

Introduction to ANSYS Mechanical

Realize Your Product Promise®

ANSYS Goals

The goal of this workshop is to use several techniques to create named selections that will then be used to set up the boundary conditions shown below.

- Two holes at one end of the model will be used to apply a fixed support.
- On one of the remaining holes we will apply a radial displacement to simulate the effect of a fastener that has been press fit into it.



ANSYS Project Schematic

Begin a new Workbench session and, from the Project page, choose "Restore Archive . . . " and browse to the file "Named_Selections.wbpz" and Open (location provided by instructor).

When prompted, "Save" using the default name in the same location as the archive file.





 From the Static Structural system double click (or RMB > Edit) the "Model" cell.



2. When Mechanical opens, verify the units are set to "Metric (mm, kg, s, mV, mA)".



When Mechanical opens note the model's orientation with respect to the global coordinate system:

3. Expand the Coordinate Systems branch and highlight "Global Coordinate System".





The first named selection will be created so that the constraints can be added to the geometry and conveniently modified.

4. Highlight the cylindrical face of the hole nearest the global coordinate system origin.

5. RMB > Create Named Selection.



- 6. In the Selection dialog enter the name "Fixture".
- 7. Choose "Apply geometry items of same:".
- 8. Check the box "Size".
- 9. Click the "Ok" button.

10. In the tree, highlight the new named selection "Fixture" and note the scope of the selection is 4 faces.

Since our goal is to apply the constraints only to the 2 holes closest to the origin, we need to add a location filter the worksheet.





In order to proceed we need to first determine the location of the features to be filtered. While there are a number of ways we might accomplish this, we'll use the "selection information" feature.

10. Click back into the Graphics tab and then highlight the cylindrical face used previously.

1 k

11. In the top menu click the box to activate "Selection Information".

The summary shows the face centroid is located at an X coordinate of 8 mm. Also note the radius of the cylinder is 2.5 mm.

Selection Information										
Coordinate System: Global Coordinate System 💌 🧭 Show Individual and Summary 💌										
Entity	Surface Area (mm²)	Centroid X(mm)	Centroid Y(mm)	Centroid Z(mm)	Body	Туре	Radius (mm)			
1 Cylinder, Summary	31.326	8.	8.	2.						
Face 1	31.326	8.	8.	2.	1	Cylinder	2.5			





Click into the worksheet tab and note the initial selection (select by size) is represented by the first row.

12. RMB in the worksheet table and "Add Row".



By inspection we can see that the centroid of both required holes must be at the same X location. Instead of using that criteria directly we'll illustrate the use of a "filter".

13. Configure the row to Filter (i.e. retain) the selection based on X location in a range of





- 15. With the filter applied verify the scope of the selection is now 2 faces.
- 16. In the graphics window review the selection.
- 17. To view the mesh on highlighted Named Selections click the Annotation Preferences button in the toolbar and enable "Plot Elements Attached to Named Selections" and click OK. Then de-select and re-select the NS to see the surface mesh on the NS.

Annotation Preferences

✓ Plot Elements Attached to Named Selections

Note this preference is saved for future use.





Again, there are a number of ways we might proceed with this step, but we'll create a geometry based named selection.

Create a named selection where the press fit simulation will be applied:

- **17.** Select the cylindrical face shown here.
- 18. RMB > Insert > Named Selection.

19. In the tree RMB > Rename the new selection to "PressFace".





ANSYS Environment

20. Highlight the Static Structural branch, RMB > Insert > Fixed Support.



- 21. In the details change the scoping method to "Named Selection".
- 22. From the drop down list choose the named selection "Fixture".

In the graphics window note the fixed support is scoped to the 2 holes closest to the global origin.





ANSYS ... Environment

23. Highlight the Static Structural branch, RMB > Insert > Pressure.

Static Structural (A5) Analysis Settings Solution (A6)	Insert 🔶	Ressure	23.	
Solution Inform	🧚 Solve			
	2 Clear Generated Data a∐o Rename (F2)			E - 2 Static Structural (A5)
	Open Solver Files Directory			

- 24. In the details change the scoping method to "Named Selection".
- 25. From the drop down list choose the named selection "PressFace".
- 26. Choose for "Define By" the option "Components" and put 2 MPa for X direction and 1 Mpa for Z direction



	Scope				
	Scoping Method	Named Selection			
	Named Selection	PressFace			
	Definition				
	Type	Pressure			
	Define By	Components			
	Coordinate System	Global Coordinate System			
	X Component	2, MPa (ramped)			
1	Y Component	0, MPa (ramped)			
1	Z Component	1, MPa (ramped)			
	Suppressed	No			

26

ANSYS Results

27. Highlight the Solution branch, RMB > Insert > deformation > Total



A: Static Structural Total Deformation Type: Total Deformation Unit mm Time: 1

0,039465 Max 0,03508 0,03695 0,02631 0,021925 0,01754 0,013155 0,0087699 0,004385 0 Min



28. Highlight the Solution branch, RMB > Insert > stress > Equivalent (von-Mises)





ANSYS Go further!

If you finish this workshop and find yourself with extra time, you could try the following steps:

- 1. Create named selection on a specific zone where you want to check the results.
- 2. Then insert results and define scoping method with the named selection option in order to check the result in a precise zone.