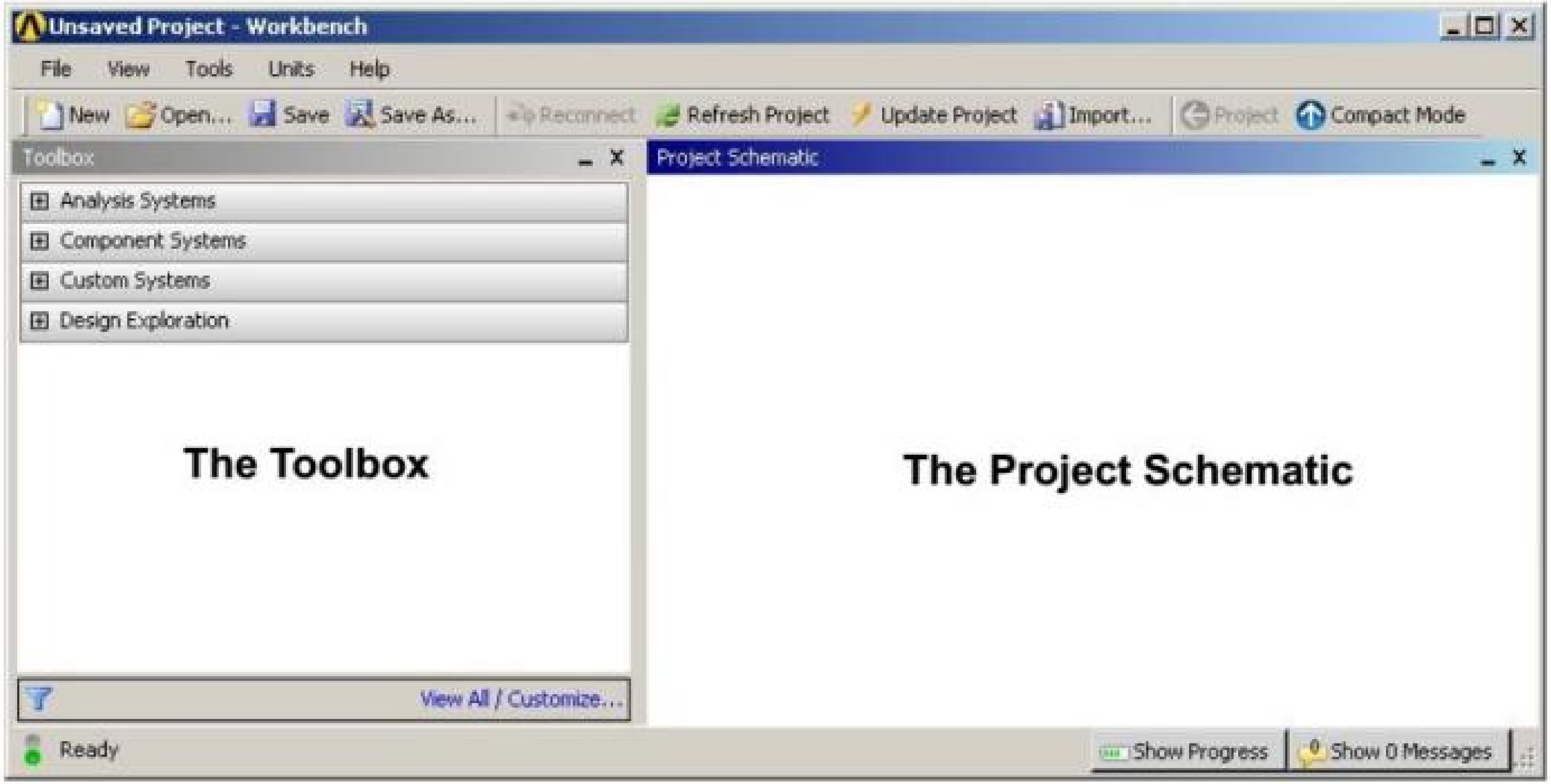
Computational Fluid Dynamics
AE 433

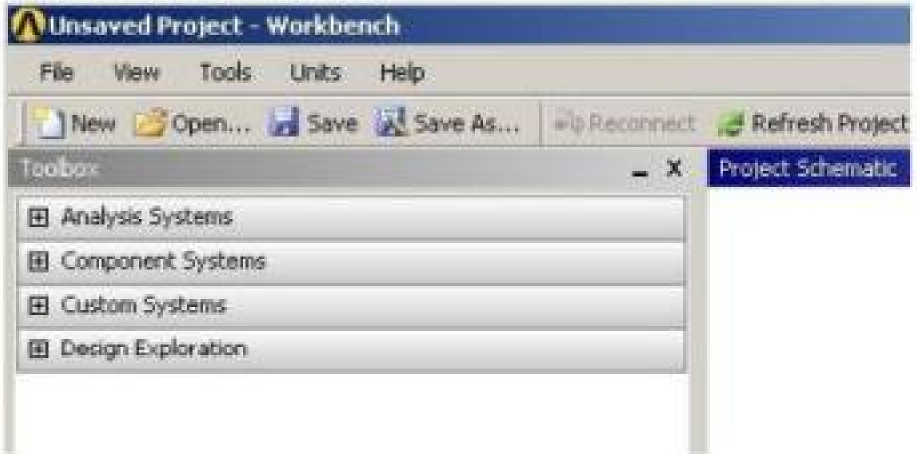
CHAPTER 9
–
Introduction to GUI & Meshing

by
Dr. Emre Kara , Univ. of Gaziantep, TURKEY

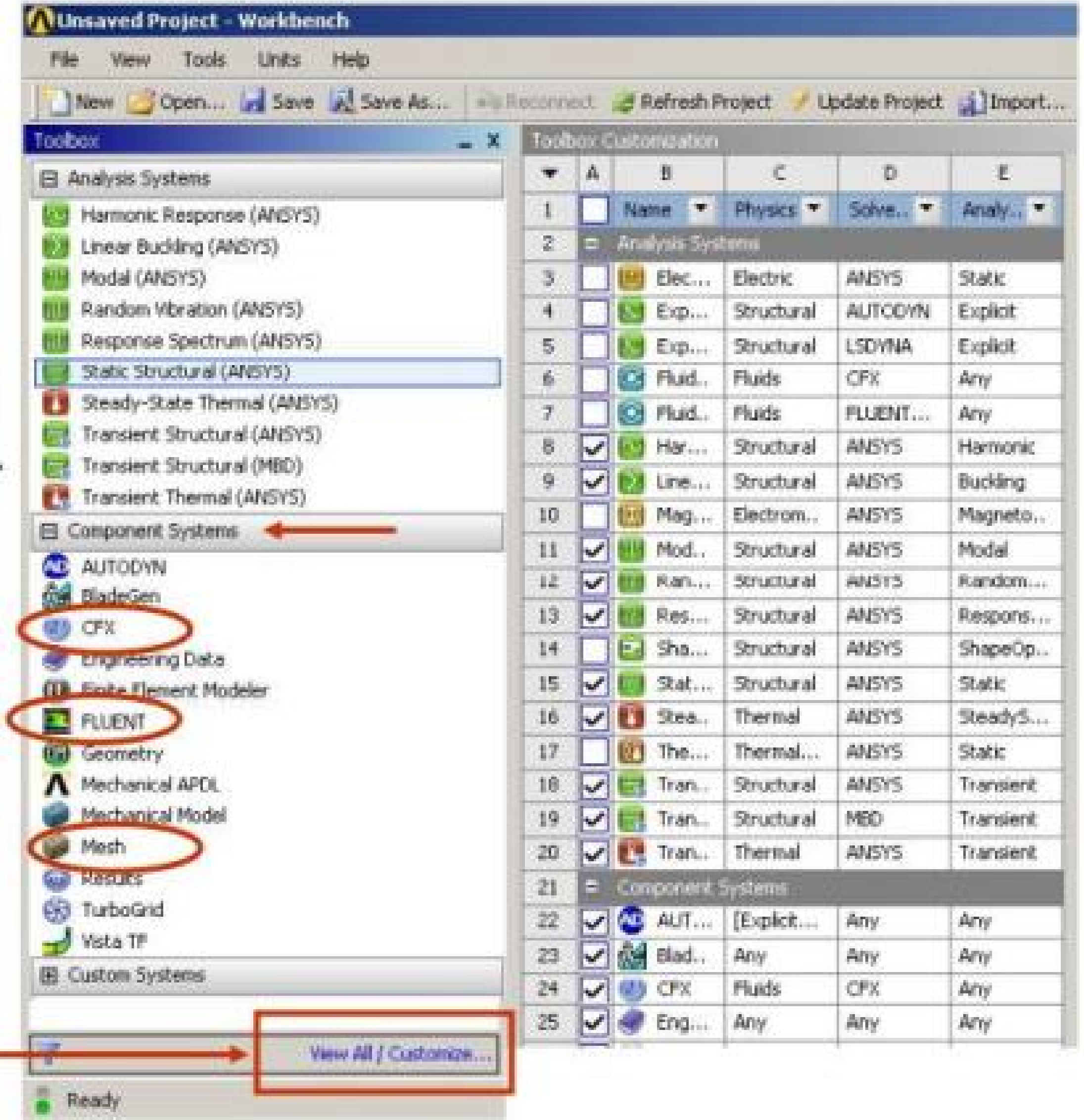
- For most situations the Workbench GUI is divided into 2 primary sections (there are other optional sections we'll see in a moment):



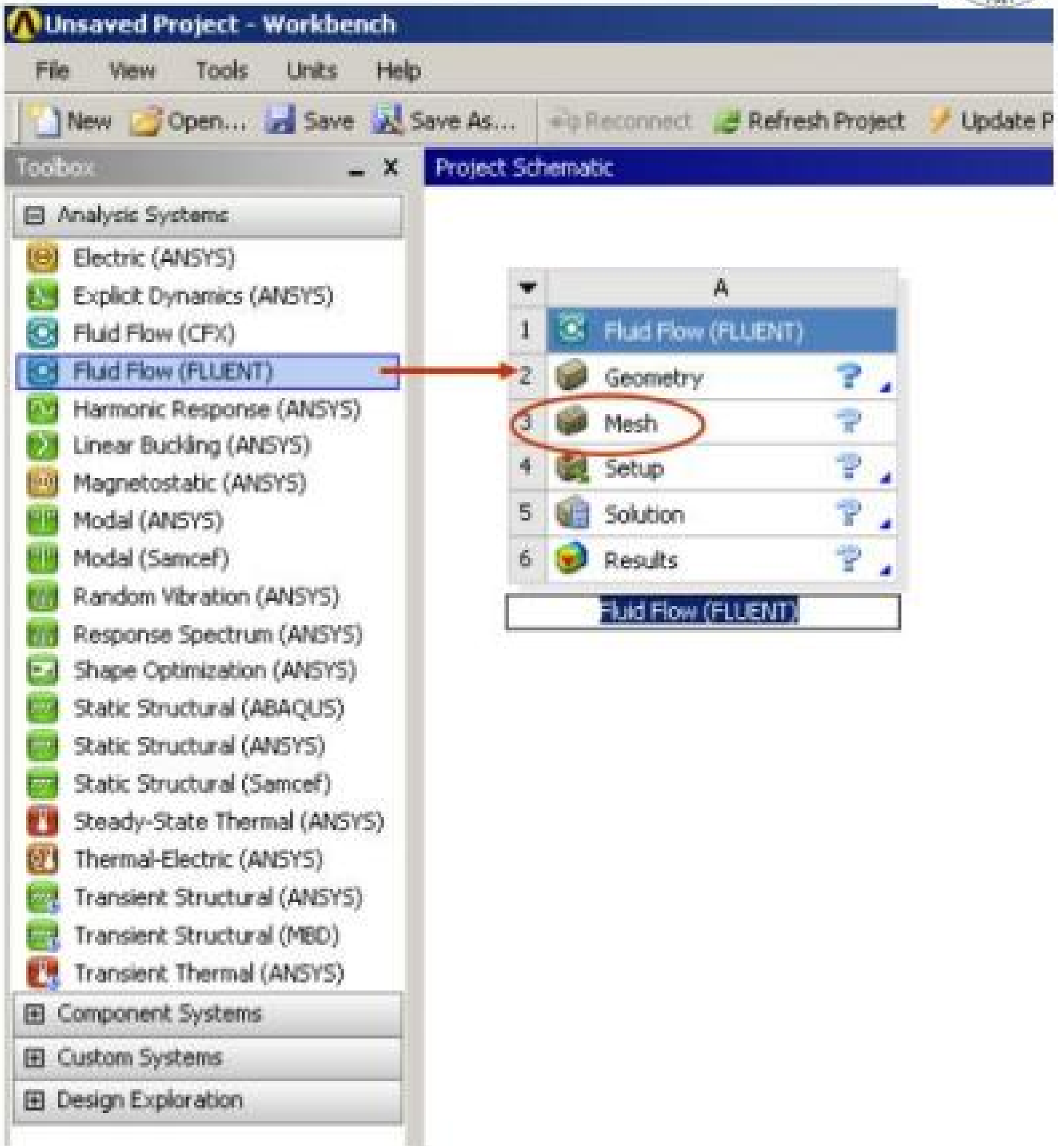
- The toolbox contains 4 subgroups:
- Analysis systems: predefined templates that can be placed in the schematic.
- Component systems: various applications that can be accessed to build, or expand, analysis systems.
- Custom Systems: predefined analysis systems for coupled applications (FSI, thermal-stress, etc.). Users can also create their own predefined systems.
- Design Exploration: Parametric management and optimization tools.



