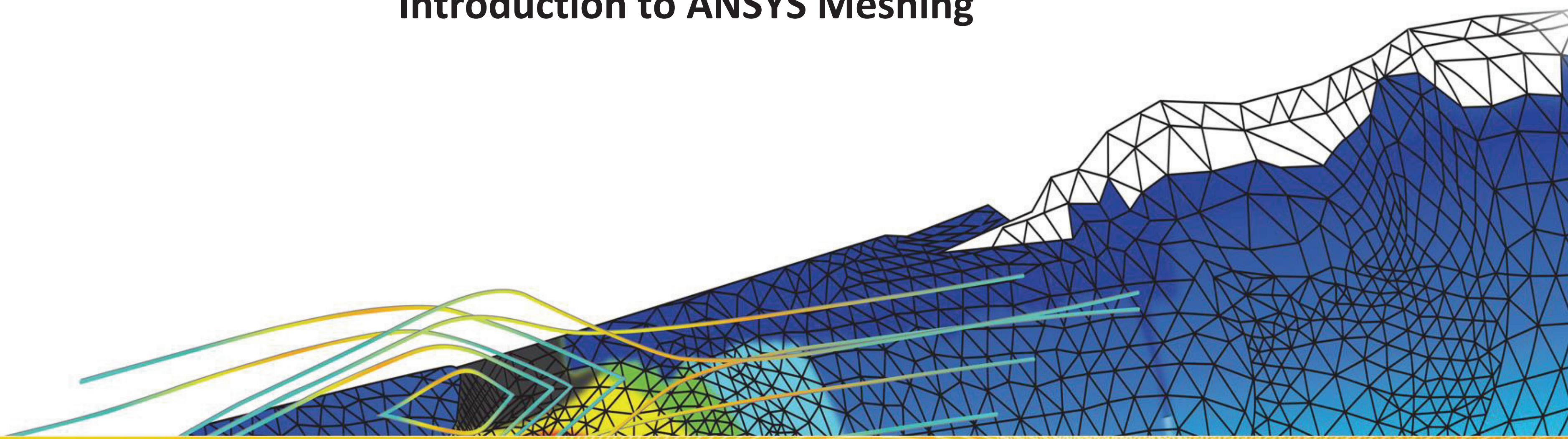
Realize Your Product Promise®

17.0 Release

Tutorial 6: ANSYS Meshing Methods

Introduction to ANSYS Meshing



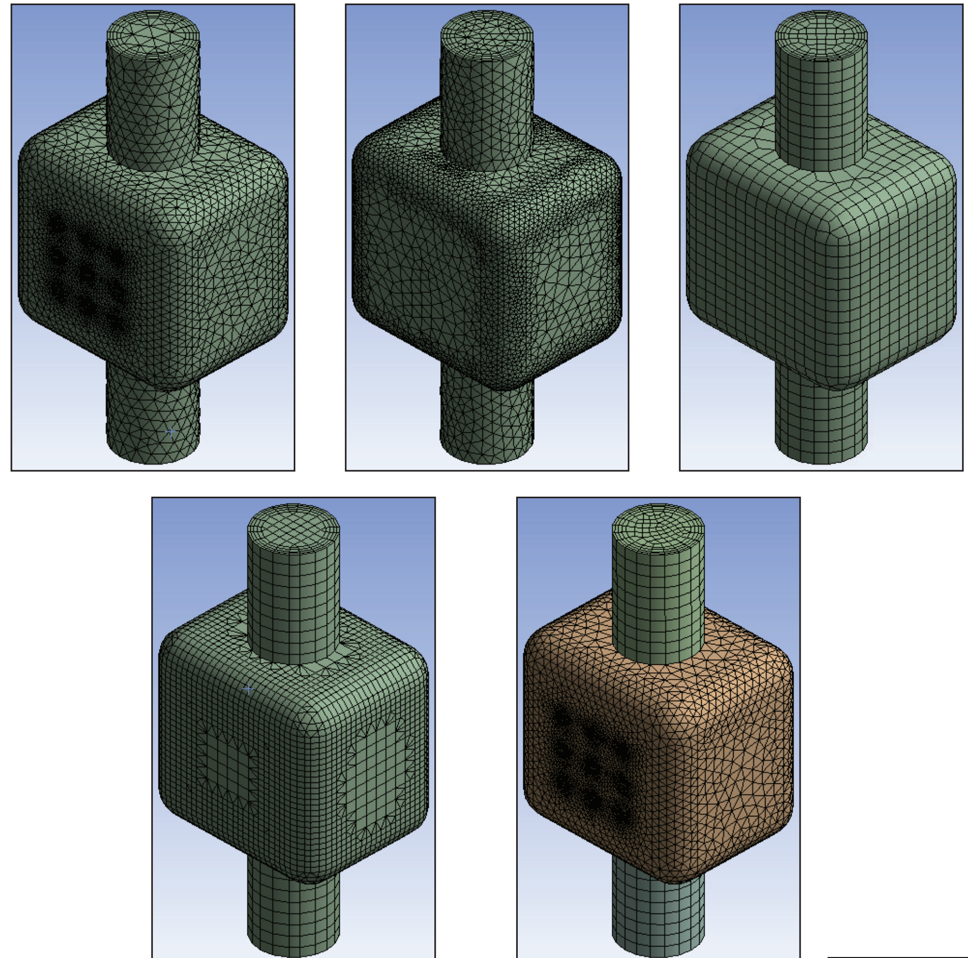
Introduction

Background

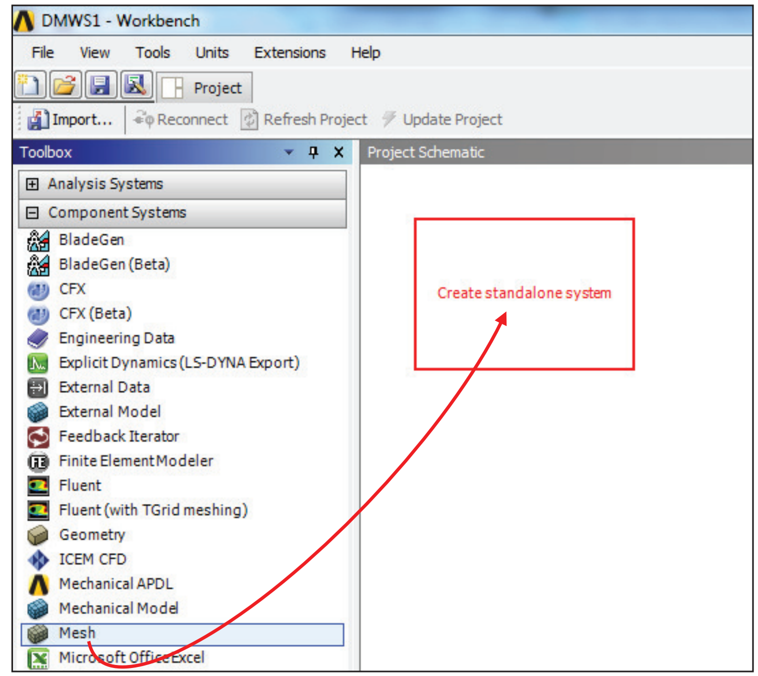
- **This workshop will introduce different meshing methods available in ANSYS Meshing**

Objectives

- **Testing and comparing the mesh generated by the different methods:**
  - **Automatic (Tet Patch Conforming)**
  - **Tet Patch Independent**
  - **Multizone**
  - **Assembly Meshing (CutCell)**
  - **Decomposition for Sweep Meshes**
  - **Automatic (Tet & Sweep)**

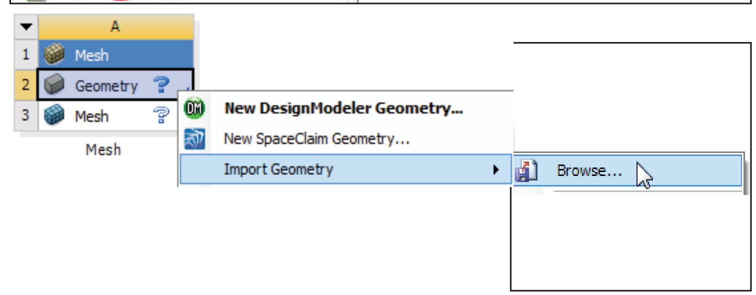


Project Startup



Create the Project

- Start Workbench
  - Start → All Programs → ANSYS 17.0 →Workbench 17.0
  - Drag and drop a Mesh Component System into the Project Schematic



