

# Workshop 4.2 : Mesh Control

16.0 Release



Fluid Dynamics

Structural Mechanics

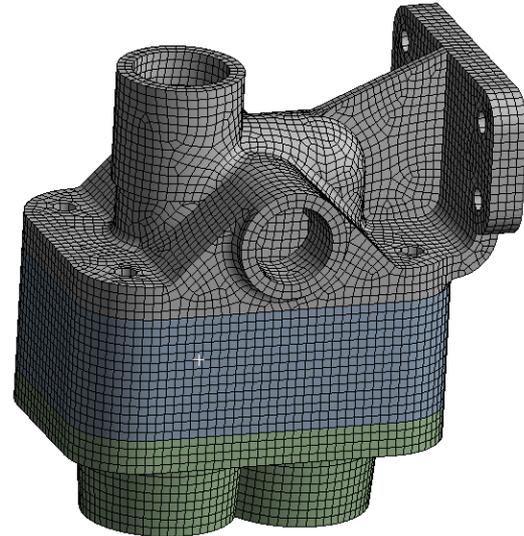
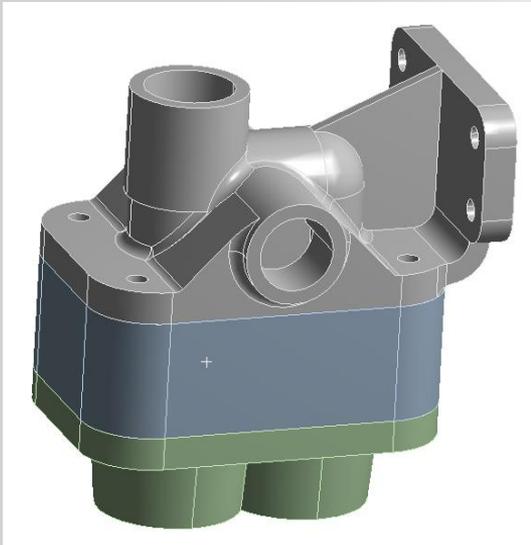
Electromagnetics

Systems and Multiphysics

## Introduction to ANSYS Mechanical

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

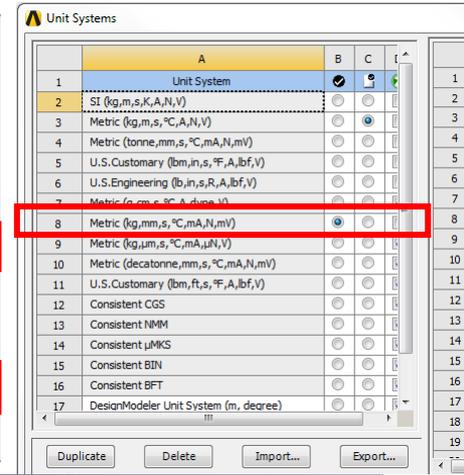
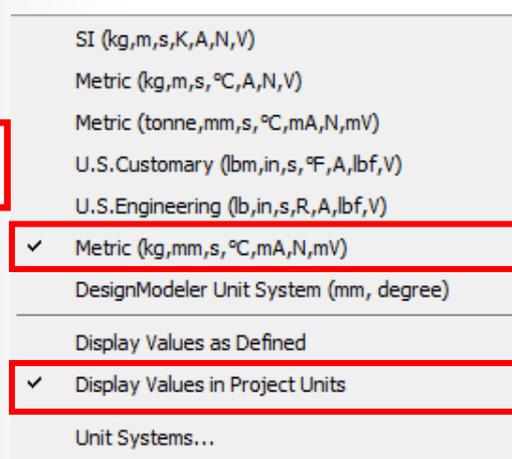
- Our goal is to use meshing controls in order to deal with some geometry specification defects.



## 1. From the “Units” menu verify:

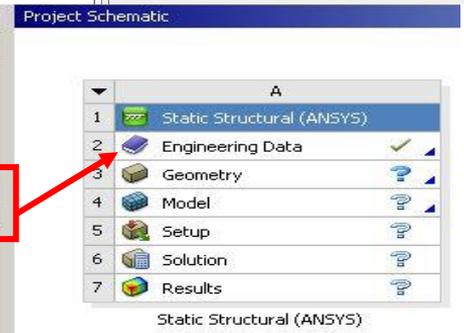
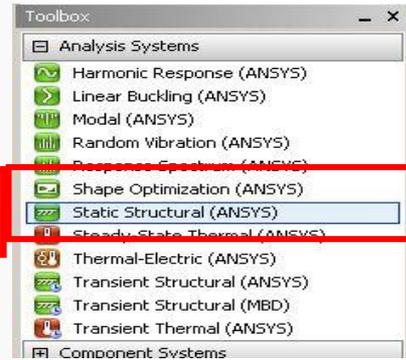
- Project units are set to “Metric (kg, mm, s, °C, mA, N, mV)”. If not available in the menu, click on “Unit Systems...” and set them for the active project.
- “Display Values in Project Units” is checked (on).

1.



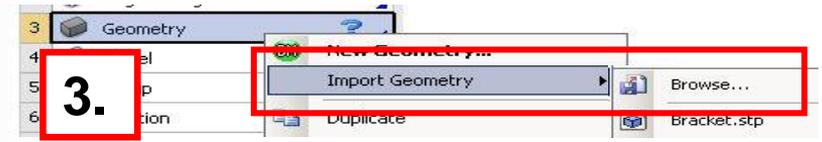
## 2. From the Toolbox double click “Static Structural” to create a new system.

2.

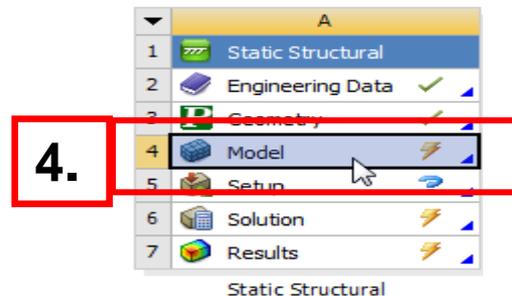


## 3. RMB the geometry cell and “Import Geometry” and browse to “assembly\_solid.stp”.

3.



