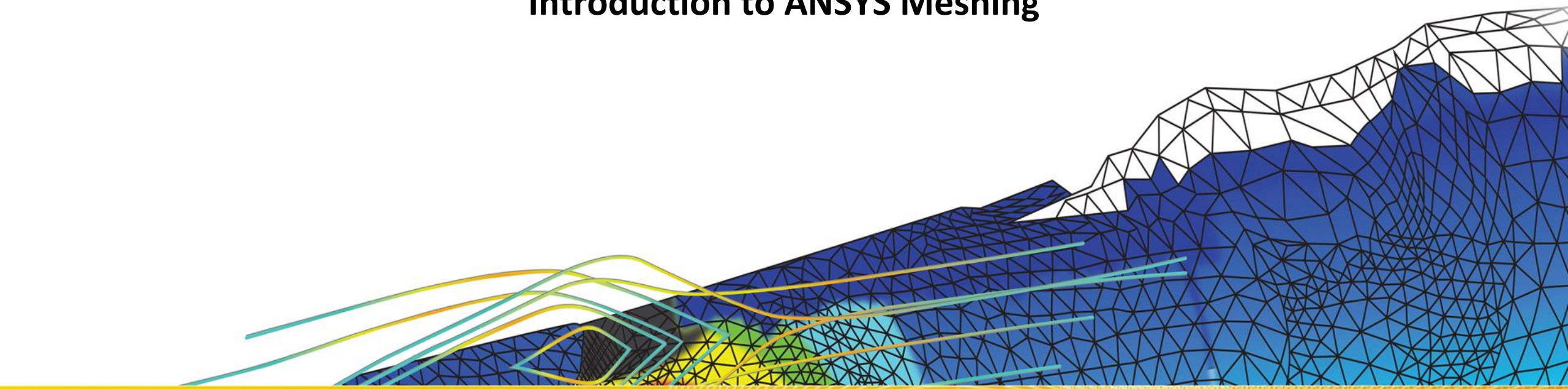
Realize Your Product Promise®

17.0 Release

Tutorial 5: ANSYS Meshing Basics

Introduction to ANSYS Meshing



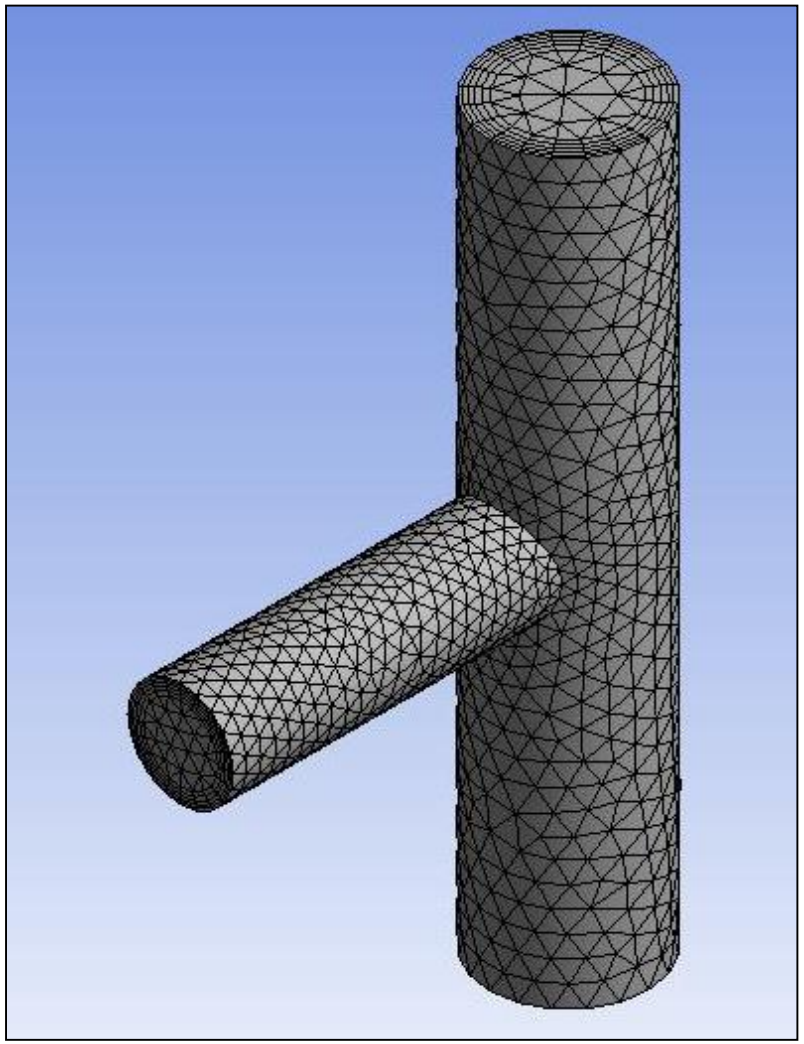
Introduction

Background

- This workshop assumes little or no prior experience of ANSYS Meshing. Basic workflow will be demonstrated

Objectives

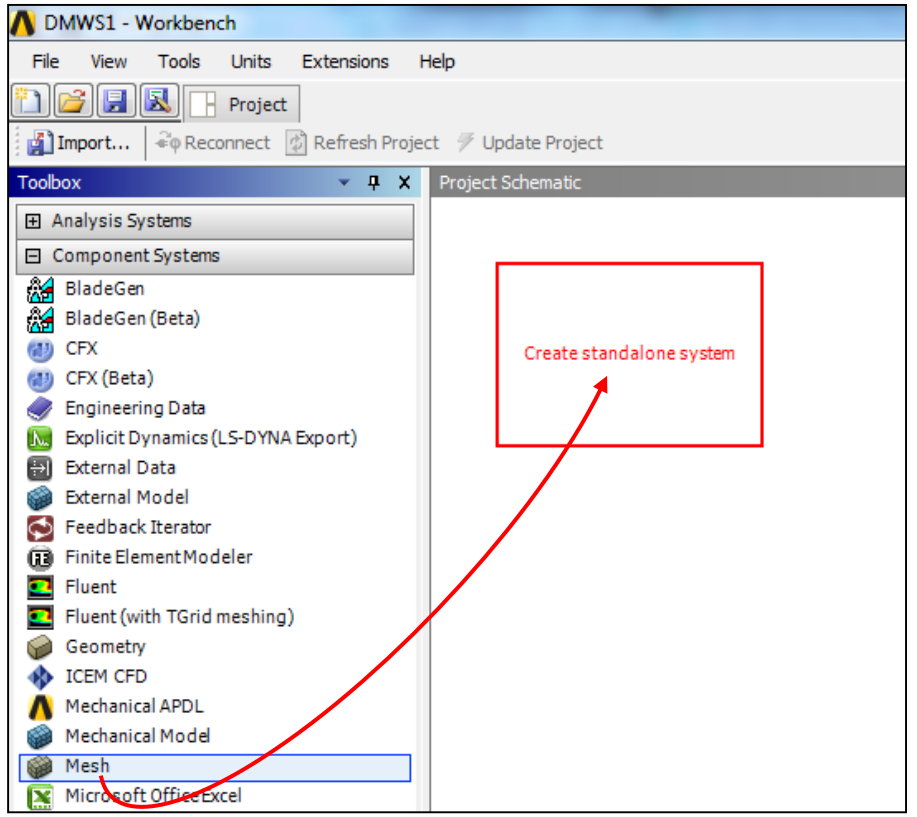
- Generating a mesh
- Creating Named Selections
- Using Inflation
- Checking mesh quality
- Saving the project