- Right click on the Geometry cell (A2) and select Import Geometry → Browse

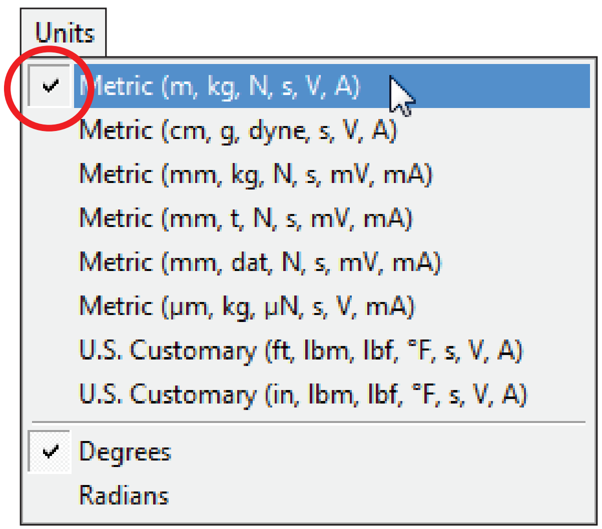
- Locate the file “component.stp” in the Meshing workshop input files (Module02) folder and select it. The geometry cell will show a check mark indicating it is up to date

- Double click the mesh cell (A3) to start Meshing

Units

Set Units

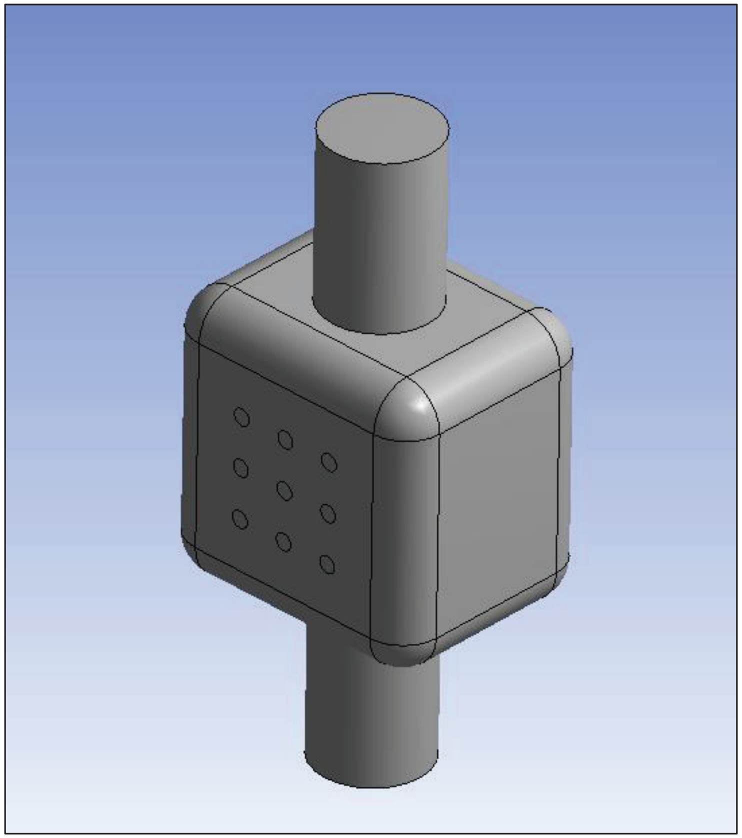
- **In Meshing window from the main menu select Units and, if it is not already set, specify Metric (m…)**



Preparation

Planning

- This CAD file contains a single body representing a component in a ventilation system
- Flow enters through the upper pipe, passes through the chamber and exits through the lower pipe. There a several small circular vents imprinted on the chamber front face
- Almost all mesh methods can be applied here, the selection of which method to use depends on speed, memory usage, requirement for defeaturing and any mesh specific solver restrictions
- We will demonstrate how different methods are applied (including inflation) and highlight important differences in the resulting meshes

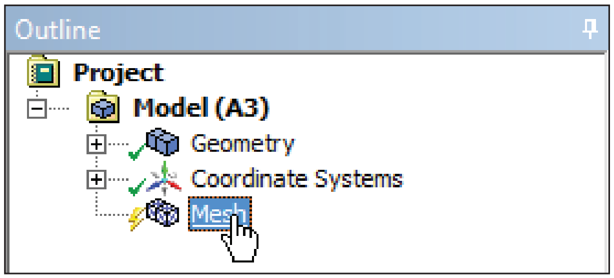


ANSYS

Global Mesh Settings (1)

Mesh

- **In the Outline, select the Mesh object to display Details of "Mesh"**



- **In Details of "Mesh", set the following under Defaults**
  - **Physics Preference: CFD**
  - **Solver Preference: FLUENT**

- **Under Sizing, set**
  - **Relevance Center: Medium**
  - **Set Size Function to Curvature**

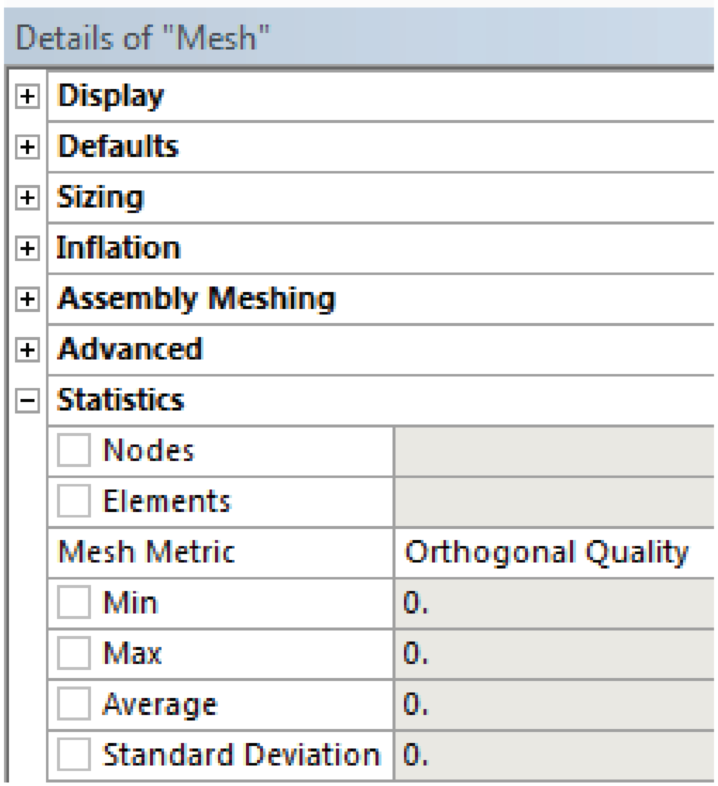
Details of "Mesh"

Display	
Display Style	Body Color
Defaults	
Physics Preference	CFD
Solver Preference	Fluent
Relevance	0
Export Format	Standard
Shape Checking	CFD
Element Midside Nodes	Dropped
Sizing	
Size Function	Curvature
Relevance Center	Medium
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Slow
Span Angle Center	Fine
Curvature Normal Angle	Default (18.0 °)
Min Size	Default (3.172e-005 m)
Max Face Size	Default (3.172e-003 m)
Max Tet Size	Default (6.3441e-003 m)
Growth Rate	Default (1.20)
Automatic Mesh Based Defeaturing	On
Defeaturing Tolerance	Default (1.586e-005 m)
Minimum Edge Length	9.4248e-003 m
Inflation	
Assembly Meshing	
Advanced	
Statistics	

Global Mesh Settings (2)

Mesh

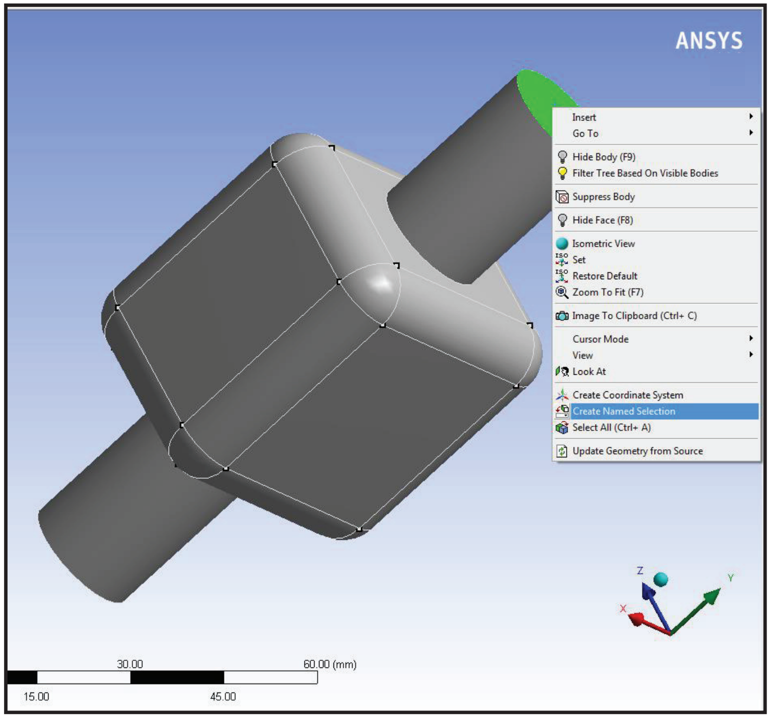
- Under Statistics, set
  - Mesh Metric: Orthogonal Quality

