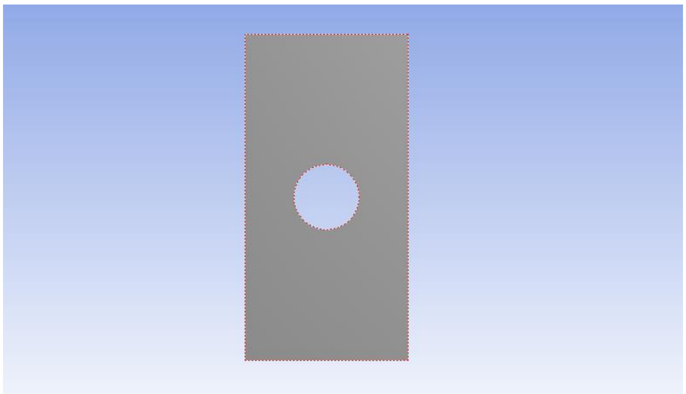
ENGSCI 344 TUTORIAL 5 – A plate with a hole

This tutorial covers assigning material properties, meshing, loads and boundary conditions.

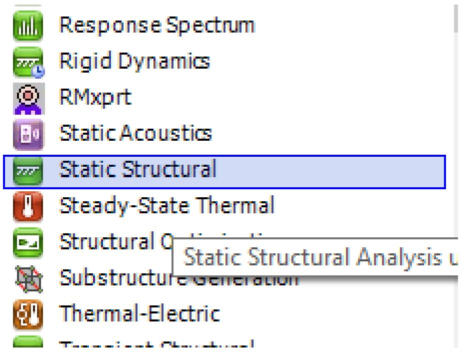


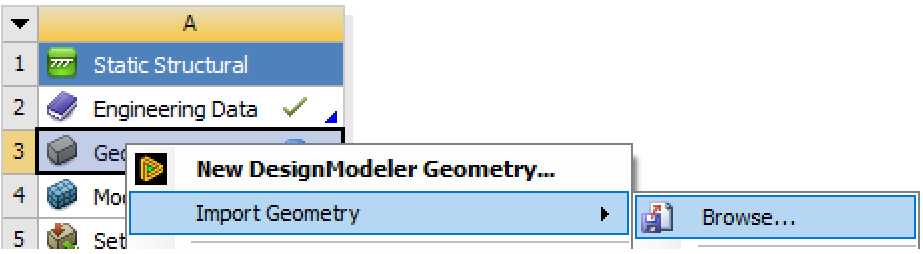
PART A

Import the design modeler file (.adgb file extension) provided on Canvas into ANSYS.

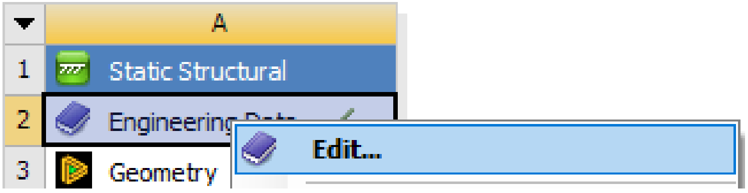
Geometry import, material selection and meshing

1. To import, create a Static Structural template in Workbench (present in the Analysis Systems toolbox on the left), right-click on geometry and import the necessary file.

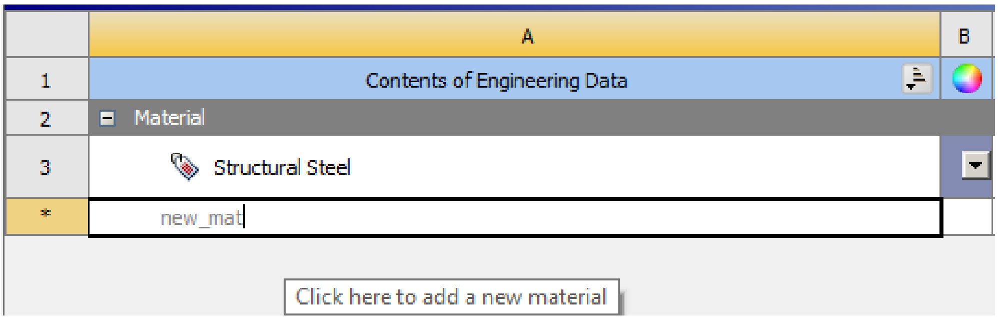




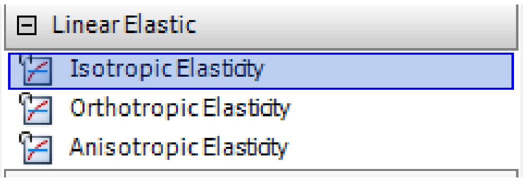
2. Open the Engineering data tab in the Static Structural template.