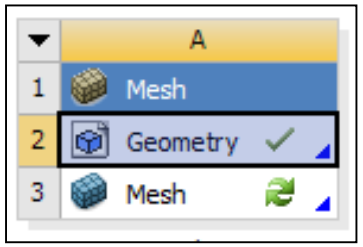


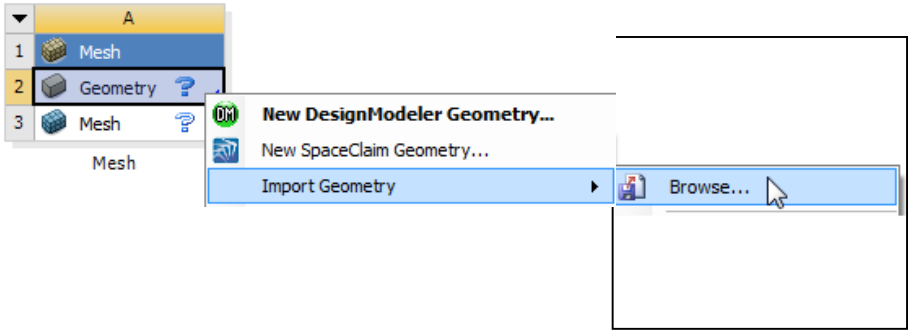
Project Startup (1)

Create the Project

- Drag and drop a Mesh component system into the Project Schematic
- Right click on the Geometry cell (A2) and select Import Geometry → Browse
- Locate the file “pipe-tee.stp” in the Meshing Workshops Input Files folder (Module 01) and select it. The geometry cell will show a check mark indicating it is up to date



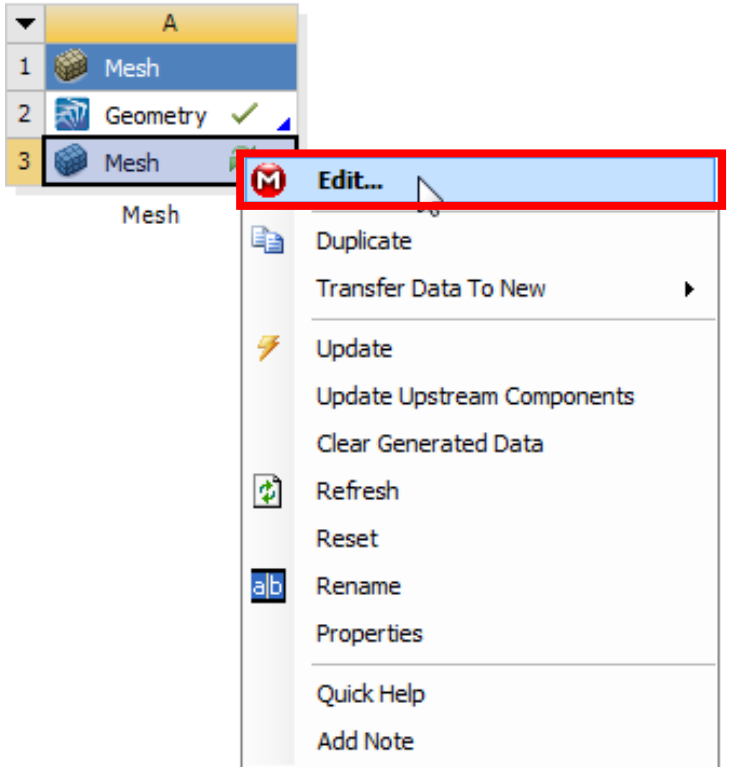


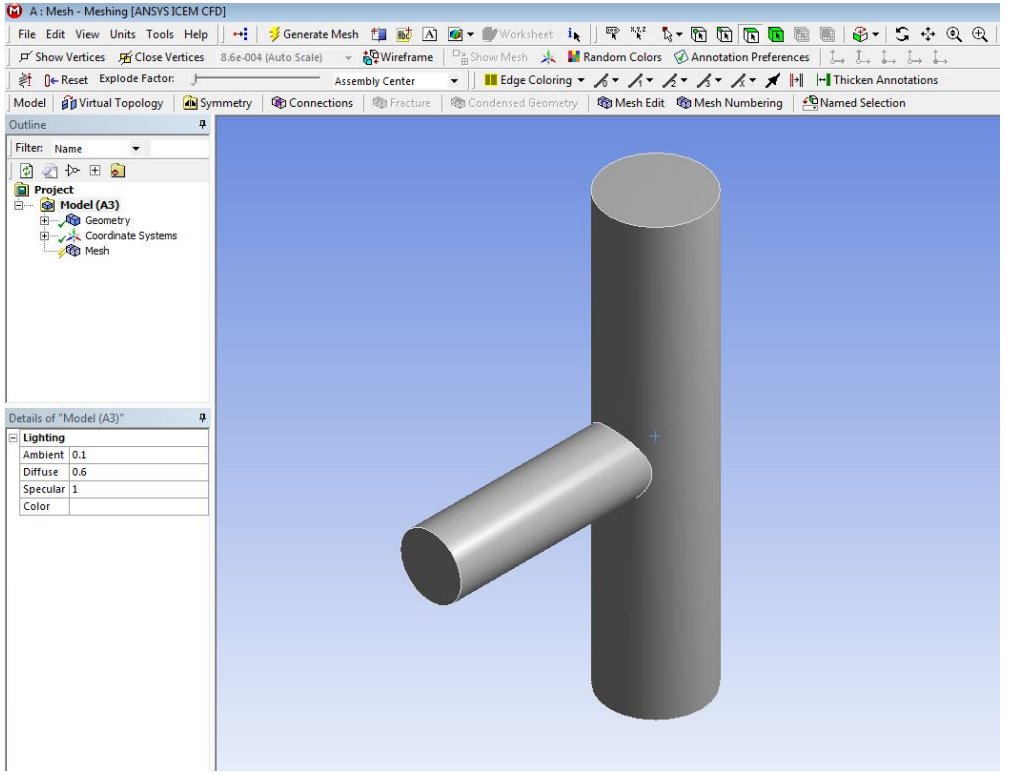


Project Startup (2)

Start Meshing

- On the Mesh cell right click and select Edit
  - *Note: Double clicking on the Mesh cell can also startup Meshing*
- ANSYS Meshing will start up and load the geometry

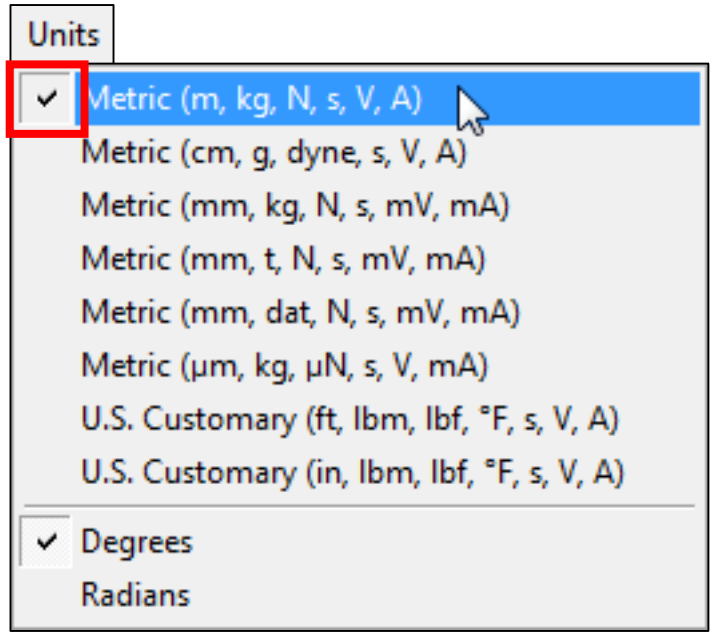




Units

Set Units

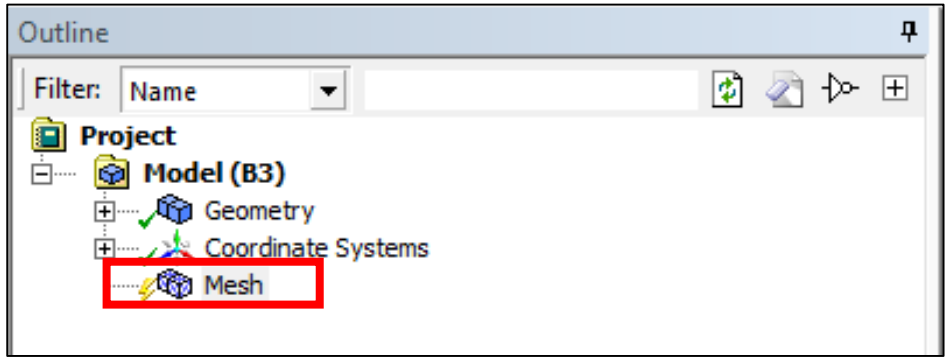
- It’s good practice to first check and, if required, set the units.
- From the main menu select Units and, if it is not already set, specify Metric (m...).