- The systems and components displayed in the toolbox will depend on the installed products.
- Using the check boxes in the “View All / Customize” window, the items displayed in the toolbox can be toggled on or off.
- The toolbox customization window is normally left closed when not in use.



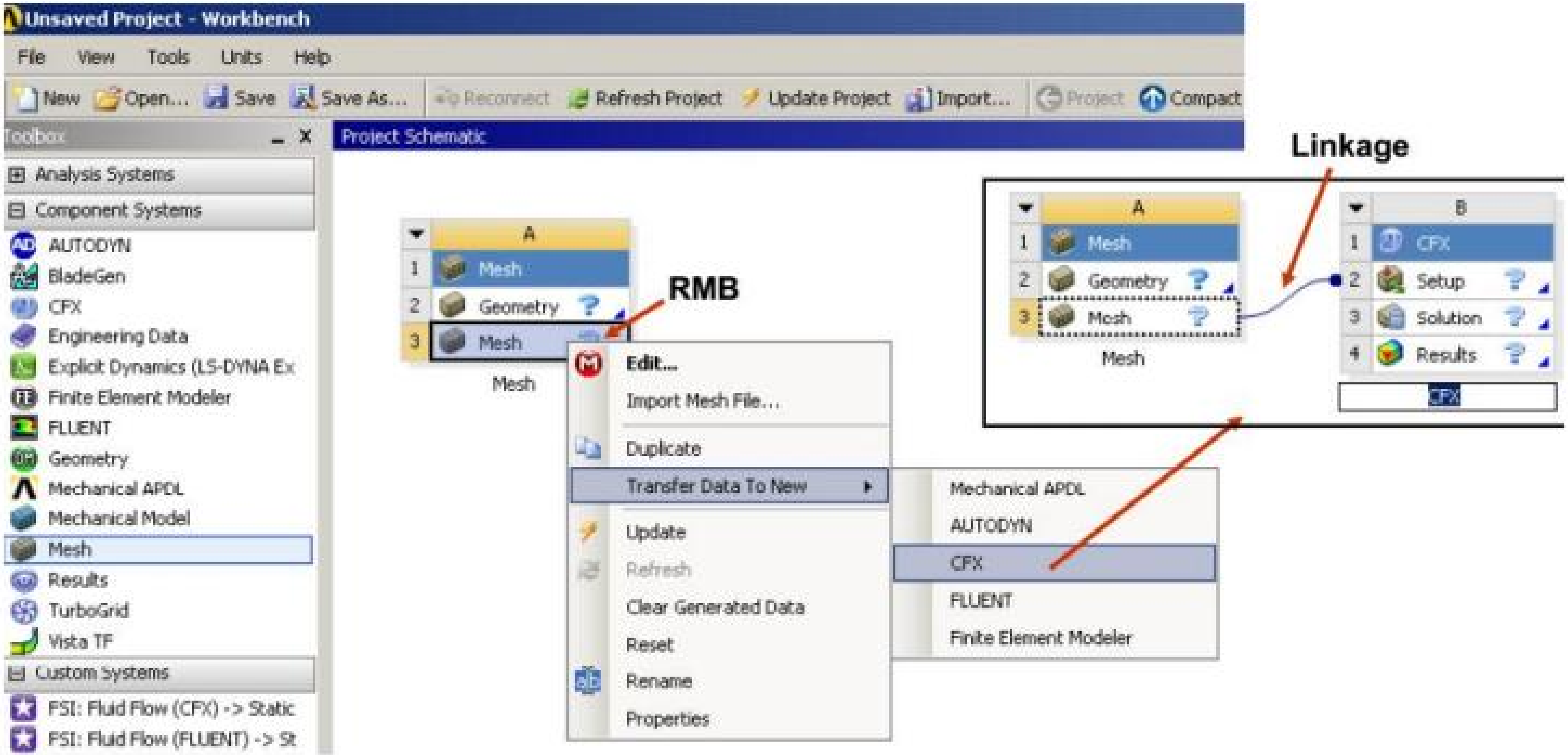
- In this example a Fluid Flow (FLUENT) analysis type is selected for the project schematic.
- From the toolbox the selection can be dragged and dropped onto the schematic or simply double clicked.
- Note that a Mesh item appears in the Fluid Flow block



- Note that you can also create an entry on the Project Schematic that is a standalone instance of the ANSYS Meshing Application

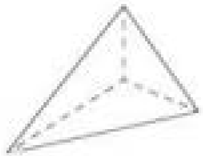
- In the example shown, the Meshing entry in the Component Systems toolbox is dragged onto the Project Schematic

- Linkages between different entries in the schematic can be established in a number of ways
- The RMB will reveal various choices to you

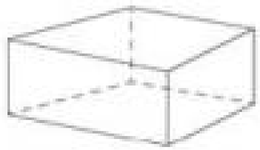


Purpose

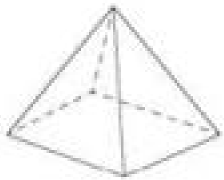
- For both CFD (fluid) and FEA (solid) modelling, the software performs the computations at a range of discrete locations within the domain.
- The purpose of meshing is to decompose the solution domain into an **appropriate** number of locations for an accurate result.
- The basic building-blocks for a 3D mesh are:



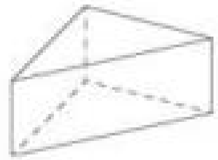
Tetrahedrons (unstructured)



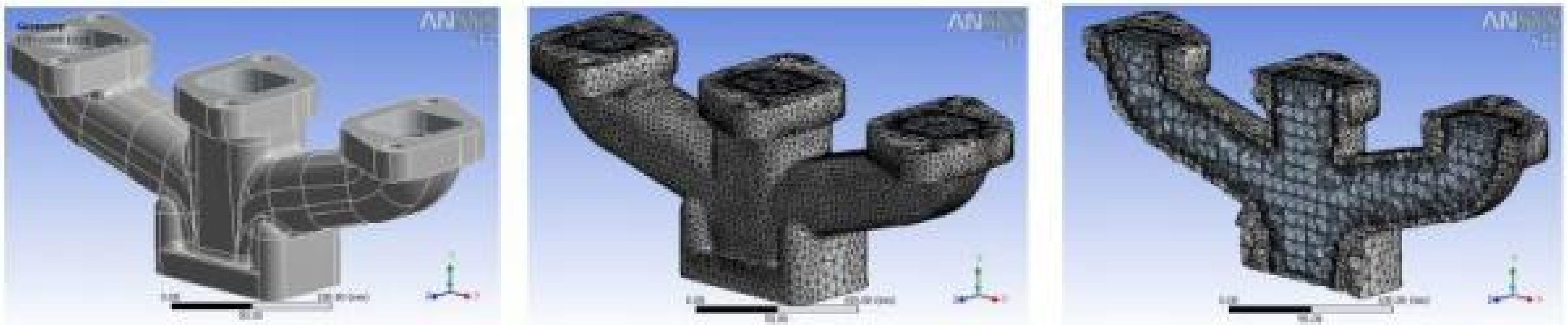
Hexahedrons (usually structured)



Pyramids (where tet. and hex. cells meet)



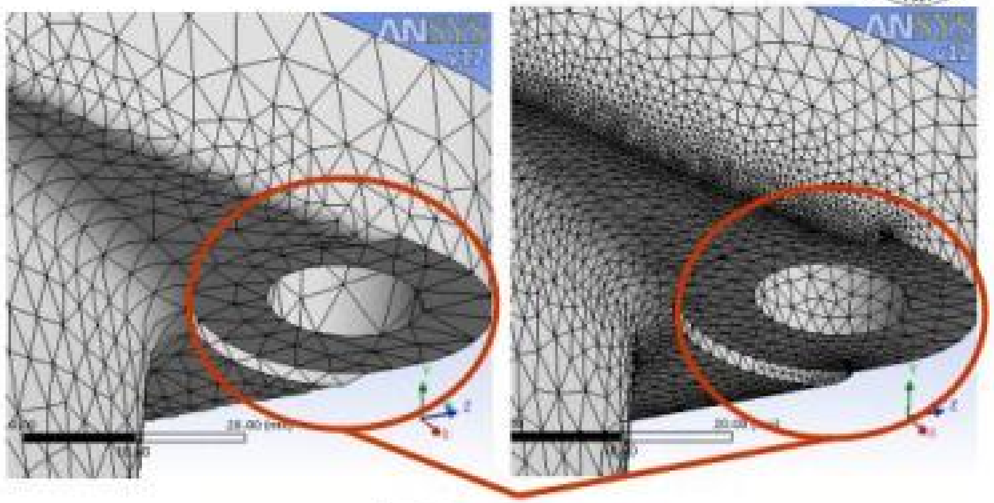
Prisms (formed when a tet mesh is extruded)



Manifold Example: Outer casting and internal flow region are meshed for coupled thermal/stress gas flow simulation

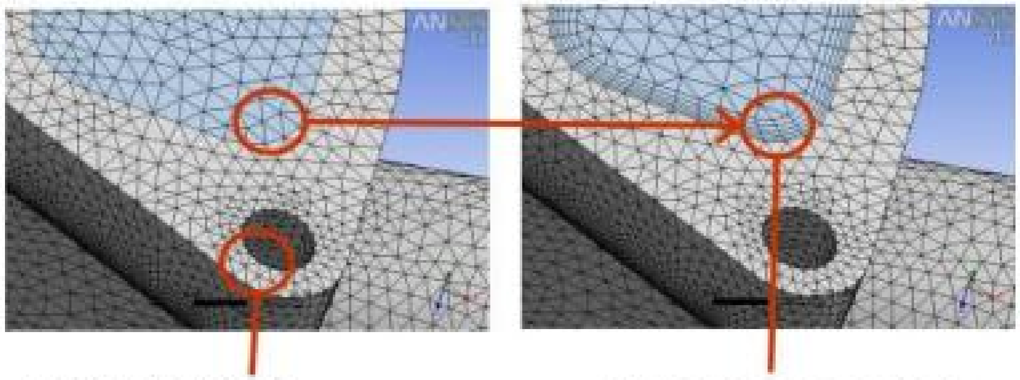
Considerations

- Detail:
  - How much geometric detail is relevant to the simulation physics.
  - Including unnecessary detail can greatly increase the effort required for the simulation.



Is it necessary to resolve this recess?

