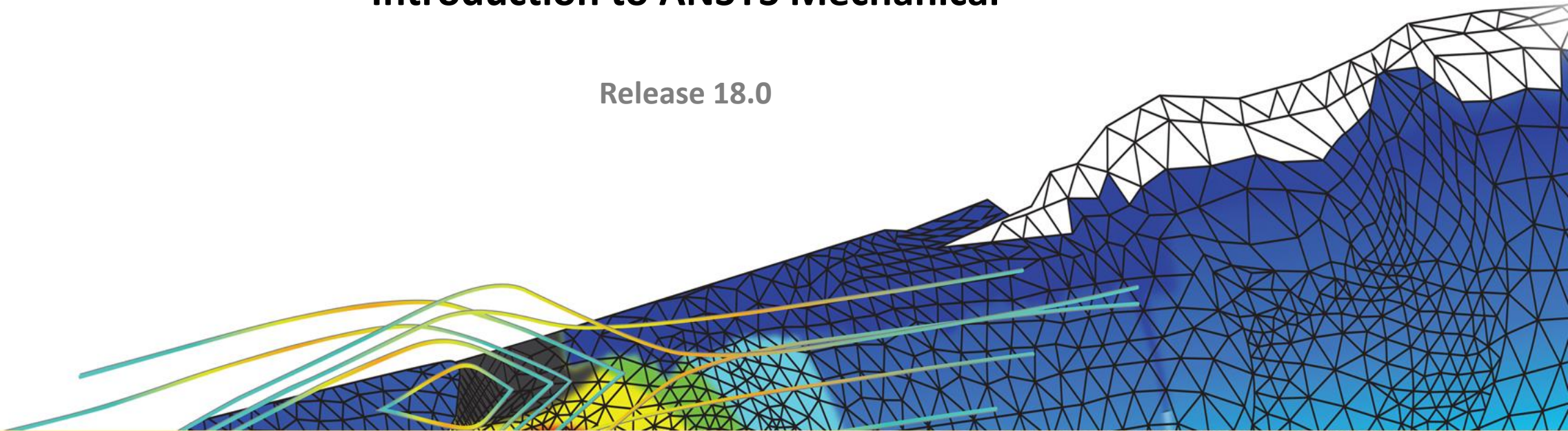




# Workshop 05.2: Mesh Control

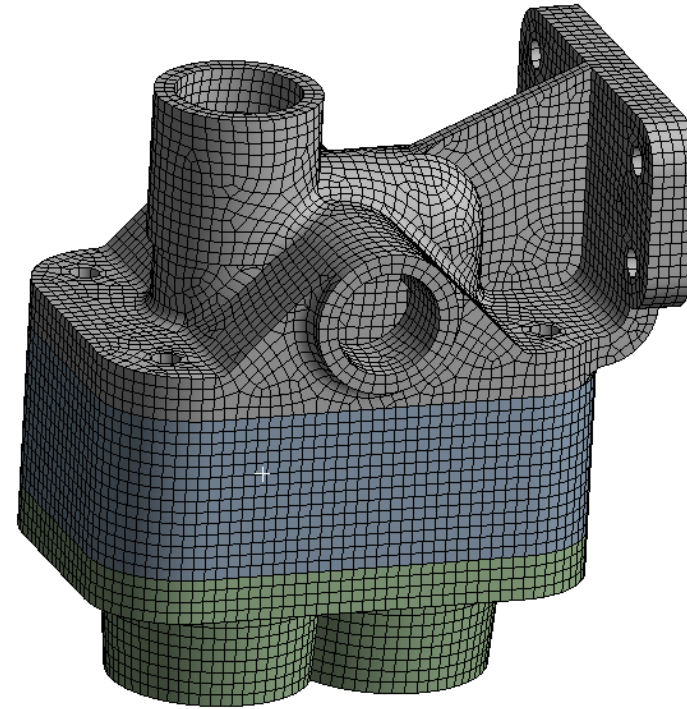
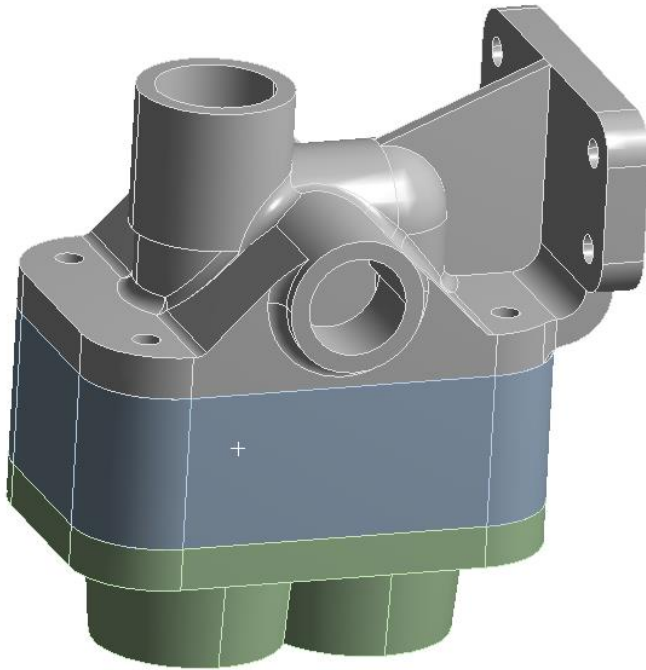
## Introduction to ANSYS Mechanical

Release 18.0



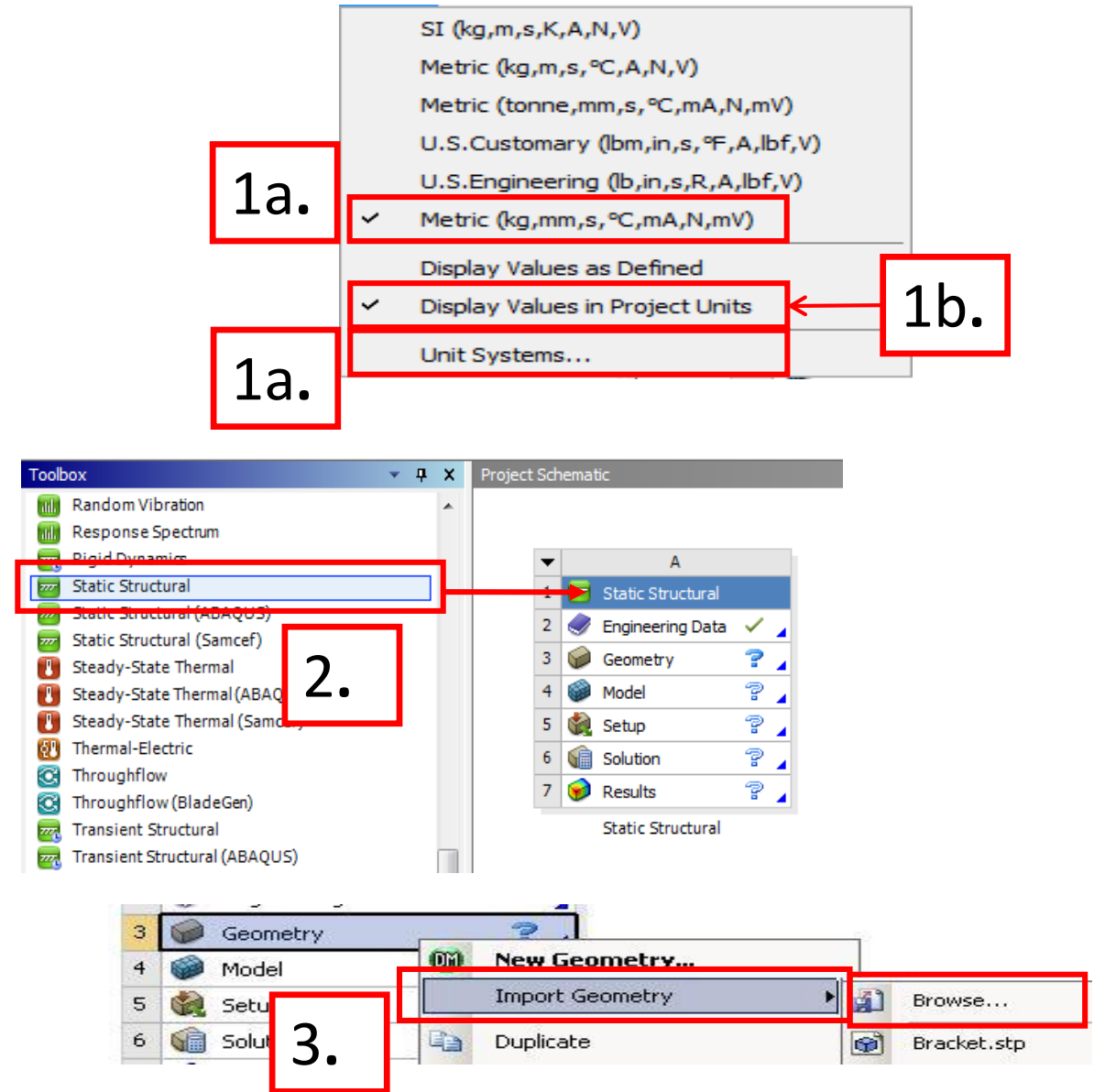
# Goals

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.



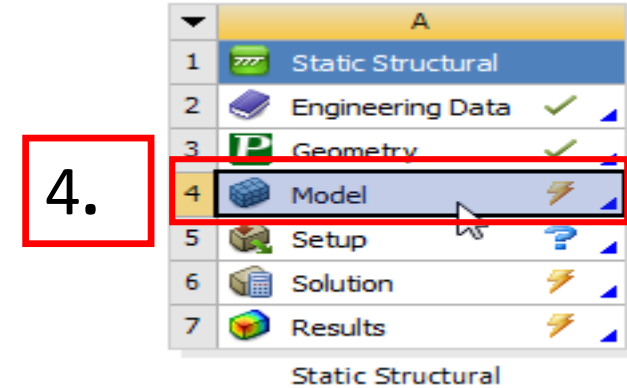
# Project Schematic

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.”

