

GLOBAL
EDITION



Finite Element Analysis

Theory and Application with ANSYS

FOURTH EDITION

Saeed Moaveni

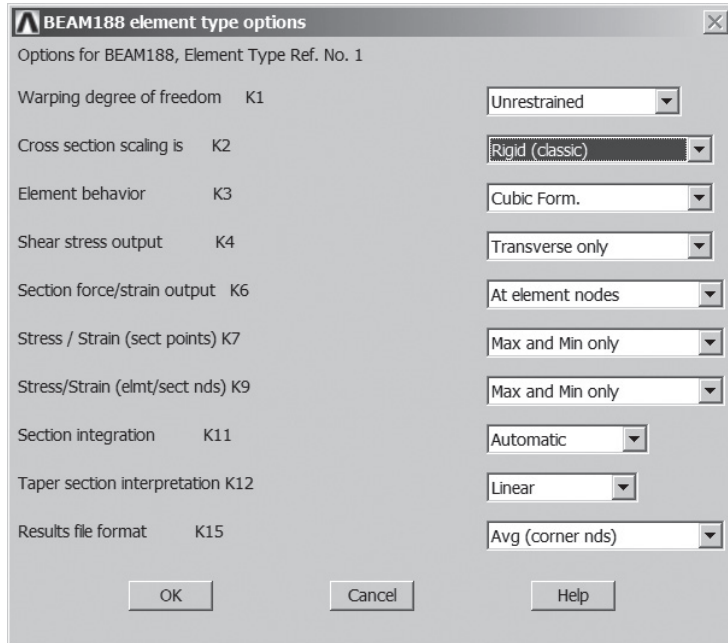


ALWAYS LEARNING

PEARSON

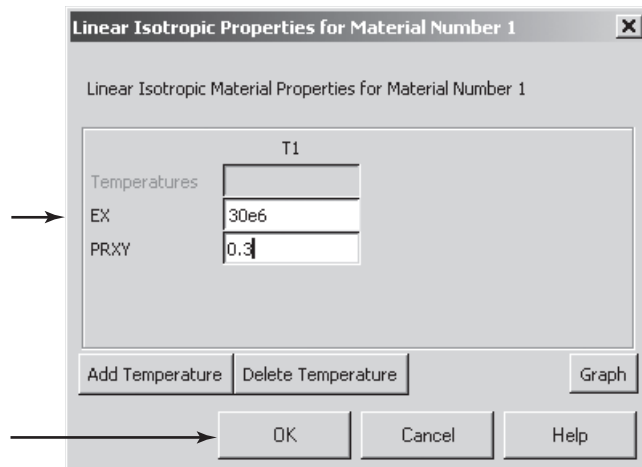
FINITE ELEMENT ANALYSIS

Next click on **Options . . .** button and set K1, K2, . . . , K15 option as shown.



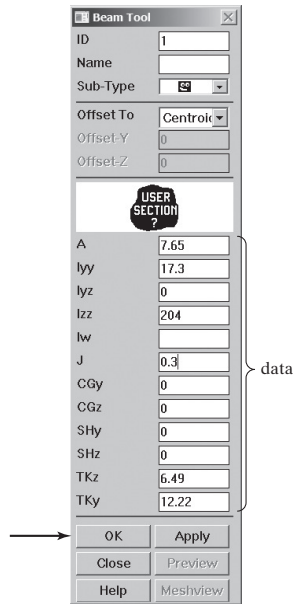
Assign the modulus of elasticity by using the following commands:

main menu: **Preprocessor** → **Material Props** → **Material Models** →
Structural → **Linear** → **Elastic** → **Isotropic**



Close the “Define Material Model Behavior” window.