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



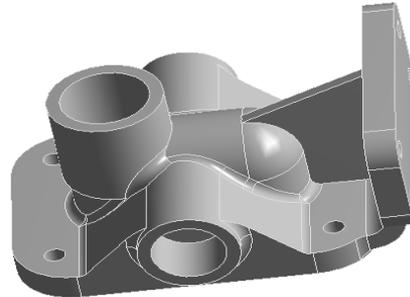
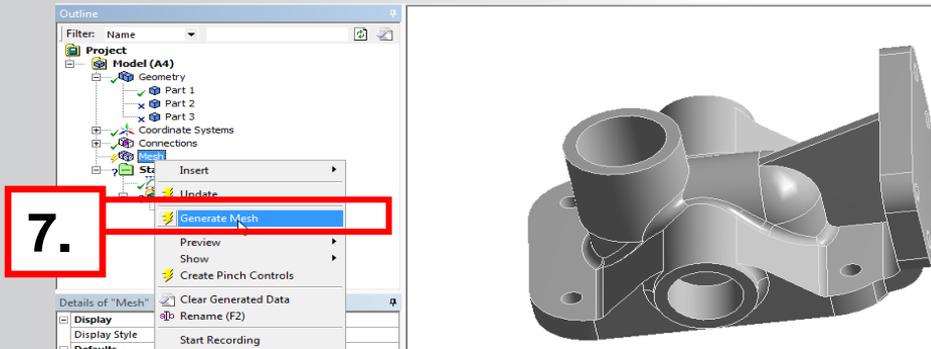
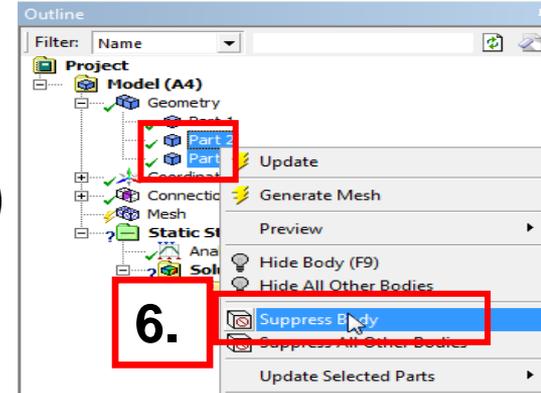
5. Set/verify the working unit system:
  - “Units > Metric (mm, kg, N, s, mV, mA)”.



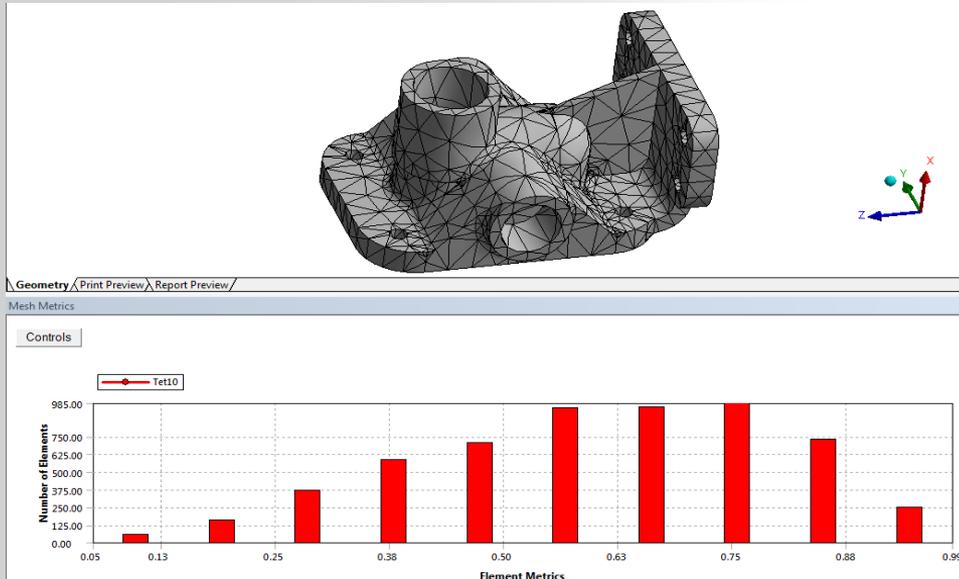
6. Select Part 2 and Part 3 in the Geometry details.

RMB and suppress body. (we'll focus on Part 1 in a 1<sup>st</sup> 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.



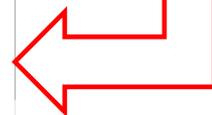
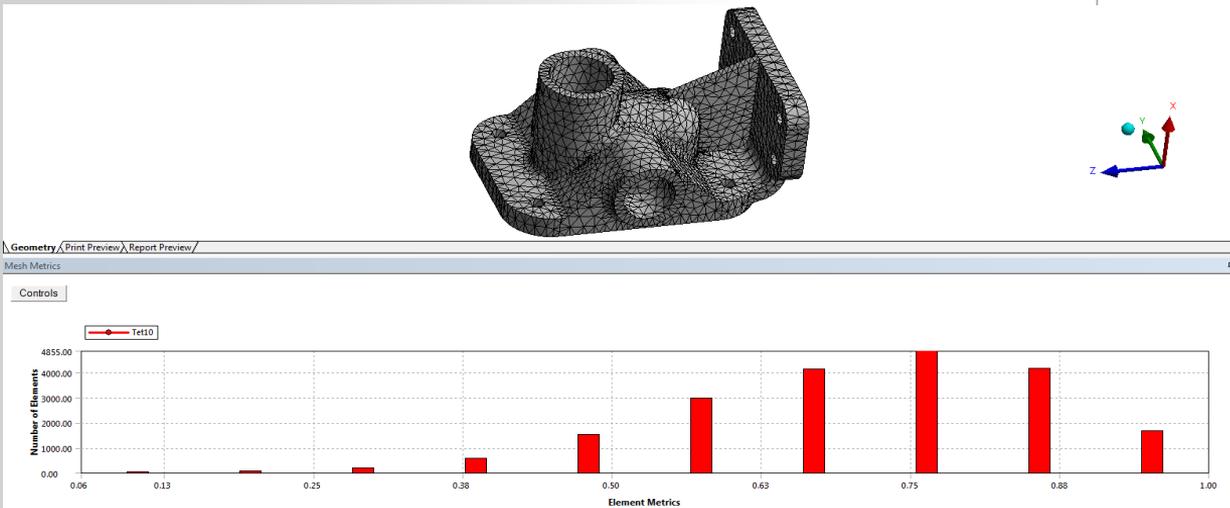
The screenshot shows the ANSYS software interface. The 'Outline' panel on the left shows the project hierarchy with 'Mesh' selected. The 'Details of Mesh' panel on the right shows various settings. A red box highlights the 'Statistics' section, and another red box highlights the 'Mesh Metric' dropdown menu, which is set to 'Element Quality'. A large red '8.' is placed next to the 'Mesh Metric' dropdown.

