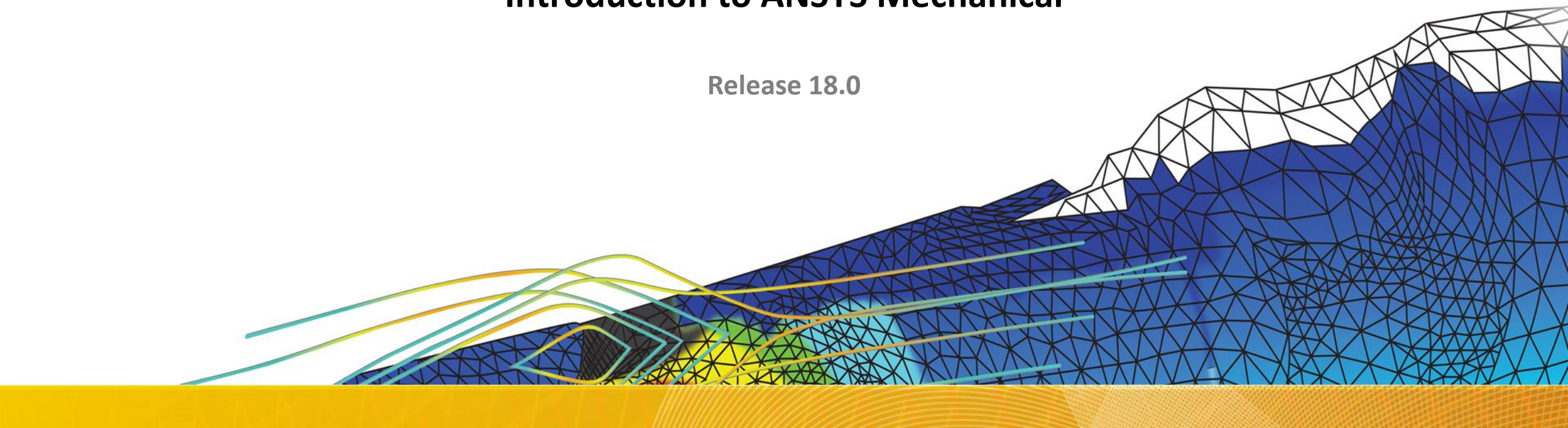
Workshop 05.2: Mesh Control

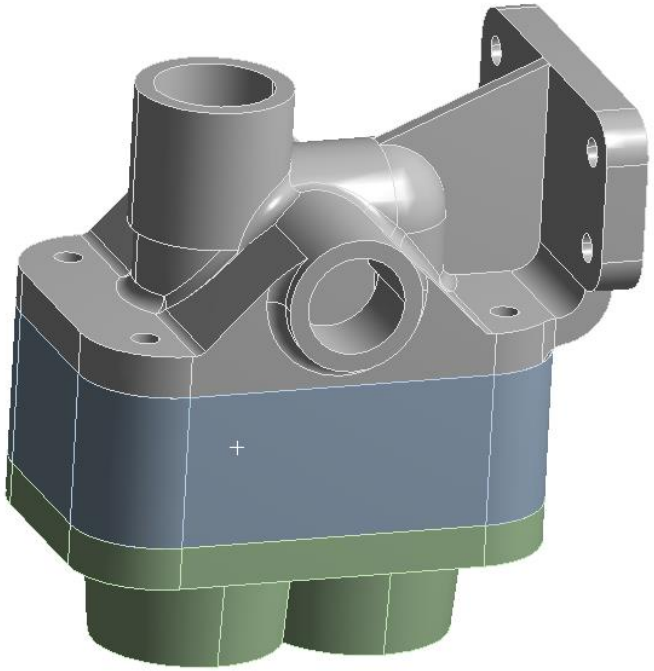
Introduction to ANSYS Mechanical

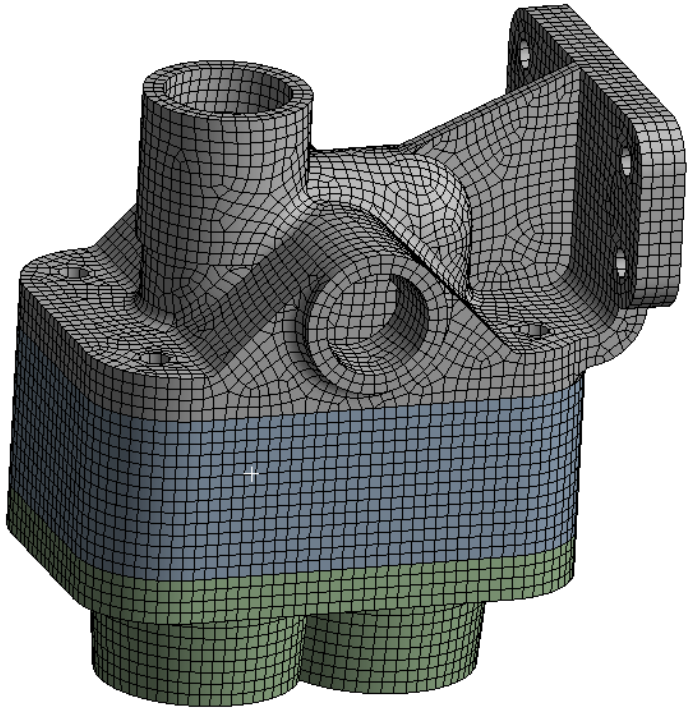
Release 18.0



Goals

Use various ANSYS Mechanical mesh controls to enhance the mesh for the model below. Our goal is to use mesh controls to deal with some defects in the imported geometry.

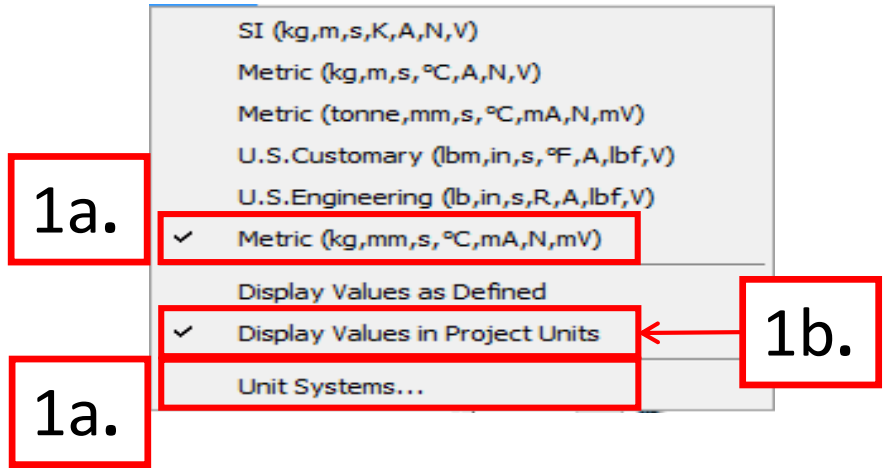


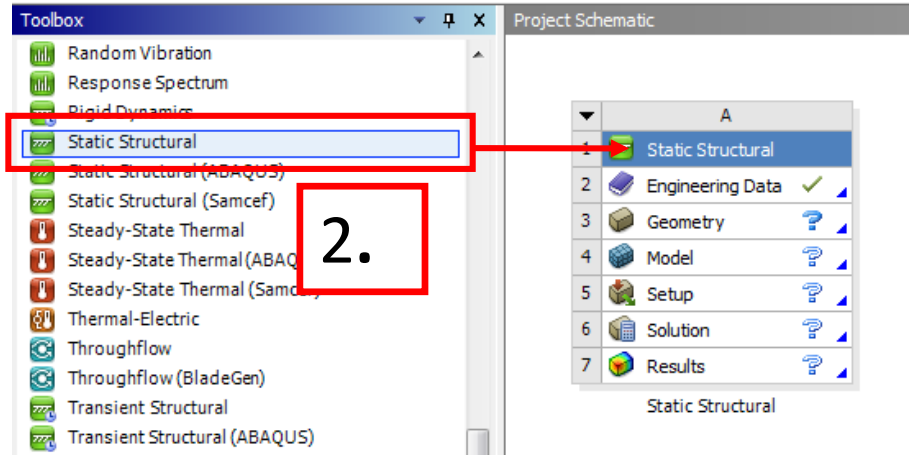


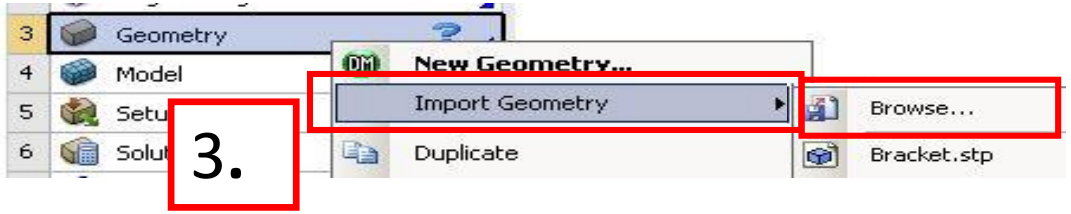
© 2017 ANSYS, Inc.

Project Schematic

1. From the Units menu, verify that:
   a. Project units are set to “Metric (kg, mm, s, °C, mA, N, mV). If this system is not displayed in the Units menu, click “Unit Systems… ” and unsuppress it.
   b. “Display Values in Project Units” is checked.
2. From the Toolbox, double-click “Static Structural” to create a new analysis system.
3. RMB on the Geometry cell, then “Import Geometry” and browse to “assembly_solid.stp.”



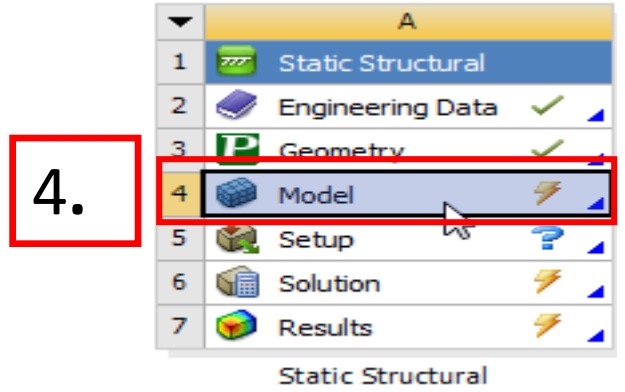




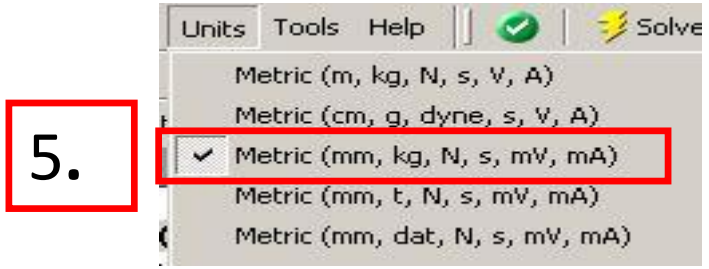
© 2017 ANSYS, Inc.