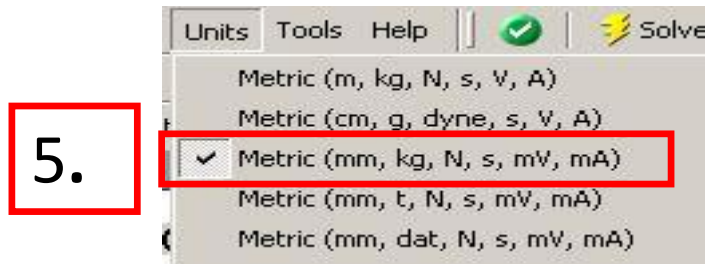


# Project Schematic

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



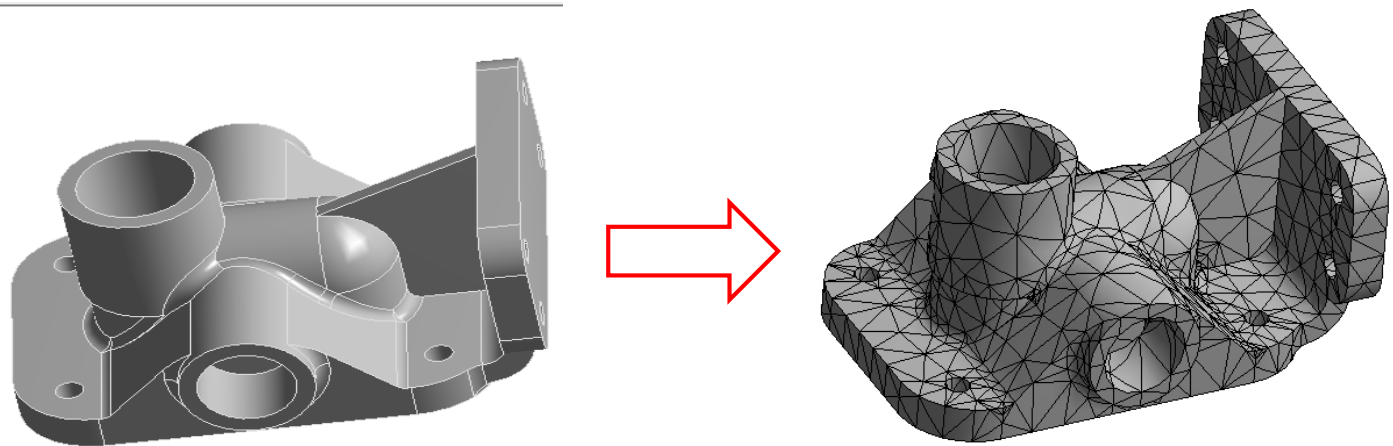
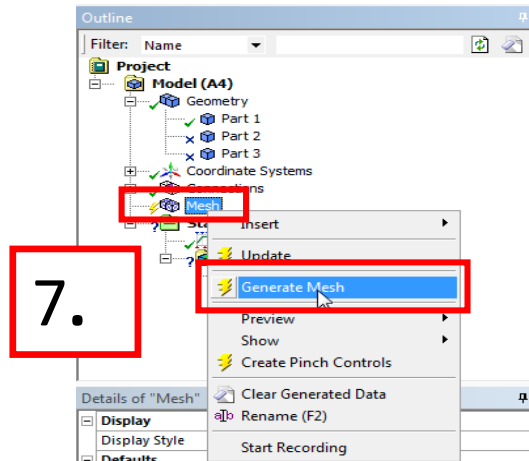
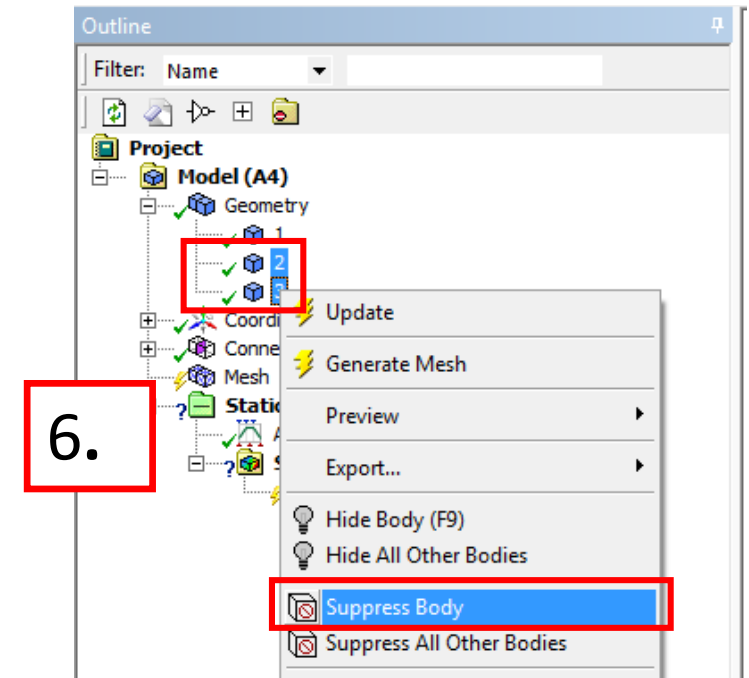
5. Set the working unit system to “Metric (mm, kg, N, s, mV, mA).”





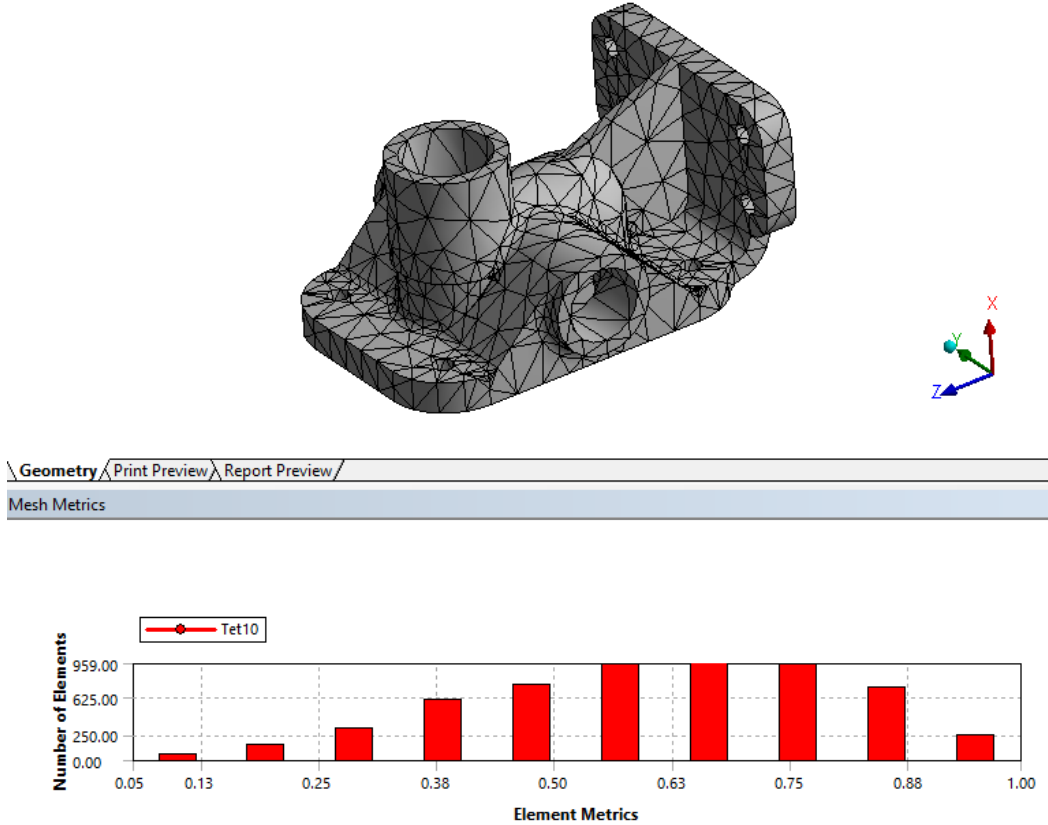
# Mesh Control

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.



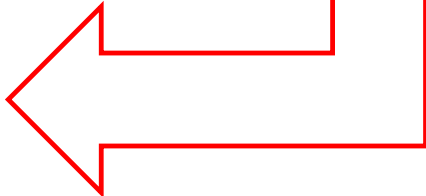
# Mesh Control

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



Details of "Mesh"

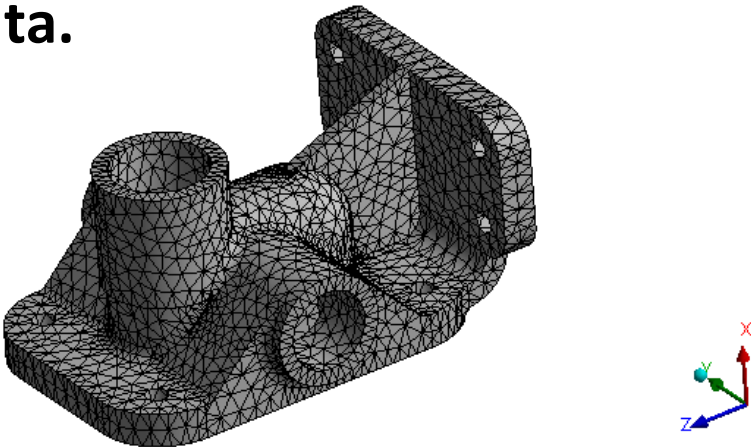
Display	
Display Style	Body Color
Defaults	
Physics Preference	Mechanical
<input type="checkbox"/> Relevance	0
Element Midside Nodes	Program Cont
Sizing	
Quality	
Check Mesh Quality	Yes, Errors
Error Limits	Standard Mechanical
<input type="checkbox"/> Target Quality	Default (0.050000)
Smoothing	Medium
Mesh Metric	Element Quality
<input type="checkbox"/> Min	5.126e-002
<input type="checkbox"/> Max	0.99534
<input type="checkbox"/> Average	0.6074
<input type="checkbox"/> Standard Deviation	0.19257



8.

# 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.



Coordinate Systems  
Connections  
**Mesh**  
Static Structural (A5)

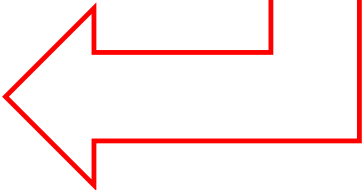
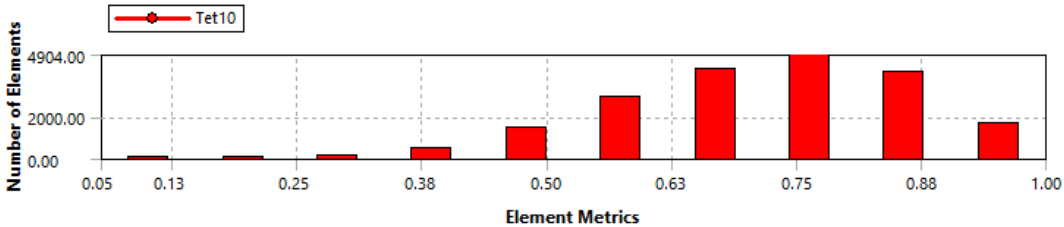
Details of "Mesh"

Display	
Display Style	Body Color
Defaults	
Physics Preference	Mechanical
Relevance	0
Element Midside Nodes	Program Controlled
Sizing	
Size Function	Adaptive
Relevance Center	Coarse
<b>Element Size</b>	<b>4.0 mm</b>
Initial Size Seed	Active Assembly
Transition	Fast
Span Angle Center	Coarse
Automatic Mesh Based D	On

9.

