



INGENTA

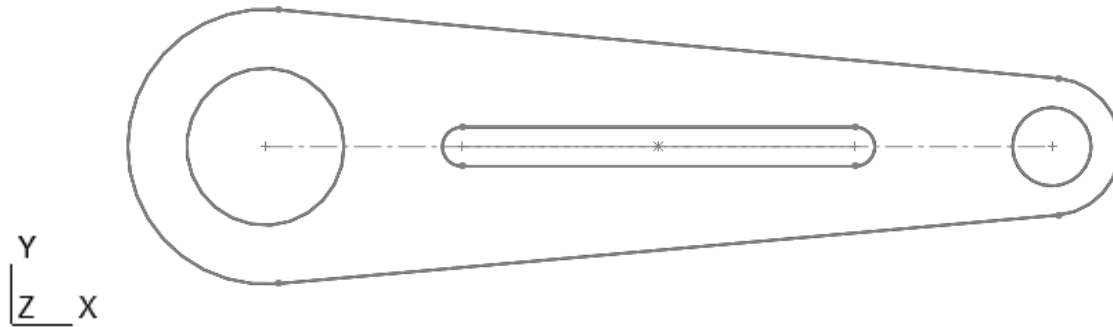
Engineering Solutions

SCIFESOL Tutorial : Static Analysis of Torque Arm



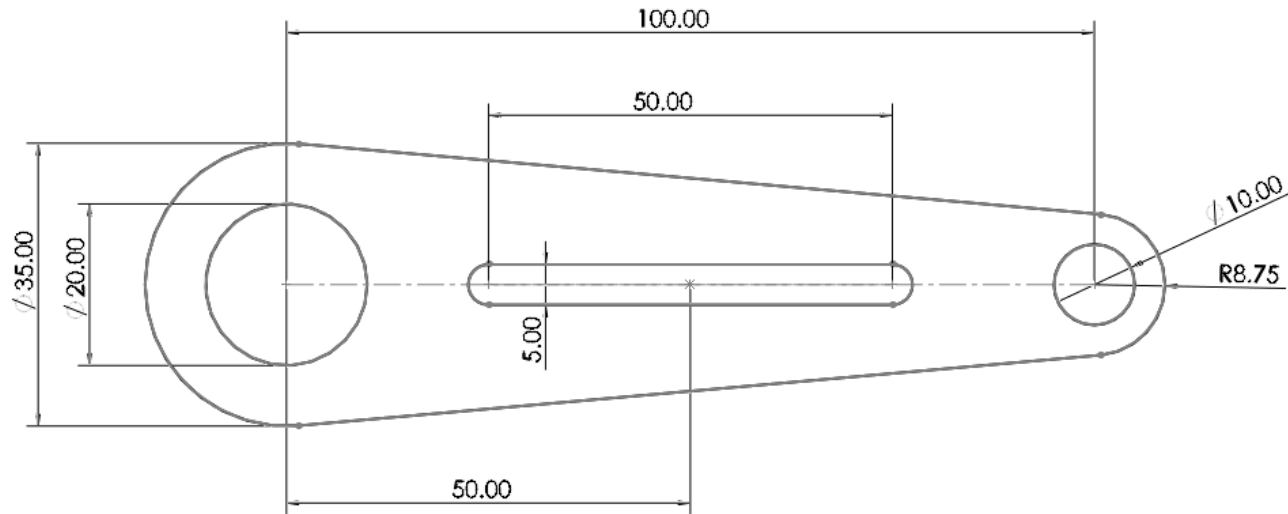
Problem Description

A torque arm of an automotive component is to be verified for its structural integrity . The torque arm is fixed at its left hole and loads are applied on the right side hole.