Project Schematic

4. **Double click the “Model” cell to open the Mechanical application.**



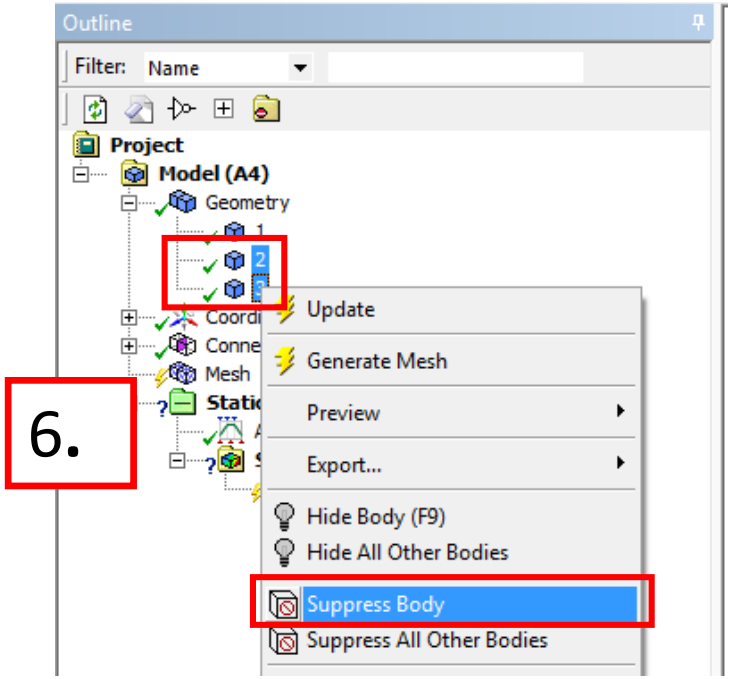
5. **Set the working unit system to “Metric (mm, kg, N, s, mV, mA).”**



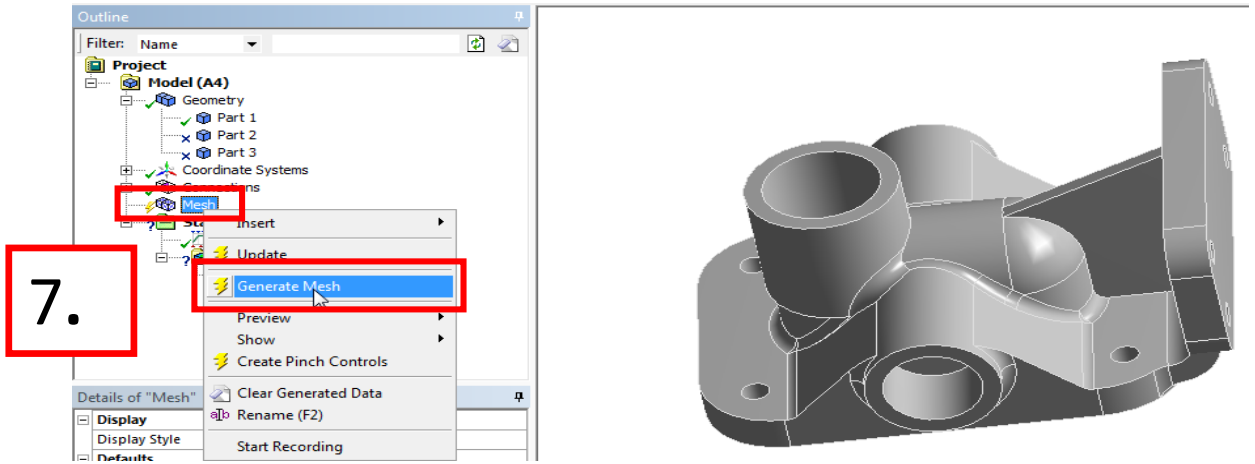
© 2017 ANSYS, Inc.

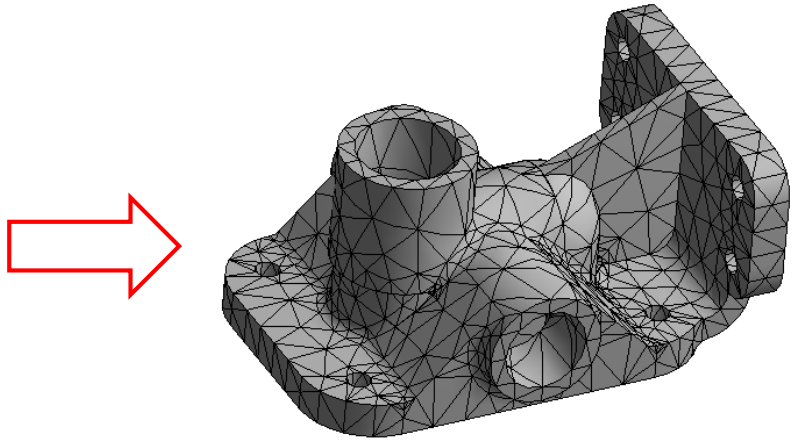
Mesh Control

6. Expand the Geometry branch, select part “2” and part “3,” and RMB > Suppress Body. This will allow us to isolate part “1” for attention in this workshop.



7. Click on Mesh, then RMB > Generate Mesh to generate a default mesh.

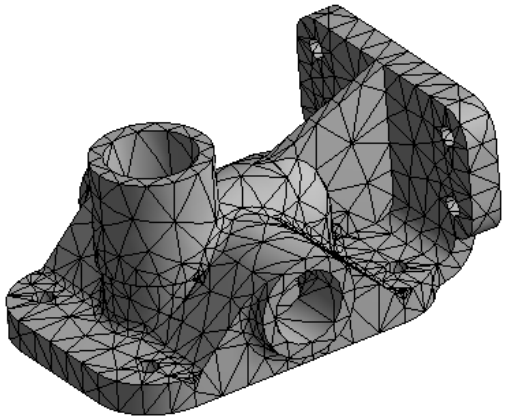


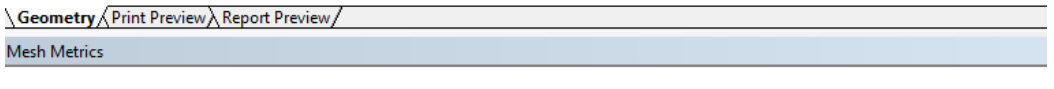


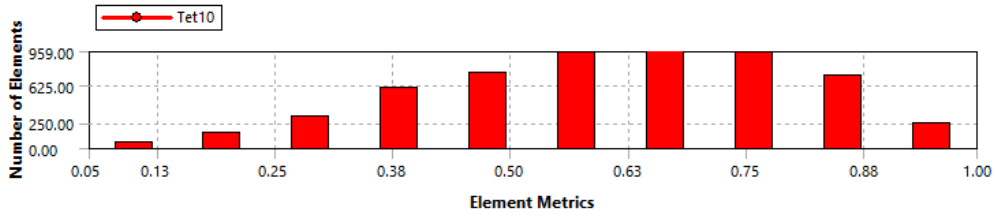
© 2017 ANSYS, Inc.

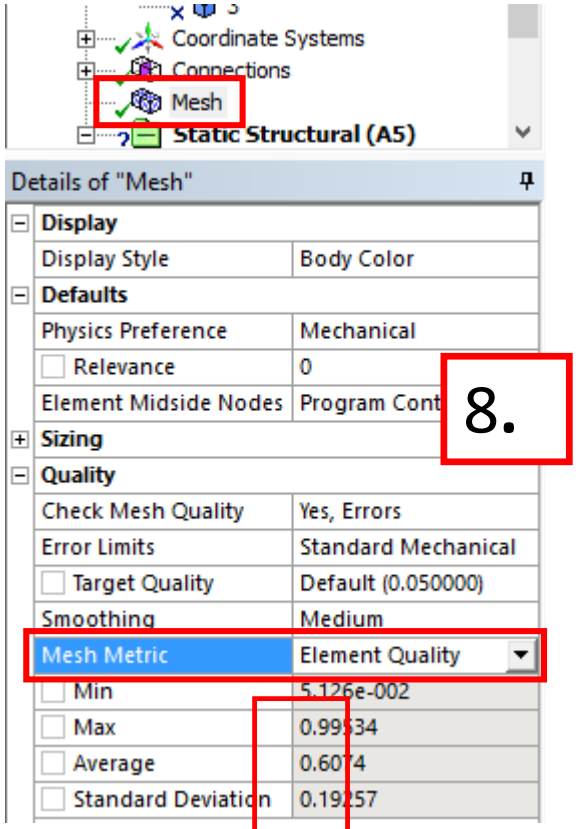
Mesh Control

8. From the Mesh Details view, expand Quality, then set Mesh Metric = Element Quality and review the element quality data.





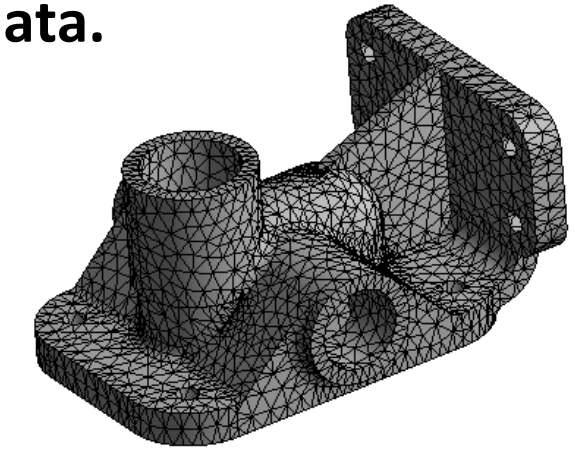


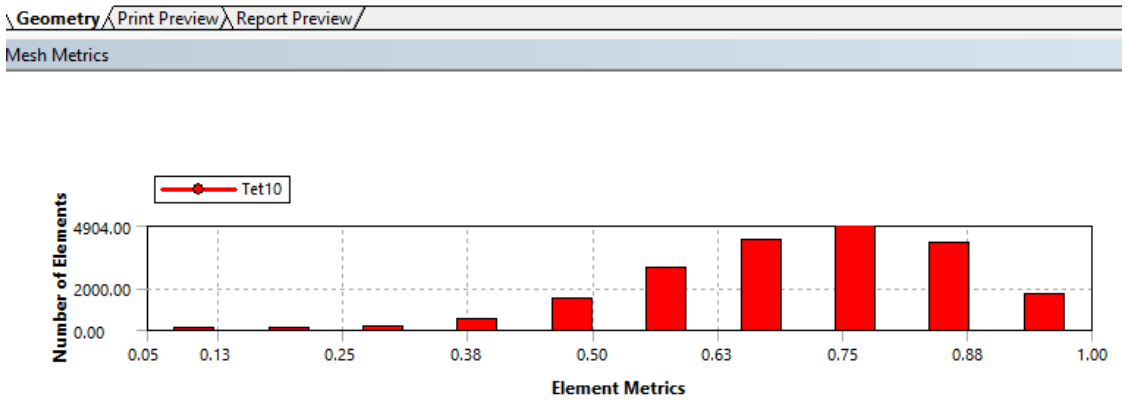


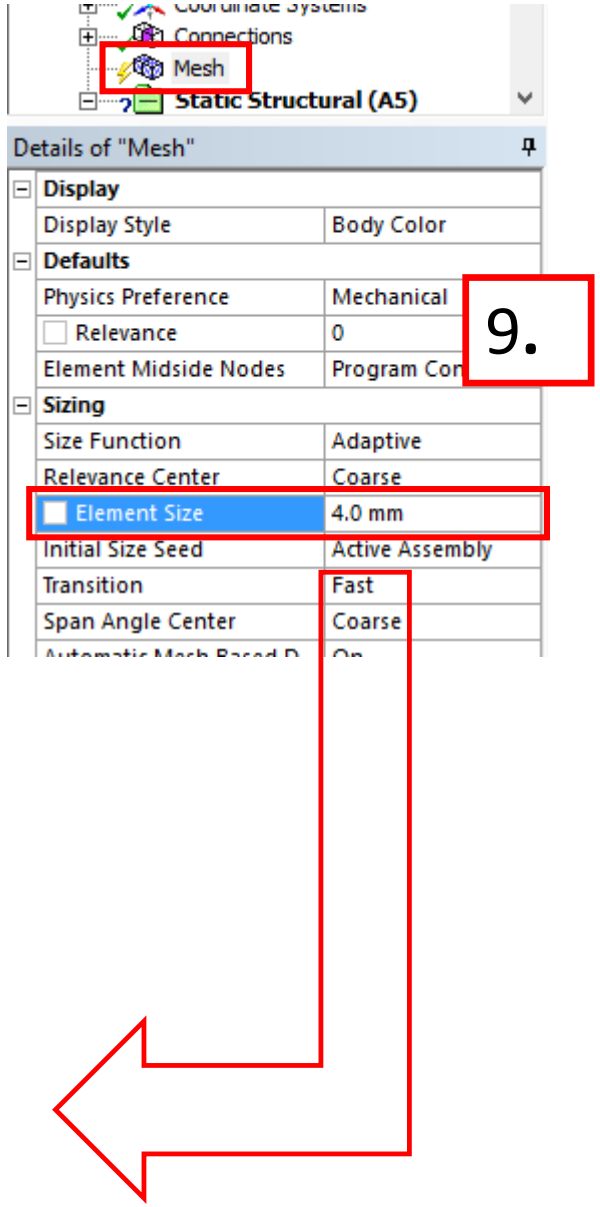
© 2017 ANSYS, Inc.

