ANSYS - Orthotropic plate with a hole - Step 4

Step 4: Solve!

Before solving, we need to exit the model creator preprocessor (`/PREP7`). To do this we will use command `FINISH`, which is also part of the original file.

```
FINISH
```

We can now enter solution module and solve:

```
/SOL
SOLVE
FINISH
```

This is equivalent to entering the solution tab in the ANSYS Main Menu and solving the current LS.

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,
FINISH
/SOL
SOLVE
FINISH
```

Verify Progress

Let's verify that a solution is generated with the list of commands we have created so far. Again, 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. ANSYS performs the solution and a yellow window should pop up saying "Solution is done!".

Go to Step 5: Postprocess the results

Go to all ANSYS Learning Modules