Property	Value
Nodes	10291
Elements	5727
Mesh Metric	Element Quality
Min	5.0794e-002
Max	0.99187
Average	0.60508
Standard Deviation	0.19457



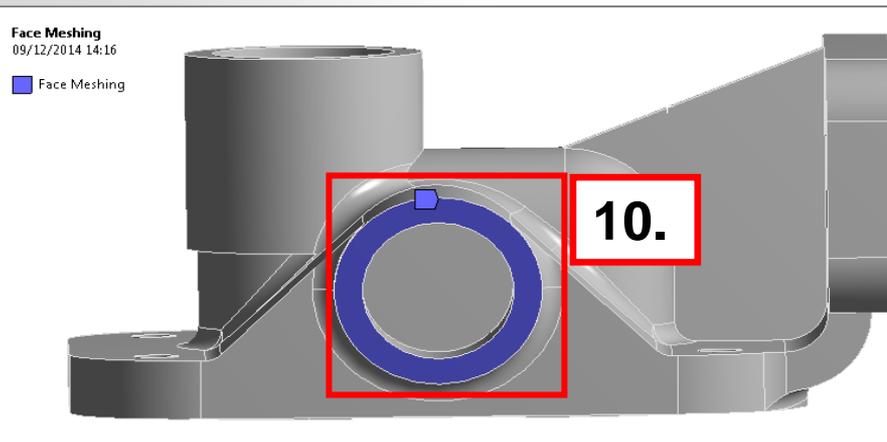
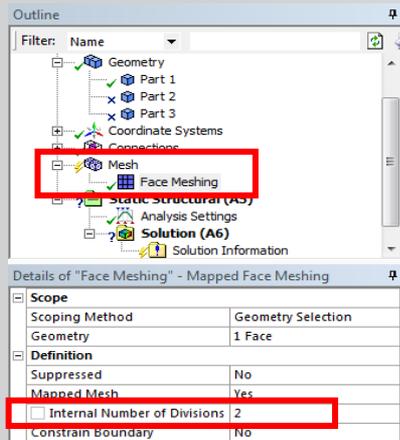
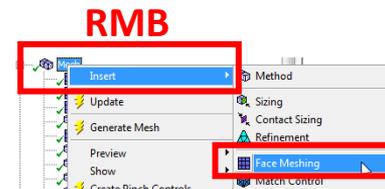
Details of "Mesh"	
[-] Display	
Display Style	Body Color
[-] Defaults	
Physics Preference	Mechanical
<input type="checkbox"/> Relevance	0
[-] Sizing	
Use Advanced Size Function	Off
Relevance Center	Coarse
<input checked="" type="checkbox"/> Element Size	4.0 mm
Initial Size Seed	Assembly
Smoothing	
Transition	

9.



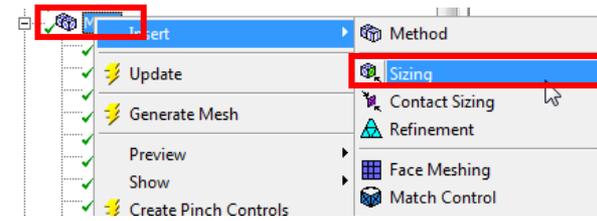
10. Insert a Face Meshing at the cylindrical surface shown below

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.

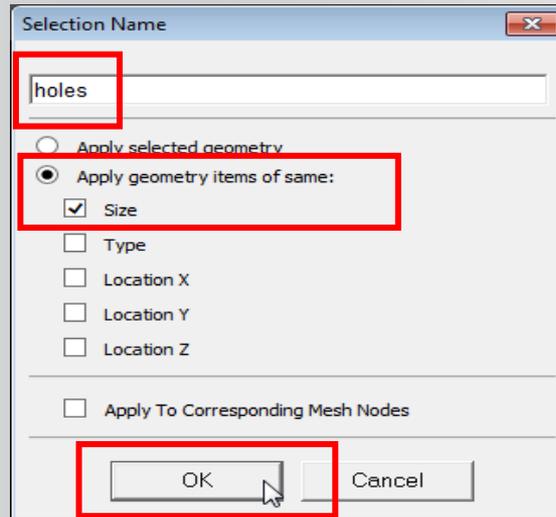
The screenshot shows the ANSYS software interface with the 'Face Sizing' details panel on the left and a 3D model of a mechanical part on the right. The 'Face Sizing' details panel shows the following information:

Details of "Face Sizing" - Sizing	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Suppressed	No
Type	Element Size
Element Size	3. mm
Behavior	Soft

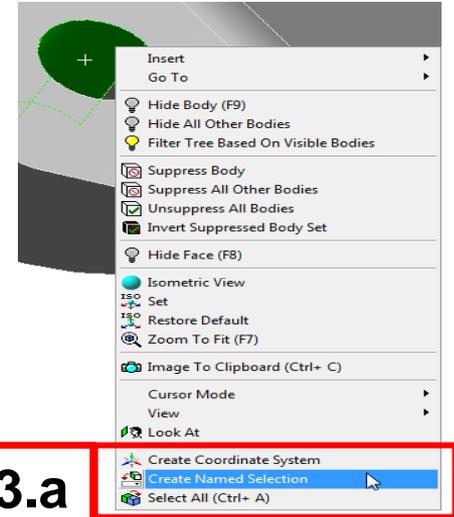
The 3D model shows a mechanical part with a face highlighted in blue. A red box labeled '12.' is placed over the highlighted face, indicating the location where the face sizing should be applied.

13. a. Select one of the 2 surfaces in the 8 holes (half of the hole) and RMB create named selection.

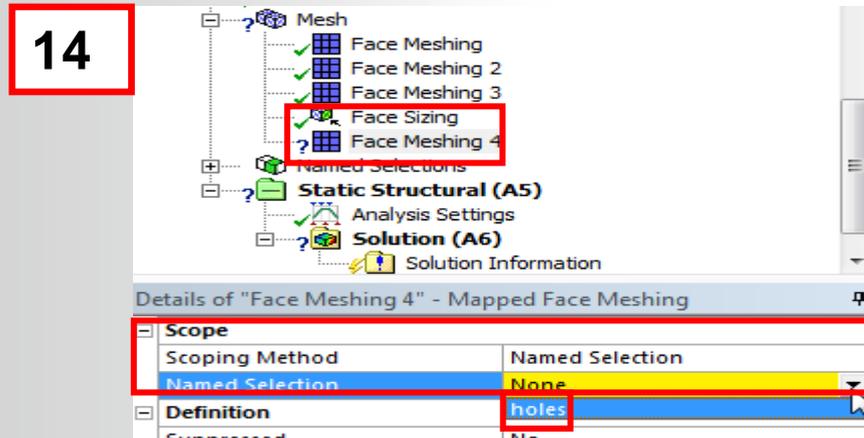
13. b. In the named selection details, change the name to 'holes' and choose apply geometry items of same size.



