

ANSYS - Vibration Analysis of a Frame - Step 2

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

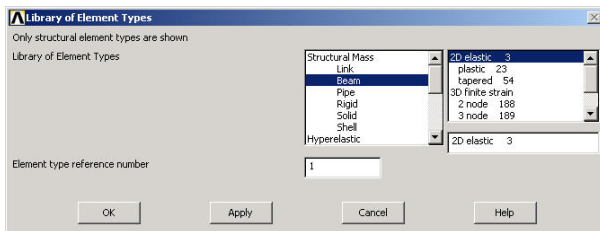
Step 2: Specify element type and constants

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*:*

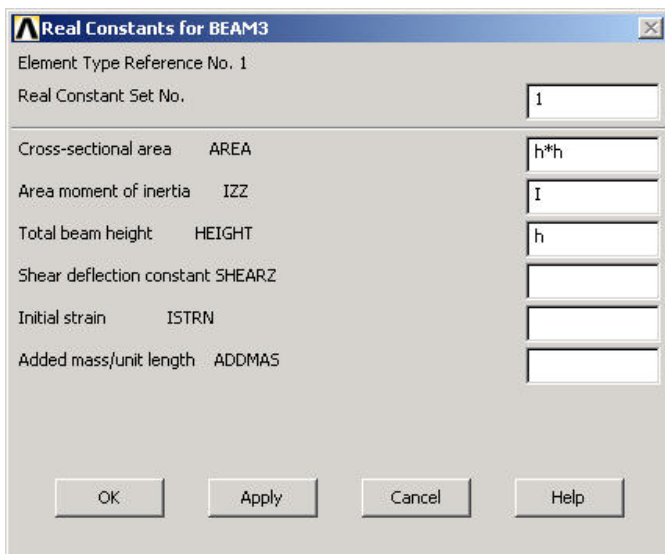
Real Constants > Add/Edit/Delete > Add...

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](#)