Geometry | Print Preview | Report Preview |

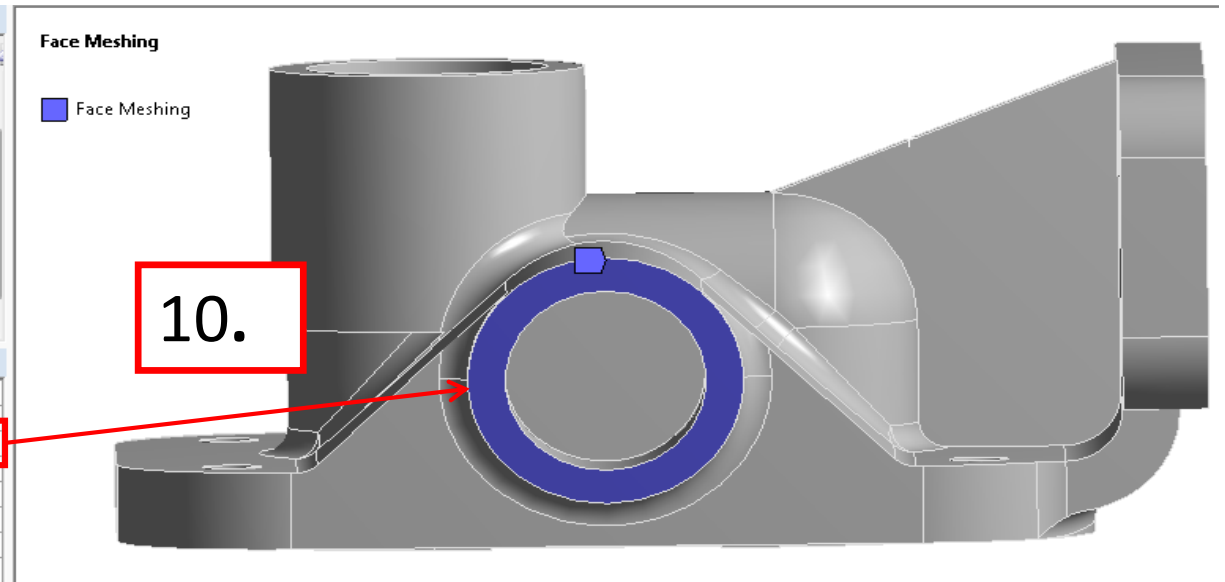
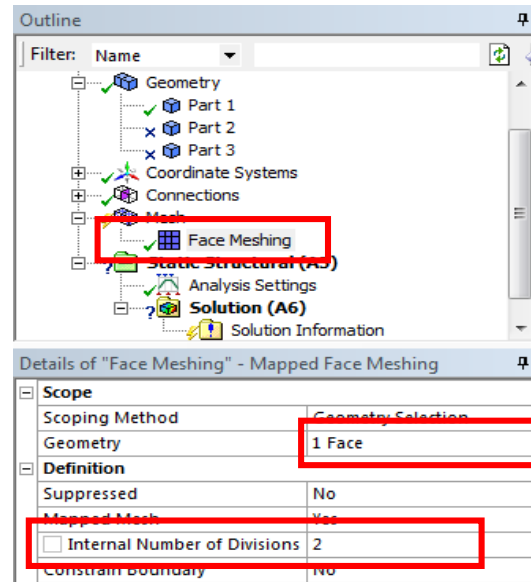
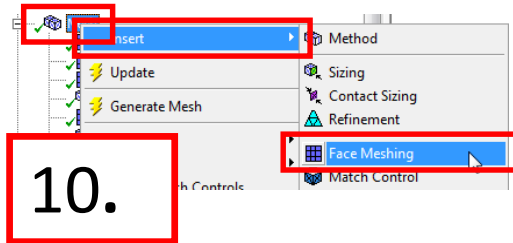
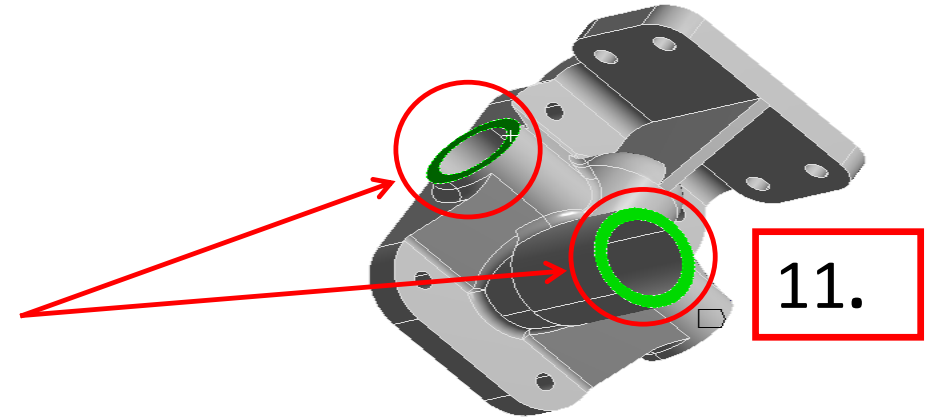
Mesh Metrics



# 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.





# Mesh Control

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

12.

Insert

Method

Sizing

Update

Generate Mesh

Preview

Show

Create Pinch Controls

Contact Sizing

Refinement

Face Meshing

Match Co

Name

Geometry

Part 1

Part 2

Part 3

Coordinate Systems

Connections

Mesh

Face Meshing

Face Meshing 2

Face Meshing 3

Face Sizing

Static Structural (A5)

Details of "Face Sizing" - Sizing

Scope

Scoping Method: Geometry Selection

Geometry: 1 Face

Definition

Suppressed: No

Type: Element Size

Element Size: 3. mm

Behavior: Sort

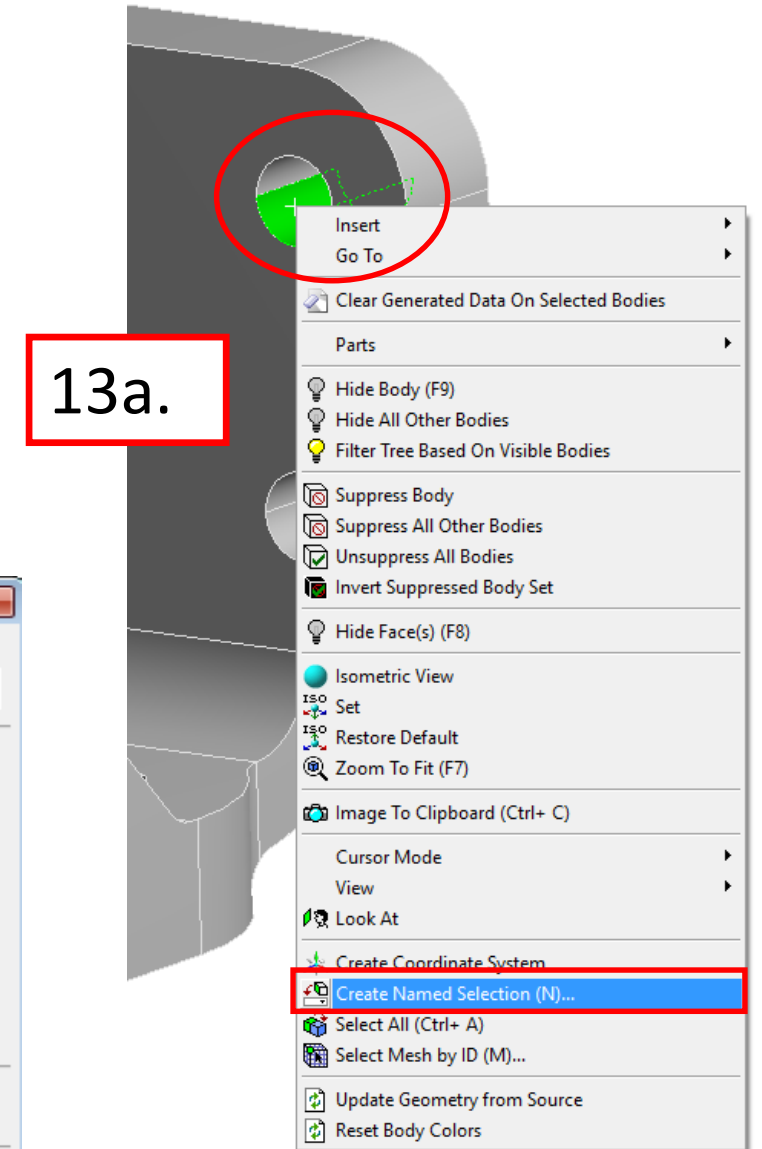
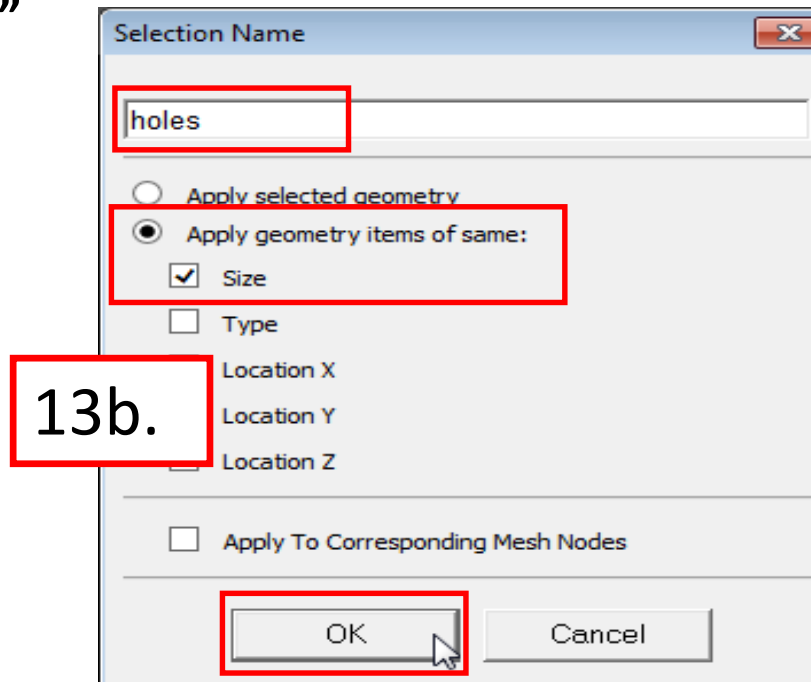
Face Sizing

