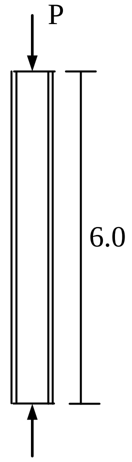
8. Compression Buckling

Applicable CivilFEM Product:	All CivilFEM Products
Level of Difficulty:	Moderate
Interactive Time Required:	25 minutes
Discipline:	Compression Buckling
Analysis Type:	Linear static
Element Type Used:	BEAM3
Active Code	British Standard 5950 (2001)
Units System	kN, m, s.
CivilFEM Features Demonstrated:	Units selection, code selection, material definition, user hot rolled section definition, section introduction inside the program library and compression buckling checking.

Problem Description

Check the column shown in the figure bellow to compression buckling using Grade 43 steel. It is pin ended about both x-x and y-y axes.



Loads:	
P = 2500 kN	
Section properties	(Section= 356x368x129 UC)
Dimensions of section:	
Width	L = 368.3 mm
Depth	D = 356 mm
Web thickness	B = 10.7 mm
Flange thickness	D = 17.5 mm
Radius of fillet	R = 15.4 mm
Material:	
Steel reinforcement	Gr43

Given

The load distribution, the section geometrical dimensions, material properties.

Approach and Assumptions

We will use 2D elastic beam elements for this analysis. Model geometry will be defined by generation of nodes and elements.

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 Member property
8. Define Beam & Shell properties
9. Define model geometry
10. Mesh
11. Save the database

Solution

12. Apply displacement constraints

13. Apply load
14. Solve

Postprocessing

15. Enter the postprocessor and read in results
16. Checking in compression buckling
17. Review compression capacity
18. Review section class
19. Review criterion
20. Review slenderness
21. Exit the ANSYS program

Interactive Step-by-Step Solution

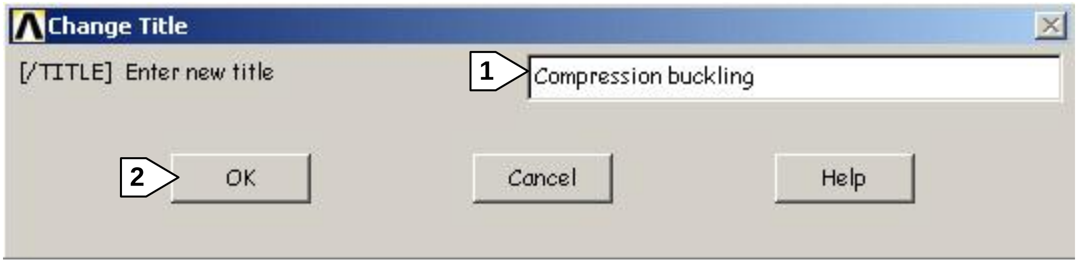
■ Preprocessing

1. Specify title

Utility Menu: **File →Change title**

1 Enter the title: “Compression buckling”

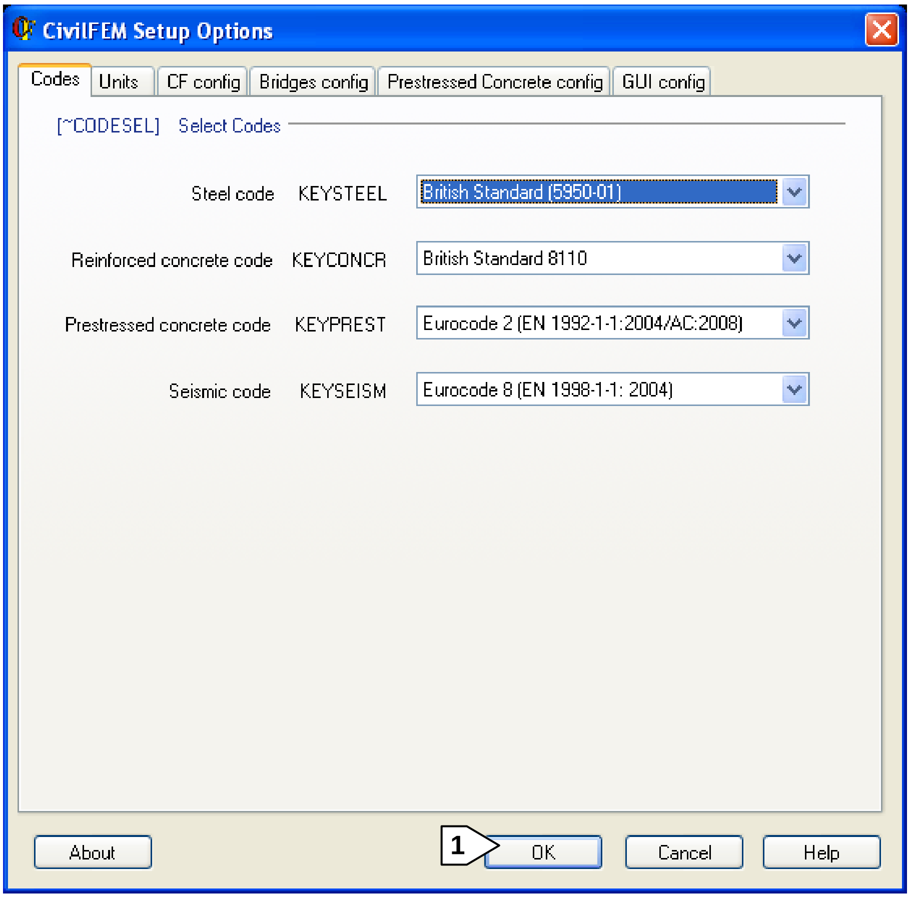
2 OK to define the title and close the dialog box.