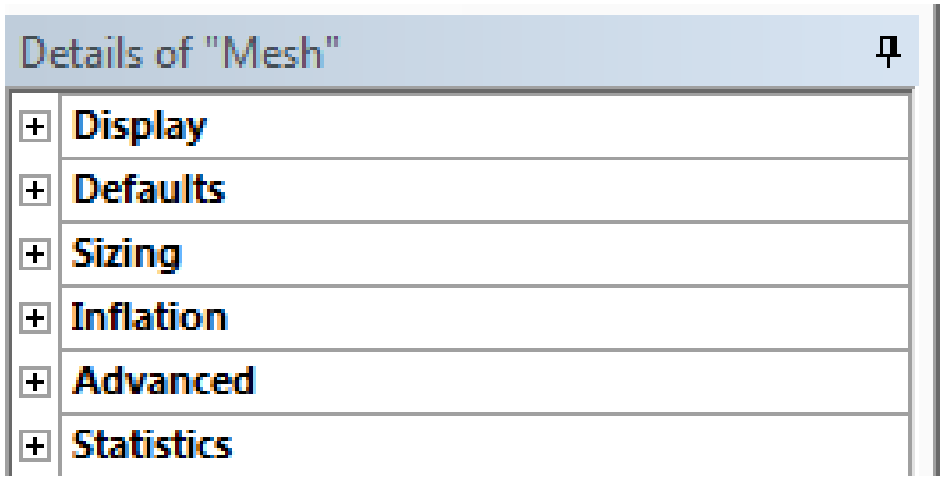


Defaults

Set Defaults

- We'll first demonstrate how a basic mesh can quickly be generated with the minimum of input.
- Click Mesh in the Outline
- This will display the Details of "Mesh" panel which contains Global Mesh Control settings.

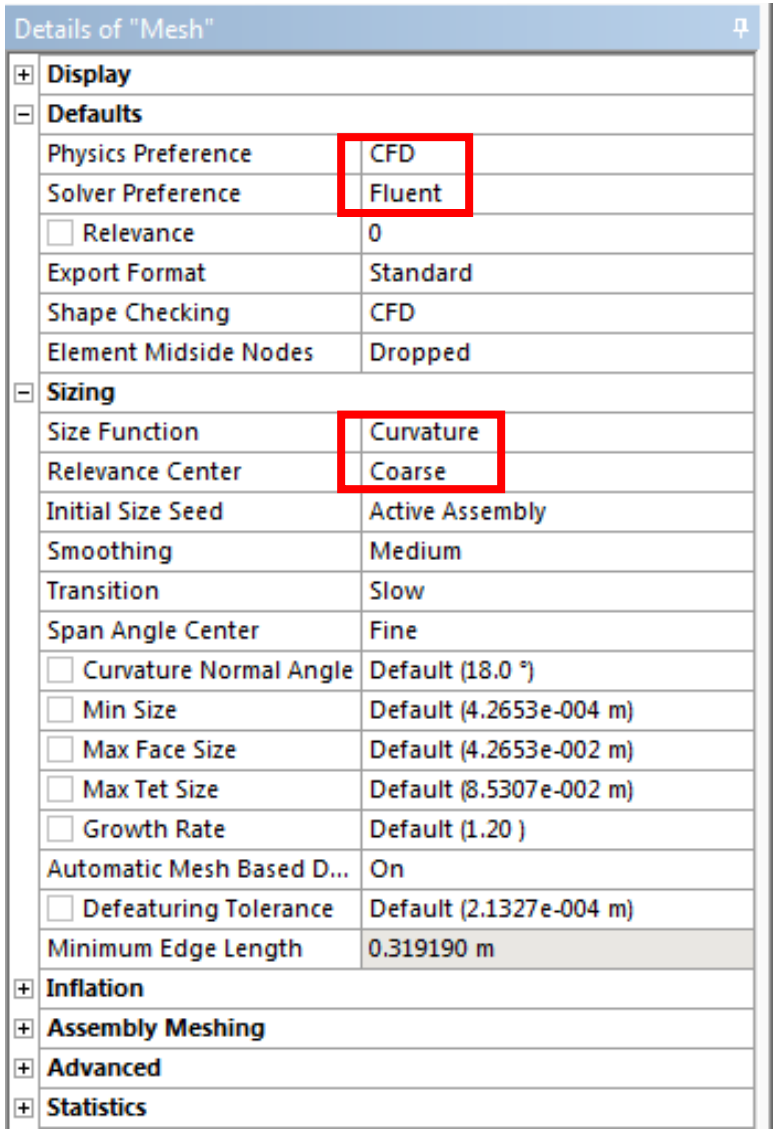




Mesh Settings

Set Defaults

- In the Details of Mesh panel expand Defaults and click in the box to the right of Physics Preference to activate the drop down box
  - Select CFD
  - Set the Solver Preference to Fluent
- Under Sizing set the following.
  - Use Advanced Size Function: On: Curvature
  - Relevance Center: Coarse
- Click the “Update” button located in the toolbar