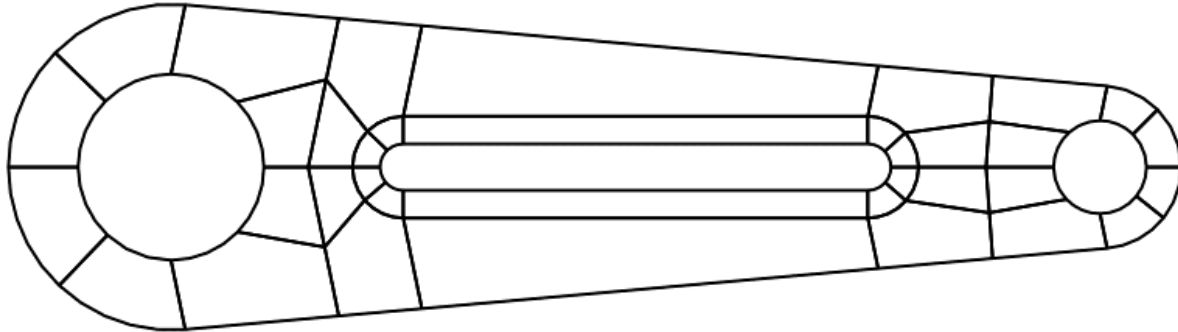
Geometry

A 2D plane stress model of the torque arm having thickness 10 mm is prepared as per the shown dimensions.



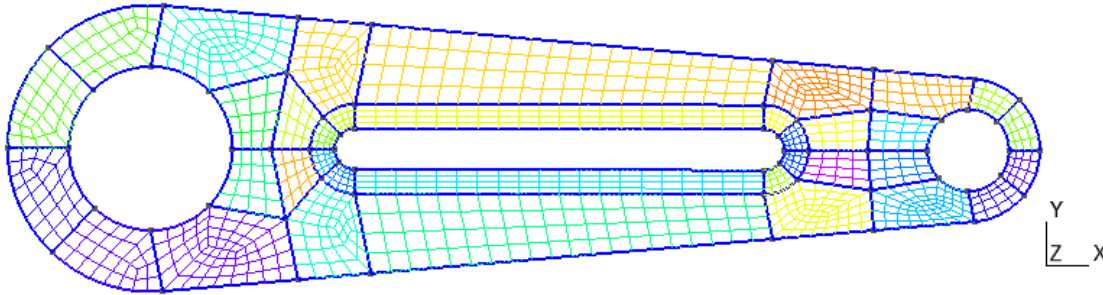
Mesh Generation

Meshing of the torque arm is done in GMSH. We mesh the arm with 4 node quadrilateral elements only. The arm is split to create good quality elements . We need to create physical groups of the curves so that SCIFESOL can recognize regions for load and boundary condition application.



Mesh Generation

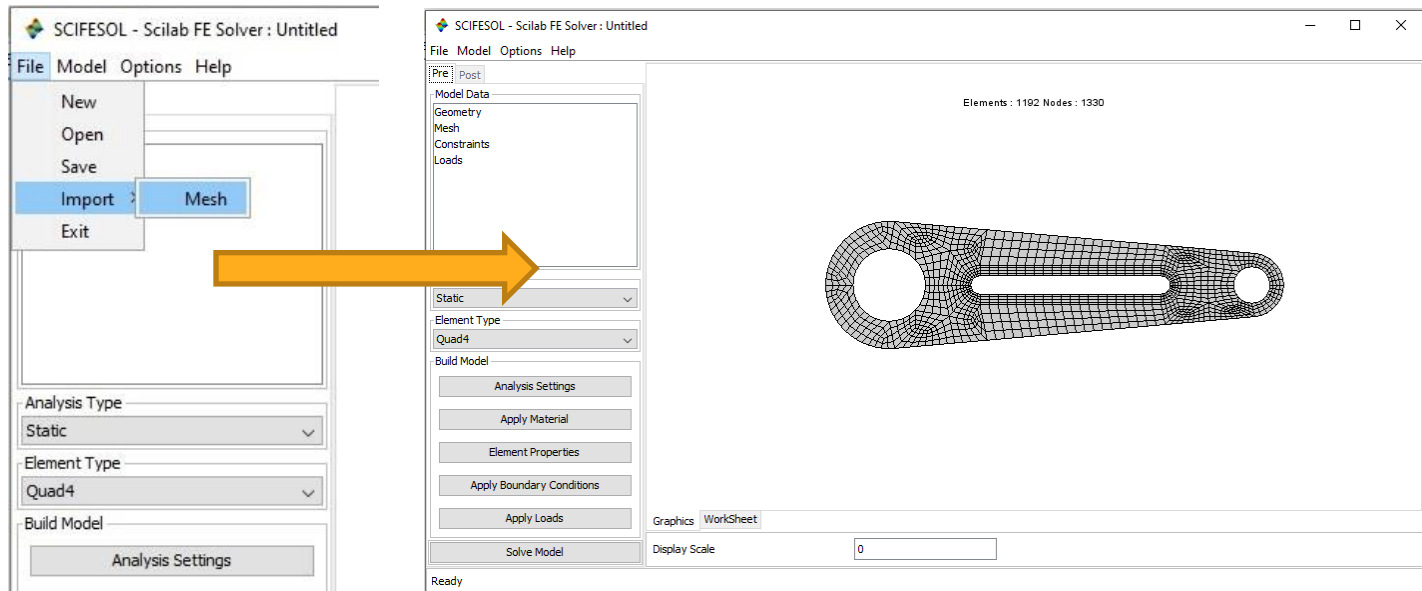
Meshing of the torque arm is done in GMSH.



