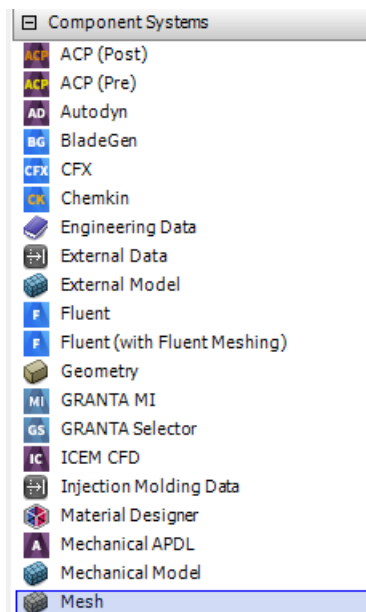


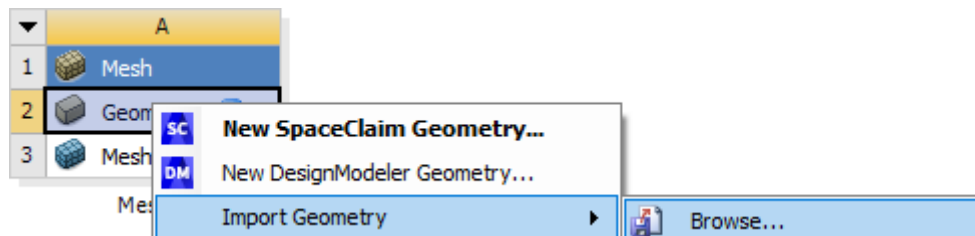
# ENGSCI 344: Tutorial 4 – Part 1

This tutorial will introduce different meshing methods in ANSYS.

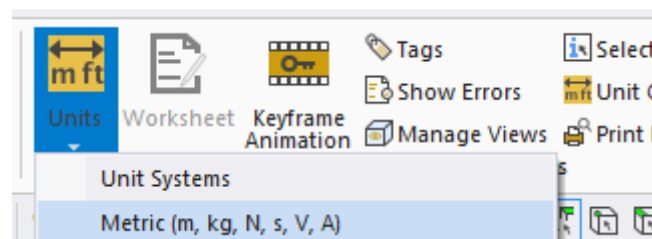
1. Download the component.stp file from Canvas.
2. Start Workbench and create a Mesh Component system (using the tree on the left).



3. Right click on the Geometry tab and import the component.stp file.

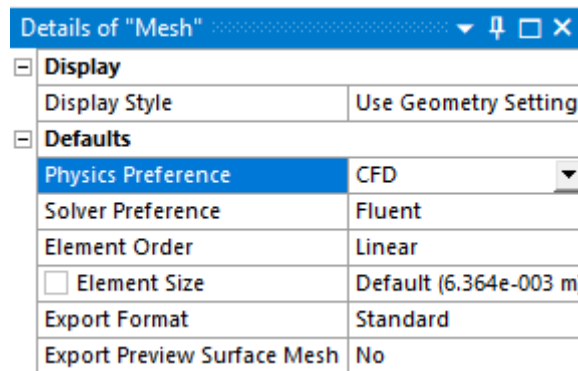


4. Double click the Mesh to start Meshing.
5. Change the units to Metric (m,kg,N,s,V,A).

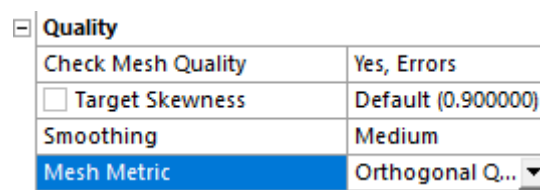


6. Visualise the geometry. Several meshing methods can be applied to this body and few of those methods are demonstrated in this tutorial.