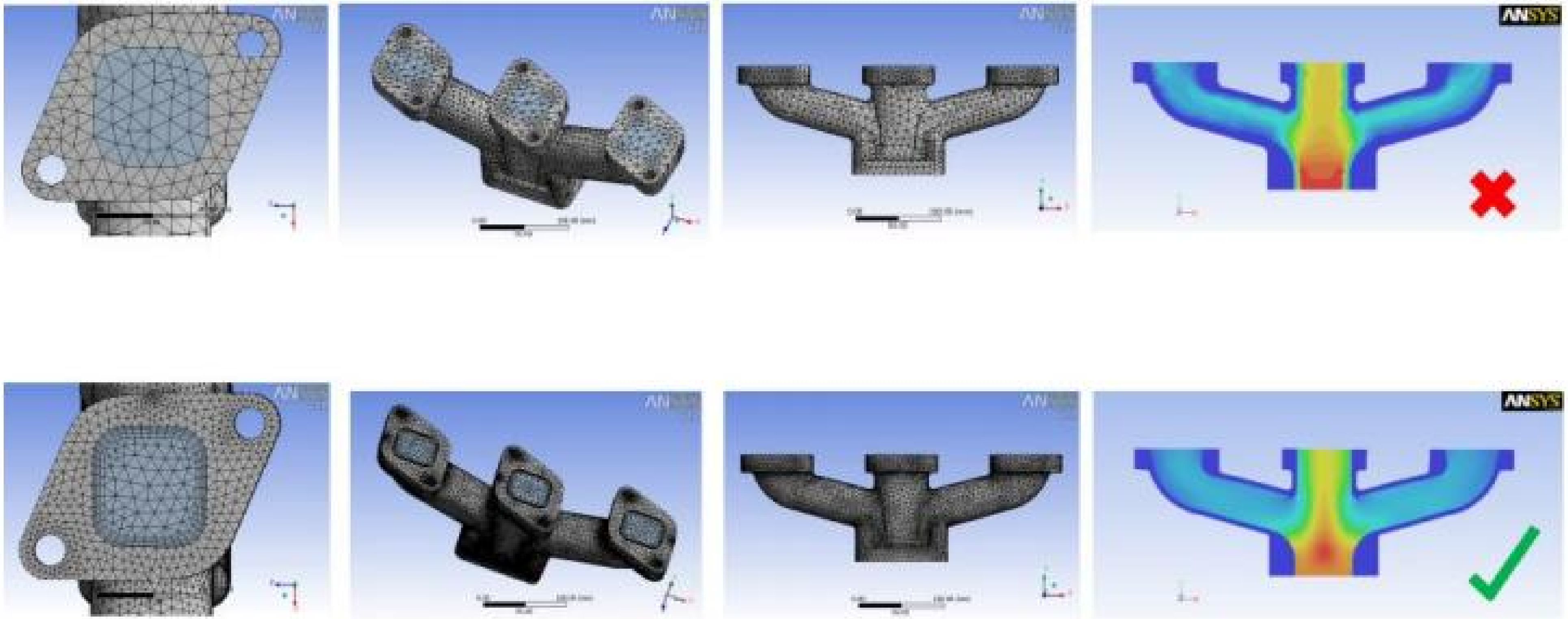
- Refinement
  - Where in the domain are the most complex stress/flow gradients? These areas will require higher densities of mesh elements.



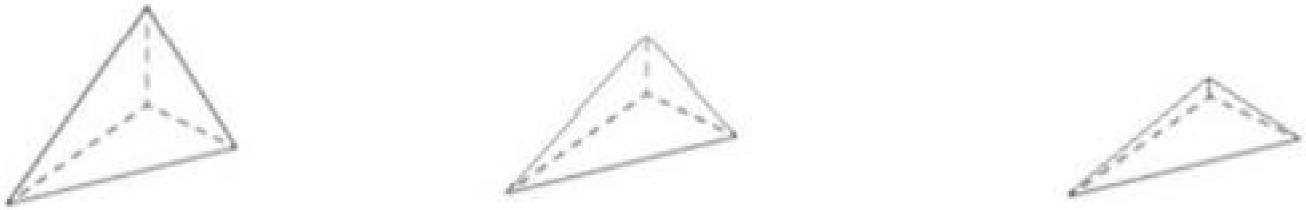
Refined mesh around bolt-hole

Extra mesh applied across fluid boundary layer

- Efficiency
  - Greater numbers of elements require more compute resource (memory / processing time). Balance the fidelity of the simulation with available resources.



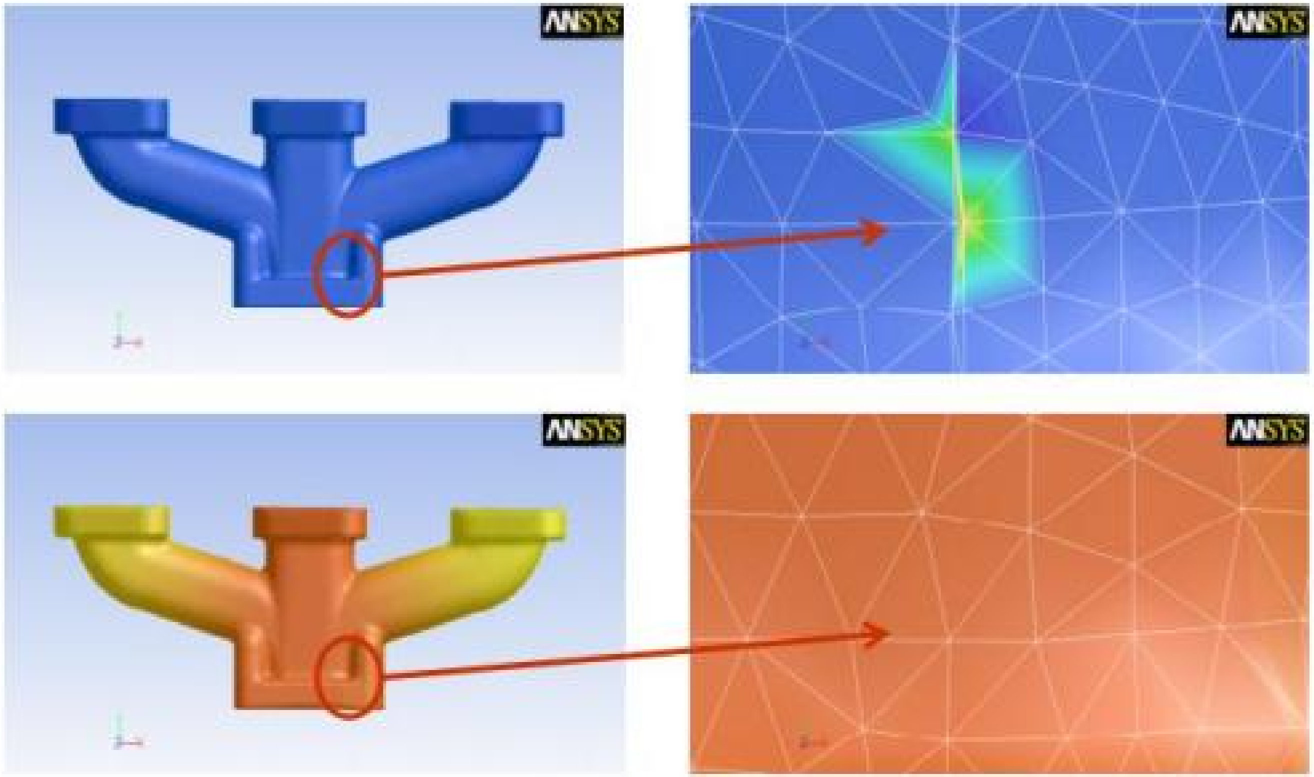
- Quality
  - In areas of high geometric complexity mesh elements can become distorted. Poor quality elements can lead to poor quality results or, in some cases, no results at all!
  - There are a number of methods for measuring mesh element quality (mesh metrics*). For example, one important metric is the element 'Skewness'. Skewness is a measure of the relative distortion of an element compared to its ideal shape and is scaled from 0 (Excellent) to 1 (Unacceptable).



0-0.25	0.25-0.50	0.50-0.80	0.80-0.95	0.95-0.98	0.98-1.00
Excellent	very good	good	acceptable	bad	Unacceptable

*Further information on mesh metrics is available in the documentation and training lecture appendices

Example showing difference between good and poor meshes:



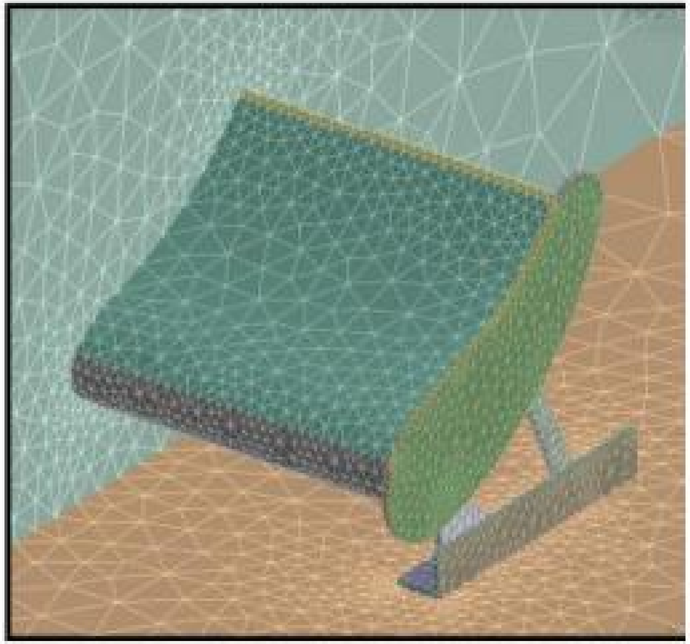
This example illustrates an unconverged thermal field in a manifold solid casting. On closer inspection it is clear that the simulation is unable to resolve a sensible data field in the region of poor quality elements.

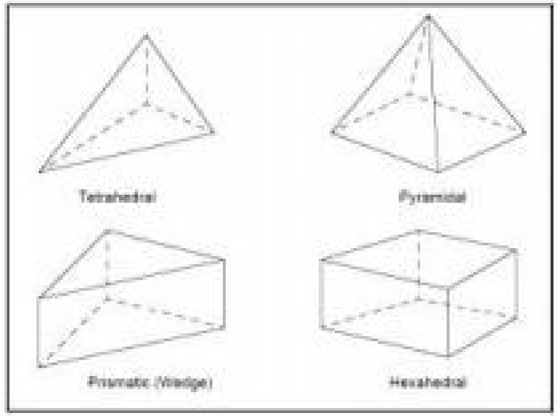
The example with good quality elements demonstrates no problems in the solution field.