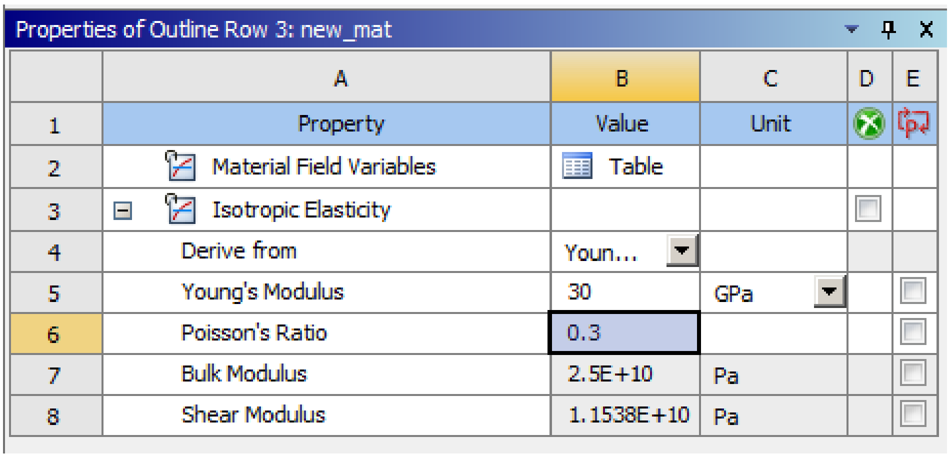
3. Create a new material.



4. Insert new isotropic elasticity definition using the toolbox on the left.

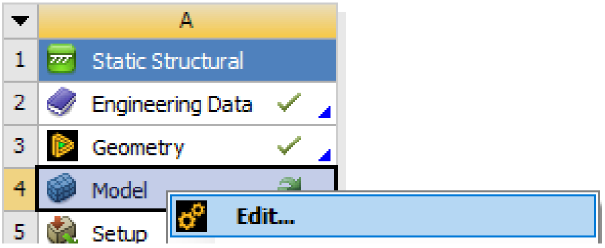


5. Set the Young's modulus and Poisson's ratio to the following values. Do we need to set any other values for linear elastic analysis?

