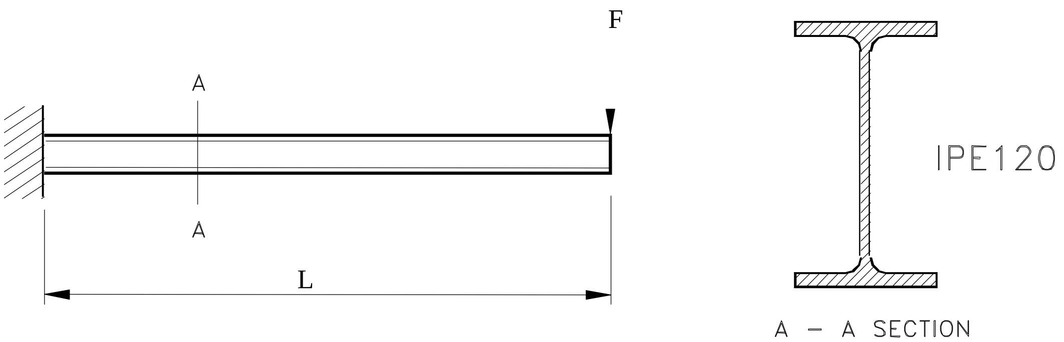
1. Stress Analysis of a Cantilever Steel Beam

Applicable CivilFEM Product:	All CivilFEM Products
Level of Difficulty:	Easy
Interactive Time Required:	15-20 minutes
Discipline:	Structural Steel
Analysis Type:	Linear static
Element Type Used:	BEAM3
Active code:	Eurocode 3
Units System:	N, m, s
CivilFEM Features Demonstrated:	Units selection, code selection, material definition, section definition from library, postprocessing of forces and stresses

Problem Description

This problem analyzes the forces and stresses in a steel cantilever beam. Such a beam is subjected to a vertical force of 2500 newton at its free end. No plasticity effects are taken into account.



■ Given

The geometry and loads of the cantilever beam are shown in the previous figure. The beam is a hot rolled shape IPE 120 and is made of Fe430 steel. The following is a list of all the input parameters:

Material	Fe 430
Section type	Hot Rolled IPE 120
Length	L = 2 m
Load	F = 2500 N

Approach and Assumptions

We will discretize the beam with elastic 2D beam elements. Model geometry is defined with elements and nodes.

Summary of Steps

Preprocessing

1. Specify title
2. Set code
3. Set units
4. Define material
5. Define element type
6. Define section
7. Define Beam properties
8. Define Nodes and Elements
9. Save the database

Solution

10. Apply displacement constraints
11. Apply force load
12. Solve

Postprocessing

13. Enter the postprocessor and read results
14. Plot bending moment
15. Plot the bending stress in Y top
16. List bending extreme stresses
17. Plot bending stress distribution inside the cross-section
18. Exit the ANSYS program

Interactive Step-by-Step Solution

■ Preprocessing

A typical CivilFEM analysis begins providing data such as the units system, active code, materials, element types, section and model geometry definition.

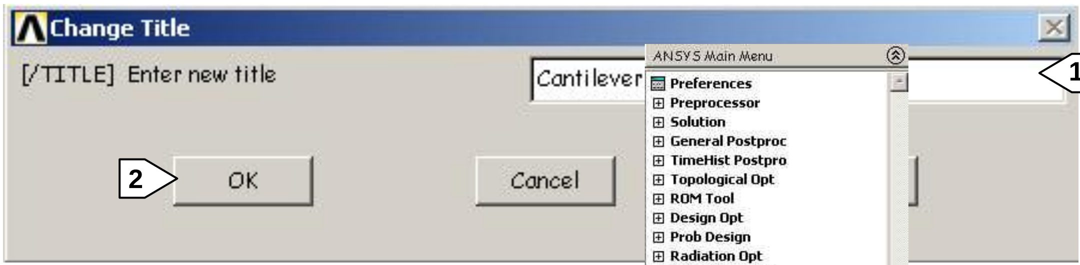
1. Specify title

Although this step is not required for a CivilFEM analysis, we recommend that you make it part of all your analyses.

Utility Menu: **File→Change title**

1 Enter the title: "Cantilever Steel Beam"

2 OK to define the title and close the dialog box.

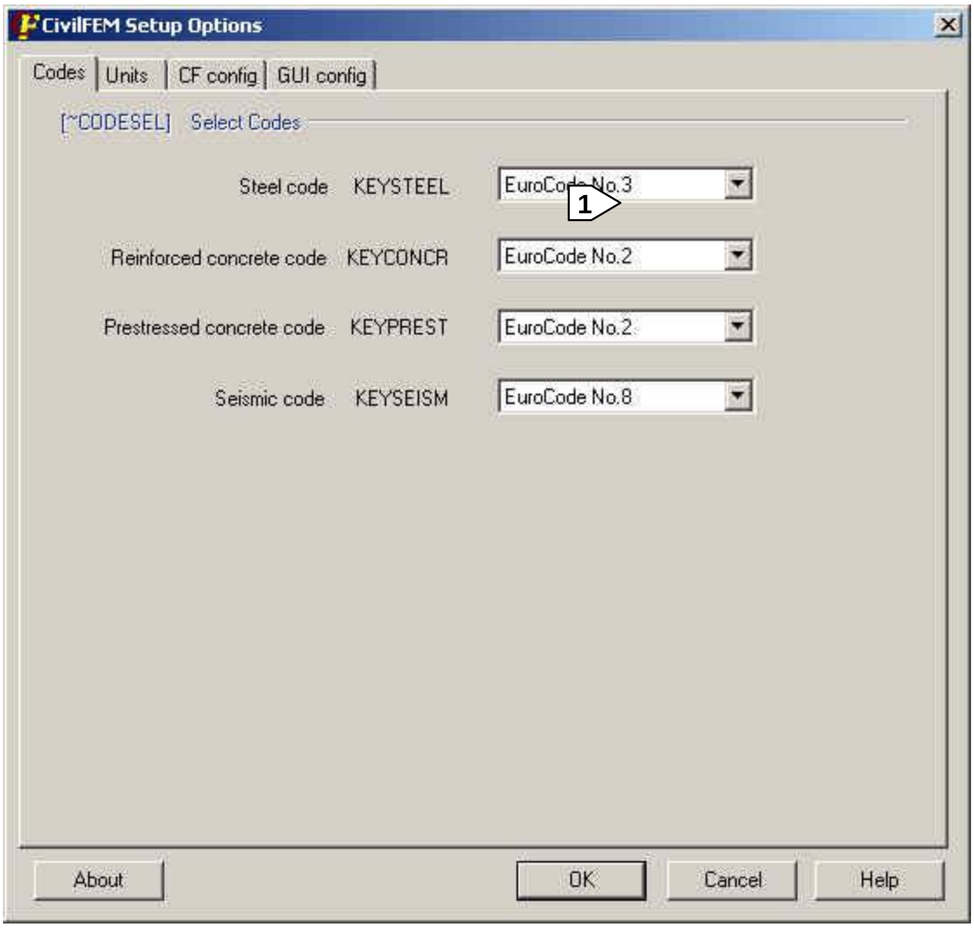


2. Set code

In CivilFEM you can choose between differen[t] g and designing. CivilFEM allows you to uphold diffe simultaneously, one for concrete calculations another one for and a third one for seismic design. In this example the active 3, which is the default option.

Main Menu: – CivilFEM – **Civil Setup**

1 Select Civil Setup



2 to set active code and close the code dialog box

3. Set units

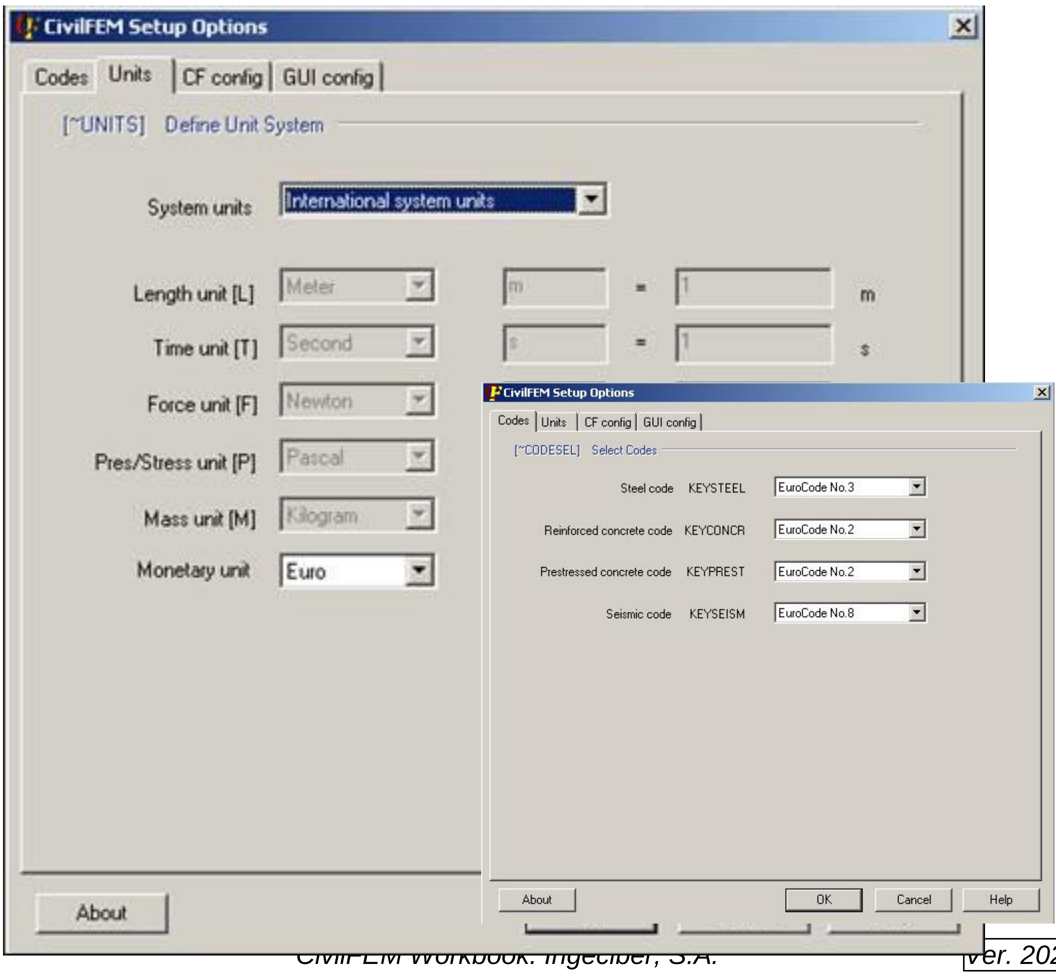
In CivilFEM you must define a unit system. CivilFEM will need such system to perform calculations according to Code. You should maintain it during the entire design. In this analysis, we will select SI units, that is, meters, seconds and newtons.

Main Menu: – CivilFEM – **Civil Setup**

1 Choose Units Library

1

2 OK to accept units and close the units dialog box



2

4. Define material

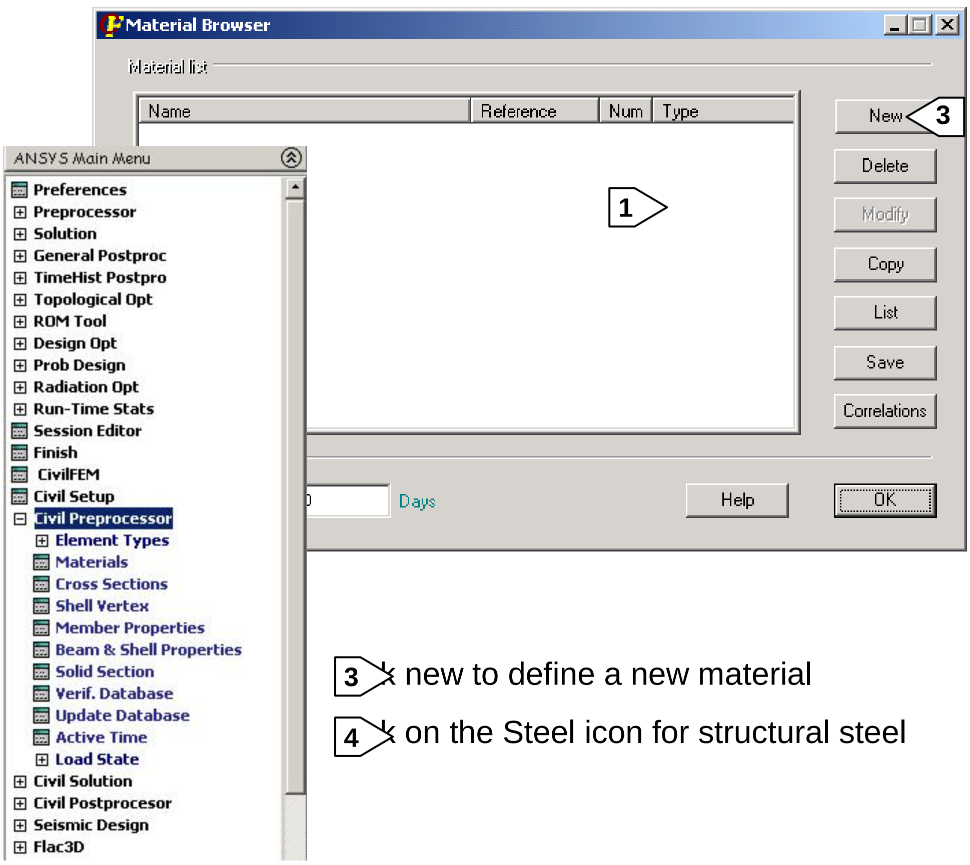
Material properties definition is performed using the CivilFEM **~CFMP** command. This command automatically defines the ANSYS material properties (density, Young's modulus, Poisson's ratio and thermal expansion coefficient) and the CivilFEM material properties necessary for code checking. In this case we will select Fe 430 steel.

The CivilFEM **~CFMP** command allows us to define stress-strain diagrams, to define safety coefficients, to control the linear or non-linear behavior of the material and to select the activation time of the material.

