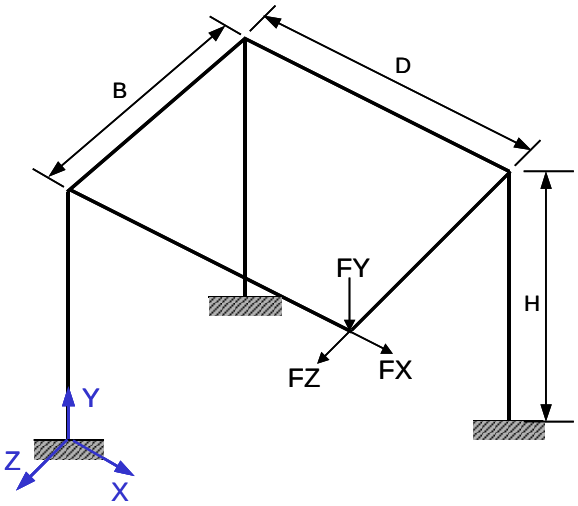
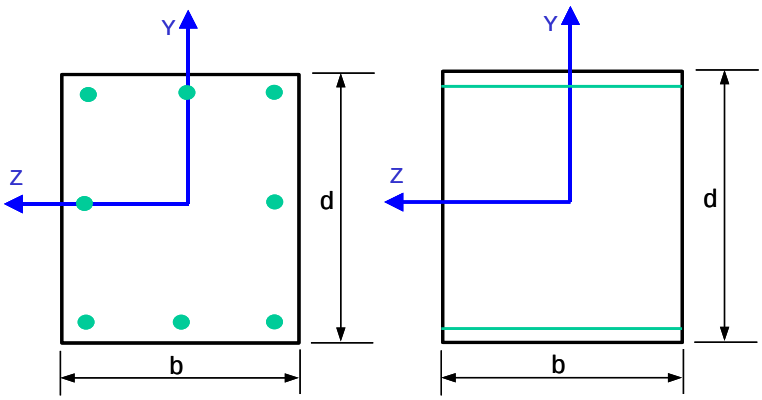
7. Biaxial Bending + Axial Force Checking According to BS 8110

Applicable CivilFEM Product:	All CivilFEM Products
Level of Difficulty:	Moderate
Interactive Time Required:	35 minutes
Discipline:	Structural Concrete
Analysis Type:	Linear static
Element Type Used:	BEAM4
Active Code	British Standard 8110
Units System	Ton, m, s.
CivilFEM Features Demonstrated:	Units selection, code selection, material definition, section definition by dimensions, axial and biaxial bending reinforcement checking.

Problem Description

This example illustrates the British Standard 8110 checking criteria for a concrete frame given bellow. We will use the program skills to check the frame reinforcement for biaxial bending and axial force. The frame has two different cross sections, one for the piers and other for the beams, and it is subjected to the loads shown bellow.





Loads:

FX = 6 Ton

FY = 15 Ton

FZ = 6 Ton

Geometric dimensions:

Length B = 4 m

Depth D = 5 m

Height H = 3 m

Section properties:

Dimensions of section 1:

Width B = 0.3 m

Depth D = 0.3 m

Dimensions of section 2:

Width B = 0.25 m

Depth D = 0.3 m

Material:

Concrete C40

Steel reinforcement Gr460A

Reinforcement:

Bending reinforcement: Group 1 Section 1 3 ϕ 16 mm at face number 1 (Scalable)

Bending reinforcement: Group 2 Section 1 3 ϕ 16 mm at face number 2 (Scalable)

Bending reinforcement: Group 3 Section 1 3 ϕ 16 mm at face number 3 (Scalable)

Bending reinforcement: Group 4 Section 1 3 ϕ 16 mm at face number 4 (Scalable)

Bending reinforcement: Group 1 Section 2 Total area at top = 6E-3 m^2

Bending reinforcement: Group 2 Section 2 Total area at bottom = 1E-3 m^2

Given

The load distribution, the section geometrical dimensions, material properties and reinforcement distribution are shown in the previous table.

Approach and Assumptions

We will use 3D elastic beam elements for this analysis. Model geometry will be defined by direct 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 sections
7. Define bending reinforcement properties
8. Define Beam & Shell properties
9. Define model geometry
10. Save the database

Solution

11. Apply displacement constraints
12. Apply load
13. Solve

Postprocessing

14. Enter the postprocessor and read in results
15. Checking under 3D biaxial bending moment and axial force
16. Review elements OK and No OK
17. Review BS8110 total criterion
18. Plot an interaction diagram
19. Exit the ANSYS program

Interactive Step-by-Step Solution

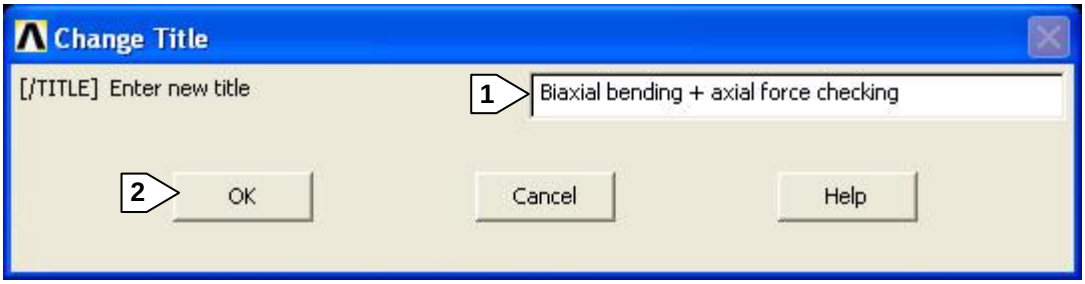
■ Preprocessing

1. Specify title

Utility Menu: **File →Change title**

1 Enter the title: “Biaxial bending + axial force checking”

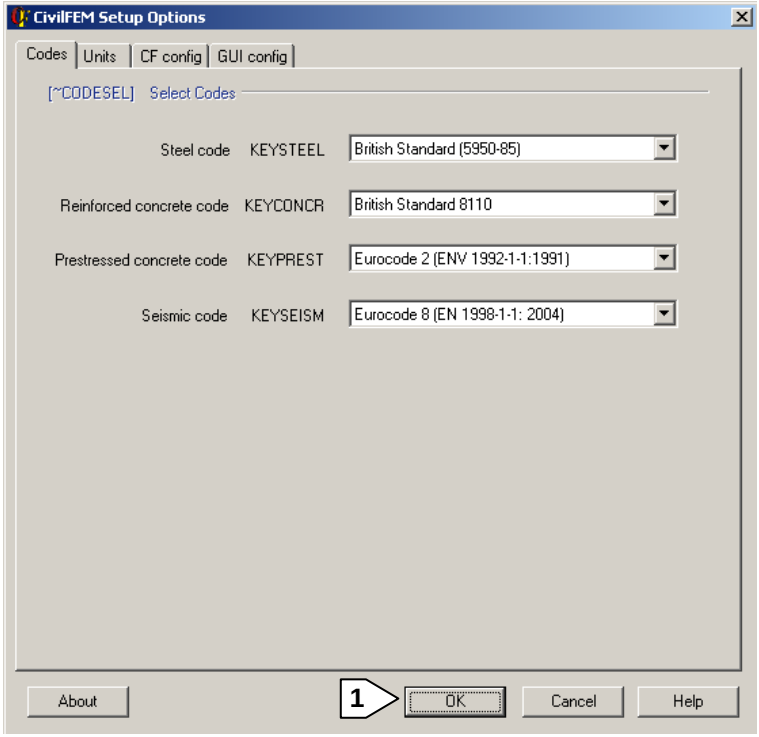
2 OK to define the title and close the dialog box.