2. Set code

Utility Menu:– CivilFEM – **Civil Setup**

1 OK to set active code and close the code dialog box

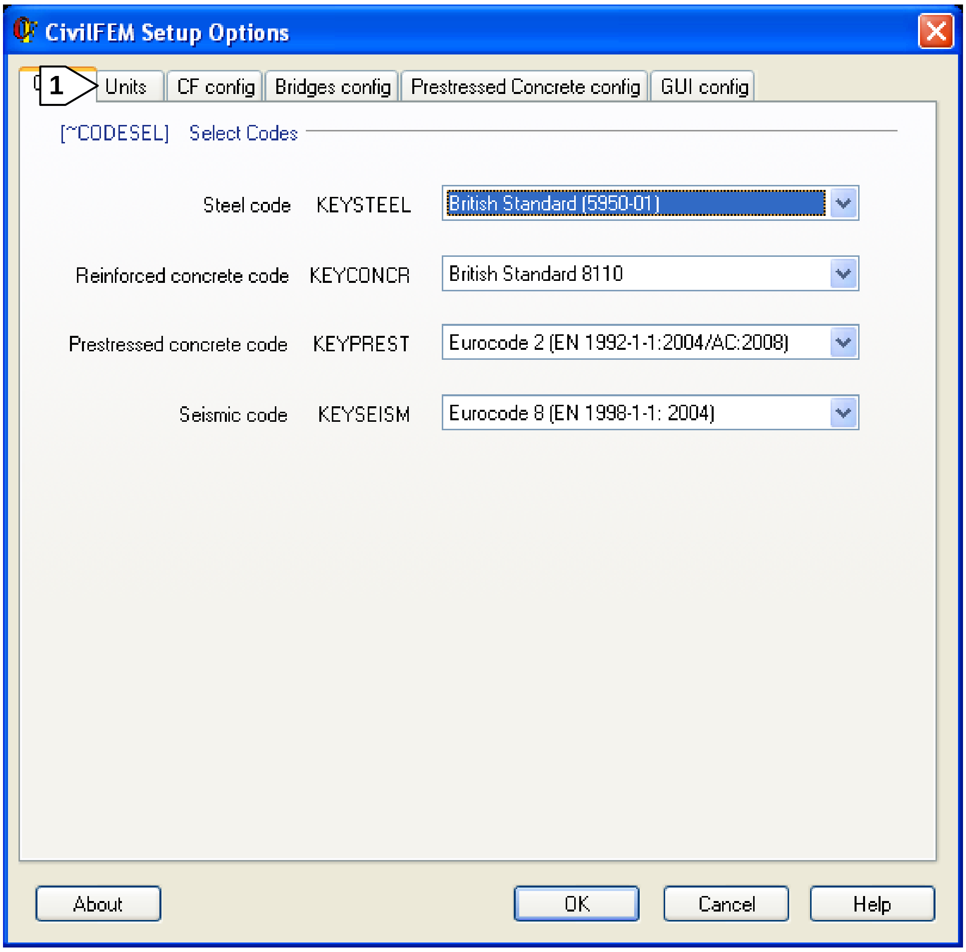


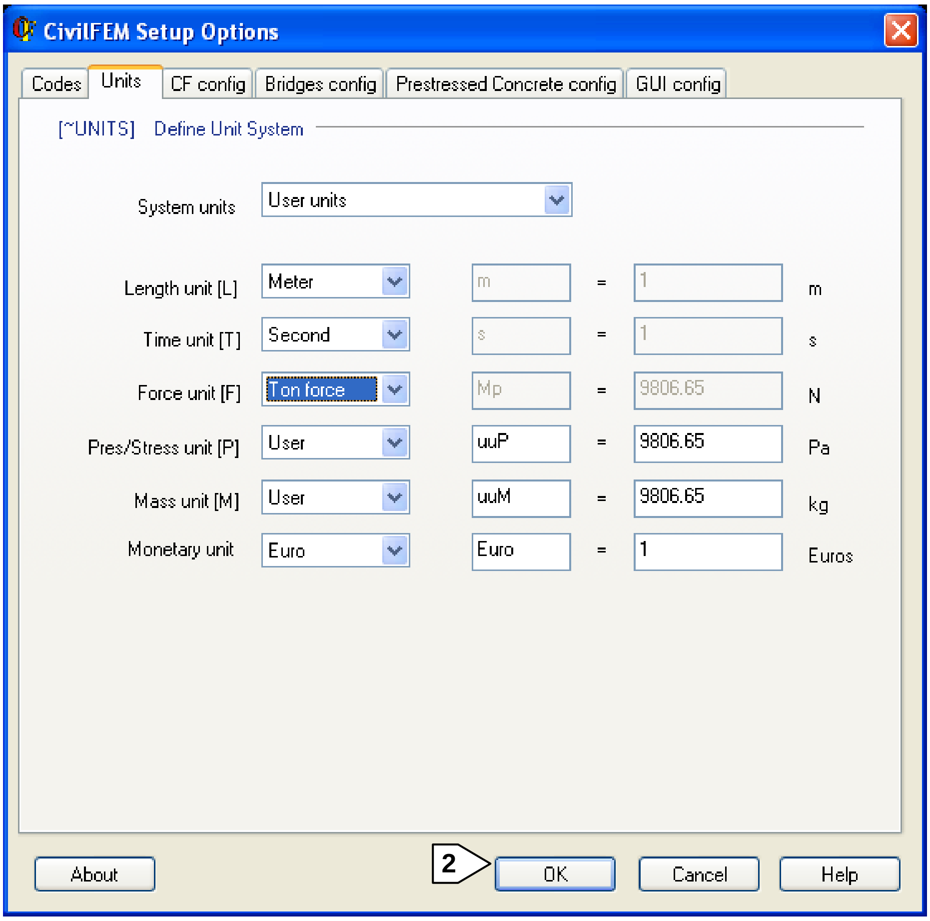
3. Set units

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

1 Pick on the Units tab

2 Define kN, m and s as user units and OK to close the units dialog box





4. Define material

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

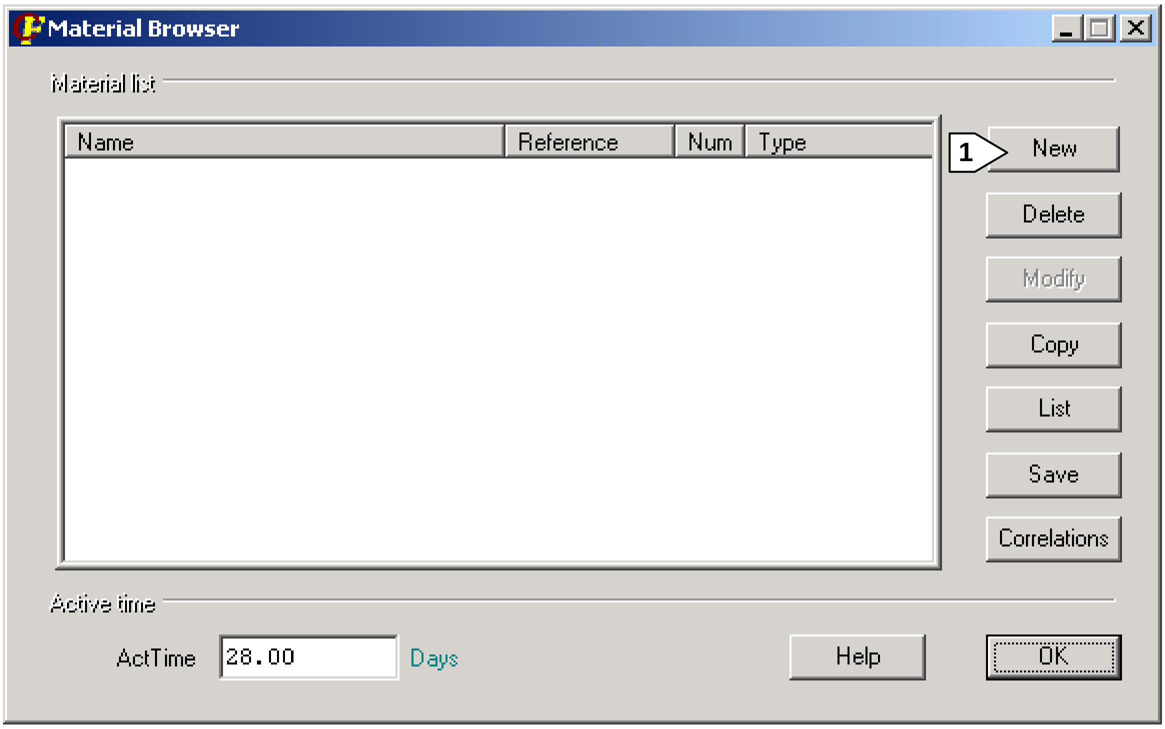
1 Choose New

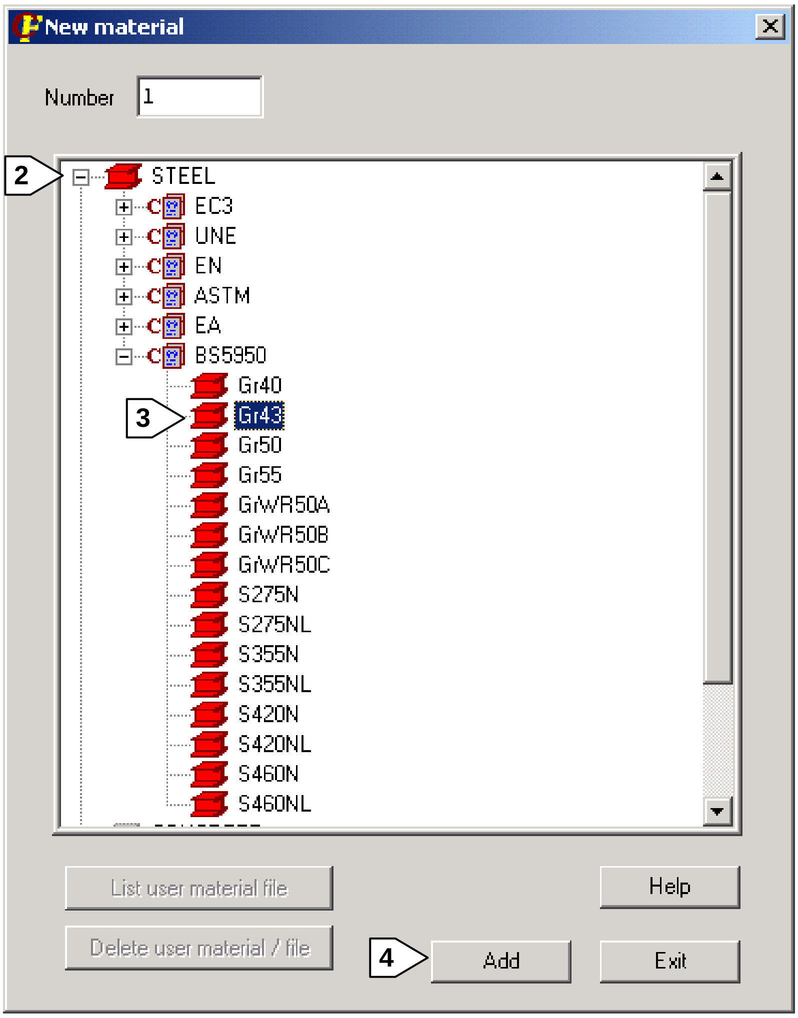
2 Select steel material type

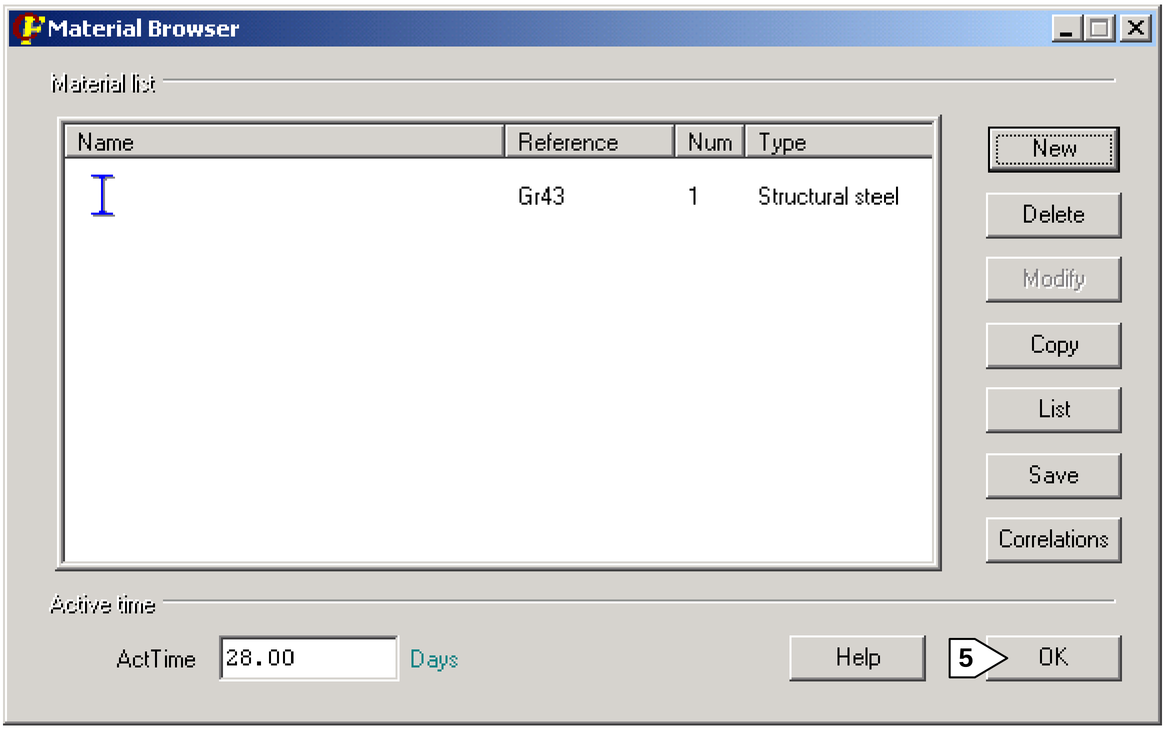
3 Choose BS 5950: Gr43

4 Add to define material 1 and Exit.

5 Ok to define material properties and close the dialog box


Material Browser
Material list
Name
Reference
Num
Type
1
New
Delete
Modify
Copy
List
Save
Correlations
Active time
ActTime
28.00
Days
Help
OK


New material
Number
1
2
STEEL
EC3
UNE
EN
ASTM
EA
BS5950
Gr40
3
Gr43
Gr50
Gr55
GrWR50A
GrWR50B
GrWR50C
S275N
S275NL
S355N
S355NL
S420N
S420NL
S460N
S460NL
List user material file
Help
Delete user material / file
4
Add
Exit


Material Browser
Material list
Name
Reference
Num
Type
Gr43
1
Structural steel
New
Delete
Modify
Copy
List
Save
Correlations
Active time
ActTime
28.00
Days
Help
5
OK

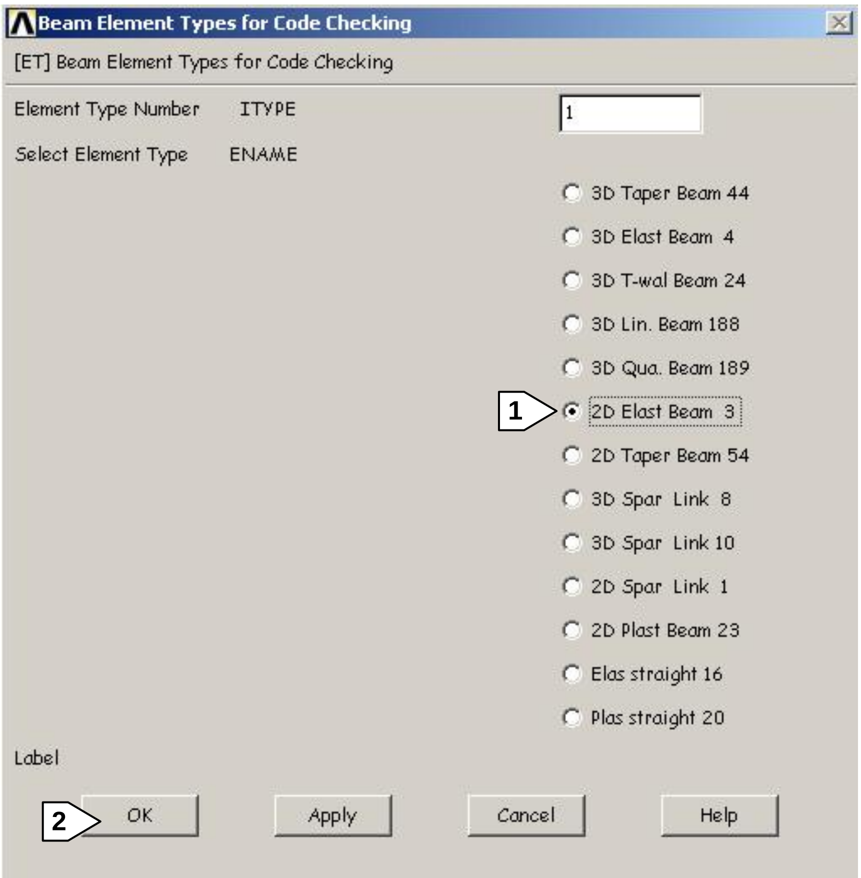
5. Define element type

We will make use of BEAM3.

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

1 Select 2D Elastic Beam 3

2 OK to define element type



6. Define section

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

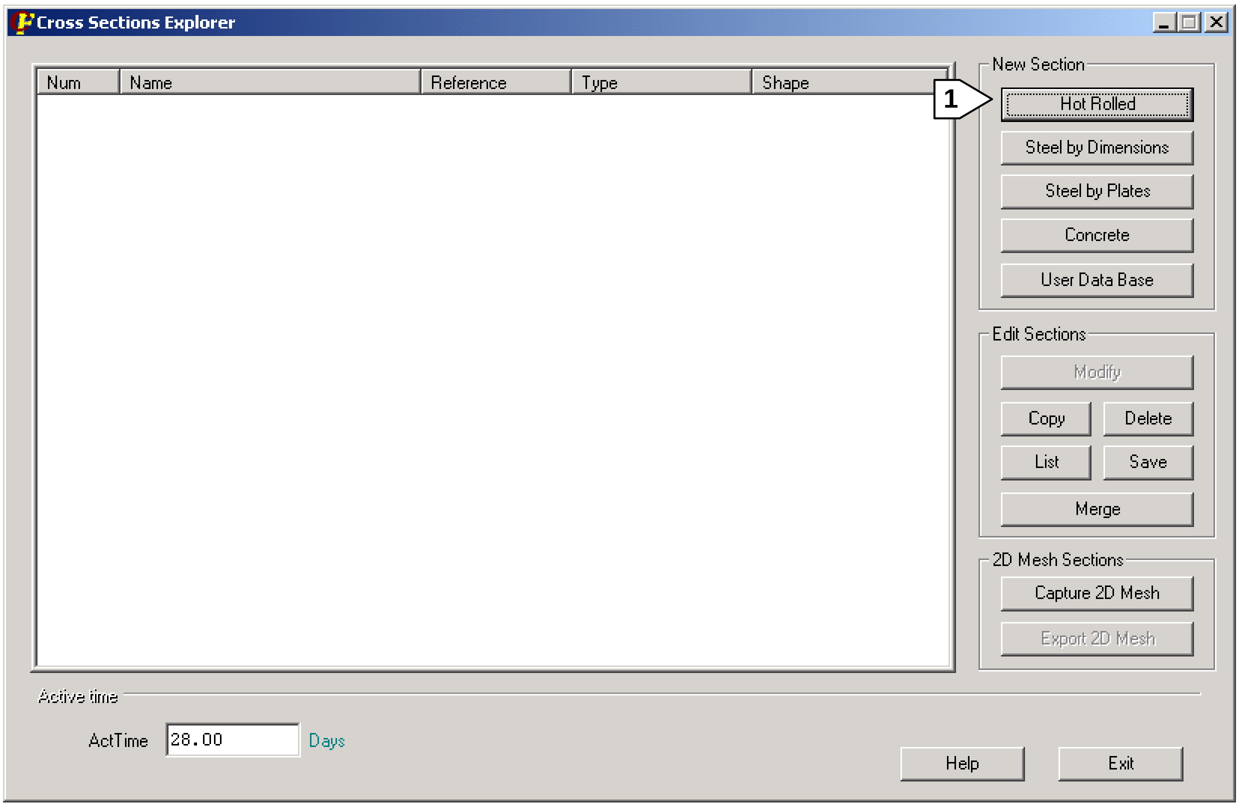
1 Pick on the Hot Rolled button to define the section