The ANSYS Meshing Application provides many tools to help maximise mesh quality

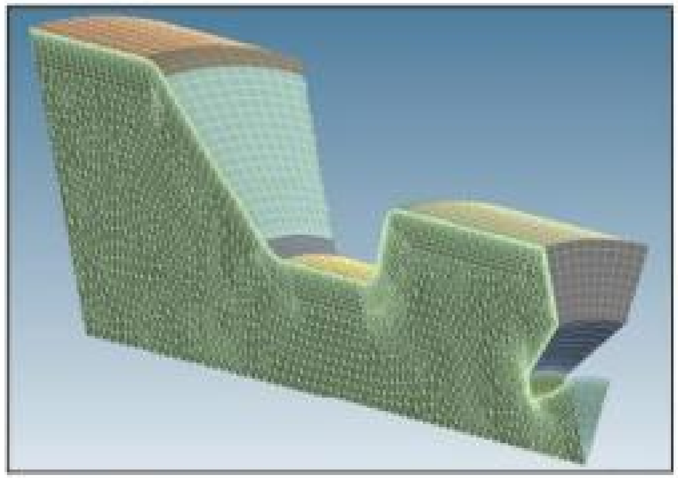
CFD

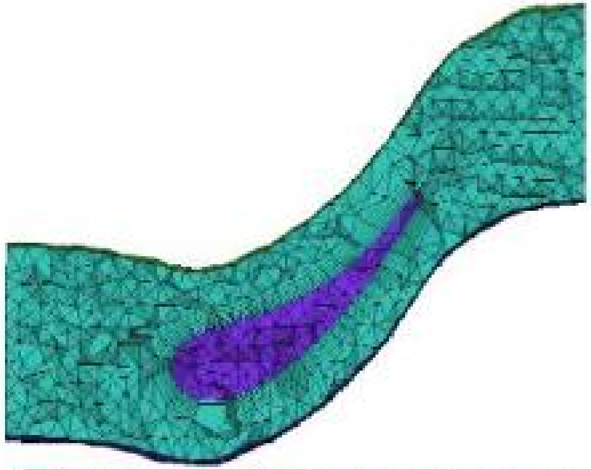
- Refine mesh to capture gradients of concern
  - E.g. Velocity, pressure, temperature, etc.
- Mesh quality and smoothness critical for accurate results
  - This leads to larger mesh sizes, often millions of elements
- tet mesh dominated, but hex elements still preferred
- tet meshes for CFD are usually first order (no mid-side nodes on element edges)

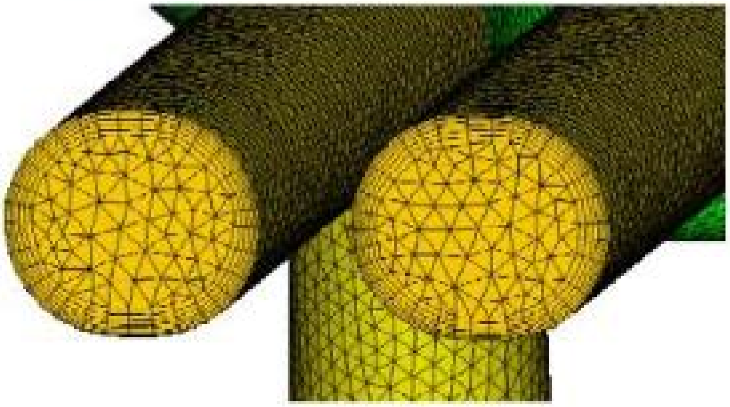


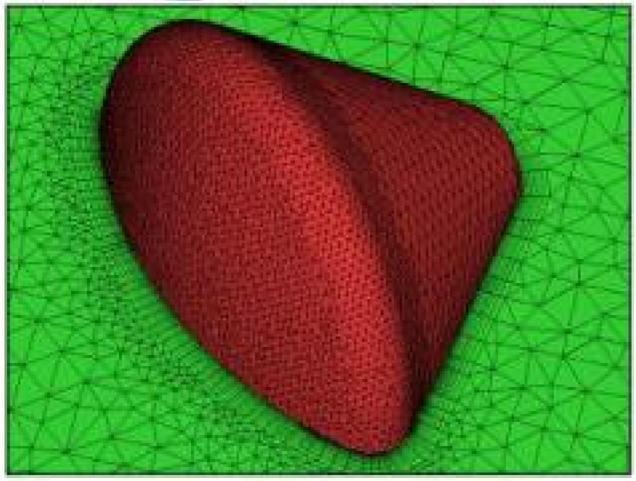


- **Tet Mesh and Tet/Prism hybrid**

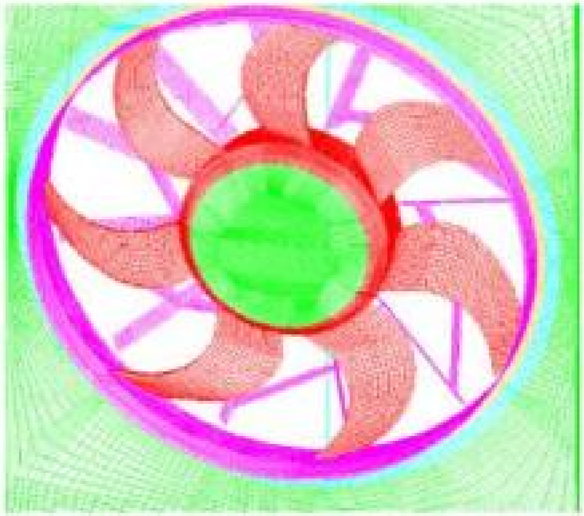


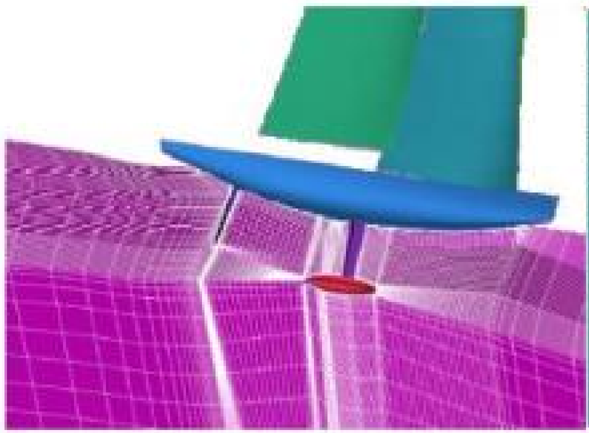


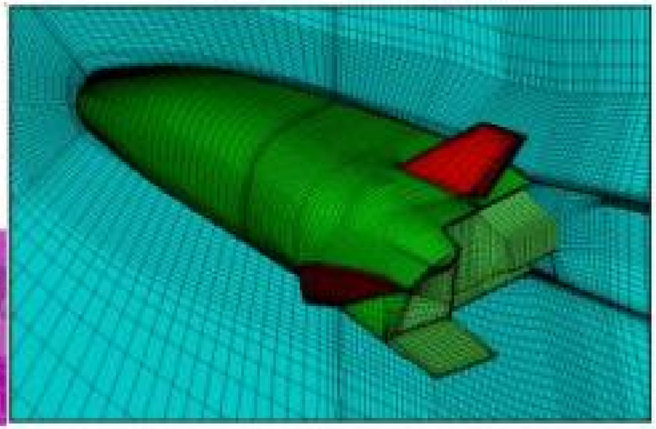




- **Hex Mesh**





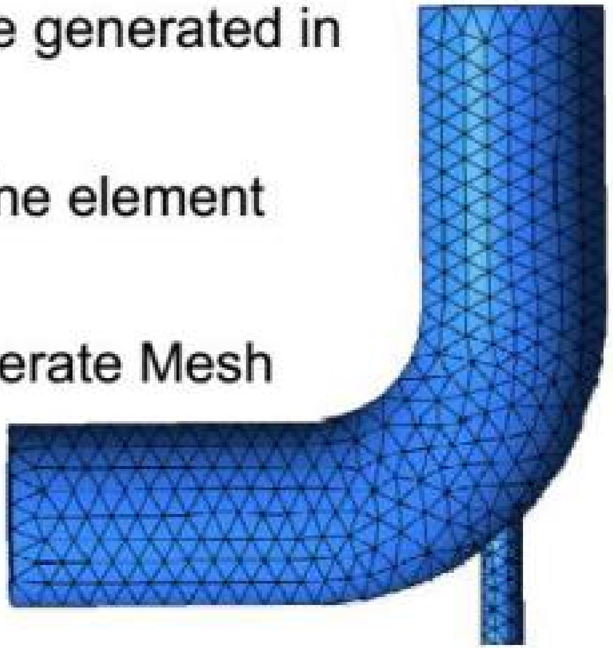


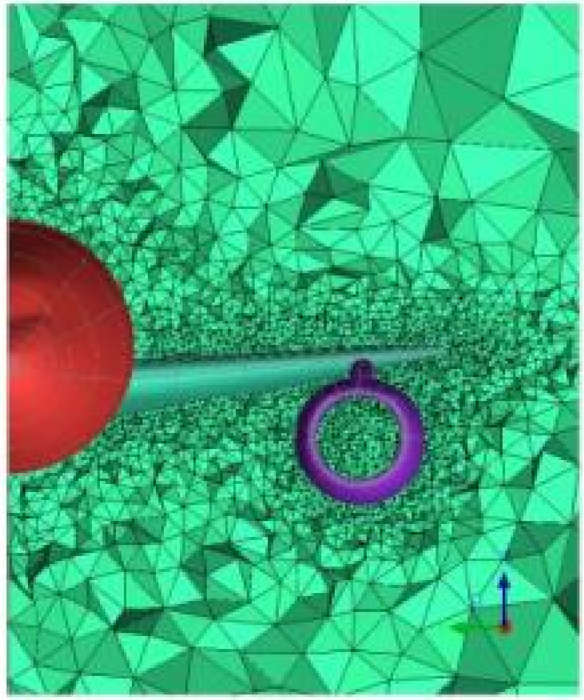
- **Tet Mesh**
  1) Can be generated quickly, automatically, and for complicated geometry

Mesh can be generated in 2 steps:

Step 1: Define element sizing

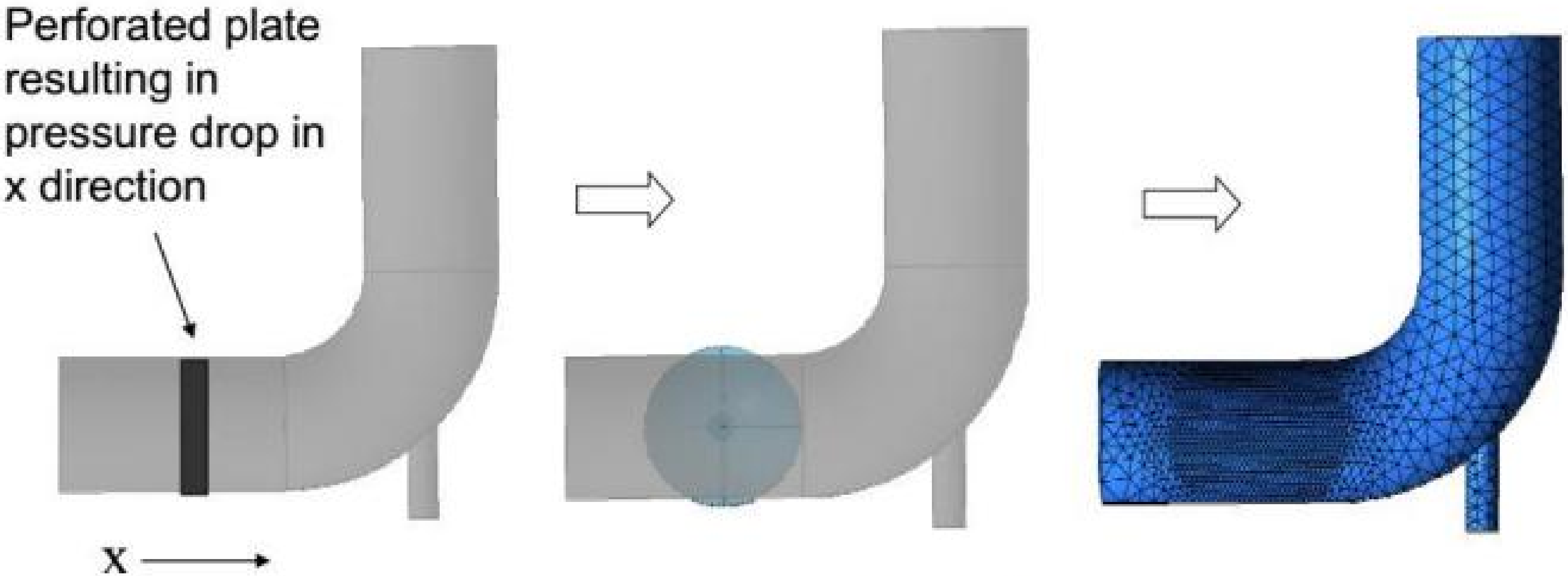
Step 2: Generate Mesh



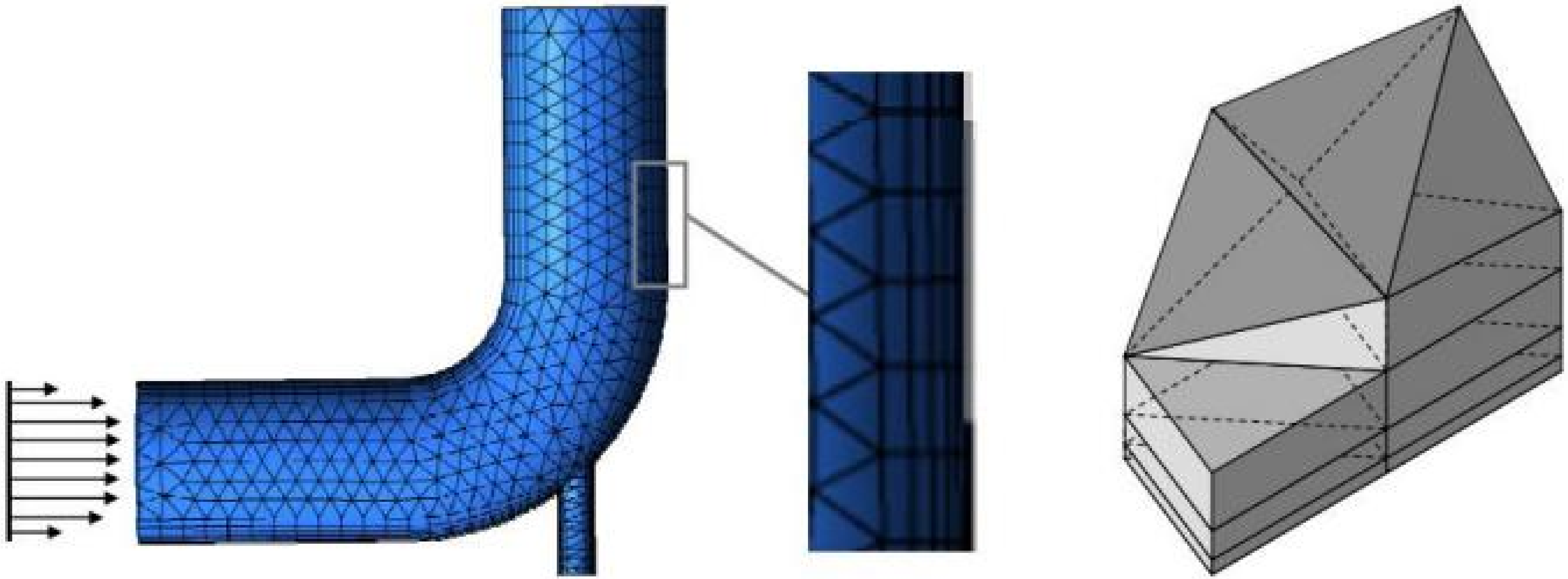


- **Tet Mesh**

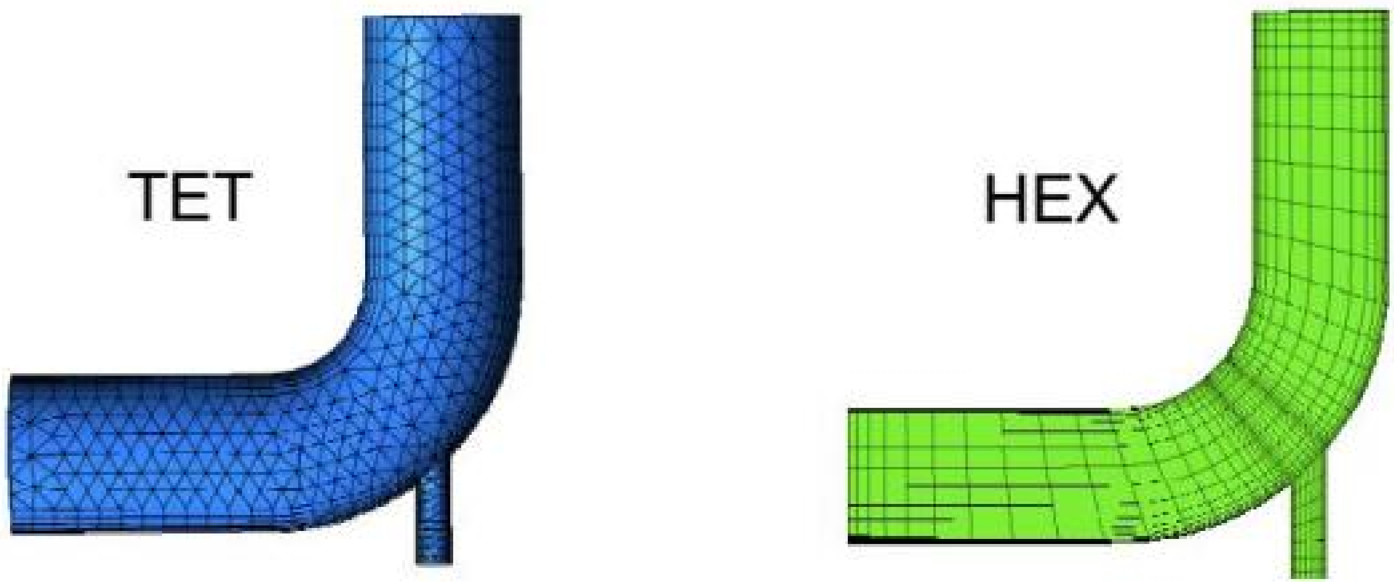
  2) Isotropic refinement – in order to capture gradients in one direction, mesh is refined in all three directions – cell counts rise rapidly



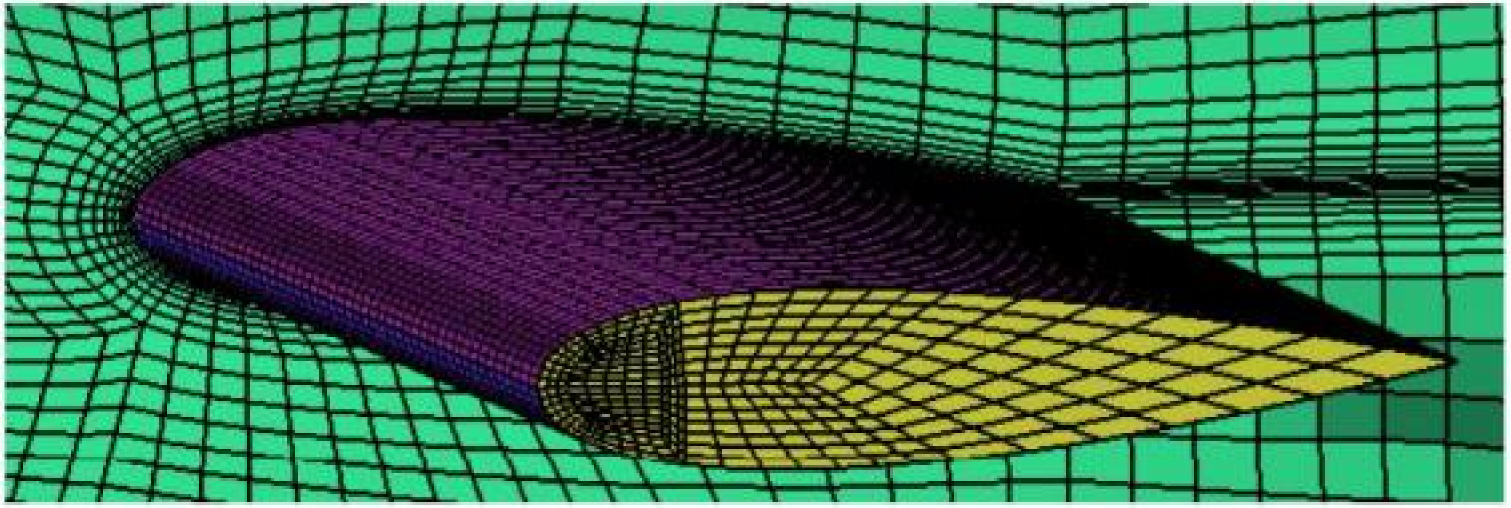
- **Tet Mesh**
  3) Inflation layer helps with refinement normal to the wall, but still isotropic in 2-D (surface mesh)



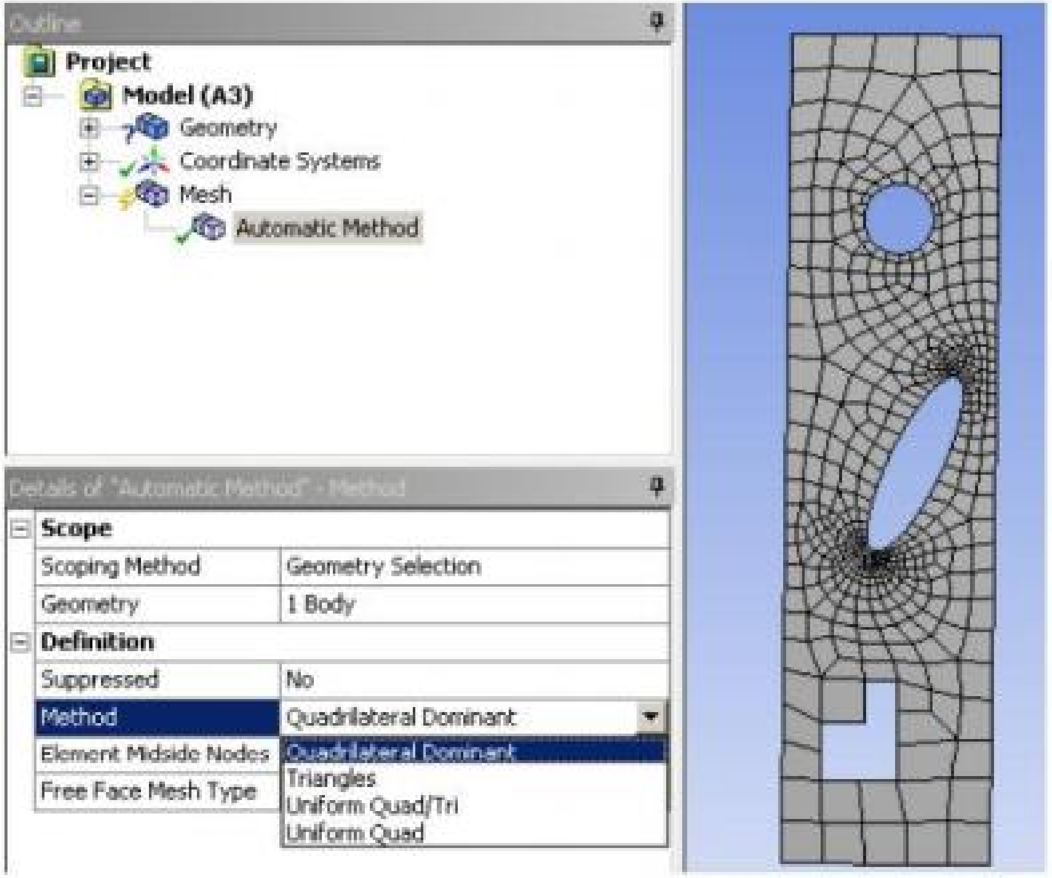
- **Hex Mesh**
  - Fewer elements required to resolve physics for most CFD applications
    - This hexahedral mesh, which provides the same resolution of flow physics, has LESS than half the amount of nodes as the tet-mesh)



- **Hex Mesh**
  - Fewer elements required to resolve physics for most CFD applications.
    - Anisotropic elements can be aligned with anisotropic physics (boundary layers, areas of tight curvature like wing leading and trailing edges)