Main Menu: – CivilFEM – **Civil Preprocessor →Materials**

1 Select Civil Preprocess

2 Choose Materials



3 k new to define a new material

4 k on the Steel icon for structural steel

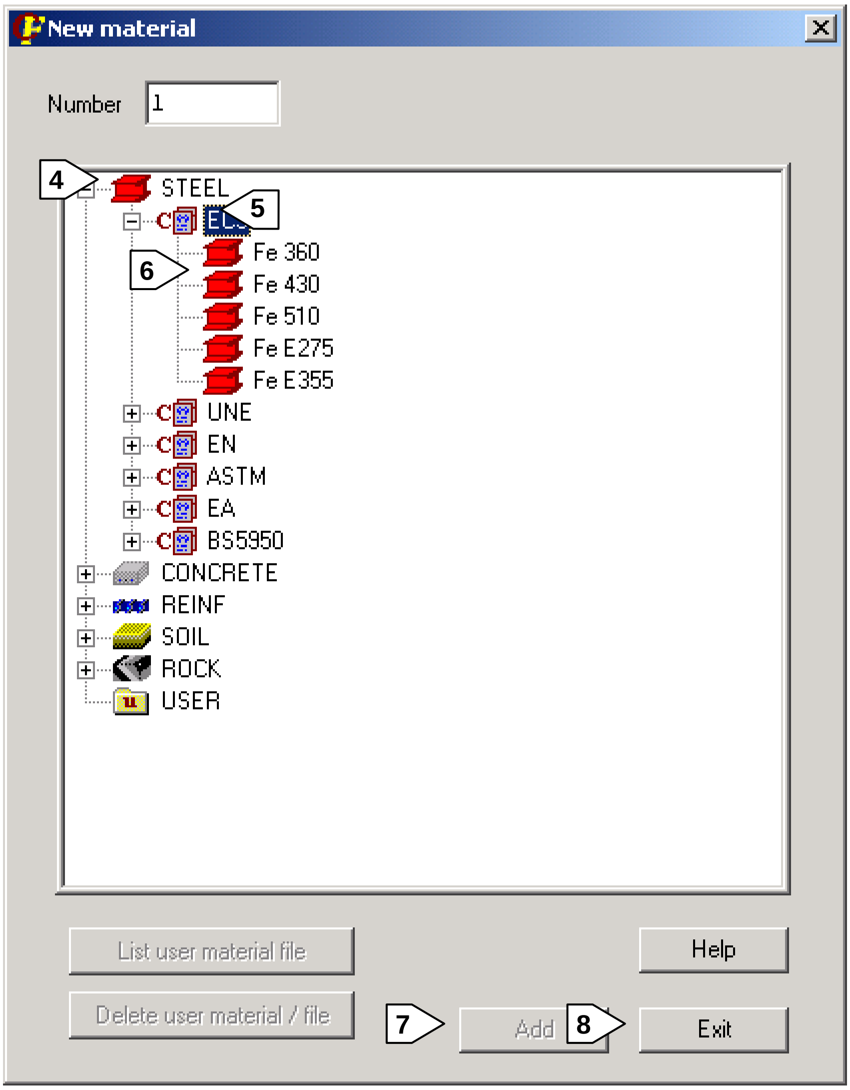
5 Pick on the EC3 icon to choose steel from EC3 code

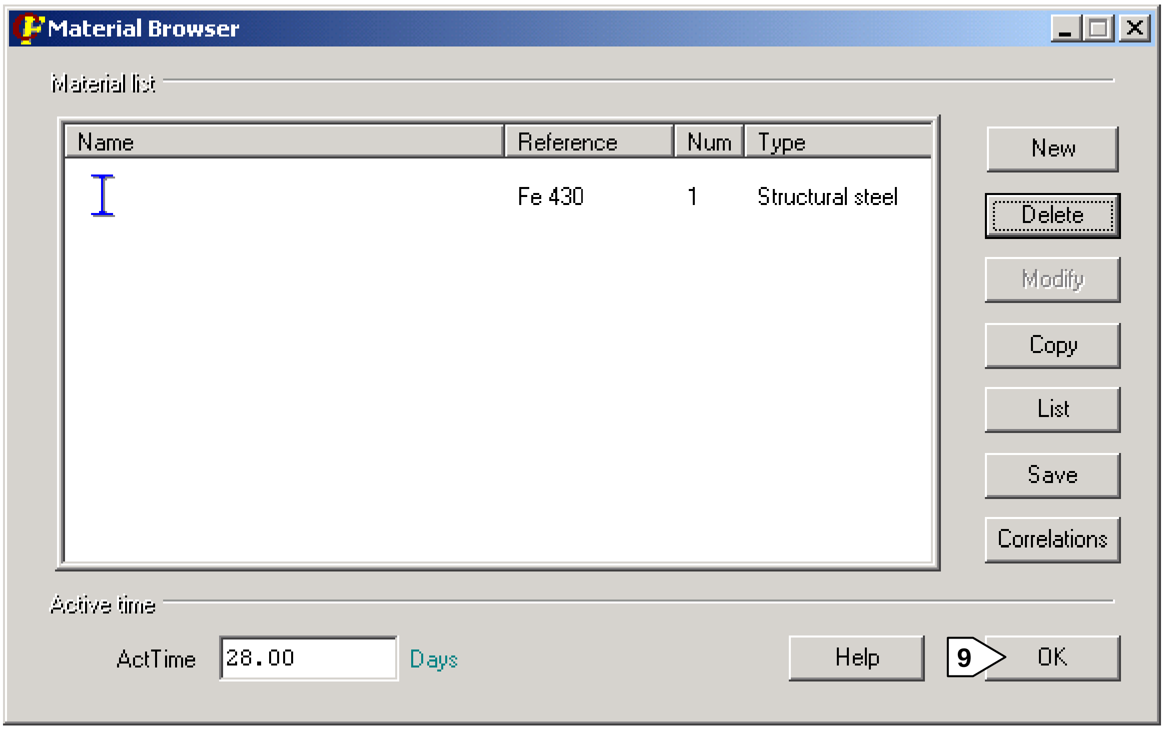
6 Choose Fe430 Steel and all the material properties corresponding to Fe430 steel are automatically calculated according to Eurocode 3 (active code)

7 Add to define the material properties set

8 Exit to close the dialog box

9 OK





5. Define element type

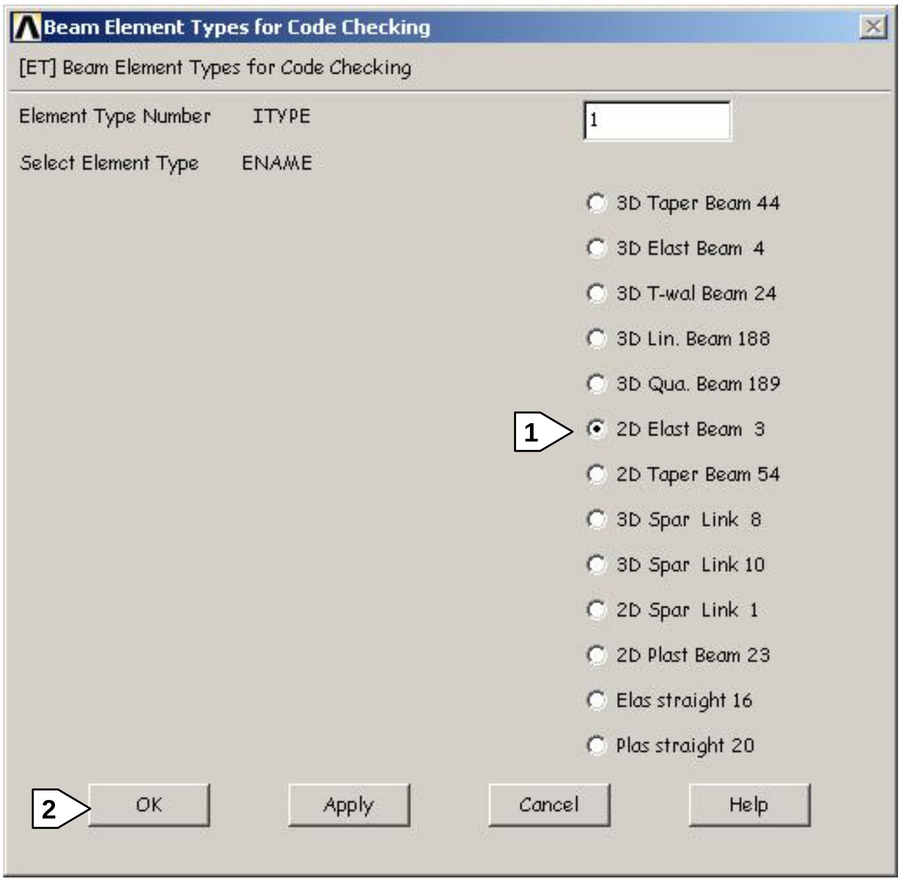
Checking and designing according to codes is performed only on CivilFEM supported element types, although you can use any ANSYS element to define your model, only the CivilFEM supported elements will be checked according to codes. In the element type menu you can see the CivilFEM supported beam elements.

We will use 2D Elastic Beam 3 for this analysis.

Main Menu: – CivilFEM – **Civil Preprocessor → Element Types →Civil Beams**

1 Select 2D Elastic Beam 3

2 OK to define element type



6. Define section

CivilFEM allows an automatic section definition, calculating its mechanical properties and defining its real constants.

There are three ways to select steel rolled shapes from CivilFEM Library: the graphical selection is an easy way to select a shape from more than 6000, but you can also select it by name or by index (in this case, the shape can be used as a design variable).

We will use IPE 120 and it will be selected from the Library using the *Cross Sections Explorer.*

Main Menu: – CivilFEM – **Civil Preprocessor →Cross Sections**

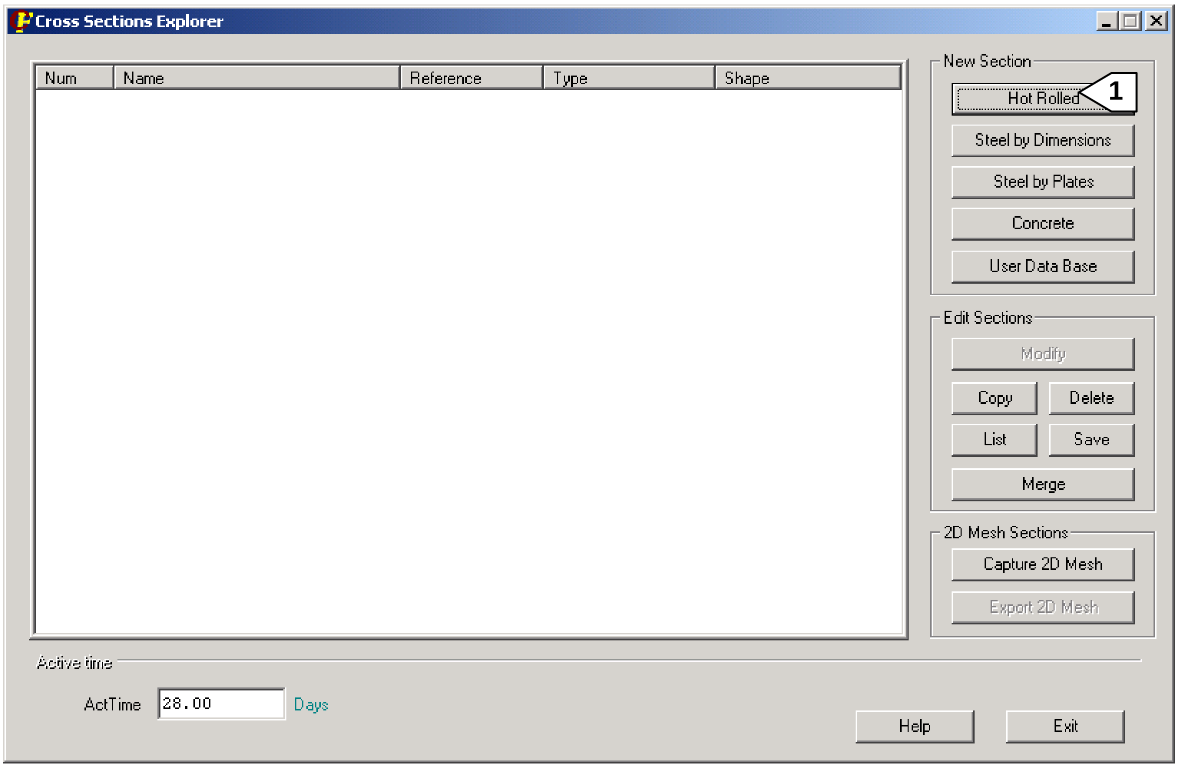
1 Click on the Hot Rolled Button

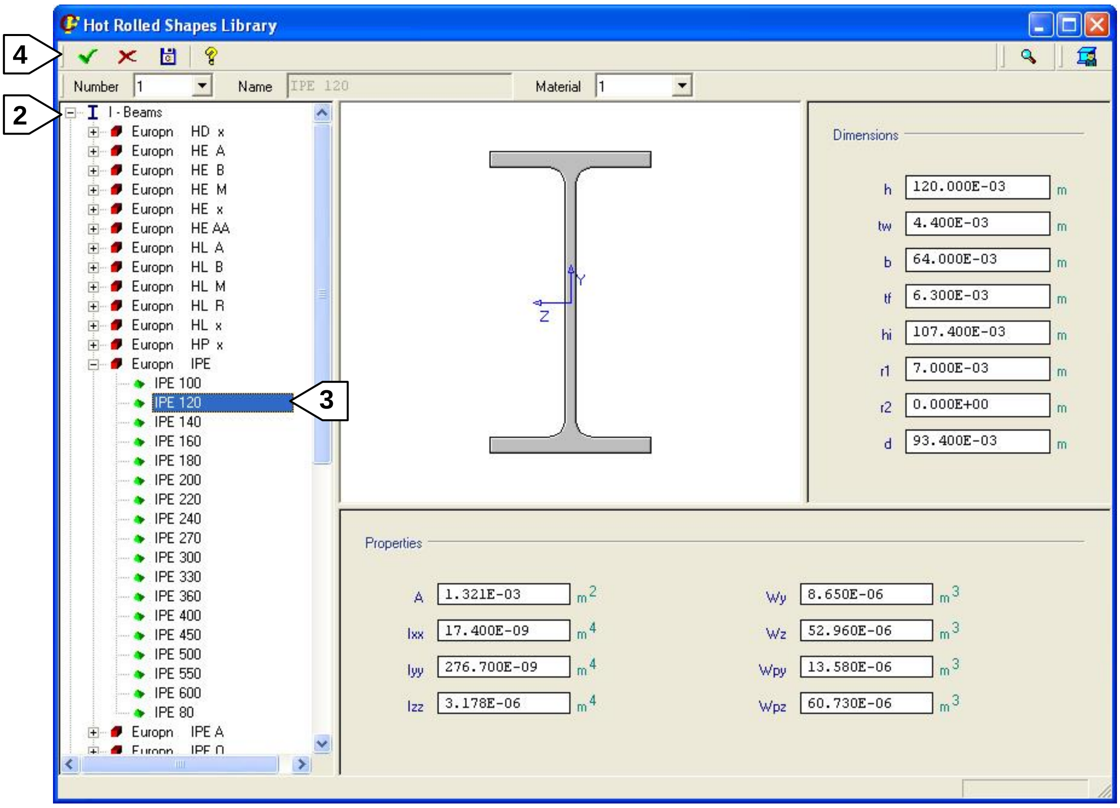
2 Select I Beams group.