Face Sizing

# Mesh Control

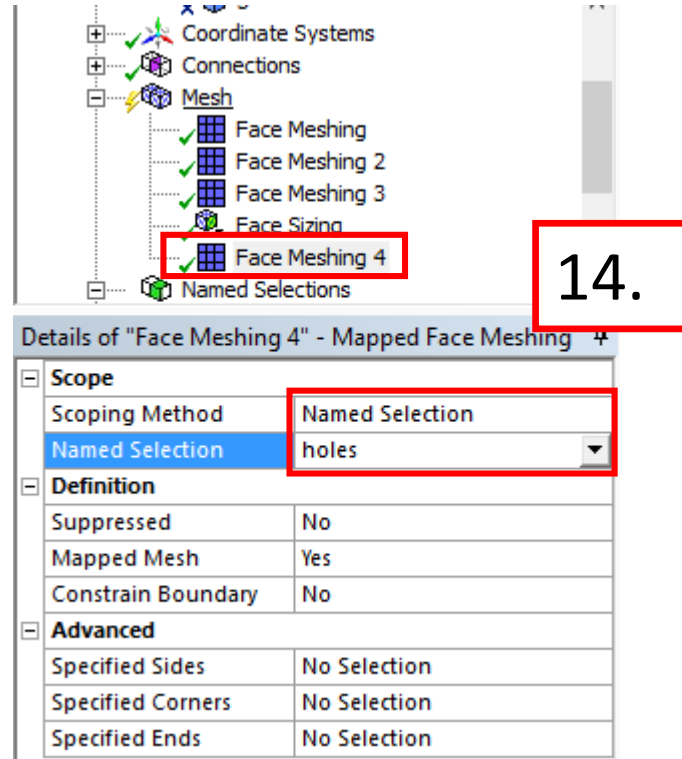
## 13. Create a Named Selection:

- 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.
- In the Selection Name dialog, change the name to “holes,” select “Apply geometry items of same,” check “Size,” and click “OK.”



# Mesh Control

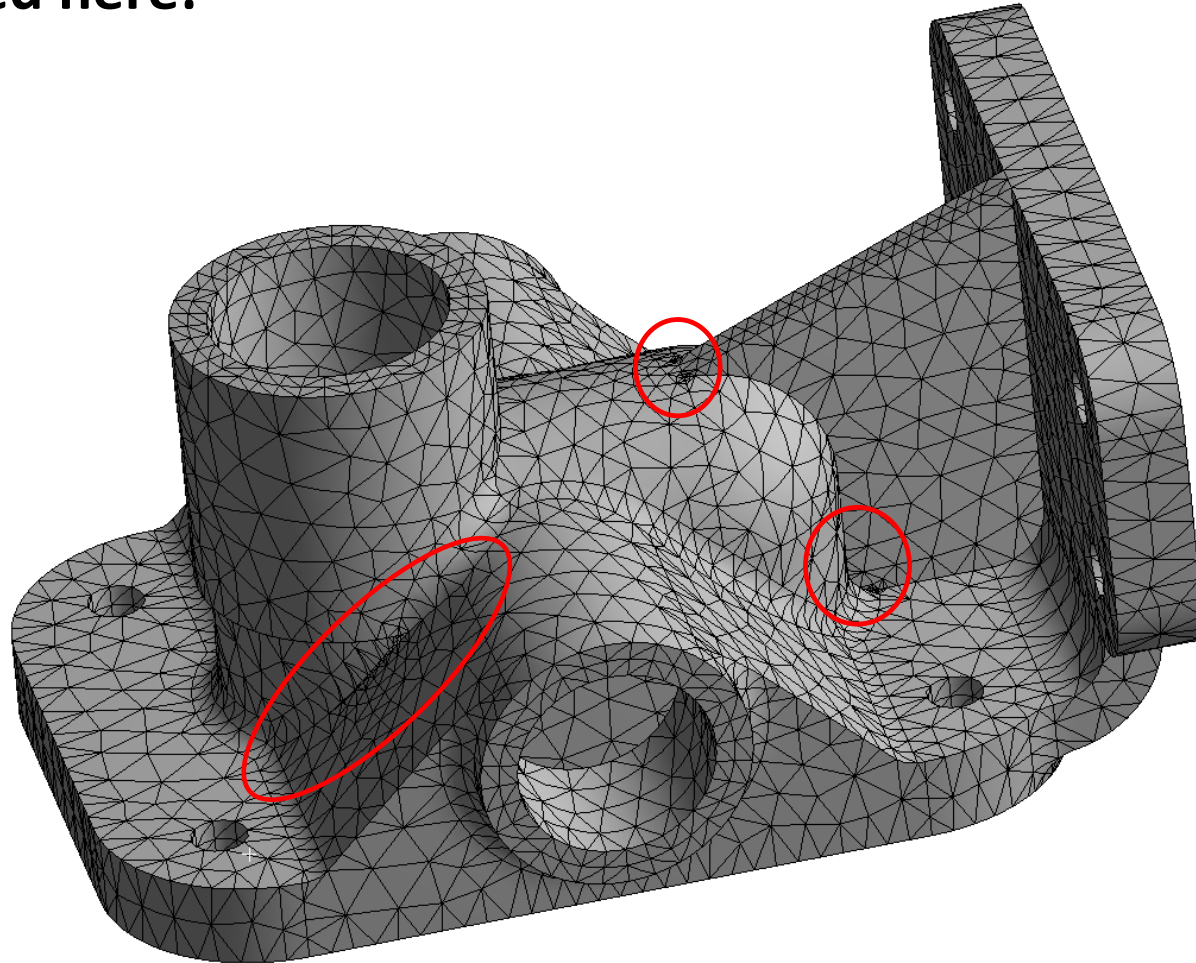
**14. Insert a Face Meshing control. Change the Scoping Method to “Named Selection” and change the Named Selection to “holes.” Generate the mesh.**



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

# Geometry Correction

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





# 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.

Mesh

- Face Meshing
- Face Meshing 2
- Face Meshing 3
- Face Sizing
- Face Meshing 4
- Pinch
- Named Selections
- Static Structural (A5)
- Analysis Settings
- Solution (A6)

Details of "Pinch" - Pinch

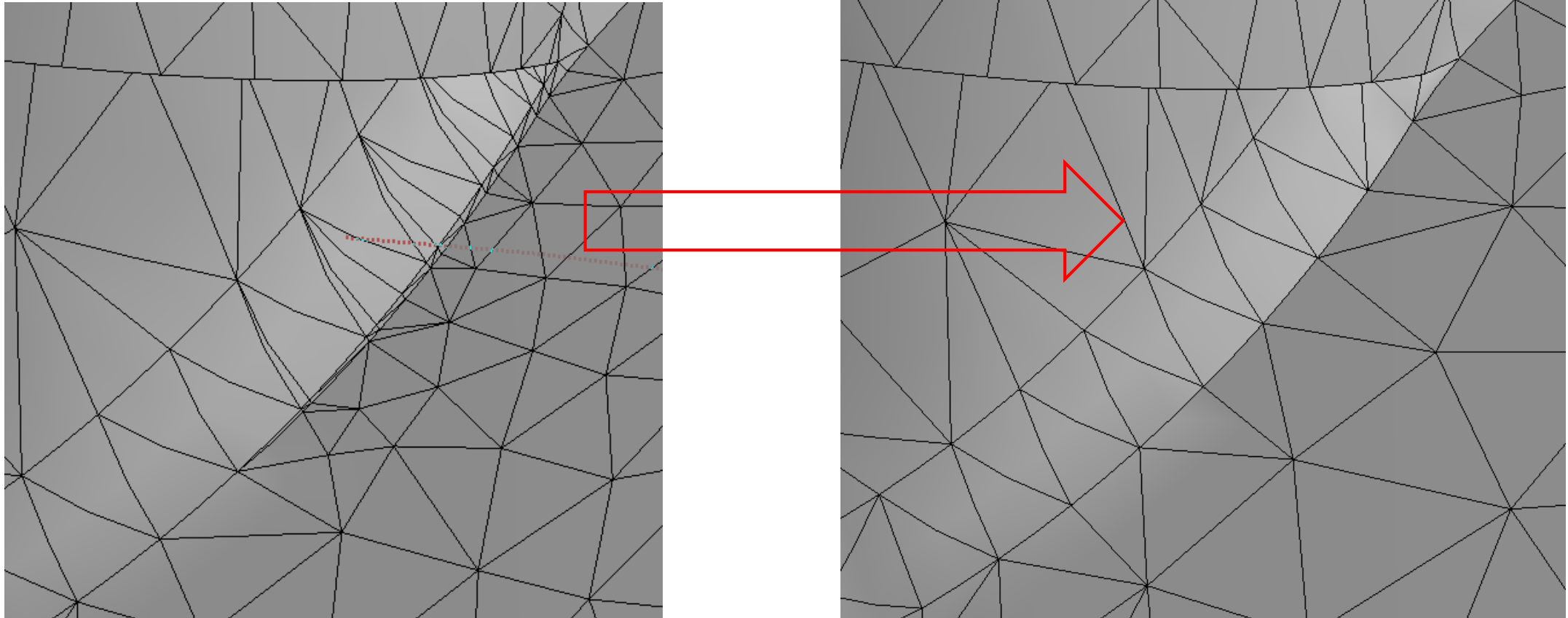
Scope	
Master Geometry	2 Edges
Slave Geometry	2 Edges
Definition	
Suppressed	No
<input type="checkbox"/> Tolerance	2. mm
Scope Method	Manual

Pinch

ANSYS

# Geometry Correction: Pinch

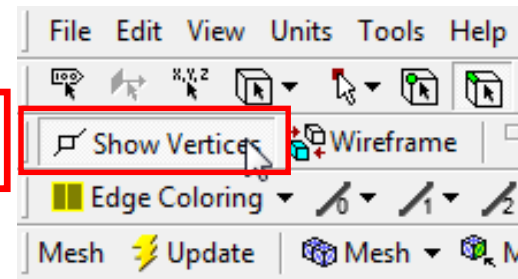
After generating the mesh, review the results in the region of the Pinch control and compare them to the original mesh in the same region.