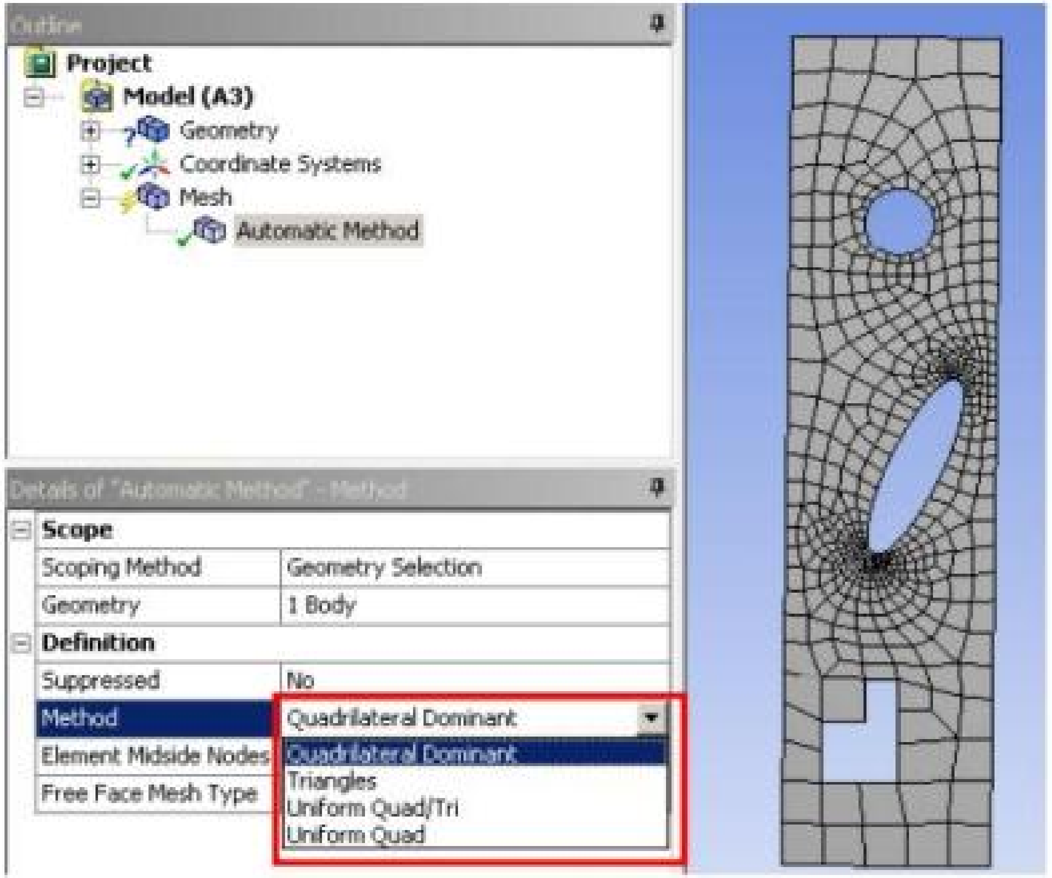
- There are four different meshing methods in the ANSYS Meshing Platform for 2D Geometry which can be applied to Surface Bodies or Shells:
  - Automatic Method (Quadrilateral Dominant)
  - All Triangles
  - Uniform Quad/Tri
  - Uniform Quad



- Both Fluent and the Mechanical Products in the ANSYS Portfolio accept 2D (Surface) Meshes for 2D Analyses
- Surface meshes or shell meshes are created for Surface Bodies created in DesignModeler or other CAD package
- CFX does not accept Surface Meshes as it is inherently a 3D Code. To do a 2D analysis in CFX, create a volume mesh (using Sweep) that is 1 element thick in the symmmetry direction for a
  - Thin Block for Planar 2D
  - Thin Wedge (< 5°) for Axisymmetric 2D
- The Surface Meshing methods discussed in this lecture are therefore appropriate for Fluent CFD or ANSYS Mechanical simulations

Meshing Methods for Surface Bodies

- Automatic Method (Quad Dominant)
- All Triangles
- Uniform Quad/Tri
- Uniform Quad



The following mesh controls will likely be most useful for Surface Meshing

- Edge Sizings
  - Use Bias Factors and Hard Divisions as necessary
- Mapped Face Meshing
  - To try to enforce a structured rather than a paved mesh
  - Note that sides, corners, and ends can be specified as Advanced options

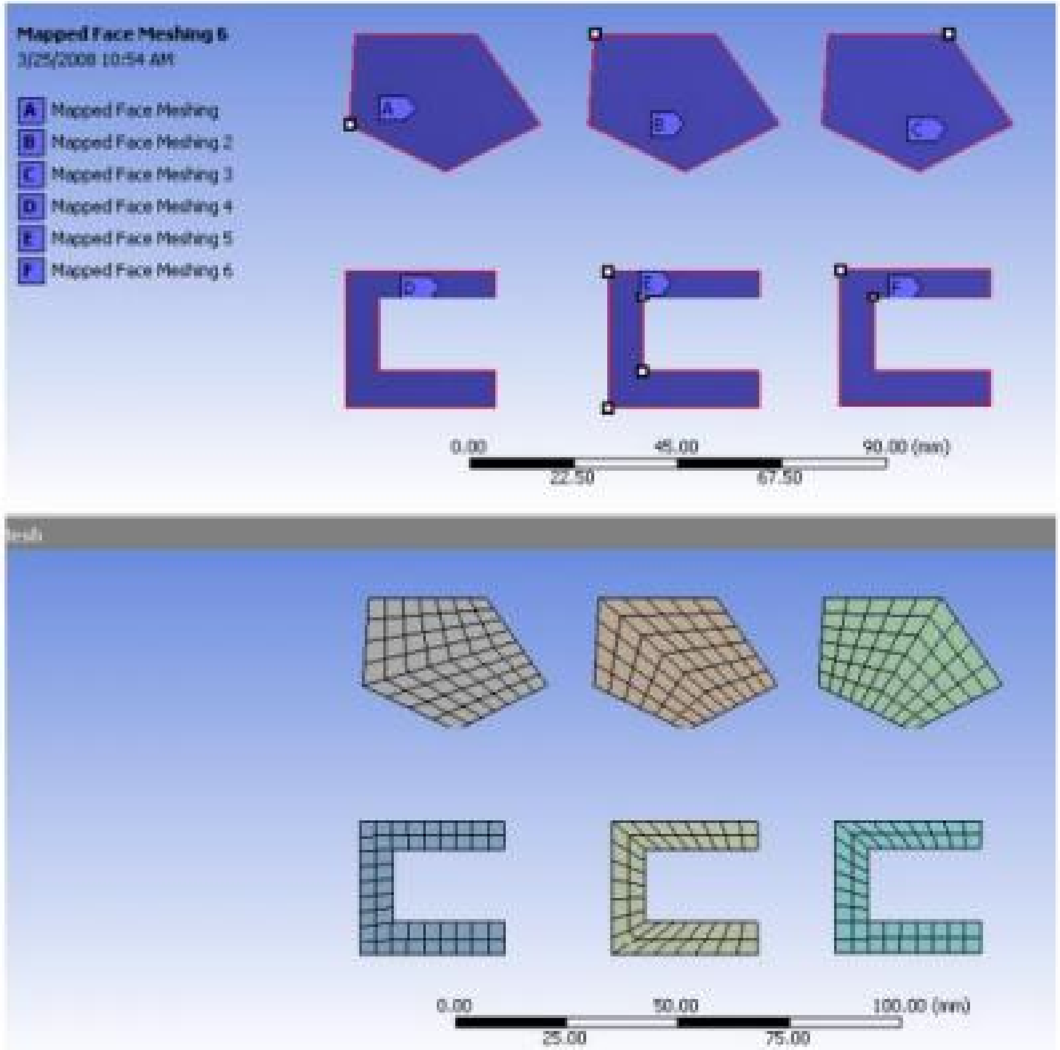
Details of "Edge Sizing 3" - Sizing	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
Definition	
Suppressed	No
Type	Number of Divisions
Number of Divisions	12
Behavior	Hard
Bias Type	___ __ _ _
Bias Factor	1.5

Details of "Mapped Face Meshing" - Mapped Face Meshing	
Scope	
Scoping Method	Geometry Selection
Geometry	4 Faces
Definition	
Suppressed	No
Method	Quadrilaterals
Constrain Boundary	No
Advanced	
Specified Sides	None
Specified Corners	None
Specified Ends	None

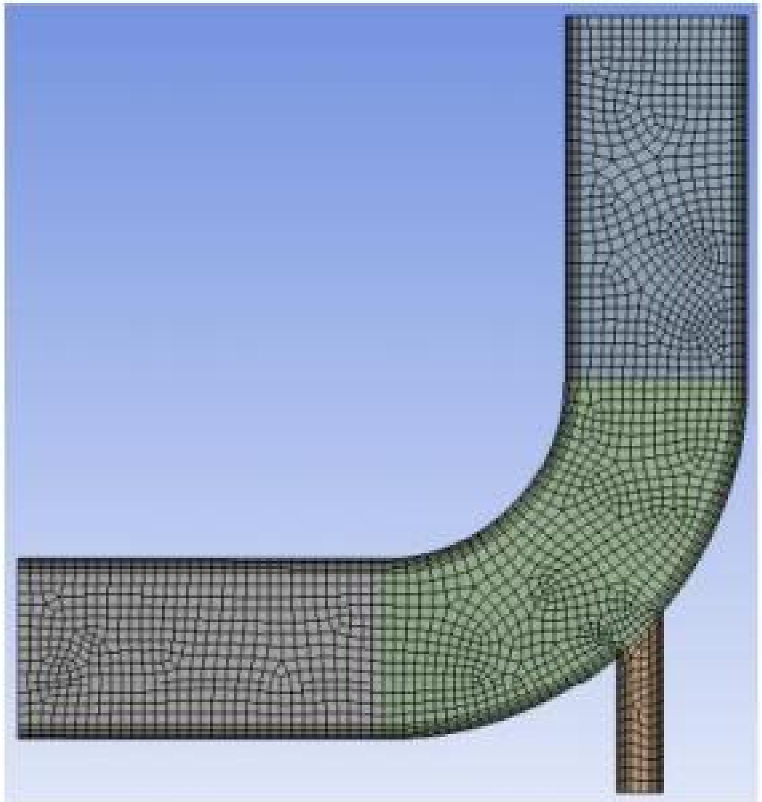
- Support for side/corner controls to define strategy for sub-mapping

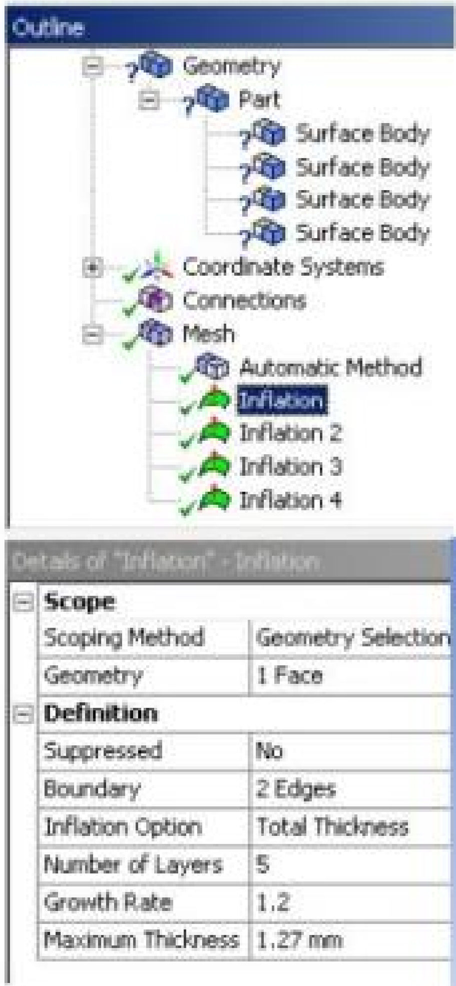
Details of "Mapped Face Meshing" - Mapped Face ...

