. Running the solver for **AOA** = 5 and **Mach** = 0.65.
3. Computing Force and Moment information.
4. Visualizing the results in Post-Processing.
5. Running the 6DOF tool.

a) Starting the Project

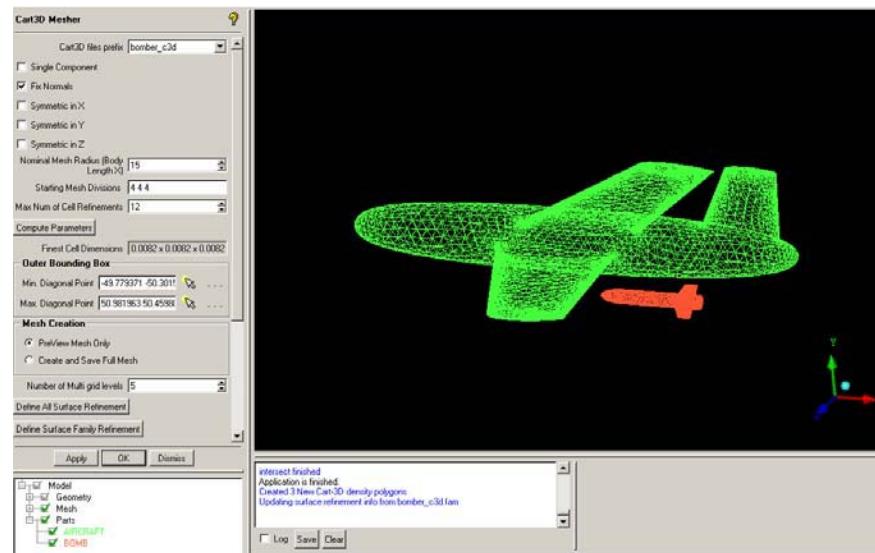
The input files for this tutorial can be found in the Ansys Installation directory, under `../docu/Tutorials/CFD_Tutorial_Files/Cart3D_Examples`. Note: It is preferable to create a separate folder **bomber** and put only the **bomber.uns** (mesh) file in that folder before performing this Tutorial.

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 here.

**Figure
4-580
Cart3D
Main
GUI**



Toggle 'ON' Fix Normals enabled.

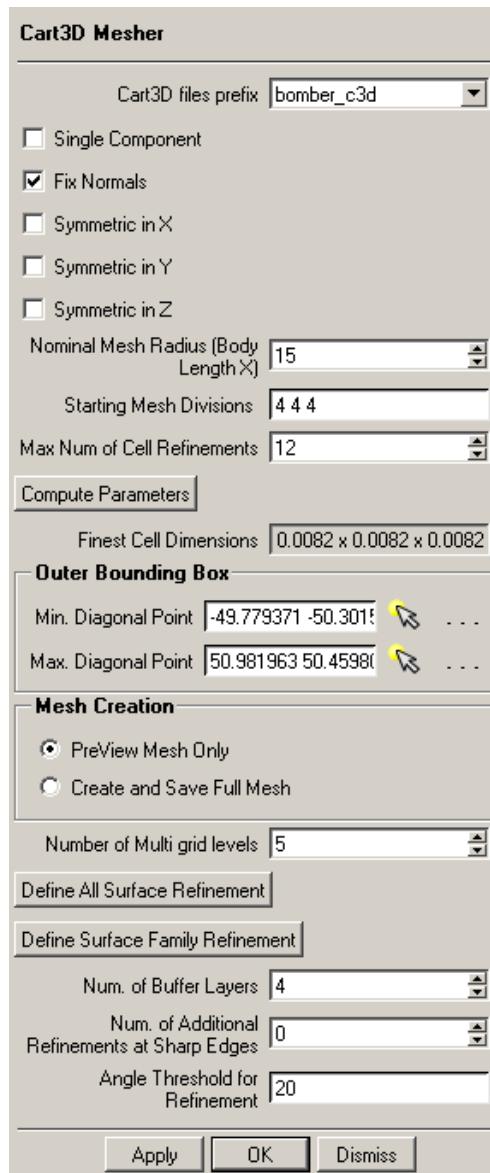
This case is to be run with Mach Number = 0.65. For such subsonic flow, the far field can be 15 times the body length, so specify 15 for Nominal

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 it into Cart3D format and determines the intersections. At the end, it displays the Finest Cell Dimensions as shown here.

Figure 4-581
Compute Parameter



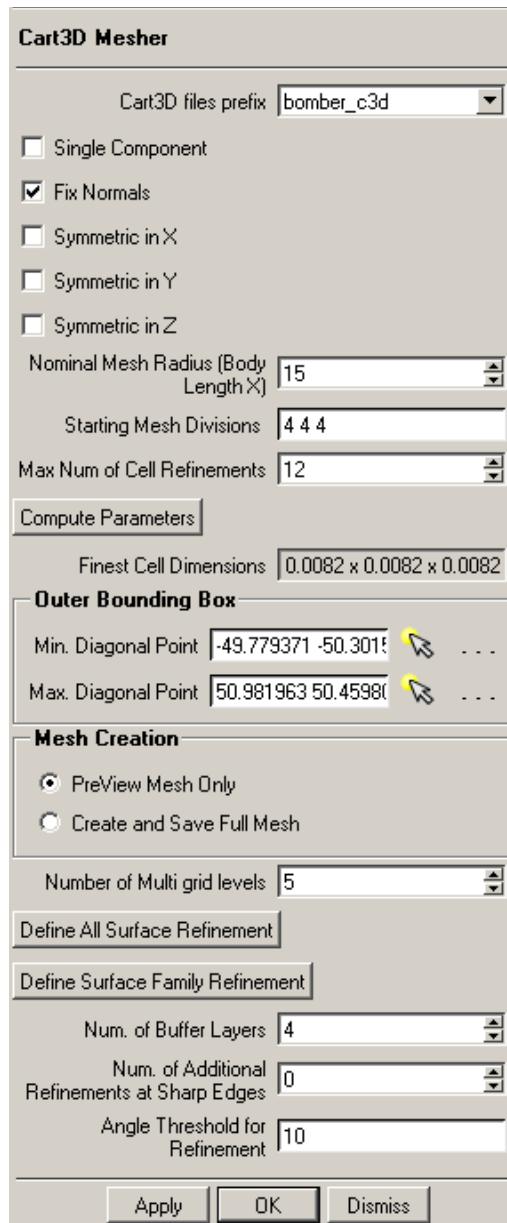
This will create 5 density polygons by default for mesh density control, which can be viewed via Geometry > Densities in the Display Tree.

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).

Set the Angle Threshold for Refinement to 10 as shown here.

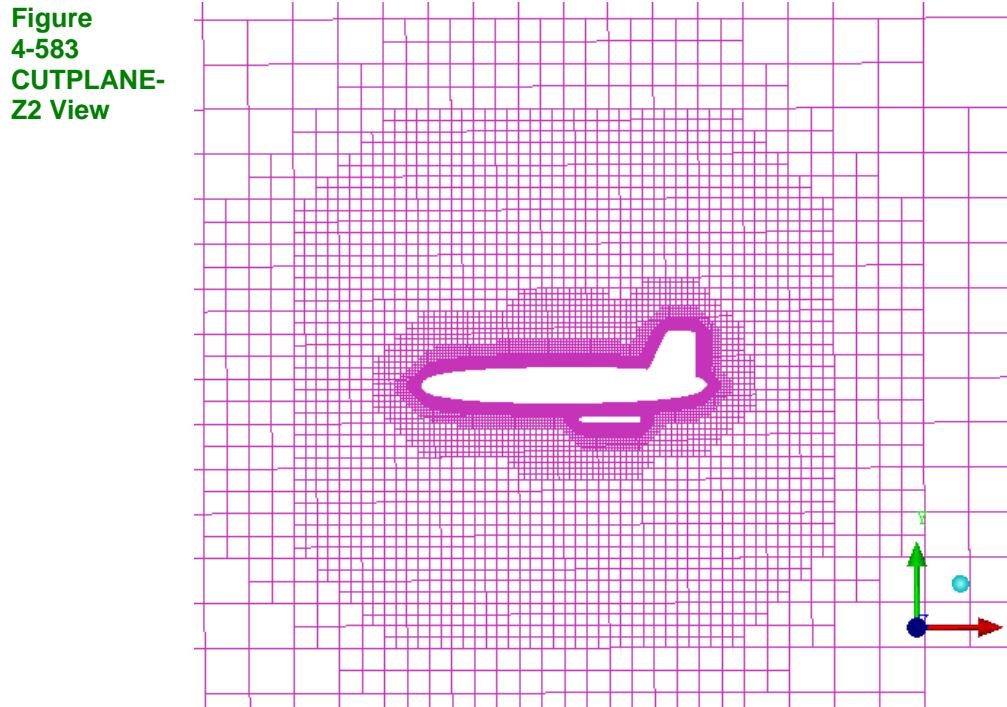
Figure 4-582
Angle Threshold for Refinement
Window



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

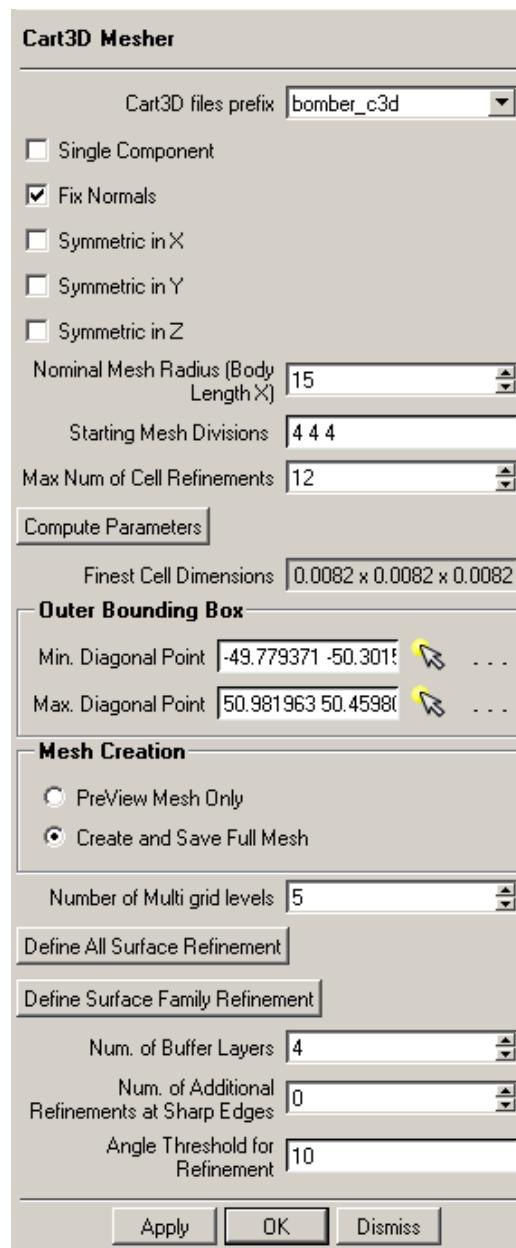
One such view is shown below.



c) Mesh Generation Full Mesh

Enable Create and Save Full Mesh 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 4-584
Create and Save Full Mesh



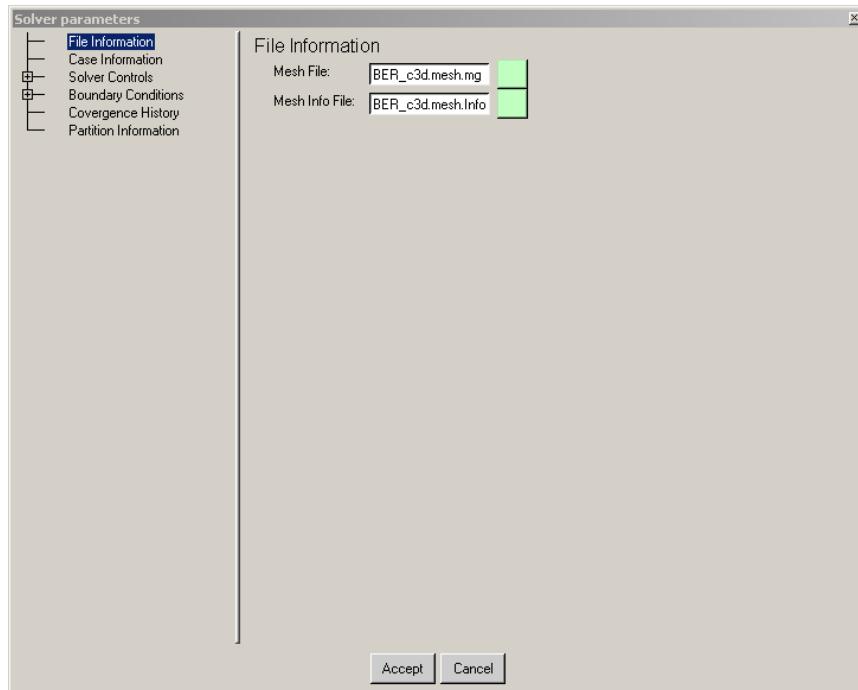
Press Apply. The Cart3D Mesh window appears which asks us about loading the cart3D Full Mesh. Press Yes.

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.

**Figure
4-585
Solver
Parameter
Window**



Choose File Information>Mesh File as **BOMBER_c3d.mesh.mg** (this should be default).

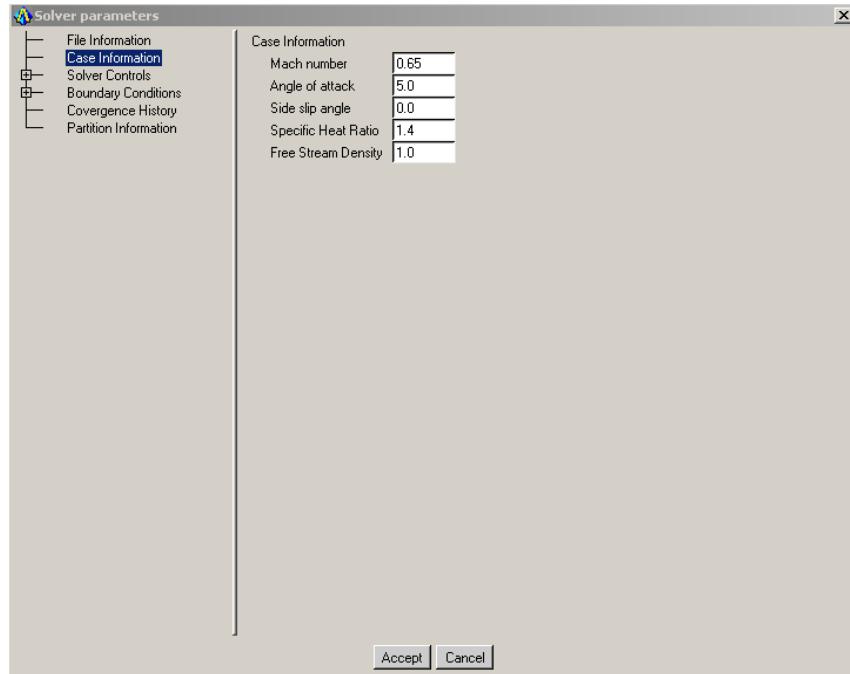
Click on Case Information and enter the following parameter values:

Mach number = 0.65

Angle of Attack = 5.0

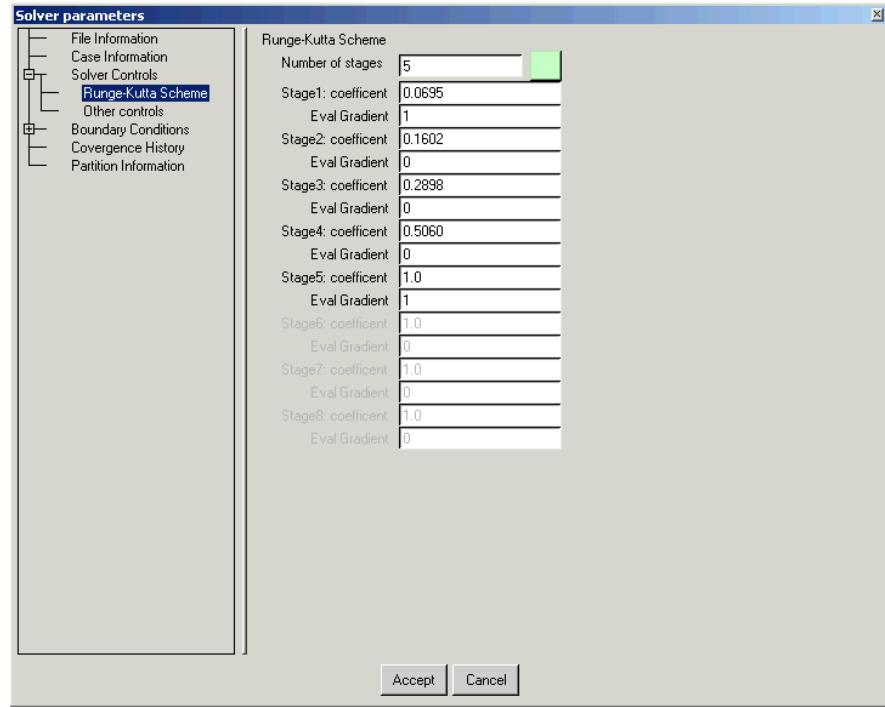
Side Slip angle = 0.0
Specific Heat Ratio = 1.4
Free Stream Density = 1.0

**Figure
4-586
Case
Information
Window**



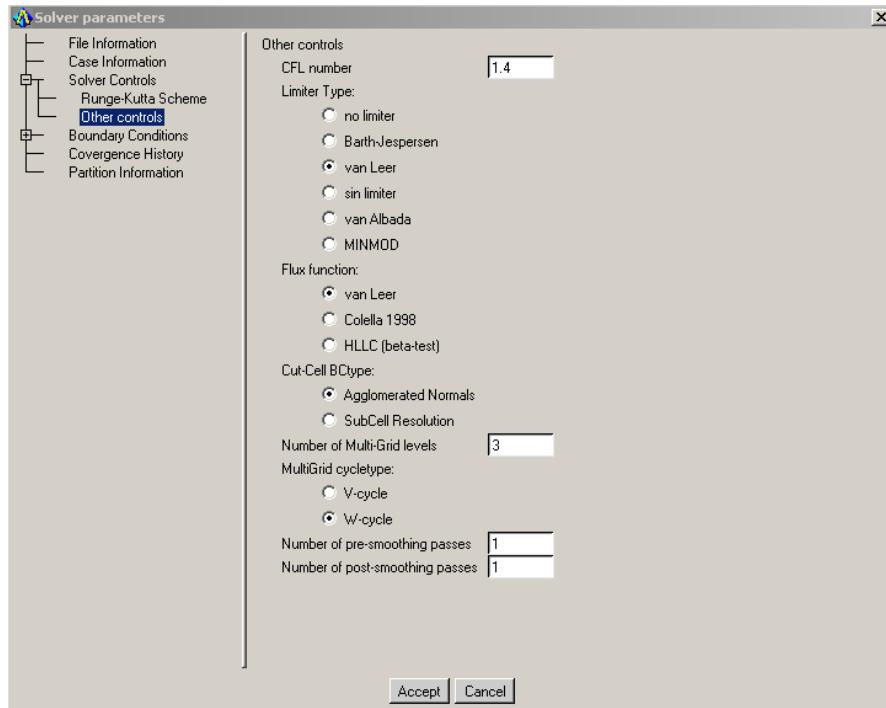
Click on Solver Controls > Runge-Kutta Scheme. Evaluate the gradients only at Stages 1 and 5 as shown.

**Figure
4-587
Runge-
Kutta
Scheme**



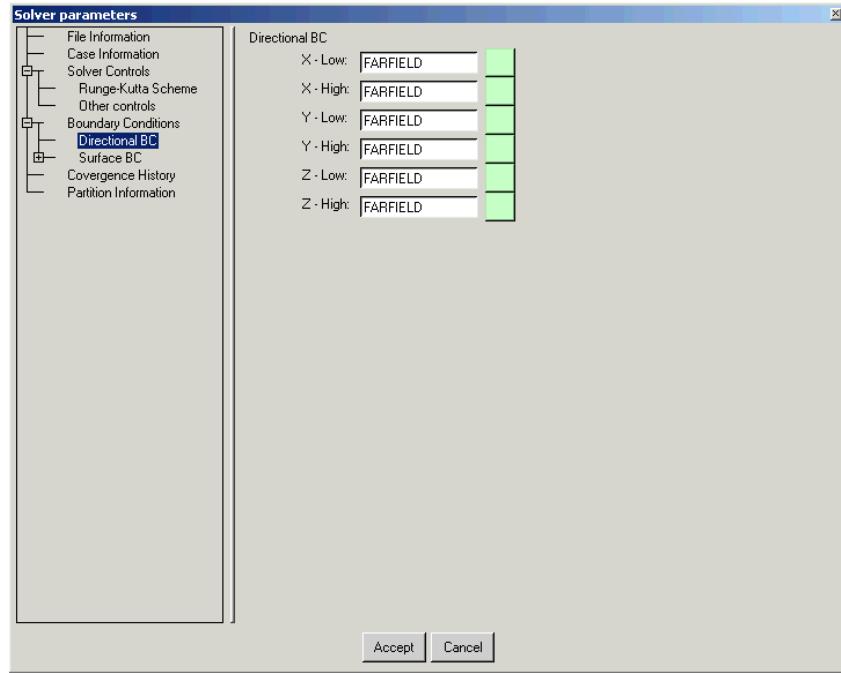
Click on Other controls. Set CFL number to 1.4 and Number of Multigrid levels to 3. Leave the remaining values as default.

**Figure
4-588
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.

**Figure
4-589
Boundary
Condition
Window**



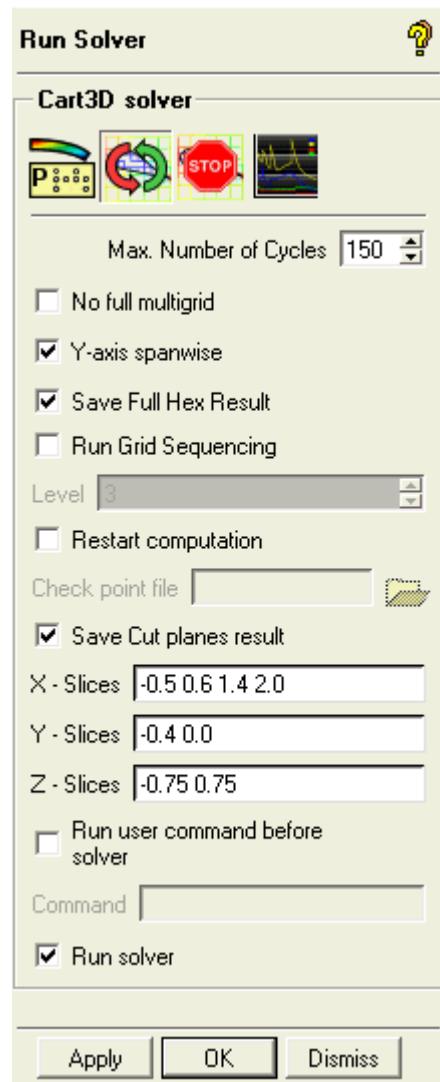
Leave the Convergence History and Partition Information as default. Click on Accept in the Solver parameters window.

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 the figure below.

Figure 4-590
Flow Cart Solver Window



Click on **Apply** to start the solver and output the results files.
The convergence history plot window should automatically open.

f) Visualizing the results

FlowCart writes two output files:

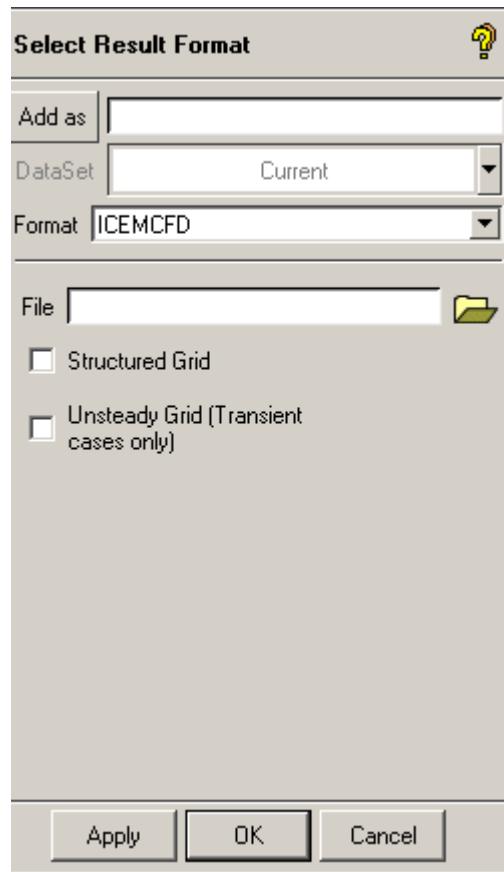
i) **BOMBER_c3d.i.triq** - Contains Pressure, Velocity and Density extrapolated to the surface triangles. This can be converted to domain file format via Edit> Cart3D Tri file->Domain file. The resultant file will be **BOMBER_c3d.uns**.

ii) **slicePlanes.dom** – cut plane results.

Go to File > Results > Open Result File.

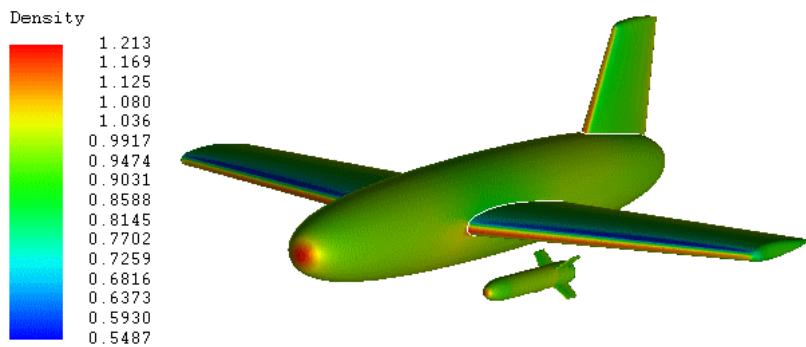
A Select Result Format window pops up, Select ICEM CFD as the Format.

**Figure 4-591
Results Window**



Select the result file **surface_results.dom** and Apply to get the default result as shown here.

**Figure
4-592
The result
Generated**



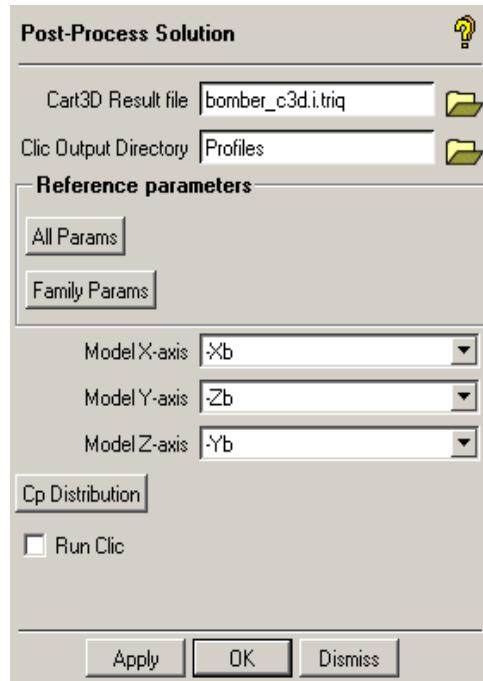
g) Clic - Computing Force and Moment

Aerodynamics and Body forces can be calculated by using the Clic utility.



Select Cart3D > Integrate Cp to open the Post-Process Solution window.

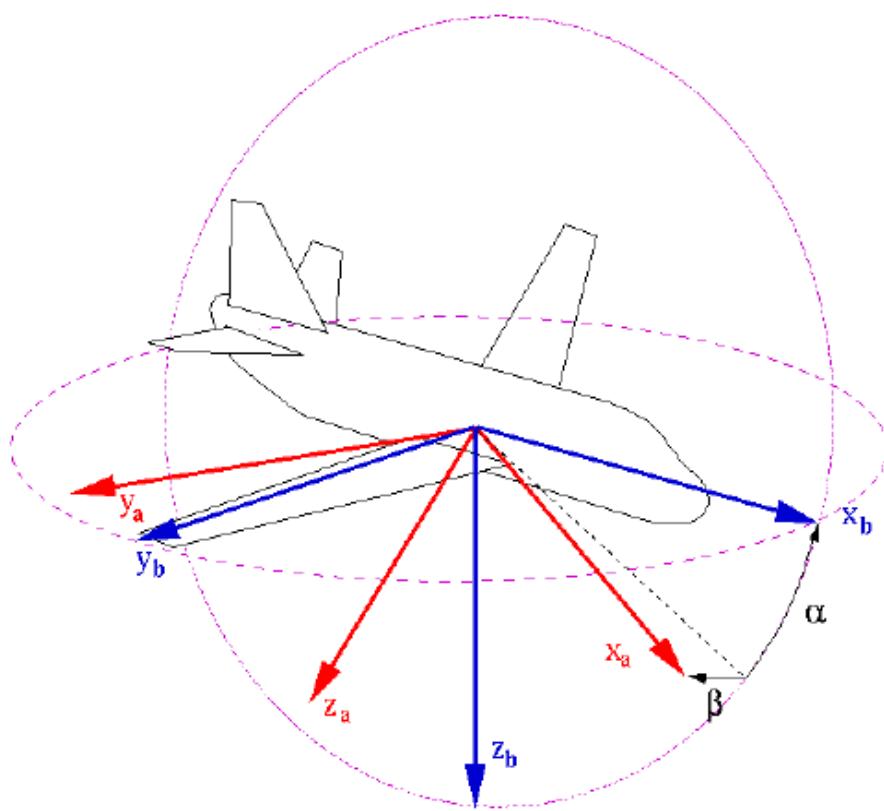
Figure 4-593
Post Process Solution Window



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 the figure below. Our model coordinate should be mapped on to Clic's coordinates. In this case, the Display X-axis is $-X_b$ in Clic's coordinates. Similarly Display Y-axis is $-Y_b$ in Clic's coordinates and Display Z-axis is $-Z_b$ in Clic's coordinates.

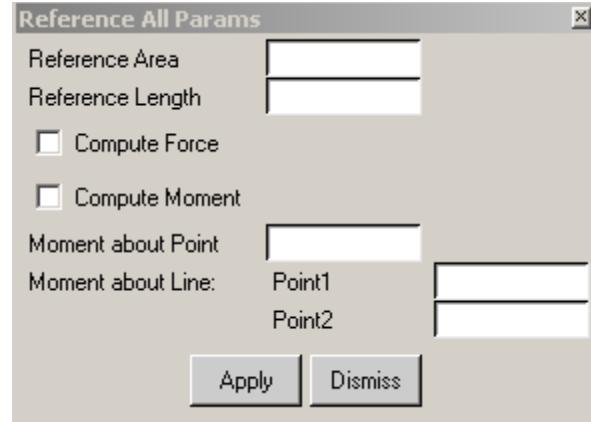
**Figure
4-594
Axes**



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 .

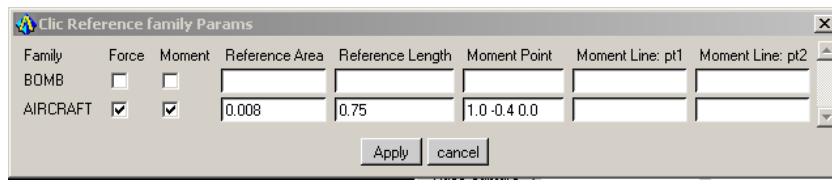
Figure 4-595
Reference All Param
Window



For this case Force and Moment are only calculated for the **Bomb** component.

Click on Family Params under Reference parameters. The Clic Reference family Params window opens.

Figure
4-596
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.

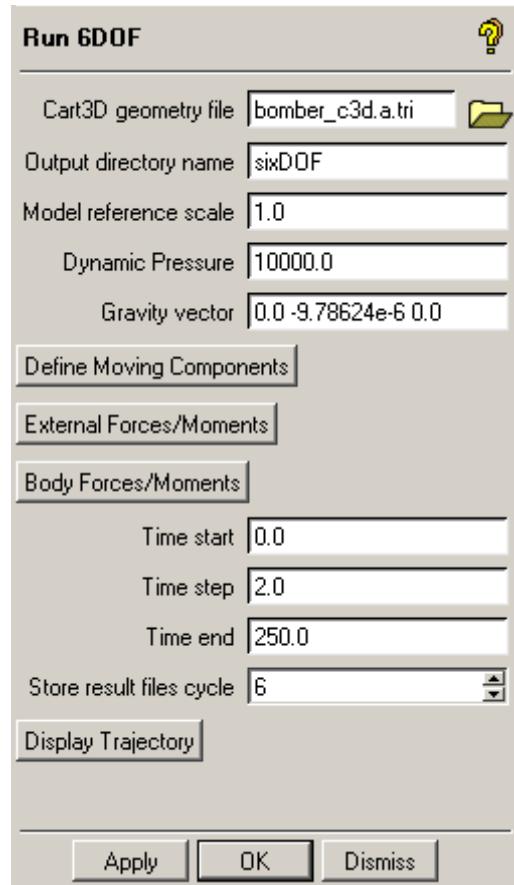
h) Six Degrees of Freedom

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 here.

Figure 4-597
Run SixDOF window

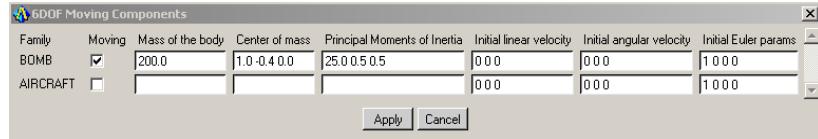


Choose the un-intersected surface tri file **bomber_c3d.a.tri** as the Cart3D geometry file.

Use the non-dimensional value for 9.81 m/s^2 (0.0, -8.667e-05, 0.0) for the Gravity vector.

Click on ‘Define Moving Components’ from the Run 6DOF window shown above. This will open up the 6DOF Moving Components window as shown in figure below.

Define Moving Components



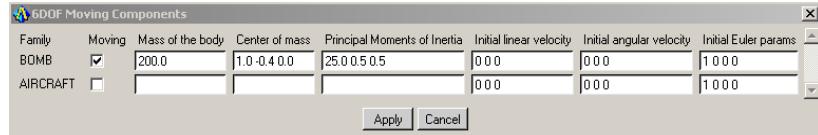
Select **BOMB** as the Moving component. Set Mass of the body = 200.0. Specify the BOMB Center of mass as (1.0, -0.4, 0.0). The Principal moments of Inertia have to be specified for three Components as (10.0, 200.0, 200.0). Leave the default values for Initial linear velocity, Initial angular velocity 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.

Define Moving Components

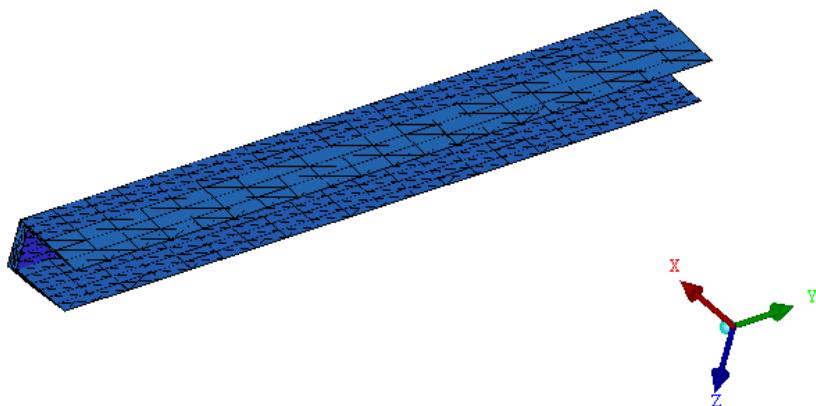


4.7.7: Advanced Pitot Intake Tutorial

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

The input files for this tutorial can be found in the Ansys Installation directory, under `../docu/Tutorials/CFD_Tutorial_Files/Cart3D_Examples`. 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**.

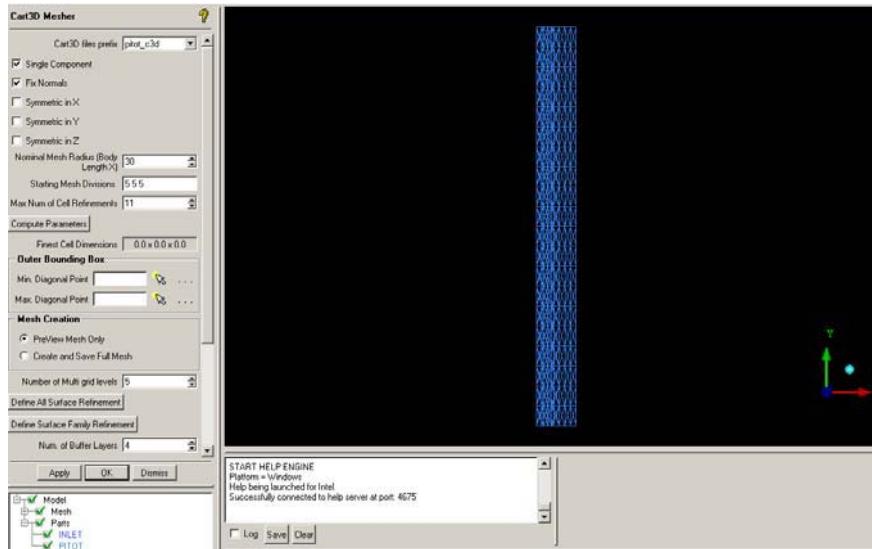
Select Open Geometry  to load **Density_boxes.tin** which contains density boxes set up for bow and lip shock resolution.

Turn on Geometry > Densities in the Display Tree 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 here.

**Figure
4-598
Cart 3D
GUI
window**



Toggle ON Fix Normals. This will fix the triangle orientations such that their normals are pointing outward.

Enable Single Component .

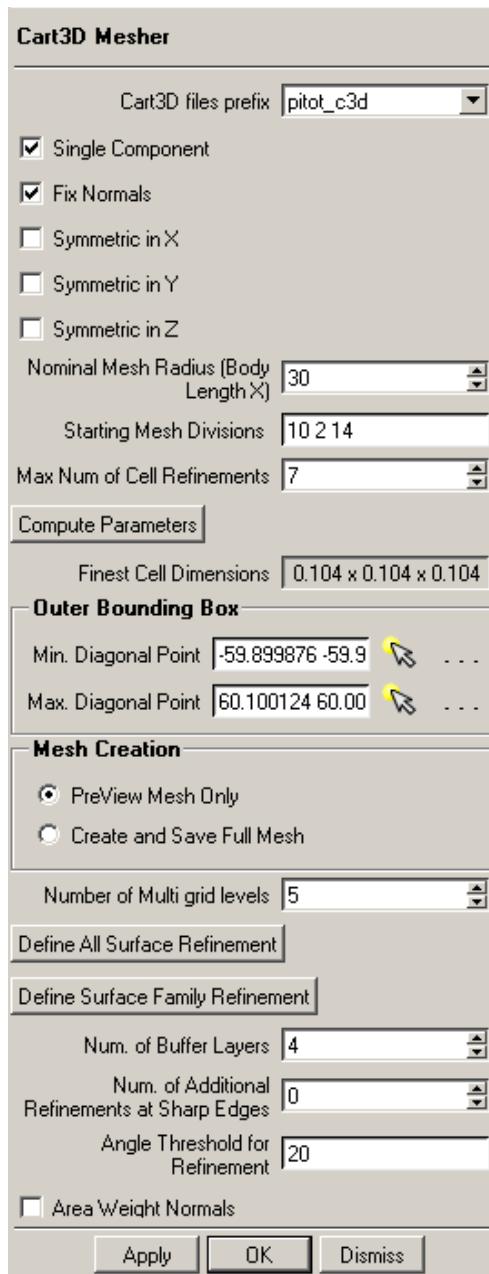
Accept default value of Nominal Mesh Radius (Body Length X) = 30, Starting Mesh Divisions = 10 2 14 and Max Num of Cell Refinements = 7

Leave the other values as default.

Click Compute Parameters. This saves the mesh in the local directory and converts it into Cart3D format. At the end, it displays the Finest Cell Dimensions as shown here.

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	688
------------------------	--	-----

Figure 4-599
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

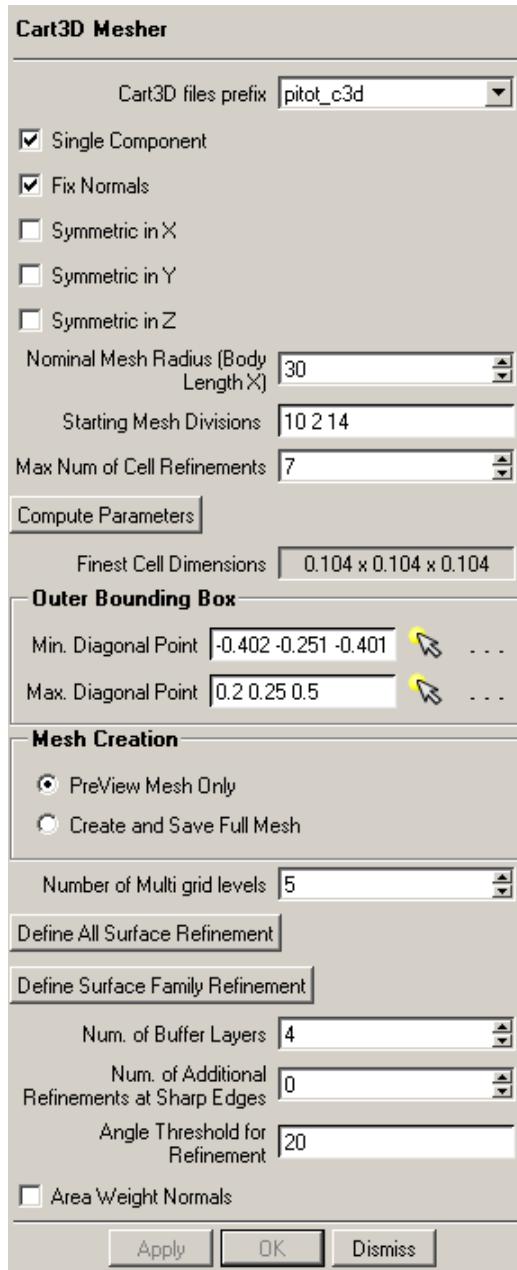
Leave the Angle for Threshold Refinement = 20

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 4-600.

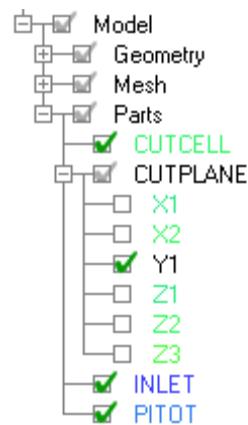
Click Apply to run the mesher. This will create a domain file with 6 Cut Planes (Quad Elements)

Figure 4-600
**Change Maximum/Minimum
 Diagonal Point**



In the Part Menu under the Display Tree perform the operation Parts > Hide All and then turn on only the Part CUT PLANE Y1 as shown here.

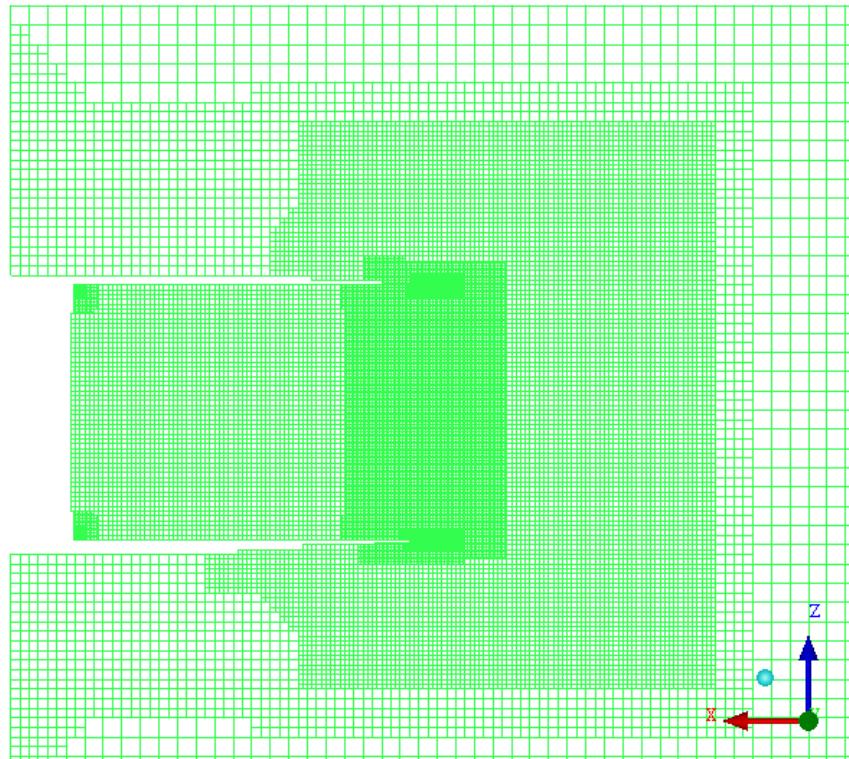
Figure 4-601
Display Tree



The mesh projected onto CUTPLANE-Y1 is shown below.

Note: The mesh in this figure can be viewed by View > Top.

**Figure
4-602
CUT
PLANE
Y1
Mesh**

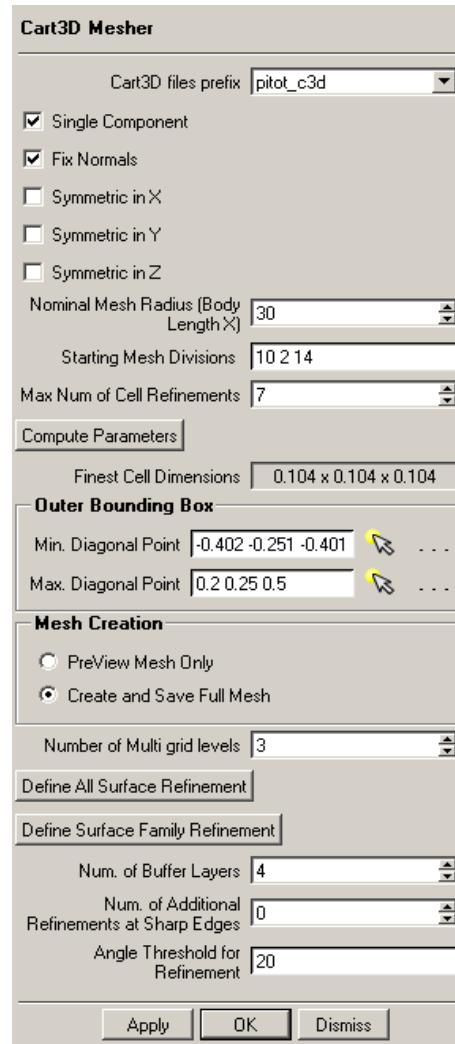


Perform the operation Parts>Show All after viewing the mesh.

c) Mesh Generation-Full Mesh

Now in the Cart3D Mesher window enable Create and Save Full Mesh as shown. Set the Number of Multi grid levels to 3. This will create 3 levels of coarsened mesh, which can be read by the solver.

Figure 4-603
Create and Save Full Mesh



Press Apply. The Cart3D Mesh window appears which asks us about loading the cart3D Full Mesh. Press Yes.

Figure 4-604
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.

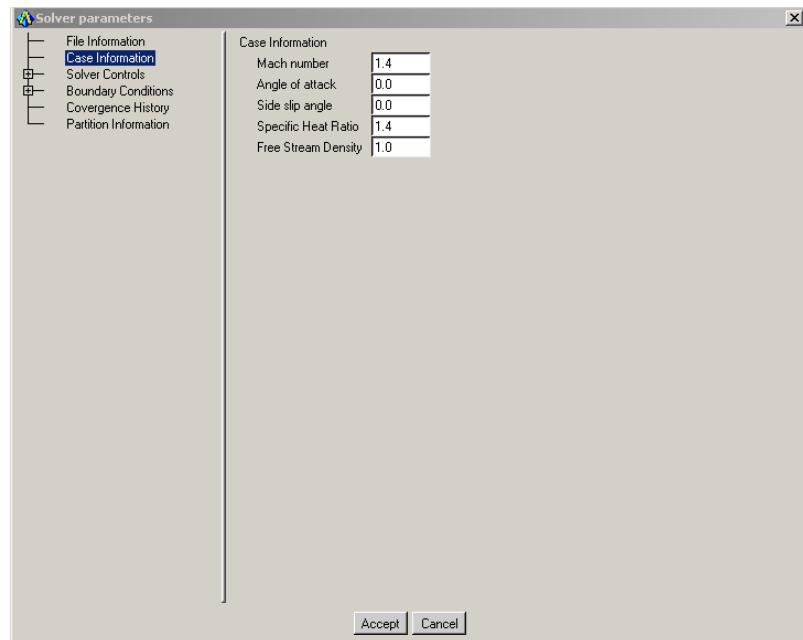


Go to Solver  > Define solver parameters  (The parameters panel may open automatically.)

In the Solver parameters window do the following:

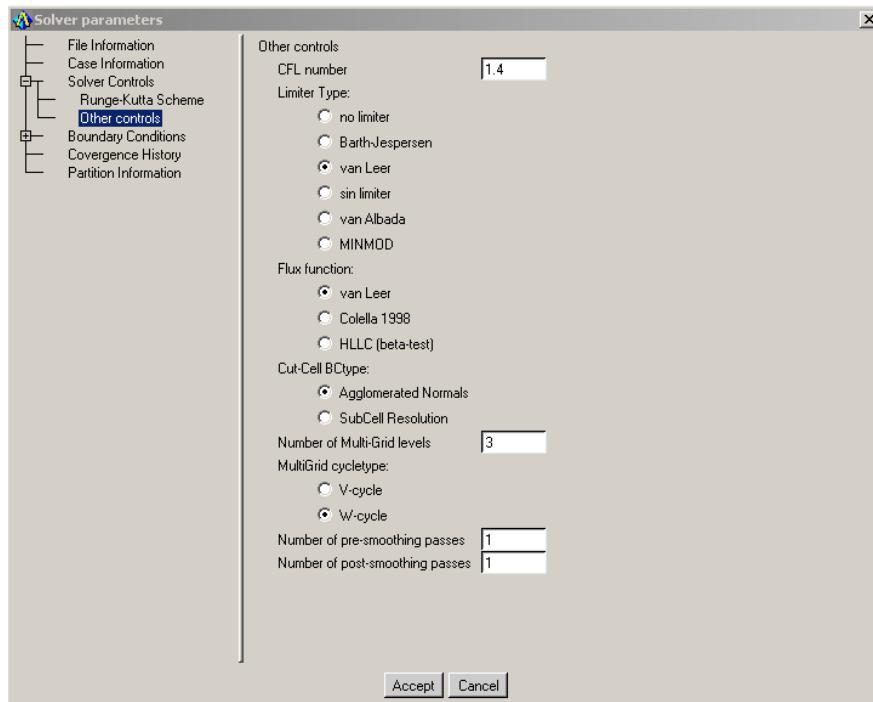
- i) In Case information: set Mach number = 1.4. Use the other defaults as shown below.

**Figure
4-605
Case
Information
Window**



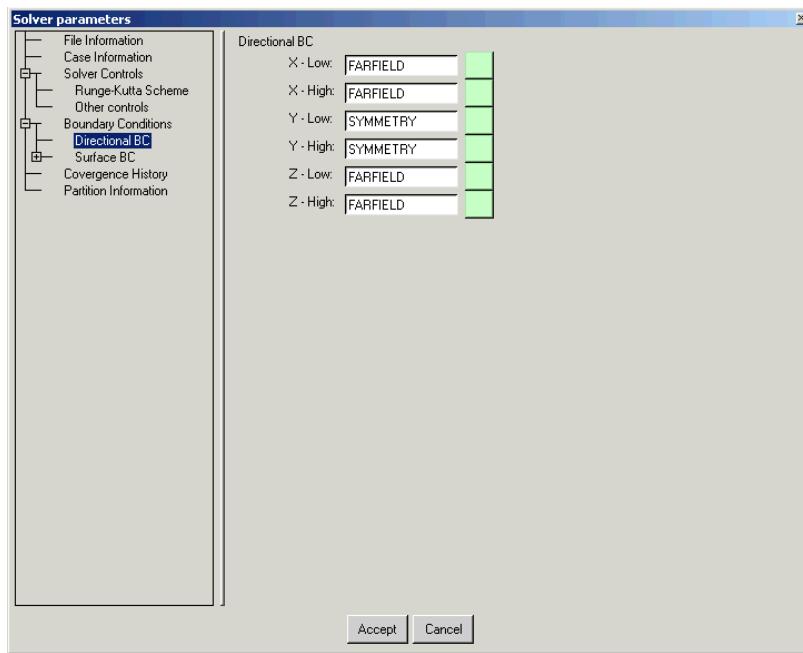
- ii) In Solver Controls > Other controls: set Number of Multi-Grid levels =3. Use the other defaults as shown here.

**Figure
4-606
Other
Control
Window**



- iii) In Boundary Conditions>Directional BC: set Y-Low and Y-High to SYMMETRY. Leave the others as default as shown.

**Figure
4-607
Directional
Boundary
Condition
Window**



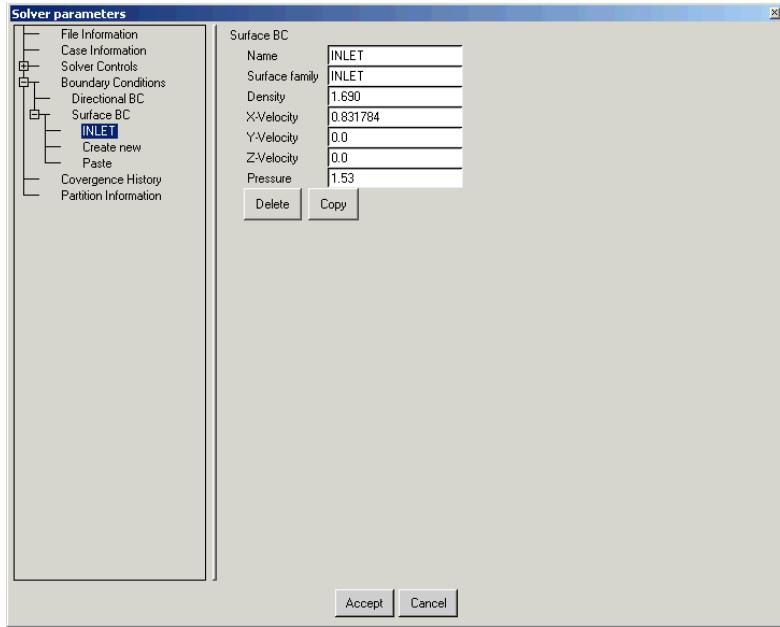
iv) In Surface BC select Create new and specify the following:

Name	INLET
Surface family	INLET
Density	1.690
X-Velocity	0.831784
Pressure	1.53

The others should be the default values.

Click Accept from the Solver parameters window.

**Figure
4-608
Surface
Boundary
Condition**



e) Run solver

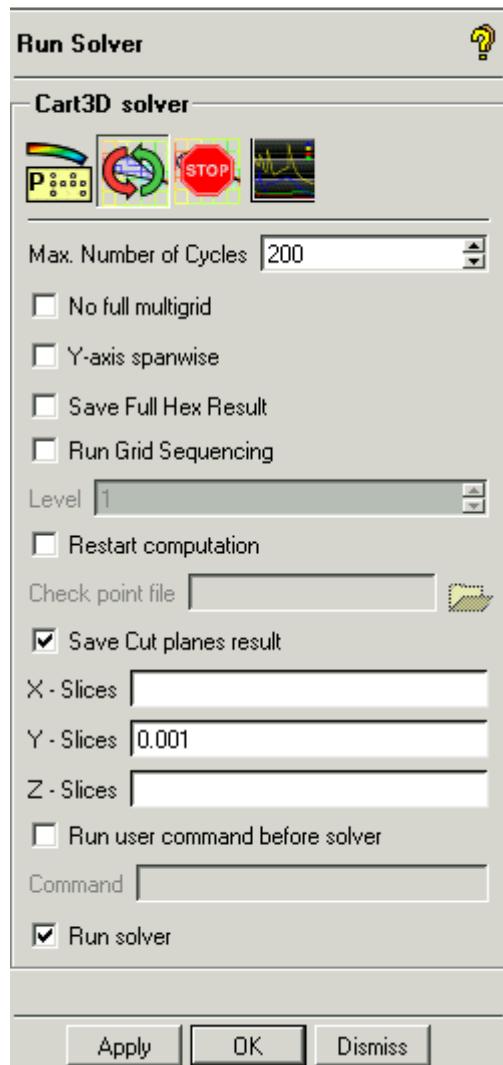


Click on Run solver  to get the Run Solver window as shown.

Specify Max. Number of Cycles as 200.

Enable Save Cut planes result and specify Y-Slices = 0.001 as shown in the figure below and press Apply.

Figure 4-609
Run Solver

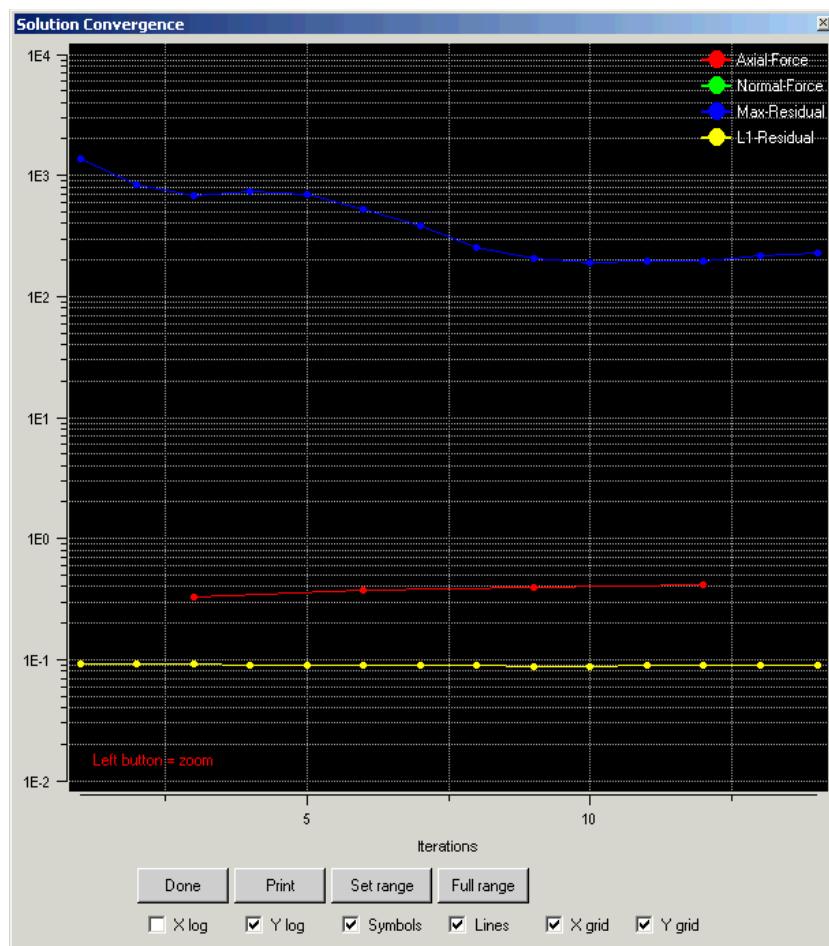


The user can view the convergence by clicking on the Convergence



Monitor to view the plot as shown. (The monitor may open automatically.)

Figure 4-610
Solution
Convergence
Window



f) Visualization of Results

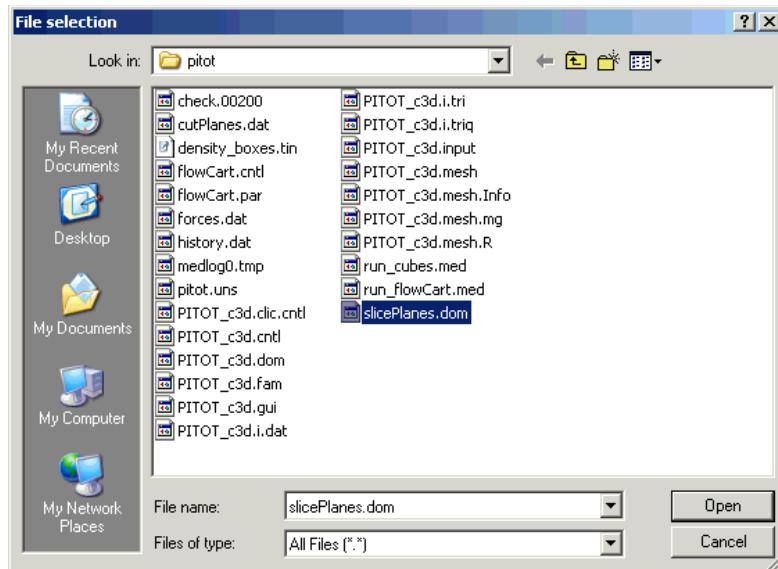
Go to File > Results > Open Results... The Select Result Format window opens as shown here. Select ICEM CFD as the Format.

Figure 4-611
Select Result
Window



Select the file slicePlanes.dom from the **Critical** run as shown in the figure below and press ‘Open’.

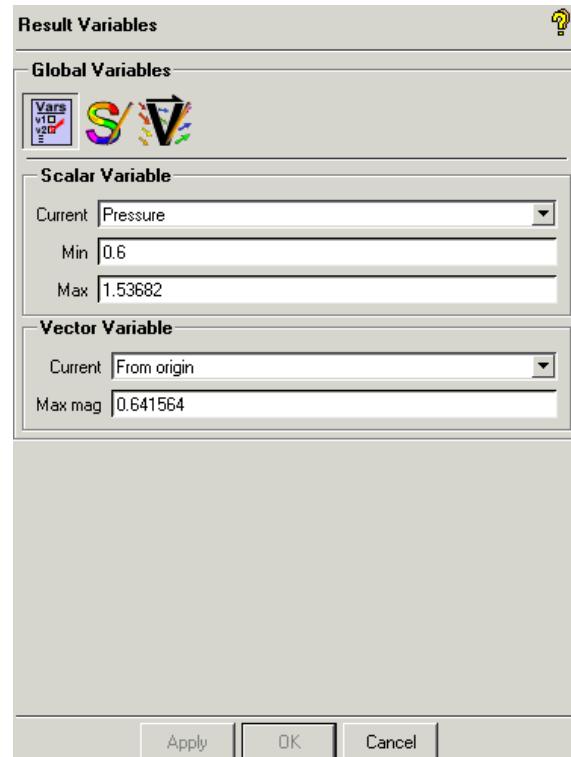
**Figure
4-612
File
Selection**



Press Apply in the Select Result Format window.

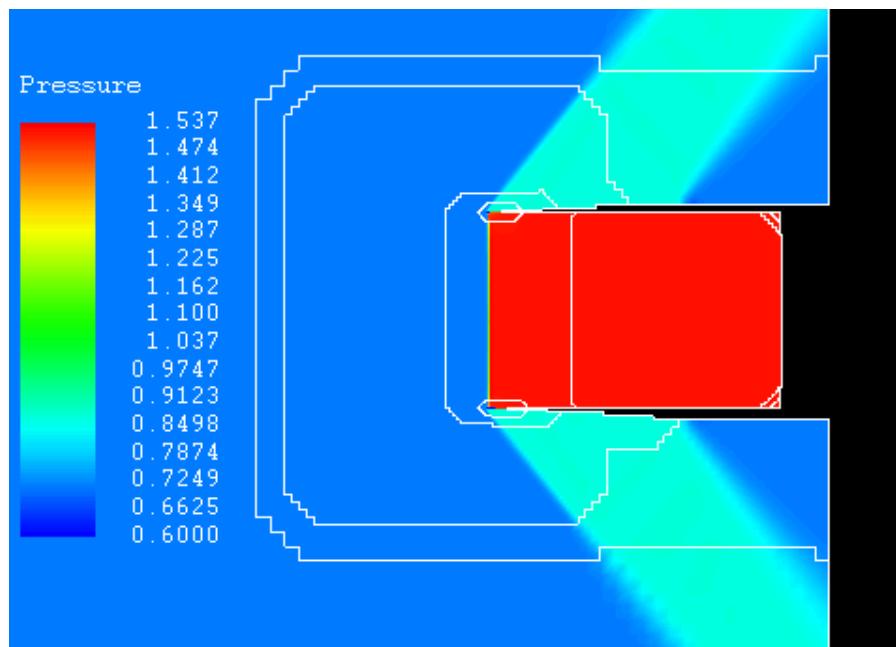
From the Post-processing tab select Variables and Select Variables . In the Scalar Variable panel select Pressure and set Min = 0.6 as shown below.

Figure 4-613
Result Variable Window



Press Apply in the Result Variables window to get the image shown here.

**Figure
4-614
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.

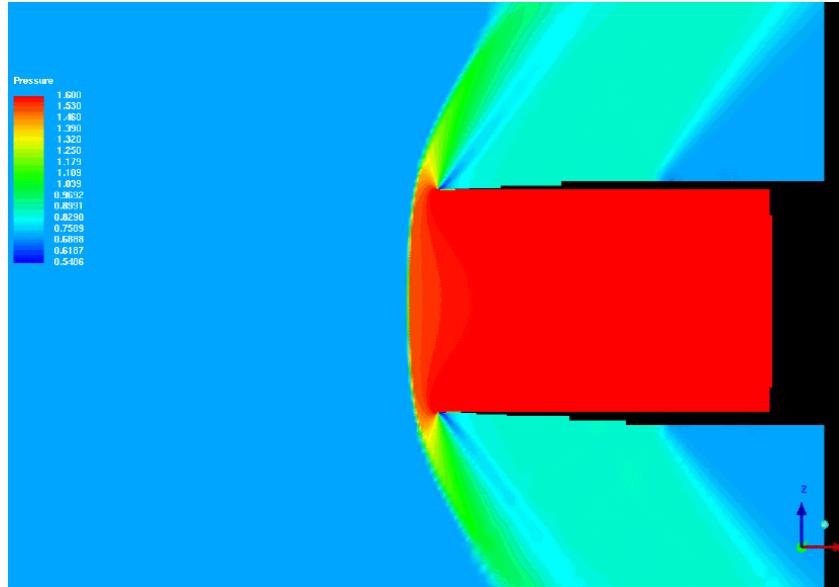
Change the working directory via File > Change Working Dir... and set the location to the folder **Sub-Critical** into which the original files were copied.

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 Cycles = 200.

The Pressure result is shown here. Note the Min and Max values for Pressure in the figure are 0.5406 and 1.600, respectively.

**Figure
4-615
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.

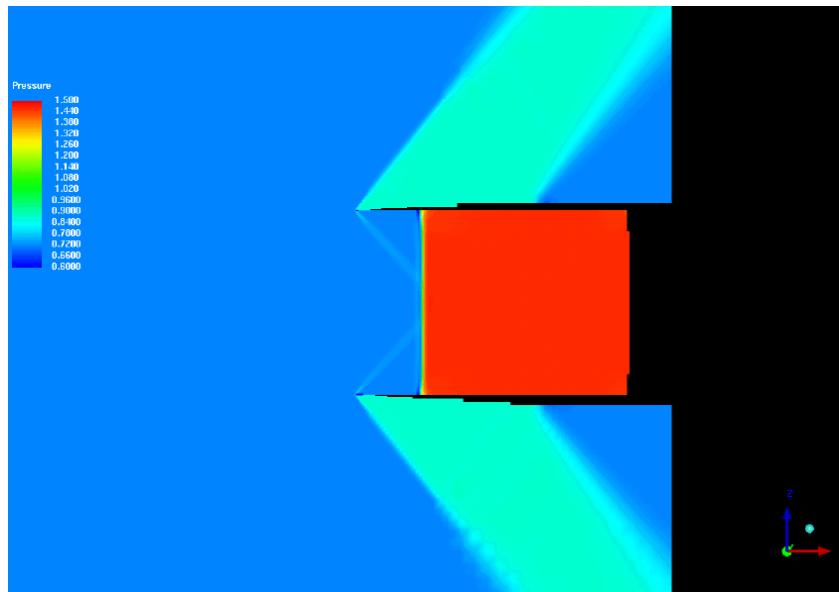
Change the working directory via File>Change Working Dir... and set the location to the folder **Super-Critical** into which the original files were copied.

From the Cart3D menu, select Solver>Define solver params. In Boundary Conditions>Surface BC>INLET set Pressure = 1.42 as shown. 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 Cycles = 200.

The Pressure result is shown below. Note the Min and Max values for Pressure in the figure are 0.600 and 1.500, respectively.

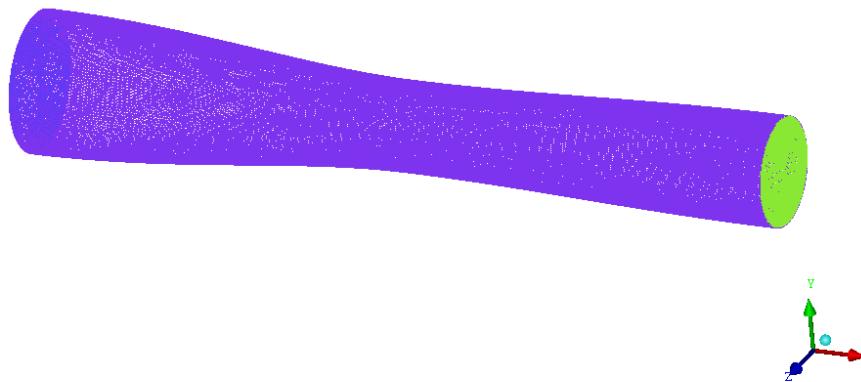
**Figure
4-616
Super
Critical
Post
Processor
Result**



4.7.8: 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, $p_{exit}/p_{total} = 0.89$

Case B: Transonic, $p_{exit}/p_{total} = 0.75$

Case C: Supersonic, $p_{exit}/p_{total} = 0.16$

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 ($x = 0 \text{ in}$), an area of 1.0 in^2 at the throat ($x = 5 \text{ in}$), and an area of 1.5 in^2 at the exit ($x = 10 \text{ in}$). The nozzle Area varies using a Cosine function and has the form:

If $x < 5.0$ then Area = $1.75 - 0.75 * \text{Cos}((0.2 * x - 1.0) * \pi)$.
 If $x \geq 5.0$ then Area = $1.25 - 0.25 * \text{Cos}((0.2 * x - 1.0) * \pi)$.
 This nozzle comes from MS Liou's paper AIAA 87-0355.

a) Starting the Project

The input files for this tutorial can be found in the Ansys Installation directory, under `../docu/Tutorials/CFD_Tutorial_Files/Cart3D_Examples`. Note: It is preferable to create a separate folder **nozzle** and put only **nozzle.uns** (domain file) in that folder before performing this tutorial.



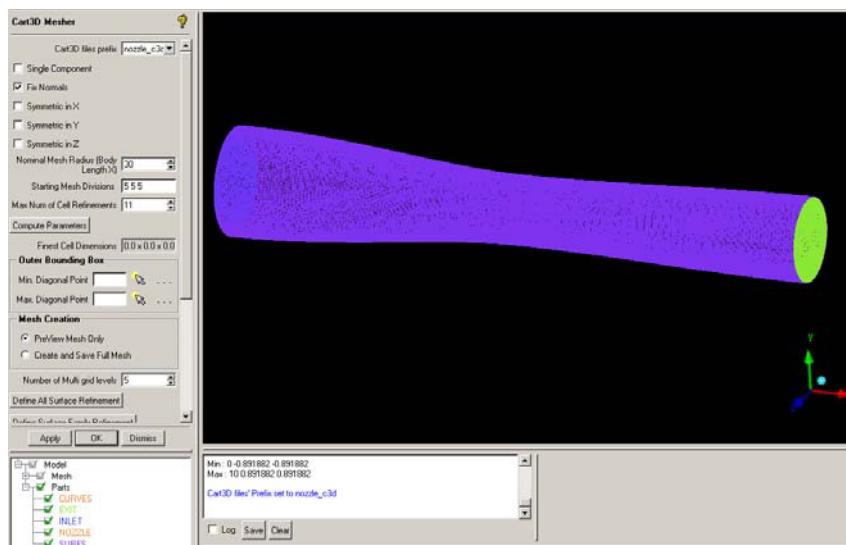
Select Open Mesh from the main menu and select **nozzle.uns**.

b) Mesh Generation Preview Only



Click on Cart3D from the main menu. Select Volume mesher . We get the Cart3D Mesher GUI as shown here.

**Figure
4-617
Cart3D
GUI
window**



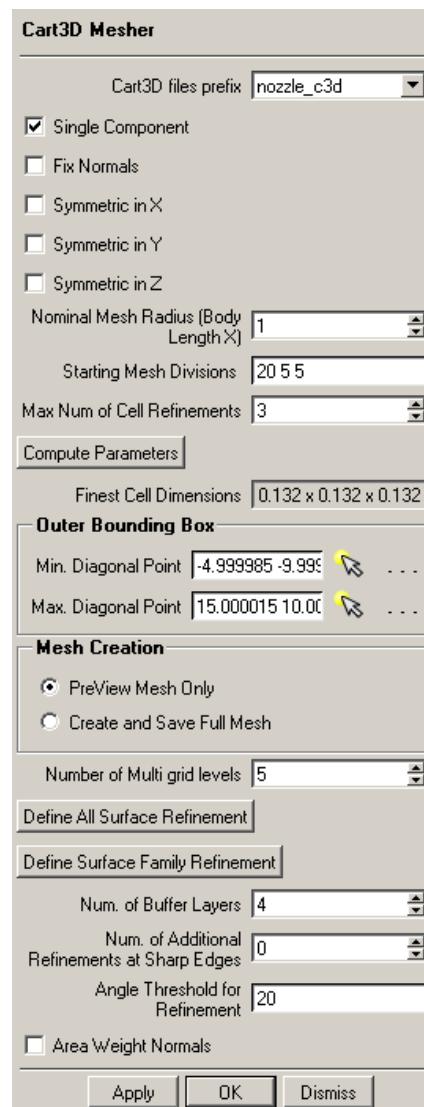
Enable Single Component

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 below.

Figure 4-618
Cart3D Mesh Window

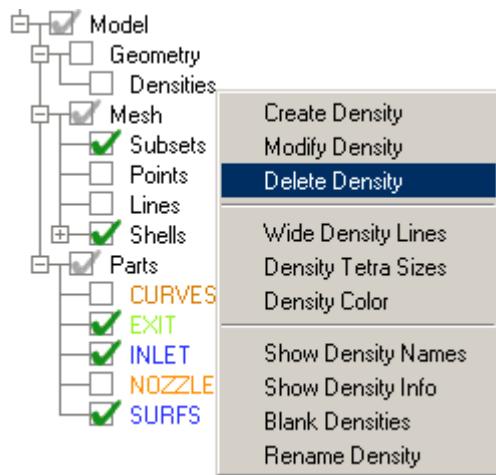


This will create 4 density polygons by default for mesh density control. These can be viewed by enabling Geometry >Densities in the Display Tree.

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 as shown here.

Figure 4-619
Display Tree Delete



The Delete Density panel opens as shown below. When in selection mode, select all the densities with the hotkey ‘a’ on the keyboard and press Apply.

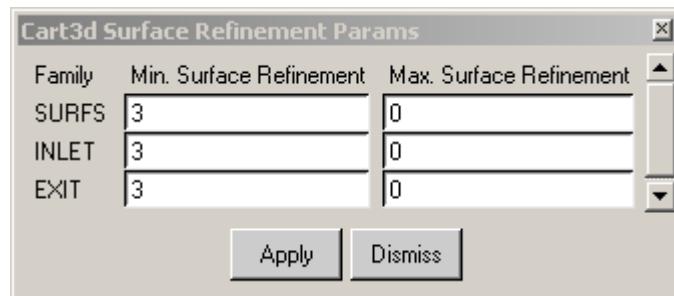
Figure 4-620
Delete Density Window



From the Cart3D menu select Volume Mesher . In the Cart3DMesher window select Define Surface Family Refinement In the Cart3d Surface

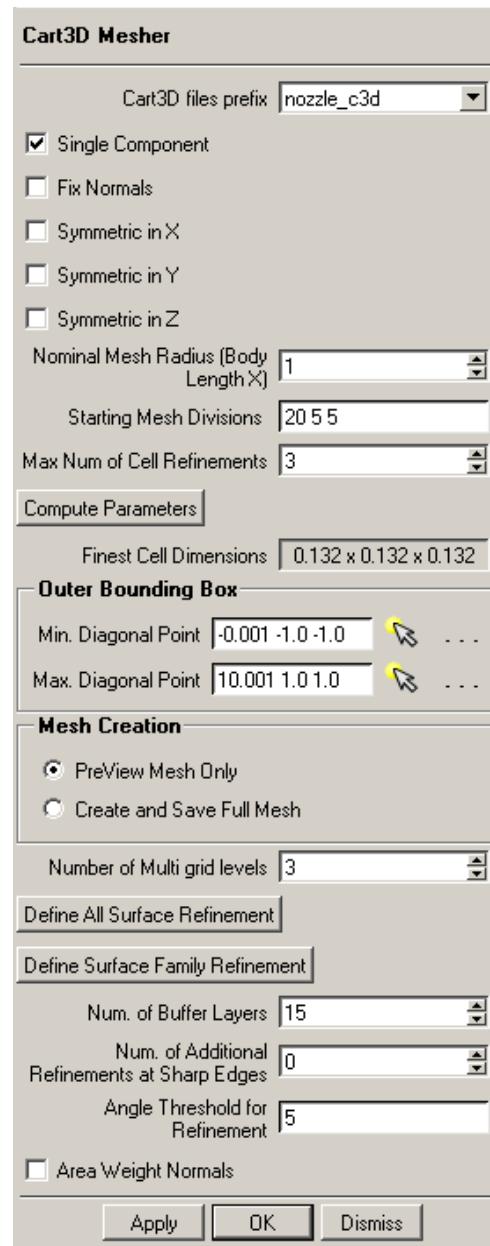
Refinement Params window set Min. Surface Refinement for SURFS, INLET, and EXIT to 3 as shown below. Press Apply and Dismiss.

Figure 4-621
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.

Figure 4-622
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 here.

Figure

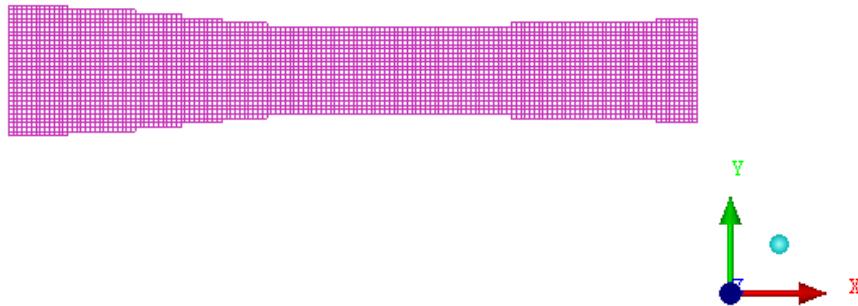
4-623

Cut

Plane

Z2

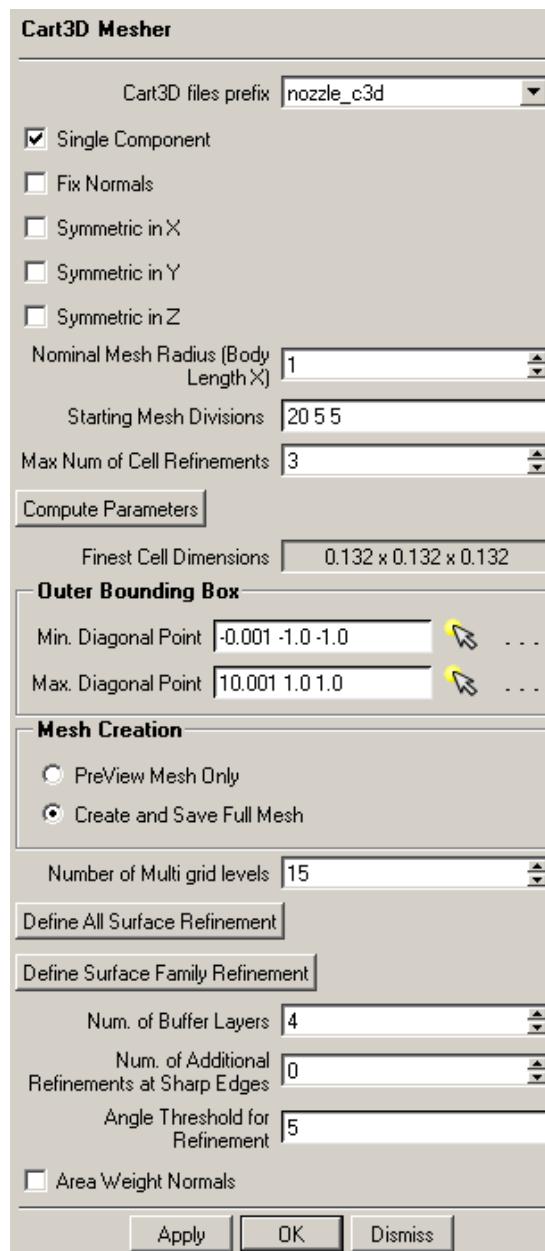
View



c) Mesh Generation Full Mesh

In the Cart3D Mesher window enable Create and Save Full Mesh as shown below and press Apply. This will create 3 levels of coarsened mesh which can be read by the solver.

Figure 4-624
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 $p_{inlet} = 1.0$, $p_{inlet} = 1/\gamma$ 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

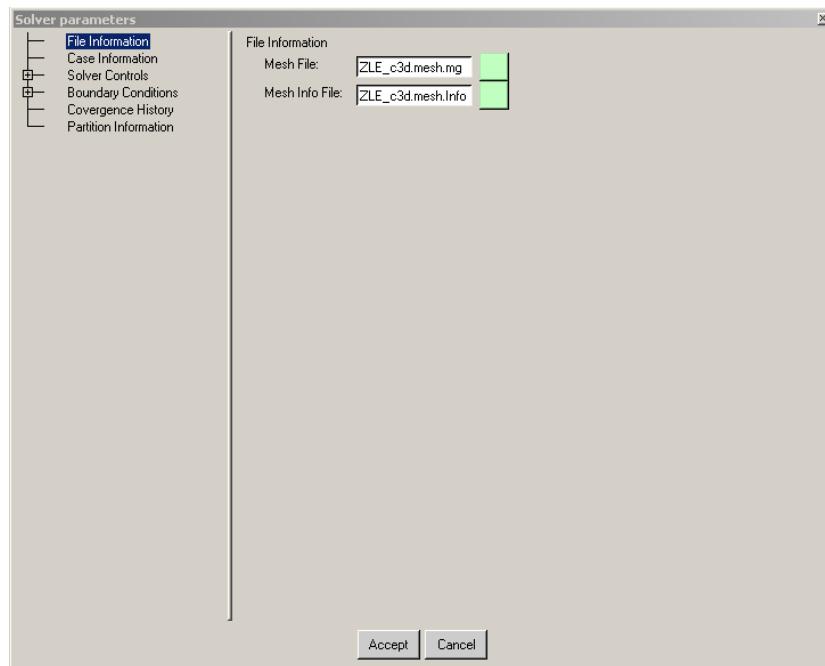
All cases are done using the INLET/EXIT surface Boundary condition for both inflow and outflow. Since there are no cells that get the far-field boundary condition, Mach number, Alpha, and Beta are used to define the initial condition. The subsonic and transonic cases use the subsonic initial condition. 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 will appear. (This window may open automatically.)

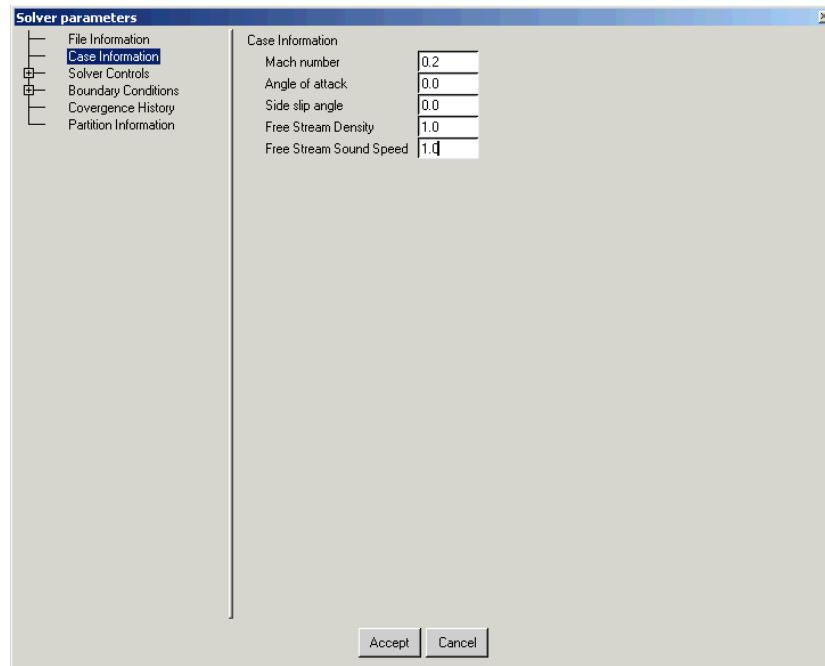
**Figure
4-625
Solver
Parameters
Window**



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.

**Figure
4-626
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

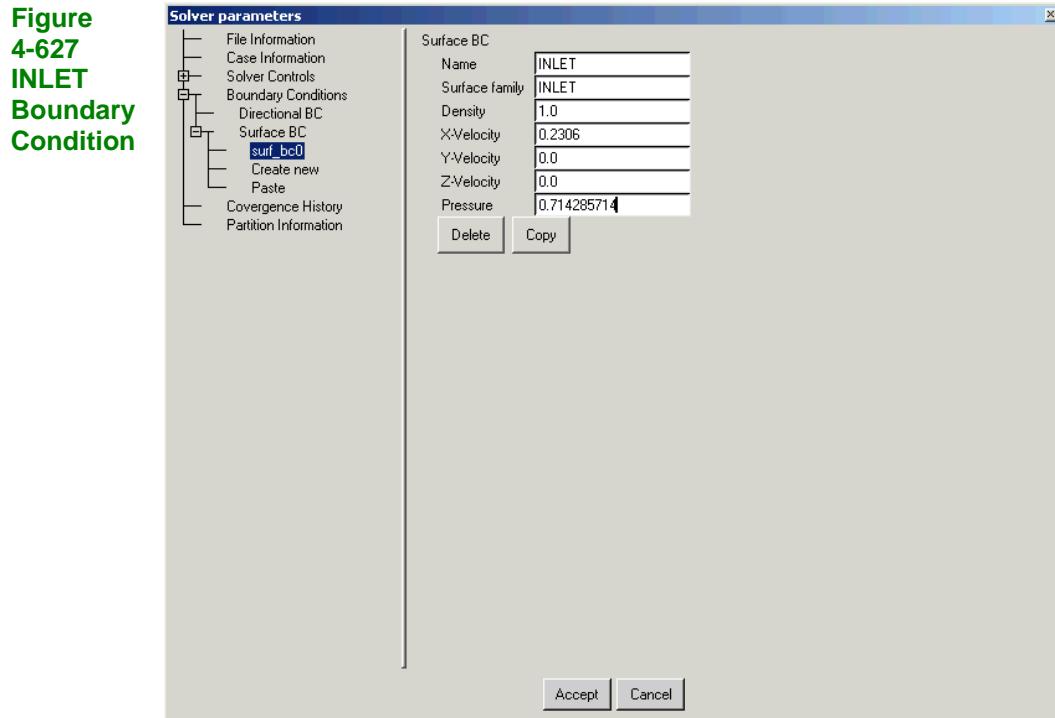
Surface family INLET

Density 1.0

X-Velocity 0.2306 (from exact solution)

Pressure 0.714285714

The values are shown here.



Since the flow is fully subsonic there is no shock wave present in the nozzle and

ISENTROPIC relations hold good.

$$t_{\text{total}} = p * (1 + (\gamma - 1) * M^2 / 2)^{\gamma / (\gamma - 1)} = 0.7412294$$

$$p_{\text{exit}} / p_{\text{total}} = 0.89; \Rightarrow p_{\text{exit}} = .65969403$$

$$p_{\text{inlet}} / (p_{\text{inlet}})^{\gamma} = p_{\text{exit}} / (p_{\text{exit}})^{\gamma}$$

$$\rho_{\text{exit}} = .944801$$

Conservation of mass:

$$\rho_{\text{inlet}} * \text{Area}_{\text{inlet}} * V_{\text{inlet}} = \rho_{\text{exit}} * \text{Area}_{\text{exit}} * V_{\text{exit}}$$

$$M_{\text{exit}} = .41002478$$

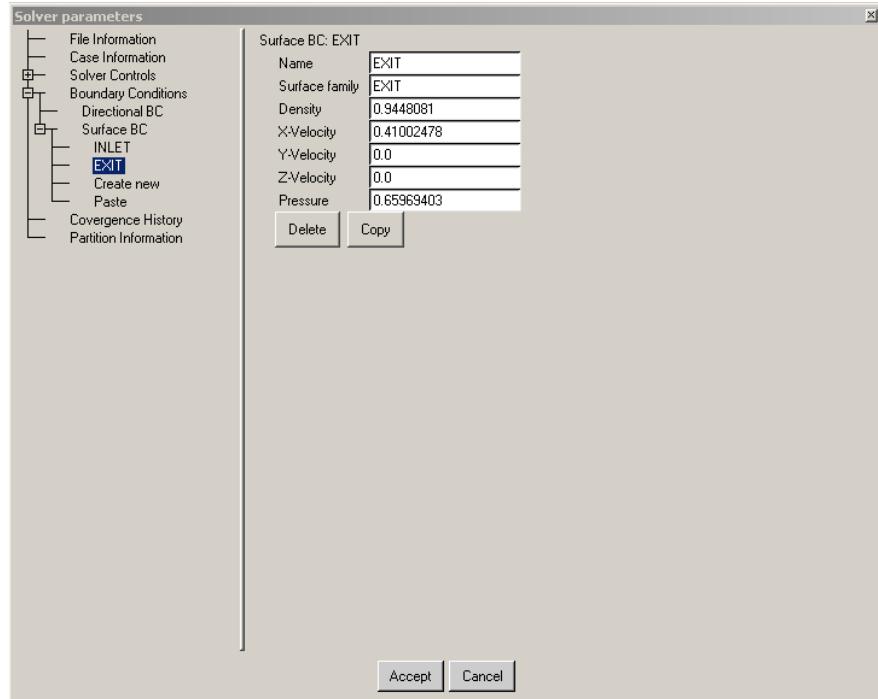
Select Create new and specify the Surface BC for EXIT by entering the following values as shown below.

Name EXIT

Surface family EXIT

Density	0.944801
X-Velocity	0.41002478
Pressure	0.65969403

**Figure
4-628
Exit
Boundary
Condition**



Use the other defaults and press Accept in the Solver parameters window.

f) Running the FlowCart Solver



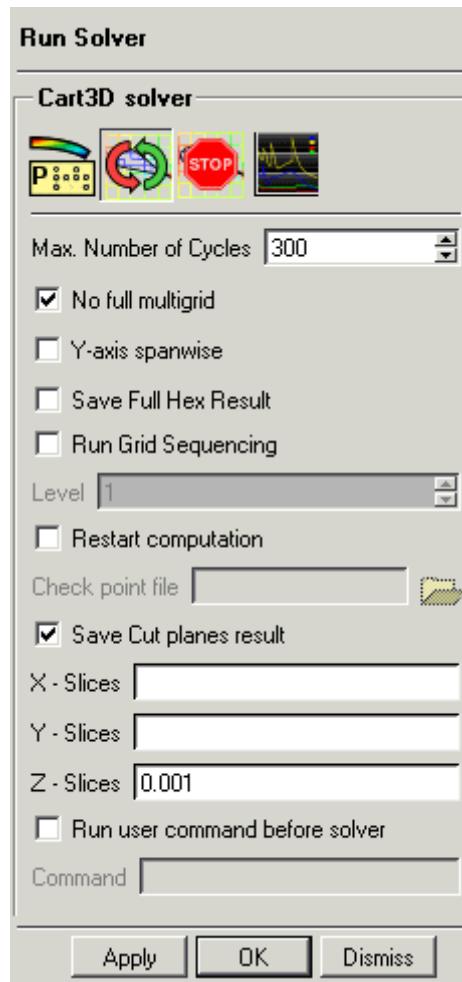
Select Solver > Run solver to open the solver panel.

Specify Max. Number of Cycles = 300.

Enable No full multigrid.

Enable Save Cut planes result, and specify Z-Slices = 0.001. Press Apply.

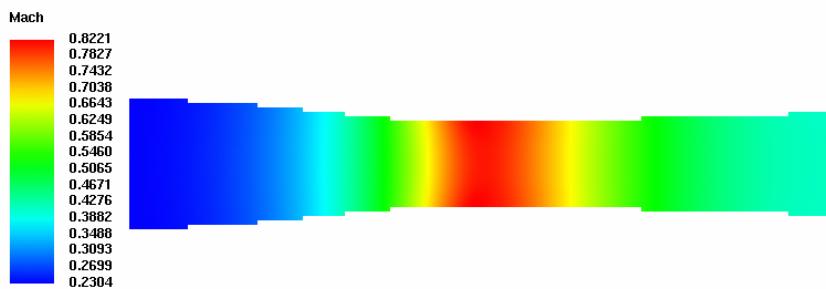
Figure 4-629
Run Solver
Window



Note: Post processing is explained in previous tutorials. Follow the same procedure to view the results.

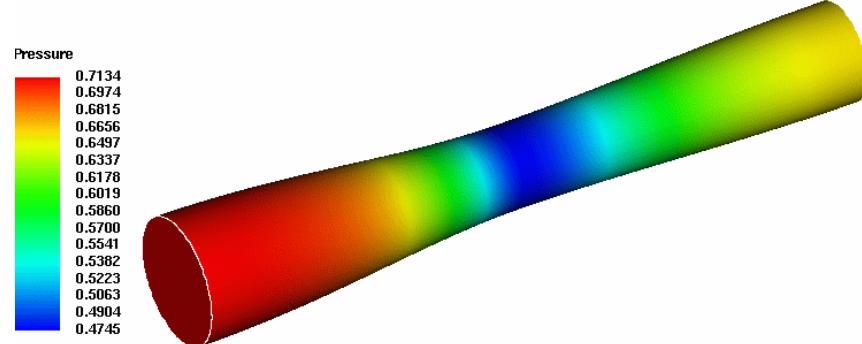
Mach number results in **slicePlanes.dom** for the Sub-Sonic Flow case are shown here.

**Figure
4-630
Sub
Sonic
Result
Mach
Number**



Pressure results in **surface_results.dom** for the Sub-Sonic Flow case are shown below.

**Figure
4-631
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 $1/\gamma = 1/1.4 = 0.714285714$

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.

$$t_{\text{total}} = p * (1 + (\gamma - 1) * M^2 / 2)^{\gamma / (\gamma - 1)} = 0.743390$$

$$p_{\text{exit}}/p_{\text{total}} = 0.75$$

$$p_{\text{exit}} = .55754252$$

From 1-D normal shock relations:

$$\rho_{\text{exit}} = 0.81060$$

Conservation of mass

$$\rho_{\text{inlet}} * \text{Area}_{\text{inlet}} * V_{\text{inlet}} = \rho_{\text{exit}} * \text{Area}_{\text{exit}} * V_{\text{exit}}$$

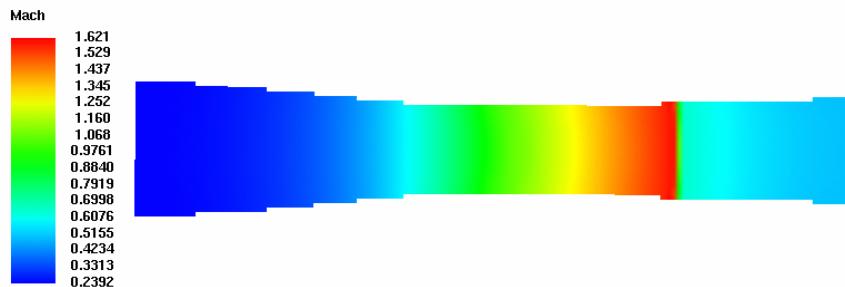
$$M_{\text{exit}} = .492519$$

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.

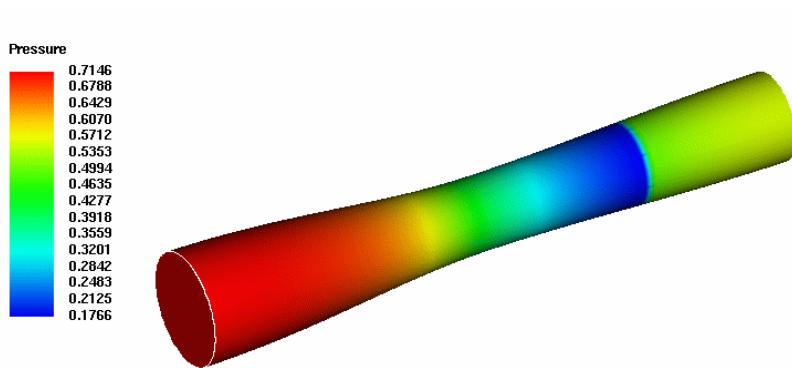
Mach number results from **sliceplanes.dom** for the Trans-Sonic Flow case are shown here.

**Figure
4-632
Trans
sonic
Result
Mach
Number**



Pressure results in **surface_results.dom** for the Trans-Sonic Flow case are shown in the figure below.

**Figure
4-633
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.
 $t_{\text{total}} = p * (1 + (\gamma - 1) * M^2 / 2)^{(\gamma / (\gamma - 1))} = 0.743390$

$$p_{\text{exit}} / p_{\text{total}} = 0.16; \Rightarrow p_{\text{exit}} = 0.1189424$$

$$\rho_{\text{inlet}} / (\rho_{\text{inlet}})^{\gamma} = p_{\text{exit}} / (\rho_{\text{exit}})^{\gamma}$$

$$\rho_{\text{exit}} = 0.278127$$

Conservation of mass:

$$\rho_{\text{inlet}} * \text{Area}_{\text{inlet}} * V_{\text{inlet}} = \rho_{\text{exit}} * \text{Area}_{\text{exit}} * V_{\text{exit}}$$

$$\Rightarrow M_{\text{exit}} = 1.435448$$

A change in the EXIT boundary condition needs to be incorporated.

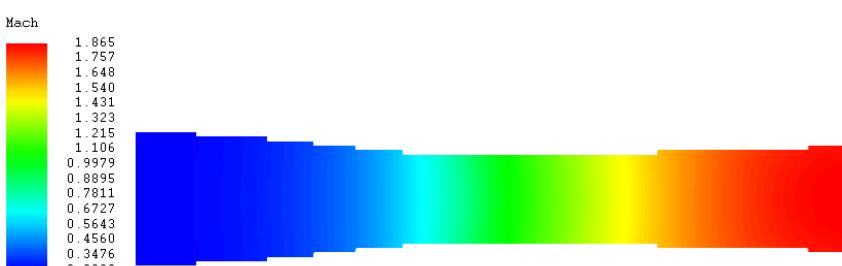
Expand Boundary Conditions>Surface BC

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 here.

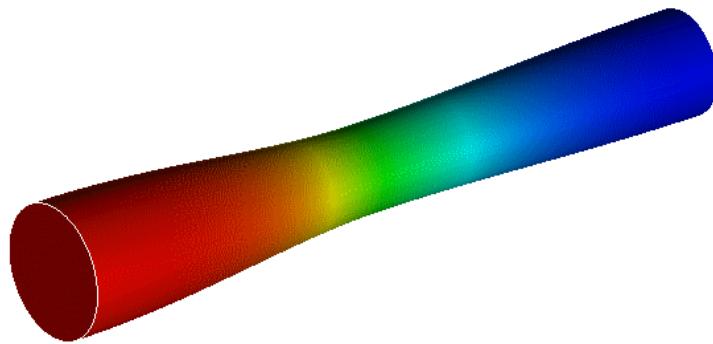
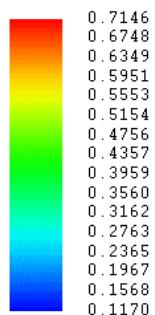
**Figure
4-634
Supersonic
Flow Mach
Number**



Pressure results in **surface_results.dom** for the Supersonic Flow case are shown below.

**Figure
4-635
Super
Sonic
Flow
Pressure**

Pressure

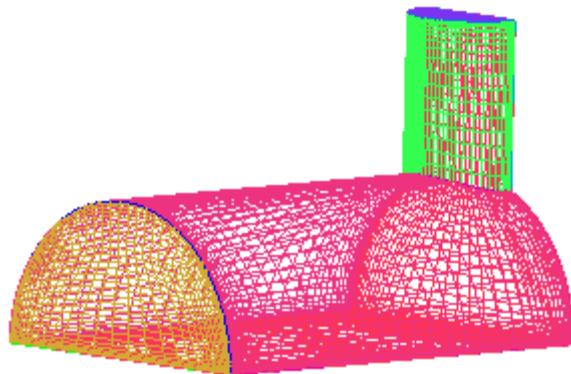


4.8: Output to Solvers

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 4-636

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 conditions.
- 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.ansys.com/products/icemcfoutput-interfaces.asp> 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.

4.8.1: Unstructured Mesh

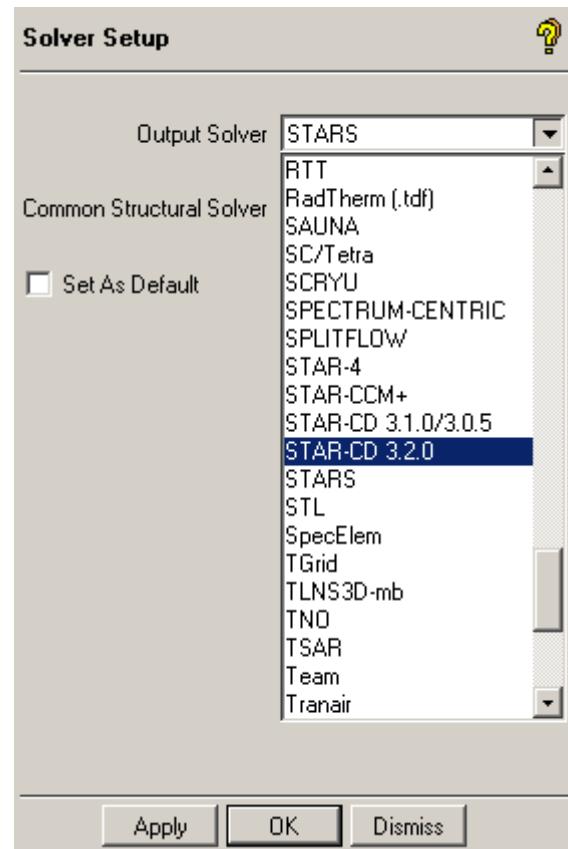
The input files for this tutorial are found in the Ansys Installation directory, under/docu/Tutorials/CFD_Tutorial_Files. Copy the 3DPipeJunct files to your working directory. Choose File > Open Project and, from the File selection window, choose 3DpipeJunct and press Accept.

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 here

Figure 4-637
Select STAR-CD



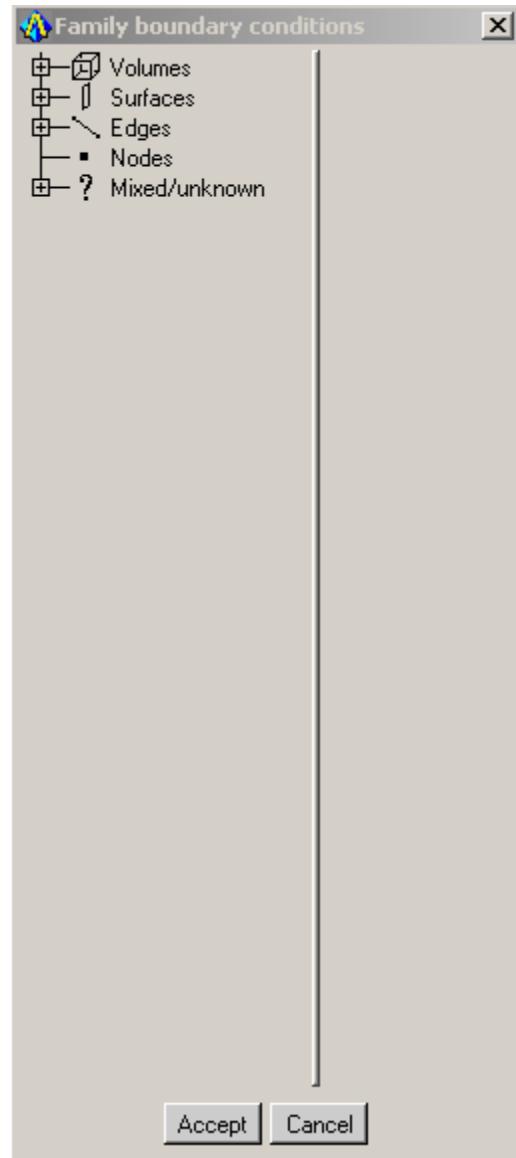
From the Selection window, select STAR-CD 3.2.0 and then press Okay.

b) Editing Boundary Conditions

With the solver set, the user can browse and set solver-specific boundary conditions in the Mesh Editor.

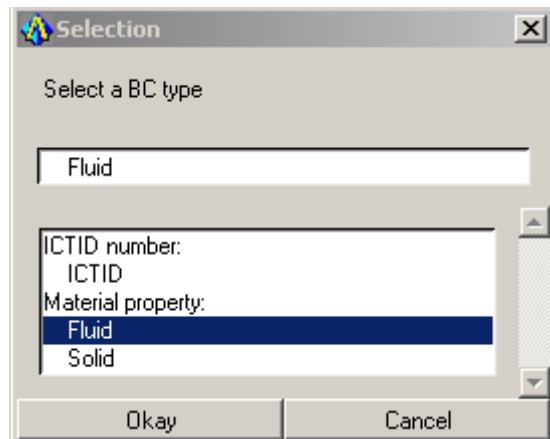
Press Output > Boundary conditions . This will bring up the Family boundary conditions window as shown here.

Figure 4-638
The family boundary condition window



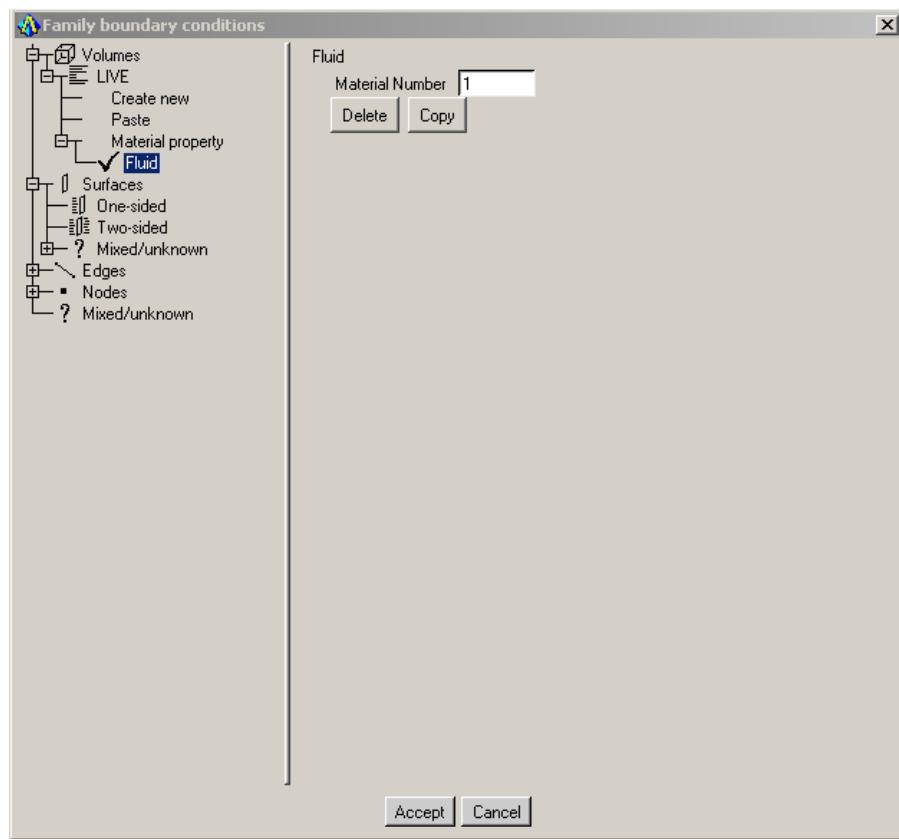
In the Family boundary conditions window, select Volumes > LIVE > Create new. This will open up a window to select the type of cells you have. Select Fluid BC type in this window.

**Figure 4-639
Select the FLUID BC
to LIVE family**



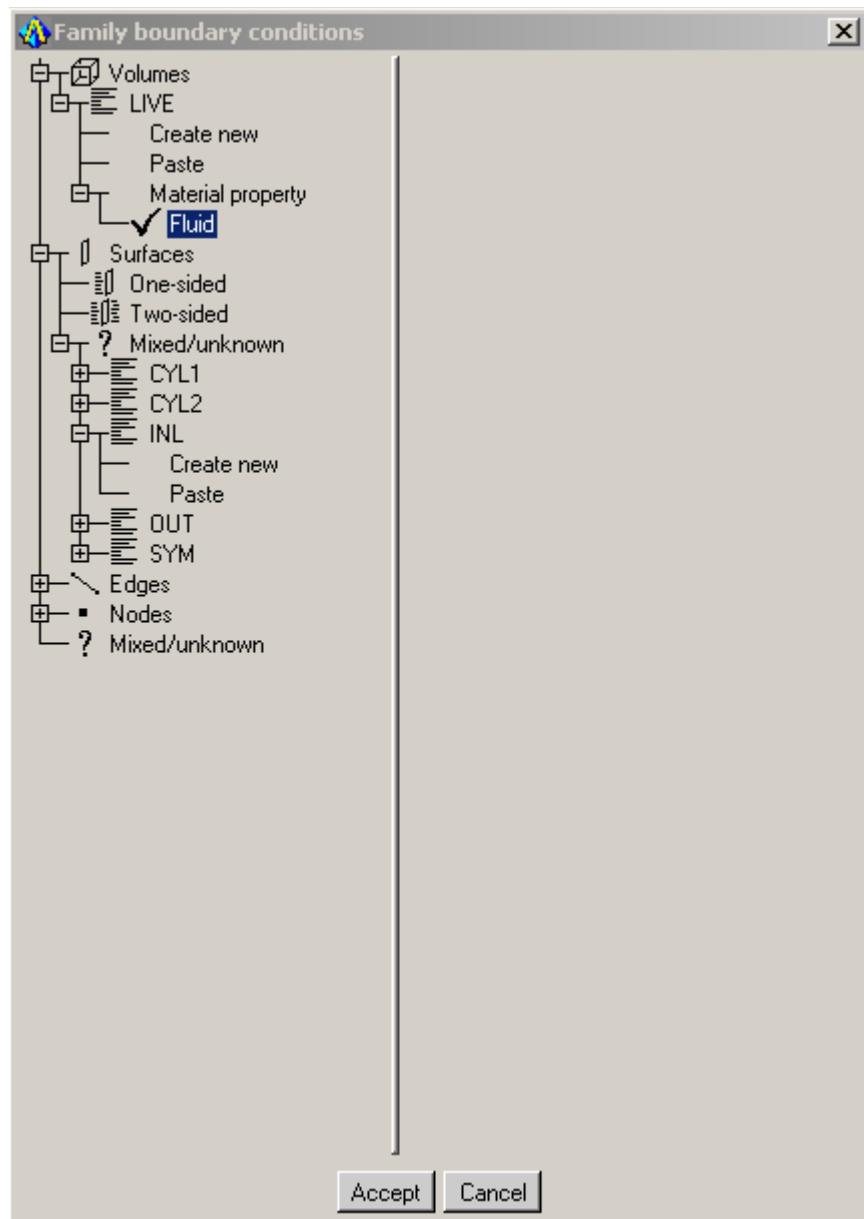
Press Okay and this should change the Family Boundary conditions window as shown below.