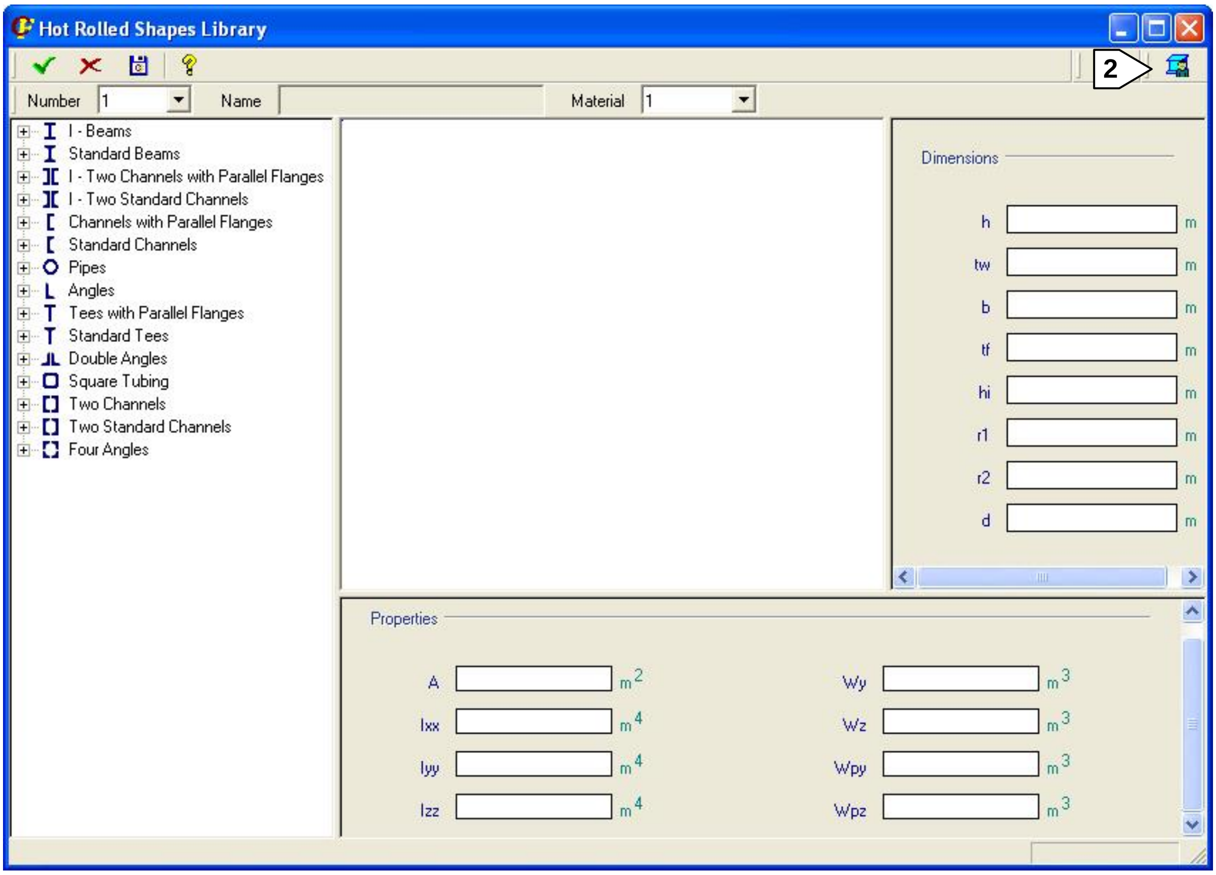
2 Pick on Add shape button

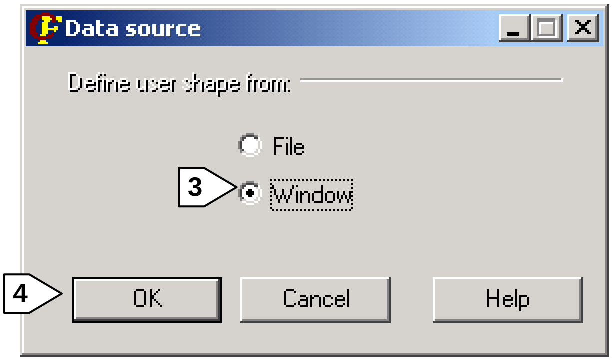
3 Choose Window

4 OK

5 Enter 501 as Group number


Cross Sections Explorer
Num
Name
Reference
Type
Shape
1
New Section
Hot Rolled
Steel by Dimensions
Steel by Plates
Concrete
User Data Base
Edit Sections
Modify
Copy
Delete
List
Save
Merge
2D Mesh Sections
Capture 2D Mesh
Export 2D Mesh
Active time
ActTime
28.00
Days
Help
Exit


Hot Rolled Shapes Library
2
Number
1
Name
Material
1
I - Beams
Standard Beams
I - Two Channels with Parallel Flanges
I - Two Standard Channels
Channels with Parallel Flanges
Standard Channels
Pipes
Angles
Tees with Parallel Flanges
Standard Tees
Double Angles
Square Tubing
Two Channels
Two Standard Channels
Four Angles
Dimensions
h
tw
b
tf
hi
r1
r2
d
m
Properties
A
Ixx
Iyy
Izz
Wy
Wz
Wpy
Wpz



6 Write I-Beam as Group name

7 Write 356x368x129 UC as shape name

8 Choose I-Beams in section type

9 Enter 356E-3 as h

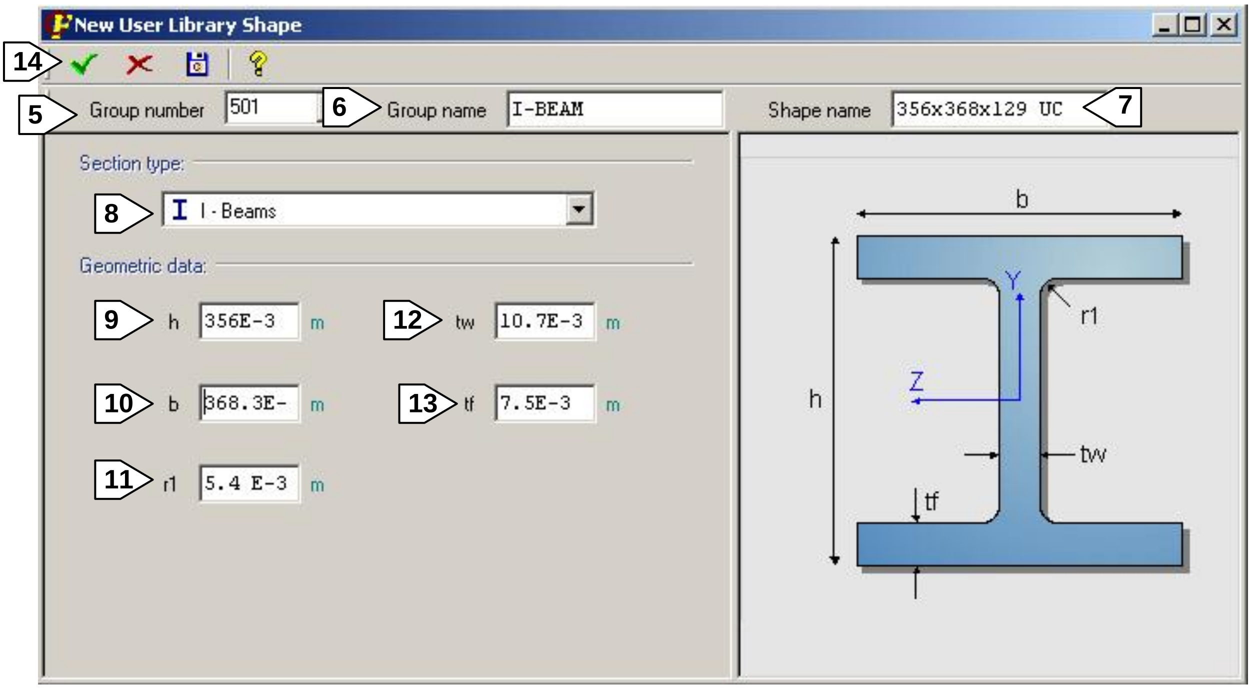
10 Enter 368.3E-3 as b

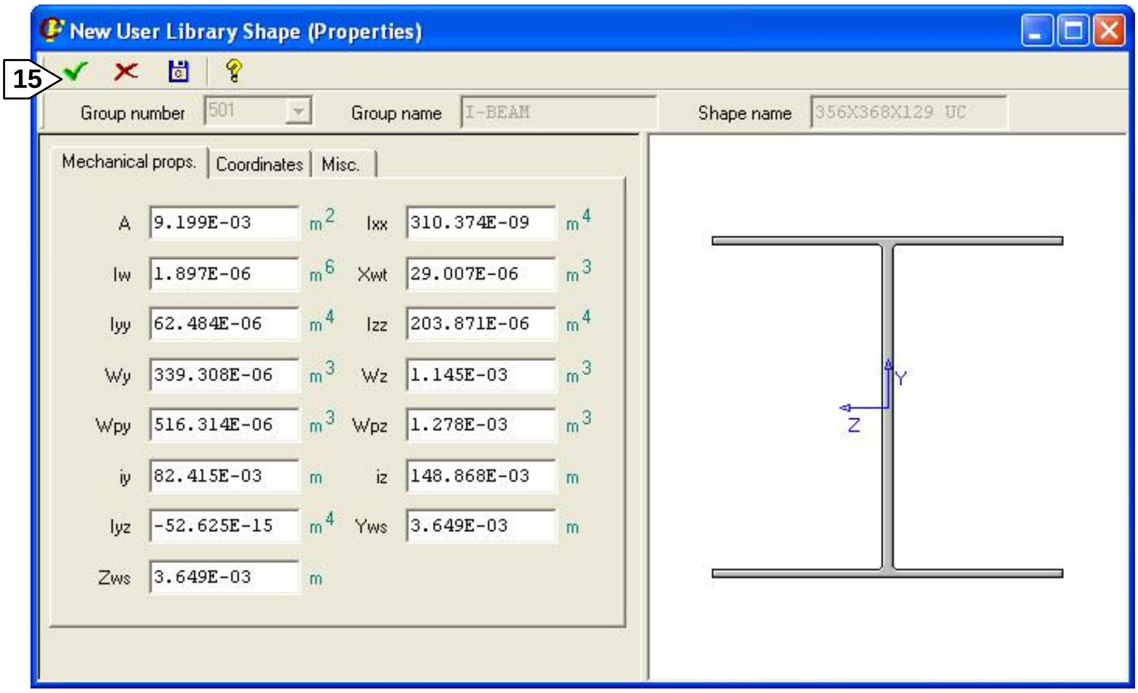
11 Enter 15.4 E-3 as r1

12 Enter 10.7E-3 as tw

13 Enter 17.5E-3 as tf

14 OK



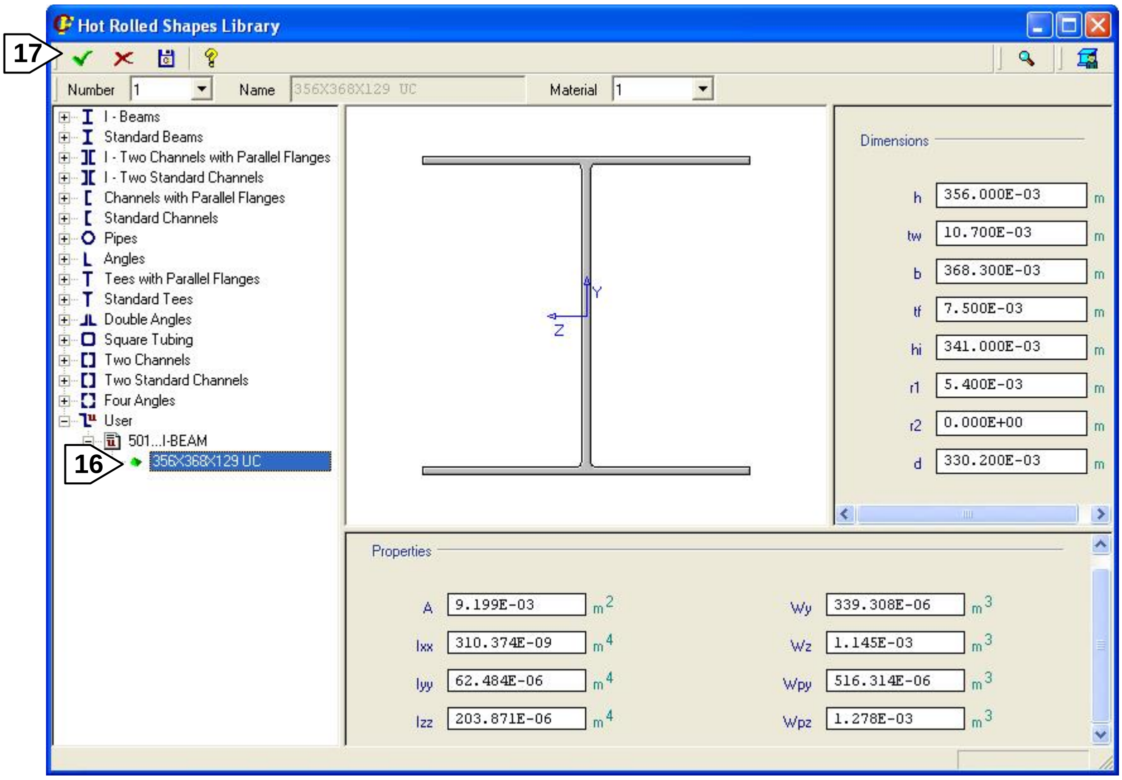


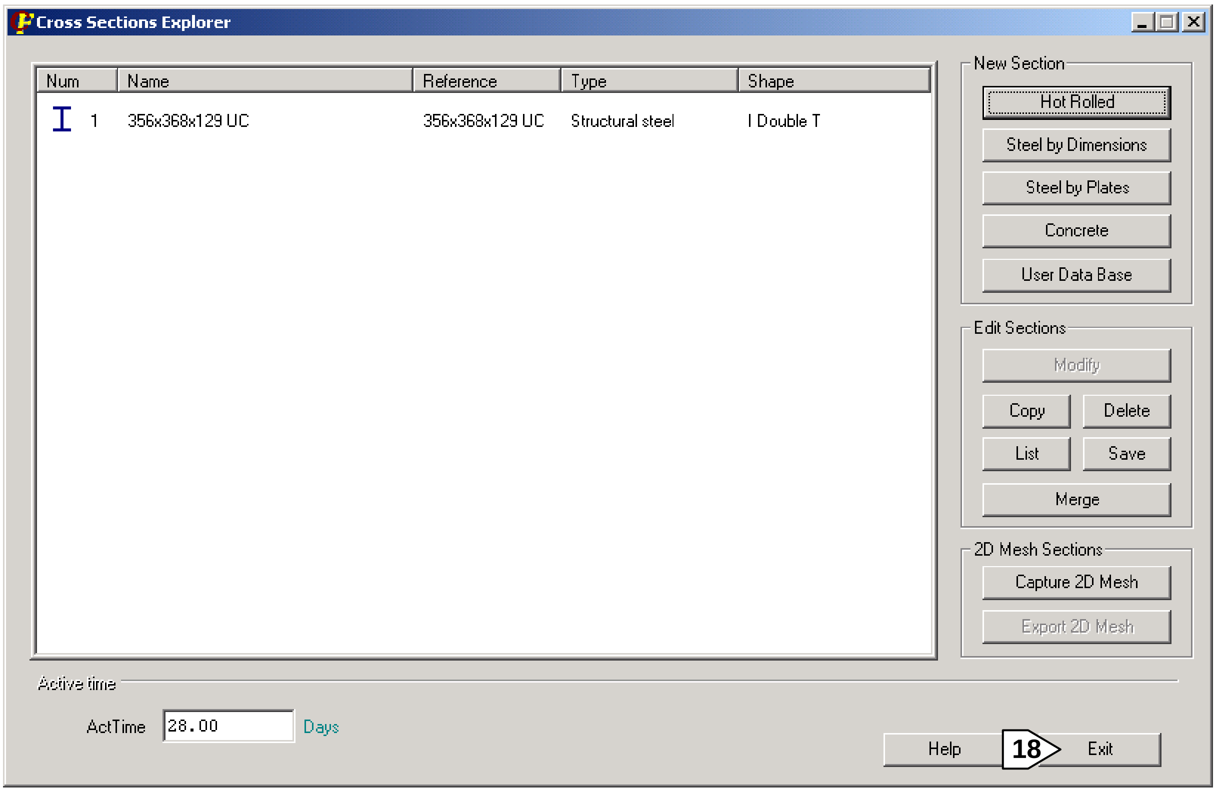
15 OK

16 Choose User > 501 > 356x368x129 UC

17 OK

18 Exit





7. Define Member property

CivilFEM command **~MEMBPRO** will be used to define member properties.