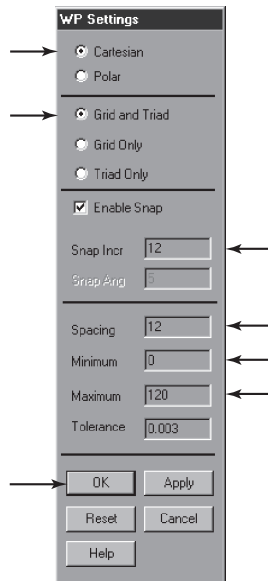
main menu: **Preprocessor** → **Sections** → **Beam** → **Common Sections**



ANSYS Toolbar: **SAVE_DB**

Set up the graphics area (i.e., work plane, zoom, etc.) with the following commands:

utility menu: **Workplane** → **WP Settings ...**



utility menu: **Workplane** → **Display Working Plane**

Bring the workplane to view by the command

utility menu: **PlotCtrls** → **Pan, Zoom, Rotate . . .**

Click on the small circle until you bring the workplane to view. Then create the nodes and elements:

main menu: **Preprocessor** → **Modeling** → **Create** → **Nodes**
→ **On Working Plane**



[WP = 0,108]



[WP = 120,108]



[WP = 120,0]

OK

main menu: **Preprocessor** → **Modeling** → **Create** → **Elements** →
Auto Numbered → **Thru Nodes**



[pick node 1]



[pick node 2]



[apply anywhere in the ANSYS graphics window]



[pick node 2]



[pick node 3]



[anywhere in the ANSYS graphics window]

OK

utility menu: **Plot** → **Elements**

Toolbar: **SAVE_DB**

Apply boundary conditions with the following commands:

main menu: **Solution** → **Define Loads** → **Apply** → **Structural** →
Displacement → **On Nodes**



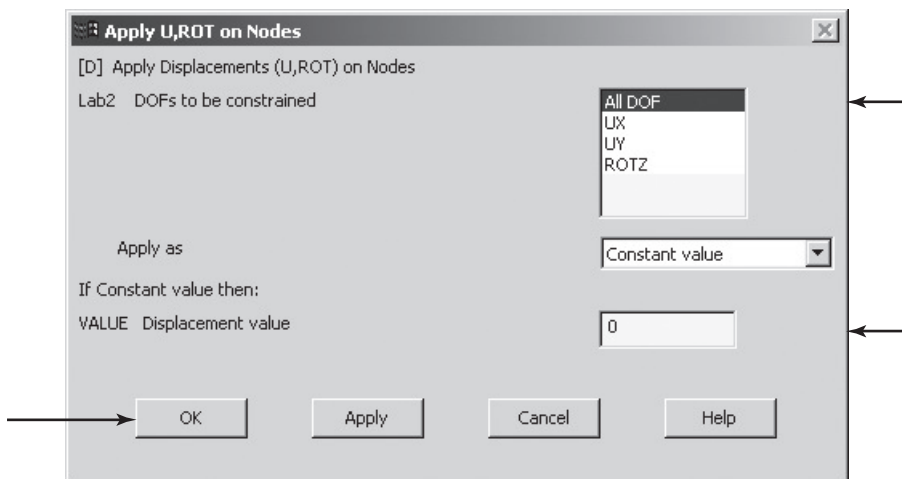
[pick node 1]



[pick node 3]



[anywhere in the ANSYS graphics window]



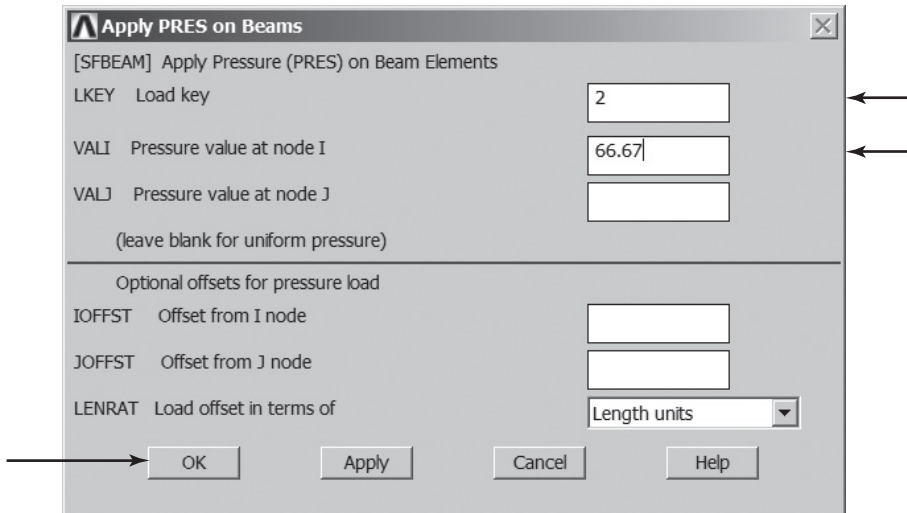
main menu: **Solution** → **Define Loads** → **Apply** → **Structural** →
Pressure → **On Beams**