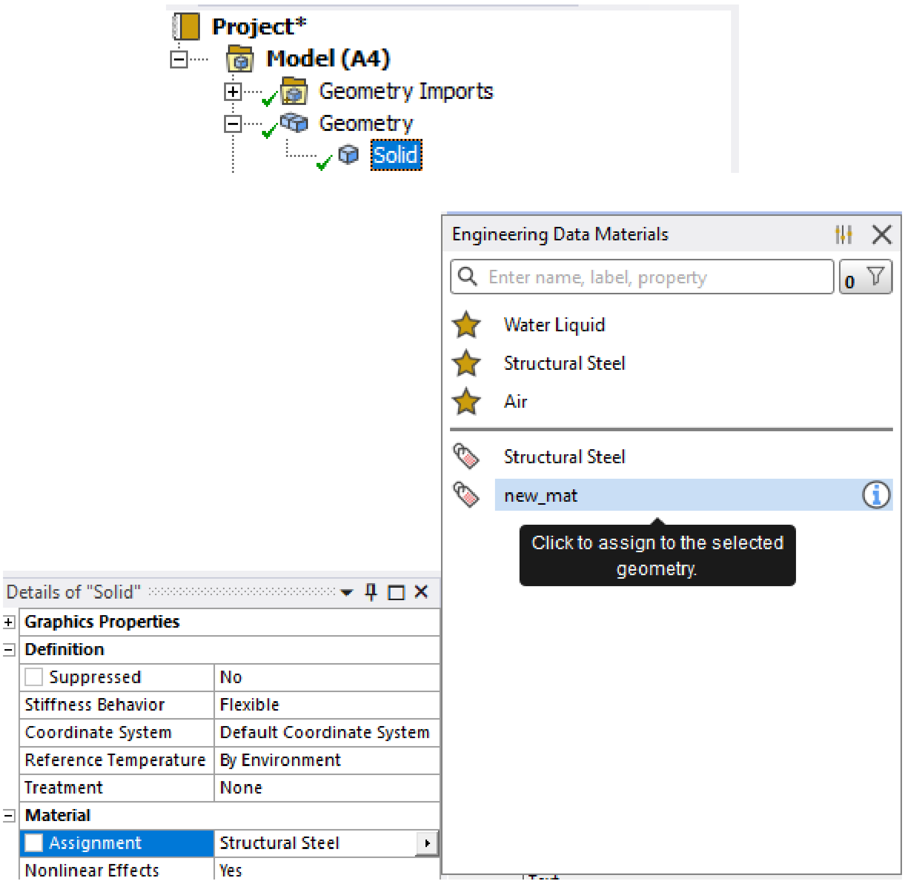

Properties of Outline Row 3: new_mat

	A	B	C	D	E
1	Property	Value	Unit		
2	Material Field Variables	Table			
3	Isotropic Elasticity				
4	Derive from	Youn...			
5	Young's Modulus	30	GPa		
6	Poisson's Ratio	0.3			
7	Bulk Modulus	2.5E+10	Pa		
8	Shear Modulus	1.1538E+10	Pa		

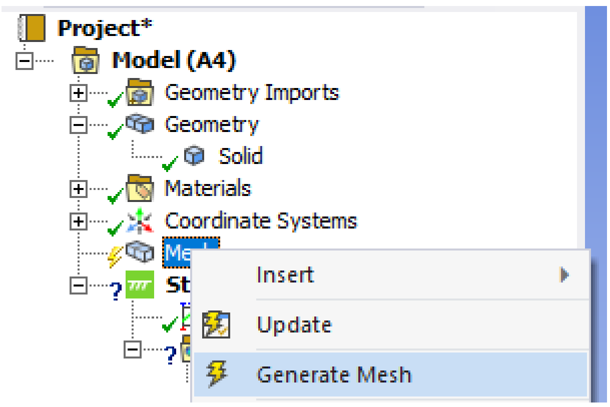
6. Open the Model section and check that the units are set to Metric (mm, kg, N, s, mV, mA).



7. Expand the Geometry tab in the tree and assign the newly created material to the plate.

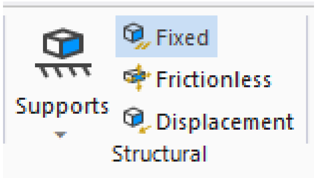


8. Mesh the body using the default options.

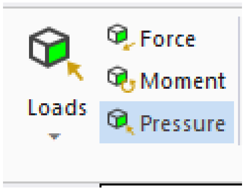


Boundary conditions and Forces

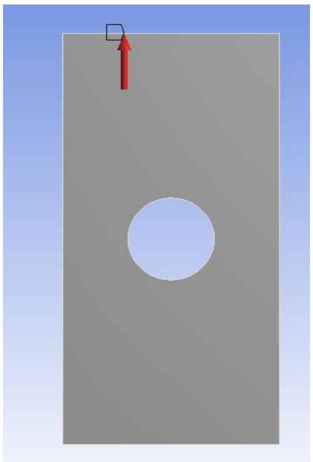
9. Select the bottom face and assign a fixed support boundary condition (Y-axis is upwards). This is available in the supports section. This constrains the motion of the face to zero in all degrees of freedom (3 translational and 3 rotational degrees of freedom). The Supports option is found in the Environment tab.



10. Select the top face and apply a pressure of 1 MPa. The pressure should be acting outwards. Change the 'Define by' section to Vector and assign the direction. The end result should look like the pictures below. The faces where the supports and loads are applied are in the X-Z plane. The Loads option is found in the Environment tab.