Main Menu: – CivilFEM – **Civil Preprocessor →Member Properties**

1 Pick on the New Property button

2 Enter 1 as Number

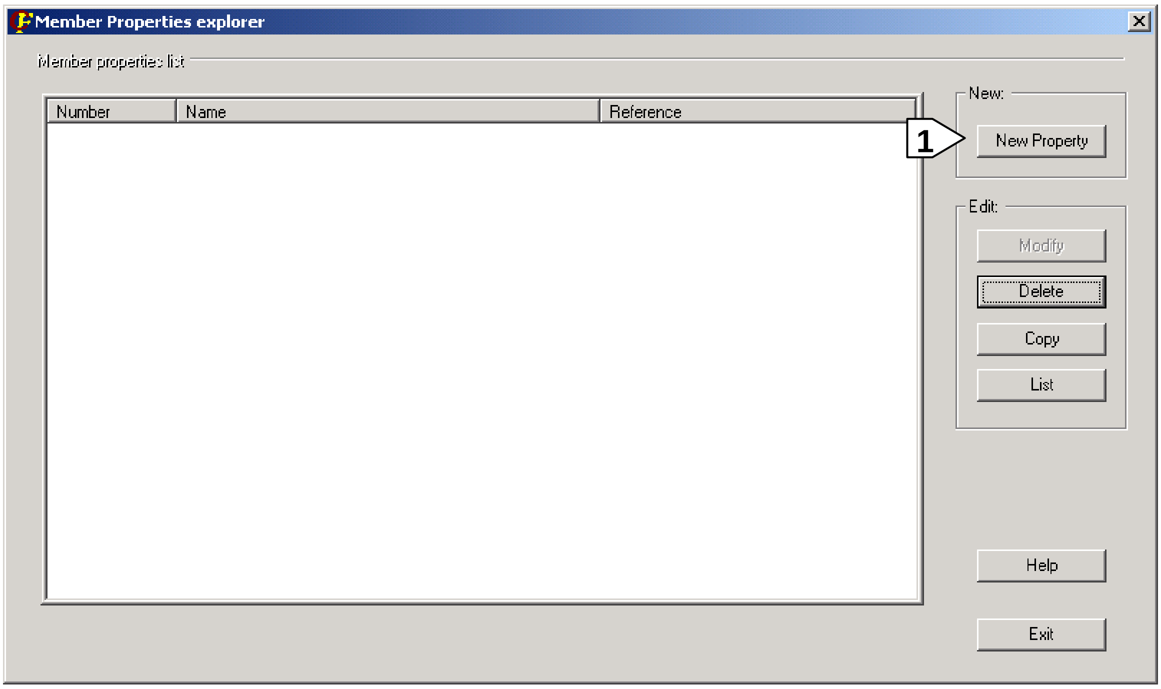
3 Write Member Property 1 as Name

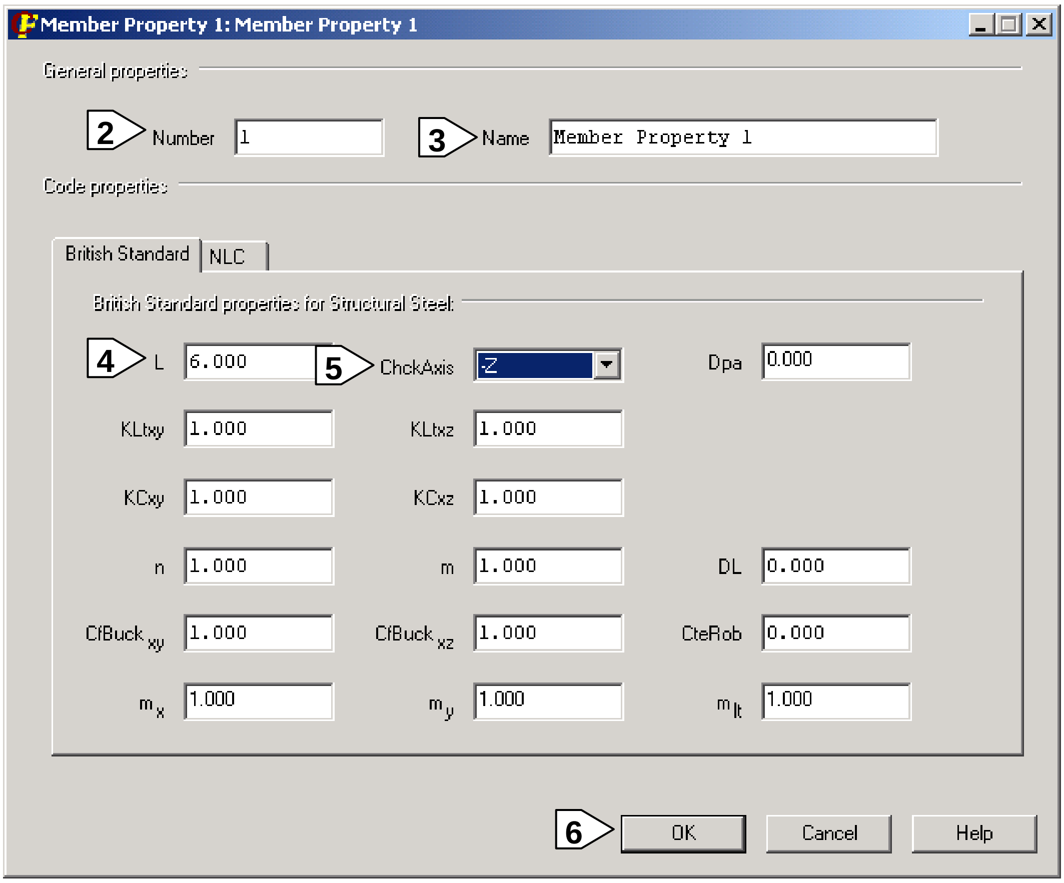
4 Enter 6.0 as L

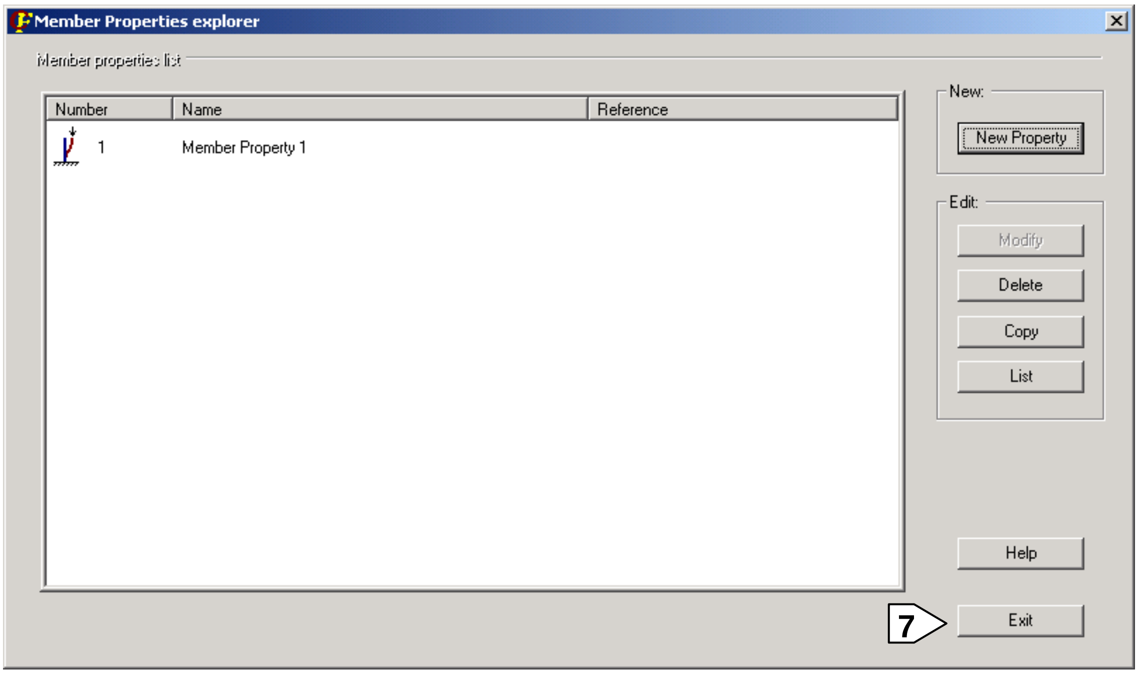
5 Choose –Z as ChckAxis

6 OK

7 Exit


Member Properties explorer
Member properties list
Number
Name
Reference
New:
1
New Property
Edit:
Modify
Delete
Copy
List
Help
Exit


Member Property 1: Member Property 1
General properties
2
Number
1
3
Name
Member Property 1
Code properties
British Standard
NLC
British Standard properties for Structural Steel
4
L
6.000
5
ChckAxis
-Z
Dpa
0.000
KLtxy
1.000
KLtxz
1.000
KCxy
1.000
KCxz
1.000
n
1.000
m
1.000
DL
0.000
CfBuck xy
1.000
CfBuck xz
1.000
CteRob
0.000
m x
1.000
m y
1.000
m lt
1.000
6
OK
Cancel
Help

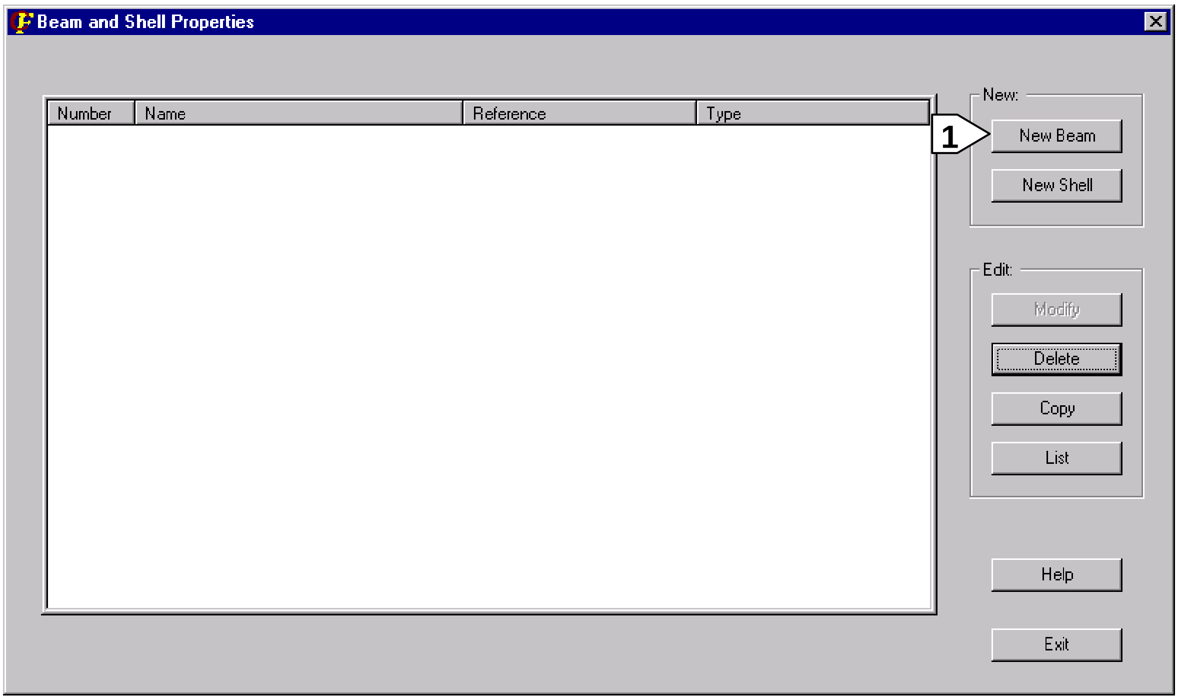

Member Properties explorer
Member properties list
Number
Name
Reference
1
Member Property 1
New:
New Property
Edit:
Modify
Delete
Copy
List
Help
7
Exit

8. Define Beam & Shell properties

CivilFEM command **~BMSHPRO** will be used to define ANSYS real constants.