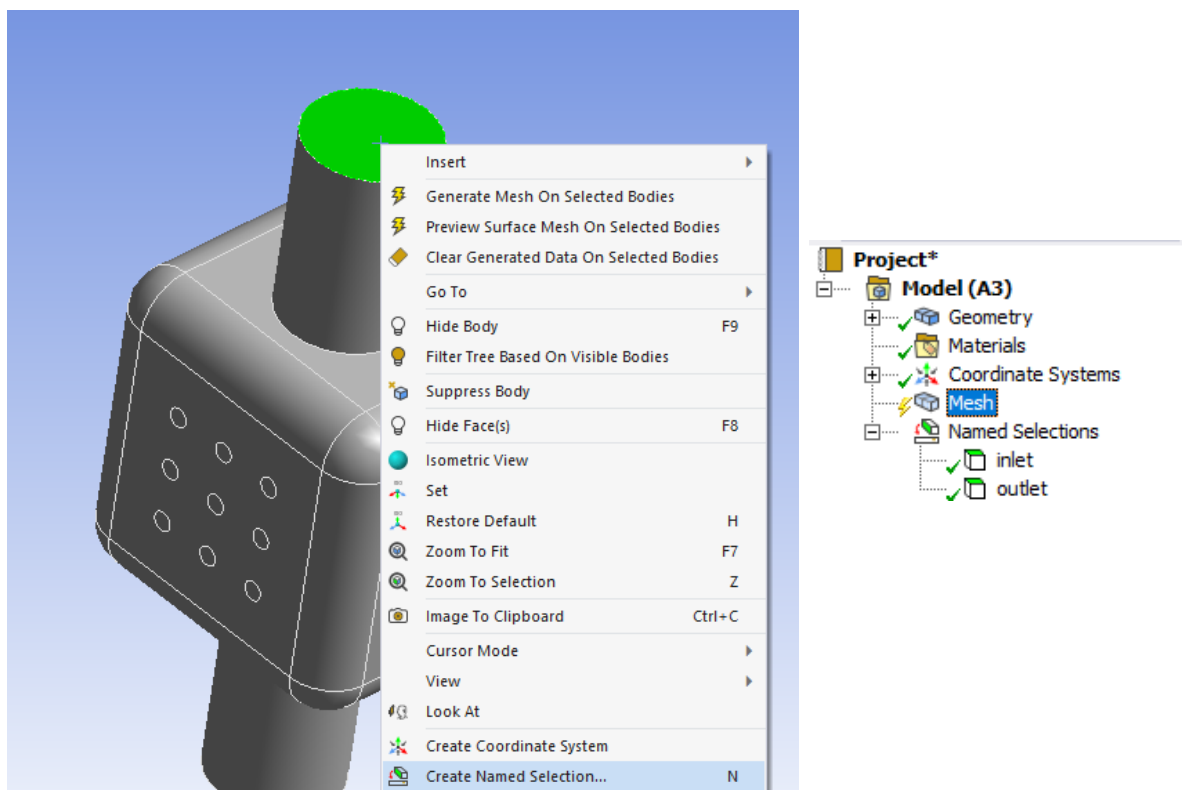
7. In the Details of 'Mesh' (Highlight the Mesh tab in the outline tree), set the Physics Preference to CFD and Solver Preference to FLUENT.



8. Expand the quality tab and set the Mesh Metric to Orthogonal Quality.



9. Define Inlet and Outlet named selections. Named selections can be used to define boundary conditions while solving CFD problems in FLUENT.
- Select the face on the +Y side and name it as inlet.
  - Similarly, select the face on the -Y side (the face that is opposite to the inlet face selected earlier) and name it as outlet.
  - Save screenshot of named selections.



10. Set global inflation.

- a. In the details tab of 'Mesh', set the inflation settings as shown in the figure below:

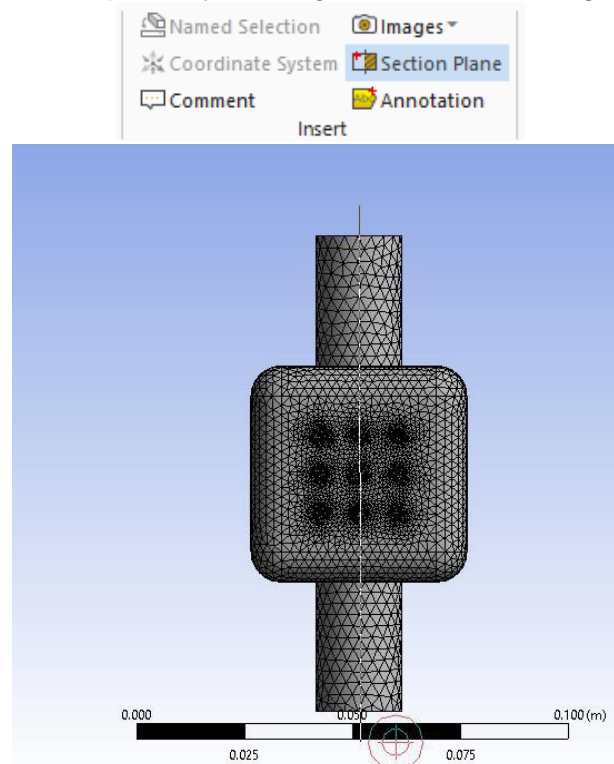
| Inflation                                  |                    |
|--|--------------------|
| Use Automatic Inflation                    | Program Control... |
| Inflation Option                           | Total Thickness    |
| <input type="checkbox"/> Number of Layers  | 4                  |
| <input type="checkbox"/> Growth Rate       | 1.2                |
| <input type="checkbox"/> Maximum Thickness | 3.e-003 m          |
| Inflation Algorithm                        | Pre                |
| View Advanced Options                      | No                 |

11. Generate the mesh and record the regions where the inflation layers are present.

What elements (cells) were used to mesh the body?