2. Set code

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

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



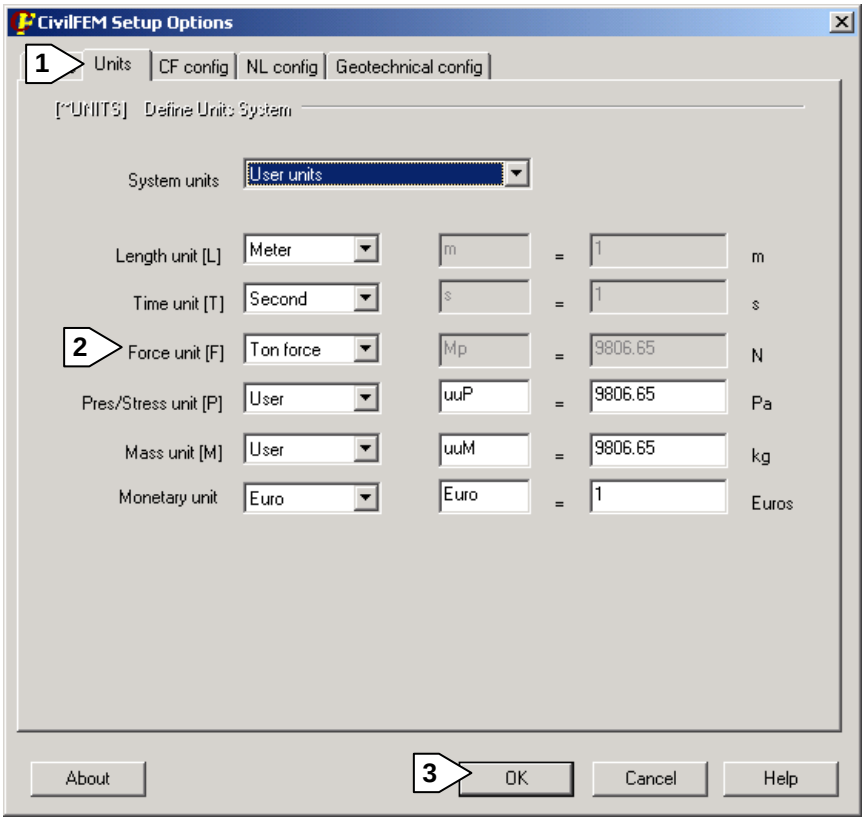
3. Set units

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

1 Pick on the Units tab

2 Change to user units and choose Ton force

3 Ok



4. Define material

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

1 Choose New

2 Select concrete material type

3 Choose BS 8110: C40 concrete

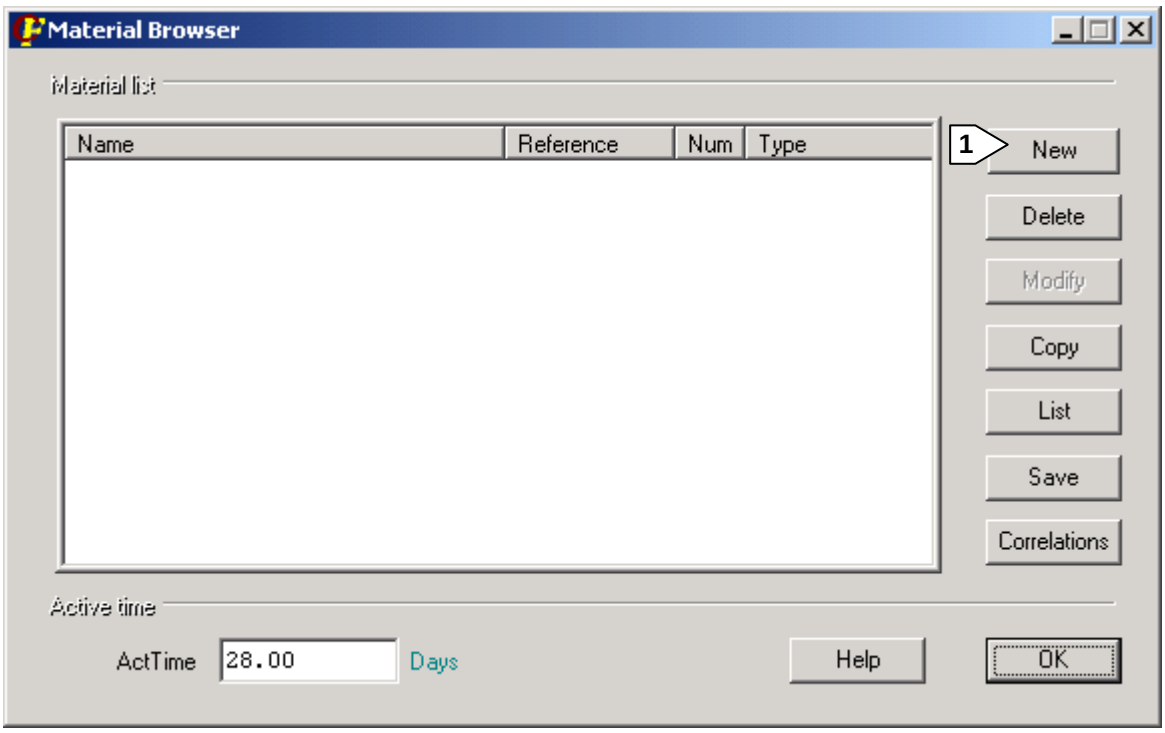
4 Add to define material 1

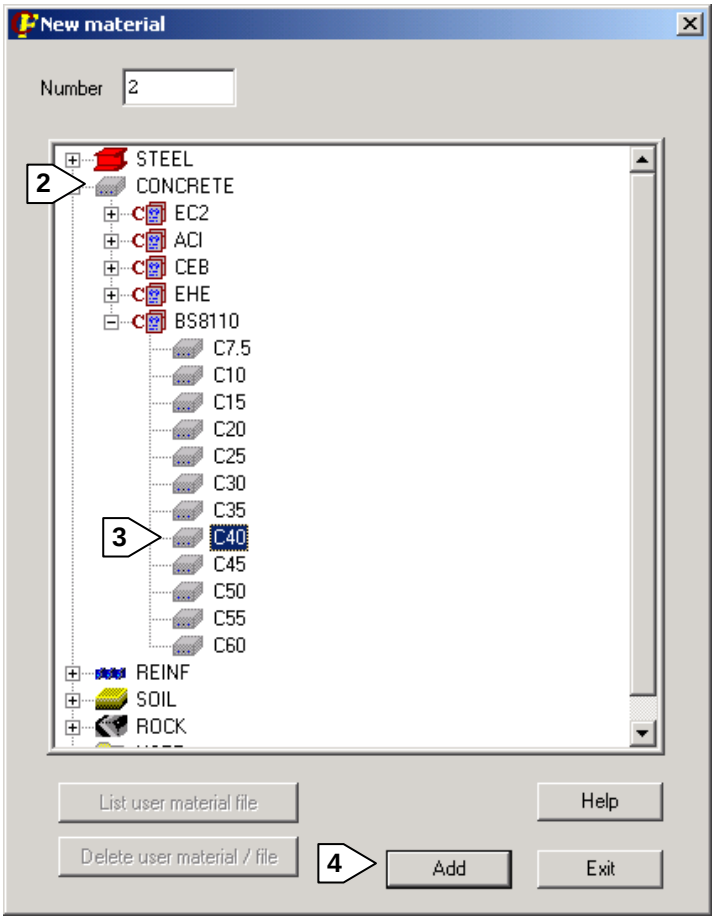
5 Select reinforcing steel material type