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

1 Pick on the New Beam button



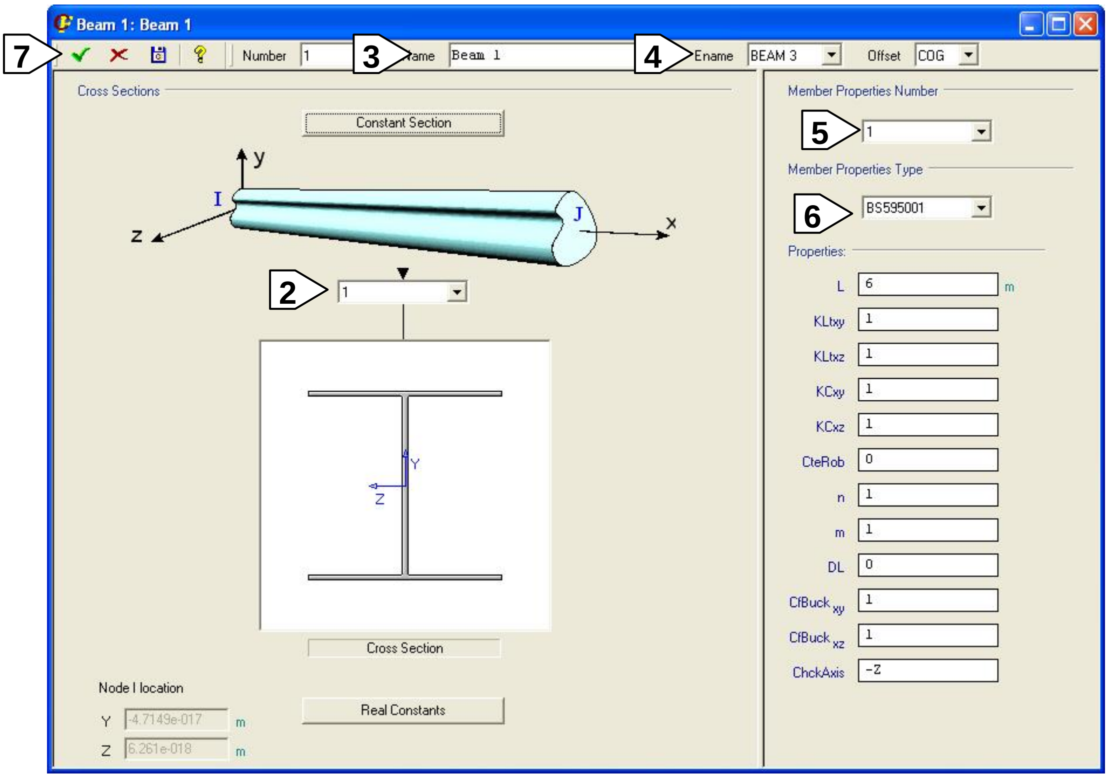
2 Select cross section number

3 Enter “Beam 1” as Name for the Beam property

4 Select element type 3

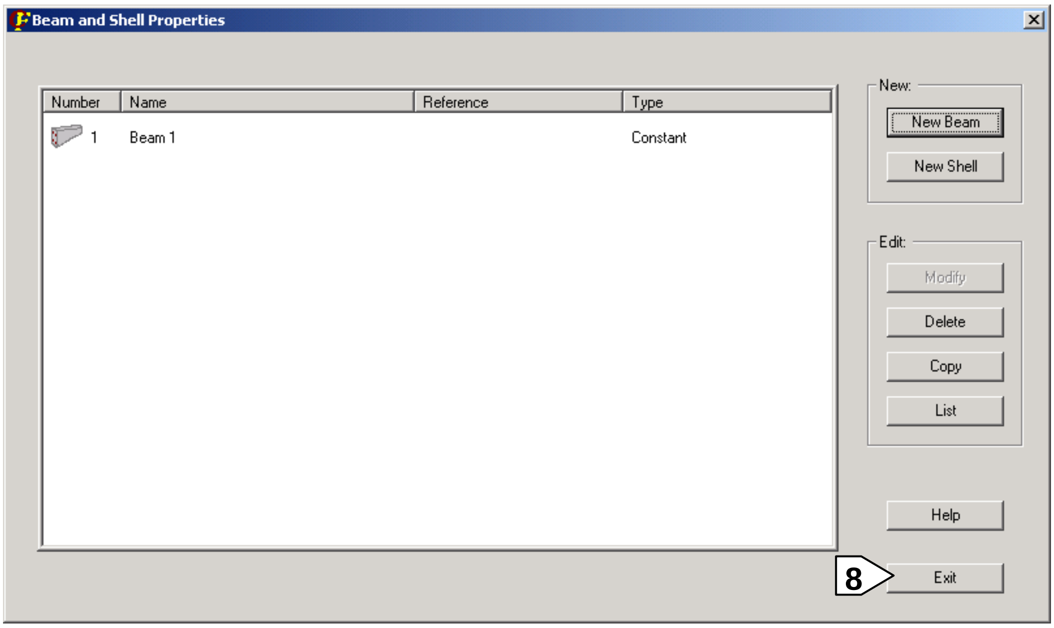
5 Choose 1 as Member Properties Number

6 Choose BS595001 as Member Properties Type


Beam 1: Beam 1
7
Number 1
3
Name Beam 1
4
Ename BEAM 3
Offset COG
Cross Sections
Constant Section
y
I
z
J
x
2
1
Y
Z
Cross Section
Node I location
Y -4.7149e-017 m
Z 6.261e-018 m
Real Constants
Member Properties Number
5
1
Member Properties Type
6
BS595001
Properties:
L 6 m
KLtxy 1
KLtxz 1
KCxy 1
KCxz 1
CteRob 0
n 1
m 1
DL 0
CfBuck xy 1
CfBuck xz 1
ChckAxis -Z

7 Ok

8 Exit


Beam and Shell Properties
Number
Name
Reference
Type
1
Beam 1
Constant
New:
New Beam
New Shell
Edit:
Modify
Delete
Copy
List
Help
8
Exit

9. Define model geometry

Geometry definition implies the generation of nodes and elements.

The steps that you must follow are:

a) Define keypoints

b) Define lines

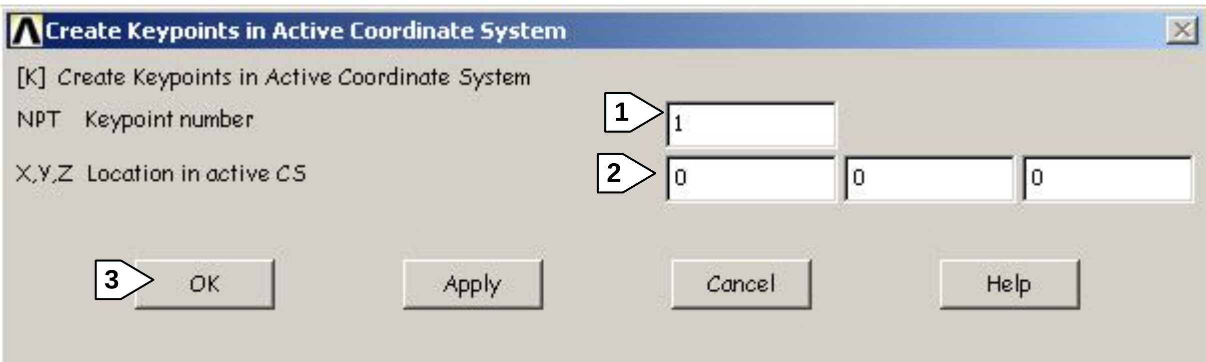
c) Mesh lines

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

1 Enter 1 as keypoint 1

2 Introduce the coordinates of keypoint 1

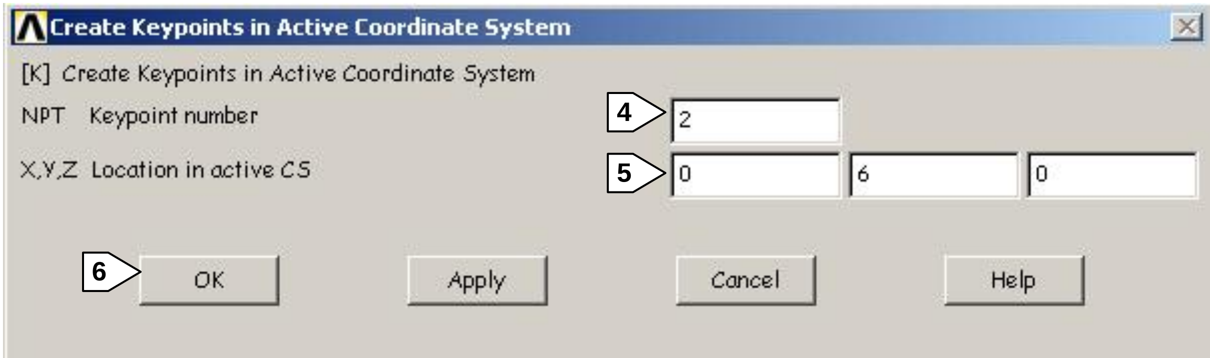
3 Pick on apply



4 Enter 2 as keypoint 2

5 Introduce the coordinates of keypoint 2

6 Pick on OK

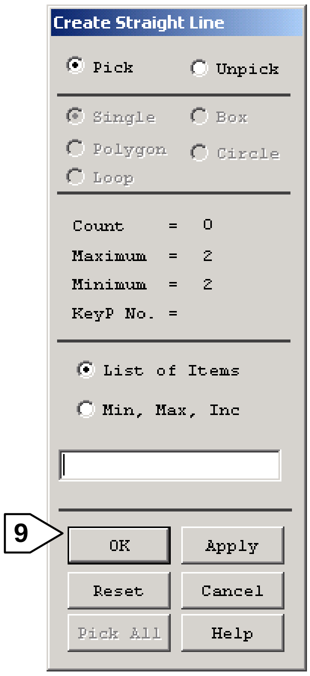


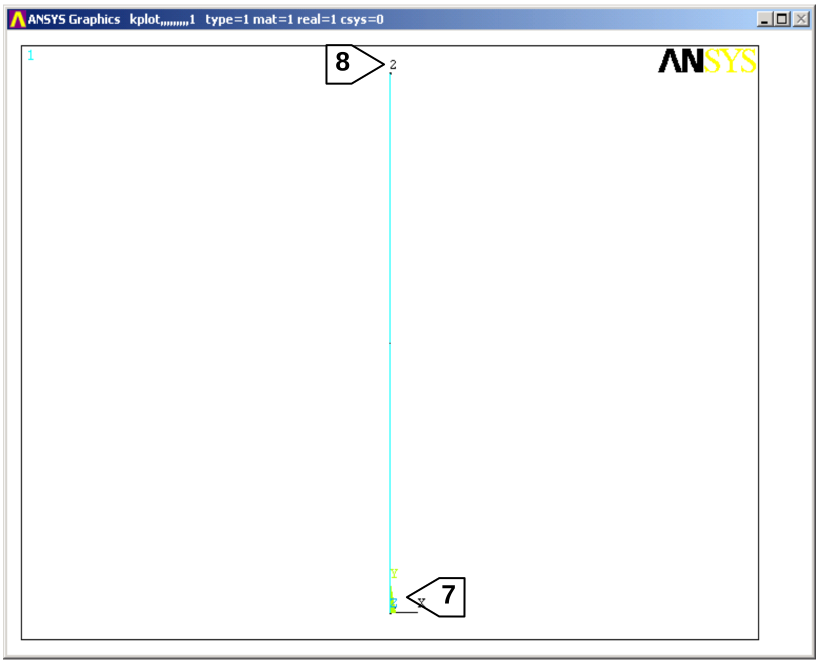
Main Menu: **Preprocessor** → –Modeling– **Create** → **Lines** → **Straight Line**

7 Select keypoint 1

8 Select keypoint 2

9 OK





10. Mesh

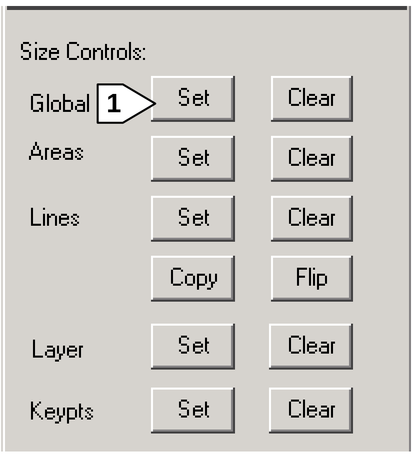
We first specify the meshing control.