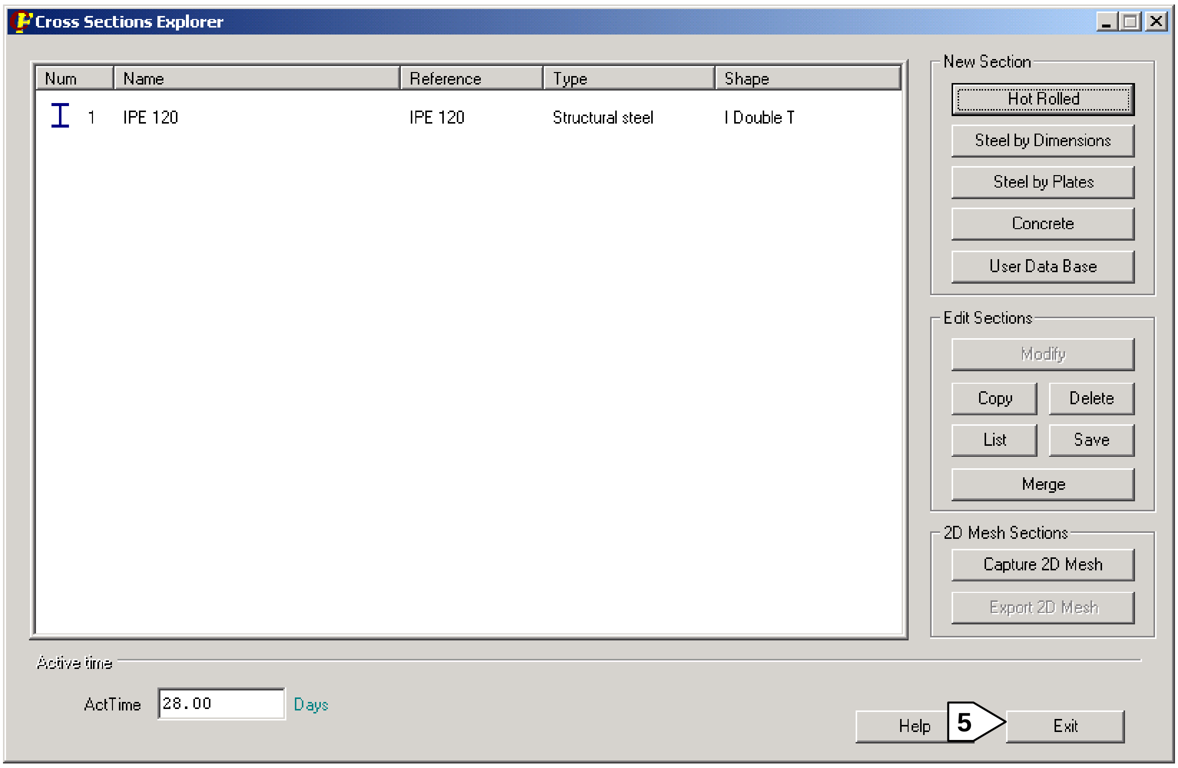
3 Select IPE 120 Shape

4 OK to define cross section 1.

5 Exit to close cross section explorer





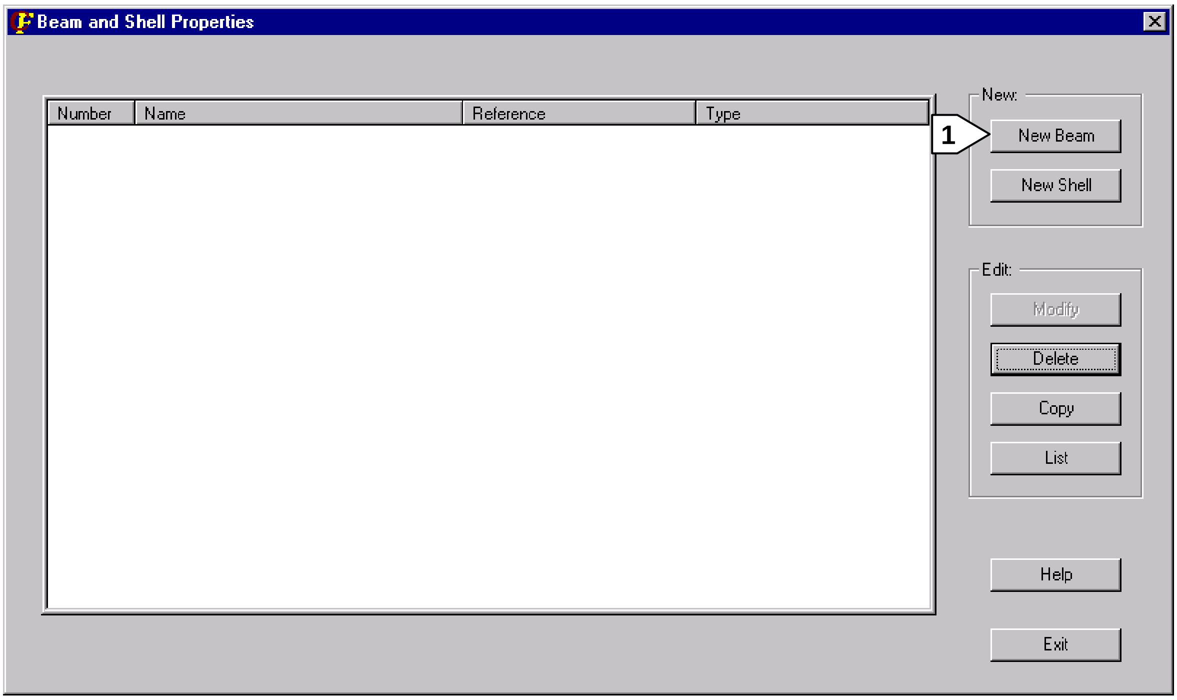


7. Define Beam & Shell properties

CivilFEM command **~BMSHPRO** will be used to define ANSYS real constants. You must take into account that CivilFEM creates a real constant set for every Beam & Shell property and with the same number.

Main Menu: – CivilFEM – **Civil Preprocessor →Beam & Shell pro**

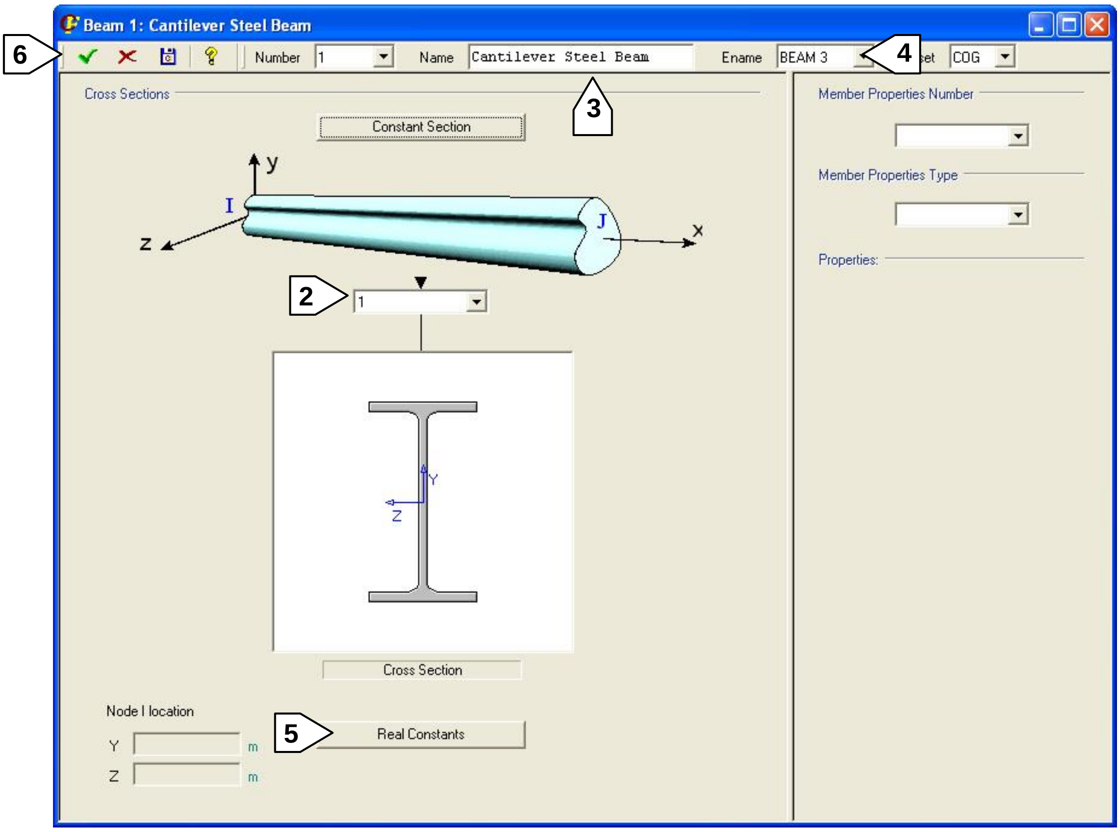
1 Pick on the New Beam button



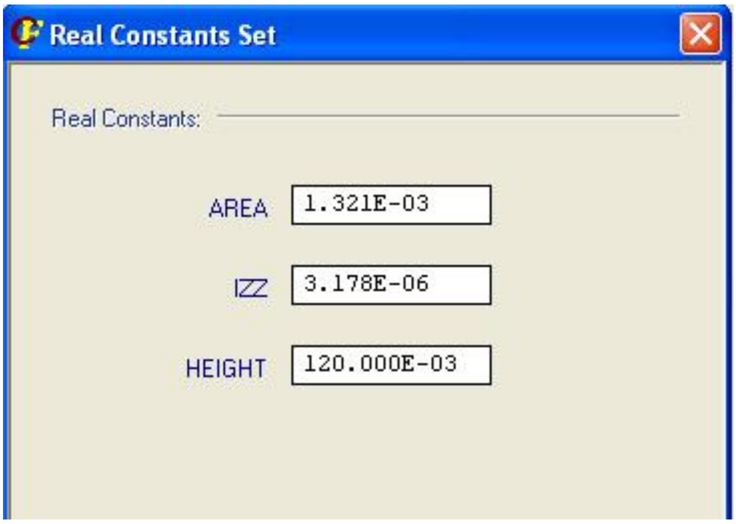
2 Select cross section number

3 Enter a Name for the Beam property

4 Select element type BEAM 3

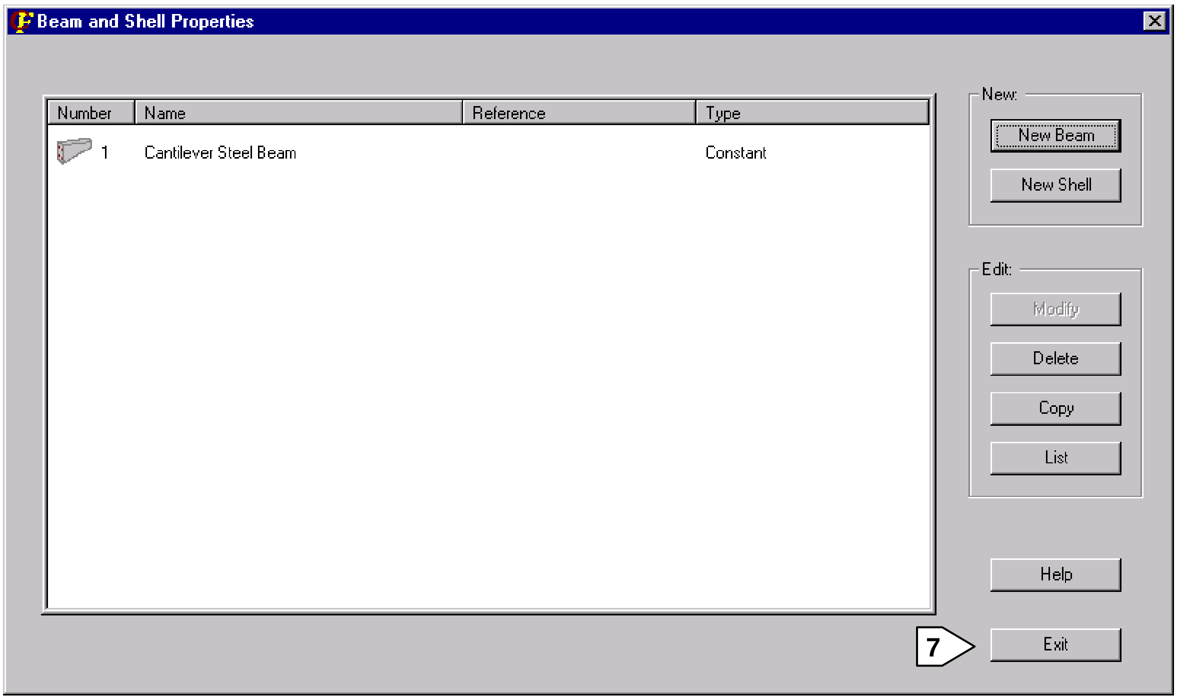


5 You can review ANSYS real constants by picking on the Real Constants button



6 OK to define Beam property on the Beam & Shell properties window

7 EXIT to close Beam & Shell properties window



8. Define nodes and elements

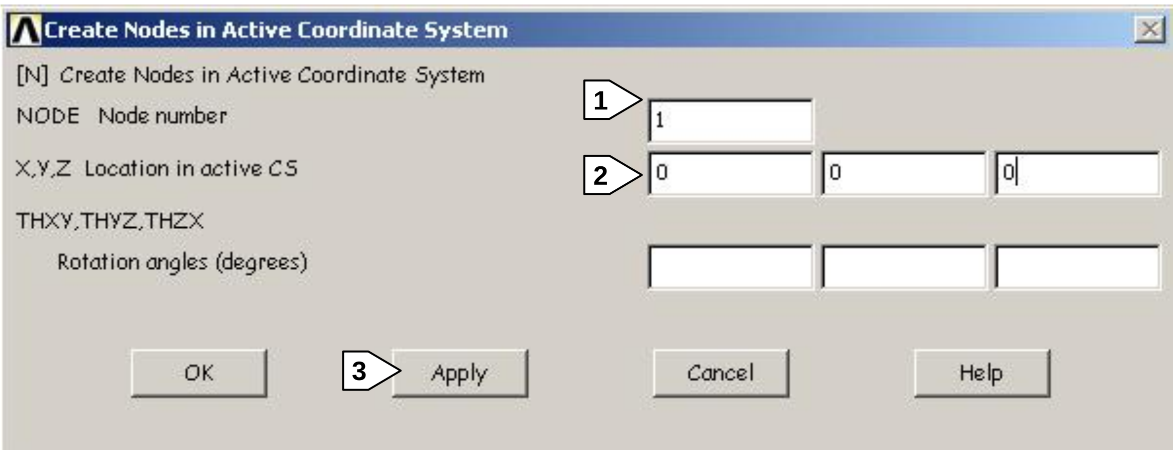
Model geometry will be established through direct elements and nodes generation. We will discretize the cantilever beam with 11 nodes and 10 elements with the ANSYS commands.