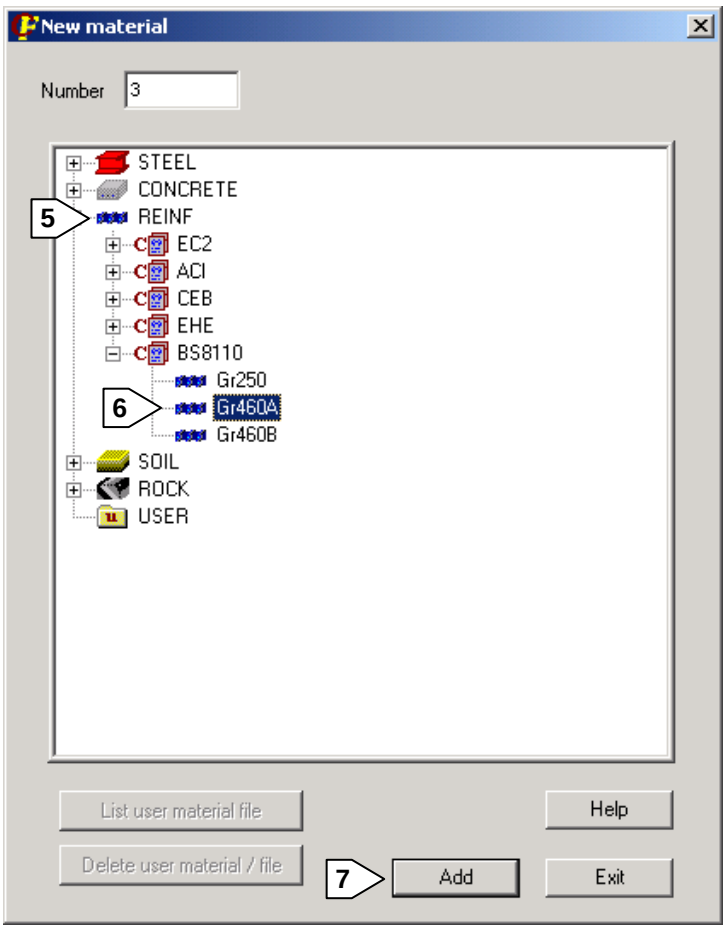
6 Choose BS 8110: Gr460A reinforcing steel

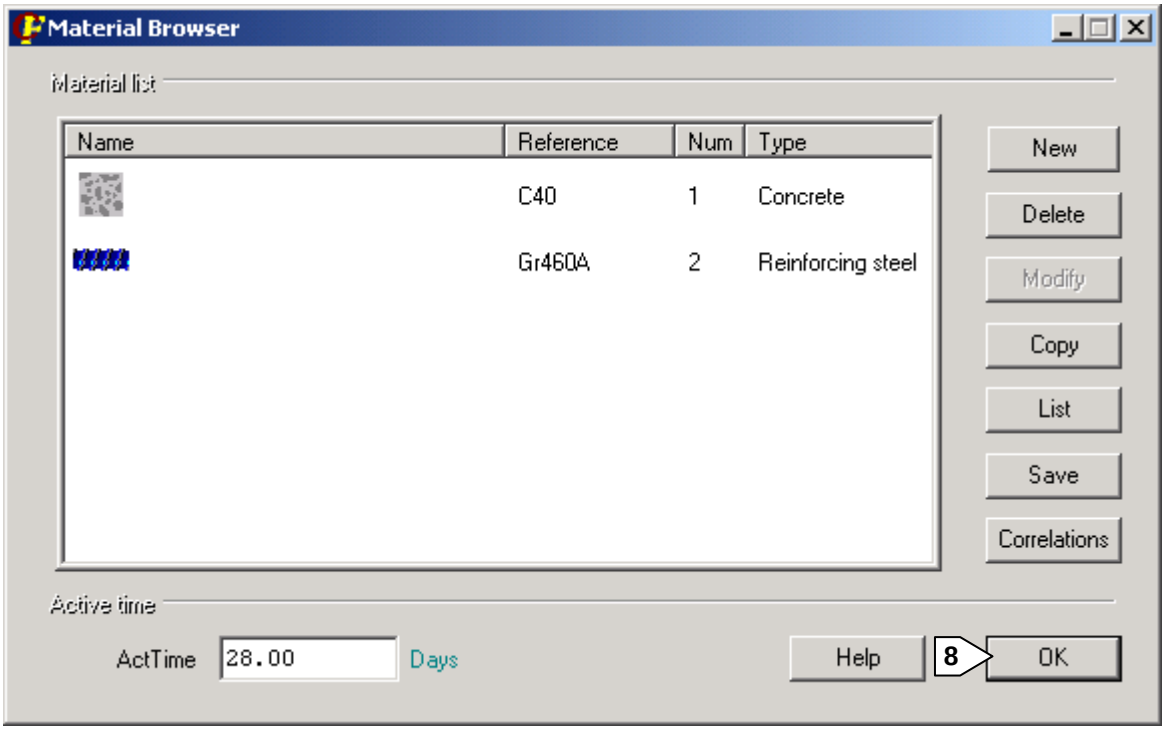
7 Add to define material 2 and Exit.

8 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
2
STEEL
2
CONCRETE
EC2
ACI
CEB
EHE
BS8110
C7.5
C10
C15
C20
C25
C30
C35
3
C40
C45
C50
C55
C60
REINF
SOIL
ROCK
List user material file
Help
Delete user material / file
4
Add
Exit


New material
Number
3
STEEL
CONCRETE
5
REINF
EC2
ACI
CEB
EHE
BS8110
Gr250
6
Gr460A
Gr460B
SOIL
ROCK
USER
List user material file
Help
Delete user material / file
7
Add
Exit


Material Browser
Material list
Name
Reference
Num
Type
C40
1
Concrete
Gr460A
2
Reinforcing steel
New
Delete
Modify
Copy
List
Save
Correlations
Active time
ActTime
28.00
Days
Help
8
OK

