ANSYS - Plate with a Hole - Step 1

- **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 1: Start-up and preliminary set-up

Start ANSYS

Create a folder called plate at a convenient location. We'll use this folder to store files created during the ANSYS session.

Start > All Programs > ANSYS 12.0 > Mechanical APDL Product Launcher

Note that in version 11, it is at

Start > Programs > ANSYS 11.0 > ANSYS Product Launcher

In the window that comes up, enter the location of the folder you just created as your Working Directory by browsing to it. All files generated during the ANSYS run will be stored in this directory/folder.

Specify plate as your Job Name. The job name is the prefix used for all files generated during the ANSYS session. For example, when you perform a save operation in ANSYS, it'll store your work in a file called *plate.db* in your working directory.

Click on Run. This brings up the ANSYS interface. To make best use of screen real estate, move the windows around and resize them so that you approximate this screen arrangment. This way you can read instructions in the browser window and implement them in ANSYS.

You can resize the text in the browser window to your taste and comfort.

In Internet Explorer, use Menubar > View > Text Size, then choose the appropriate font size.

In Mozilla Firefox, use Menubar > View > Zoom.

Set Preferences

As before, we'll more or less work our way down the Main Menu.

Main Menu > Preferences

In the Preferences for GUI Filtering dialog box, click on the box next to Structural so that a tick mark appears in the box. Click OK.

Arelerences for GUI Filtering	the second s	2
[KEYW]]/PMETH] Preferences for Gut Pittering		
Individual discipline(s) to show in the Gut		
	🖓 Structural	
	[" Thermal	
	T ANSIS PAR	
	F PLOTRAN OFD	
Electronagretic:		
	T Magnetic-Nodal	
	T Hagnetic-Edge	
	F High-Frequency	
	F Electric	
Note: If no individual disciplines are selected they will	al show.	
Discipline options		
	(* h-Method	
	C p-Method Strutt.	
	C p-Method Dectr.	
×	Carcel Help	
		100

Recall that this is an optional step that customizes the graphical user interface so that only menu options valid for structural problems are made available during the ANSYS session.

Enter Parameters

For convenience, we'll create scalar parameters corresponding to the plate half-width a, hole radius r, pressure p, and material properties E and v.

Utility Menu > Parameters > Scalar Parameters

Enter the parameter value for *a*: a=10e-3 Click *Accept*.

Similarly, enter the other parameter values and click Accept after each.

r=7e-3 p=1e6 E=1e13 nu=0.3

A	= 1.00000000E-02
NU	= 1.000000000000000000000000000000000000
R	= 1000000 = 7.00000000E-03
Sele	ction
Sele	ction
Sele	ction = 1.00000000E-02

Close the Scalar Parameters window.

We can now enter these variable names instead of the corresponding values as we set up the problem in ANSYS. This is also helpful in carrying out parametric studies where one looks at the effect of changing a parameter.

Go to Step 2: Specify element type and constants

See and rate the complete Learning Module

Go to all ANSYS Learning Modules