Mesh Control

9. From the Mesh—Sizing—Element Size Detail, change the element size to 4 mm, regenerate the mesh, and re-check the mesh metric data.





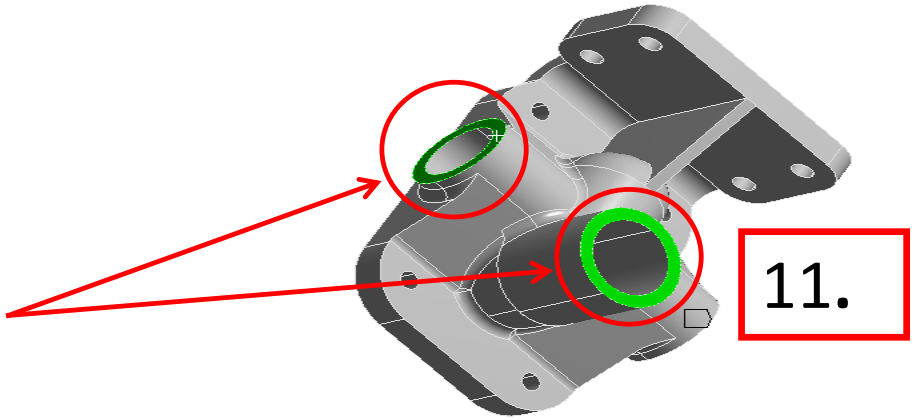


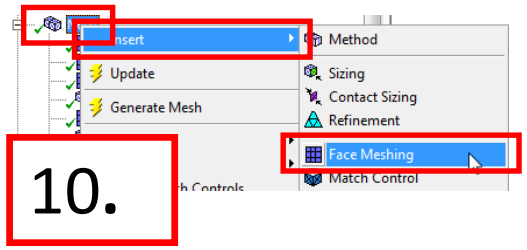
© 2017 ANSYS, Inc.

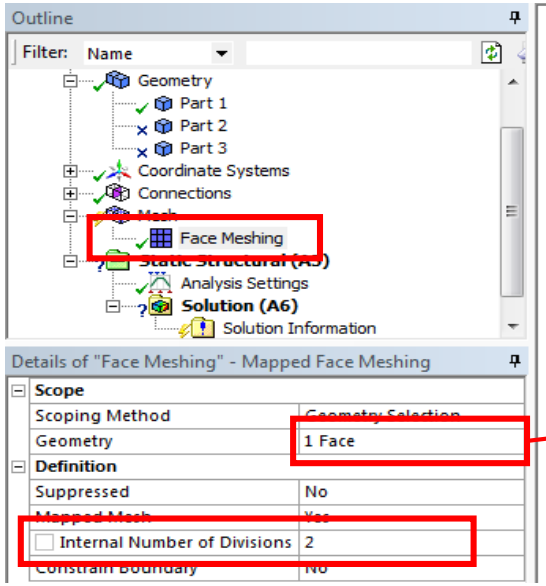
Mesh Control

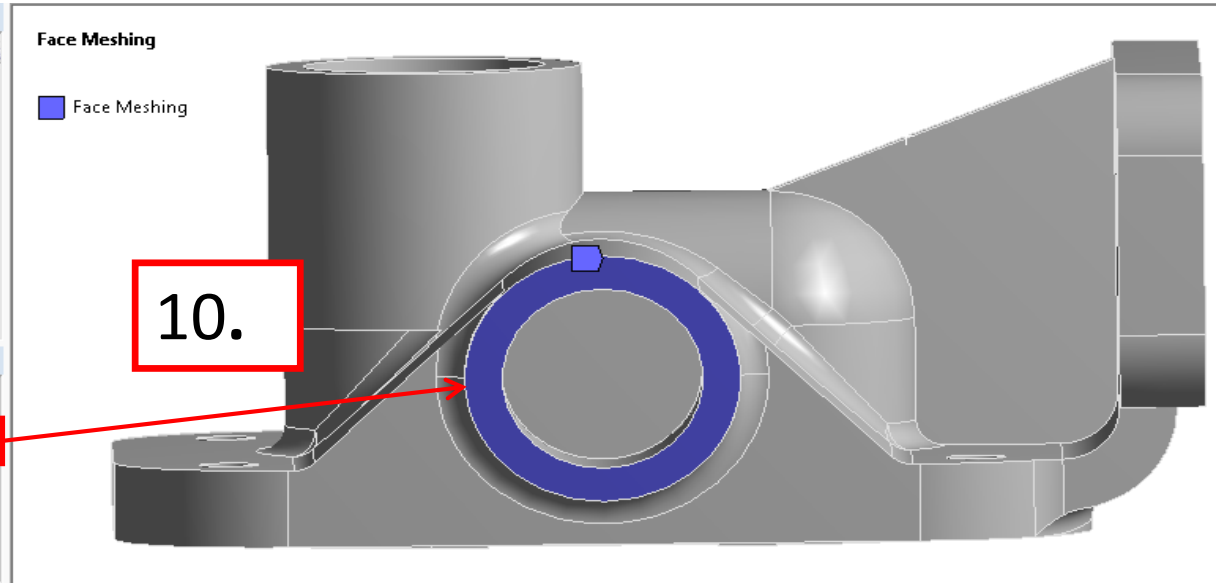
10. Insert a Face Meshing branch scoped to the annular surface shown below and set the Internal Number of Divisions to 2.

11. Repeat Step 10 for the 2 annular surfaces shown.





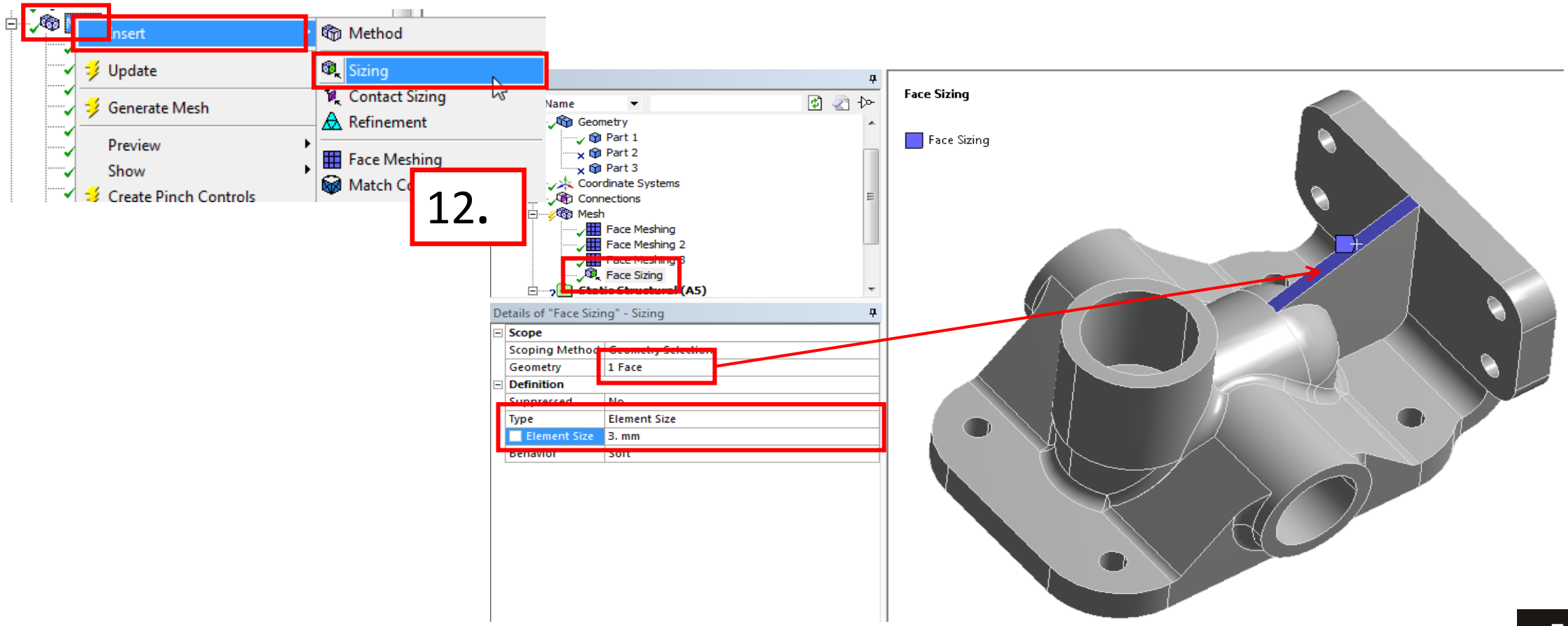




© 2017 ANSYS, Inc.

Mesh Control

12. Insert a Sizing control scoped to the rectangular surface shown below and set the Element Size to 3 mm.



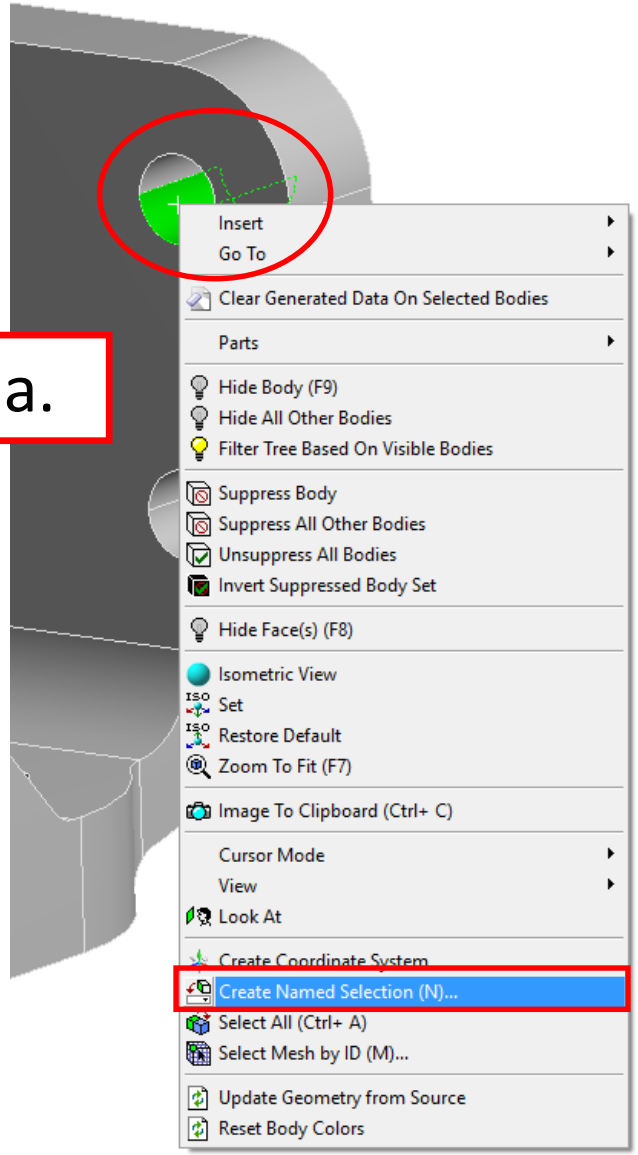
© 2017 ANSYS, Inc.