Main Menu: **Preprocessor** → – Modeling – **Create** →**Nodes** →**In Active CS**

1 Enter 1 for first node

2 Enter x=0, y=0, z=0 for node 1 coordinates.

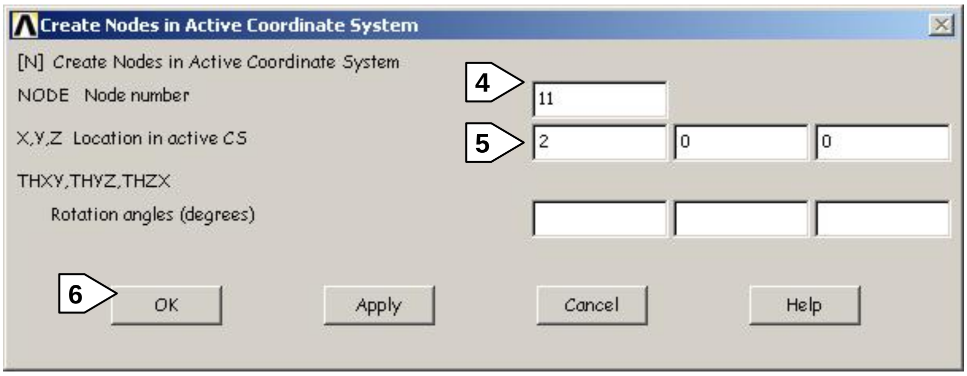
3 Apply to create the first node



4 Enter 11 for last node

5 Enter x=2, y=0, z=0 for node 11 coordinates.

6 OK to create the last node and close the dialog box



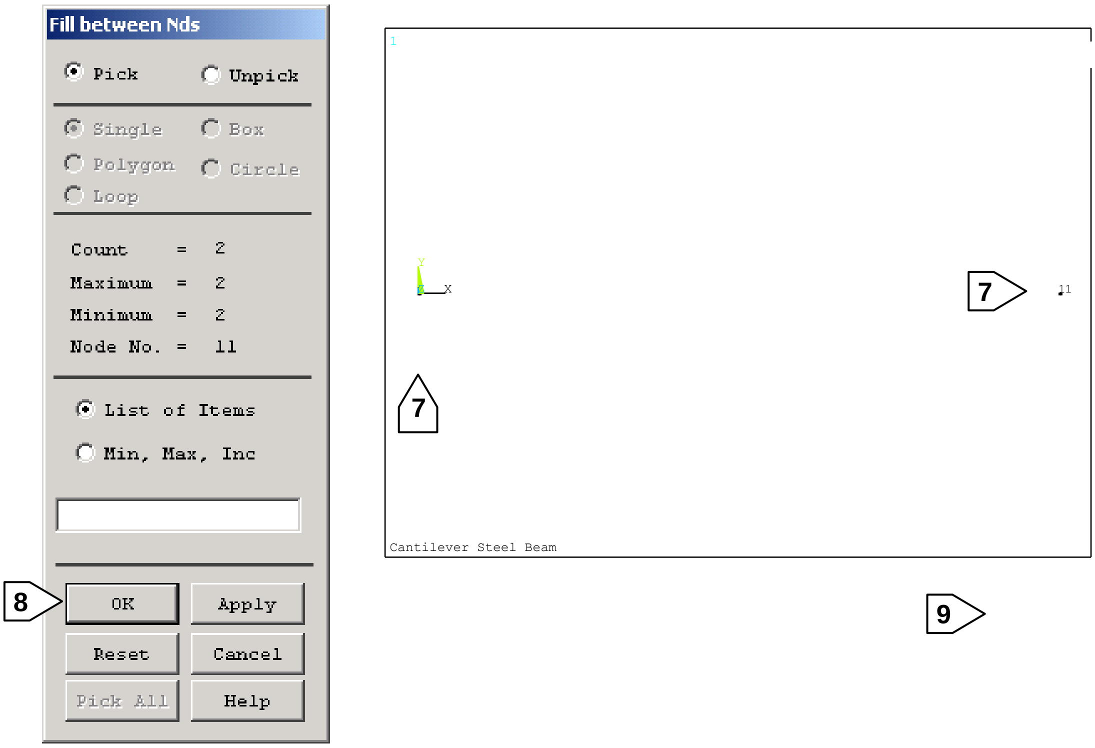
The cantilever ends have been defined. We will fill the rest of the nodes that define our model between these two points.

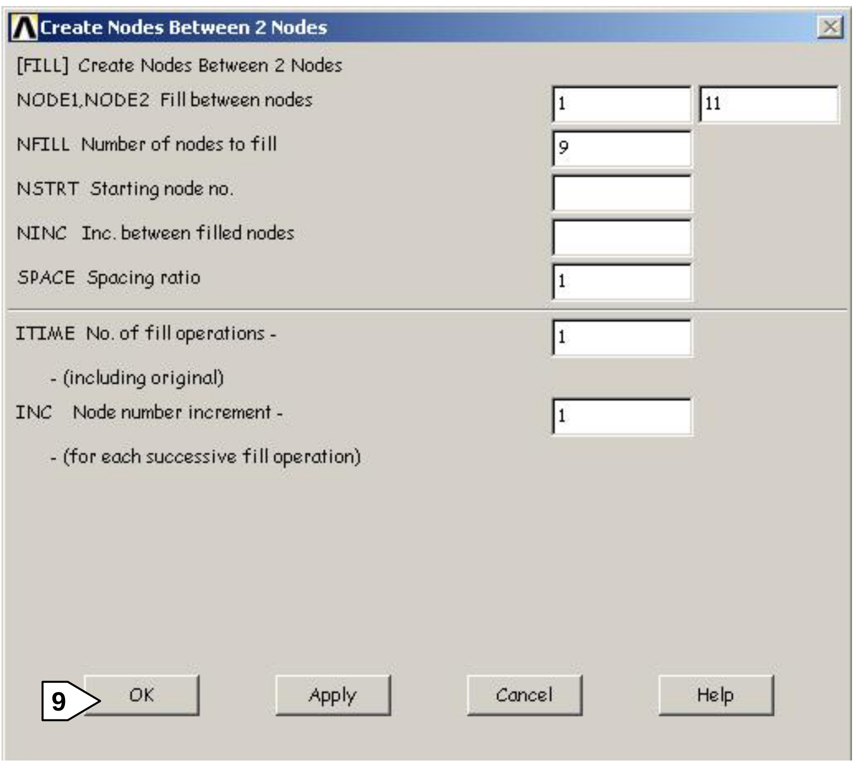
Main Menu: **Preprocessor →** – Modeling – **Create→Nodes→Fill between Nds**

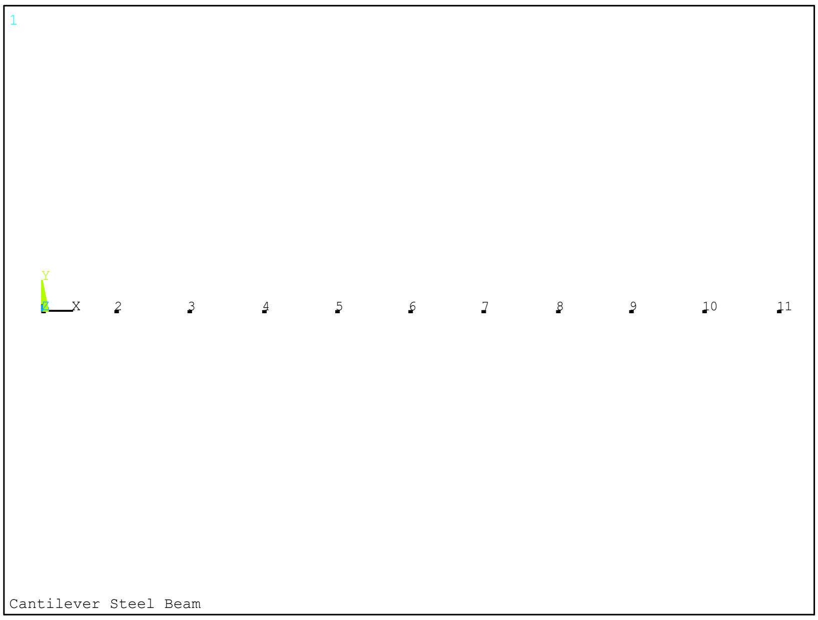
7 Pick node 1 (on the coordinate system origin) and node 11

8 OK to finish picking nodes

9 OK to fill between nodes





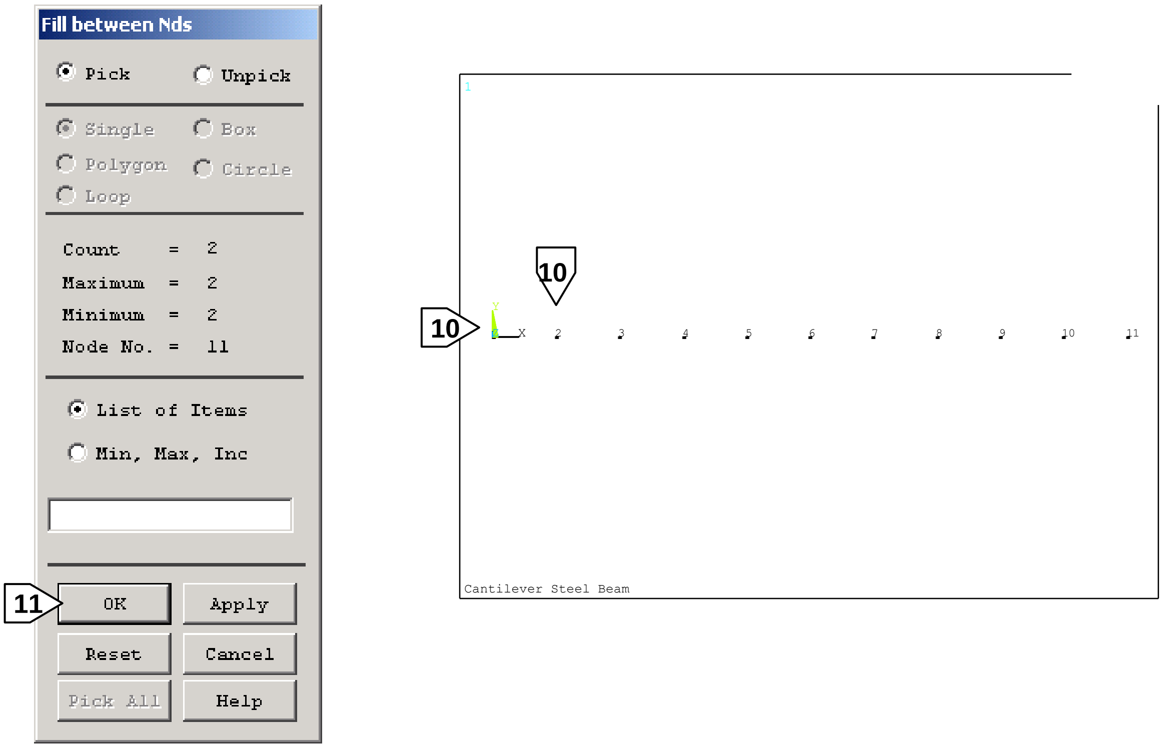


Then, we create the elements by defining the first one and copying the rest.

