Workshop 4.2: Mesh Control

Introduction to ANSYS Mechanical
Use the various ANSYS Mechanical mesh controls to enhance the mesh for the model below.

• Our goal is to use meshing controls in order to deal with some geometry specification defects.
1. From the “Units” menu verify:
   • Project units are set to “Metric (kg, mm, s, °C, mA, N, mV). If not available in the menu, click on “Unit Systems...” and set them for the active project.
   • “Display Values in Project Units” is checked (on).

2. From the Toolbox double click “Static Structural” to create a new system.

3. RMB the geometry cell and “Import Geometry” and browse to “assembly_solid.stp”.
4. Double click the “Model” cell to open the Mechanical application.

5. Set/verify the working unit system:
   - “Units > Metric (mm, kg, N, s, mV, mA)”.

Mesh control

6. Select Part 2 and Part 3 in the Geometry details. RMB and suppress body. (we’ll focus on Part 1 in a 1st step)

7. RMB on Mesh and click on Generate Mesh to generate a default mesh
8. From the mesh details -> Statistics, select Mesh Metric and check the element quality.
9. From the mesh sizing details, change the element size to 4 mm and regenerate mesh. Re-Check the mesh metric.
10. Insert a Face Meshing at the cylindrical surface shown below and set the internal Number of divisions to 2.

11. Repeat the same control to the 2 other cylindrical surfaces shown here.
12. Insert a Face sizing of 3 mm as element size at the surface shown below.
13. a. Select one of the 2 surfaces in the 8 holes (half of the hole) and RMB create named selection.

13. b. In the named selection details, change the name to ‘holes’ and choose apply geometry items of same size.

- After re-generating the mesh, you can see that the number of distorted elements has been reduced by checking the mesh metric.
Geometry correction technics

- We will now focus on dealing with some geometric specifications like the ones shown here:
15. Insert a pinch control and choose as master Geometry the 2 edges in blue. As Slave Geometries, choose the two edges in red. Set the tolerance to 2 mm.
• After generating the mesh check the result and compare it to the original mesh on the corresponding area.
16. Clear generated mesh and Turn the Show vertices on:

17. Insert another pinch control and choose as master Geometry the vertex in blue. As Slave Geometries, choose the three following vertices. Set the tolerance to 2 mm and Mesh.
18. Repeat the pinch control at the following points: Change the tolerance value if needed.

- Compare the mesh at the corresponding area
Virtual topologies

- The Pinch controls done here can sometimes be replaced by virtual topologies.
- We will create a virtual topology in the other side of the 2 created pinch controls:
19. From the model control RMB and insert Virtual Topology.

20. a. Select the 3 edges in the corresponding area.
20. b. Click on Merge Cells.

Generate the mesh and Compare.
Virtual topologies vs Pinch

- Pinch and VT are complementary tools. In some cases if one could not be applied, try the other one:

Select the 3 edges and try to merge the cells.
• In these cases, the pinch control should be used:

21. Insert a Pinch Control and select the 2 following points as master geometry:

And the other two points as slave geometry with a tolerance of 1 mm.
...Virtual topologies vs Pinch

• Check the difference between the generated meshes before and after this operation:
Go Further!

- In this exercise we focused on the sizing and the cleaning operations.
- The user may test some methods in order to have hex elements.

Test the Hex Dominant method on the Part 1 body.
• Unsuppress Part 2 and Part 3 bodies.

Test the Multizone method on both bodies.
Insert a body sizing of 5 mm as element size to both bodies
Test the inflation on the Part 2 body. With a body sizing of 3mm.