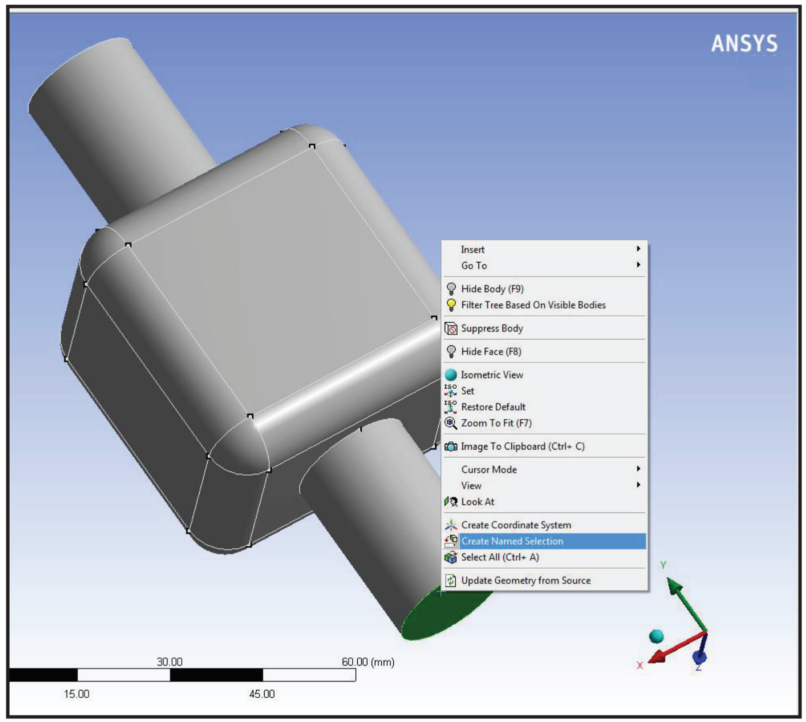


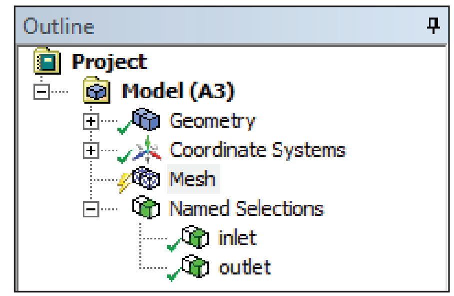
Named Selection

Define Inlet, Outlet to faces shown below by Named Selection

- Face on +Y side named as “inlet”
- Face on –Y side named as “outlet”







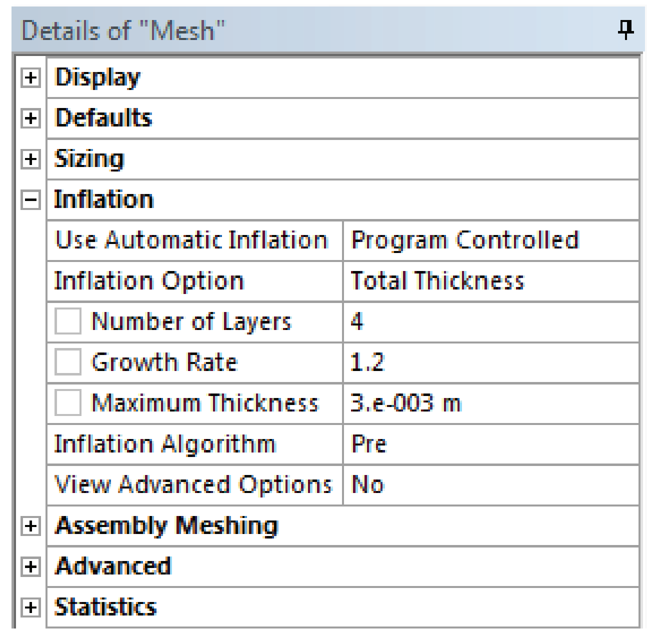
Global Inflation & Automatic Method

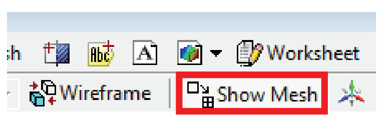
Setup Global Inflation

- In Details of “Mesh” set the following under Inflation
  - Use automatic Inflation: Program Controlled
  - Inflation option: Total Thickness
  - Number of Layers 4
  - Maximum Thickness 0.003m

- This will generate inflation layers on all faces excluding those we have created Named Selections for

- Generate the mesh



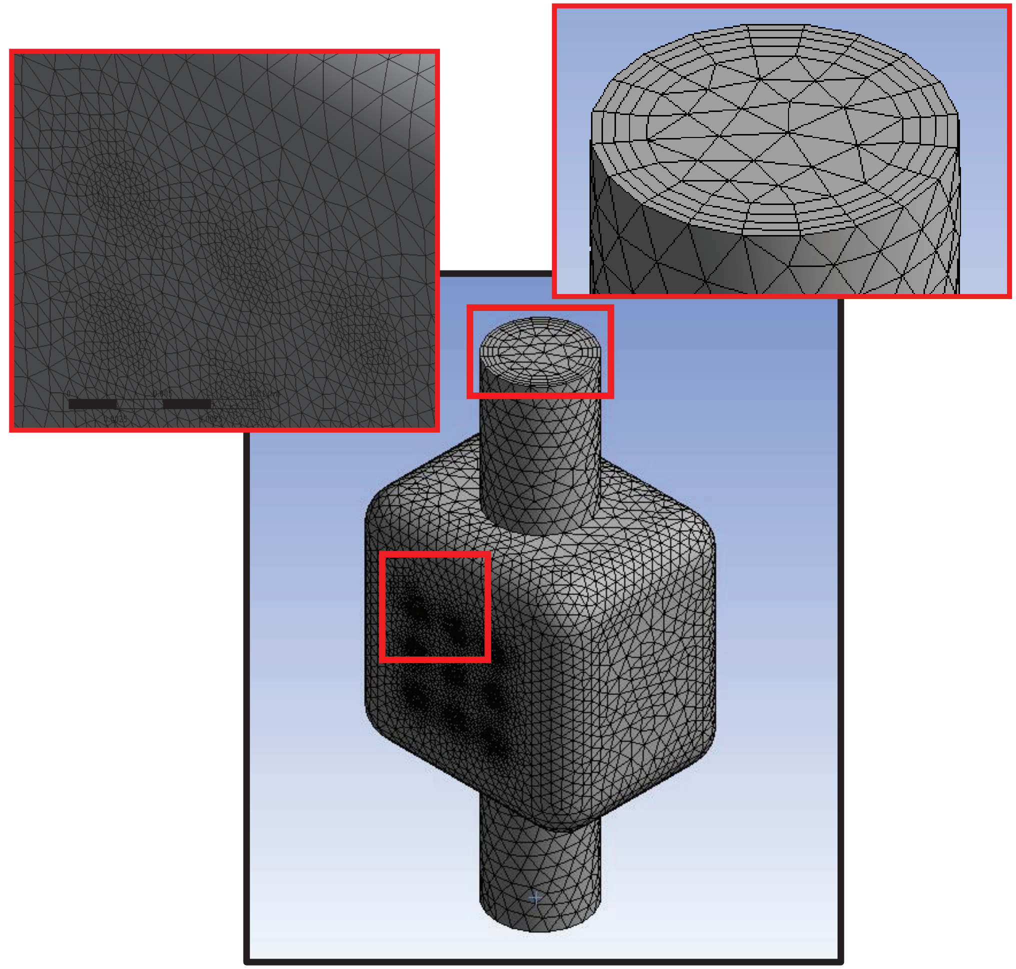


Automatic Method (1)

View the Mesh

- Right click in the Graphics Window and select Isometric View from the Context Menu

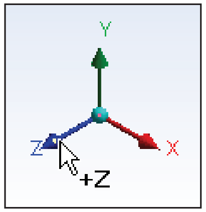
- The Automatic Method has selected the Tetrahedrons Method using the Patch Conforming Algorithm
- The mesh has conformed to all geometric details
- All surfaces except the inlet and outlet Named Selections have been inflated



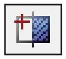
Automatic Method (2)

Create a Section Plane

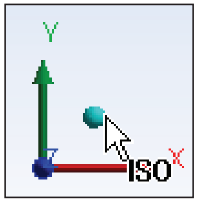
- Snap to the +Z view using the Axis Triad