Main Menu: **Preprocessor →** – Modeling – **Create →Elements →** – Auto numbered – **Through nodes**

10 Pick node 1 (on the coordinate system origin) and node 2

11 OK to finish picking nodes



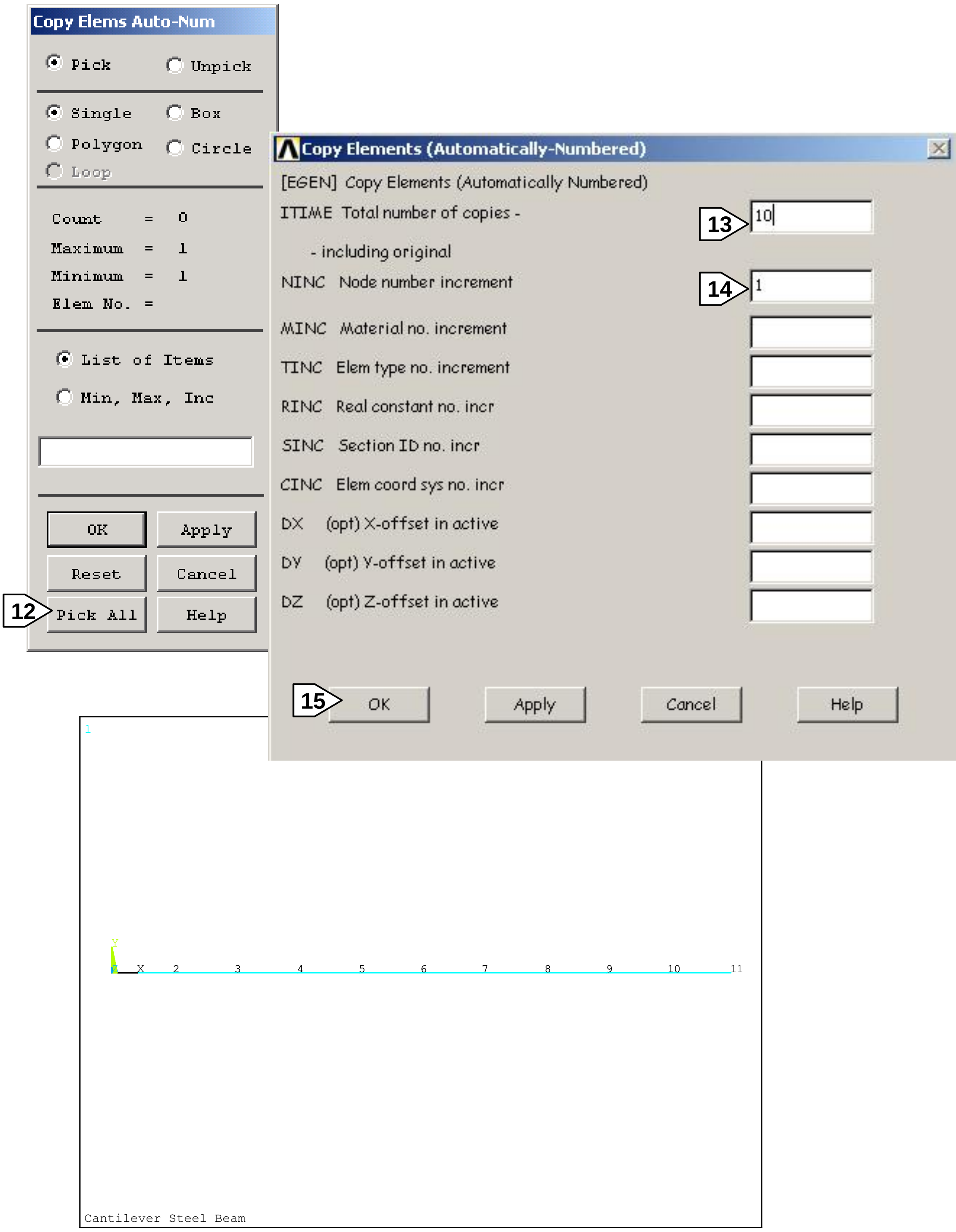
Main Menu: **Preprocessor →** – Modeling – **Copy →** – Elements – **Auto Numbered**

12 Pick All (to select the element)

13 Enter 10 (number of copies including original)

14 Enter 1 (node number increment)

15 OK to generate elements



9. Save the database

Before moving to the next step, we will save all we have done so far. The save operation will save the database to file.db and file.cfdb.

Toolbar: **CFSAVE**

■ Solution

In this step we will define the analysis type and its options, apply loads and initiate the finite element solution. A new, static analysis is the default option, so we will not need to specify the analysis type for this problem. Moreover, there are no analysis options for this problem.

10. Apply displacement constraints

The model is a cantilever beam so you have to constrain all degrees of freedom at node 1.

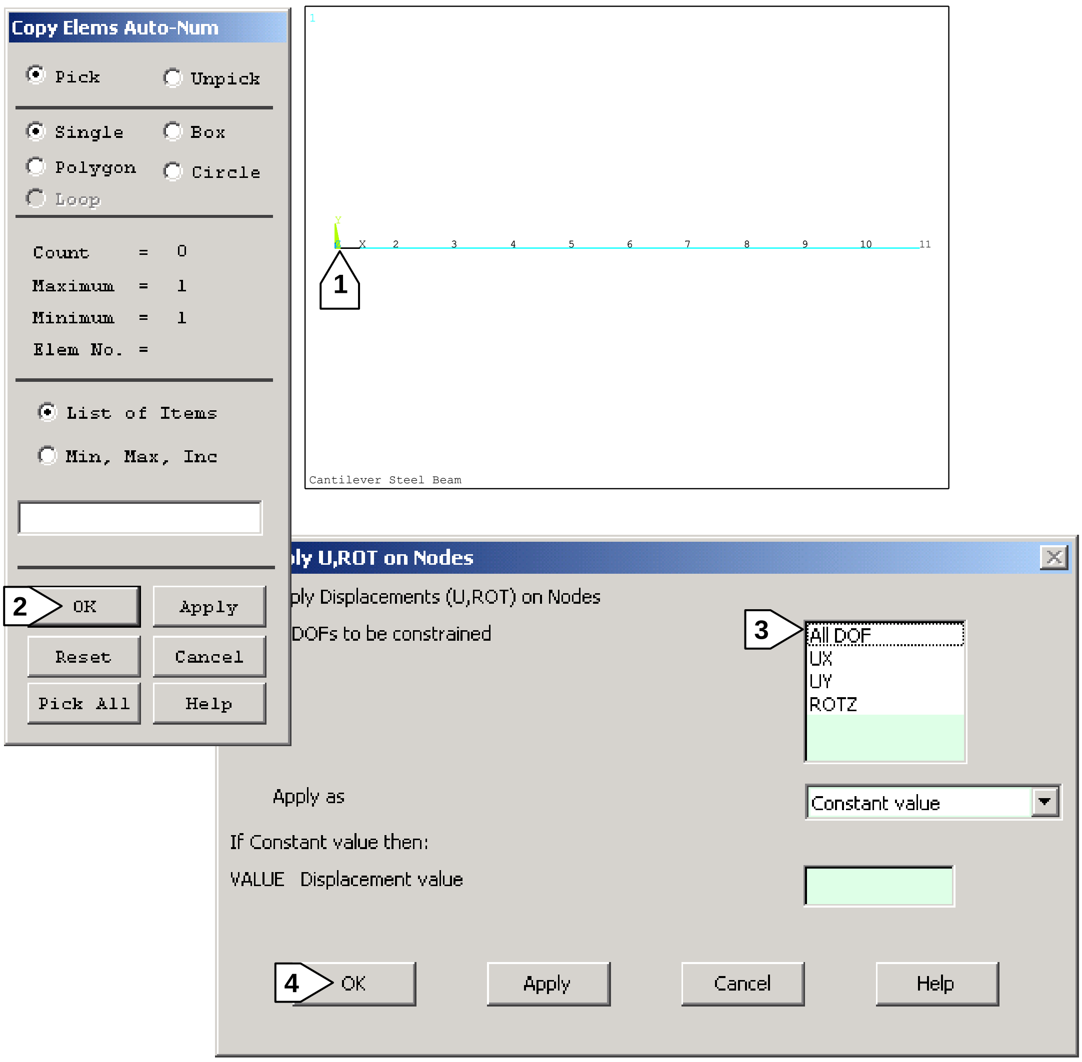
Main Menu: **Solution** → –Define Loads – **Apply** → – Structural – **Displacement** →**On Nodes**

1 Pick node 1

2 OK to finish picking nodes

3 Select All DOF

4 OK to apply constraints (zero displacement for all DOF) and close dialog box

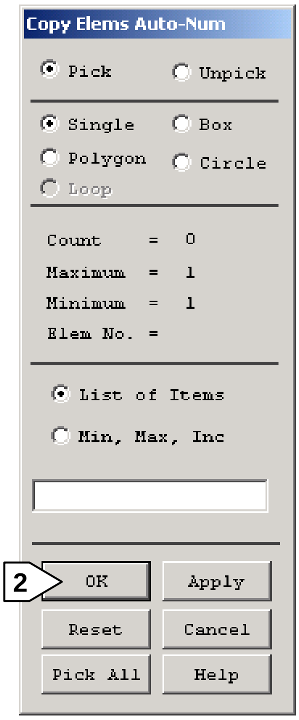


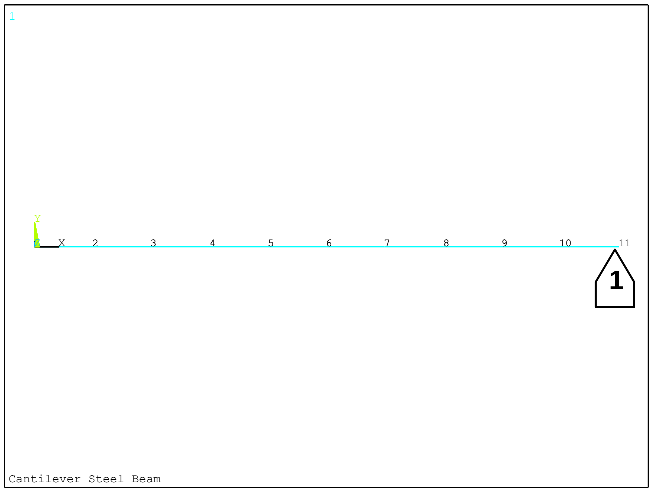
11. Apply force load

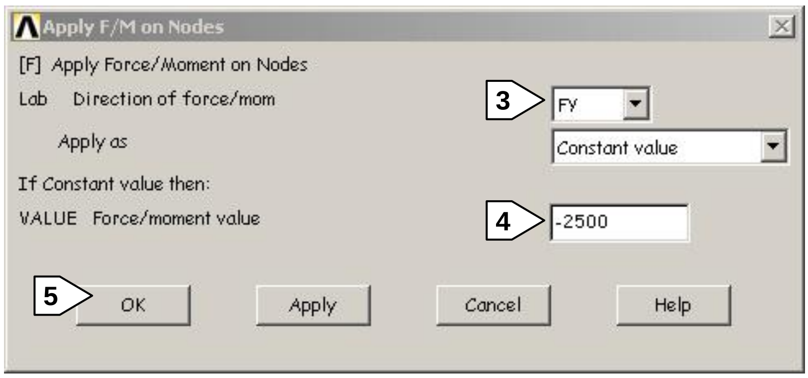
You have to apply a concentrated load at node 11 with a value of 2500 N.

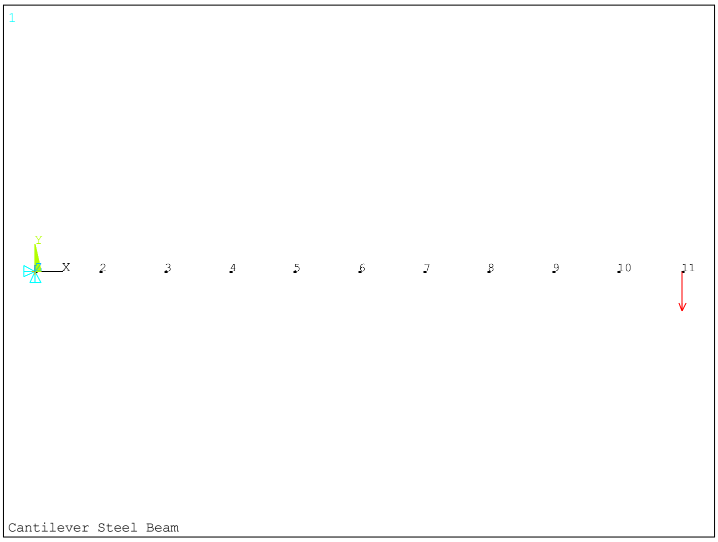
Main Menu: **Solution** → – Define Loads – **Apply** → – Structural – **Force/Moment** →**On Nodes**

1. Pick node 11
2. OK to finish picking nodes
3. Select FY as the force in the Y direction
4. Enter –2500
5. OK to apply force and close dialog box





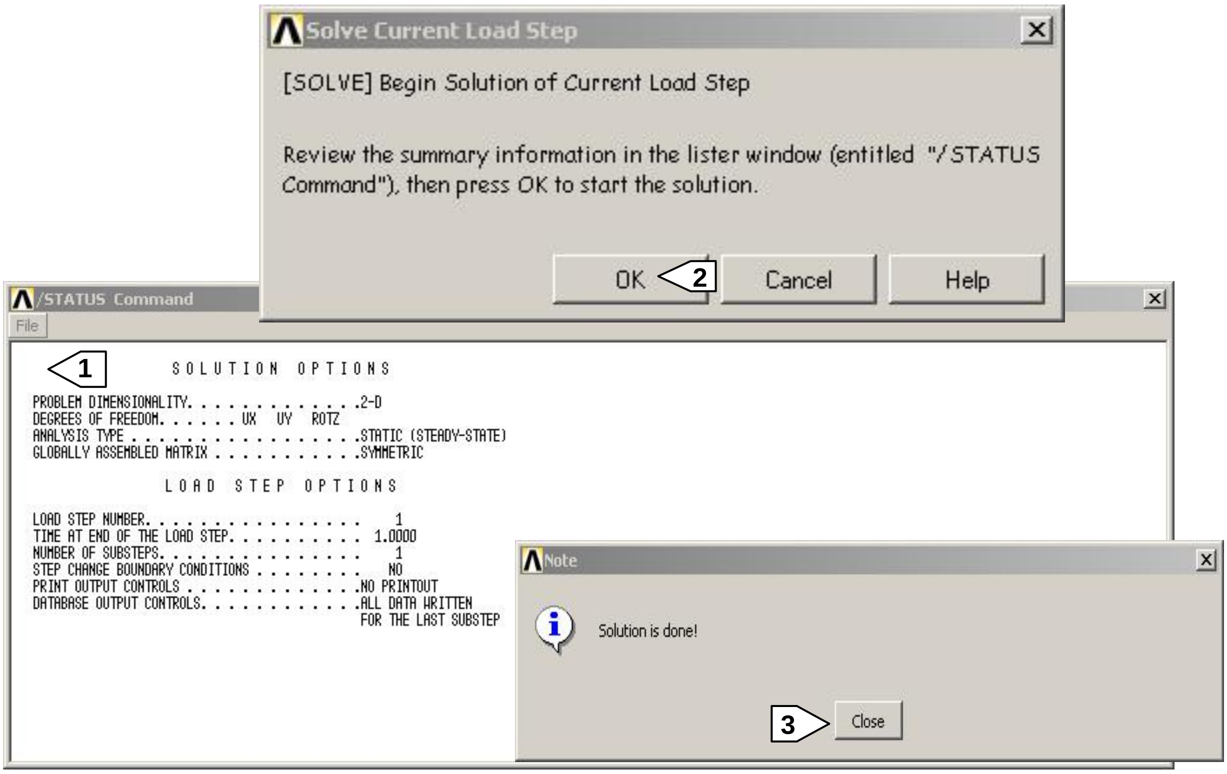




12. Solve

Main Menu: **Solution** → – Solve – **Current LS**

1. Review information in the status window, and then pick File →Close to close the window
2. OK to begin the solution
3. Close the information window when solution is done



■ Postprocessing

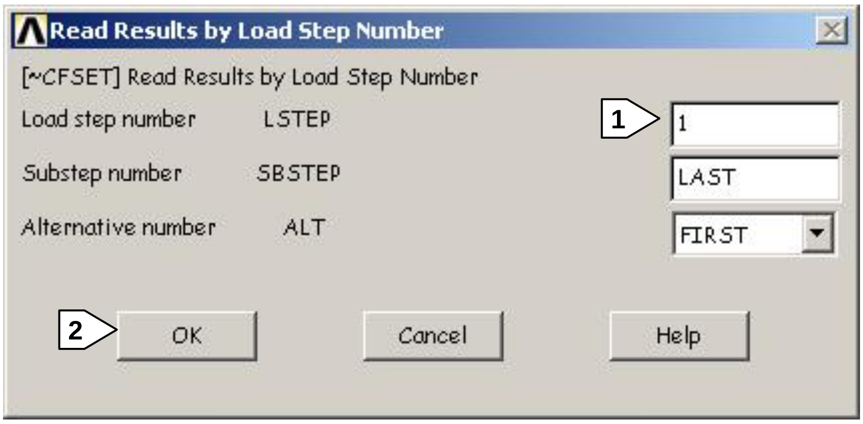
Postprocessing is where you review the analysis results through graphic displays and tabular listings. CivilFEM saves all this data in its own results file called file.RCV. To review its results you must define the dataset to be read from this file using the CivilFEM command **~CFSET**. This command points to both ANSYS and CivilFEM data.

