ANSYS - 3D Curved Beam 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

Create a folder

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

Start ANSYS

Start > All Programs > ANSYS 11.0 > ANSYS Product Launcher

Enter the location of the folder cbeam that you just created as your Working Directory by browsing to it.

Enter cbeam as your Job Name. So all files generated during this ANSYS session will have cbeam as the prefix. Click on Run.

Resize the ANSYS windows as shown in this snapshot so that you can read instructions in your browser window and implement them in ANSYS.

Set Preferences

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.

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

Go to Step 2: Specify element type and constants

See and rate the complete Learning Module

Go to all ANSYS Learning Modules