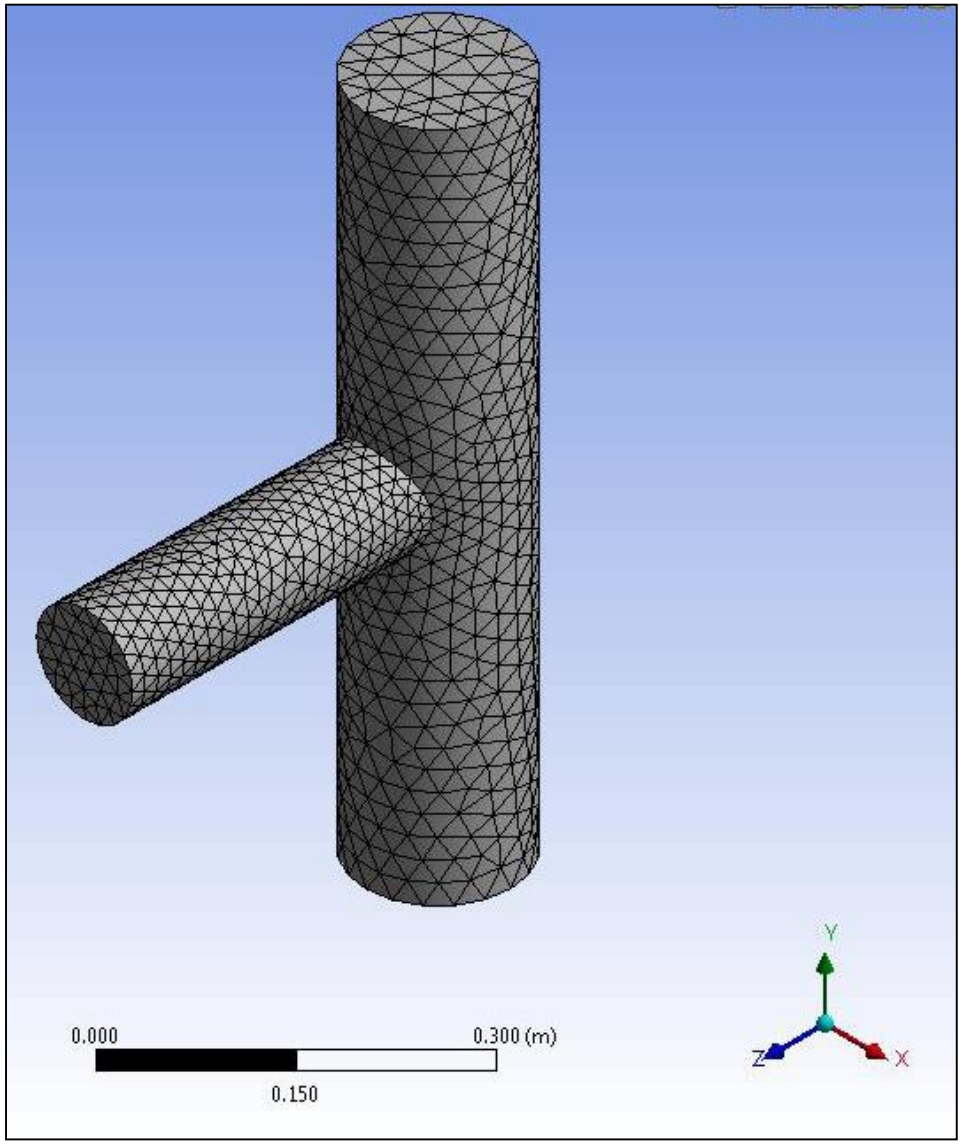


View the Mesh

Mesh

- Check that the generated mesh matches that shown
  - This is a fully automatic tetrahedron mesh
  - Appropriate sizing and parameters have been set and calculated based upon the selected physics preference and geometry characteristics

- The next step is to specify which geometric entities will form boundaries for use in the solver by creating Named Selections

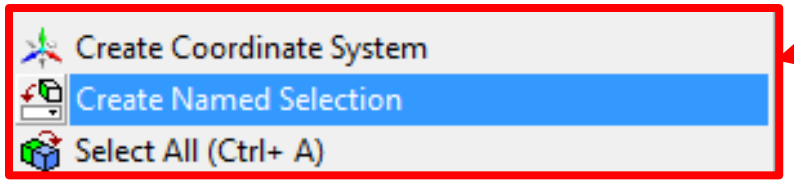


Named Selections (1)

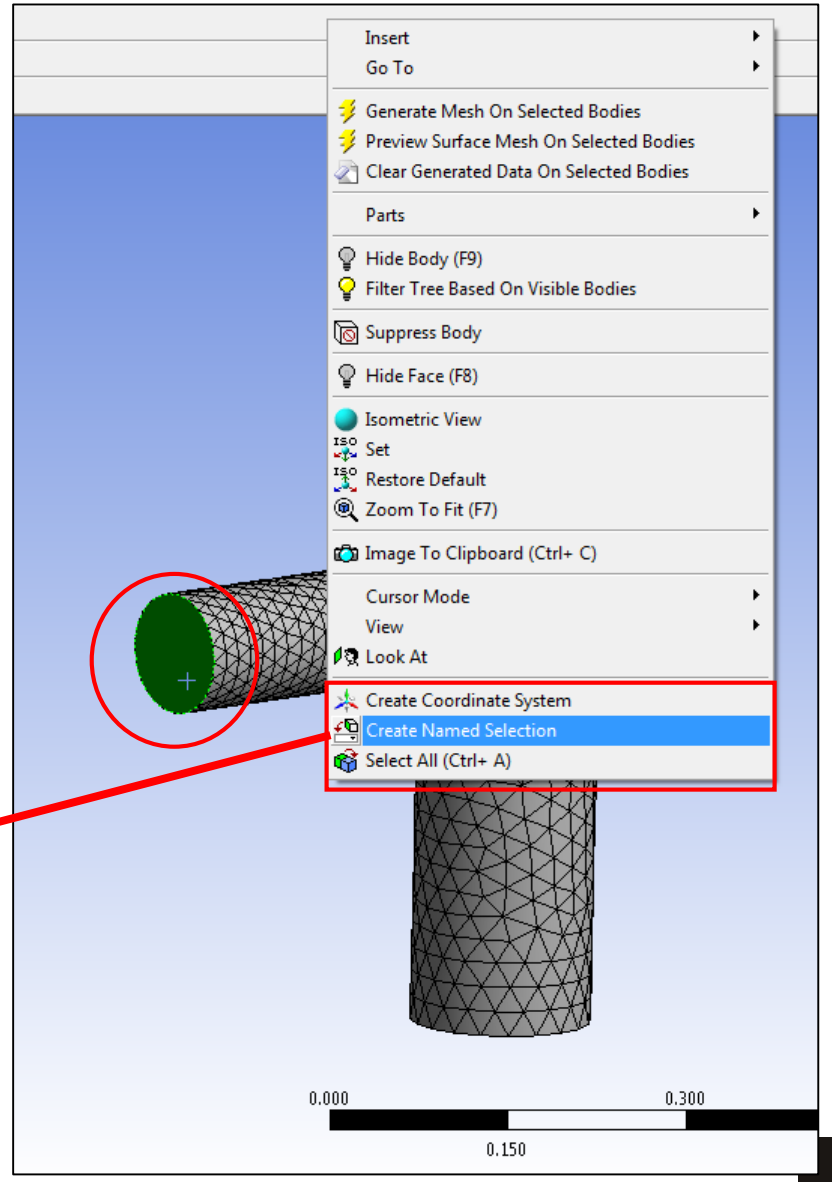
Add Named Selections

- From the top toolbar select the Face Selection Filter

- Select the face as shown by left clicking over it (will turn green)
- With the face selected, right click and select Create Named Selection from the Context Menu that appears



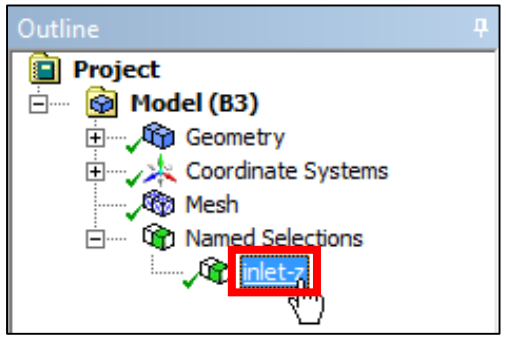
- The Named Selection Dialog Box will appear



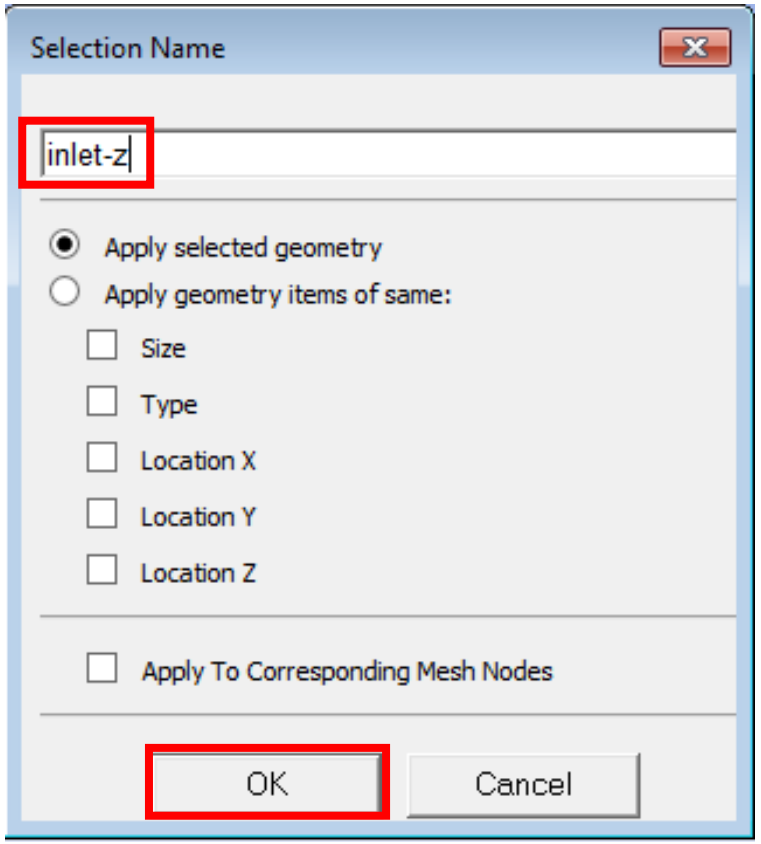
Named Selections (2)