Statistics			
Geometry	Mesh	Post-processing	
1330	Nodes		
60	Points		
528	Lines		
0	Triangles		
1192	Quadrangles		
0	Tetrahedra		
0	Hexahedra		
0	Prisms		
0	Pyramids		
0	Trihedra		
0	Time for 1D mesh		
0.0780549	Time for 2D mesh		
0	Time for 3D mesh		
Press Update	SICN	Plot	X-Y 3D
Press Update	Gamma	Plot	X-Y 3D
Press Update	SIGE	Plot	X-Y 3D
<input type="checkbox"/> Compute statistics for visible entities only			
Memory usage: 137.277Mb			Update ↻

Import Mesh

- Now import the mesh file saved in file TorqueArm.m which is exported from GMSH.



Define Analysis Settings

➤ Specify type of Analysis and Analysis Settings

The screenshot displays the SCIFESOL - Scilab FE Solver interface. The main window shows a model with 1192 elements and 1330 nodes. The 'Analysis Settings: Static' dialog box is open, showing the following configuration:

- Solver Control:** Large Deflection is set to Off.
- Define Load Steps:** Start Time is 0, End Time is 1, and No. of Sub Steps is 1.
- Load Steps Table:**

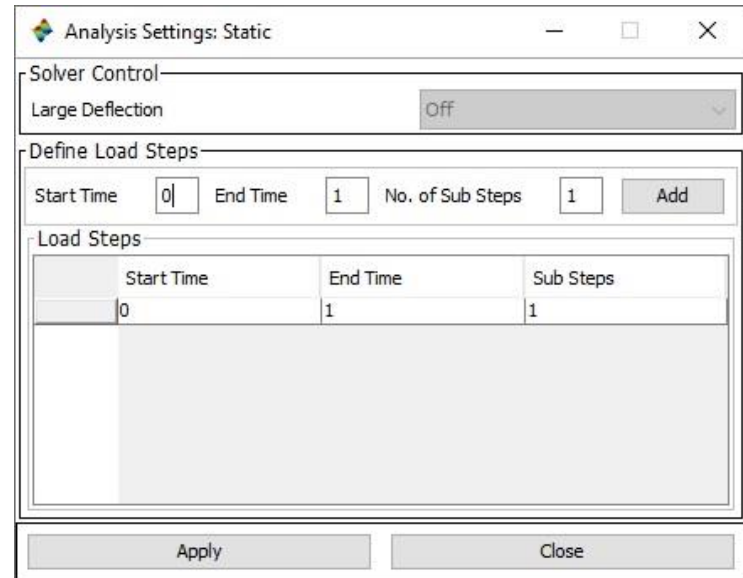
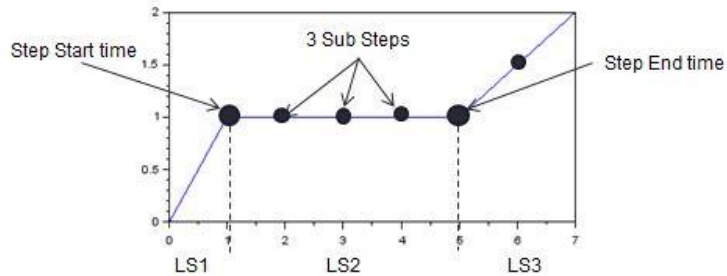
	Start Time	End Time	Sub Steps
	0	1	1

An orange arrow points from the 'Analysis Settings' button in the 'Build Model' section of the main window to the 'Analysis Settings: Static' dialog box. The 'Analysis Type' is set to Static, and the 'Element Type' is Quad4.

