ANSYS - Orthotropic plate with a hole - Step 5

Step 5: Postprocess the Results

In order to access the results, we need to enter the database results postprocessor. This is equivalent to entering the postprocessing module.

```
/POST1
```

To determine the stress concentration along the hole, we will first select the nodes attached to the line that defines the hole and then obtain the value of the circumferential stress at each of these nodes.

Modify Output Options

Since we are interested in obtaining the circumferential stress, we need to change the options for output of results from cartesian to cylindrical.

```
RSYS,1
```

Select Lines and Nodes

We'll use select logic to first select the line that defines the hole (5) and then the nodes attached to this line.

```
LSEL,S,LINE,,5
NSLL,,1
```

Sort Nodal Data

If we were to list the nodal results now, we will obtain a list of the circumferential stresses as a function of the node number. However, we are interested in the circumferential stress as a function of the angle (0 to 90 deg). Since the y coordinate of the nodes along the hole increases as the angle increases, to obtain the circumferential stress as a function of the angle we can sort the results based on the y coordinate of the nodes.

```
NSORT,LOC,Y,1,,
```

Recall that in step 3, we divided the line that defines the hole into 40 elements and that the elements were equally spaced (no grading). Therefore, since we know that the angle varies from 0 to 90 deg and that the line was divided into 40 elements, we can determine the angle at each node.

List Circumferential Stress

The last step is to list the results.

```
PRNSOL,S,COMP
```

This command generates a list containing the X,Y,Z,XY, YZ, and XZ stress components at each node. Since we changed the options for output of results from cartesian to cylindrical, the circumferential stress is shown in the second column (Y component).

The modified and final 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,
FINISH
/SOL
SOLVE
FINISH
/POST1
RSYS,1
LSEL,S,LINE,,5
NSLL,,1
NSORT,LOC,Y,1,,
PRNSOL,S,COMP
```

Verify Progress

Restart ANSYS or go to **Utility Menu > File > Clear & Start New** and select *Do not read file*.

Copy the list of commands and paste them in the ANSYS Command Input window. The list of commands will generate the following:

Analysis of Results

We will use the theoretical solution developed by Greszczuk, L.B (see reference below) to verify the results obtained with ANSYS. To do this, we need to import the results obtained into Excel or a similar application.

After the solution is performed, save the list generated (*PRPATH command* window). Go to File > Save as. Enter `plate2.lis` as the file name. Open this file using Excel or a similar application and delete all columns except the SY column (circumferential stress). You will need to create a new column to specify the angle. Recall that the angle at each node can be determined based on the number of divisions (90deg/40div=2.25 increments). The file will look like this:

Angle (deg)	SY Circum Stress
0.00	-668540
2.25	-664290
4.50	-653420
6.75	-635420
9.00	-609630
11.25	-577290
13.50	-538470
15.75	-493860
18.00	-443690
20.25	-388380
22.50	-328530
24.75	-264370
27.00	-196280
29.25	-124330
31.50	-48549
33.75	31190
36.00	115130
38.25	203640
40.50	297240
42.75	396600
45.00	502490
47.25	615760
49.50	737630
51.75	869300
54.00	1012100
56.25	1167400
58.50	1336900
60.75	1521800
63.00	1723500
65.25	1942500
67.50	2179300
69.75	2432700
72.00	2700100
74.25	2975400
76.50	3253200
78.75	3517000
81.00	3750700
83.25	3953500
85.50	4098400
87.75	4188200
90.00	4225200

Create a text file (results.txt) with these results. Use Matlab or a similar application to import/read the results.txt file and plot them along with the theoretical solution. Refer to the reference below for a detailed description of the theoretical solution and associated equations.
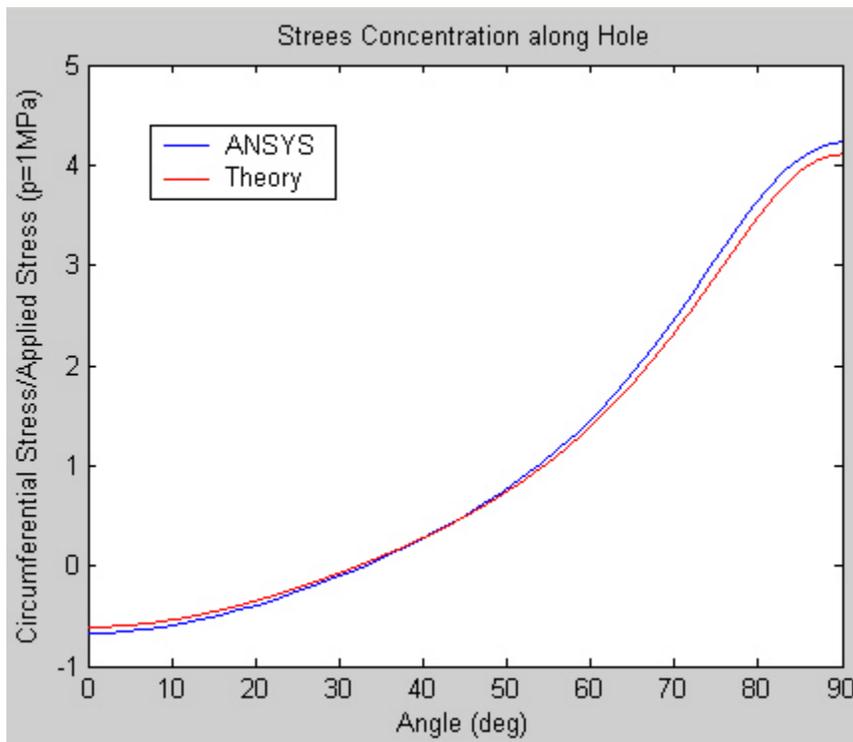
Stress Concentration along Hole

As we can see, the solution obtained with ANSYS compares well with the theoretical solution. The highest variation between the theoretical solution and the results obtained with ANSYS occurs at 90 deg. At this angle, the value obtained with ANSYS varies by less than 3% with respect to the theoretical value.

Reference

Greszczuk, L.B., "Stress Concentrations and Failure Criteria for Orthotropic and Anisotropic Plates with Circular Openings", *Composite Materials: Testing and Design (Second Conference), ASTM STP 497,* American Society for Testing and Materials, 1972, pp. 363-381.

Go to all ANSYS Learning Modules