13.b



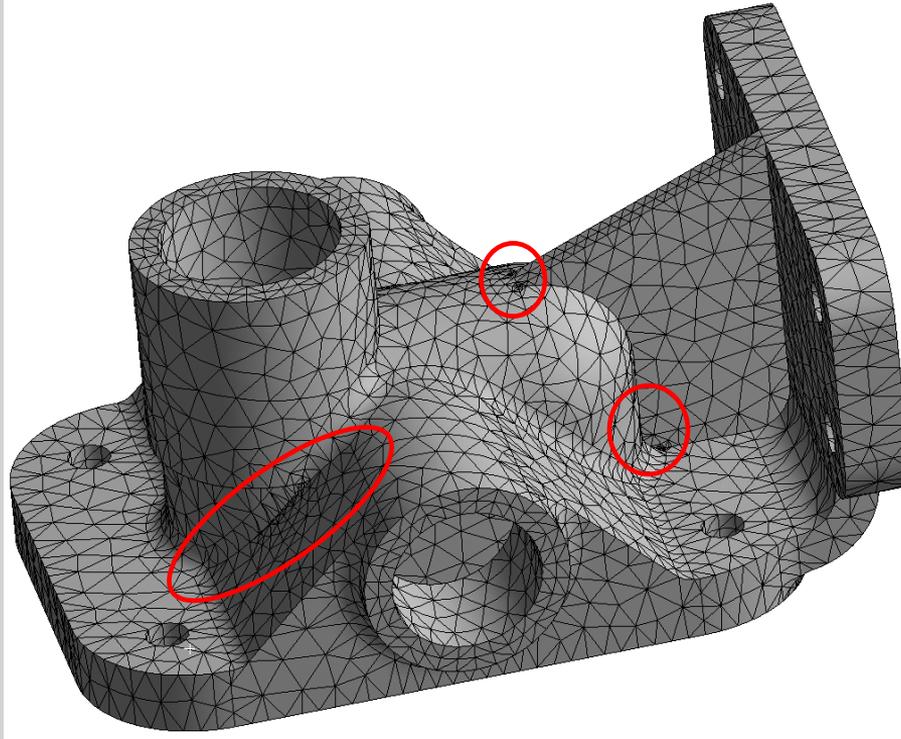
14. Insert a Face Meshing control and change the Scoping Method to Named Selection. In the Named Selection select holes.



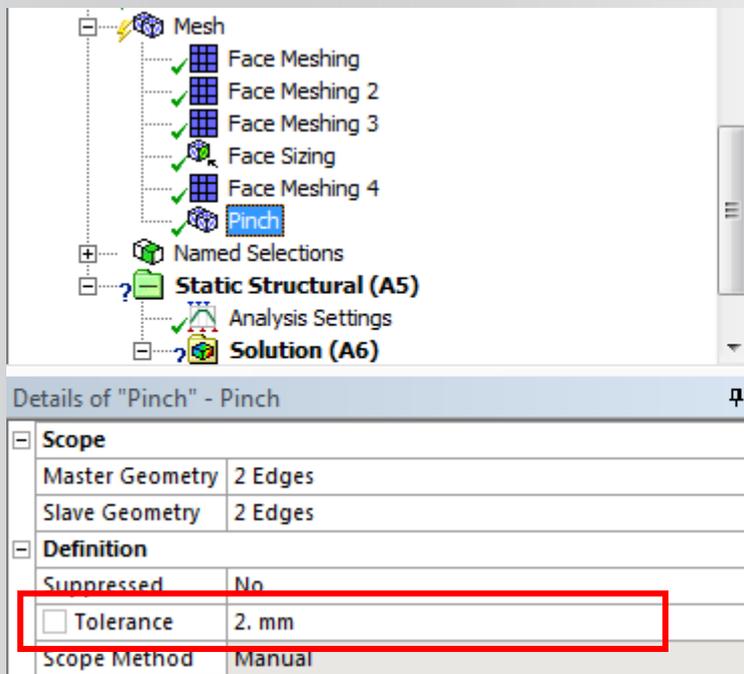
- After re-generating the mesh, you can see that the number of distorted elements has been reduced by checking the mesh metric.

# Geometry correction technics

- We will now focus on dealing with some geometric specifications like the ones shown here:

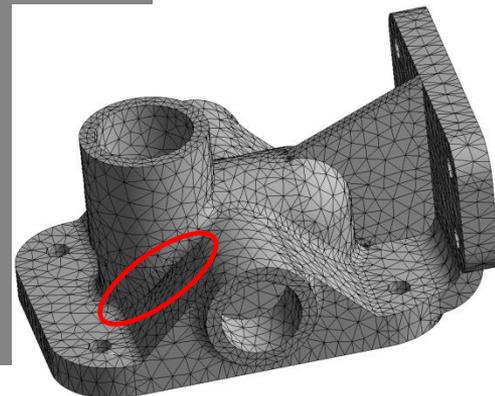
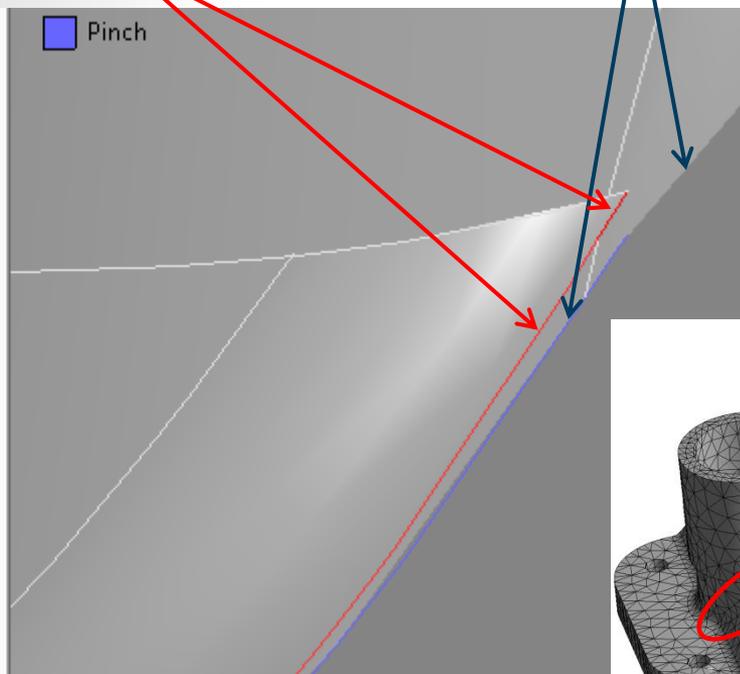


15. Insert a pinch control and choose as master Geometry the 2 edges in blue. As Slave Geometries, choose the two edges in red. Set the tolerance to 2 mm.