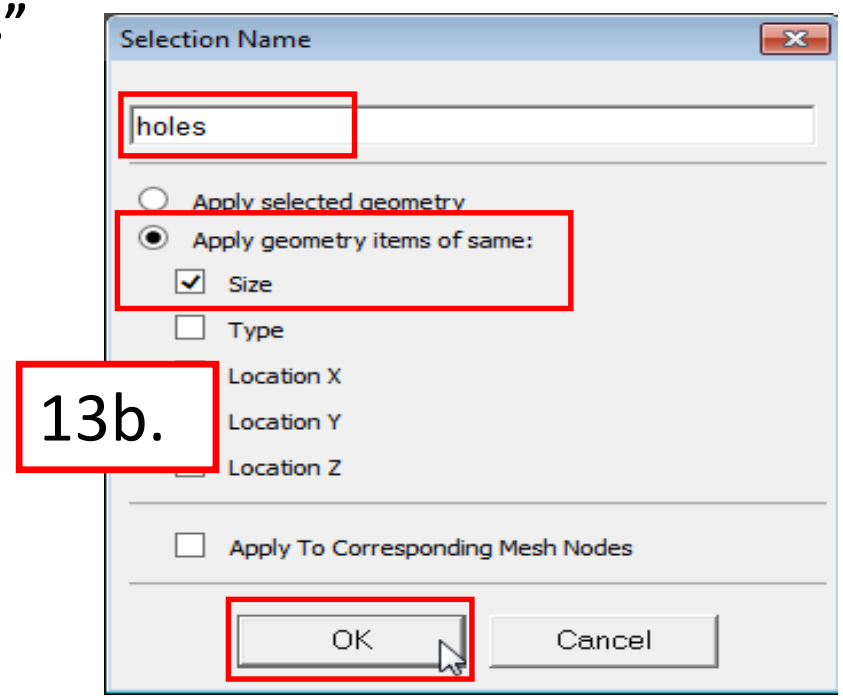
Mesh Control

13. Create a Named Selection:

a. **Select one of the 2 surfaces (i.e., half of the hole) in any one of the 8 smallest holes and RMB > Create Named Selection.**

b. **In the Selection Name dialog, change the name to "holes," select "Apply geometry items of same," check "Size," and click "OK."**

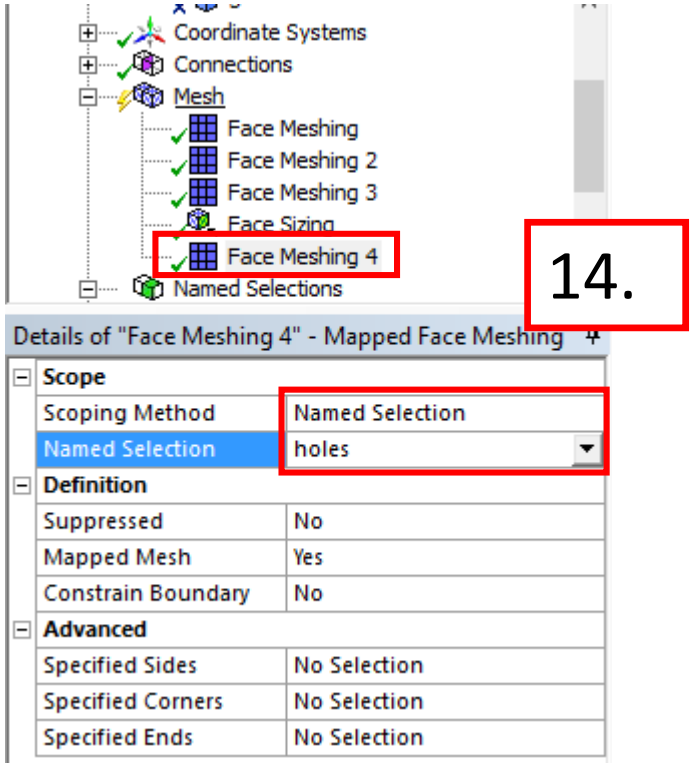




© 2017 ANSYS, Inc.

Mesh Control

14. Insert a Face Meshing control. Change the Scoping Method to “Named Selection” and change the Named Selection to “holes.” Generate the mesh.

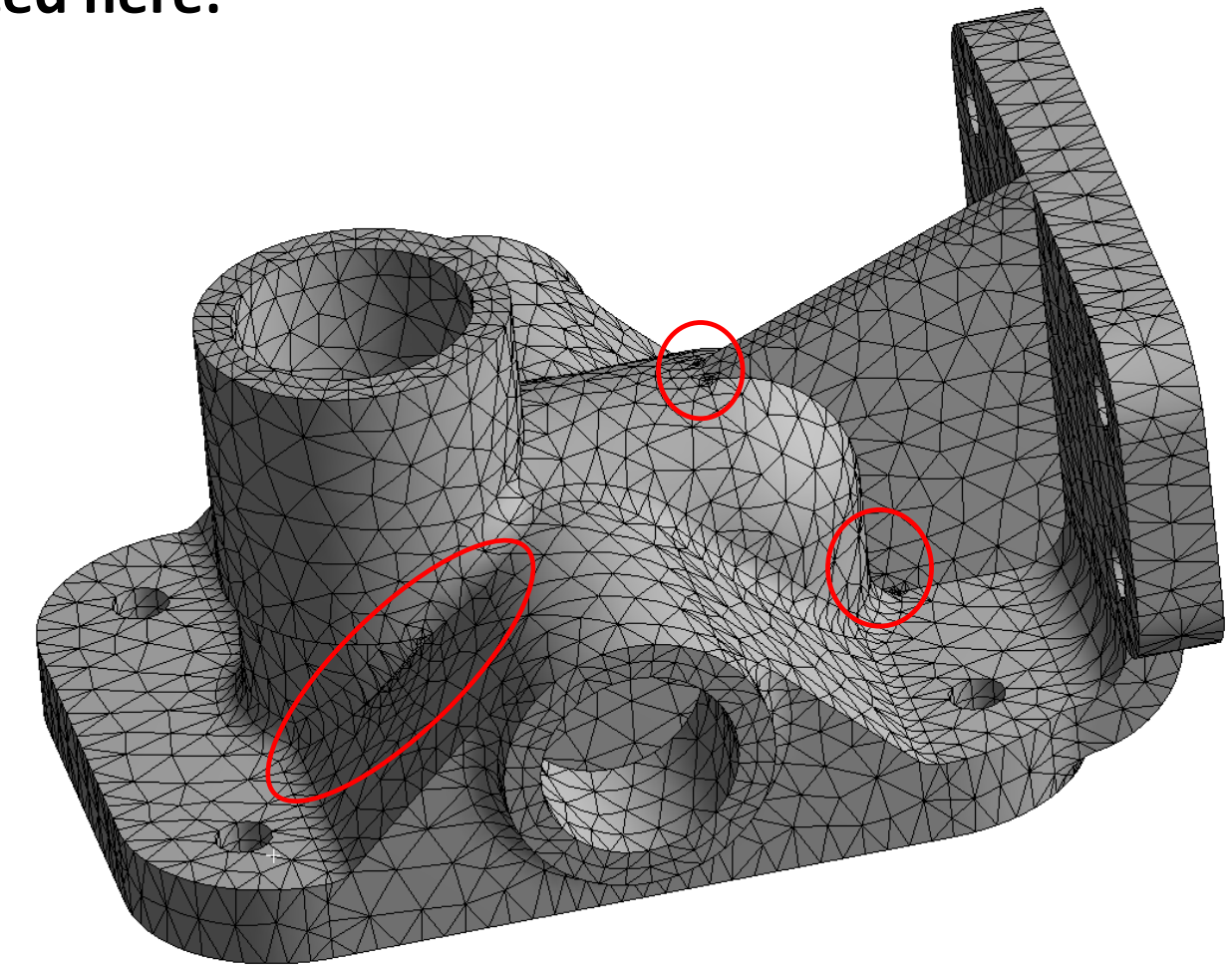


Observe that the number of distorted elements has been reduced by checking the mesh metric.

© 2017 ANSYS, Inc.

Geometry Correction

We will now focus on dealing with some small geometric features like the ones indicated here:



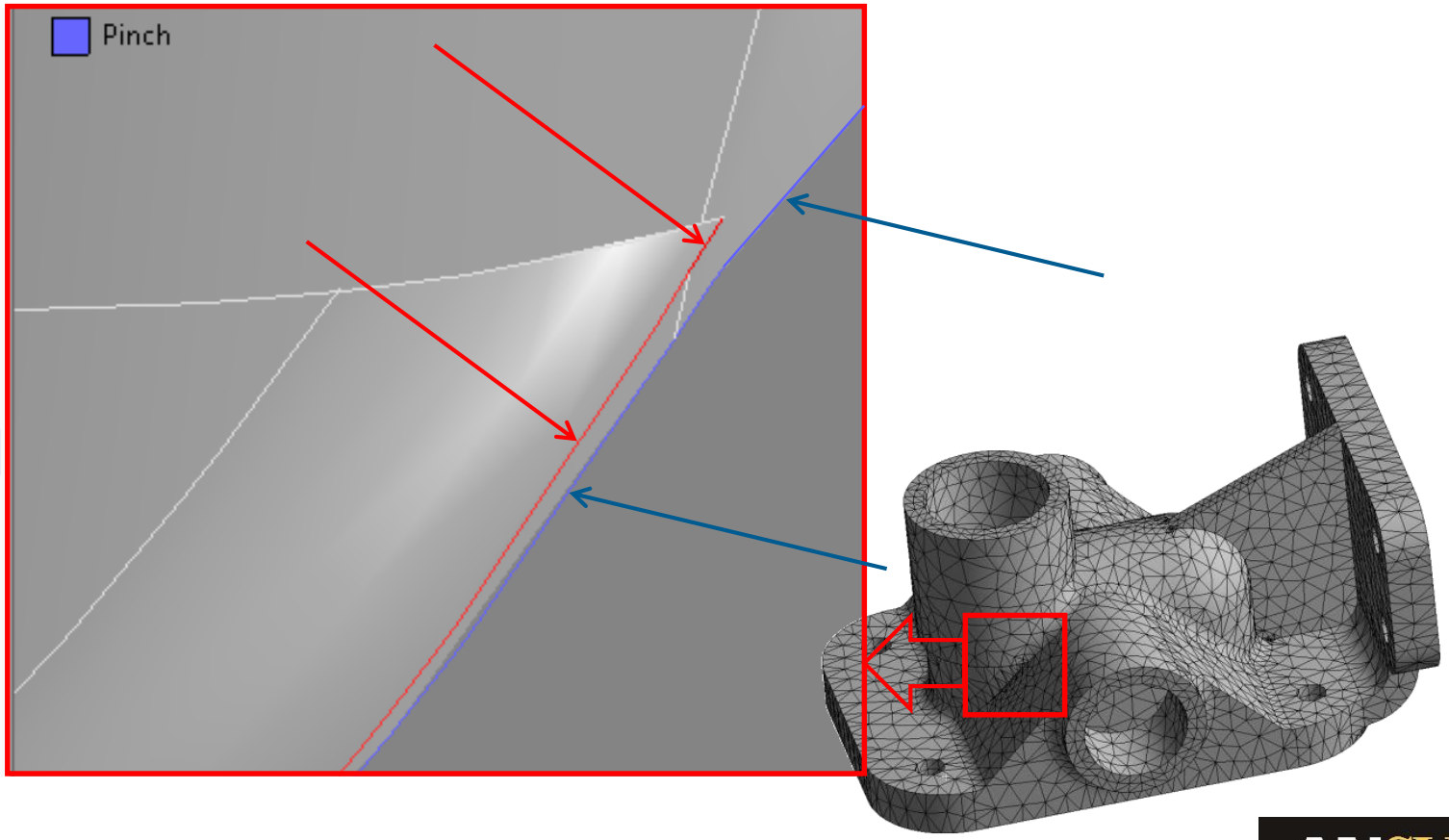
© 2017 ANSYS, Inc.

Geometry Correction: Pinch

15. Insert a Pinch control. Select the two edges shown in blue as the Master Geometry and the two edges shown in red as the Slave Geometry. Set the Tolerance to 2 mm.