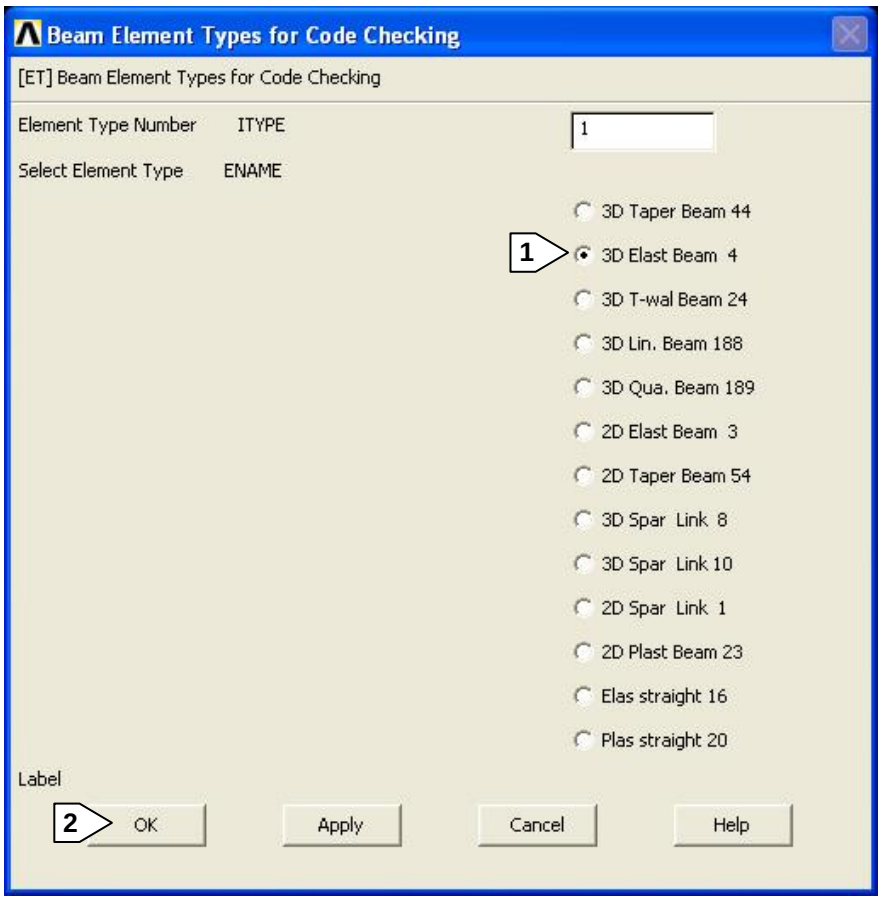
5. Define element type

We will make use of BEAM4.

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

1 Select 3D Elastic Beam 4

2 OK to define element type



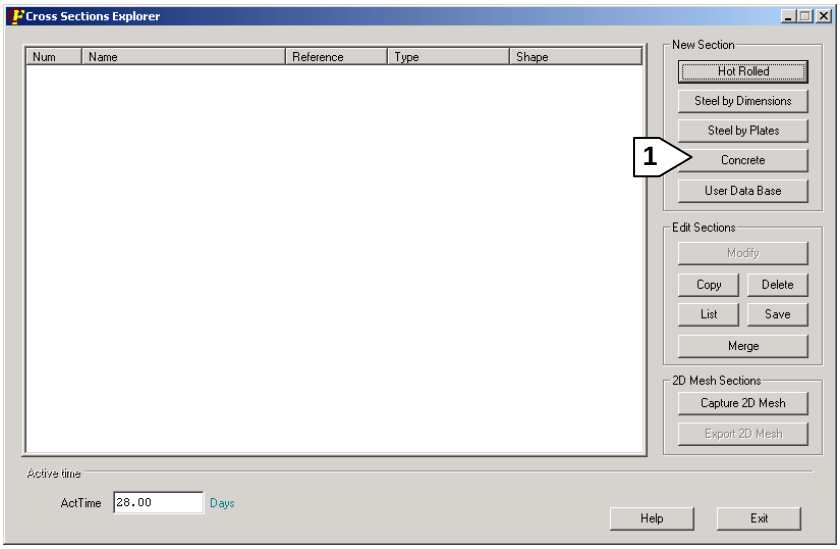
6. Define section

First, we will define the section geometry and then we will introduce the reinforcements.

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

1 Pick on the Concrete button to define a concrete section

2 Choose rectangular shape



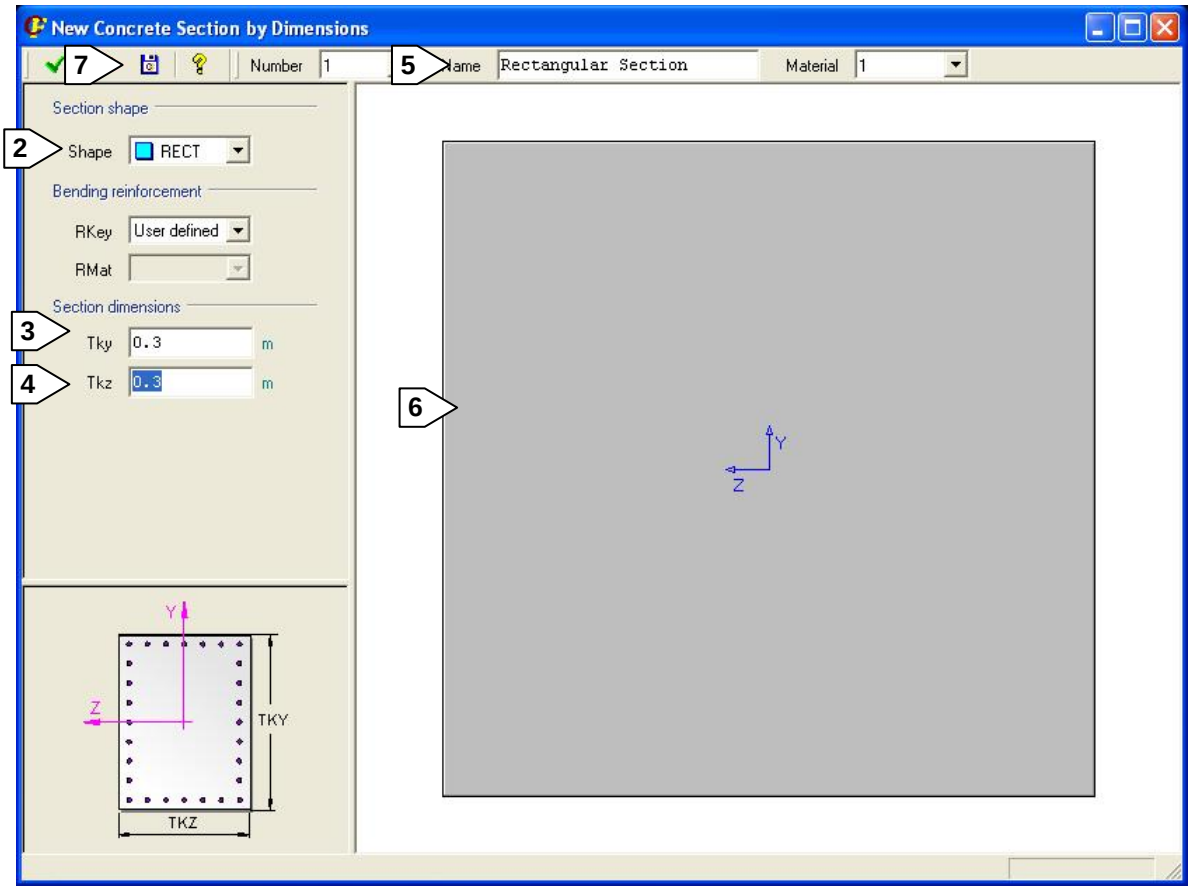
3 Enter 0.3 as depth (TKY)

