- **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.

Library of Element Types		×
Only structural element types are shown		
Library of Element Types	Structural Mass Link Bean Pipe Rigid Shel Hyperelastic 2 D elastic 2 D elastic 2 D elastic 2 agered 3 O finite stra 1 n node 1 2 D elastic 2 node 1 3 node 1 2 D elastic 2 node 1 3 node 1 2 D elastic 2 node 1 3 node 1	3 54 in 88 89 3
Element type reference number	1	
OK	Cancel	db

#### 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* = 1 *HEIGHT* = h

ziement Type Reference No. 1	
Real Constant Set No.	1
Cross-sectional area AREA	h*h
Area moment of inertia IZZ	I
Total beam height HEIGHT	h
5hear deflection constant SHEARZ	
initial strain ISTRN	
Added mass/unit length ADDMAS	
	ancel Help

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