Details of "Pinch" - Pinch

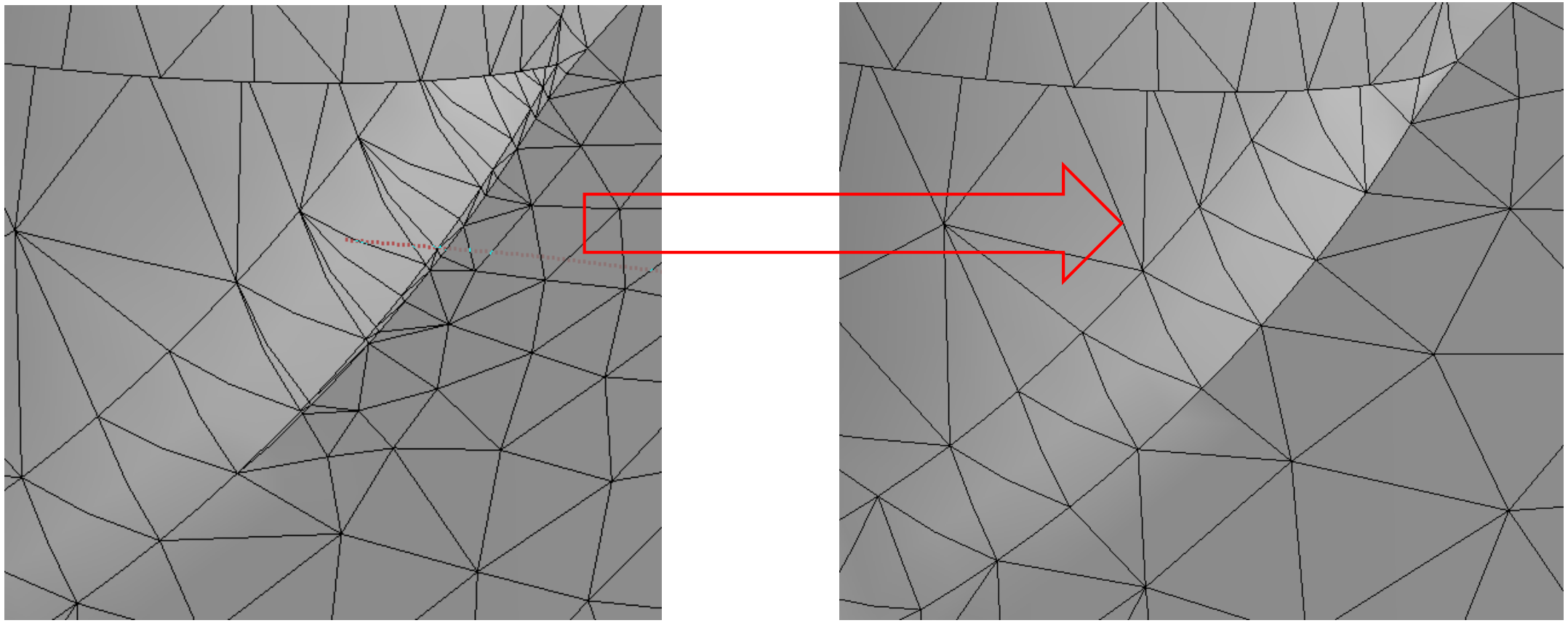
Scope	
Master Geometry	2 Edges
Slave Geometry	2 Edges
Definition	
Suppressed	No
Tolerance	2. mm
Scope Method	Manual



© 2017 ANSYS, Inc.

Geometry Correction: Pinch

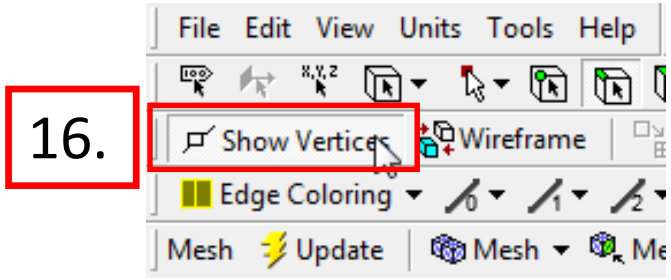
After generating the mesh, review the results in the region of the Pinch control and compare them to the original mesh in the same region.

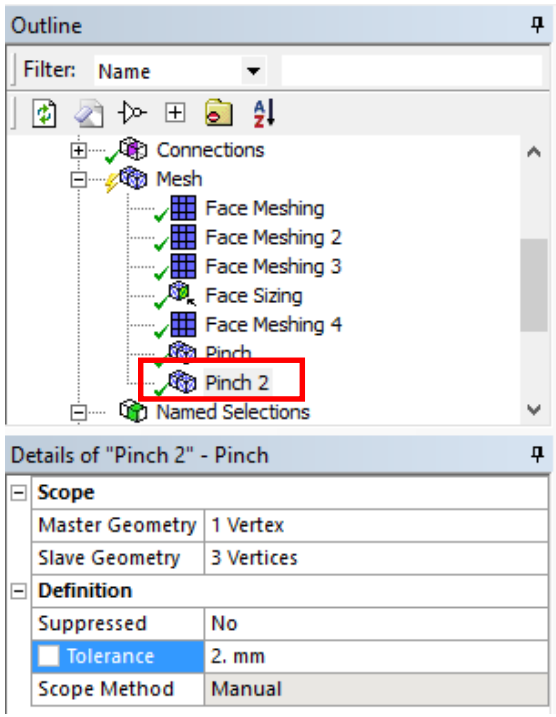


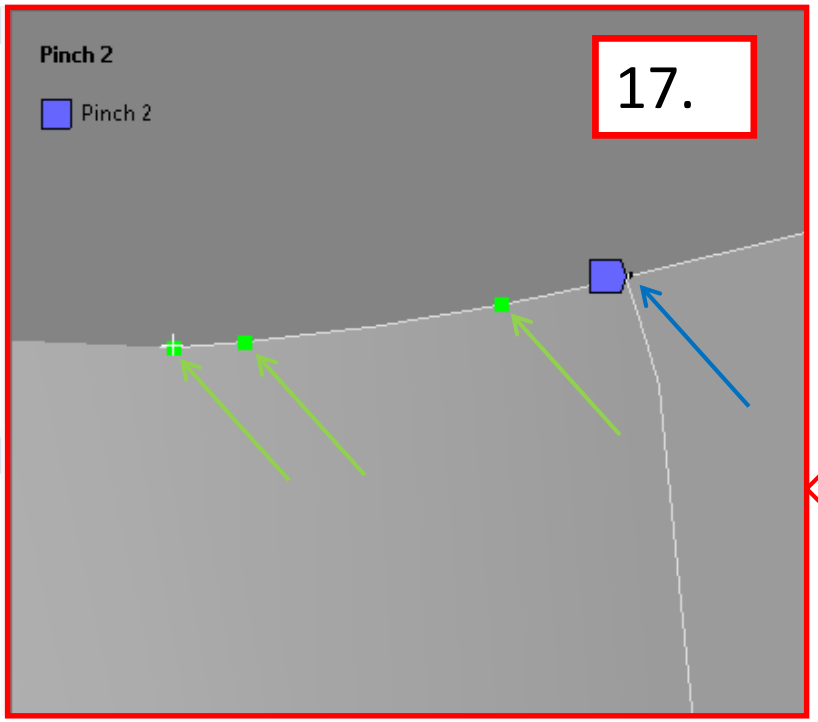
© 2017 ANSYS, Inc.

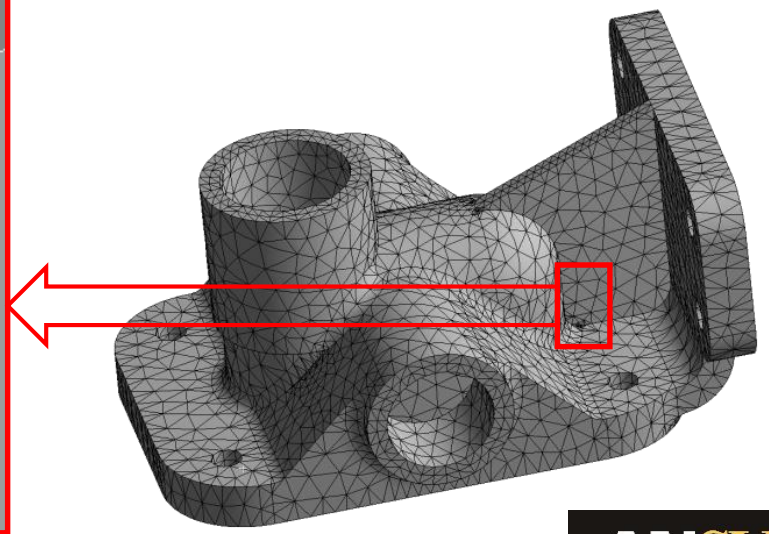
Geometry Correction: Pinch

16. Clear the mesh (RMB > Clear Generated Data) and activate Show Vertices.
17. Insert another Pinch control. Select the vertex indicated by the blue flag as the Master Geometry and the three vertices highlighted in green as the Slave Geometry. Set the Tolerance to 2 mm and generate the mesh.









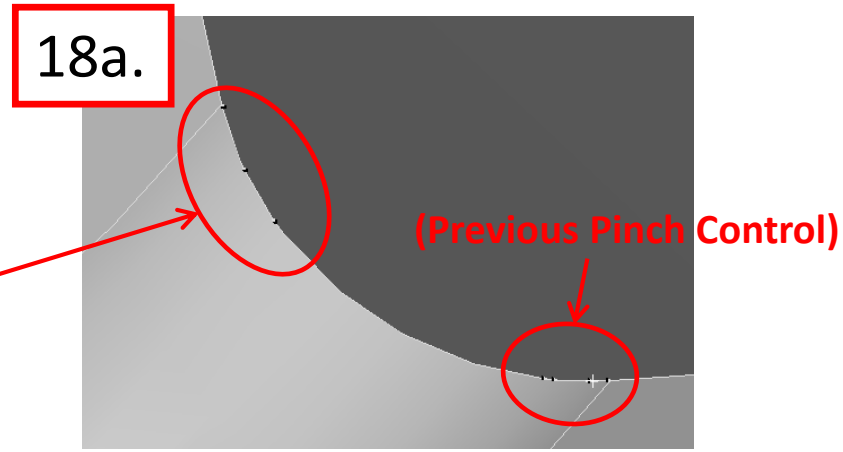
© 2017 ANSYS, Inc.

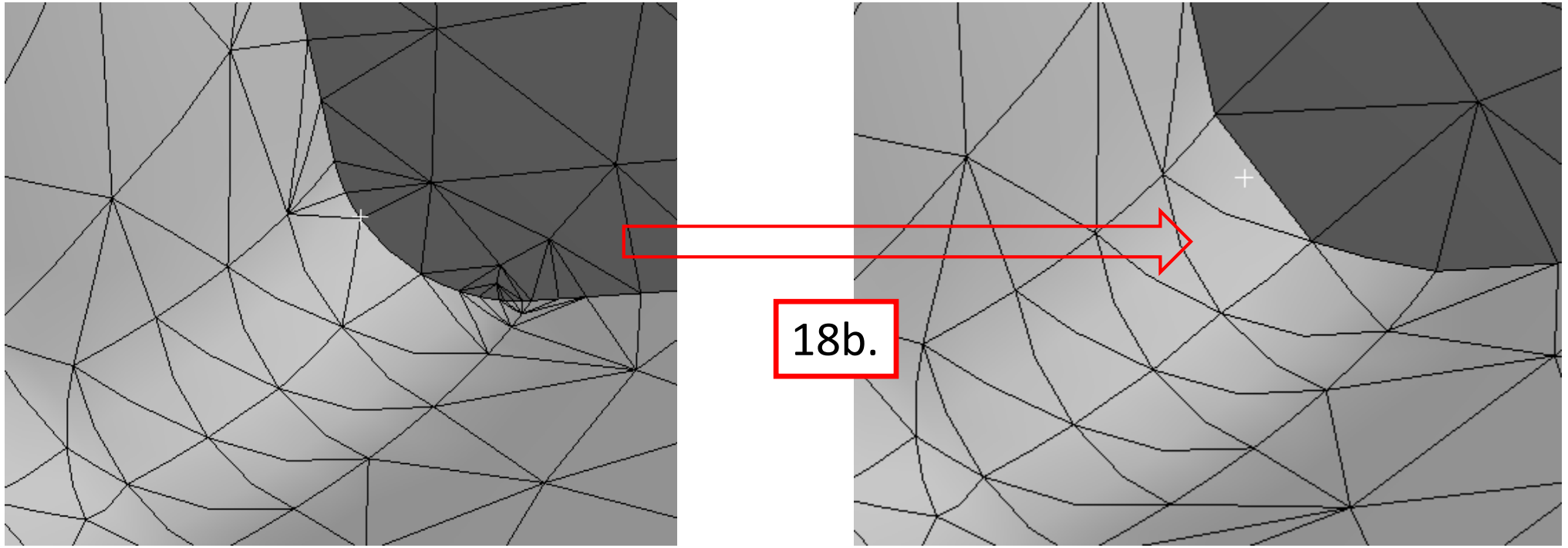
Geometry Correction: Pinch

18. Create another Pinch control:

a. Scope the Pinch control to the vertices shown. Adjust the tolerance value if necessary.

b. Compare the mesh results in this region.