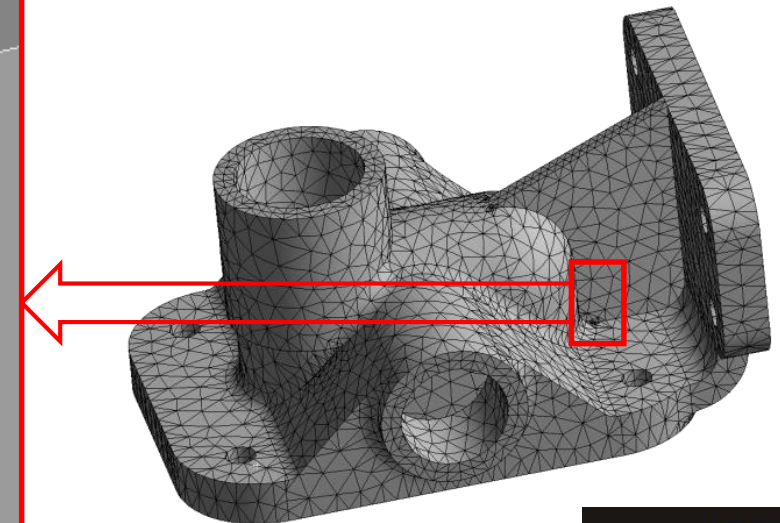
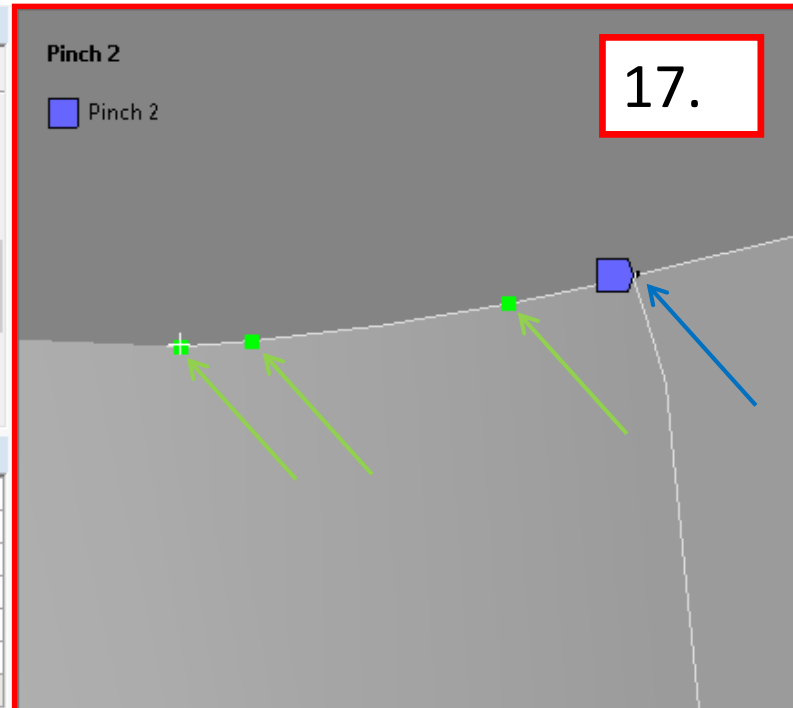
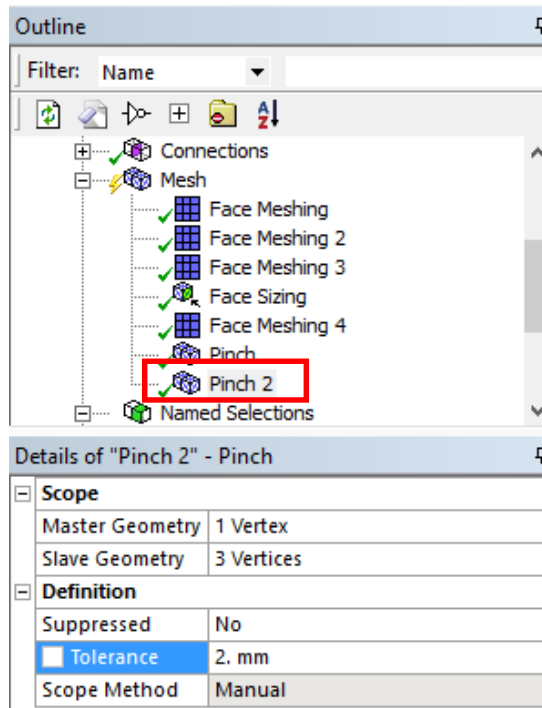
# Geometry Correction: Pinch

16. Clear the mesh (RMB > Clear Generated Data) and activate Show Vertices.

16.



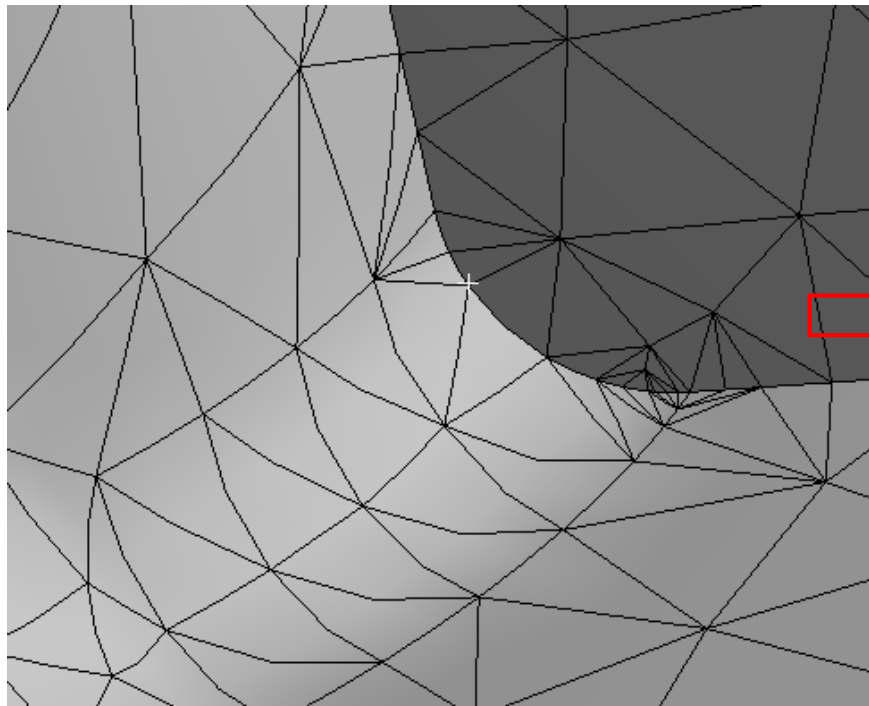
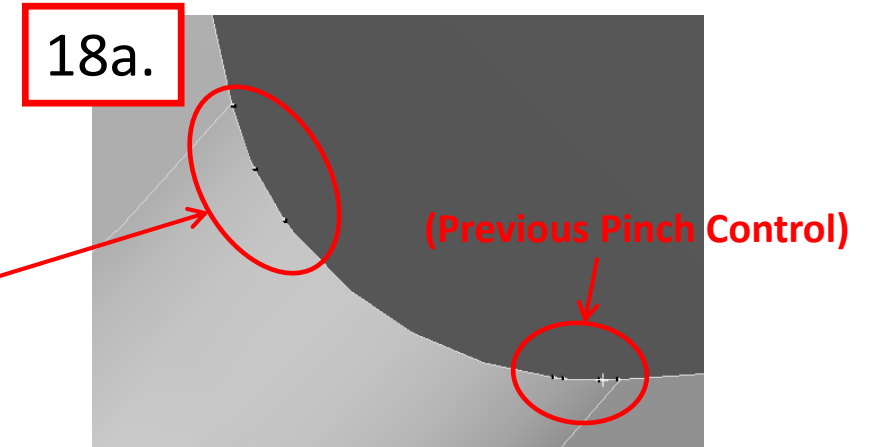
17. Insert another Pinch control. Select the vertex indicated by the blue flag as the Master Geometry and the three vertices highlighted in green as the Slave Geometry. Set the Tolerance to 2 mm and generate the mesh.



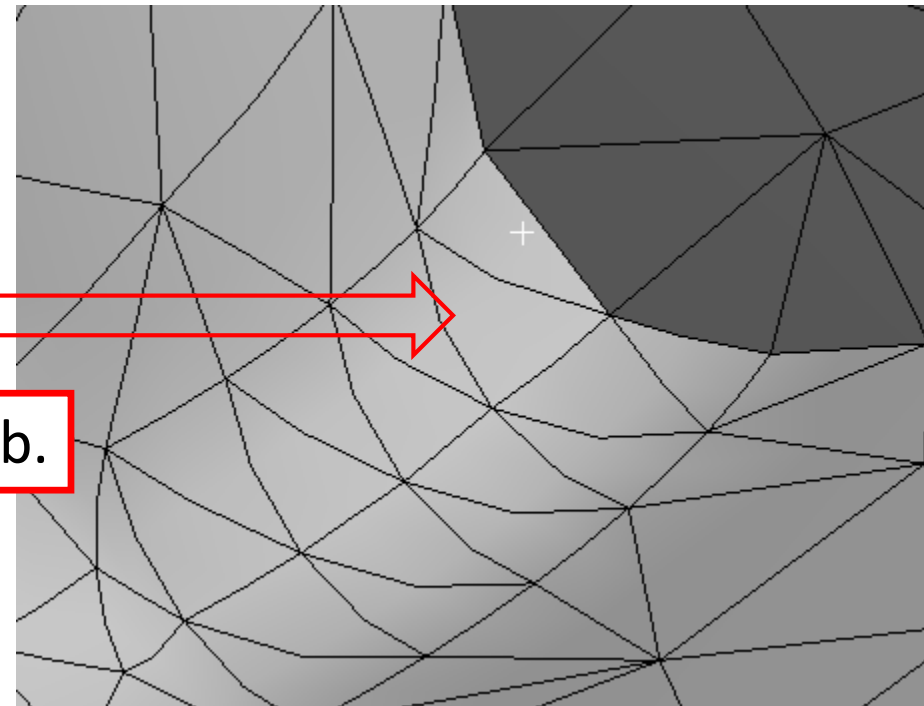
# Geometry Correction: Pinch

## 18. Create another Pinch control:

- Scope the Pinch control to the vertices shown. Adjust the tolerance value if necessary.
- Compare the mesh results in this region.



