

ANSYS - Orthotropic plate with a hole - Step 2

Problem Specification
1. Create Command Log file
2. Modify Log file - part 1
3. Modify Log file - part 2
4. Solve
5. Postprocess the results

Step 2: Modify Log file - part 1

Delete Special Commands

We'll start by deleting unnecessary commands. ANSYS generates certain commands which are either generated by default or intended for use in Batch mode (i.e. no user interface). Since we are not going to be working in batch mode, we will ignore the following commands. Delete the following commands:

```
/BATCH
/input,menust,tmp,'',,,,,,,,,,,,,,1
WPSTYLE,,,,,,,,,0
/NOPR
/PMETH,OFF,0
```

The next eleven commands in the original input file start with KEYW. This set of commands customizes the graphical user interface. Since we are not going to be using the graphical user interface, we'll ignore these commands. Delete commands that start with KEYW.

Modify Specified Parameters

We'll now modify the parameters specified in the Plate with a Hole tutorial. We'll do so by modifying the commands that start with *SET. The scalar parameters created as part of the Plate with a Hole tutorial are the plate half-width a , hole radius r , applied pressure p , and material properties E and ν . We will redefine these parameters and add new ones to match the geometry and material properties of our problem:

```
*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
```

Note that the plate half-width a changed from $10e^{-3}$ to $60e^{-3}$ and the hole radius r remained unchanged. Also, the material properties were modified to include the orthotropic material properties: Young's modulus in the fiber direction $E1=Ex=59.3\text{GPa}$, Young's modulus in the transverse direction $E2=Ey=22\text{GPa}$, in-plane shear modulus $G12=Gxy=8.9\text{GPa}$ and minor Poisson ratio $\nu21=0.047$.

At this point, the modified input 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
```

Note that the `/Title, Orthotropic Plate with a Hole` command has been added. This command allows us to add a title which will be shown in the various user interface windows.

Specify Element Type

Before specifying the element type, we need to enter the model creator preprocessor. This is equivalent to entering the preprocessor tab in the ANSYS Main Menu. The following command, which was part of the original input file, accomplishes this task and we'll keep it for our analysis.

```
/PREP7
```

The ET command defines the element type. For the Plate with a Hole tutorial, we used **PLANE42**, which is a 4-node 2D structural solid element. For this tutorial, we will use **PLANE82**, which is a 8-node 2-D structural solid element.

```
ET,1,PLANE82
```

Specify Material Properties

The next set of commands in the original file, specify the material properties by using `MPTEMP` and `MPDATA`. For our analysis, however, we will use the `MP` command instead. We'll also use the previously defined parameters.

```

MP,EX,1,E1
MP,EY,1,E2
MP,NUXY,1,NU21
MP,GXY,1,G12

```

Note that we have added two more commands: `MP,EY,1,E2` and `MP,GXY,1,G12`. These define the Young's modulus in the transverse direction and the in-plane shear modulus, respectively. Also, note that we have specified the minor Poisson ratio rather than the major Poisson ratio by using the `MP,NUXY` command.

Create Geometry

The next set of commands in the original file, generate the geometry by creating a square area of side a and then subtracting a circular sector of radius r . Since the geometry we want to generate is the same, we will use the same set of commands. In addition, since the geometry, material properties and loading of our problem are also all symmetric with respect to the horizontal and vertical centerlines, we will model only a quarter of the plate.

```

RECTNG,0,a,0,a,
CYL4,0,0,0,0,x,90
ASBA,1,2

```

Note that the list of commands is the same. The fact that we changed the value for the plate half-width when we modified the parameters, allows us to keep the same list of commands to create the desired geometry.

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,x,90
ASBA,1,2

```

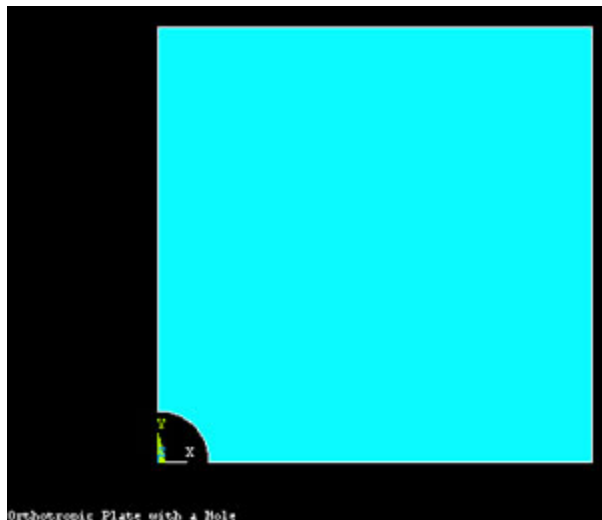
Verify Progress

Let's verify the list of commands we created have so far. To do this, you will need to start a new ANSYS session. Restart ANSYS or go to **Utility Menu > File > Clear & Start New** and select *Do not read file* (if you have been working on another 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.



Next, we will continue to modify the original file to mesh the areas and specify boundary conditions.

Go to [Step 3: Modify log file - part 2](#)

Go to [all ANSYS Learning Modules](#)