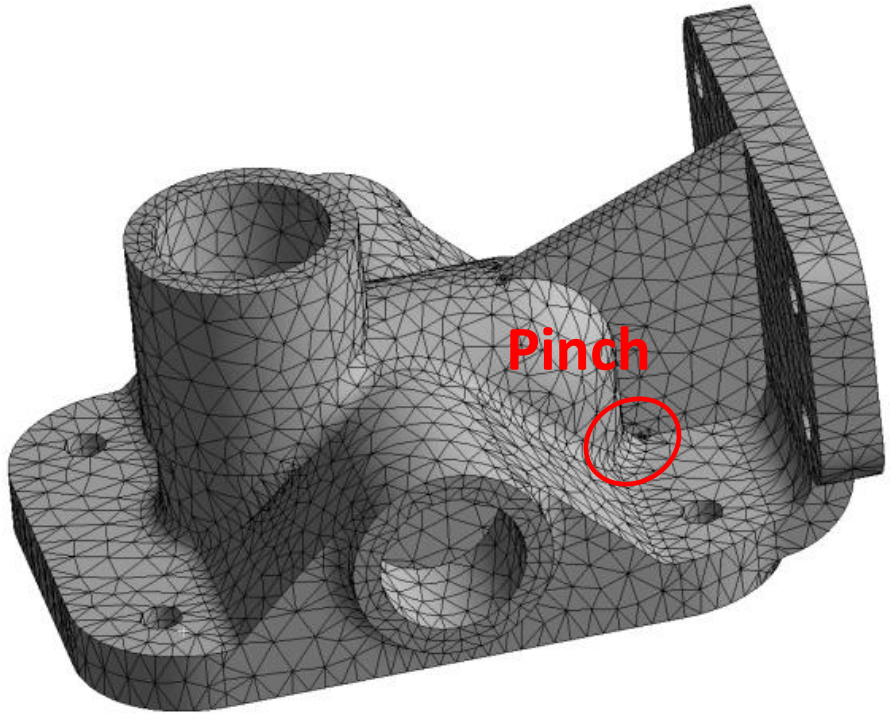


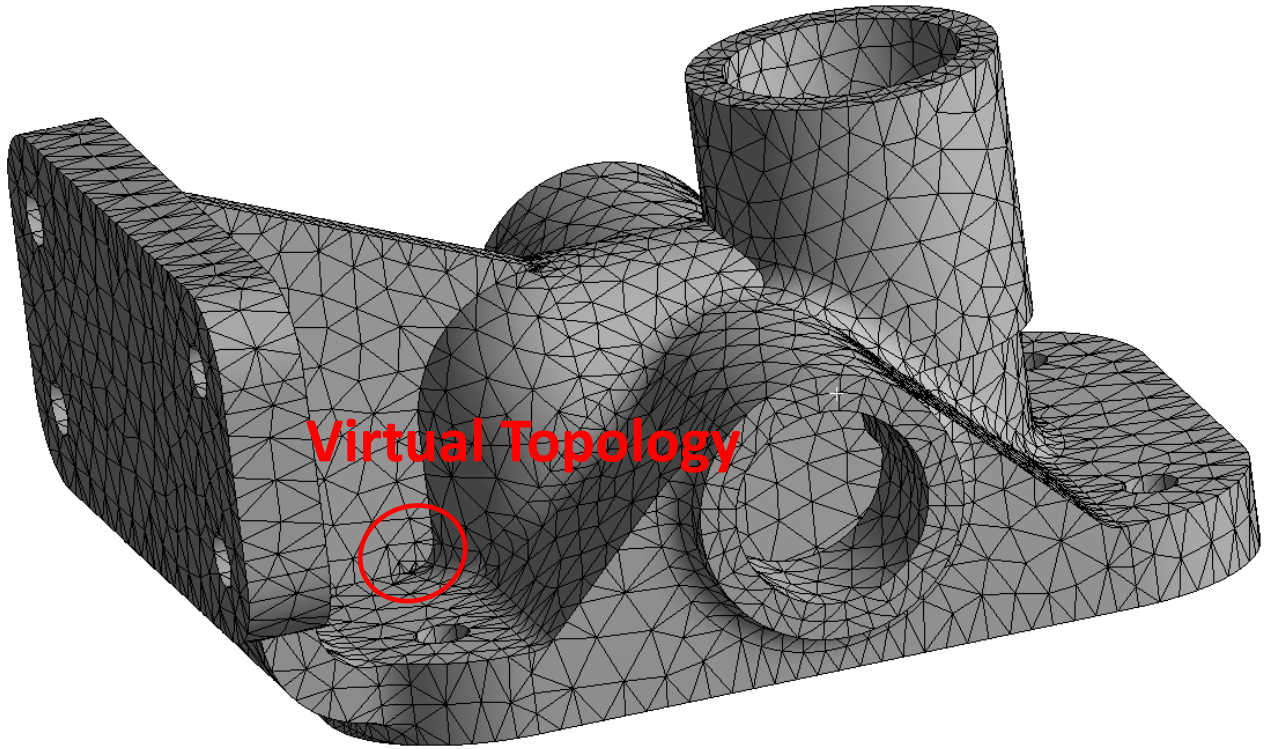


© 2017 ANSYS, Inc.

Geometry Correction: Virtual Topology

Virtual Topology offers another method of consolidating small geometric details. We will use a Virtual Topology object to mimic the behavior of the two Pinch controls on the opposite side of the part:





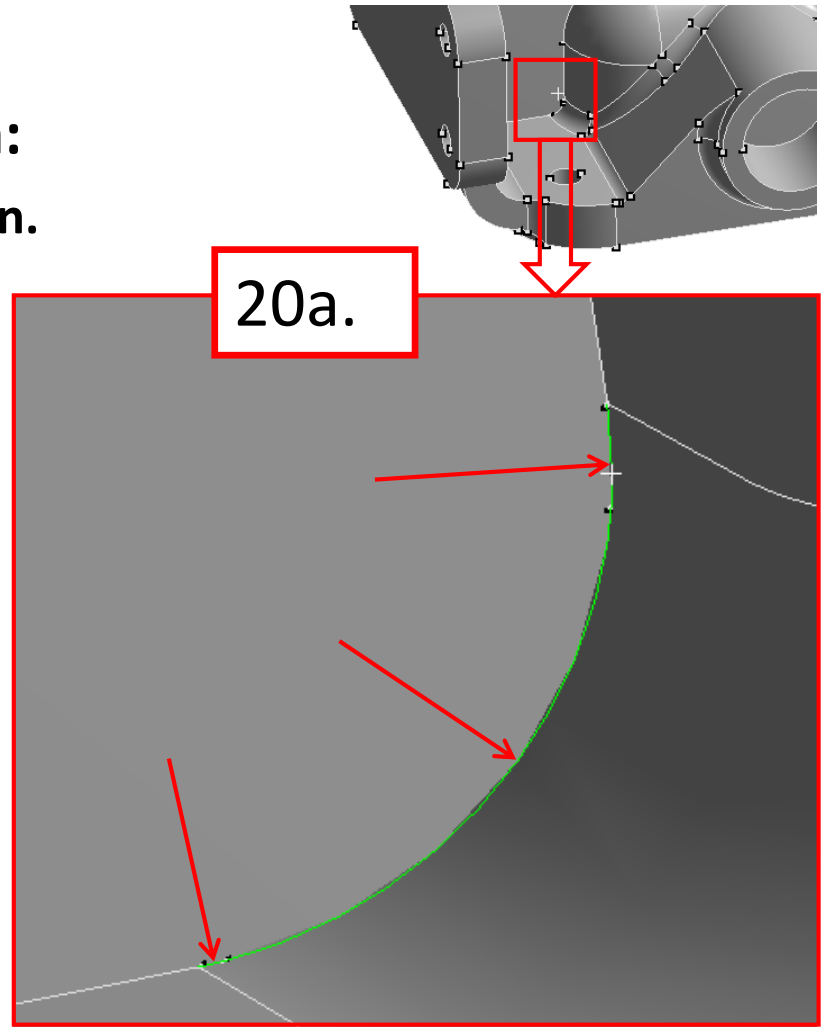
© 2017 ANSYS, Inc.

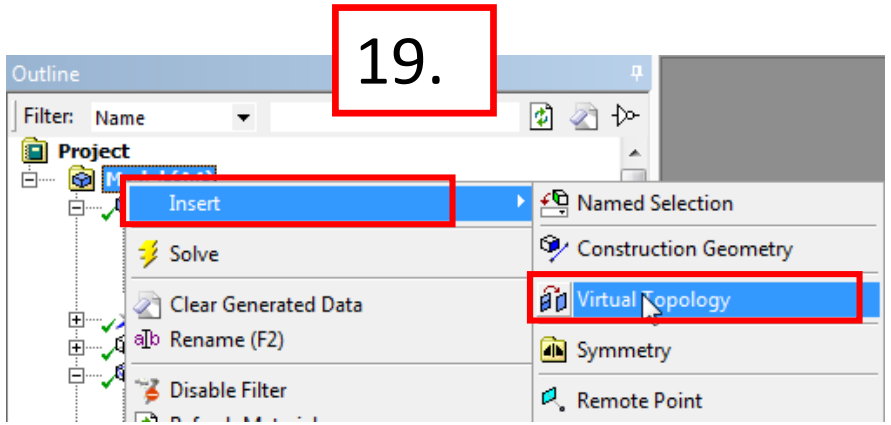
Geometry Correction: Virtual Topology

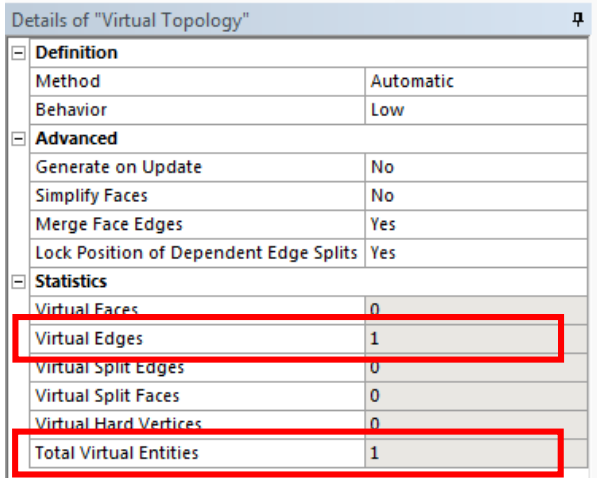
19. From the Model branch, RMB > Insert > Virtual Topology.

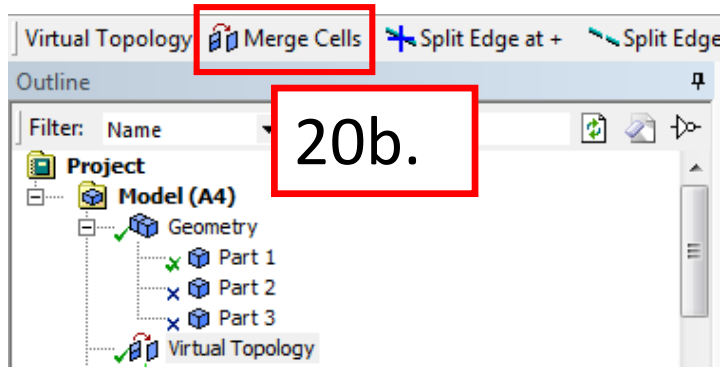
20. Complete the definition:
    a. Select the 3 edges shown.
    b. Click on Merge Cells.

Generate the mesh and compare.