Define Analysis Settings

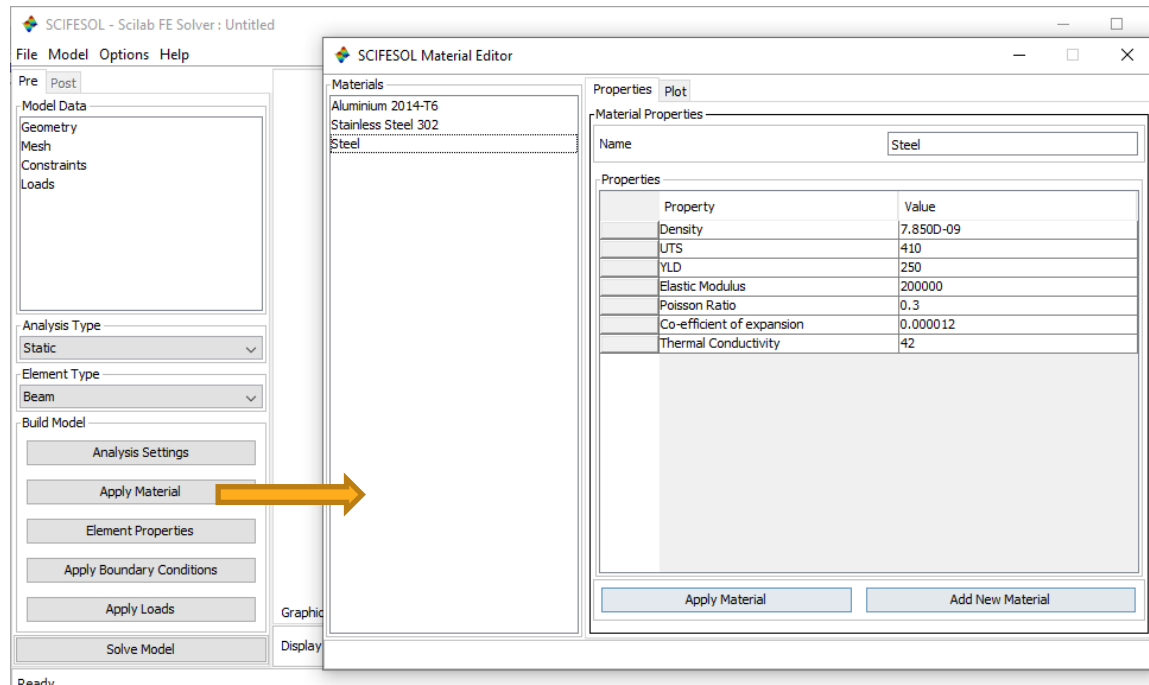
➤ Analysis Settings > Define Load Steps

Load steps is used to define loading conditions for time varying loads.