4 Enter 0.3 as width (TKZ)

5 Enter Rectangular Section as Name

6 To draw the rectangular section click with the right button on the window

7 Apply to define Concrete Section



8 Choose rectangular shape

9 Enter 0.3 as depth (TKY)

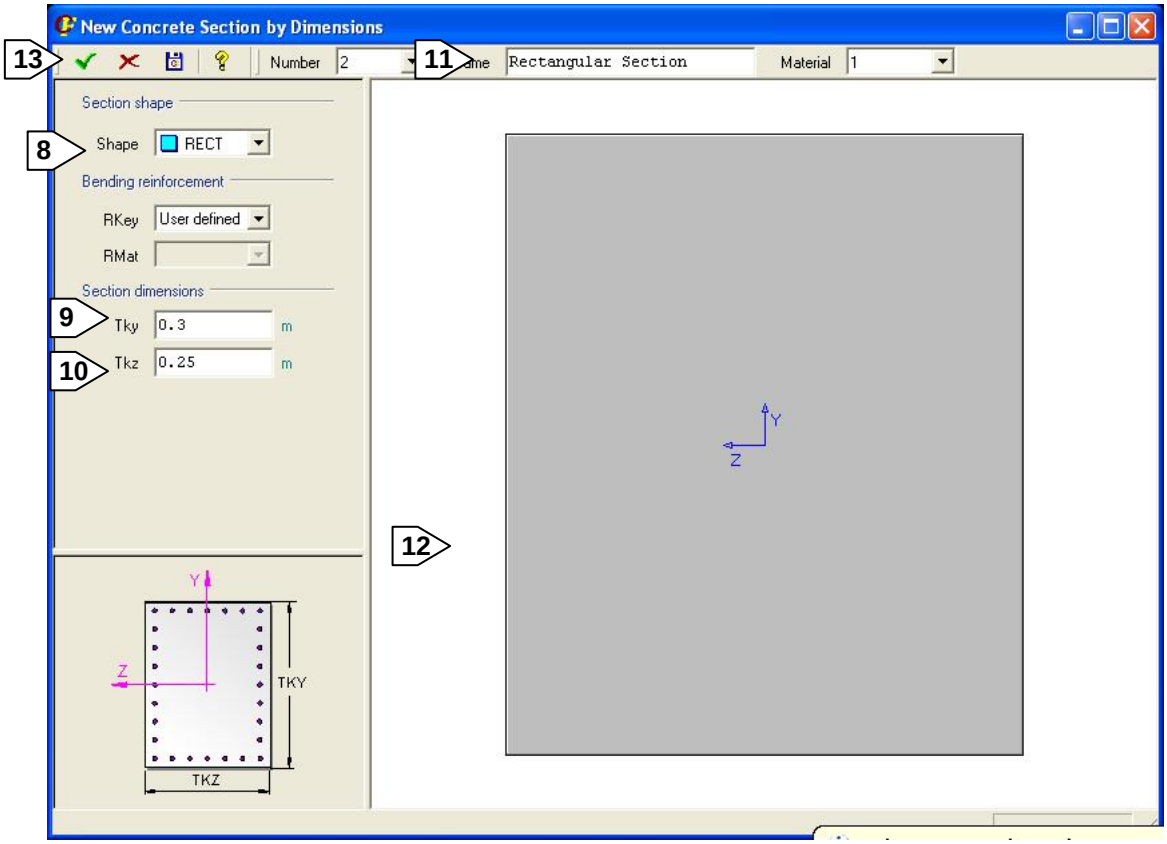
10 Enter 0.25 as width (TKZ)

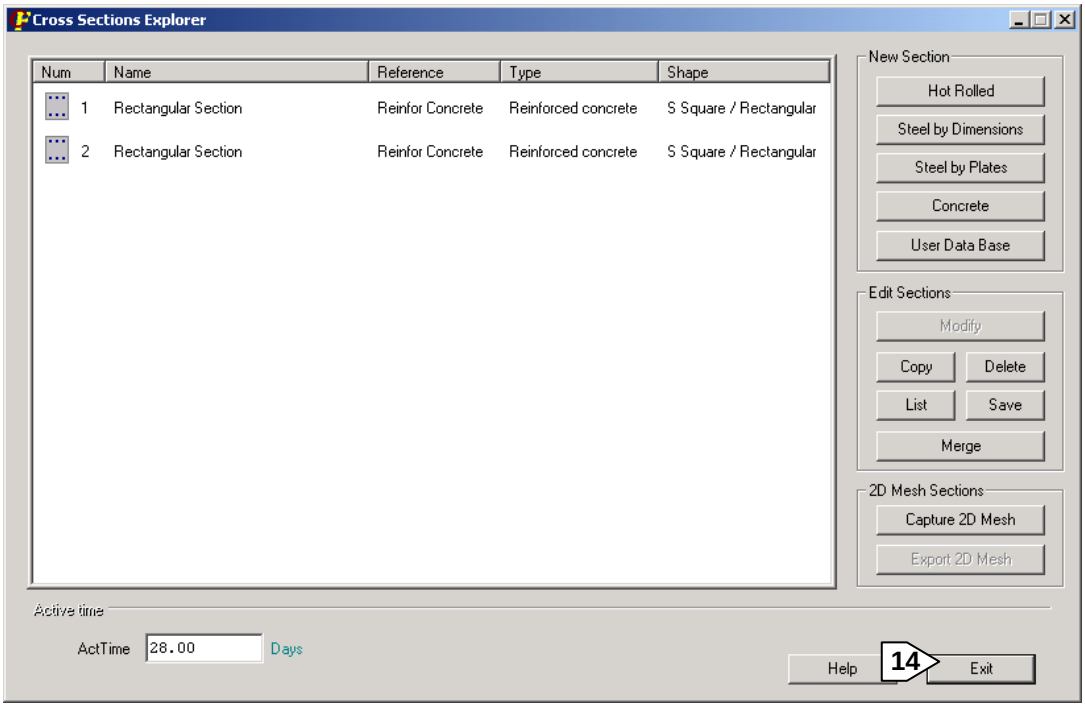
11 Enter Rectangular Section as Name

12 To draw the rectangular section click with the right button on the window

13 Ok to define Concrete Section

14 Exit to close dialog box





7. Define bending reinforcement properties

To define the bending reinforcement we have to take into account the following facts:

a) Number of reinforcement groups to be defined.
b) Material of reinforcement group.
c) Class of reinforcement group.
d) Amount of reinforcement per group.
e) Location of reinforcement group.

The characteristics of the reinforcement of group 1 of the section 1 are shown in the following table:

Group	**1**
Reinforcement material	Material 2 (Gr460A)
Reinforcement class	Scalable
Diameter	16 mm
Number of bars	3
Face number	1
Geometrical cover	0.03 m

The characteristics of the reinforcement of group 2 of the section 1 are shown in the following table:

Group	**2**
Reinforcement material	Material 2 (Gr460A)
Reinforcement class	Scalable
Diameter	16 mm
Number of bars	3
Face number	2
Geometrical cover	0.03 m

The characteristics of the reinforcement group 3 of the section 1 are shown in the following table:

Group	**3**
Reinforcement material	Material 2 (Gr460A)
Reinforcement class	Scalable
Diameter	16 mm
Number of bars	3
Face number	3
Geometrical cover	0.03 m

The characteristics of the reinforcement group 4 of the section 1 are shown in the following table:

Group	**4**
Reinforcement material	Material 2 (Gr460A)
Reinforcement class	Scalable
Diameter	16 mm
Number of bars	3
Face number	4
Geometrical cover	0.03 m

The characteristics of the reinforcement group 1 of the section 2 are shown in the following table:

Group	**1**
Reinforcement material	Material 2 (Gr460A)
Reinforcement class	Scalable
Total reinforcement area	0.6E-2
Face number	2
Geometrical cover	0.03 m

The characteristics of the reinforcement group 2 of the section 2 are shown in the following table:

Group	**2**
Reinforcement material	Material 2 (Gr460A)
Reinforcement class	Scalable
Total reinforcement area	0.1E-2
Face number	4
Geometrical cover	0.03 m