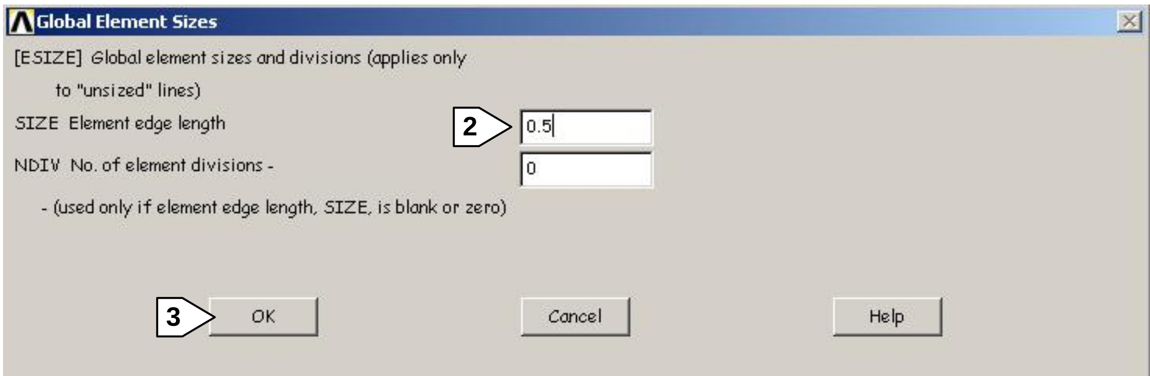
Main Menu: **Preprocessor** → **MeshTool**

1 Choose Global Set

2 Enter Size 0.5

3 OK



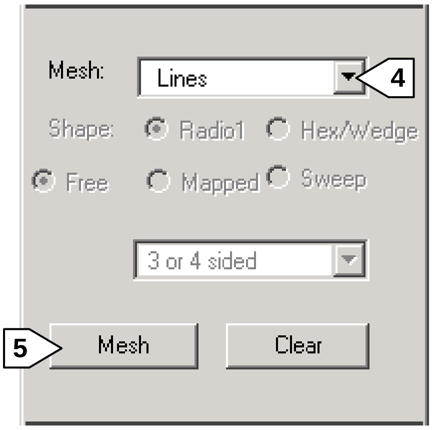


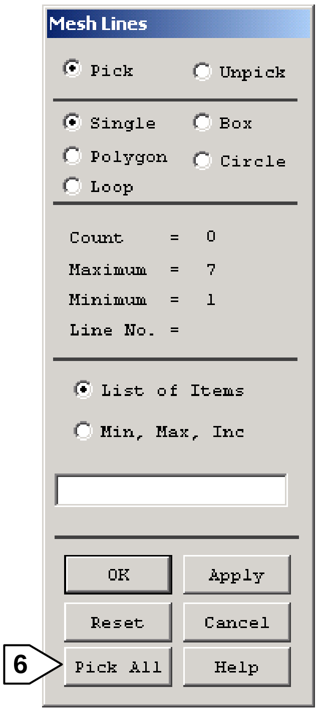
Main Menu: **Preprocessor → MeshTool**

4 Choose Lines

5 Pick on Mesh

6 Pick on Pick all





11. Save the database

Before going 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

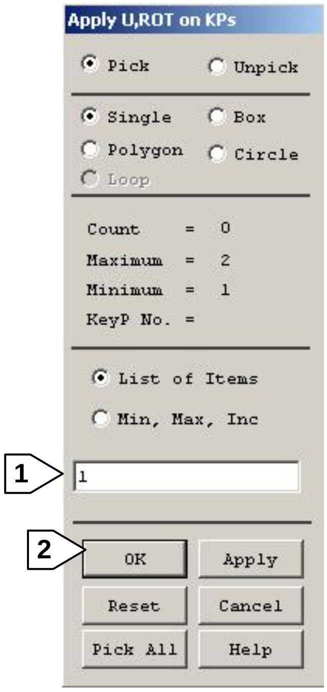
Toolbar: **SAVE (Command line ~CFSAVE)**

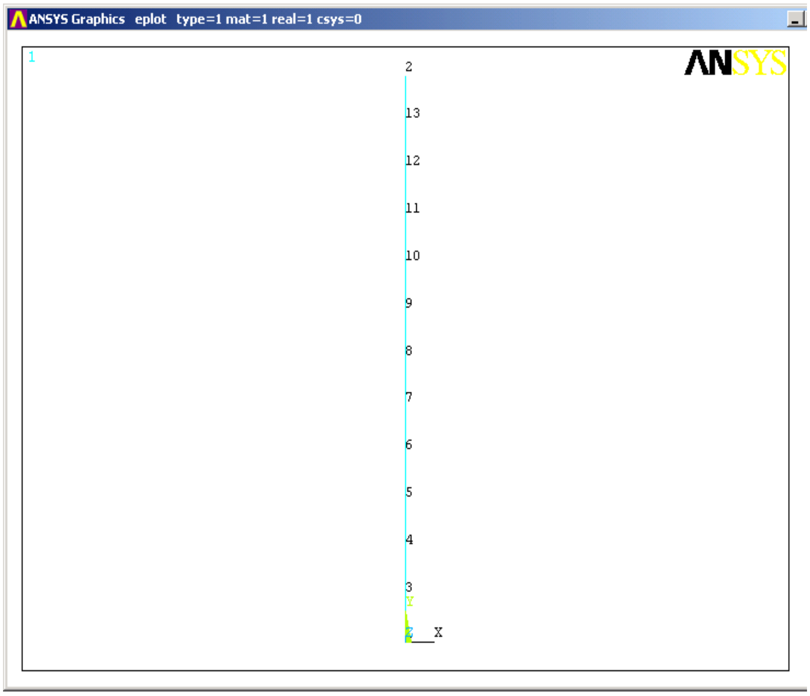
■ SOLUTION

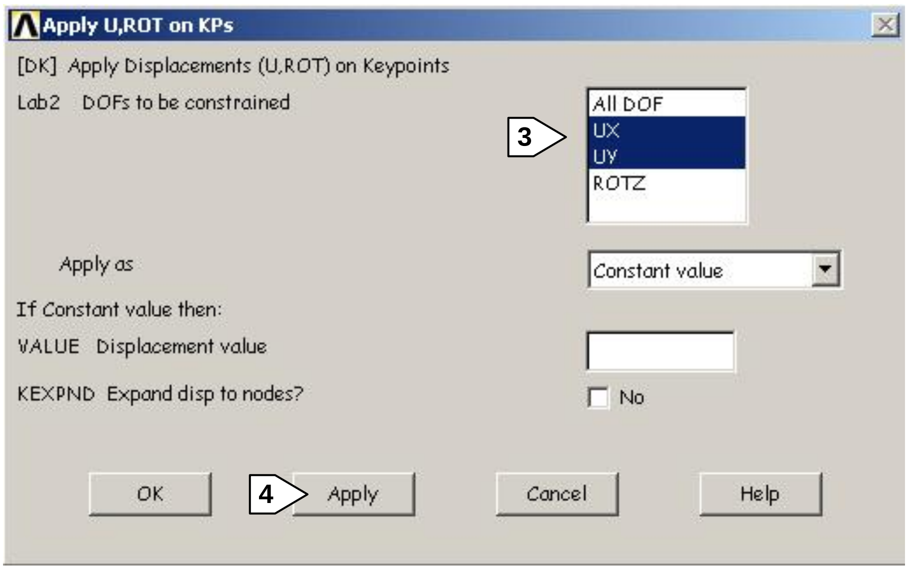
12. Apply displacement constraints

Main Menu: **Solution** → – Loads – **Apply** → – Structural – **Displacement** →**On Keypoints +**