Add Named Selections (Continued)

- In the Named Selections Dialog Box enter the name “inlet-z” as shown
- Click OK
- The Named Selection you have just created will be listed under the Named Selections object in the Outline



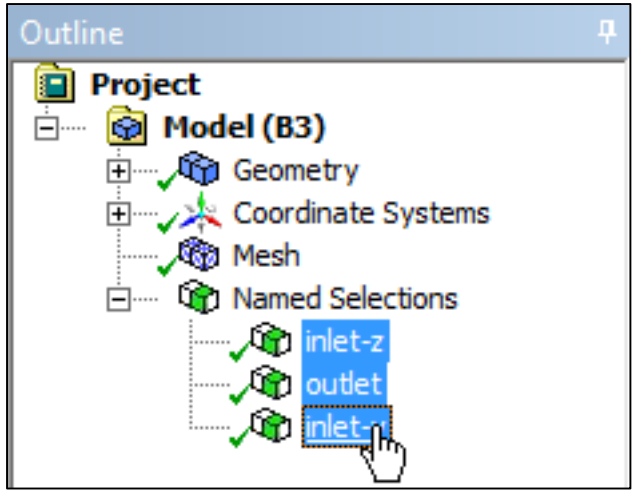
- Selecting it will highlight the corresponding faces



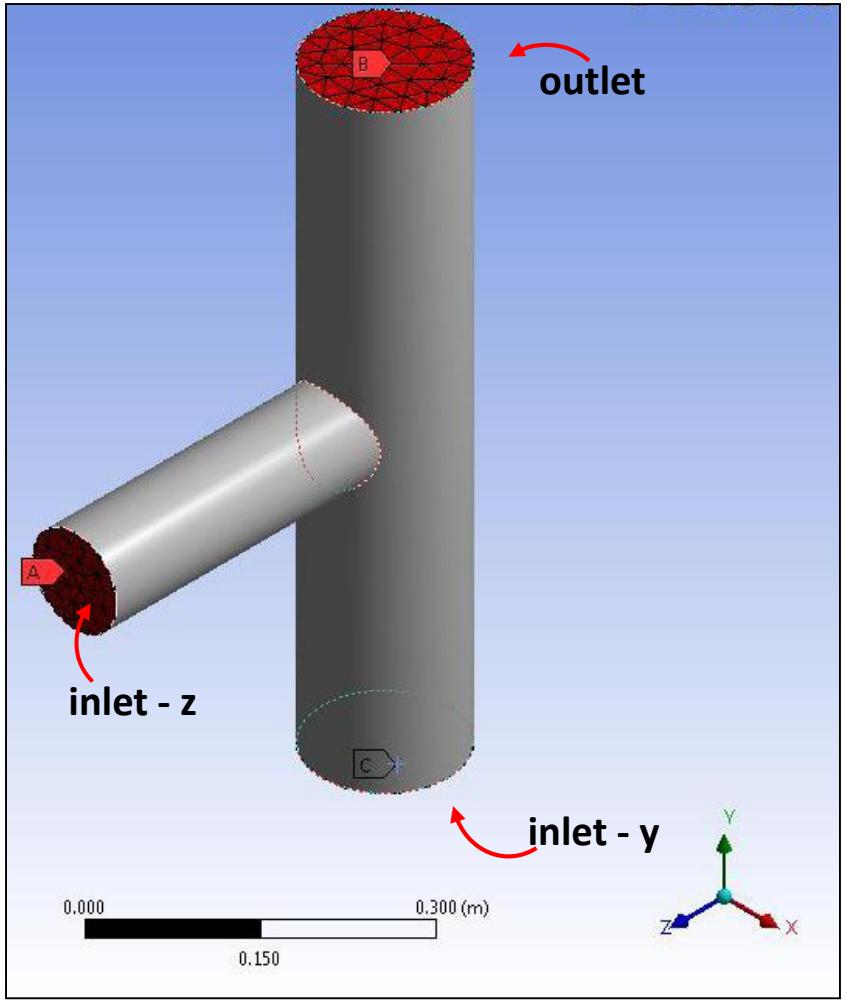
Named Selections (3)

Add Named Selections (Continued)

- Use exactly the same procedure to create two more Named Selections “inlet-y” & “outlet”



- These Named Selections will be used to define boundary conditions in the fluid solver



Review

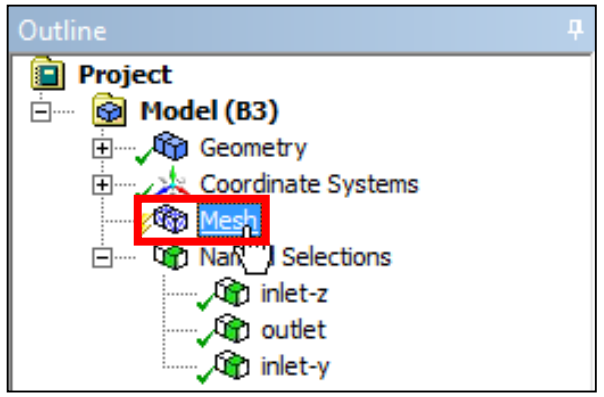
- The mesh we have just generated may be suitable for a simple laminar flow calculation
- There are many ways in which a mesh can be generated to accommodate for the requirements of specific applications
- More complex analyses, for example CFD solutions containing turbulent flow, have additional requirements
- We'll now look at an example

Inflation

Mesh with Inflation Layers

- The fluid simulation will concern a wall bounded turbulent flow. To adequately resolve flow gradients near the wall we need smaller mesh cells near the wall
- An efficient way to achieve this is by ‘inflating’ the wall surface mesh to produce layers of thin prismatic cells called Inflation Layers
- In the Outline, click mesh to display the Details of Mesh panel
- Expand the Inflation Section and set Use Automatic Inflation to Program Controlled
  - Click Update 