13. Enter the postprocessor and read results

Main Menu: – CivilFEM – **Civil Postproces →Read Results → By Load Step**

1 Enter 1 in the Load Step number box

2 OK to read load step 1

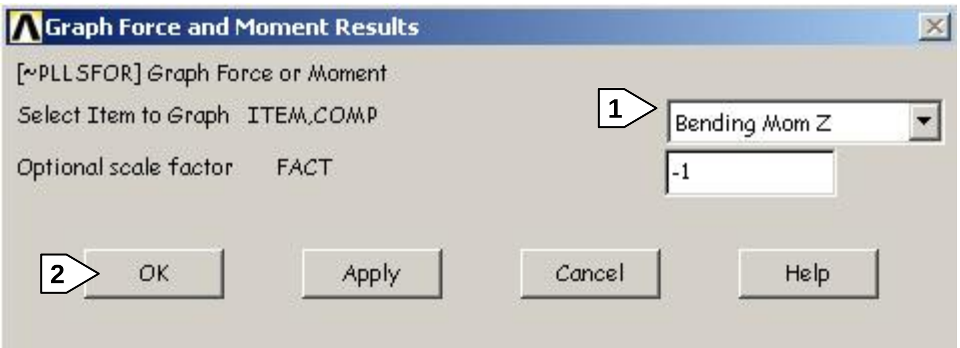


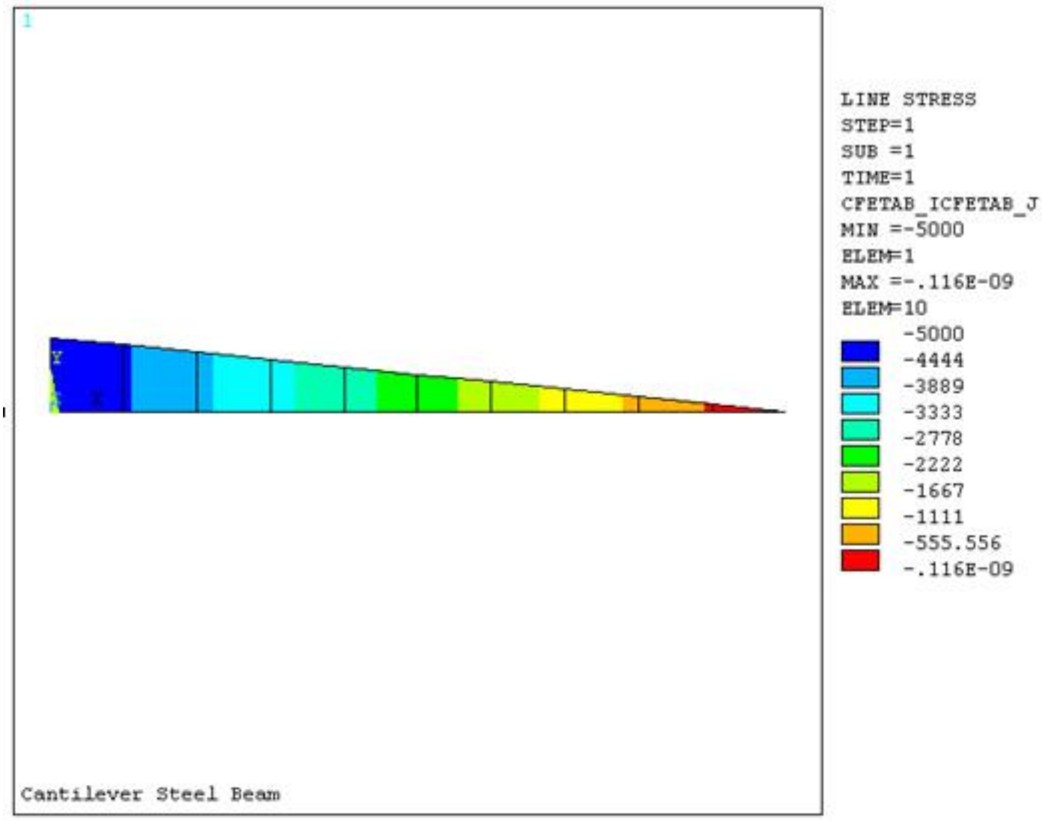
14. Plot bending moment

Main Menu: – CivilFEM – **Civil Postprocess →Beam Utilities →** GRAPH RESULTS: **Forces & Moments**

1 Choose bending moment Z

2 OK



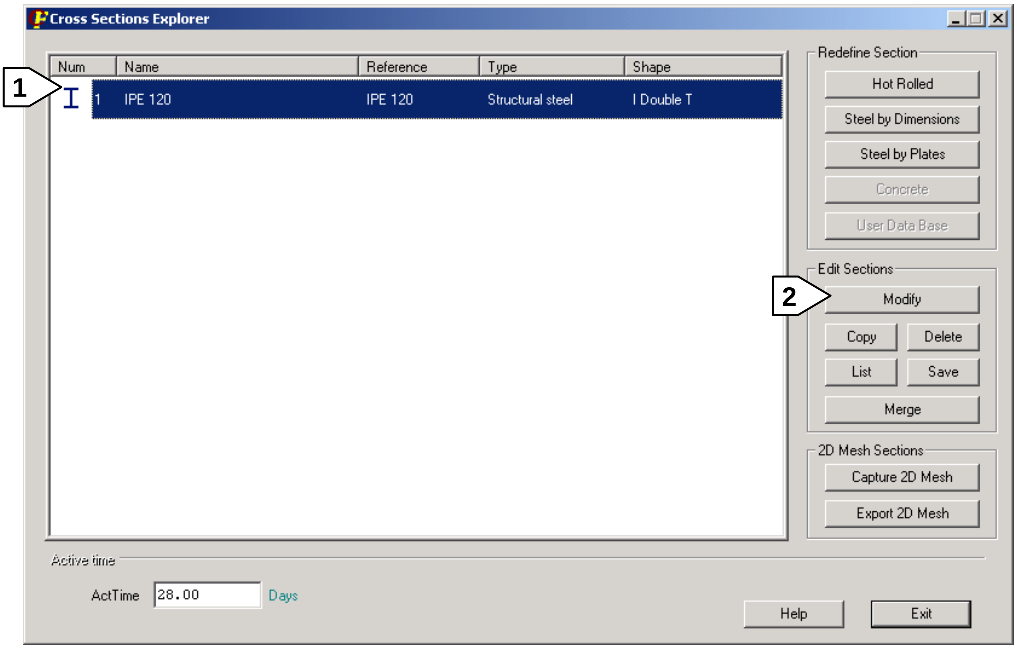


15. Plot the bending stress in Y top fiber

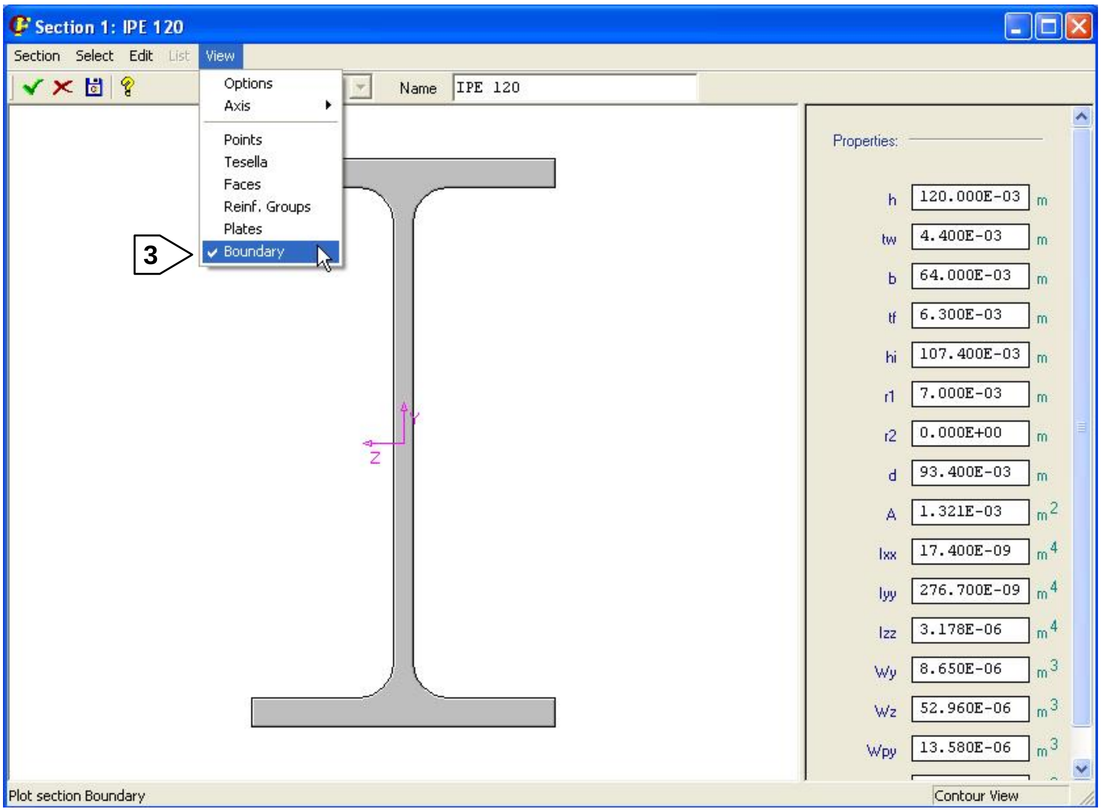
Main Menu: – CivilFEM – **Civil Preprocessor →Cross Sections**