**Figure
4-640
After
defining
cell
type**



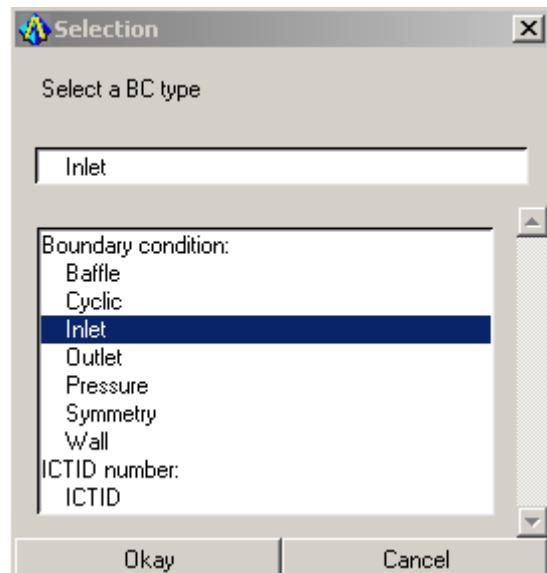
Now, from the Selection window, select Surfaces>Mixed/unknown > INL > Create new as shown in the following figure.

**Figure
4-641
Defining
INL
family
type**



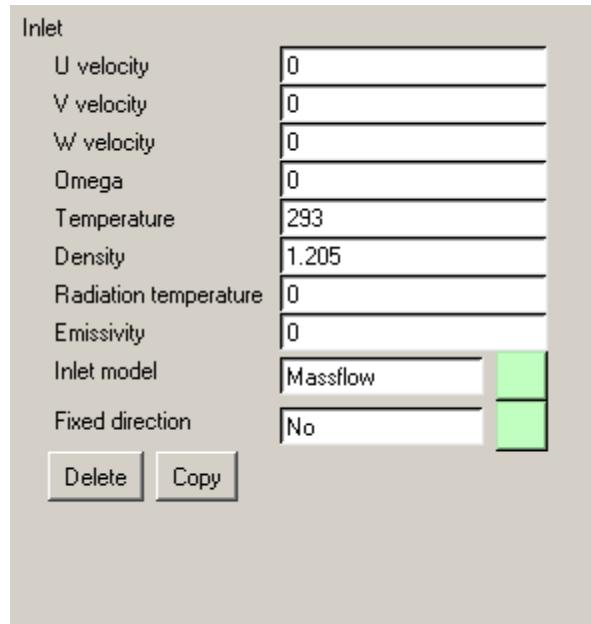
This will open up the bc selection window where select the BC type as Inlet as shown in the following figure.

Figure 4-642
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 here and then press Accept.

Figure 4-643
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.

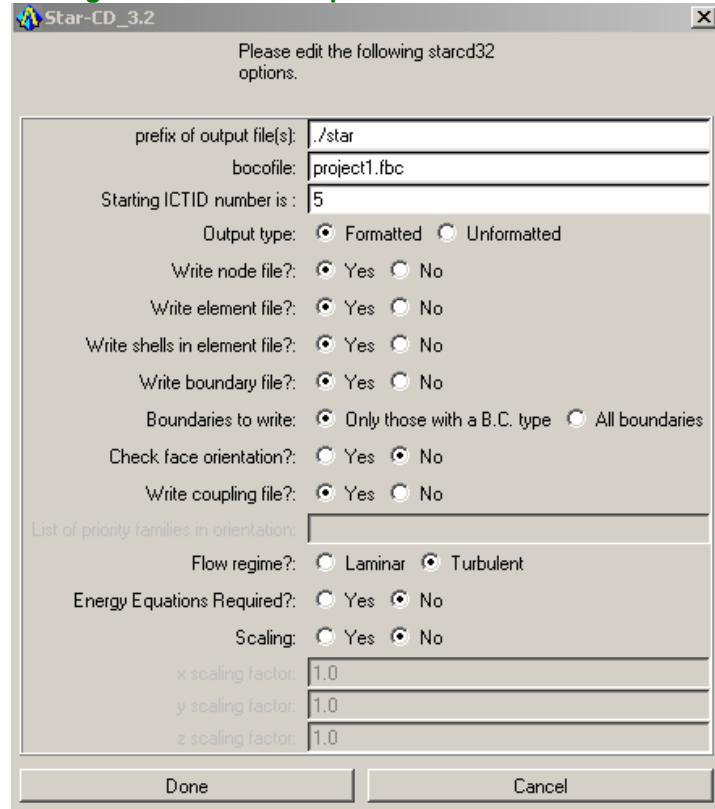
Some solvers accept input files that store information (solver execution, etc.) unrelated to ICEM CFD grid information. If the user's solver is one of those for which we support output of this additional file, you can set up the solver parameters in this file by selecting the params button adjacent to the Boundary conditions button.

c) Writing the Solver Input File

Next, choose Output > Write Input (double click on the icon to open). Select the project saved earlier. It will open a window as shown below, Star-CD_3.2, 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 as shown here.

Assign the parameters for the STAR-CD input file set. Make sure that the boundary condition file is selected as family_boco.fbc.

Figure 4-644
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 ICEM CFD window. The remainder of this section deals with writing output files for structured mesh.

4.8.2: Structured Mesh

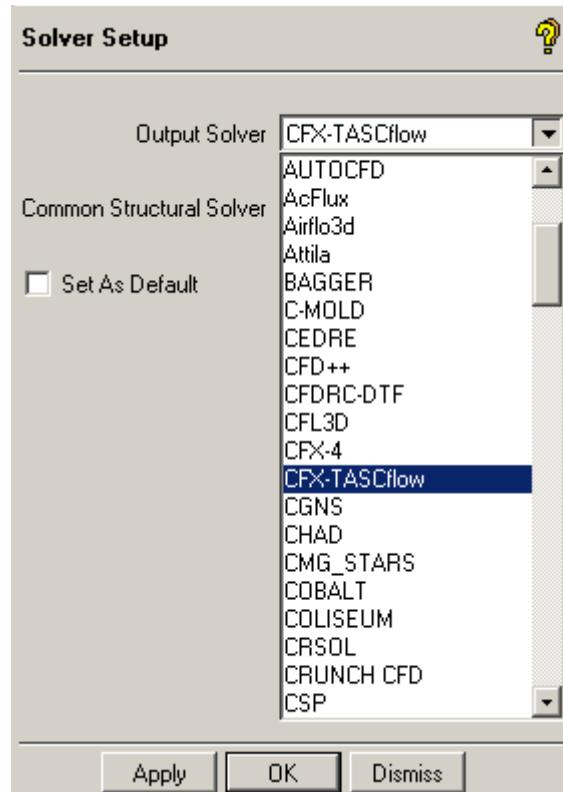
The input files for this tutorial are found in the Ansys Installation directory, under/docu/Tutorials/CFD_Tutorial_Files. Copy the 3dpipeJunct files to your working directory. Go to File > Open > Project and, from the File selection window, choose 3DpipeJunct. Press Accept. Load the Tetin file geometry.tin 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 4-645
Select your second solver, CFX-TASCflow



From the Selection window, select CFX-TASCflow. Press Okay.

b) Solver-Specific Boundary Conditions

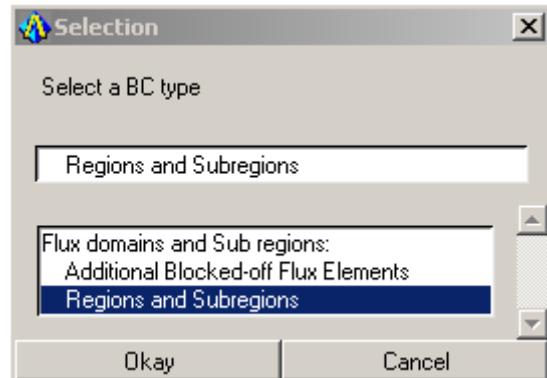
With the solver set to CFX-TASCflow, the user can define regions that CFX-TASCflow will recognize.

Press Output > Boundary conditions . This will bring up a file selection window to select an existing boundary condition file. This will bring up the Family boundary conditions window.

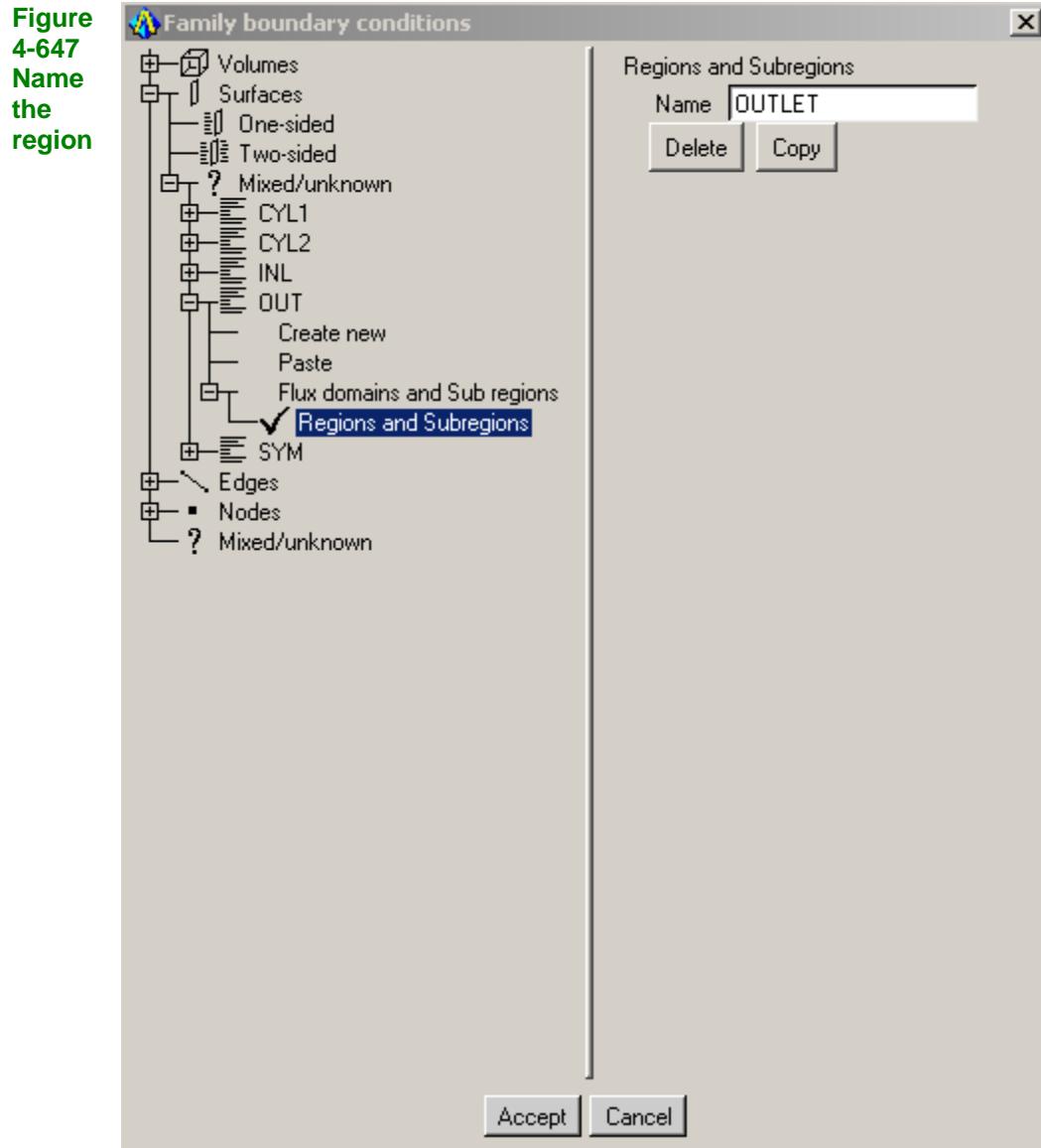
From the Family boundary conditions window select Surfaces > Mixed/unknown > OUT > Create new. This will open up a window to

choose the Boundary Condition type. Double-click Regions and Sub regions under Flux domains and Sub regions.

Figure 4-646
Creating a region on
the OUT family



Name this region “OUTLET” as shown here.



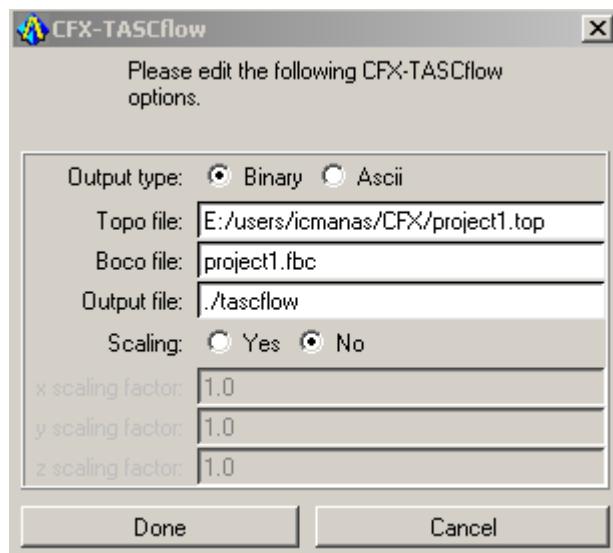
When the user has finished defining the desired regions, press Accept from the Family boundary conditions window.

Select File > Save Project.

Finally choose Output > CFX-TASCflow input.

Set the parameters for input to the solver, as shown below.

Figure 4-648
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.

4.9: Post Processing Tutorials

4.9.1: 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

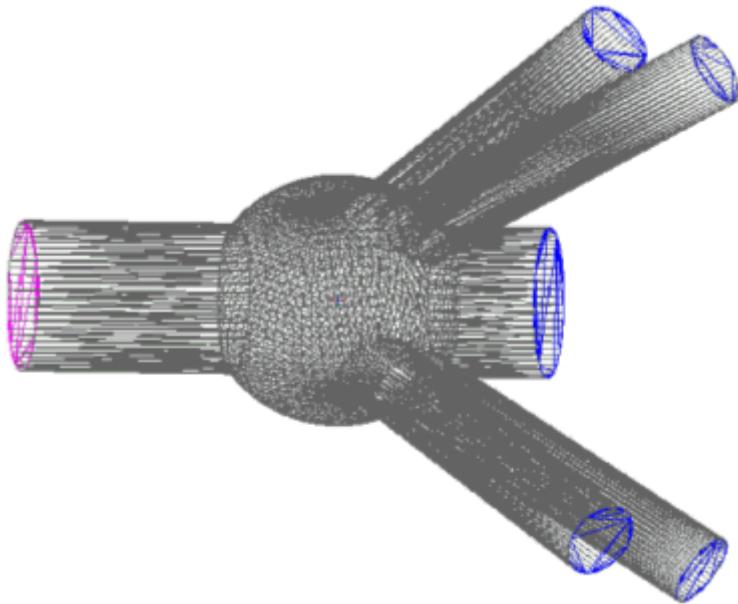
The input files for this tutorial is found in the Ansys Installation directory, under .../docu/Tutorials/Visual3_Files/Visual3_Examples/Pipe_Network.

Copy these files to your working directory.

a) Case Description

The geometry of the Fluent file used for this tutorial 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.

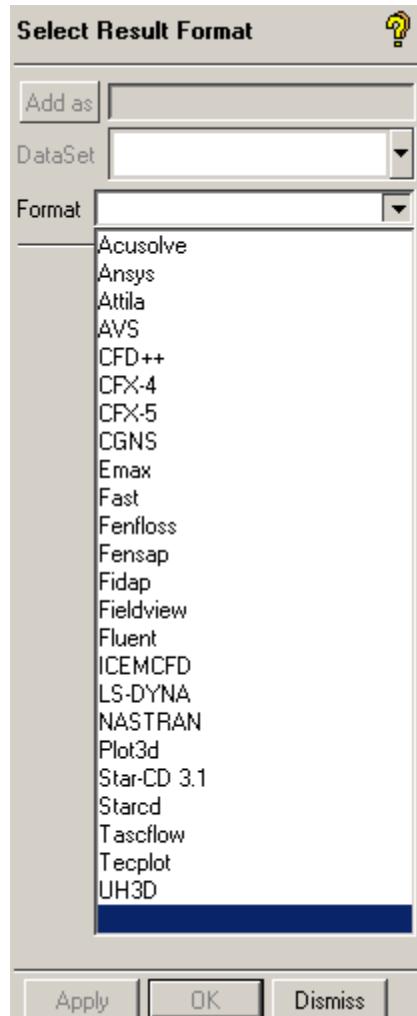
**Figure
4-649
Geometry
for the
Fluent
file**



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 where the user can select different solver formats for which user wants to do post-processing of the results.

Figure 4-650
Select Result Format
window

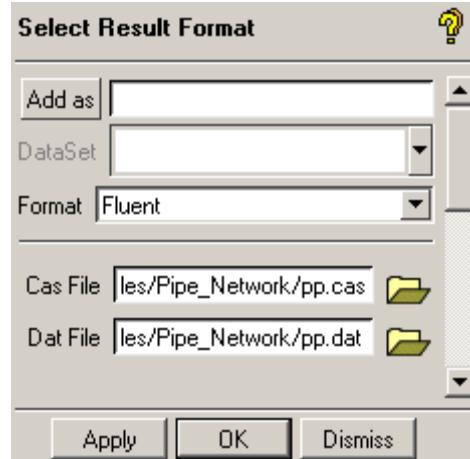


c) Reading Fluent Files

From the **Select Result Format** window, select Format as **Fluent** and press **Apply**. This will pop-up Fluent files selection window as shown below. Through this window, browse the directories and select the **pp.cas** file with the square button towards right side of the Cas file option.

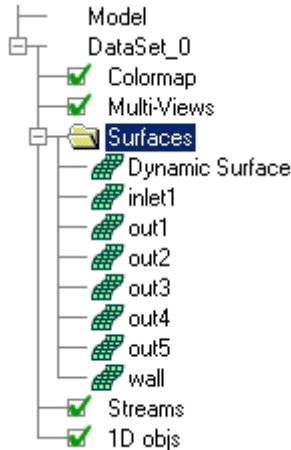
Similarly, read **pp.dat** as the data file for Dat file. Finally, press **Apply** from Fluent files window. This will load the fluent case in the main window.

Figure 4-651
Reading Fluent Case and Data file



The boundary names (Family names) are read in by ANSYS ICEM CFD for easy post-processing. These names are organized in the model tree of the data set.

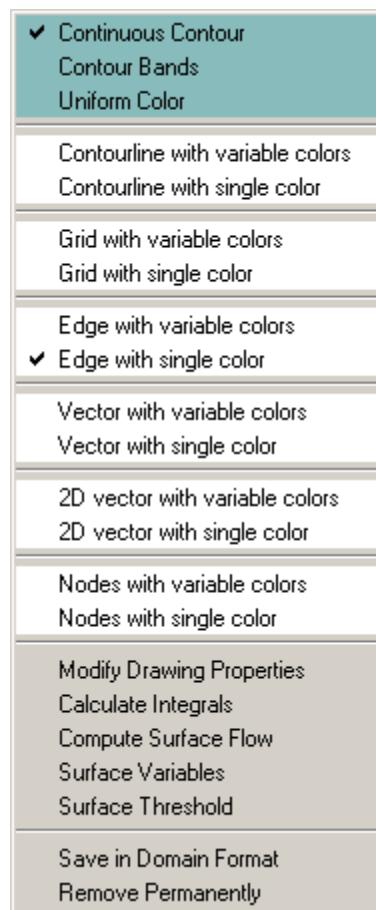
Figure 4-652
Surface Manager under Display Tree



d) Visualization of Surface Grid and Solid Contours

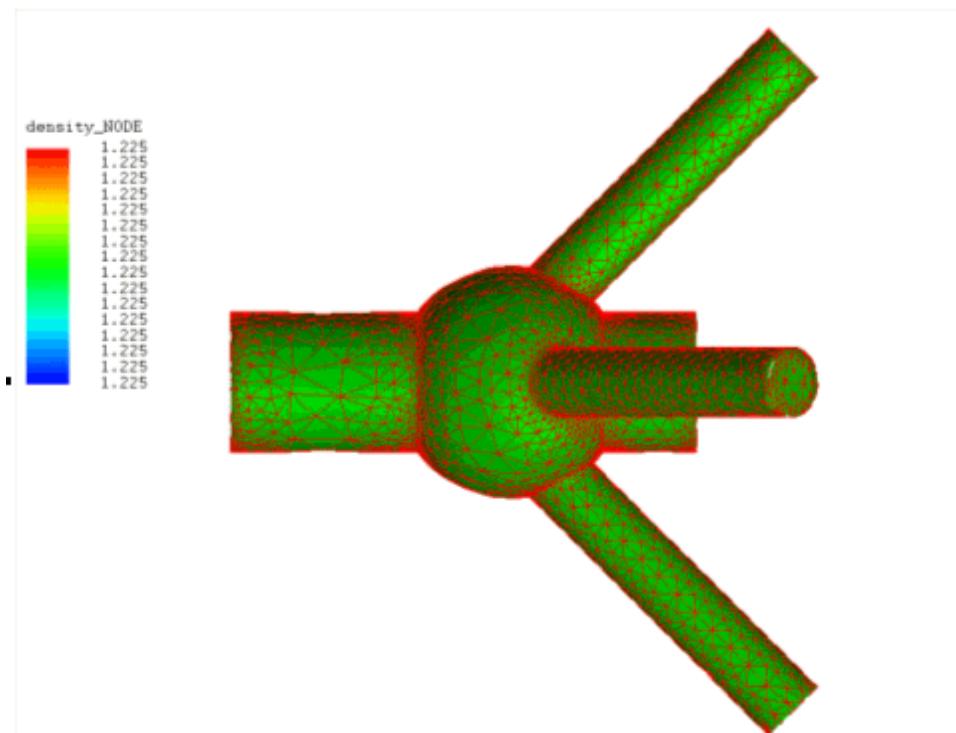
For post processing, the display controls are provided for individual surfaces. User can access those options from the entry to the surfaces in the Display Tree. To access these options, user has to click right hand mouse button after selecting the Surface from the tree. To do changes globally, user has to select “Surfaces” in the tree and click right mouse button on it. The different options to control the display of the result variable are listed here.

Figure 4-653
Right click options on
Surfaces branch



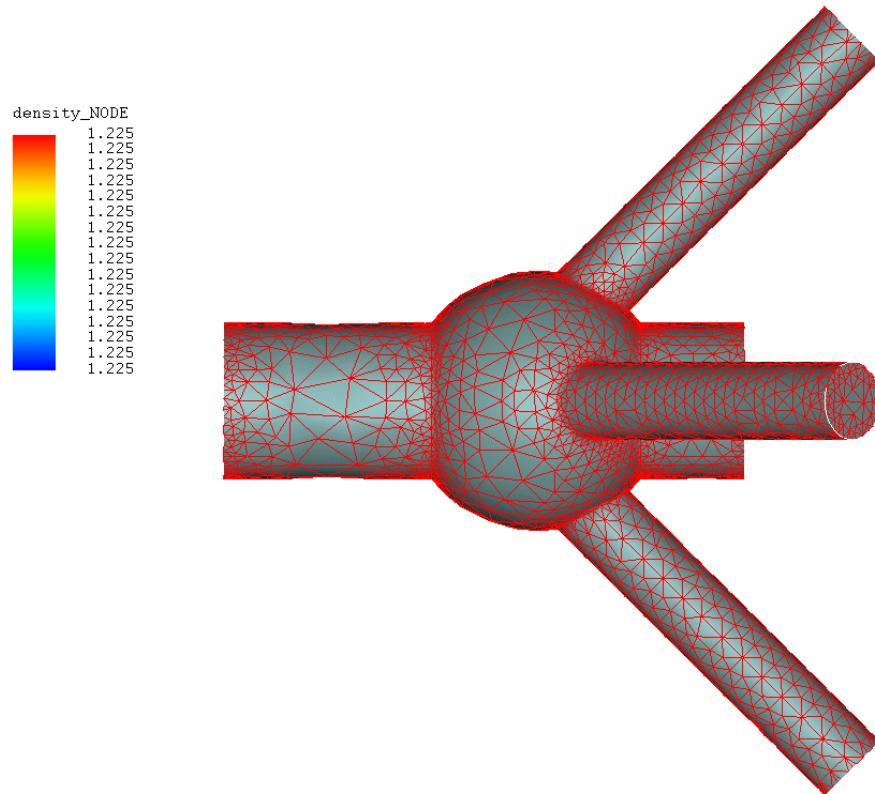
Select the option Grid with variable color from the pop-up menu list. The surfaces would now be displayed with the mesh in scalar variable color. The user would find the mesh in one color since the default scalar variable density_NODE is not changing in this example.

Figure 4-654
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 here.

Figure 4-655
Surface displayed with mesh and Solid color



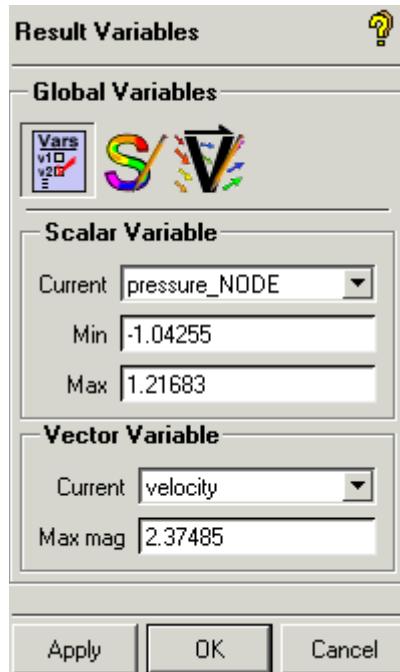
Switch OFF the Grid with variable colors and Uniform colors options by selecting them again.

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. This will open the Result Variables window shown below.

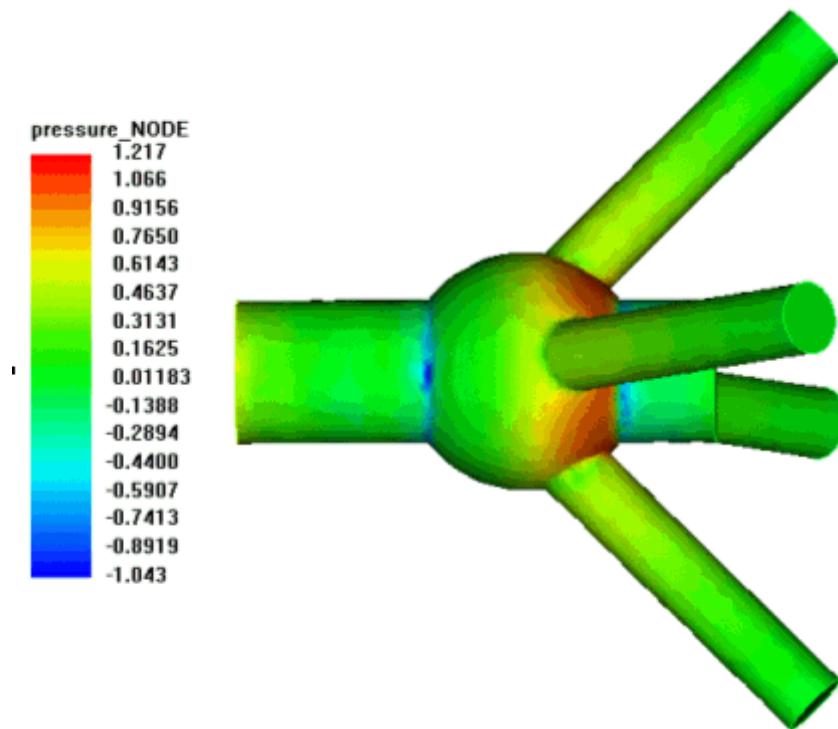
Figure 4-656
Variables
window



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 here.

Figure 4-657
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. From this window, click on Use flat shading button to display the result.

Figure 4-658
Surface Properties
panel

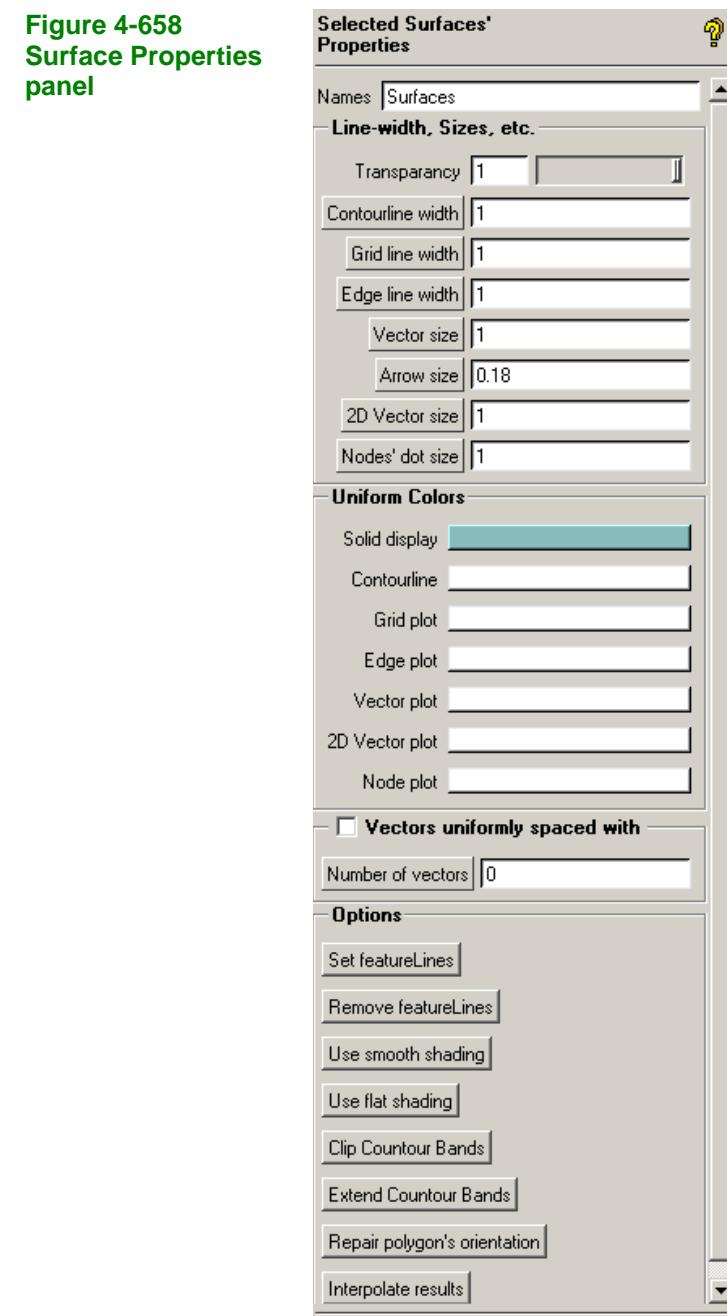
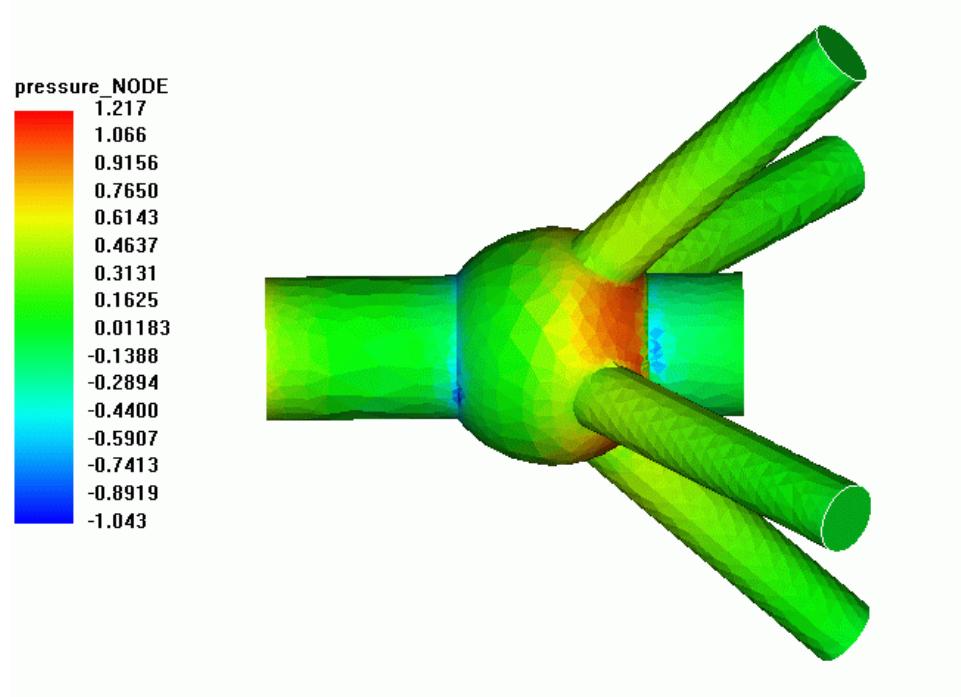


Figure 4-659
Solid contours with Flat shading



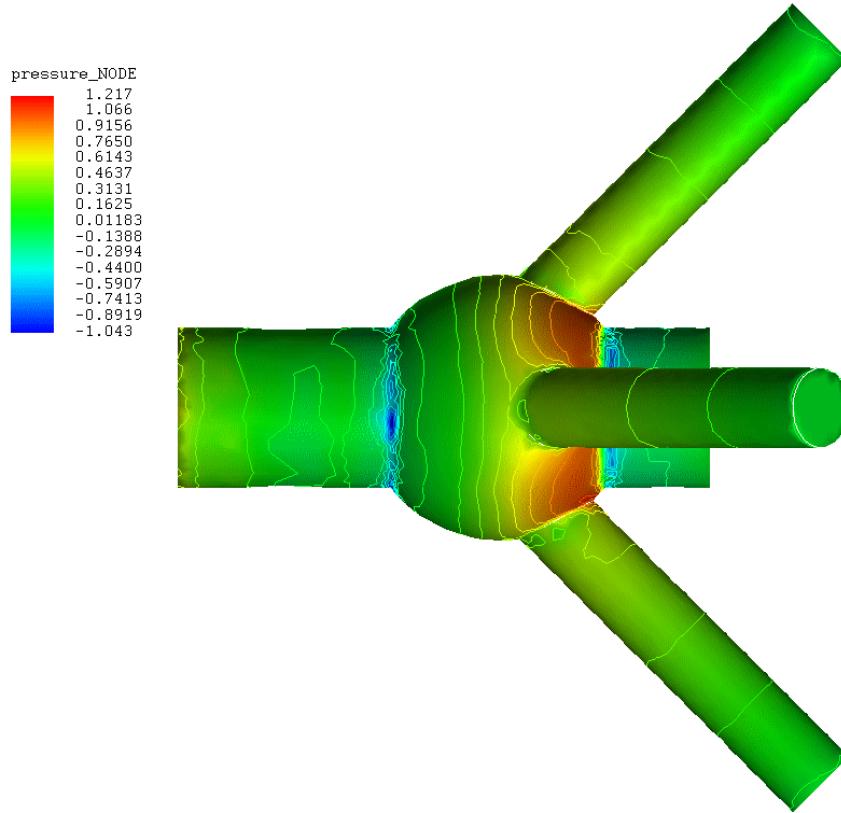
f) Displaying surfaces with contour lines

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.

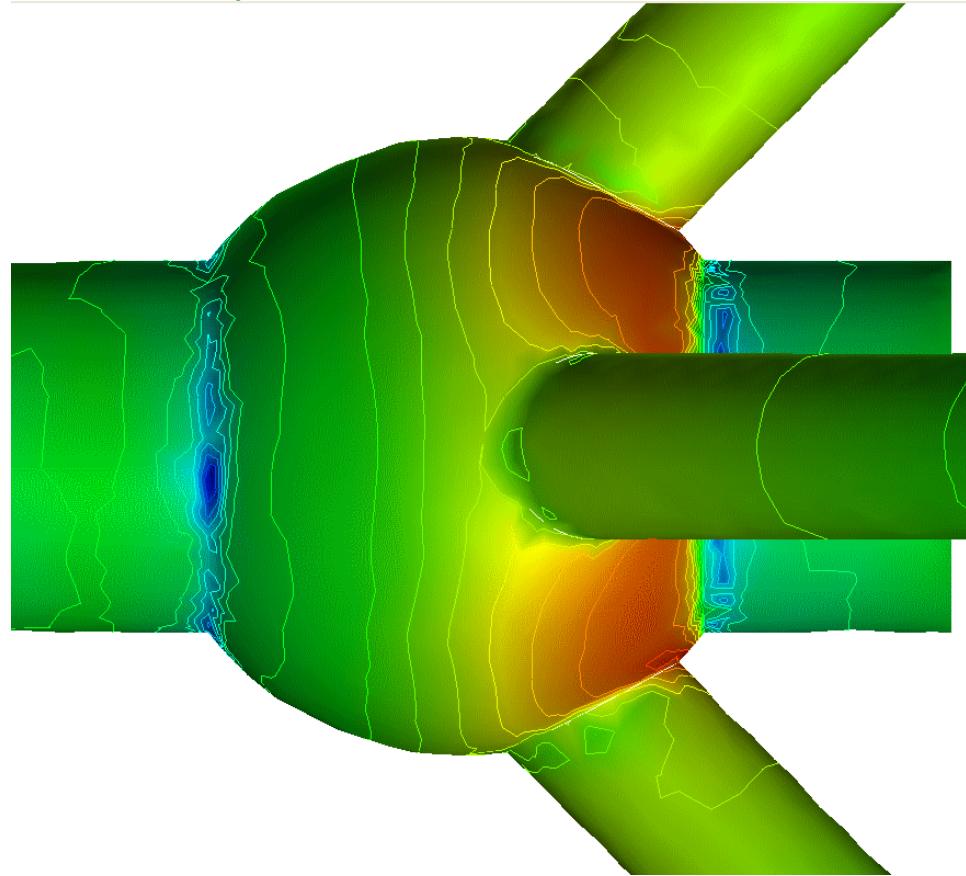
Select the option Contour line with variable colors for the Surfaces in the Display Tree. The contour bands on the surface will look like the figure below.

Figure 4-660
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 figure below is zoomed in on the spherical region. The user can use right mouse button to zoom into the region required.

Figure 4-661
Zoomed view of spherical location



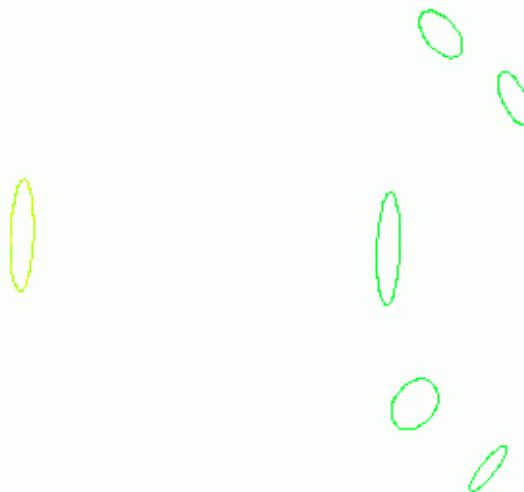
Toggle OFF the Contourlines with variable colors as well as Continuous Contour option.

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. Boundaries

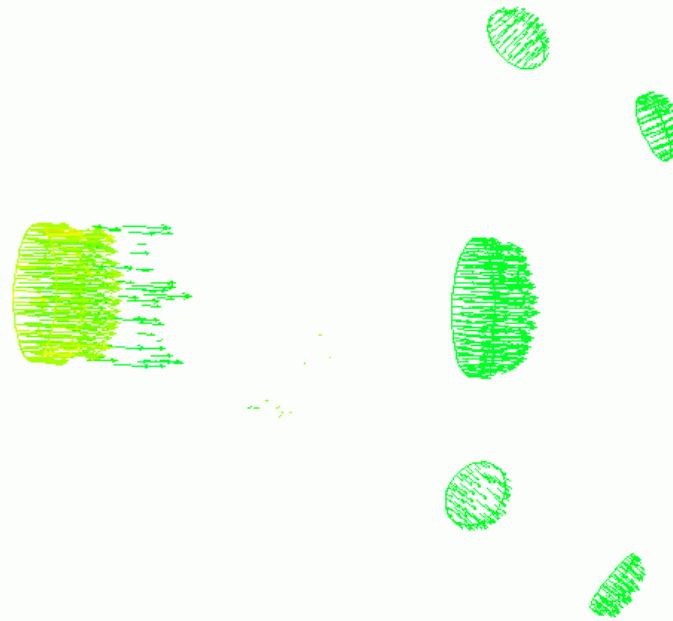
are sorted out from the feature lines provided by the Fluent case and data files.

**Figure
4-662
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 here.

**Figure
4-663
Surface
displaying
Vector
with
variable
color**



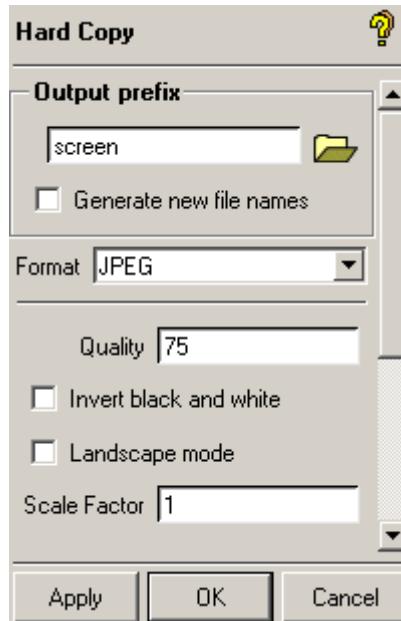
Note: The direction of the arrows corresponds to the currently selected vector variable and the colors of the vectors indicate the present scalar variable. The length of the vector arrows is relative to the magnitude of the vector variable.

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.

Figure 4-664
Output Window



Select the Format as JPEG and click on Apply. This will save the image in the current directory User can view the image file in any image viewer software later.

4.9.2: 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.

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	759
------------------------	--	-----

Load the case file pp.cas and the data file pp.dat.

b) Point Probing Technique

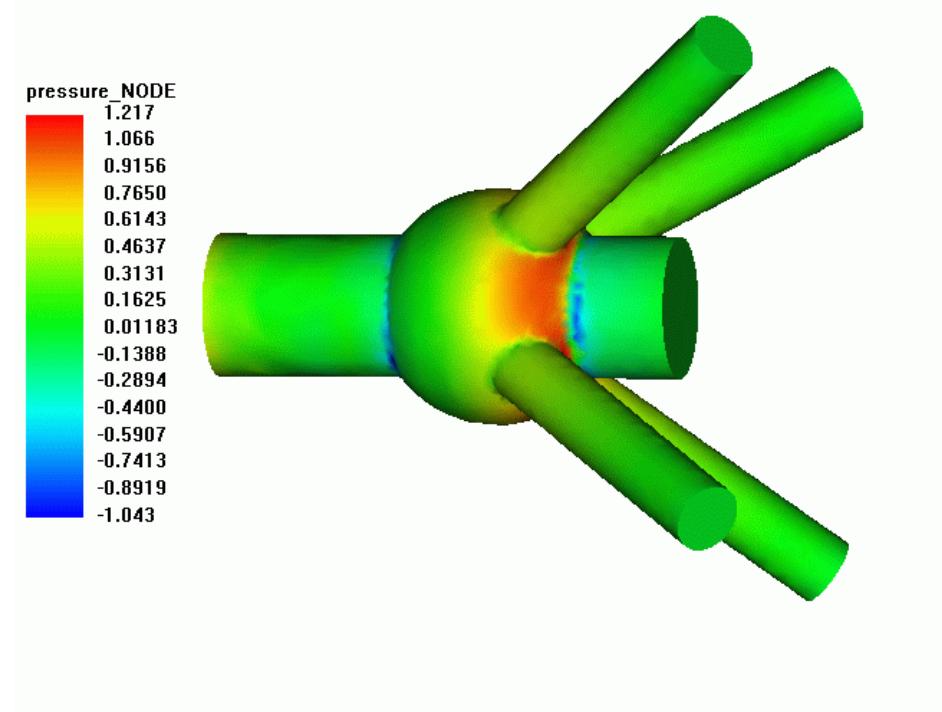
The point probe returns the point's coordinates at the cursor position, the value of the active scalar, and vector functions. Here Point Probing Technique would be explained using the scalar variable pressure.

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.

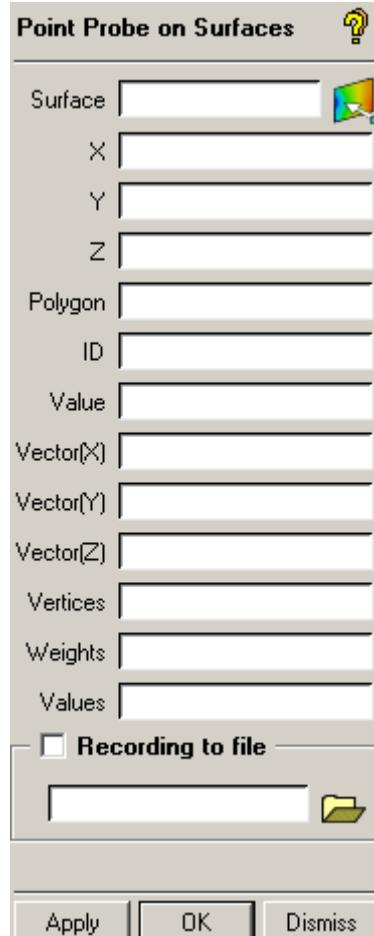
The surface display using pressure_NODE variable is presented below.

Figure 4-665
Surface display for the variable pressure



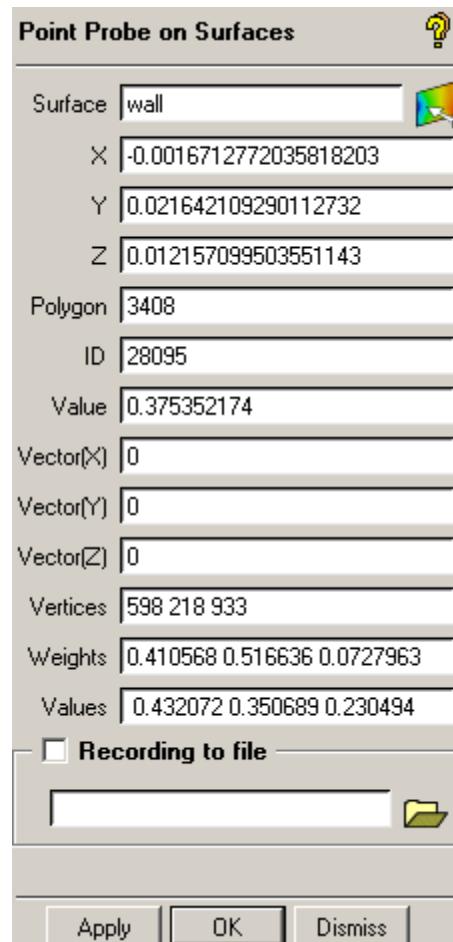
Go to Point Probe icon On the Post-processing tab menu. This will open the Point Probe on Surfaces window.

Figure 4-66
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 here.

Figure 4-667
Probe window
displaying the
coordinates



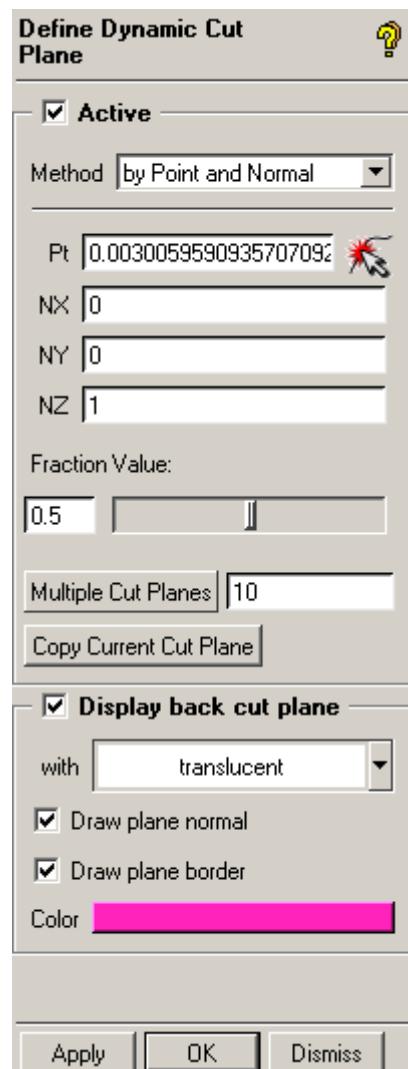
The parameter Value(s) specifically describes the value of the currently chosen scalar variable at the particular location of the surface under probe.

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. Define Dynamic Cut Plane window is presented below.

Figure 4-668
Define Dynamic Cut
Plane window

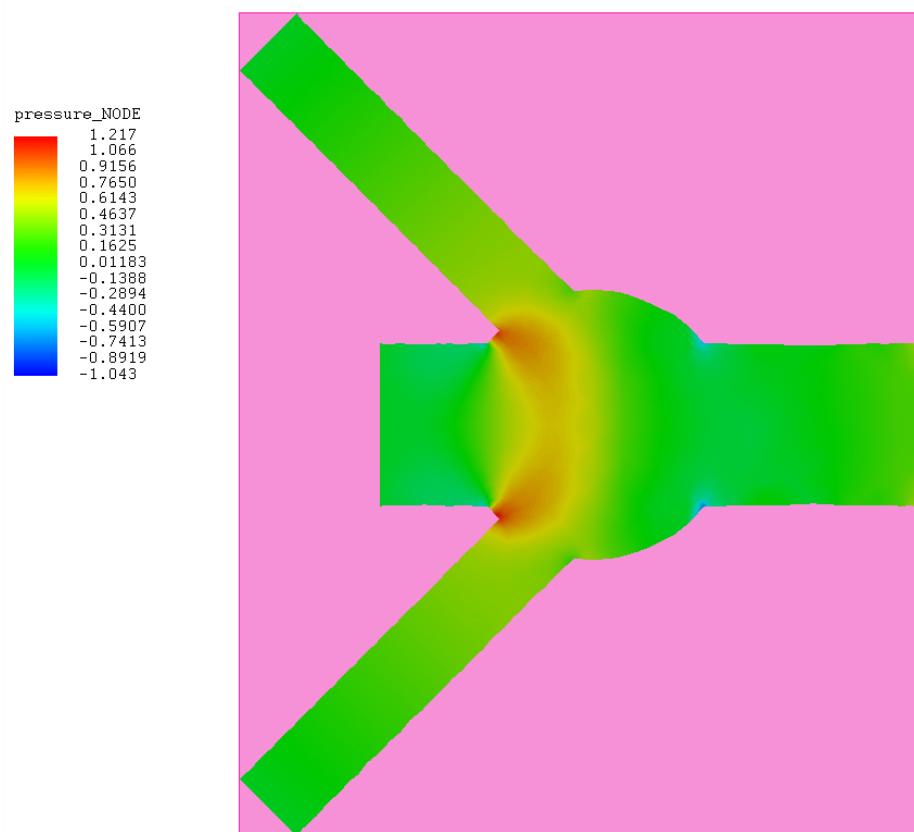


d) Contours on Dynamic Surface

Switch off the Continuous Contour option from the Surfaces, instead turn on Continuous Contour on the display options of Dynamic Surface (right click).

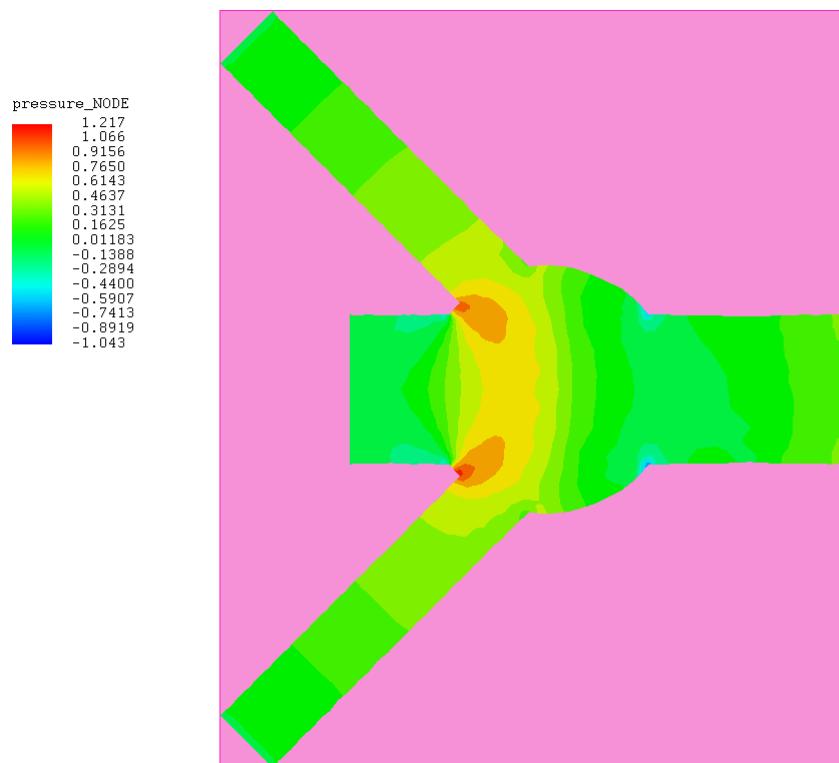
Continuous Contour for the Dynamic Surface will display the Cut Plane in the display window as shown.

Figure 4-669
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 shown below.

Figure 4-670
Cut plane surface displayed with Contour Bands

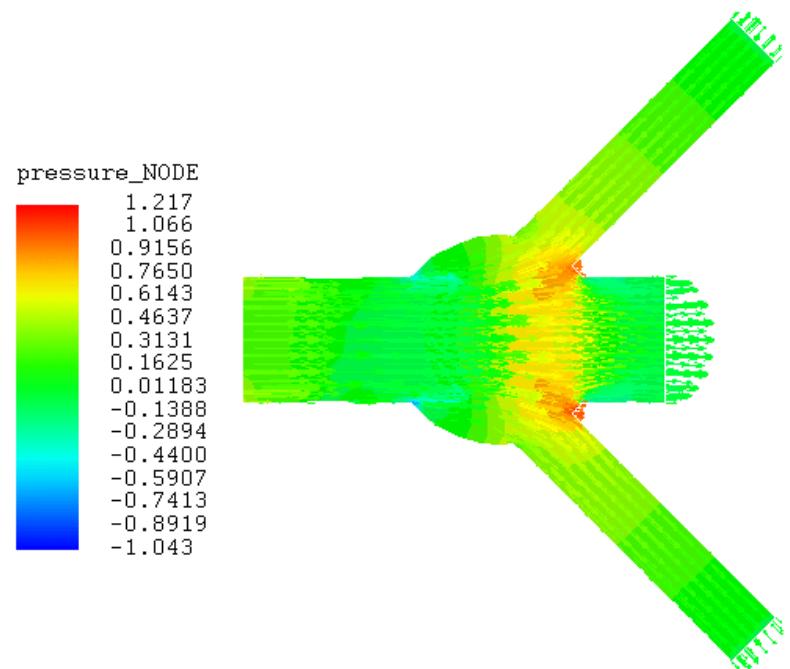


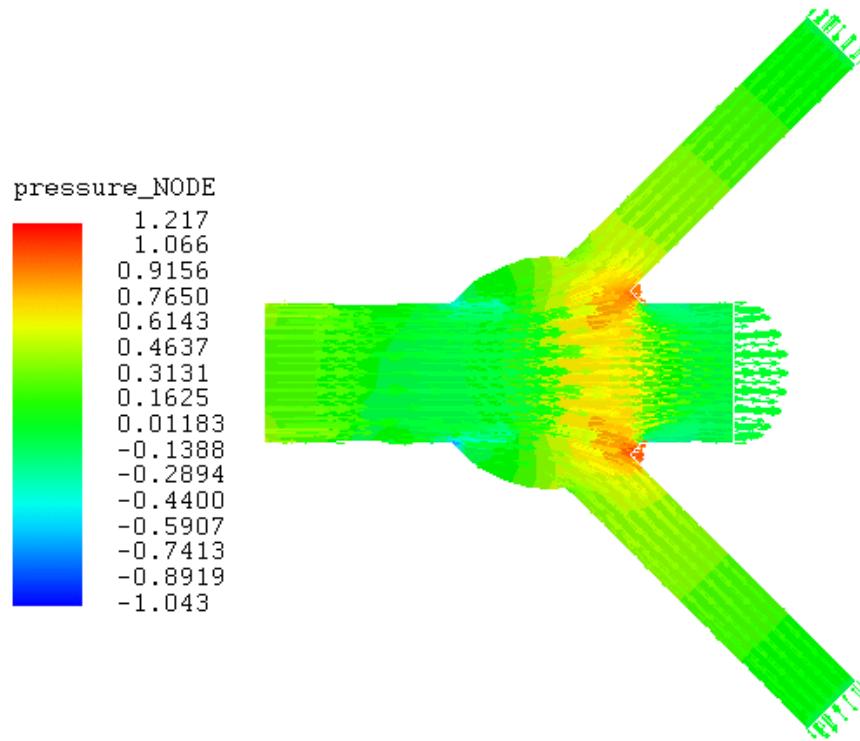
e) Velocity Vectors on Dynamic Surface

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 here.

Figure 4-671
Dynamic Surface display with Vectors of Variable Colors

Output to Solvers



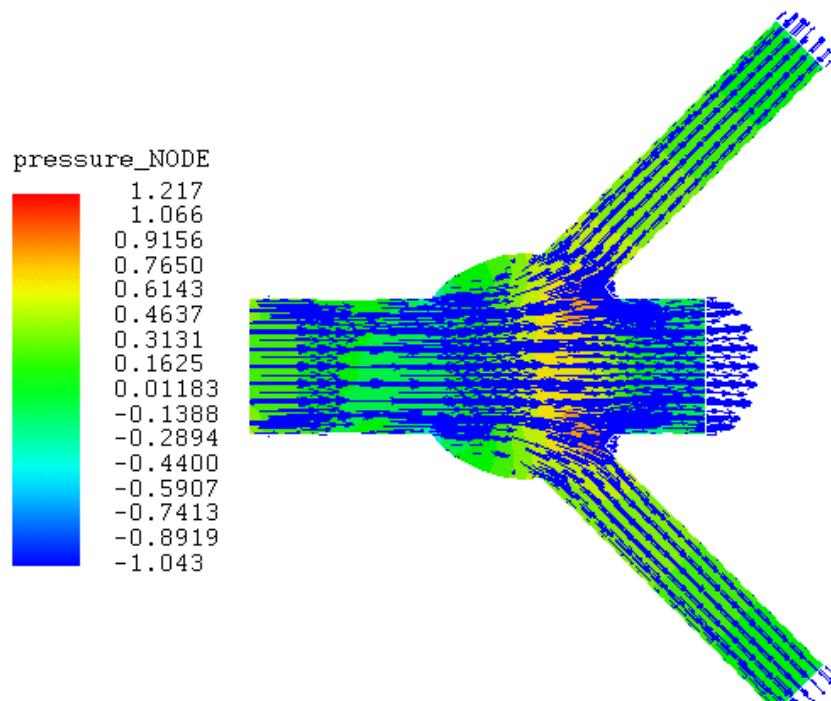


Here, contour bands and vectors display similar colors and hence it becomes difficult to distinguish them from each other. In order to observe the clear display, one can change the color of the vectors as follows. On the display control options for Dynamic Surface, select 2D Vector with single color.

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.

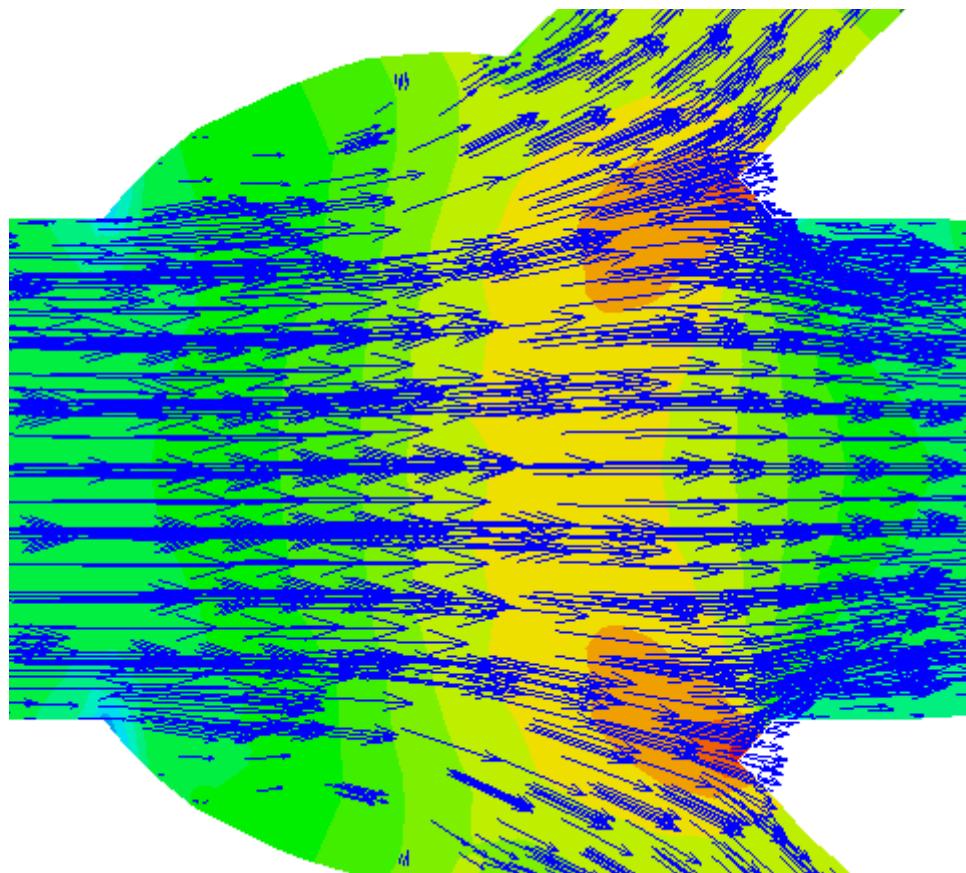
Note : One can control the Vector size and Arrow size by scaling the corresponding properties to desired value with the options Vector size and Arrow size from the Properties tab. A value of 0.4 for Vector size gives good results here.

Figure 4-672
Dynamic surface display with vectors of uniform color



A closer look at the spherical region shown below. The user can use right mouse button to zoom in on the region required.

Figure 4-673
Cut plane displaying the effects of pressure at different areas

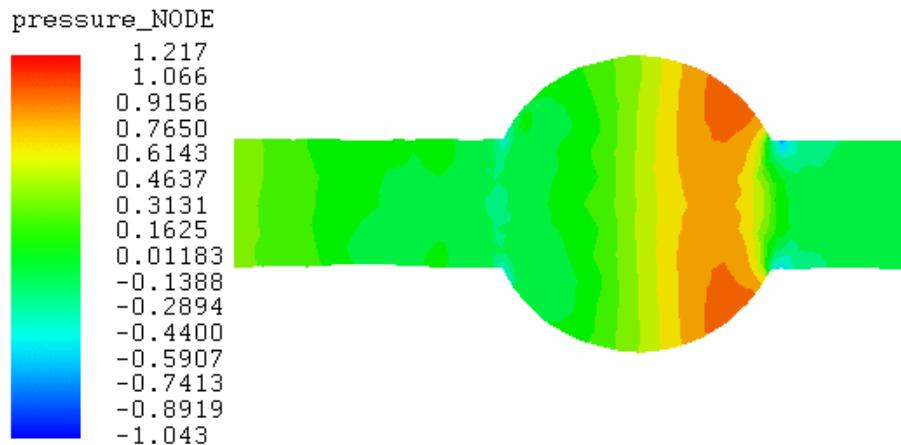


f) Movement of Cut plane

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.

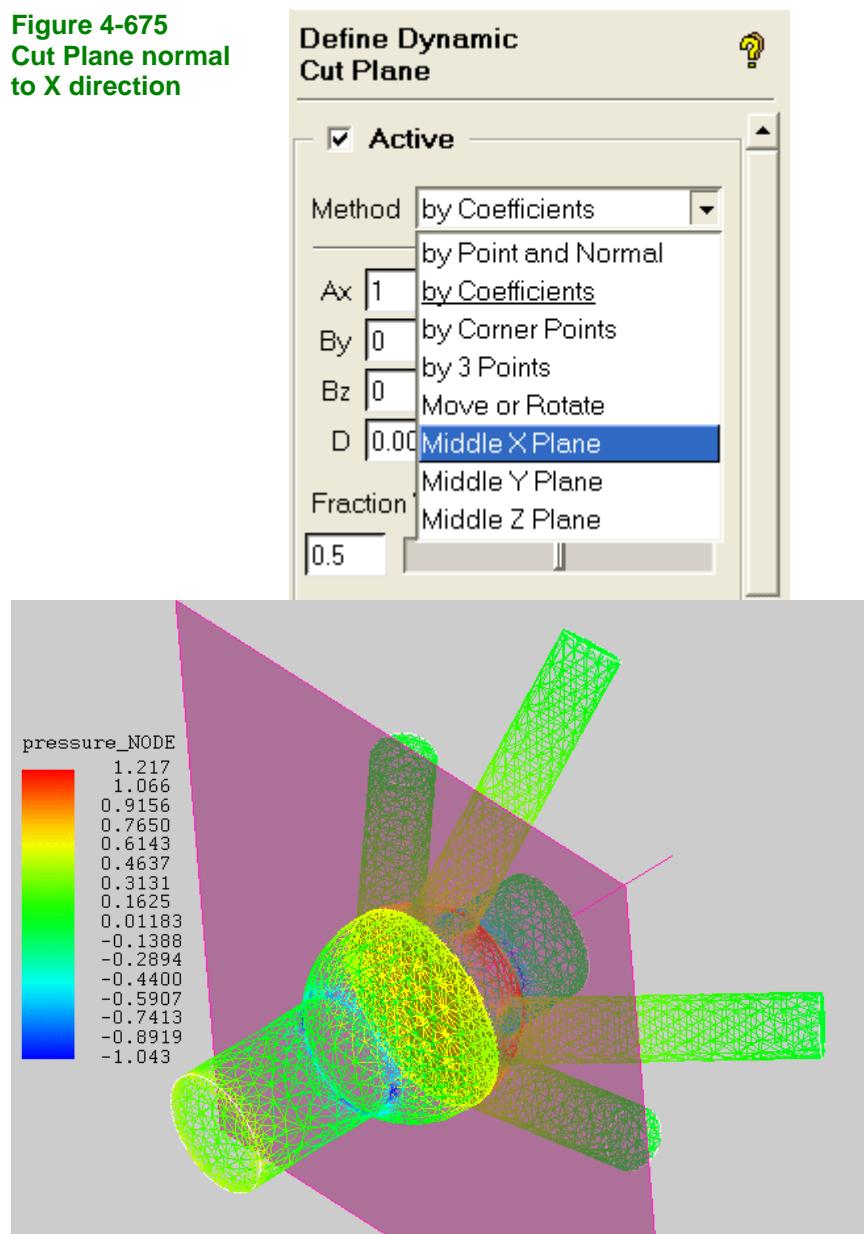
It's best to switch off the vectors and to switch On the cut plane from Tools to notice the cutplane movement. The output is as shown below.

Figure 4-674
Cut Plane at a fraction value of 0.572



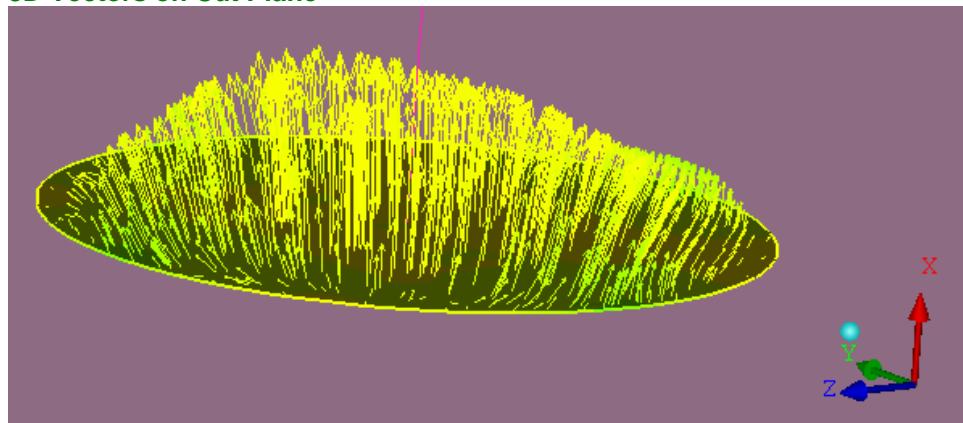
To switch the cut plane in the X direction, on Define Cut Plane panel, from Methods menu, select Middle X Plane as shown here. To see the relative position of this cut plane, display the grid with variable color and show the back cut plane.

Figure 4-675
**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. The figure below indicates that the vectors largely point in the X direction.

Figure 4-676
3D Vectors on Cut Plane



4.9.3: 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

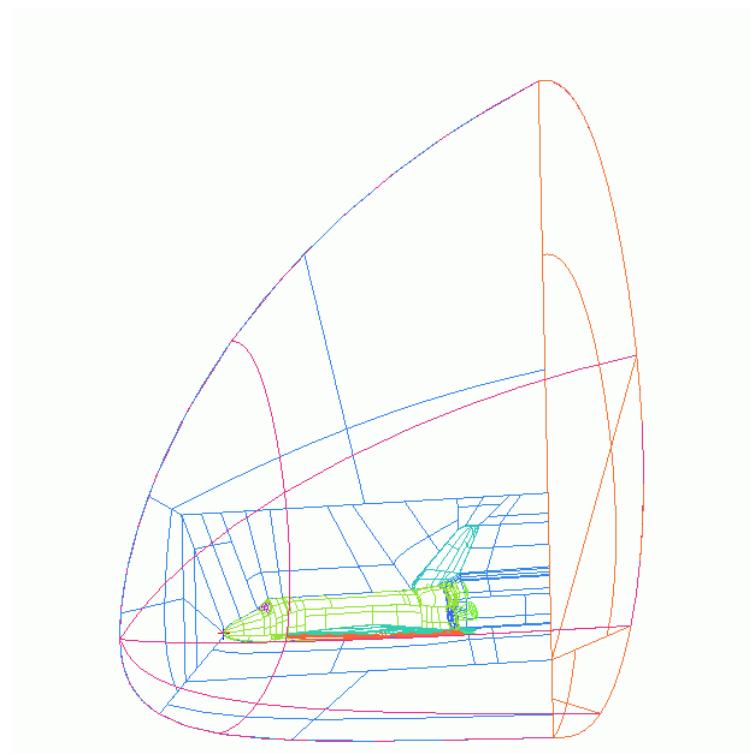
Creating Movies

The input files for this tutorial is found in the Ansys Installation directory, under .../docu/Tutorials/Visual3_Files/Visual3_Examples/Space_Shuttle. Copy these files to your working directory.

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 the figure below, only half of the model is simulated because of the symmetry conditions prevailing.

Figure 4-677
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 here.

**Figure 4-678
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

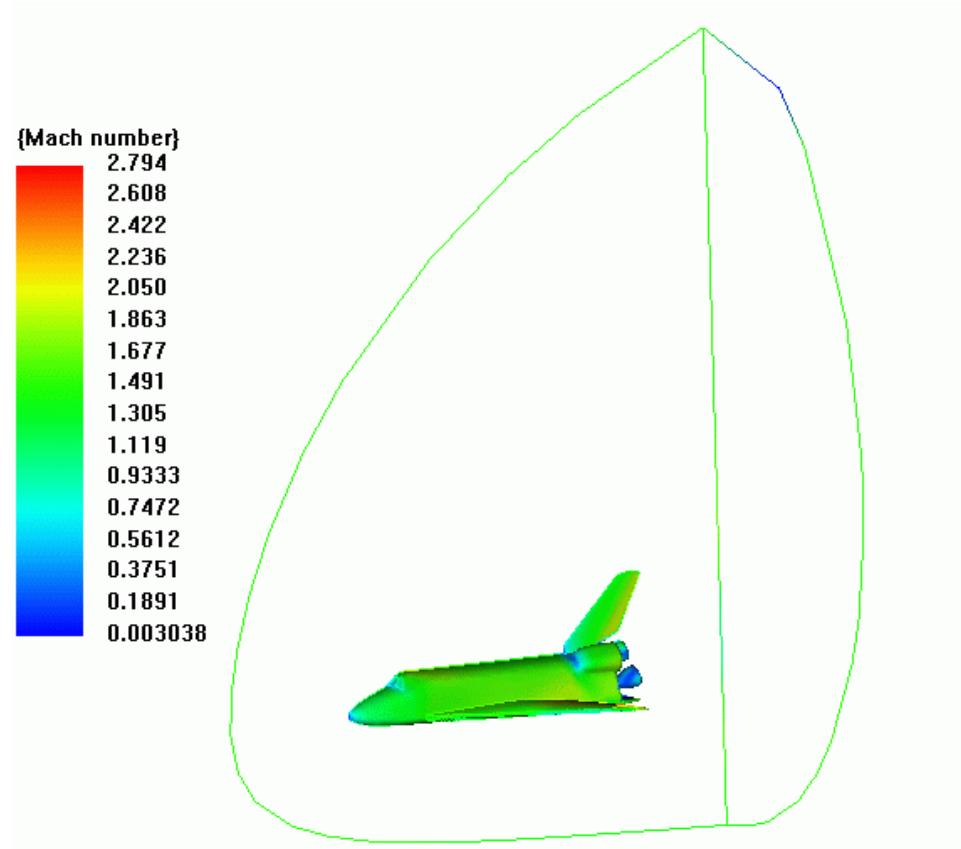
In the Surfaces display options, Continuous contour and Edge with single color will be ON by default.

Switch OFF the Continuous contours for the surfaces beginning with SYM, OUTER, OUT and ORFN.

From the Post-processing menu bar, select Variables icon.

Select Mach number from the Scalar Variable dropped down menu of the Result Variables window.

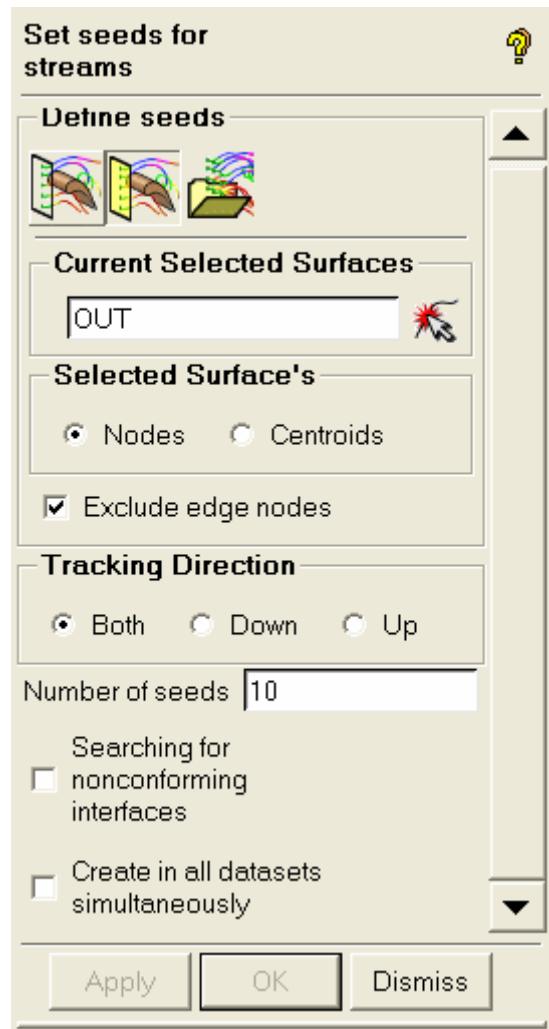
**Figure 4-679
Surfaces SYM, OUTER and OUT with no solid contours**



Select the Streams icon from the post processing tab menu. This would bring out the Set Seeds window.

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. Select OUT surface for this tutorial.

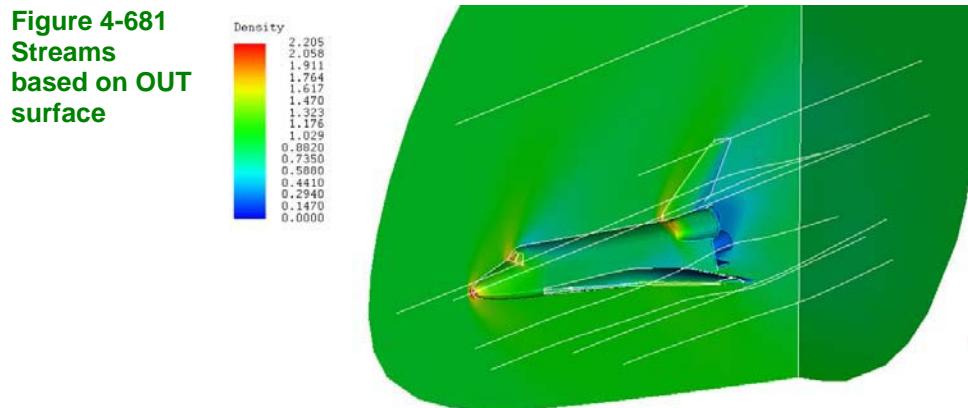
Figure 4-680
Streams Manager



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.



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 user will be prompted to confirm about the delete Group 1 entry of Streams.

After pressing Confirm, it will delete the streams.

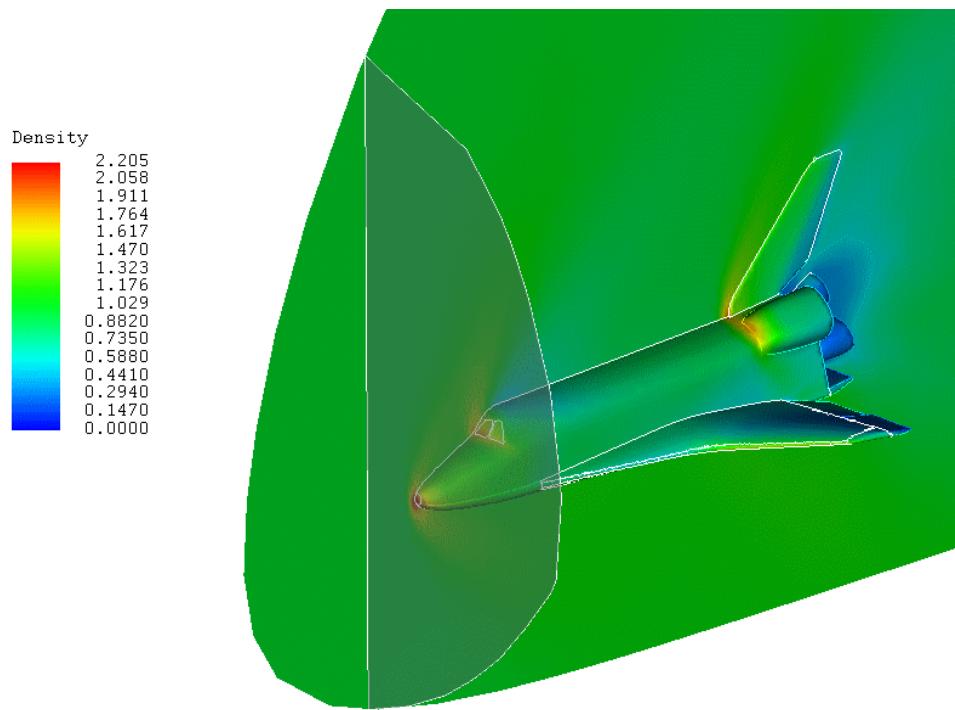
d) Creating Streamlines along 2D container:

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 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.

Figure 4-682
Cut Plane

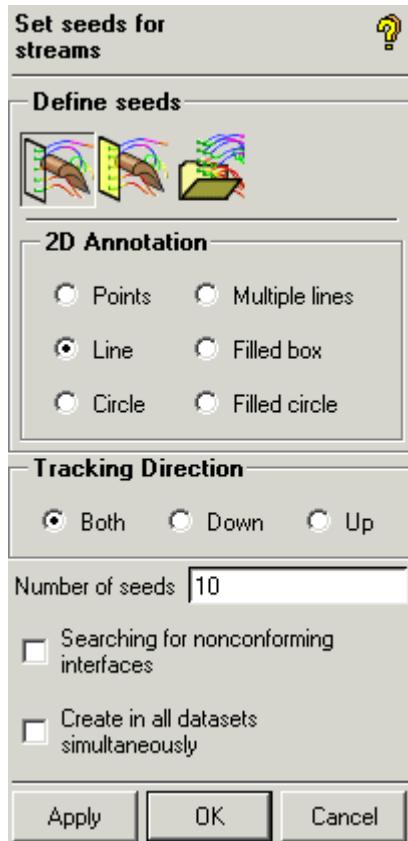


Now, this cut plane will be consider as the Dynamic Surface, under the Surfaces option of Display Tree.

Ensure that the Dynamic Surface is selected from the Display Tree and then go to Streams option of Post-processing menu bar.

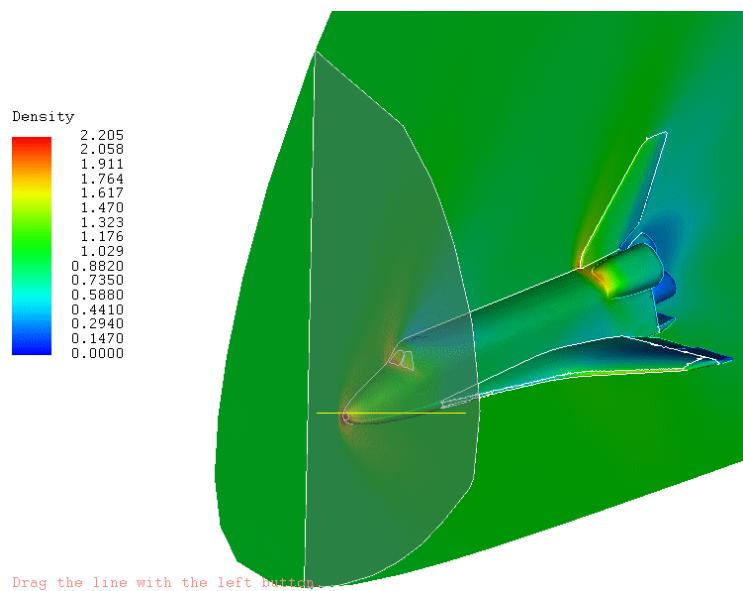
As shown below, select the option Set seeds with 2D Annotation from Define Seeds panel for streams window. Choose the radio button Line to define the type of seeds.

Figure 4-683
Streams from
2D Annotation
window



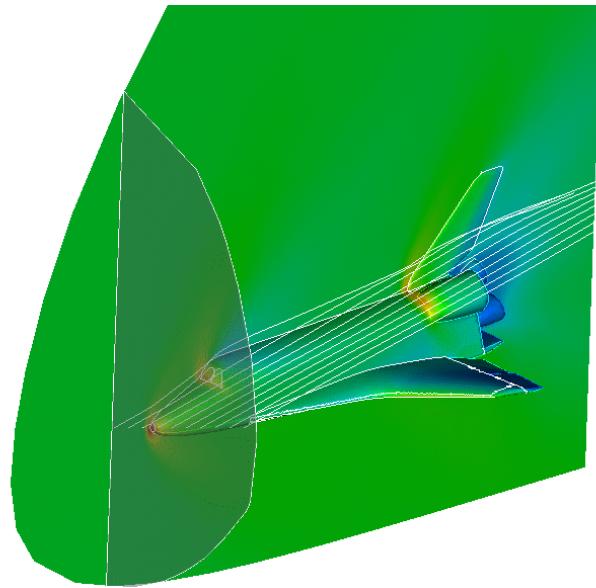
Press Apply. This would display a message on the main graphics window prompting the user to define the starting and ending points of the line. Click the left mouse button and drag it to define the length of the line as shown here. The horizontal line is drawn from left side beginning till the back side outlet going through shuttle.

Figure 4-684
Dynamic
Surface with
the line
defined



The Streamlines would appear in the graphics window as shown here. This will update the Streams branch of the Display Tree. If desired, from Group 2 in the model tree, switch to various display options.

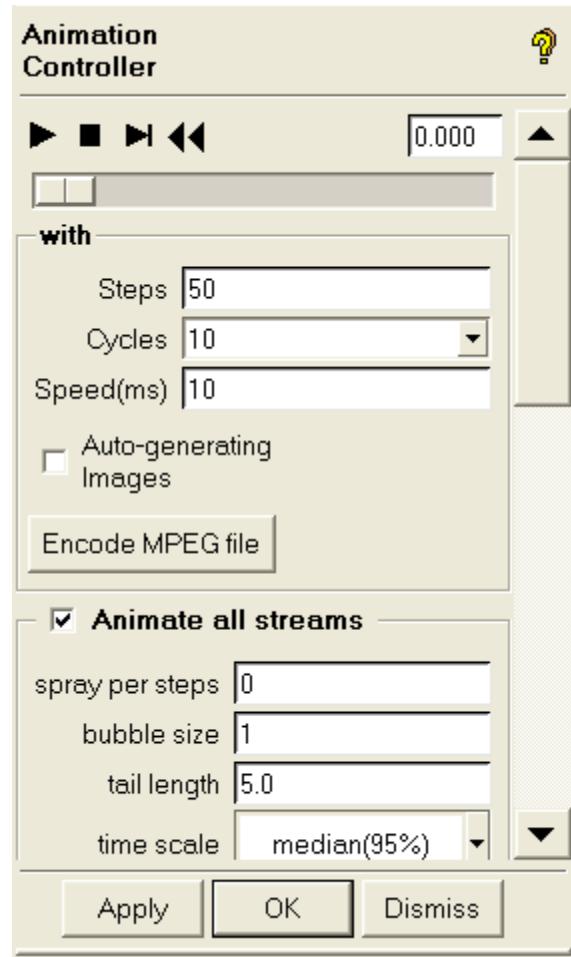
**Figure 4-685
Streamlines
based 2D
option**



e) Animating the Streamlines

Select the Animation Controller icon from the post processing menu. This will bring out the following selection windows as shown here.

Figure 4-686
All in one
Animation
Controller
window



Select the option Animate All Streams on the controller panel.

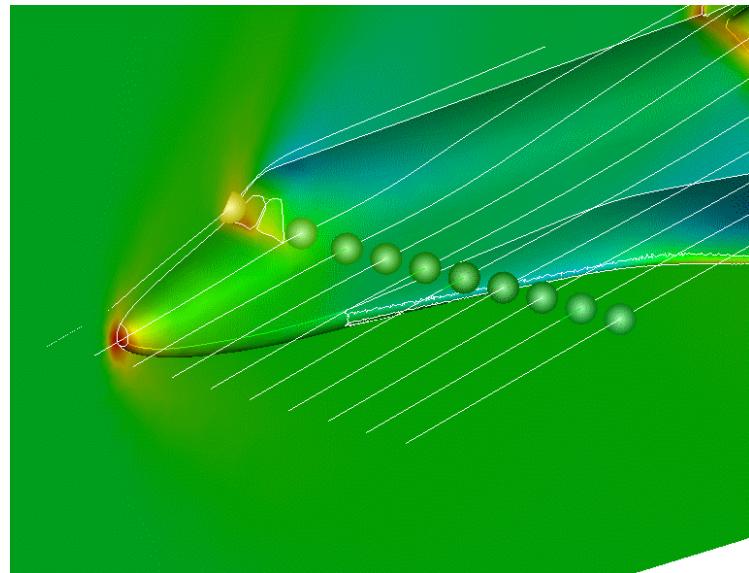
Set bubble size to 0.5.

Press Apply

On the top of Animation Controller, click on play button. Now bubbles begin moving along the streams.

The smoothness of the animation can be controlled by Steps parameter, and the speed per step can be adjusted by Speed parameter.

**Figure 4-687
Bubbles on 2D**



4.9.4: 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

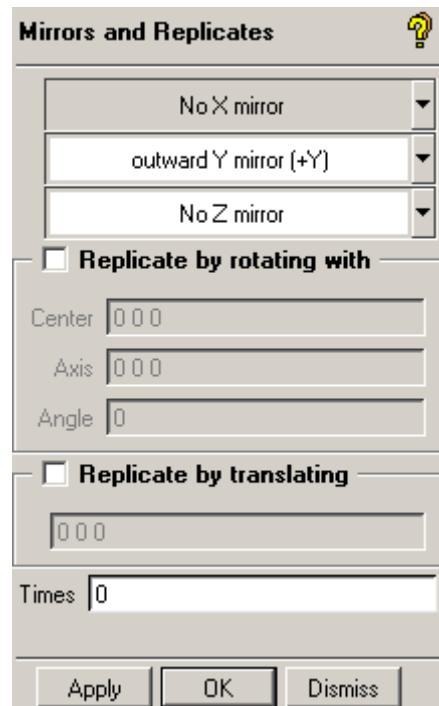
Movies

a) Loading the CFD++ file

Start Visual3 application and read the CFD++ results as in the previous tutorial.

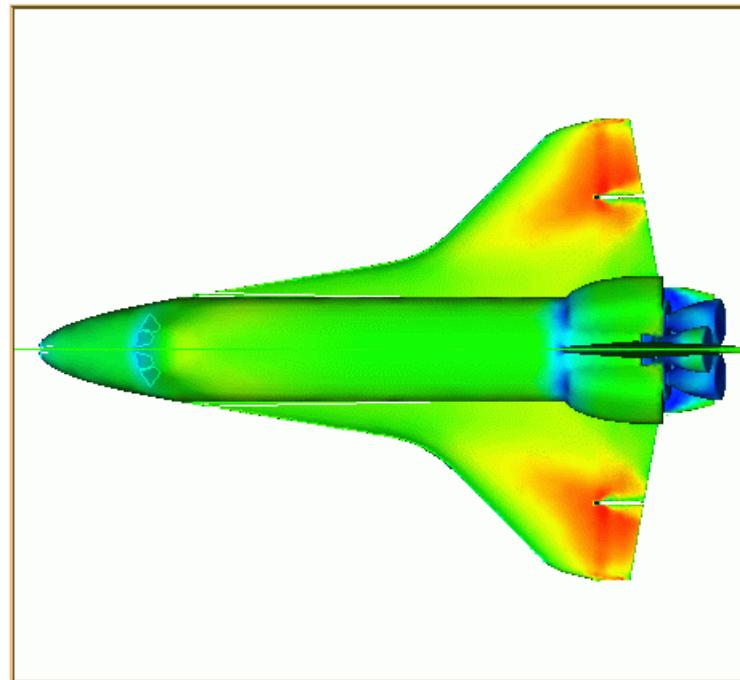
From View, select mirrors and replicates. This will bring out the Mirrors and Replicates window as shown here.

Figure 4-688
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 below.

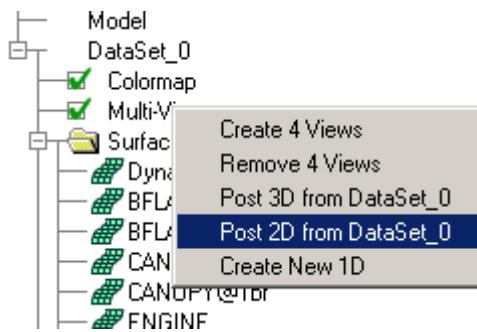
**Figure 4-689
Mirror Image**



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.

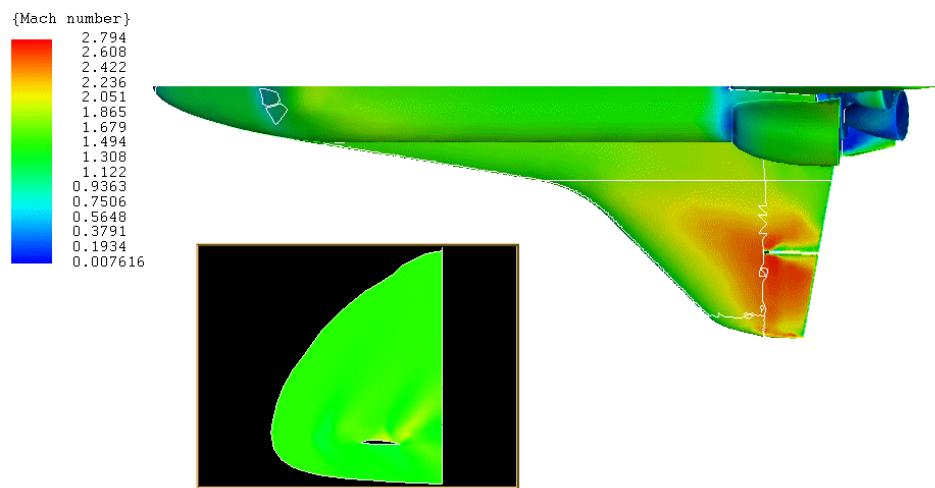
**Figure 4-690
Multi-Views options**



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 seen below.

The 2D graphics window's name will appear in the model tree under Multi-Views. This will also have its own right click options which are shown in the figure below.

Figure 4-691 Multi-Views > Post 2D window**Figure 4-692 Post 2D window display options.**

Select the option Remove Vframe to clear the newly created window.
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

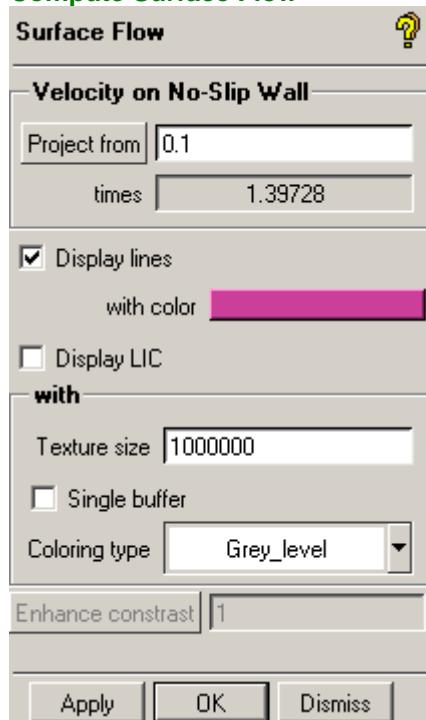
From the model tree, select the surface FUSEL, by issuing a right click on the name.

From the display options, choose Compute Surface Flow which invokes the panel displayed here.

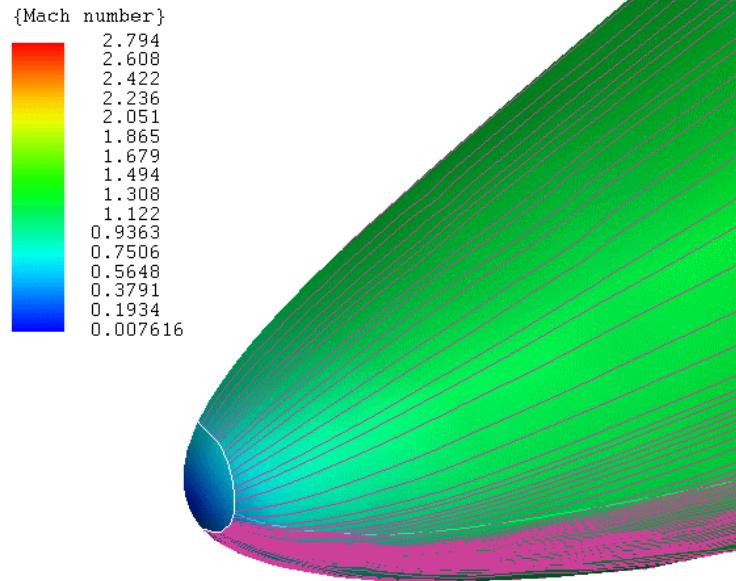
Click in the Display Lines check box.

Press Apply. This would render the selected surface with colored flow lines.

Figure 4-693
Compute Surface Flow



**Figure 4-694
Compute
Surface Flow
Display Lines**



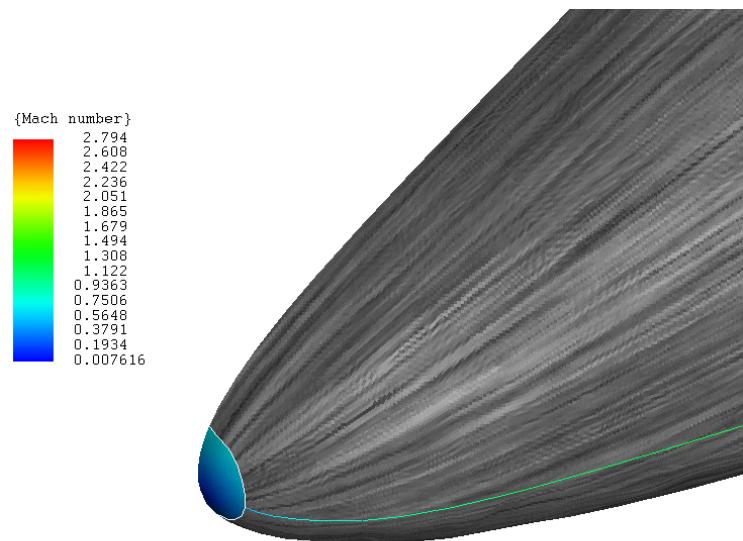
e) Line Integral Convolution (LIC) of the selected surface

From the Compute Surface Flow panel (Figure 4-693), 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 below.

Figure 4-695
Compute
Surface Flow
Display LIC



ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	794
------------------------	--	-----

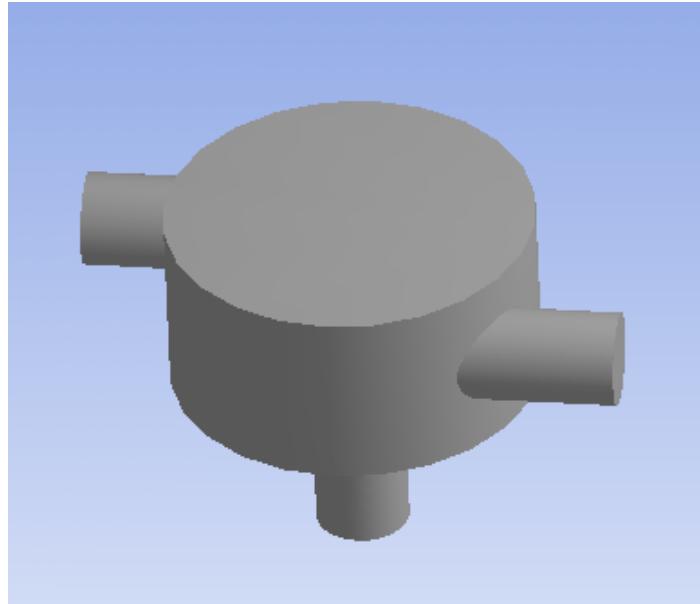
5: ANSYS ICEMCFD - CFX Tutorial Manual

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 **Advanced 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 696:
Static Mixer Geometry



a) Steps Involved in this Example

Creating Geometry in DesignModeler.

Taking Geometry to Advanced Meshing.

Assigning Mesh parameters

Generating tetrahedral mesh in Advanced Meshing.

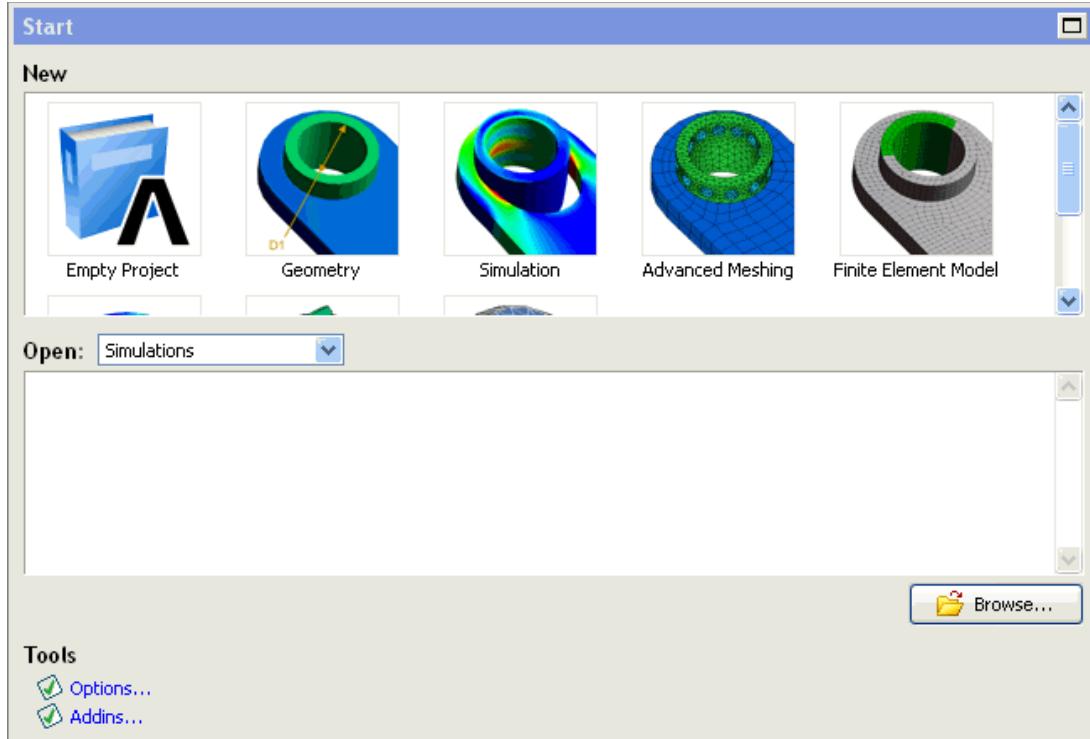
Writing input mesh file for CFX-5.

5.1.2: Starting a New Project

a) Launch ANSYS ICEMCFD - CFX

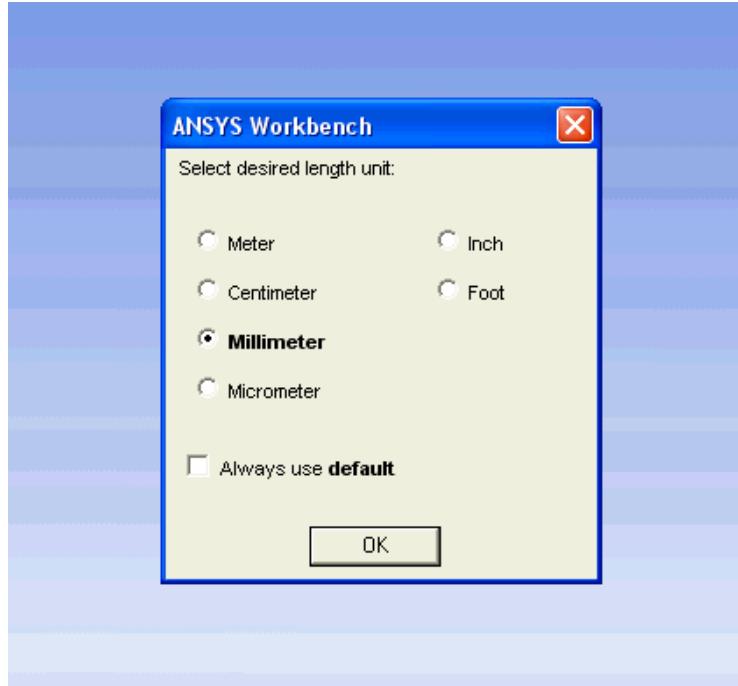
Launch the ANSYS Workbench, ANSYS Workbench window will appear as shown in the figure given below.

Figure 697
Selection window



Select the Geometry tab. This will open DM [DesignModeler] window. Another ANSYS Workbench window will pop up for selection of desired length unit, select Millimeter and press OK. Design modeler and desired unit window is shown in the figure given below.

**Figure 698
Workbench
window**



5.1.3: 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.

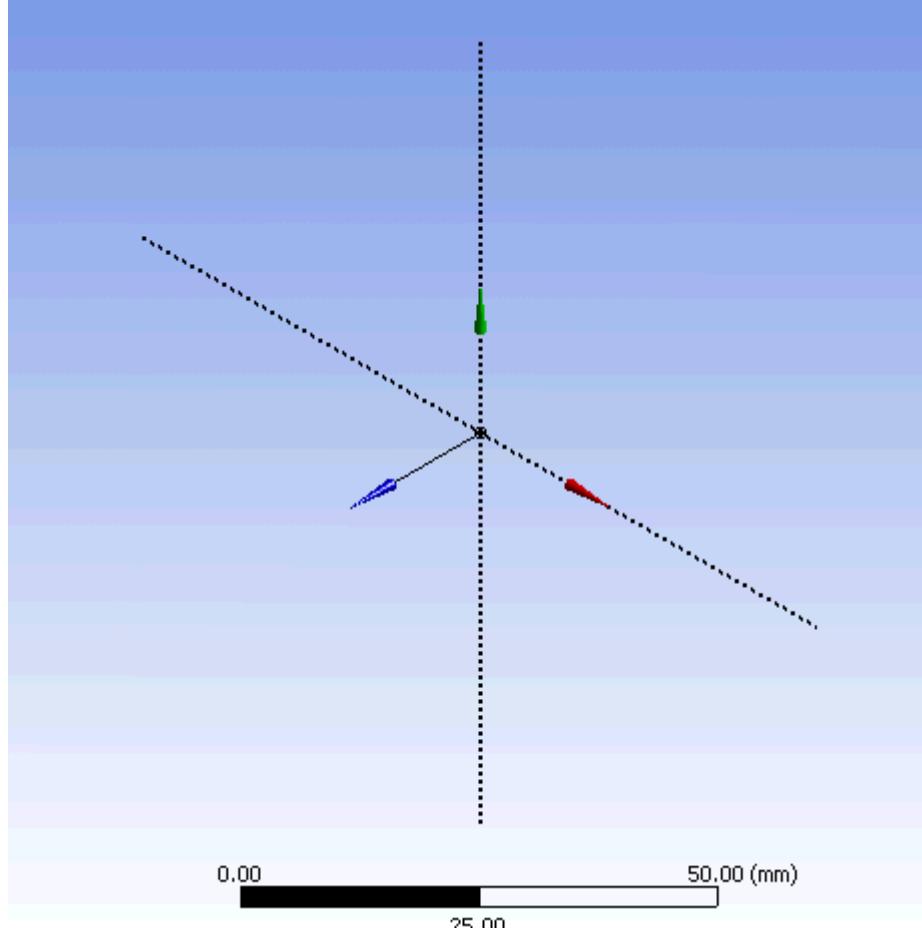
The geometry for this tutorial is divided in two sub steps. Revolving the profile curve about the vertical axis of the mixer will generate the main body of the mixer. Then the inlet pipe will be generated. The detailed description is as follows.

a) Creating Main Mixer Body

Creation of Profile Curves

Select XYPlane 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 the figure given below.

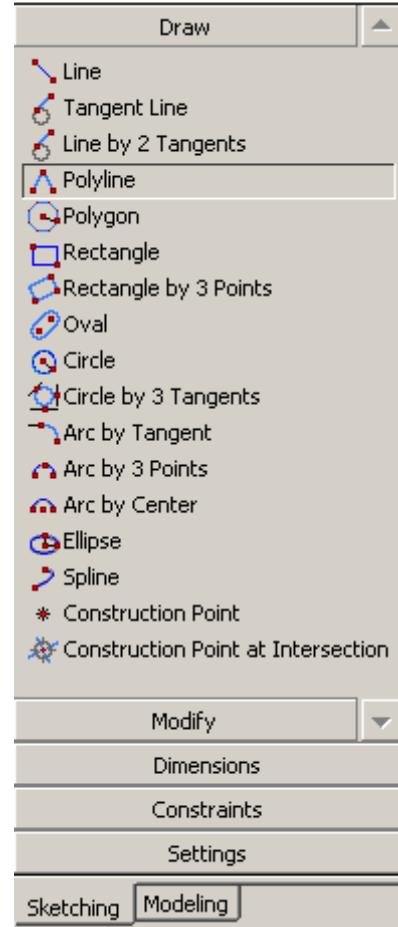
Figure 699
Workbench
graphics
window



Select (Look at Face/Plane/Sketch) icon from main tool bar.

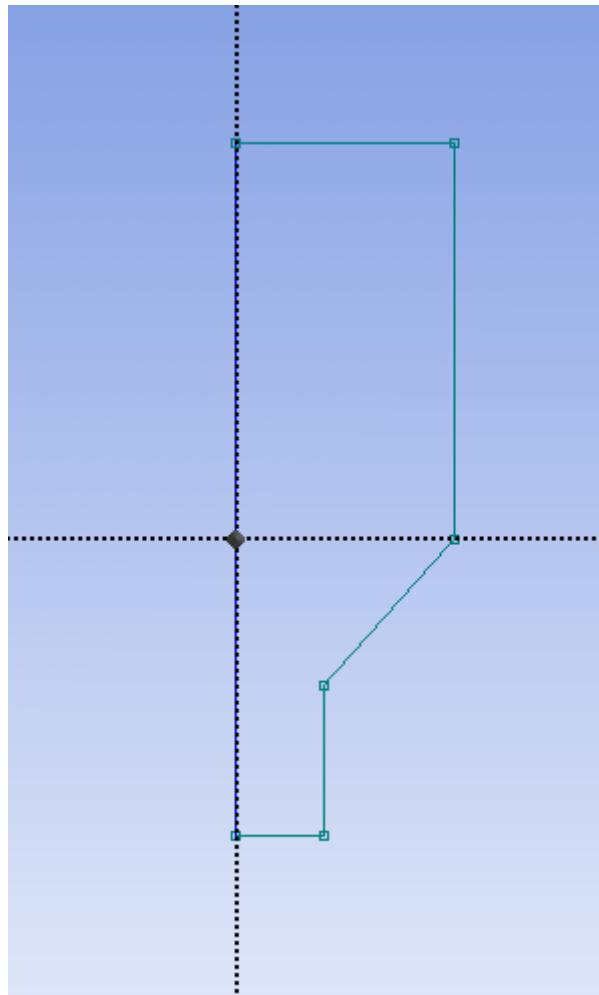
Select *sketching* from project tree. It will open the draw tool bar. Now select *polyline* from draw tool bar as shown in the figure given below.

Figure 700
Dimensions window



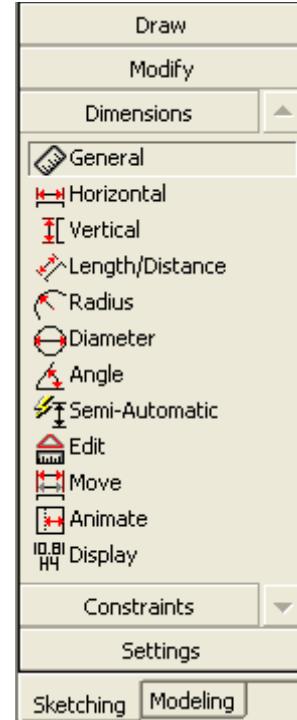
Now draw the approximate shape with the help of cursor as shown in the figure given below.

Figure 701
Approximate
diagram of
static mixture
body.