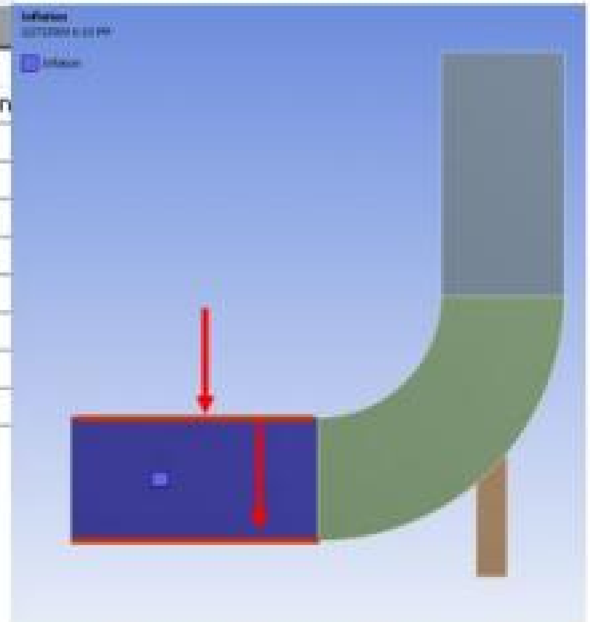
Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Suppressed	No
Method	Quadrilaterals
Constrain Boundary	No
Advanced	
Specified Sides	1 Vertex
Specified Corners	None
Specified Ends	None



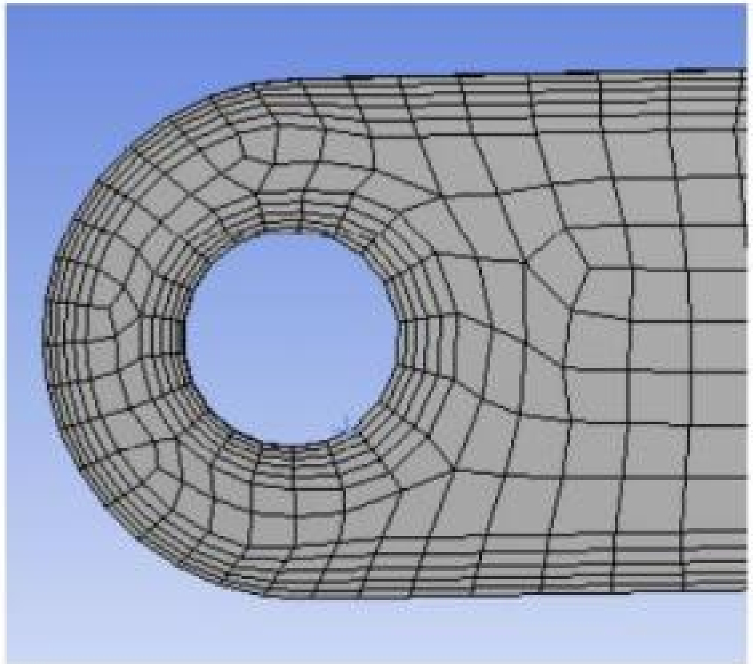
- Inflation can be defined for a Surface Meshing method
- It is scoped to the surface and defined for the edges as for a swept mesh



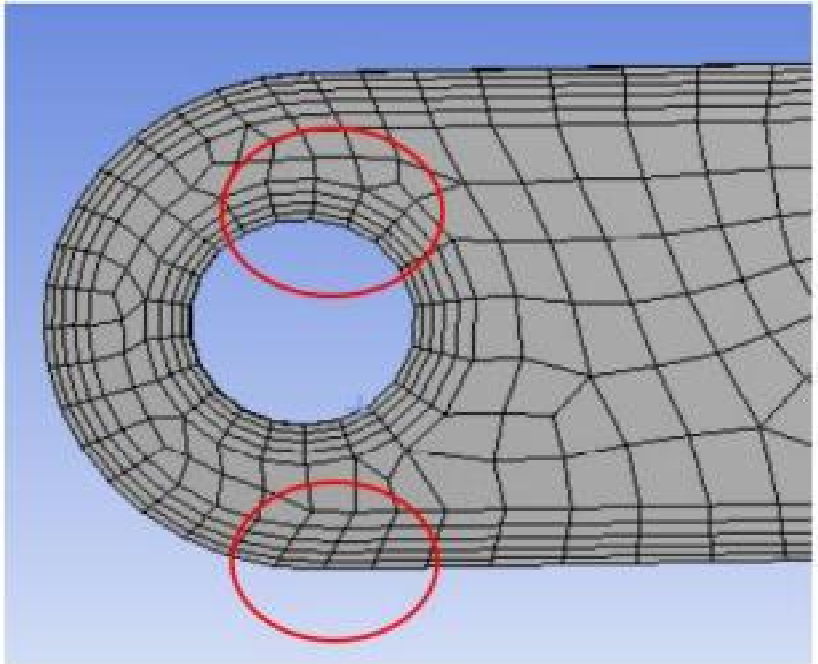




- 2-D inflation controls
  - 2-D planar models (Qmorph)
  - 2-D inflation for Sweep



Layer Compression

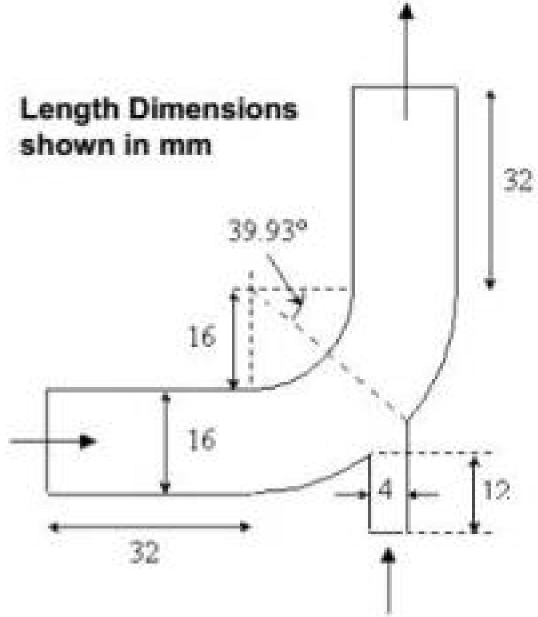


Stair-step

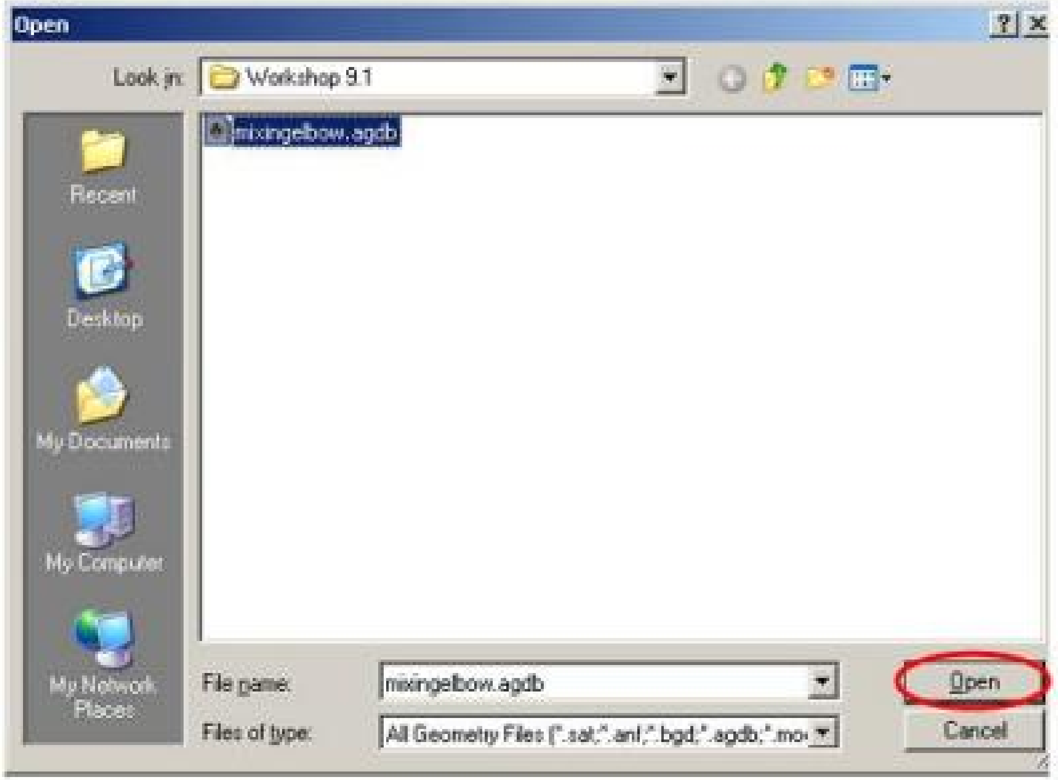
- Example: 2D mixing elbow configuration

This workshop will take you through the process of generating a mesh for a 2-D mixing elbow. The mixing elbow configuration used here is a 2-D representation of piping systems often encountered in power plants. The flow and temperature field in the neighborhood of mixing region is of interest to the CFD analyst.

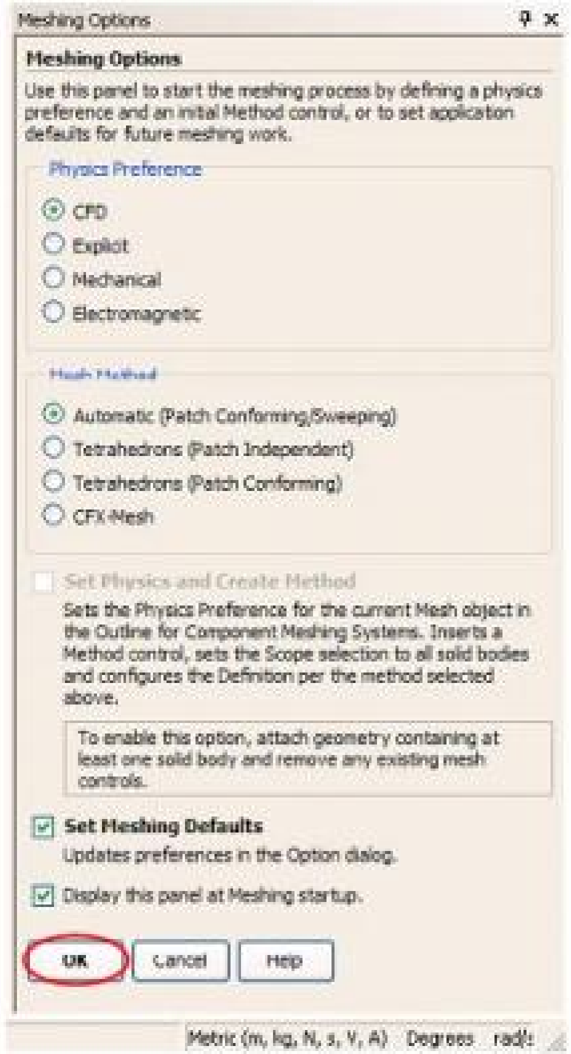
- Goals:
  - Import the 2D geometry created in DesignModeler to ANSYS Meshing.
  - Create a 2-D mesh on the surface body.



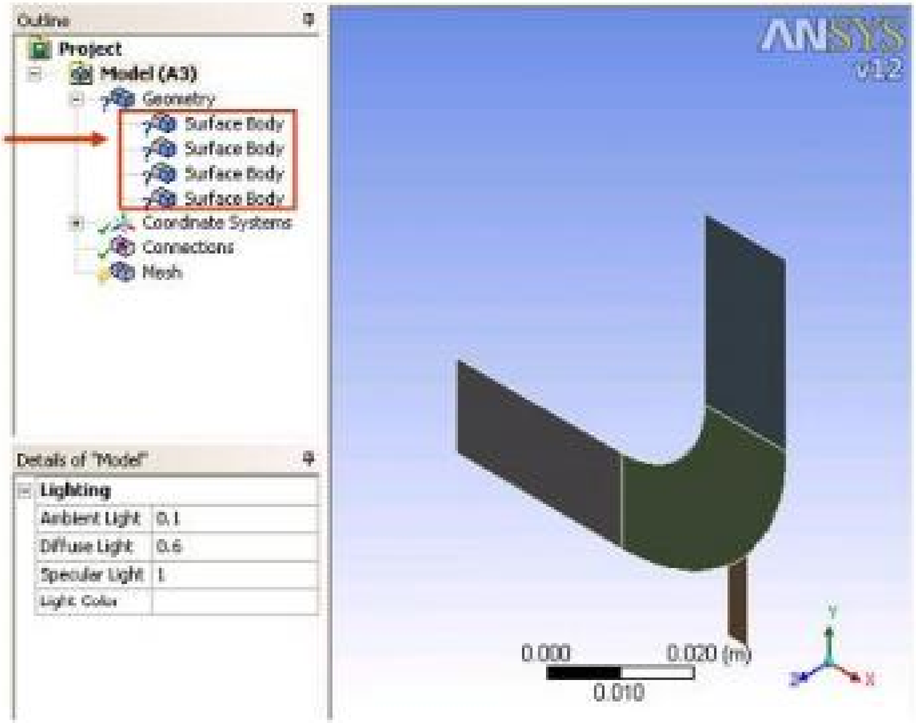
1. Copy the *mixingelbow.agdb* file from the Tutorial Files folder to your working directory
2. Start Workbench and double-click the Mesh entry in the Component Systems panel
3. Right-click on Geometry in the Mesh entry in the Project Schematic and select Import Geometry/Browse
4. Browse to the block and mixingelbow.agdb file you copied and click Open. Note that the Geometry entry in the Project Schematic now has a green check mark.