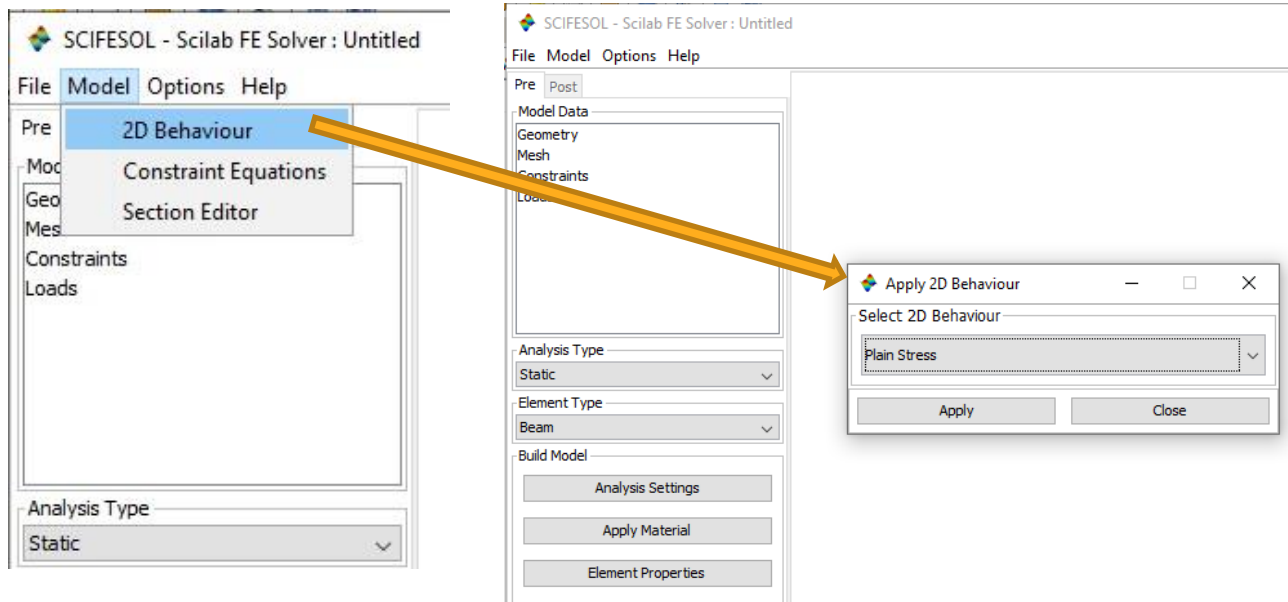
Define Material Properties

- Select material of torque arm from material editor. We can add new material if other material is required.



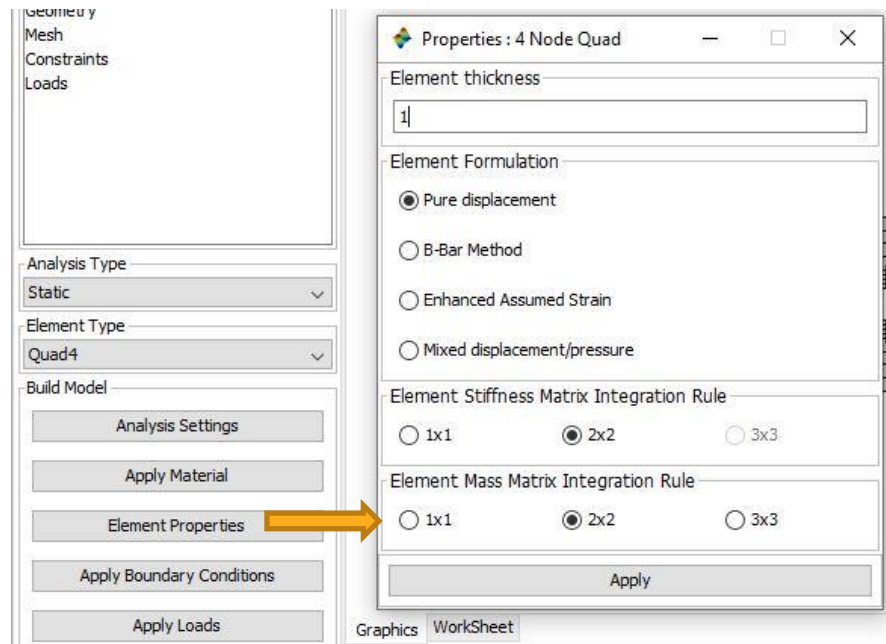
Define Element Properties

- We define the behavior of 2D 4 node quadrilateral elements as plane stress.