- The mesh will be regenerated with the new settings

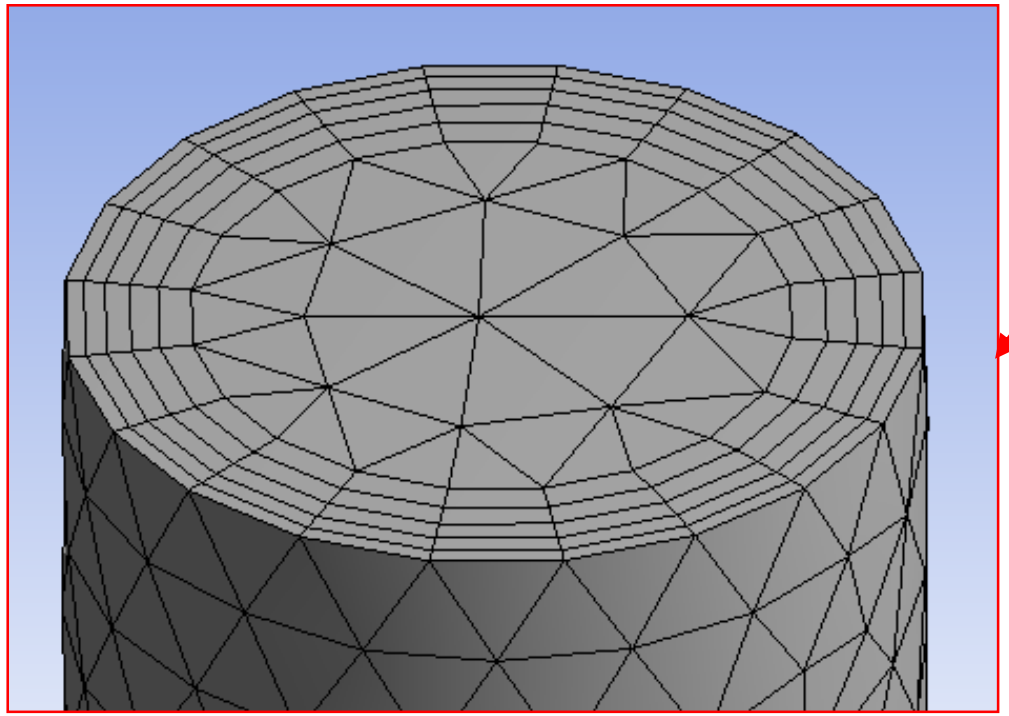


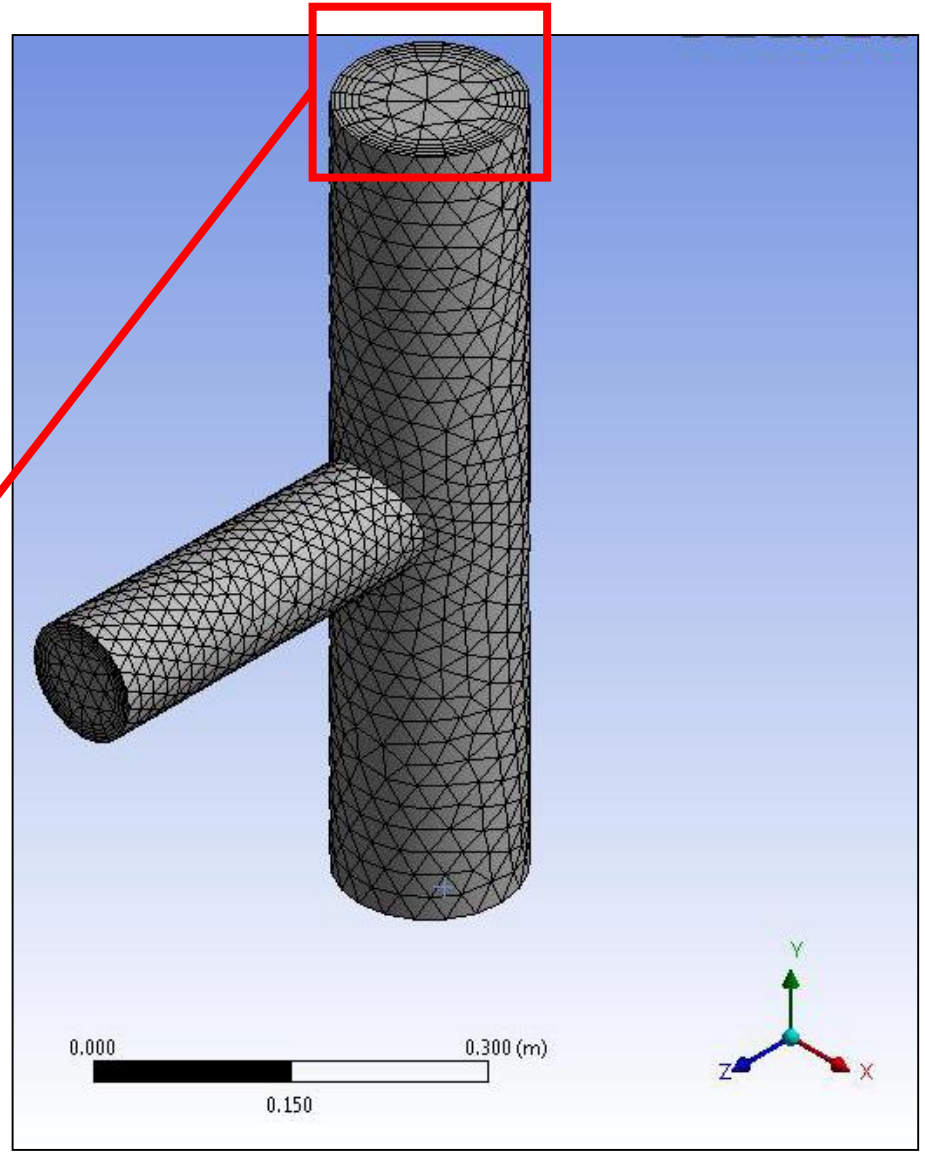
Inflation	
Use Automatic Inflation	Program Controlled
Inflation Option	Smooth Transition
Transition Ratio	0.272
Maximum Layers	5
Growth Rate	1.2
Inflation Algorithm	Pre
View Advanced Options	No

Inflated Mesh

Review the Inflated Mesh

- The program controlled inflation has automatically excluded the Named Selection faces creating layers on the unnamed wall faces

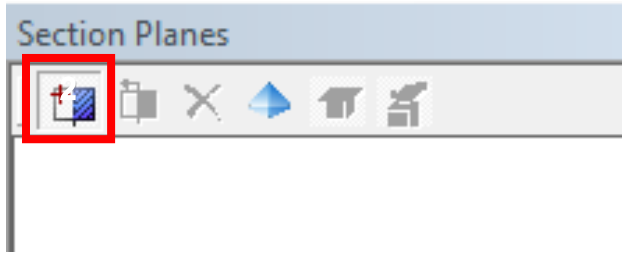




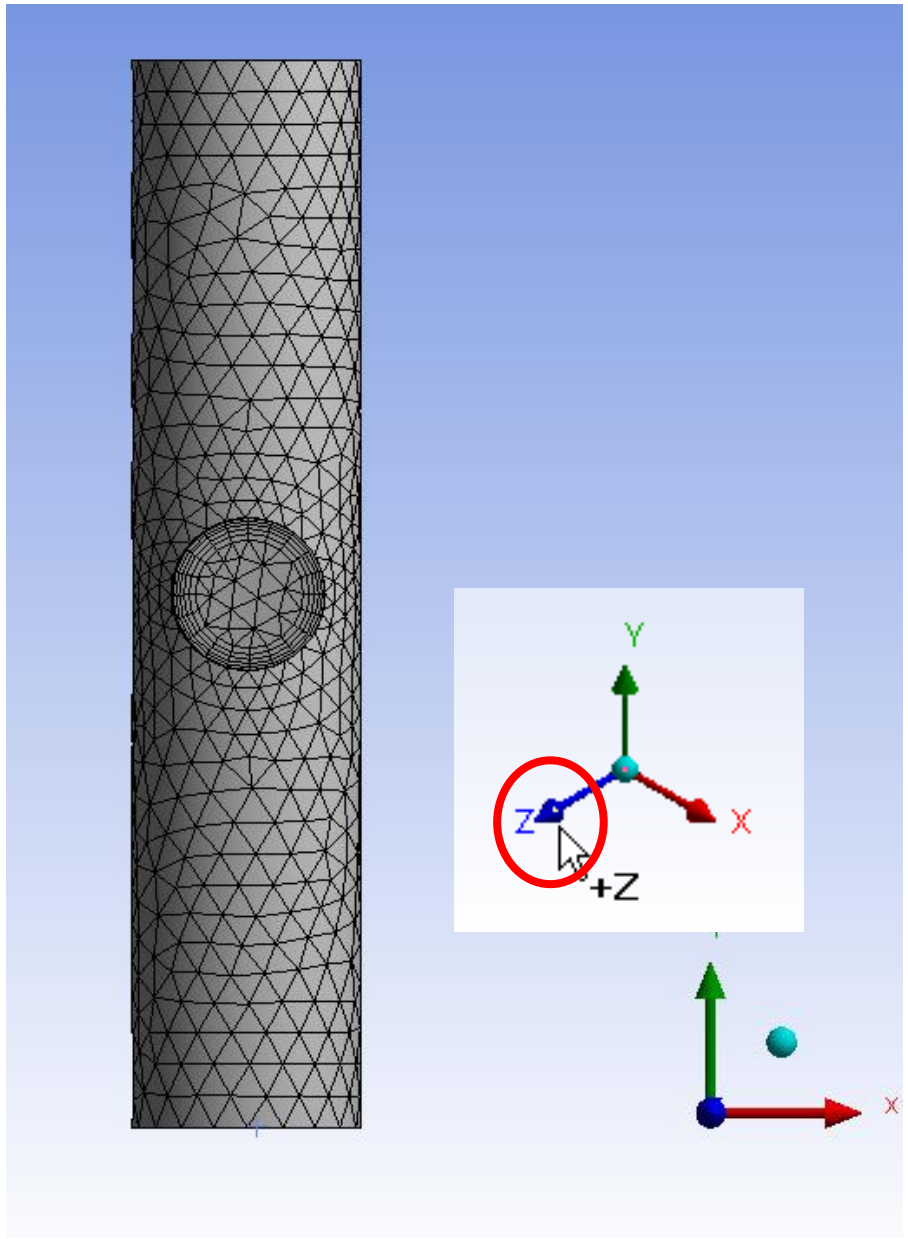
Section Planes (1)