12. Create a section plane and slice the body vertically and visualise the mesh.

- a. Snap to the +Z view by using the Coordinate axis on the bottom right of the window.
- b. Create the section plane by drawing a vertical line through the body.



13. View the Mesh Interior using the Show Whole Elements option.

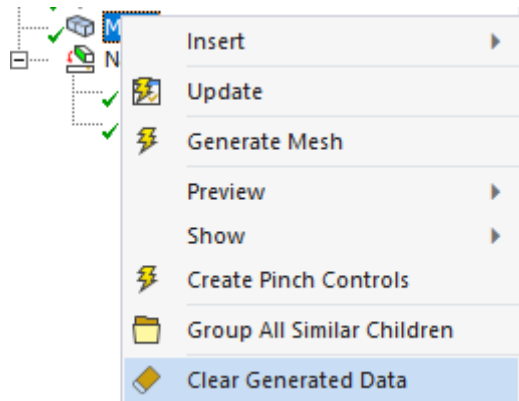


14. Switch off the section plane by unchecking the box in the Section Planes panel (bottom left). Save screenshot of the interior mesh.



15. The object is meshed using an automatic method. Now, we will mesh the same body using different manual methods.

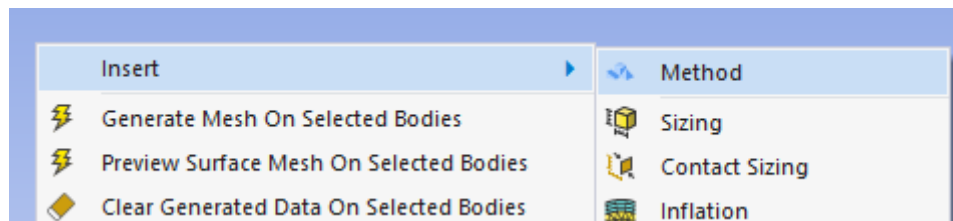
16. Clear the generated mesh data.



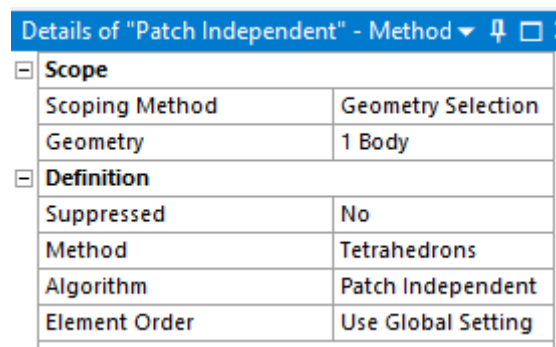
17. Use the Body Selection Filter and select the entire body.



18. Right click on the Graphics Window and insert a new Method.



19. Change the method to Tetrahedrons from Automatic and change the Algorithm to Patch Independent using the details tab. The rest of the settings can be left to the default values.



20. Generate the mesh. What do the messages mean?

21. View the mesh. How is it different from the mesh generated using the Automatic method? Save a screenshot of the mesh.
22. Change the Method to MultiZone and set the Free Mesh Type to Tetra/Pyramid.

|                       |                    |
|-----------------------|--------------------|
| [-] <b>Scope</b>      |                    |
| Scoping Method        | Geometry Selection |
| Geometry              | 1 Body             |
| [-] <b>Definition</b> |                    |
| Suppressed            | No                 |
| Method                | MultiZone          |
| Mapped Mesh Type      | Hexa               |
| Surface Mesh Method   | Program Controlled |
| Free Mesh Type        | Tetra/Pyramid      |

23. Generate the mesh. If there is an info message, what does it mean?
24. Slice the mesh using the Section Plane option and compare it with previous meshes. Capture a screenshot of the MultiZone mesh.
25. Close the meshing module and return to Workbench.
26. You can read more about meshing from ANSYS help (click F1 and type 'Specialised Meshing' in the search window).