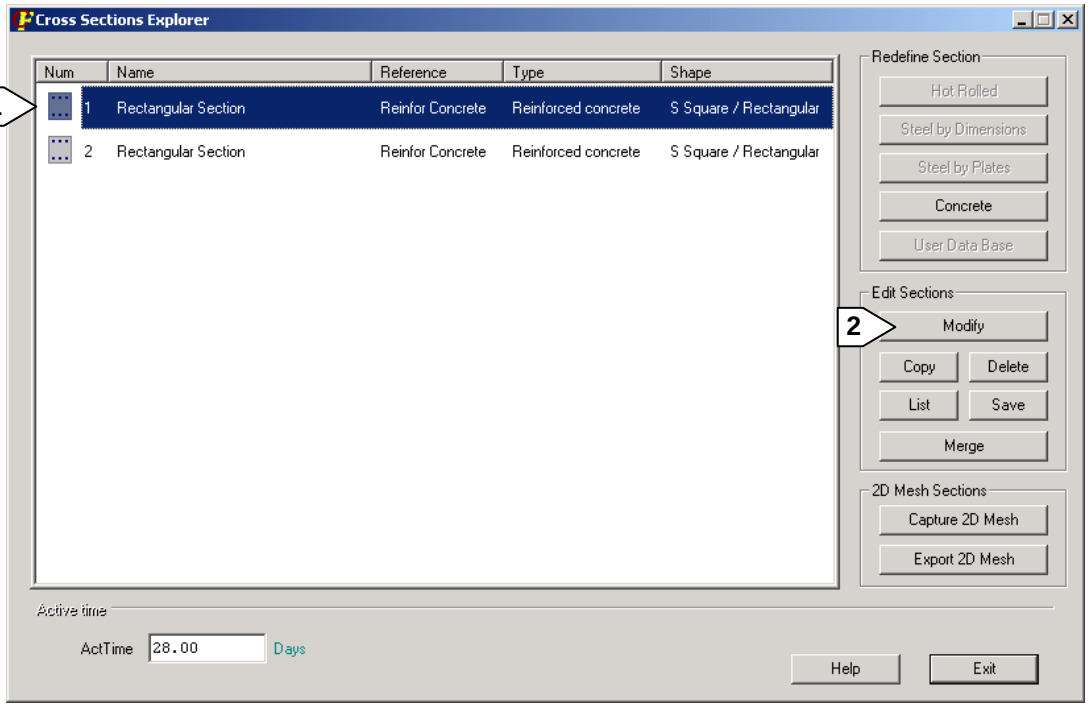
First, we will define the reinforcement group 1:

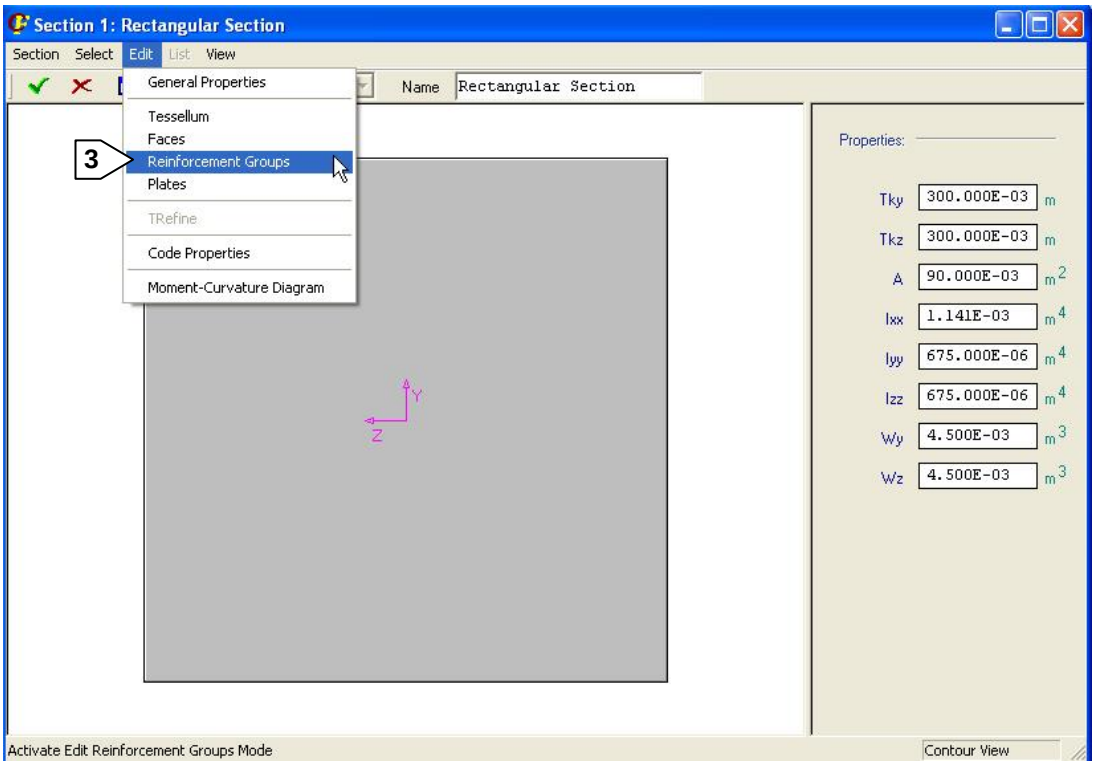
Main Menu: – CivilFEM – **Civil Preprocess → Cross Sections**