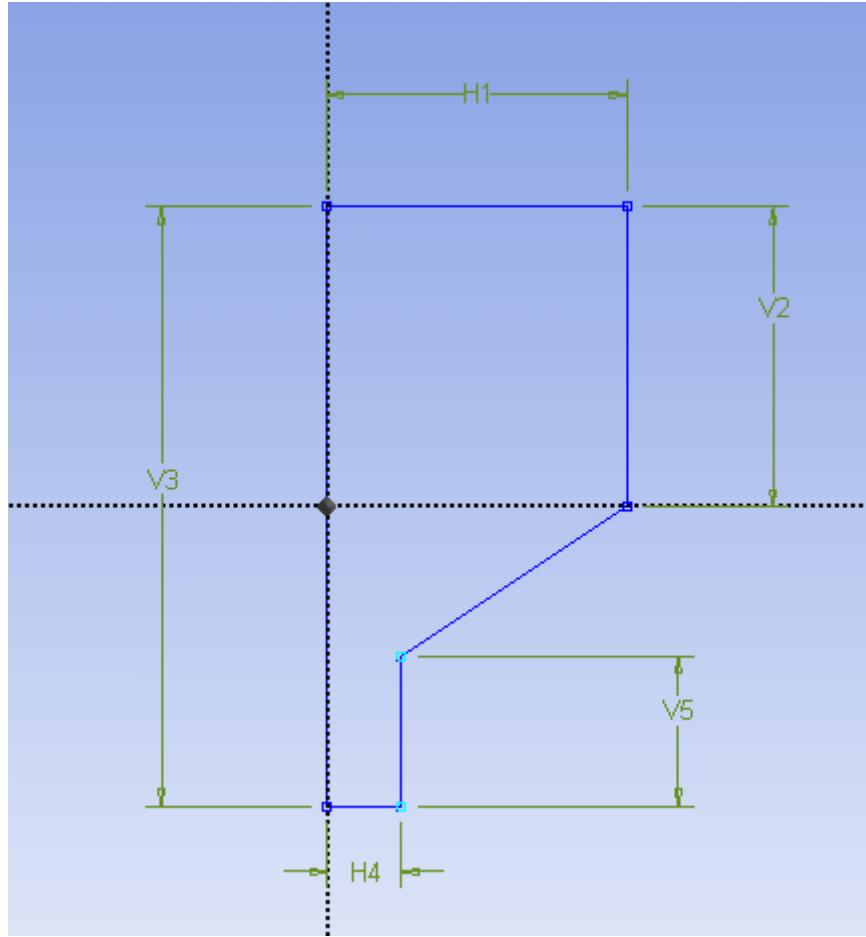
After drawing the approximate shape to revolve, the user has to define the exact dimension to the curves so that shape of the revolved component will match the geometry. Click on dimensions in the sketching tab. One dimension window will pop up as shown below.

Figure 702
Dimensions Window



Select general as default option. Take cursor on to the screen; move on to the edge on which you want to apply the dimensions. Apply the dimensions as shown in the figure below.

Figure 703
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 shown in the figure below.

Figure 704
Exact Dimensions window

Details View	
Details of Sketch1	
Sketch	Sketch1
Sketch V...	Show Sketch
Show C...	No
Dimensions: 5	
<input type="checkbox"/> H1	20 mm
<input type="checkbox"/> H4	5 mm
<input type="checkbox"/> V2	20 mm
<input type="checkbox"/> V3	40 mm
<input checked="" type="checkbox"/> V5	10 mm
Edges: 7	
Line	Ln7
Line	Ln8
Line	Ln9
Line	Ln10
Line	Ln11
Line	Ln12
Line	Ln13

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 given in the figure below.

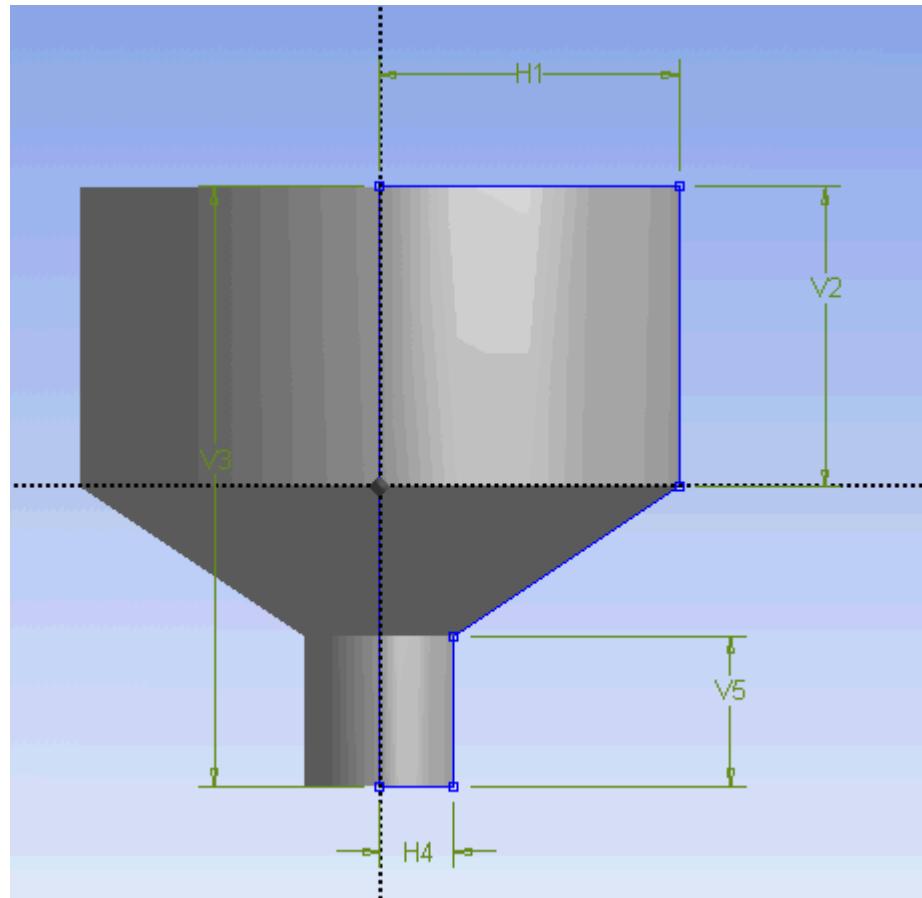
Figure 705
Revolve Details window

Details View	
Details of Revolve1	
Revolve	Revolve1
Base Object	Sketch1
Axis	Selected
Operation	Add Material
Direction	Normal
<input type="checkbox"/> FD1, Angle (>0)	360 °
As Thin/Surface?	No
Merge Topology?	Yes

Click on the Axis and select the axis as XY plane from the screen with the help of cursor. Press **Apply**.

Press **Generate** so that it will generate the mixture body as shown in the figure below.

Figure 706
Geometry
after
revolution



This is the generated mixture body. Now the user has generate the inlet pipes for the mixture.

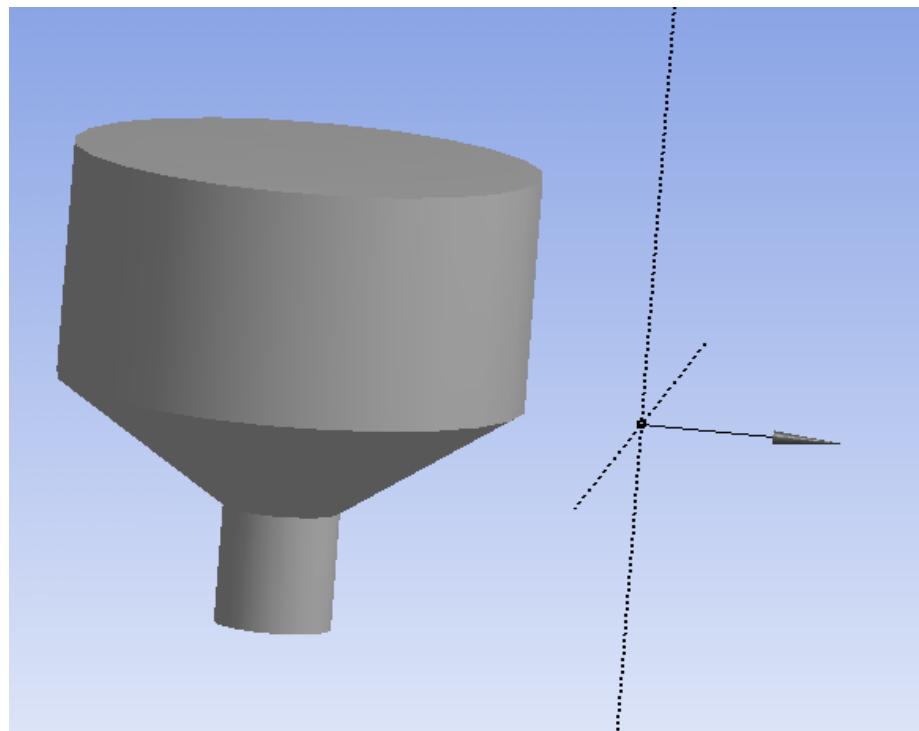
2. Generating the inlet pipes:

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 **XYplane** in the project tree and then click on new plane in the 3D features toolbar. This will create the new plane. Offset this plane from original xyplane according the entering values in the details view as shown in the figure below.

Figure 707
Offsetting parameter for plane

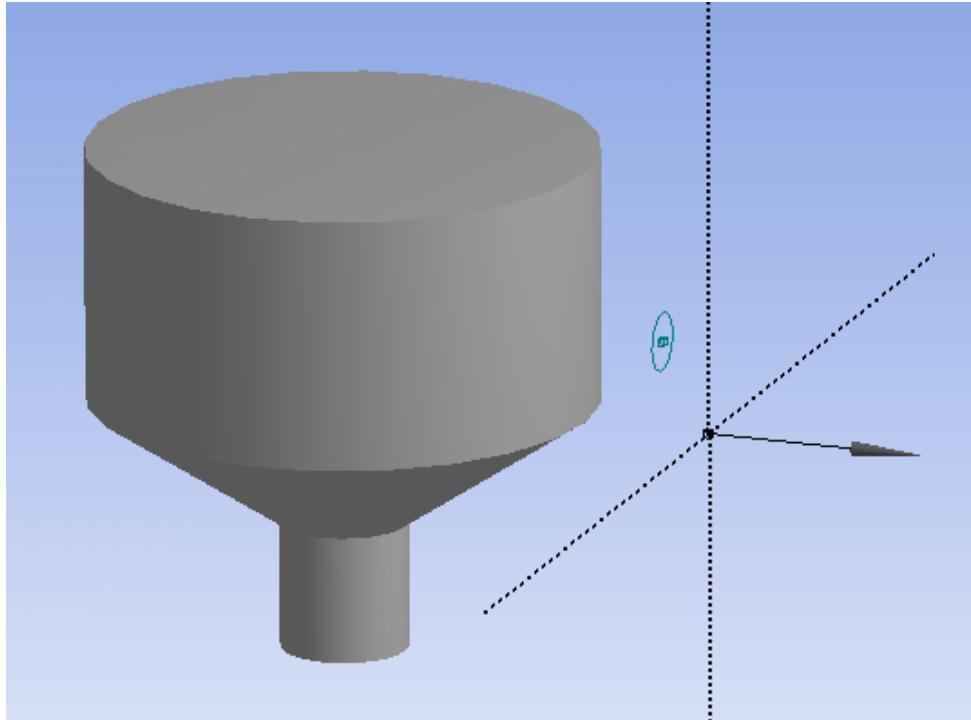
Details View	
Details of Plane4	
Plane	Plane4
Type	From Plane
Base Plane	XYPlane
Transform 1 (RMB)	Offset Z
FD1, Value 1	30 mm
Transform 2 (RMB)	None
Reverse Normal/Z-Axis?	No
Flip XY-Axes?	No
Export Coordinate System?	No

**Figure
708
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 below.

**Figure
709
Created
Circle**



Press dimensions select general and select the horizontal and vertical two dimensions. Then select the sketch and apply dimensions according to the figure given below.

Figure 710
General dimensions
To
circle

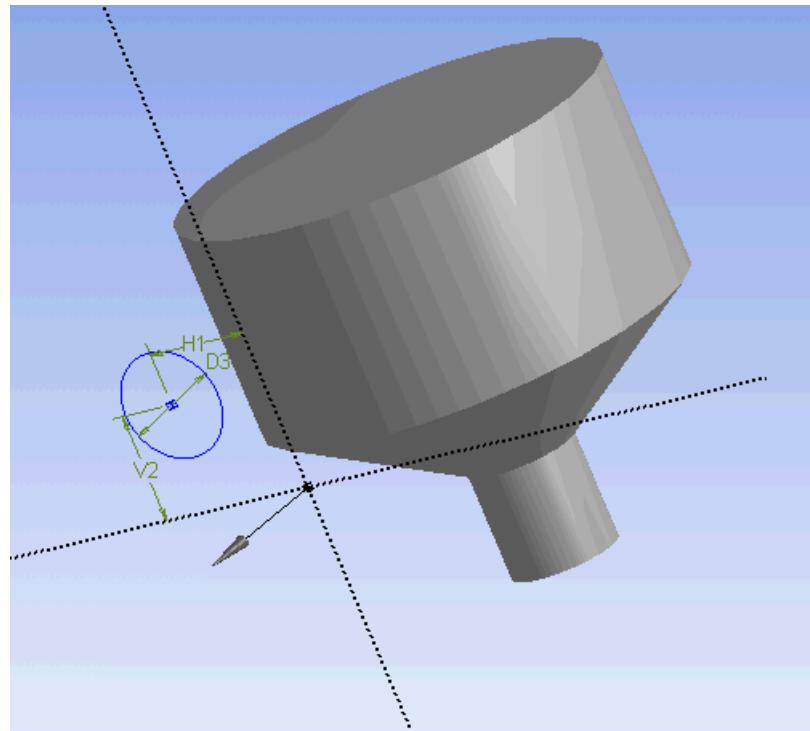


Figure 711
Detail view for creating circle

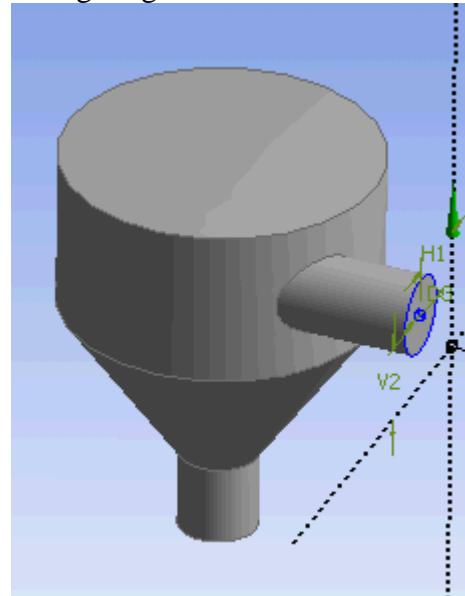
Details View	
Details of Sketch2	
Sketch	Sketch2
Sketch Visibility	Show Sketch
Show Constraints?	No
Dimensions: 3	
<input type="checkbox"/> D3	10 mm
<input type="checkbox"/> H1	10 mm
<input type="checkbox"/> V2	10 mm
Edges: 1	
Full Circle	Cr15

Figure 712
Extrude details for first curves

Details View	
Details of Extrude1	
Extrude	Extrude1
Base Object	Sketch2
Operation	Add Material
Direction Vector	None (Normal)
Direction	Reversed
Extent Type	To Faces
Target Faces	1
As Thin/Surface?	No
Merge Topology?	Yes

After extruding the geometry will look like the figure given below.

Figure 713
Geometry after extrusion of the circle



Now we have to create the same type of extruded pipe on the other side. Select the xyplane in the project tree and select new plane from the main toolbar. This will create a new plane on which we will create circle. Enter the details as per the figure given below.

Figure 714
Create Second Offset Plane Details

Details View	
Details of Plane5	
Plane	Plane5
Type	From Plane
Base Plane	XYPlane
Transform 1 (RMB)	Offset Z
FD1, Value 1	-30 mm
Transform 2 (RMB)	None
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 same dimensions as we applied for the previous circle.

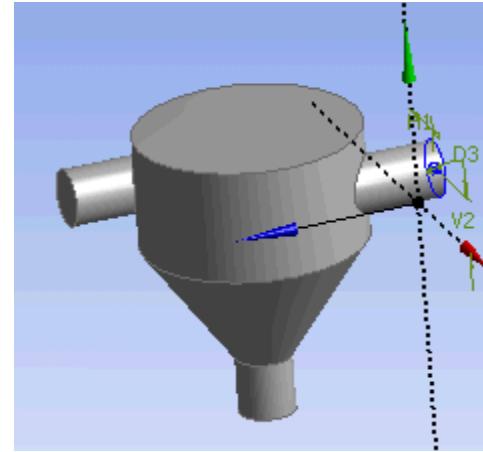
After applying the dimensions press extrude from the main toolbar and enter the detail in detail view as shown below.

Figure 715
Extrude details

Details View	
Details of Extrude2	
Extrude	Extrude2
Base Object	Sketch3
Operation	Add Material
Direction Vector	None (Normal)
Direction	Normal
Extent Type	To Faces
Target Faces	1
As Thin/Surface?	No
Merge Topology?	Yes

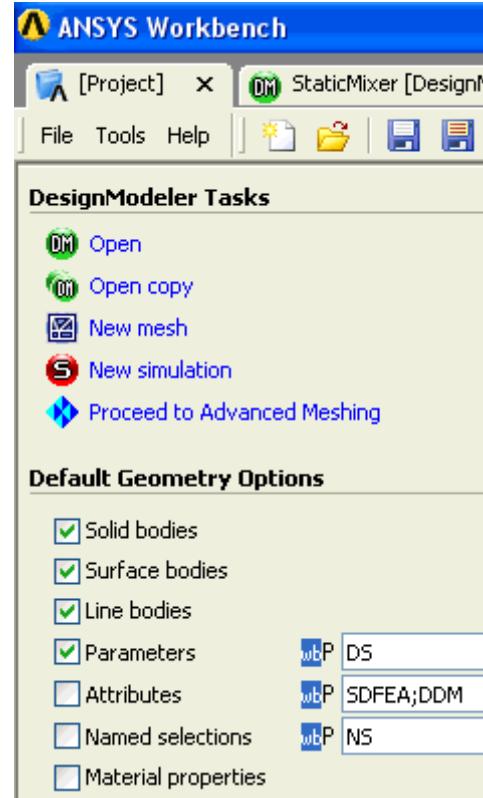
After extrusion, the geometry will look like the figure below.

Figure 716
Final geometry after complete extrusion



Now we are done with the geometry creation. We now proceed to the Advanced Meshing tab for meshing. Before going to Project Page, save the DM project with the name "static_mixer". Click on **project** in the main menu. Select the **Proceed to Advanced Meshing**. This will take user in the advanced meshing GUI where the user can repair the geometry, mesh it and write the output file for CFX.

Figure 717
Project options window



Now the Graphics User Interface for Advanced meshing opens, and the user has to run the build topology to get the necessary curves and points.

Go to Geometry > Repair Geometry > Build Topology. Enter the values as shown and press Apply.

Figure 718
Repair geometry window

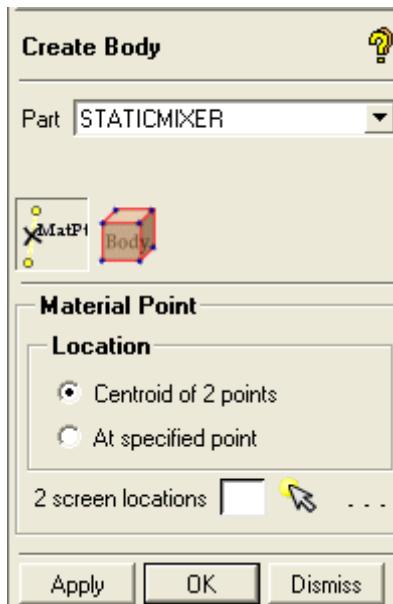


b) Creating Body

Go to the **Geometry** tab menubar and select **Create Body** .

Give *STATICMIXER* as the Part, click on Material Point (default) and toggle on the **Mid Points** option (default) as shown below.

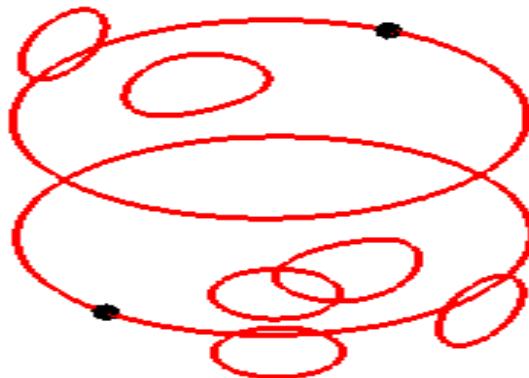
Figure 719:
Create Body
Window



Turn off all Surfaces and Points and display only curves from the Display Tree.

Click on and select two opposite locations on the screen as suggested in the figure given below and press the middle mouse button.

Figure 720:
**Two Opposite
points for
Material point**



- Press Apply.

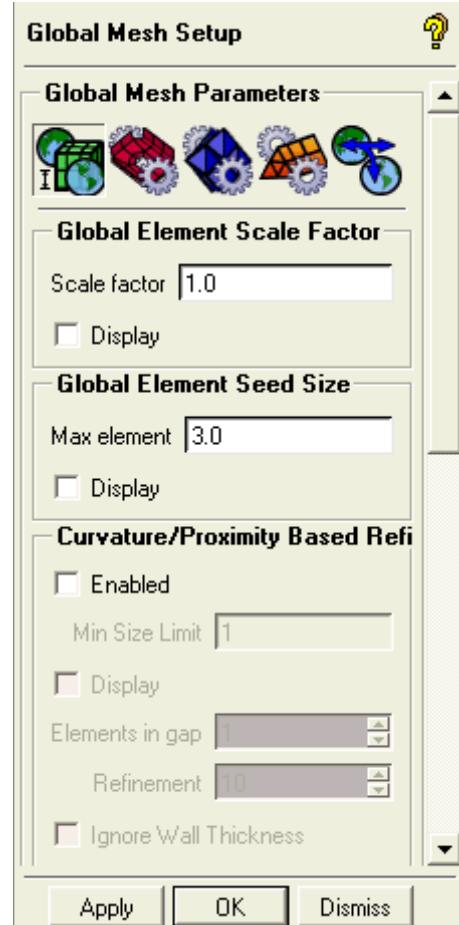
5.1.4: Mesh Generation

a) Assigning Mesh Parameters

From the **Mesh** tab menubar click on **Global Mesh Setup** .

In that window, change **Max Element** to **3** and leave the other fields as the default as shown in the figure given below.

Figure 721:
Global Mesh Size window



Press **Apply**.

The **Scale Factor** is used to scale the mesh size. Please note that all the sizes in **ANSYS ICEMCFD - CFX** get multiplied with the Scale Factor. Thus, it's important to keep a note of the Scale Factor all the time.

b) Saving the Project

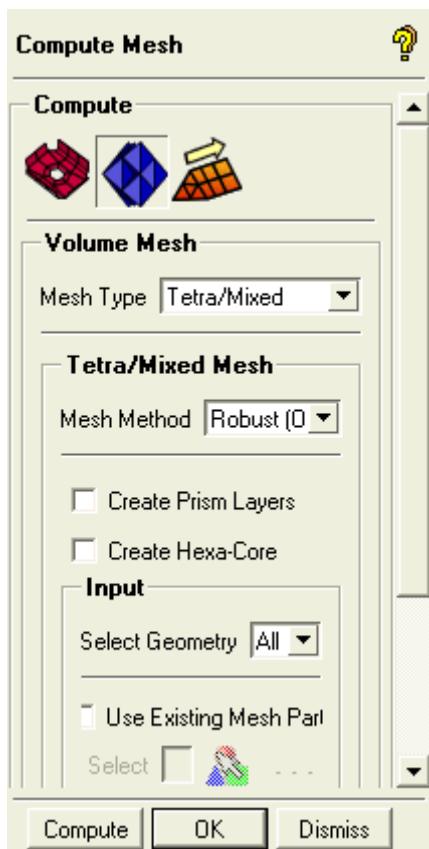
Select Save  from the main menubar.

c) Meshing

Select **Compute Mesh**  from the **Mesh** tab menubar and then click on **Volume Mesh** .

In the **Volume Mesh** window, leave all fields as the default as shown in the figure given below.

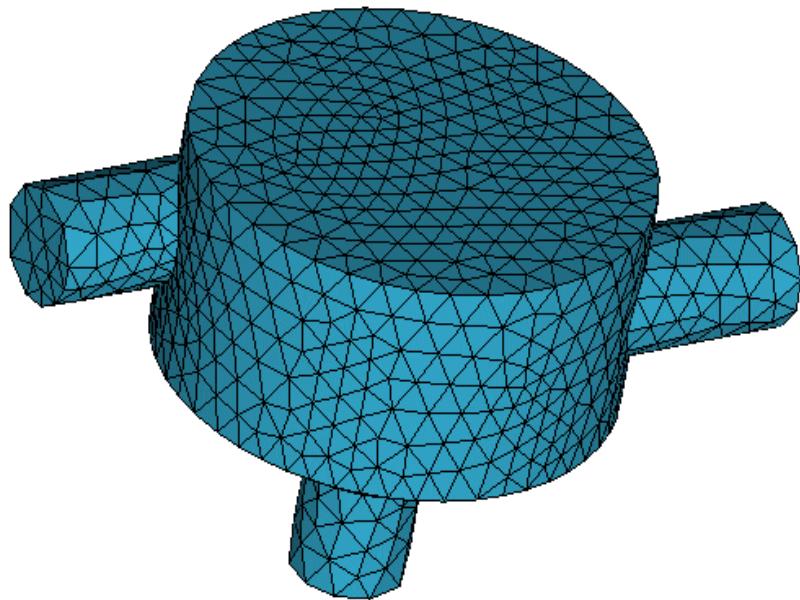
Figure 722:
Mesh Tetrahedral
Window



Press **Apply** to start the Tetra Meshing.

The tetra mesh generated is shown in the figure given below.

Figure 723:
The
Generated
Tetra Mesh

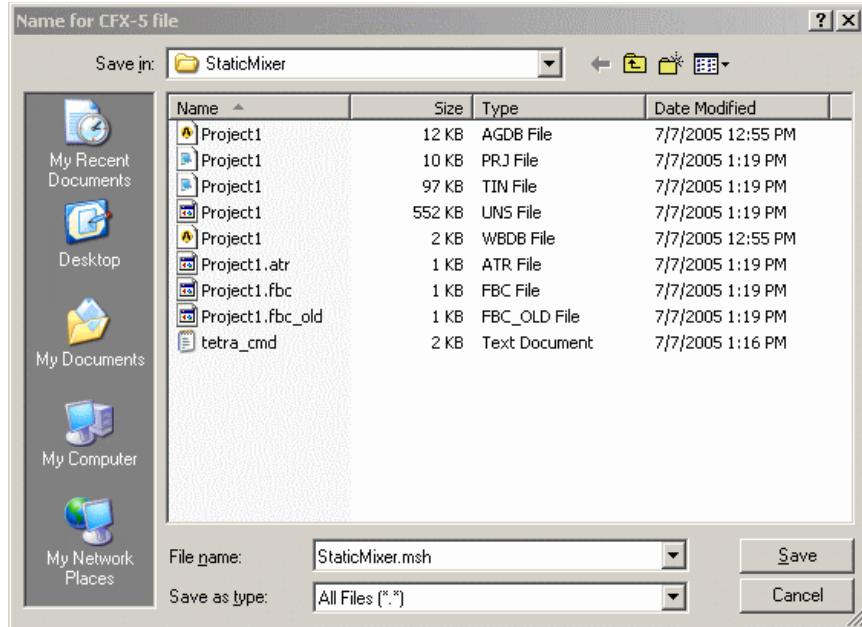


5.1.5: Writing Output

From the **Output** tab menubar, click on Output to CFX .

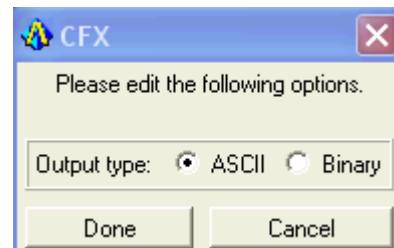
Accept the default name for File name (*StaticMixer.msh*) as shown in the figure below and press **Save**.

Figure 724:
Name for
CFX-5 file
Window



A pop-up window will appear as shown below.

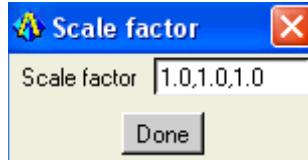
Figure 725: Output type window



Keep the default settings and press Done.

While writing the CFX file, user can scale the output through the **Scale factor** window shown. Press **Done** as no scaling is required for this tutorial.

Figure 726:
Scale Factor for
CFX-5 output



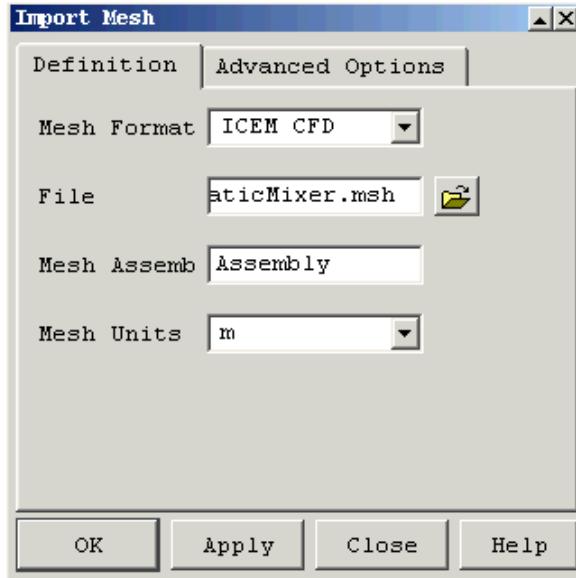
5.1.6: Exiting ANSYS ICEMCFD - CFX

Select **File > Exit** from the main menu to quit **ANSYS ICEMCFD - CFX**.

5.1.7: 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 the figure below.

Figure 727 :
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

- Creating parts for prism generation
- Defining the meshing parameters
- Creating the refined tetra mesh with prism layers
- Checking for mesh quality
- Smoothening the mesh
- Writing input file for CFX-5

5.2.2: Starting a New Project

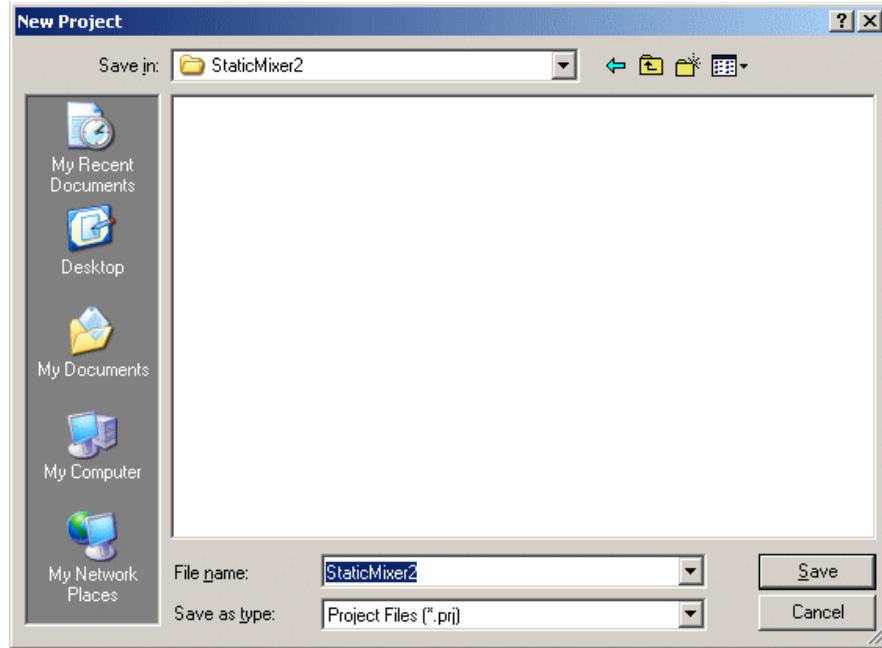
a) Creating a New Project

Launch ANSYS ICEMCFD - CFX.

Select **File > New Project** from the Main menu and enter File name as shown in the figure given below.

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	823
------------------------	--	-----

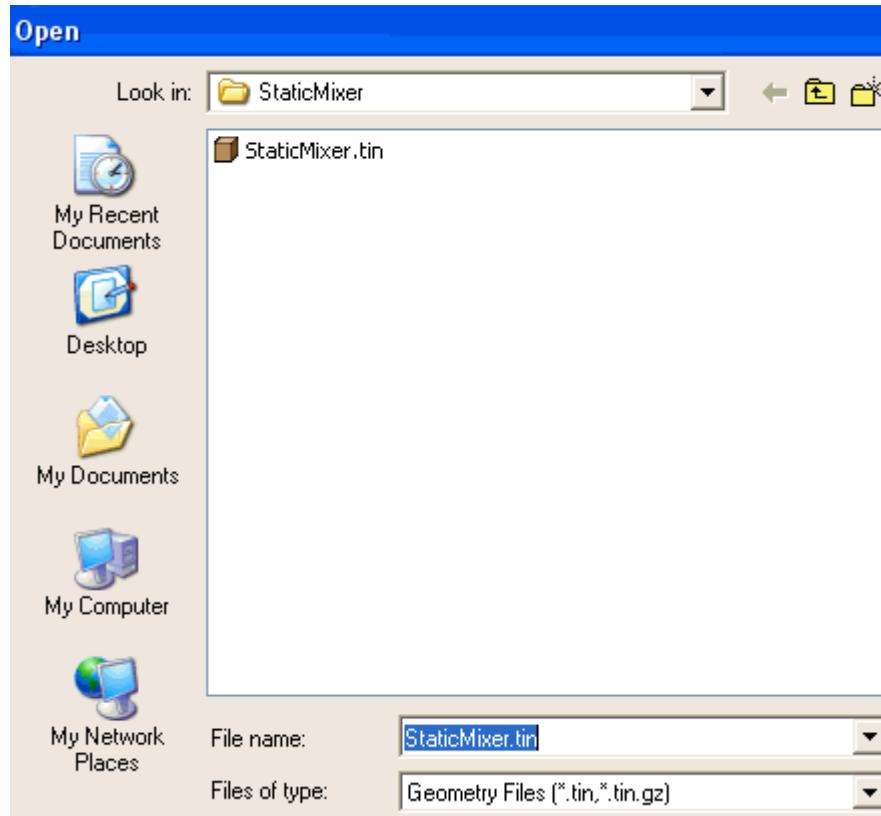
Figure 728:
New project
window



b) Loading a Geometry File

From the Main Menu, click on and select the geometry file StaticMixer.tin created in the previous tutorial.

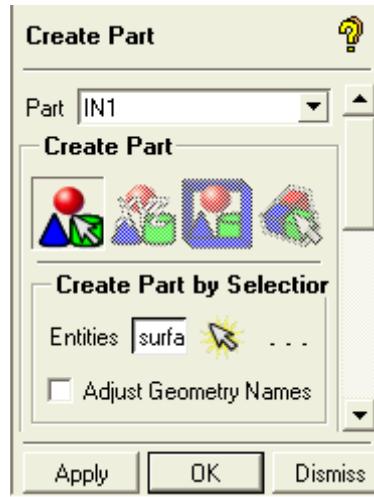
**Figure 729:
Loading the
previous
Tutorial
Geometry
File**



5.2.3: Creating Parts for Prism layers

Go to the display tree and right click on Parts > Create Part. It will open a window. Give the part name as "IN1" as shown in the figure given below.

Figure 730: Part Creation Window



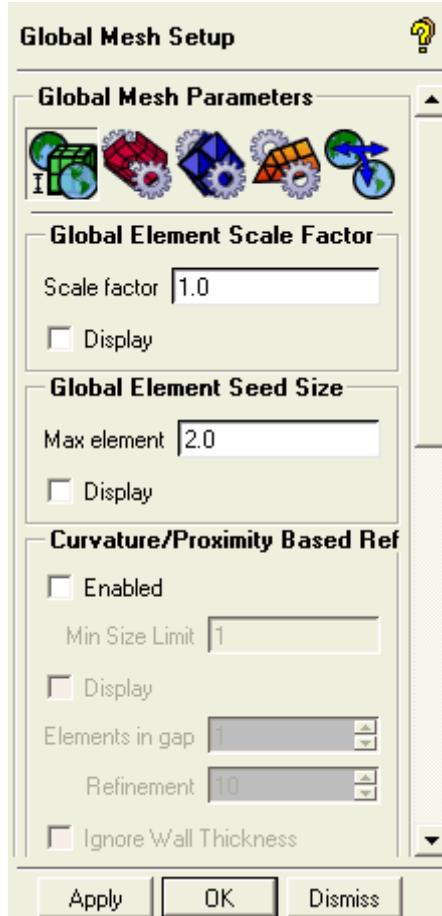
Select one of the two parallel cylindrical surfaces and Press Apply. Similarly select the other parallel cylindrical surface, give the part name as "IN2" and press Apply. Again do it for third cylindrical surface and name it as "OUTLET".

5.2.4: Mesh Generation

a) Reassigning Mesh Parameters

From the **Mesh** tab menubar click on Global Mesh Setup . Leave Scale Factor as *1* for the Global Element Scale Factor. Change **Max Element** to *2* as shown in the figure given below.

Figure 731:
Global Mesh Size window



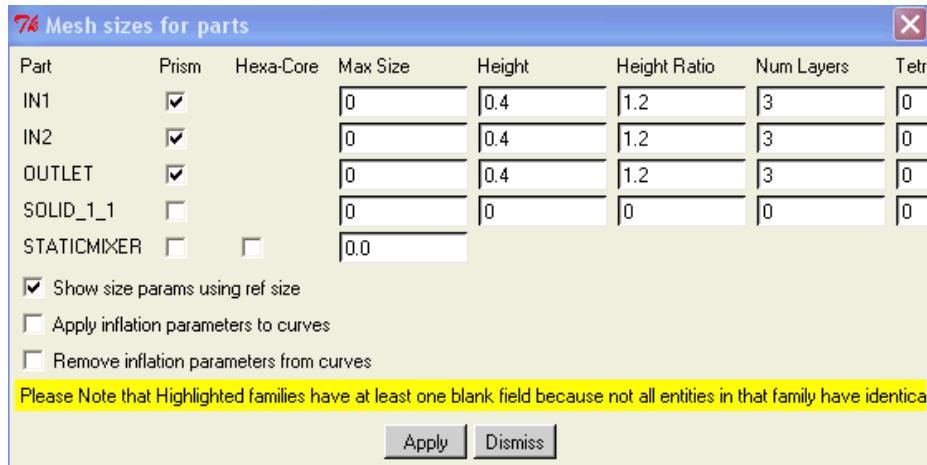
Click **Apply** to save this setting.

b) Assigning Parts for Prism Meshing

- From the **Mesh** tab menubar click on Mesh Sizes for Parts

- Check in the parts for prism layers and enter the mesh parameters as shown below.

Figure 732:
**Prism
Mesh
parameter
window**



- Press Apply and then press Dismiss.

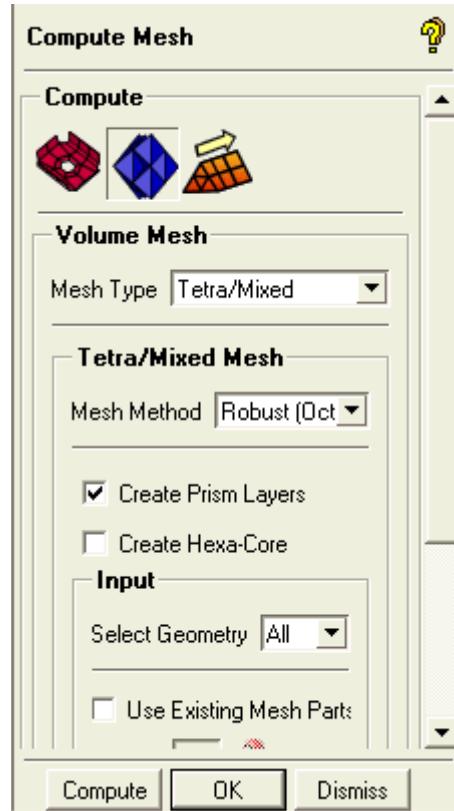
c) Saving the Project

Save the project by clicking on **Save**  from the Main Menu.

Select **Compute Mesh**  under Mesh tab menubar and then click on **Volume Meshing**  to create the refined tetrahedral mesh with prism layers on this geometry.

Toggle on "Create Prism Layers" and keep rest of the settings as default as shown in the figure given below.

Figure 733:
Compute
Mesh
Window



Click **Apply** to create the tetrahedral mesh with prism layers. Once the mesh is created, it gets loaded on the screen.

d) Verifying Mesh Quality

Click on **Display Mesh Quality** from the **Edit Mesh** tab menubar to check the quality of the mesh.

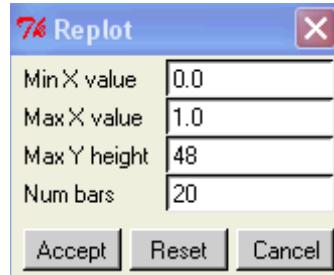
In the Quality Histogram window, right click and then click on **Replot**.

Change **Min X** value to *0*.

Change **Max X** value to *1*.

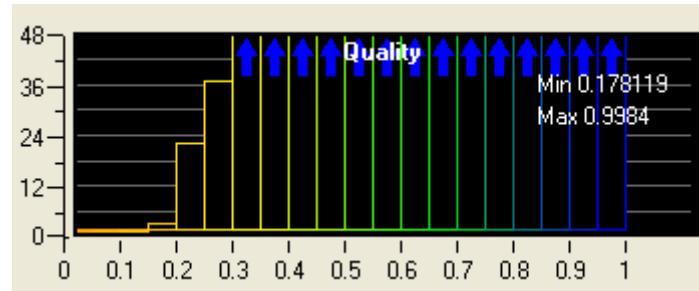
Change **Max Y** height to *48* as shown in the figure below.

Figure 734: Replot Window



Click **Accept** to replot the Histogram as shown in the figure below.

Figure 735: Quality Histogram

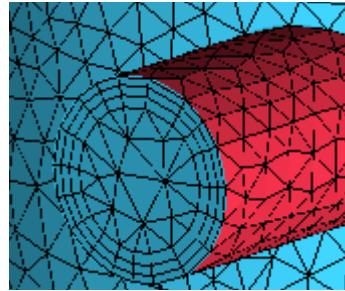


e) Saving the Project

Save the project by clicking on **Save** from the Main Menu.

After prism mesh generation, the mesh at one of the inlets will be similar to that shown the figure given below.

Figure 736:
Prism Layers at
inlet



Note: To view the mesh as solid/wire mode, right-click on **Mesh > Shells** in the Display Tree and select **Solid & Wire** option.

f) Editing the Prism Mesh

Go to the **Edit Mesh** tab menubar and click on **Smooth Mesh Globally**  to check the mesh quality.

Toggle on the options as shown in the figure given below and press **Apply**.

Figure 737: Smooth Elements Window

g) Checking Mesh Quality

- Click on  from the **Edit Mesh** tab menubar to check the quality of the mesh.

In the Quality Histogram, right click and then click on **Replot** which pops up the Replot window.

Keep the default settings and press **Apply**. It will show the quality histogram as shown in the figure given below.

Figure 738:
Histogram after
Smoothing



h) Saving the Project

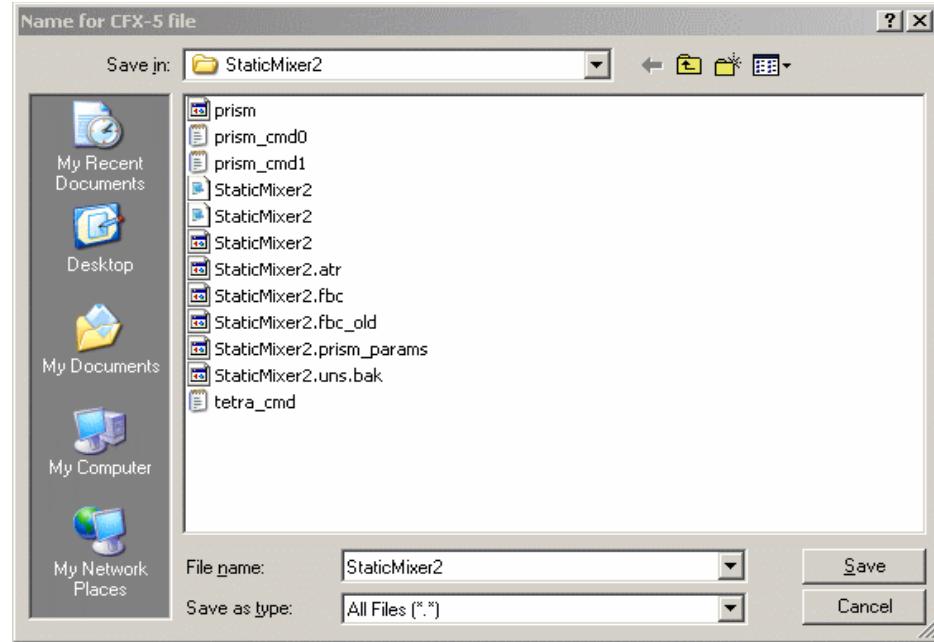
Save the project by clicking on **Save**  from the Main Menu.

5.2.5: Writing Output

From the **Output** tab menubar; click on **Output to CFX** .

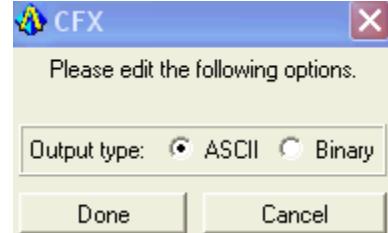
Accept the default file name as shown in the figure given below and press **Save**.

**Figure
739:
Output to
CFX
window**



A pop-up window will appear as shown in the figure given below.

Figure 740: Output type window



Keep all the settings as default and press Done.

For this tutorial, no need to do scaling of the mesh, select the default-scaling factor in the Scaling factor window and press **Done** on it to convert mesh into CFX-5 format.

5.2.6: Exiting ANSYS ICEMCFD - CFX

Select **File > Exit** from the main menu to quit **ANSYS ICEMCFD - CFX**.

5.2.7: Continuing with the Static Mixer (Refined Mesh) Tutorail

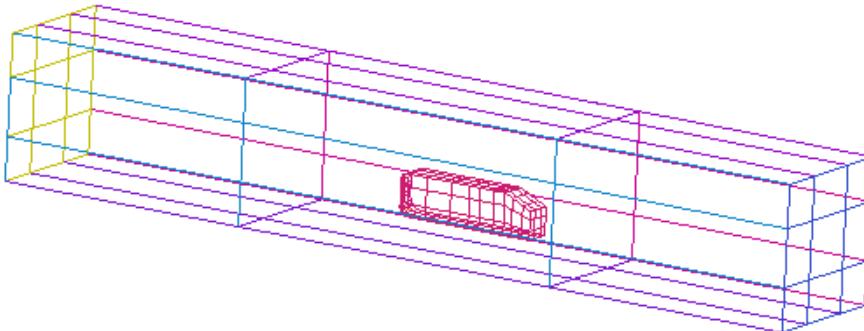
As described in previous tutorial, the user can continue CFX-5 Static Mixer (Refined Mesh) Tutorial from the section *Defining the Simulation in CFX-Pre*.

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 sections of the CFX-5 Blunt Body tutorial, picking up with *Defining the Simulation in CFX-Pre*.

**Figure
741:
Geometry
Model**



a) Steps Involved in this Example

- Importing the Geometry
- Geometry modification
- Assigning mesh parameters
- Creating the refined tetra mesh with prism layers
- Splitting prism layers
- Checking the mesh quality
- Writing output file for CFX-

5.3.2: Starting a New Project

a) 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 742
**Ansys workbench desired unit
length window**

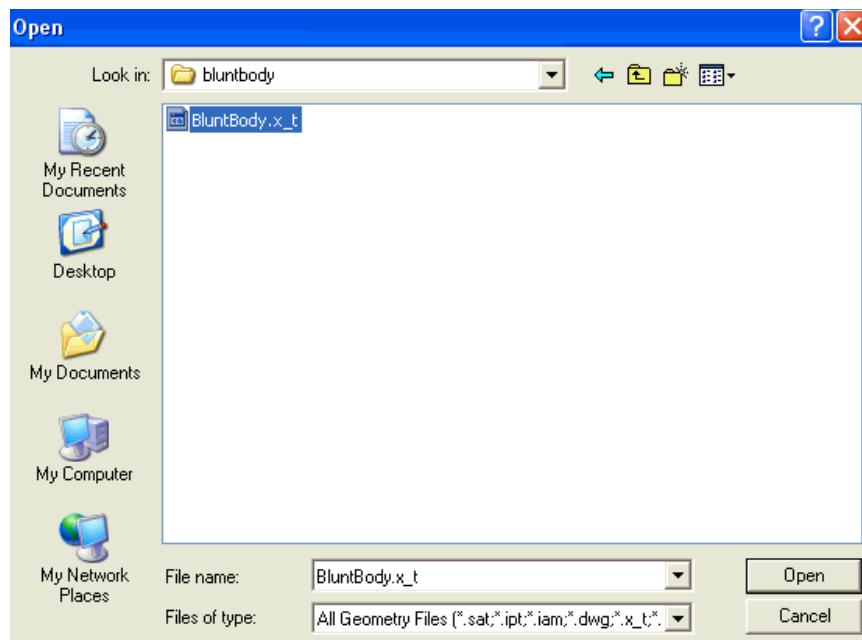


5.3.3: Geometry

a) Importing a Geometry File

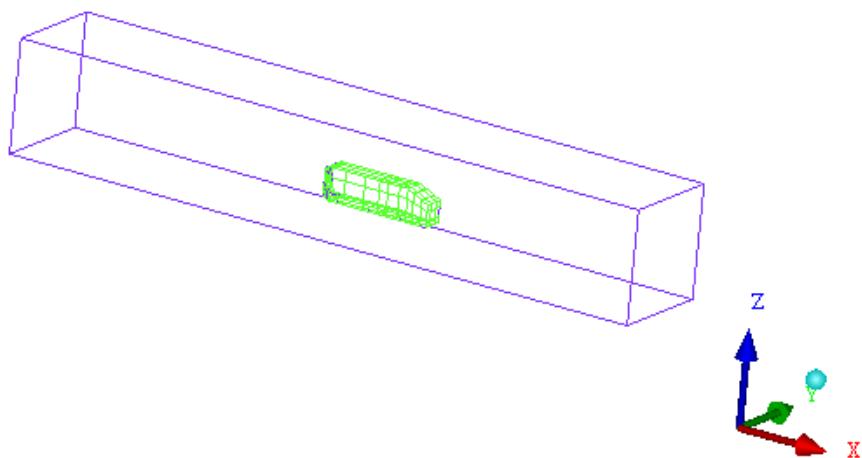
From the Main Menu, select **File > Import External Geometry file**. Select the `BluntBody.x_t` file as shown in the figure give below. The input files for this tutorial are found in the Ansys Installation directory, under `../docu/Tutorials/AICFX_Tutorial_Files/parasolid`.

Figure 743:
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 design modeler space as shown in the figure given below.

Figure 744:
Imported
Geometry



Now we have to proceed for geometry repairing and meshing in advanced meshing. To go into the Advanced meshing, Click on **Project** in the upper left corner. After opening project window, click on **Proceed to Advanced meshing**. This will open up the advanced-meshing interface. The project window is shown in the figure given below.

Figure 745
DesignModeler tasks window



b) 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 defined only by curves.

The rest of the boundary is made of several curves. Instead of joining the curves and creating surfaces from joined curves, a simpler approach is taken.

Go to the Geometry tab menu bar and select **Delete Curves** .

Click on the selection icon  and press the hotkey "a" on the keyboard to select all the curves in the model. Note that the mouse cursor must be in the Display window before pressing hotkeys on the keyboard.

Click **Apply** to delete all the curves.

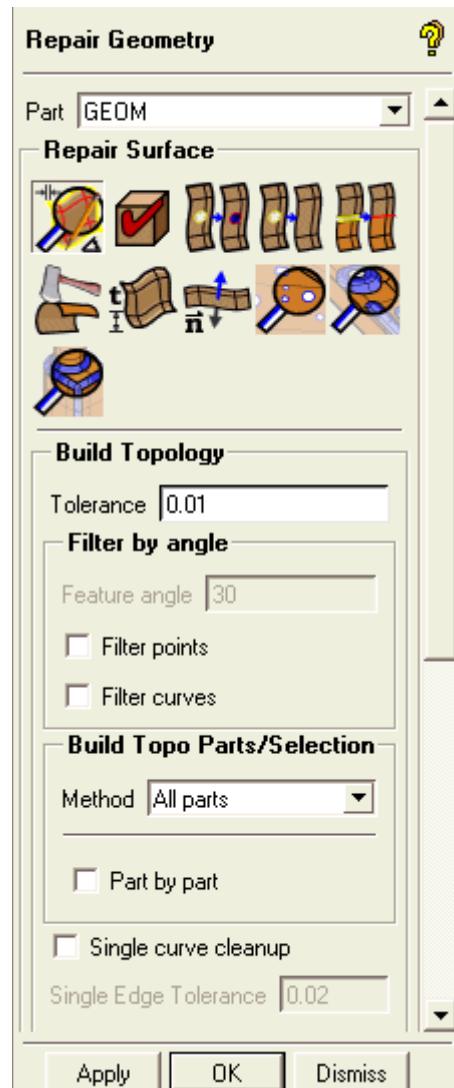
Similarly delete all the points.

To get only the necessary curves and points,

Click on **Repair Geometry**  from the **Geometry** tab menubar.

Go to **Build Topology** . Toggle on "Join edge curves" and keep rest of the settings as default as shown in the figure given below.

Figure 746:
Repair Geometry window



Press **Apply**.

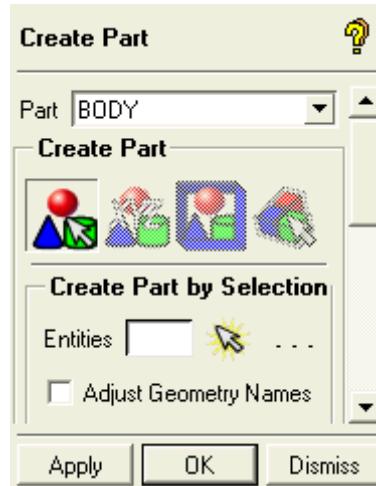
This creates only the necessary curves and points.

Note: Build Topology will turn ON the **Color by Count** and **Show Wide** option of the Curves Display. User can turn OFF them for the normal display of Curves.

c) Part Creation for Prism layers

Next right click on Parts in the Display tree and select the option **Create Part**. It will open the window as shown in the figure given below.

Figure 747:
Create Part window

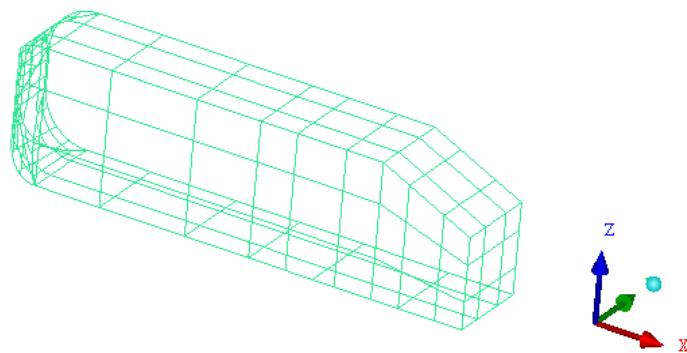


Give the Part name as *BODY* and click on **Create Part by Selection** .

Switch off **Curves** and **Points** in the Display Tree.

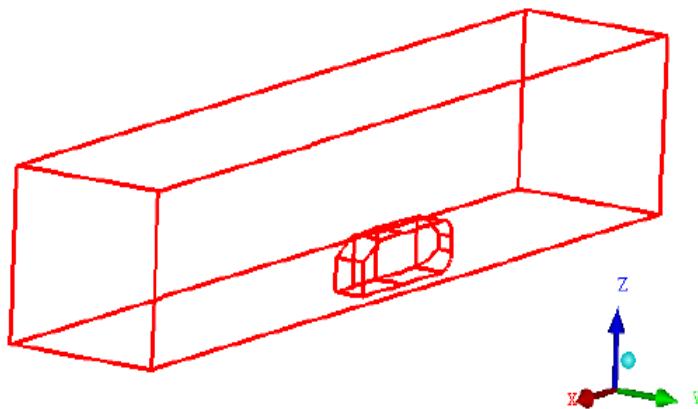
Click on the selection icon  and select the surfaces as shown in the figure given below. Only the selected surfaces need to be put into BODY.

Figure 748:
Surfaces for BODY



Press **Apply** to move the surfaces into the part **BODY**
Turn OFF the display of Surfaces and Points in the tree to make sure that all the curves are in red as shown in the figure given below.

Figure 749:
**All curves in
Red color**



d) Body Creation

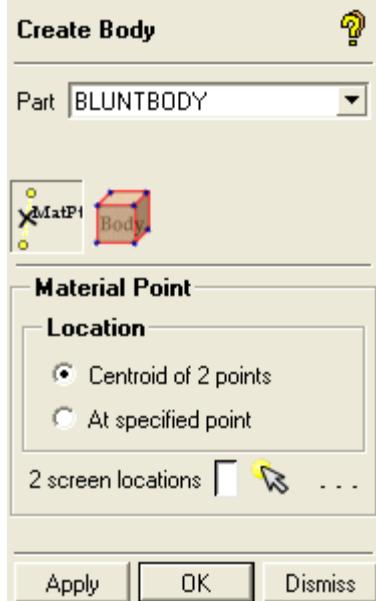
Go to the **Geometry** tab menubar and select **Create Body** .

Switch on Surface in the Display Tree.



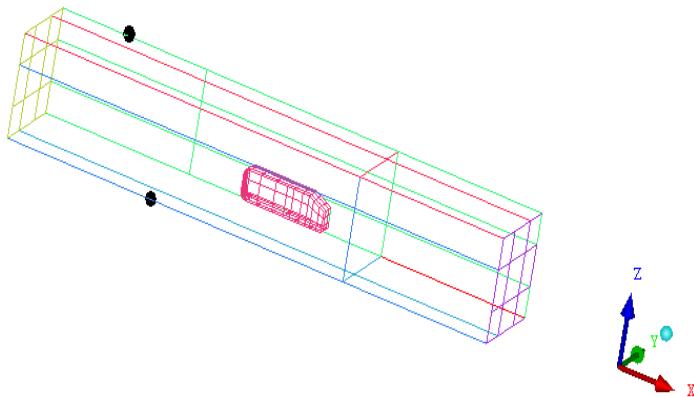
Give the Part name as *BLUNTBODY* and Click on (default) and toggle on the "**Centroid of 2 Points**" (default) option as shown in the figure given below.

Figure 750:
Create Body
Window



Click on the selection icon and select two opposite corners on the screen as shown in the figure given below and press the middle mouse button.

Figure 751:
Two Opposite
points for
Material point



Press **Apply**.

5.3.4: Mesh Generation

a) Global Mesh Parameters

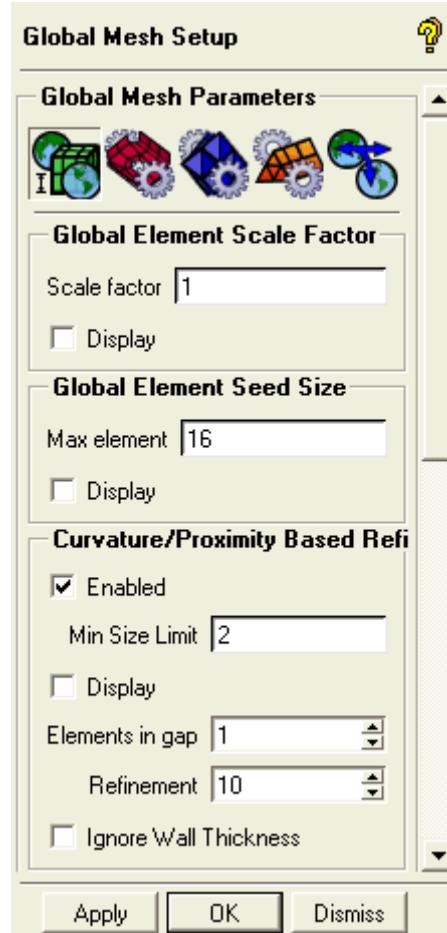
From the **Mesh** tab menubar click on **Global Mesh Setup** .

Leave Scale Factor as *1* for Global Element Scale Factor.

Change Max Element to *16* for Global Element Seed Size.

Switch on "Enable" option under 'Curvature/Proximity Based Refinement' and give "Min Size Limit" of *2* as shown in the figure given below.

Figure 752:
Global Mesh Size window



Press **Apply** to save these settings.

b) Surface Mesh Parameters

Click on **Surface Mesh Setup**  from the **Mesh** tab menubar.

Click on the selection icon  and select all the surfaces of the BODY. This will open "Select geometry" toolbar as shown in the figure given below.

Figure 753:
Select
geometr
y
toolbar



Click on **Select items in a part** , select "BODY" part as shown in the figure given below.

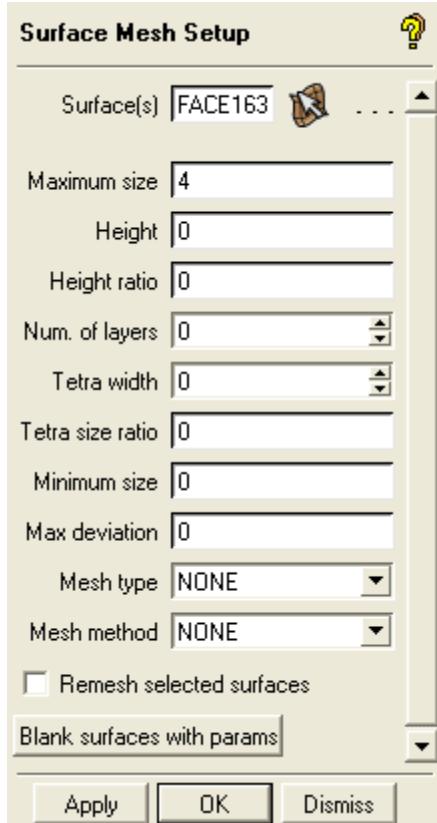
Figure 754: Part
selection window



Press Accept.

Enter Maximum Size as 4 as shown in the figure given below and keep rest of the settings as default.

Figure 755:
Surface Mesh
Size window



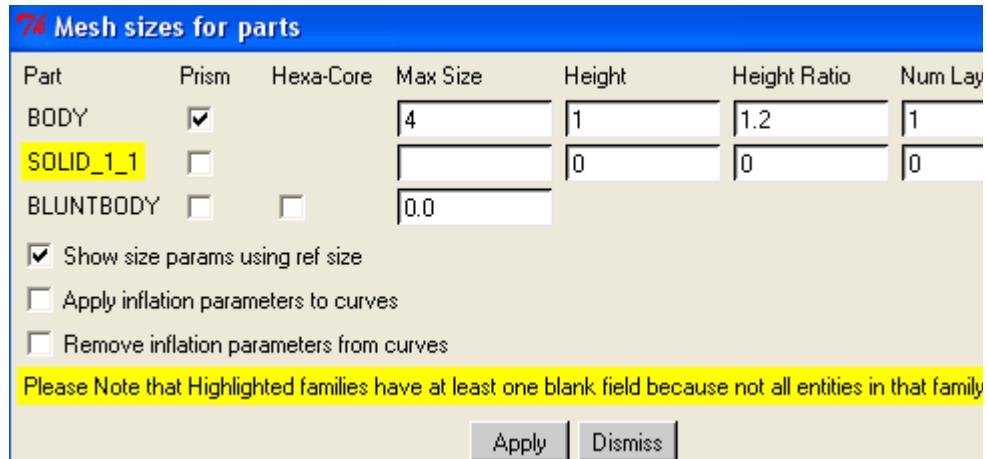
Press Apply.

Similarly assign maximum size 10 to the part 'SOLID_1_1'.

c) Assigning Prism Mesh Parameters

Click on **Part Mesh Setup**  for setting prism mesh parameters. Enter the values as shown in the figure given below.

**Figure
756:
Prism
Mesh
Parameter
window**



Press Apply.

d) Saving the Project

Save the project by clicking on **Save**  from the Main Menu.

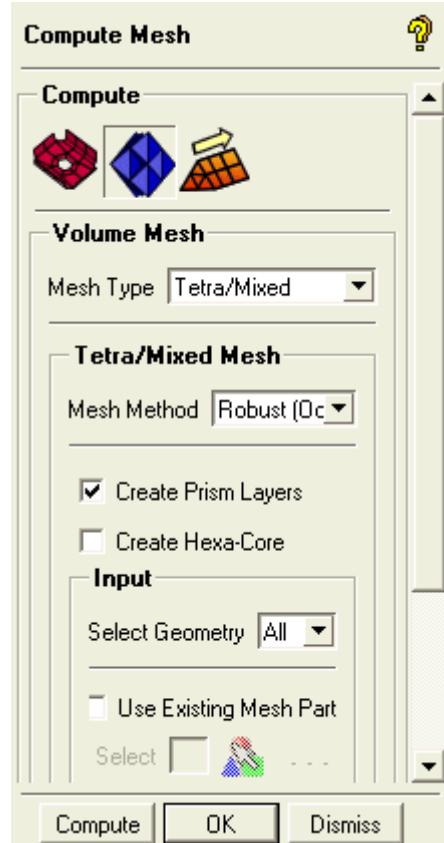
e) Meshing

Select **Compute Mesh**  from the **Mesh** tab menubar to create the refined tetrahedral mesh with prism layer on this geometry.

Under the Compute Mesh window, choose **Volume Meshing** .

Toggle on "Create Prism Layers" and keep rest of the settings as default as shown in the figure given below.

Figure 757: Compute Mesh window



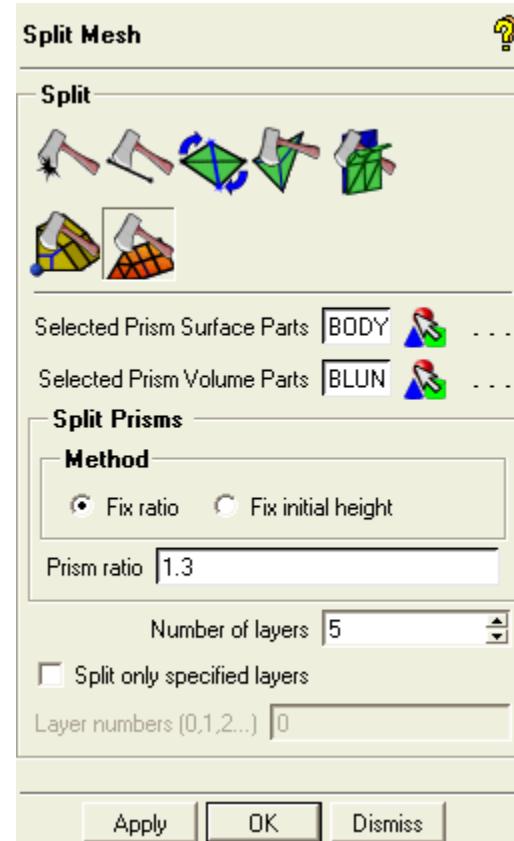
Click **Apply** to create the tetrahedral mesh with prism layer. Once the mesh is created, it gets loaded on the screen.

5.3.5: Splitting Prism Layer

Click on **Split Mesh**  from Edit Mesh tab menubar and select **Split Prisms** .

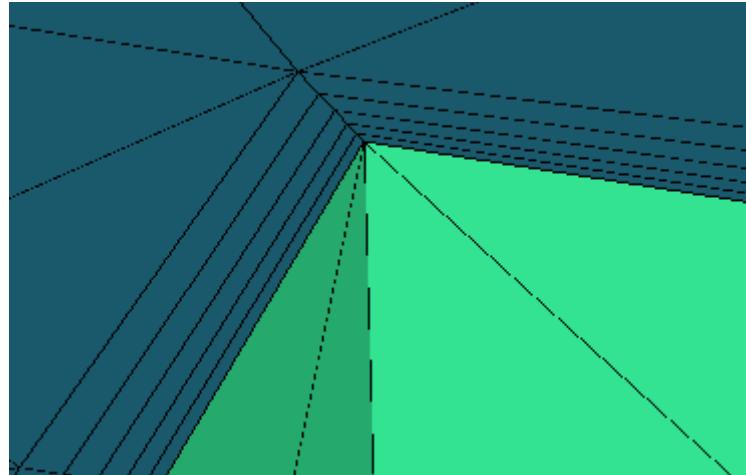
Select Prism Surface Parts as "BODY" & Prism Volume Parts as "BLUNTBODY", enter Prism ratio 1.3 and Number of Layers 5 as shown in the figure given below.

Figure 758: Split Prism window



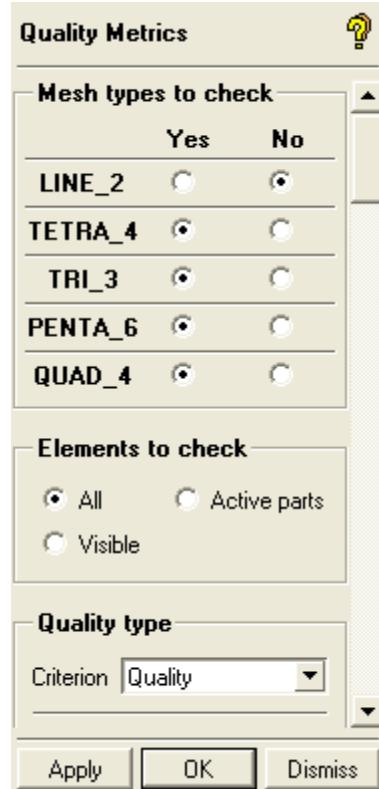
- Press Apply. It will create the Prism layers as shown in the figure given below.

Figure 759: Prism layers after split

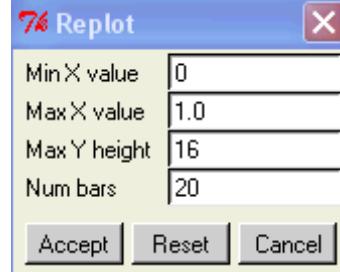


5.3.6: Checking Mesh Quality

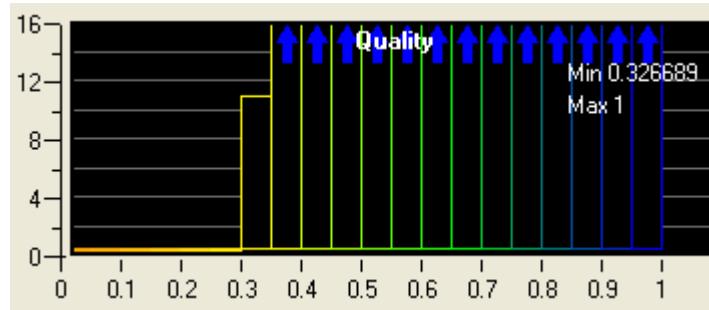
- Click on **Display Mesh Quality**  from **Edit Mesh** tab Menubar. A quality metrics window will get open as shown in the figure given below.

Figure 760: Quality Metrics window

- Keep all the settings as default and Press Apply.
- Right click on Quality Histogram and click on Replot. A pop-up window will appear, enter the values as shown in the figure given below.

Figure 761: Replot window

- This will show the quality histogram as shown in the figure given below.

Figure 762: Quality histogram window

5.3.7: Saving the Project

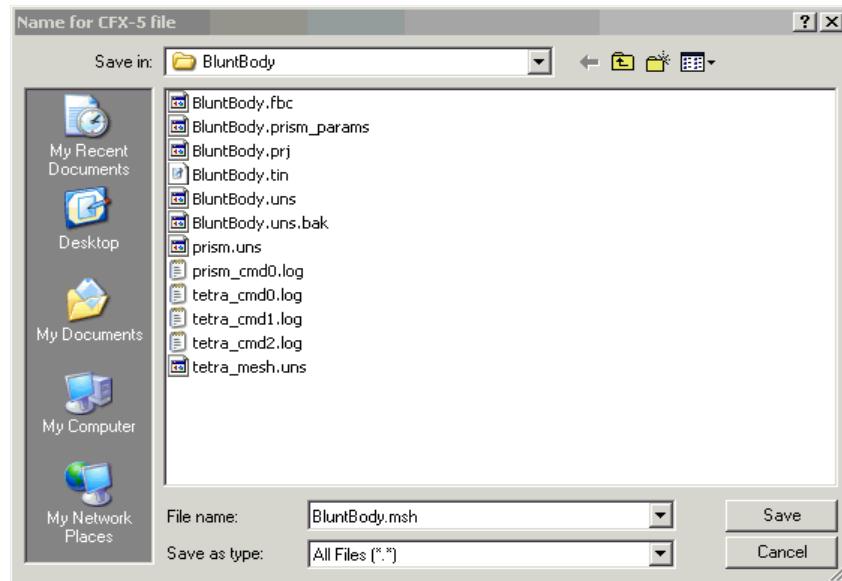
Save the project by clicking on **Save** from Main Menu. If Overwrite window occurs, press Yes.

5.3.8: Output

From **Output** tab menubar; click on **Output to CFX** .

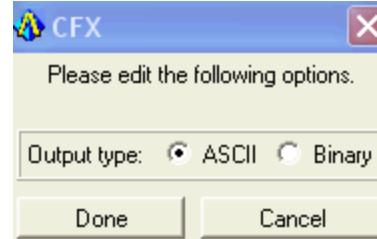
Accept the default File name for CFX file as shown in the figure given below and press **Save**.

Figure 763:
Output to
CFX window



A pop-up window will appear as shown in the figure given below.

Figure 764: Output type window



Keep the settings as default and press Done.

When it asks for the scaling factor, select default factor i.e. (1.0, 1.0 ,1.0) to complete the translation.

5.3.9: Continuing with the Blunt Body Tutorial

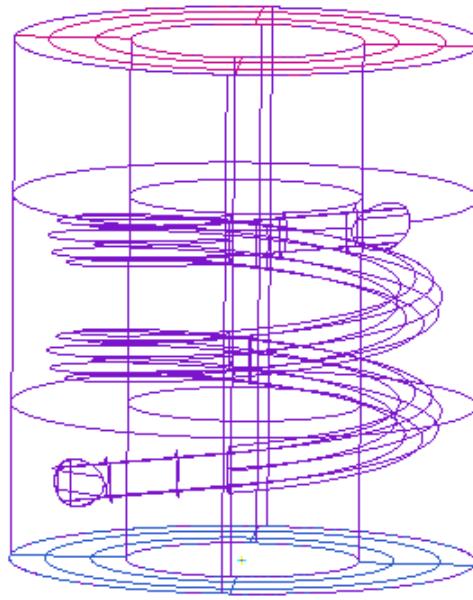
As described in Tutorial 1, the user can continue CFX-5 Flow Around a Blunt Body Tutorial from the section *Defining the Simulation in CFX-Pre*.

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 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 the figure given below.

Figure 765:
Geometry
Model



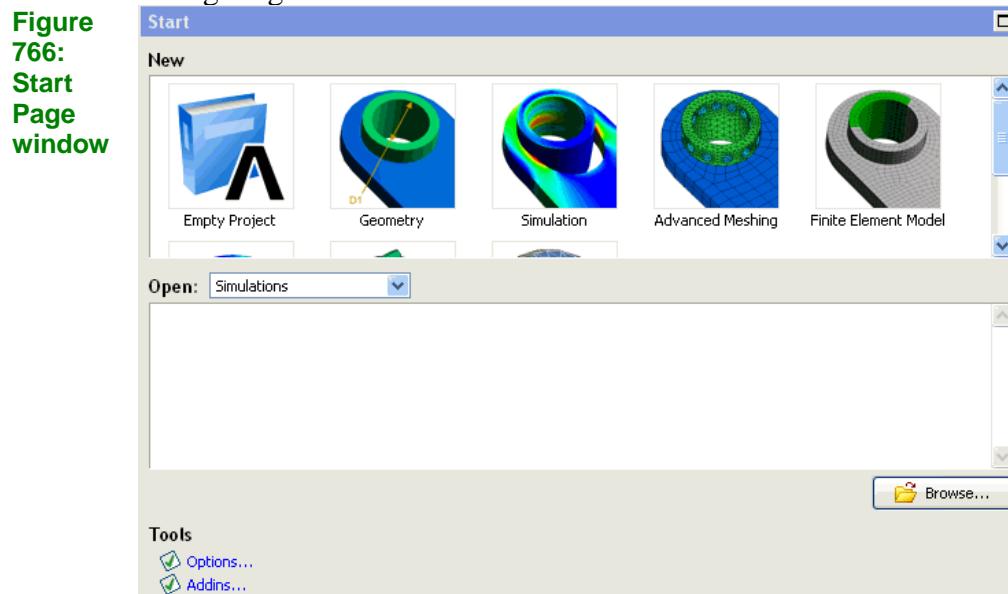
a) Steps Involved in this Example

- Importing the geometry in **Design modeler**
- Proceeding to the Advanced meshing
- Assigning mesh parameters
- Creating the Tetra mesh with prism layers
- Splitting prism layer
- Checking for quality
- Writing Output file for CFX-5

5.4.2: Starting a New Project

a) Creating a New Project

Launch the Ansys Workbench. It will open the start page window as shown in the figure given below.



Select "Empty Project"

Click on "New Geometry" to open Design Modeler. A pop-up menu will appear asking for the unit selection as shown in the figure given below.

Figure 767: Unit Selection window



Keep the default unit as Millimeter and Press OK.

5.4.3: Geometry

a) Importing a Geometry File

From the Main Menu, select File > Import external Geometry file. The input files for this tutorial are found in the Ansys Installation directory, under/docu/Tutorials/AICFX_Tutorial_Files/parasolid. Select the HeatingCoil.x_t file supplied by browsing and Press "open".

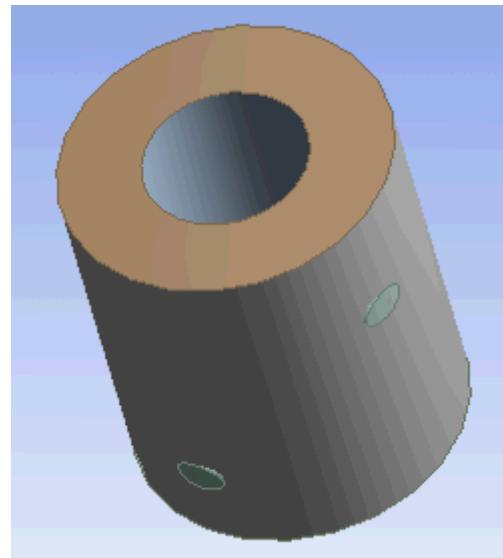
Press **Generate** from top menu.

This loads the geometry file automatically after the conversion.

The imported geometry appears as shown in the figure given below.

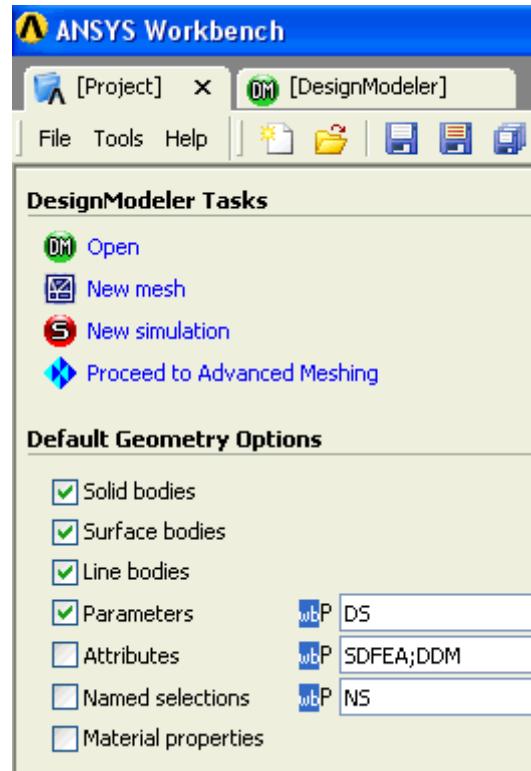
Heating Coil

Figure 768:
Imported Geometry



- Click on the Project Tab. This will open the Project Page as shown in the figure given below.

Figure 769
Proceed to advance meshing window



- Click on "Proceed to Advanced Meshing". This will shift geometry into the ICEMCFD _CFX environment.

b) Geometry Manipulation

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. 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 Curves**.

Click on the selection icon 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.
Similarly delete all the points.

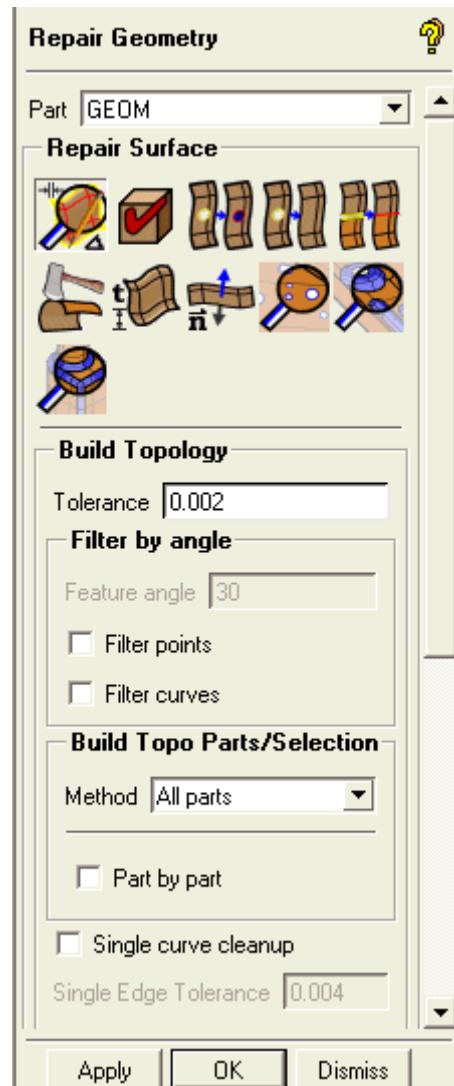
c) **Creating Curves and Points**

To get only the necessary curves and points,

Click on **Repair Geometry**  from the Geometry tab menubar.

Click on **Build Topology**  and keep all the settings as default as shown in the figure given below.

Figure 770:
Repair Geometry window



Click **Apply**.

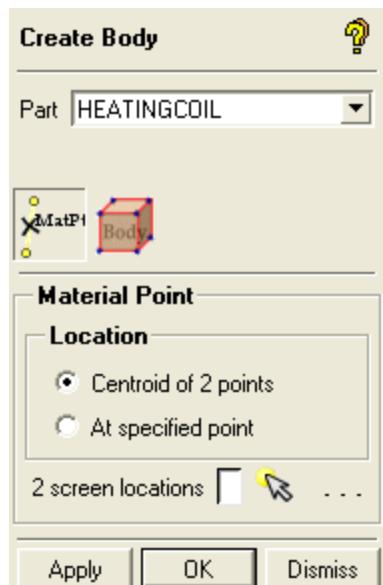
Note: Build Topology will turn ON the **Color by Count** and **Show Wide** option of the Curves Display. Right-click on **Curves** in the display tree to change the display options. User can turn OFF them for the normal display of Curves.

d) Body Creation

Go to the **Geometry** tab menubar and select **Create Body** .

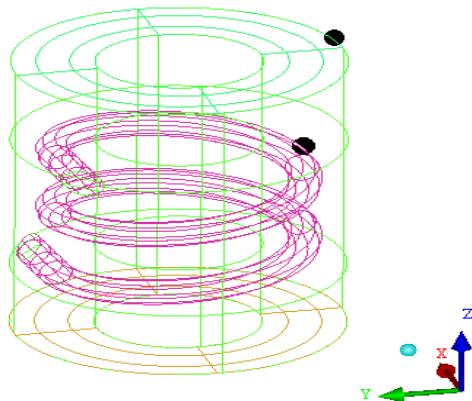
Give *HEATINGCOIL* as the Part Name and click on **Material Point**  and toggle on the **Centroid of 2 points** option (default) as shown in the figure given below.

Figure 771:
Create Body
window



Click on the selection tool , select two opposite corners on screen as suggested in the figure given below and press the middle mouse button.

Figure 772:
**Two Opposite
points for
material
points**



Press **Apply** to create the Material point.

5.4.4: Mesh Generation

a) Global Mesh Parameters

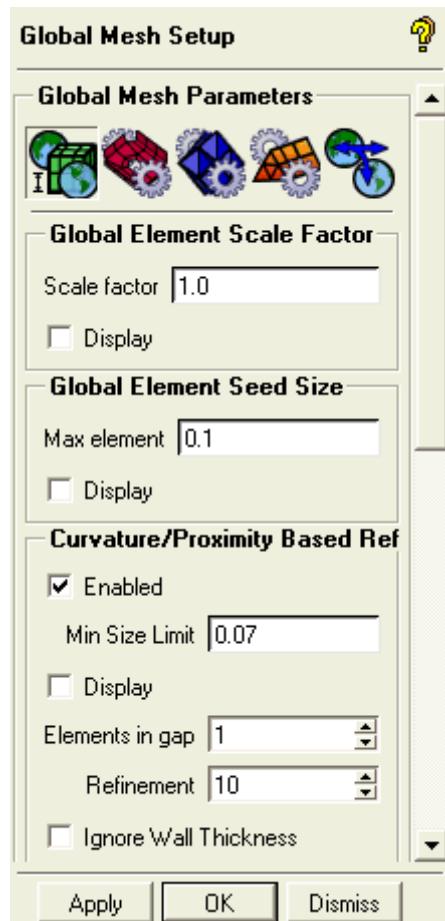
From the **Mesh** tab menubar, click on .

Leave Scale Factor as *1* for Global Element Scale Factor.

Change Max Element to *0.1* for Global Element Seed Size.

Switch on **Enable** option under 'Curvature Proximity Based Refinement' and give **Size** of *0.07* as shown in the figure given below.

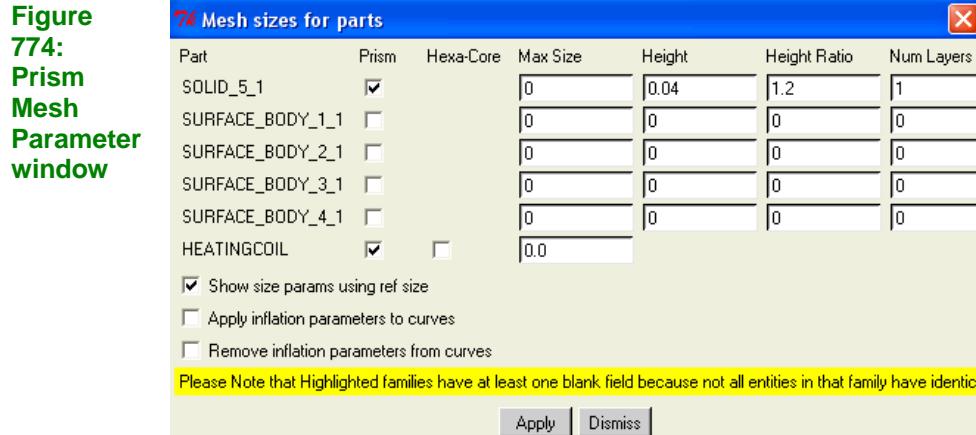
Figure 773:
Global Mesh Size
window



Click **Apply** to save this setting.

b) Defining Prism Mesh Parameters

- Click on **Part Mesh Setup** . A pop-up window will appear. Enter the values as shown in the figure given below.



- Press Apply and then press Dismiss.

c) Saving the Project

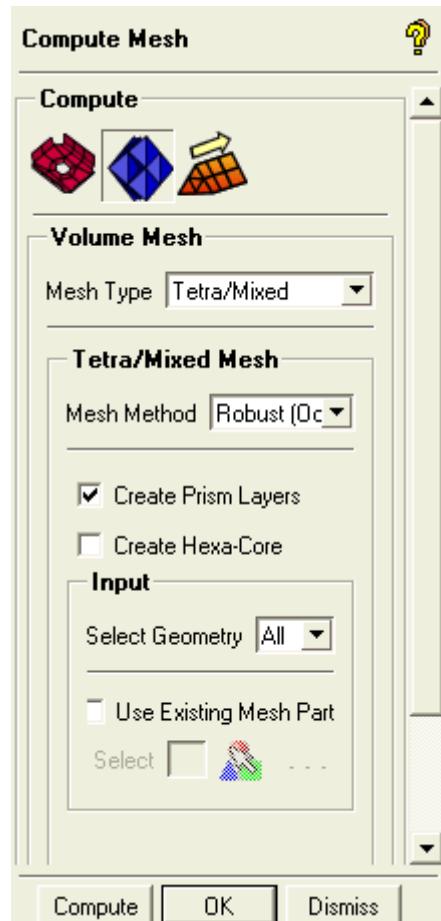
Save the project by clicking on **Save**  from Main Menu. Enter the file name as **HeatingCoil** and press Save.

d) Meshing

Select **Compute Mesh**  from the **Mesh** tab menubar and then click on **Volume Meshing**  to create the refined tetrahedral mesh with prism layer on this geometry.

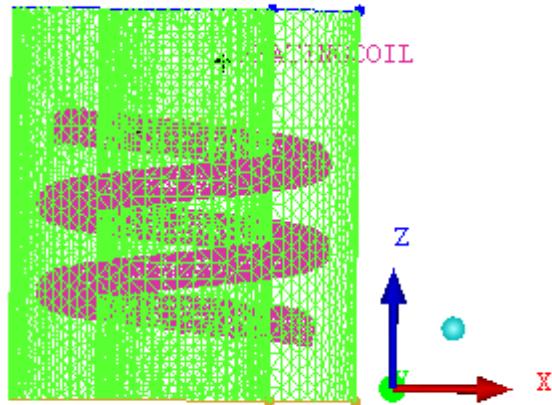
Toggle on "Create Prism Layers" and keep rest of the settings as default as shown in the figure given below.

Figure 775: Compute Mesh window



Click **Compute** to create the tetrahedral mesh. Once the mesh is created, it gets loaded on the screen as shown in the figure given below.

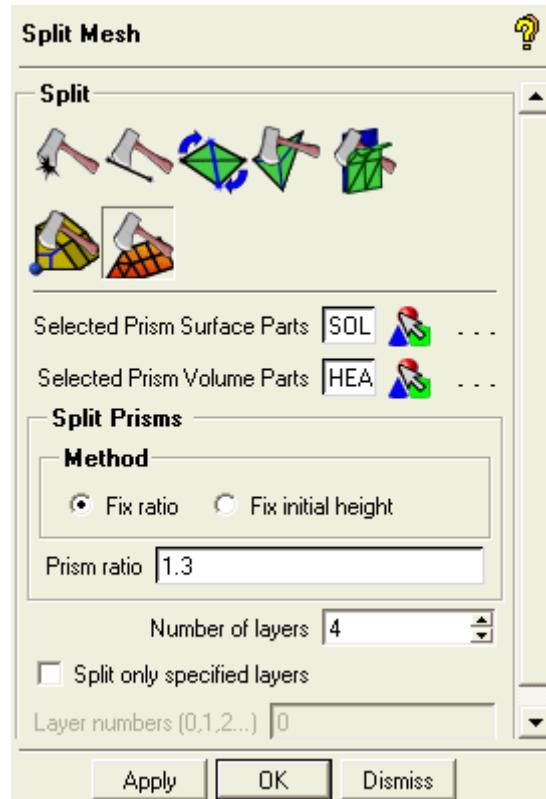
Figure 776:
Mesh



5.4.5: Spliting Prism layer

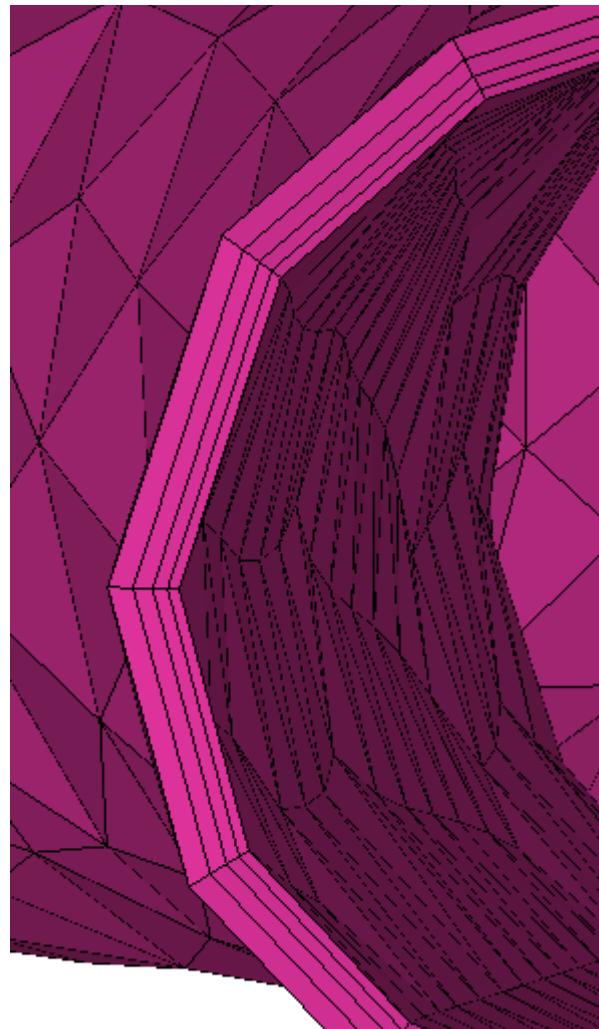
- Click on **Split Mesh** and then select **Split Prisms** .
- A pop-up window will appear. Select Prism Surface Parts as "SOLID_5_1" and press Accept. Similarly select Prism Surface Parts as "HEATINGCOIL" and Press Accept.
- Enter the values as shown in the figure given below.

Figure 777: Split Mesh window



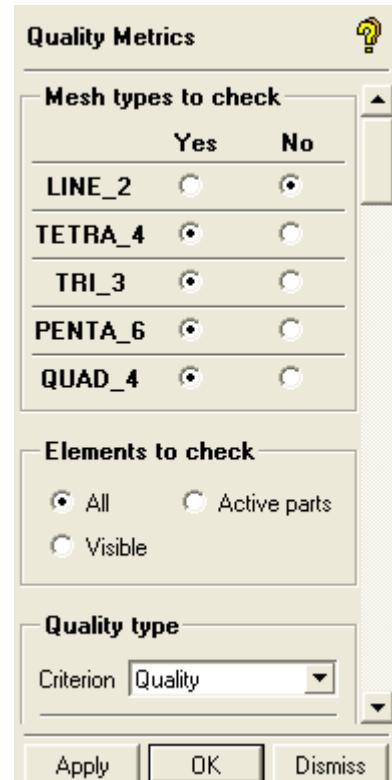
- Press Apply to split the Prism layer. The prism layer will look like as shown in the figure given below.

Figure 778: Prism Layers

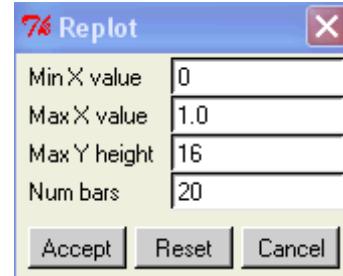


5.4.6: Checking Mesh Quality

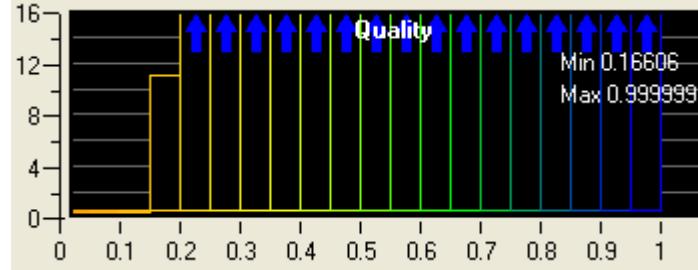
- Click on **Display Mesh Quality** under Edit Mesh menubar. This will open Quality Metrics window as shown in the figure given below.

Figure 779: Quality Metrics window

- Press Apply. A quality histogram window will appear at the lower left corner of the screen. Right click on that window, click on Replot and enter the settings as shown in the figure given below.

Figure 780: Replot window

- Press Accept. This will show the quality as shown in the figure given below.

Figure 781: Quality histogram window

a) Saving the Project

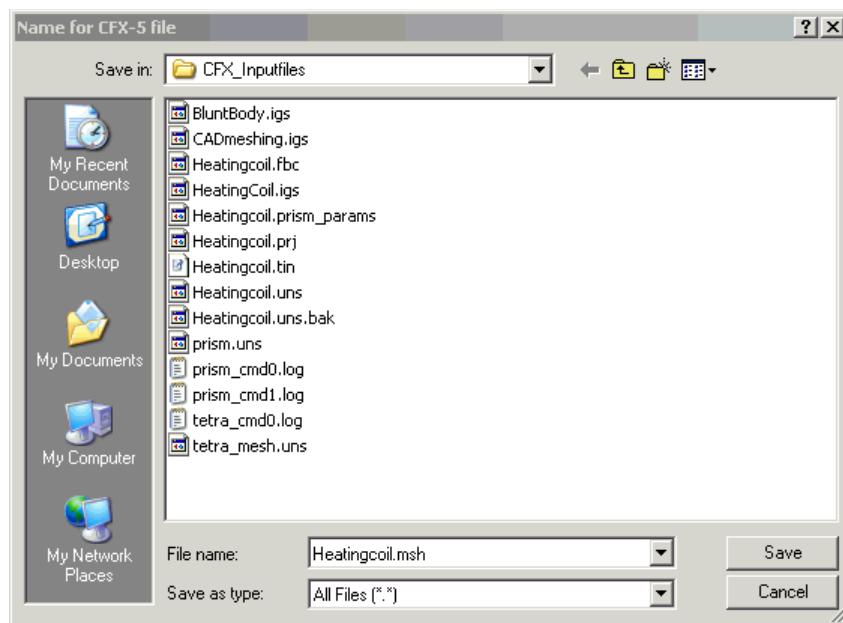
Save the project by clicking on **Save** from the Main Menu.

5.4.7: Writing Output

From **Output** tab menubar and click on **Output to CFX** .

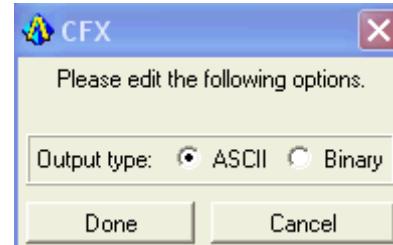
Accept the default name for CFX file as shown in the figure given below and press **Save**.

**Figure
782:
Output
to CFX**



A pop-up window will appear as shown in the figure given below.

Figure 783: Output type window



Keep the settings as default and press Done.

To translate without any scaling, select default option in the Scale Factor window.

It will complete the translation of mesh file in to CFX-5 format.

5.4.8: Exiting ANSYS ICEMCFD - CFX

Select **File > Exit** from the main menu to quit out of **ANSYS ICEMCFD - CFX**.

5.4.9: Continuing with Heating Coil Tutorial

As described in Tutorial 1, the user can continue CFX-5 Conjugate Heat Transfer in a Heating Coil Tutorial from the section *Defining the Simulation in CFX-Pre*.

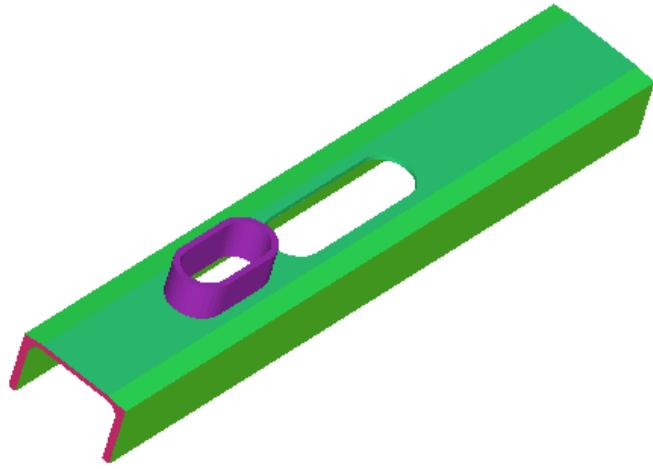
6: FEA Tutorials

6.1: Structural Meshing Tutorials

6.1.1: T-Pipe

This exercise includes meshing of T-Pipe geometry by simplifying the thickness using Mid-Surface technique. The geometry of the model is shown here.

Figure 6-1
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 in your AI*Environment installation directory:

`..\docu\Tutorials\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.

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
 Save the Project.

b) Launch AI*Environment

The input files for this tutorial can be found in your Ansys Installation directory, under/docu/Tutorials/AI_Tutorial_Files. Copy these files to your working directory and load the tetin file ‘Tpipe.tin’.

c) Geometry Editing

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 Menubar**. Enter Part as **MID** as shown in the figure below in the **Create/Modify Surface** window and leave the Name blank.



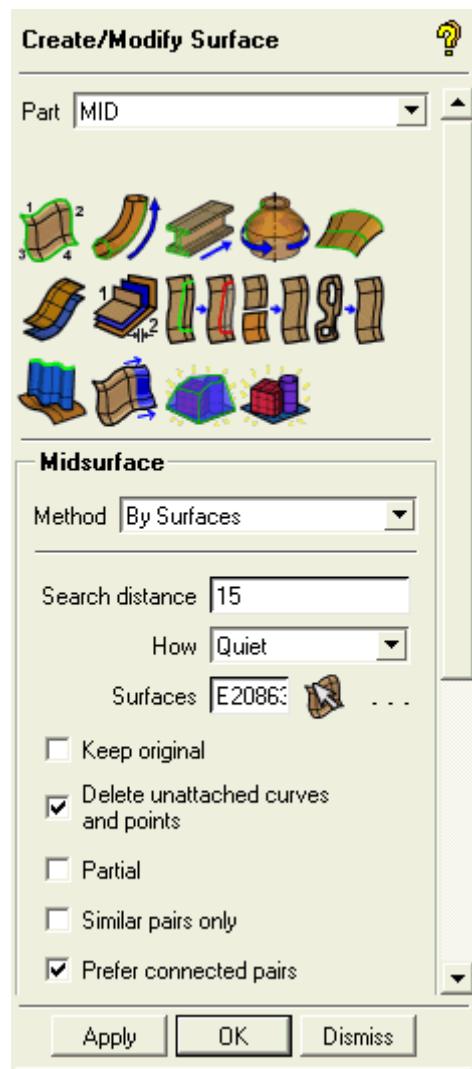
Click on  (Mid surface) icon. In the **Method** window select **By Surface**.

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.

Press ‘Surf button’  , select all the Surfaces using hotkey ‘**a**’
Enable Delete unattached Curves and Point and Prefer Connected Pairs
press Apply. The rest of the default values should be chosen as shown in
the figure below.

Note: The thickness can be measured using (Measure Distance) icon .

Figure 6-2
Create/Modify
Surface Window



Note:

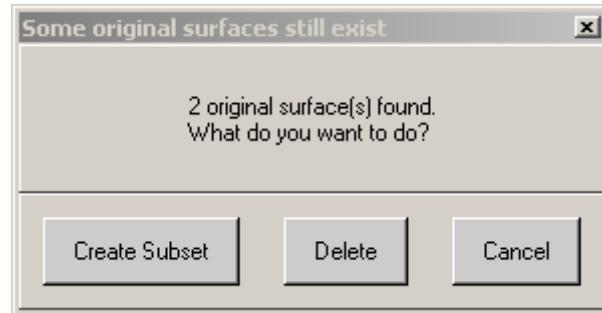
- a) From the supplied Surfaces, **AI*Environment** will automatically try to find the pair of **Surface** within the supplied distance of **15** and will quietly create the Mid-Surface without asking any questions.

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.

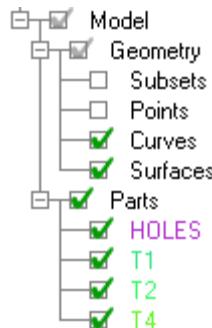
When it asks to ‘**Delete**’ some original surfaces, press Delete.

Figure 6-3
Some original Surface Exist



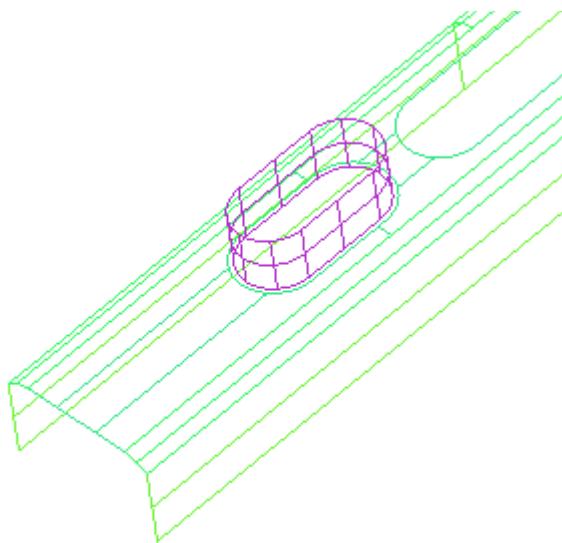
The Display Tree should appear as shown below.

Figure 6-4
Model Tree display



The corresponding model should be displayed as shown here.

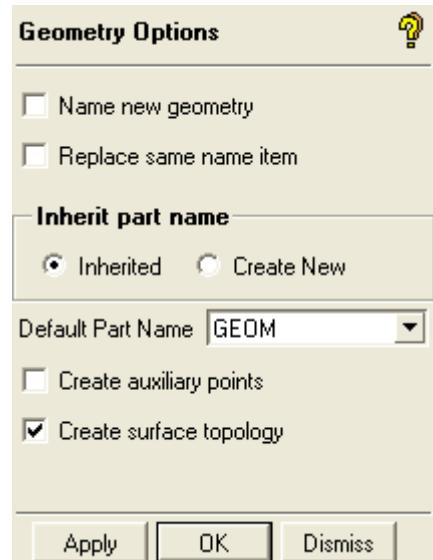
Figure 6-5
Geometry Display
Part



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.

Settings>Geometry Options

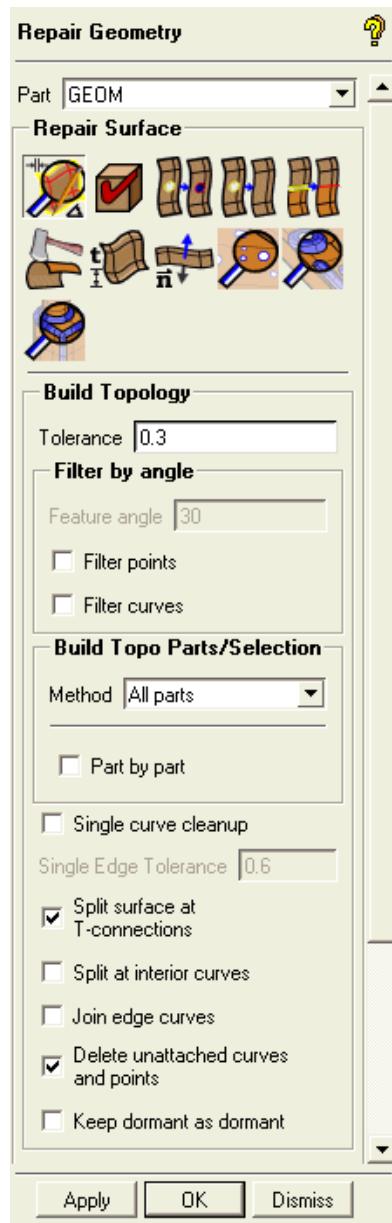
Geometry Options

Press Apply

Click on (Repair Geometry)  icon from **Geometry Menubar**. By

default the Build Topology function  is highlighted. Press Apply to extract Curves and Points from the current Surface model.

Figure 6-6
Repair
Geometry
Window



Note: Curves can be colored, and displayed by connectivity

Green = Unattached

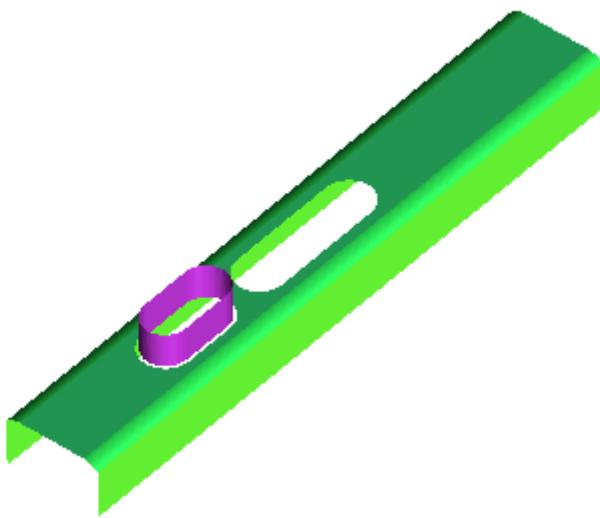
Yellow = Single

Red = Double

Blue = Multiple

In the Model Tree, turn off **Points** and **Curves** and click right mouse button on Geometry > Surfaces> Solid to display geometry modified so far.

Figure 6-7
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.

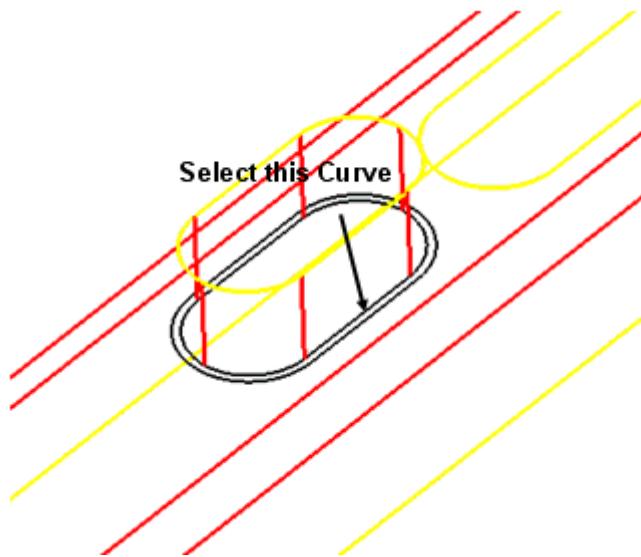
 Click on  (Remove Holes) icon in the Repair Geometry window.

**Figure 6-8
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 and press Apply to remove the hole.

Figure 6-9
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.

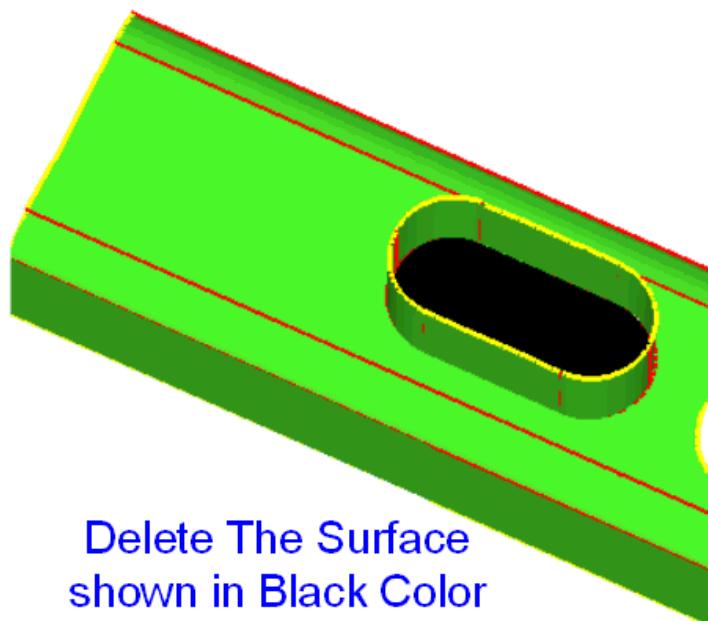
Note: User can notice that the yellow curve there has turned blue now since it's attached to surfaces from more than 2 sides.