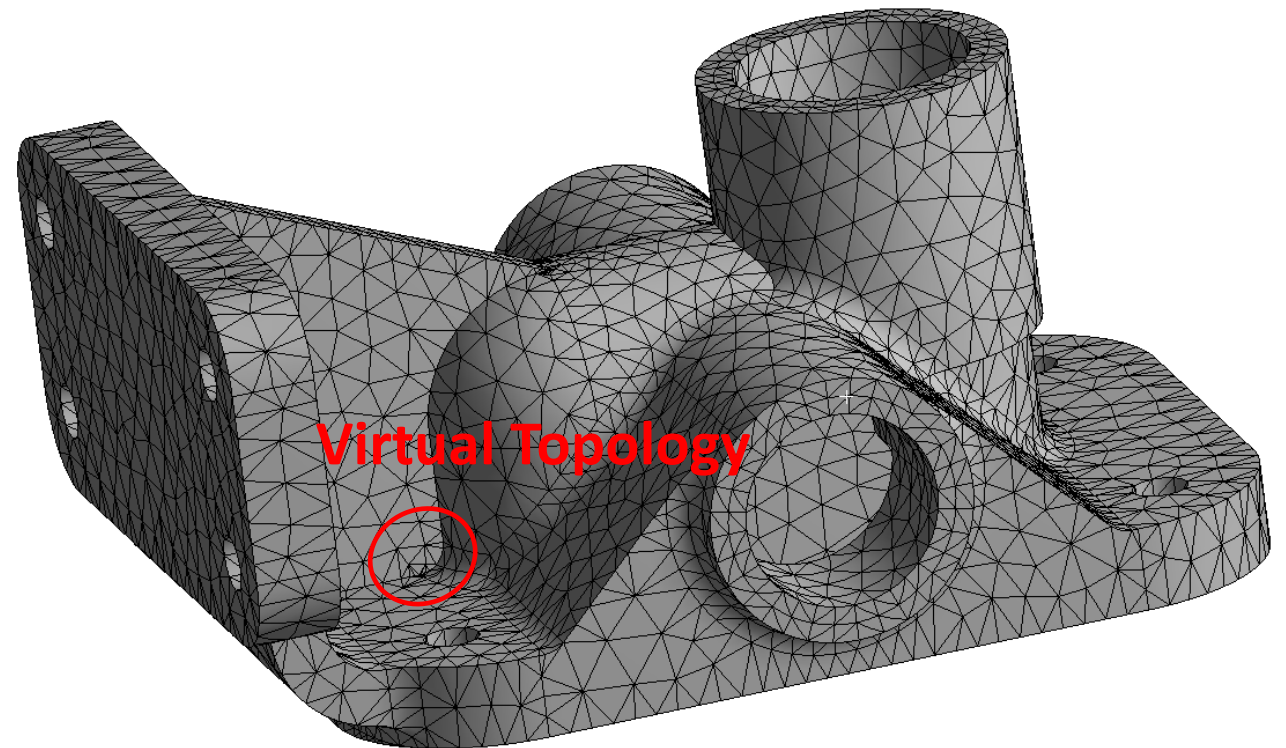
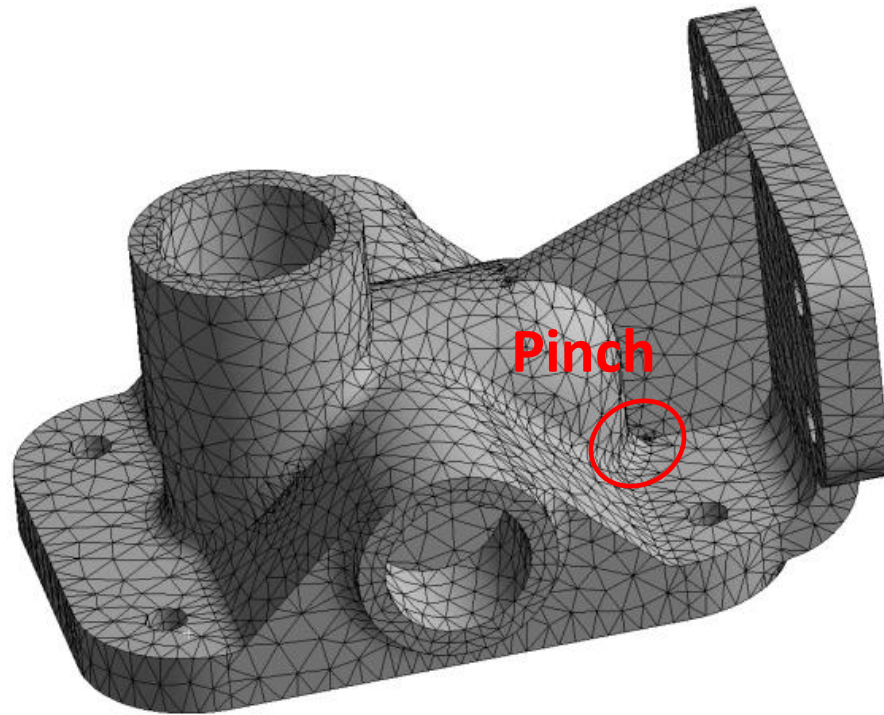
18b.





# 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:



# Geometry Correction: Virtual Topology

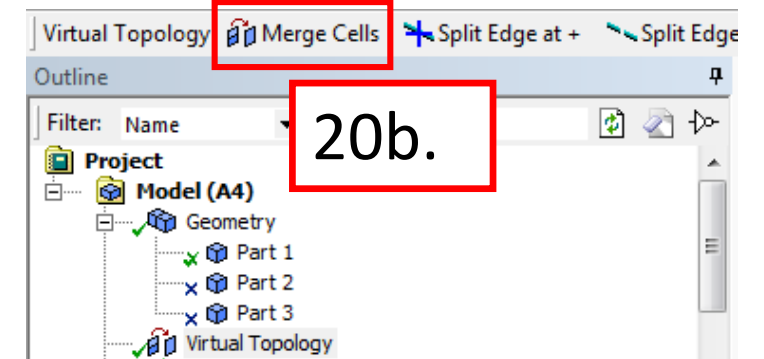
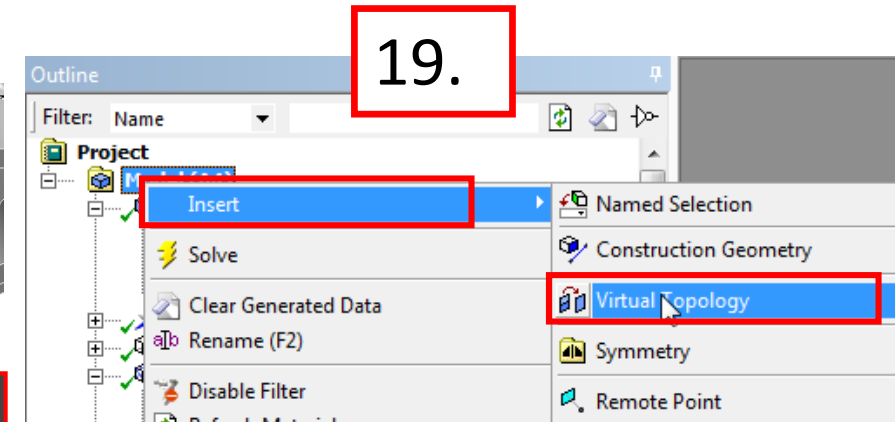
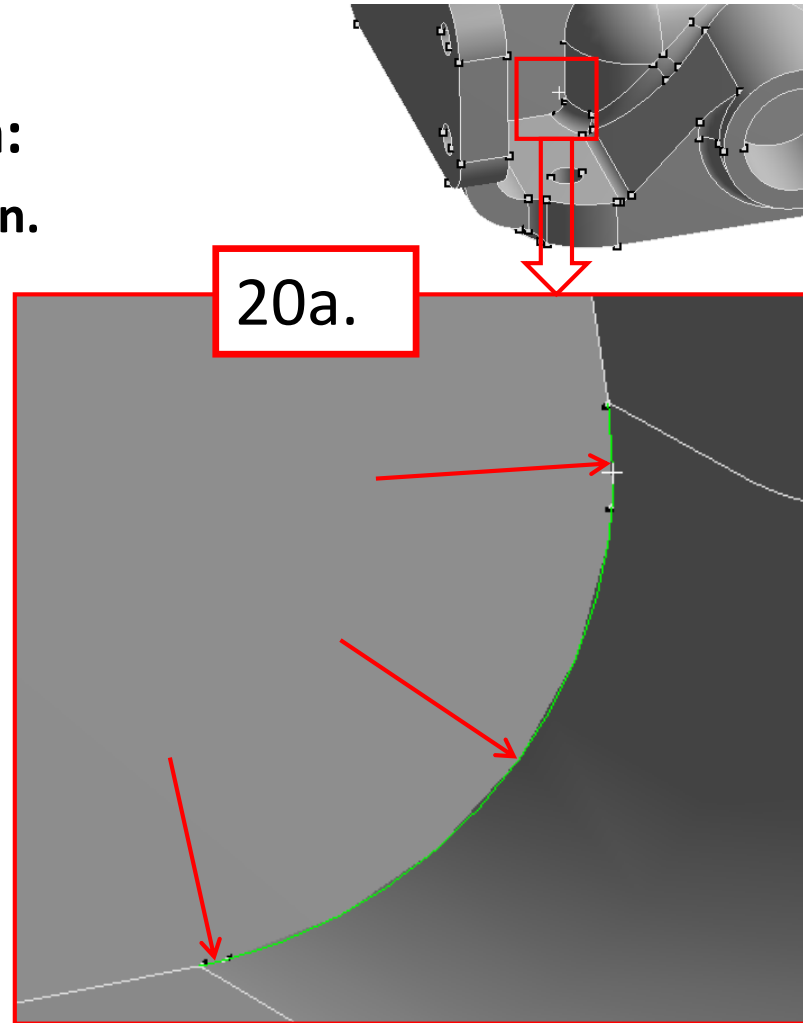
19. From the Model branch, RMB > Insert > Virtual Topology.

20. Complete the definition:

- Select the 3 edges shown.
- Click on Merge Cells.

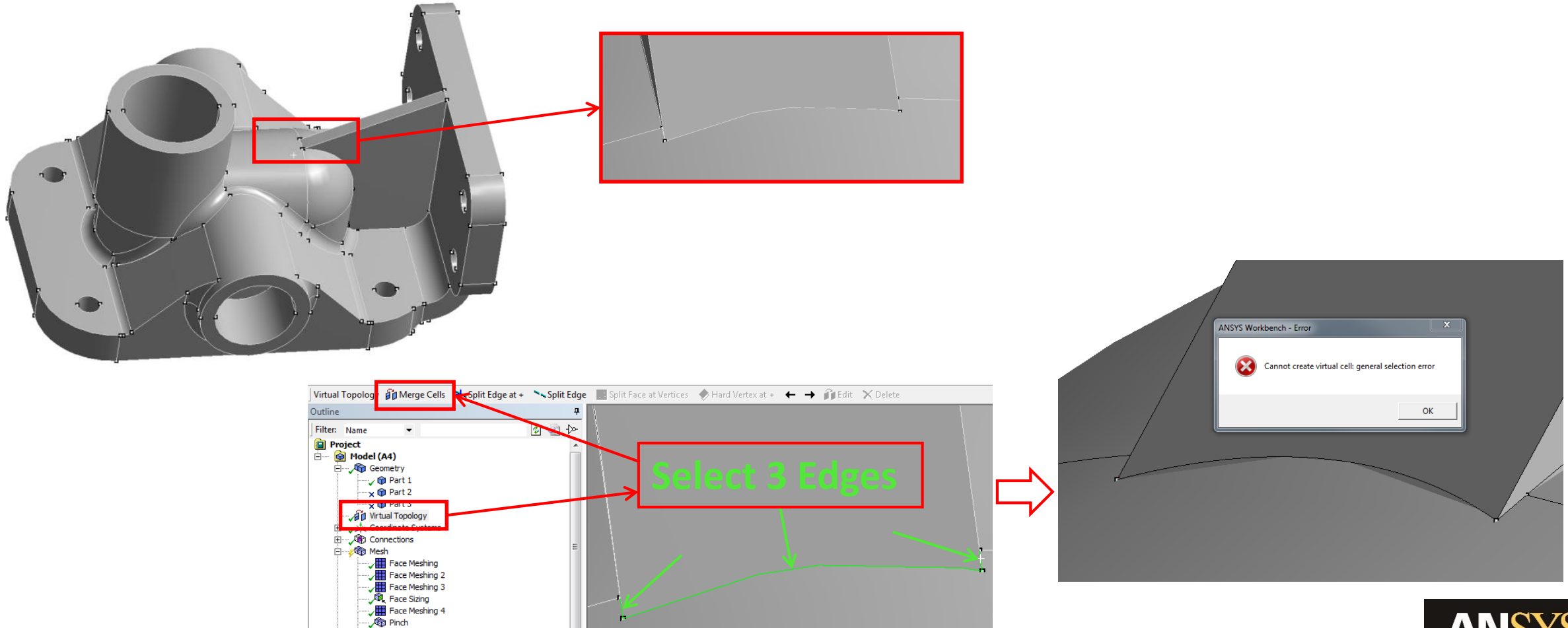
Generate the mesh and compare.