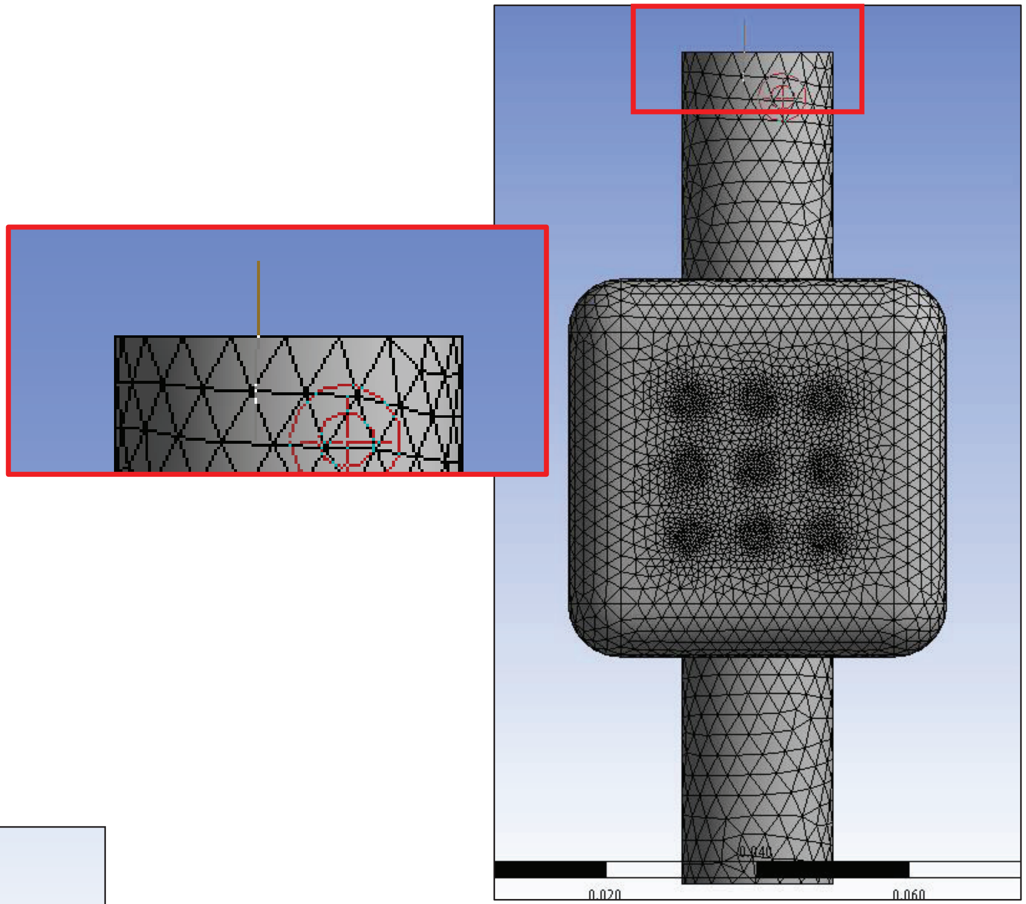


- Select the Section Plane button



- Create a Section Plane by clicking, dragging and releasing as shown vertically down through the mesh
- Restore the Isometric View using the right click Context Menu or the Axis Triad

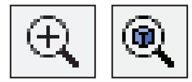




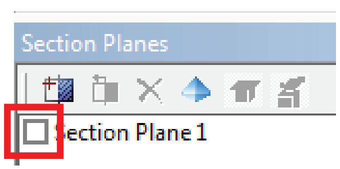
Automatic Method (3)

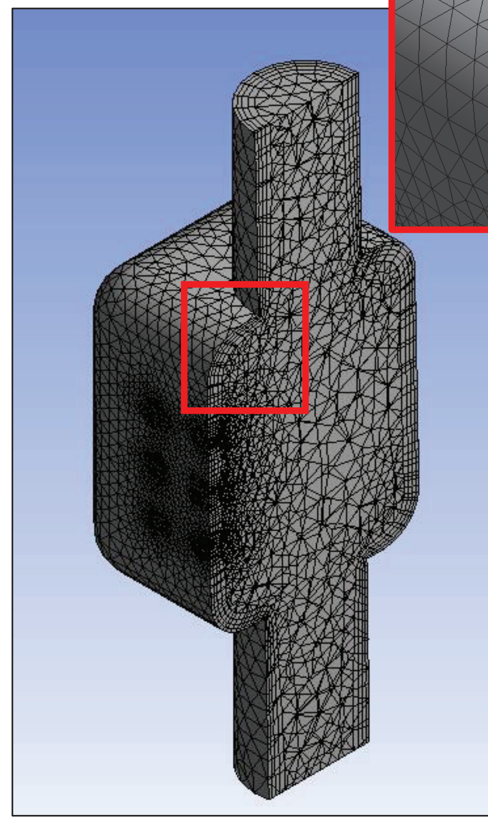
View the Mesh Interior

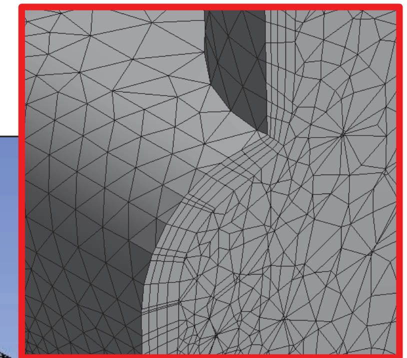
- Zoom in using the Box Zoom button to inspect the mesh. Use the Zoom to Fit button to restore the view extents



- The default Automatic Method always uses Patch Conforming Tetrahedrons and/or Sweep depending if the geometry is sweepable or not; in this non-sweepable case just Patch Conforming Tetrahedrons were used. We'll now look at how other methods can be accessed
- Switch off the Section Plane by unchecking the box in the Panel





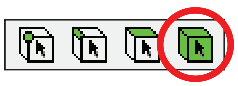


Inserting Methods Manually

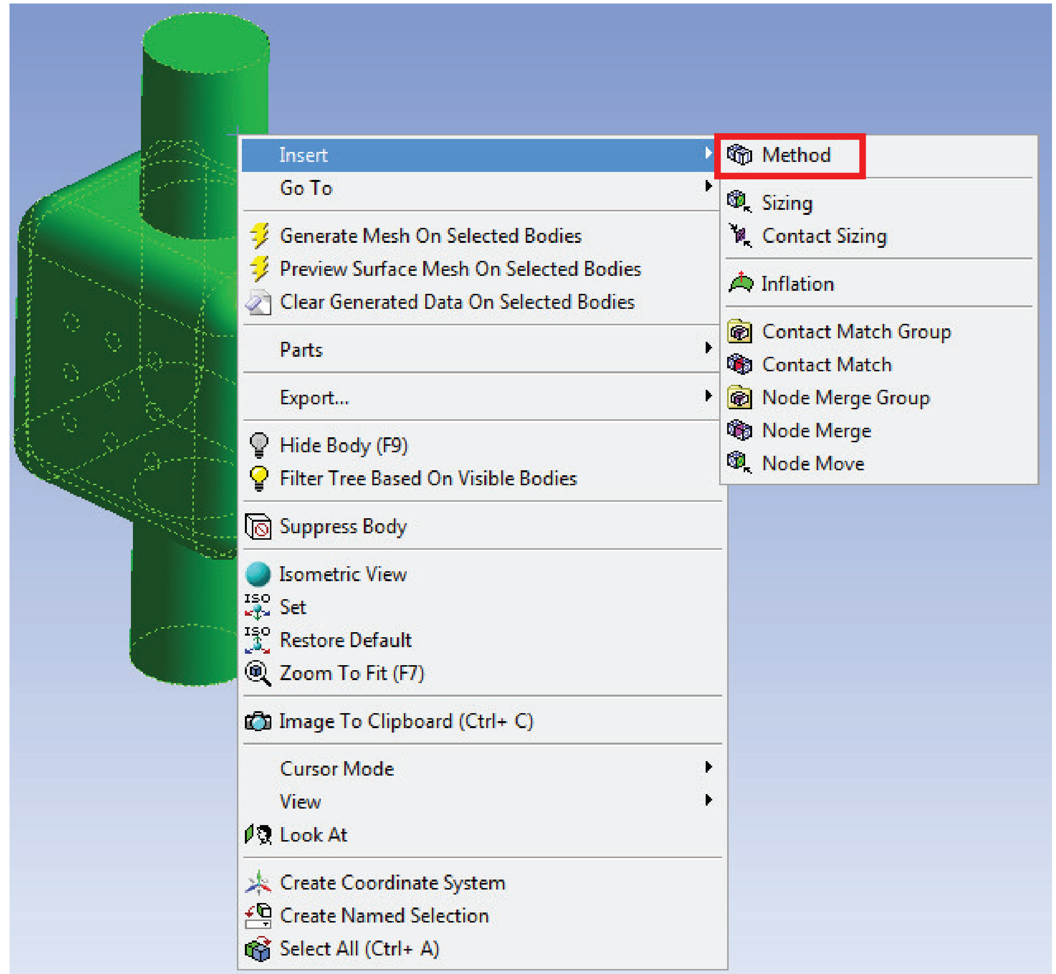
Insert a Method

- **Right click on the Mesh object in the Outline and select Clear Generated Data from the Context Menu selecting Yes when prompted to confirm**

- **Select the Body Selection Filter**



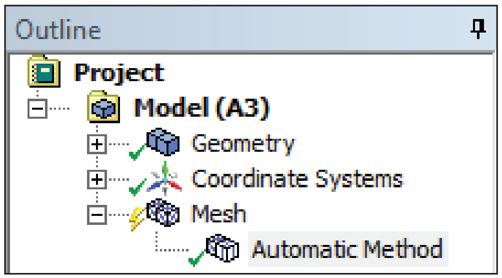
- **In the Graphics Window, select the body, right click and select Insert → Method from the Context Menu as shown**