1. Enter node 1
2. Pick OK
3. Choose UX and UY
4. Apply





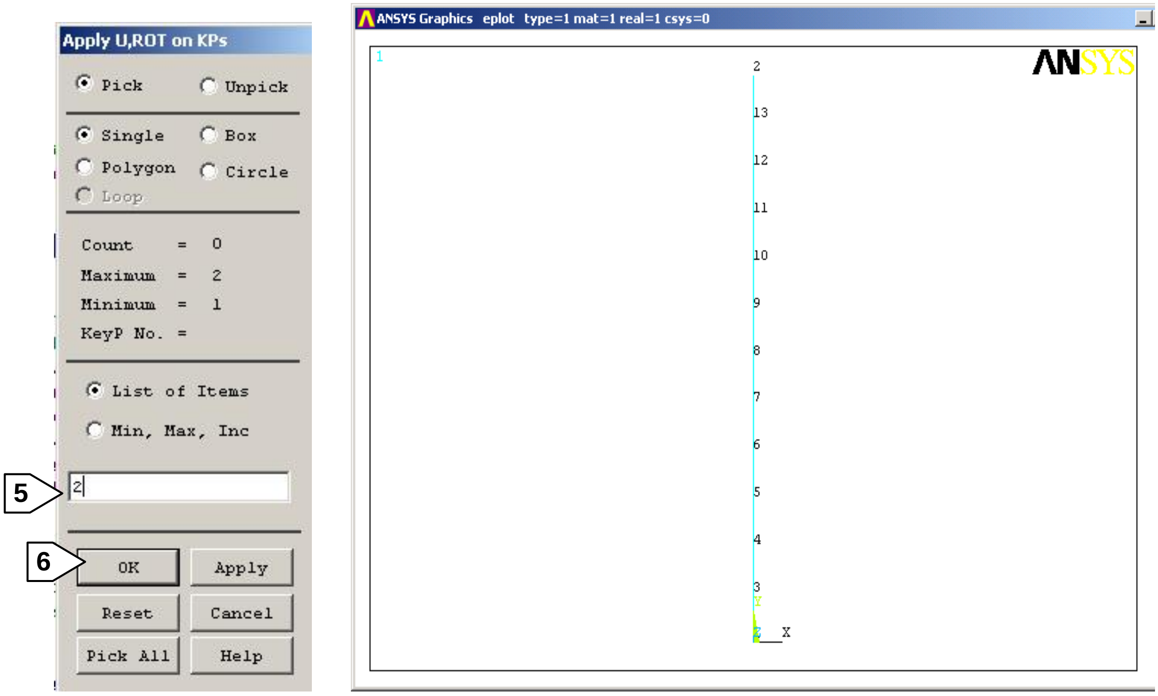


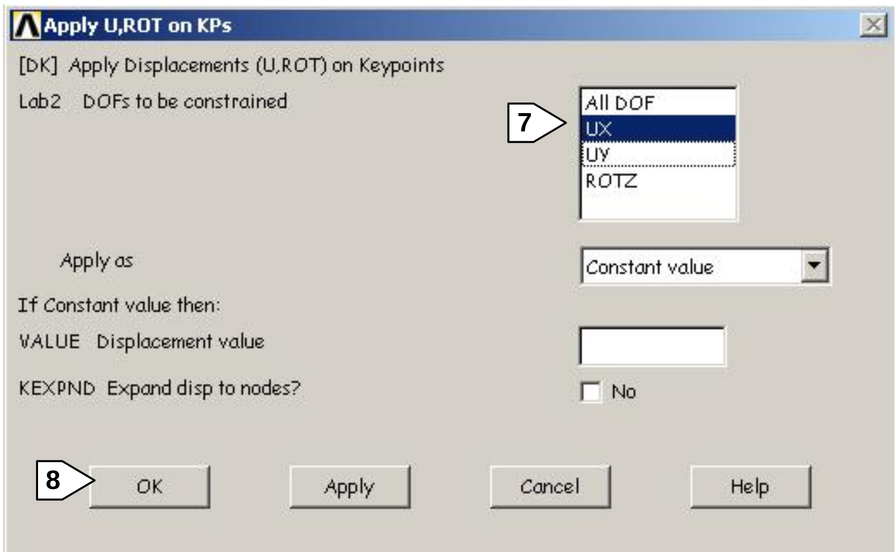
5 Enter node 2

6 Pick OK

7 Choose UX

8 OK





13. Apply load

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

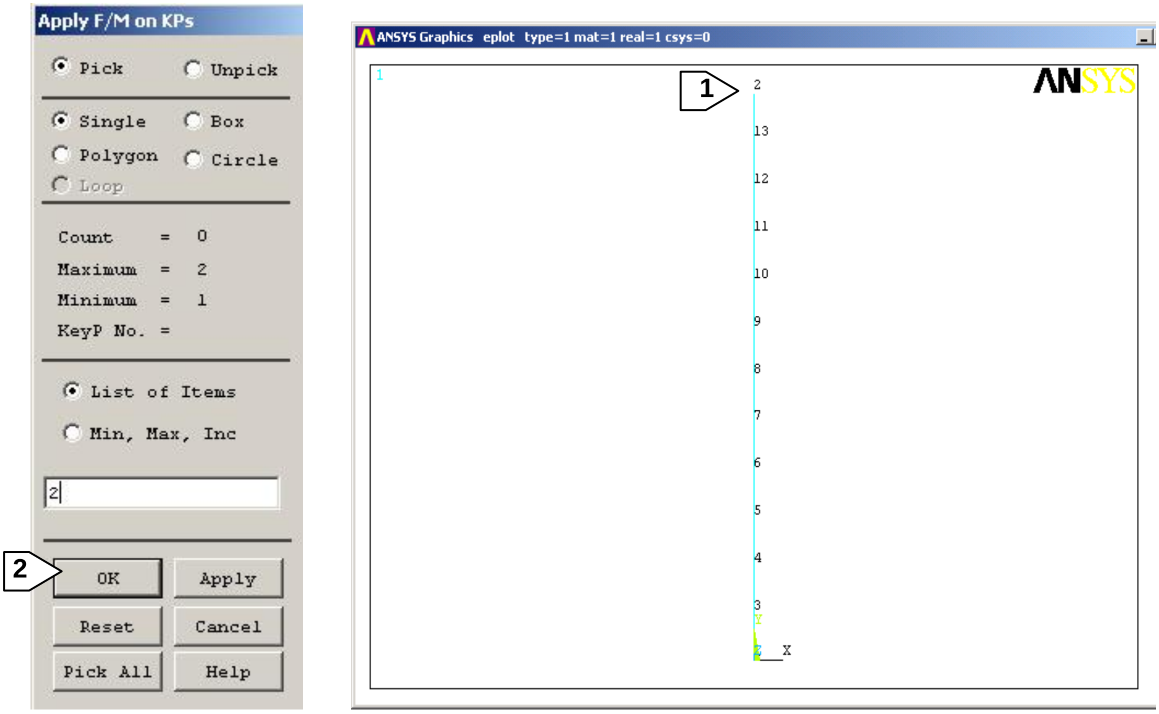
1 Pick on the keypoint 2

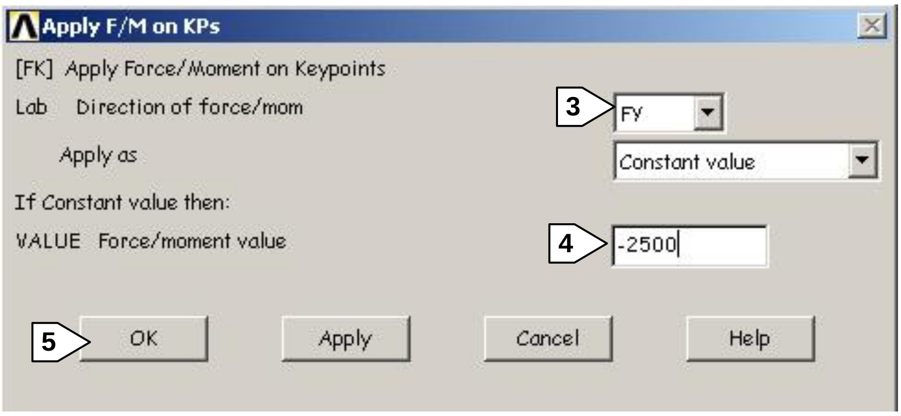
2 OK

3 Choose FY

4 Enter -2500 like force

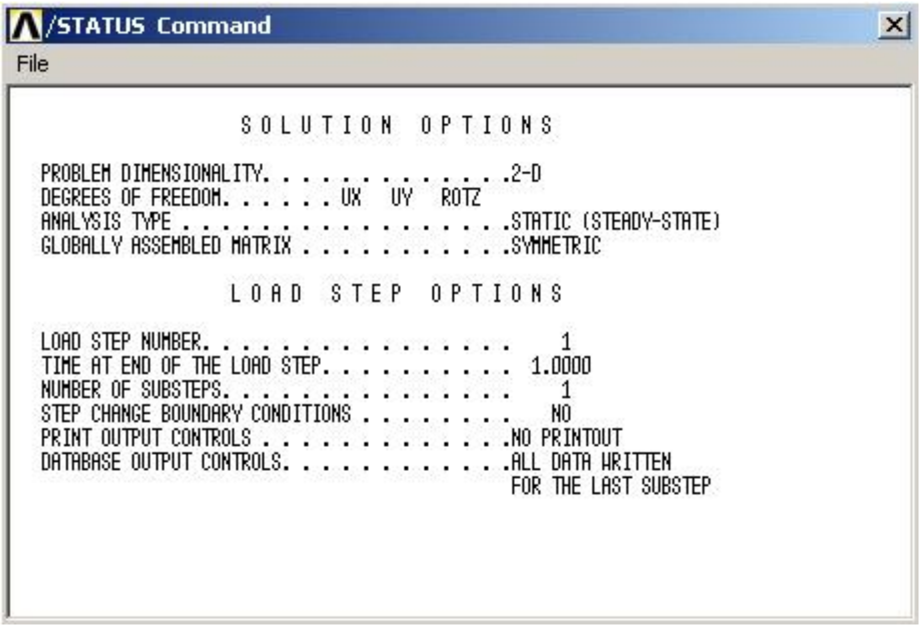
5 OK

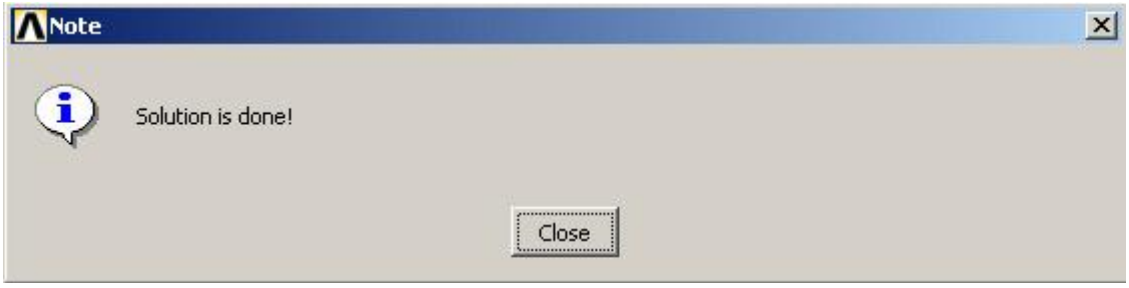




14. Solve

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





Postprocessing

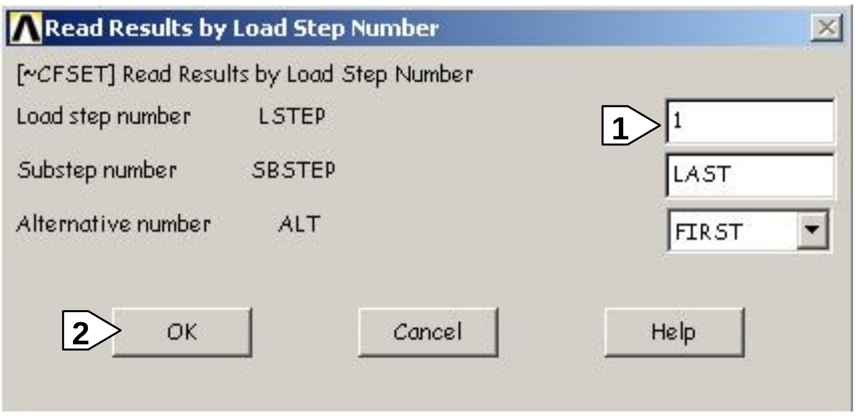
15. Enter the postprocessor and read in results

You must select the load step from you want to read results data from CivilFEM results file. This results file contains the calculated forces, moments and stresses.

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

1 Enter 1 in the Load Step number box

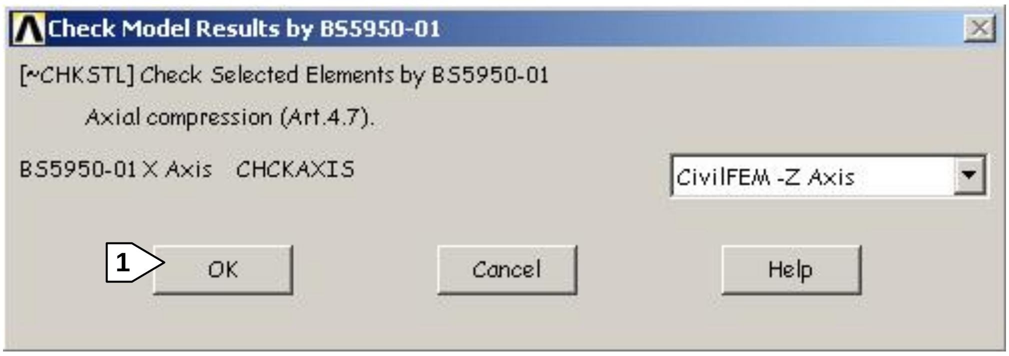
2 OK to read load step 1



16. Checking in Compression buckling

Main Menu: -CivilFEM – **Civil Postprocessor → Code Checking→ BS 5950-01→** CHECK BY CODE: **Compress buckling**

1 OK to check according to BS5950-01



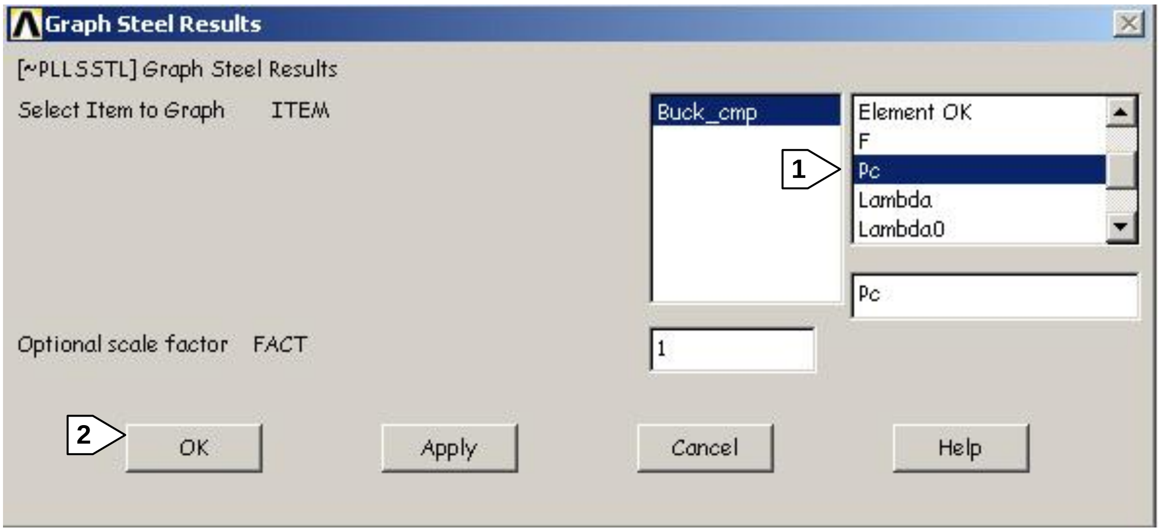
17. Review compression capacity

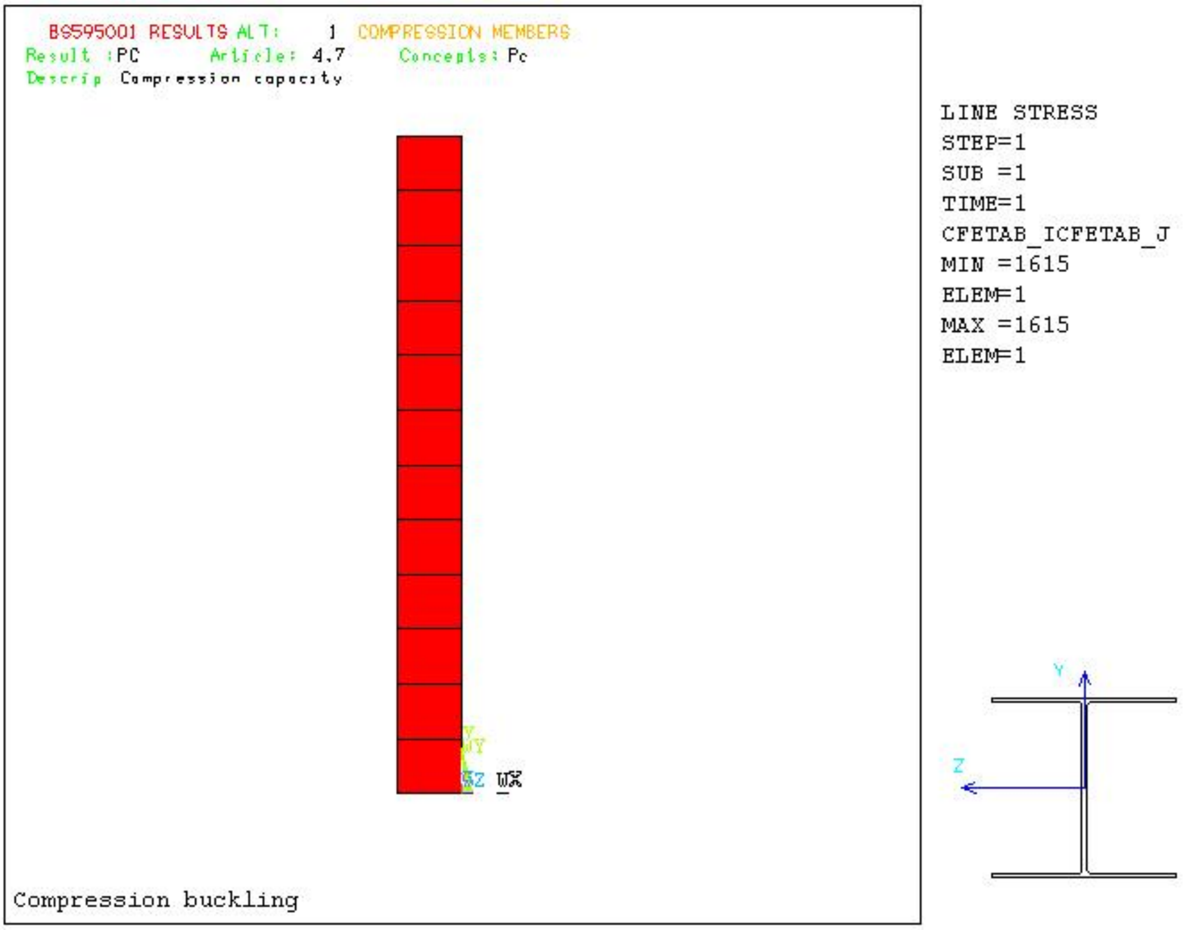
We are going to plot the compression capacity

Main Menu: - CivilFEM – **Civil Postprocessor** → **Code Checking**→ **BS5950-01** → BEAM RESULTS: **Plot Results**