1 Select the IPE section

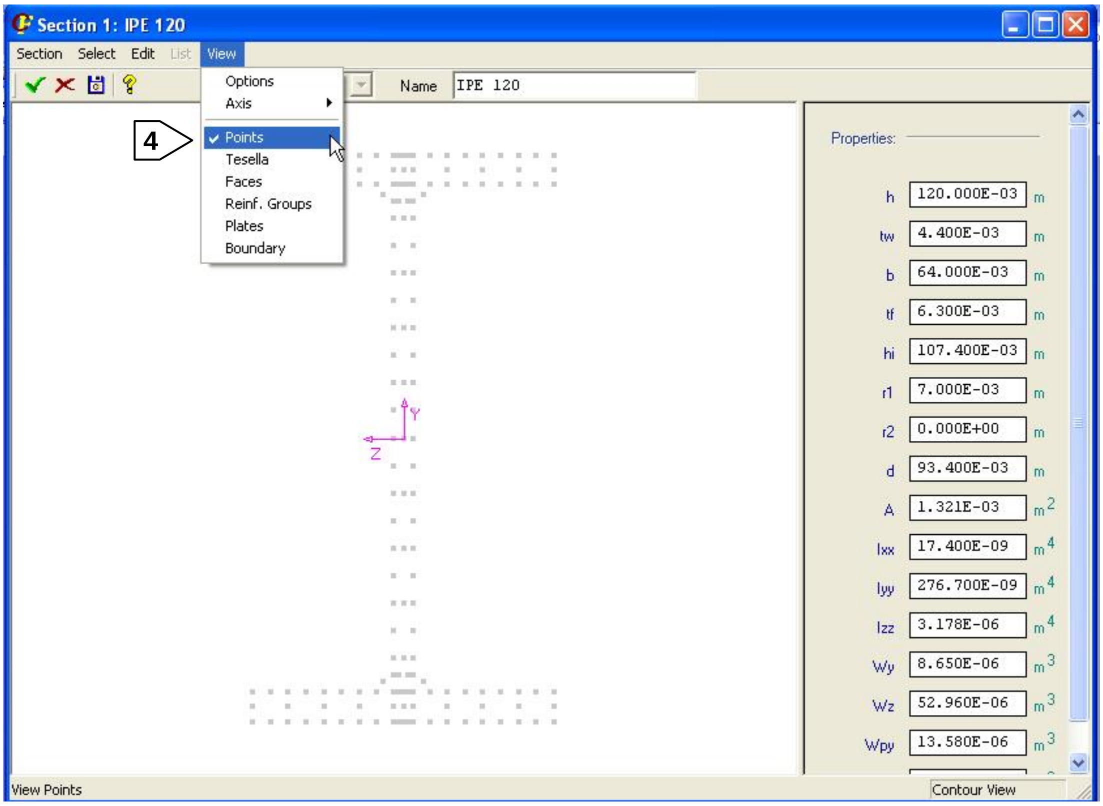
2 Modify



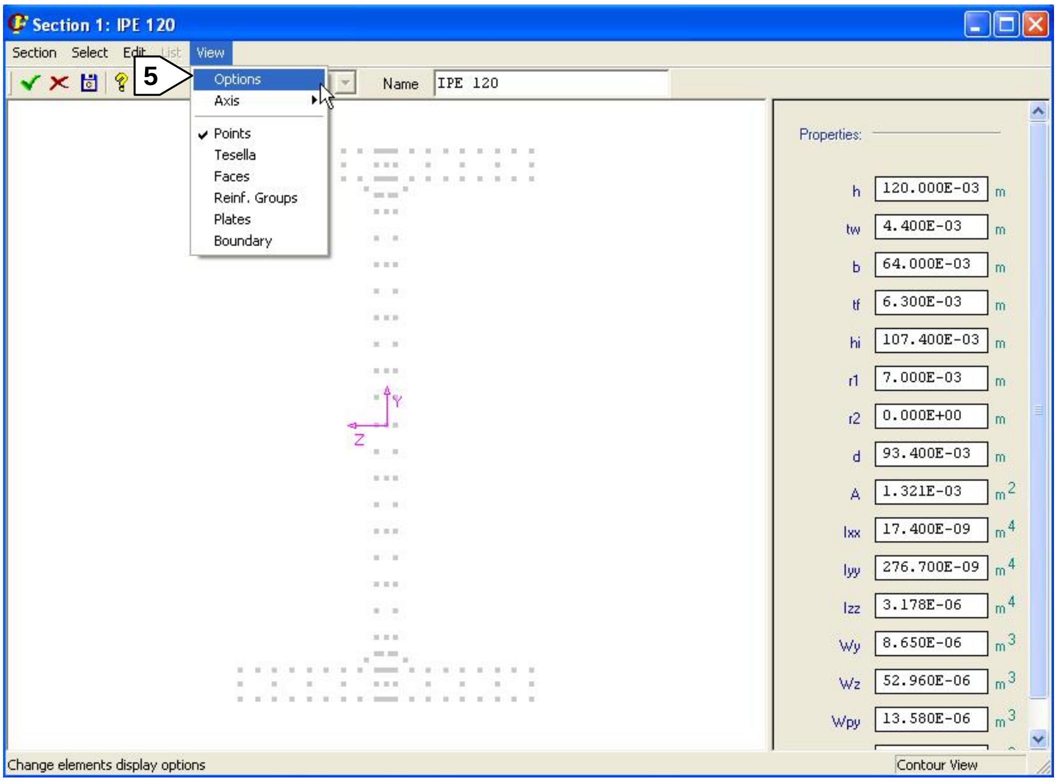
3 Uncheck boundary view



4 Check view Points

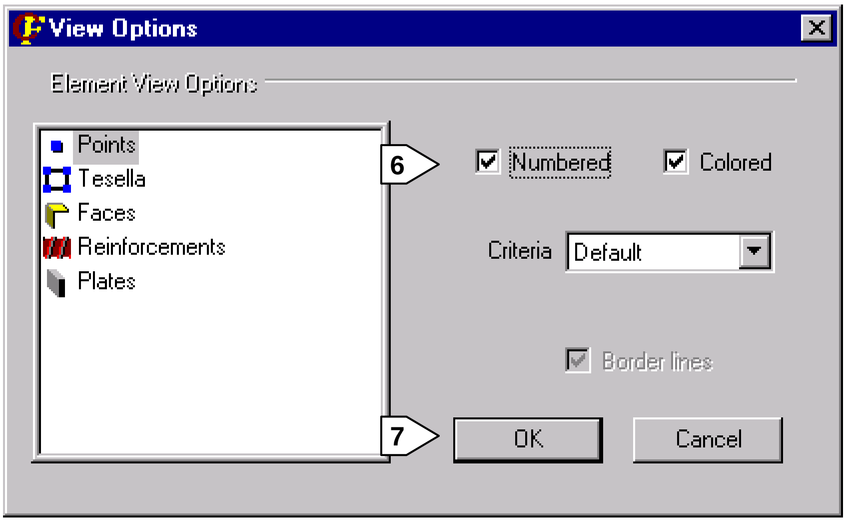


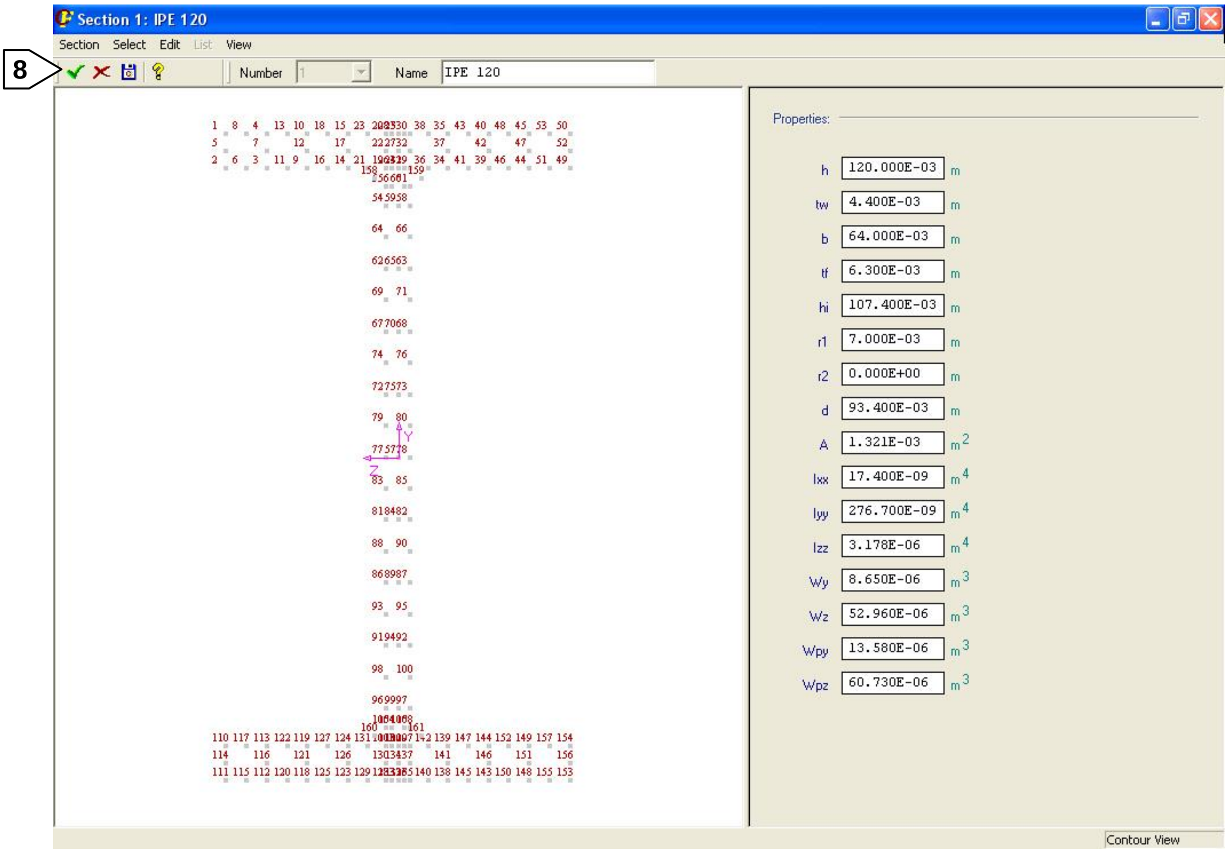
5 Select Options



6 Select Numbered to review points with its numbers

7 OK

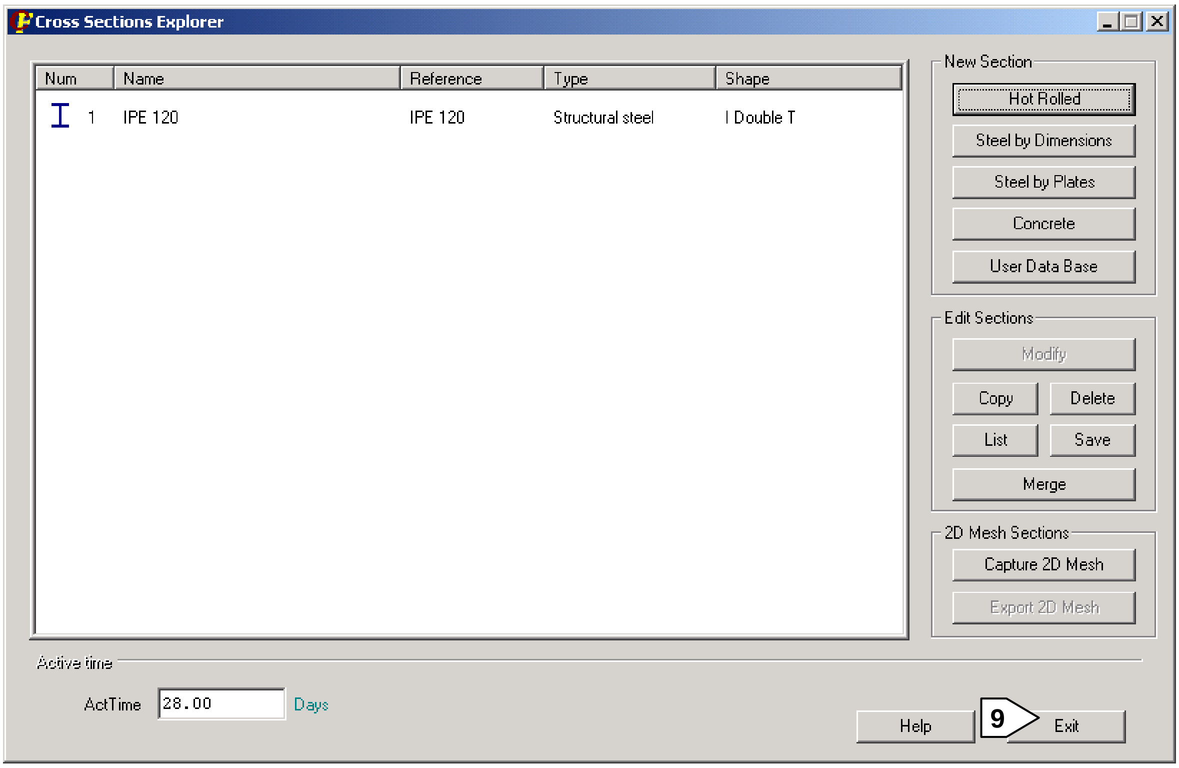




We are going to plot stresses in the point 50 on the upper right corner of the section.

8 OK

9 Exit

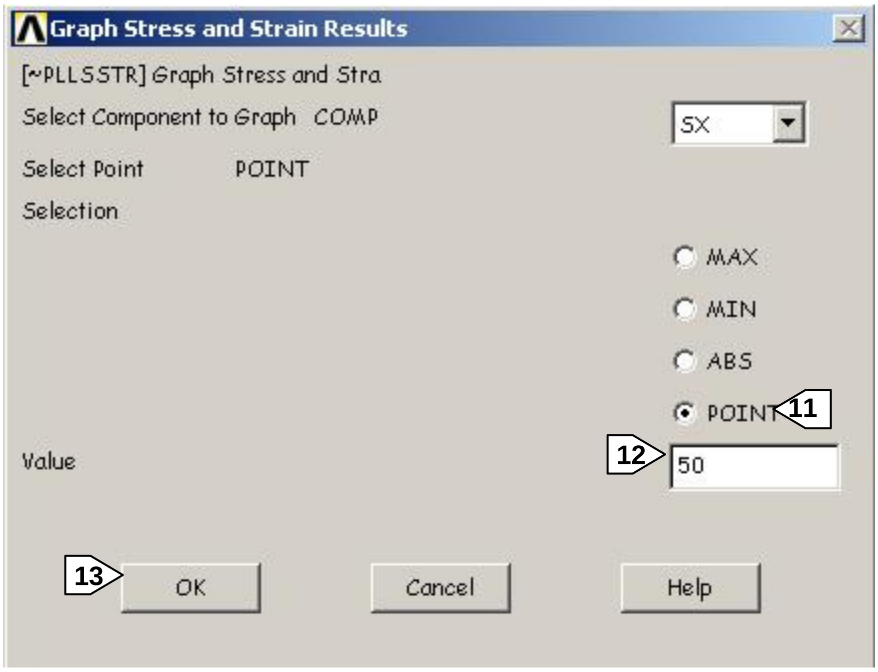


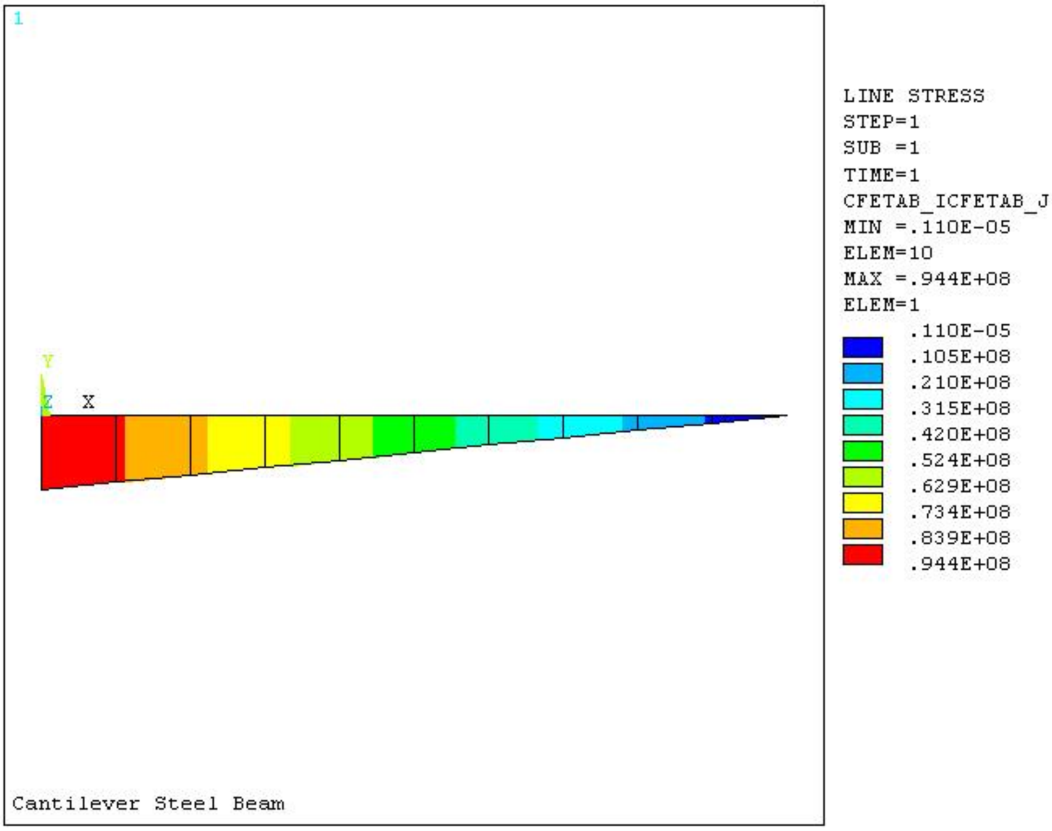
Main Menu: – CivilFEM – **Civil Postprocess** →**Beam Utilities** →GRAPH RESULTS: **Stress & Strain**