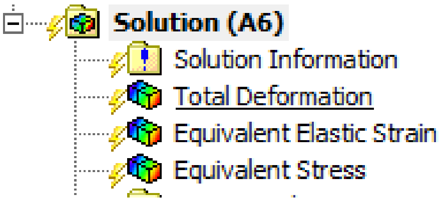


Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Type	Pressure
Define By	Vector
Magnitude	1. MPa (ramped)
Direction	Click to Change
Suppressed	No



Solution

11. Insert Total deformation, equivalent stress and equivalent elastic strain in the Solution section.



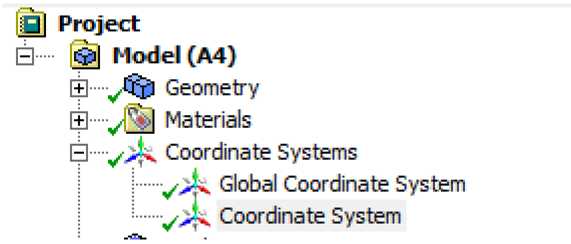
12. Solve the problem.
13. Note down the maximum equivalent stress and total deformation values. These will be used later.

Mesh refinement

Mesh refinement should be done as a part of verification, since refining the mesh and checking for changes to the output variables provides a very good estimate of convergence.

One of the easiest ways to perform this is to refine the mesh of the entire domain. However, at times, global mesh refinement could turn out to become counter-productive as it would add unnecessary computational costs to the problem. Here, strategies such as local mesh refinement can prove useful.

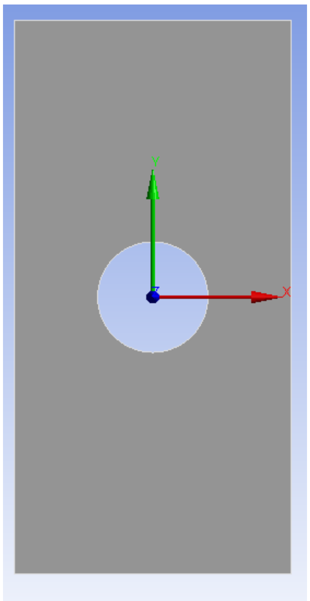
14. To perform local mesh refinement, first, identify the regions of interest in the plate. This is very important before meshing as it could save hours (even days) while solving the problem. The goal should be to refine the mesh in regions of interest and have a reasonable (coarser) mesh in regions that are away from those regions. For this specific problem, the region around the hole is the region of interest, since that is where the stresses are highest and where large stress gradients arise.
15. First, select Coordinate systems in the Outline and create a new coordinate system (Right-click on the Coordinate Systems section and insert a new coordinate system).



16. Make changes to the Details section such that it replicates the picture below:

Details of "Coordinate System"	
Definition	
Type	Cartesian
Coordinate System	Program Controlled
APDL Name	
Suppressed	No
Origin	
Define By	Global Coordinates
Origin X	50. mm
Origin Y	100. mm
Origin Z	5. mm
Location	Click to Change
Principal Axis	
Axis	X
Define By	Global X Axis
Orientation About Principal Axis	
Axis	Y
Define By	Default
Directional Vectors	
Transformations	
Base Configuration	Absolute
Transformed Configuration	[50. 100. 5.]

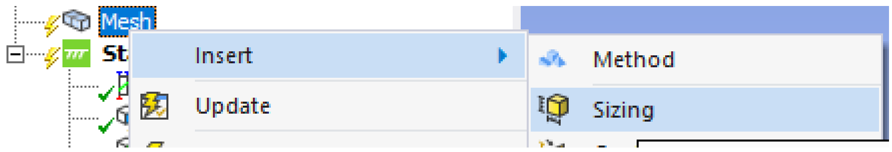
17. A new coordinate system should be generated at the center of the hole.

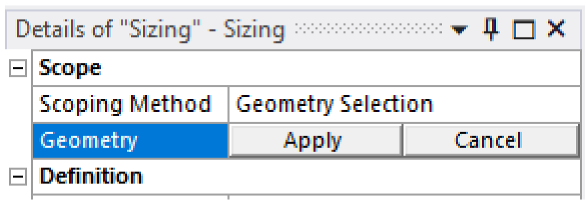


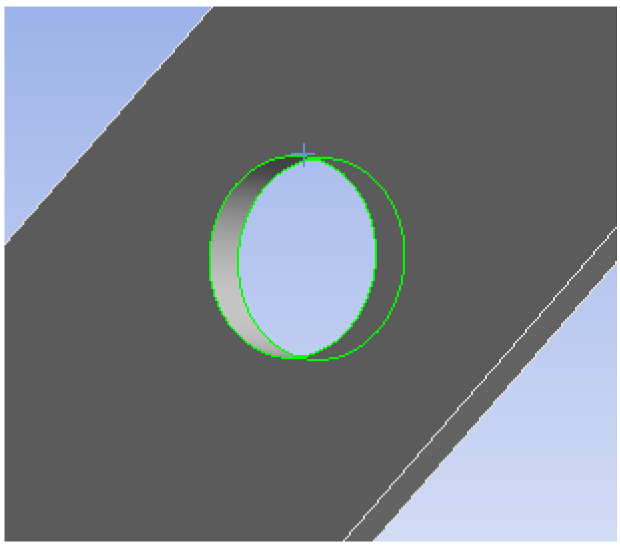
18. Change the default global size of the mesh to 3 mm using the same method followed in tutorial 1.