Details of "Virtual Topology"	
Definition	
Method	Automatic
Behavior	Low
Advanced	
Generate on Update	No
Simplify Faces	No
Merge Face Edges	Yes
Lock Position of Dependent Edge Splits	Yes
Statistics	
Virtual Faces	0
Virtual Edges	1
Virtual Split Edges	0
Virtual Split Faces	0
Virtual Hard Vertices	0
Total Virtual Entities	1



# Geometry Correction: Virtual Topology versus Pinch

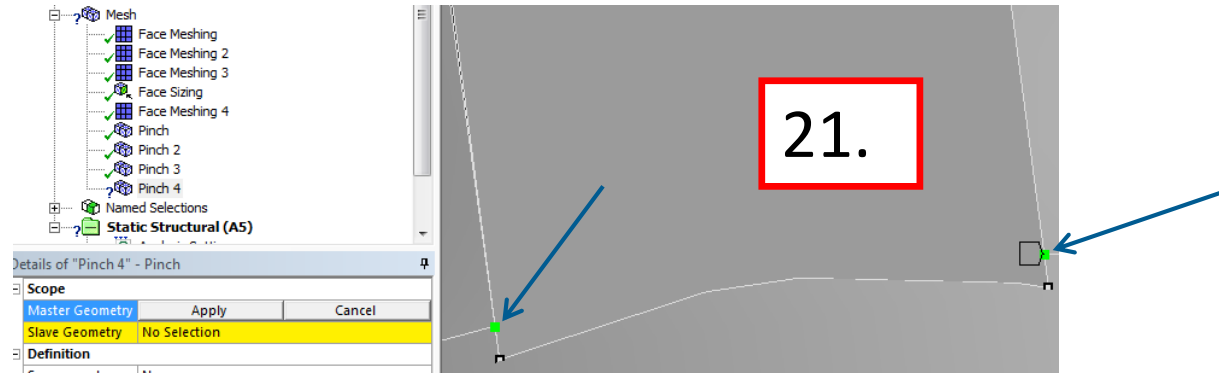
Pinch and Virtual Topology are often complementary tools. If one cannot be applied successfully in a particular application, the other may work better. An example:



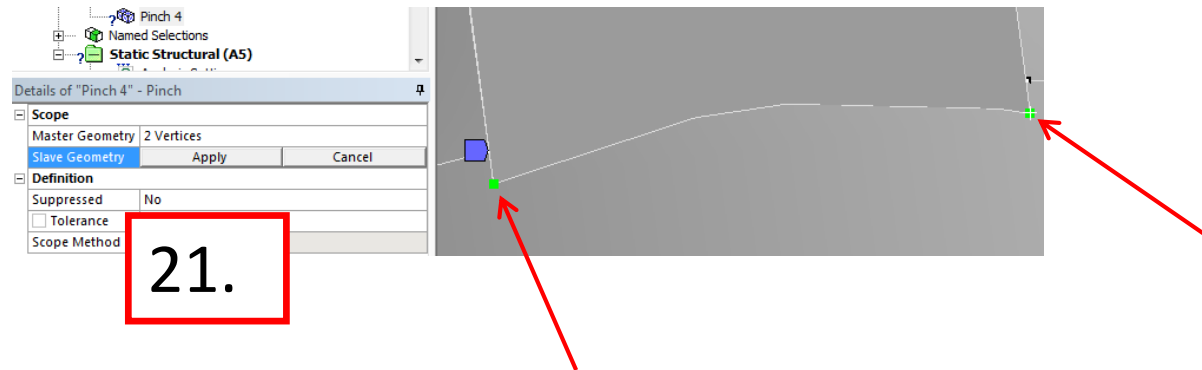
# Geometry Correction: Virtual Topology versus Pinch

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:

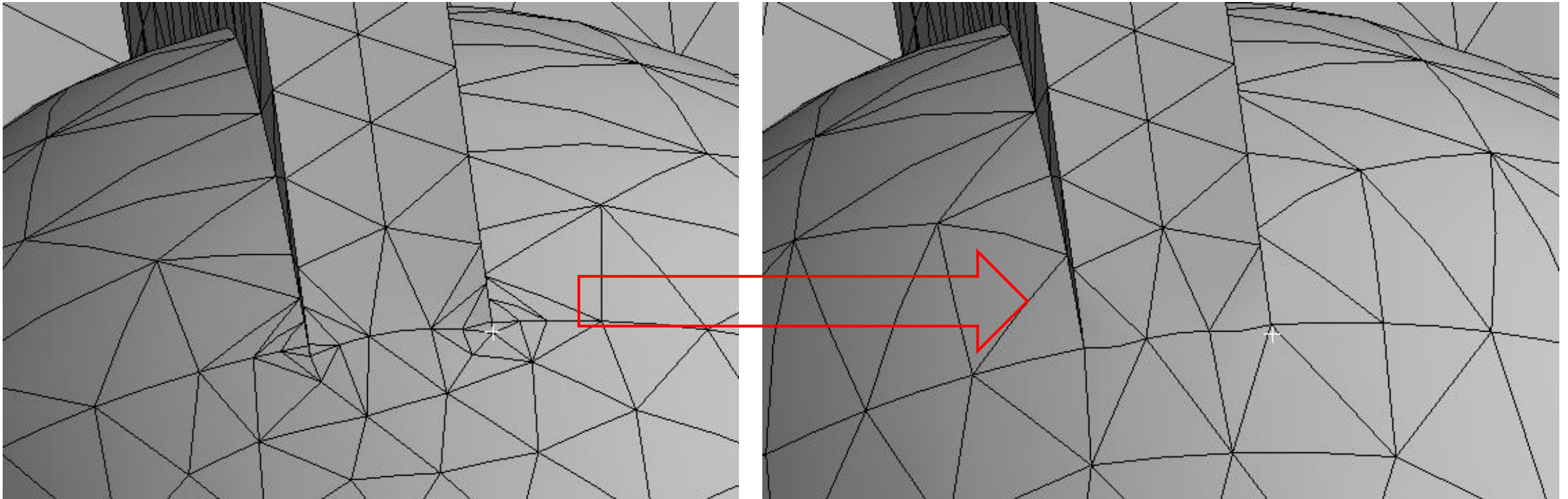


Use a Tolerance of 1 mm.



# Geometry Correction: Virtual Topology versus Pinch

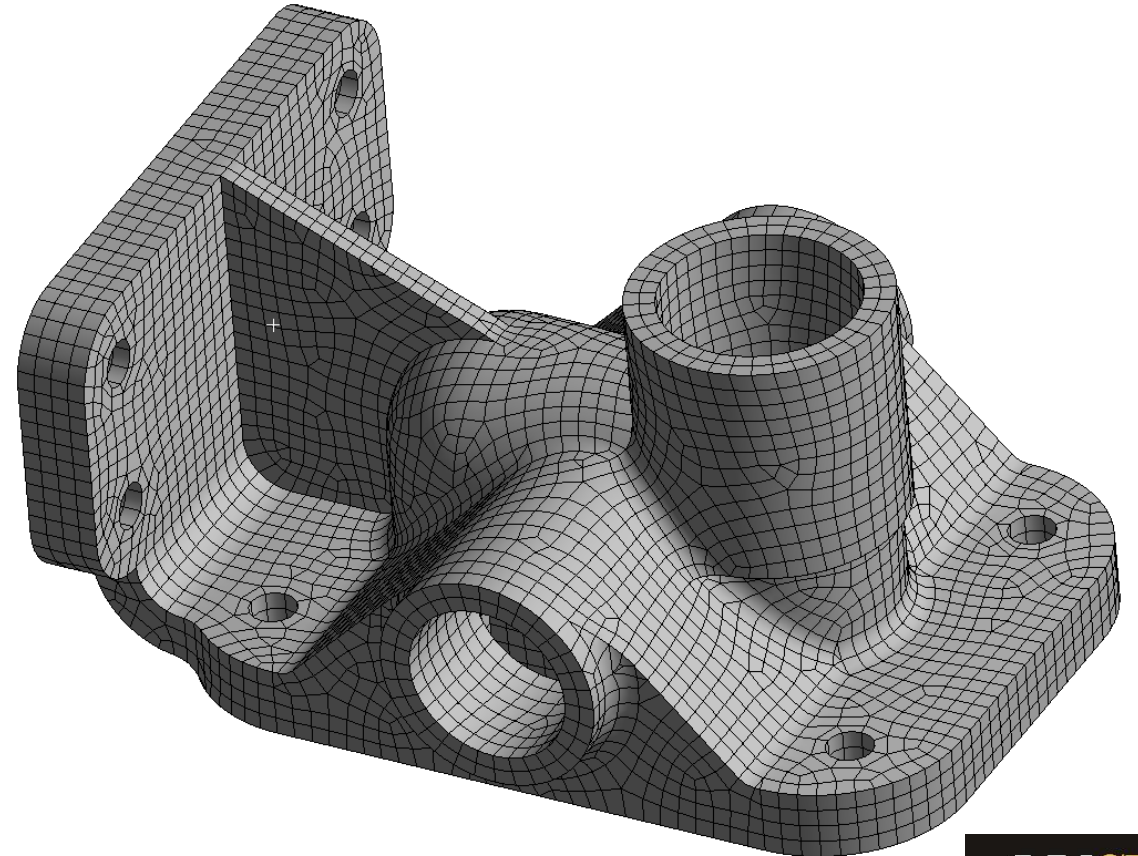
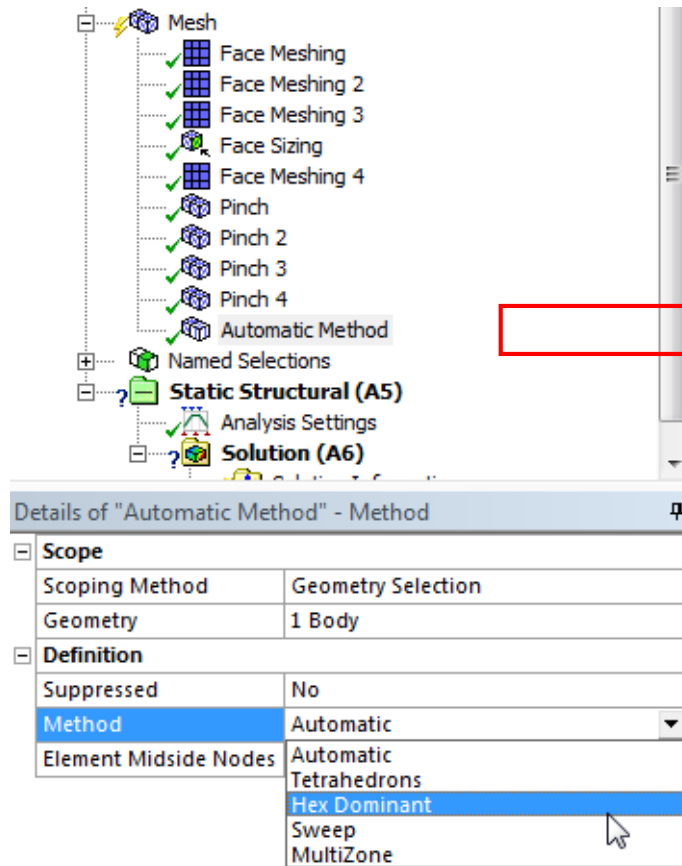
Generated the mesh and compare:



# Go Further!

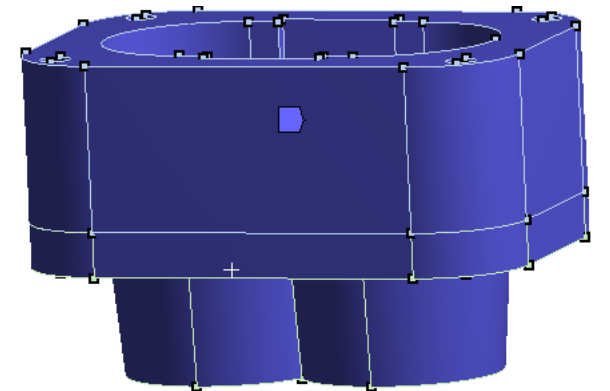
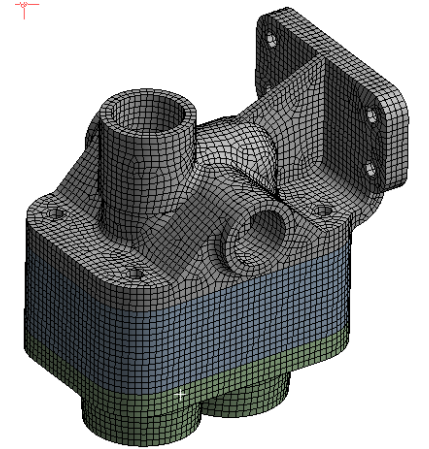
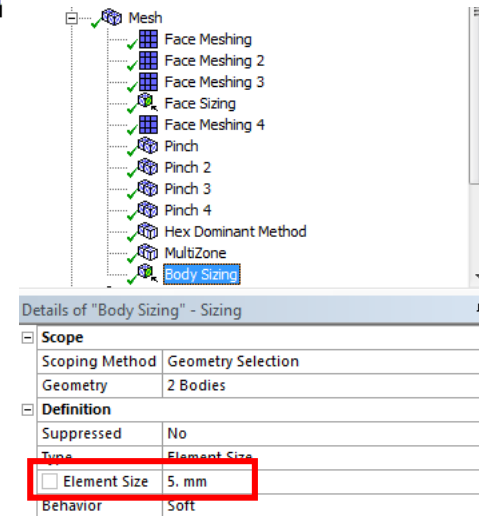
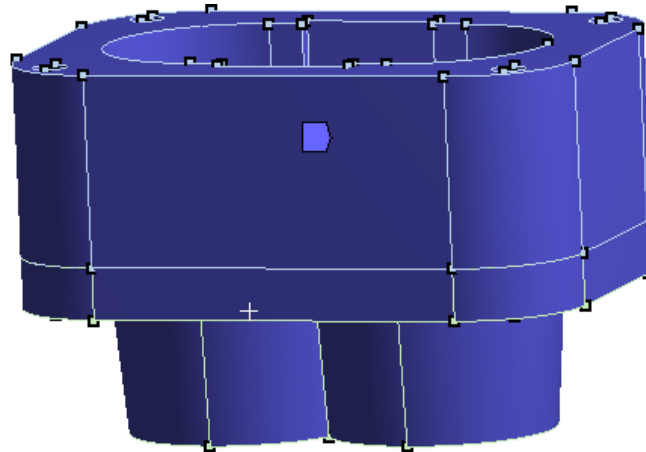
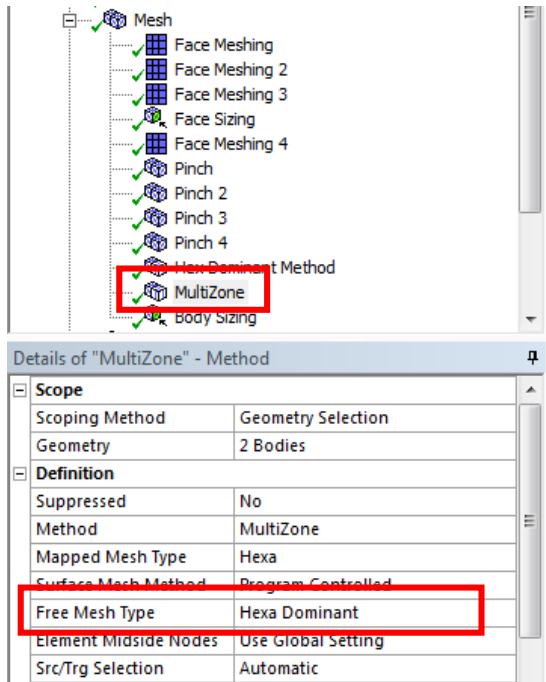
In this workshop we focused on sizing and cleaning operations. If you find yourself with extra time, try the following:

## 1. Test the Hex Dominant method on part “1:”



# Go Further!

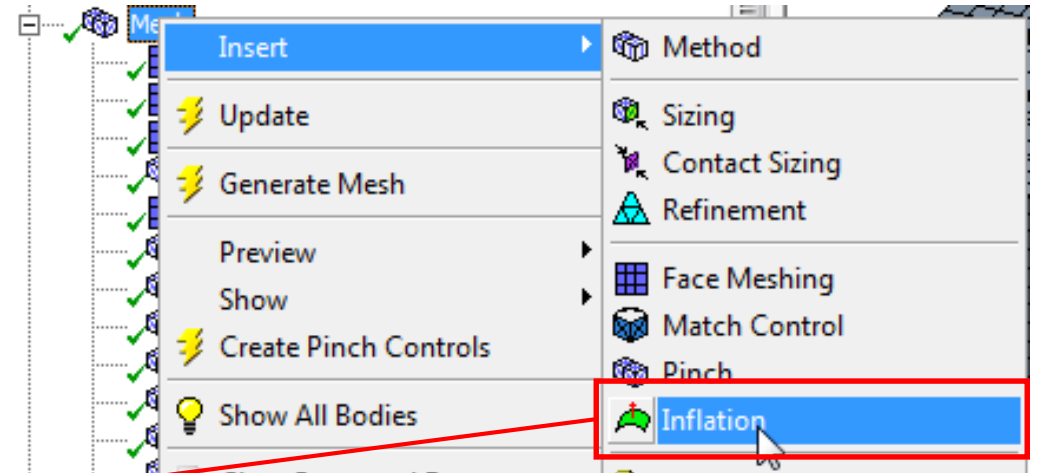
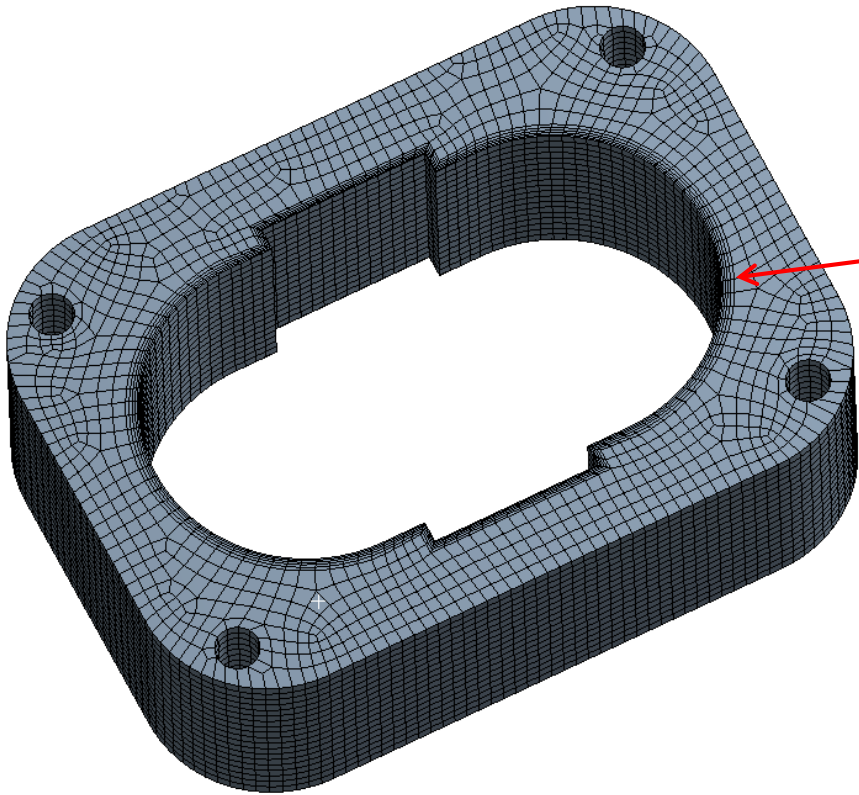
2. Unsuppress part “2” and part “3.” Insert a Sizing control with Element Size = 5 mm scoped to both bodies, and test the Multizone method with Free Mesh Type = Hexa Dominant on both bodies.





# Go Further!

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







## Workshop 05.2: Mesh Control

**END**

