ANSYS - Plate with a Hole - Step 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.

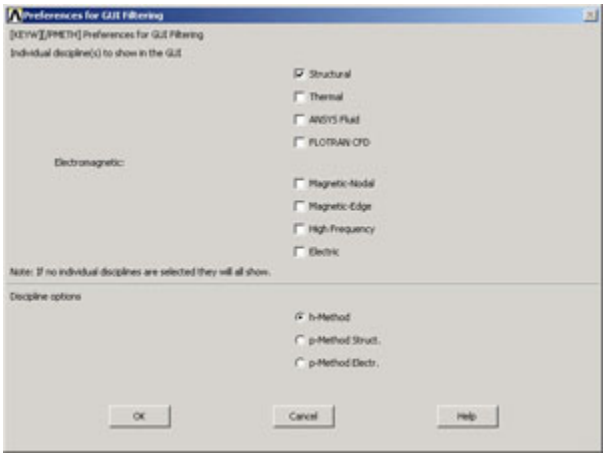
In Mozilla Firefox, use **Menubar > View > Zoom.**

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***.



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

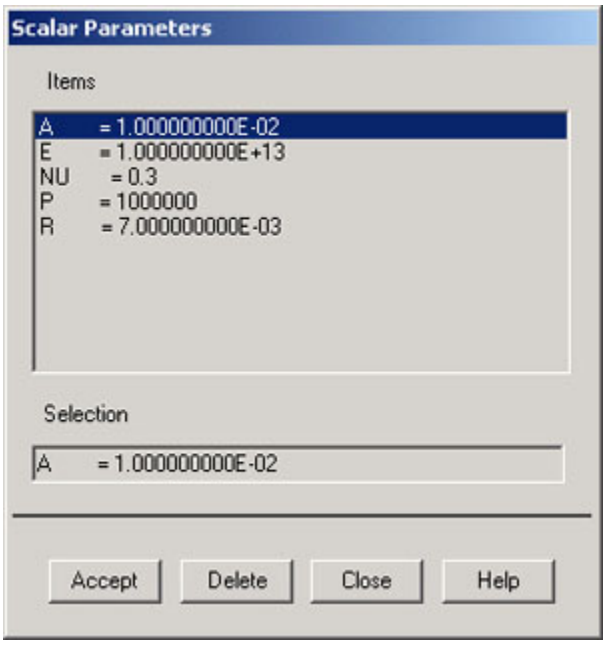
For convenience, we'll create scalar parameters corresponding to the plate half-width *a,* hole radius *r,* pressure *p,* and material properties *E* and *v.*

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
```



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