

ANSYS - Plate with a Hole - Step 6

Problem Specification

1. Start-up and preliminary set-up
 2. Specify element type and constants
 3. Specify material properties
 4. Specify geometry
 5. Mesh geometry
 - 6. Specify boundary conditions**
 7. Solve
 8. Postprocess the results
 9. Validate the results
- Problem Set 1

Step 6: Specify boundary conditions

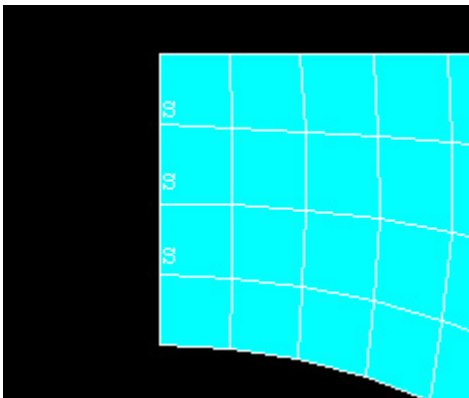
Next, we step up to the plate to define the displacement constraints and loads. Recall that in ANSYS terminology, the displacement constraints are also "loads". As in the truss tutorial, we'll apply the loads to the geometry rather than the mesh. That way we won't have to reapply the loads on changing the mesh.

Apply Symmetry Boundary Conditions

ANSYS provides the option of applying a "symmetry boundary condition" along lines of symmetry.

Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > Symmetry B.C. > On Lines

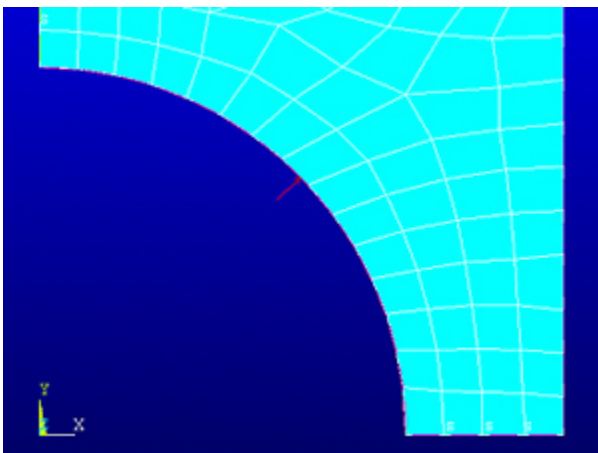
Select the straight lines corresponding to the left and bottom edges (which are the lines of symmetry for this problem) by clicking on them. Click **OK** in the pick menu. The symbol s appears along these lines indicating that the symmetry B.C. is applied along these lines.



Apply Pressure

Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Pressure > On Lines

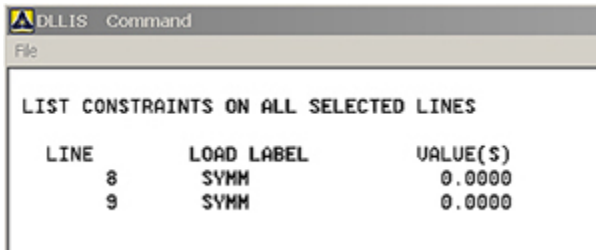
Select the circular arc and click **OK**. This brings up the *Apply Pressure on Lines* menu. Enter p for **Value** and click **OK**. A single red arrow denotes the pressure and the direction in which it is acting.



Check Loads

Let's check that the displacement constraints have been applied correctly.

Utility Menu > List > Loads > DOF constraints > On All Lines



Symmetry BCs are applied on lines 8 and 9. Turn on line numbering:

Utility Menu > PlotCtrls > Numbering

Turn on **Line numbers** and click **OK**. Are lines L8 and L9 the ones on which you want the symmetry BCs?

Similarly, check that the pressure is applied correctly using **Utility Menu > List > Loads > Surface Loads > On All Lines**. Note that **VALI** and **VALJ** would be different if the applied pressure were linearly varying along the line.

Turn off line numbering: **Utility Menu > PlotCtrls > Numbering**. Turn off Line numbers and click **OK**.

Save Your Work

Toolbar > SAVE_DB

[Go to Step 7: Solve](#)

[See and rate the complete Learning Module](#)

[Go to all ANSYS Learning Modules](#)