ANSYS - Vibration Analysis of a Frame - Step 2

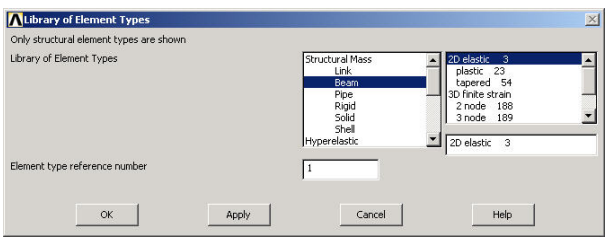
Step 2: Specify element type and constants

Specify Element Type

In the *Preprocessor* Menu, Select:

Element Type > Add/Edit/Delete > Add...

Pick ***Beam*** in the left field and ***2D elastic 3*** in the right field.



Click ***OK***.

Close the *Element Types* dialog box and also the *Element Type* menu.

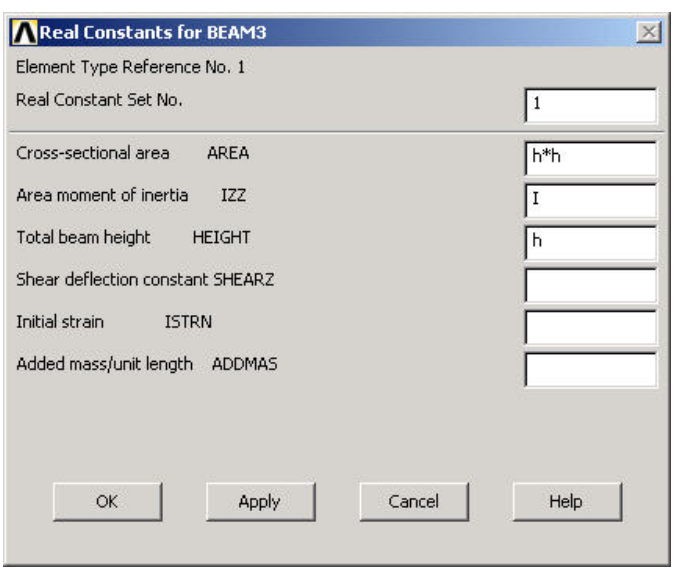
Specify the Constants

In the *Preprocessor* menu, Select*:*

Real Constants > Add/Edit/Delete > Add...

This brings up the *Element Type for Real Constants* dialog box with a list of the element types defined in the previous step. Click ***OK*** to select the ***BEAM3*** element. Enter the following values:

AREA = h*h
IZZ = I
HEIGHT = h



Save your work by clicking on the ***Save_DB*** button in the *ANSYS Toolbar.*

Go to Step 3: Specify material properties

Go to all ANSYS Learning Modules