Click on  (Delete Surface) icon from **Geometry Menubar**. Click on



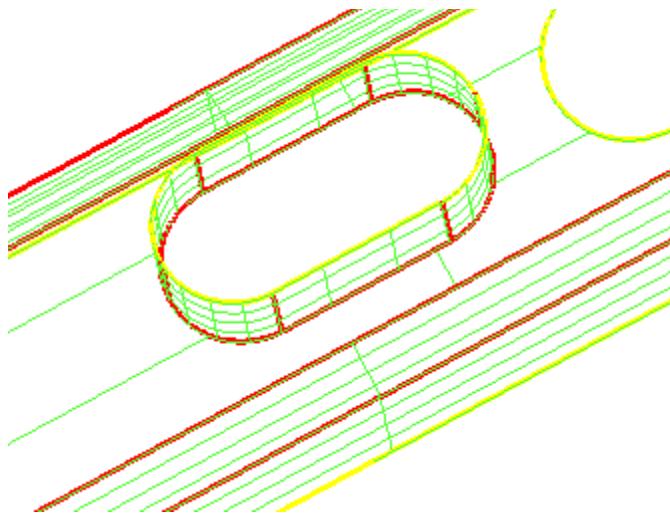
(Select Surface(s)) button to select surfaces to 'Delete'. Select the surface highlighted in the figure below and press Apply. This would remove the internal piece of the Surface, which is not required. The user can notice the changes in Color for the Curves around this surface from blue to red after deleting it.

**Figure
6-10
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 here.

Figure 6-11
Geometry
after deleting
surface



d) Mesh Parameters and Meshing

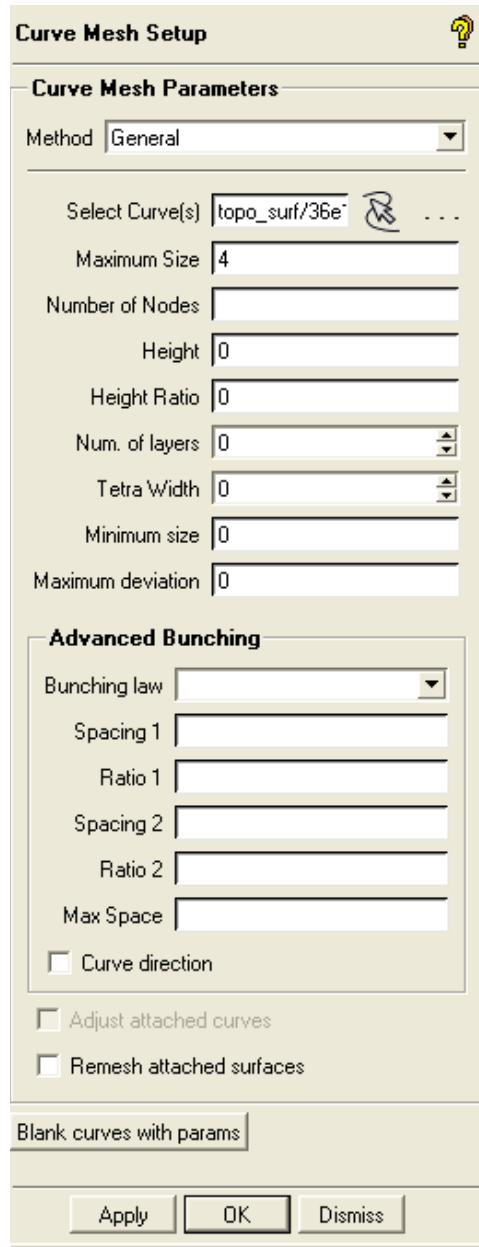
Since this is a shell model, Mesh Size information needs to be assigned to the curves. The mesh generated will be **Quad** Dominant i.e. it will have more number of QUAD elements than TRI elements. The mesh generated is associated with the geometry.

Mesh sizing

Select (Set Curve Mesh Size) icon from **Mesh Menu bar**, which pops up **Curve Mesh Size** window.

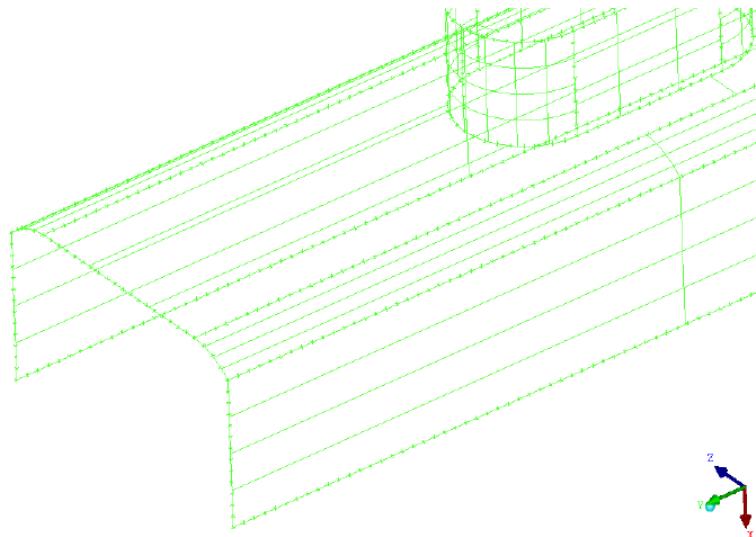
In the Method select ‘General’, click on (Select Curve(s)) button to select Curves. Place the mouse cursor in display window and press “a” from keyboard to select all curves. Enter **Maximum Size** of **4** for this case and press Apply.

Figure 6-12
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.

Figure 6-13
Curves with
Node
Spacing ON



Now de-select Curves > Curves Node Spacing under the the Model Tree.

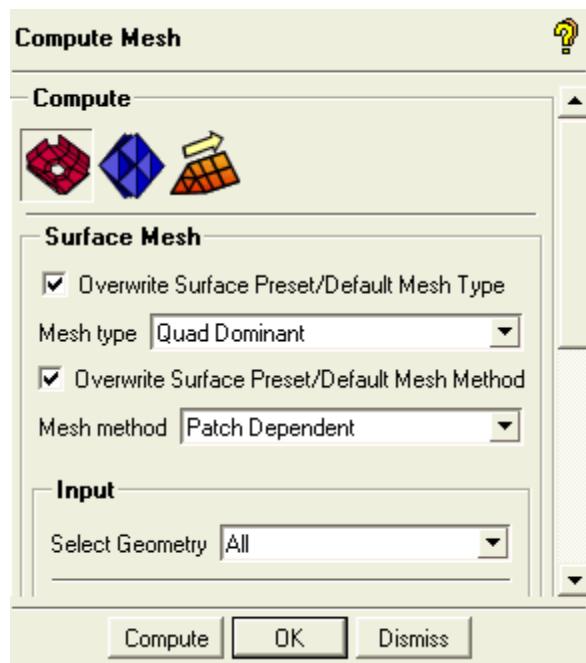
Meshing



Select the Compute Mesh > Surface Mesh icon from **Mesh Menu bar**. Select the Settings as shown in the below figure and press Apply

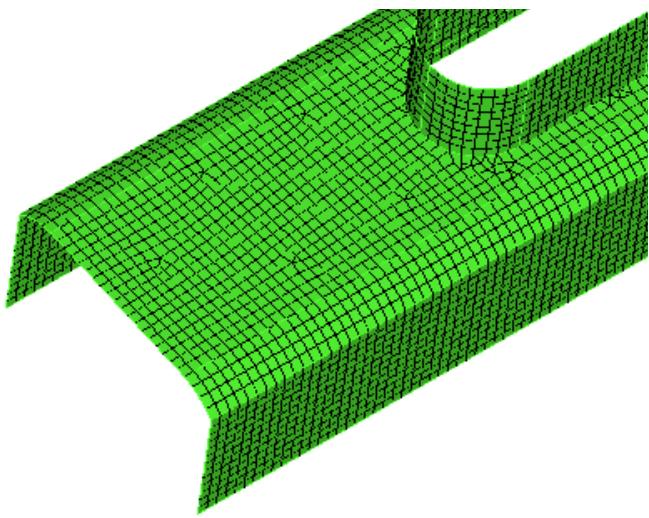
Note: If Surfaces are not selected then it considers all the Surfaces.

Figure 6-14
Mesh Surface
window



Turn off Geometry branch in the Model Display Tree
In the Model Tree, expand the Mesh branch of the tree by clicking on the +. Click right mouse button on **Shells** and select **Solid and Wire**, the mesh appears as shown here.

Figure 6-15
Mesh in Solid
and Wire Frame
mode

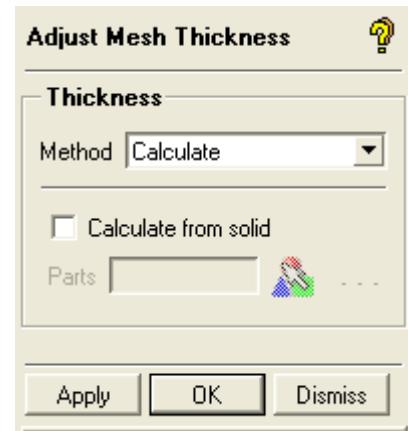


Assigning Shell Thickness

For analysis purpose it is better to assign the thickness to the shell.

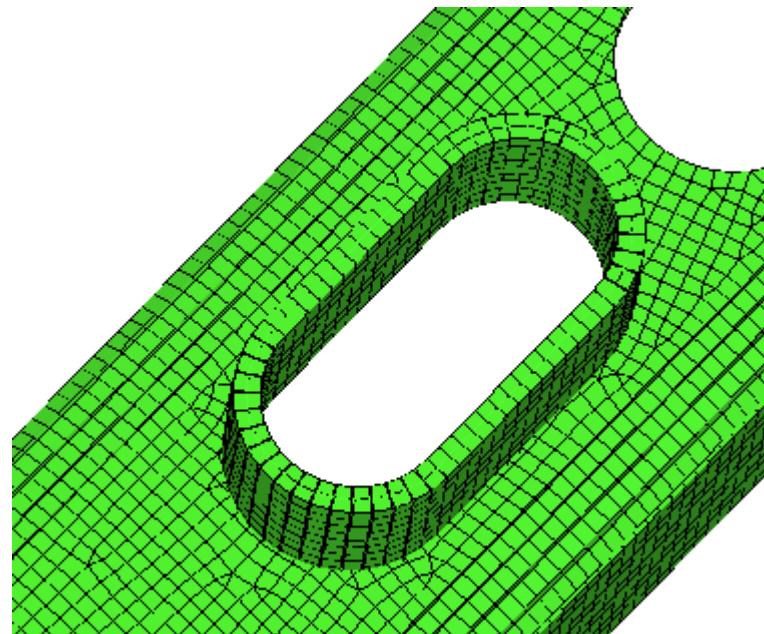
Select  (Assign Mesh Thickness) icon from **Edit Mesh Menubar**, which pops up the **Adjust Mesh Thickness** window. From **Method**, select Calculate and press the Apply. It will automatically calculate the original thickness of the geometry and assigns it to the mesh

Figure 6-16
Adjust Mesh Thickness Window



Note: If the users wants to see the assigned mesh thickness Click right mouse button on Mesh > Shell and select **Shell Thickness**.

Figure 6-17
Geometry
Showing Mesh
Thickness



Now to turn off Mesh>Shell>Shell Thickness from the Model Display Tree.

e) Save the Project

File>Save Project as>TPipe.prj

Note:-

Tpipe.prj is the input data for the Ansys and Nastran Tutorial.

6.1.2: Bar

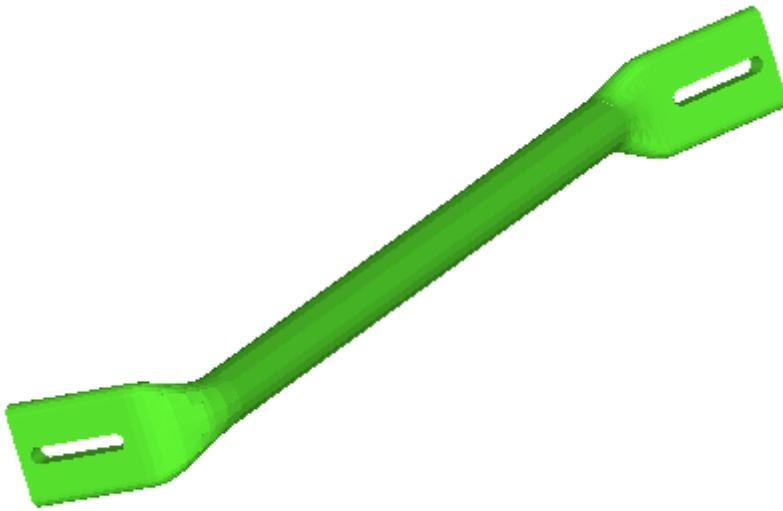
This exercise explains Tetrahedral meshing of the bar geometry displayed below.

Figure

6-18

Bar

Geometry



a) Summary of Steps

Launch AI*Environment and load geometry file

Geometry Editing

Repair

Mesh Parameters and Meshing

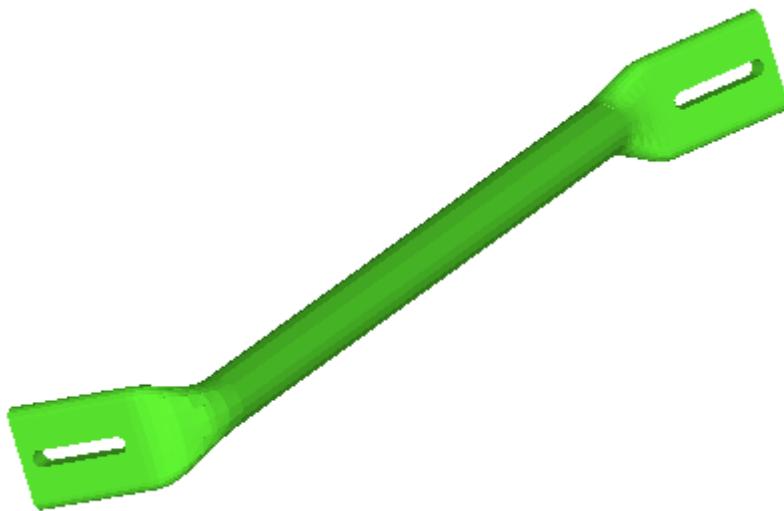
Mesh Sizing

Meshing

b) Launch AI*Environment

The input files for this tutorial can be found in the Ansys Installation directory, under/docu/Tutorials/AI_Tutorial_Files. Copy the files to your working directory and load the tetin file ‘Bar.tin’.

**Figure
6-19
The Bar
geometry**



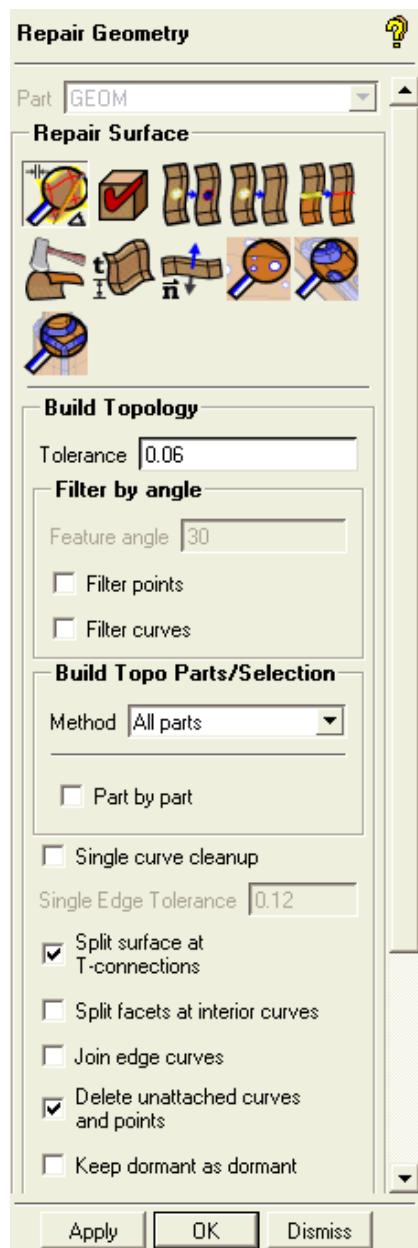
c) Geometry Editing

Repair

Expand the Geometry branch of Model Tree and turn Surfaces on.
Settings>Geometry Options>**Toggle On** the Inherited

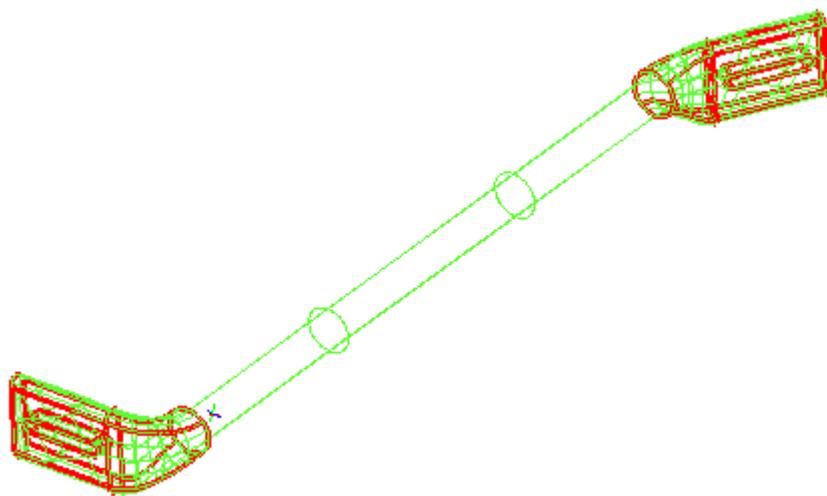
Click on  (Repair Geometry) icon from Geometry Menu bar, which  will pop up **Repair Geometry** window. By default Build Topology is highlighted and Press Apply to extract curves and points.

Figure 6-20
Repair
Geometry
window



The geometry will be displayed in the Main Display window as shown here.

**Figure
6-21
Geometry
after
Build
Topology**



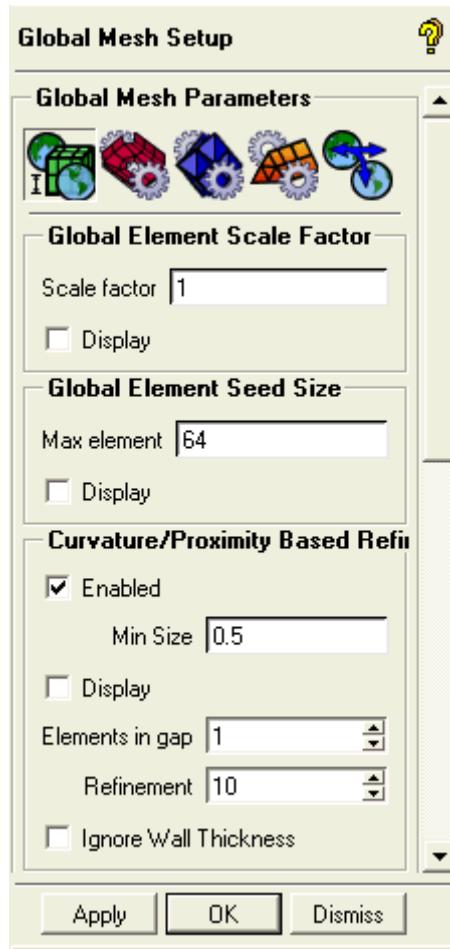
d) Mesh Parameters and Meshing

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 Menubar** and toggle **ON** Enabled 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 and press **Apply**.

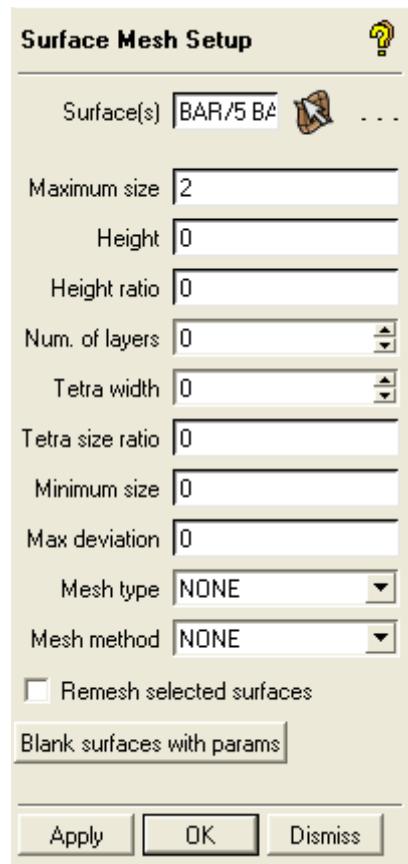
Figure 6-22
Global
MeshSize
window



Select (Set Surface Mesh Size) icon from Mesh Menubar, which will pop **Surface Mesh Size** window.

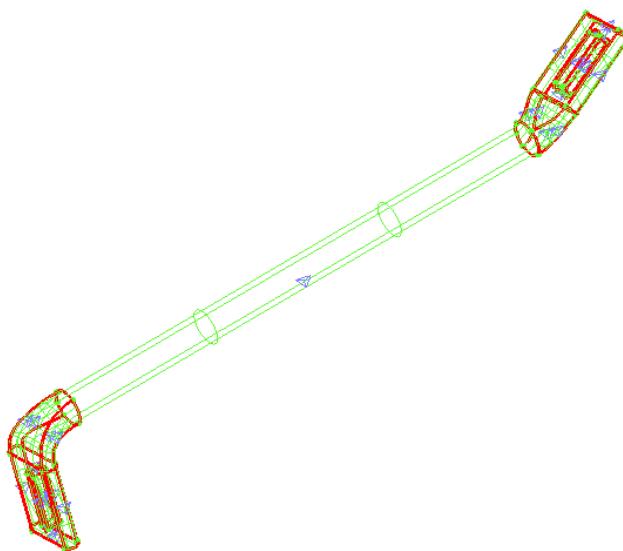
Click on (Choose an item) and select all the surfaces by pressing “a” (ensure that the mouse cursor is in display window). Enter **Maximum Size** of **2** as shown below and press **Apply**.

**Figure 6-23
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 and then turn 'off', the **Surface Tetra Sizes** by deselecting Surface > Tetra Sizes in Model Tree.

**Figure
6-24
Surfaces
with Tetra
Sizes ON**



Meshing

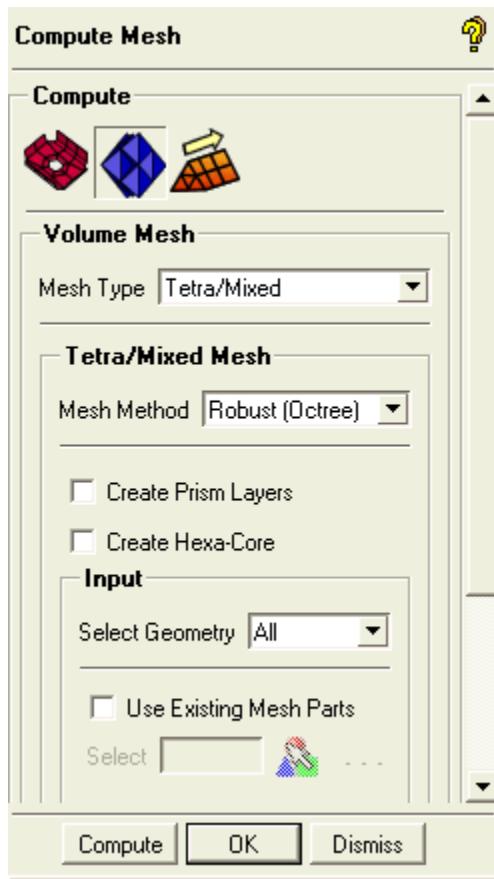
Select Mesh > Compute Mesh > (Volume Mesh) icon from Mesh Menubar.

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 below.

Enable Run as Batch Process and Load mesh after completion.

Figure 6-25
Mesh with
Tetrahedral
window

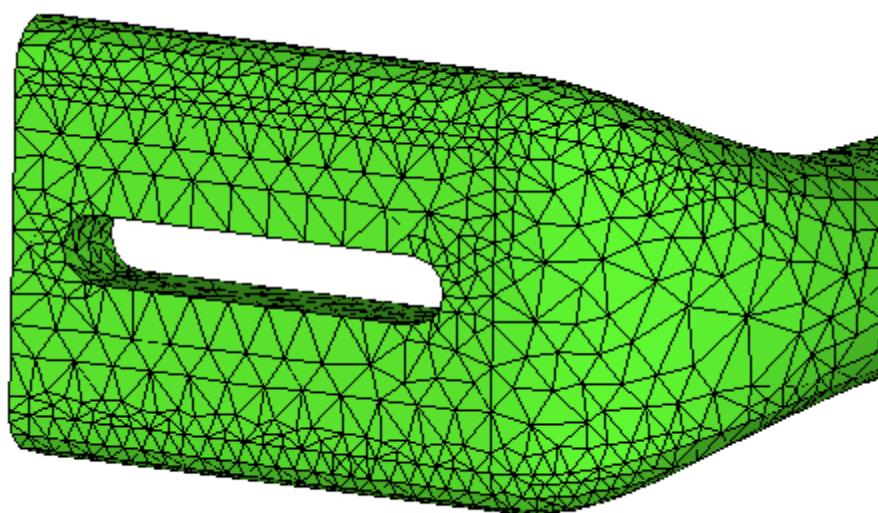


Press **Compute** to start meshing.

Switch off Geometry from the Model Tree.

In Display Tree, click on + to expand the Mesh menu. Click right mouse button on **Shells** and select **Solid and Wire**. Now, the mesh should look like the figure below.

**Figure
6-26
Mesh
in
Solid
and
Wire
mode**



e) Save Project

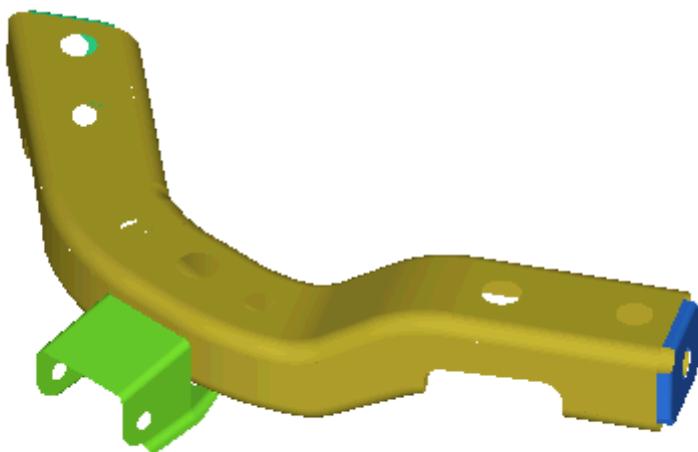
File > Save Project as > Bar.prj.

Note:Bar.prj is the input data for the Nastran Tutorial.

6.1.3: Frame

This exercise explains meshing of Frame geometry including the Seam and Spot welds.

**Figure
6-27
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

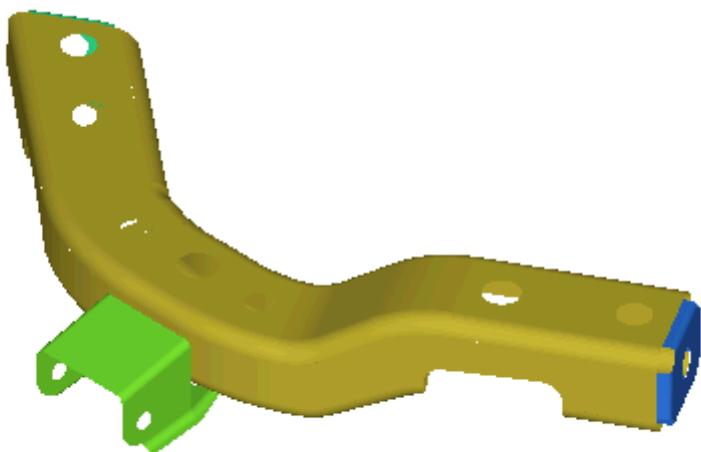
Checking mesh quality

Improving mesh quality

b) Launch AI*Environment

The input files for this tutorial can be found in the Ansys Installation directory, under/docu/Tutorials/AI_Tutorial_Files. Copy the tetin file Frame.tin to your working directory and open it.

**Figure
6-28
Frame
Geometry**



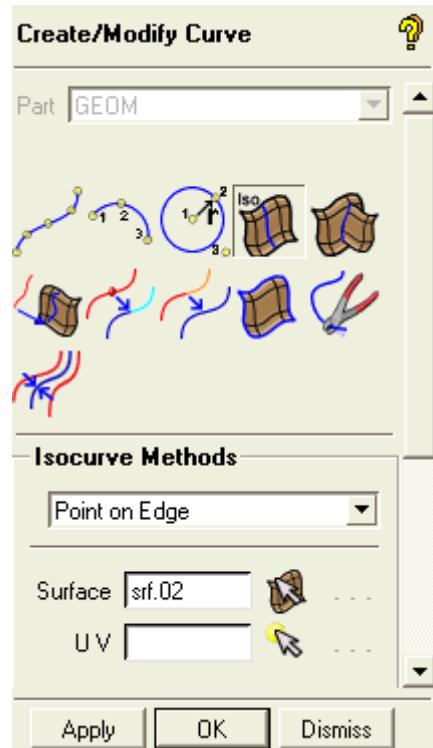
c) Geometry Editing

Create ISO curve

Use Create/Modify Curve ->Surface Parameters >Point on Edge as shown in the below Figure

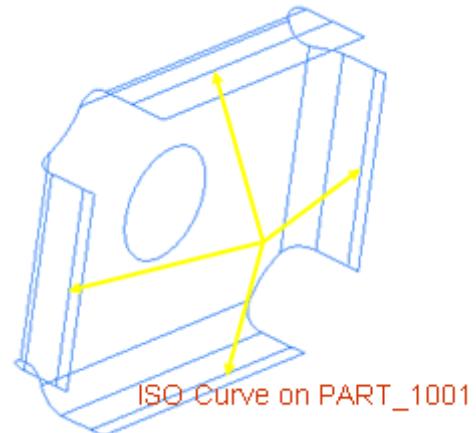
Toggle Off all parts in the Tree and Toggle on PART_1001 only and Toggle On Surfaces and Press Fit window.

Curve/Modify Curve



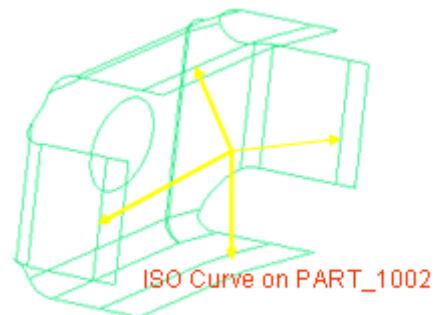
Create ISO curve as like below Figure

ISO Curve on PART_1001



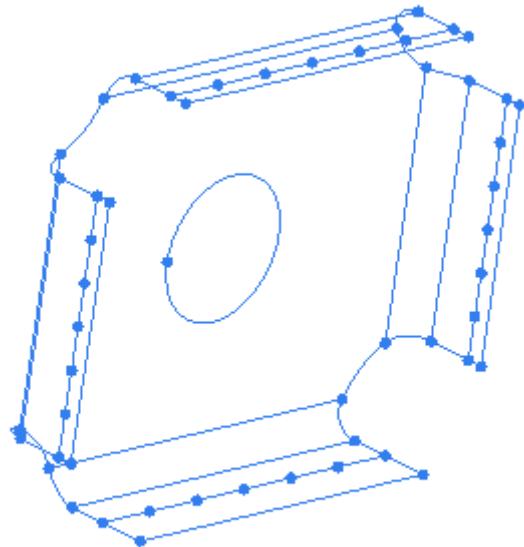
Toggle Off PART_1001 and Toggle On the PART_1002, Press Fit window Create ISO curve to the PART_1002 as like you create in the PART_1001 ,the figure is shown below

ISO Curve on PART_1002

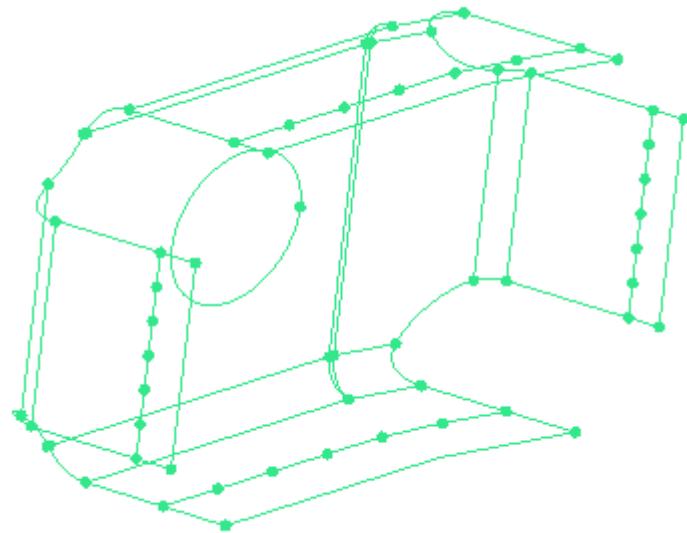


Create Points on those iso curves

Create Point  > Parameter along a Curve  , Select N point option in Method Point and Enter 7 points for each curveToggle On PART_1001 only and select ISO curve
Toggle ON points in the Tree



Create Points for the PART_1002 as Step you follow to Create Points on PART_1001



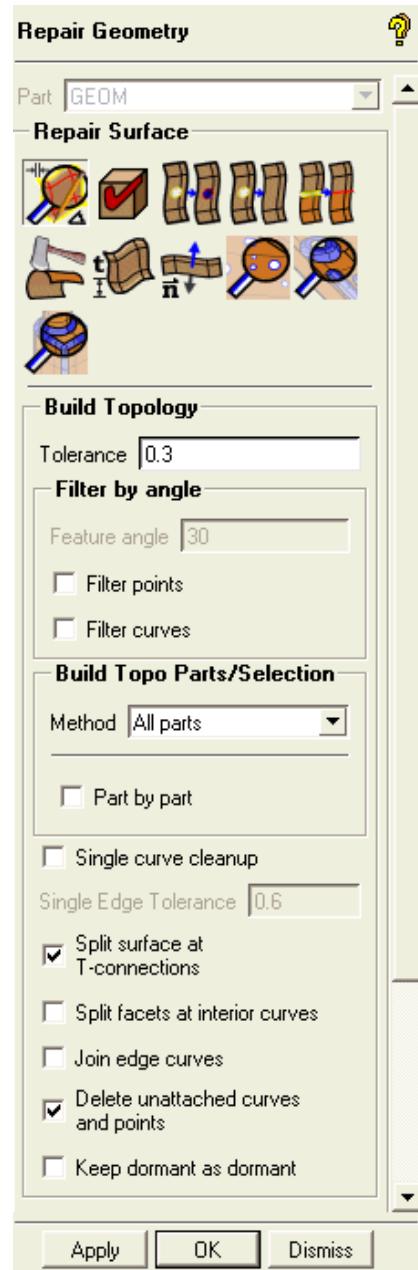
Toggle ON all Parts

Repair

Settings>Geometry Options>Toggle On the Inherited

Click on  (Repair Geometry) icon from **Geometry Menubar**, which pops up **Repair Geometry** window, by default Build Topology option

 is highlighted. Now, make sure that **Tolerance** is set to 0.3, as shown in Figure and press Apply.

**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 separate parts, it is desirable 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 Menu bar**. Click on  (Seam Weld) icon from **Define Connectors** window.

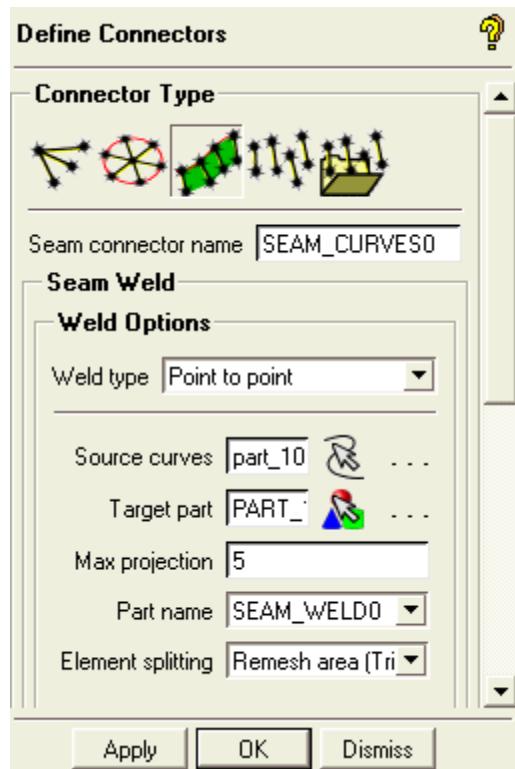
Turn the Surface display ‘Off’ from Display Model Tree by clicking on Geometry > Surfaces and also turn ‘Off’ Points from Model Tree.

Turn ‘Off’ Curves > Show Wide and Color by Count, and in Parts turn ‘On’ only PART_1003.

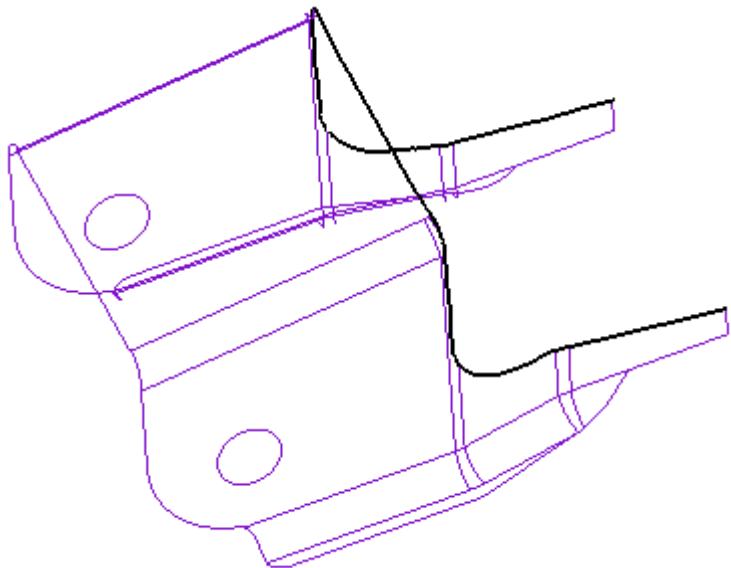
Toggle ON the PART_1003 only 

Click on (Select Curve(s)  button, and then select the curves as in the figure below as Source Curves and notice that the New Part Name for Curves comes default as **SEAM_CURVES0**.

Figure 6-29
Define
Connectors
window for
Seam Weld



**Figure
6-30
urves
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 PART_1001 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

Select (Define Connectors) icon from **Mesh Menu bar**. Now click on (Spot Weld) icon from **Define Connectors** window.

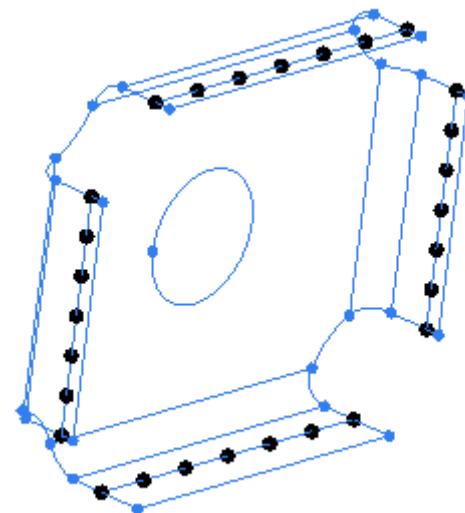
Turn ‘OFF’ all Parts except PART_1001 in the Model Tree.

Note: Make sure that Points and Curves are switched ‘On’ in the Model Tree.

Spot Weld Name: SPOT_POINTS0 (It will appear by default)

Source Points: Select the 28 points created as shown below.

Target Parts: Select the parts PART_1001 and PART_1004



Connector Part Name: SPOT_WELD0 (It will appear by default)

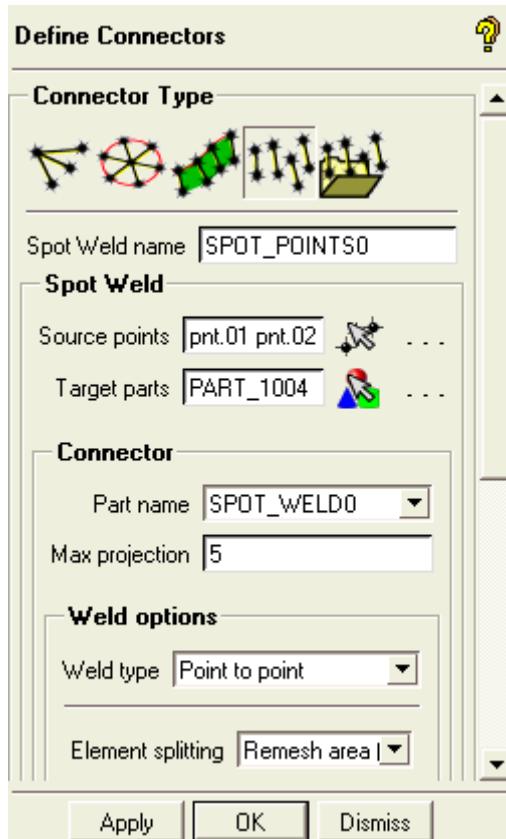
Max Projection: 5

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 here.

**Define
Connectors
window**

The target weld parts here would be **PART_1002** and **PART_1004** and the Points selected are shown

Turn '**OFF**' all Parts except PART_1002 in the Model Tree.

Spot Weld Name: SPOT_POINTS1 (It will appear by default)

Source Points: Select all the 28 points created as shown above.

Target Parts: Select PART_1002 and PART_1004

Connector Part Name: SPOT_WELD1 (It will appear by default)

Max Projection: 5

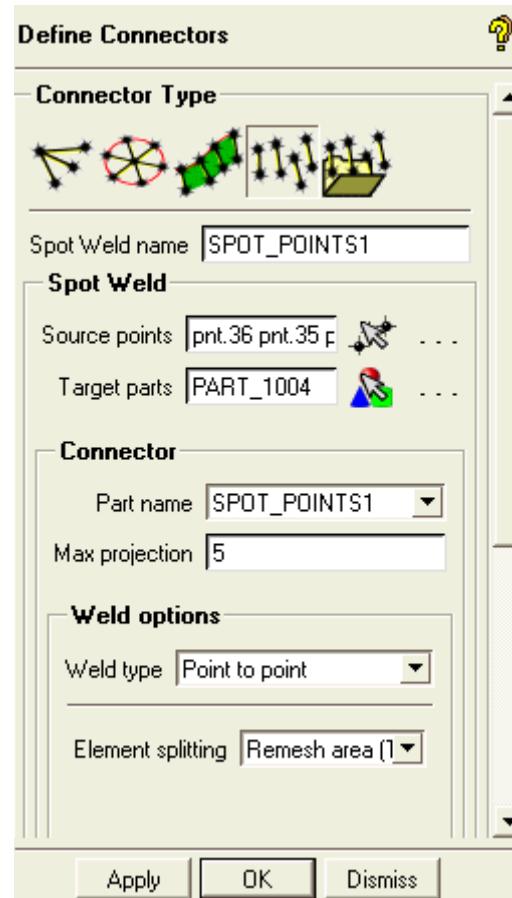
Weld Type: Point to Point (It will appear by default)

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 below.

**Define Connector Window for
Spot Weld Part 1002 and
Part1004**

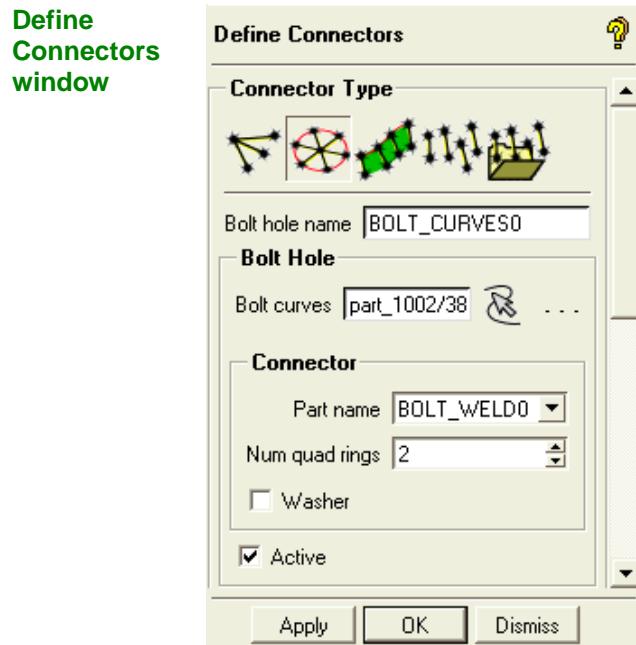


Switch ‘Off’ Connectors in the Display Tree.

Create Bolt Connectors

Now from the Display Tree turn **OFF** Points and turn **ON** all parts. The two flanges (**PART_1001** and **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 here.



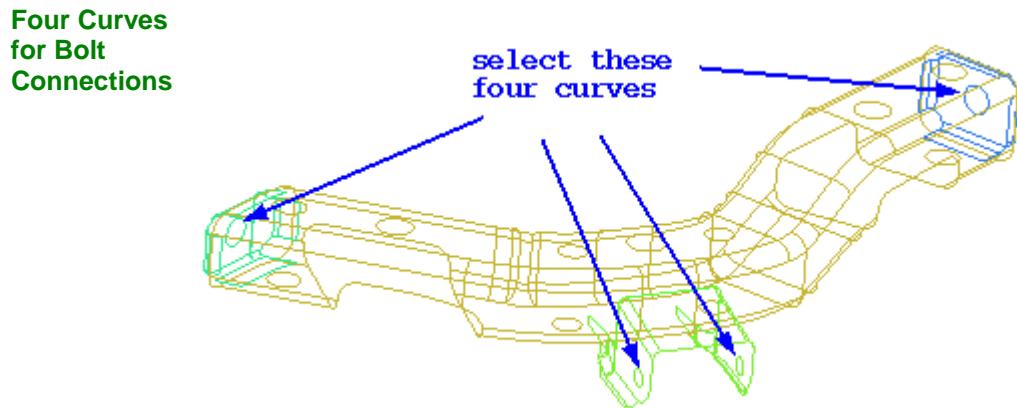
Enter the following parameters in this window. Click on  button and select each of four curves, as shown below and press **Apply**.

New Part Name for Curve: **BOLT_CURVES0** (this comes by default).

Connectors Part Name: **BOLT_WELD0** and

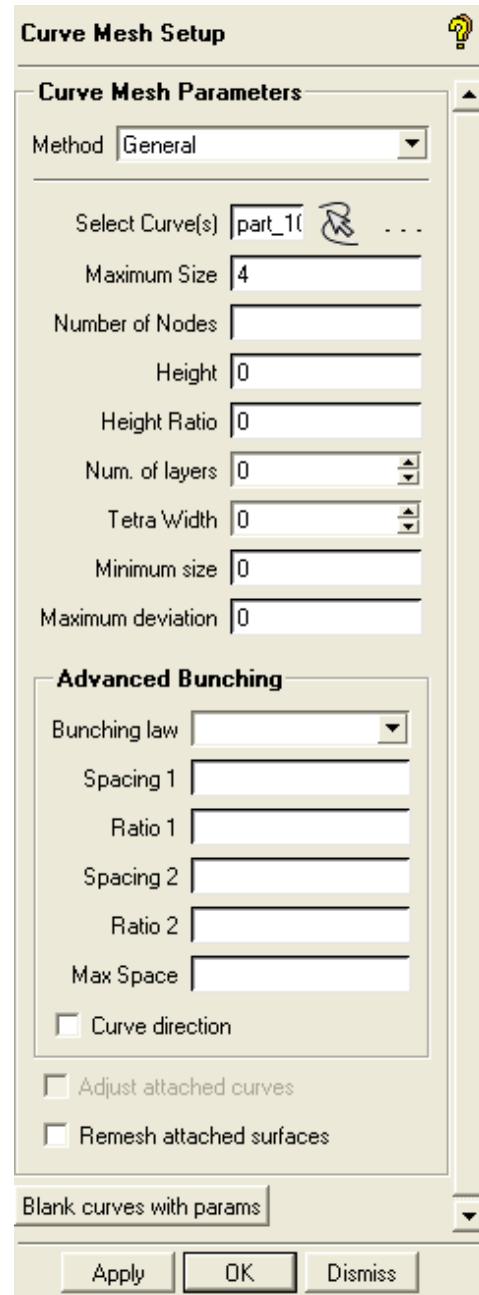
Enter No of Quad Layers as 2 and press **Apply**.

This will create Bolt Hole connector at all the four curves.



Mesh Parameters

Select (Set Curve Mesh Size) icon from **Mesh Menubar**, which pops up **Curve Mesh Size** window. 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.

**Curve
MeshSize
window**

e) Meshing

From the Model Tree turn on display of Surfaces.

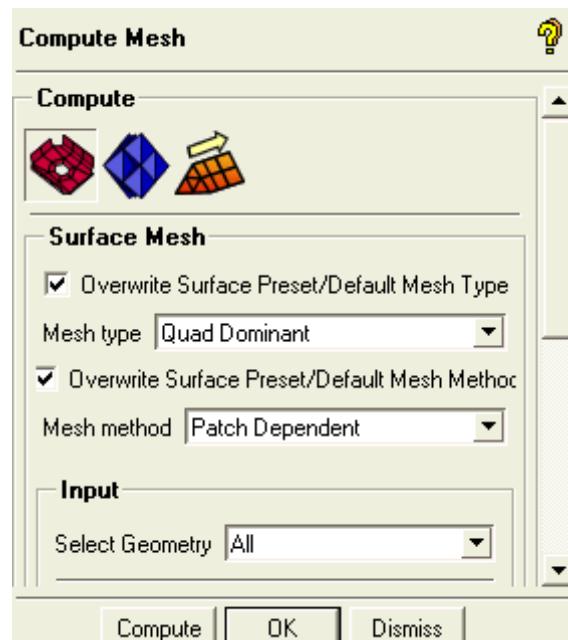
Select  Compute Mesh and then Select the  (Surface Mesh Only) icon.

Select>Mesh type as Quad Dominant

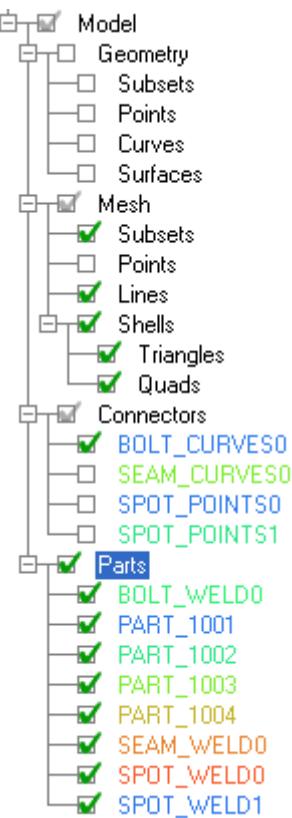
Mash Method>Patch Dependent

Now, press Compute.

Mesh Surface window



The Display Tree Should appear like below Figure

Display Tree

In Display Tree, click on ‘+’ to expand the Mesh options. Click right mouse button on **Shells** and select **Solid and Wire**. Turn ‘Off’ Surface The mesh is shown here

Mesh in
Solid
and
Wire
mode

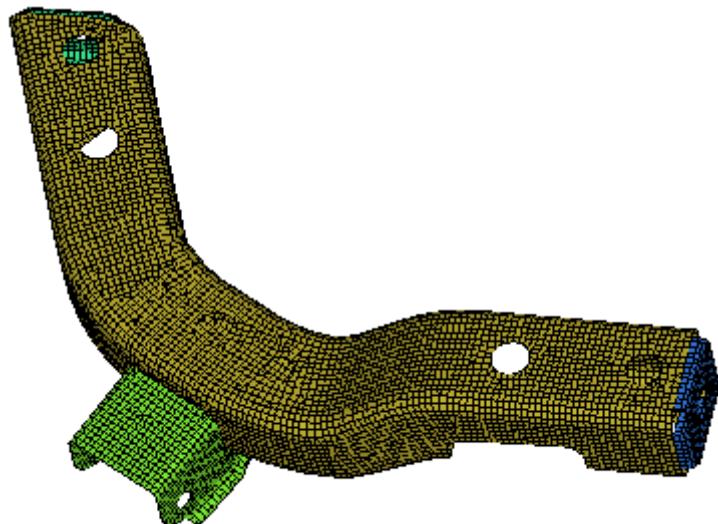
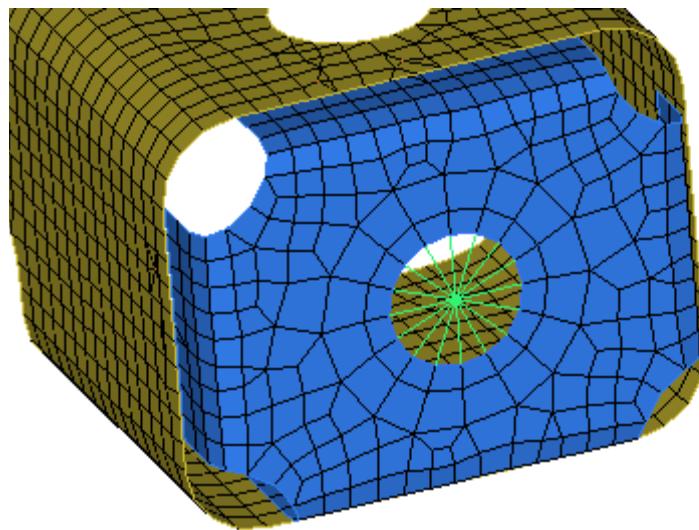
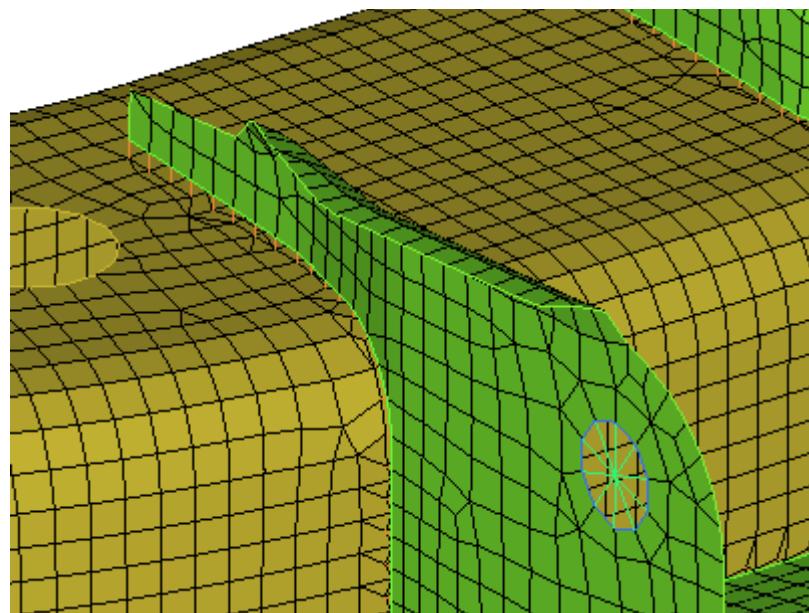


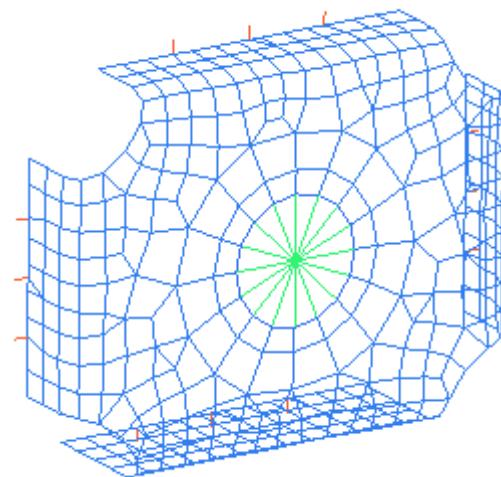
Figure
6-31
Bolt
hole



**Figure
6-32
Seam
Weld**



**Figure 6-33
Spot Weld**



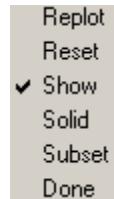
f) Mesh Quality

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, Edit Mesh> select (Smooth mesh globally) icon. 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. There are six options when we right click in the Histogram Bar as shown here.

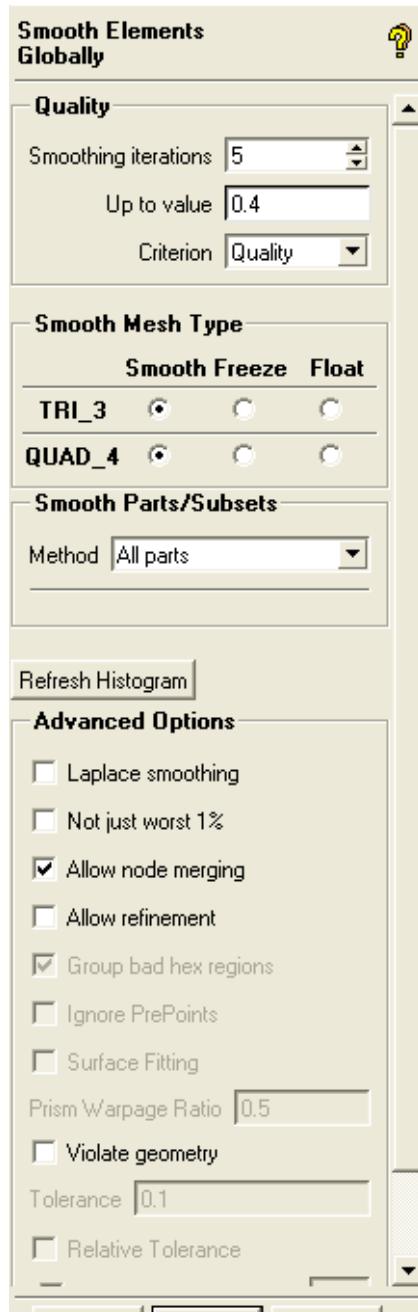
Figure 6-34
Histogram Option



To differentiate the display of element quality, click 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, 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.

Smooth Elements window

Now if there is no elements below 0.2 Quality 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) Save Project

File>Save Project as>Frame.prj

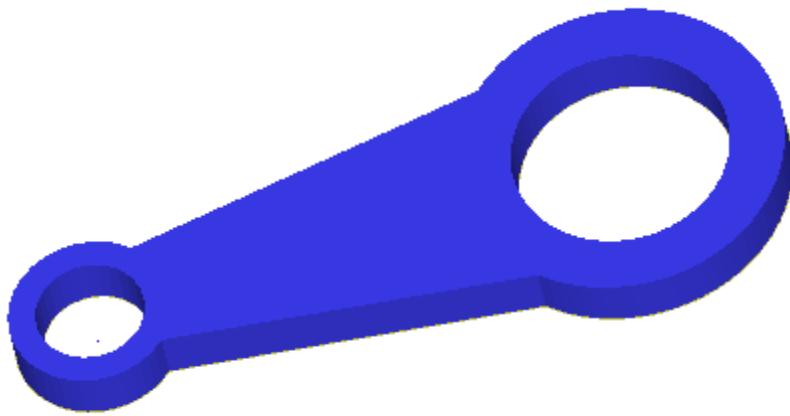
Note:-

Frame.prj is the input data for the Nastran Tutorial.

6.1.4: Connecting Rod

This exercise explains Hexahedral meshing of Connecting Rod geometry by extruding the shell elements.

**Connecting
Rod Model**



a) Summary of Steps

Launch AI*Environment and load geometry file

Geometry Editing

Repair

Mesh parameters and Meshing

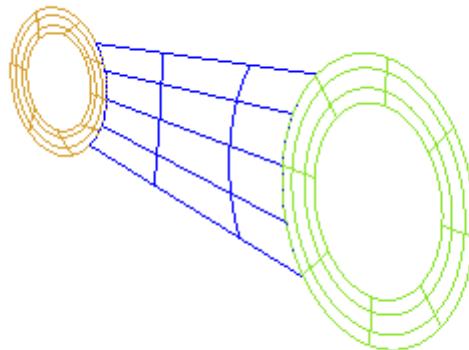
Mesh Sizing

Meshing

Extrusion of the surface mesh

b) Launch AI*Environment

The input files for this tutorial can be found in the Ansys Installation directory, under/docu/Tutorials/AI_Tutorial_Files. Copy and open the tetin file ‘Conrod tin’ in your working directory.

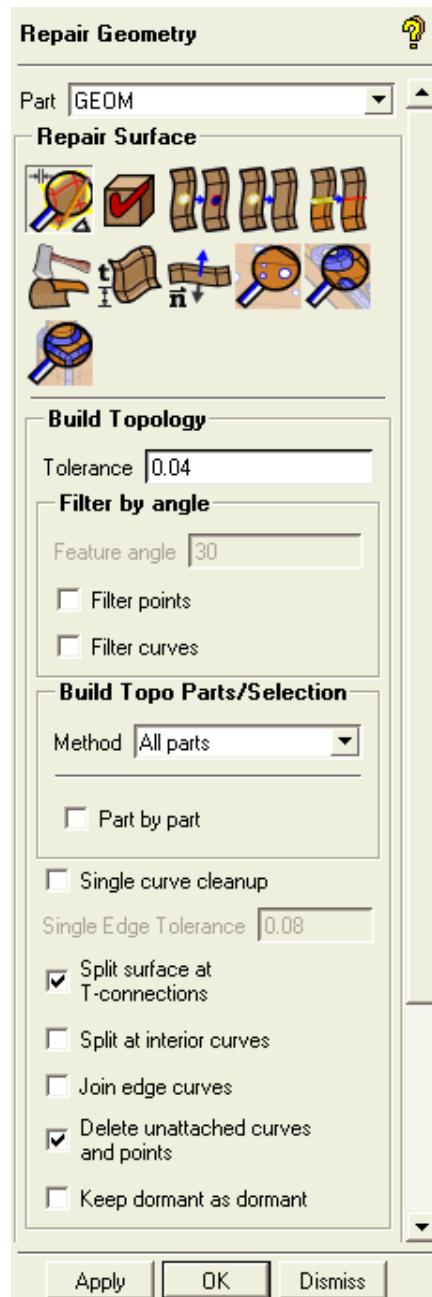
Conrod.tin Model**c) Geometry Editing**

For this tutorial, user is requested to use Conrod.tin file lying in the AI_Tutorial_Files directory as mentioned in Tpipe tutorial.

Repair

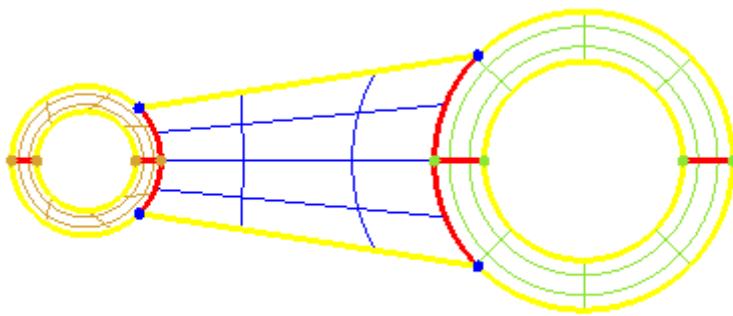
Settings>Geometry Options>Toggle On the Inherited

Click on  (Repair Geometry) icon from Geometry Menubar, which  pops up **Repair Geometry** window. By default **Build Topology**  option is selected.

**Repair
Geometry
window**

Now the geometry appears as shown here.

**Geometry
after
Build
Topology**



d) Mesh Parameters and Meshing

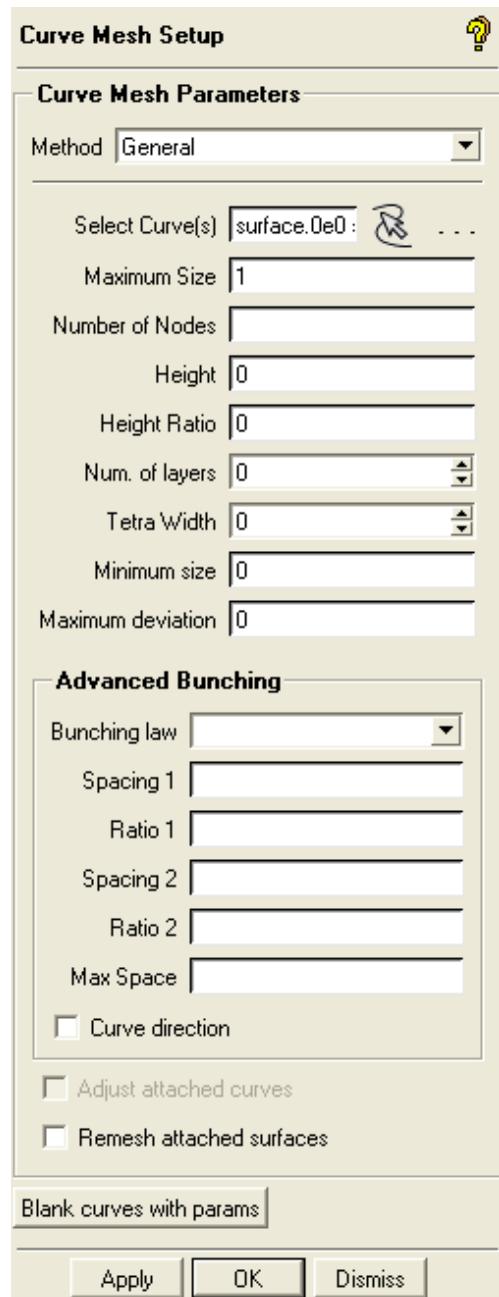
Even though this will be a 3D model, right now it is only 2D geometry, so mesh size should be given on curves. Once the surface mesh is ready then it should be extruded to get the volume mesh. This can be done as follows:

Mesh Sizing

Select (Set Curve Mesh Size) icon from **Mesh Menubar**, which pops up **Curve Mesh Size** window as shown.

Click on (Select 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.

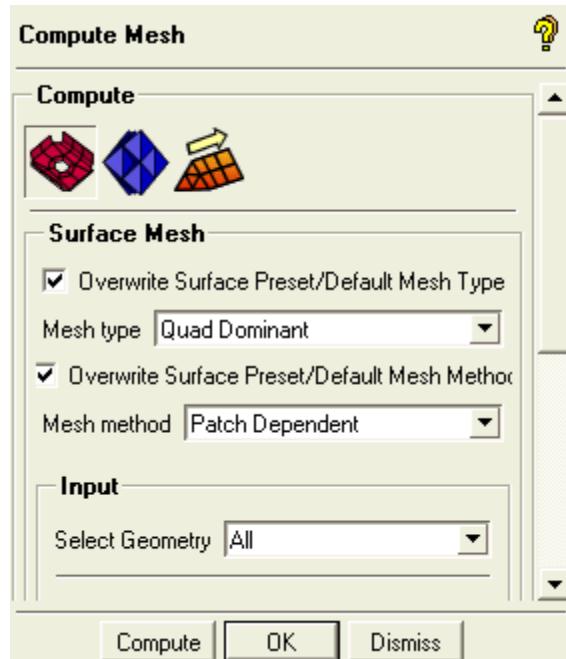
**Figure 6-35
Curve
MeshSize
window**



Meshing

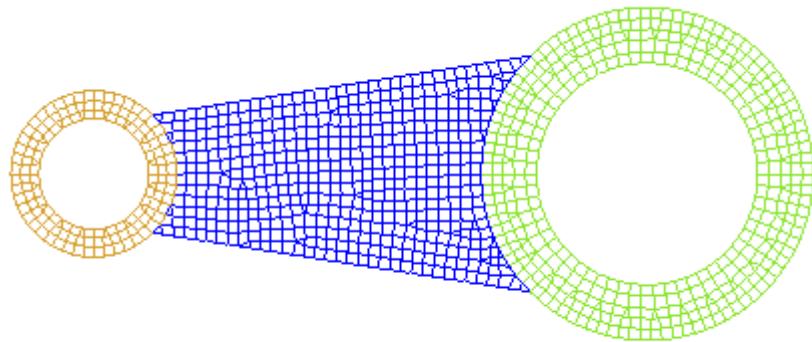
Select Mesh >  Compute Mesh > Select the  (Mesh Shell) icon and Press Compute in the **Compute Mesh** window as shown.

Figure 6-36
Mesh Surface
window



In Display Tree, click on branch of Mesh. Click right mouse button on Mesh > Shells and select **Solid and Wire** and Toggle OFF the Geometry. The mesh looks as shown here.

**Mesh
in
Solid
and
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**'.

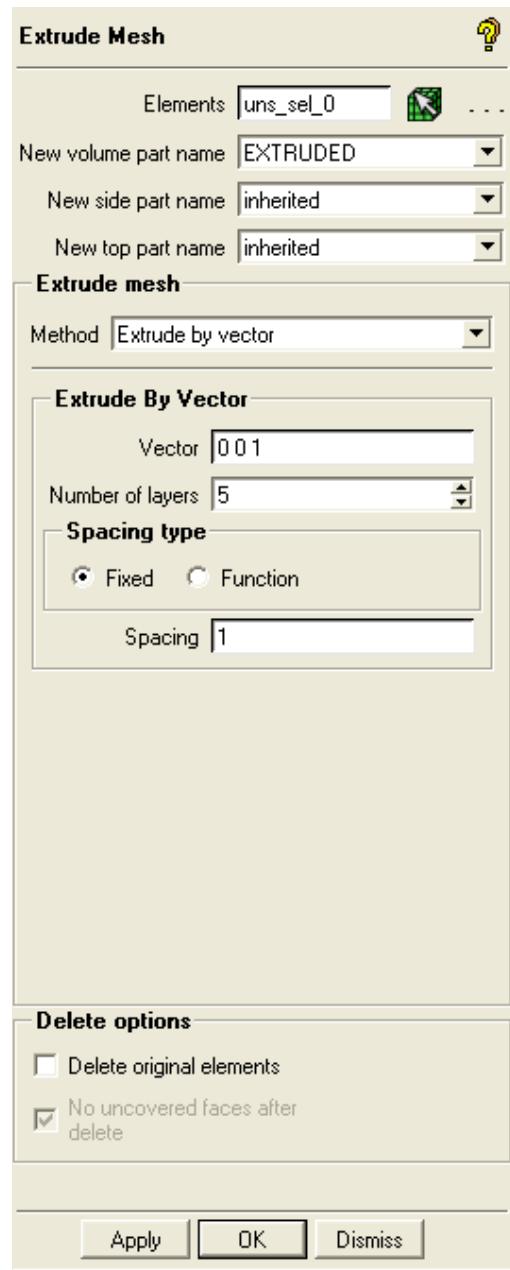


Click on (Extrude Mesh) icon from Edit Mesh which pops up **Extrude Mesh** window.

Note: Before proceeding for extrusion make sure all the line, point under mesh in the Display Tree are turned **Off**.and similarly points and lines under Geometry are Turned **Off**.

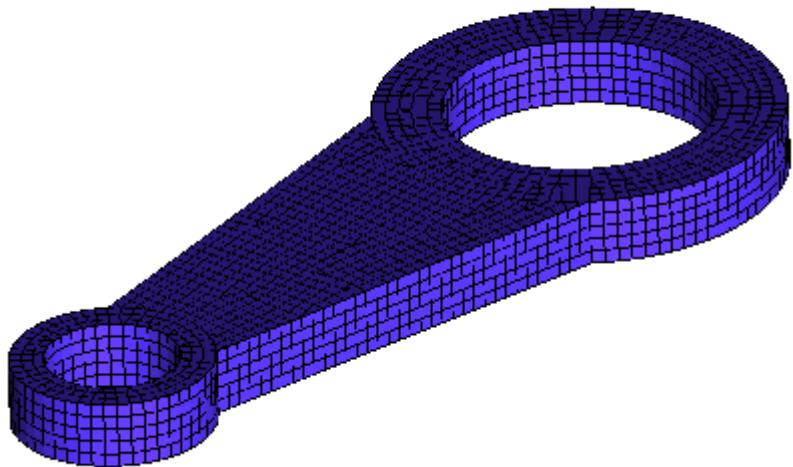
Click (Select Element) and in the **Select Mesh Element** window press Select all Surface Element and to select the entire Surface mesh elements.
 Enter the **New volume Part Name** as EXTRUDED
 Select the **Method of extrusion** as Extrude by vector,
 Enter Vector as **0 0 1**.
 Number of Layers as **5**.
 Spacing as **1** and rest of the option as default.
 Press Apply.

**Figure 6-37
Extrude Mesh
window**



Switch Off - Shell and Geometry under the Display Tree and Switch ‘On’ Volume > Solid and Wire.

**Extruded
mesh**



Switch Off Mesh and then Switch On Mesh.

Note: User can view the default setting of the Mesh in the Display Tree by switching **Off** the Mesh and then switching **On** the Mesh, by default only Shell and Line are switched ‘**On**’.

e) Save Project

File>Save Project as>Connecting_Rod.prj

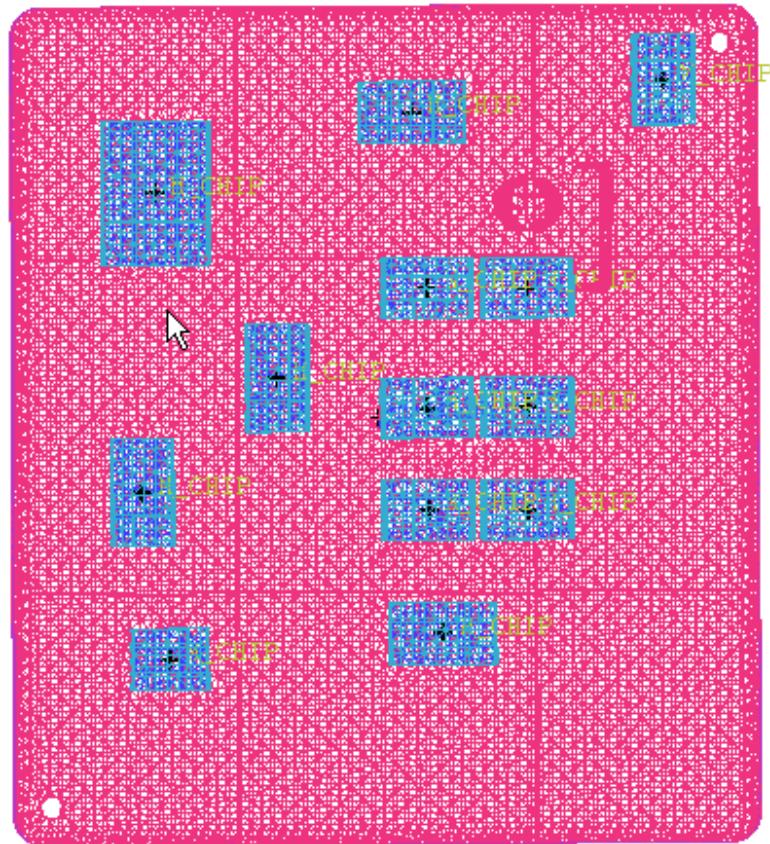
Note:-

Connecting_Rod.prj is the input data for the Ansys and Nastran Tutorial.

6.1.5: PCB-Thermal Analysis

Overview

In this tutorial, it is shown that how easy to create a mesh in the PCB model 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
Saving the project

b) Starting the project

The input files for this tutorial can be found in the Ansys Installation directory, under..../docu/Tutorials/AI_Tutorial_Files > PCB. Copy these files to your working directory and load the tetin file geometry.tin.

c) Repairing the geometry

Settings>Geometry Options>Toggle On the Inherited

For repairing geometry, select Geometry > Repair geometry  > Build

 topology.

Run the build topology with the default parameters

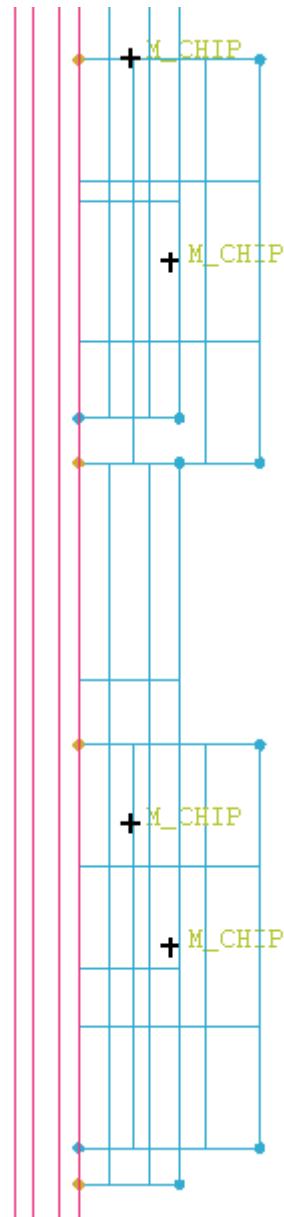
d) Assigning the mesh sizes

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, Enter part as M_CHIP, select by

 topology and method as Entire model. Press Apply.

M_CHIP BODY

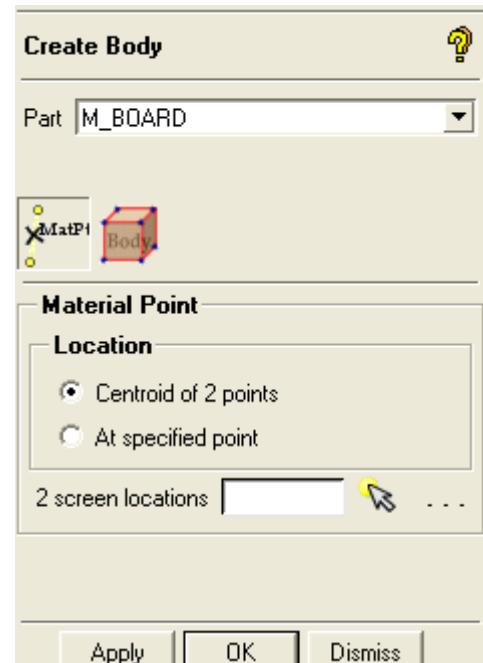


To see the bodies in the geometry on screen, please make bodies visible from the model tree.

In create body window, Enter part as M_BOARD, select method as **Material Point**.

In Tree, Right-click the Point under Geometry and Select Show Point Names

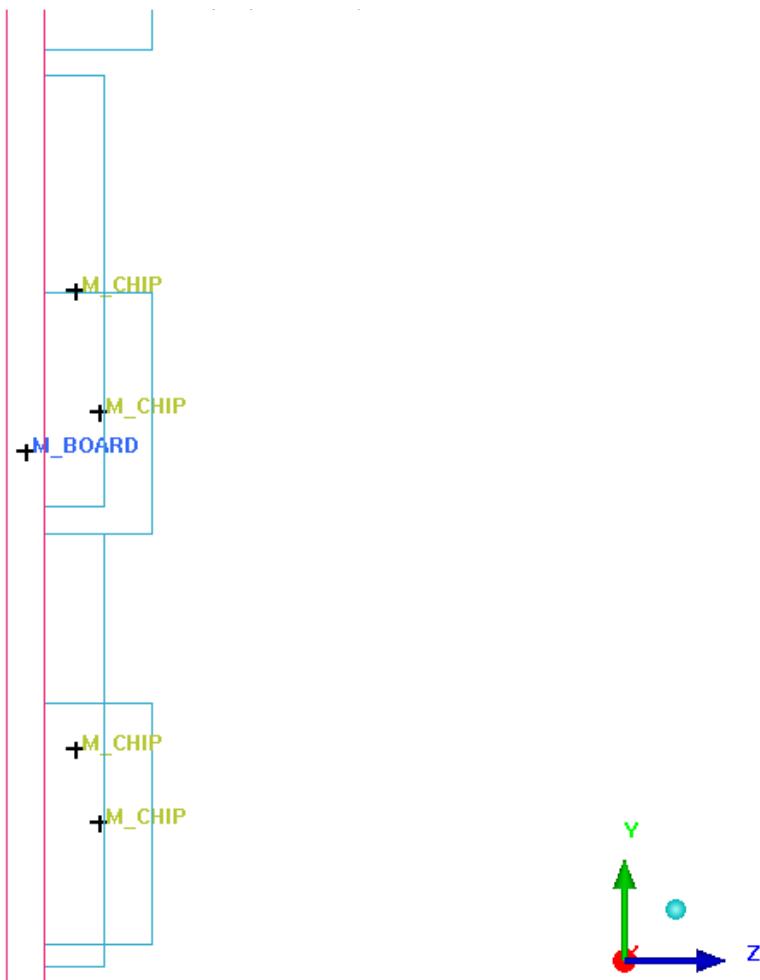
Select Points>Board_SURF.146 & Board_SURF.219



Press Apply.

Then Unselect the Show Point Names

Created M_CHIP & M_BOARD



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 1.5 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 0.75 and select only visible parts by using option ‘v’

After assigning mesh sizes.

Toggle ON all Parts

f) Generating the tetrahedral mesh

To generate the tetrahedral mesh, Select Mesh >  Compute Mesh >Mesh

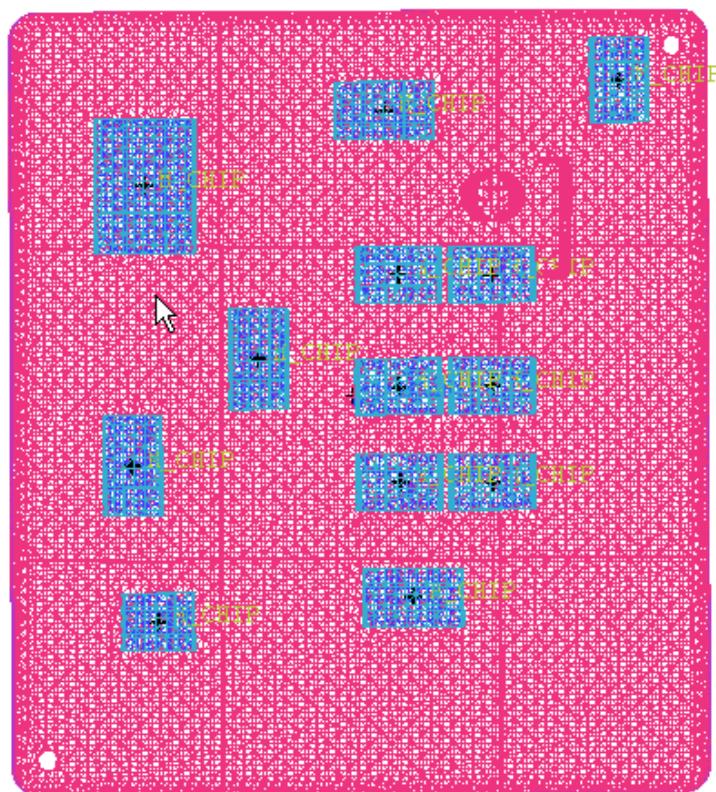
Tet  SelectMesh Type > Tetra/Mixed

Mesh Method > Robust(Octree)

Select Geometry > All

This will generate the tetrahedral mesh on the geometry as shown here.

**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.

h) Save Project

File>Save Project as>PCB.prj

Note:-

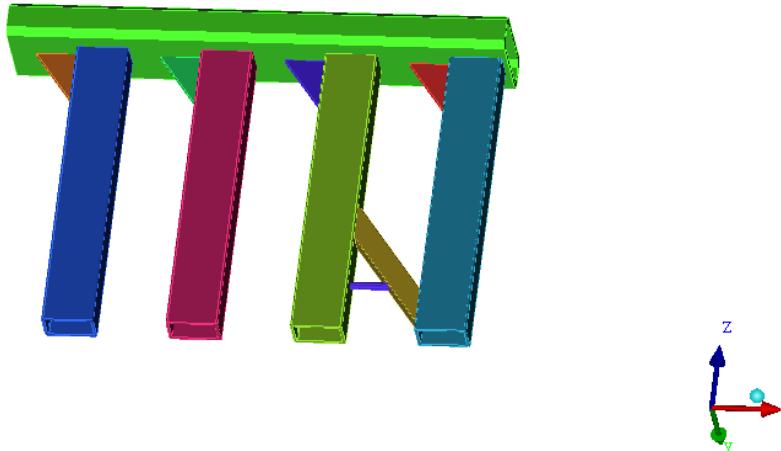
PCB.prj is the input data for the Ansys Tutorial.

6.1.6: Tube Frame

Overview

In this example, the user will generate a Quad Surface mesh on a Tube Frame. The frame represents a part fabricated of steel having a thickness of about 10 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 and then generate shell mesh on that surface.

Figure 6-38:Tube Frame



a) Summary of steps

Starting the project
Running Build Topology
Determining the mid surfaces
Extending Surfaces
Defining Mesh Parameters
Generating Shell Mesh on surfaces
Checking the Mesh Quality
Defining Material Property
Editing the Shell Element thickness
Saving the project

b) Starting the Project

The input files for this tutorial can be found in the Ansys Installation Directory, under ..\v110\docu\Tutorials\AI_Tutorials\TubeFrame project.
Copy the geometry file to your working directory and open it.

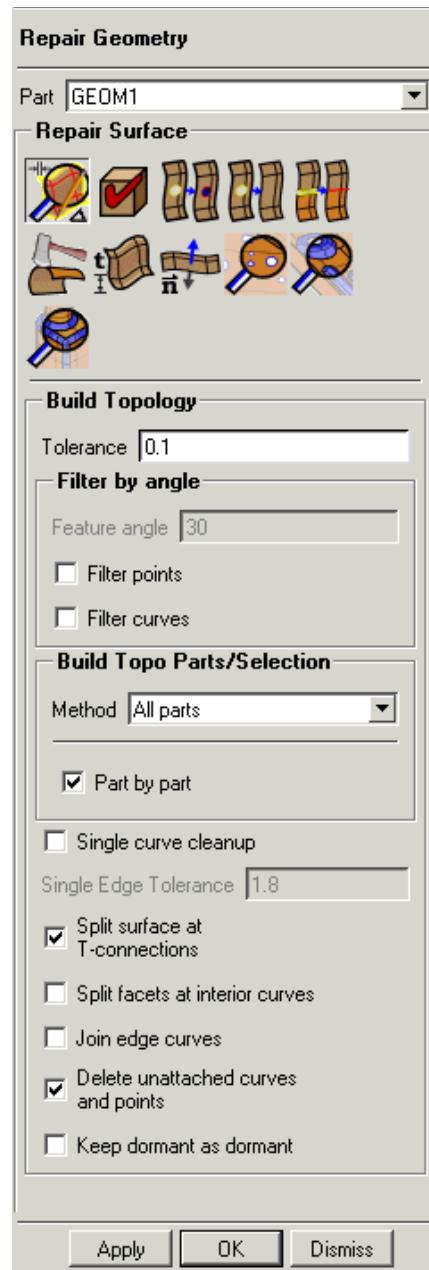
c) **Repairing Geometry- Running Build Topology**

The user can notice that there are no curves or points in the geometry. So build topology should be run with appropriate tolerance to create curves,points on the surface.

Select Geometry > Repair geometry > Build Diagnostics Topology

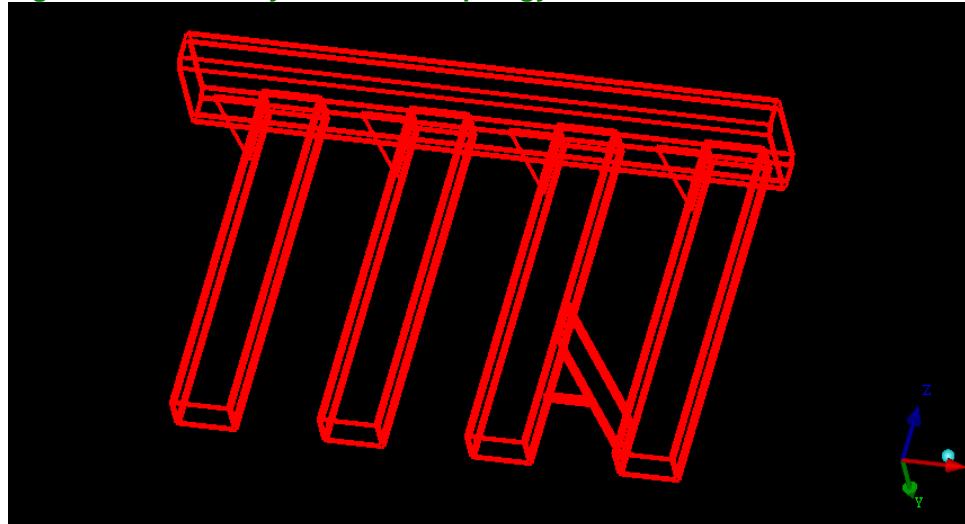


.Assign Tolerance of **0.1** as shown. Toggle ‘**ON**’ Part By Part Option as shown in the Repair Geometry GUI.

Figur.6-39: Repair Geometry

After the Build topology is run, curves and points get created which can be viewed from Display Tree, Model > Geometry> Point, Surfaces. All the curves should be in Red Coloured- there should be no curves in Yellow or Blue coloured. Curves in Red coloured indicates that the curve is shared by 2 surfaces. with two other surfaces .

Figure 6-40:Geometry after Build Topology



D) Determining Mid Surface.

The user should verify the thickness by View > Distance or distance icon



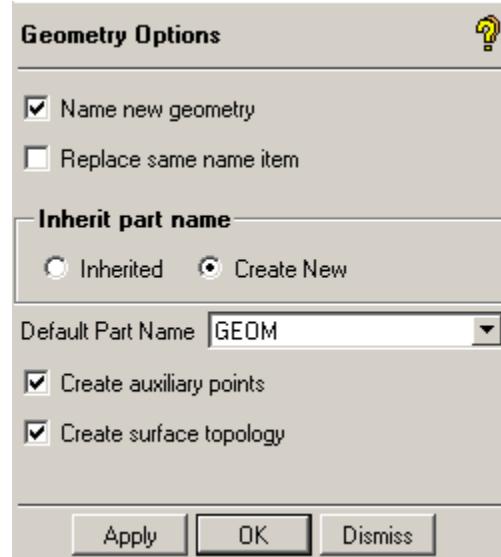
by clicking on two sides spanning the thickness of the geometry. The part will be modeled by the quad shell elements that are computed on the mid surface of the geometry. However, mid surface creation depends on the gap between the two spanning surfaces. The user can safely take a value of **15** to determine the mid surface.

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	947
------------------------	--	-----

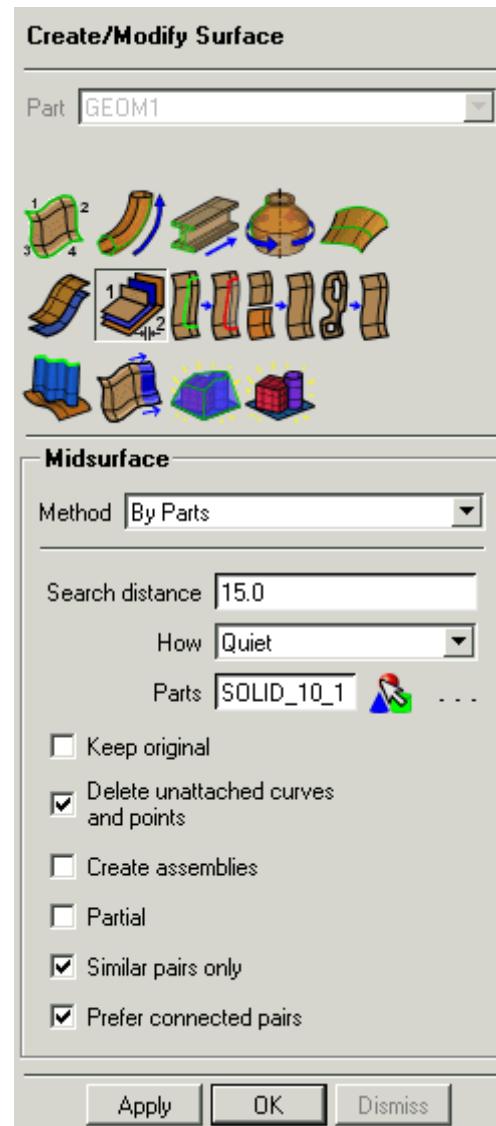
- Display Surfaces from the model tree. The user may type "h" from the key-board to get the home view.
- Select Settings >Geometry Options.

Toggle 'ON' –Name new Geometry and Toggle 'ON' Create New under Inherit Part Name as shown in the figure below.

Figure 6-41 Geometry Options



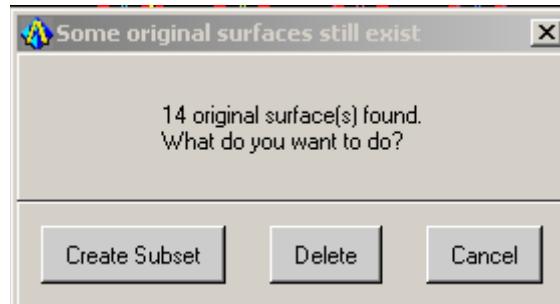
Geometry >Create/Modify Surface . Press Mid Surface. This will open up a window as shown. In the panel, put the **Search distance** as **15** .Keep the Method>By Parts. Next, select **Quiet** in the **How** window, press on Parts picker to select the all parts by hotkey ‘a’.
Toggle ‘ON’ **Similar Pairs Only** and **Prefer Connected Pairs**.

Figure 6-42:Mid Surface

Click **Apply**.

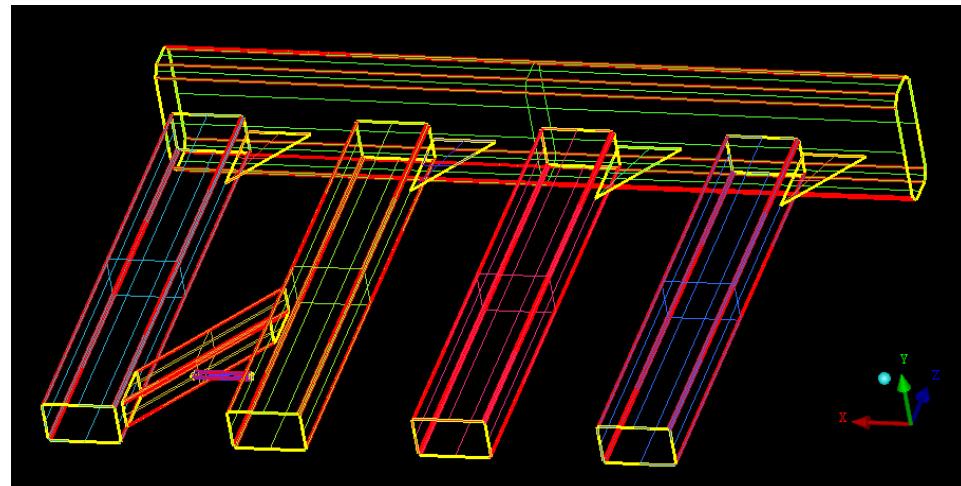
When prompted by the delete window, press **Delete** to delete all the original entities. This operation will also remove the original part name.

Figure 6-43:Delete Original surfaces



The geometry would now look as shown in below. The geometry will look like this after making all the parts visible.

Figure 6-44: Geometry After Mid Surfacing



e) Surface Thickness:

Now make the surfaces visible. From Model Tree, Geometry >Surfaces, Toggle ON the surfaces.

Right Click Surface > Show Surface Thickness .This will open up a GUI as shown below.

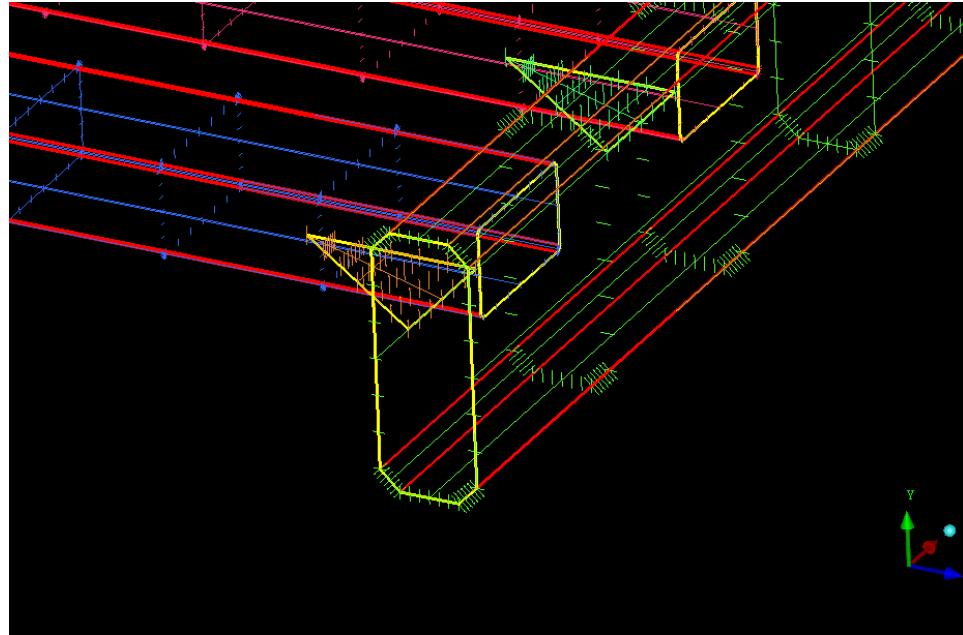
Figure 6-45:Show Surface Thickness



This will show the different thickness assigned on surfaces as shown. The arrows, displays the varied thickness assigned to each surfaces.

Figure 6-46:Thickness Assigned to Surfaces

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	951
------------------------	--	-----

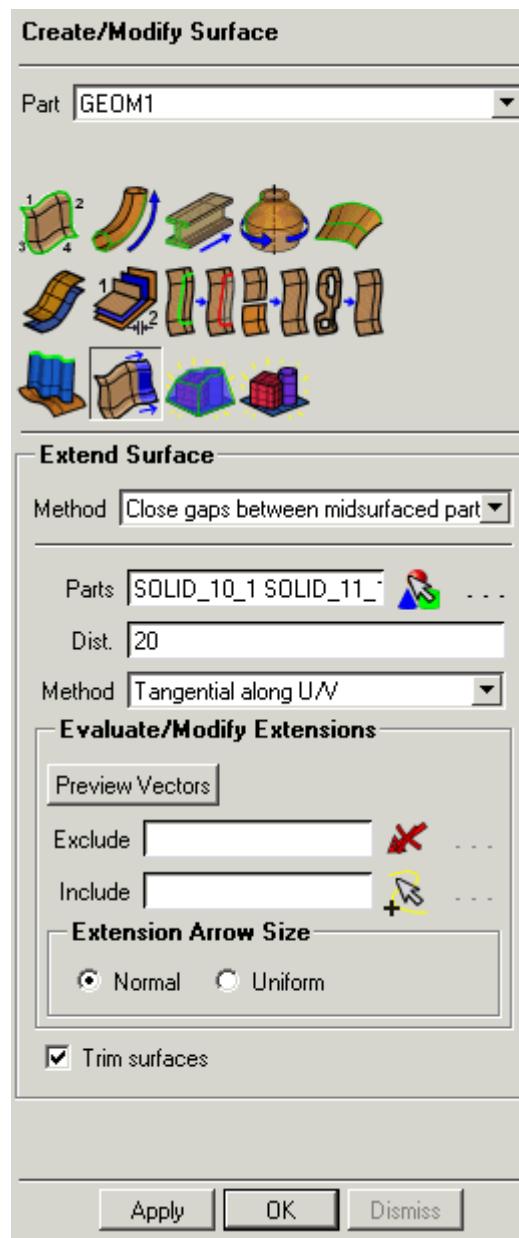


f) Create/Modify Surface >Extend Surface:

After the mid surfaces operation there are gaps created between the different surfaces. Now the user must use the Option Extend Surface to close the gaps. For each part selected, single edges within the specified Distance from another surface will be extended to that surface.

- Under Geometry Geometry >Create/Modify Surface . Press Extend Surface . In the panel, put the **Distance** as **20** . Keep the Extend Surface Method> Close gaps between midsurfaces part. Next, select **Method** as **Tangential along U/V** , press on Parts picker to select the all parts by hotkey ‘a’ . Leave the rest of the default values.

Figure 6-47
:Extend Surface

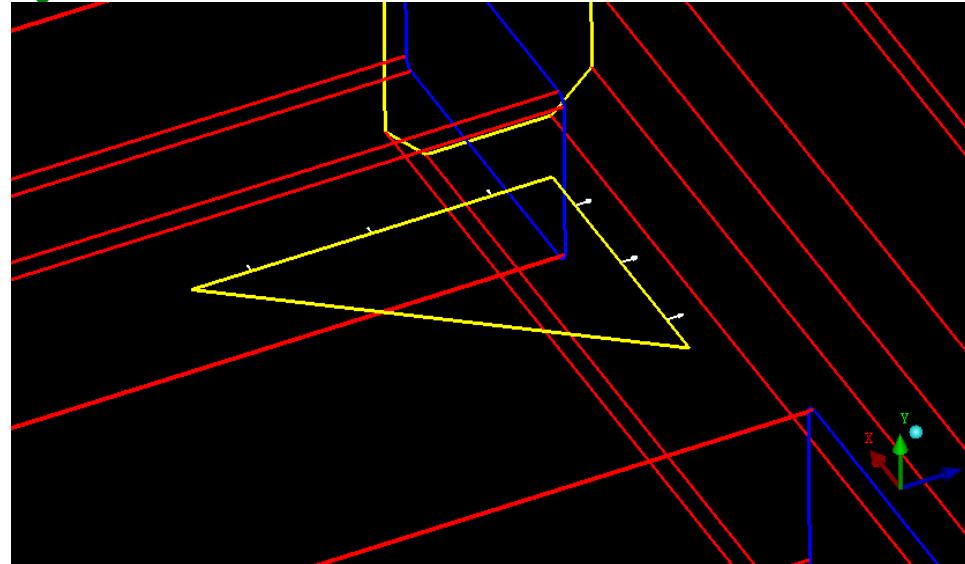


Now Turn **OFF** Surfaces from the Display Tree.

Now press Preview Vector icon in Extend Surface GUI, to see the connections.

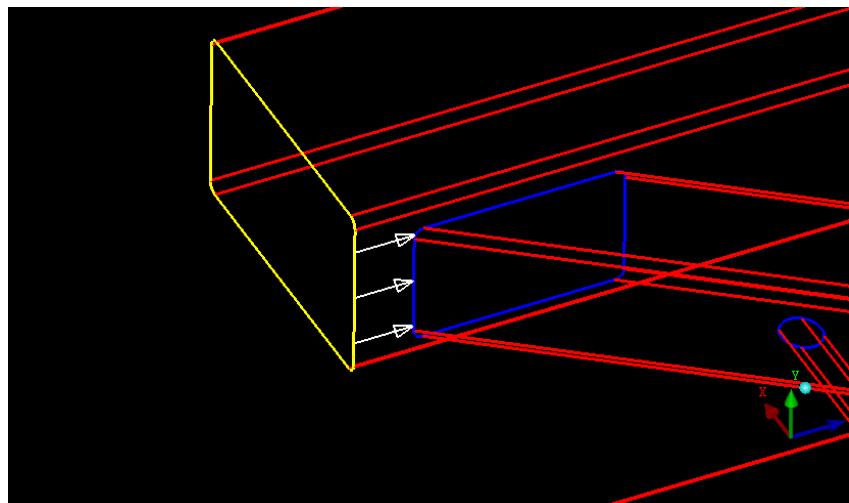
The actual Preview vectors can be seen in the GUI as shown below.

Figure 6-48:Preview Vectors



Click On the '**Exclude**' Icon in the Extend Surface GUI, and click on the large vectors as shown below that are not part of the midsurfaces.

**Figure
6-49:Exclude
Vector**

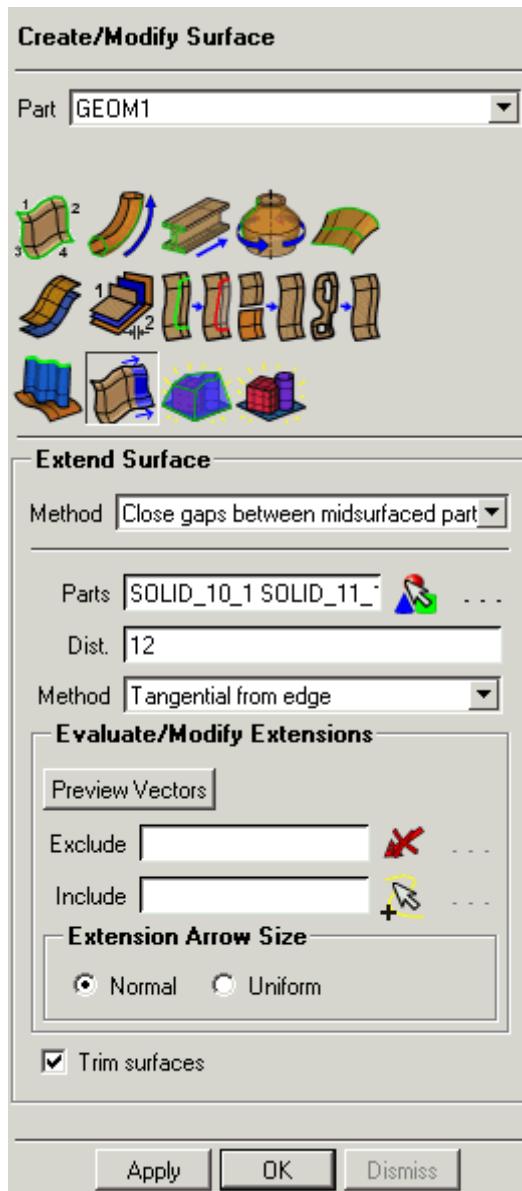


Press **Apply** to Extend the surfaces. It will extend the surfaces, and the extended surfaces will be shown in blue colour indicating they are properly connected.

Create/Modify Surface >Extend Surface(Contd.):

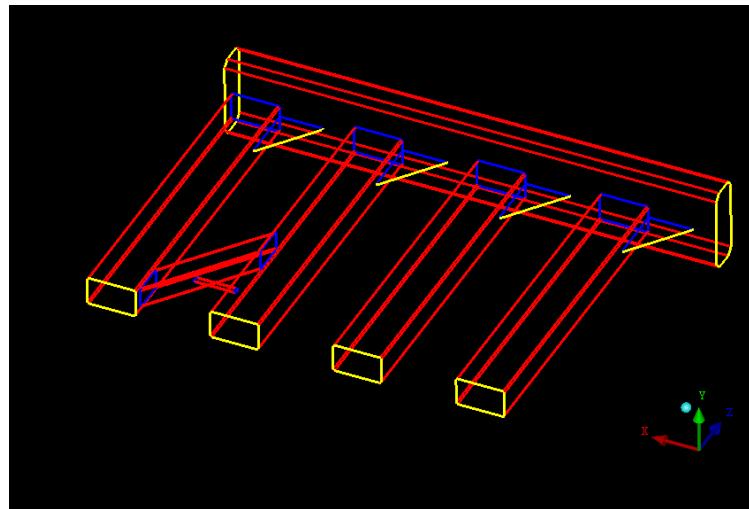
Now in the Extend Surface GUI, change the Distance to 12.0, and change the **Method to Tangential From Edge**. Press Apply to extend the rest of the unconnected edges.

Figure 6-50:Extend Surface(Tangential from Edge)



The geometry should look as shown below.

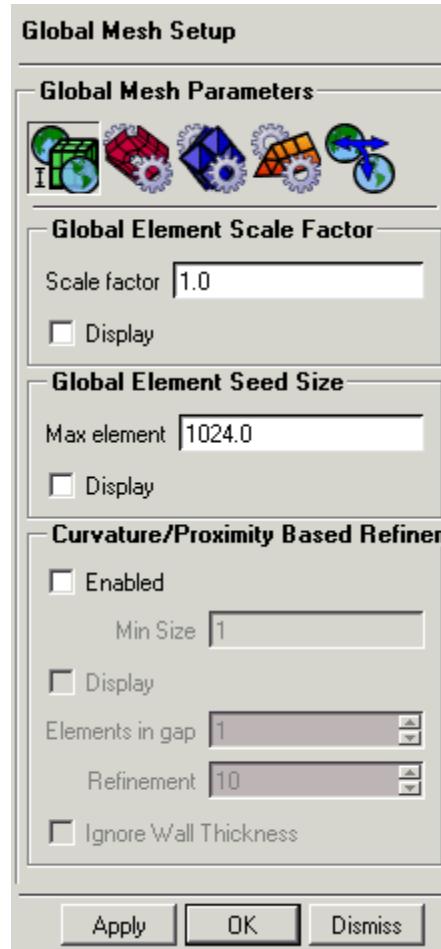
**Figure
6-51:Geometry
after Extend
Surface**



g) Surface Meshing:-

The user must define mesh sizes before mesh generation. Under Mesh

> Set Global Mesh Setup > Global Mesh Size , Set Scale factor to **1** and **Max Element** to **1024.0** as shown. Press Apply followed by Dismiss to close the window.

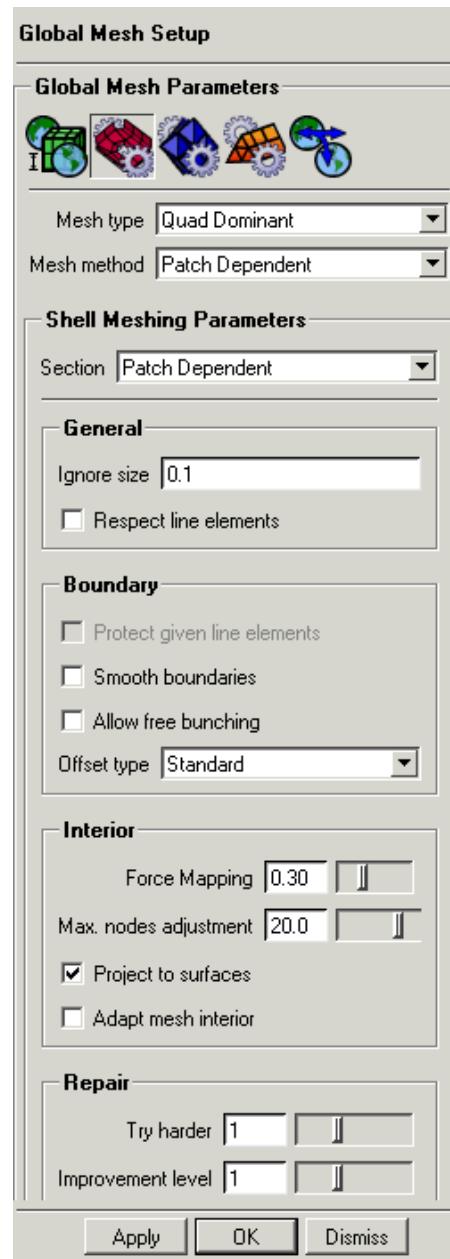
Figure 6-52:Global Mesh Size

- From **Global Mesh Setup** , Select **Shell Meshing**

Parameter  . Now choose **Mesh Method** as **Patch Dependent**, the **Mesh Type** as **Quad Dominant**.

Now, from section General, keep the Ignore Size=0.3.