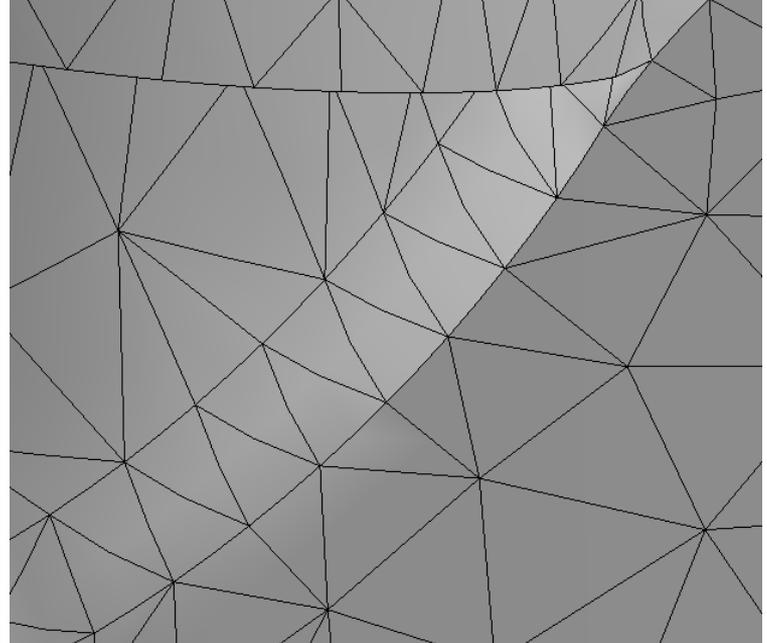
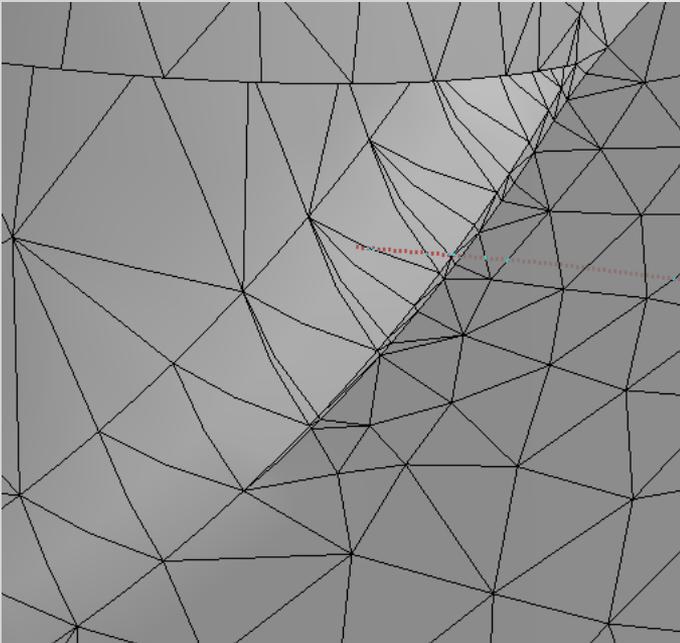


The screenshot shows the ANSYS software interface. On the left, the Mesh tree is expanded to show the 'Pinch' control. Below the tree, the 'Details of "Pinch" - Pinch' panel is visible. The 'Scope' section shows 'Master Geometry' as 2 Edges and 'Slave Geometry' as 2 Edges. The 'Definition' section shows 'Suppressed' as No and 'Tolerance' as 2. mm, which is highlighted with a red box. The 'Scope Method' is set to Manual.

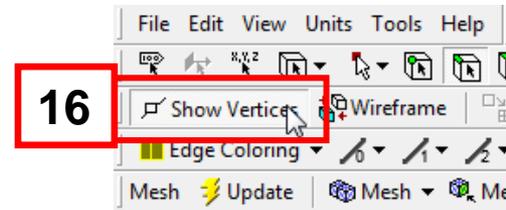
Details of "Pinch" - Pinch	
<b>Scope</b>	
Master Geometry	2 Edges
Slave Geometry	2 Edges
<b>Definition</b>	
Suppressed	No
<input type="checkbox"/> Tolerance	2. mm
Scope Method	Manual



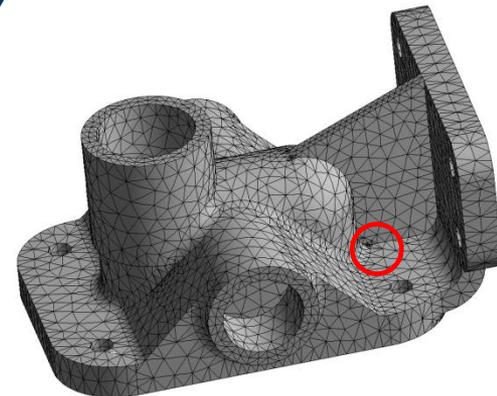
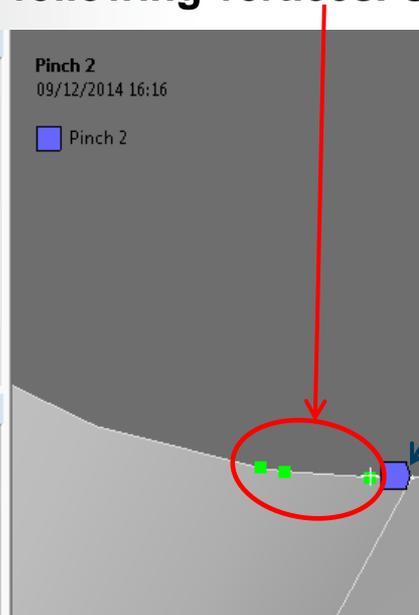
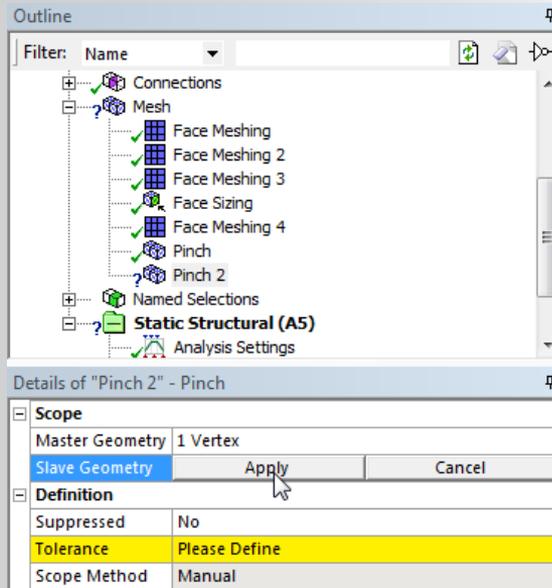
- After generating the mesh check the result and compare it to the original mesh on the corresponding area.