[pick element 1]



[anywhere in the ANSYS graphics window]

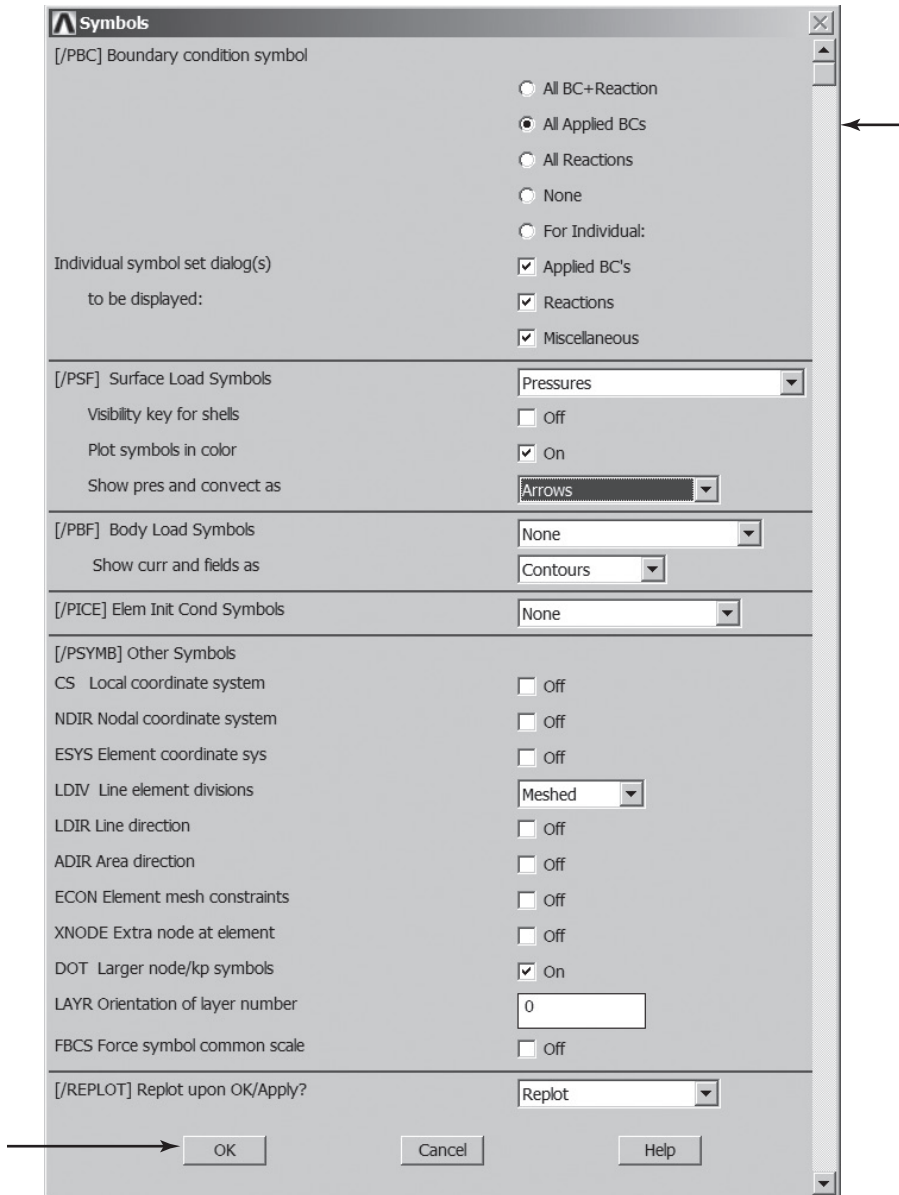


To see the applied distributed load and boundary conditions, use the following commands:

utility menu: **PlotCtrls** → **Symbols . . .**

utility menu: **Plot** → **Elements**

ANSYS Toolbar: **SAVE_DB**



Solve the problem:

main menu: **Solution** → **Solve** → **Current LS**

OK