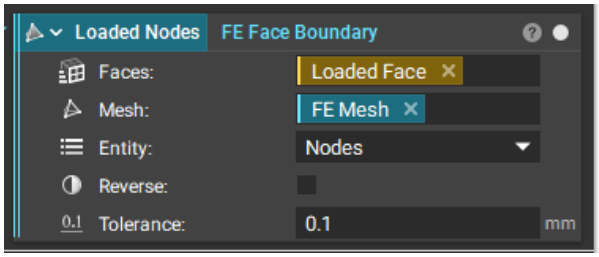
Follow Along: Boundary Conditions

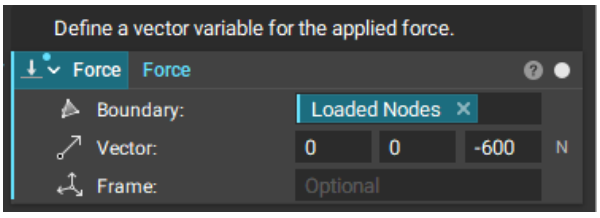
In this video, we will add boundary conditions to the brake pedal used in the previous Follow Along Video. To review selecting boundaries, please revisit our Guide to Meshing Course.

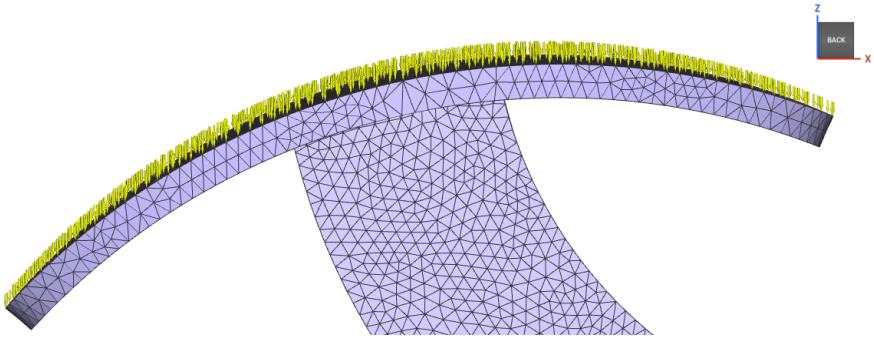
Please use the completed FE Model to follow along with the tutorial.

Step 1: The starter file already has our FE Mesh created (in the FE Model Section), and we will use this to select our boundaries. Add the **FE Face Boundary** Block to the workflow under a new section labeled Boundary Conditions. For the face option, use the Loaded Face variable in the Geometry section of the notebook. For the Mesh, use the FE Mesh defined above and use Nodes as the entity. You can make this a variable by right clicking on the block and choosing “Make Variable” or using the hotkey CTRL M.

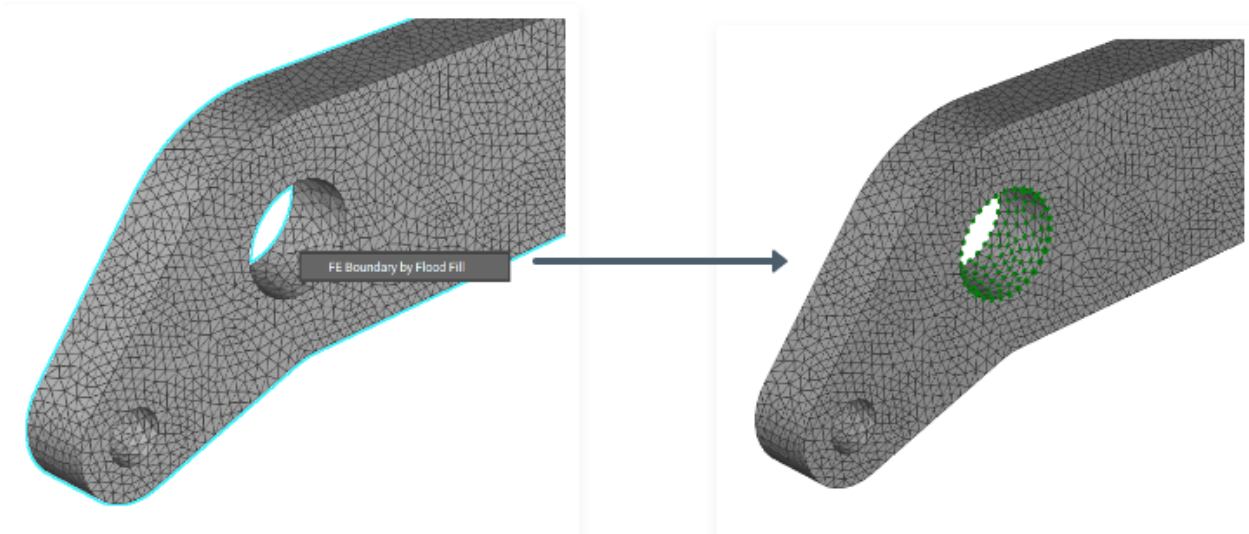


Step 2: Add a **Force** block to the workflow. Insert the **FE Face Boundary** block into the Boundary input and set the Vector to (0,0,-600N). This represents a foot acting on the brake pedal head with a factor of safety.The downward arrows represent the -600N of Force acting in the negative Z-direction over the entire brake pedal head. If you want more control over the direction of the force, you can edit it using the 'Frame' input. The yellow arrows represent the location and direction of the force.





Step 3: To select the boundary for the displacement restraint, turn on only the visibility of the FE Mesh by selecting that block and typing I. Then, right click on the face of the hole shown below. Choose FE Boundary by Flood Fill and it will select that entire region.



Step 4: Add a **Displacement Restraint** block to the notebook, and insert the **FE Boundary by Flood Fill** block into that Boundary input. The red icons on the hole represent the restrained boundary.