In the section Interior, keep Force Mapping as 0.30 and Max. Nodes adjustment value as 20.0, leave the rest as default values. Press Apply.

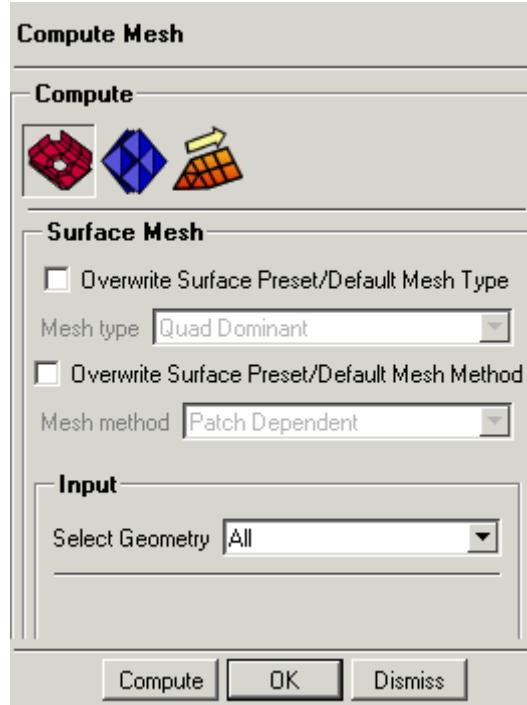
Figure 6-53:Shell Mesh Parameter

Now, go for Mesh > Surface Mesh Setup  > Select all Surfaces(s) by hot key ‘a’. Assign Surface Size of 5.

Now select Mesh > Compute Mesh  > Surface Mesh

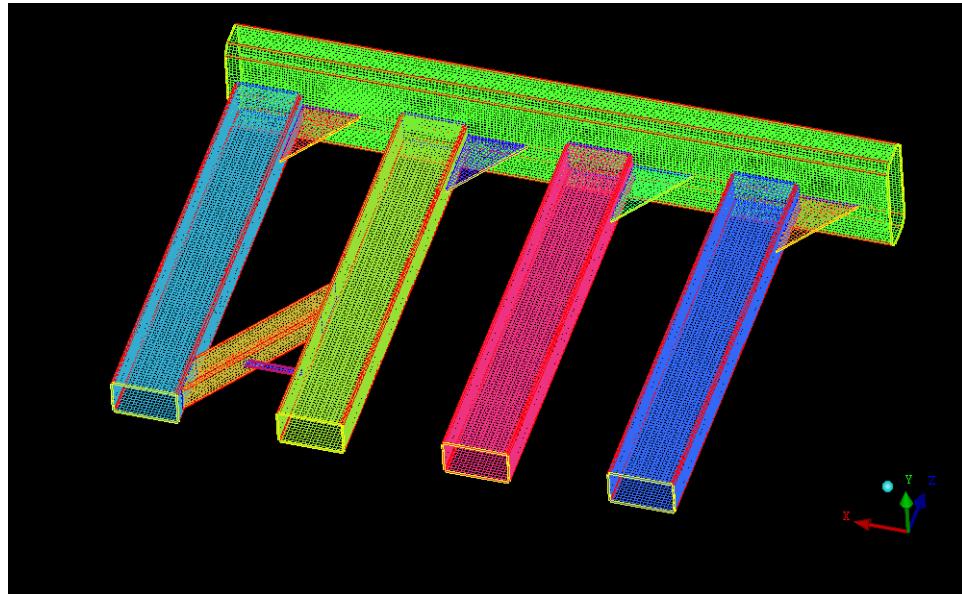
Only  , Input Geometry “All”, rest all default settings. Press ‘Compute’ button to generate the mesh as shown.

Figure 6-54:Compute Mesh



This would create a mesh as shown.

Note: User can see the mesh by Mesh>Shell>Solid and Wire in the Display tree

Figure 6-55: Final mesh

h) Define Material and Element Property:

Before applying Constraints and Loads on the elements, we have to define the type of material and assign properties to the elements.

Material Definition

Select the **Properties> Create Material Property** .

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**.

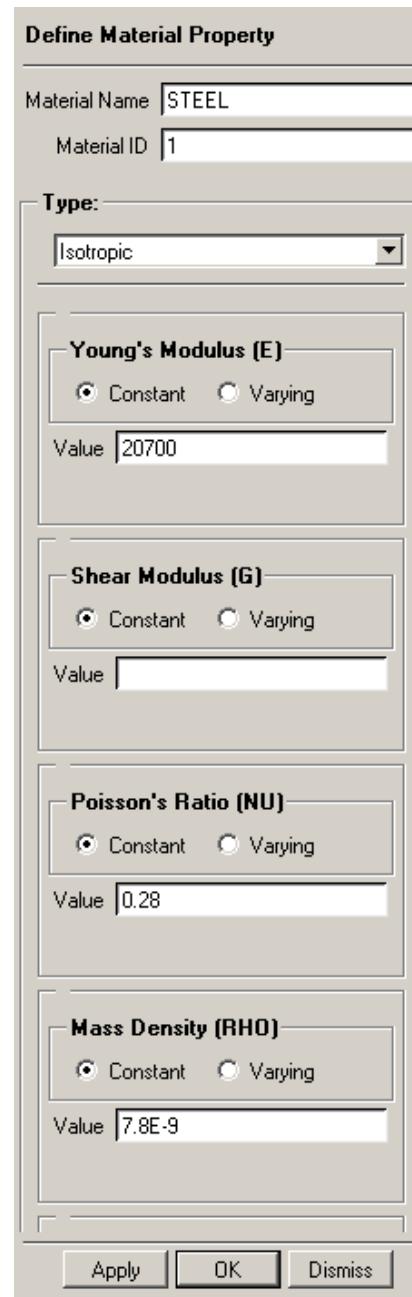
Define the **Thermal Expansion** Coefficient as a Constant **12.22e-6**.

Enter the **Reference Temperature** (TREF) as **298**.

Enter the values in the **Define Material Property** window as shown below.

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	963
------------------------	--	-----

Figure 6-56:Define Material Property Window



Press **Apply**.

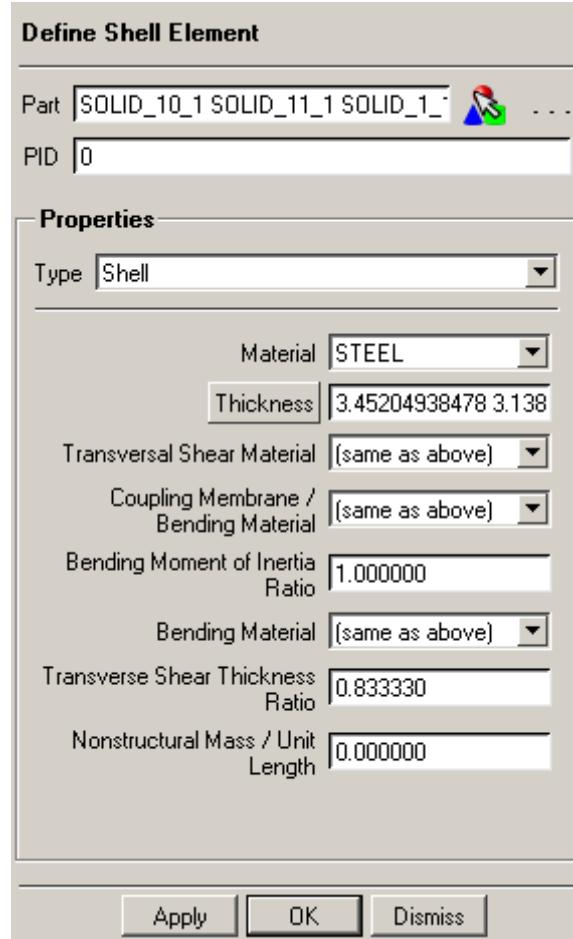
i) Define Shell Element Properties:

Now we have to define the thickness on the shell elements. Under

Properties > Define 2D Element Properties , we have the option to define the Properties for the Shell Elements.

In the Part selection mode, we can Press ‘a’ to select all the Parts, assign the Material as ‘STEEL’, and press Apply.

Figure 6-57:Define Shell Element



This will apply the properties of **Steel** on the created shell elements.

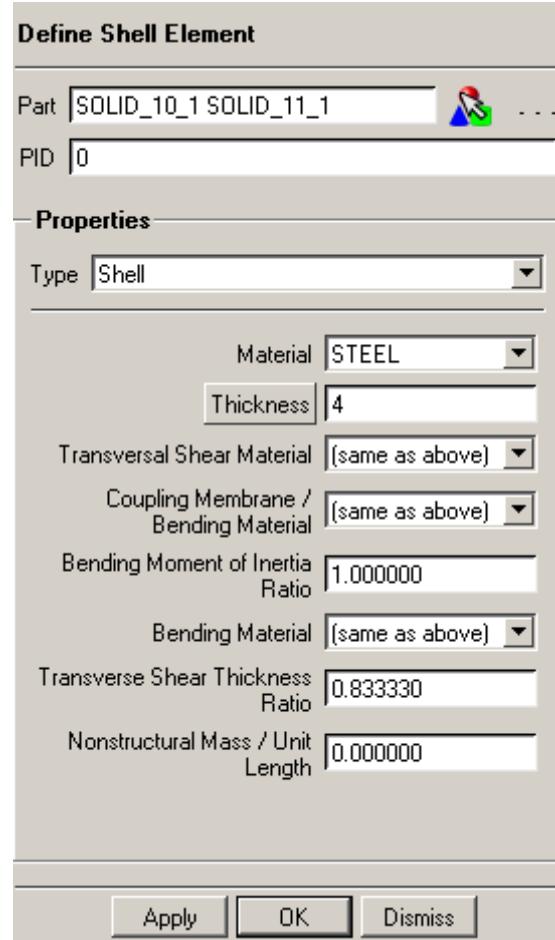
Now from Model Tree, Right Click on Mesh>Shell > Display Shell Thickness. This will display the existing shell thickness.

Now we can modify the shell thickness in two ways:-

1. **Thickness:** Option from Define Shell Element window. There we can choose any of the Part and change it's Thickness or also can

choose multiple parts. For example for parts, SOLID_10_1 and SOLID_11_1 the thickness has been changed to 4 units from previous 3.45205 and 3.13811 respectively.

Figure 6-58:Change Shell Thickness



2. **Edit Attribute:** options from the Part Display Tree. From Model Tree, Right Click on Part Display Tree >Edit Attribute. This will open up a window as shown here.

**Figure
6-59:Edit
Attributes**

part	color	property	material	element type	thickness
SOLID_10_1	#c9b433	surface	STEEL	ISHE	3.45205
SOLID_11_1	#5c33fb	surface	STEEL	ISHE	3.13881
SOLID_1_1	#33acd0	surface	STEEL	ISHE	4.18774
SOLID_2_1	#96e033	surface	STEEL	ISHE	4.18726
SOLID_3_1	#ed337f	surface	STEEL	ISHE	4.18726
SOLID_4_1	#3368f7	surface	STEEL	ISHE	4.17934
SOLID_5_1	#52fd33	surface	STEEL	ISHE	7.6516
SOLID_6_1	#fe413d	surface	STEEL	ISHE	10
SOLID_7_1	#5633fc	surface	STEEL	ISHE	10
SOLID_8_1	#33f66c	surface	STEEL	ISHE	10
SOLID_9_1	#eb8333	surface	STEEL	ISHE	10

Now we can change the thickness of any part from the table. Here for example for the part SOLID_5_1 the thickness is 7.6516 units which can be changed to say for example 10 units as shown in here.

**Figure
60:Edited
Attribute**

part	color	property	material	element type	thickness
SOLID_10_1	#c9b433	surface	STEEL	ISHE	3.45205
SOLID_11_1	#5c33fb	surface	STEEL	ISHE	3.13881
SOLID_1_1	#33acd0	surface	STEEL	ISHE	4.18774
SOLID_2_1	#96e033	surface	STEEL	ISHE	4.18726
SOLID_3_1	#ed337f	surface	STEEL	ISHE	4.18726
SOLID_4_1	#3368f7	surface	STEEL	ISHE	4.17934
SOLID_5_1	#52fd33	surface	STEEL	ISHE	10
SOLID_6_1	#fe413d	surface	STEEL	ISHE	10
SOLID_7_1	#5633fc	surface	STEEL	ISHE	10
SOLID_8_1	#33f66c	surface	STEEL	ISHE	10
SOLID_9_1	#eb8333	surface	STEEL	ISHE	10

6.1.7: Tibia

Overview

In this example, the user will generate pure Hexa Dominant mesh on the Bone geometry with Body Fitted Cartesian Mesher. The bone, shown below, represents a STL geometry of a actual human bone. It is of amorphous nature which makes it difficult to mesh. This Body Fitted Cartesian Meher generates quickly pure Hexa elements on bad geometry like this.

Figure 6-61:Tibia



a) Summary of steps

Starting the project

Setting up Body Fitted Cartesian Mesh Setup

Assigning Surface Sizea

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	970
------------------------	--	-----

Generating Volume Mesh
Running Check Mesh
Quality Check
Automatic Smoothing
Saving the Project

b) Starting the Project

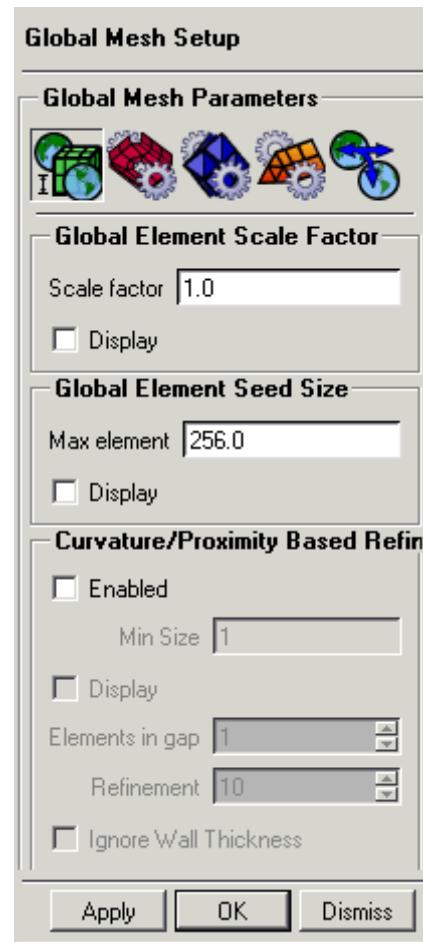
The input files for this tutorial can be found in the Ansys installation directory, under ..\v110\docu\Tutorials\AI_Tutorial_Files > Tibia project. Copy the geometry (*.tin) file to your working directory and open it. Turn ON the surfaces if the geometry is not available.

c) Setting up Body Fitted Cartesian Mesh Setup

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 **256.0**. Press Apply followed by Dismiss to close the window.

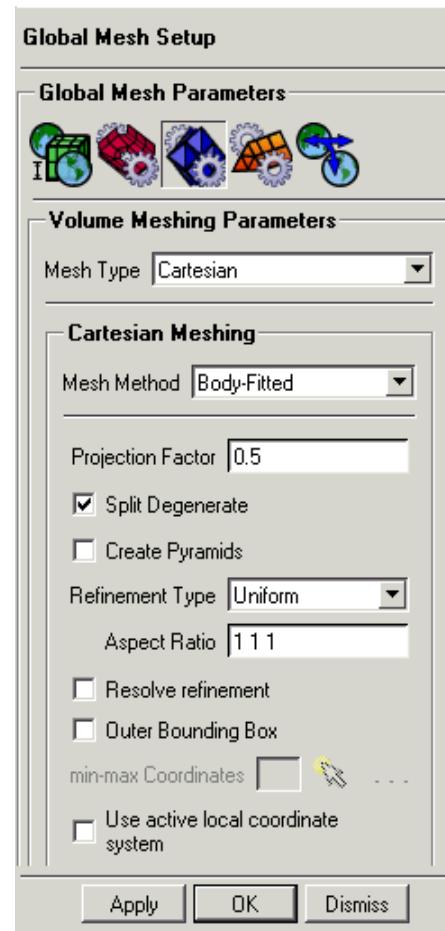
Figure 6-62:Global Mesh Size

- From **Global Mesh Setup**, Select **Volume Meshing**

 Now choose Mesh Type as **Cartesian**, the Mesh Method as **Body Fitted**.

Now, keep projection Factor as 0.5 .

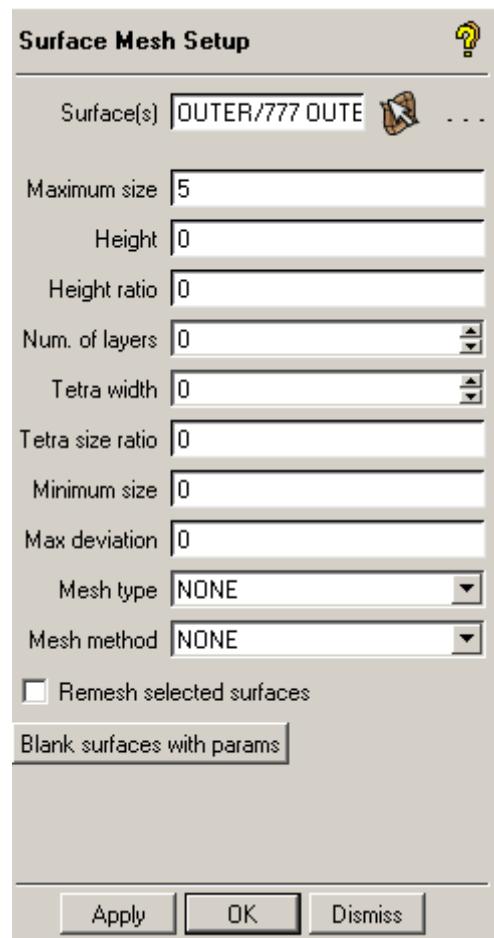
Toggle on Split Degenerate as demonstrated below.

Figure 6-63:Volume Meshing Parameters

Press Apply to apply the settings followed by Dismiss to close the GUI.

d) Surface Mesh Setup:

Now from Mesh>Surface Mesh Setup  , select all the surfaces by hot key ‘a’ . Apply surface size of ‘5’ on all the surfaces as shown below.

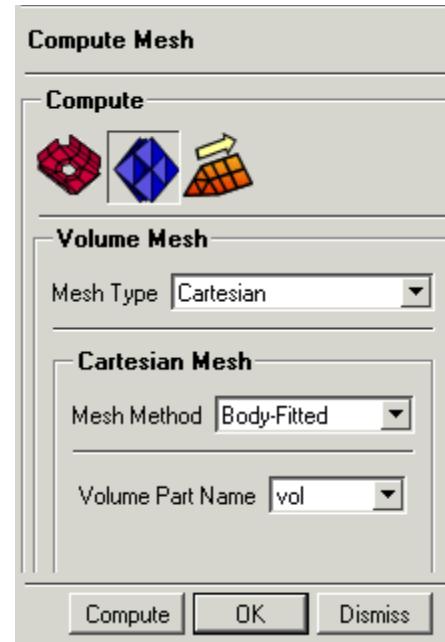
Figure 6-64:Surface Mesh Setup

Press Apply button to apply uniform size of 5 on all the surfaces. This can be checked from Model Tree, Right Click on Surfaces> Tetra Sizes.

e) Compute Mesh:

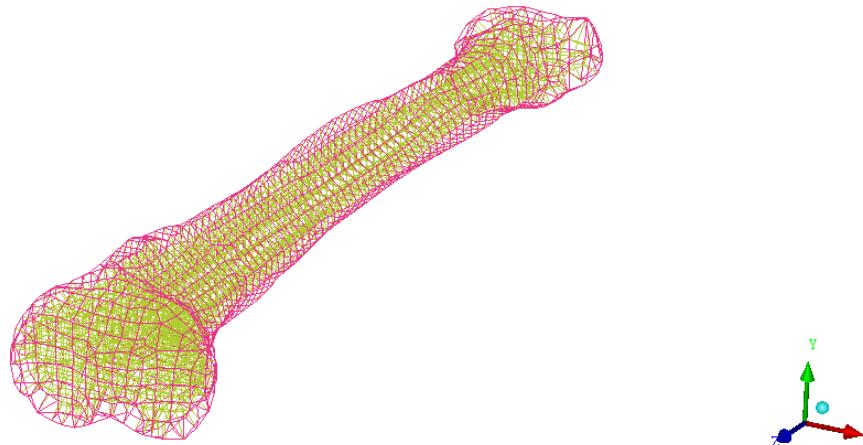
Now select Mesh>Compute Mesh > Volume Mesh . Change the Mesh Type to Cartesian, Mesh Method to Body-Fitted and Volume Part Name ‘vol’ as shown here.

Figure 6-65:Compute Mesh



Press ‘Compute’ button to start for generating volume mesh. Upon asking to save the settings file, user can save on the existing geometry file or can save in a new name as desired. The volume mesh generated is shown here.

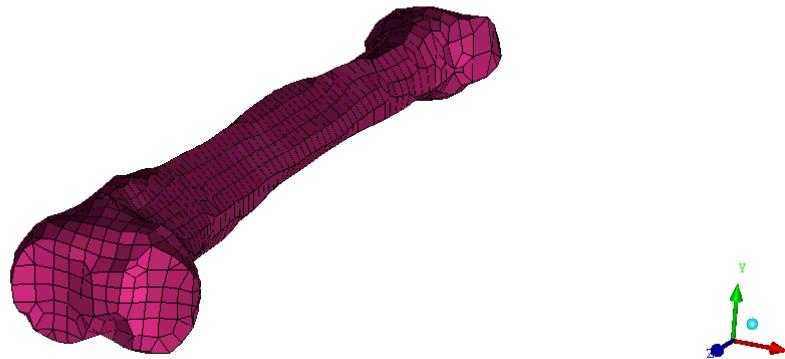
Figure 6-66: Mesh Generated



Turn OFF surfaces from model tree.

From Model Tree, Right Click on Shell and Turn ON Solid and Wire mode. The mesh will look like this as shown.

**Figure
6-67:Shell
Mesh
only**

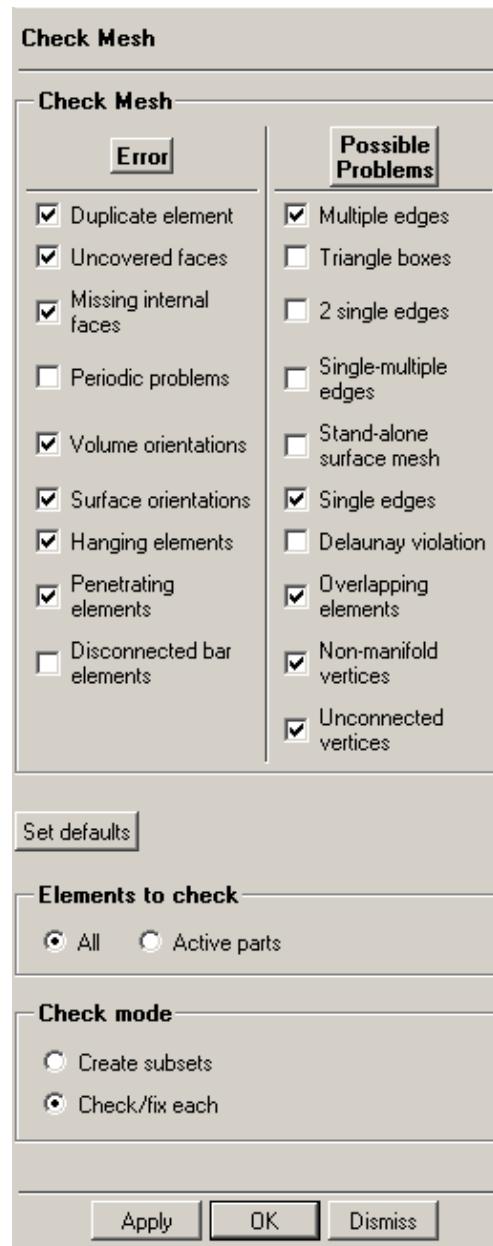


f) Check Mesh:

Now the mesh so created should be checked for any errors. Go for Edit



Mesh > Check Mesh options to check all the errors and possible problems present in the mesh created. This should open a window as shown in

Figure 6-68:Check Mesh

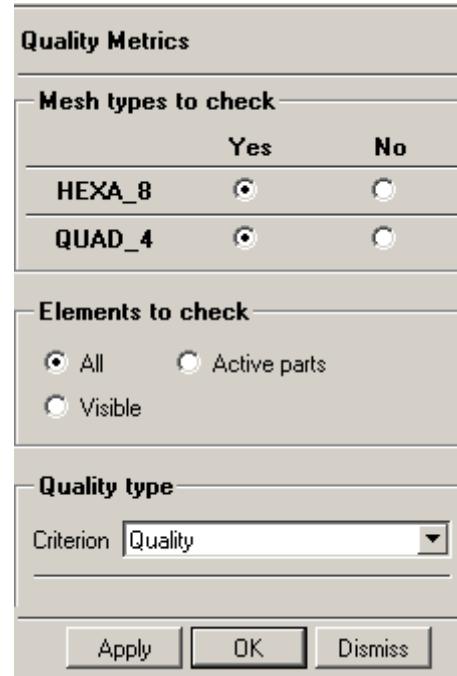
Choose the options for Check mesh as shown in the above figure. There can be some errors for 'Volume Orientation'. Click on 'Fix' tab to correct the volume orientation errors.

g) Mesh Quality:-

The mesh so created should be checked for its quality. Go to Edit Mesh >

Display Mesh Quality  . It should open a window as shown in

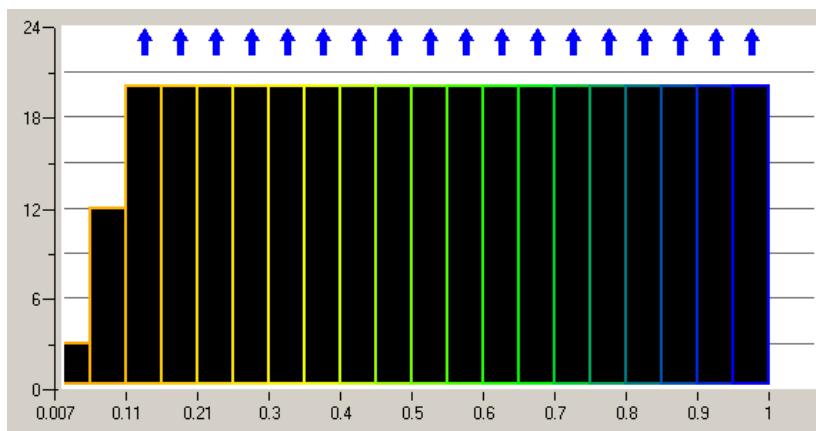
Figure 6-69:Quality Metric



Press 'Apply' to get the Quality Histogram.

The Quality achieved for the mesh elements will be displayed in the Quality Histogram as shown in the

**Figure
6-70:Quality
Histogram**

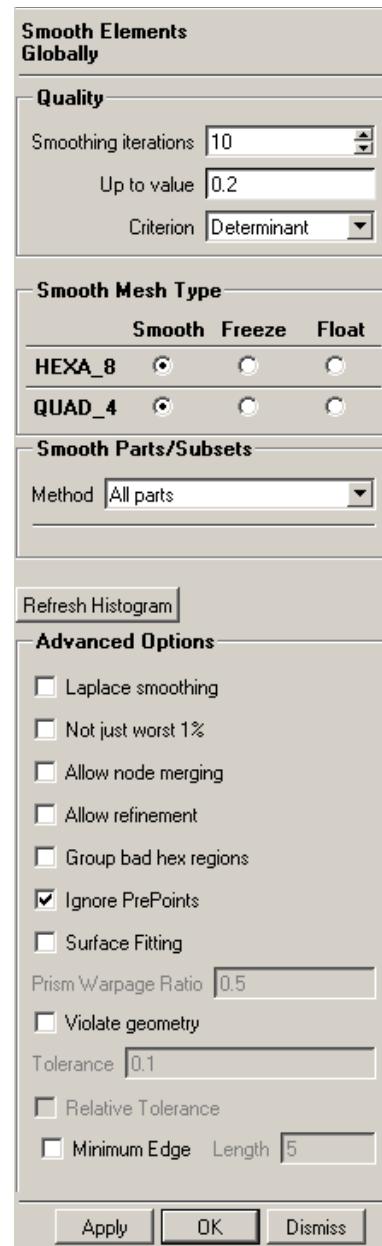


The Quality obtained is as Min = 0.00702059, max = 0.999892, mean = 0.776753440131.

h) Smooth Elements Globally:

The Quality so achieved can be increased further to higher level. For that Smooth Element Globally options should be used. Go for Edit Mesh >

Smooth Elements Globally  to get the Global Smoothing window as shown in

Figure 6-71:Smooth Element Globally

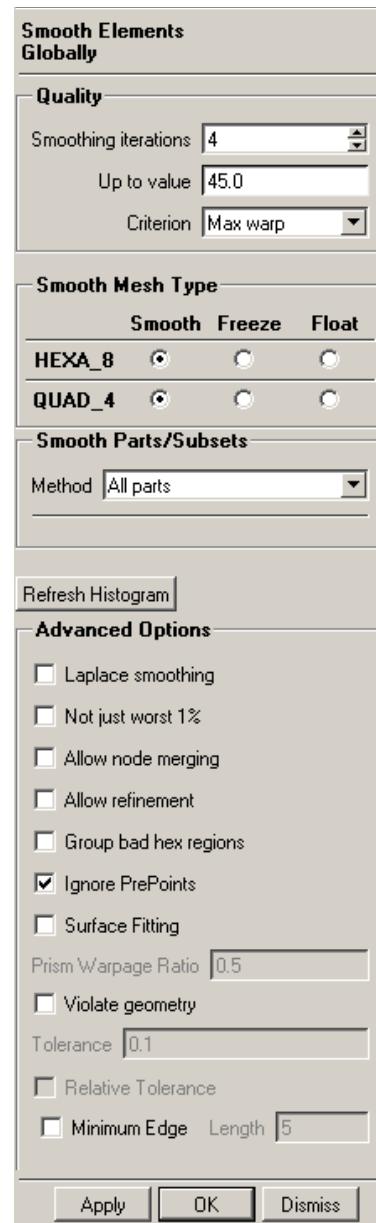
From the Smooth Element Globally window, Change the Criterion of Smoothing from Quality to Determinant as shown in the above figure.

Turn OFF Group bad hex regions.

Turn ON Ignore Pre Points.

Press '**Apply**' to run the smoother. Do the smoothing operations till you get desired mesh quality.

After getting satisfactory result through smoothing operations for Determinant criterion, change the smoothing criterion to Max Warp from Determinant as shown in

Figure 6-72:Smooth Element-Max Warp

Keep the Up to Value to 45 as shown in the above figure.

Press ‘Apply’ to start the smoothing operation.

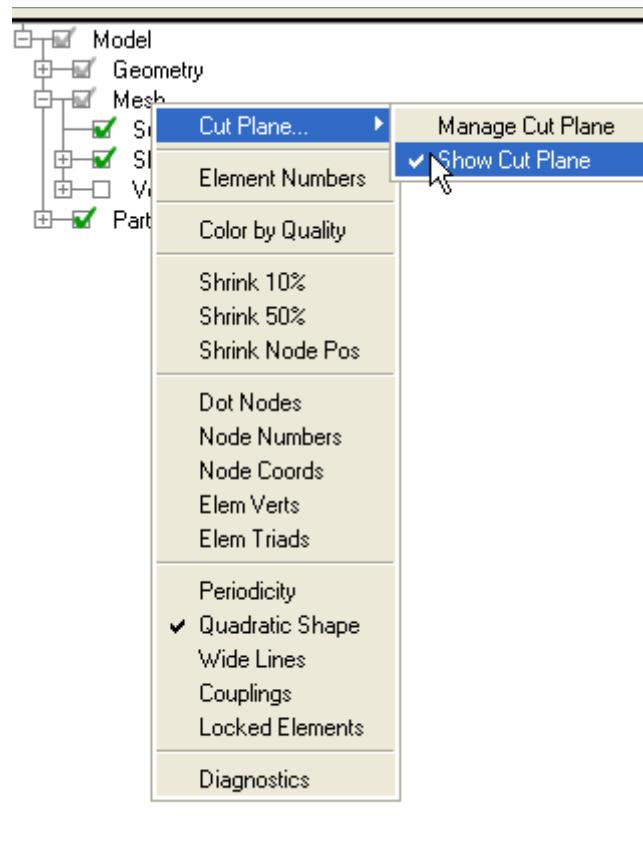
Do the smoothing operation by changing different parameters till the desired quality is achieved.

Re-check Determinant again and do smoothing if necessary.

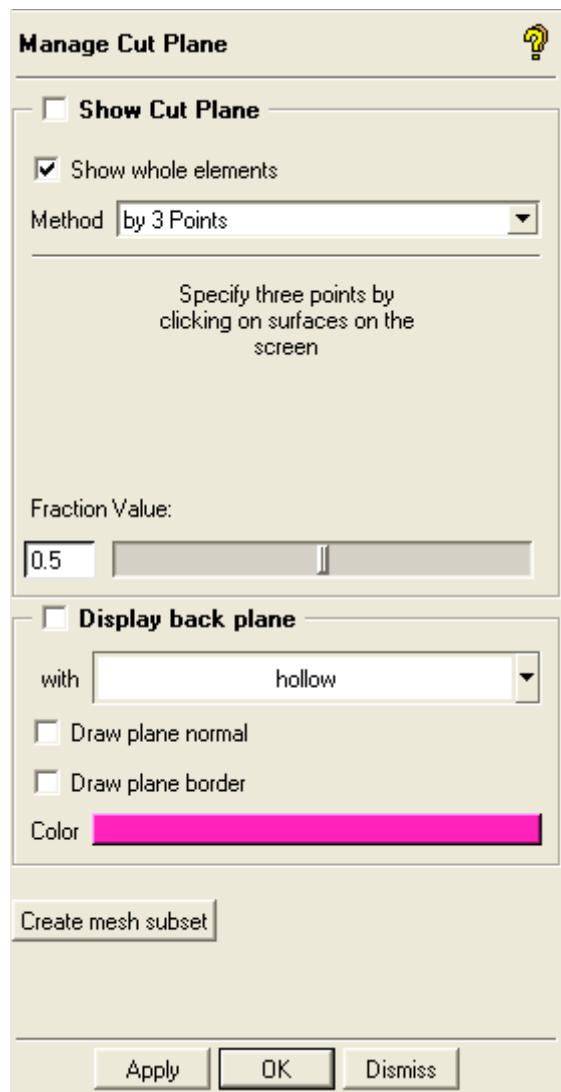
i) Display Volume Mesh:

Turn ON Surfaces. Now Right Click on Mesh > Cut Plane >Show Cut Plane as shown in

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	984
------------------------	--	-----

Figure 6-73:Cut Plane

This will open up the cut plane GUI as shown in

Figure 6-74:Manage Cut Plane

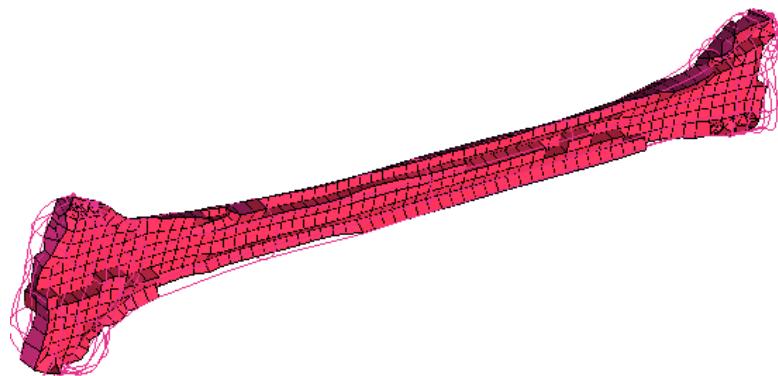
Change the method to by 3 Points as shown in the above figure. Select any 3 points along the surface to describe a plane along the length of the bone.

Turn ON Mesh >Volumes in the Model Tree.

Scroll Fraction Value bar to scan through and get a proper view along the full length of the bone as shown in the below figure.

The Hexa Hedrals generated can be viewed from the cut plane as shown in the

**Figure
6-75:Cutaway
view of Hexa
Hedrals**



6.2: Ansys Tutorials

6.2.1: T-Pipe(Nastran Modal): 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 here.

T-Pipe
model



a) Summary of Steps

Data Editing

Launch AI*Environment and import an existing Nastran data file

Verification of imported data

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	988
------------------------	--	-----

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.

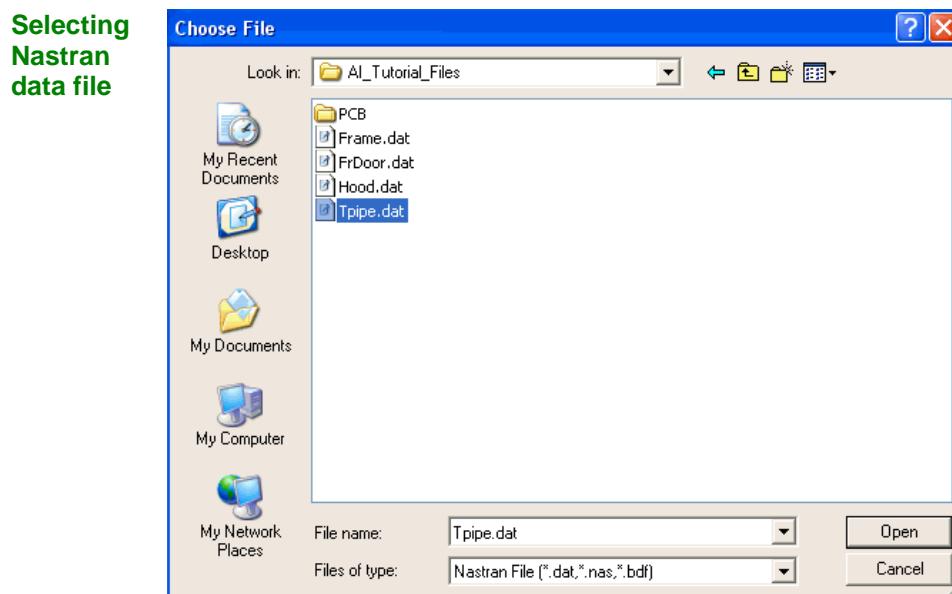
Launch AI*Environment.

Select **File > Import Mesh > From Nastran** from the main menu, which will open the **Import Nastran File** window.

Import Nastran File window



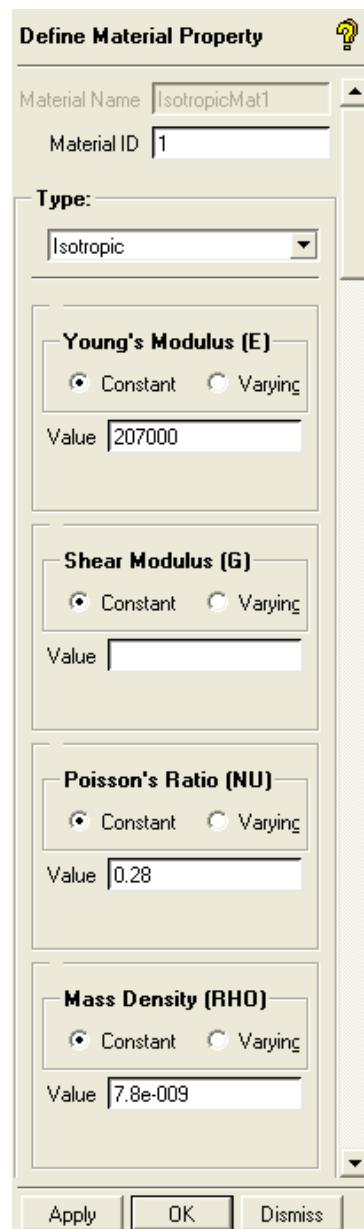
Click on the open file icon for the file-browsing window. Select the file **Tpipe.dat** from the **AI_Tutorial_Files** directory.



After the Tpipe.dat, Press Apply in the Import Nastran File window

c) Verification of imported data

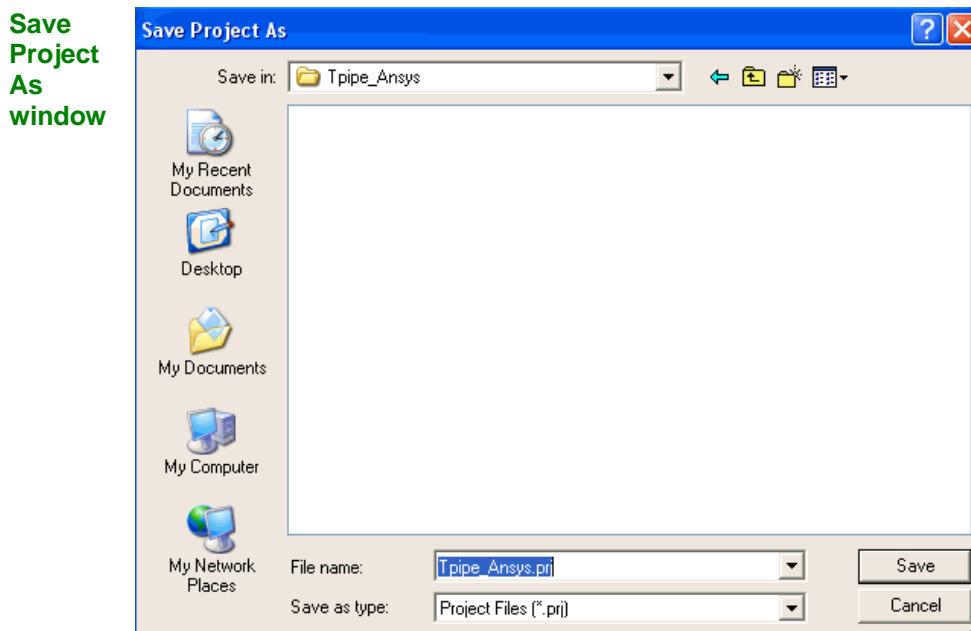
Expand the **Material Properties** in the Model Tree by clicking on the +. To open the **Define Material Property** window for **IsotropicMat1**, double click on the material, **IsotropicMat1**, with the left mouse button, or right click and select **Modify**.

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 below. 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.



e) Solver Setup

First, the user should select the appropriate solver before proceeding further. Select **Settings > Solver** from the main menu and select **Ansys** from the Common Structural Solver dropdown arrow. Then press **Apply**.

Click the Solve Options tab, then the Setup Analysis Type icon. The window that appears is shown below.

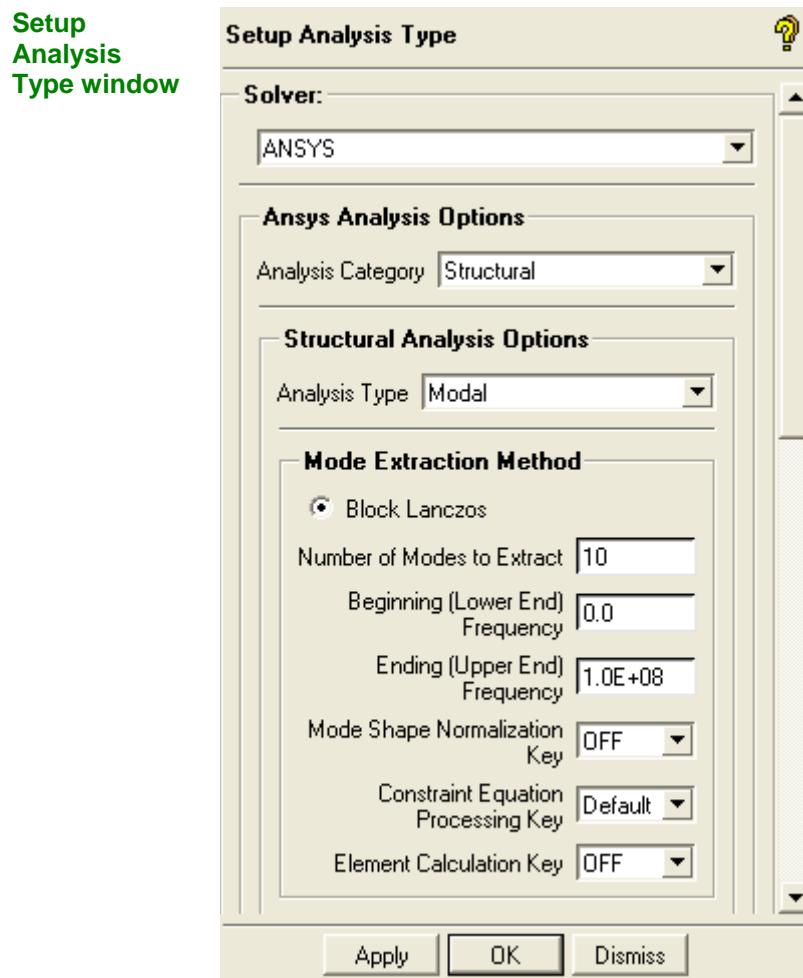
Enter the following:

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.



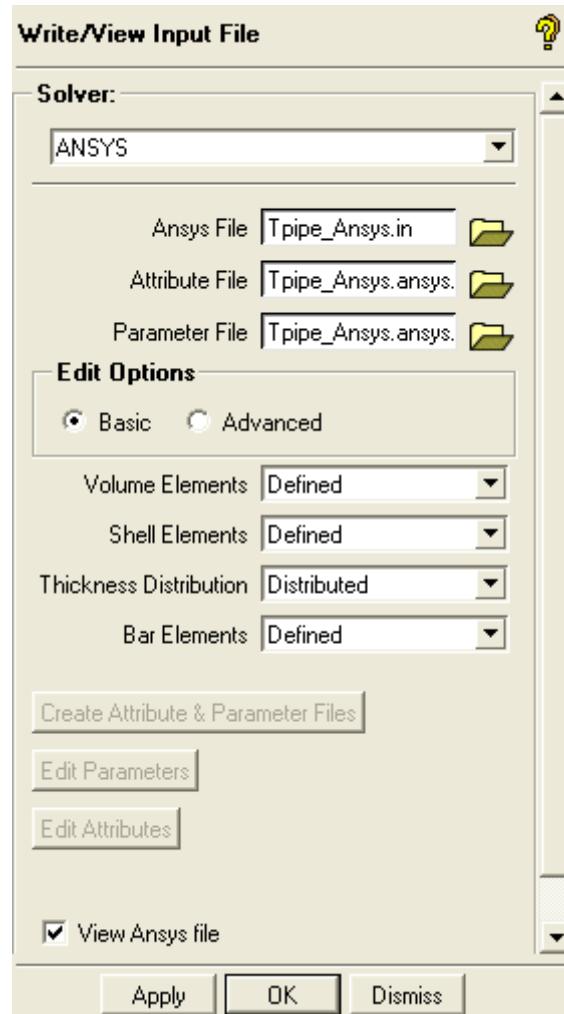
f) Write Ansys Input File

Press the **Write/View Input File** icon  from the **Solve Options** Menu bar.

The Ansys File name should be Tpipe_Ansys.in.

Scroll to the bottom and switch **ON View Ansys file**.

Keep the other options as the default and press **Apply**.

**Write/View
Input File
window**

Toggle ON the view Ansys File and Press Apply

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 ICEM CFD.

Solving the problem

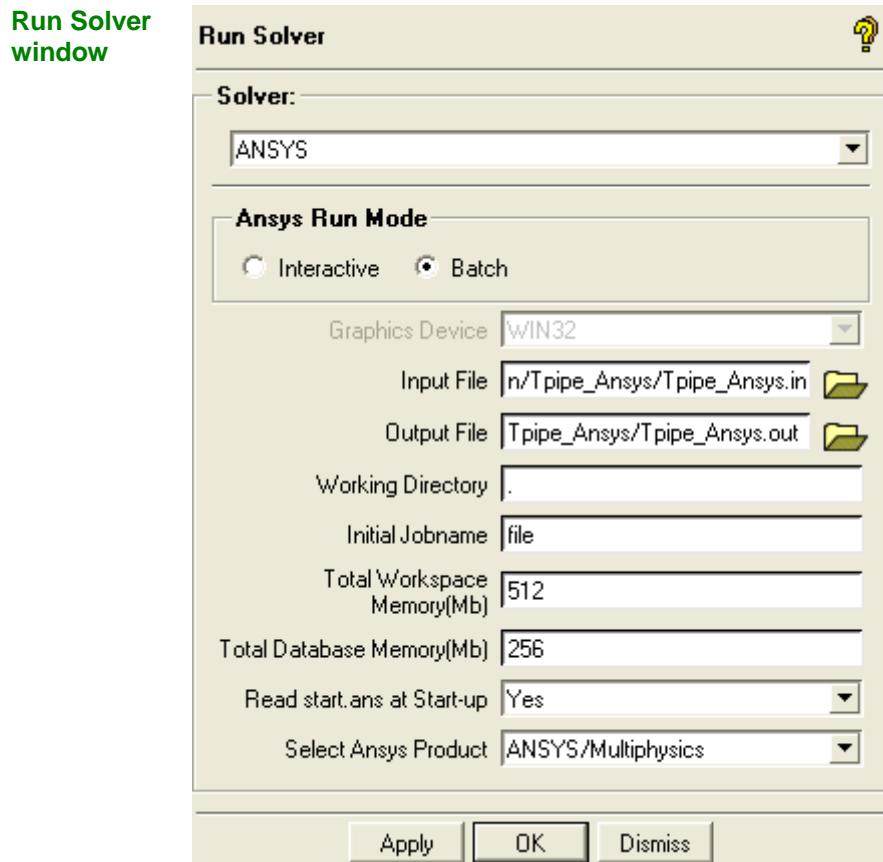
Click on the **Submit Solver Run** icon  from the **Solve Options** Menu bar to open the window shown below.

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.

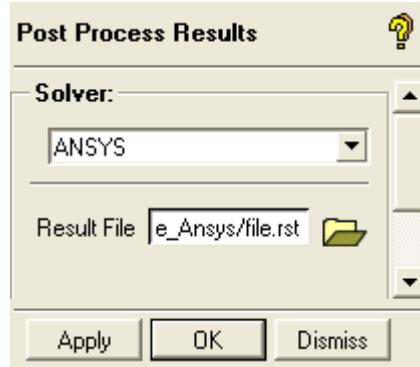
The ANSYS_EXEC_PATH environment variable may have to be set to the full path to the Ansys executable for ICEM CFD to be able to run Ansys.

Press **Apply** to run the Ansys solver in batch mode.

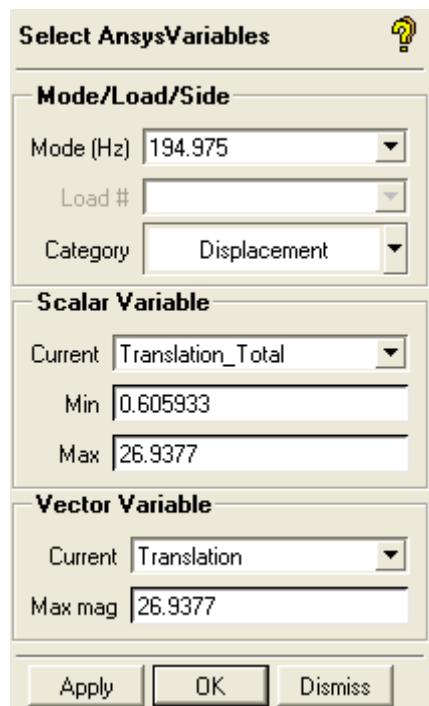


Post Processing of Results

Click on the **Post Process Results** icon  from the **Solve Options** Menu bar, which will open the **Post Process Results** window given below. 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.

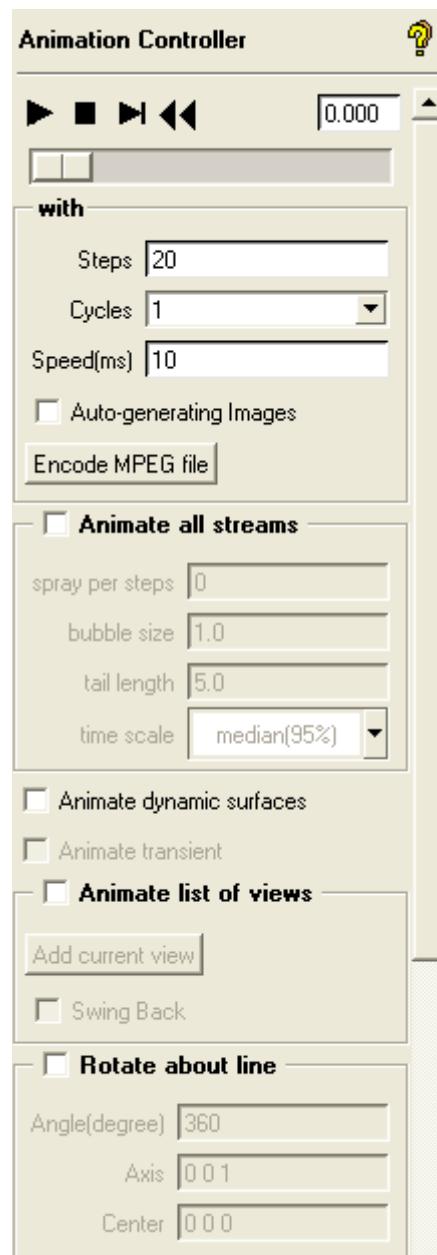
Post Process window

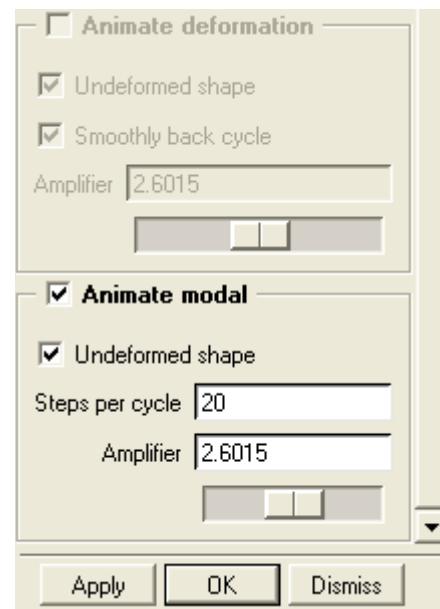
Click on the **variables** icon from the **Post processing** tab to display the **Select AnsysVariables** window. Select the pull down next to **Mode(Hz)** to display the modal frequencies that resulted from the Ansys solution. The numbers near zero are the solutions to the homogeneous equation that 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 here.

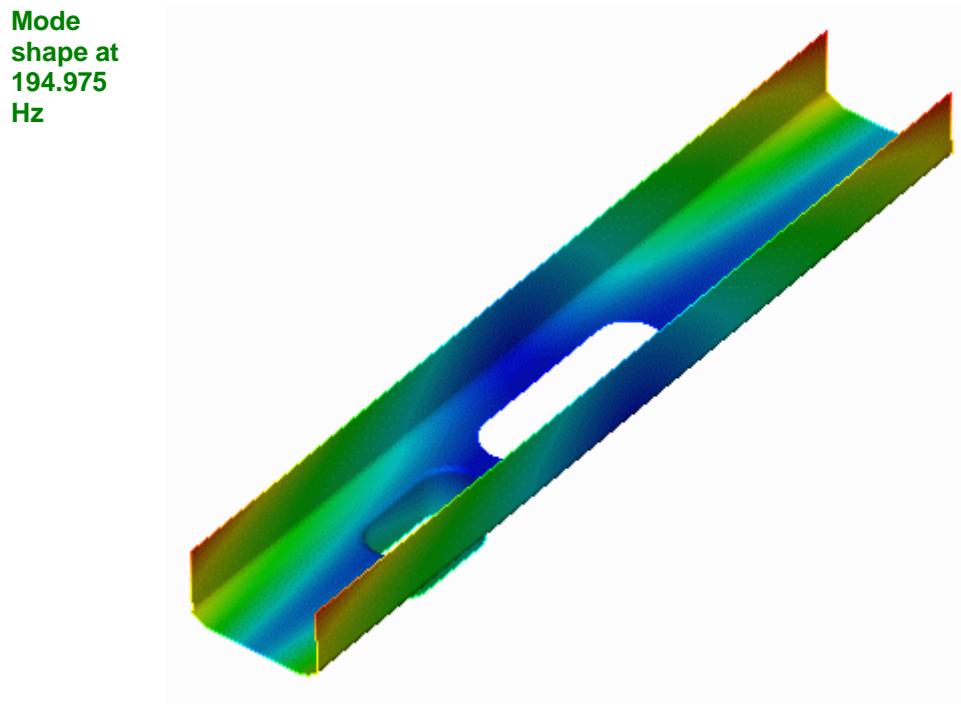
**Select Ansys
Variables
window**

Click on the **Control all Animations** icon  from the **Post-processing** menu bar. Select  **Animate** (play button).

**Animation
Setup and
Controller
window**

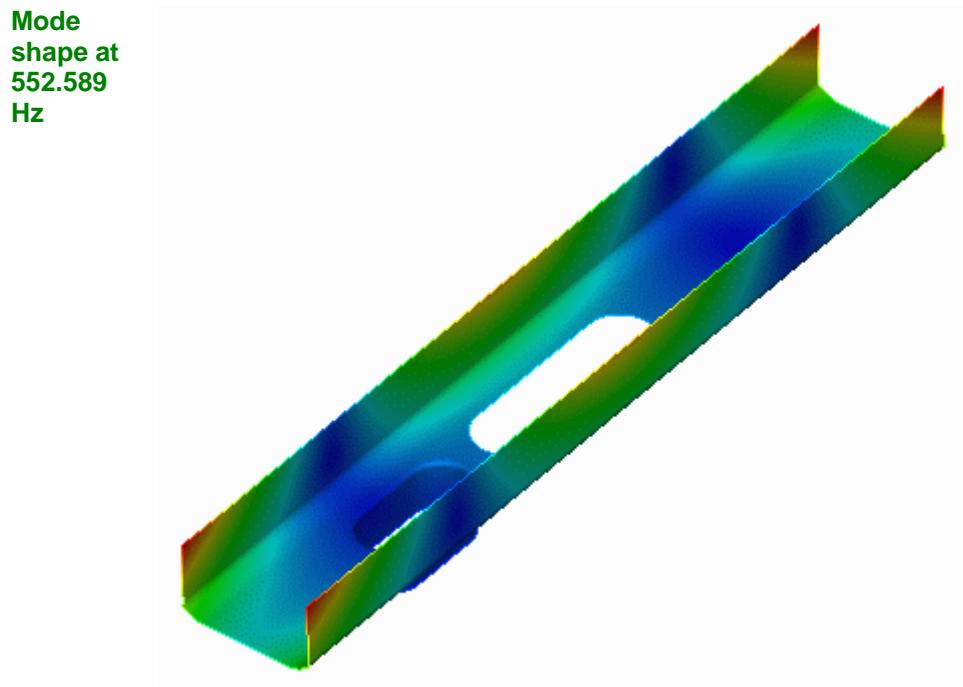






Similarly to view another mode shape, select the next frequency 552.589 Hz from the **Select Ansys Variables** window and animate the mode shape as shown below.

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	1002
------------------------	--	------

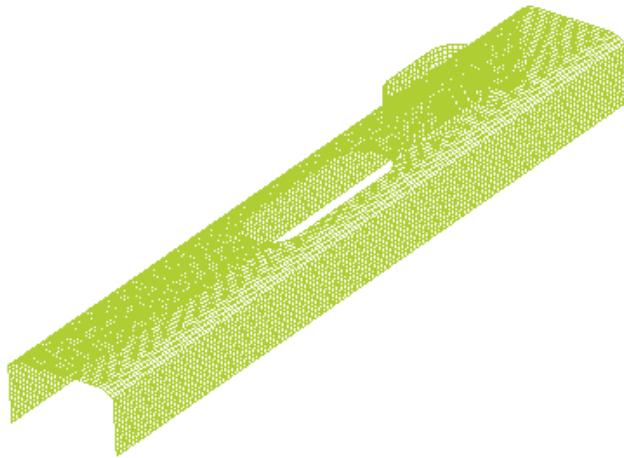


Finally, select **File > Results > Close Result** to quit the post-processor.

6.2.2: T-Pipe(Abaqus Modal): Modal Analysis

The main objective of this tutorial is to demonstrate legacy conversion from a Abaqus model to an Ansys model. It also highlights the ease of use with **AI*Environment** in translating a model from one solver to another. An Abaqus modal analysis data file is provided as input. Once the *.inp File is imported into **AI*Environment** and the solver is changed to Ansys, the shell element materials, which are defined for Abaqus, are converted to the corresponding Ansys materials. The imported mesh is shown here.

T-Pipe



a) Summary of Steps

Launch AI*Environment and import an existing Abaqus data file

Verification of imported data

Solver setup

Setting Solver Parameters

Write Ansys Input File

Solution and Results

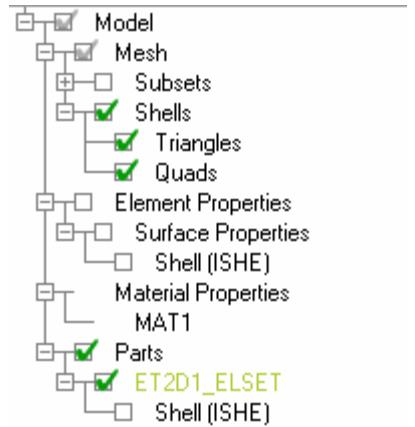
Solving the problem

Visualization of Results

From Main Menu, Select File > Import Mesh > From Abaqus from the main menu, which will open the Open Menu > Select TPipe.inp

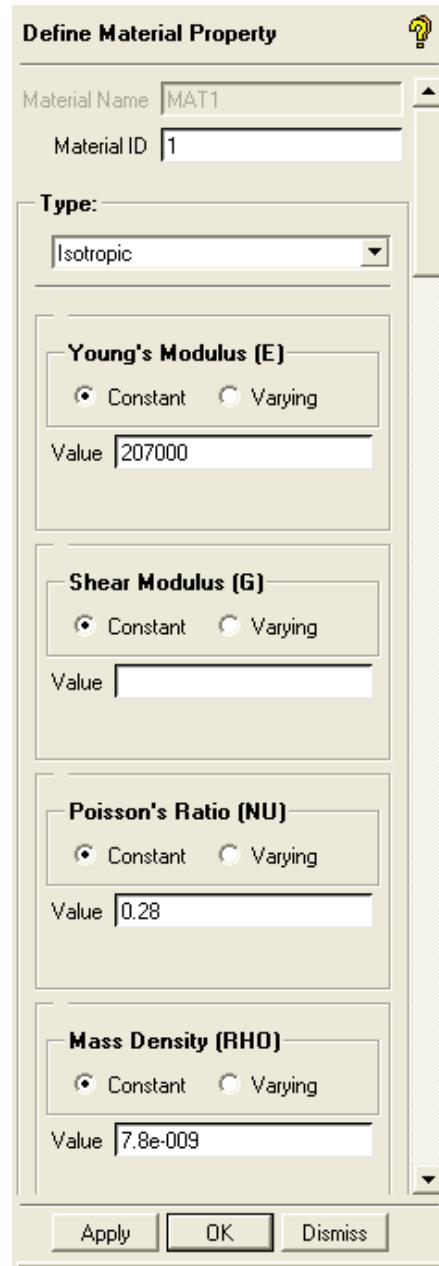
The Display Control Tree after you import the TPipe.inp is shown below.
Expand the Material Properties in the Display Control Tree by clicking on the +.

Display Control Tree



b) Verification of imported data

To open the Define Material Property window for MAT1, double click on the material - MAT1 with the left mouse button or right click and select **Modify**

Define Material Property window

c) 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 Common Structural Solver as **Ansys** from the dropdown arrow. Then press **Apply**.

Setting Solver Parameters

Click the Solve Options tab, then the Setup Analysis Type icon.  The window that appears is shown below.

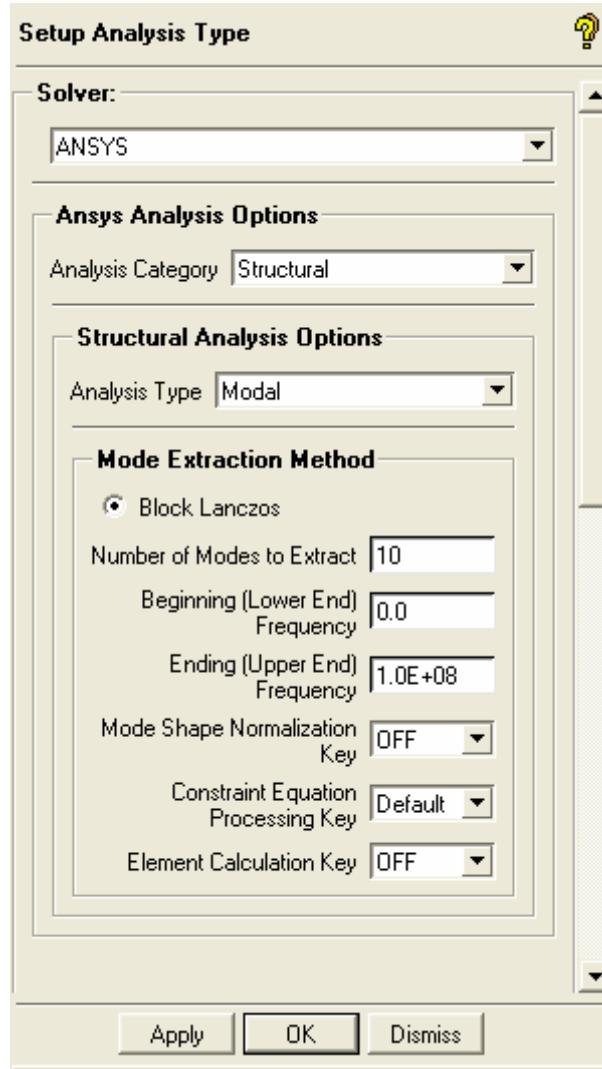
Enter the following:

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.

Setup Analysis Type 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**, then enter that folder and enter **TPipe.prj** as the project name and Press the **Save** button.

Along with the TPipe.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.

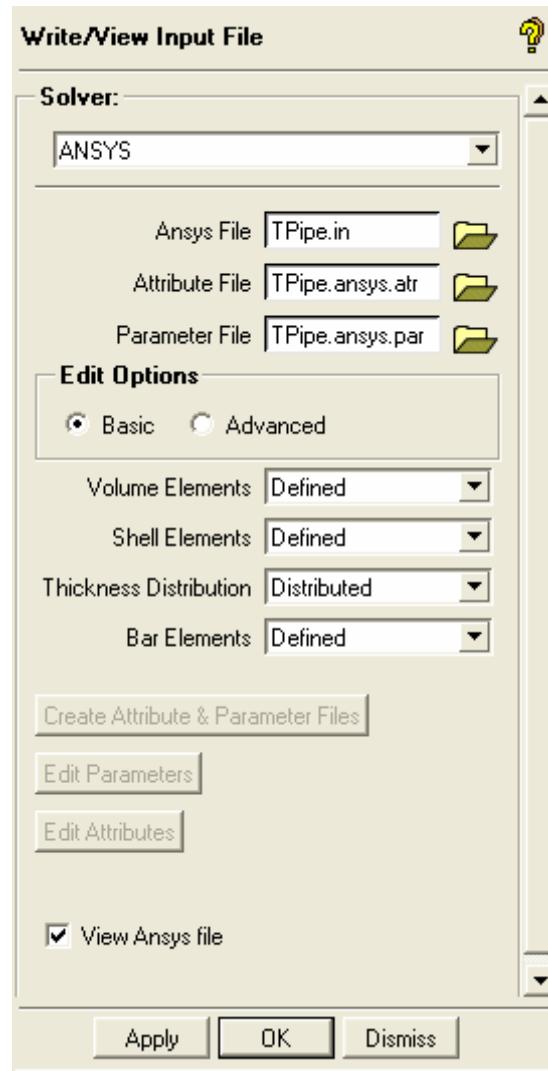
e) Write Ansys Input File

Press the **Write/View Input File** icon  from the **Solve Options** Menu bar.

The Ansys File name should be TPipe.in.

Scroll to the bottom and switch **ON View Ansys file** as shown here.

Keep the other options as the default and press Apply.

Write/View Input File**f) Solution and Results**

A modal analysis will be performed in Ansys on this model and the results will be visualized within ANSYS ICEM CFD.

Solving the problem

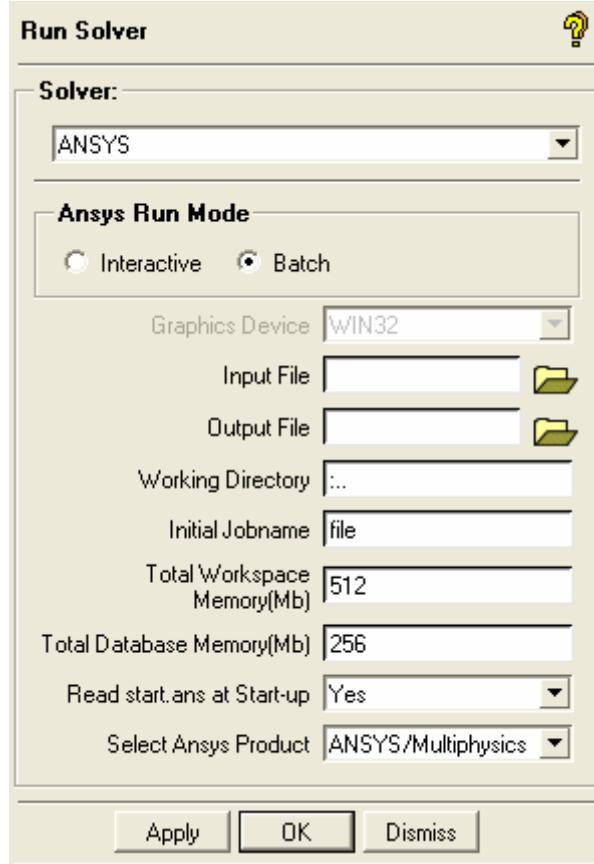
Click on the **Submit Solver Run** icon  from the **Solve Options** Menu bar to open the window shown.

Select **Batch** under **Ansys Run Mode**. Next to **Input File**, the name of the previously written input file should appear, **TPipe.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 ICEM CFD to be able to run Ansys.

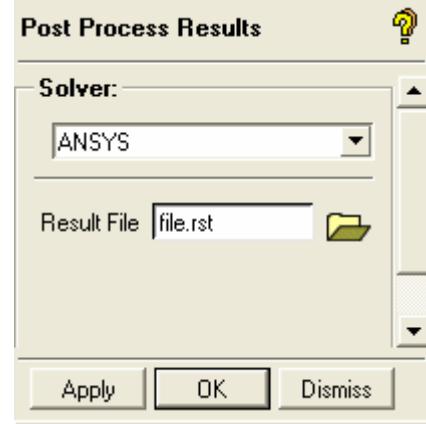
Press Apply to run the Ansys solver in batch mode.

Figure 6-76 Run Solver

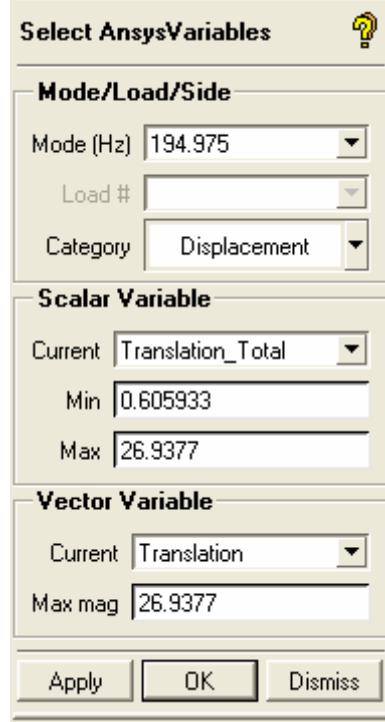
Post Processing of Results

Click on the **Post Process Results** icon  from the **Solve Options** Menu bar, which will open the **Post Process Results** window. Confirm that the **Solver** is set to Ansys.

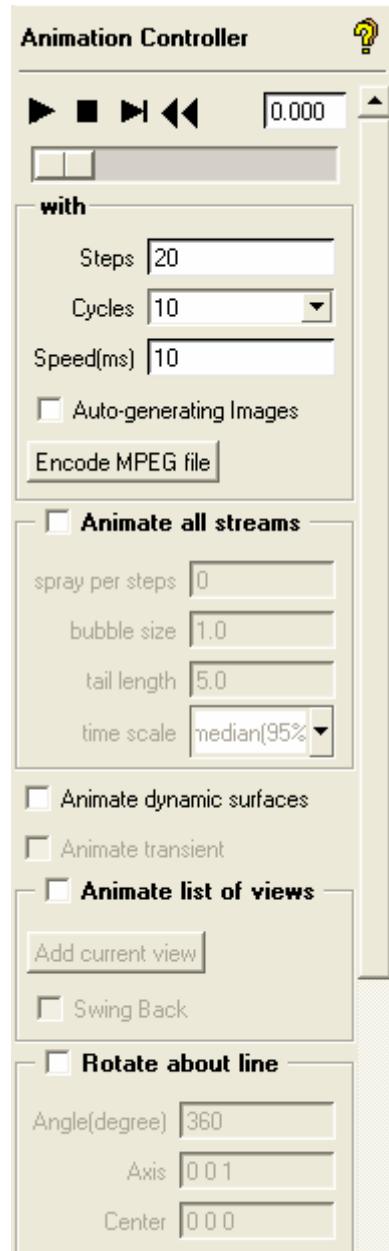
The Ansys **Result file** should be set to **file.rst** where “file” is the **Initial Jobname** specified in the previous menu. Press **Apply** to launch the Post processor with the Ansys result file.

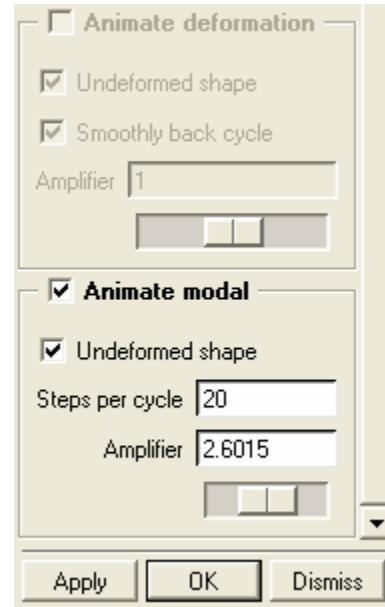
Post Process window

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 here.

**Select Ansys
Variables window**

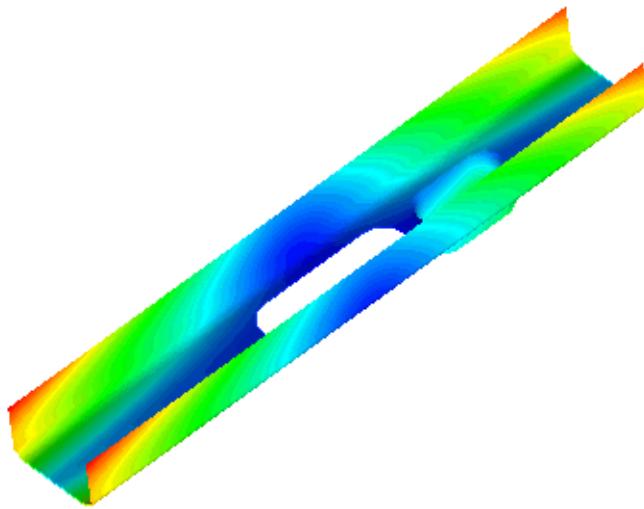
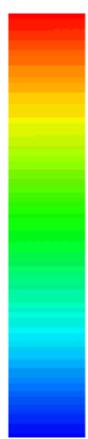
Click on the Control all Animations icon  from the Post-processing menu bar. Select Animate (play button). ►

Animation Setup and Controller window

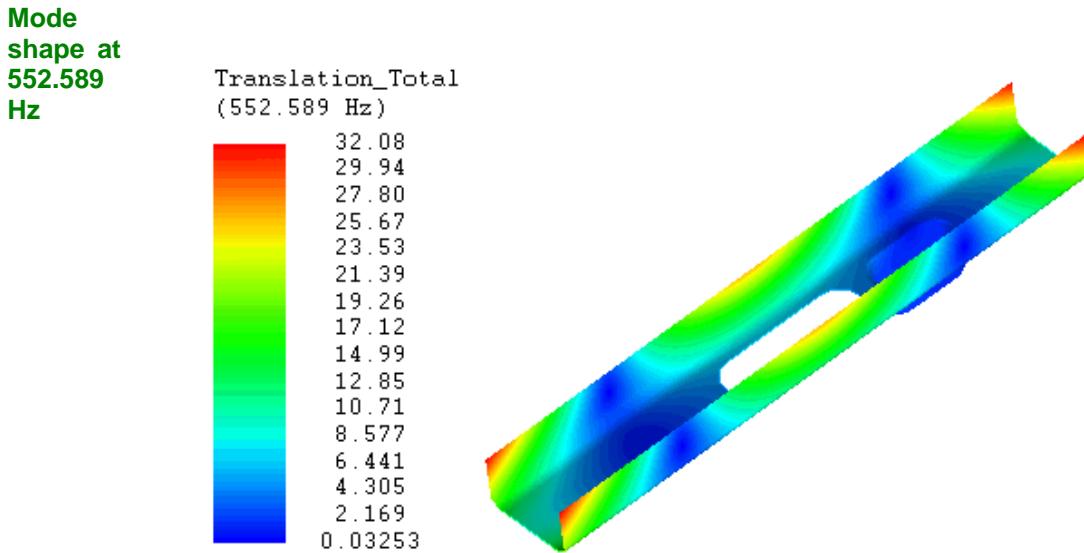


Mode
shape at
194.975
Hz

Translation_Total
(194.975 Hz)



Similarly to view another mode shape, select the next frequency 552.589 Hz from the **Select Ansys Variables** window and animate the mode shape as shown below.

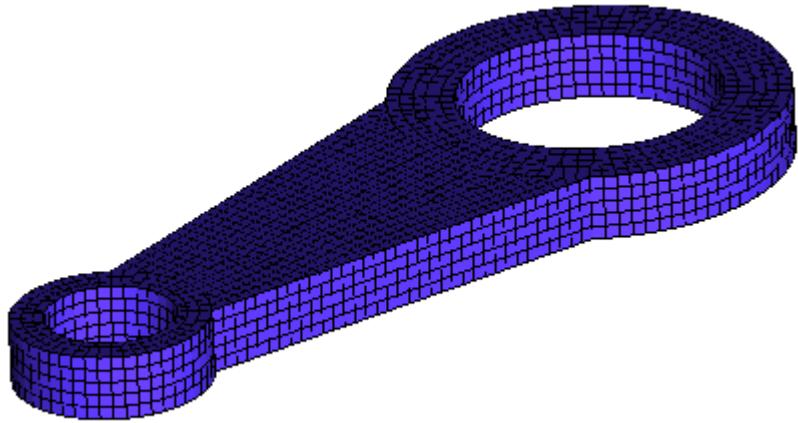


Finally, select File > Results > Close Result to quit the post-processor.

6.2.3: Connecting Rod: Thermal Boundary Condition

AI*Environment can be used to see the thermal 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).

Connecting Rod



a) Summary of Steps

- Launch AI*Environment and load Project file
- Materials and Element Properties
 - Selection of Material
 - Element Properties
- Subsets
 - Subset1
 - Subset2
- Constraints and Loads
 - Constraints
- 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

Proceed the Tutorial as the Continue Before Connecting Rod:Structural Meshing Tutorials.

b) Open Project File

Open the Project File > ConnectingRod.prj as you Earlier Completed in the Structural Meshing Tutorials

c) 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 **MAT1** 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**,

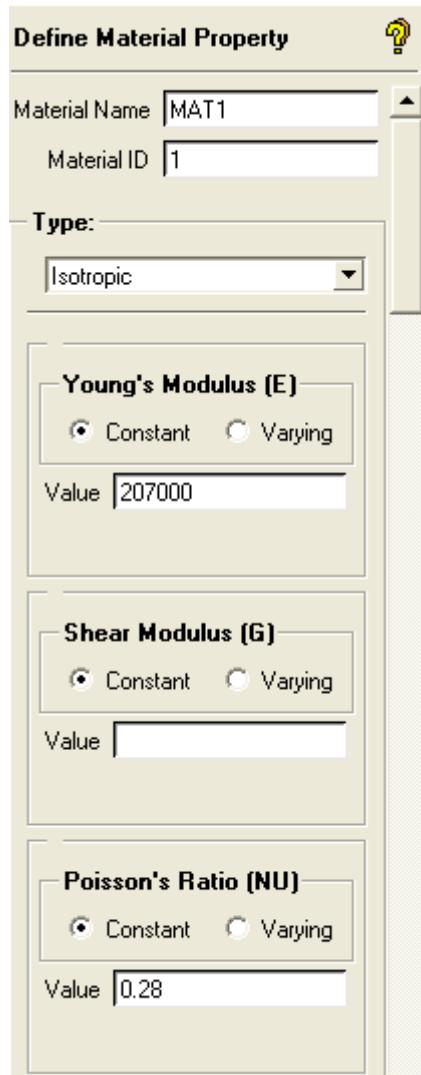
Define Poisson's ratio as a **Constant 0.28**,

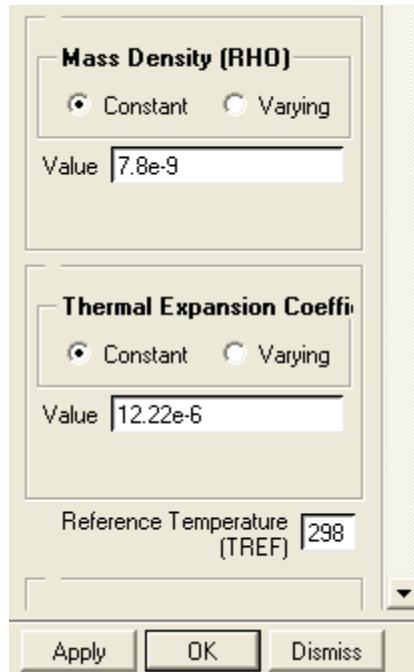
Define the Mass Density as a **Constant 7.8e-9**.

Define the Thermal Expansion Coefficient as a **Constant 12.22e-6**.

Enter the Reference Temperature (TREF) as 298.

Press Apply.

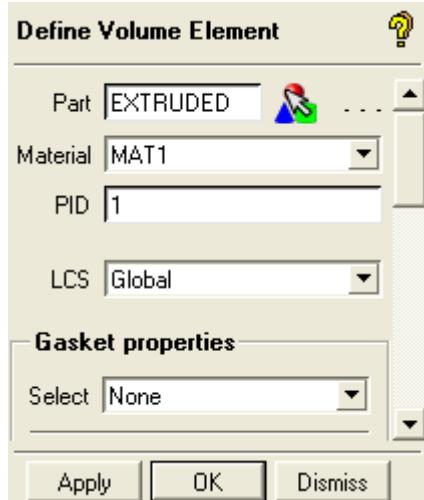
Define Material Property window



Element Properties

Select the **Properties> Define 3D Element Properties** icon, and the **Define Volume Element** window as shown below 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.

Define Volume Element window

d) 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 below.

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**

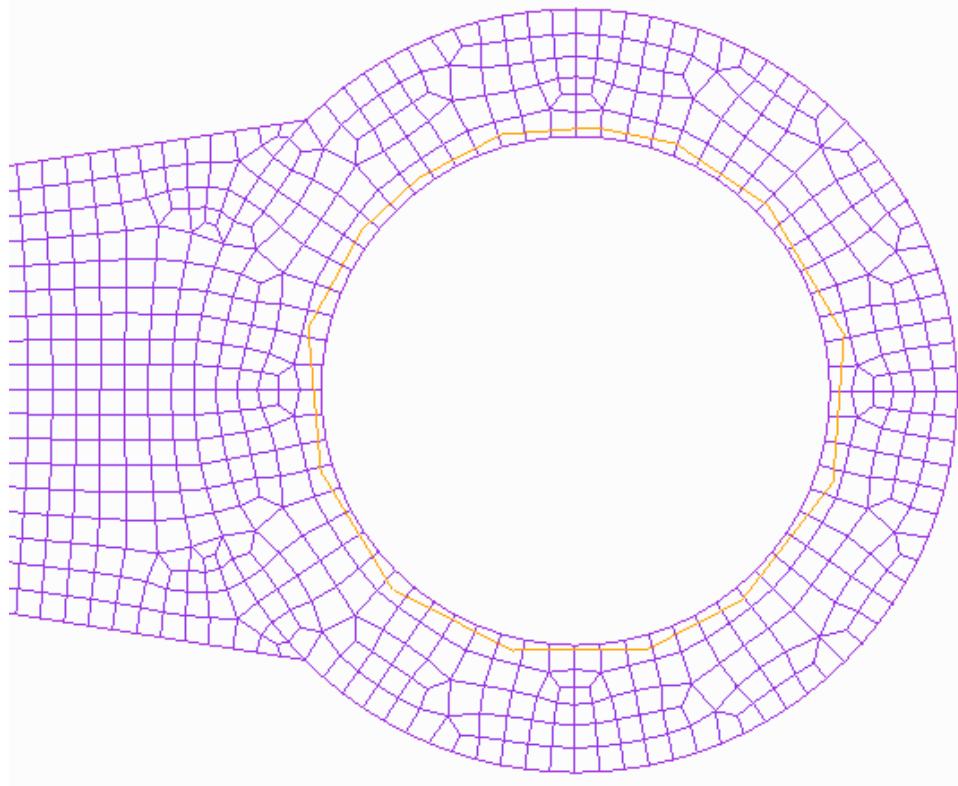
 button, then press "p" from key board (ensure that the mouse cursor is

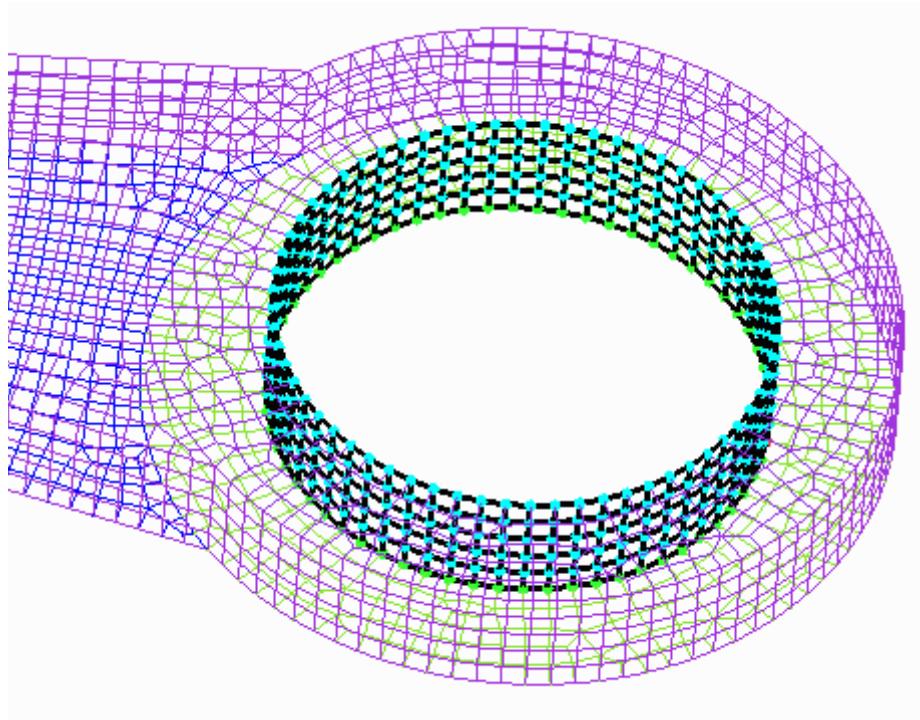
in the display window) or click the polygon selection icon  , then keep left mouse clicking on the screen to draw a polygon as shown below. Middle mouse click to finish the polygon and then middle mouse click again to finish all selection. Then press Apply to create the subset. You'll see the name **Subset0** appear under **Mesh > Subsets** in the Model Tree.

Create subset window



Elements
selection
by
polygon
and
elements
selected
for
Subset0

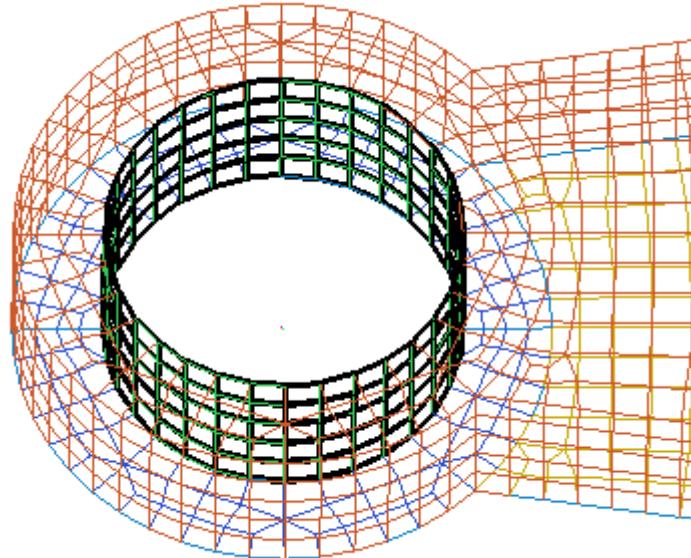




Subset1

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 shownhere. Press Apply to create Subset1.

Elements
selected for
Subset1



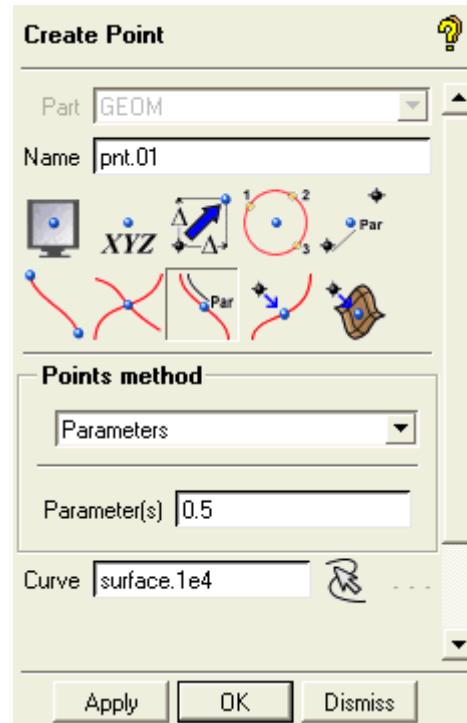
e) Cylindrical Coordinate System

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.

Select **Settings > Geometry Options** from the main menu and Toggle ON the Name new geometry. Then press Apply.

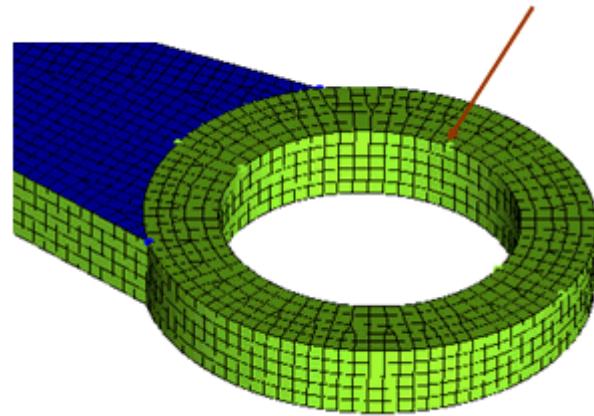
Note:- Create point by using Parameter along curve - Press Geometry>Create point >Parameter along a curve, Select Point Method>Parameters and Enter Parameter value as 0.5

Create Point



Make sure that the Points &Curves that are ON in the Model Tree.

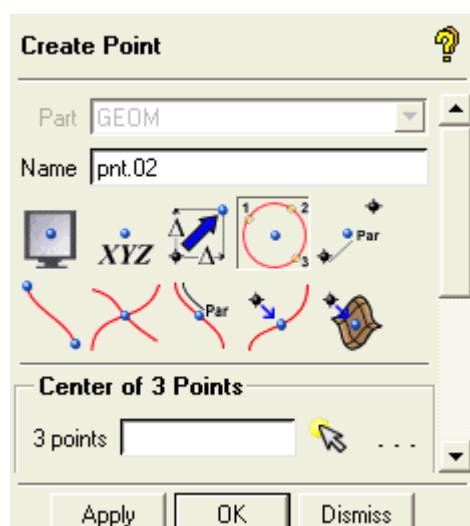
Select Curve on the Model



The Point will Appear in the GUI

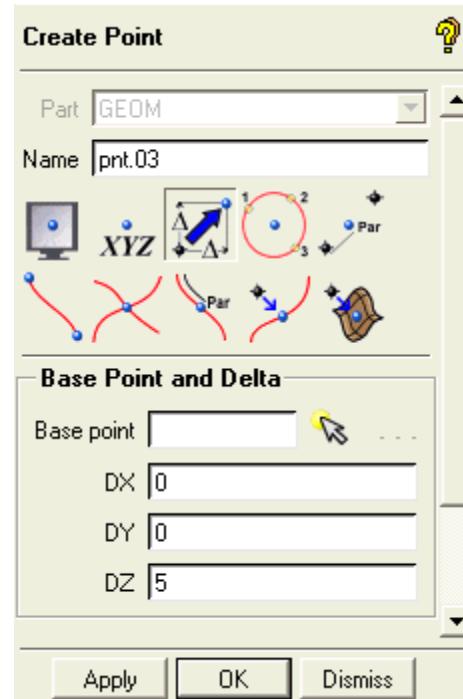
Geometry > Create point > Center of 3 Points. You should see the window shown below.

**Center point
menu**

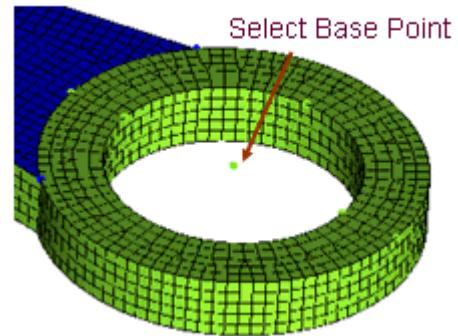


Select 3 points on the nodes at one side of the large hole as shown here. Then press Apply to create the center point.

After Create Pnt.02, Create pnt.03 as like below.

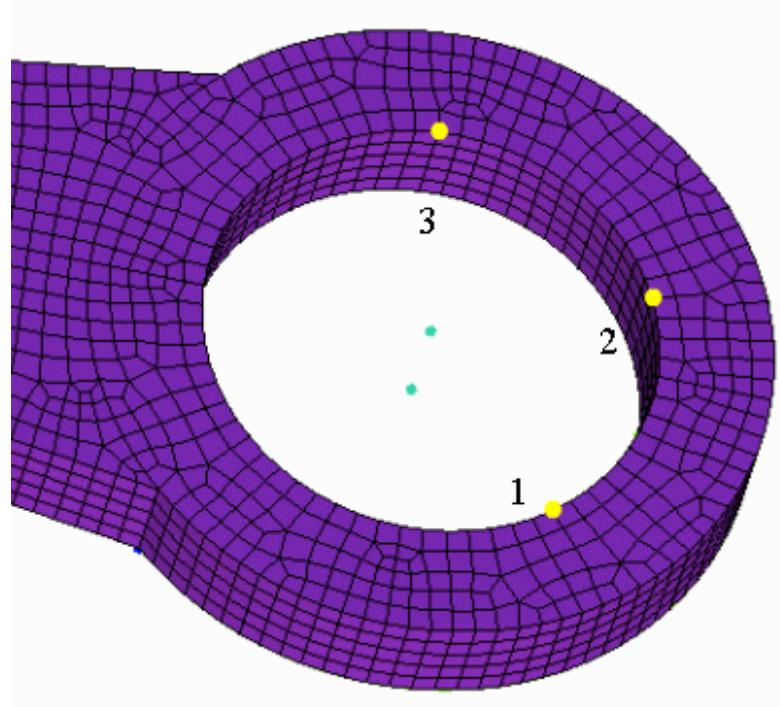


Select Base Point as indicate in the below Figure and Enter DZ as 5.



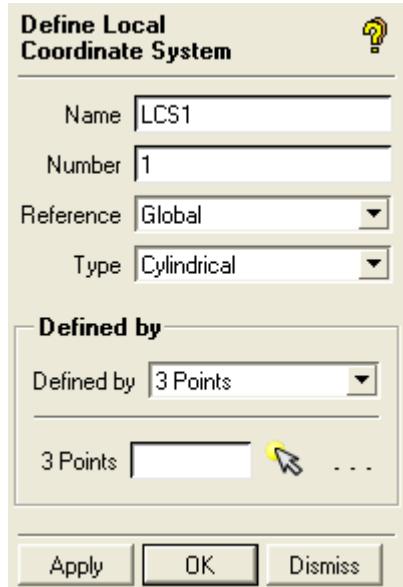
In Tree, Toggle ON only Points under Geometry and Volume under Mesh, the Model should appear as the figure below.

Three points



Press the Local Coordinate Systems button  from the main menu. You should see the window shown here.

**Define Local
Coordinate
System Window**

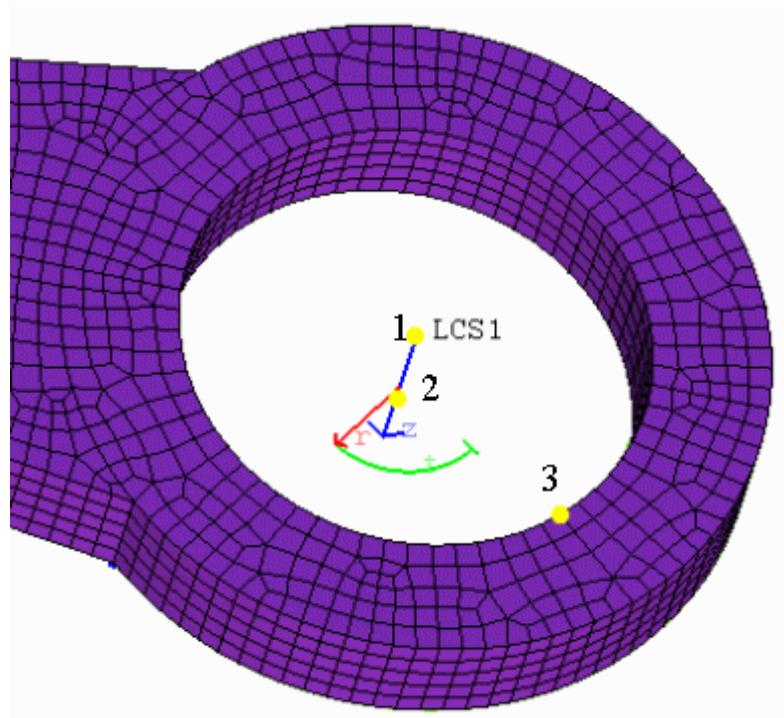


The Name should read **LCS1**.

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 here. Press the middle mouse button to accept, and then Apply.

Figure 6-77
Cylindrical coordinate system



After creating this, turn OFF Local Coord Systems > LCS1 and Points in the Display Tree.

f) Constraints and Loads

Constraints

Click on the Constraints > Create Constraints/Displacements > Displacement on Subset icon. 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.

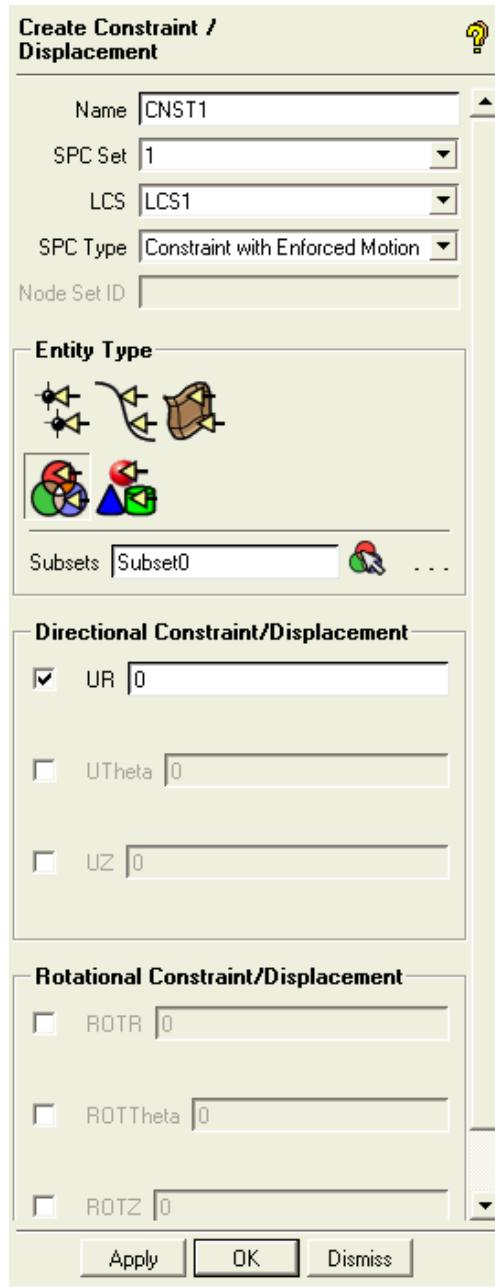
Click on the subset selection button and select **Subset0** as shown in the figure below.

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 6-78
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>  Create Constraints/Displacements >Displacement on Point  button. Enter the Name, FIXED_MOTION.

Make Sure that Point is Toggle ON under Mesh in Display Tree

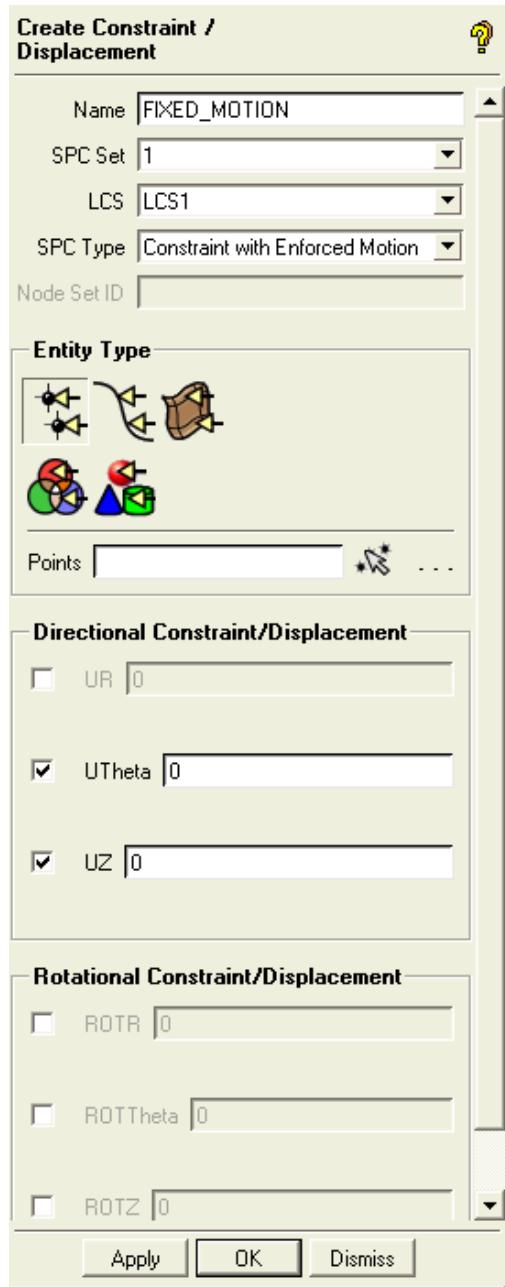
The **LCS** should be set as **LCS1**.

Check ON the **Directional Displacement**, **UTheta**, and **UZ**, and leave the numbers at “0.”

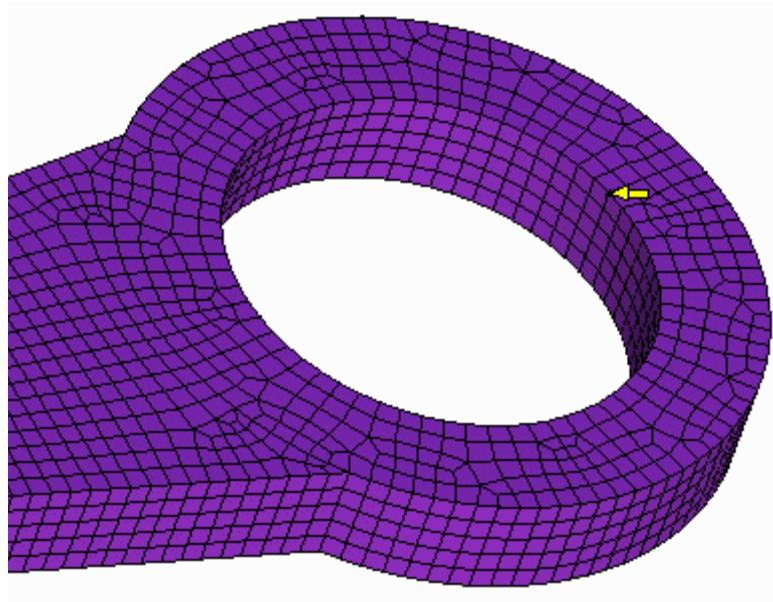
For the **Points**, select one node on the symmetry plane of the model that is on the large hole. See the figure below as a reference. Then press Apply.

Turn **OFF Displacements** from the Model Tree to simplify the display.

Create
Displacement on
Subset window



**Node to constrain
solid body motion**



Loads

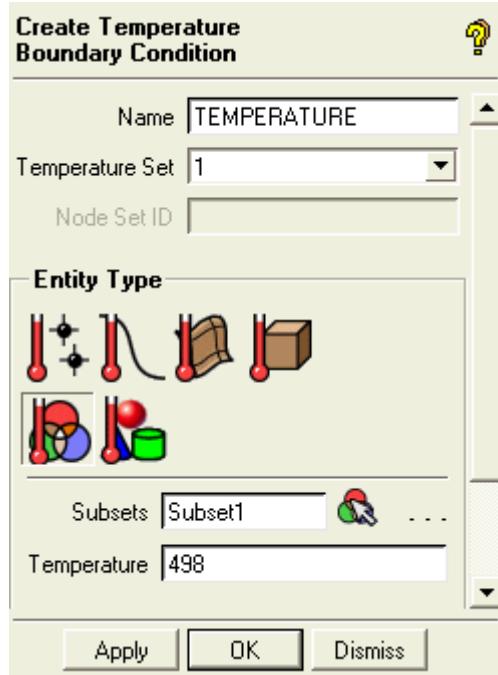
Click on Loads > Create Temperature Boundary Condition > Create Temperature Boundary Condition on Subset , which will bring up the Define Temperature Boundary Condition on Subsets window. Enter the Name as **TEMPERATURE**.

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 6-79
Define Temperature Boundary Condition on Subsets window



g) 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** from the Common Structural Solver and press **Apply**.

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.

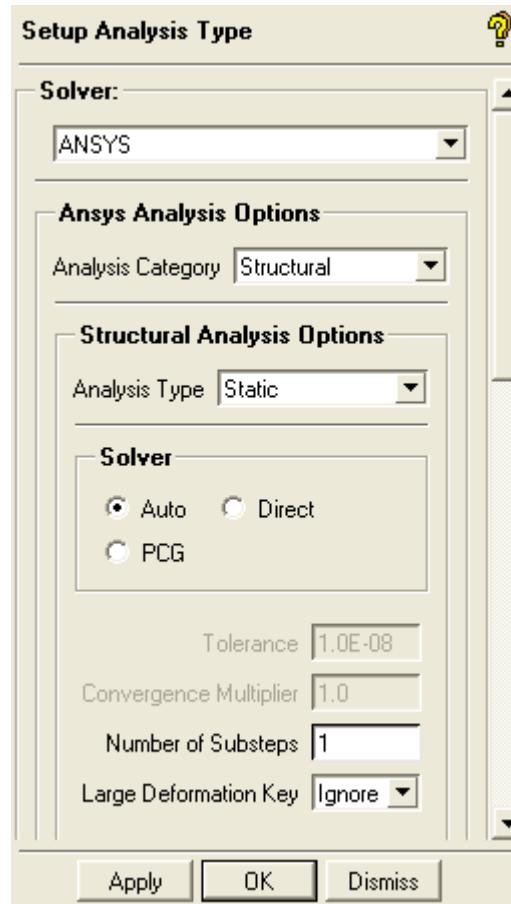
The solver should be set as **Ansys**.

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.

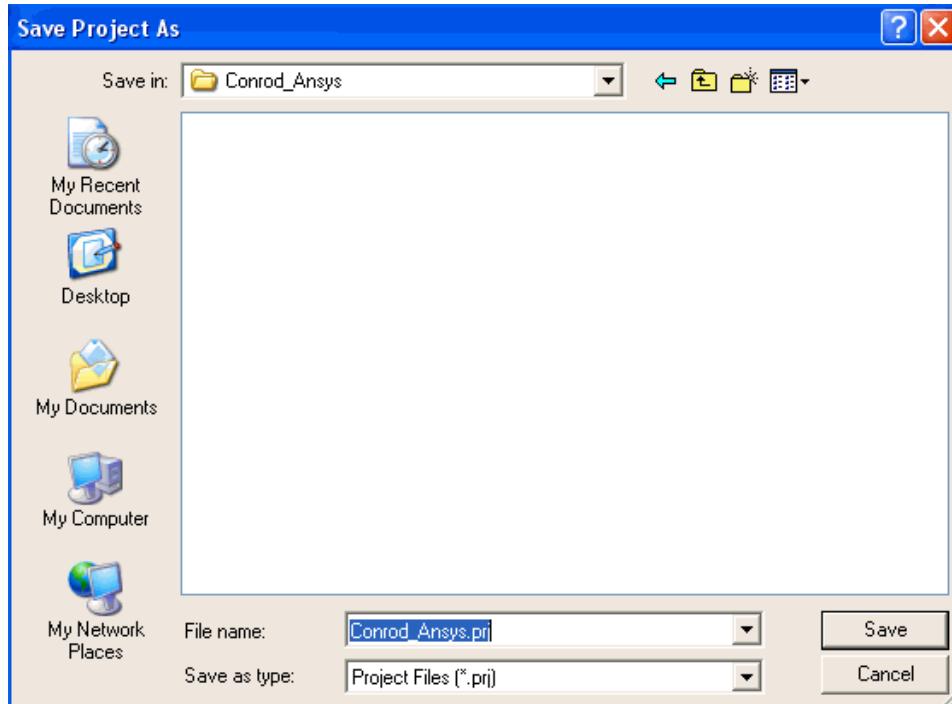
Setup Analysis Type window**Save Project**

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 below.

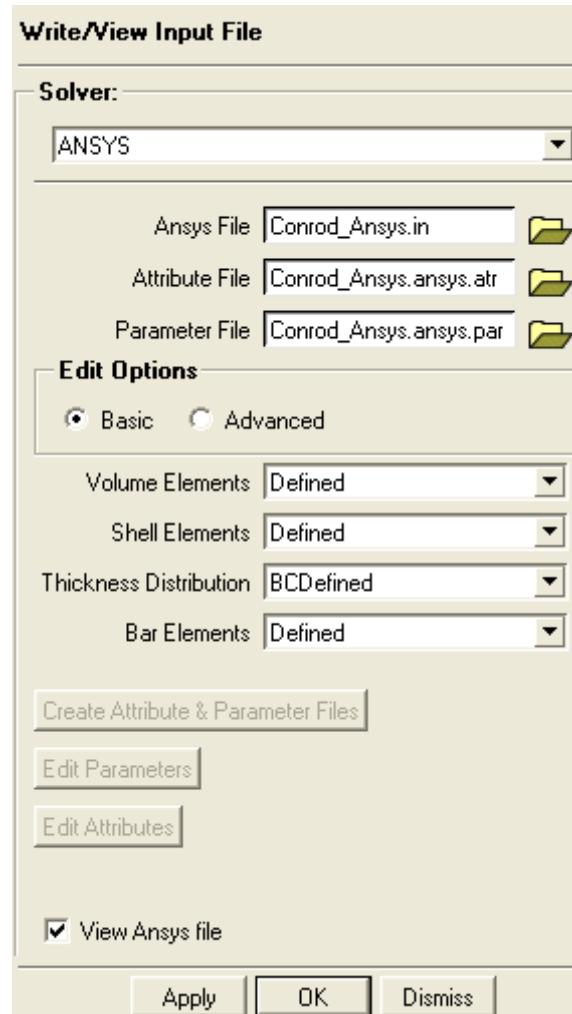
It will save six files: Geometry file (.tin), Mesh file (.uns), Attribute file (.atr), Parameter file (.par), boundary conditions file (.fbc), and the project settings file (.prj).