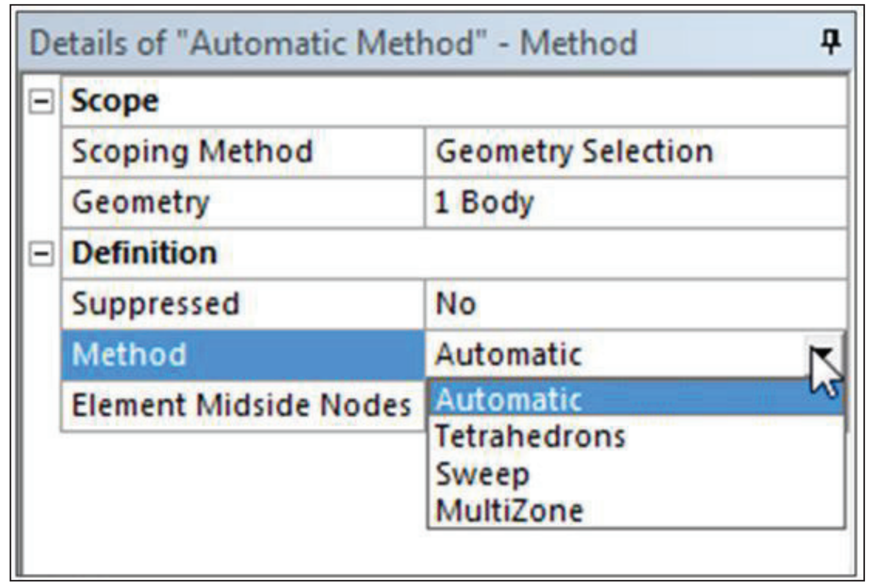
Setting Method Details

Set Method Details

- A Method Object, defaulting to Automatic, is inserted in the Outline



- In the Method Details the Method Type is accessed via a drop down box exposing the available mesh methods
- The Method set here is applied to the body initially selected



ANSYS

Tetrahedrons Patch Independent (1)

Set Method Details

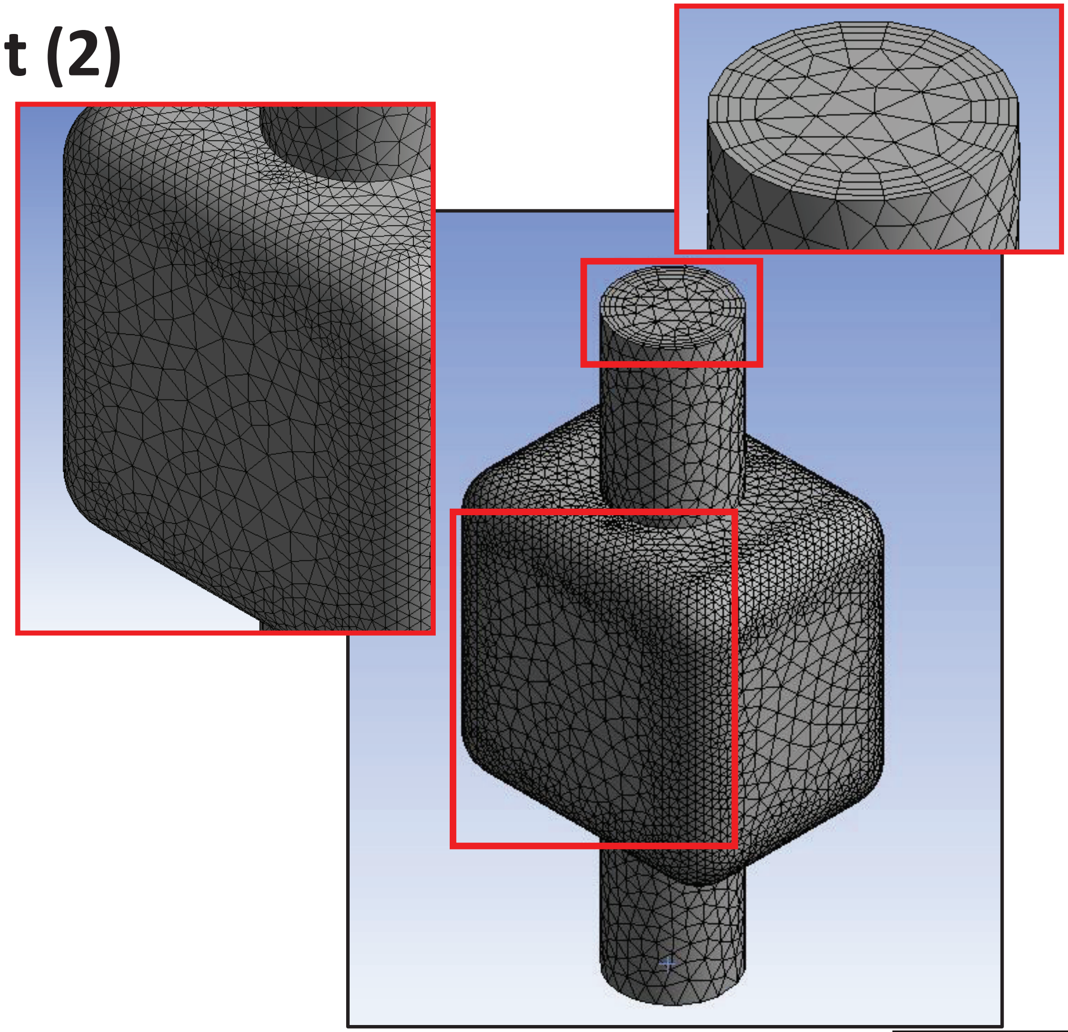
- The Automatic Method is the same as the default we have just demonstrated
- Set Method to Tetrahedrons
- Set Algorithm to Patch Independent
  - We have already seen Patch conforming used with the Automatic Method
  - When Patch Independent is selected, a number of Patch Independent specific settings appear controlling sizing and defeaturing. These can be left to the defaults
- Generate the mesh

Mesh Update

Details of "Patch Independent" - Method	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Method	Tetrahedrons
Algorithm	Patch Independent
Element Midside Nodes	Use Global Setting
Advanced	
Defined By	Max Element Size
Max Element Size	Default(6.3441e-003 m)
Feature Angle	30.0 °
Mesh Based Defeaturing	Off
Refinement	Proximity and Curvature
Min Size Limit	Default
Num Cells Across Gap	Default
Curvature Normal Angle	Default
Smooth Transition	Off
Growth Rate	Default
Minimum Edge Length	9.4248e-003 m
Write ICEM CFD Files	No

ANSYS

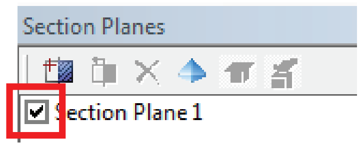
Tetrahedrons Patch Independent (2)

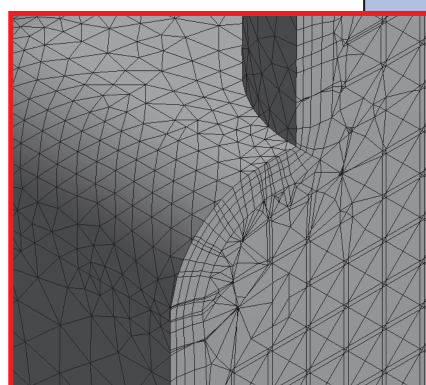


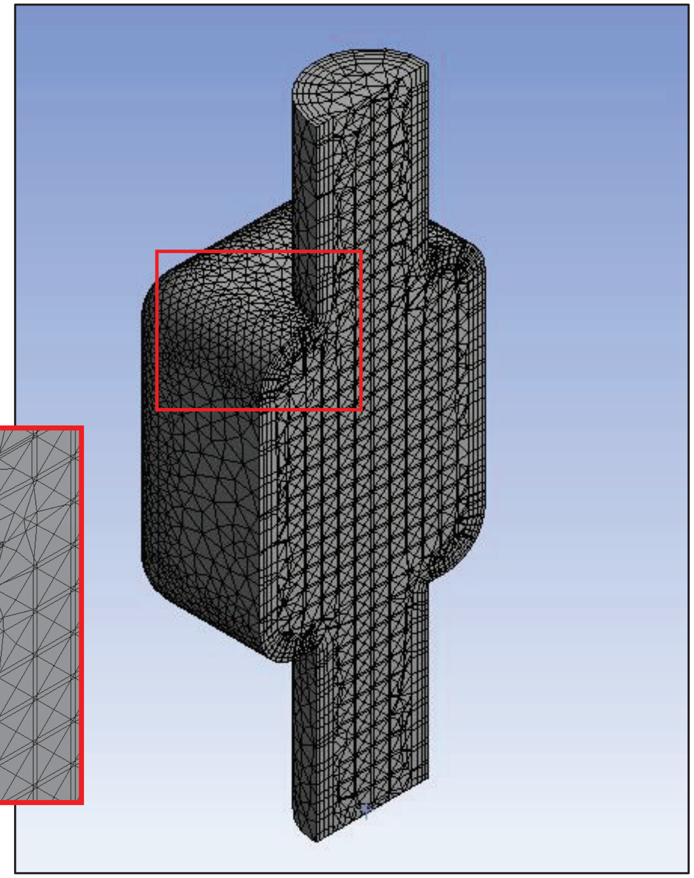
View the Mesh

- The Patch Independent Method uses a different and specific type of inflation algorithm (Post), the message box may indicate this along with a warning regarding mesh resolution. You may have also noticed it has taken more time to generate the mesh
- Note the automatic defeaturing of the small imprinted faces present on the geometry
- This behaviour can be useful for defeaturing large CAD models
- Switch on the Section Plane to view the mesh interior