5. In the Projects window, double-click on Mesh to open ANSYS Meshing
6. Select the Physics Preference to CFD and select the Automatic mesh method in the Meshing Options panel and click OK.

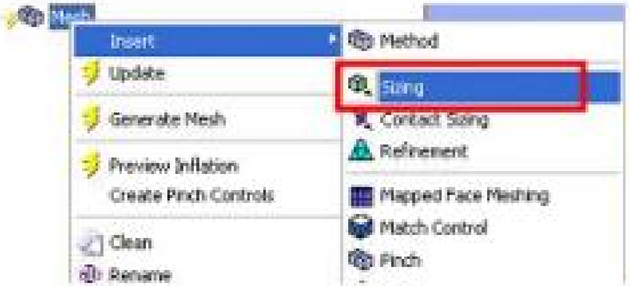


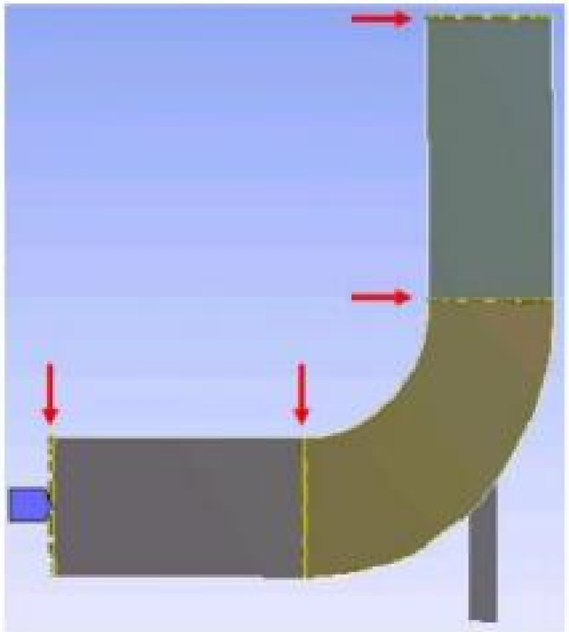
4 Surface Bodies



7. Right-click on Mesh in the Outline, and select *Insert/Sizing*. Change the Selection Filter to Edges and select the 4 edges as shown.

8. Set the Type for the Ege Sizing to Number of Divisions and enter 10 for the number. Set the behavior to Hard. Set the Bias Type to shrink towards the edges and set a Bias factor of 10.

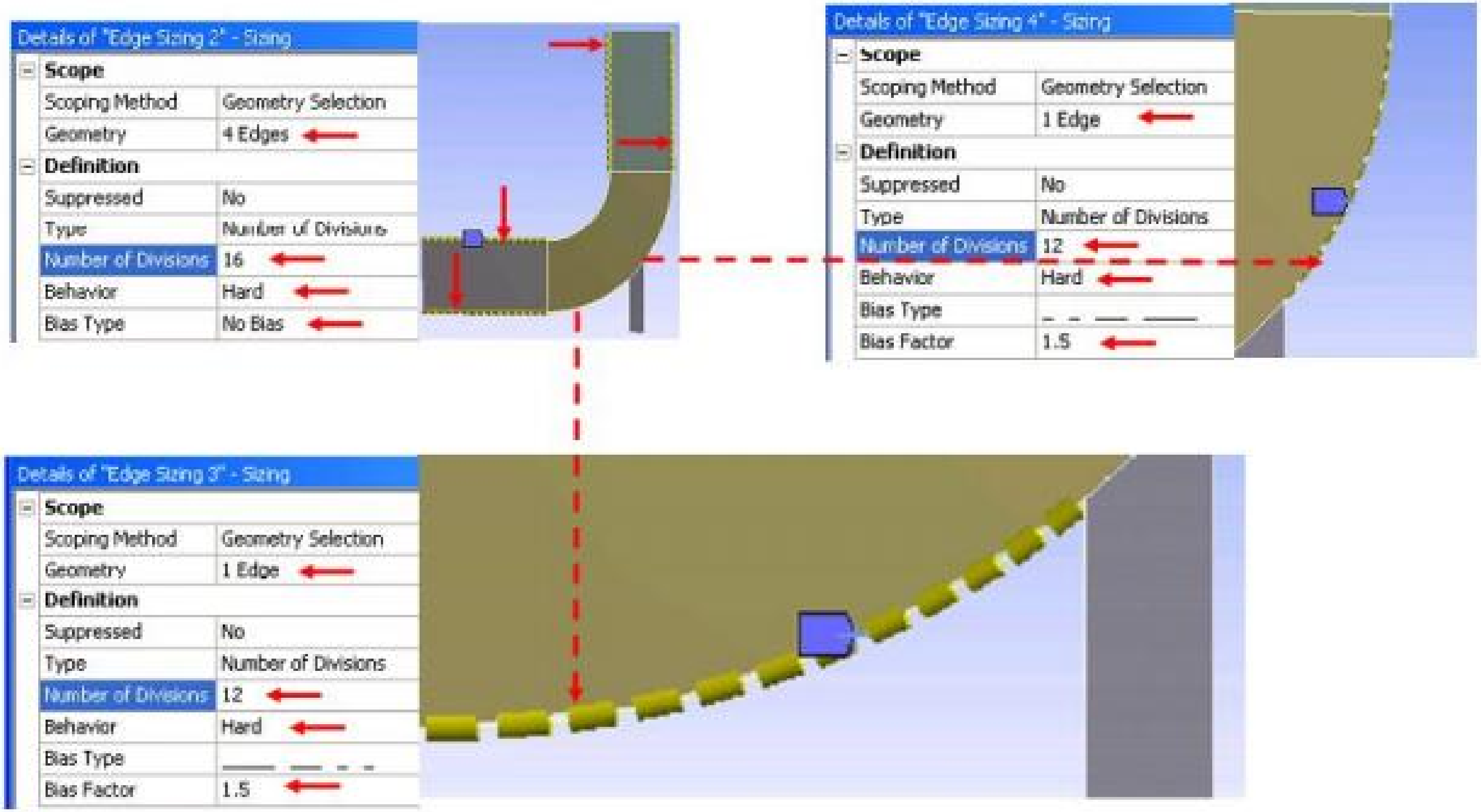





Details of "Edge Sizing" - Sizing

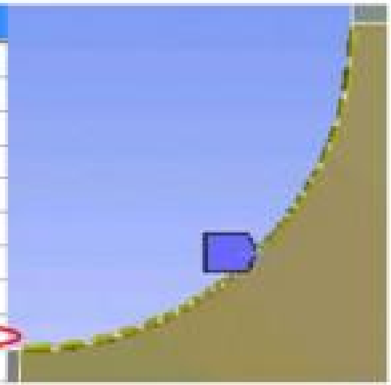
Scope	
Scoping Method	Geometry Selection
Geometry	4 Edges
Definition	
Suppressed	No
Type	Number of Divisions
Number of Divisions	10
Behavior	Hard
Bias Type	
Bias Factor	10.

9. Repeat the process for the other edges as shown.



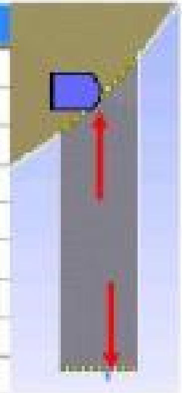
Details of "Edge Sizing 5" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
Definition	
Suppressed	No
Type	Number of Divisions
Number of Divisions	34
Behavior	Hard
Bias Type	— — - — —
Bias Factor	10.



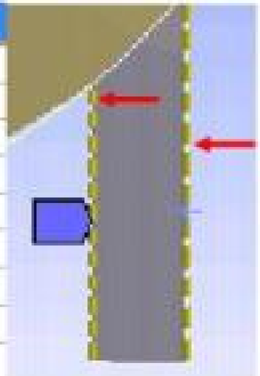
Details of "Edge Sizing 6" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	2 Edges
Definition	
Suppressed	No
Type	Number of Divisions
Number of Divisions	10
Behavior	Hard
Bias Type	No Bias



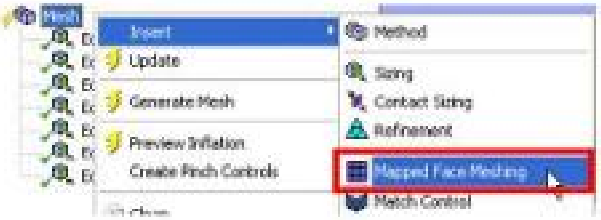
Details of "Edge Sizing 7" - Sizing

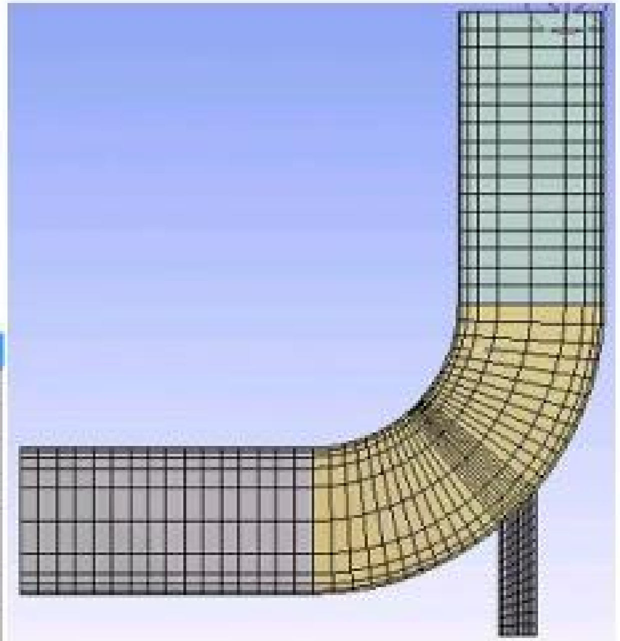
Scope	
Scoping Method	Geometry Selection
Geometry	2 Edges
Definition	
Suppressed	No
Type	Number of Divisions
Number of Divisions	12
Behavior	Hard
Bias Type	No Bias

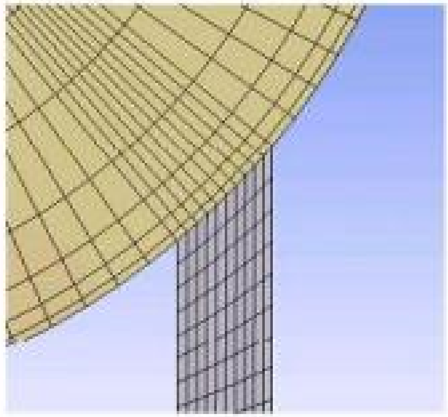


10. Insert a Mapped Face Meshing as shown. Select all four faces.

11. Generate the mesh. Note the inflation-like approach from the edge sizings







Details of "Mapped Face Meshing" - Mapped Face Meshing

Scope	
Scoping Method	Geometry Selection
Geometry	4 Faces
Definition	
Suppressed	No
Method	Quadrilaterals
Constrain Boundary	No
Advanced	
Specified Sides	None
Specified Corners	None
Specified Ends	None