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. Press **Apply**.

**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.

h) Solution and Results

A Linear Static analysis will be performed on this model and the results will be visualized within ANSYS ICEM CFD.

Solving the problem

Click on the **Solve Options> Submit Solver Run**  button to display the window shown in the figure below.

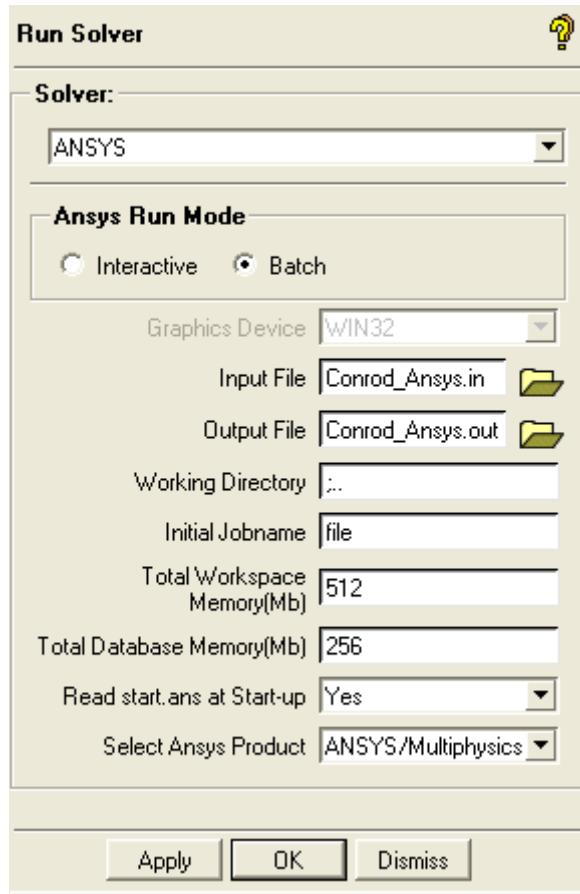
Select the **Batch** option and ensure that the **Input File** is set to the **Conrod_Ansys.in** file created in the above step.

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 ICEM CFD to be able to run Ansys.

Press **Apply** to run the Ansys solver in batch mode.

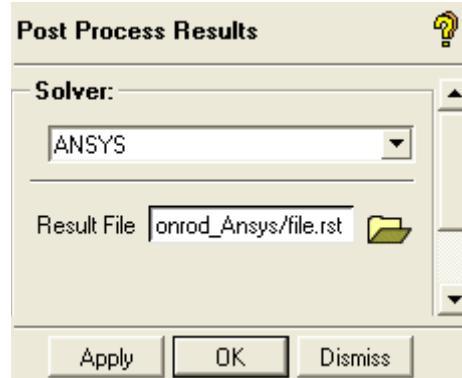
Figure 6-80
Run Solver window



Post Processing of Results

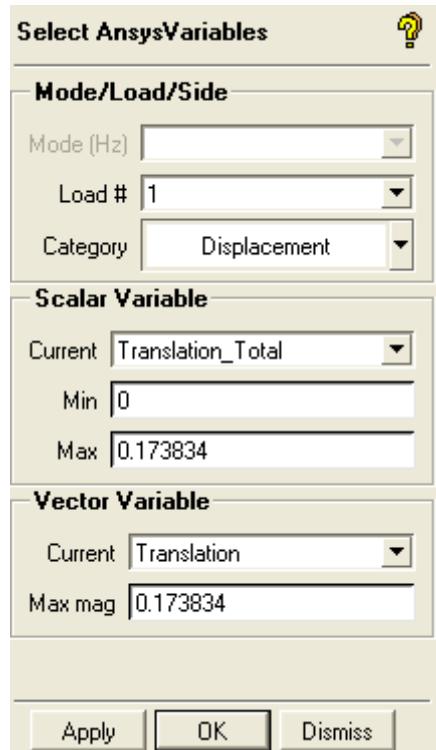
Click on the **Solve Options>Post Process Results**  button, which opens the **Post Process Results** window.

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.

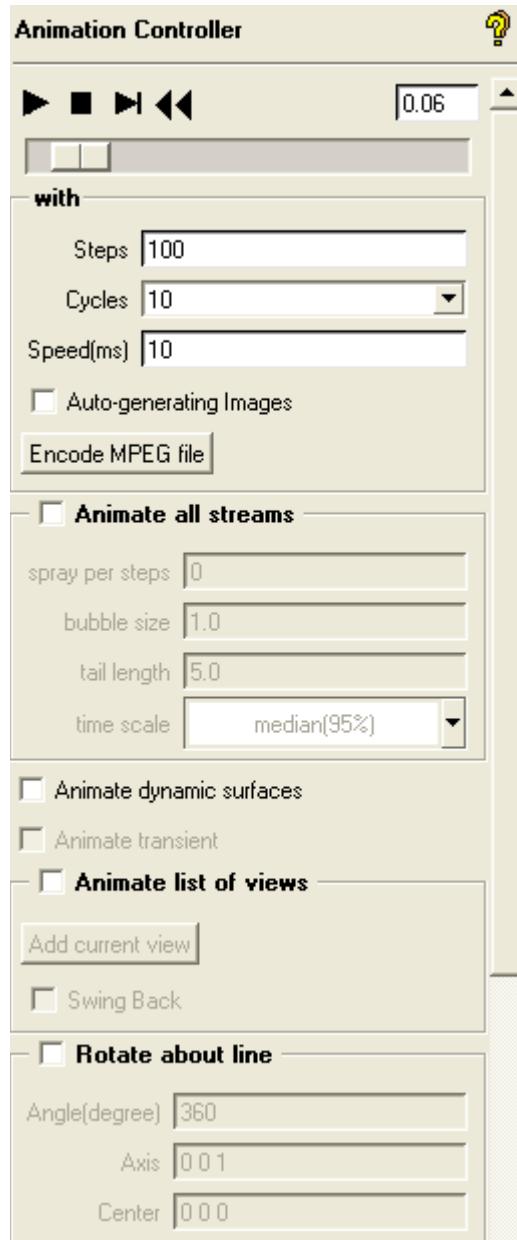
**Post Process Results
window**

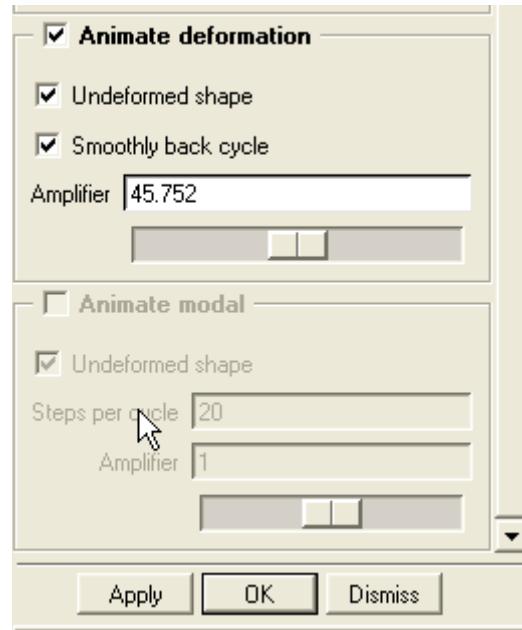
Click on **Variables** from the **Post-processing** menu bar.

To display the **Total Translation** Displacement, select **Load#** as **1** and **Category** as **Displacement** in the **Select AnsysVariables** window. This is the default.

**Ansys Variables
window**

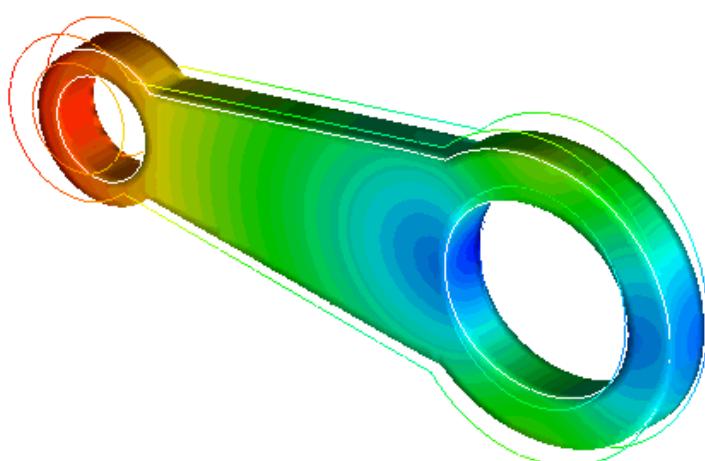
Click on Control All Animations from the Post-processing menu bar.
Select **Animate**. The deformation is shown below.

**Animation Setup
and Controller
window**



Animated model of Total Translation

Translation_Total
(Load #1)

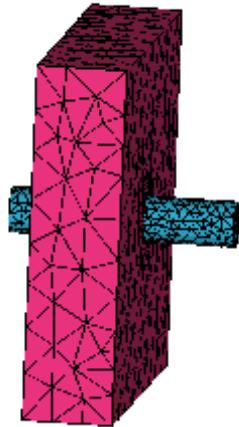


Finally, select **File > Results > Close Result** to quit the post processor.

6.2.4: 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 here.

Figure 6-81
Pin Block
Geometry



a) Summary of Steps

- 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

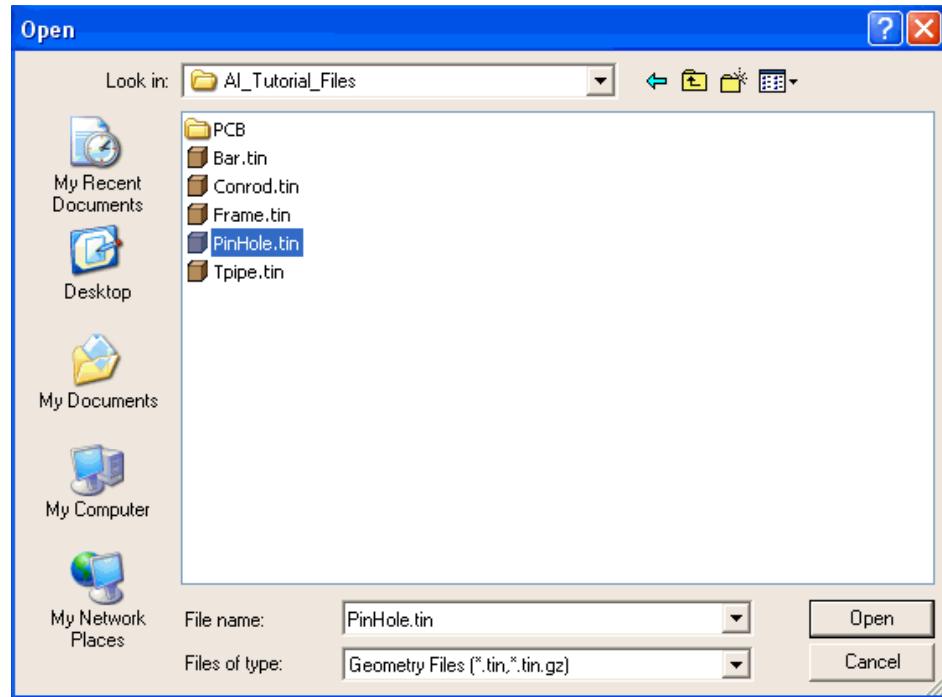
Displacement
Contact
Solver setup
 Setup Ansys Run
 Save Project
 Write Ansys Input File
Solution and Results
 Solving the Problem
 Post processing of Results

b) Geometry Editing

Launch AI*Environment

The input files for this tutorial can be found in the Ansys installation directory, under
..../v110/docu/Tutorials/AI_Tutorial_Files. Copy and open the **PinHole.tin** file in
your working directory.

Open
Geometry
File
window

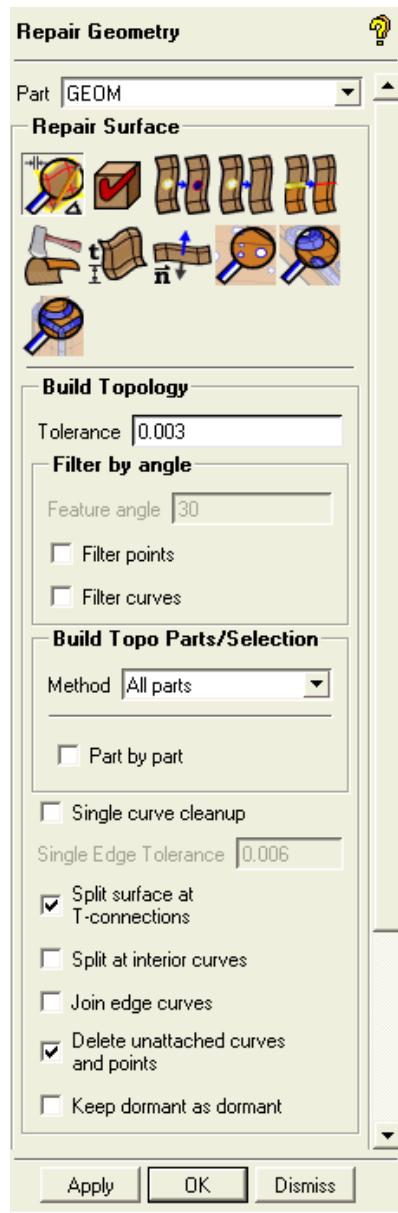


Repair

From Main Menu, Settings>Geometry Options>Toggle On the Inherited

Click on the **Geometry>Repair Geometry** button, which will bring up the **Repair Geometry** window. The default **Tolerance** of 0.003 should work fine here. Then press Apply.

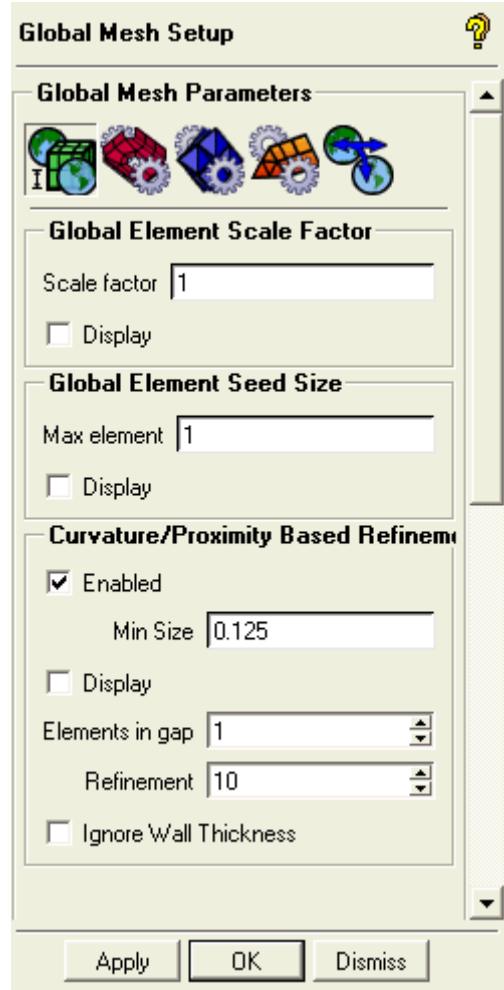
Figure 6-82
Repair Geometry
window



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 below and press **Apply**.

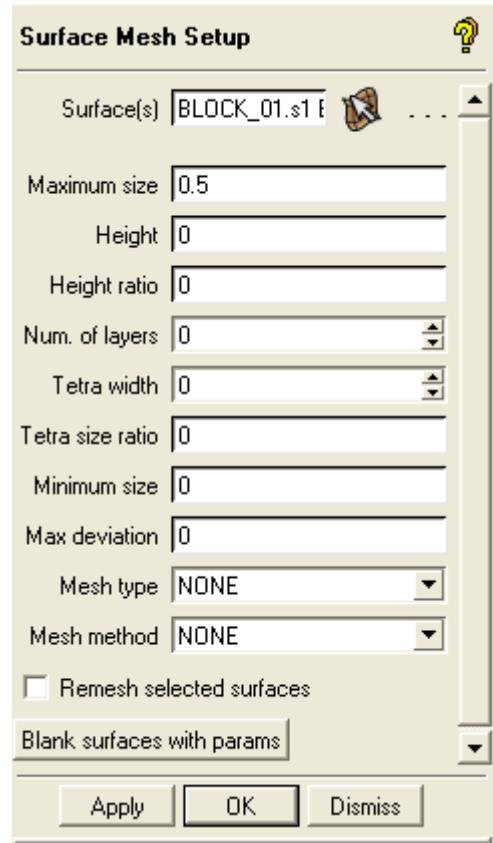
Figure 6-83
Global Mesh Size window



Select the **Mesh > Set Surface Mesh Size** button, which brings up the **Surface Mesh Size** window.

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 here and press **Apply**.

Figure 6-84
Surface Mesh Size window

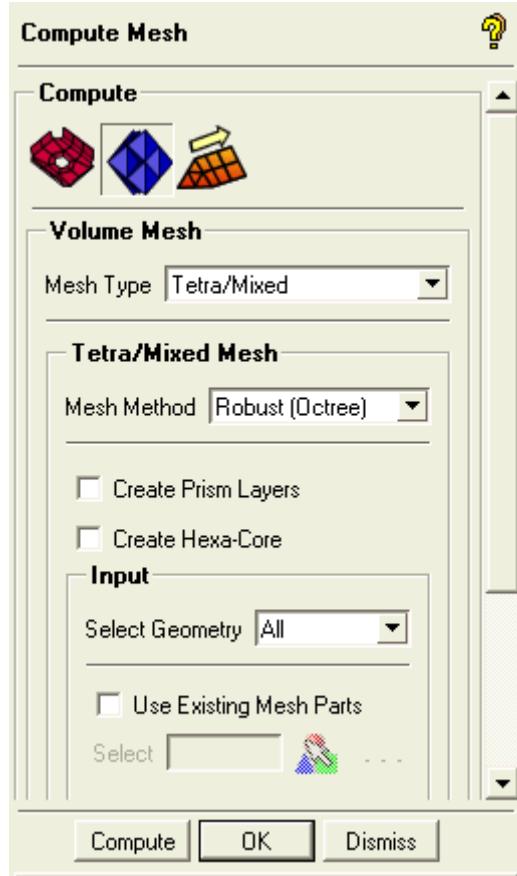


c) Meshing and Internal Wall

Tetra Meshing

Select the Mesh > Compute Mesh button. It opens the Compute Mesh window. Ensure that the Mesh type is set to Tetra/Mixed.

Figure 6-85
Mesh Volume

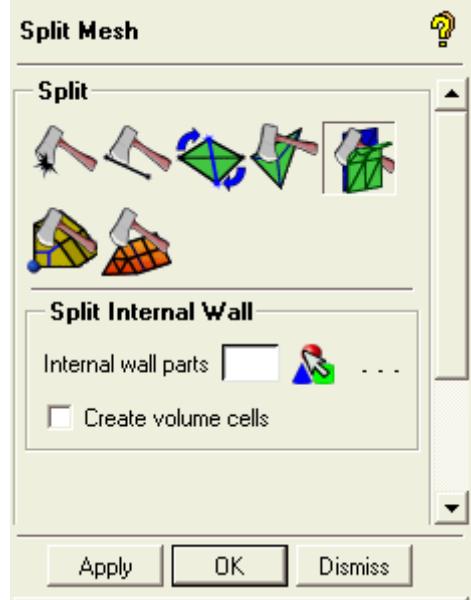


For this tutorial, don't change anything here. Leave the default parameters as they are and press Compute to start meshing.

Creating Internal wall

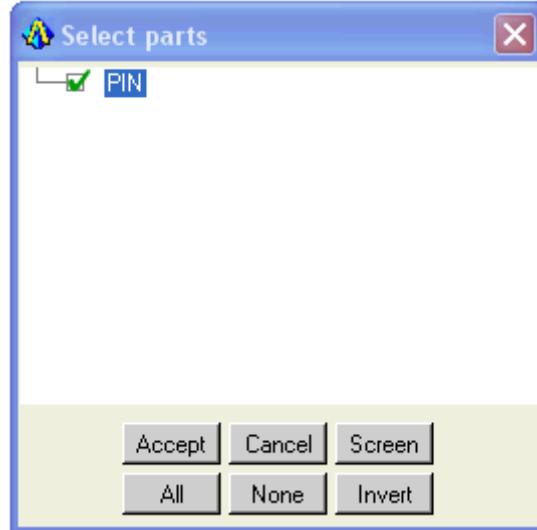
Click on the Edit Mesh > Split Mesh > Split Internal Wall button. It opens the Split Mesh window.

Figure 6-86
Split Mesh window



Click on the part selection  button. A window with the current parts in the model will appear. Select the part, **PIN** and press Accept to close the **Select parts** window.

Figure 6-87
Internal Wall Parts
Selection window

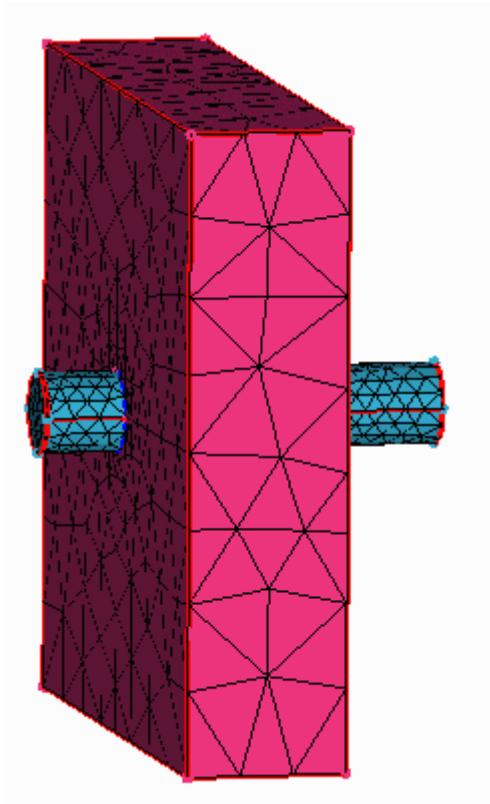


Now, Click Apply to split the internal wall.

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 and 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 here.

Figure 6-88
Mesh in Solid and
Wire mode



d) Material and Element Properties

Definition of the Material

Select **Properties> Create Material Property** .

Define the Material Name as MAT1.

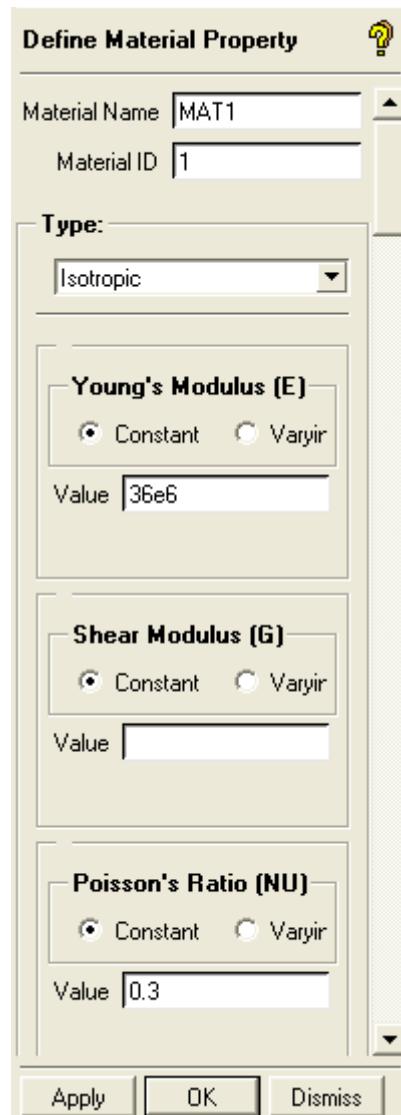
The **Material ID** can be left as 1,

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. Then press Apply.

**Define Material Property
window****Element Properties**

Select **Properties> Define 3D Element Properties** 

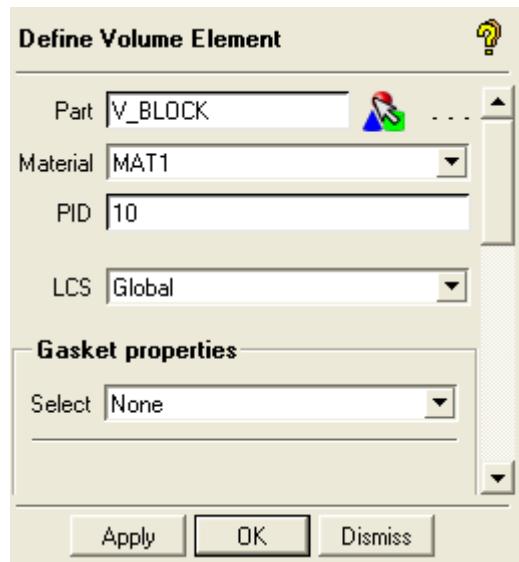
Select the **Part** as **V_BLOCK**.

Select the **Material** as **MAT1**.

Set the **PID** to **10**.

The **Define Volume Element** window should look like the figure below when you are finished. Then press **Apply**.

Figure 6-89
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>Create Constraints/Displacements>Create Constraints/Displacement on Surface  button, which will give the window as presented below.

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. Toggle **ON** all options of X, Y and Z for the Directional displacement. Press Apply. Switched **OFF** the **Displacements** from the Model Tree after the constraint has been applied.

Figure 6-90
Create
Displacement on
Surface window

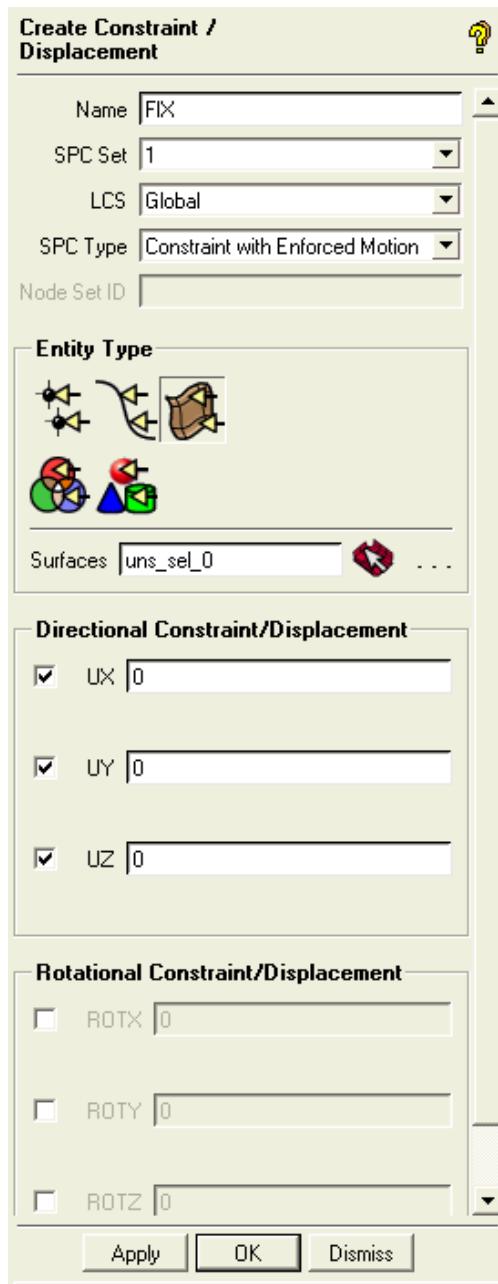
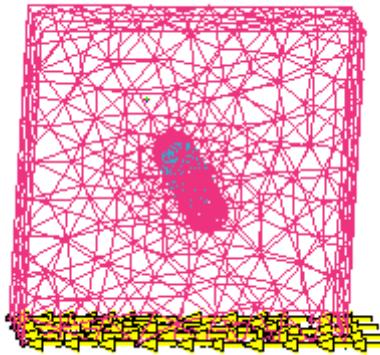


Figure 6-91
Constraint
Display



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 below. Press **Apply**.

Switched **OFF Displacements** in the Model Tree.

Figure 6-92
Create
Displacement on
Surface window

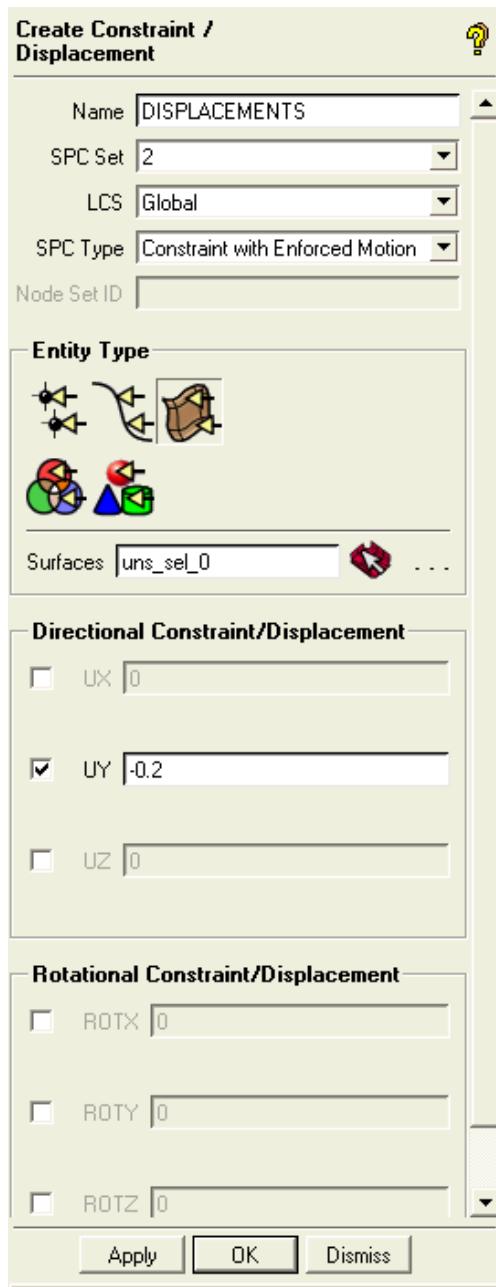
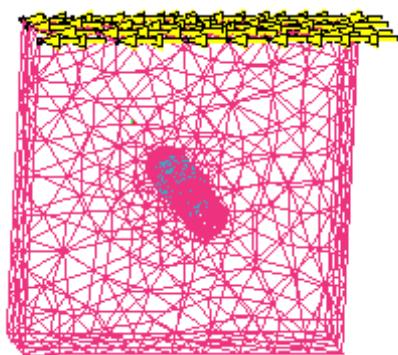
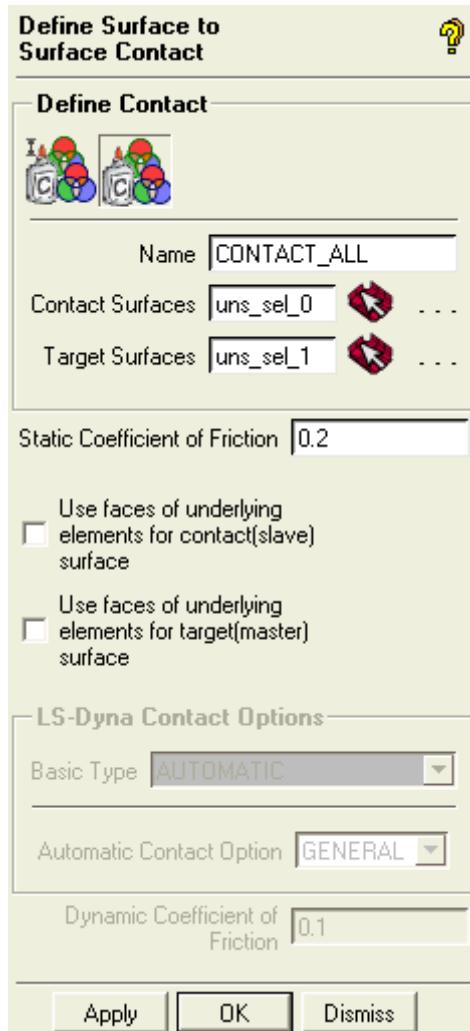


Figure 6-93
Displacement
Display



Contact

Click on Constraints>Define Contact >Manual Definition (the second one).
Enter the Name as **CONTACT_ALL**.

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 Common Structural Solver by using dropdown arrow. Press **Apply**.

Click on the **Solve Options> Setup Analysis type**  button. This will bring up the **Setup Analysis Type** window as shown below.

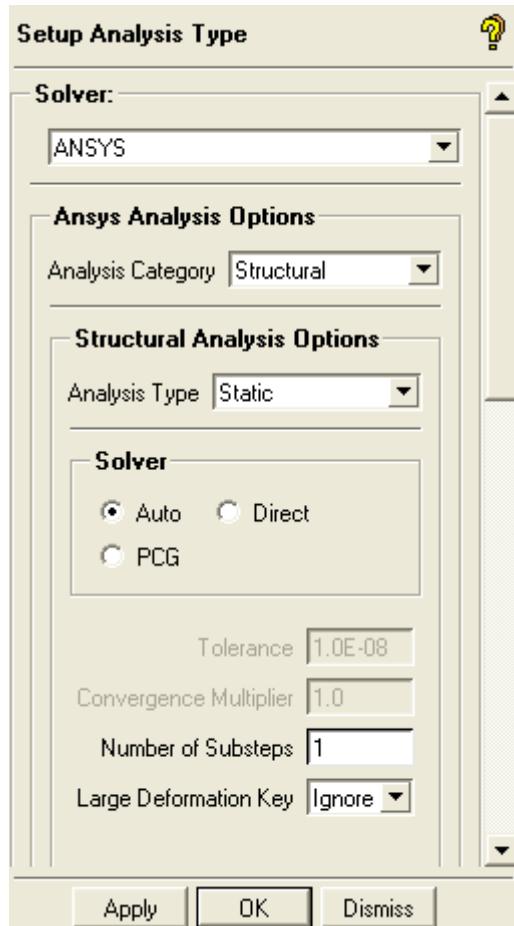
The solver should read as ANSYS.

Select the Analysis Category as Structural.

Select the Analysis Type as Static.

Keep all other options as default, and press **Apply** to complete the setup.

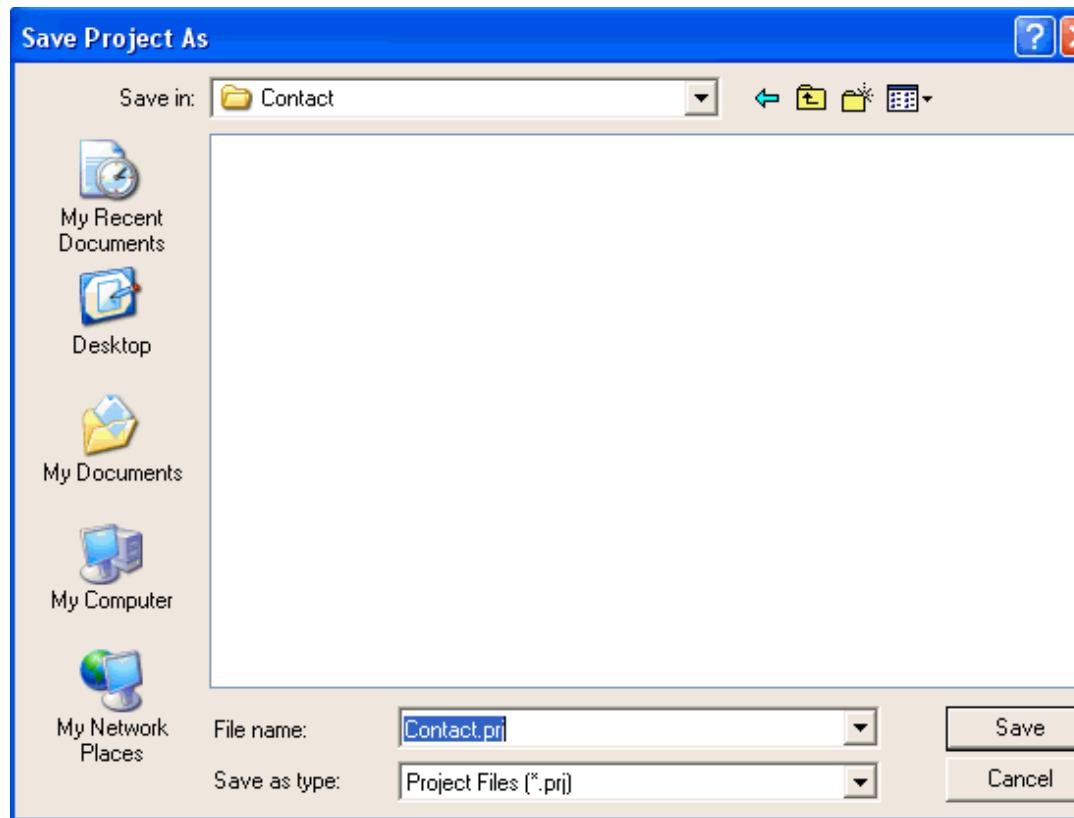
Setup Analysis Type window



Save Project

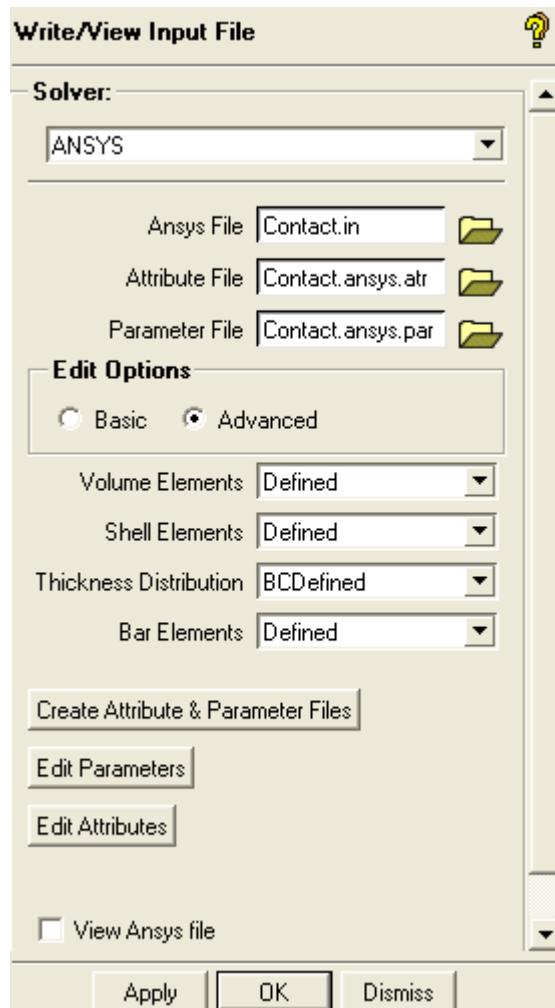
Select **File > Save Project As**, and in the new window press the icon to create a new folder. Name this folder **Contact** and enter into it. Then enter the file name, **Contact**.

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 and Parameter Files**.

**Ansys Input File
window**

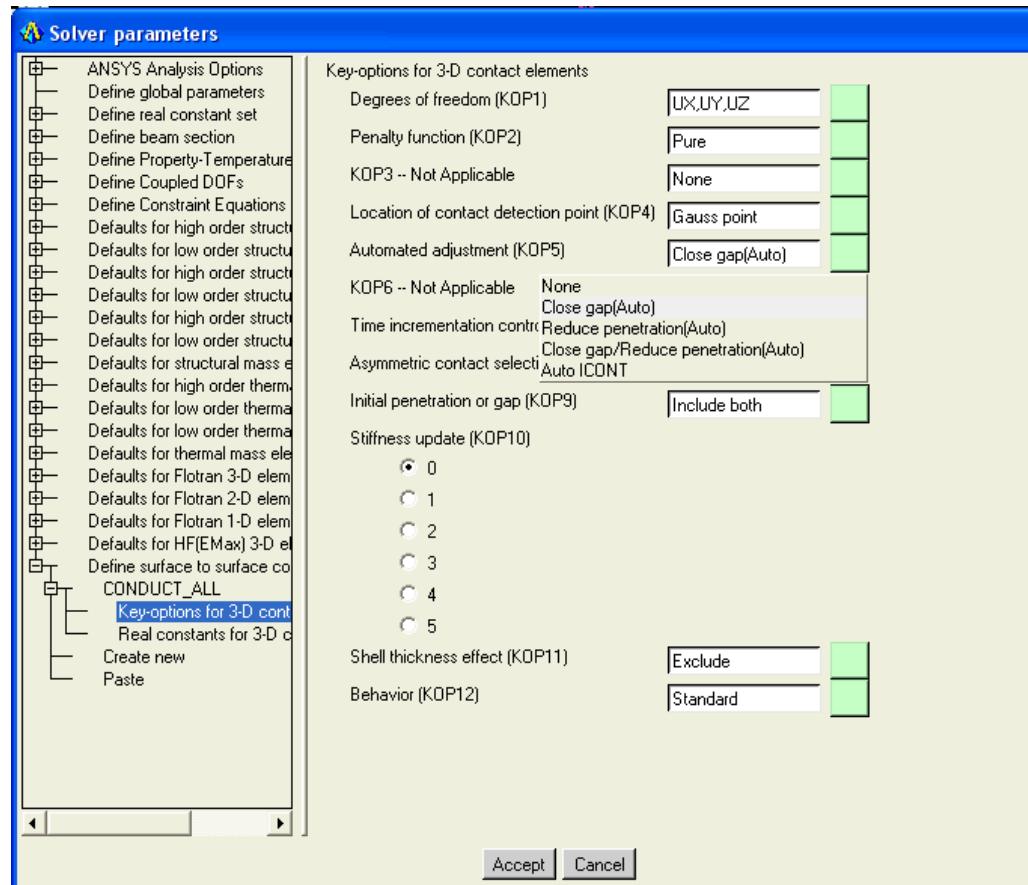
Click on the **Edit Parameters** button, which will open the Solver Parameters window.

Expand under Define surface-to-surface contact configuration, then expand under the part name CONTACT_ALL, and select on the words, Key-options for 3-D contact elements.

Change the option for Automated Adjustment (KOP5) to Close gap (Auto).

Press Accept to save these changes in the parameter file (.par file) and close the Solver Parameters window.

**Figure 6-94
Solver
Parameters
window**



Also, switch ON the View Ansys file option in the Write/View Input File window 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 ICEM CFD post processor.

Solving the problem

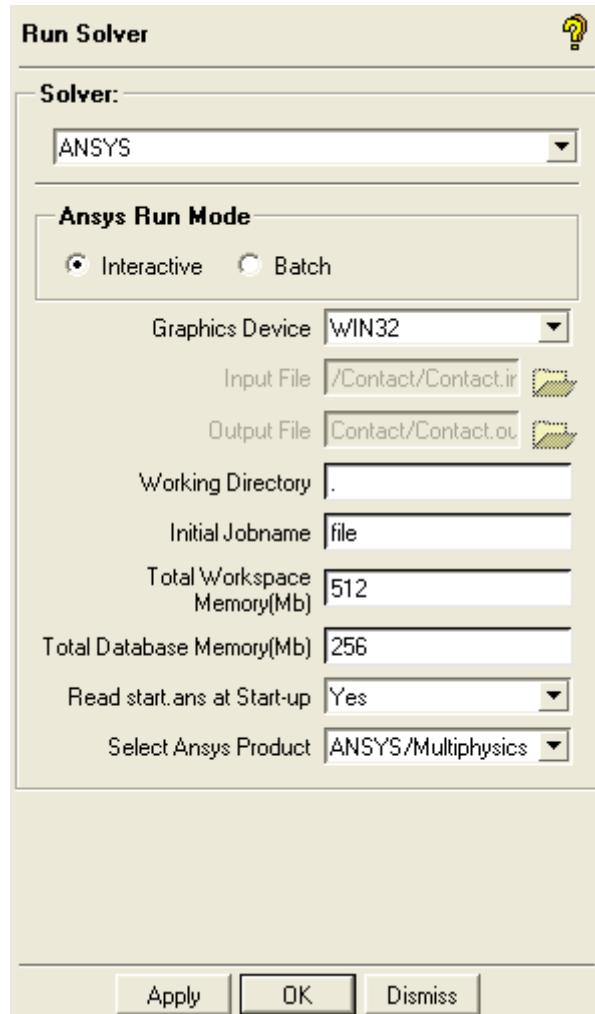
Click on the **Solve Options> Submit Solver Run**  button, which should display the **Run Solver** window as shown below.

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 ICEM CFD to be able to run Ansys.

Press Apply to start the Ansys solver in Interactive mode.

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.

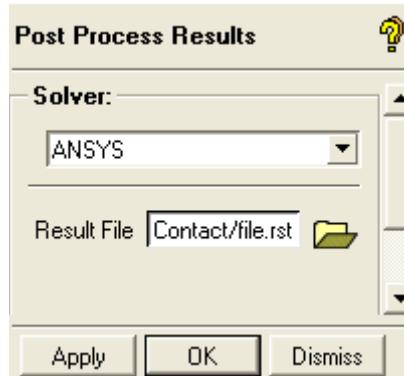
The user can work in Ansys after this, if desired, or exit out of Ansys and Post Process the results in ICEM CFD.

Post-Processing of Results

Click on the **Solve Options > Post Process Results**  button, which opens the **Post Process Results** window given below.

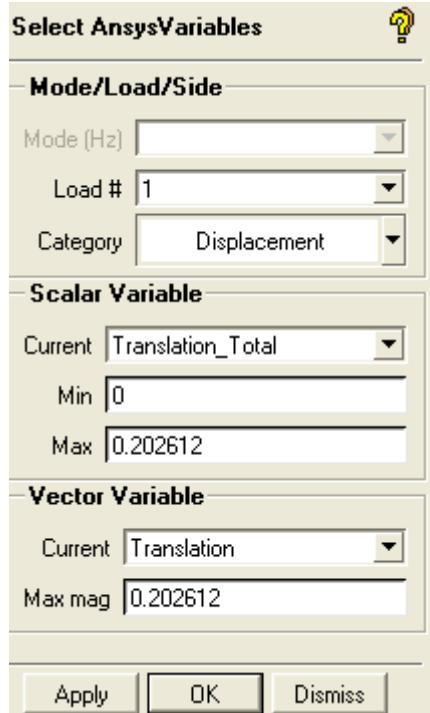
Select the folder button to browse for the file, **file.rst**, where “file” is the **Initial Jobname** specified earlier. Press **Apply** to launch the Visual3p Post processor with the Ansys result file.

Post Process window



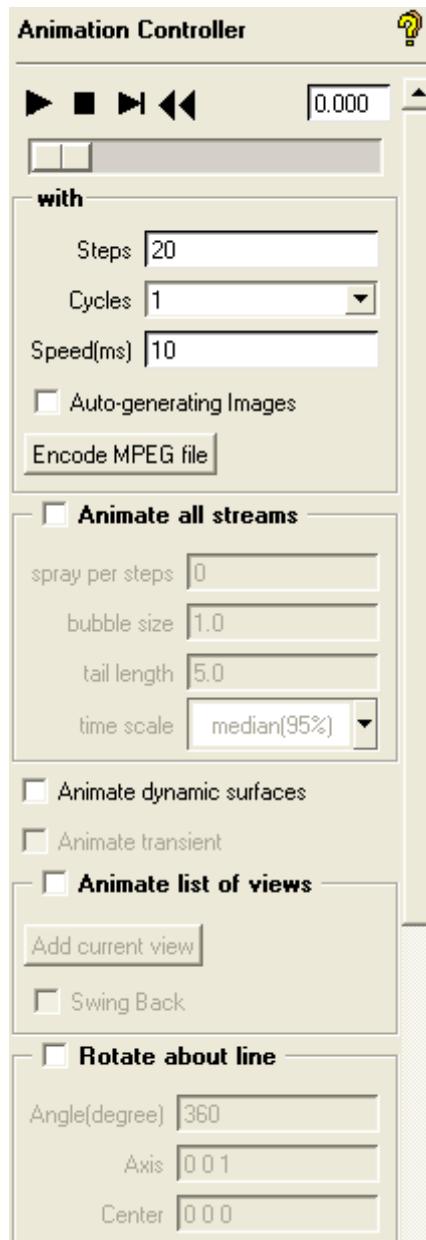
Click on  **Variables** from the **Post-processing** menu bar.

To display the Total Translation Displacement, select the **Load#** as **1** and **Category** as **Displacement** in the **Select AnsysVariables** window as shown here. This should be the default.

**Ansys Variables
window**

Click on Control All Animations from the Post-processing menu bar.

Select **Animate** to see the deformation. The deformed shape is shown below.

Animation Setup and Controller window

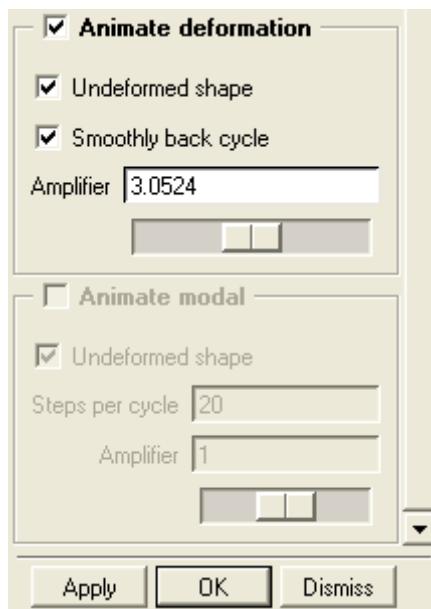
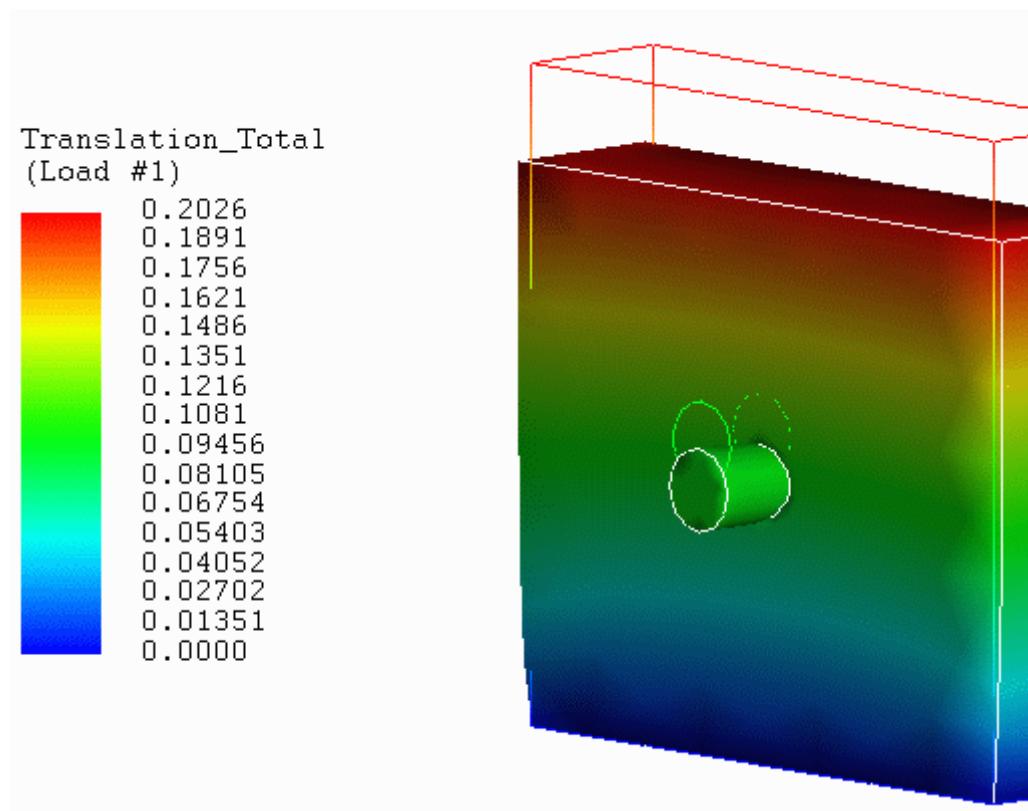


Figure 6-95
Animated
model of
Total
Translation

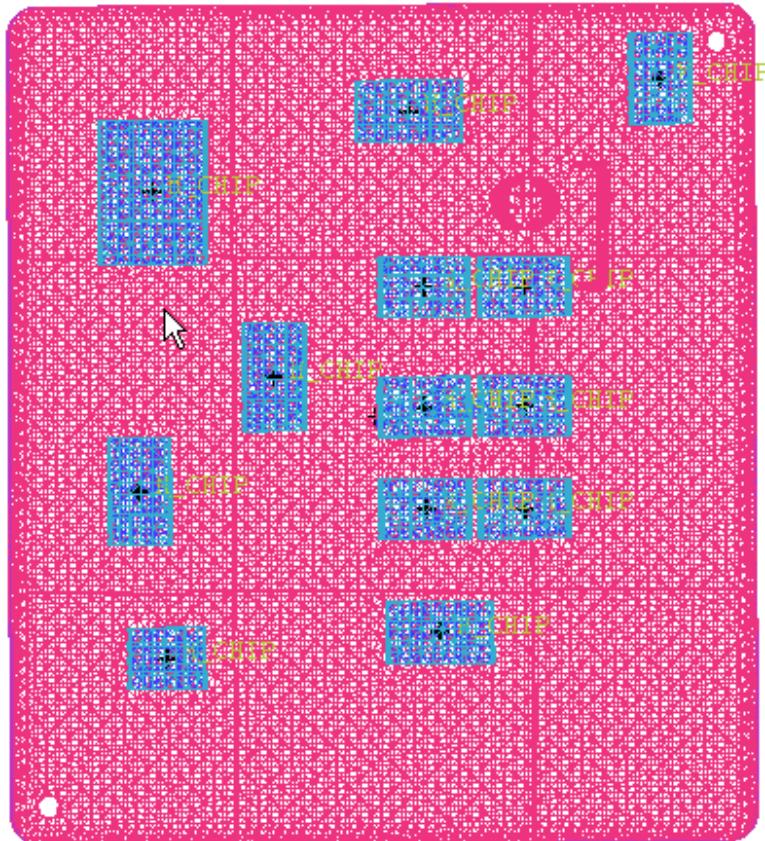


Finally, select **File > Results > Close Result** to quit the post processor

6.2.5: PCB-Thermal Analysis

Overview

In this tutorial, it is shown that how to write Ansys input for the previous PCB Tutorial in Structural Meshing Tutorial and then do thermal analysis in Ansys using AI*Environment.



a) Summary of steps

Open the project

Defining the material properties

Setting the solver parameters

Writing the input file

Solution and results

Saving the project

Proceed the Tutorial as the Continue Before PCB:Structural Meshing Tutorials.

b) Open Project File

Open the Project File > PCB.prj as you Earlier Completed in the Structural Meshing Tutorials

c) 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

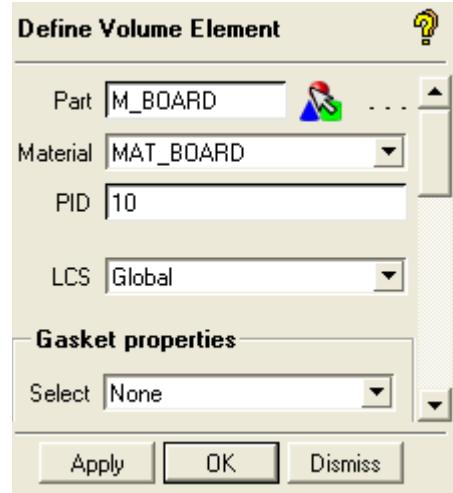
After defining material property for board material, **MAT_BOARD**, we have to define material property for **MAT_CHIP**.

Enter Material name as **MAT_CHIP**, Material ID as **1**, Type as **Isotropic**, Young modulus as **70000**, Poisson'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 the figure below.

Figure 6-96
Define volume element window



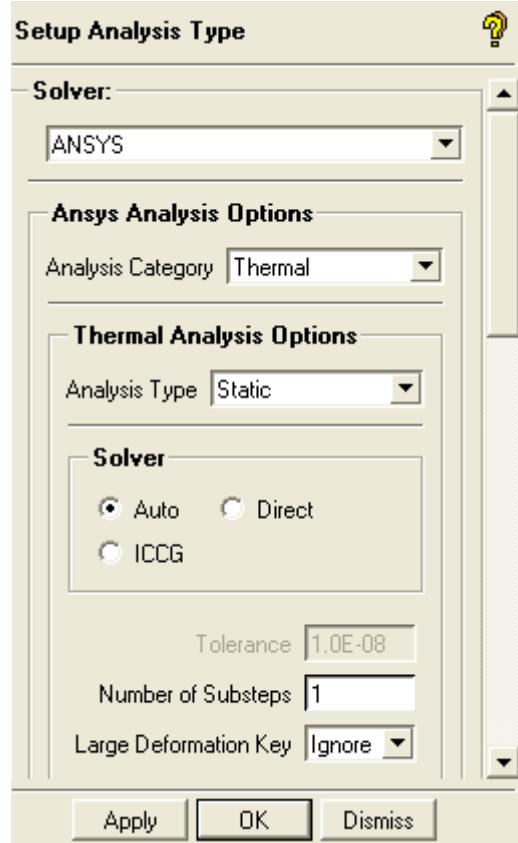
Now after defining volume element properties for **M_BOARD**, we have to define properties to the **M_CHIP** also. So in the same window, enter Part as **M_CHIP**, Material as **MAT_BOARD**, PID as 11 and with default option press Apply.

d) 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 here.

Setup Analysis Type window

After entering the parameters, press Apply.

Save Project

Select **File > Save Project As**, and in the new window press the icon to create a new folder. Name this folder **PCB_Thermal** and enter into it. Then enter the file name, **PCB_Thermal.prj**

e) 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.

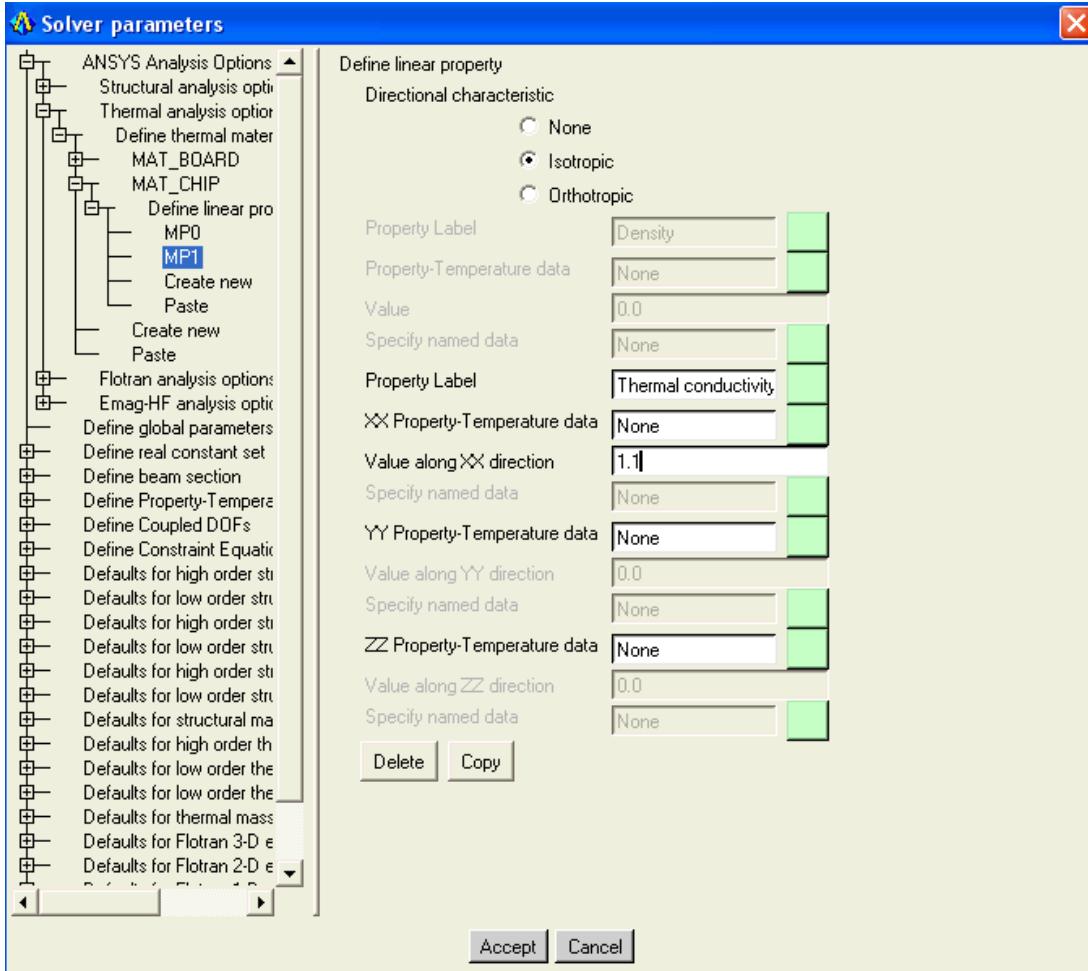
Now click on Edit parameters, which will invoke a solver parameters window.

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.

Solver parameters window for MAT_CHIP

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	1085
------------------------	--	------



Do the same operation to define the thermal conductivity to the MAT_BOARD material.

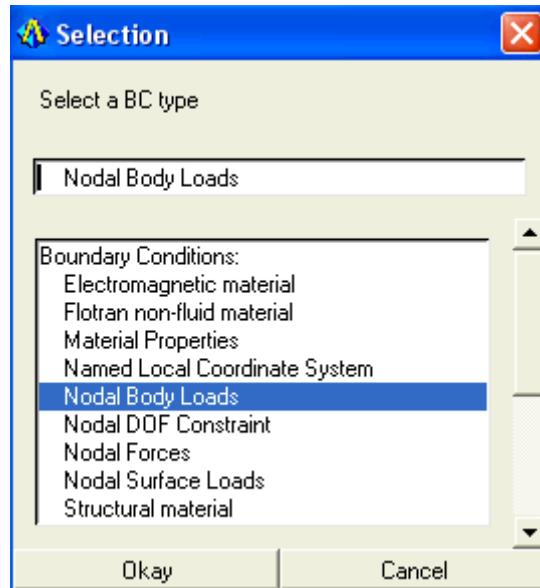
Then click on Accept in solver parameters window.

Now press Edit attributes which will invoke a Boundary Conditions window.

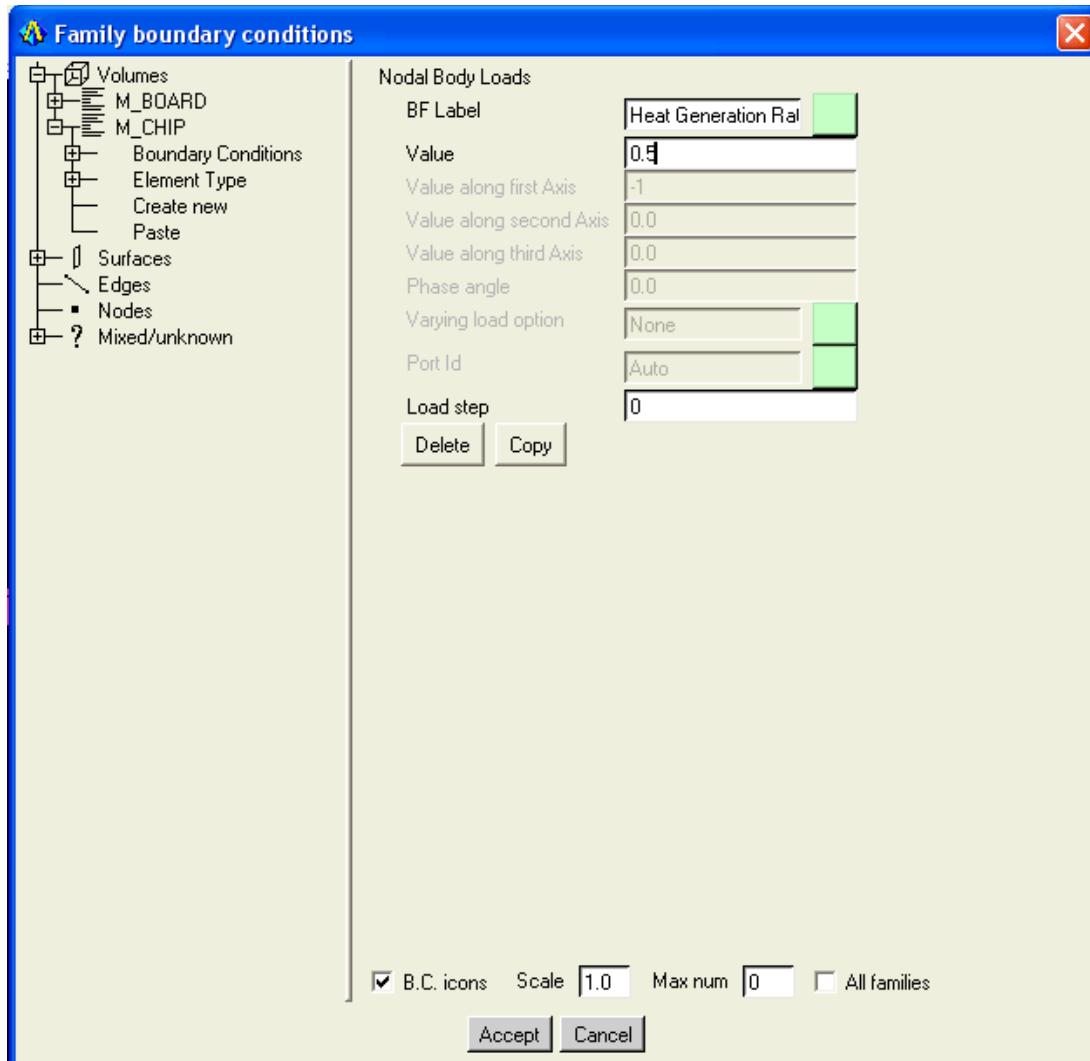
Please follow the images to define the boundary condition for M_CHIP and BOARD_SURF part.

Click on Volumes > M_CHIP > Create New option to select the boundary condition type. Select the Nodal Body Loads as shown here.

Boundary condition selection window for MAT_CHIP

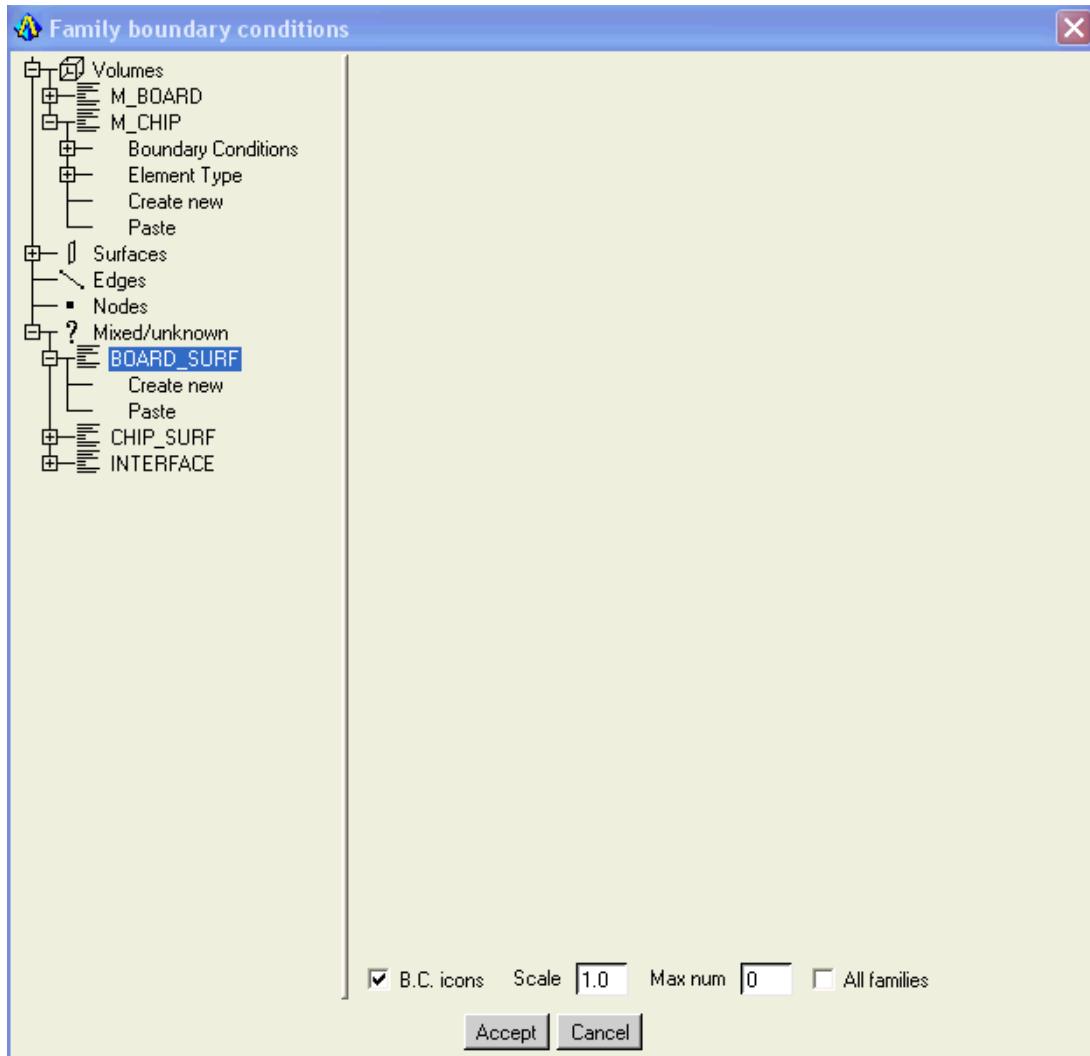


Family Boundary Condition window for M_CHIP

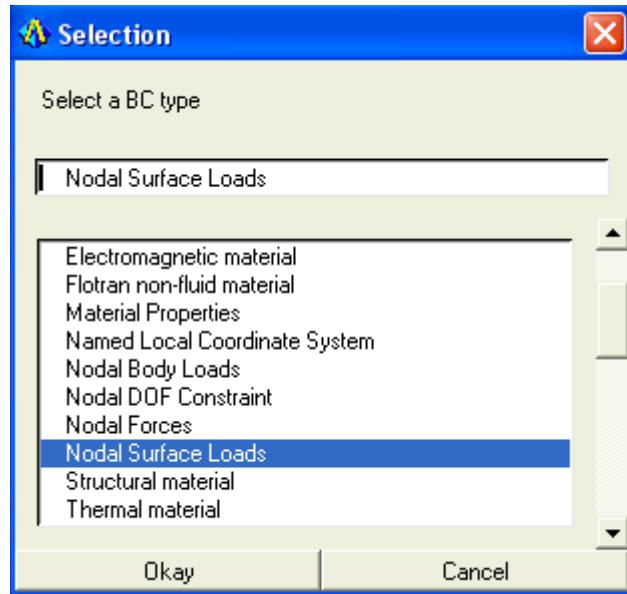


In the same window, now got to Mixed/Unknown > BOARD_SURF > Create new and select Nodal Surface Loads type from the list.

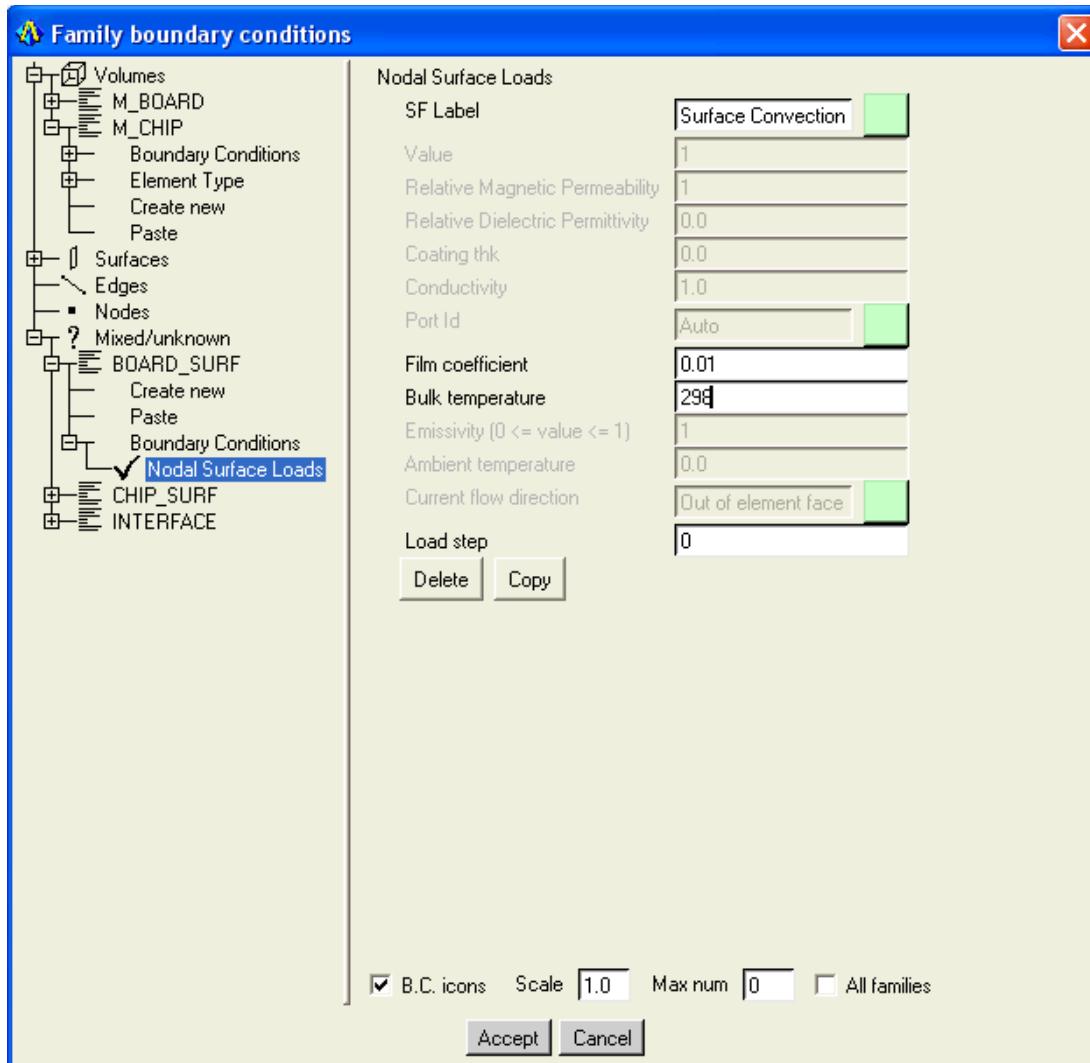
Family Boundary Condition window for BOARD_SURF



**BC selection window for
BOARD_SURF**



Bulk Temperature defining for BOARD_SURF



Press **Accept** in Edit attributes window.

Now in the Write/view input file window, select None for Ignore BAR elements, keep View Ansys file ON and other option as default. Press Apply.

f) 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 Menubar to start the Ansys as shown below.

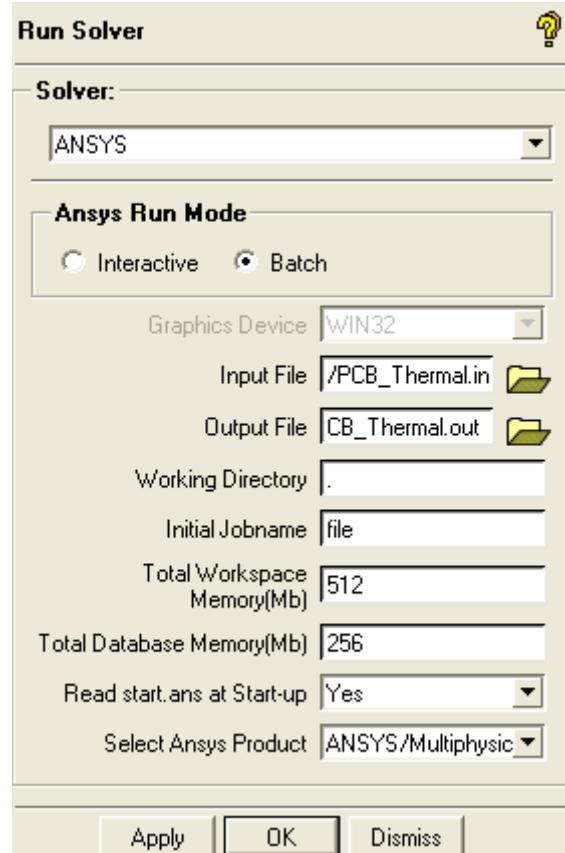
Press Apply in Run Solver window.

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 6-97
Run Solver window



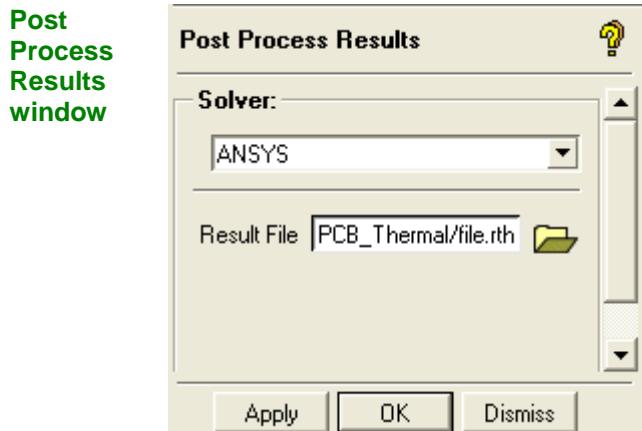
As it launches the Ansys Interactive window, load the ansys.in file in Ansys for the analysis through File > Read Input From option. User can check out the convergence graph in the Ansys window during the process.

After solution is done, user can do post processing in Visual 3p.

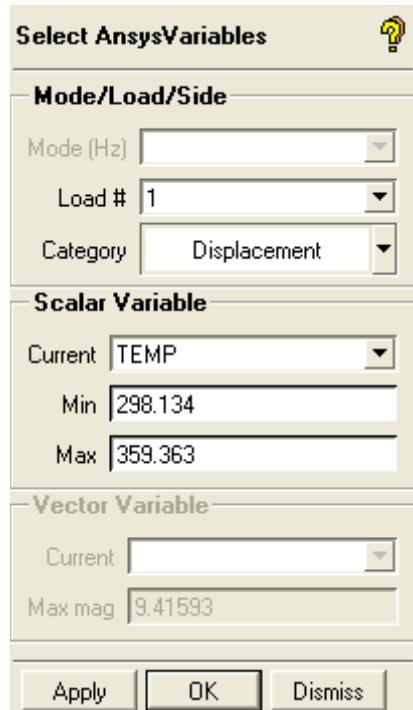
Post-Processing of Results

Click on  (Post Process Result) icon from the Solve Options Menubar, which opens Post Process Results window given below.

Supply Ansys Result file **file.rth** in this window and press Apply to launch Visual3p Post processor with Ansys result file.

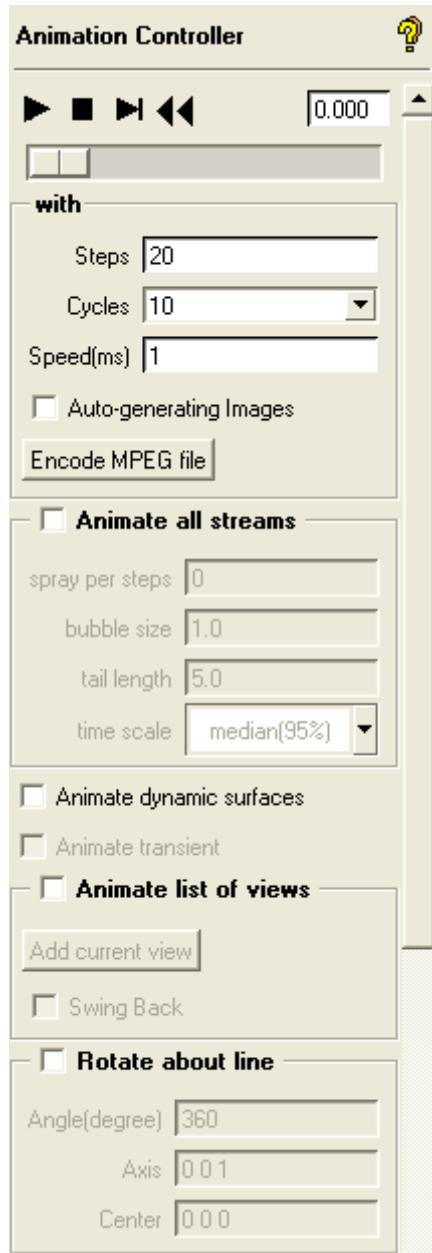


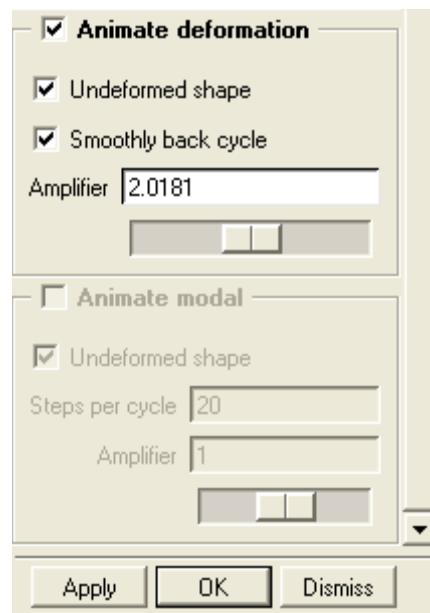
Select **variables**  from **Post processing** tab menu bar, to display the Total Translation Displacement, select Load as **1** and Category as **Displacement** in Ansys Variables window as shown here.

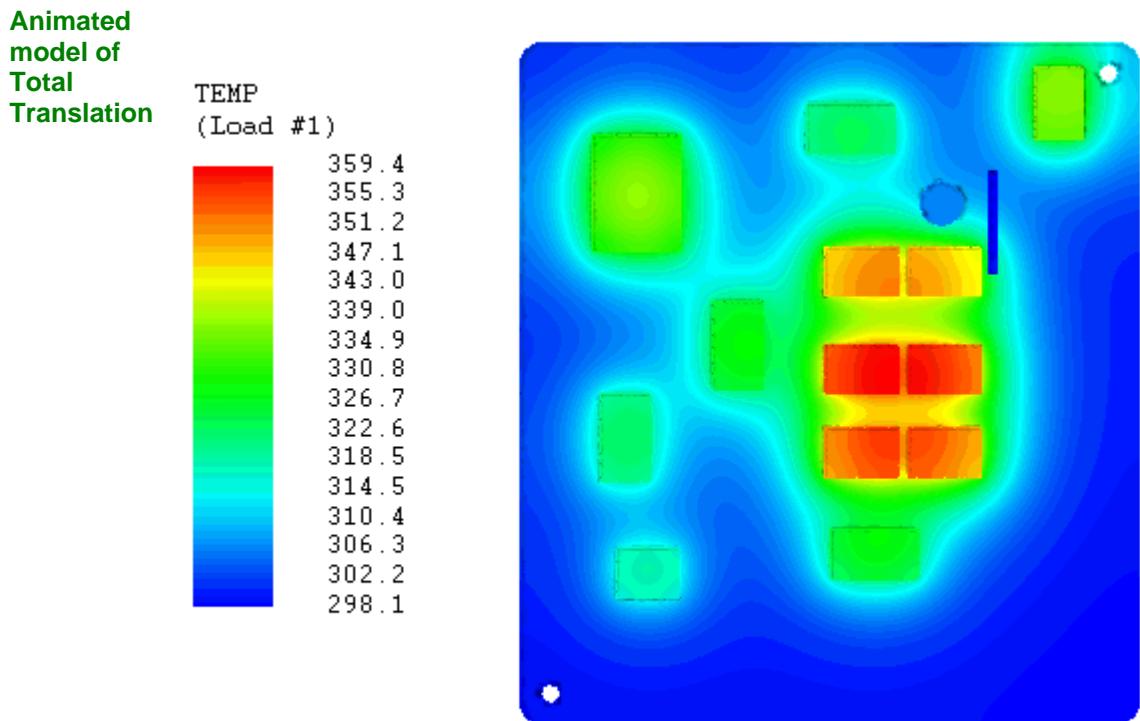
Select Ansys Variables window

Select Animate , which pops up the Setup Animations window as shown.

Now press Animate  to view the deformation. The deformation is shown below.

Animation Setup and Controller window





Finally select **Exit** to quit the post processor.

g) Saving the project

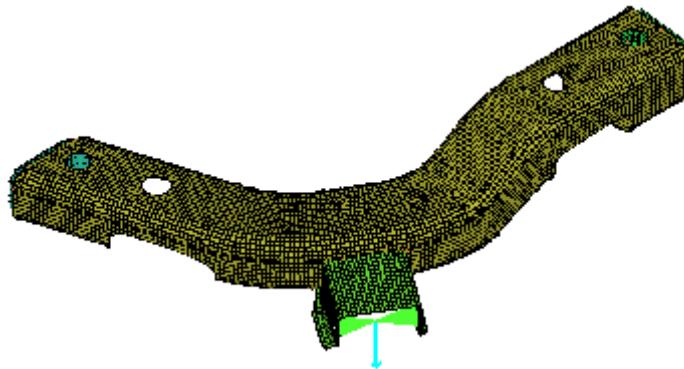
File > Save Project and close.

6.3: LS-Dyna Tutorials

6.3.1: 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 here.

Figure 6-98
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

The input files for this tutorial can be found in the Ansys installation directory, under ..\v110\docu\Tutorials\AI_Tutorial_Files.

Copy the file Frame.dat to your working directory and use File > Import Mesh > From Nastran to open it.

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. All defaults and options applicable for LS-Dyna will be made active.

Define Constraints

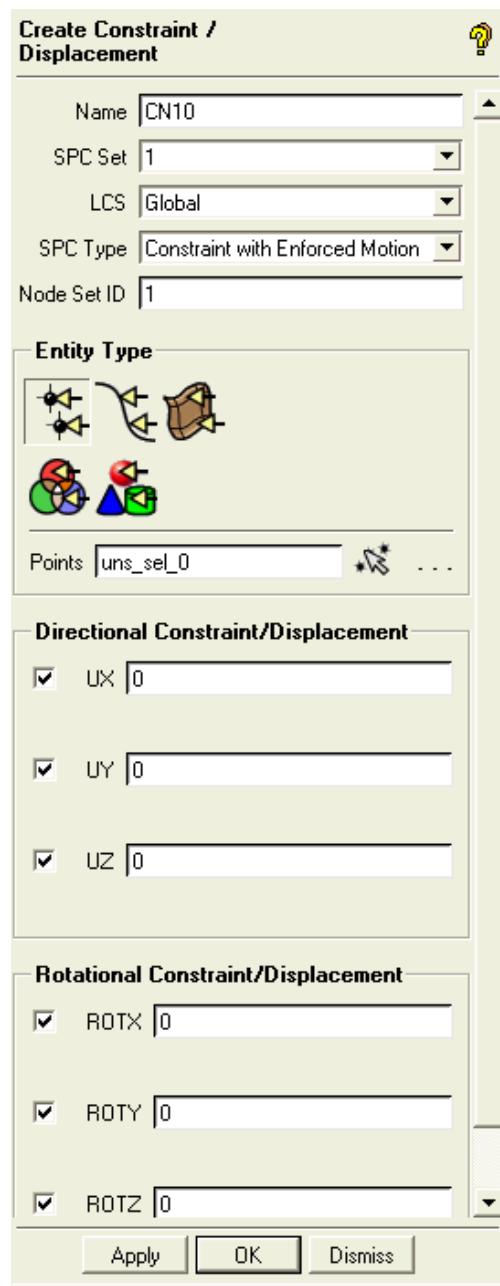
First a displacement constraint will be applied to one bolt hole. Another bolt hole was already defined in the original Nastran deck.

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 >  Create Constraints/Displacements >Create

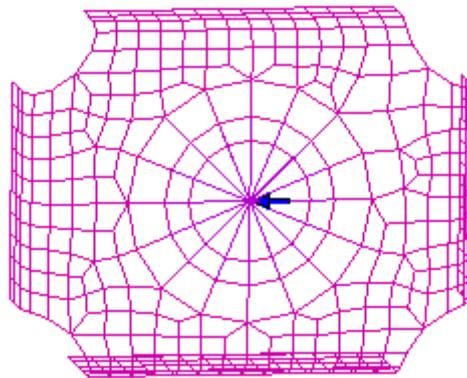
Constraints/Displacemets on Point  as presented below. Enter CN10 for the Name. Toggle on options UX, UY, UZ, ROTX, ROTY, ROTZ. Leave all settings and values as default.

Figure 6-99
Create Displacement on Point window



Click on Points > Select node(s)  ; select the node as shown below and Apply.

Figure 6-100
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 Surface Contact  as presented below.

**Figure 6-101
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 here.

Define Material Property window**Define Material Property**

Material Name IsotropicMat1

Material ID 1

Type:

Isotropic

Young's Modulus (E) Constant Varying

Value 207000

Shear Modulus (G) Constant Varying

Value

Poisson's Ratio (NU) Constant Varying

Value 0.28

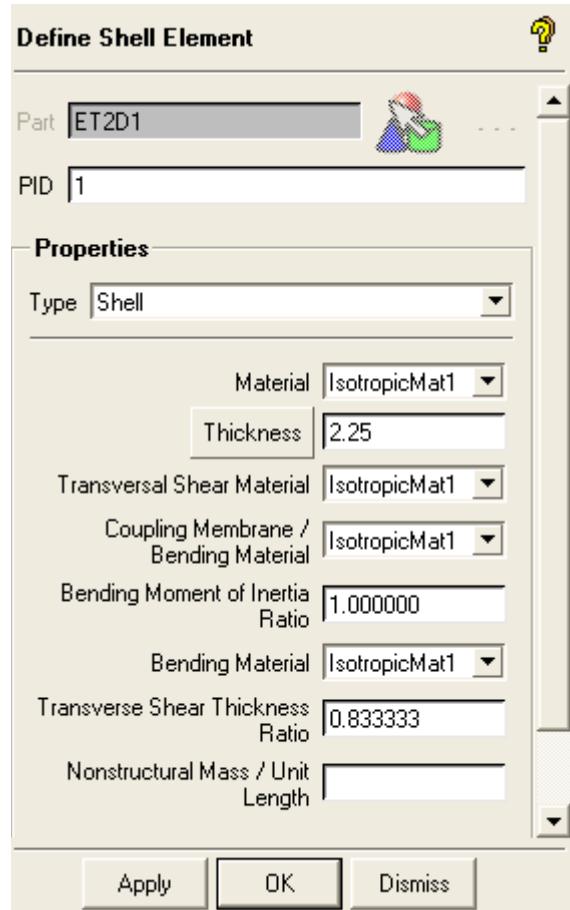
Mass Density (RHO) <input checked="" type="radio"/> Constant <input type="radio"/> Varying Value 7.8e-006	Stress Limits for Tension (ST) <input checked="" type="radio"/> Constant <input type="radio"/> Varying Value
Thermal Expansion Coefficient (A) <input checked="" type="radio"/> Constant <input type="radio"/> Varying Value	Stress Limits for Compression (SC) <input checked="" type="radio"/> Constant <input type="radio"/> Varying Value
Reference Temperature (TREF)	Material Coordinate System (MCSID)
Structural Element Damping Coefficient (GE) <input checked="" type="radio"/> Constant <input type="radio"/> Varying Value	Stress Limits for Shear (SS) <input checked="" type="radio"/> Constant <input type="radio"/> Varying Value
LS-Dyna Material type Select Type Type 24: *MAT_PIECEWISE_LINEAR	
Yield stress 210	
Failure strain 0.2	
<input type="button" value="Apply"/> <input type="button" value="OK"/> <input type="button" value="Dismiss"/>	

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 below. Note the Material assignment and Thickness. Review all other shell parts (ET2D*).

**Figure 6-102
Save Project As
window**



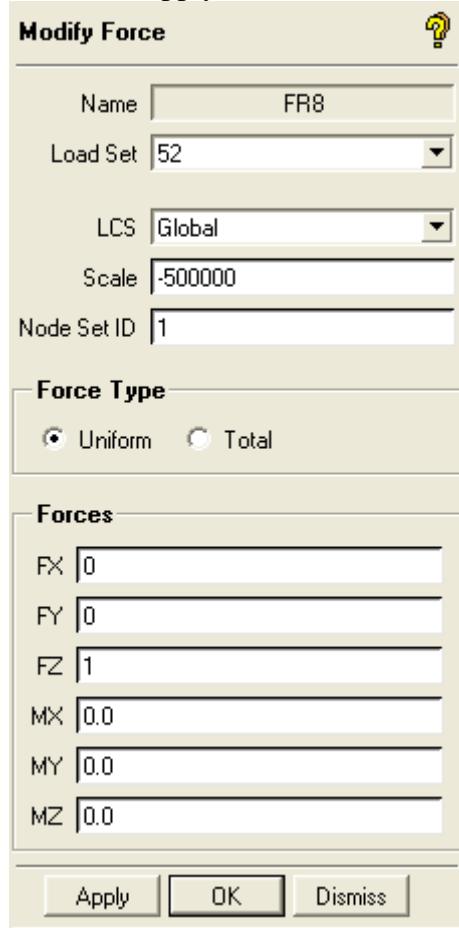
Also review the line properties for ET1D5 (bars representing the bolt holes) and RBE2 > ET1D16 (rigid bodies connecting the main shell parts). Turn on Line

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 below.

Enter Node Set ID as 1,Press Apply

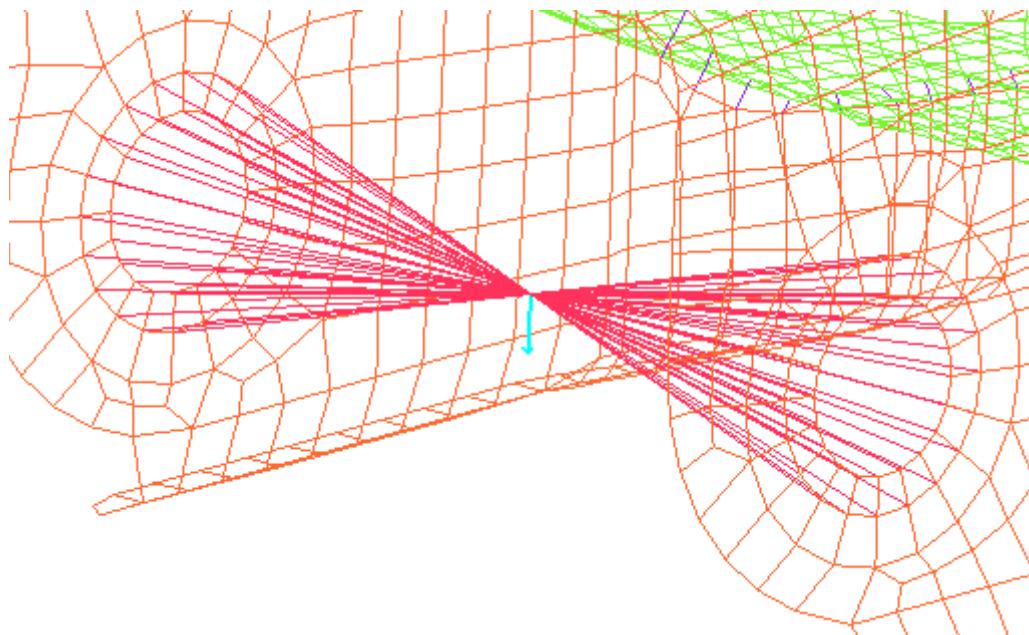
**Figure 6-103
Modify Force
panel**



Turn on Mesh > Lines. Also, turn on the Load in the Model tree and view as shown below.

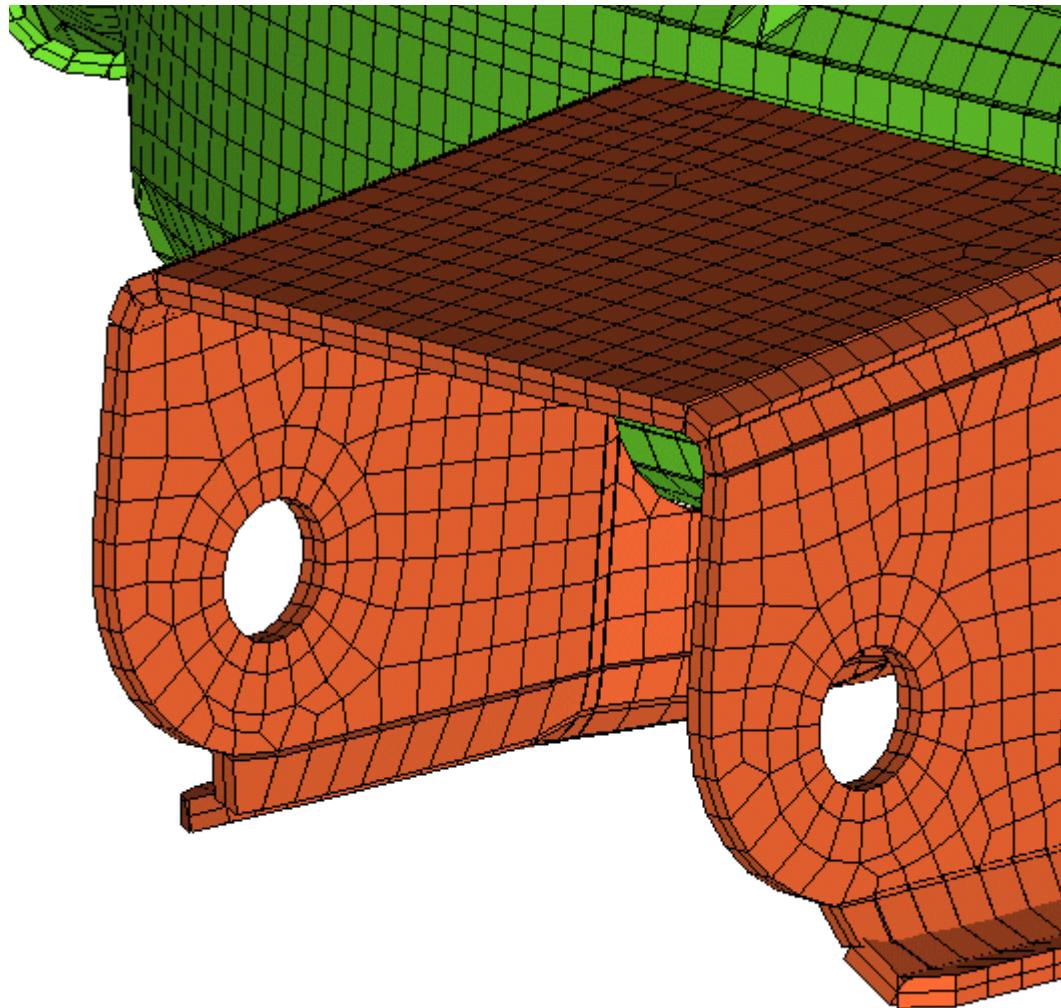
Note the downward force applied to the center of the bars representing the bolt across the flange, ET2D2.

**Figure
6-104
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.

**Figure
6-105
Modified
Force**

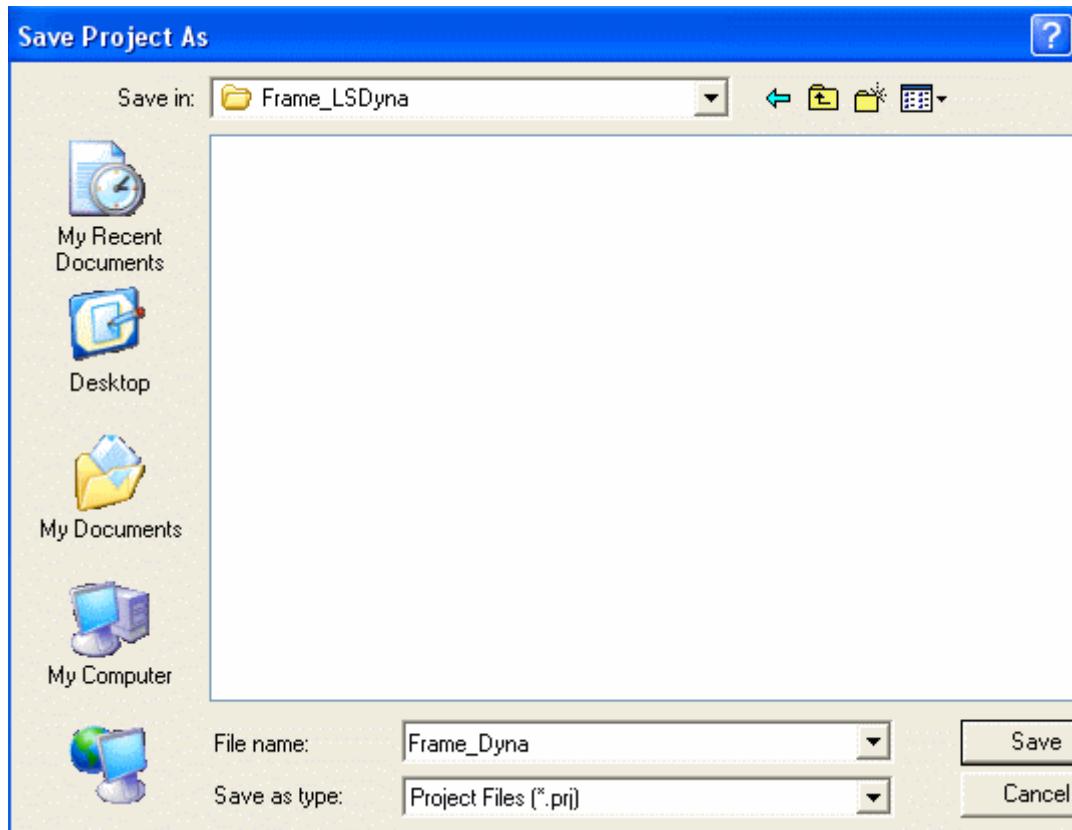


c) Save Project

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 below.

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
6-106
Save
Project
As
window**

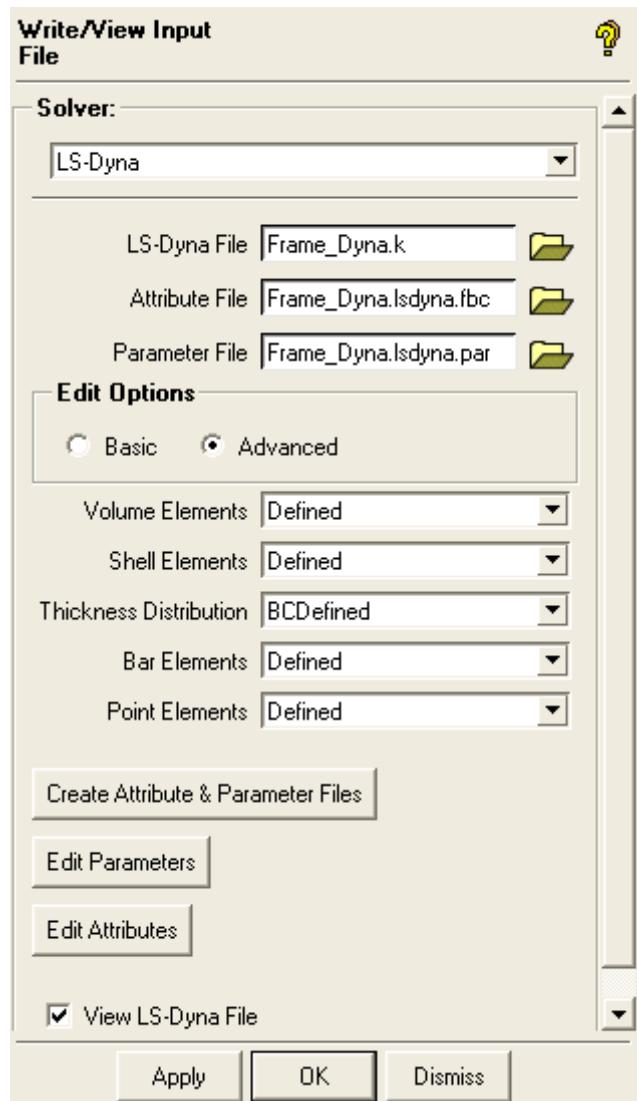


d) Solver Setup

Write LS-Dyna Input File

Select Solve Options > Write/View Input File to get the panel shown below.

Figure 6-107
Write/View Input
File window



Change Thickness Distribution to BCDefined. Turn on Edit Options > Advanced and select Create Attribute and 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

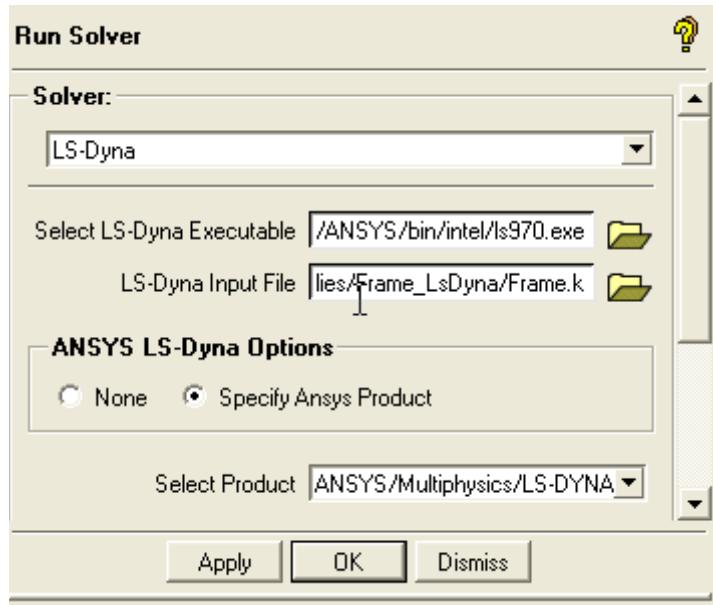
Select Solve Options > Submit Solver Run  to start LS-Dyna. Make sure the LS-Dyna Input File is selected as Frame_Dyna.k. Check and verify the location of the LS-Dyna executable.

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 6-108
RunSolver window

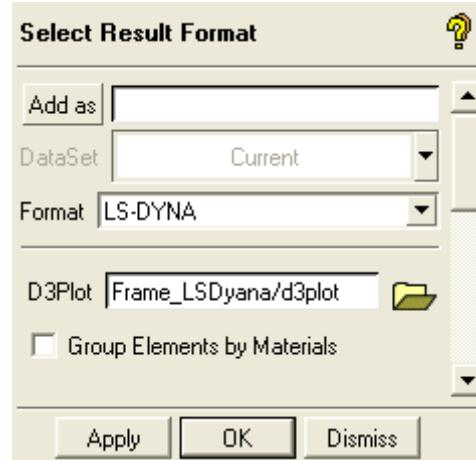


Select File > Close Project and save if prompted.

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.

Select Result Format
window



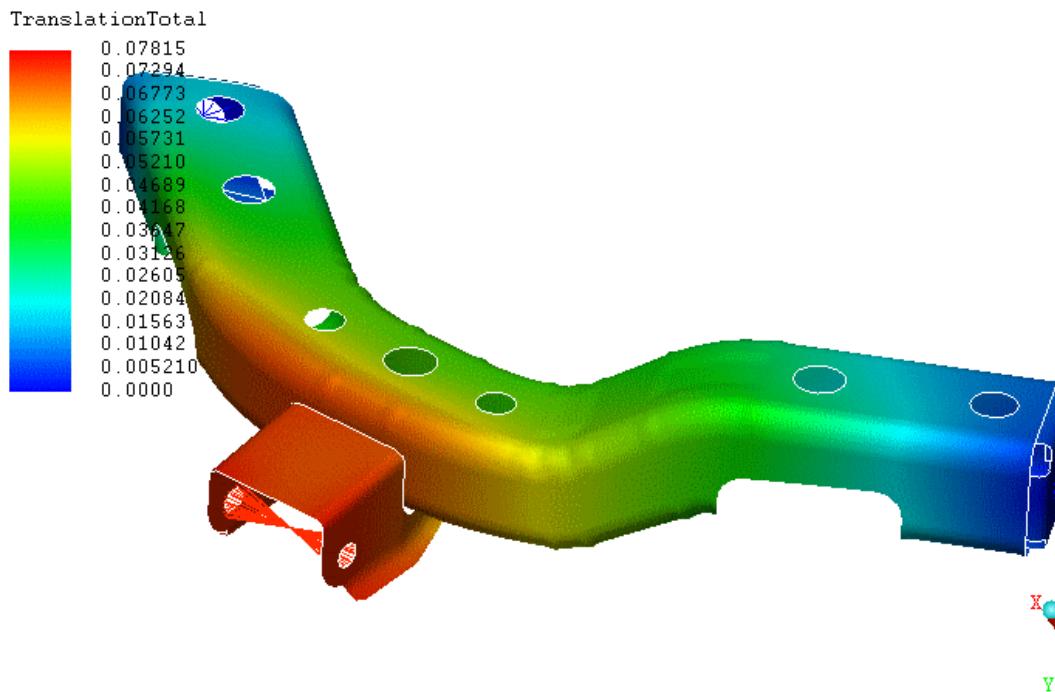
After loading, the default panel will be Select Transient Steps. The default displayed variable will be total translation. To change the displayed variable, select

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 here. For a more complete tutorial of post-processing functionality, please refer to the CFD > Post Processing or the FEA > Ansys tutorials.

Figure 6-109
Results Displayed in the Graphics window



6.3.2: 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.

**Figure 6-110
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

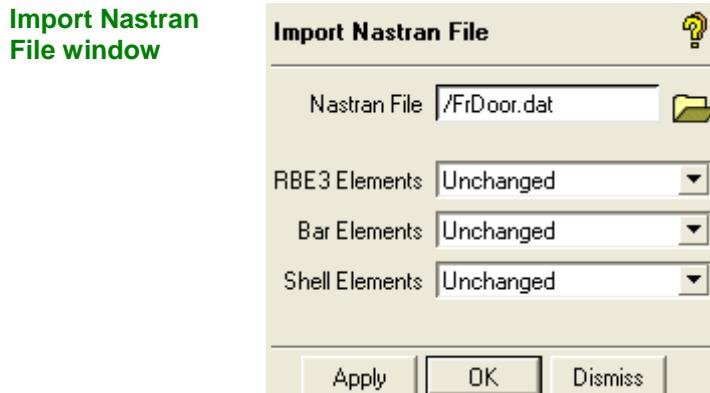
b) Data Editing

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 below. Make sure LS-Dyna is selected as the solver via Settings > Solver, and hit Apply.