View Mesh Interior

- To inspect the interior of a mesh Section planes are used
- Click the +Z axis to orientate the view as shown
- Click the New Section Plane button in the Section Planes panel (lower left)



- If the panel is not visible activate it by selecting View → Windows → Section Planes from the main menu bar

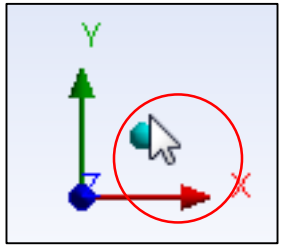


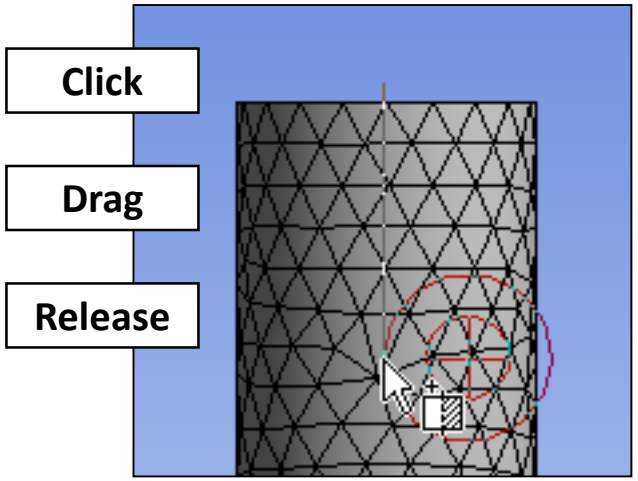
Section Planes (2)

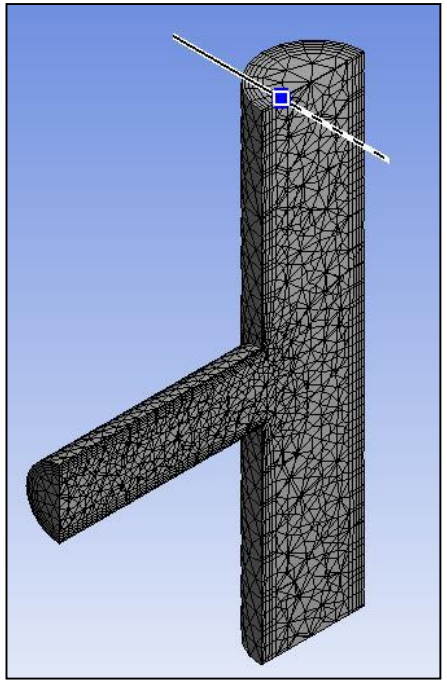
View Mesh Interior (Continued)

- Create the Section Plane by clicking and dragging a vertical line down through the geometry as shown
  - It's not necessary to drag the line all the way through the geometry – just far enough to establish a vertical line then release

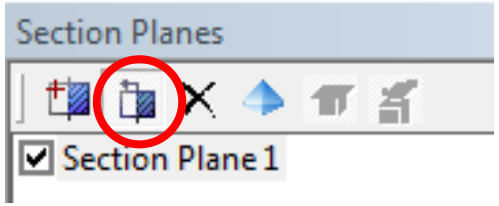
- Click the blue iso ball to snap to the isometric view



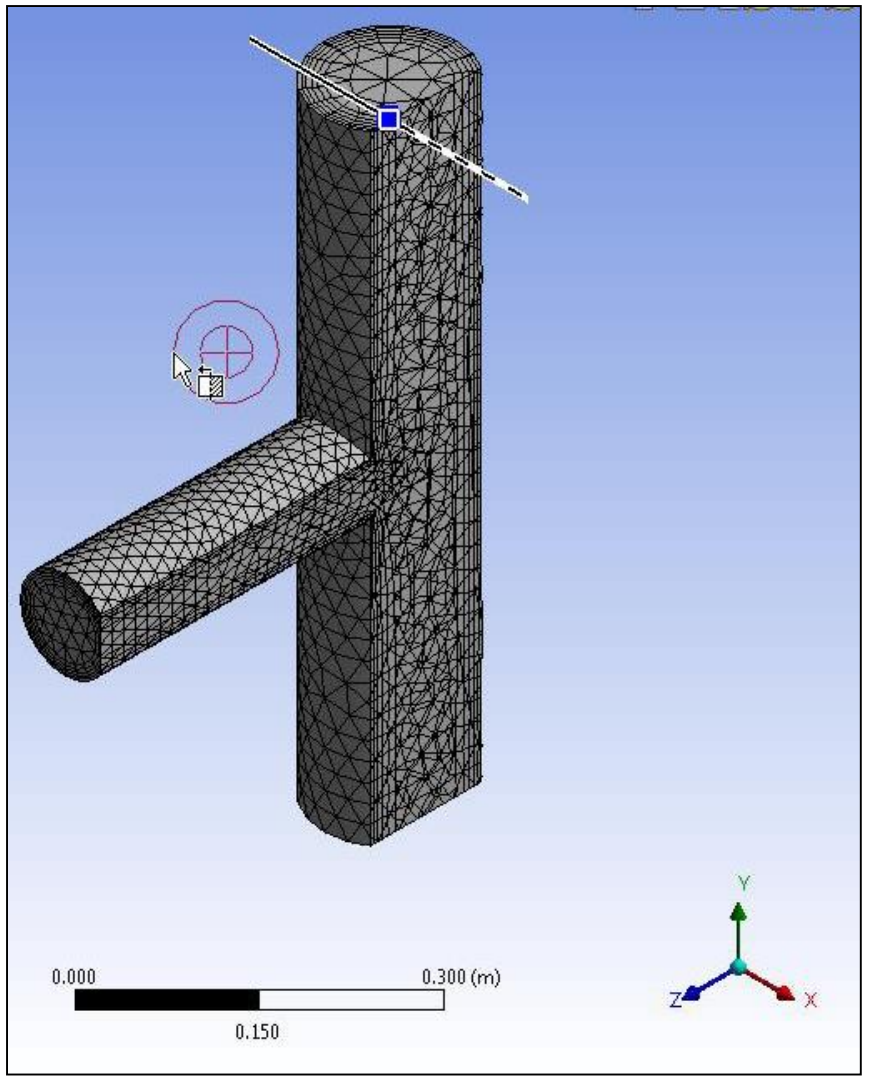




Section Planes (3)