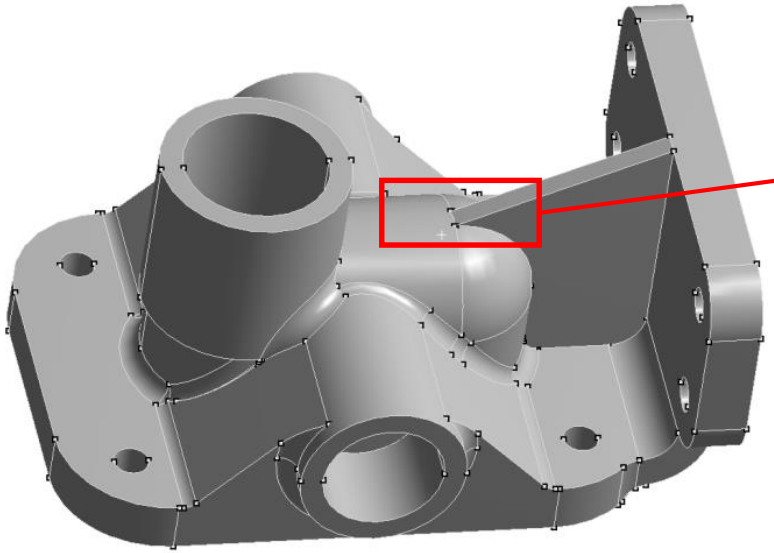


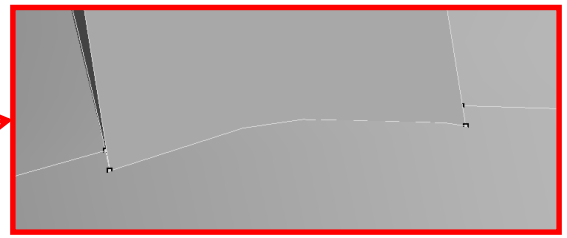


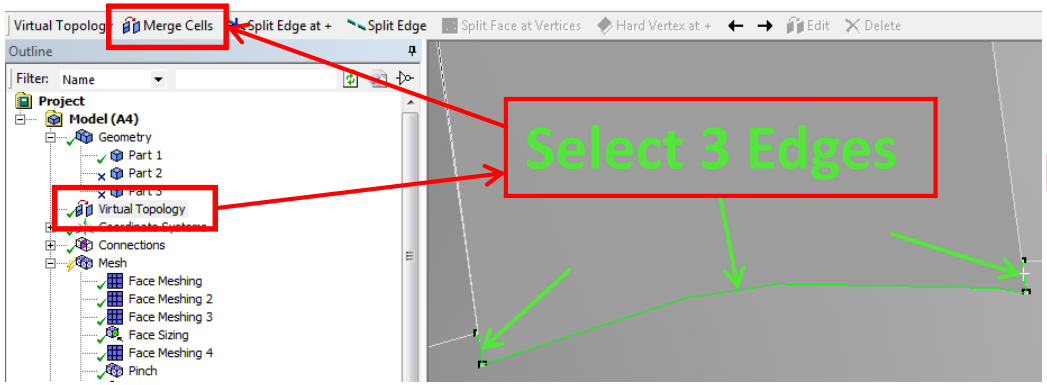
© 2017 ANSYS, Inc.

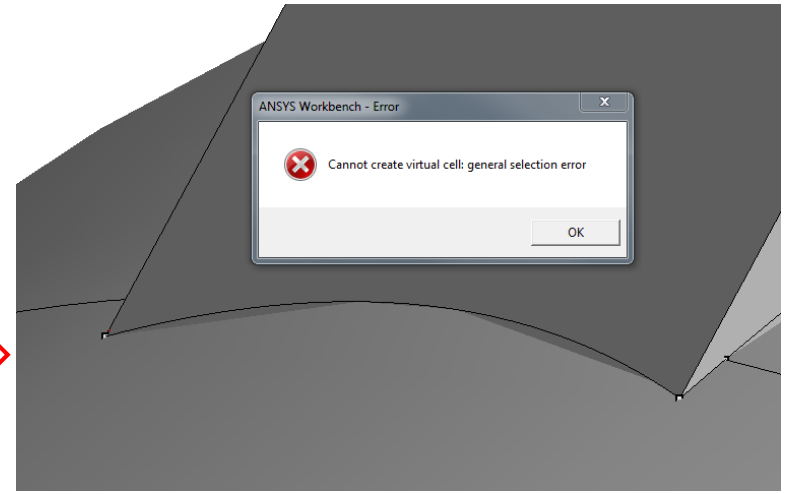
Geometry Correction: Virtual Topology versus Pinch

Pinch and Virtual Topology are often complementary tools. If one cannot be applied successfully in a particular application, the other may work better. An example:







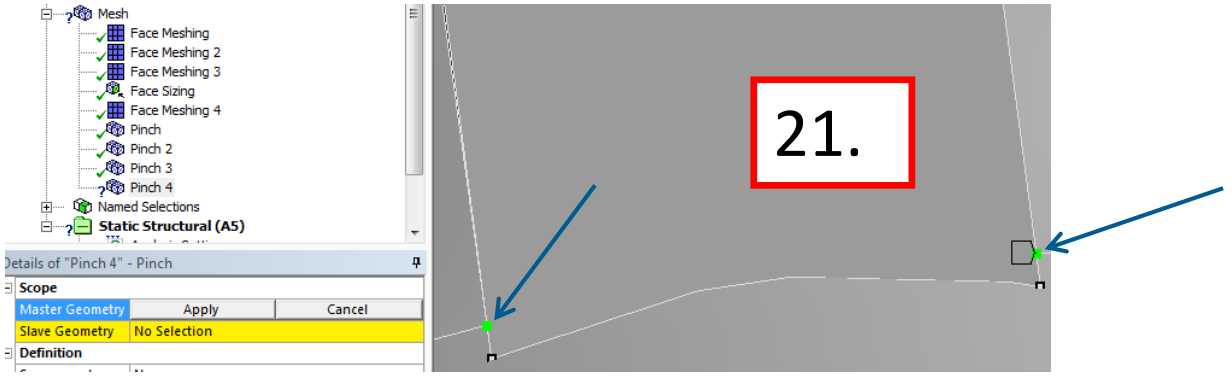


© 2017 ANSYS, Inc.

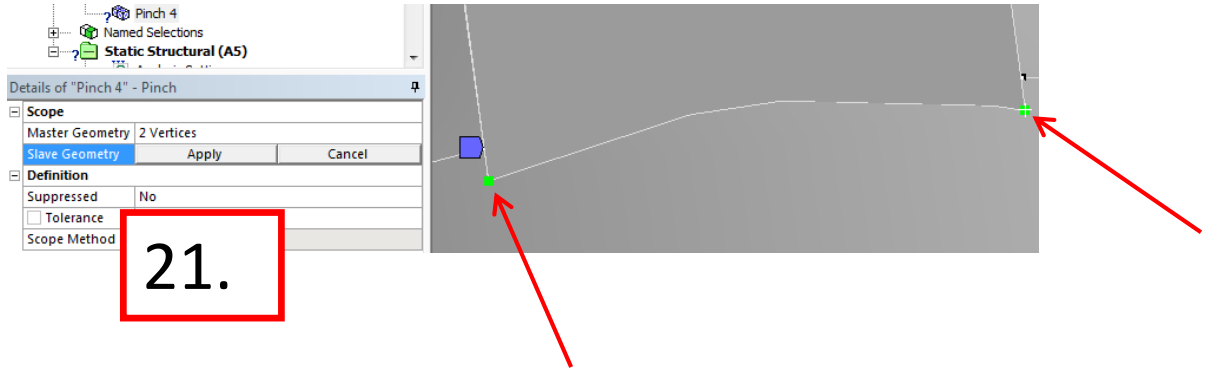
Geometry Correction: Virtual Topology versus Pinch

Try Pinch control instead:

21. Insert a Pinch control and select the two vertices shown here as the Master Geometry:



and the two vertices shown here as the Slave Geometry:

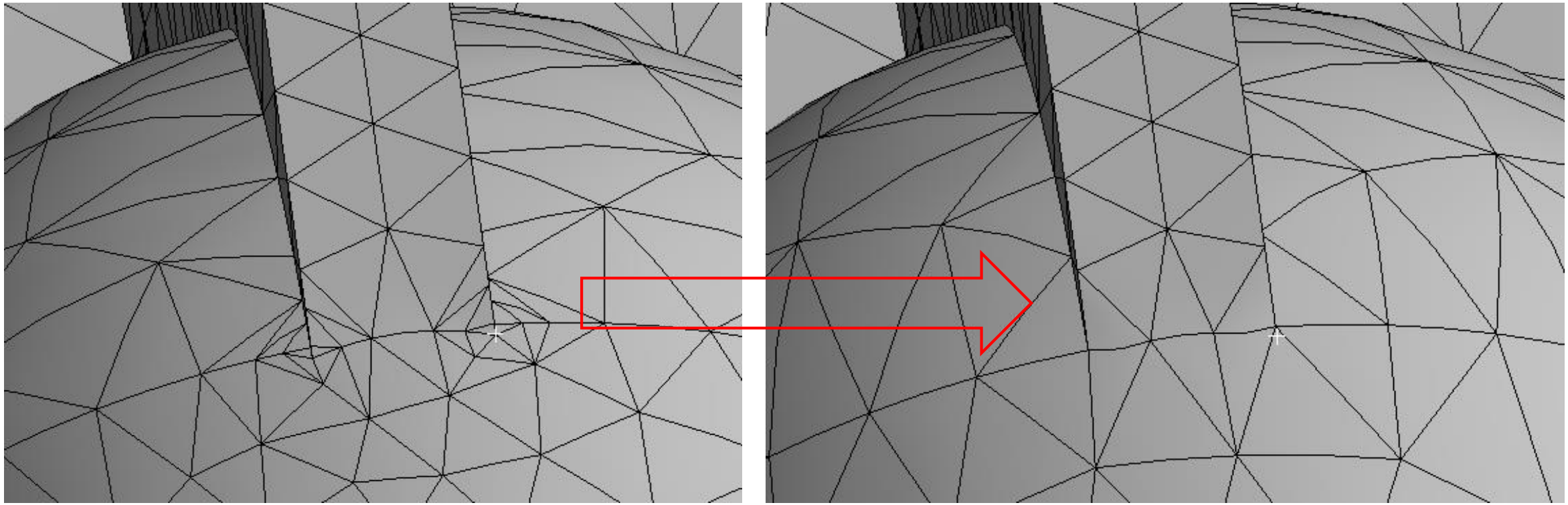


Use a Tolerance of 1 mm.

© 2017 ANSYS, Inc.

Geometry Correction: Virtual Topology versus Pinch

Generated the mesh and compare:

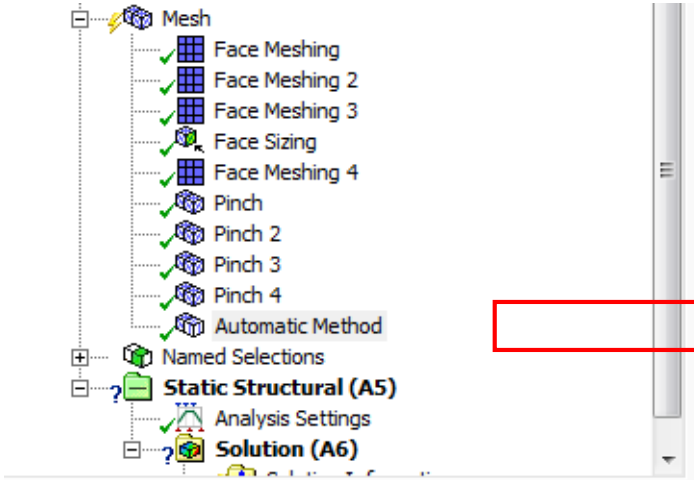


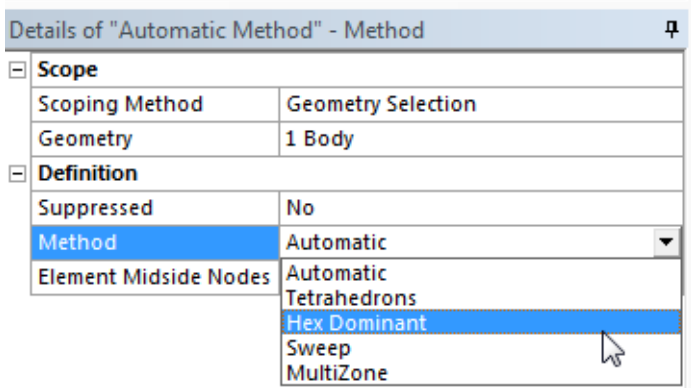
© 2017 ANSYS, Inc.