Tetrahedrons Patch Independent (3)

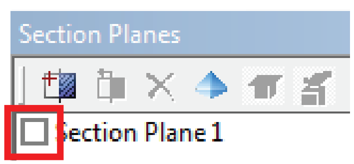






View the Mesh Interior

- Note the inflation and internal structure of the mesh
- When ready to proceed, switch off the Section Plane and clear the mesh as before. By right clicking on the Mesh Object in the Outline and Selecting Clear Generated Data from the Context Menu



MultiZone (1)

Set Method Details

- **Select the Method Object in the Outline (it assumes the name of the current set Method)**

- **In the Details, change the Method to MultiZone**
  - **The details will change to those specific to the MultiZone Method**
  - **Set Free Mesh Type to Tetra/Pyramid**
- **Generate the mesh**

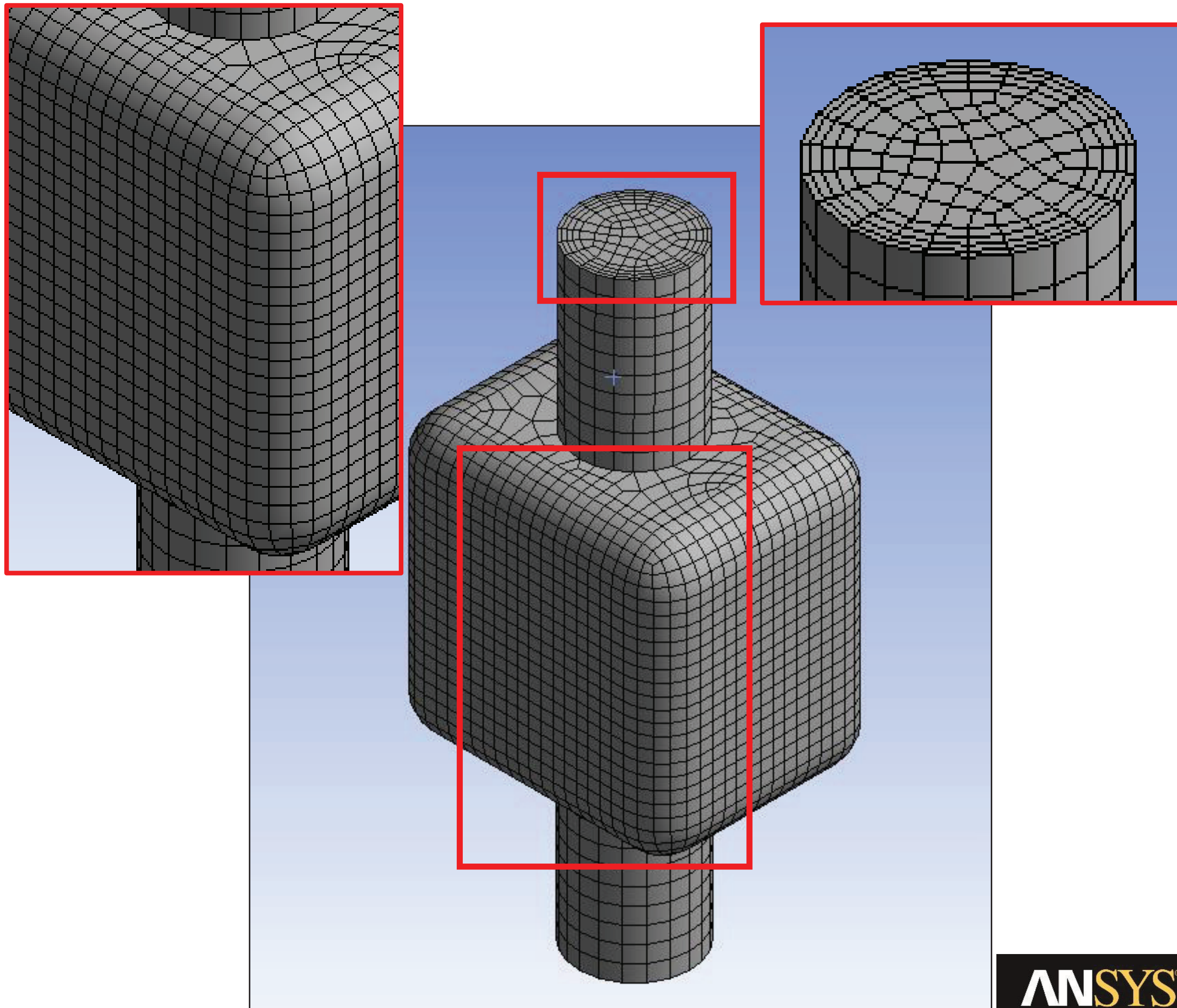
Details of "MultiZone" - Method

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Method	MultiZone
Mapped Mesh Type	Hexa
Surface Mesh Method	Program Controlled
Free Mesh Type	Tetra/Pyramid
Element Midside Nodes	Use Global Setting
Src/Trg Selection	Automatic
Source Scoping Method	Program Controlled
Source	Program Controlled
Sweep Size Behavior	Sweep Element Size
Sweep Element Size	Default
Advanced	
Preserve Boundaries	Protected
Mesh Based Defeaturing	Off
Minimum Edge Length	9.4248e-003 m
Write ICEM CFD Files	No

MultiZone (2)

View the Mesh

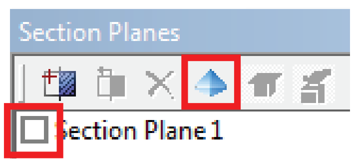
- **MultiZone has generated an all quad surface mesh**
- **Note again, the automatic defeaturing of the small imprinted faces present on the geometry**
    - **Multizone is also a method that is categorised as Patch Independent and, like the Tetrahedrons Patch Independent Method is capable of automatic defeaturing**
- **Switch on the Section Plane to view the mesh interior and turn on “Show Whole Elements”**

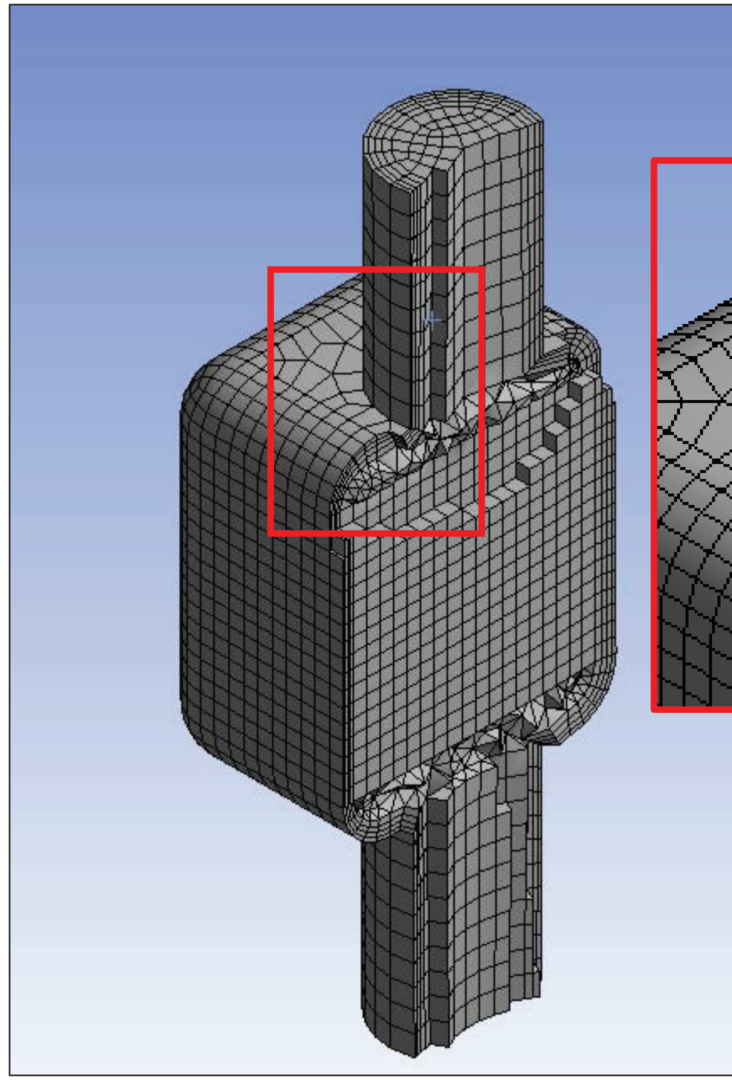


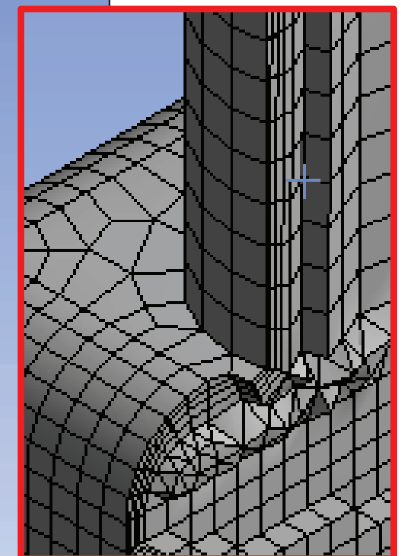
MultiZone (3)