10 Select Point to plot stresses on a section point

11 Enter point number

12 OK to plot stress results

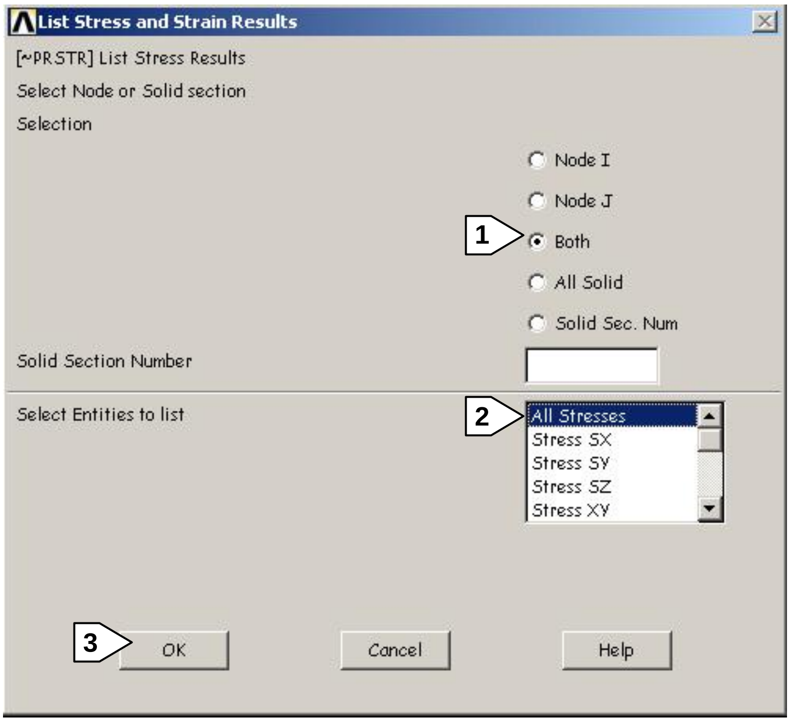


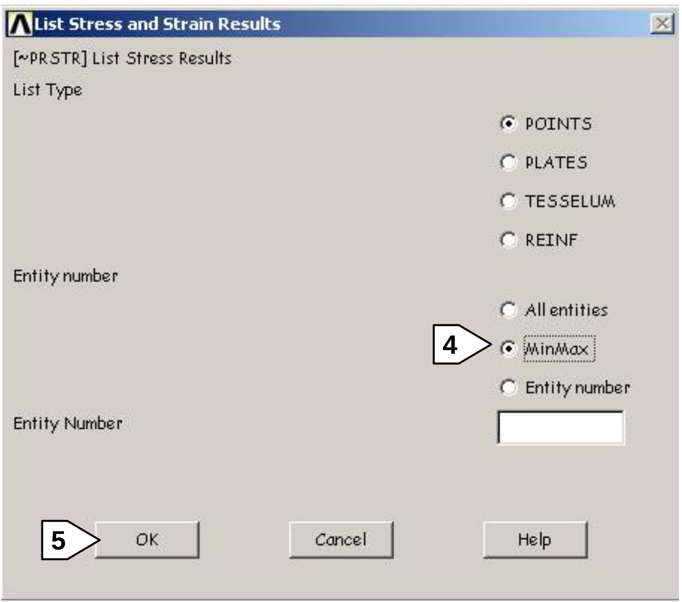


16. List bending extreme stresses

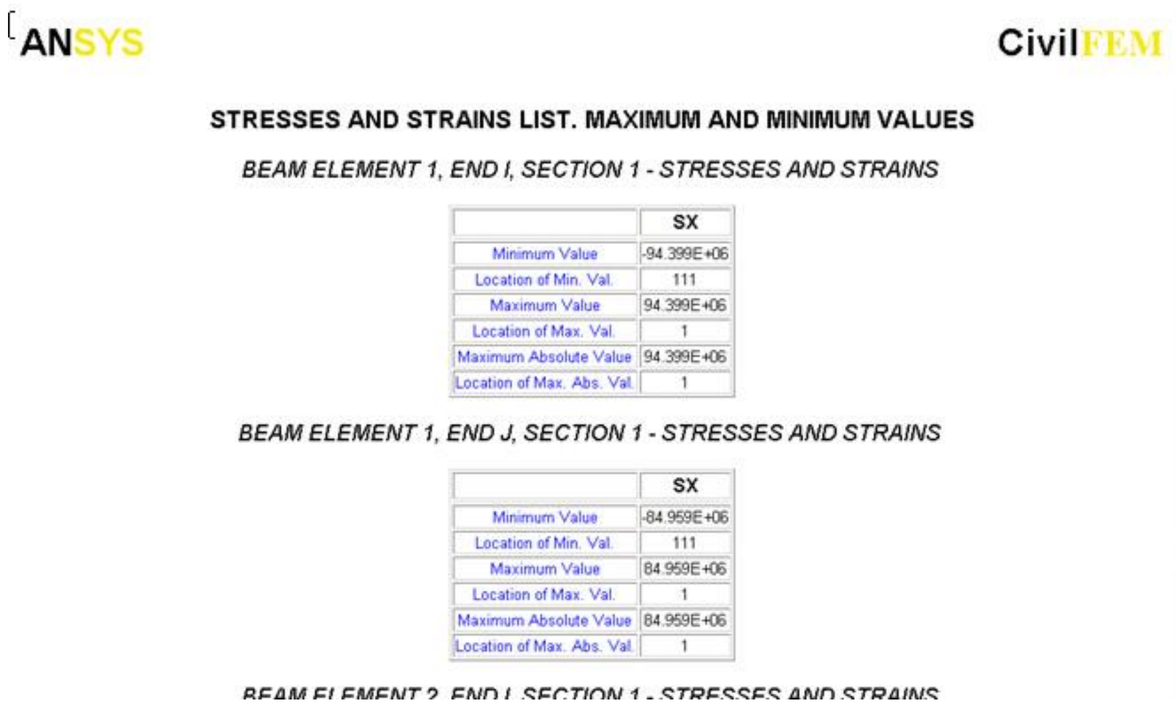
Main Menu: – CivilFEM – **Civil Postprocess** →**Beam Utilities** →LIST RESULTS: **Stress & Strain+**

CivilFEM allows to obtain the values of the different stress types at several section points. In this case we'll see the extreme values of bending stresses.





1. Select both nodes, I and J
2. Select All Stresses in Entities to list
3. OK
4. Select MinMax to list extreme values of bending stresses
5. OK to list values. An htm file will open



ANSYS CivilFEM

STRESSES AND STRAINS LIST. MAXIMUM AND MINIMUM VALUES

BEAM ELEMENT 1, END I, SECTION 1 - STRESSES AND STRAINS

	SX
Minimum Value	-94.399E+06
Location of Min. Val.	111
Maximum Value	94.399E+06
Location of Max. Val.	1
Maximum Absolute Value	94.399E+06
Location of Max. Abs. Val.	1

BEAM ELEMENT 1, END J, SECTION 1 - STRESSES AND STRAINS

	SX
Minimum Value	-84.959E+06
Location of Min. Val.	111
Maximum Value	84.959E+06
Location of Max. Val.	1
Maximum Absolute Value	84.959E+06
Location of Max. Abs. Val.	1

BEAM ELEMENT 2, END I, SECTION 1 - STRESSES AND STRAINS

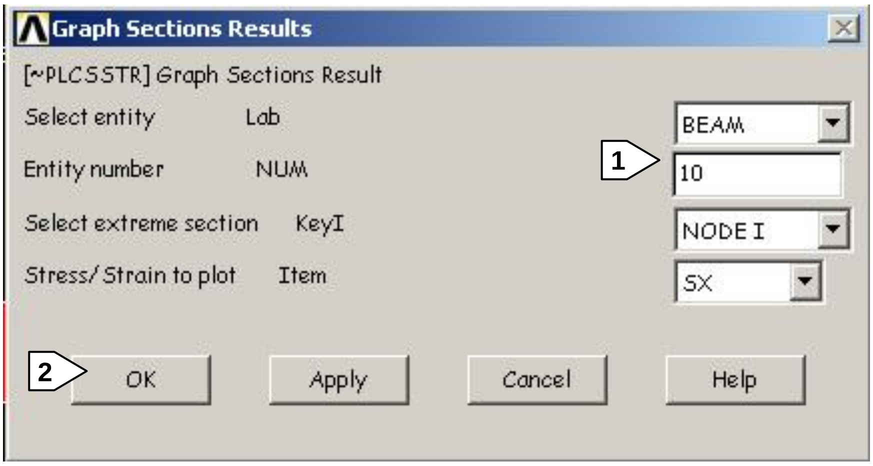
17. Plot bending stress distribution inside the cross-section

Finally we are going to plot the stress distribution inside the cross-section in the element where the bending stress is maximum. The ~ **PLCSSTR** command plots the cross-section of any desired element. Since the first element has the maximum stress, it will be plotted by default:

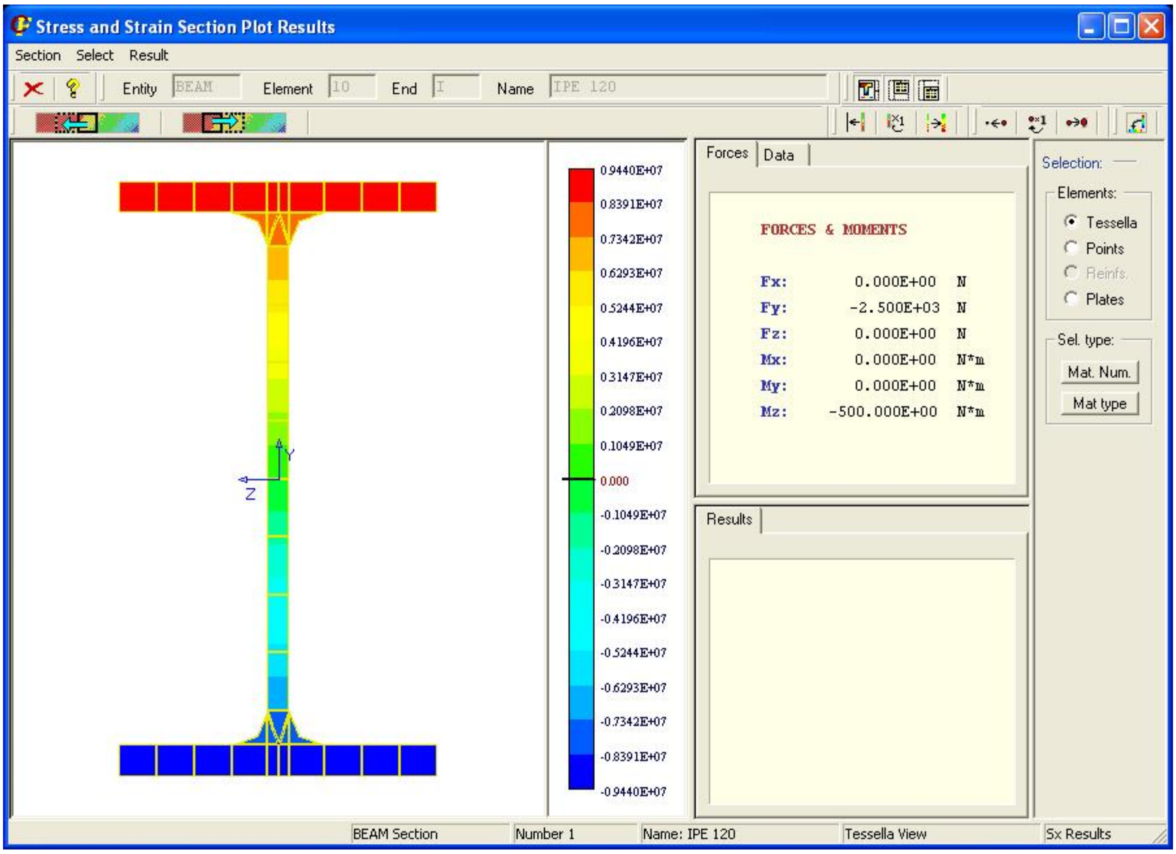
Main Menu: – CivilFEM – **Civil Postprocess** →**Beam Utilities** →GRAPH RESULTS: **Section Results…**

1 Enter Element number

2 OK



In CivilFEM the sign criteria is positive for tensile stresses and negative for compression stresses. Since the load was applied in the Y-axis direction, we will have tensile stresses in the top flange in red.

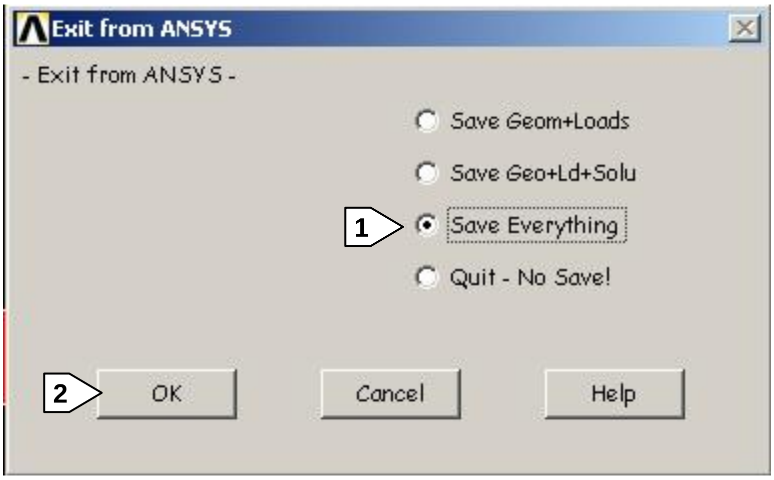


18. Exit the ANSYS program

ANSYS Toolbar: **Quit**

1 Choose save everything

2 OK


Exit from ANSYS
- Exit from ANSYS -
Save Geom+Loads
Save Geo+Ld+Solu
1
Save Everything
Quit - No Save!
2
OK
Cancel
Help