16. Clear generated mesh and Turn the Show vertices on:

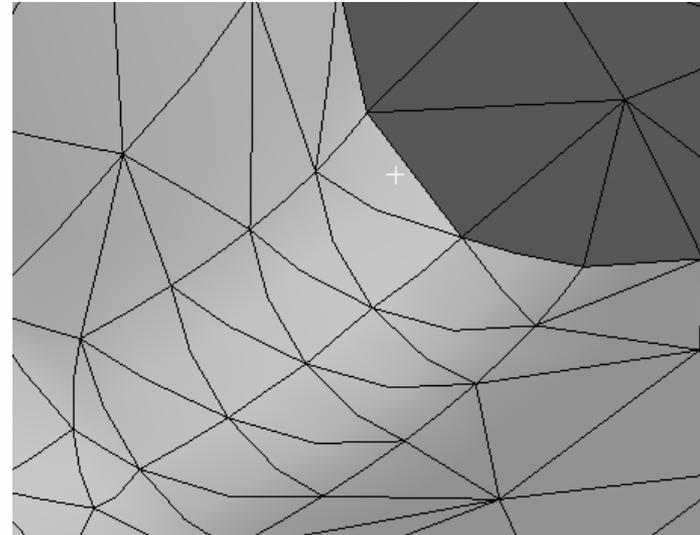
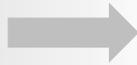
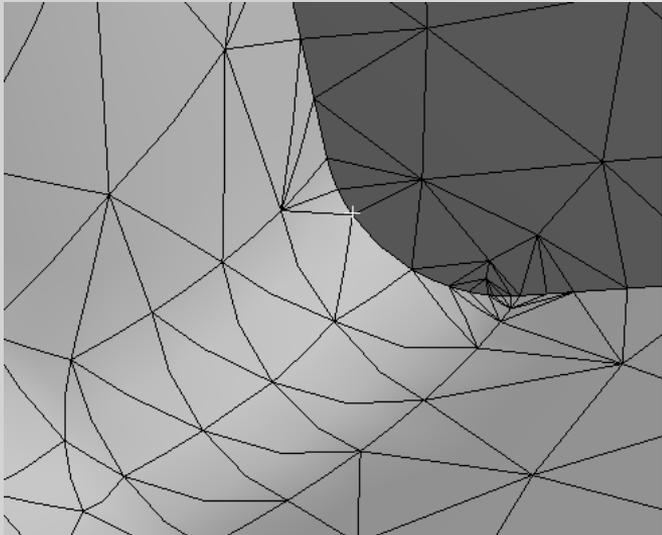
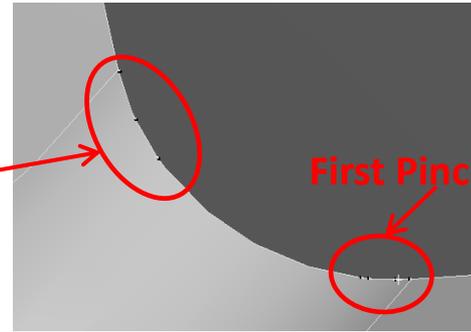


17. Insert another pinch control and choose as master Geometry the vertex in blue. As Slave Geometries, choose the three following vertices. Set the tolerance to 2 mm and Mesh.



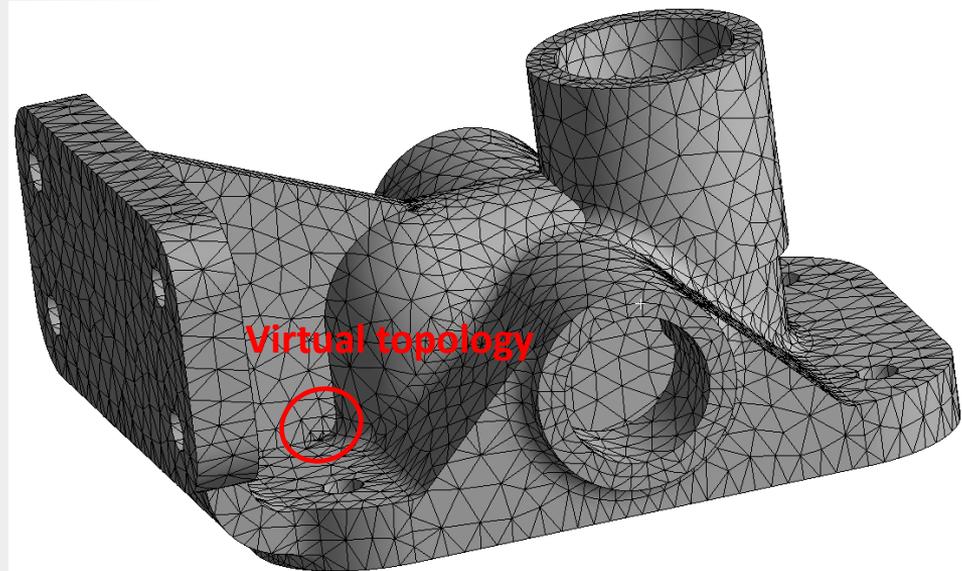
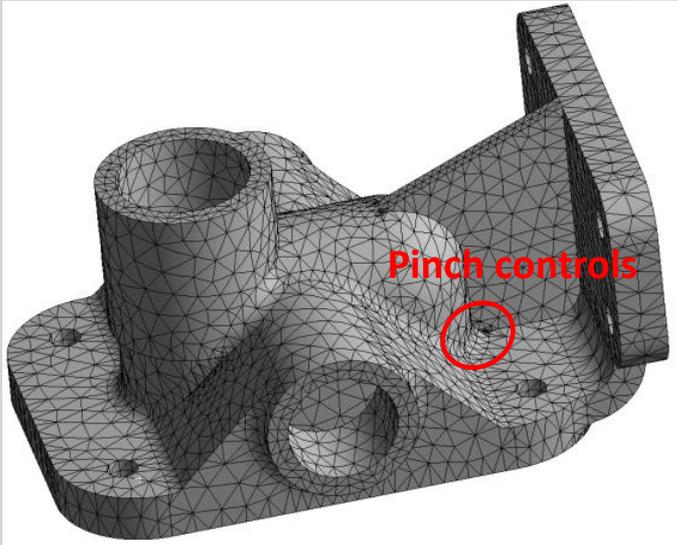
18. Repeat the pinch control at the following points:  
Change the tolerance value if needed.

- 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:



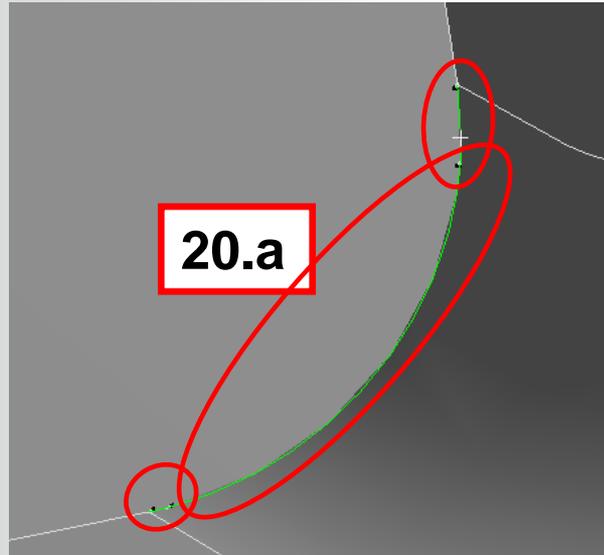
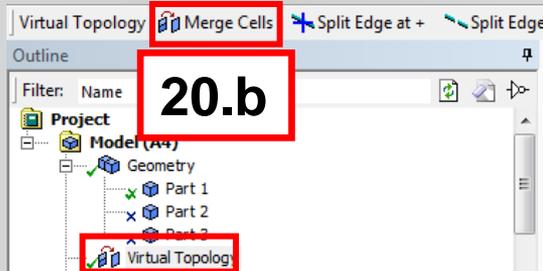
# ... Virtual topologies

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.