1 Select the Reinforced Concrete section

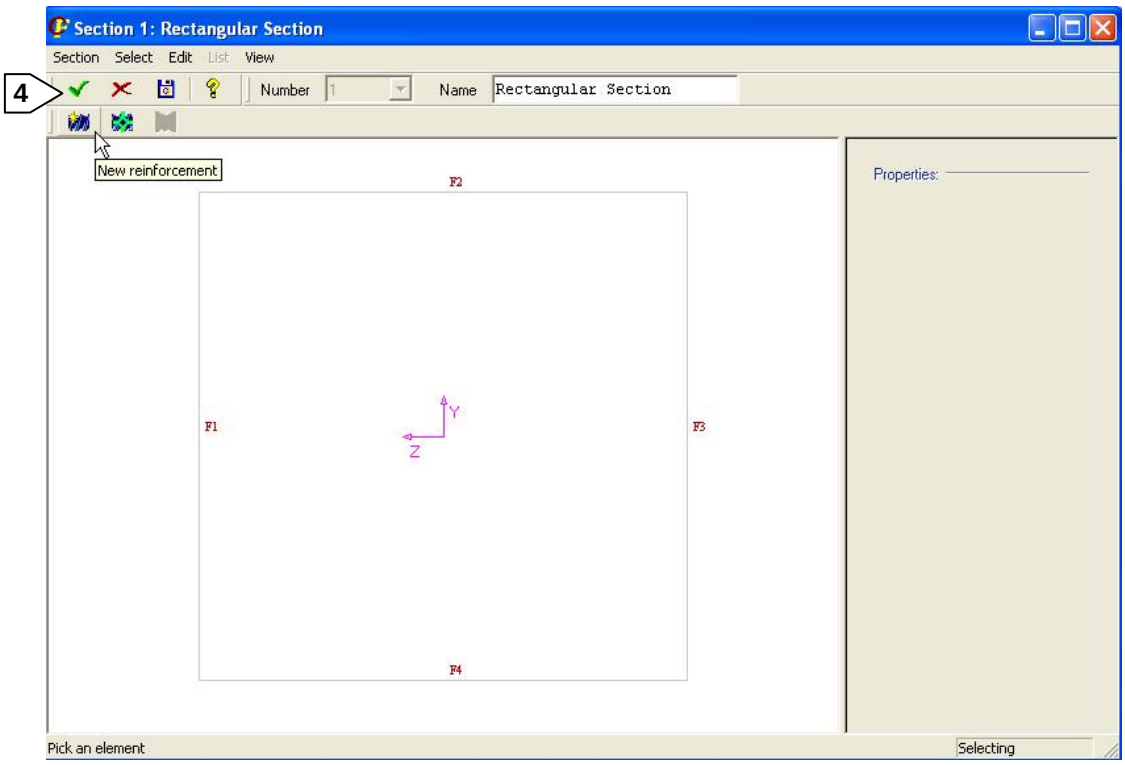
2 Pick on the Modify button

3 Edit Reinforcement Groups





4 New Reinforcement



5 Enter 0.03 for Geometrical cover

6 Select Reinforcement at Face 1

7 Select material 2 as material for bars

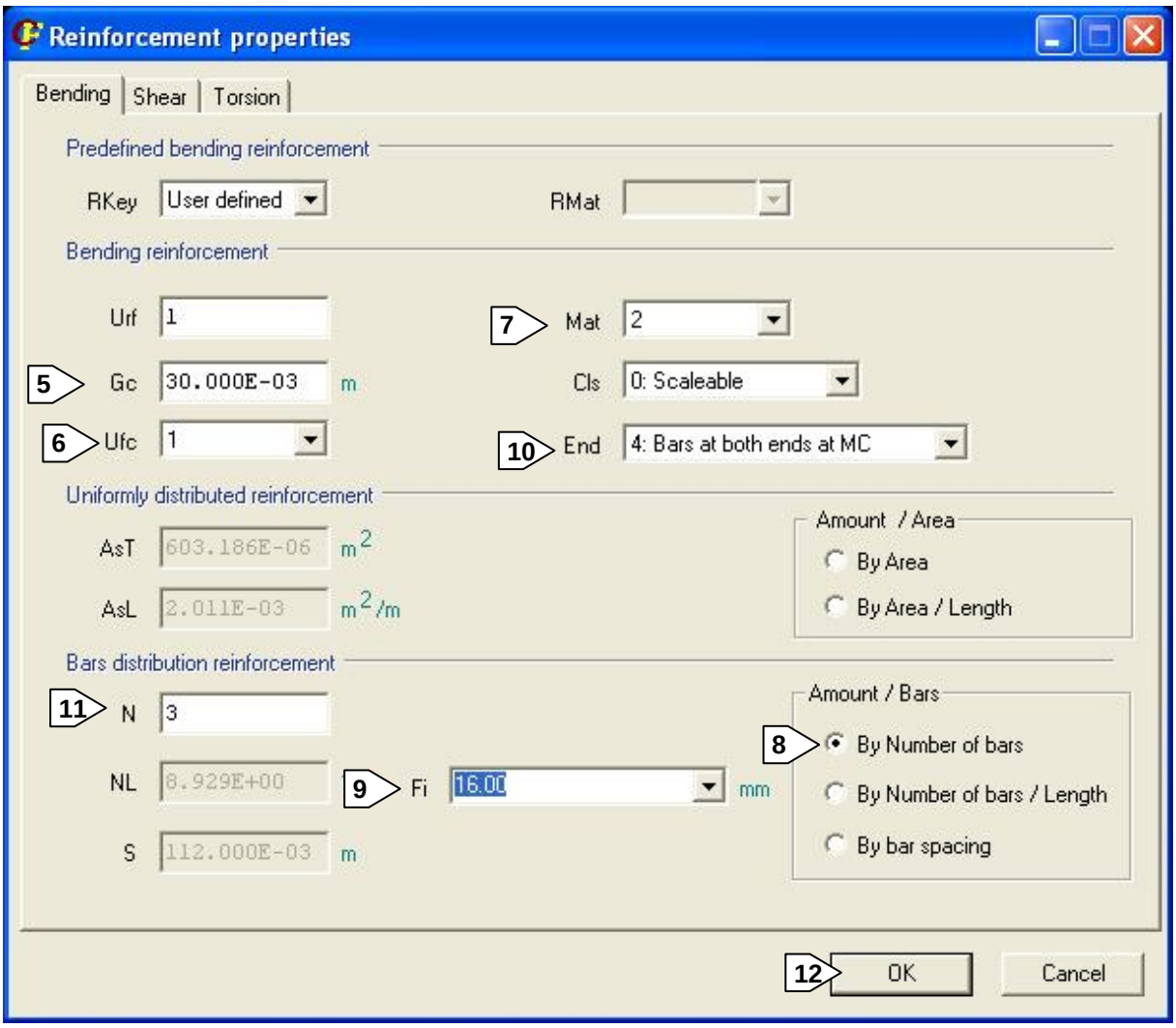
8 Specify reinforcement distribution by number of bars

9 Select 16 mm as Diameter of Bars

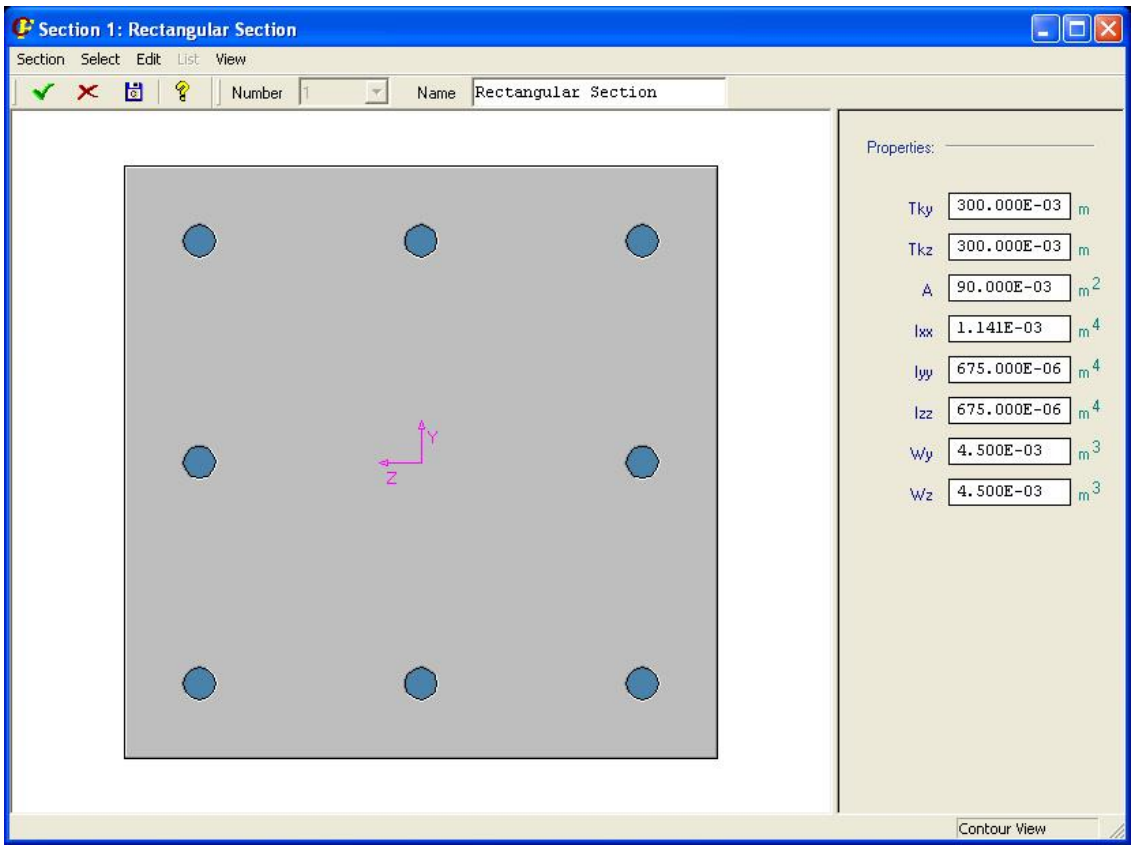
10 Select Bars at both ends at MC

11 Enter 3 as number of bars

12 Ok to define reinforcement group 1



The others three reinforcements of this section are made follow the same procedure.