View the Mesh Interior

- Note the inflation and internal structure of the mesh
- It internally divides model in blocks and fills blocks with hex wherever possible. In remaining areas it creates Tetra/Pyramid
- Switch OFF the Section Plane







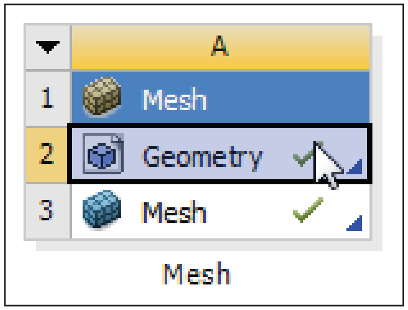
Save the Project

- From the Workbench Project Page use the file menu and save the project as “AMWS2.1_cfd.wbpj” to your working folder
  - If you have access to DesignModeler or SCDM proceed to the next slide
  - If not you can jump to slide no.29

Sweep Meshing

Sweep Requirement

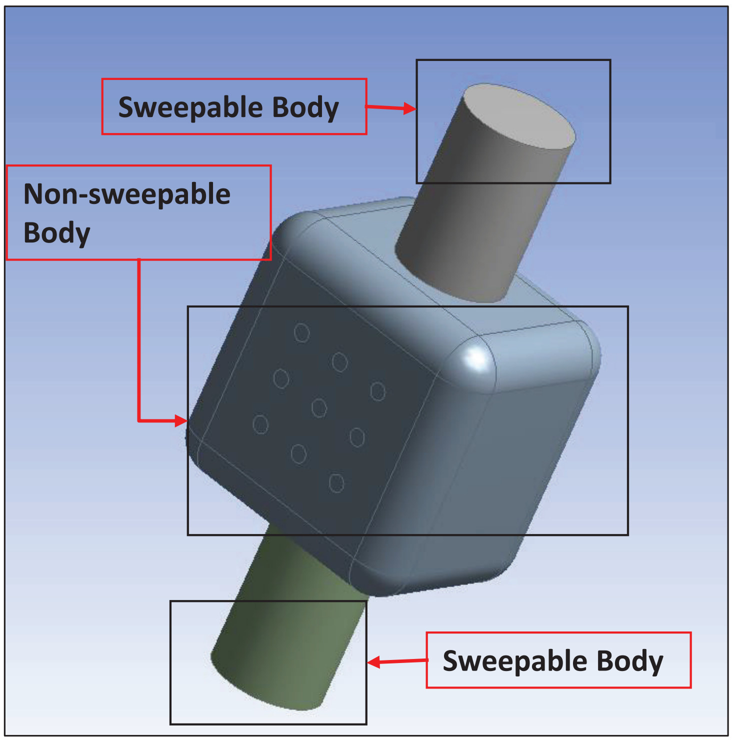
- The Sweep Method can produce high quality hex meshes
- In order to generate a sweep mesh we must have sweepable bodies
- We will decompose the geometry in DesignModeler for illustrating Sweep Method
- From the Workbench Project Page, launch DesignModeler by double clicking the Geometry Cell (A2)
  - Leave the Meshing Application Open



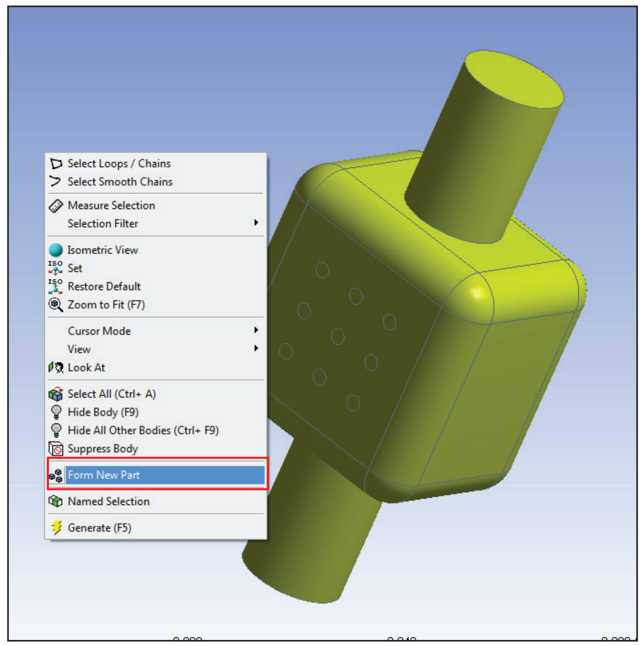
ANSYS

Geometry Decomposition

- **Slice Geometry and decompose in to sweepable and non sweepable bodies as shown in fig. you have access to DesignModeler or SCDM and you know how to use it**



Make sure that after this you make them as multi body Part

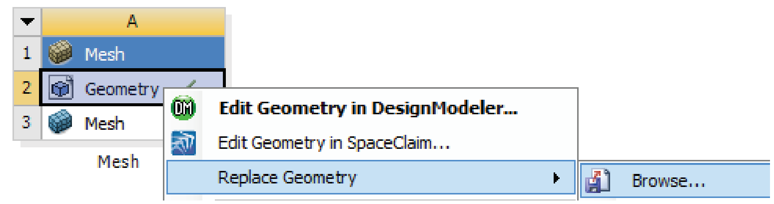


Note : **You can directly jump to meshing mode by replacing the geometry by "sweep-method.scdoc" file**

Tetra & Sweep Method (1)

If you didn't decompose geometry or you didn't have access to SpaceClaim nor to DM you can open new workbench project

- Go to workbench project page
  – Right click on Geometry/Replace Geometry/Browse
  – Browse to the sweep_method.scdoc file in the Module02 folder
  – Double click on Mesh to refresh the Mechanical Model



Method Setup

- Like Patch Conforming Tetrahedrons, we can insert the Sweep Method manually
- Since we have deactivated assembly meshing and suppressed the method we had used earlier the Meshing Application will now default to the Automatic Method

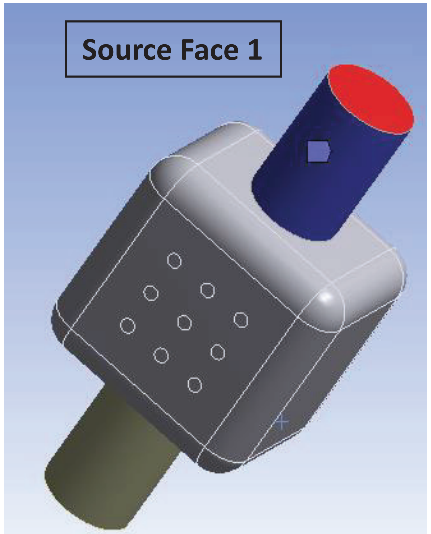
ANSYS

Tetra & Sweep Method (2)

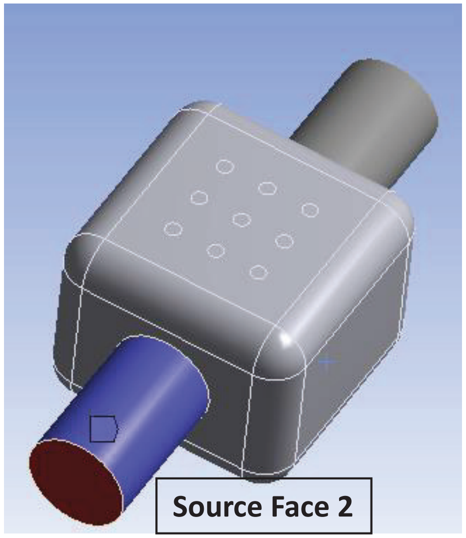
- Apply 2 Mesh methods as Sweep method
  - Select each body shown in blue for each method
  - Select corresponding face as source face in each method

Details of "Sweep Method" - Method

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Method	Sweep
Element Midside Nodes	Use Global Setting
Src/Trg Selection	Manual Source
Source	1 Face
Target	Program Controlled
Free Face Mesh Type	Quad/Tri
Type	Number of Divisions
Sweep Num Divs	Default
Sweep Bias Type	No Bias
Element Option	Solid
Constrain Boundary	No



