ANSYS - Truss Step 1

Step 1: Start-up & preliminary set-up

Take a few minutes to review the conventions used in the tutorials which are given on the main page.

Start ANSYS

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

There are two ways to launch ANSYS.

1. Using the ***ANSYS Product Launcher*** is the elegant way to start ANSYS. One specifies the working folder, job name etc. in the *Product Launcher*. We will use this approach below.
2. One can alternatively start the ANSYS graphical user interface (GUI) directly and then set the jobname etc. within the GUI. This can be messy with the beginner often not sure where the files from the session are being stored. This is not the recommended way to start ANSYS.

Start the *ANSYS Product Launcher* from the Start menu. The usual location is:

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. If you specify, say, *C:\ansys* as your working directory, all files generated during the ANSYS run will be stored in this folder.

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

Click on ***Run***. This brings up the ANSYS interface. To make the best use of screen real estate, move the windows around and resize them so that you approximate this screen arrangement. This way you can read instructions in the browser window and implement them in ANSYS. Note that this tutorial has been formatted to fit in a skinny browser window. If your monitor screen is small, you can use *Alt+Tab* keys to conveniently switch between the ANSYS and browser windows (this trick works in Microsoft Windows).

You can resize the text in the browser window to your taste and comfort as follows:

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

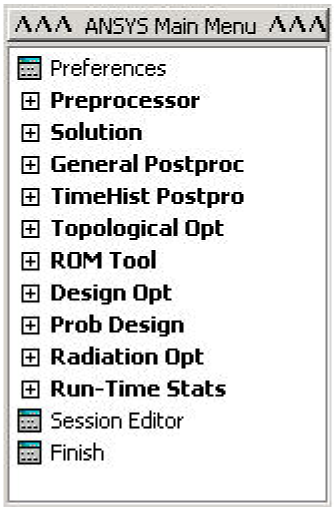
In Mozilla Firefox: **Menubar > View > Zoom**

The ANSYS interface consists of the following entities:

- ***Utility Menu***:

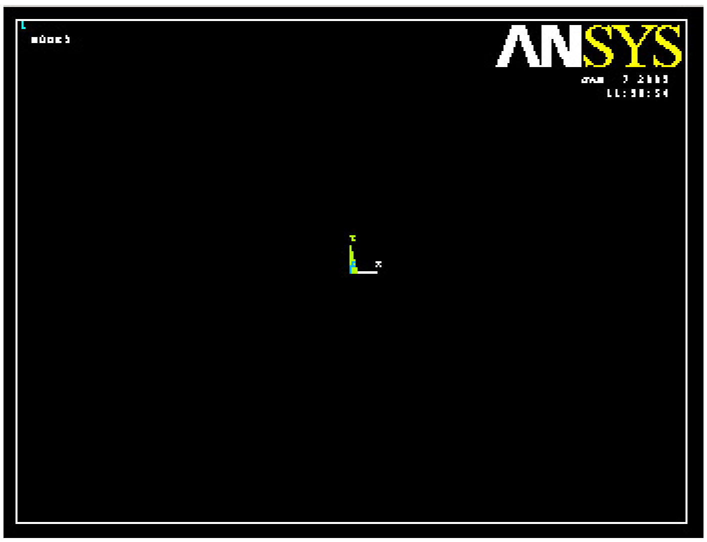
Note that the job name *truss* appears in parenthesis in the title bar of the Utility Menu.

- ***ANSYS Main Menu***:



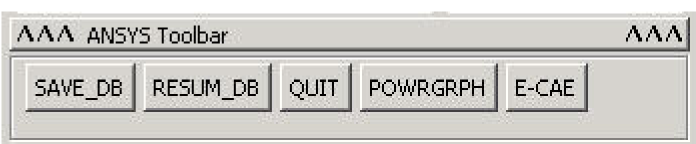
We'll more or less work our way down the ***Main Menu*** as we go through the solution steps.

- ***ANSYS Graphics:***



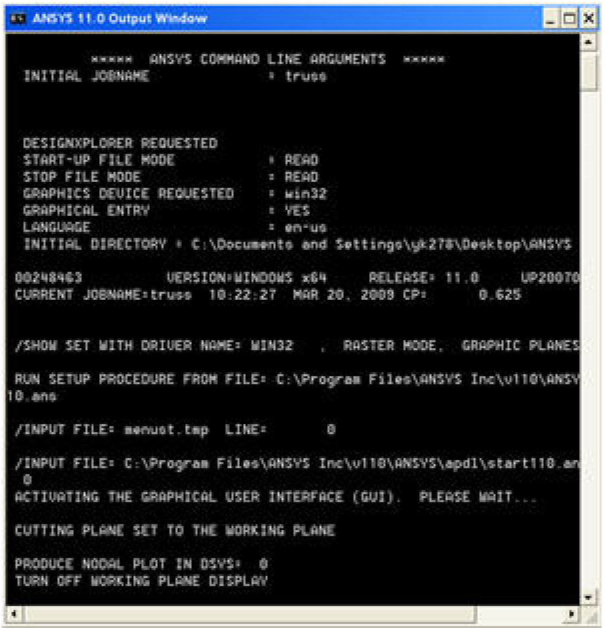
This is the window where the geometry, mesh, applied boundary conditions, and results such as stress values are displayed.

- ***ANSYS Toolbar:***



ANSYS Toolbar contains shortcuts to often used commands such as *save* and can be customized by the user for convenience.

- ***ANSYS 11.0 Output Window:***



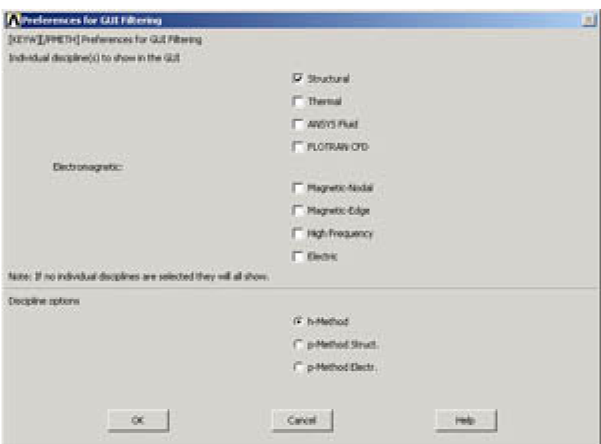
This is the window to which output from ANSYS commands is written and which provides feedback on the actions taken by ANSYS as you navigate the menus. If, at some point, you are not sure you clicked the right button or entered a value correctly, this is where to look first to figure out what you just did.

Set Preferences

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.



The effect of selecting the structural preference is that only menu options applicable to structural mechanics are shown in the graphical user interface; options related to other disciplines such as thermal, fluid and electromagnetics are grayed out. This helps a little while navigating the vast menu options within ANSYS.

Click ***OK*** to close the ***Preferences*** dialog box.

Go to Step 2: Specify element type and constants

See and rate the complete Learning Module

Go to all ANSYS Learning Modules