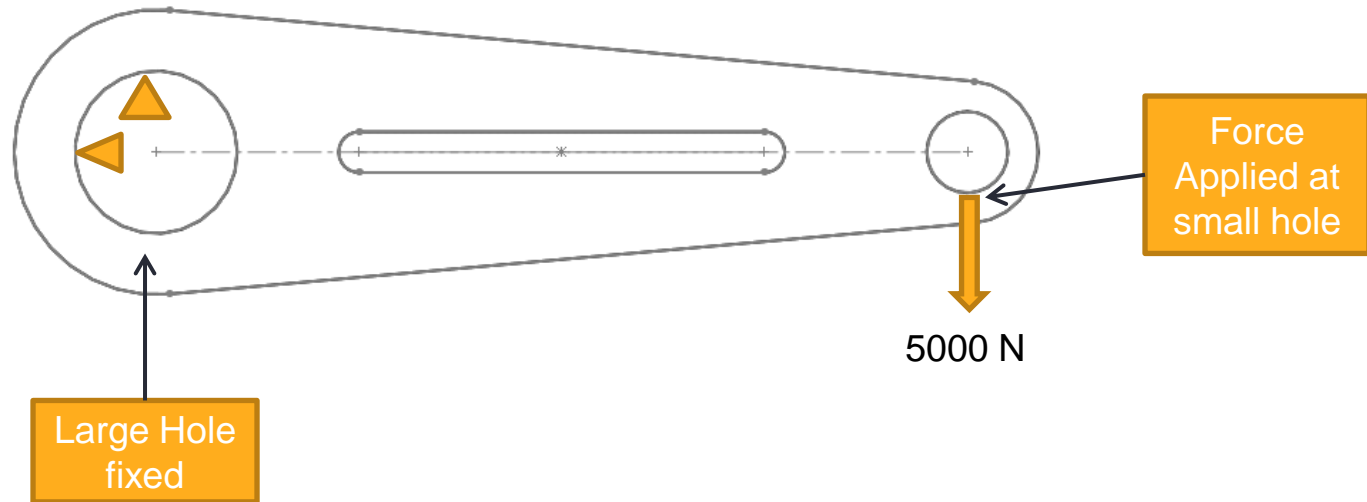
Define Element Properties

- Now we define the formulation of quadrilateral elements as pure displacement method.
- Select the stiffness and mass matrix integration rule as 2x2.
- Specify the thickness as 10 mm.



Apply Boundary Conditions

The torque arm is fixed at its left hole and force of 5000 N is applied on the right side hole.



Apply Boundary Conditions

Using the edge Id of the edges in the left side hole we select all the nodes on those edges to apply the fixity constraint.

The screenshot displays the software interface for applying boundary conditions. The 'Apply Boundary Conditions' dialog is open, showing the 'Fixed' option selected. The 'Apply Fixed Boundary Condition' dialog is also open, showing the 'Select Region' option selected and the region ID '1,2,3,4,5,6,7,8' entered. A legend on the right shows edge IDs 1-8 with corresponding colors. A circular plot on the right shows the geometry with edges colored according to the legend.

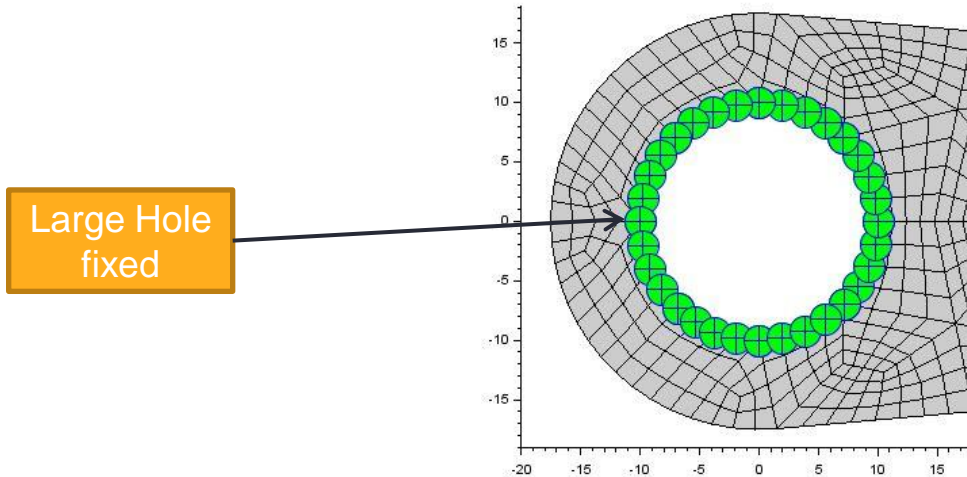
Color	Edge ID
Black	Bid: 1
Blue	Bid: 2
Green	Bid: 3
Cyan	Bid: 4
Red	Bid: 5
Magenta	Bid: 6
Yellow	Bid: 7
Grey	Bid: 8



Identify the boundary ID of the hole region from geometry link in the model data list.