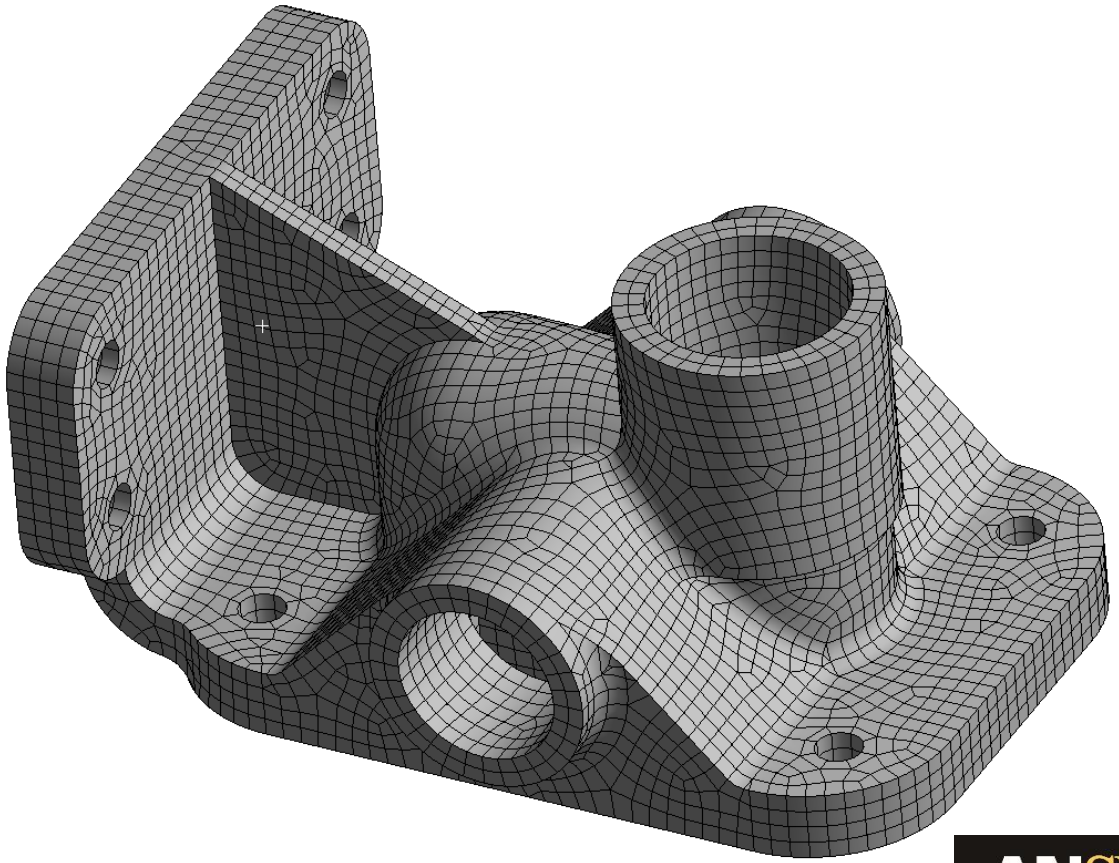
Go Further!

In this workshop we focused on sizing and cleaning operations. If you find yourself with extra time, try the following:

1. **Test the Hex Dominant method on part "1:"**



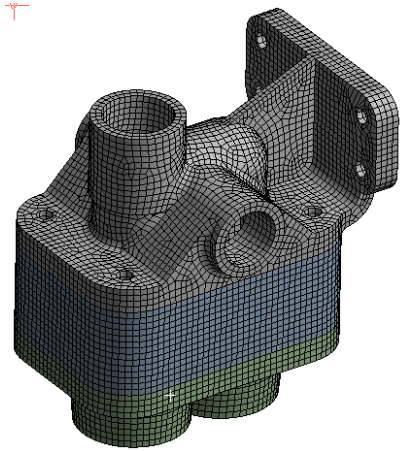


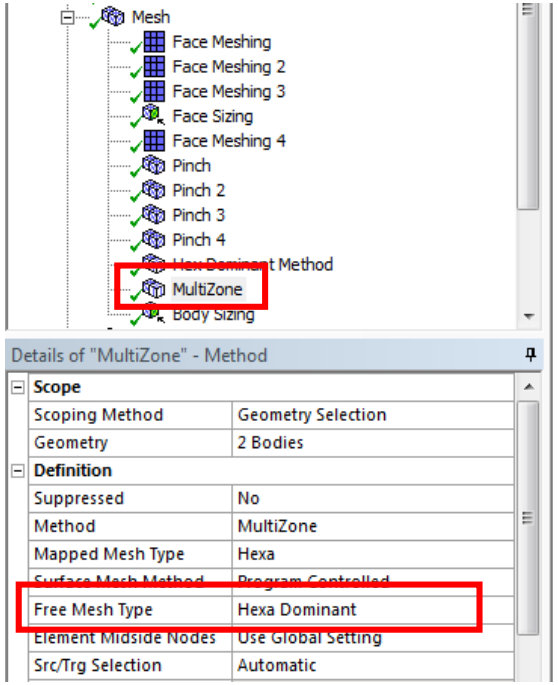


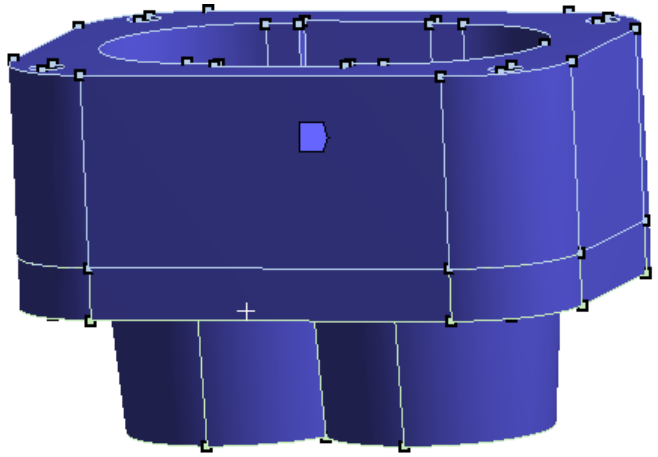
© 2017 ANSYS, Inc.

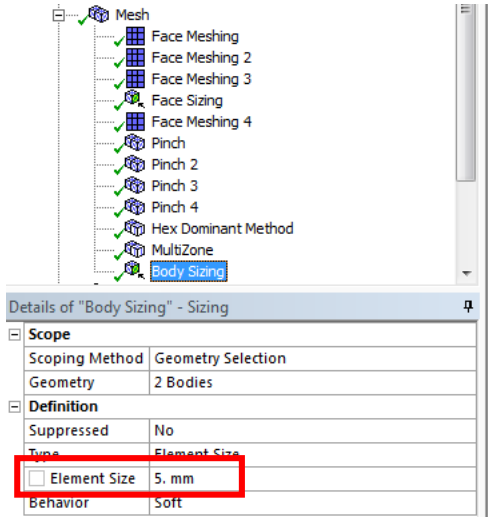
Go Further!

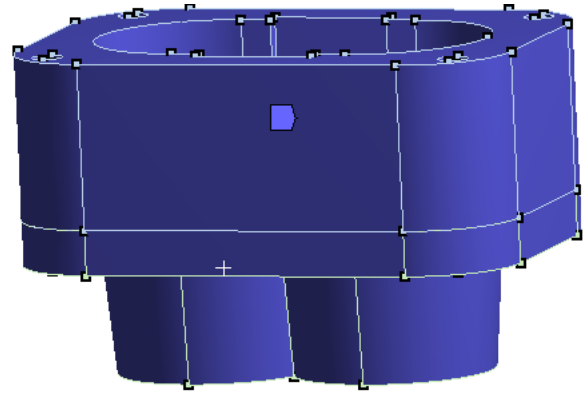
2. **Unsuppress part “2” and part “3.” Insert a Sizing control with Element Size = 5 mm scoped to both bodies, and test the Multizone method with Free Mesh Type = Hexa Dominant on both bodies.**







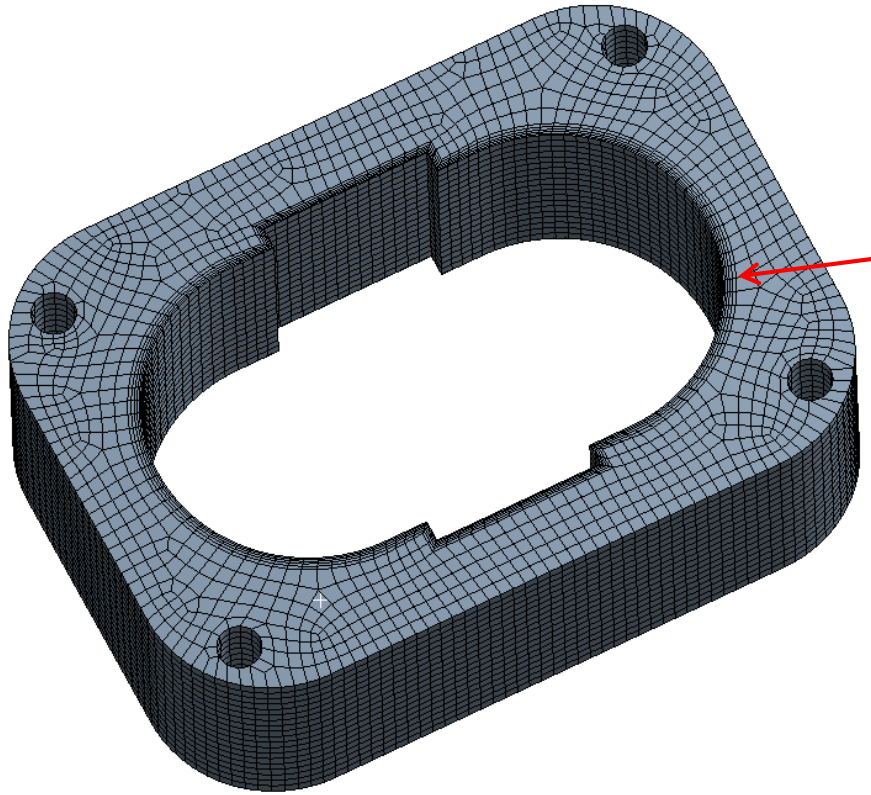


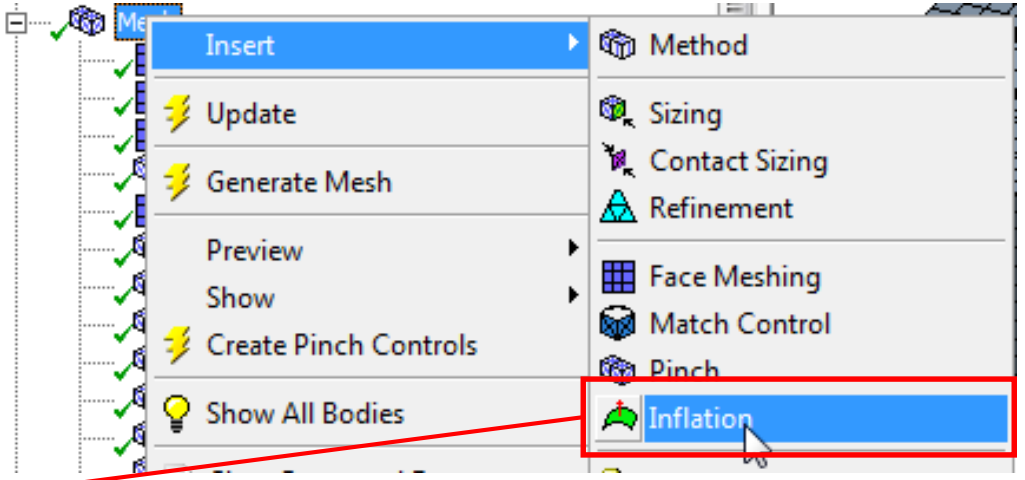


© 2017 ANSYS, Inc.

Go Further!

3. **Test Inflation on part “2.” Use a Sizing control of 3mm on the body.**





© 2017 ANSYS, Inc.

ANSYS®

Workshop 05.2: Mesh Control

END