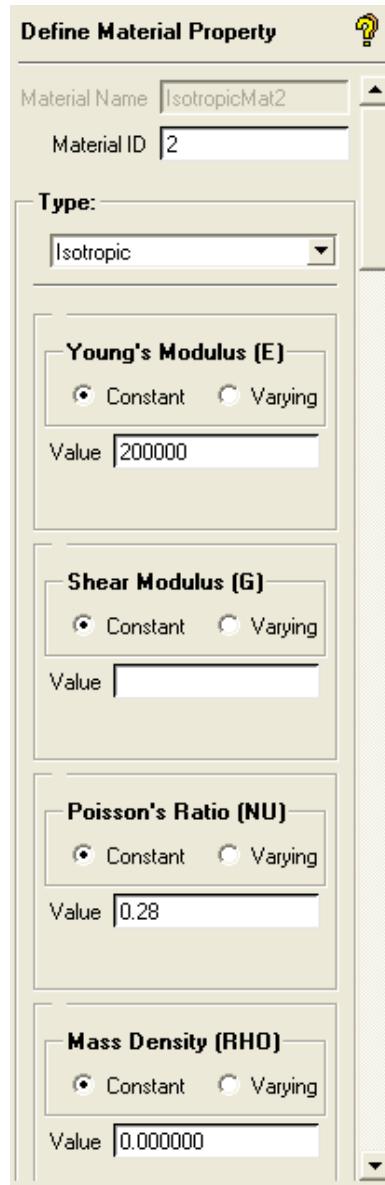
Press Apply in the Import Nastran File window.

c) Solver Setup

First, user should select the appropriate solver before proceeding further. Select Settings > Solver from the Main menu and select LS-Dyna from the Common Structural Solver and press Apply

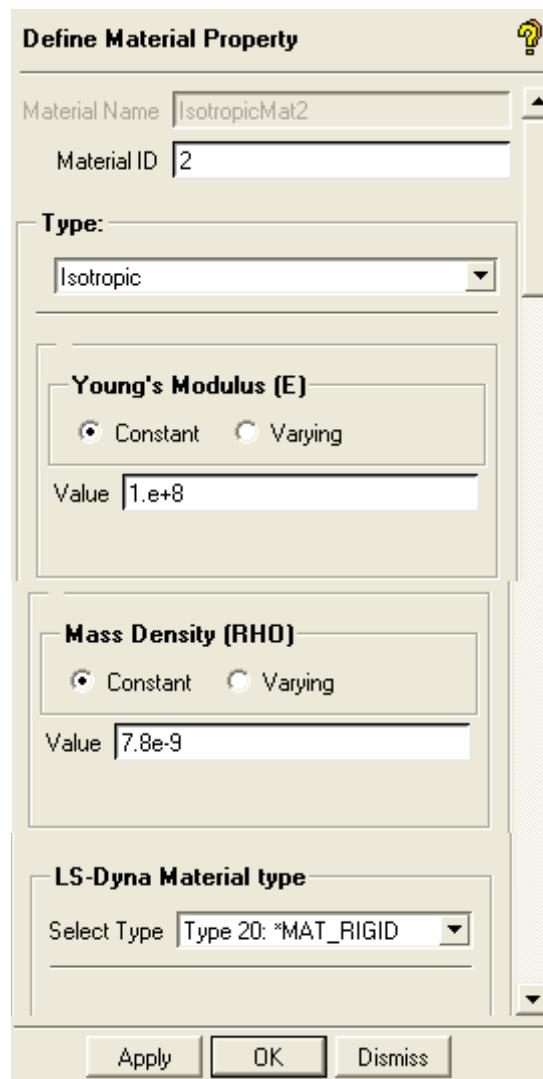
Verification of imported data

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 here.

**Define Material Property
window**

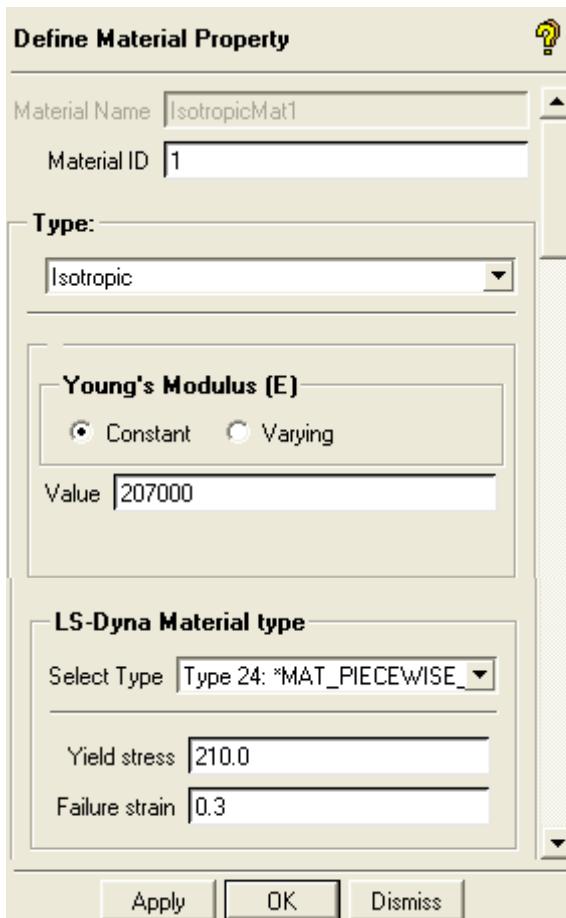
Modifying Density and LS Dyna Material Type for Materials

Change the constant value of Young's Modulus from **200000** to **1.e+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 and press Apply.

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**. Set Yield stress to 210.0 and Failure strain to 0.3 as shown and Press Apply.

Modify Isotropic Material 1



d) 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.

Define Single Surface Contact window



As shown above, supply the following information for the contact.

Name: CONTACT_ALL

Contact surfaces: Select all visible elements using hotkey “v” from the keyboard. (Make sure that Points and Lines are switched Off in the Mesh branch of the Display Tree.) The messages area should indicate “11081 elements.”

Select AUTOMATIC SINGLE_SURFACE option for Automatic contact option.

Press Apply to generate Contact Surface.

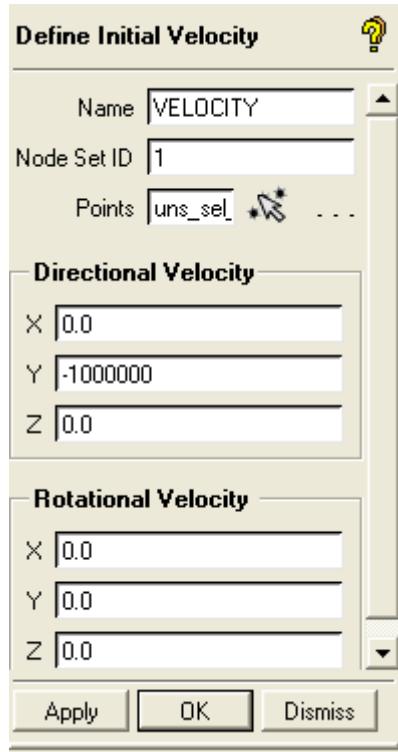
Turn **OFF** Single Surface Contacts display from Display Tree.

Velocity

From the Constraints tab click on Define Initial Velocity  to open the Define Initial Velocity window as presented below.

Expand Parts in Display Tree by clicking on +, and turn **OFF** all the parts except ET2D22.

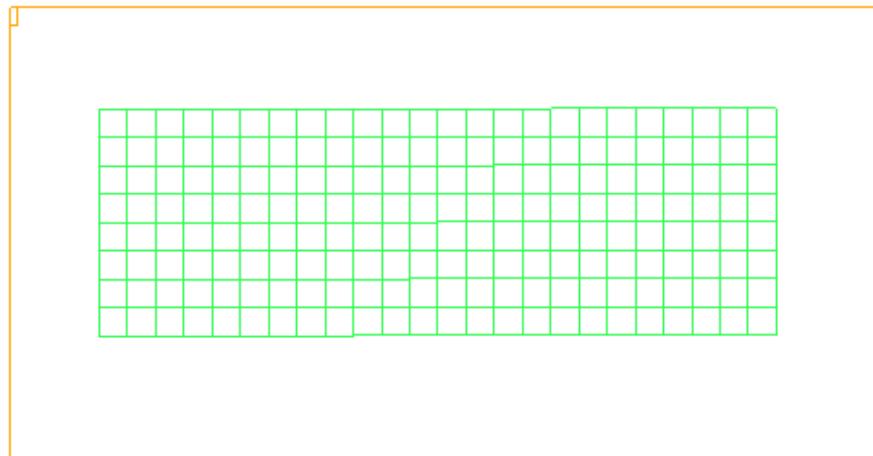
**Figure 6-111
Define Initial
Velocity window**



As shown, 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 below. (Make sure that Points and Line are switched ‘Off’ in the Display Tree). The message area should indicate “514 nodes.”

**Figure
6-112
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.

Switch **On** all the Parts by Parts > Show All in the Display Tree.

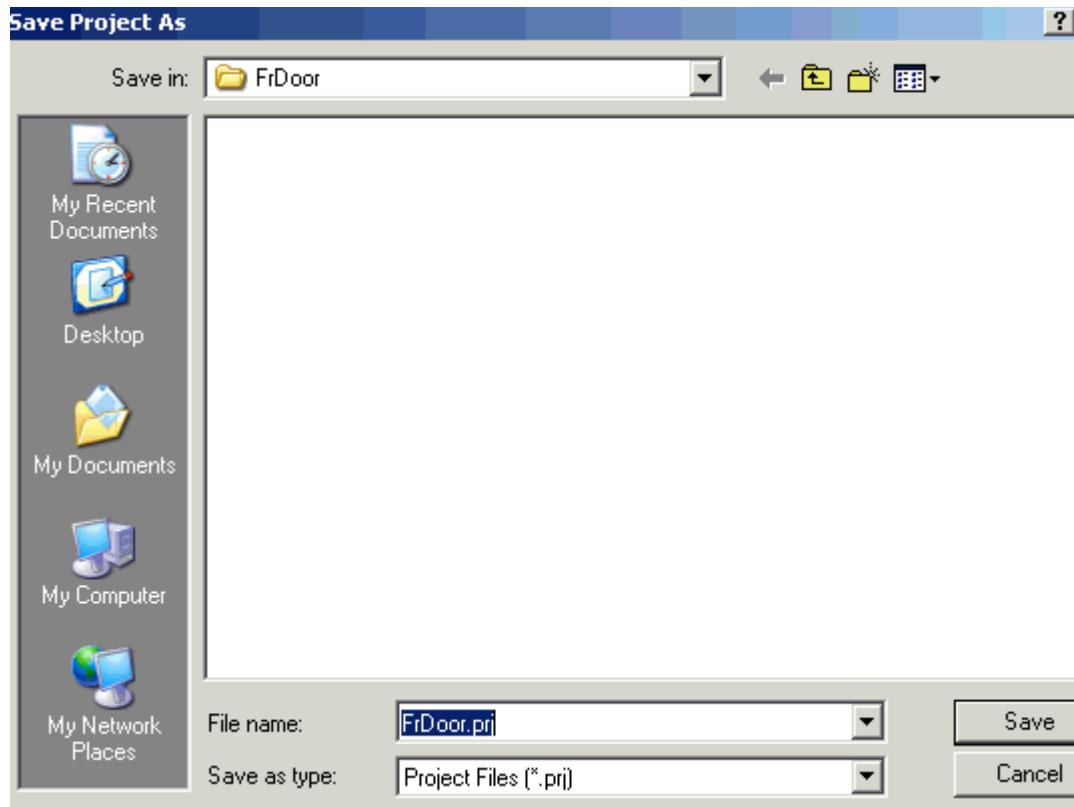
Save Project

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.

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.

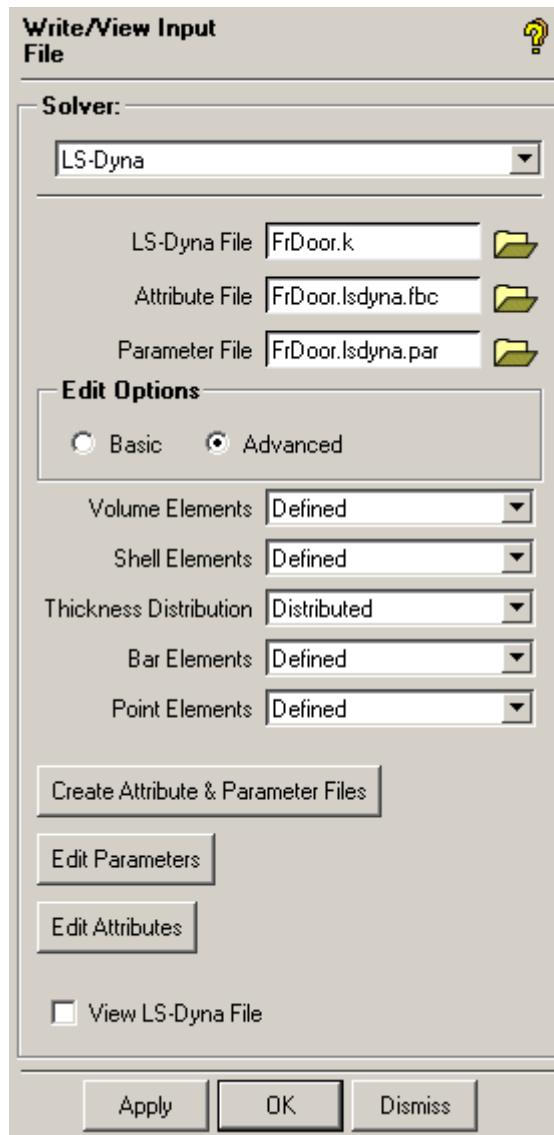
Figure
6-113
Save
Project
As
window



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 here.

Figure 6-114
Write/View Input
File window



In Edit Options, Enable **Advanced**, and click on **Create Attribute and 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.

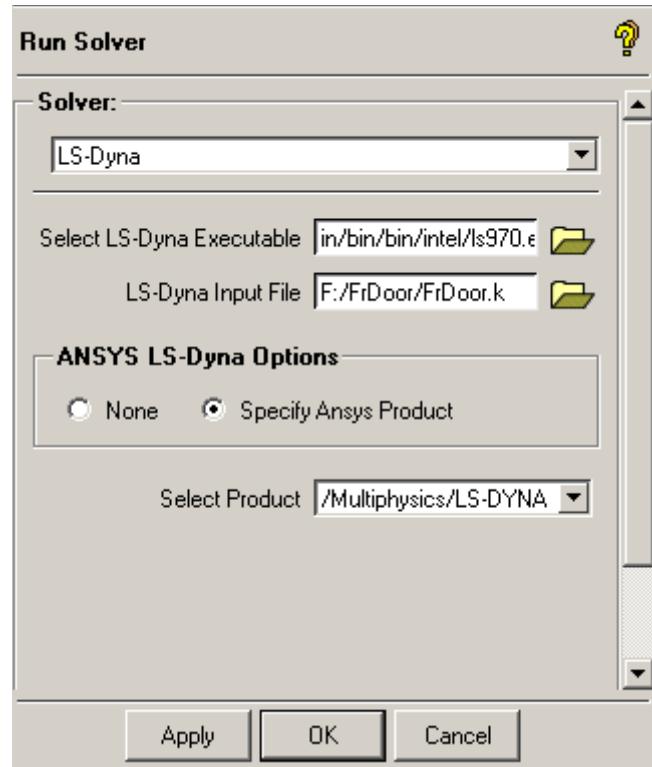
Solving the problem

From the Solve Options tab click on Submit Solver Run  to start LS-Dyna with the **Run Solver** window given below.

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 6-115
LS-Dyna Run window

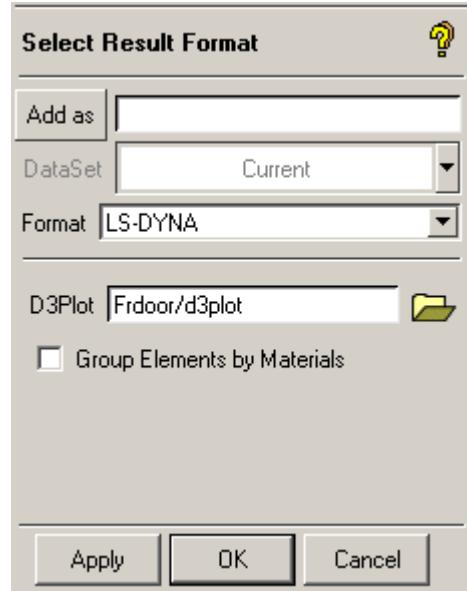


Post Processing of Results

From the main menu select File > Results > Open Results. The Select Result Format window is displayed. 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.

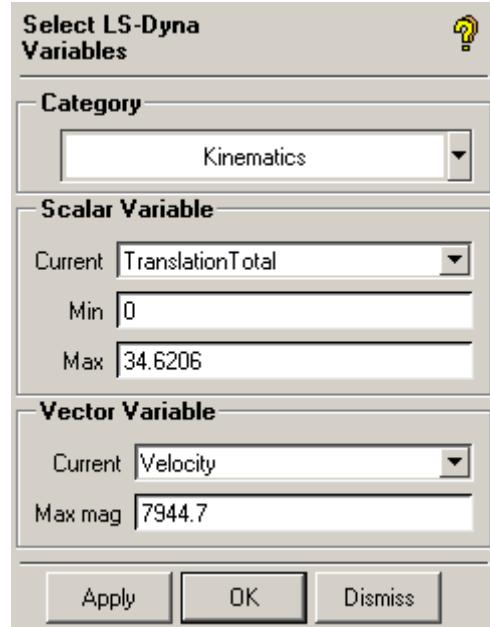
In the Select Transient Steps panel set Single-step to step#1 0.005 and enable the Display Transient Time option. (Note that Run-time instead of Single-step will process through all the time steps updating the results display at each step.)

Figure 6-116
Select Result Format



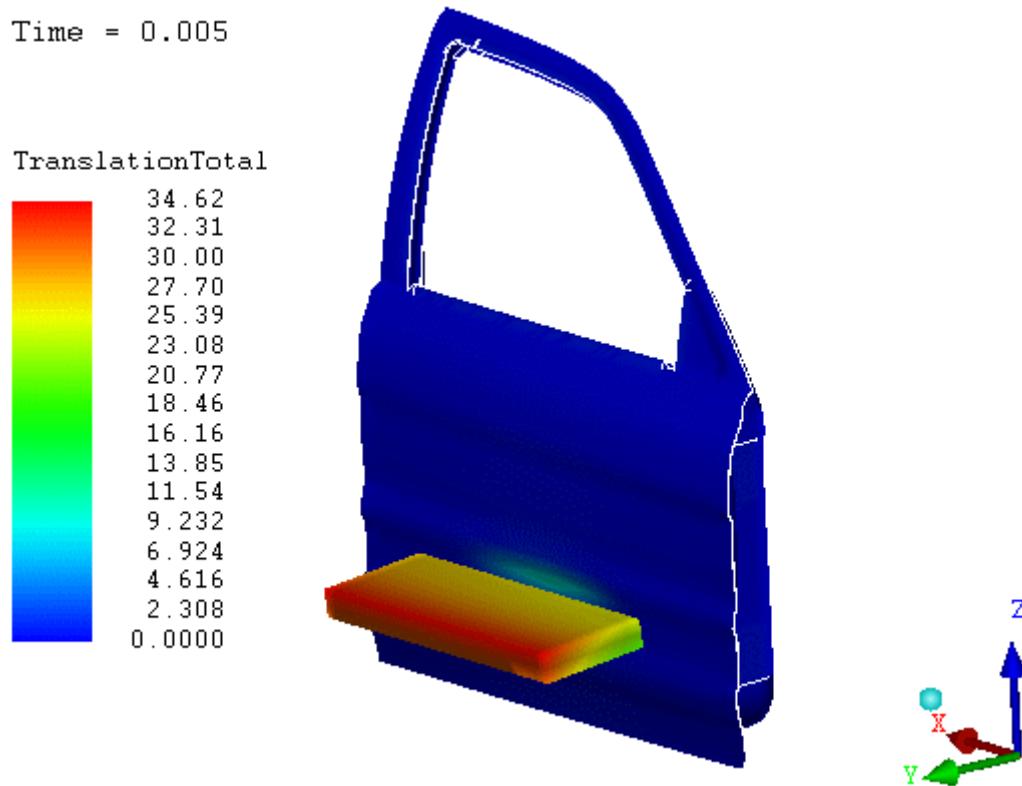
The model will be displayed in the graphics window. From the Post-processing tab click on Variables and the Select LS-Dyna Variables window will be displayed. Set the Category to Kinematics and the Current Scalar Variable to Translation Total as shown.

Figure 6-117
Result Variables window



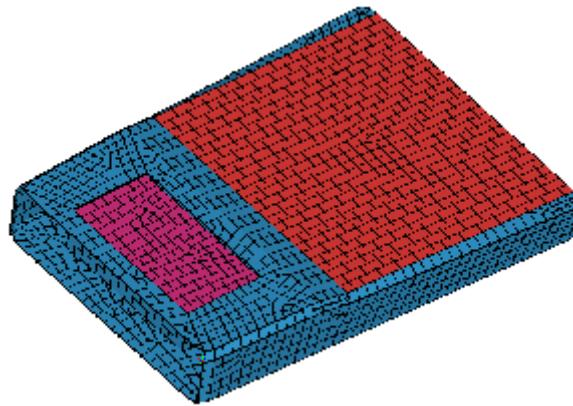
The following results can be seen in the graphics window shown.

Figure 6-118
Display Results in the Graphics window



6.3.3: 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

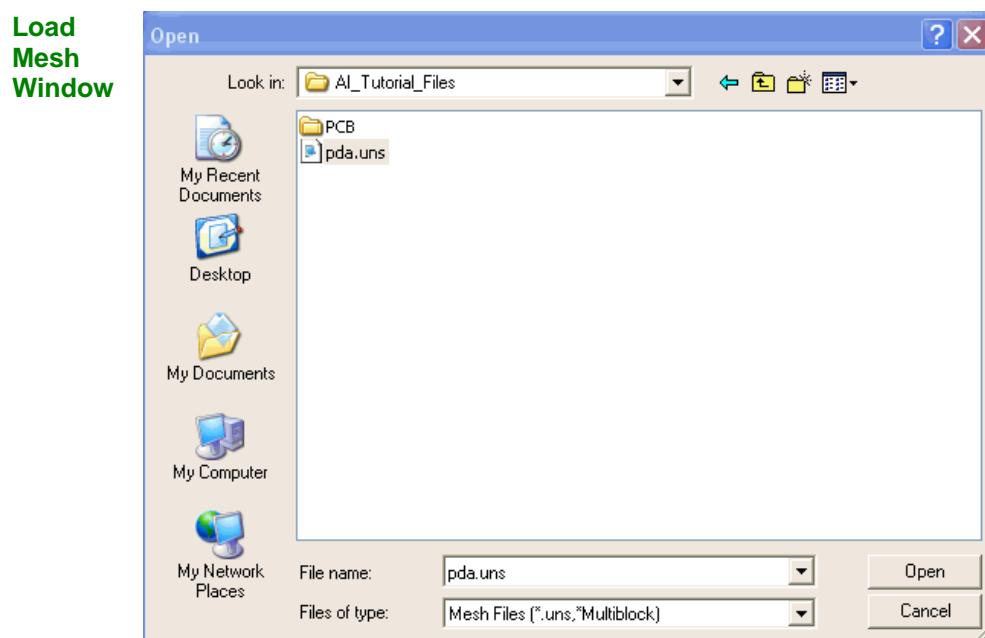
ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	1132
------------------------	--	------

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. The input files for this tutorial can be found in the Ansys installation directory, under ..\v110\docu\Tutorials\AI_Tutorial_Files. Copy the pda.uns file to your working directory and use Open Mesh  from the main menu to open the window as shown.



c) Solver Setup

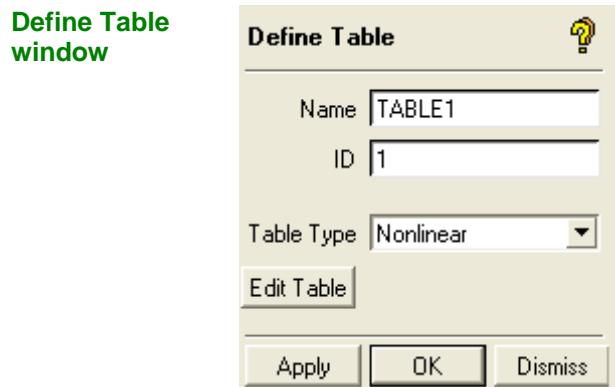
First, user should select the appropriate solver before proceeding further.

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	1133
------------------------	--	------

Select Settings > Solver from the Main menu and select LS-Dyna from the Common Structural Solver and press Apply

Go to Define Properties in Table

From the Properties tab click on Create Table  to open the **Define Table** window.



Enter the **Name** as TABLE1 and **ID** as 1. (Both will come by default)

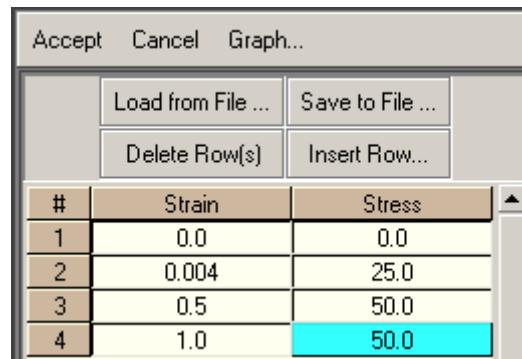
In the **Table Type** window select Nonlinear.

Click on **Edit Table** and enter the data shown. 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 6-119
Table Editor
window**



The screenshot shows a Windows-style dialog box titled "Table Editor". At the top, there are three buttons: "Accept", "Cancel", and "Graph...". Below these are two rows of buttons: "Load from File ...", "Save to File ...", "Delete Row(s)", and "Insert Row...". The main area is a table with four columns. The first column is labeled "#", the second is "Strain", and the third is "Stress". There are four rows of data:

#	Strain	Stress
1	0.0	0.0
2	0.004	25.0
3	0.5	50.0
4	1.0	50.0

d) 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.

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**

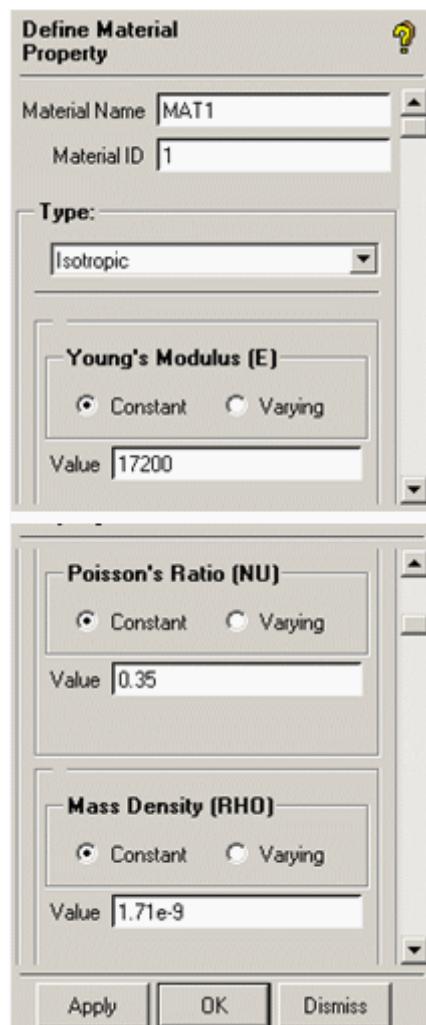
Define the Constant **Poisson's ratio** as **0.35**,

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 6-120
Define Material Property window



Create another material Named as **MAT2** in **Define Material Property** window shown.

Material ID can be left as **2**,

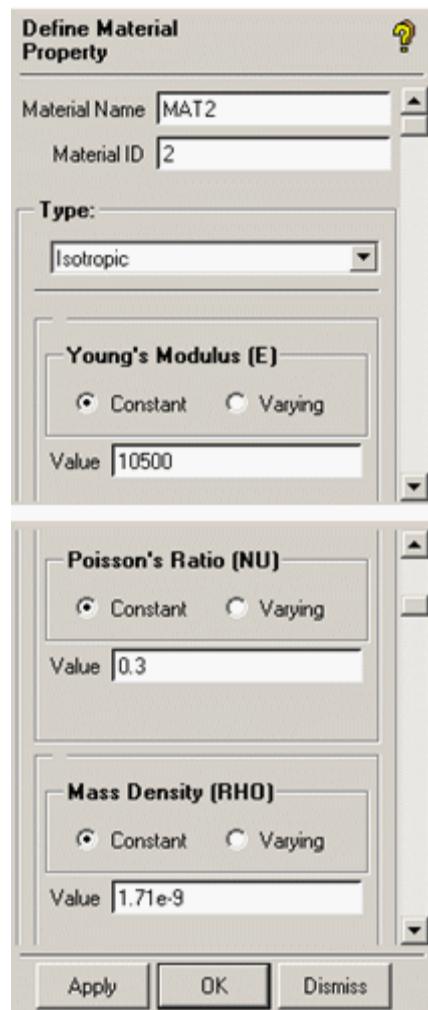
Select the type as **Isotropic** from the drop down menu.

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 6-121
Define Material Property window

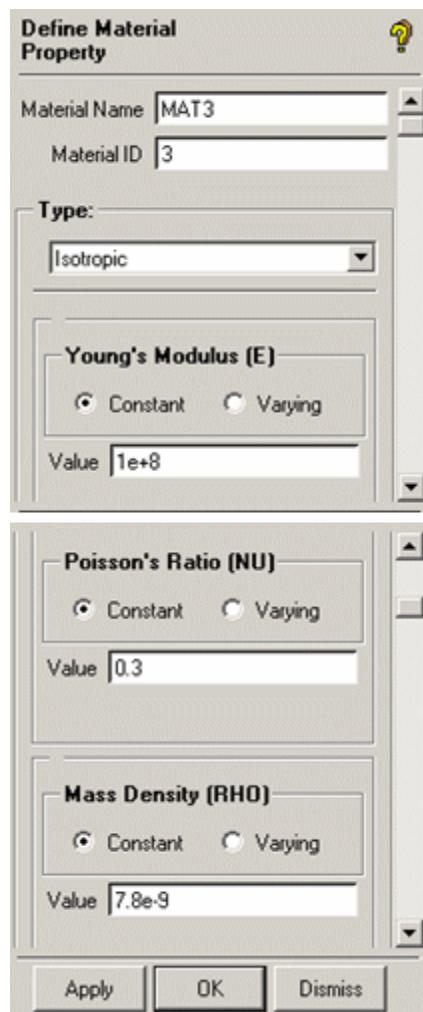


Create another material Named as **MAT3** in **Define Material Property** window shown.

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 6-122
Define Material Property window



Element Properties

From the Properties tab select Define 2D Element Properties.  The Define Shell Element window appears as shown.

Select Part as **PDA**.

Set PID as **10**.

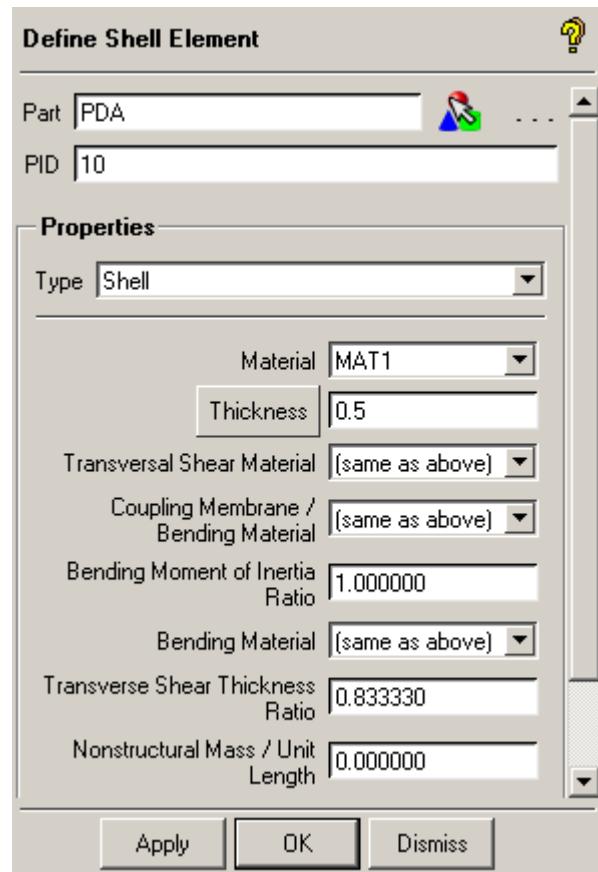
Select Type as **Shell**

Select material as **MAT1**.

Supply Thickness as **0.5**,

Keep all other parameters as default and press Apply.

Figure 6-123
Define Shell
Element
window

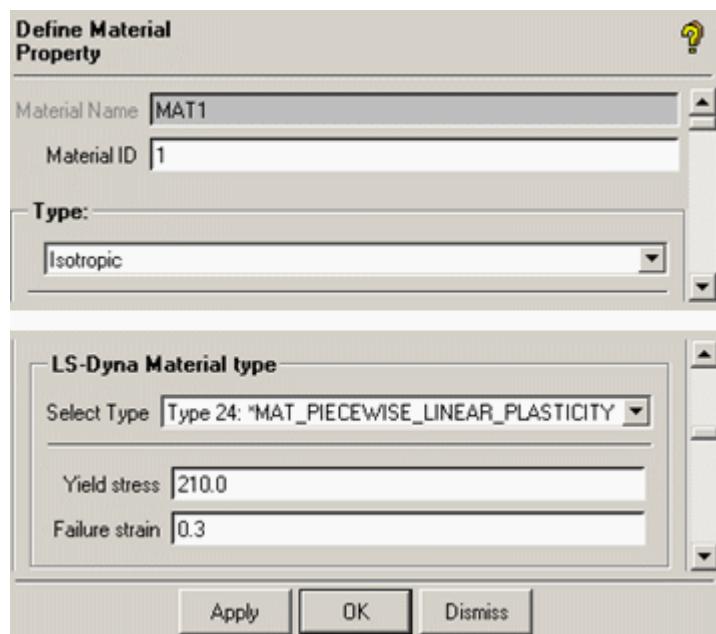


Similarly, define the shell properties on the BAT_COVER part also. All the properties are going to be the same for BAT_COVER except PID. Set the PID to **11** and supply the same thickness of **0.5**.

In the main menu select Settings > Solver and select **LS-Dyna** from the pull-down. Press Apply. Expand Material Properties in the Display Tree. Right-click on MAT1 and select Modify to open the Define Material Property window as shown, 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 6-124
**Define
Material
Property**



From the Properties tab select Define 2D Element Properties  to open the Define Shell Element window as shown.

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**.

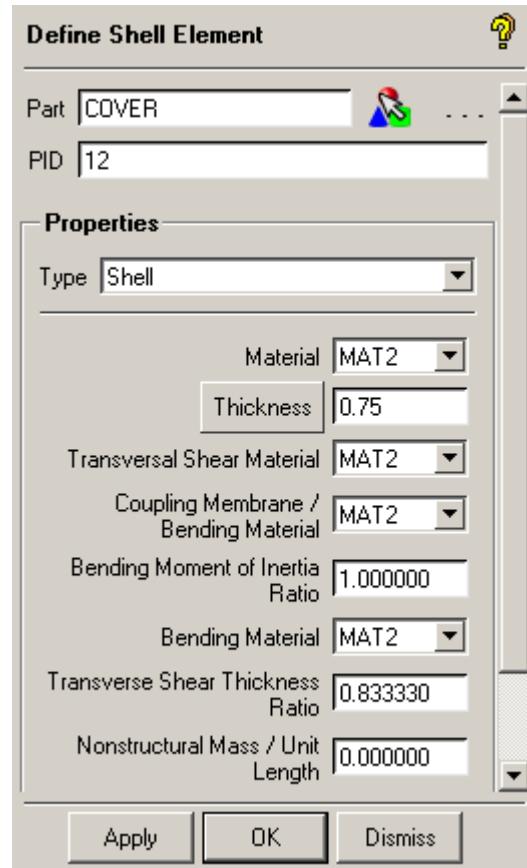
Select material as **MAT2**,

Supply thickness as **0.75**,

Select **MAT2** for the Transversal Shear Material, Coupling Membrane/Bending Material and Bending Material option.

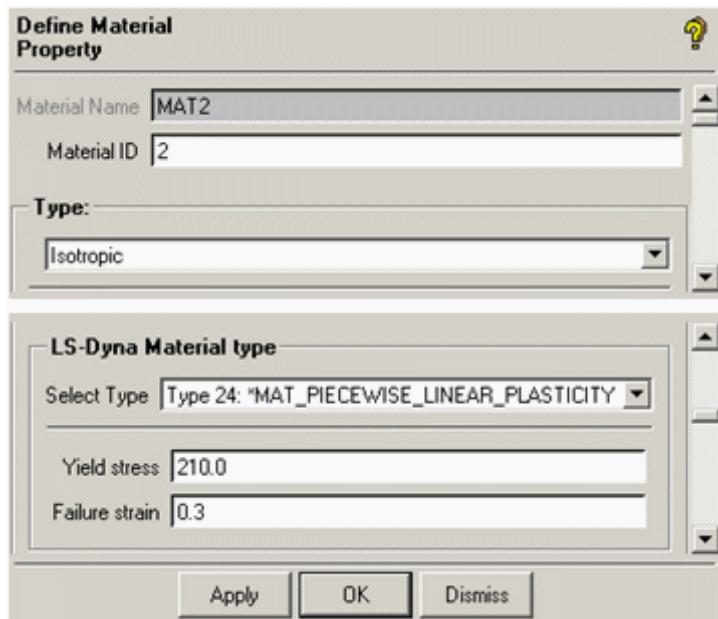
Press Apply to complete the operation.

Figure 6-125
Define Shell element window



Expand Material Properties in the Display Tree. Right-click on **MAT2** and select Modify to open the **Define Material Property** window shown below. 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 6-126
**Define
Material
Property
Window**



From the Properties tab select Define 3D Element Properties to open the **Define Volume Element** window shown below.

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

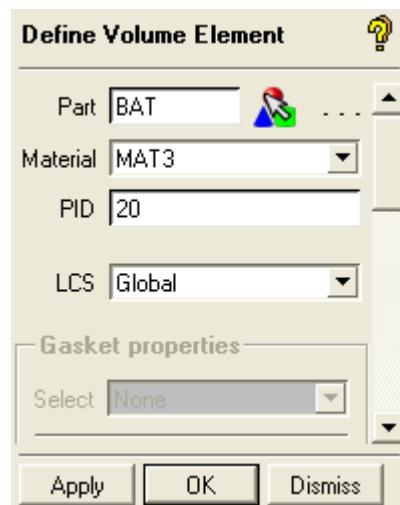
Select Part as **BAT**

Select material as **MAT3**.

Set PID to **20**.

Press Apply to complete the operation.

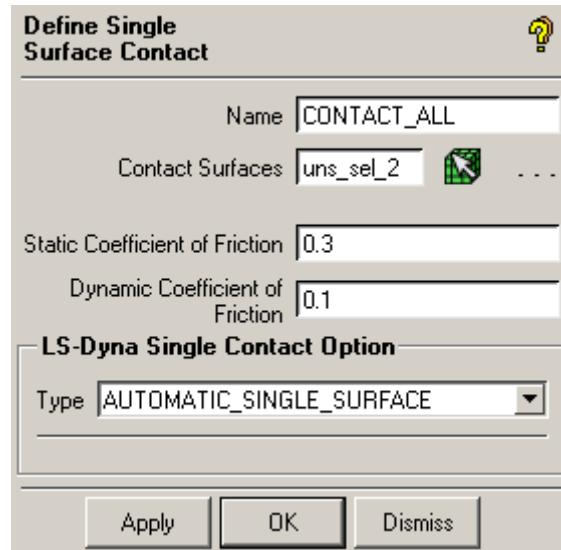
Define Solid
Element
window



e) Contact

From the Constraints tab click on Define Single Surface Contact  to open the **Define Single Surface Contact** window shown here.

**Figure 6-127
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

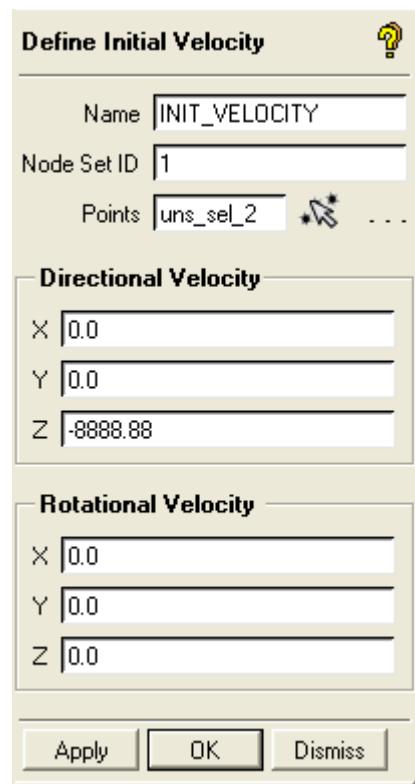
Press Apply to generate Contact information.

Turn **OFF** Single Surface Contacts display from the Display Tree.

f) Velocity

From the Constraints tab click on Define Initial Velocity to open the **Define Initial Velocity** window as shown below.

**Figure 6-128
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.

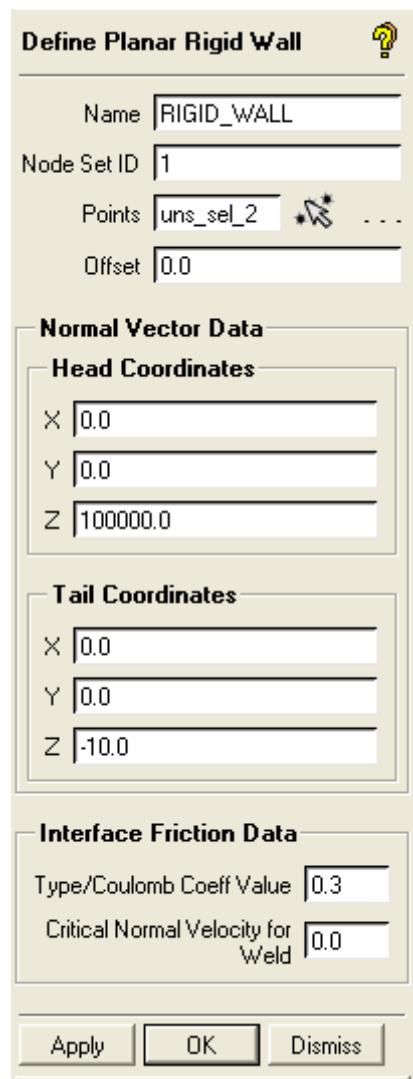
Press Apply to define Initial velocity.

Turn **OFF** Velocities display from the Display Tree.

g) Rigid Wall

From the Constraints tab click on Define Planer Rigid Wall to open the **Planar Rigid Wall** window as presented below.

Figure 6-129
Define Planar
Rigid Wall
window



Enter name as **RIGID_WALL**

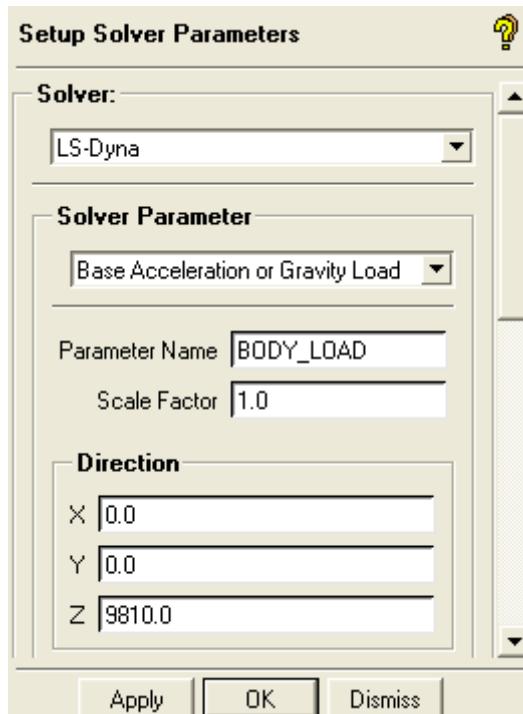
For Points click on Select node(s) and select all the nodes (0d elements) using hotkey “a”.

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.

h) Gravity Loading

Solve Options>Setup Solve Parameters and Solver Should shown as LS Dyna

Gravity Loading



Enter Load Set as **BODY_LOAD**.

Supply the Value of 9810.0 as the Z component for the Gravity.

Press Apply to define gravity.

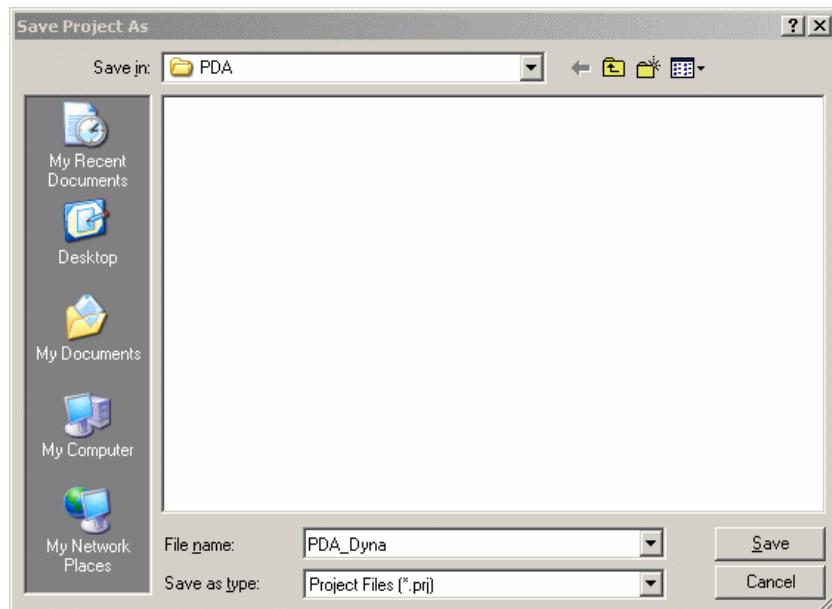
i) Save Project

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.

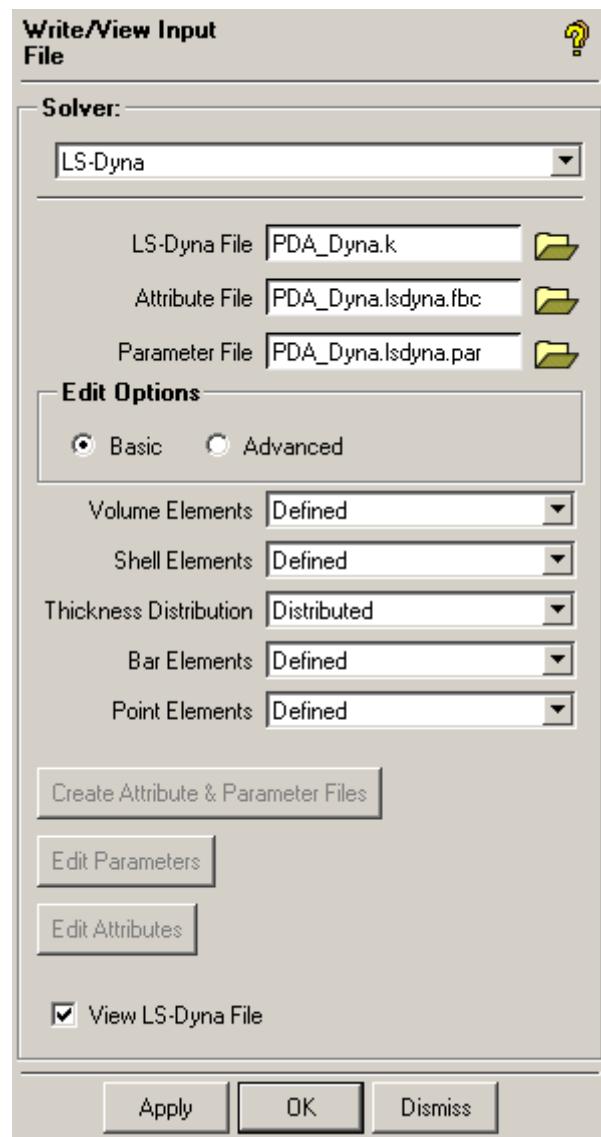
Along with the PDA_Dyna.prj file, it will also store three other files, Mesh file, Attribute file and Parameter files as PDA_Dyna.uns, PDA_Dyna.fbc and PDA_Dyna.par respectively.

**Figure
6-130
Save
Project
As
window**



From the Solve Options tab click Write/View Input File  to open the Write/View Input File window as shown here.

Figure 6-131
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.

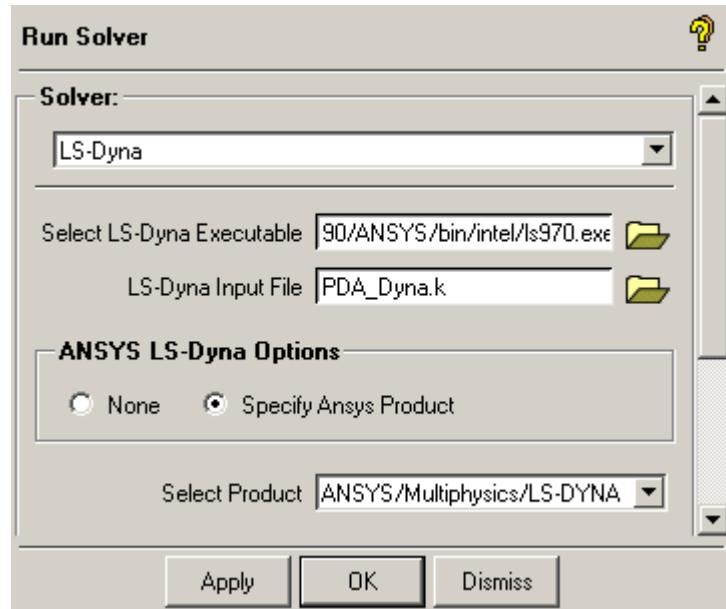
j) Solution and Results

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 below. 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.

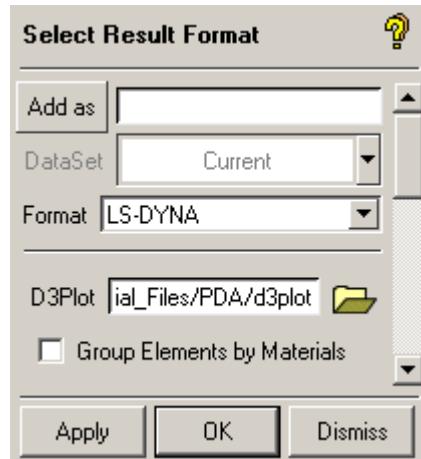
**Figure 6-132
Run Solver
Window**



Post Processing of Results

From the main menu select File > Results > Open Results  . The Select Result Format window is displayed as shown here. 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 6-133
Select Result
Format**



As soon as Apply button is pressed Select Transient Steps window will be displayed. as shown. 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.

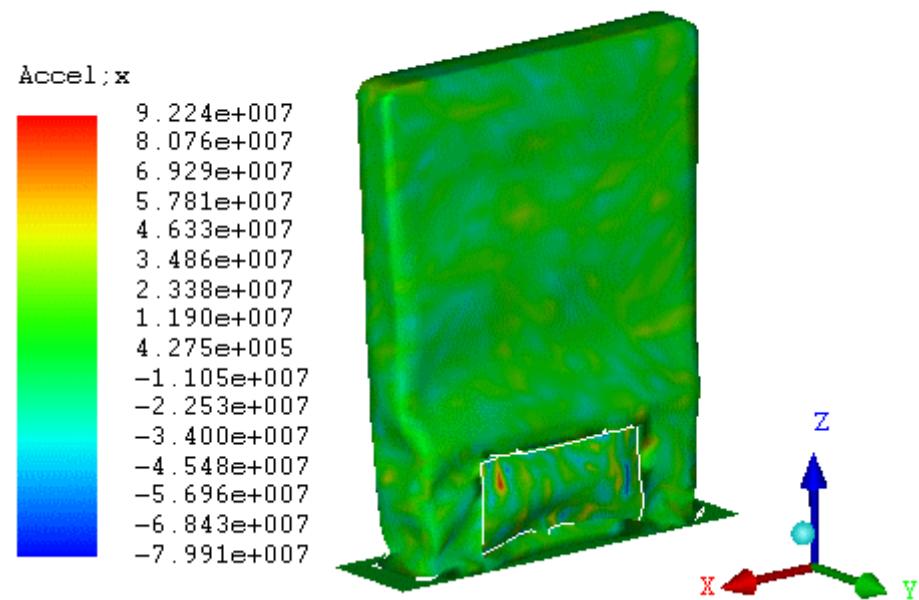
Figure 6-134
Select Transient Steps window



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.

Figure 6-135
Results Displayed in the Graphics window

Time = 0.00499887

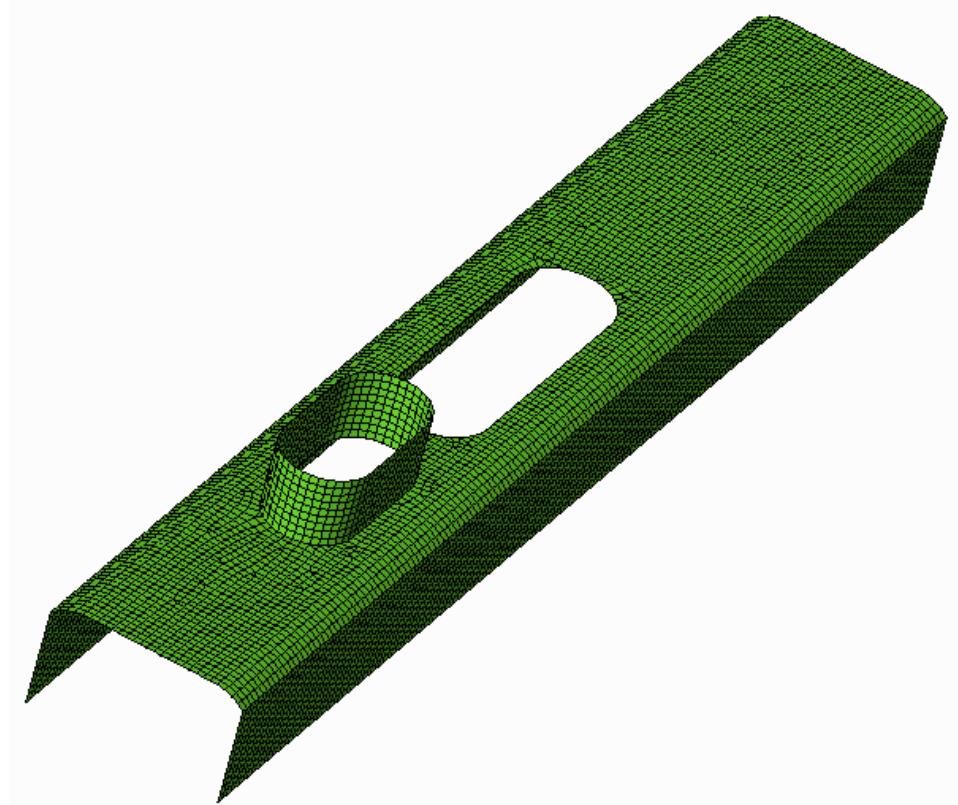


6.4: Nastran Tutorial

6.4.1: T-Pipe

This exercise is the continue of T-Pipe in the Structural Meshing Tutorial (TPipe.prj) from that how to writing the input file (*.dat) to perform Modal Analysis in NASTRAN. The visualization of results in Post Processor Visual3p is also explained. The model is shown below

T-
Pipe



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.

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	1154
------------------------	--	------

The input files for this tutorial can be found in the Ansys installation directory, under `../v110/docu/Tutorials/AI_Tutorial_Files`.

a) Summary of Steps

Launch AI*Environment and load geometry file

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

Proceed the Tutorial as the Continue Before T-Pipe:Structural Meshing Tutorials.

b) Open Project File

Open the Project File > TPipe.prj as you Earlier Completed in the Structural Meshing Tutorials

c) 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 Menubar**. Define the Material Name as **STEEL** and supply the required parameters for it in the **Define Material Property** window as shown below.

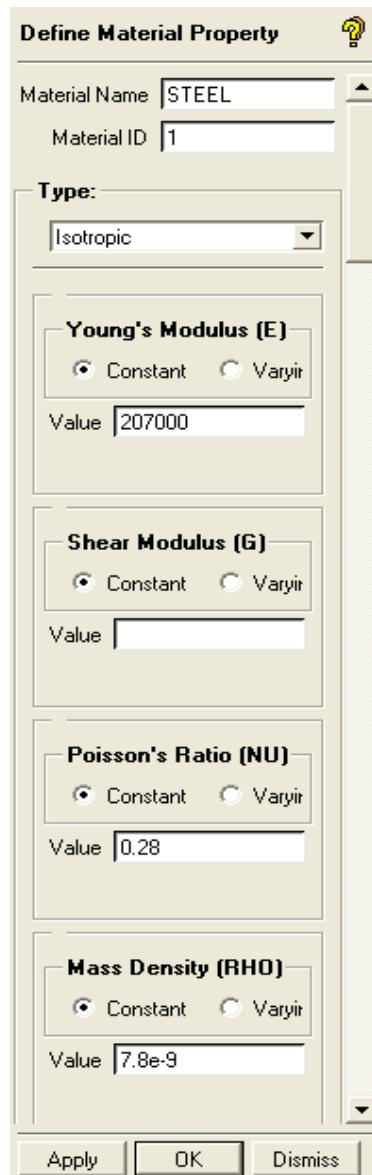
Material ID can be left as **1**.

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.

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	1156
------------------------	--	------

Figure 6-136
Define Material
Property
window



Element Properties

Select  (Define 2D Element Properties) icon from the **Properties Menu bar**.

Select Part as T4 for applying property.

Set PID as 1 in the **Define Shell Elements** window as shown below.

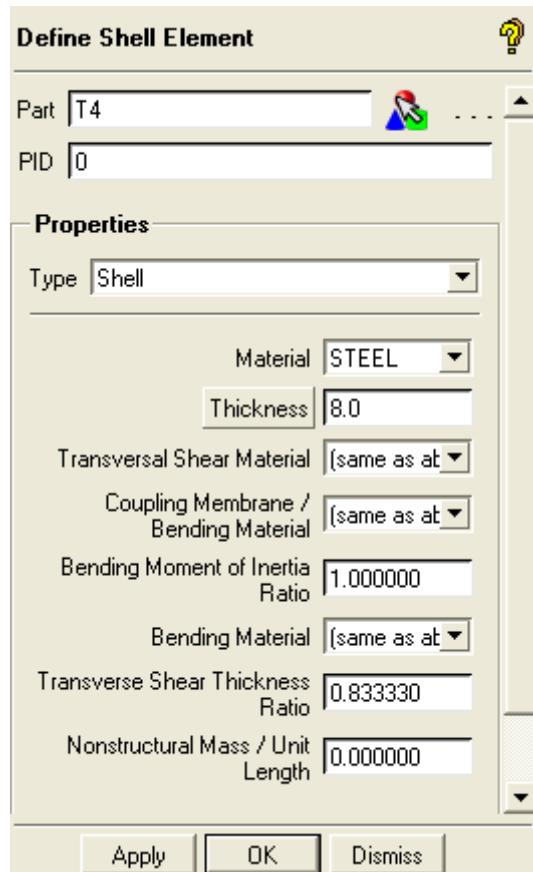
In the **Type** window select Shell

Thickness comes by default

Select Material as STEEL.

Press Apply.

Define Shell Element window



d) 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 Menu bar and select appropriate solver viz. Nastran and press Apply

Setup Solver Parameters

Click on  (Setup Solver Parameters) from **Solve Options 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.

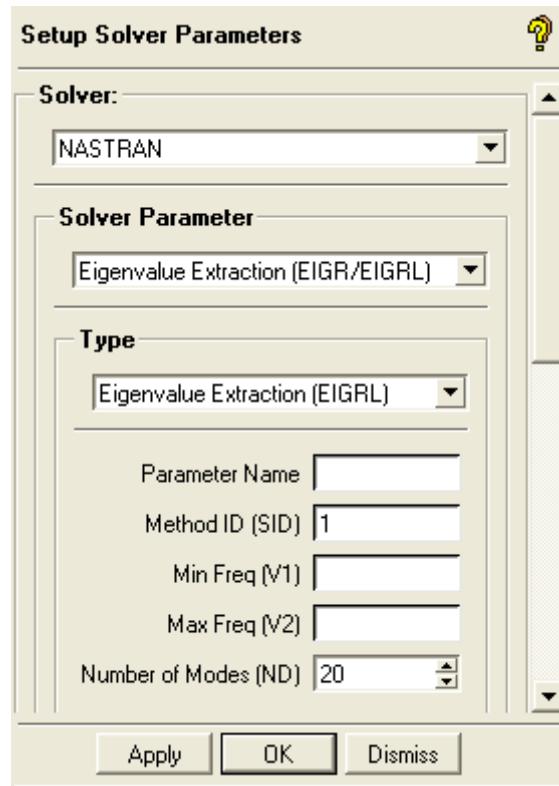
Set **Number of Modes** to 20. Min and Max could be defined to limit this, however in general it is easier to let Nastran just return first 20 frequencies.

Leave the other parameters as Default as shown here.

Press **Apply**.

A default Subset by the name of EIGRL1 is created under Parameters in the Model Tree.

Figure 6-137
Setup Solver Parameters
window

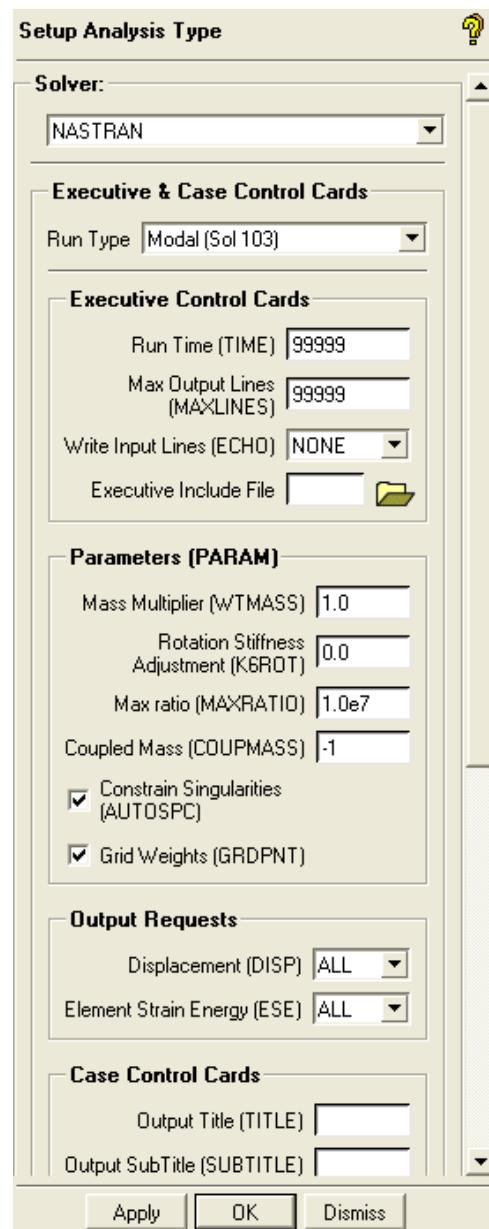


Click on  (Setup Analysis Type) icon from **Solve Options Menu bar** to setup Nastran run to do Modal Analysis that pops up **Setup Analysis Type** window shown.

Select Run Type as **Modal (Sol 103)**,

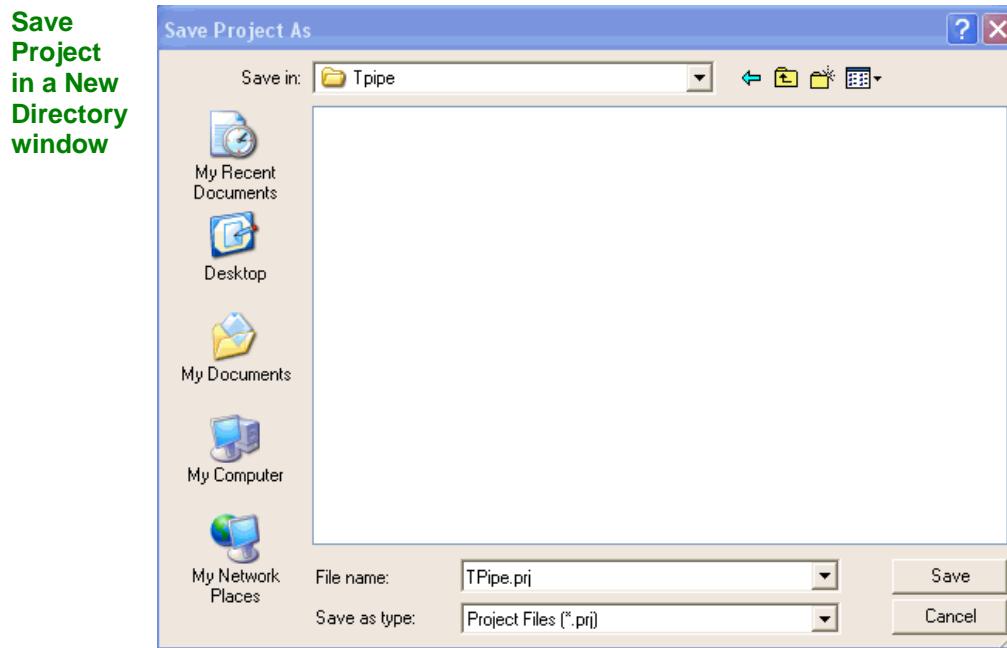
Select ALL in Output Requests section for Displacement (DISP) and Element Strain Energy (ESE). Also select the Case Control Cards as EIGRL1.

Press Apply.

**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. Now enter the project name as **Tpipe** to save all the information in Tpipe directory.



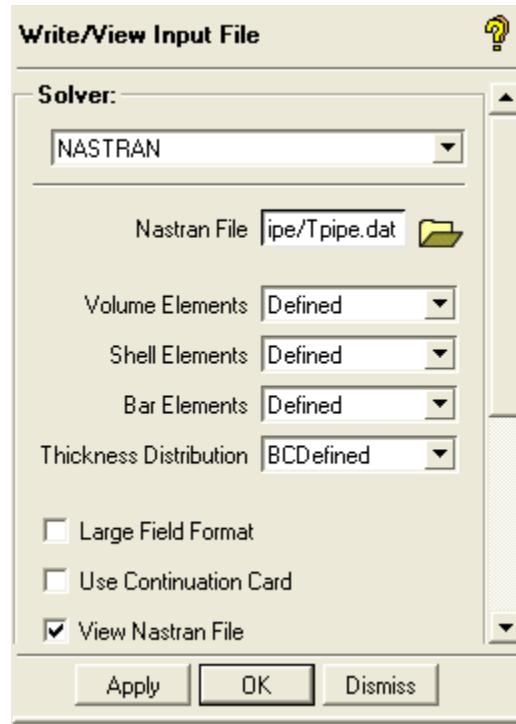
It saves additional fours files Geometry file, Mesh file, Attribute file and Parameter files as Tpipe.tin, Tpipe.uns, Tpipe.fbc and Tpipe.par respectively along with the project file, Tpipe.prj.

e) Write Nastran Input File

Click on (Write/View Input File) icon from **Solve Options Menu bar**.

Feed the Nastran file name as Tpipe.dat and switch ‘On’ View Nastran file as shown and press Apply in **Write/View Input File** window.

**Figure 6-138
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.

f) 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 Menubar** to start the Nastran as shown. 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.

Run Solver window



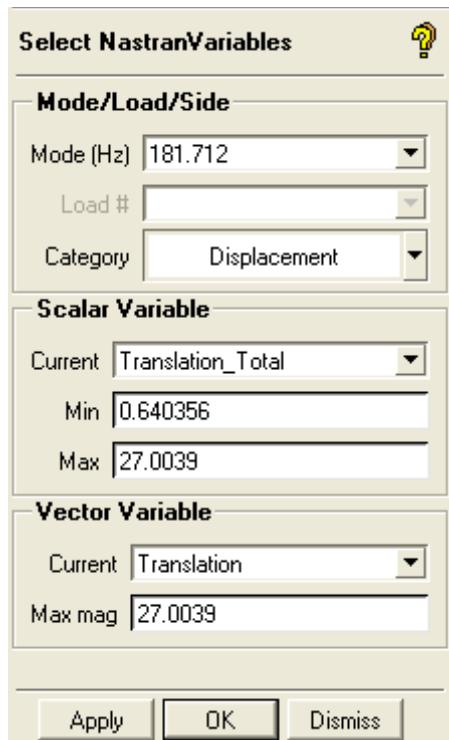
Note: If Nastran doesn't go through, Please refer the **FATAL** error in the file *.f06 and fix the issues accordingly in the dat file (*.dat).

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** menu bar. In Select Nastran Variables window, select **Category** as **Solid** and Current scalar variable as **Translation_Total** as shown.

**Select Nastran
Variables
window**

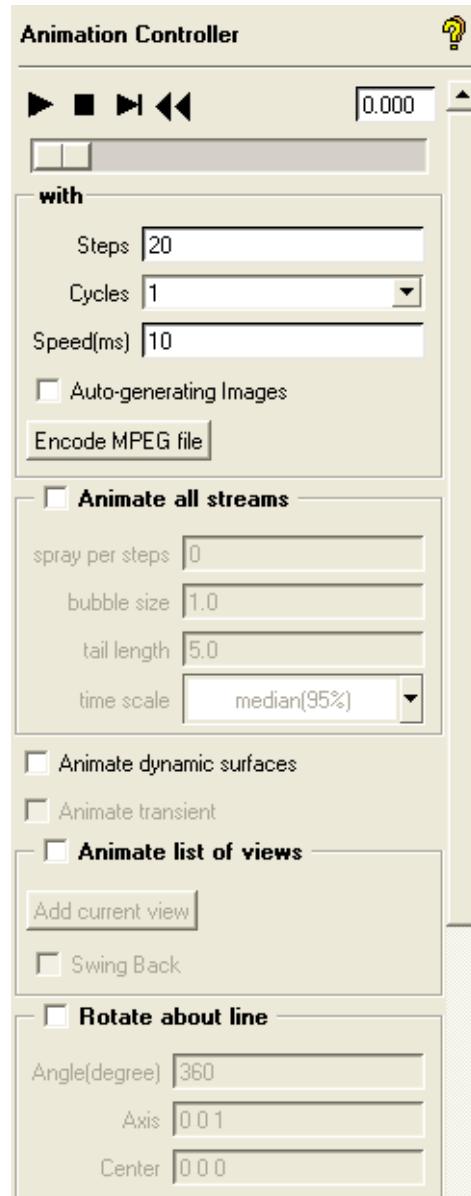


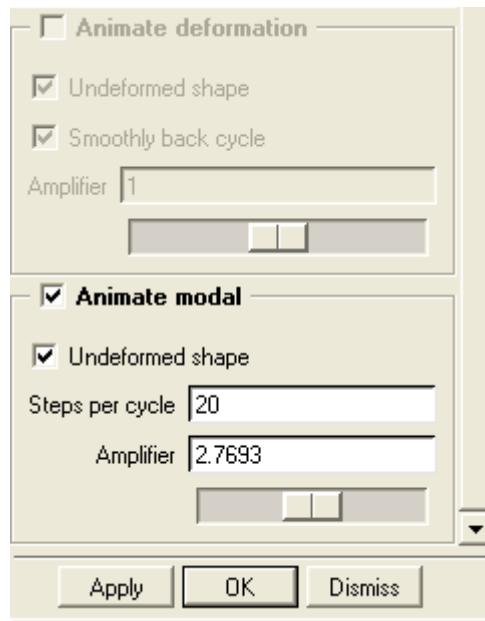
Note: MSC Nastran run obtains Results shown here. Results may differ with those of AI*Nastran run depending on the version.

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.

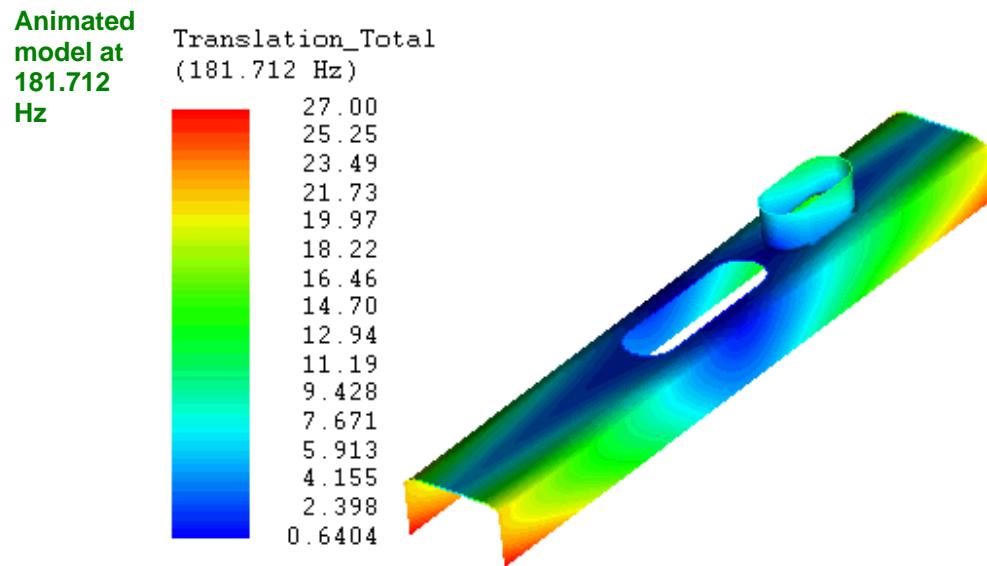


Select **Control All Animation** option from **Post-processing** tab menu bar which will open Animation Controller window as shown here.

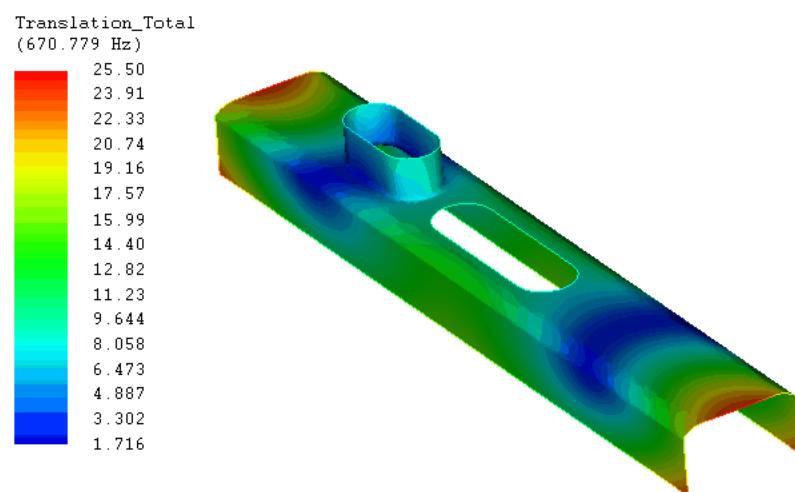
**Animation
Controller**



Set the values as shown above and press (Animate) to view the mode shape as shown in below.



**Figure 6-139
Animated model at 670.779 Hz**



Finally select Exit to quit the post processor.

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	1169
------------------------	--	------

6.4.2: Bar

This exercise explains the writing the input file to solve this Linear Static problem in Nastran and post processing the results of Bar.prj from Bar:Structural Meshing Tutorial.

a) Summary of Steps

Launch AI*Environment and load Project file

Material and Element Properties

 Selection of Material

 Element Properties

Constraints and Loads

 Constraints

 Loads

Solver setup

 Setup Nastran Run

 Write Nastran Input File

 Save Project

Solution and Results

 Solving the Problem

 Post processing of Results

Proceed the Tutorial as the Continue Before Bar:Structural Meshing Tutorials.

b) Open Project File

Open the Project File> Bar.prj as you Earlier Completed in the Structural Meshing Tutorials

Select Settings > **Solver** from Main menu, Select appropriate solver and select Nastran from Common Structural Solver and then Press Apply.

c) 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 Menubar.

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	1170
------------------------	--	------

Define the Material Name as STEEL in **Define Material Property** window shown below.

Material ID can be left as **1**,

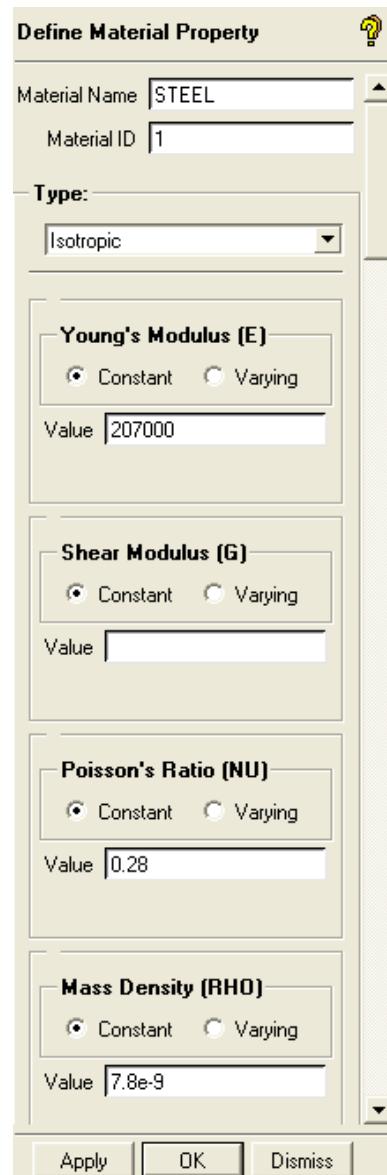
Select **Isotropic** as the type of the Material,

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.

Define Material Property window

d) Element Properties



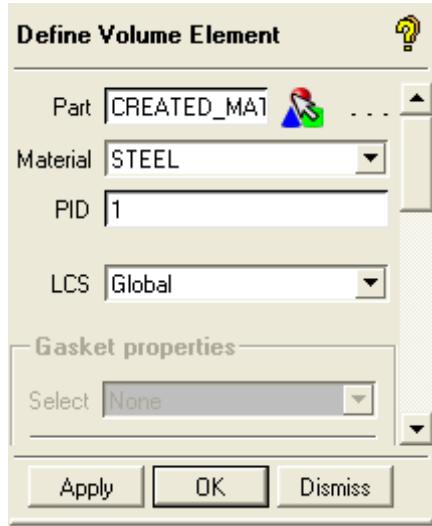
Select (Define 3D Element Properties) from the Properties Menubar.
Set PID as 1 in the **Define Volume Element** window as shown below.
Select Part as CREATED_MATERIAL_2,

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 6-140
Define Volume
Element
window



e) 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

Create  Constraints/Displacements > Click on  icon, the Create Displacement on Surface window as presented below.

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 the figures below, 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 6-141
Create
Displacement on
Surface window

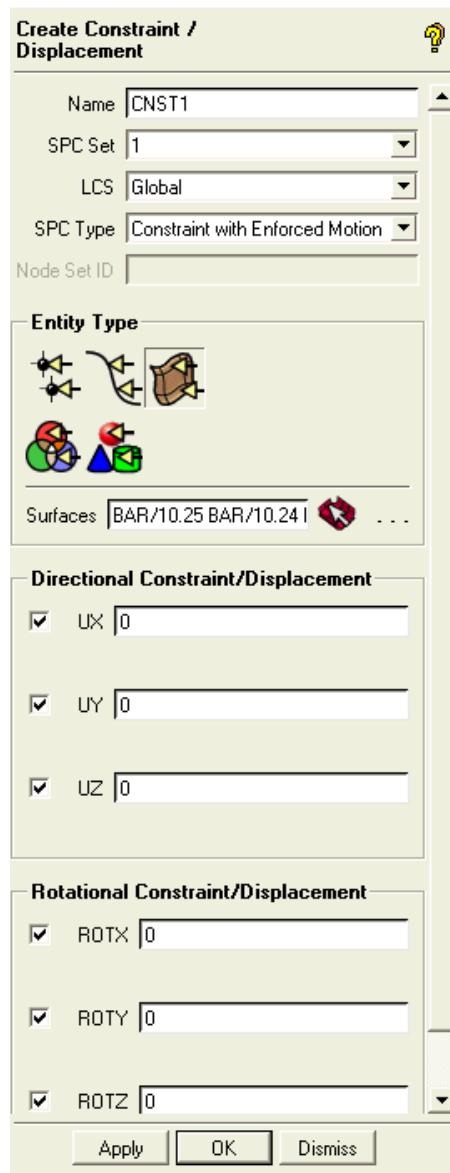


Figure 6-142
Surfaces for
Displacement
and Loads

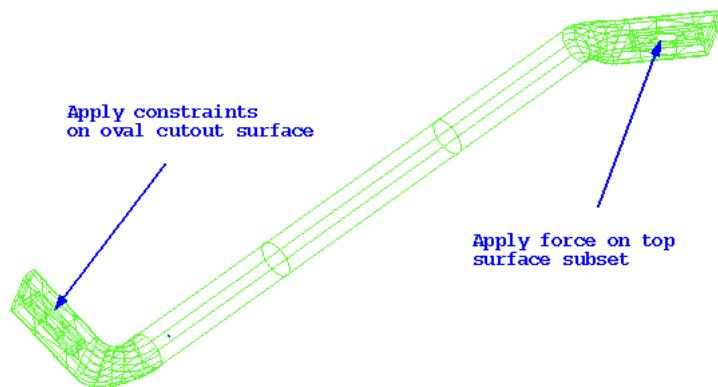
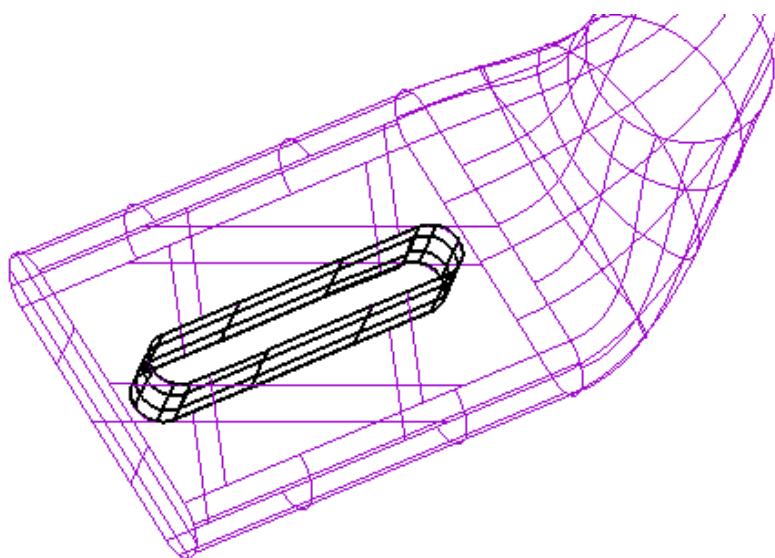
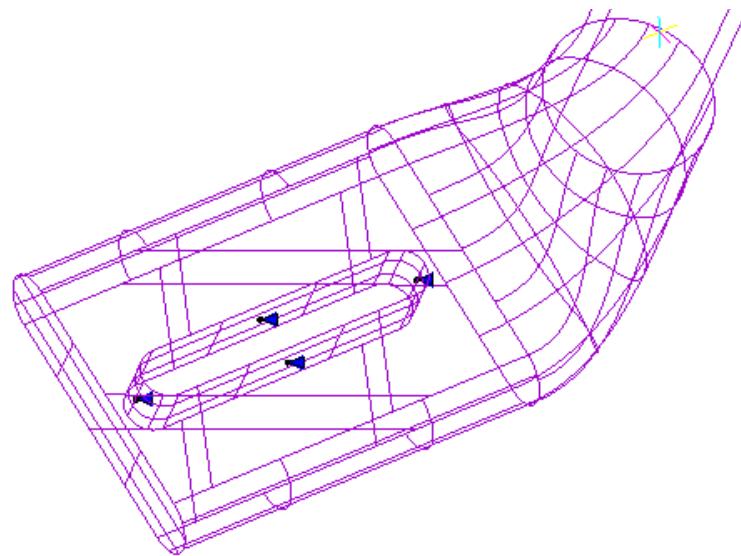


Figure 6-143
Surfaces for
Displacement
and after
applying the
Displacement





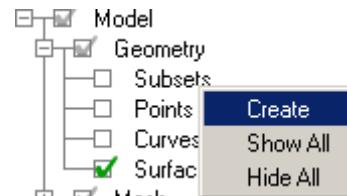
f) Loads

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.

Figure 6-144
Create Subset



Note: Even the Subset is switched 'Off' in the Display Tree it doesn't alter the appearance of the Create Subset window.

This **Create Subset** window pops up as shown below.



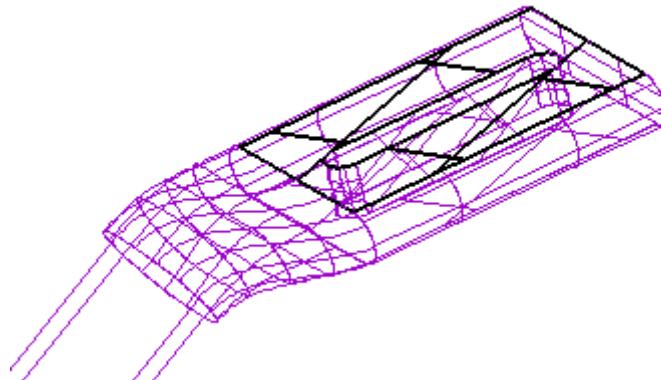
In this window, enter Subset as **LOAD_SURF** and click on (Create) Subset by Selection icon. Click on (Select Geometry) button and select the surface as shown and press Apply. This creates the subset **LOAD_SURF**.

Figure 6-145
Create Subset
window



Switch ‘Off’ Geometry>Subset in the Model Tree.

**Figure
6-146
Surface for
subset**

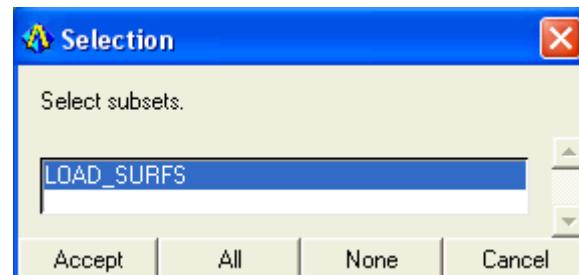


Loads

Create Force > Create Force on subset (Force on Subset) icon. In this window enter Name as **FORCE**. Enter values of **FX** as 0.467, **as 0.2 and **FZ** as -0.862, it is shown.**

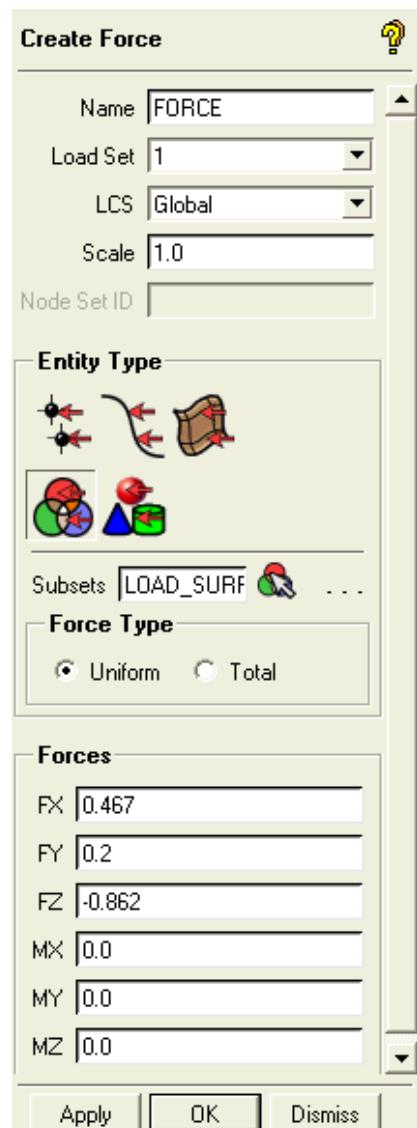
Press Select Subset In the **Selection** window select LOAD_SURFS as shown and press Accept in the Selection window.

**Selection window for
Subset**



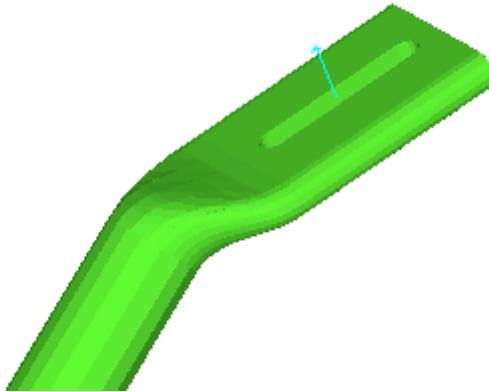
Press Apply in the Create Force on Subset window.

**Figure 6-147
Create Force
on Subset
window**



Note: If the user wants to view the applied Force as shown, he can do so by Swiching Loads 'On' in the Model Tree.

Figure 6-148
Force
applied



g) 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.

Click on  (Setup Analysis Type) icon from Solve Options Menu bar to setup Nastran run which pops up **Setup Analysis Type** window as shown. In the Setup Analysis Type window, do the following:

Select **Run Type** as Linear Static (Sol 101),

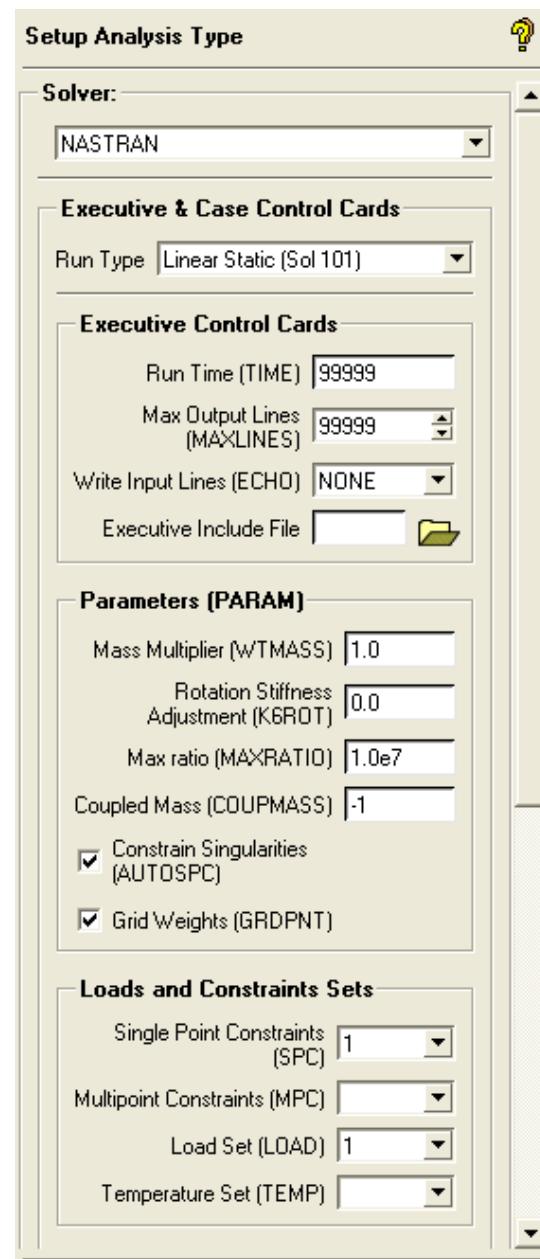
Make sure that Constrain Singularities (AUTOSPC) and Grid Weights (GRDPNT) are turned ON

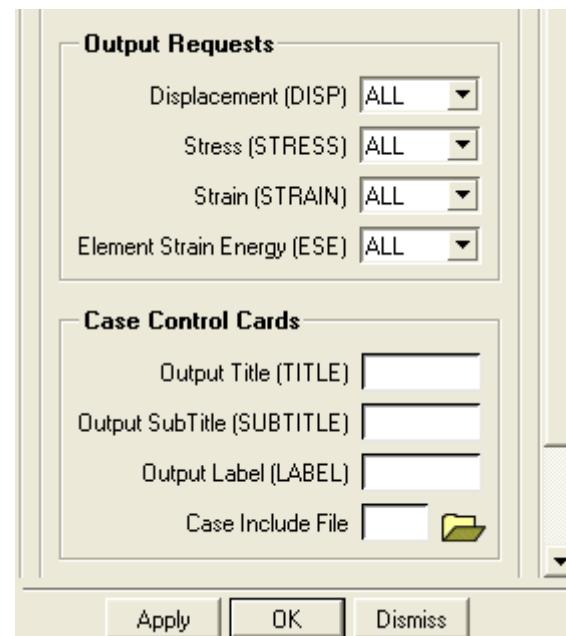
For the Default Sets, select Single Point Constraints (SPC) and Load Set (LOAD) as 1,

In the Output Requests toggle ON Displacement (DISP), Stress (STRESS) and Element Strain Energy (ESE).

In the end, press Apply to complete the setup.

Setup
Analysis Type
window





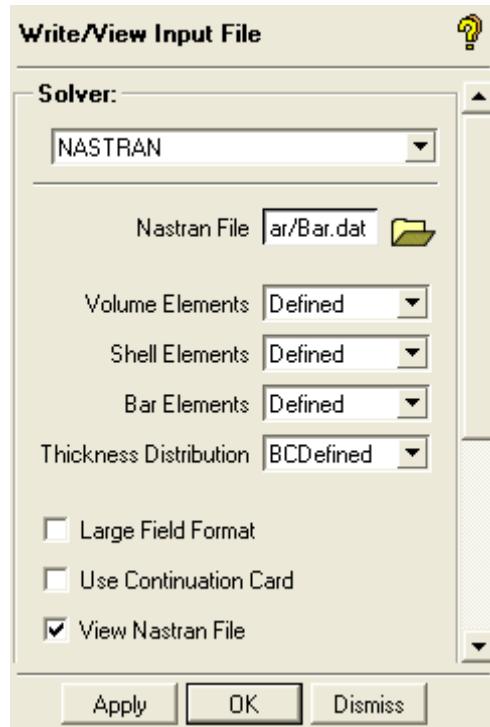
Save Project as Bar.prj

Write Nastran Input File

Click on  (Write/View Input File) icon from the Solve Options Menubar.

Enter the Nastran file name as **Bar.dat** and switch **ON** View Nastran file option in **Write/View Input File** window as shown and press Apply.

Figure 6-149
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.

h) 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 Menu bar to start the Nastran. The Nastran file will be selected by default as **Bar.dat**.

Toggle ‘On’ **Post process and View Results** and press Apply in Run Solver window.

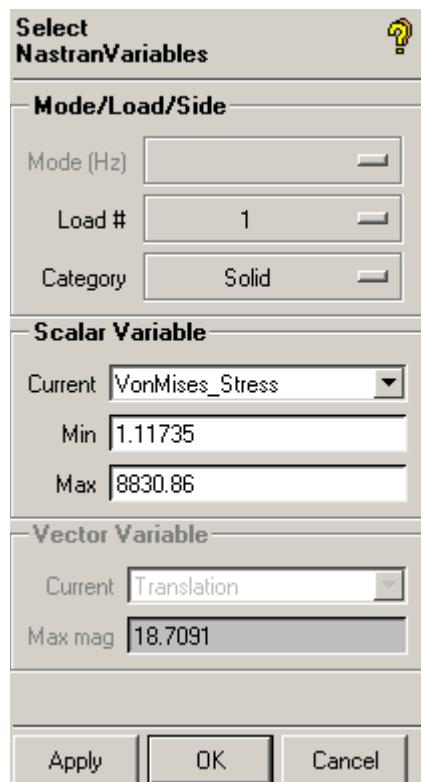


i) Post Processing of Results

After completion of Nastran run, the results will be automatically loaded into the post processor tab.

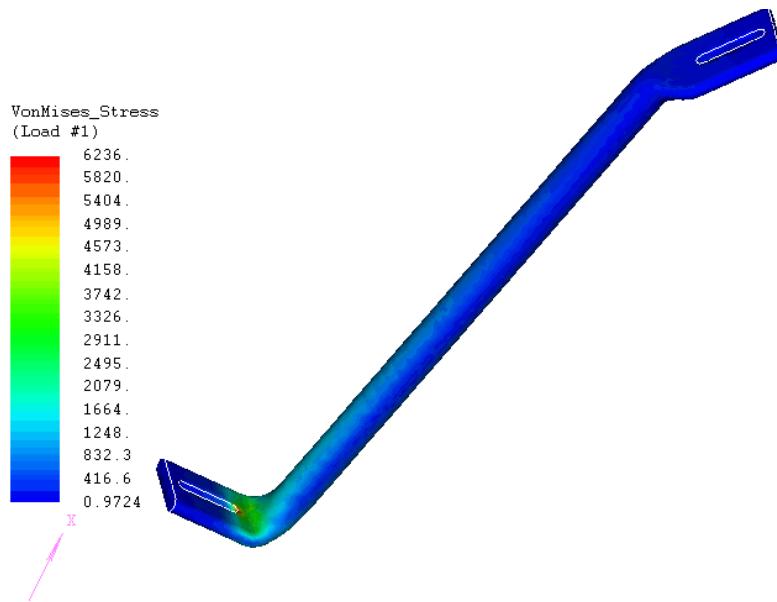
Click on  **Variables** option in **Post-processing** menu bar. In Select Nastran Variables window, select **Category** as **Solid** and Current Scalar Variable as **VonMises_Stress** as shown. The VonMises Stress distribution is shown below.

Figure 6-150
Nastran
Variables
window



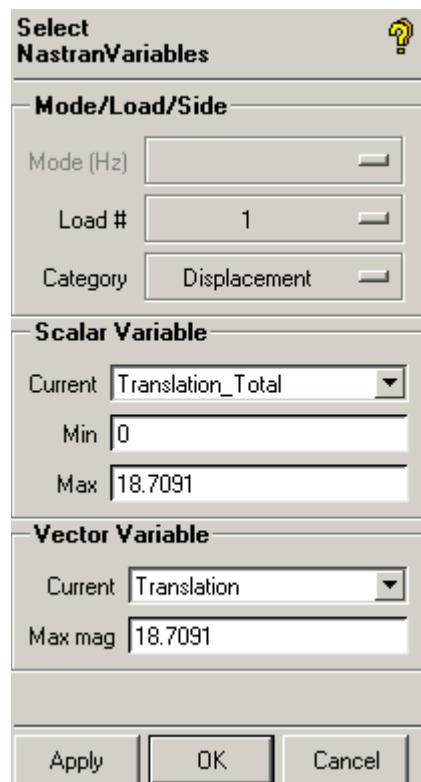
Note: Results shown here are obtained by MSC Nastran run. Results may differ with those of AI*Nastran run depending on the version.

**Figure
6-151
VonMises
Stress
distribution**

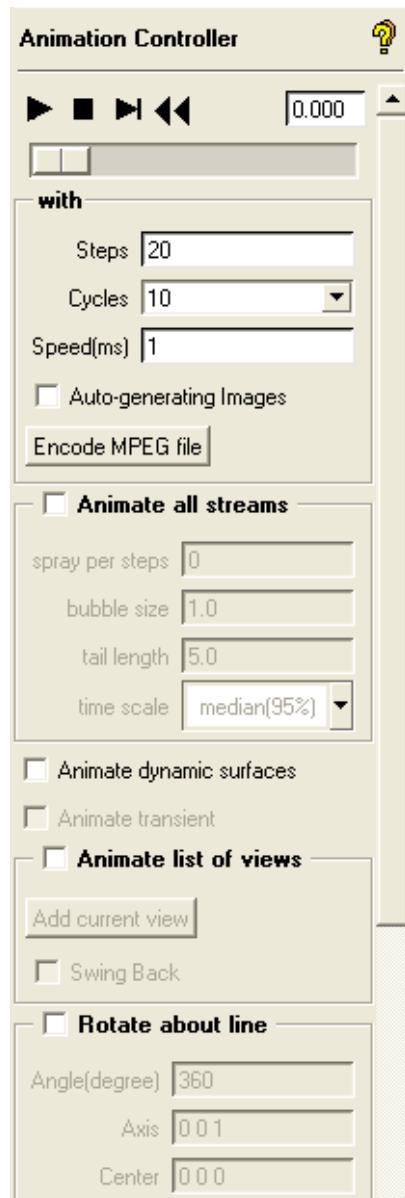


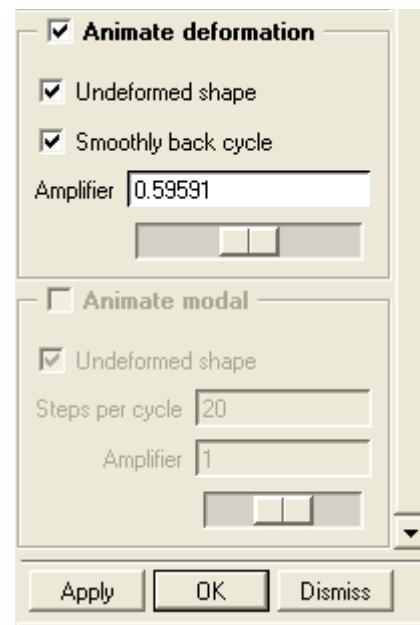
To display mode shape at Total Translation Frequency, select **Category** as **Displacement** and Current Scalar Variable as **Translation_Total** in Select NastranVariables window.

Figure 6-152
Nastran
Variables
window



Select  (Control All Animation) option from **Post-processing** tab menu bar which will open Animation Controller window as shown here.

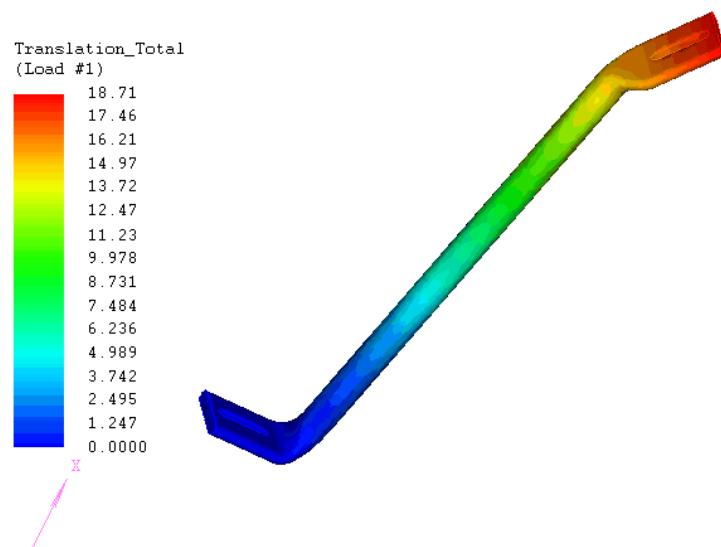
**Animation
Controller window**



Set the values as shown in the figure above and press (Animate) to view the mode shape as shown below.

Finally select **Exit** to quit the post processor.

**Figure
6-153
Animated
Model**



6.4.3: Frame

This exercise explains writing the input file to solve this Linear Static problem in NASTRAN.

a) Summary of Steps

Launch AI*Environment and load geometry file
 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
 Proceed the Tutorial as the Continue Before Frame:Structural Meshing Tutorials.

b) Open Project File

Open the Project File> Frame.prj as you Earlier Completed in the Structural Meshing Tutorials
 Select Settings > Solver from Main menu, Select > Nastran and Press Apply

c) 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

 ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	1192
---	--	------



Select **Create Material Property** icon from **Properties 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 Menu bar.

It will open **Define Line Elements** window as shown.

Select the **Part** as **SEAM_WELD0**,

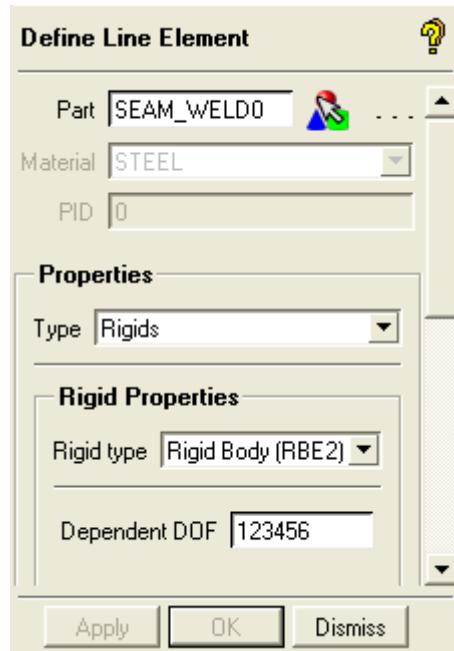
Select Type as **Rigid**,

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

Define Line Element window



Repeat these steps to define properties for other Line elements of Spot welds and BOLT HOLES.

Shell Element Properties

Select  (Define 2D Element Properties) icon from the Properties Menubar.

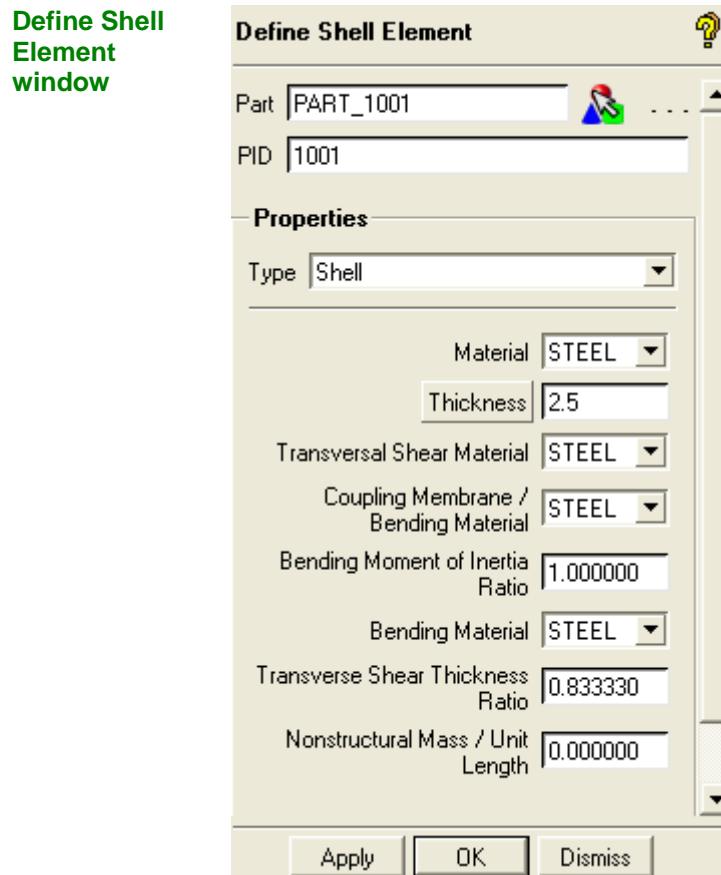
Set PID as 1001, in Define Shell Elements window.

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.



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
-----------	------	-----

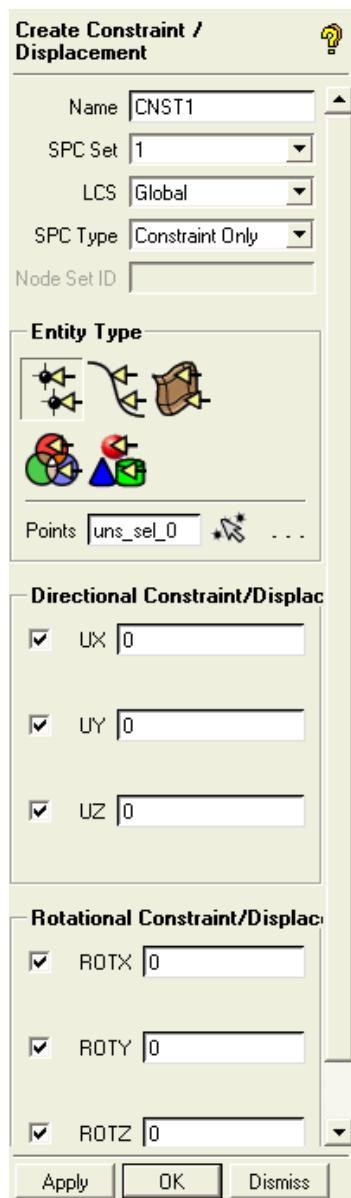
d) Constraints and Loads

Constraints

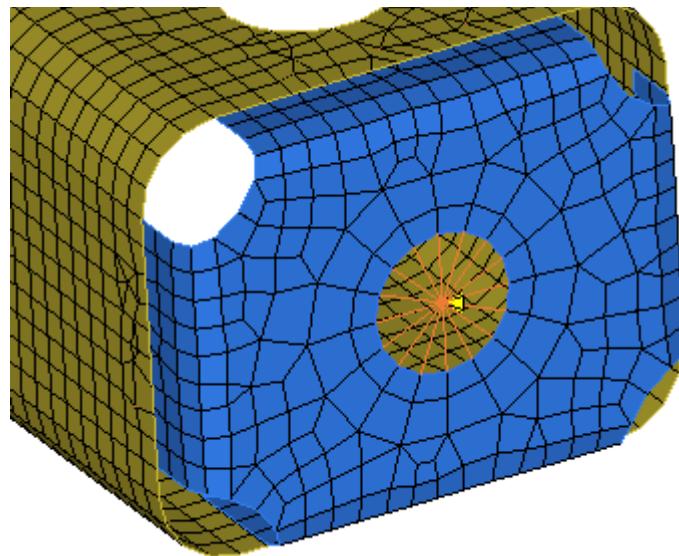
Create   Constraints/Displacements >  (Constraints/Displacements on Point) icon, the window shown here.

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_1001 and PART_1002 as shown 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 6-154
Create
displacement on
Point window



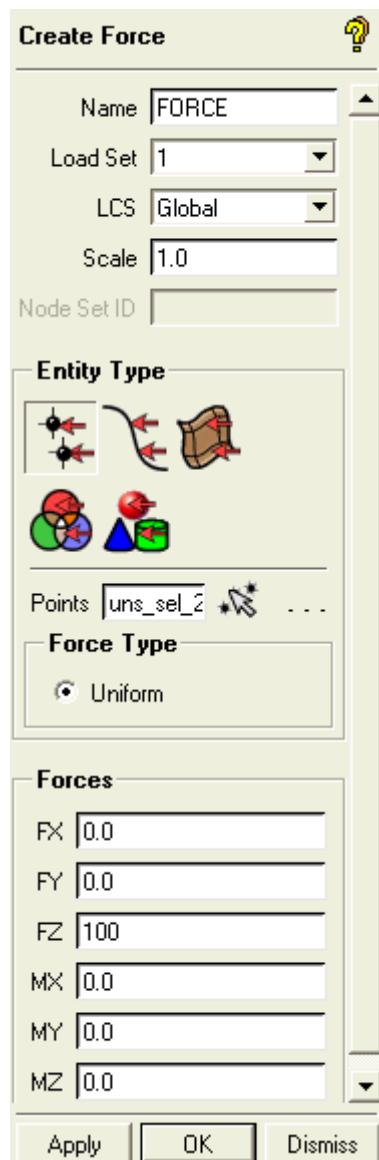
**Figure
6-155
Constraint
on Point**



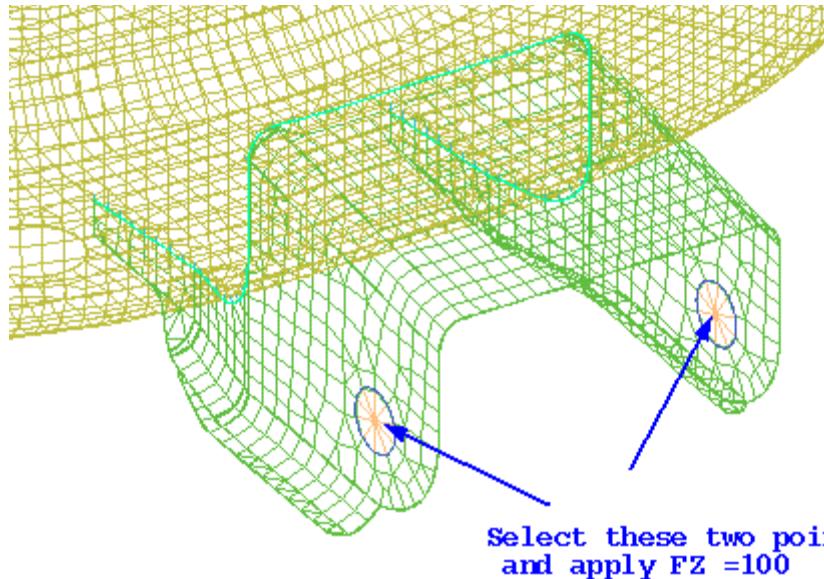
Loads

Create Force > Click on (Force on Point) icon from Loads Menu bar, which will pop up **Place Force on Point** window as shown above. In this window enter Name as **FORCE** and select the two center points of the bolt connections on **PART_1003** as shown and enter a value of **100** for FZ and press Apply.