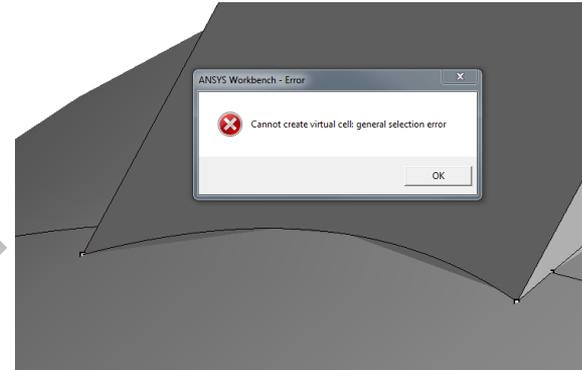
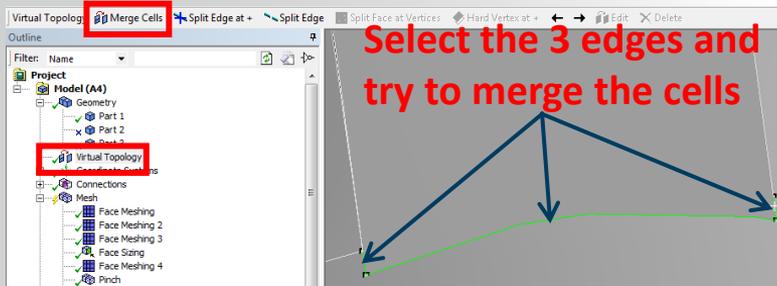
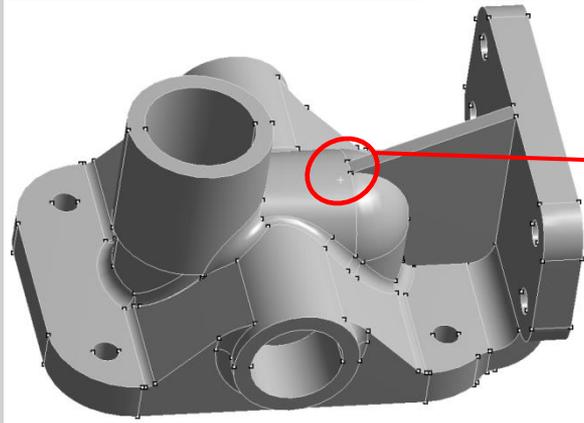
Generate the mesh and Compare.



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

# Virtual topologies vs Pinch

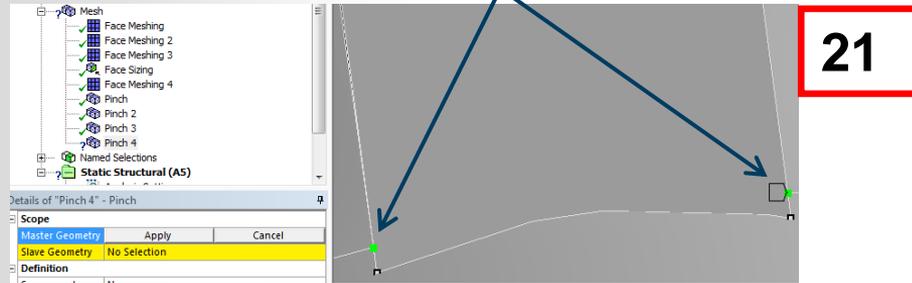
- Pinch and VT are complementary tools. In some cases if one could not be applied, try the other one:



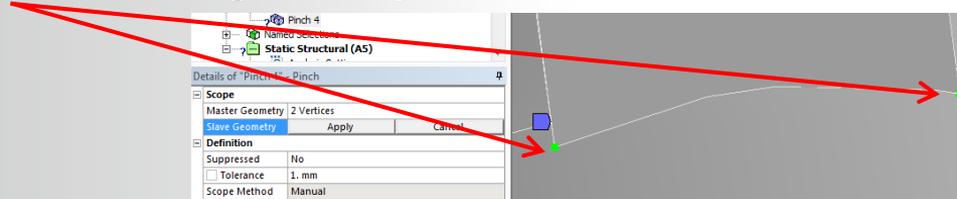
# ...Virtual topologies vs Pinch

- In these cases, the pinch control should be used:

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

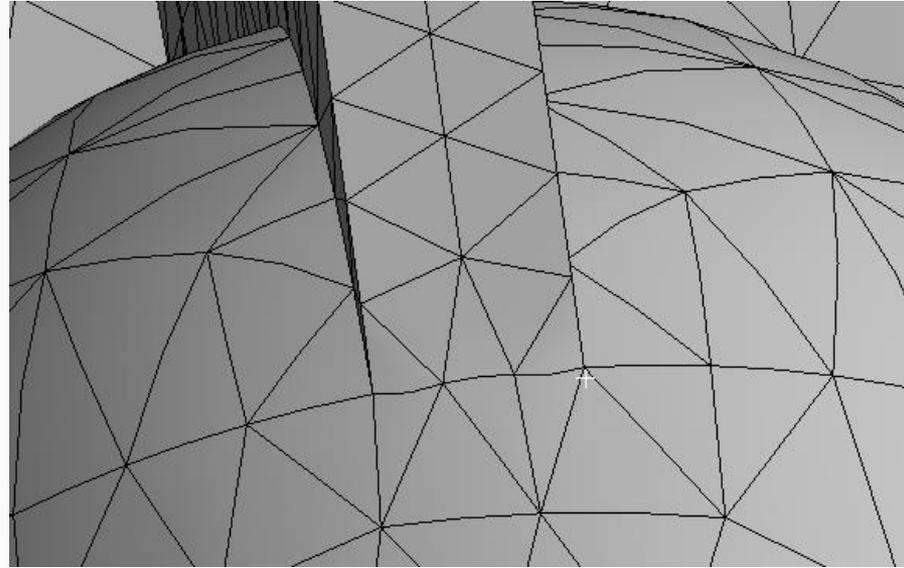
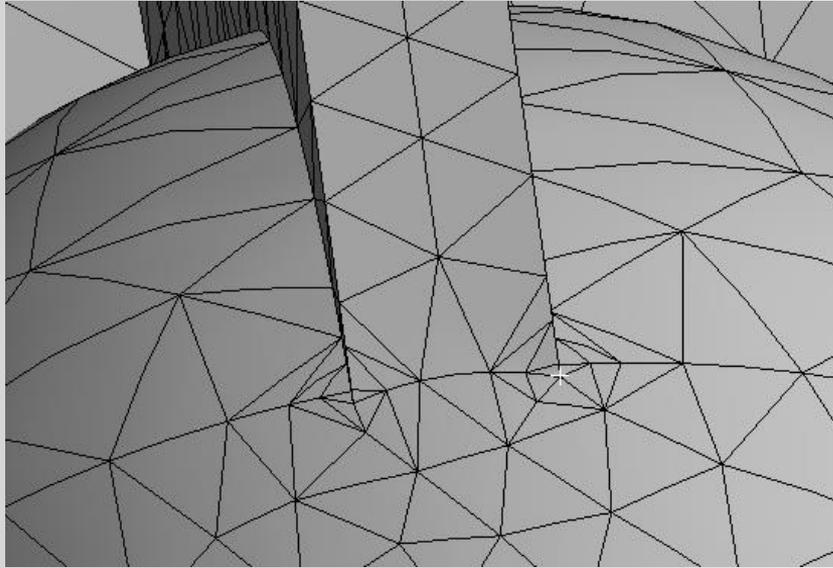