Apply Boundary Conditions

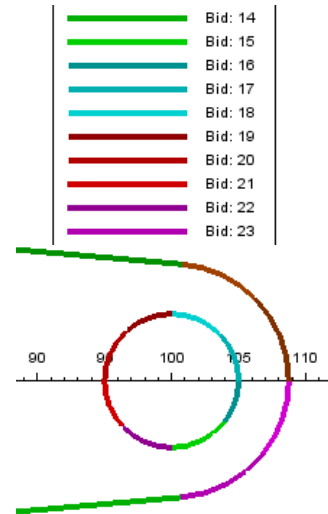
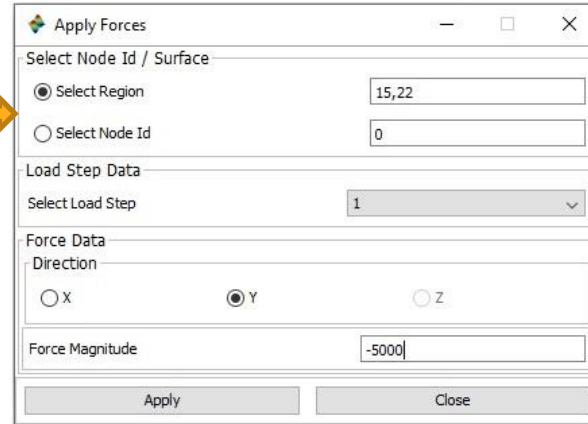
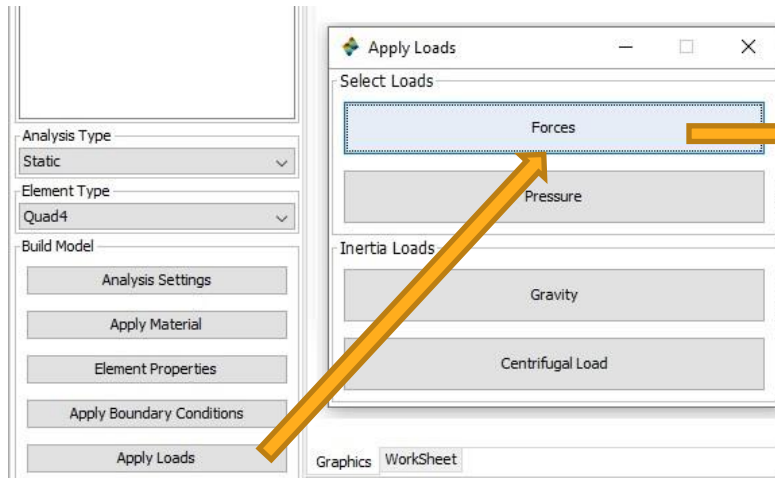
Fixed boundary condition is applied at the large hole.



After applying the boundary condition , first click on the mesh link in the model data list to activate the mesh then click on the constraints or loads link to display the constraints.