Details of "Mesh"	
Display	
Display Style	Use Geometry Setting
Defaults	
Physics Preference	Mechanical
Element Order	Program Controlled
Element Size	3.0 mm
Sizing	
Quality	
Inflation	
Advanced	
Statistics	

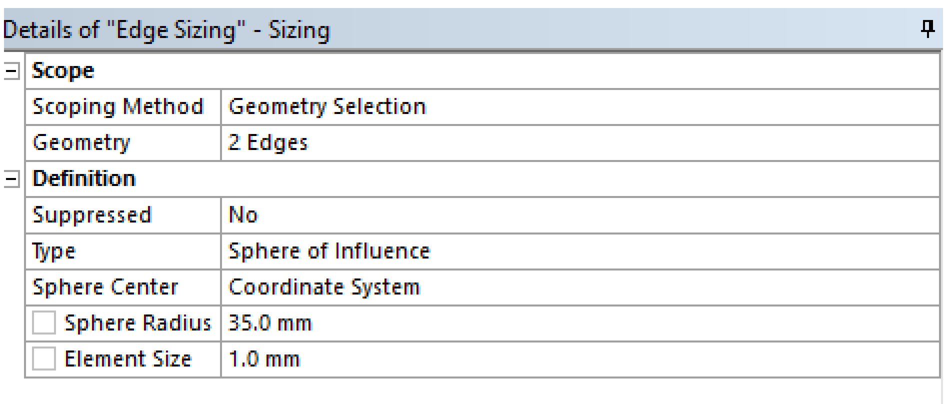
19. Select the circular edge and refine it using edge sizing. The value of element size should be lower than the mesh size of the entire plate (you wouldn't be refining the mesh if the mesh size is greater than the global size). To refine along the edge, insert a sizing method in the mesh section (right-click mesh > Insert > sizing). For the geometry, select both the edges of the hole.







20. Make changes to the "Details" section as shown below. Select the newly created coordinate system in the Sphere Center section.



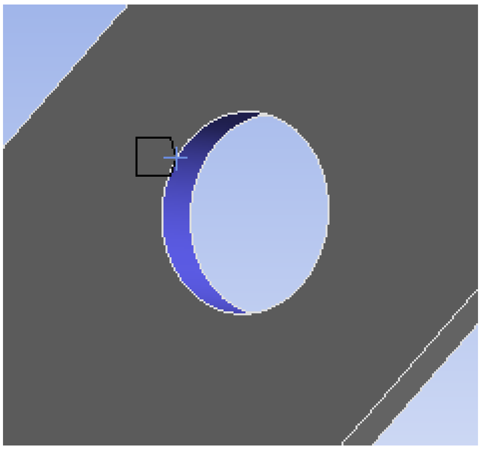
Details of "Edge Sizing" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	2 Edges
Definition	
Suppressed	No
Type	Sphere of Influence
Sphere Center	Coordinate System
Sphere Radius	35.0 mm
Element Size	1.0 mm

21. Insert another sizing option and select the inner face of the hole. Change the element size to 1.0 mm. This should set the face sizing along the inner face of the hole to 1.0 mm.

Details of "Face Sizing" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Suppressed	No
Type	Element Size
Element Size	1.0 mm



22. Use /quadratic as the element order (Details of "Mesh" section). Mesh the body.

Details of "Mesh"

Display	
Display Style	Use Geometry Setting
Defaults	
Physics Preference	Mechanical
Element Order	Quadratic
Element Size	3.0 mm

23. Solve and observe for changes.
24. Refine the mesh further along the hole and observe for changes if any.
25. Change the element order to linear and record changes if any.
26. Change the boundary condition on the bottom face to frictionless support instead of fixed support and solve. Check for warnings/messages/errors. If you encounter any warnings/errors, what do they mean?