And the other two points as slave geometry with a tolerance of 1 mm.



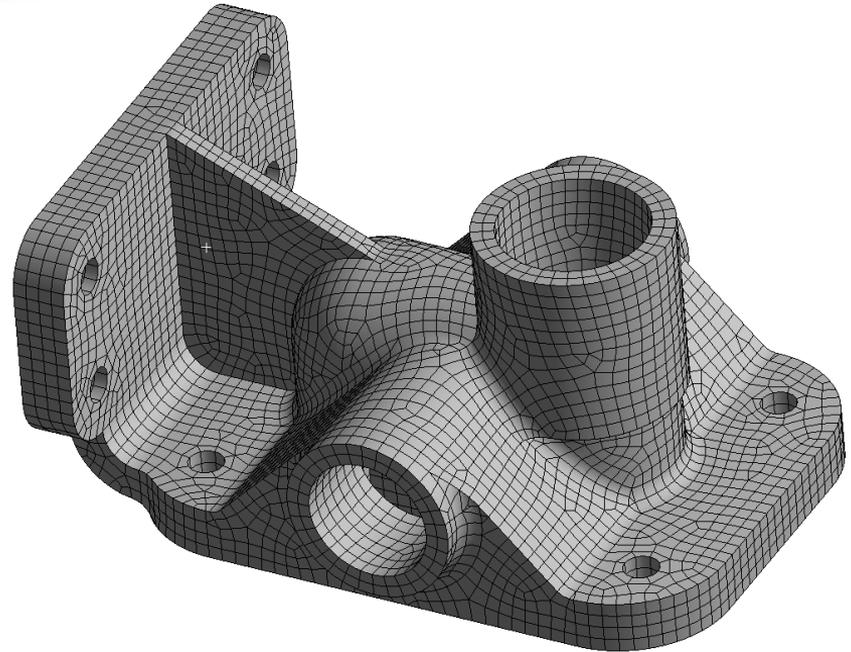
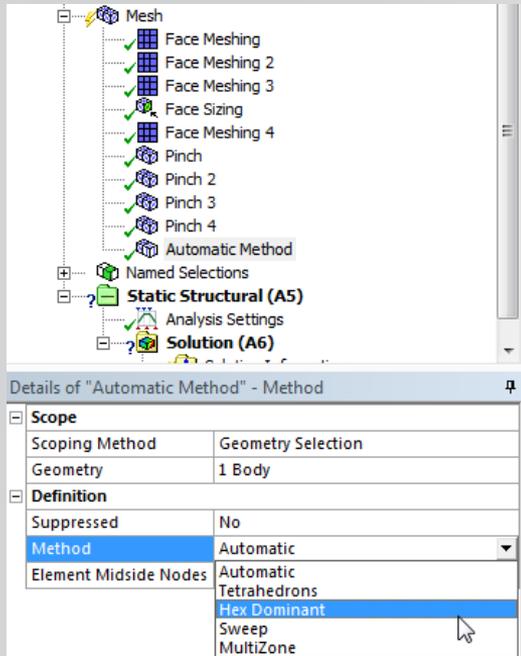
# ...Virtual topologies vs Pinch

- Check the difference between the generated meshes before and after this operation:



- 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.



# ... Go Further!

- Unsuppress Part 2 and Part 3 bodies.

Test the Multizone method on both bodies.

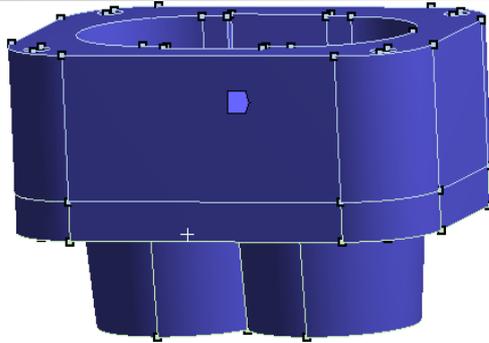
Insert a body sizing of 5 mm as element size to both bodies

Mesh

- Face Meshing
- Face Meshing 2
- Face Meshing 3
- Face Sizing
- Face Meshing 4
- Pinch
- Pinch 2
- Pinch 3
- Pinch 4
- Hex Dominant Method
- MultiZone**
- Body Sizing

Details of "MultiZone" - Method

Scope	
Scoping Method	Geometry Selection
Geometry	2 Bodies
Definition	
Suppressed	No
Method	MultiZone
Mapped Mesh Type	Hexa
Surface Mesh Method	Program Controlled
<b>Free Mesh Type</b>	<b>Hexa Dominant</b>
Element Midside Nodes	Use Global Setting
Src/Trg Selection	Automatic

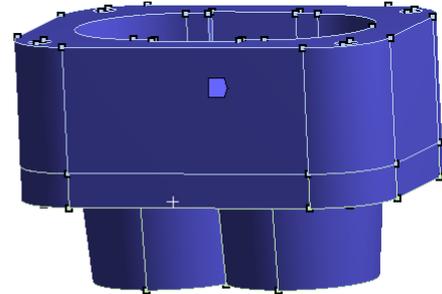


Mesh

- Face Meshing
- Face Meshing 2
- Face Meshing 3
- Face Sizing
- Face Meshing 4
- Pinch
- Pinch 2
- Pinch 3
- Pinch 4
- Hex Dominant Method
- MultiZone
- Body Sizing**

Details of "Body Sizing" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	2 Bodies
Definition	
Suppressed	No
Type	Element Size
<b>Element Size</b>	<b>5. mm</b>



Test the inflation on the Part 2 body.  
With a body sizing of 3mm.