**Figure 6-156
Force on Point
window**



**Figure
6-157
Points for
Load
application**



e) Solver Setup

Setup Nastran Run

First, user should select the appropriate solver before proceeding further.

Click on  (Setup Analysis Type) icon from Solve Options Menu bar to setup Nastran run to do Linear Static that pops up Setup Analysis Type window as shown.

In the **Setup Analysis Type** window, enter the following:

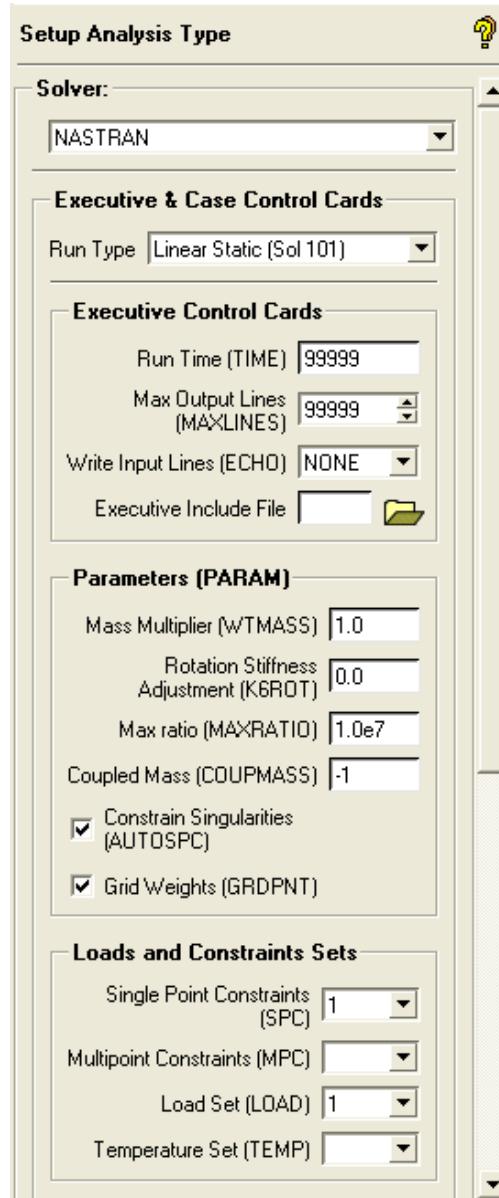
Select Run Type as Linear Static (Sol 101)

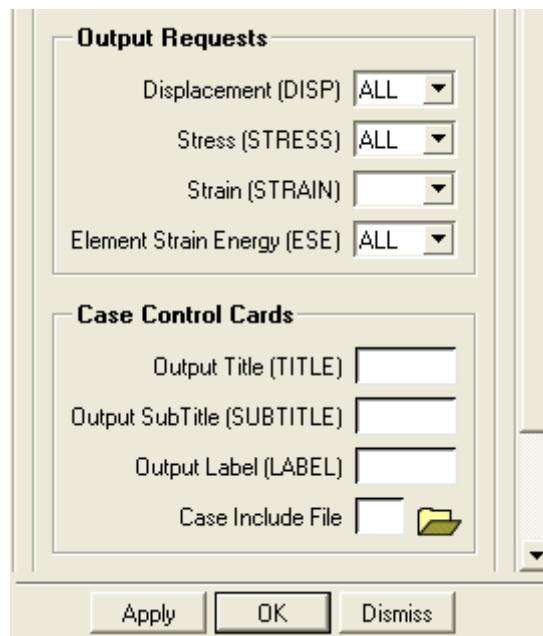
Make sure that **Constrain Singularities** (AUTOSPC) and Grid Weights (GRDPNT) are turned ‘On’.

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.

**Setup
Analysis Type
window**



Save Project

Through File > Save Project As option, create new directory Frame as said in earlier tutorials.

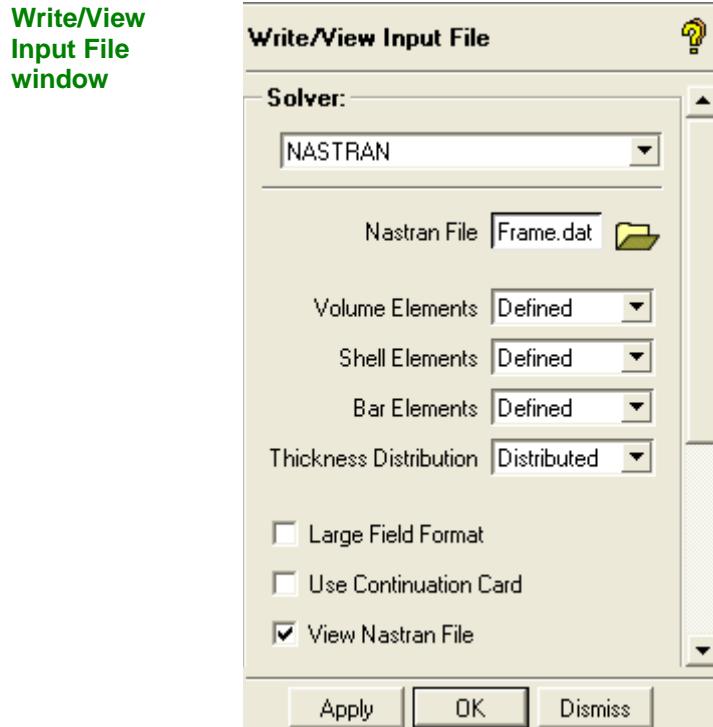
Enter **Frame** as project name and press **Save** to save all these information in this directory

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 Menu bar, which will open **Write/View Input File** window presented here.

Give the Nastran file name as **Frame.dat** and switch ‘On’ View Nastran file and press Apply.



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.

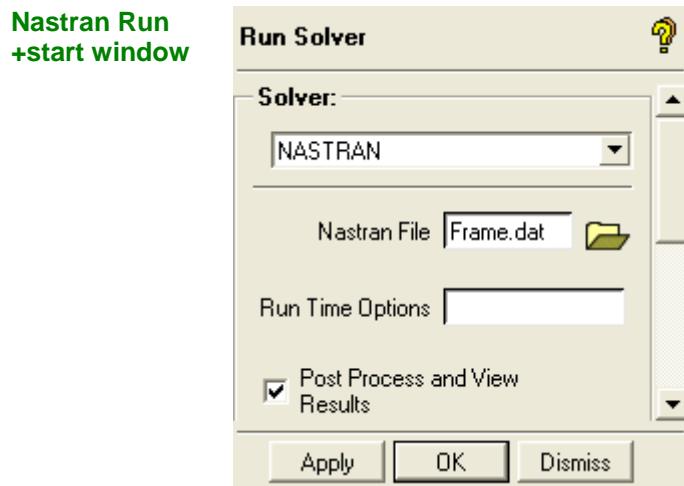
f) Solution and Results

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 Menubar to start Nastran as shown. 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.

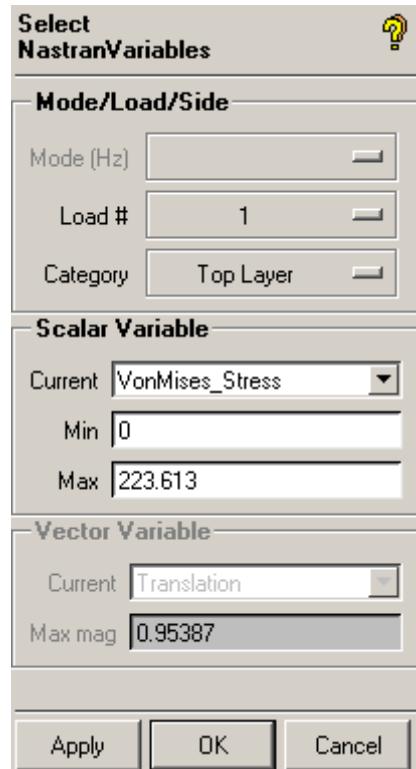


Post Processing of Results

After completion of Nastran run, the results will be automatically loaded into the post processor tab.

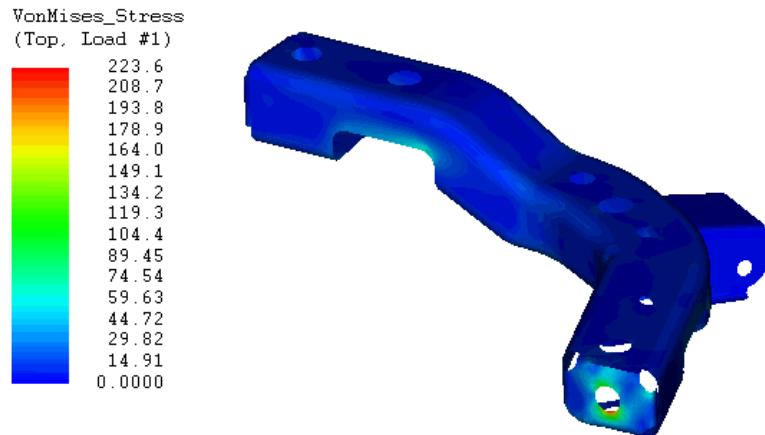
Click on  **Variables** option in **Post-processing** menu bar. In Select Nastran Variables window, select **Category** as **Solid** and Current Scalar Variable as **VonMises_Stress** as shown. The VonMises Stress distribution is shown below.

**Figure 6-158
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
6-159
VonMises
Stress
distribution**



To display mode shape at Total Translation Frequency, from the Nastran Variables window as shown. Select Category as **Displacement** and variable as **Translational_Total**.

Figure 6-160
Nastran
variables
window

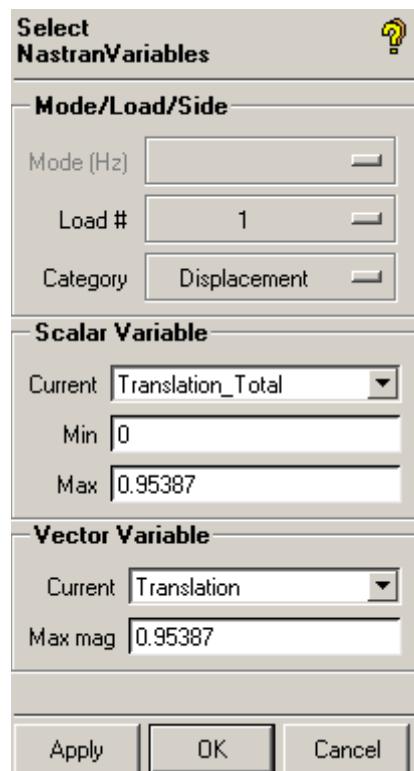
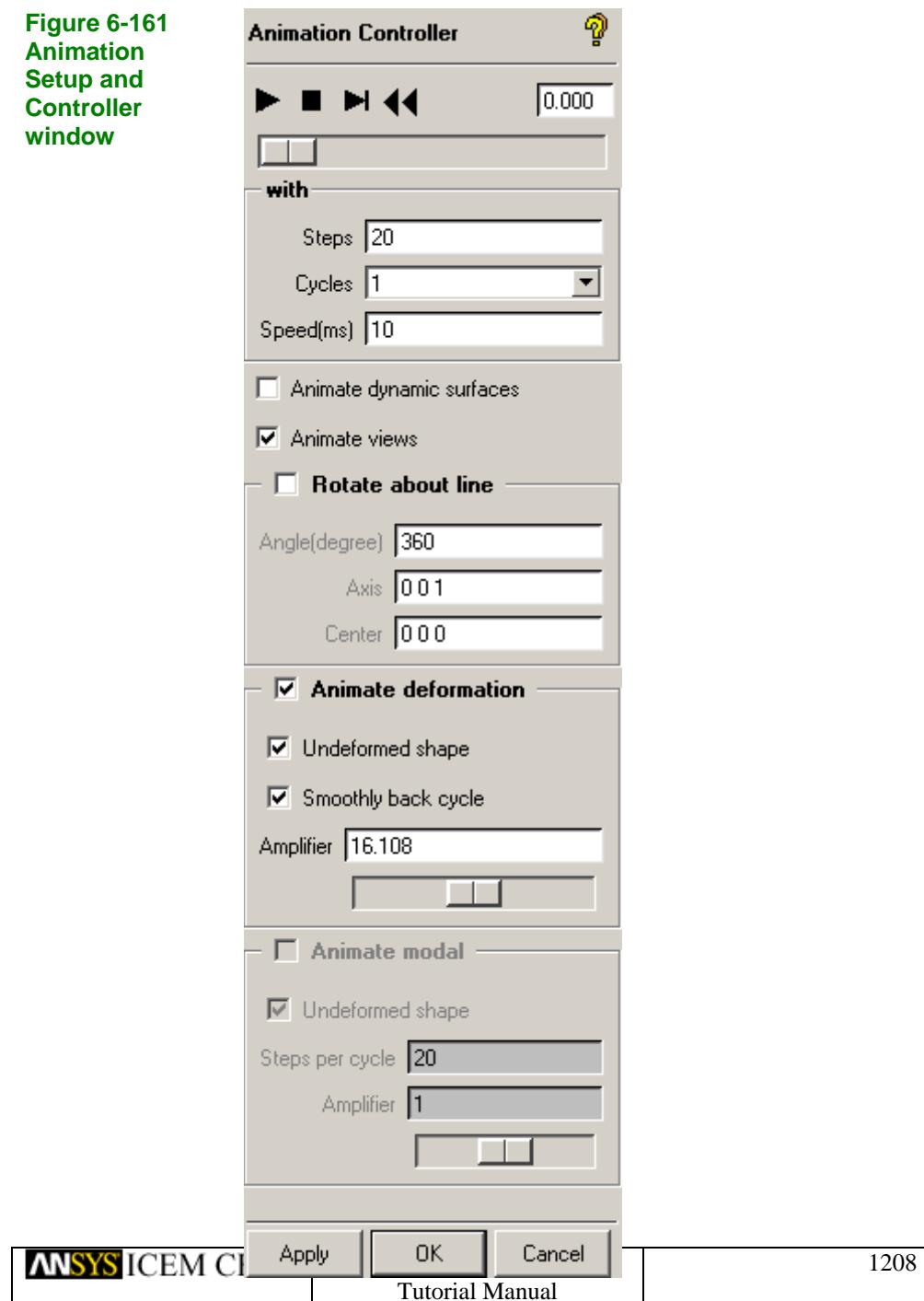


Figure 6-161
Animation
Setup and
Controller
window

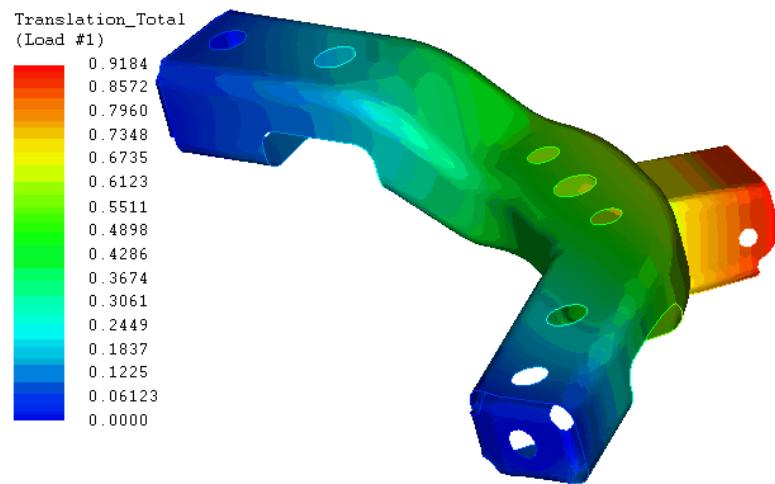


Select (Control All Animation) option from **Post-processing** tab menu bar which will open Animation Controller window as shown in.

Set the values as shown above and press (Animate) to view the mode shape as shown in the figure below.

Finally select **Exit** to quit the post processor.

**Figure
6-162
Animated
model**



6.4.4: Connecting Rod

This exercise explains writing the input file to solve this Linear Static problem in Nastran and Post Processing the results.

a) Summary of Steps

Launch AI*Environment and load geometry file

Materials and Element Properties

 Selection of Material

 Element Properties

Subsets

 Subset1

 Subset2

Constraints and Loads

 Constraints

 Loads

Solver setup

 Setup Nastran Run

 Write Nastran Input File

 Save Project

Solution and Results

 Solving the Problem

 Post processing of results in Visual3p

Proceed the Tutorial as the Continue Before Connecting Rod:Structural Meshing Tutorials.

b) Open Project File

Open the Project File> Connecting_Rod.prj as you Earlier Completed in the Structural Meshing Tutorials

Select Settings > Solver from Main menu, Select > Nastran and Press Apply

c) Material and Element Properties

Before applying Constraints and Loads on elements, define the type of material and assign properties to the elements.

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	1210
------------------------	--	------

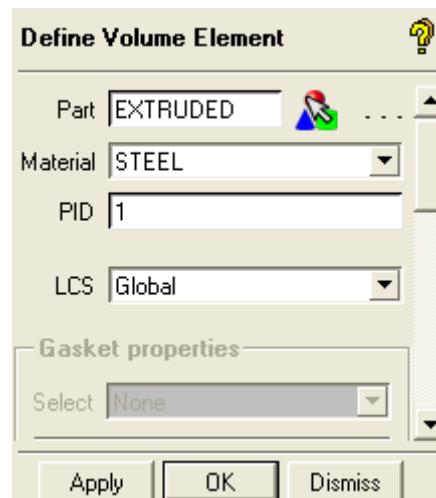
Selection of Material

Select  (Create Material Property) icon from **Properties Menu bar**. 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 Menu bar**. Set PID as **1** in the **Define Volume Element** window as shown below. Select the Part as **EXTRUDED**. Select Material as **STEEL**. Press **Apply**.

Figure 6-163
Define Volume
Element
window



d) 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 group of surfaces or elements.

Subset0

In Display Tree, click mouse right button on **Subset** under Mesh and select Create option. This will pop up **Create Subset** window as shown below.

Ensure that all the geometry entities are turned ‘Off’ from Display Model Tree.



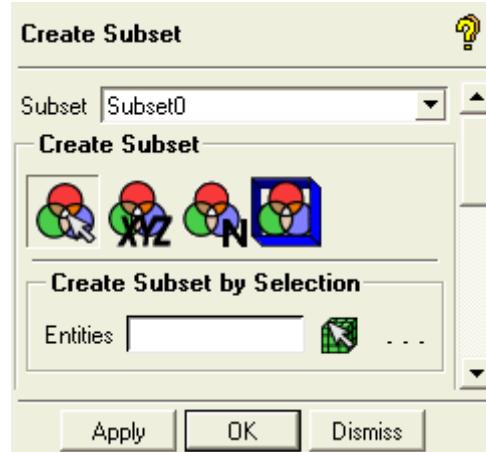
Enter Subset as **Subset0** and click on **Create Subset by Selection** icon in **Create Subset** window.

Toggle ‘Off’ Points, Lines and Volume and Toggle ‘On’ Shells under Mesh in the Display Tree.

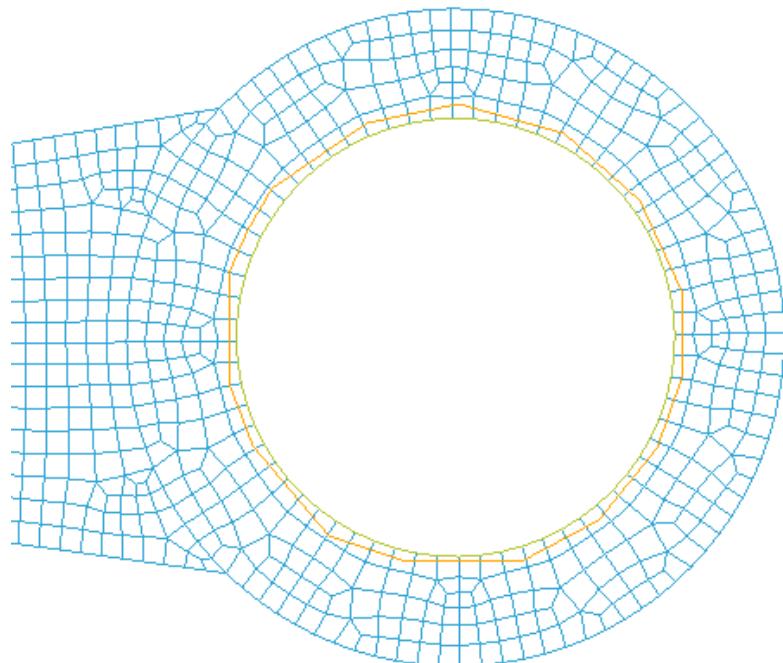


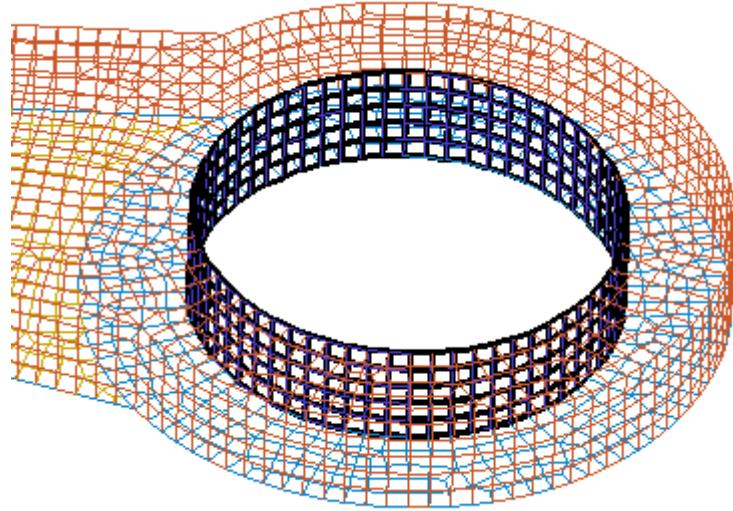
To select the elements on Crank end for this subset click on **Select Element(s)** button and press “p” from key board (ensure that the mouse cursor is in display window) which allows to select the shell elements as shown below by drawing a polygon and press Apply.

Figure 6-164
Create Subset
window



**Figure
6-165
Elements
election
by
polygon
and
elements
selected
for
Subset1**





Subset1

In Display Tree, click mouse right button on **Subset** under Mesh and select Create option. As shown in the **Create Subset** window in the figure below, 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 **Subset1** and click on  (Create Subset by Selection) icon in Create subset window.

To select the elements on Piston end for this subset click on  (Select Elemn(s)) button and press "p" from key-board (ensure that the mouse cursor is in display window), which allows selecting the Shell elements by drawing a polygon, and press Apply.

Figure 6-166
Create Subset
window

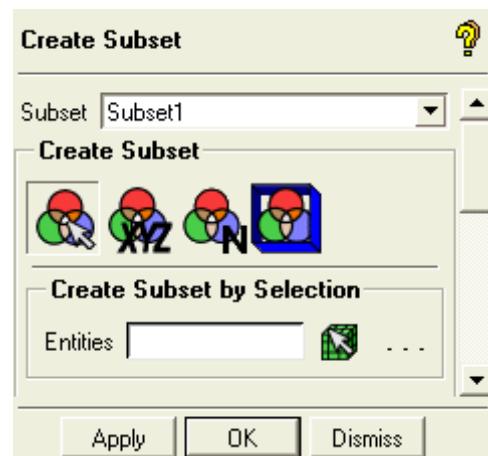
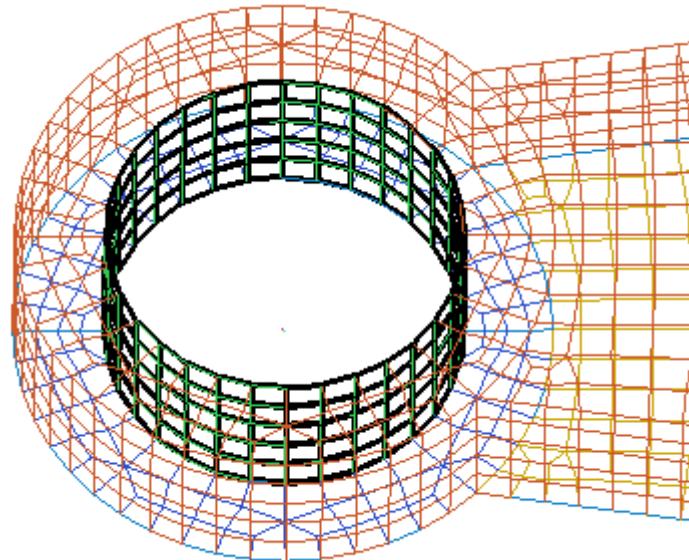


Figure
6-167
Elements
selected
for
Subset2

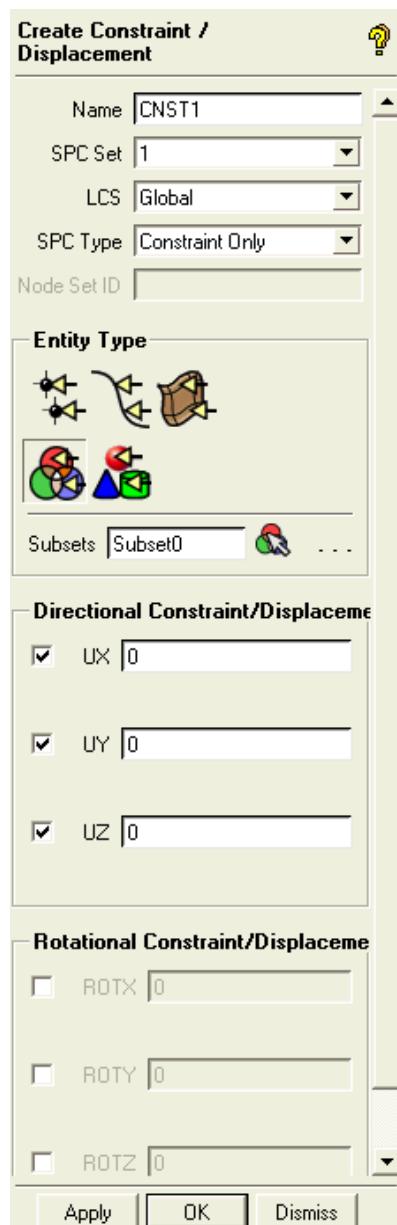


e) Constraints and Loads

Create  Constraints/Displacements >  Create Constraints/Displacements on Subset, the window given below. 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 and press Apply.
Turn '**Off**' **Displacement** display from Model Tree.

Figure 6-168
Create
Displacement on
Subset window



Loads

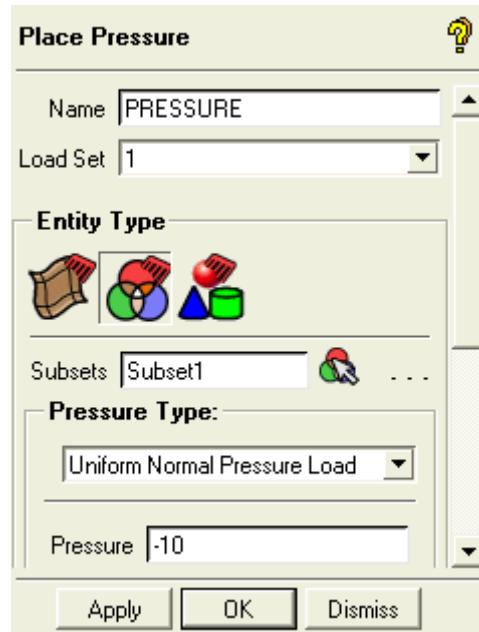
Loads > Place Pressure > Place Pressure on Subsets from window shown here.

Enter Name as **PRESSURE**.

Click on (Select Subset) button and select **Subset2** for subsets as shown.

Enter a value of “**-10**” (negative value) for Pressure and press Apply.
Turn ‘Off’ Loads display from Display Model Tree.

Figure 6-169
Place Pressure
on Subsets
window



f) Solver Setup

Setup Nastran Run

First, user should select the appropriate solver before proceeding further.

Click on (Setup Analysis Type) icon from **Solve Options Menu bar** to setup Nastran run to do Linear Static Analysis that will pop up **Setup Analysis Type** window as shown.

In the **Setup Analysis Type** window, do the following:

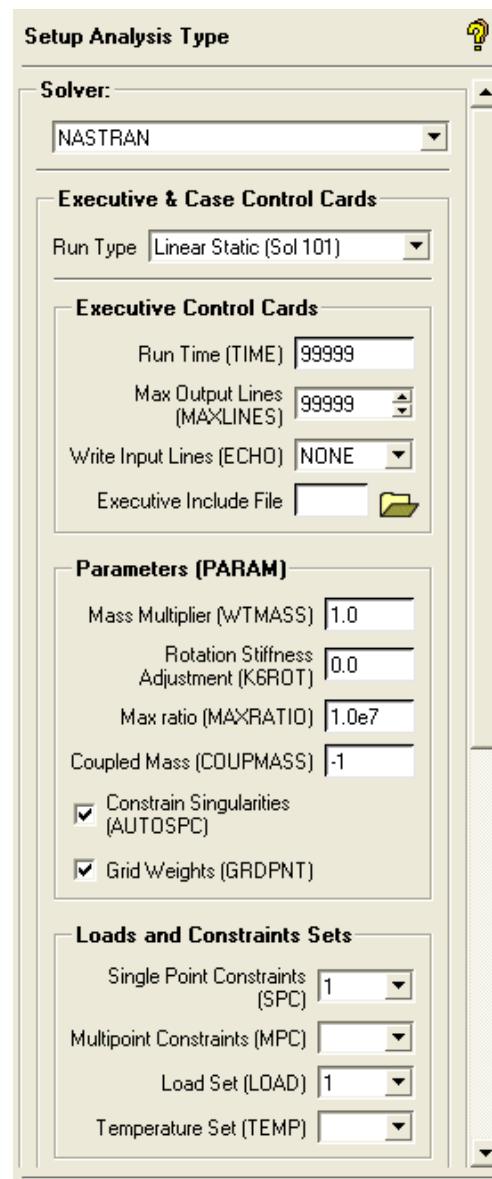
Select **Run Type** as Linear Static (Sol 101),

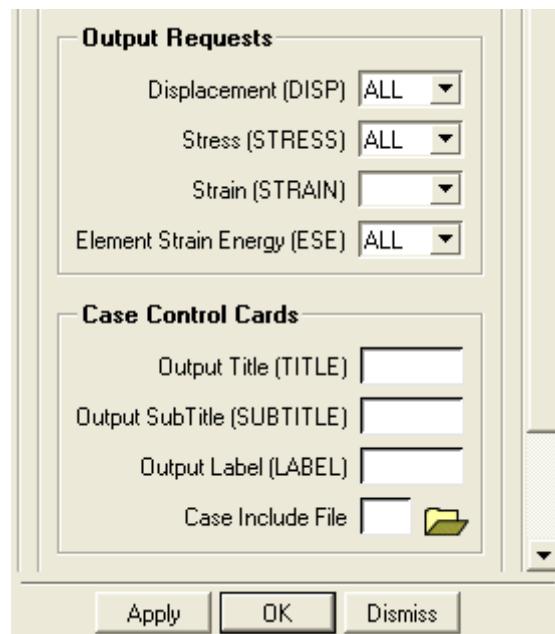
Make sure that **Constraint Singularities** (AUTOSPC) and **Grid Weights** (GRDPNT) is turned **ON**.

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.

**Setup
Analysis Type
window**

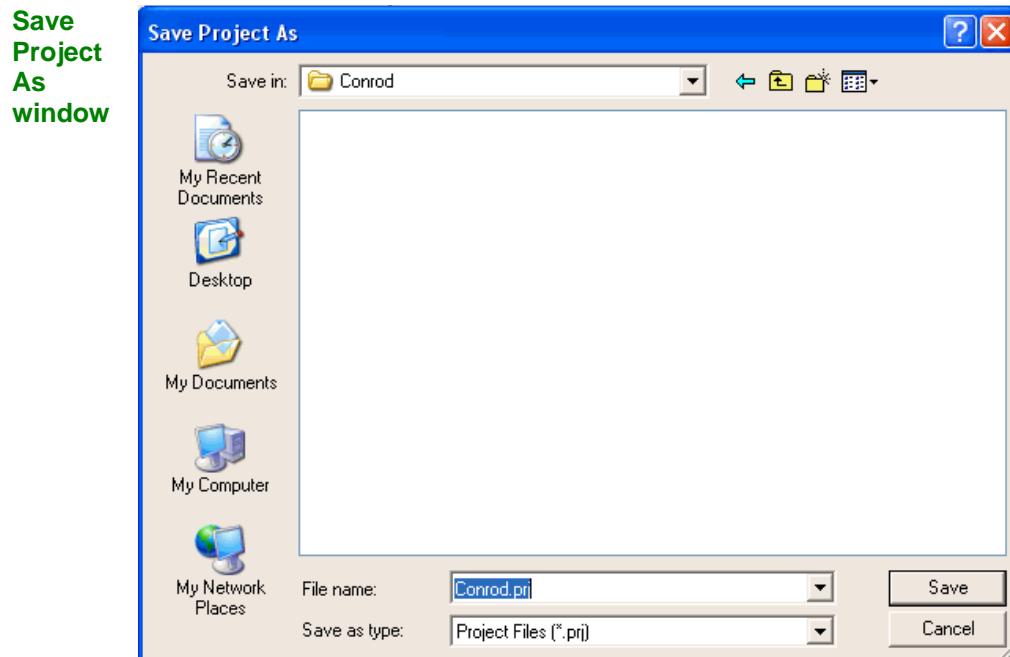


Save Project

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 here.

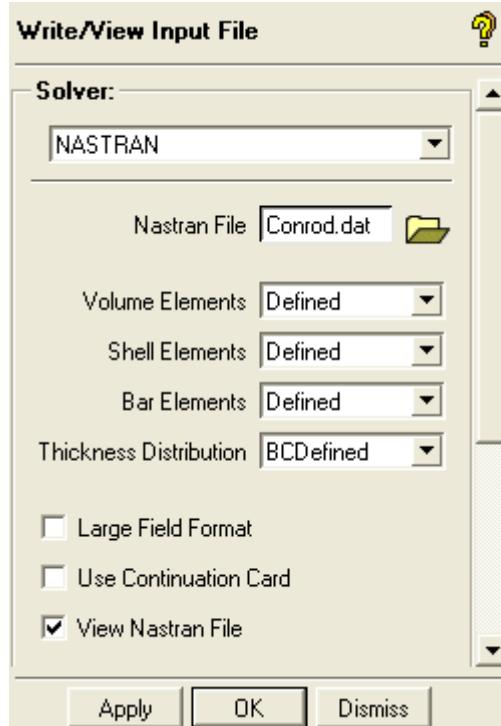
It will save four files, geometry file, mesh file, attribute file and parameter files as Conrod.uns, Conrod.fbc and Conrod.par respectively along with the supplied project file - Conrod.prj.



Write Nastran Input File

Click  (Write/View Input File) icon from **Solve Options Menubar**. Enter the Nastran file name as **Conrod.dat** and switch **ON** View Nastran file as shown in **Write/View Input File** window presented here and press **Apply**.

**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.

g) 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 Menubar** to start Nastran as shown in the figure below. 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.

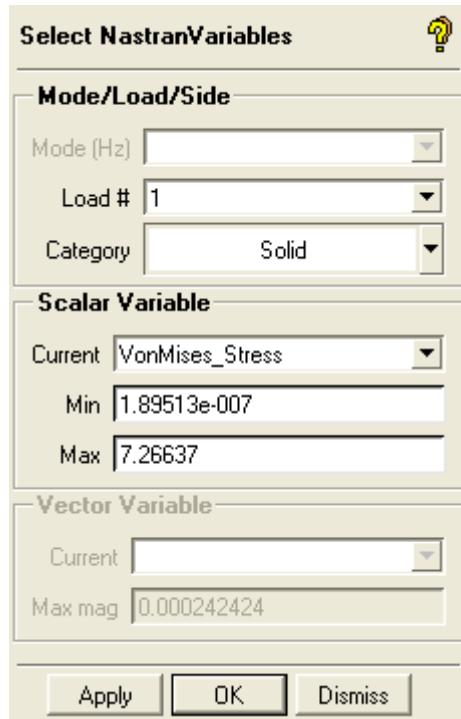
Run Solver
window



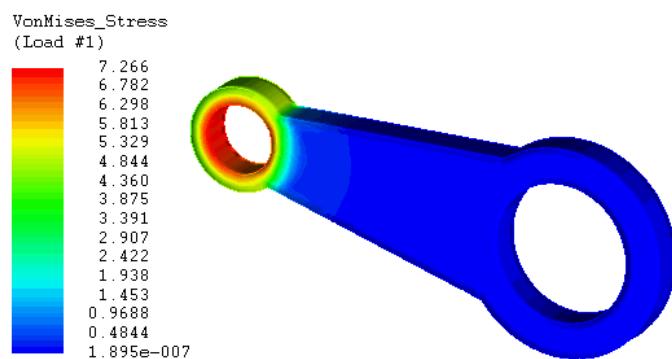
Post Processing of Results

After completion of Nastran run, the results will be automatically loaded into the post processor Visual3p.

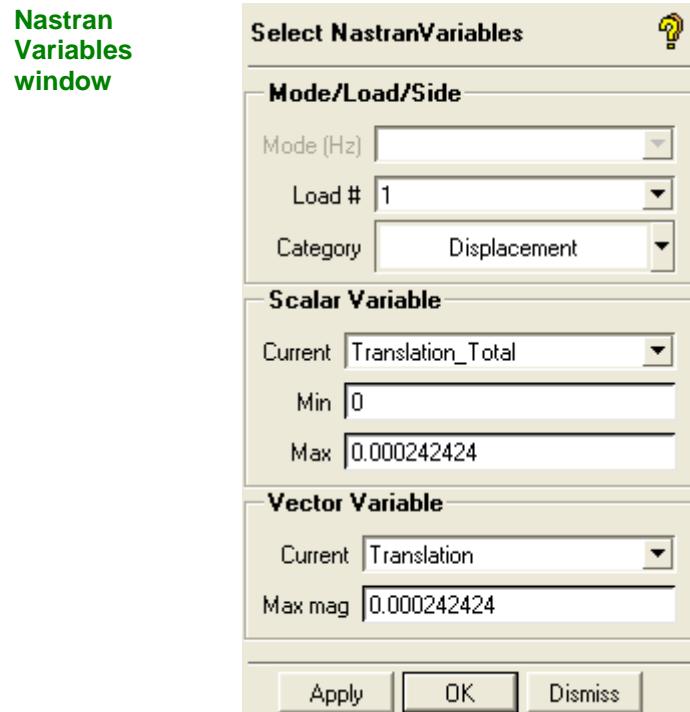
Click on Variables option in **Post-processing** menu bar. In Select Nastran Variables window, select **Category** as **Solid** and Current Scalar Variable as **VonMises_Stress** as shown here. The VonMises Stress distribution is shown below.

**Nastran
Variables
window**

Note: The results shown here are obtained by 'MSC Nastran' Solver.

**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.

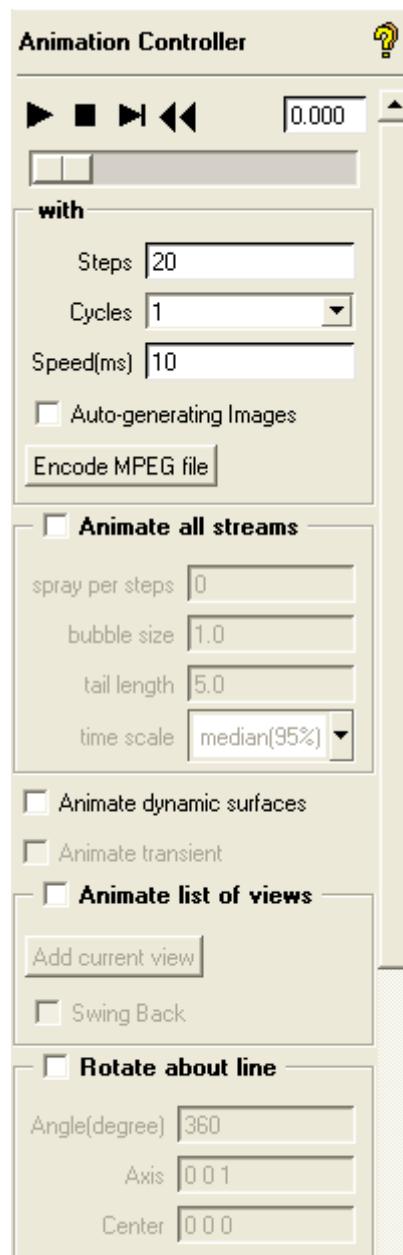


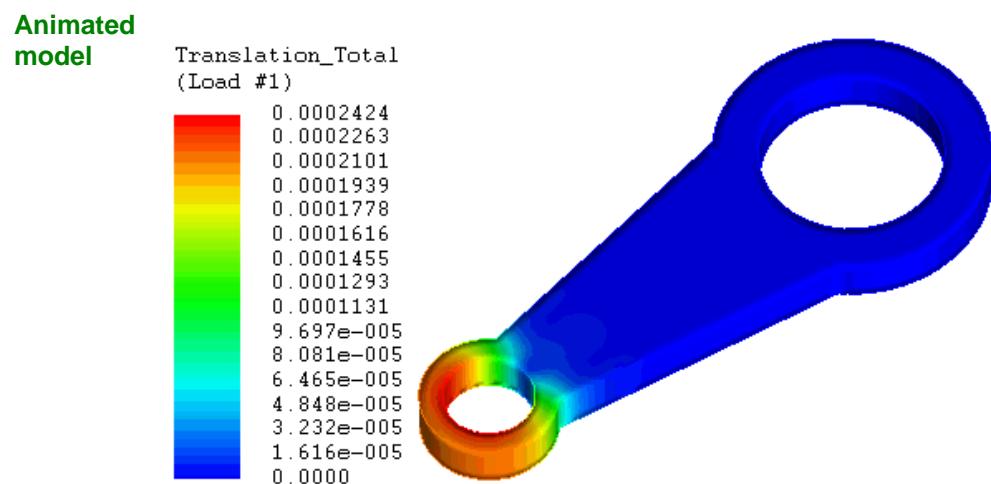
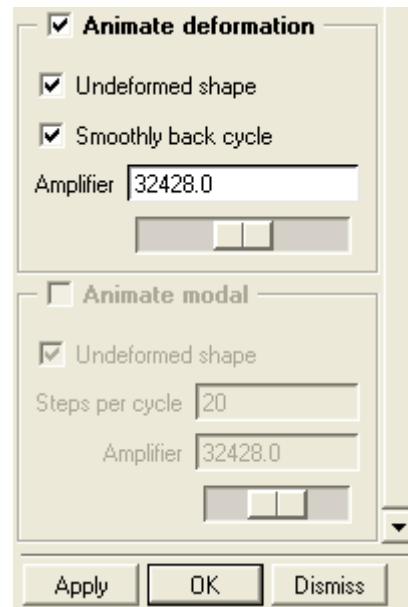
Select (Control All Animation) option from **Post-processing** tab menu bar which will open Animation Controller window as shown in. Set the values as shown below and press (Animate) to view the mode shape as shown .

Finally select **Exit** to quit the post processor.

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	1227
------------------------	--	------

**Animation
Setup and
Controller
window**





6.4.5: 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.

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

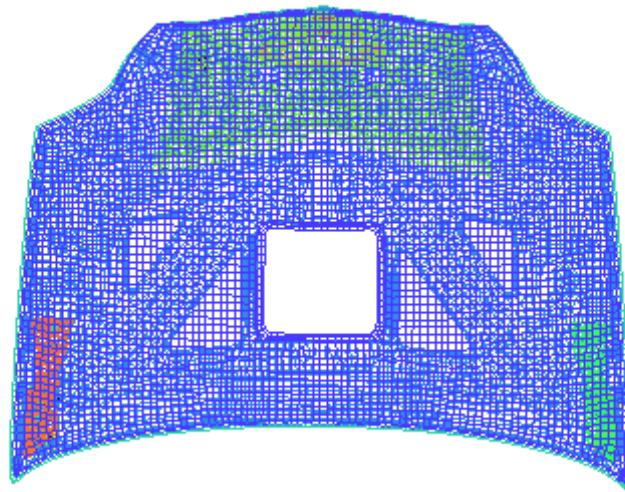
a) Launch AI*Environment and Import Data

Launch the AI*Environment from UNIX or DOS window. The input files for this tutorial can be found in the Ansys installation directory, under `../v110/docu/Tutorials/AI_Tutorial_Files`. Copy the Hood.dat file to our working directory and open it with File > Import Mesh > Nastran which pops up the window immediately below. The figure below shows the imported data in AI*Environment.

**Import Nastran
File window**



**Figure
6-170
Hood
model**



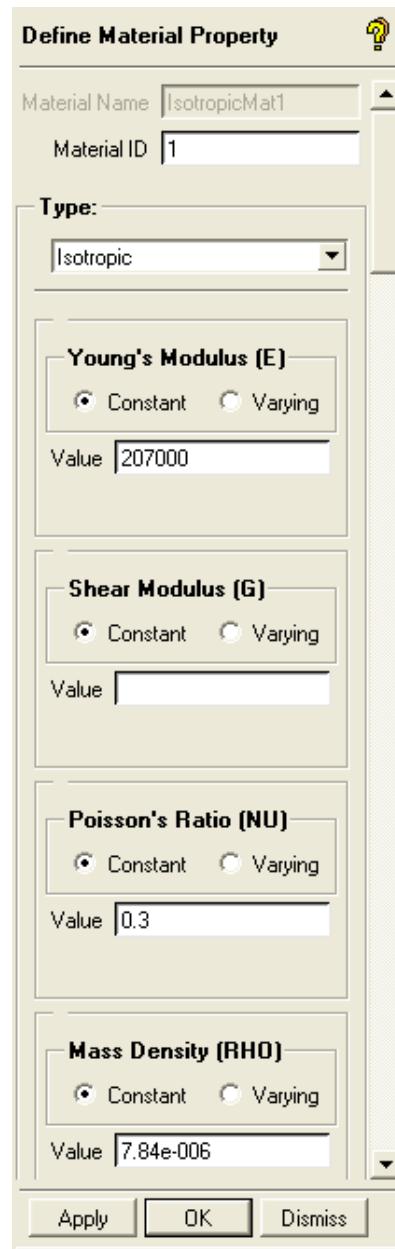
Select Settings > Solver as LS Dyna and press Apply

b) Data Editing

Verification of Imported data

Expand **Material Properties** in Model Tree by clicking on +, select **IsotropicMat1**. To open **Define Material Property** window as shown, double click on the selected Material (IsotropicMat1) with left mouse button.

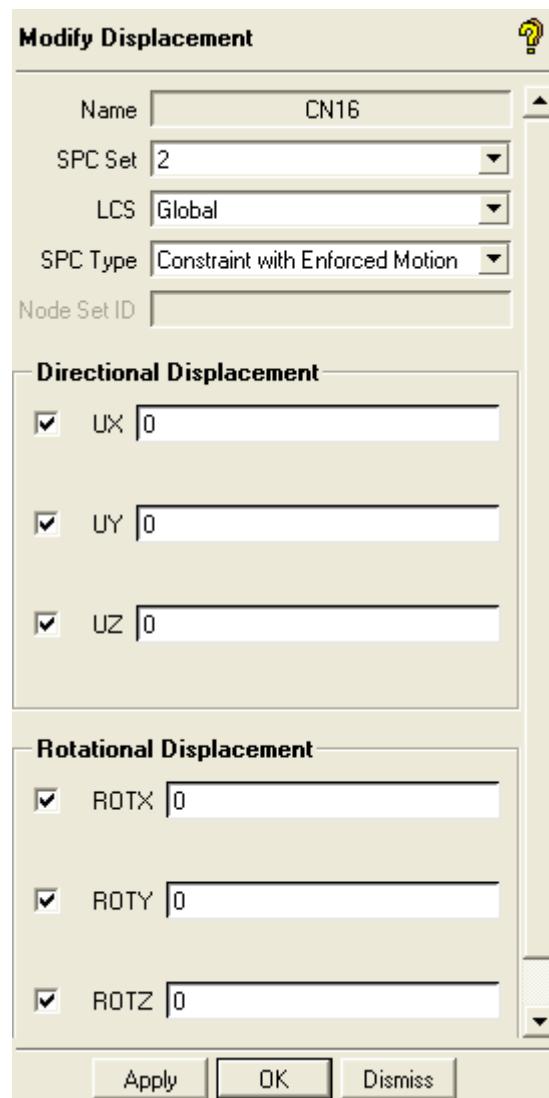
Figure 6-171
Define Material Property



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 here.

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	1233
------------------------	--	------

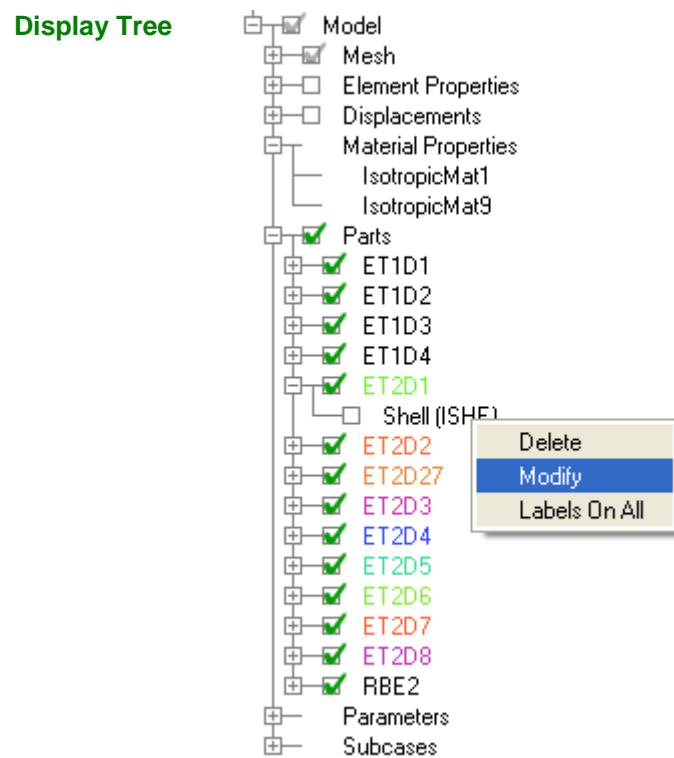
**Figure 6-172
Modify
Displacement
window**



Modifying thickness

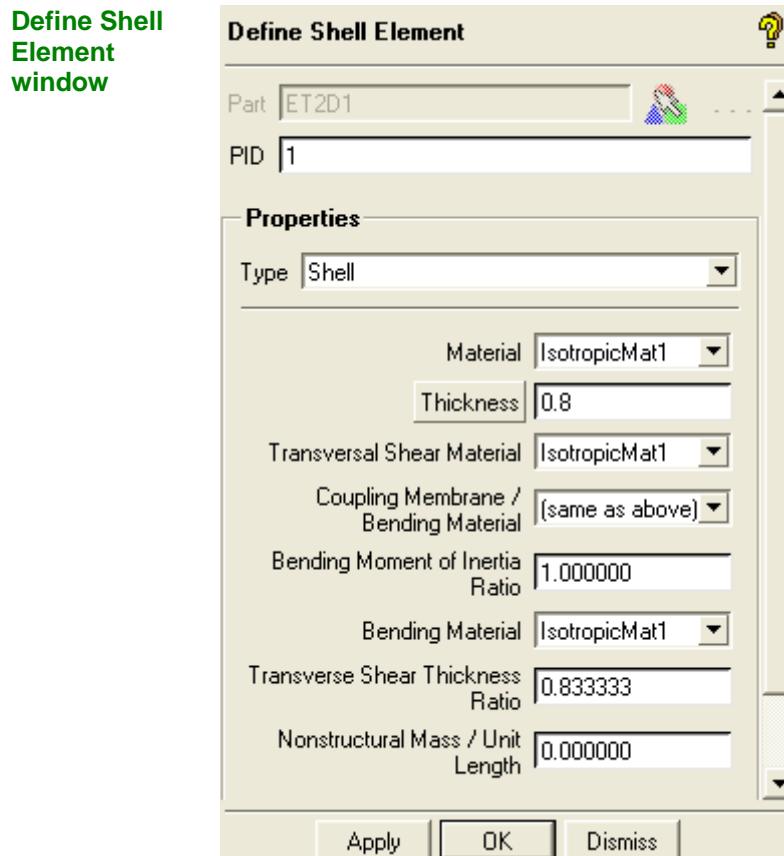
Expand the **Part** option in Model Tree by clicking on + and also expand **ET2D1** as shown below.

Right Click on ET2D1>Surface Properties>Modify.



The window that pops up is shown below.

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**

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.

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.

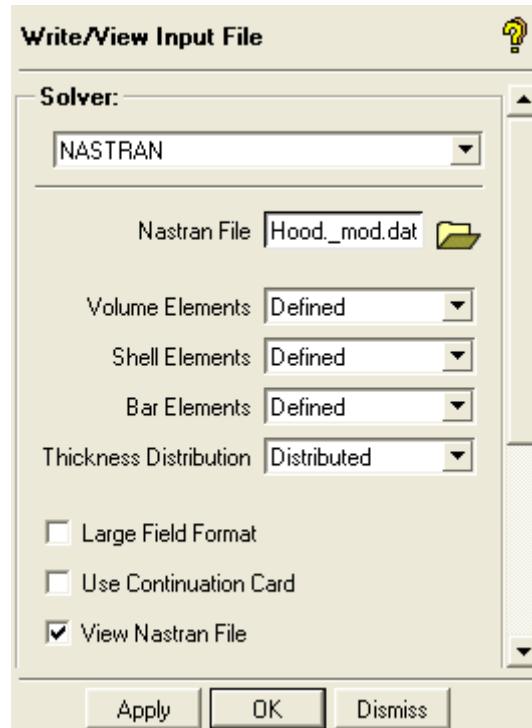
Save Project As window

c) Write Nastran Input

First, user should select the appropriate solver before proceeding further.

Click  (Write/View Input File) icon from the Solve Options Menu bar. Enter the Nastran file name as **Hood_mod.dat** and switch **ON** View Nastran file in **Write/View Input File** window as shown and press Apply.

**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.

d) 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 Menubar to start Nastran with Nastran Input File window given. Supply Nastran file as **Hood_mod.dat**.

Toggle ON Post process and View Results and press Apply in Run Solver window.

Run Solver window

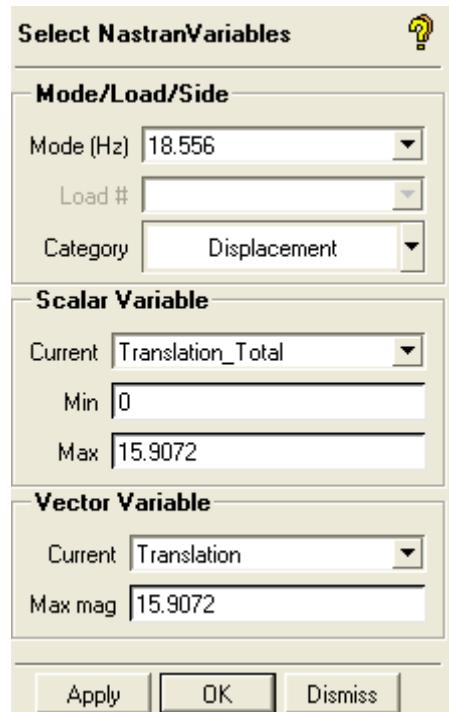


Post Processing of Results

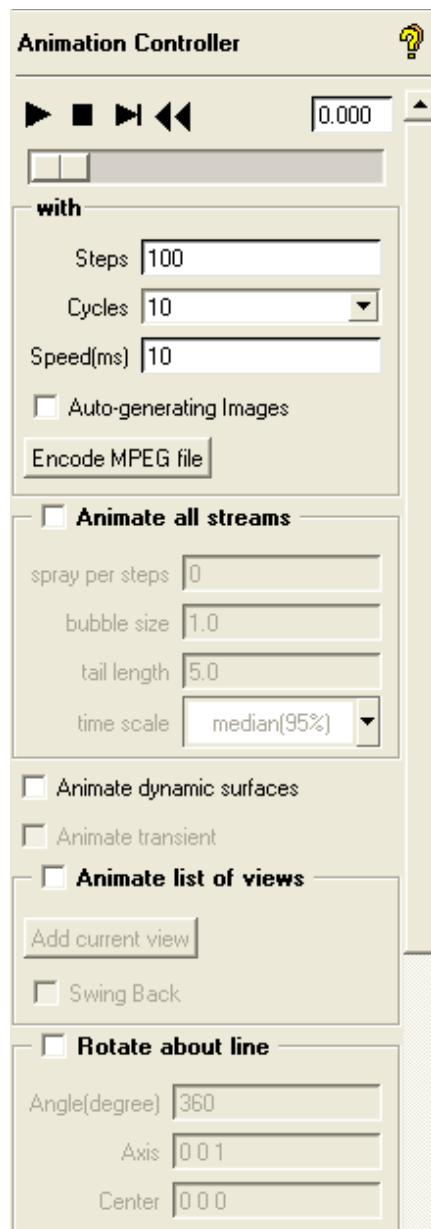
After completion of Nastran run, the results will be automatically loaded into the post processor Visual3p.

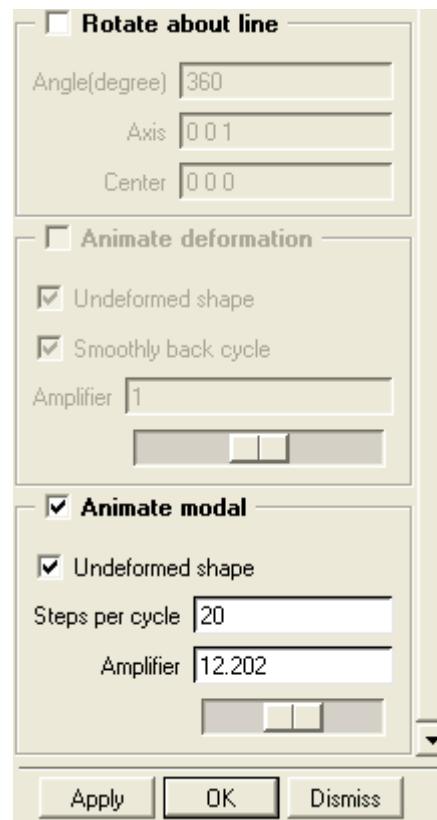
Click on  **Variables** option in **Post-processing** menu bar. In Select Nastran Variables window, set Scalar Variable as **Translation_Total**, as shown below, and press **Apply**.

Select Nastran Variables window



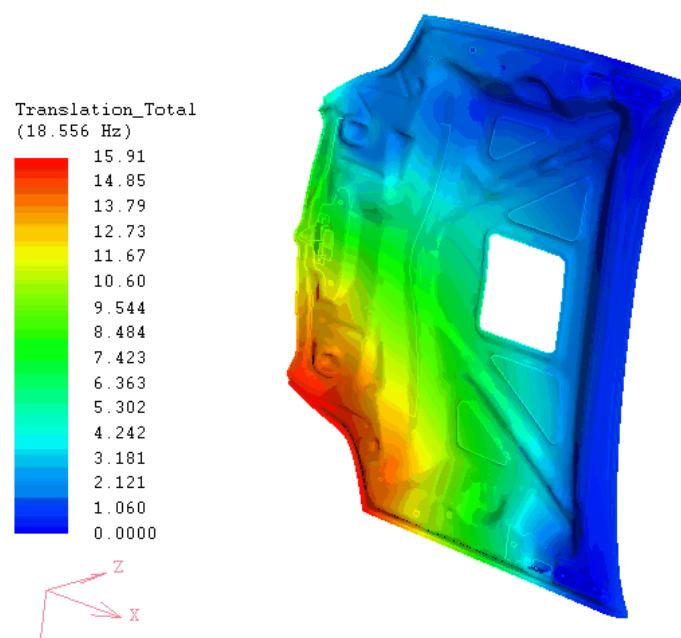
Note: Results shown here are obtained by MSC Nastran run. Results may differ with those of AI*Nastran run depending on the version.

Animation Setup and Controller window



Select (Control All Animation) option from **Post-processing** tab menu bar which will open Animation Controller window as shown below. Set the values as shown in the figure above and press (Animate) to view the mode shape as shown below.
Finally select **Exit** to quit the post processor.

**Figure
6-173
Animated
model at
18.556 Hz**



Similarly, for the frequency **104.023** Hz also, the result can be animated as shown below.

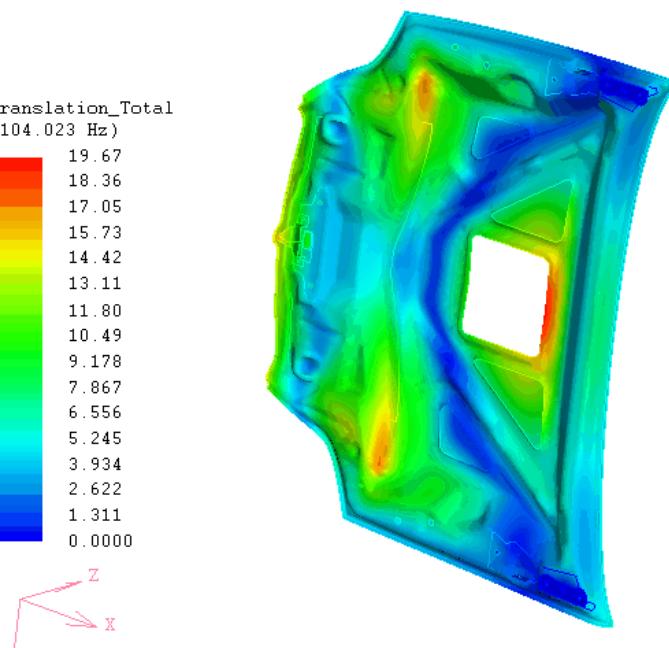
Finally select **Exit** to quit the post processor.

**Figure
6-174
Mode
shape
at
104.023
Hz**

Translation_Total
(104.023 Hz)



19.67
18.36
17.05
15.73
14.42
13.11
11.80
10.49
9.178
7.867
6.556
5.245
3.934
2.622
1.311
0.0000

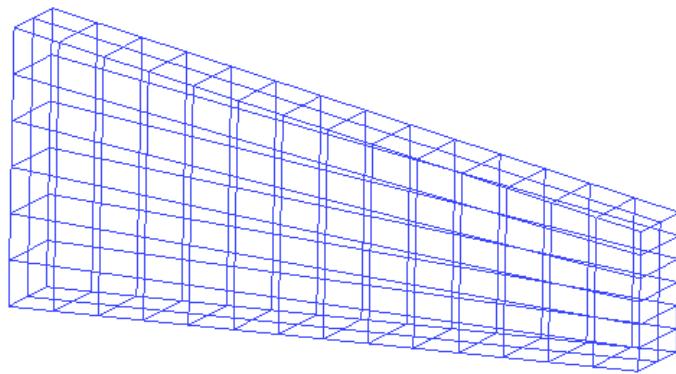


6.5: Abaqus Tutorial

6.5.1: Taper Rod Problem: Linear Static Analysis

The main objective of this tutorial is to demonstrate how to write/view input file for Abaqus for Linear static Analysis in AI*Environment. The mesh for this tutorial is shown below.

Taper
Rod
Problem



a) Summary of Steps

- Launch AI*Environment and load Mesh file
- Apply Material
- Element Properties
- Apply Constraints
- Apply Loads
- Solver setup
- Save Project

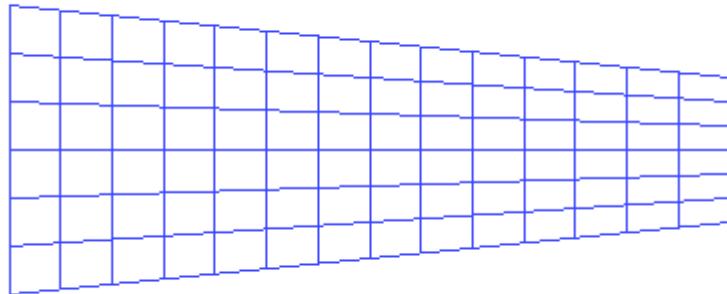
ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	1245
------------------------	--	------

Write Abaqus Input File

Launch AI*Environment

Launch the AI*Environment from UNIX or DOS window. The input files for this tutorial can be found in the Ansys installation directory, under `../v110/docu/Tutorials/AI_Tutorial_Files`. Copy and load the Mesh file ‘TaperRod.uns’ in your working directory, and the mesh will appear as shown.

TaperRod



b) Solver Setup

For solver settings, select Settings > Solver, then solver set up **window** will pop up. Select solver as **Abaqus** from Common Structural Solver and press Apply

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 Menubar.

Define the Material Name as STEEL,

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	1246
------------------------	--	------

Material ID can be left as 1,
Select Isotropic type from the drop down list,
Define Young's **modulus as** 2.03e7,
Define Poisson's ratio as 0.3 and **leave other** fields as it is.
Press Apply.
Element Properties

Select  (Define 3D Element Properties) icon from the **Properties Menu bar**.

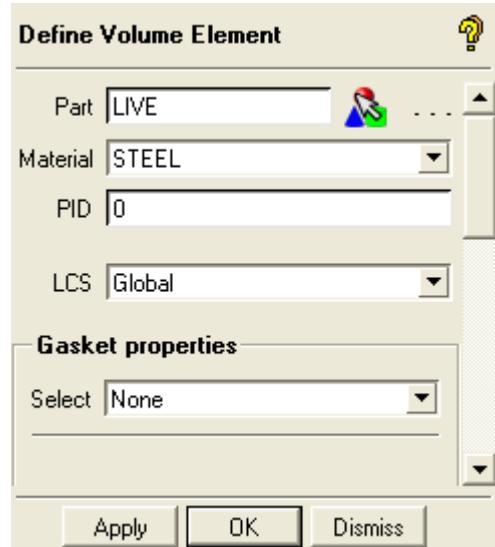
Set PID as 0 in the **Define Volume Element** window as shown.

Select the Part as LIVE.

Select Material as **STEEL**.

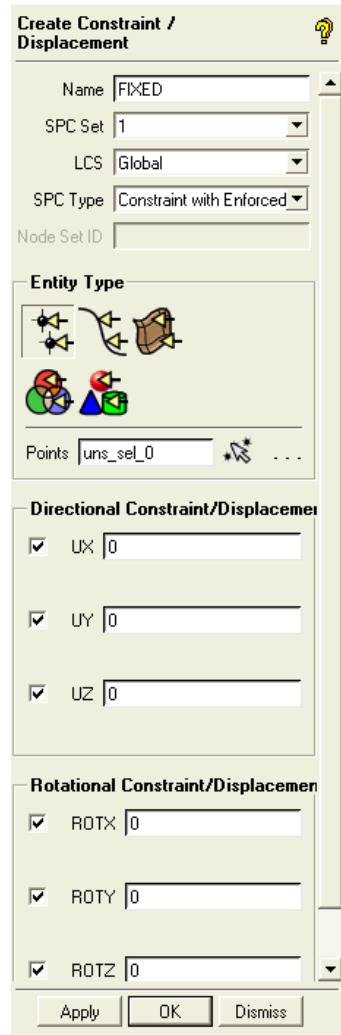
Press Apply.

Define Volume Element

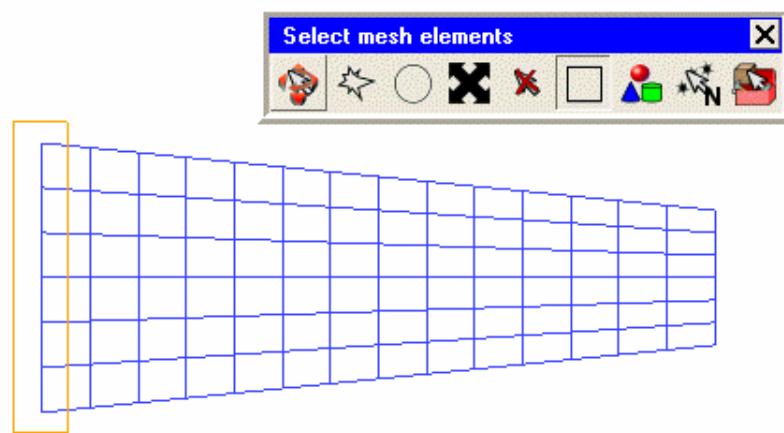


Constraints and Displacement
Constraints

Click on the Constraints>  Create
Constraints/Displacements>  Create Constraints/Displacement on Point button, which will give the window as presented here.
Select the Node(s) selection icon for Mesh
Then make sure you are using the “entire” selection method by using the  in the selection window. Box select the Node(s) at the bottom as shown.
Toggle ON all options for the Directional and Rotational displacement. Press Apply. The constraint applied is shown below.

Constraints/Displacement on Point

**Constraint
Display**



Loads

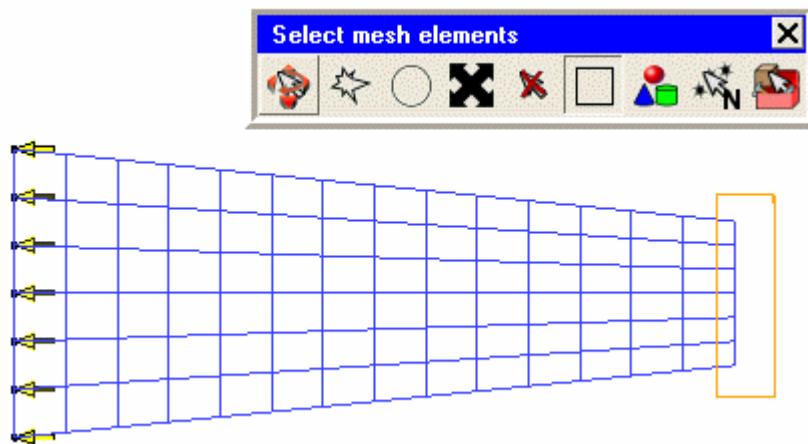
Loads > Create Force >Create Force on Point

In this window enter Name as **FORCE**. Enter values of **FX** as 20e3.

Create Force on Point

The Load applied is shown here.

**Load
applied**



Setting Analysis Type

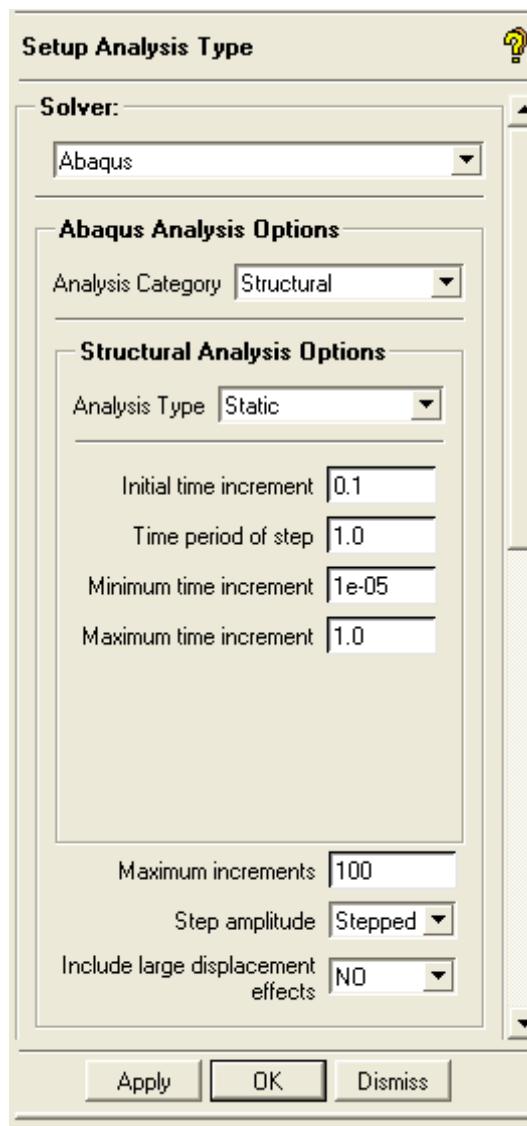
Click on the Solve Options > Setup Analysis Type button to setup an Abaqus run to do Linear Static Analysis. This will bring up the **Setup Analysis Type** window, as shown.

The solver should be set as **Abaqus**.

Set the Analysis Type to Static from the pull down.

Leave all other options as default,

Press Apply to complete the setup.

Setup Analysis Type window**Save Project**

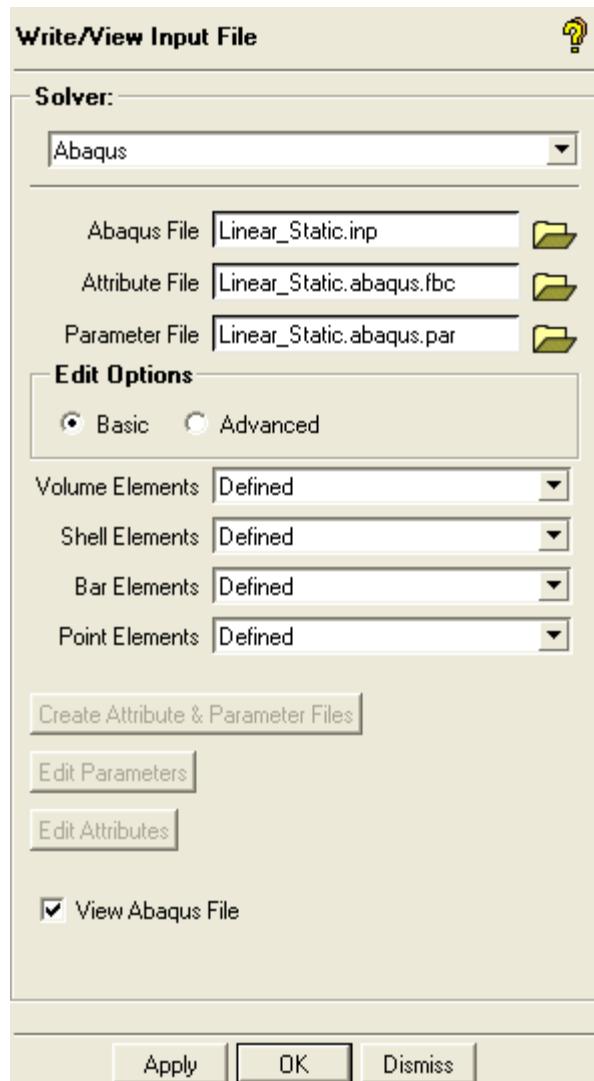
Through File > **Save Project As**, create a new directory called **Linear_Static** and enter into it.

Enter Linear_Static as the project name and press Save to save the mesh, constraints, and loads in this directory

Write Abaqus Input File

Click the Solve Options> Write/View Input File  button.

Enter the Abaqus file name as **Linear_Static.inp and switch **ON View Abaqus file** at the bottom as shown. Press Apply.**

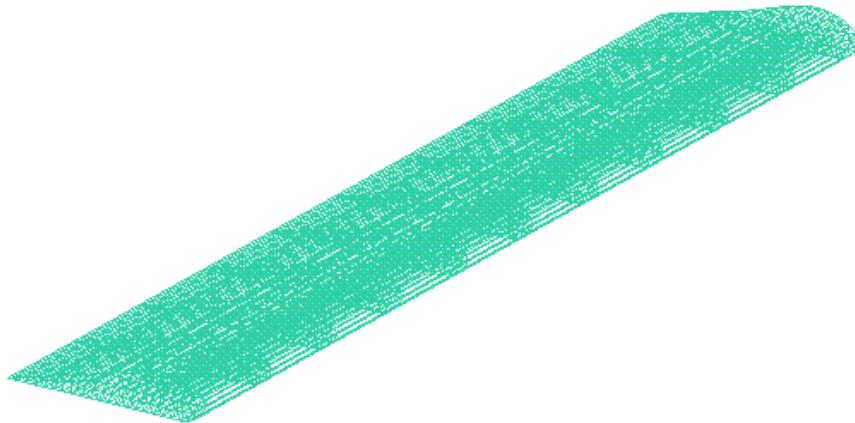
**Write/View Input File
window**

Linear_Static.inp file created in your Working Directory

6.5.2: Wing Problem: Modal Analysis

The main objective of this tutorial is to demonstrate the Modal Analysis (Natural Frequency Extraction) is well Supported by AI*E for Abaqus Solver. The mesh for this tutorial is shown.

**Wing
Problem**



a) Summary of Steps

Launch AI*Environment and load Mesh file

Apply Material

Element Properties

Apply Constraints

Apply Loads

Solver setup

Save Project

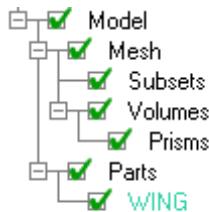
Write Abaqus Input File

Launch AI*Environment

Launch the AI*Environment from UNIX or DOS window. The input files for this tutorial can be found in the Ansys installation directory, under
..../v110/docu/Tutorials/AI_Tutorial_Files. Copy and load the Mesh file
'wing.uns' in your working directory.

Make the Display Control Tree as shown.

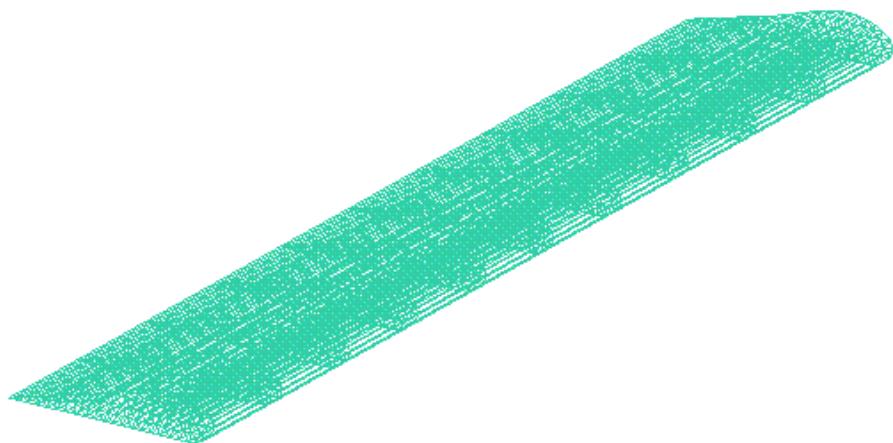
Display Control Tree



Hit Fit Window in the Utilities panel in the upper left hand corner to make the part fill the screen.

View > Isometric

The Mesh will appear as shown here.

Mesh**Solver Setup**

For solver settings, select Settings > Solver, then solver set up window will pop up. Select solver as **Abaqus** from Common Structural Solver and press Apply

Material and Element Properties**b) Selection of Material**

Select  (Create Material Property) icon from Properties Menubar.

Define the Material Name as **STEEL**,

Material ID can be left as 1,

Select Isotropic type from the drop down list,

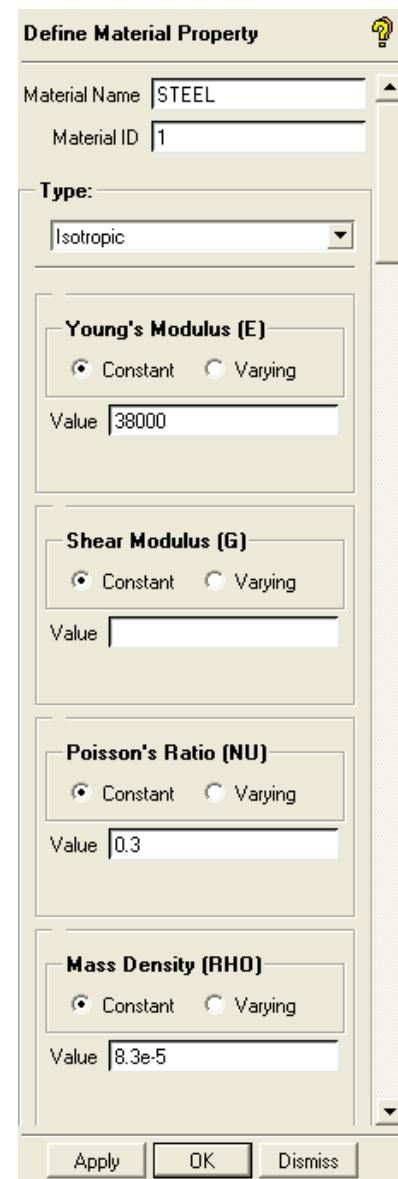
Define Young's modulus as **38000**,

Define Poisson's ratio as **0.3**

Define the Mass Density as a **Constant 8.3e-5**.

and leave other fields as it is.

Press Apply.

Material Property

c) Element Properties

Select  (Define 3D Element Properties) icon from the **Properties Menu bar**.

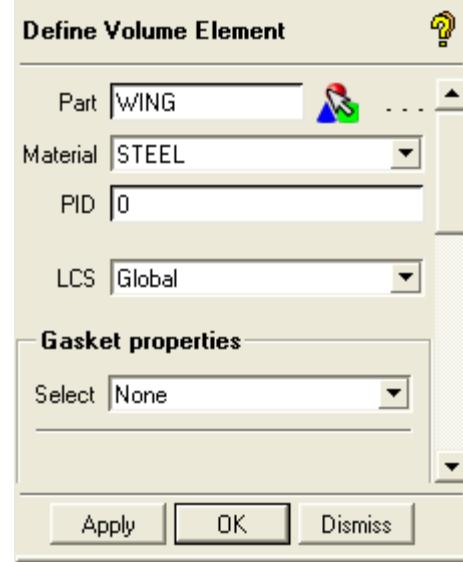
Set PID as 1 in the **Define Volume Element** window .

Select the Part as **WING**.

Select Material as **STEEL**.

Press Apply.

Define Volume Element



d) Constraints and Displacement

Constraints

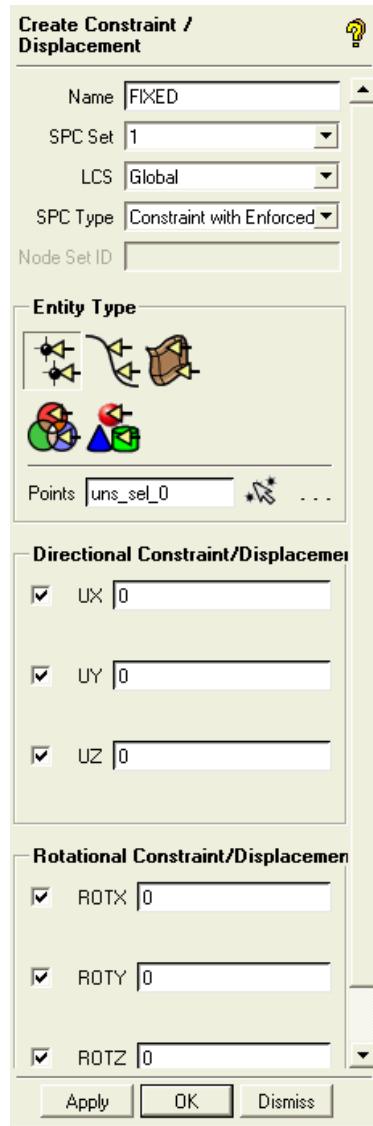
Click on the Constraints>  Create Constraints/Displacements> 
 Create Constraints/Displacement on Point button
 Main Menu > View > Left
 Select the Node(s) selection icon for Mesh as shown here.

Toggle **ON** all options for the Directional and Rotational displacement.
Press Apply. The constraint applied is shown below.

**Node(s)
selection**



Select nodes with the left button, middle = done, right = back up / cancel, '?' = list options

Constraints/Displacements**e) Setting Analysis Type**

Click the **Solve Options** tab, then the **Setup Analysis Type** icon.  The window that appears is shown below.

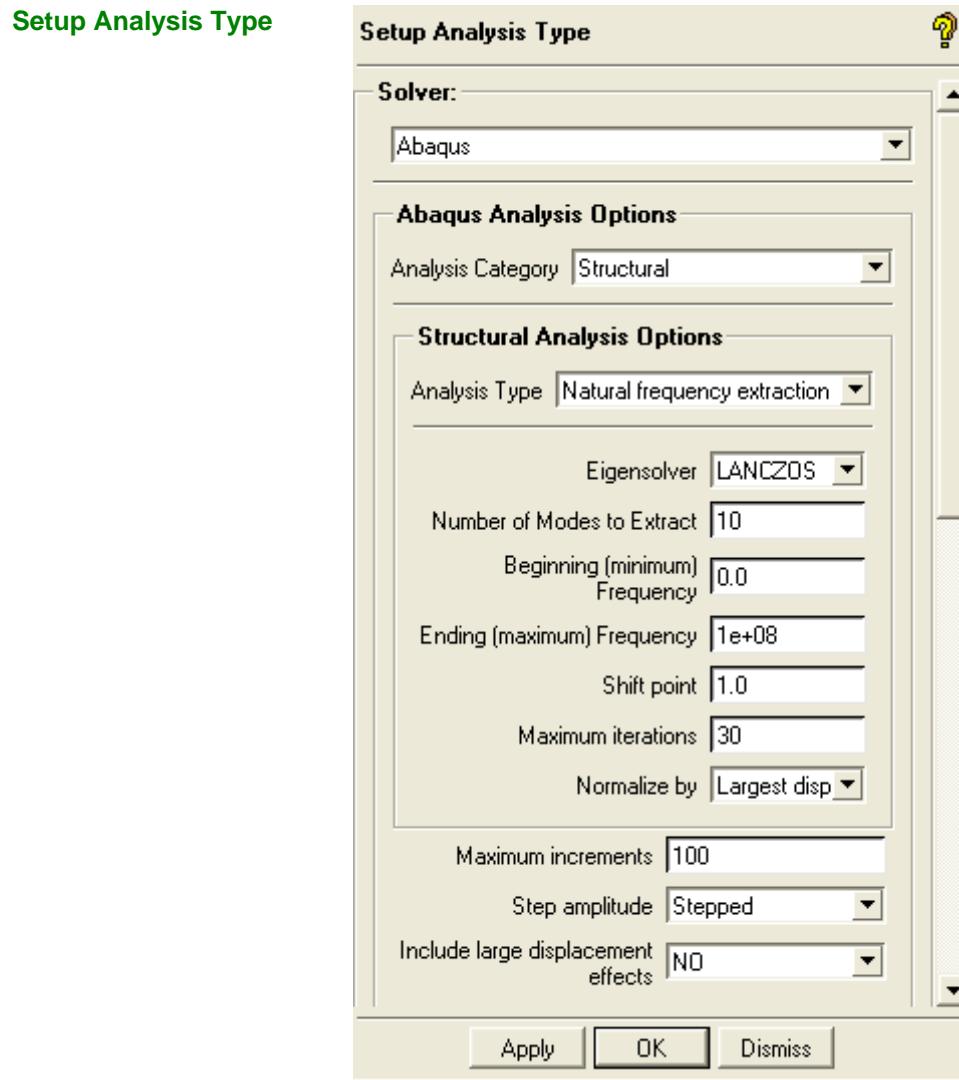
Enter the following:

Select the Analysis Category as **Structural**

Select Natural frequency extraction from the dropdown for Analysis Type and keep all the default options.

Press Apply to complete the setup.

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	1263
------------------------	--	------



f) Save Project

Through File > Save Project As, create a new directory called Wing_Modal and enter into it.

Enter Wing_Modal as the project name and press Save to save the mesh, constraints in this directory

g) Write Abaqus Input File

Click the **Solve Options> Write/View Input File**  button.
Enter the **Abaqus** file name as **Wing_Modal.inp** and switch **ON View Abaqus file** at the bottom as shown here. Press Apply

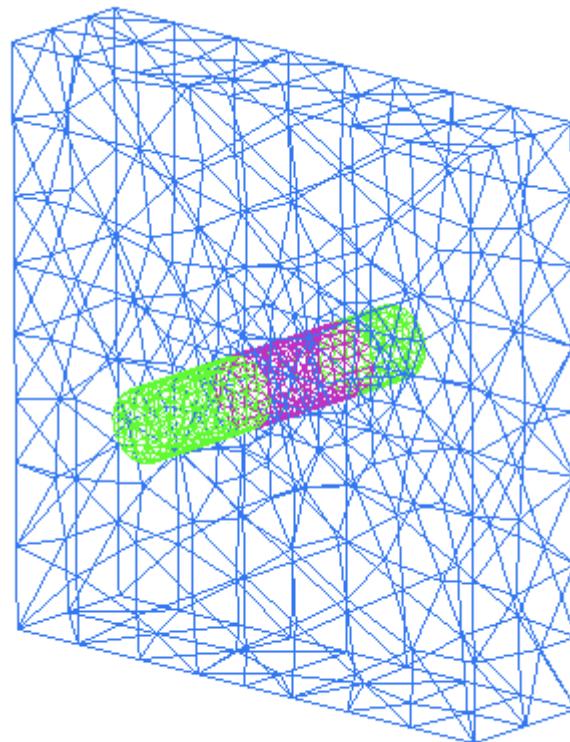


Wing_Modal.inp file is created in your Working Directory

6.5.3: PinHole: Contact Analysis

The main objective of this tutorial is to demonstrate the ease of use defining contacts in AI*Environment for Abaqus Solver. The mesh for this tutorial is shown here.

PinHole



a) Summary of Steps

Launch AI*Environment and load Mesh file

Create Mesh Subset

Apply Material

Element Properties

Apply Constraints

Apply Loads

Solver setup

Save Project

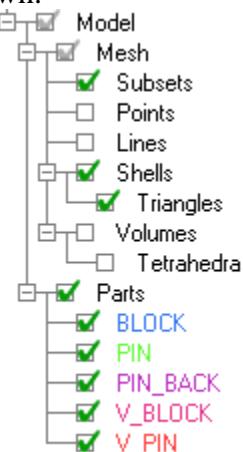
Write Abaqus Input File

b) Launch AI*Environment

Launch the AI*Environment from UNIX or DOS window. The input files for this tutorial can be found in the Ansys installation directory, under ..\v110\docu\Tutorials\AI_Tutorial_Files.

Make the Display Control Tree as shown.

Display Tree



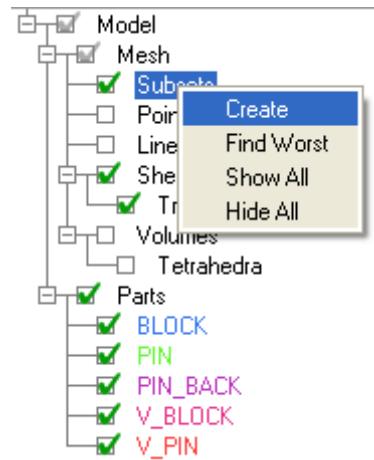
c) Solver Setup

For solver settings, select Settings > Solver, then solver set up window will pop up. Select solver as **Abaqus** from Common Structural Solver and press Apply.

d) Create Mesh Subset

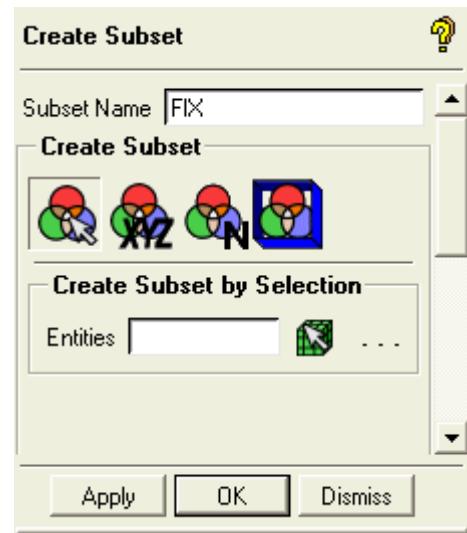
Create Mesh Subset as shown here.

Create Mesh Subset



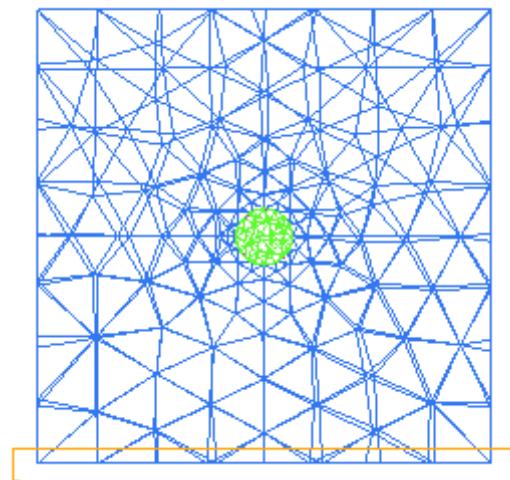
The Create Mesh Subset window will appear in the GUI and Enter FIX in the Subset Name.

Create Subset:FIX



Click Select Element(s) in the Create Subset window, and pick elements as shown in the figure below.

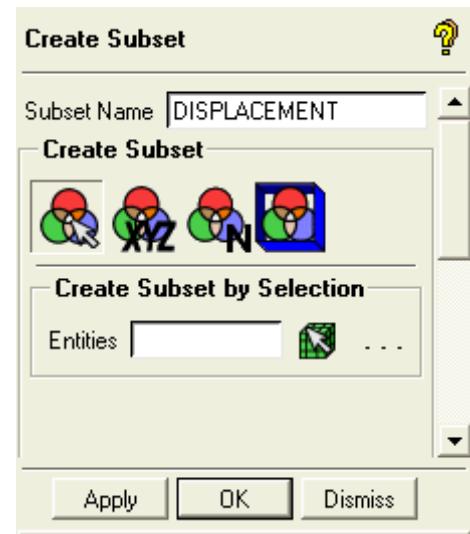
Select Element for FIX



Press Apply.

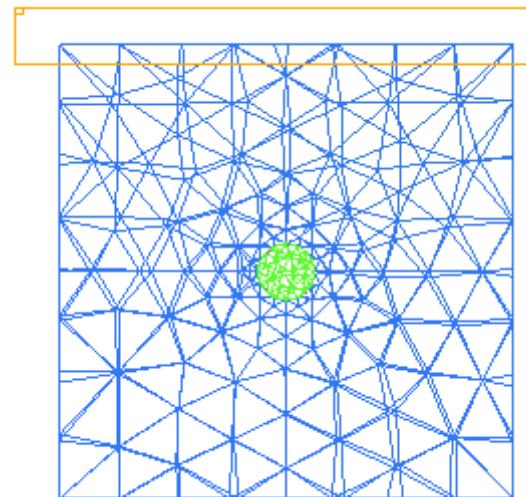
The Create Mesh Subset window will appear in the GUI and Enter **DISPLACEMENT** in the Subset Name.

Select Element for DISPLACEMENT



Click Select Element(s) and click the elements as shown below.

**Select Element for
DISPLACEMENT**



Toggle Off the Shells in the tree in which only created Subsets will appear in the GUI as shown.

Created Subsets



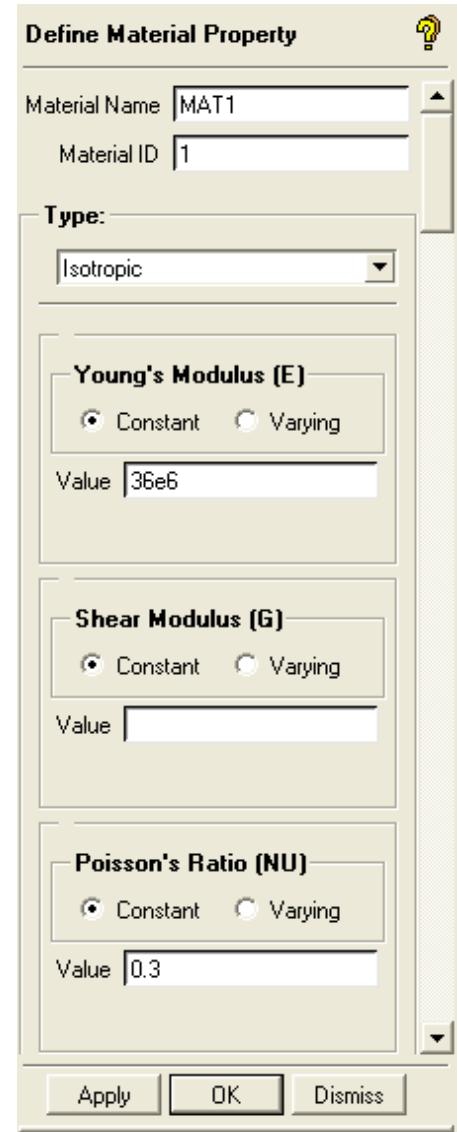
ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	1272
------------------------	--	------

e) Material and Element Properties

Selection of Material



Select (Create Material Property) icon from Properties Menubar.
Define the Material Name as MAT1,
Material ID can be left as 1,
Select Isotropic type from the drop down list,
Define Young's modulus as 36e6,
Define Poisson's ratio as 0.3
and leave other fields as it is.
Press Apply

Apply Material Property

f) Element Properties

Select  (Define 3D Element Properties) icon from the Properties Menu bar

Select Part: V_BLOCK

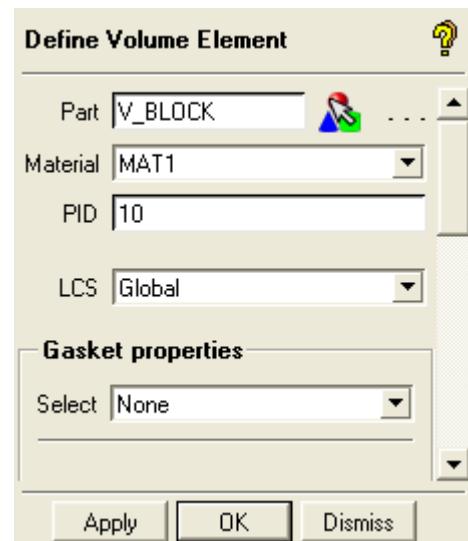
Material: MAT1

PID:10

LCS: Global

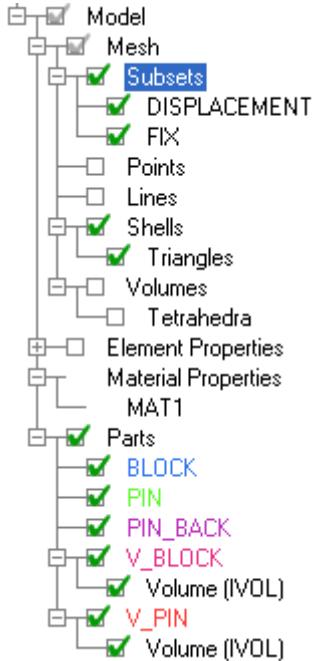
Press Apply.

Define Volume Element



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.

Toggle On the shells in the Display Tree.

Display Tree with Volume Element**g) Constraints and Displacement**

Constraints

Click on Constraints> Create Constraints/Displacements> Create Constraints/Displacement on Subset button

Enter Name as FIX

Toggle ON All Options of Directional and Rotational Constraints

Select Subset as shown below and then press Apply and Dismiss

Create Constraints/Displacements

Click on Constraints>  Create Constraints/Displacements>  Create
Constraints/Displacement on Subset button

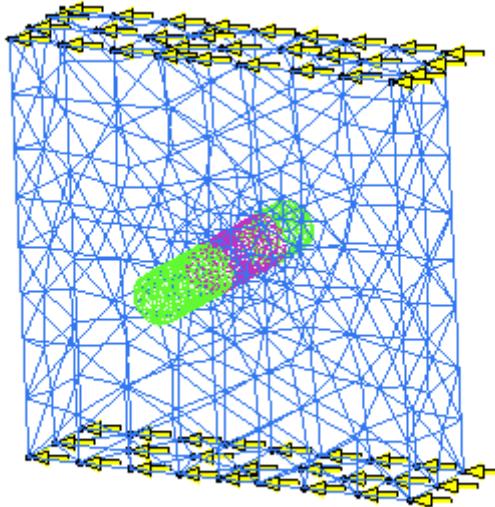
Enter Name as DISPLACEMENT
Toggle ON, UY and Enter Value as -0.2
Select **Subset** as shown below.
Press Apply.

 ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	1278
---	--	------

Constraints/Displacements
:DISPLACEMENTS

The Constraints will apply as shown and press Dismiss

Constraints will Appear



h) Contact

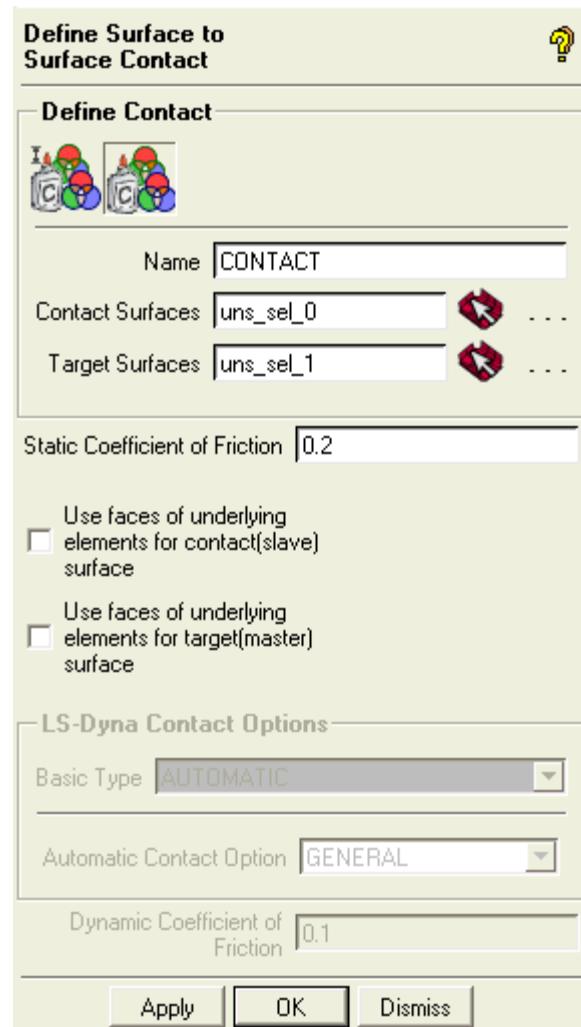
Click on **Constraints>Define Contact** >**Manual Definition** The window as presented will display.
Enter the Name as **CONTACT**

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

Then Press Apply.

Define Contact



i) Setting Analysis Type

Click on the **Solve Options > Setup Analysis Type**  button to setup an Abaqus run to do Linear Static Analysis. This will bring up the **Setup Analysis Type** window.

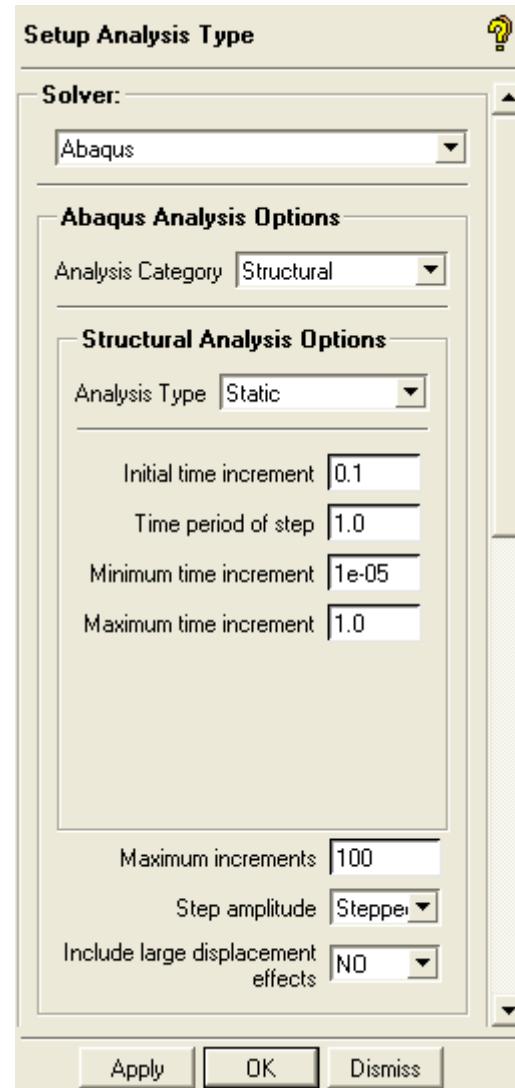
The solver should be set as **Abaqus**.

Set the **Analysis Type** to **Static** from the pull down.

Leave all other options as default,

Press **Apply** to complete the setup.

ANSYS ICEM CFD™	ANSYS ICEM CFD 11.0 Tutorial Manual	1282
------------------------	--	------

Setup Analysis Type

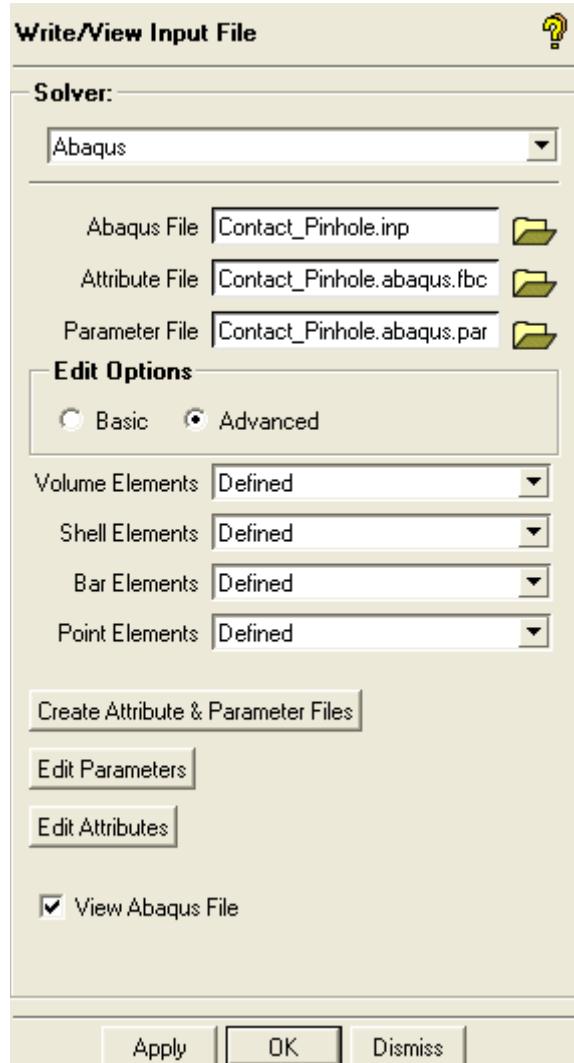
j) Save Project

Through **File > Save Project As**, create a new directory called **Contact_Pinhole** and enter into it.

Enter **Contact_Pinhole** as the project name and press **Save** to save the mesh, constraints and Contact in this directory

k) Write Abaqus Input File

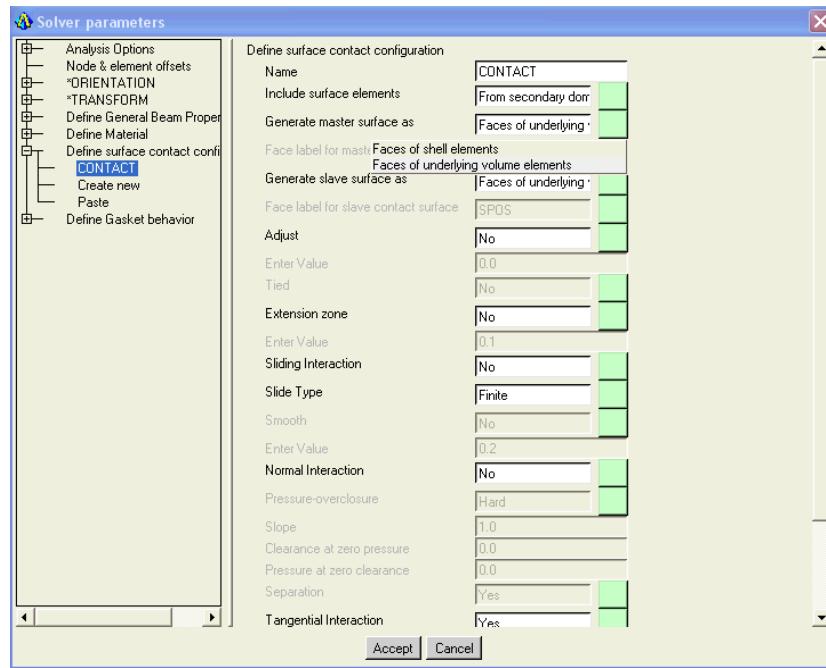
Click the **Solve Options> Write/View Input File**  button.
Enter the **Abaqus** file name as **Contact_Pinhole.inp** and switch **ON View Abaqus file** at the bottom as shown.
Toggle **ON** the Advanced option under Edit Options, and click on **Create Attribute and Parameter Files**.

Abaqus 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**.

Solver Parameters window



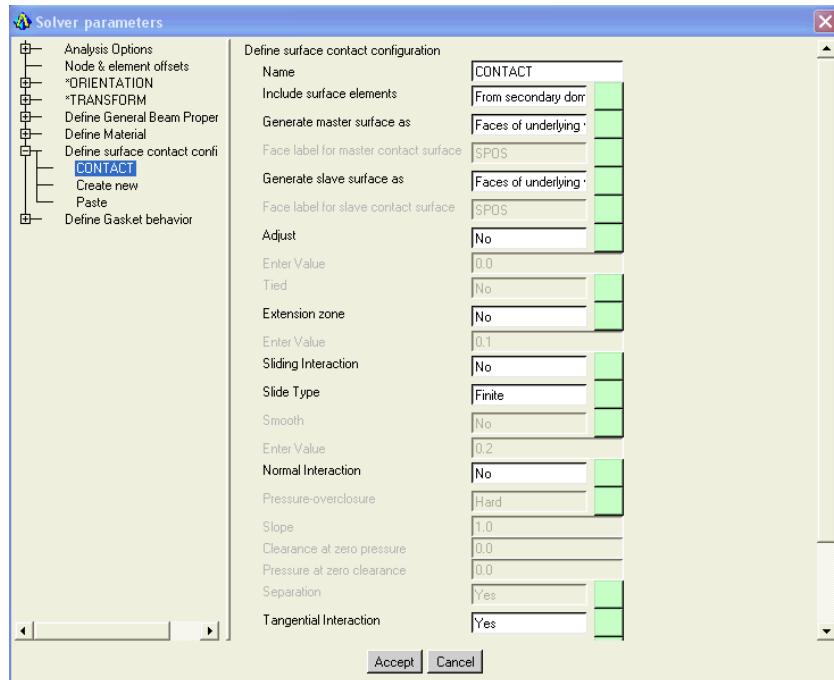
Change Solver Parameters window.

Include Surface Elements as From Secondary Domain

Generate Master Surface as Faces of Underlying Volume Elements

Generate Slave Surface as Faces of Underlying Volume Elements

Change in Solver Parameters window



Also, Switch ON the View Abaqus file option in the Write/View Input File window as shown and press Apply

The **Contact_Pinhole.inp** file is created in your working Directory.