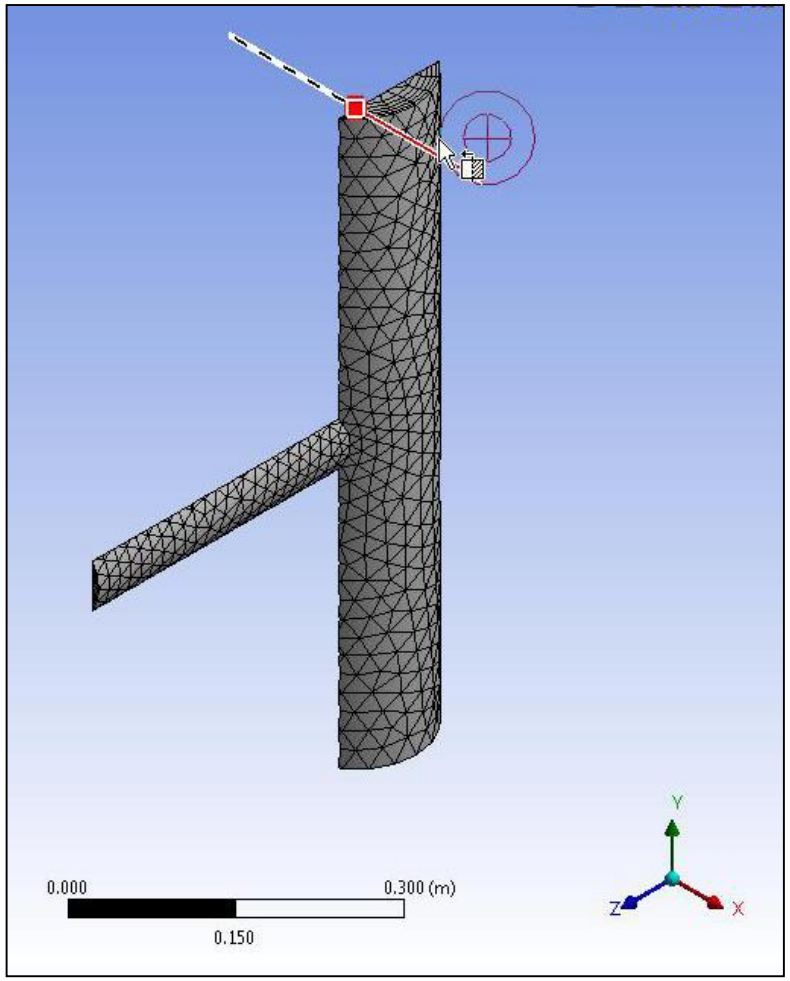


- Click on “Edit Section Plane” button, then click anywhere in the graphics window and drag to slide the Section through the mesh
- Release to set the new position



Section Planes (4)

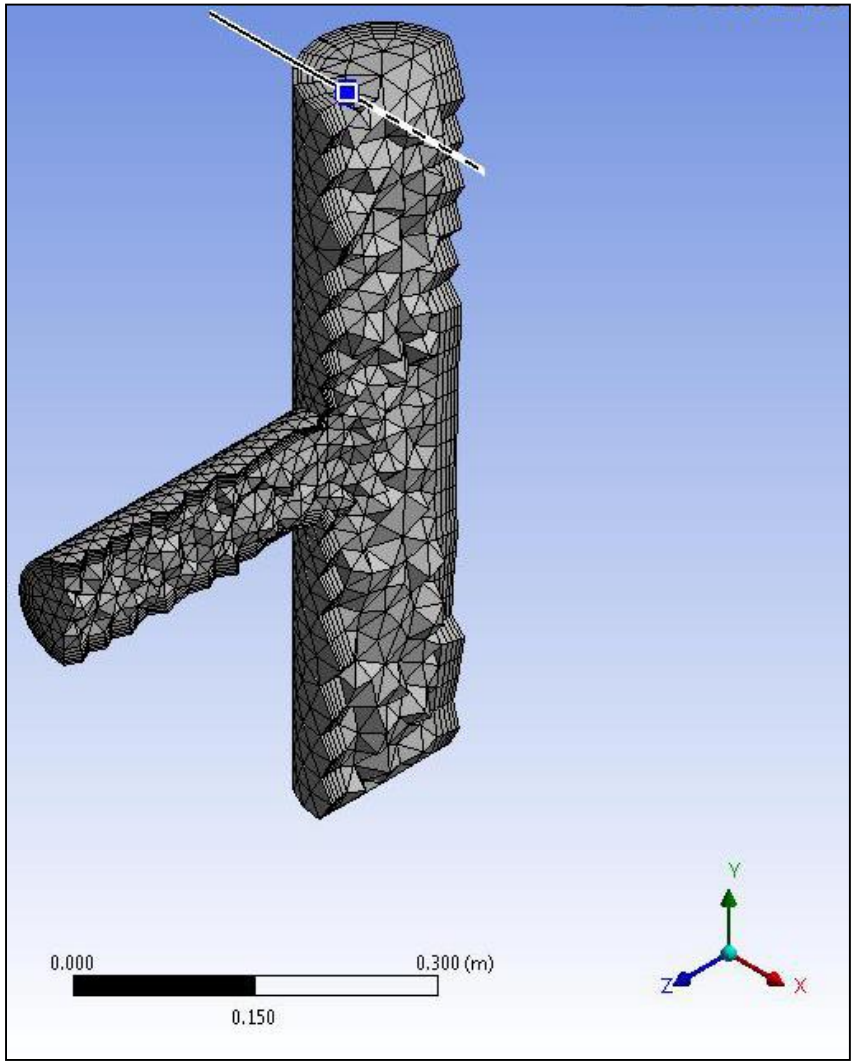
- Click either side of Section Plane tool to toggle the culling
- One done with setting section plane, click again on “Edit Section Plane” button to de-activate it



Section Planes (5)

- **Display whole elements using the button in the Section Plane Panel**

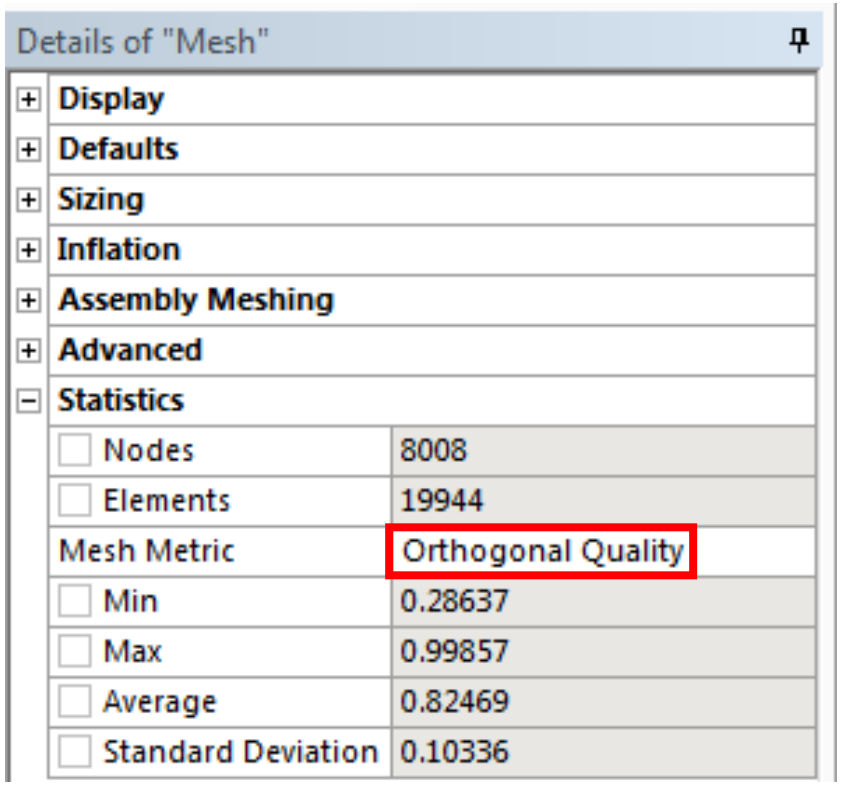
- **Deactivate the Section Plane by unchecking the box “Slice Plane 1” in the Section Plane Panel**



Mesh Statistics

Check Quality

- Before using a mesh in any solver it is important to check the mesh quality
- Quality is defined through various metrics which measure the degree to which each mesh cell varies from an ideal shape
- In the Details of Mesh panel expand Statistics. Click in the box to the right of Mesh Metric and select Orthogonal Quality
- The minimum value for Orthogonal Quality is important and should not fall below 0.05. The minimum for this mesh is acceptable



Save the Project

This completes the workshop

- **From the main menu select File → Close Meshing**
  - **Workbench will save any application data**
- **From the Workbench Project Page use the file menu and save the project as "AMWS1.1_cfd.wbpj" to your working folder**