1 Choose PC

2 OK





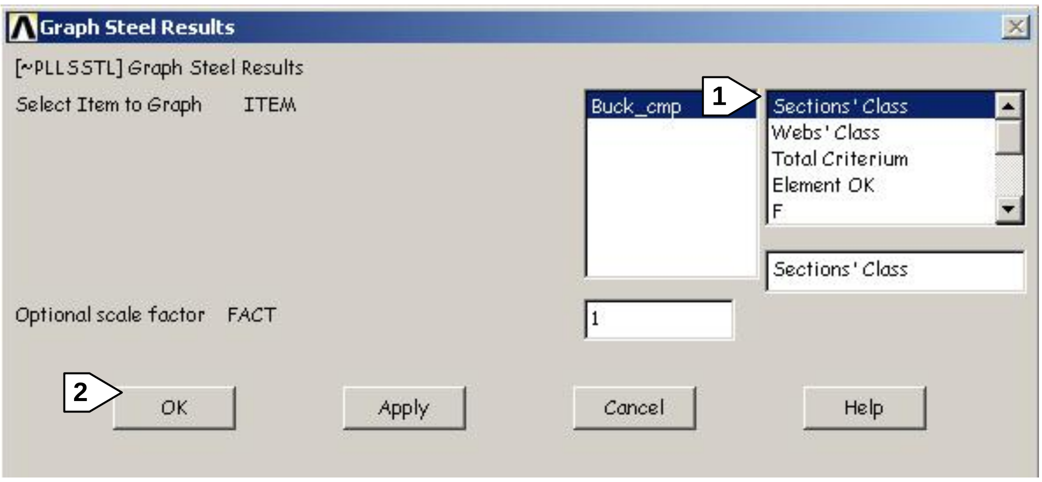
18. Review section class

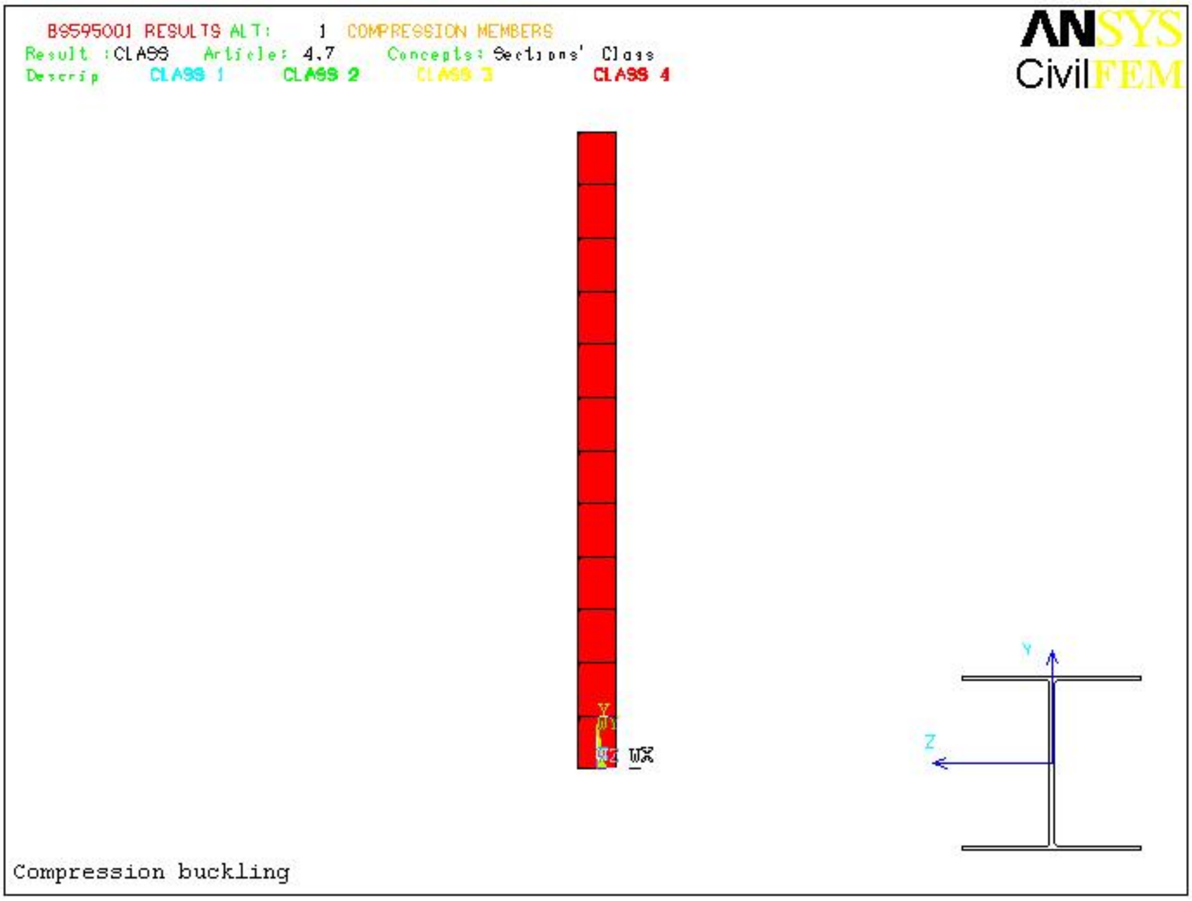
We are going to plot the section class

Main Menu: - CivilFEM – **Civil Postprocessor → Code Checking→ BS5950-01 →** BEAM RESULTS: **Plot Results**

1 Choose Section' Class

2 OK





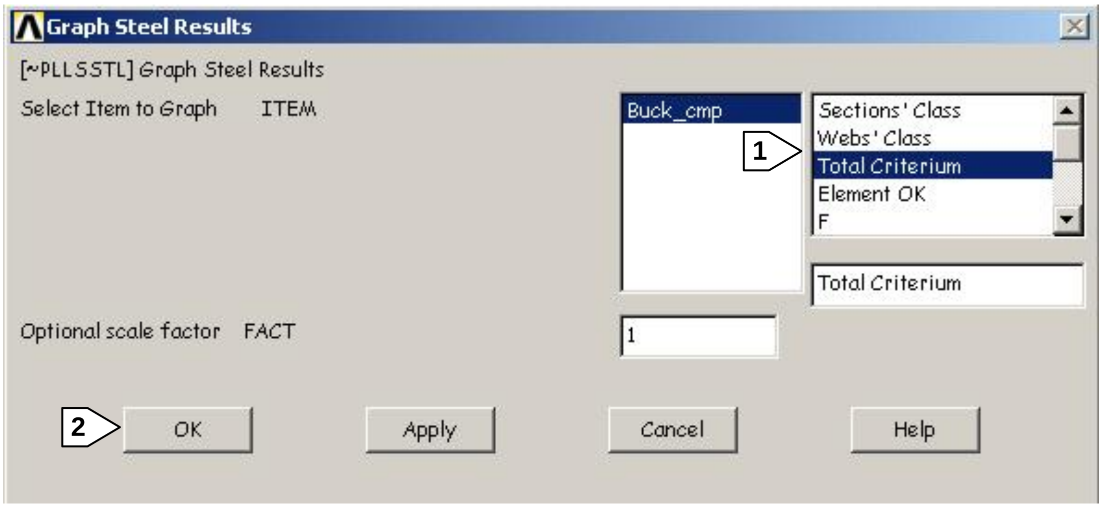
19. Review criterion

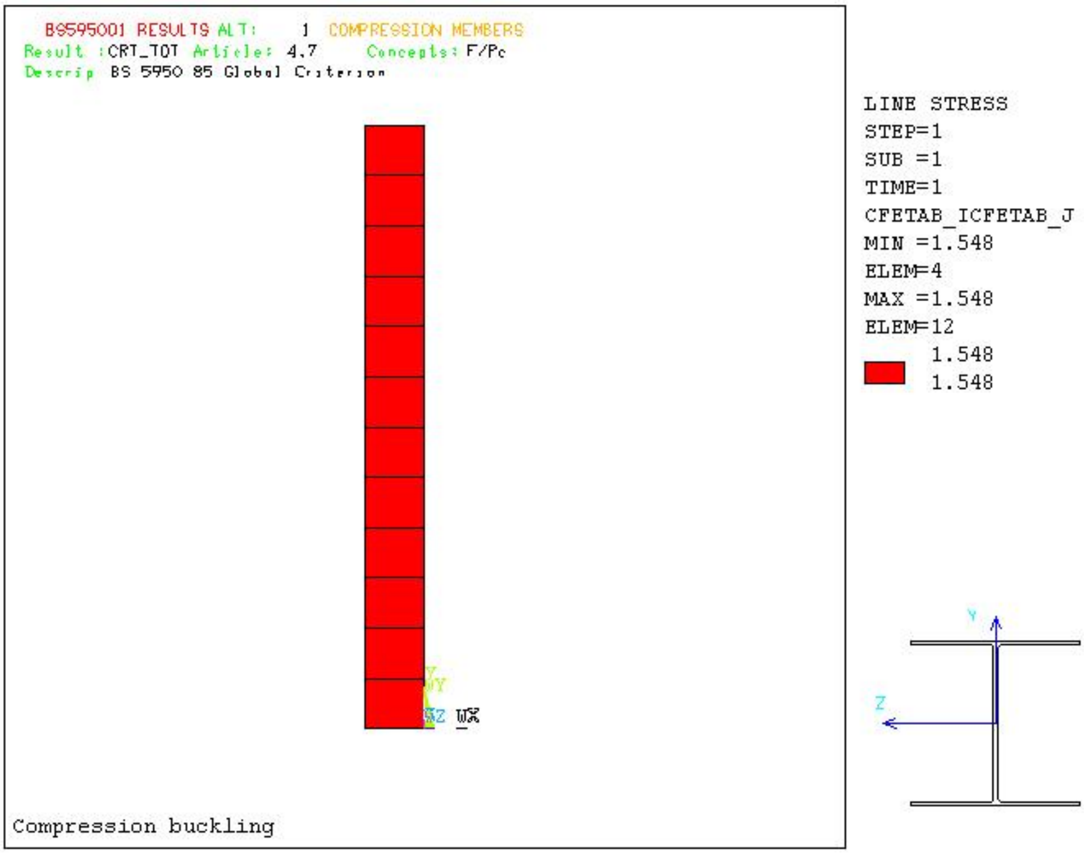
We are going to plot the criterion (must be <1.00)

Main Menu: - CivilFEM – **Civil Postprocessor** → **Code Checking**→ **BS5950-01** → BEAM RESULTS: **Plot Results**

1 Choose Total Criterium

2 OK





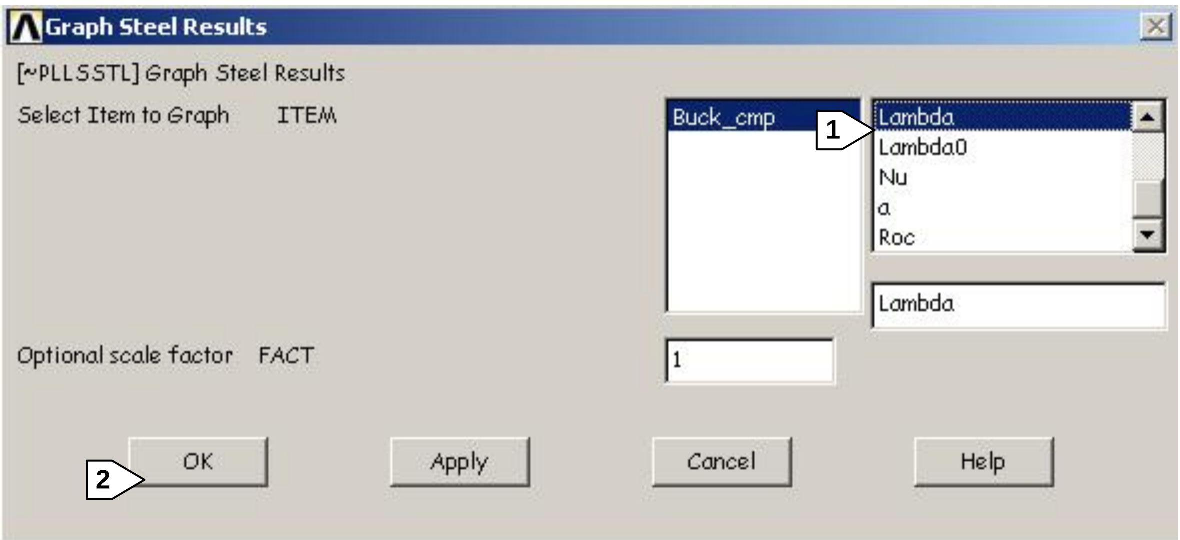
20. Review slenderness

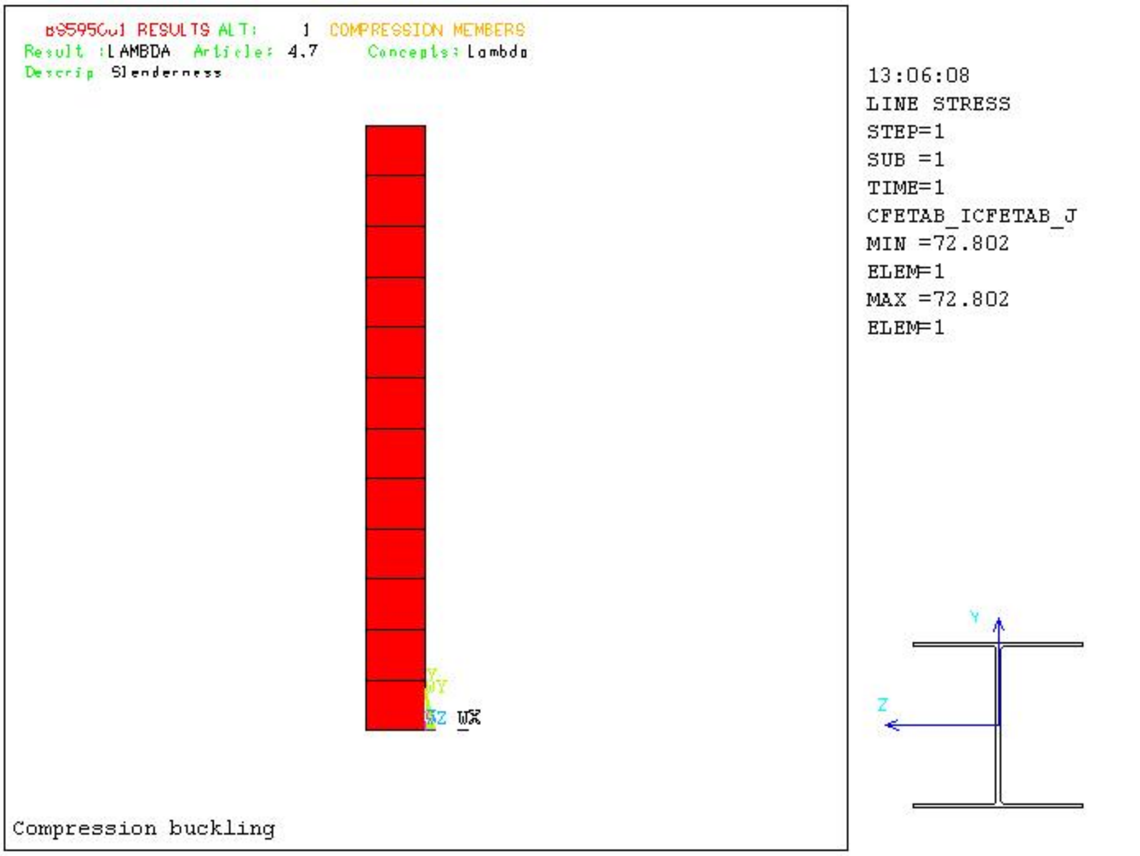
We are going to plot the slenderness

Main Menu: - CivilFEM – **Civil Postprocessor → Code Checking→ BS5950-01 →** BEAM RESULTS: **Plot Results**

1 Choose Lambda

2 OK





21. Exit the ANSYS program

ANSYS Toolbar: **QUIT**

1 Pick on Save Everything option

2 OK