

ANSYS - Orthotropic plate with a hole - Step 3

Problem Specification

1. Create Command Log file
2. Modify Log file - part1
3. **Modify Log file - part2**
4. Solve
5. Postprocess the results

Step 3: Modify Log file - part 2

Specify Line Divisions and Spacing Ratio

Since we want to determine the stress concentration on the edge of the hole, we want to create a mesh that gets finer in the region near the hole. To do this we'll use the LESIZE command, which defines the number of divisions and spacing ratio of selected lines. Recall that the *spacing ratio* parameter is used to control mesh grading. If positive, the spacing ratio is the ratio of the last division size to first division size.

For this problem, we will use grading on the left and bottom lines of the geometry, which correspond to lines 9 and 8 respectively. Since both lines are the same length, we will use the same spacing ratio=0.25. We will also use 50 divisions for both lines.

```
LESIZE, 8, , , 50, 0.25, , , 0
LESIZE, 9, , , 50, 0.25, , , 0
```

We will also divide the line that defines the hole into 40 divisions. In this case, we'll not specify the spacing ratio. This will be defaulted to one (no mesh grading).

```
LESIZE, 5, , , 40, , , , 0
```

Mesh Geometry

The next twelve commands in the original file generate the mesh.

The first command is SMRT, 5. This command sets the overall element size level for meshing. Recall that we used the *SmartSize* option and set the overall element size to 5 in the Plate with a Hole tutorial. For our problem, we will use the same command, but we will change the element size to 1.

```
SMRT, 1
```

The next two commands specify the element shape to be used for meshing (MSHAPE) and specify whether free meshing or mapped meshing should be used (MSHKEY). We will use the same commands for our analysis.

```
MSHAPE, 0, 2D
MSHKEY, 0
```

The next nine commands create the mesh, by selecting and grouping the area to be meshed into a component and then meshing the selected component (area in this case). However, since we already know the area number we want to mesh (3), we can replace these nine commands with a single command to mesh the geometry.

```
AMESH, 3
```

Specify Boundary Conditions

We will now modify the next seven commands to specify the boundary conditions of our problem.

The next four commands in the original file, specify symmetry boundary conditions along lines 8 and 9 of the original Plate with a Hole geometry. As in the case of the meshing, we can replace these four commands with two commands since we already know the lines (8 and 9) where the symmetry boundary conditions are to be applied.

```
DL, 8, 3, SYMM
DL, 9, 3, SYMM
```

The next three commands in the original file, apply a pressure load to line 5, which is the line that defines the hole. Again, we can replace these three commands with a single command since we already know the line (2) where the pressure is to be applied. Note that the pressure is now applied to line 2, which is the vertical line located on the right side of the area.

```
SFL, 2, PRES, -p,
```

Also note that the sign of the pressure changed from + to -. This is due to the pressure being applied away from the area and not towards the area.

At this point the modified log file should be as follows:

```

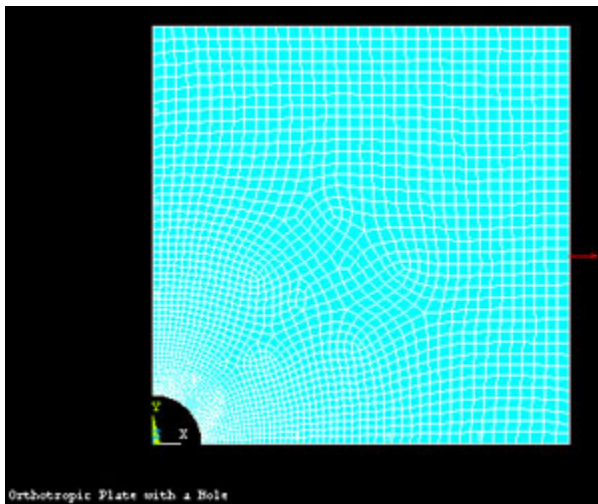
/Title, Orthotropic Plate with a Hole
*SET,a,60e-3
*SET,r,7e-3
*SET,p,1e6
*SET,E1,59.3e9
*SET,E2,22e9
*SET,G12,8.96e9
*SET,nu21,0.047
/PREP7
ET,1,PLANE82
MP,EX,1,E1
MP,EY,1,E2
MP,NUXY,1,NU21
MP,GXY,1,G12
RECTNG,0,a,0,a,
CYL4,0,0,0,0,r,90
ASBA,1,2
LESIZE,8,,,50,0.25,,,0
LESIZE,9,,,50,0.25,,,0
LESIZE,5,,,40,,,,,0
SMRT,1
MSHAPE,0,2D
MSHKEY,0
AMESH,3
DL,8,3,SYMM
DL,9,3,SYMM
SFL,2,PRES,-p,

```

Verify Progress

Let's verify the commands we have created so far. To do this, start a new ANSYS session. Restart ANSYS or go to **Utility Menu > File > Clear & Start New** and select **Do not read file**.

Copy the list of commands we have generated so far and paste them in the *ANSYS Command Input* window. The list of commands will generate the following:



Go to [Step 4: Solve](#)

Go to [all ANSYS Learning Modules](#)