Apply Loads

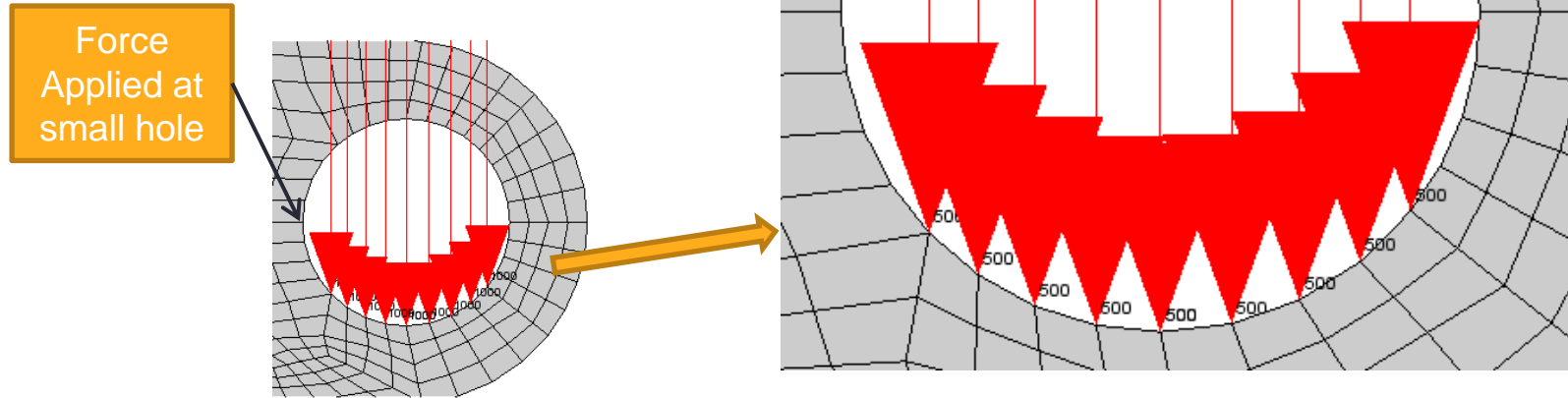
Force of 5000 N is applied on half region of the right hole. We can apply load on Nodes or Regions.



Identify the boundary ID of the hole region from geometry link in the model data list.

Apply Loads

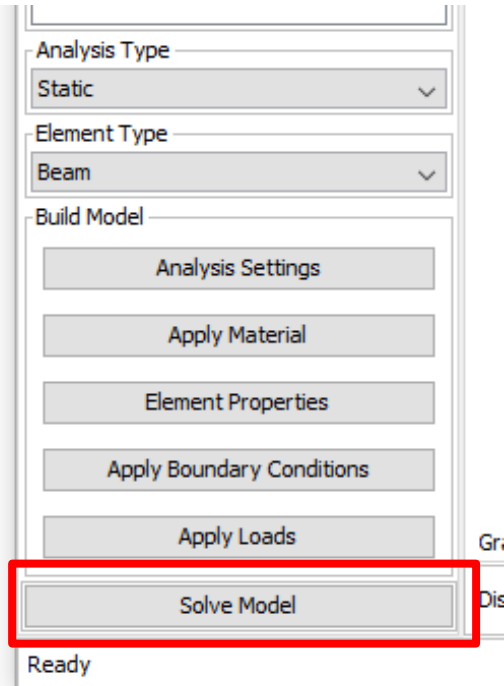
Force of 5000 N is applied in the region of the right hole. The load is divided equally on all the nodes.



After applying the boundary condition , first click on the mesh link in the model data list to activate the mesh then click on the constraints or loads link to display the constraints.

Solve

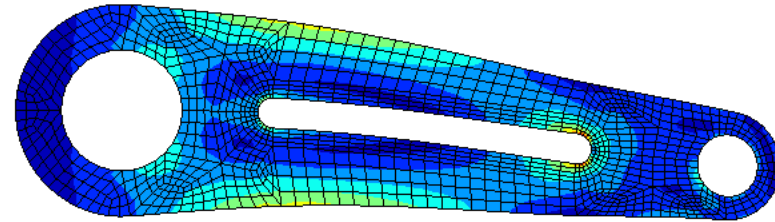
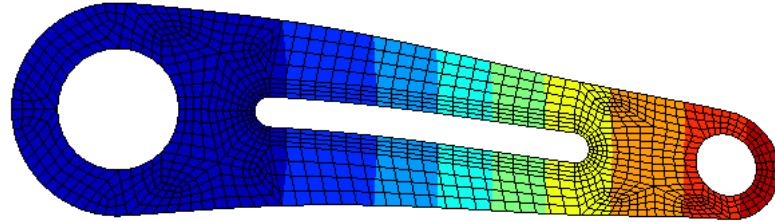
Start static solver to solve the model.



Post Process Results

Using the post processor tab, we can plot the required results.

Results
Select Load Step
1
Display Time
1
Type
Displacement
Results
UJY
Display Options
Averaged
Evaluate





Thanks!