Sweep Method

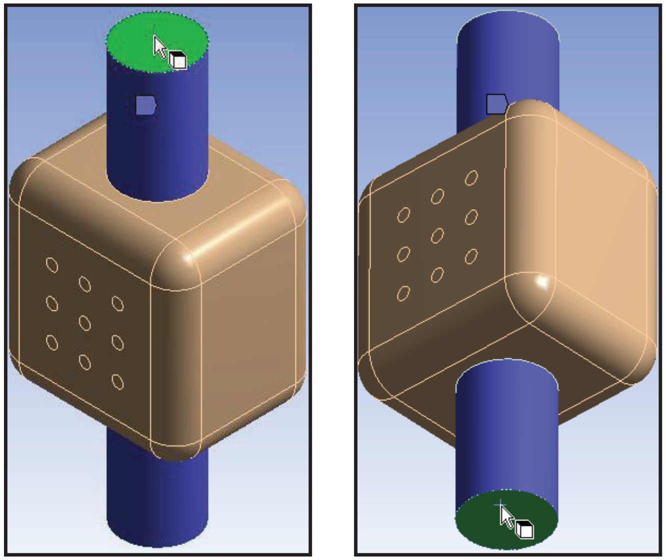


Sweep Method 2

Tetra & Sweep Method (3)

Inflation for Sweepable Bodies

- The Global Program Controlled Inflation we have used so far is not compatible with Sweep meshing so we must create a local inflation control
- Select the two faces as shown, right click and select Insert → Inflation from the Context Menu
- The Meshing Application will inflate from the boundary edges of these faces and treat them as source faces for the sweep meshes
- Inflation will set on next slide

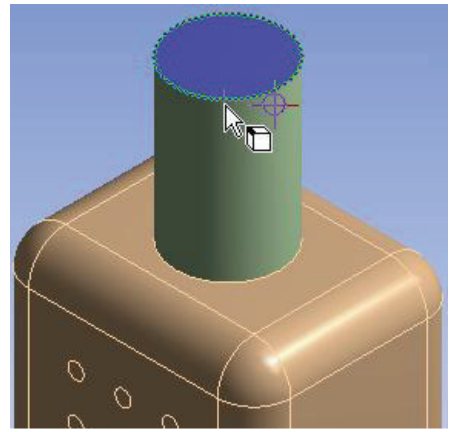


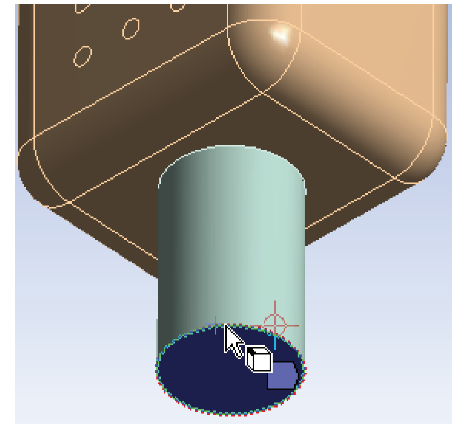
ANSYS

Tetra & Sweep Method (4)

Inflation for Sweepable Bodies (Continue…)

- **In Details of “Inflation ” activate the Boundary Selection Box, select the two edges bounding each of the source faces as shown and apply the selection**
- **Set the Following:**
  - **Inflation Option: Total Thickness**
  - **Number of Layers: 4**
  - **Maximum Thickness: 0.003m**





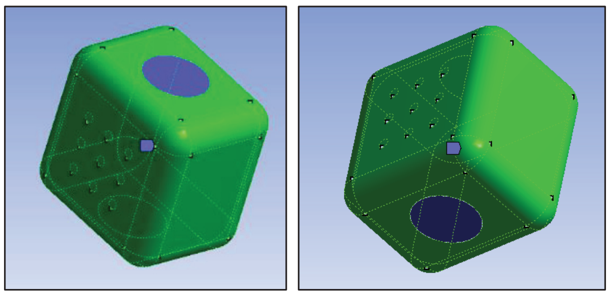
Details of "Inflation 2" - Inflation

Scope	
Scoping Method	Geometry Selection
Geometry	2 Faces
Definition	
Suppressed	No
Boundary Scoping Method	Geometry Selection
Boundary	2 Edges
Inflation Option	Total Thickness
Number of Layers	4
Growth Rate	1.2
Maximum Thickness	3.e-003 m
Inflation Algorithm	Pre

Tetra & Sweep Method (5)

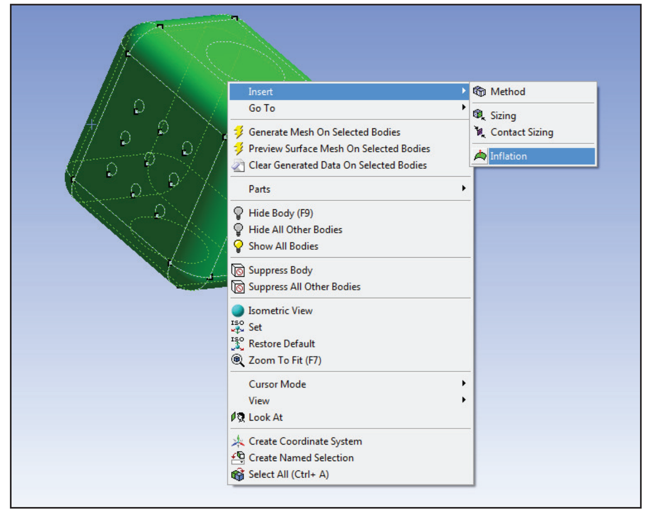
Apply Inflation to the shown body

- In details of “Inflation”
  - Select Geometry as Body as shown
  - Select 35 faces* in Boundary as shown



  - Apply inflation option to “Total thickness”
  - No. of Layers :4
  - Growth Rate : 1.2
  - Maximum Thickness : 0.003m
- Generate Mesh

Mesh Update



Details of "Inflation" - Inflation

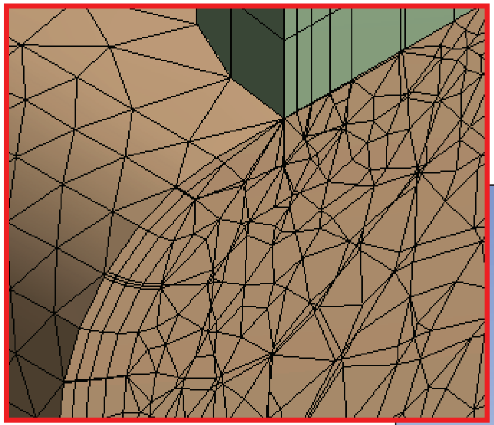
Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Boundary Scoping Method	Geometry Selection
Boundary	35 Faces
Inflation Option	Total Thickness
Number of Layers	4
Growth Rate	1.2
Maximum Thickness	3.e-003 m
Inflation Algorithm	Pre

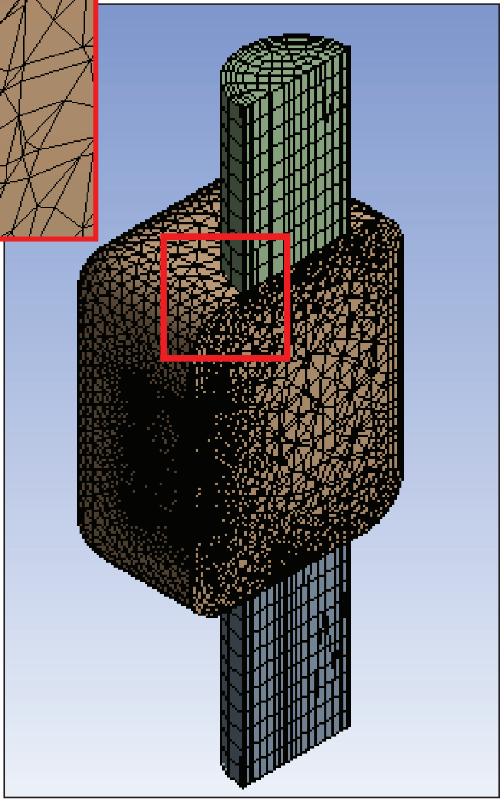
****Note: You can select all faces of visible body with Ctrl+A and deselect the two unwanted circular faces***

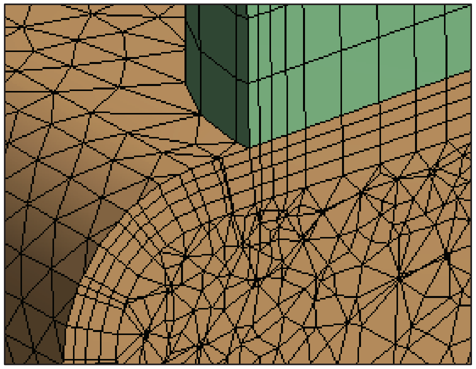
Tetra & Sweep Method (6)

View the Mesh Interior

- The global inflation used in the chamber body has ‘stair stepped’ down to the interface between the bodies (you will have seen a message indicating this)
  - You could also select top and bottom circular faces of the chamber body to avoid stair-stepping as can be seen in the bottom image







Please Note: For continuous inflation without stair-stepping for this model, we have to unite all 3 bodies and use Multizone option. For MultiZone ref. slide No. 19

  April 4, 2016

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 "AMWS2.1_cfd_B.wbpj" to your working folder