

ANSYS 11 - Crank 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

Step 1: Start-up and preliminary set-up

Start ANSYS

Create a folder called `crank` 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

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.

Specify `crank` 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 `crank.db` in your working directory.

For this tutorial, we'll use the default values for the other fields. Click **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 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:

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

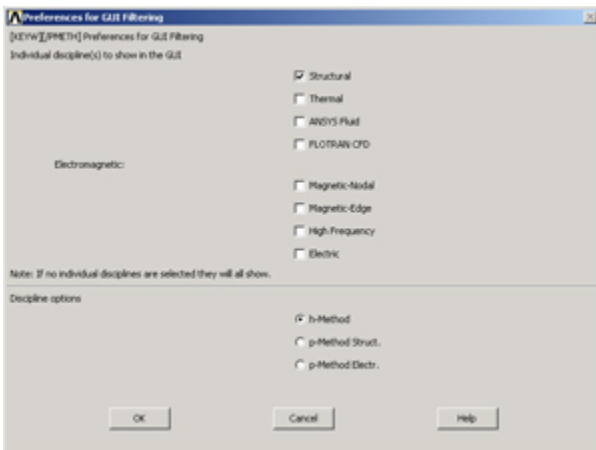
In Mozilla Firefox, select **Menubar > View > Zoom**.

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.

[Go to Step 2: Specify element type and constants](#)

[See and rate the complete Learning Module](#)

[Go to all ANSYS Learning Modules](#)