13 New Reinforcement for section 2

14 Enter 0.03 for Geometrical cover

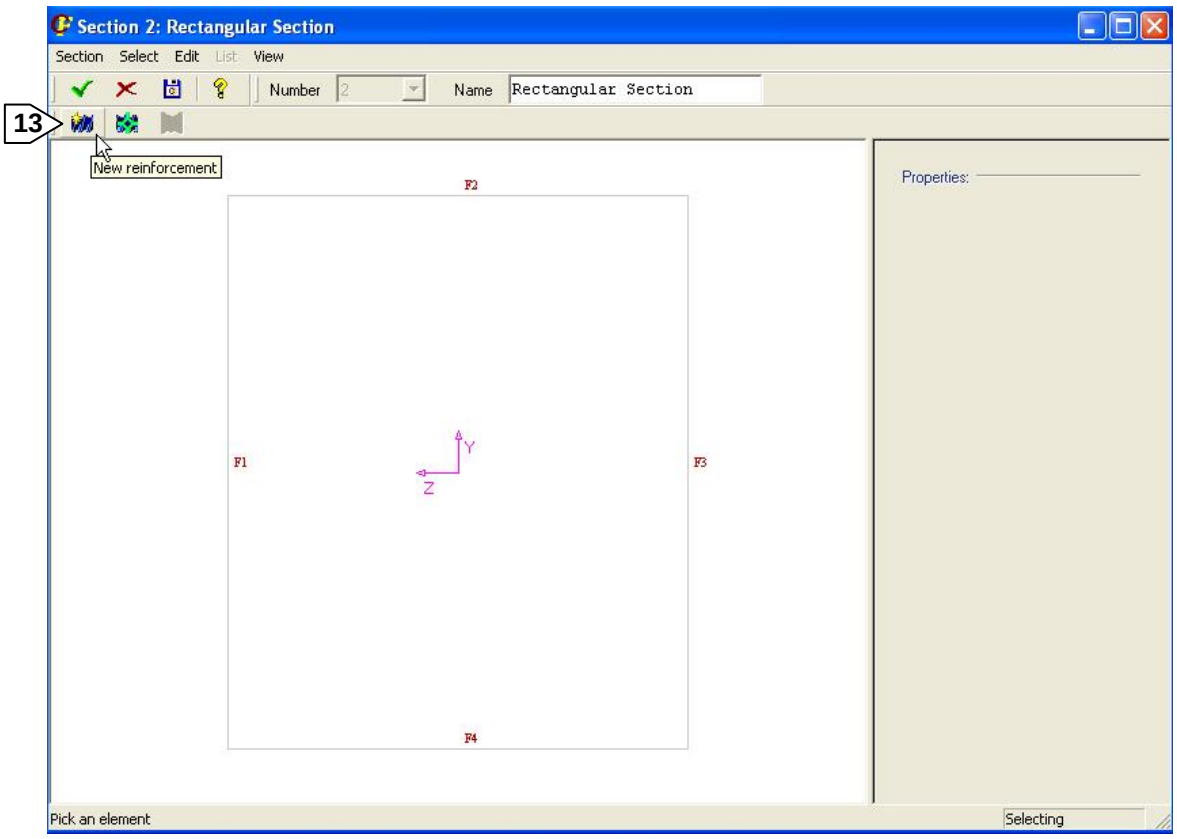
15 Select Reinforcement at Face 2

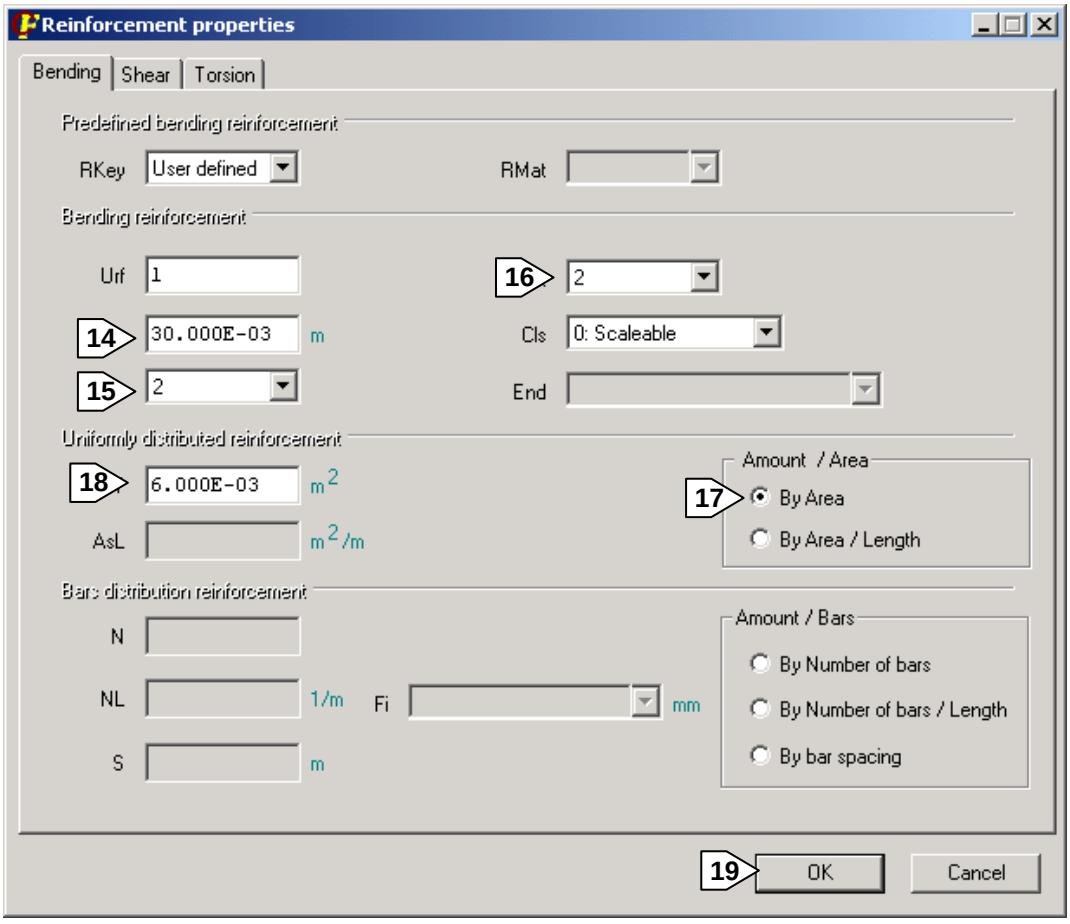
16 Select material 2 as material for bars

17 Specify reinforcement distribution by Areas

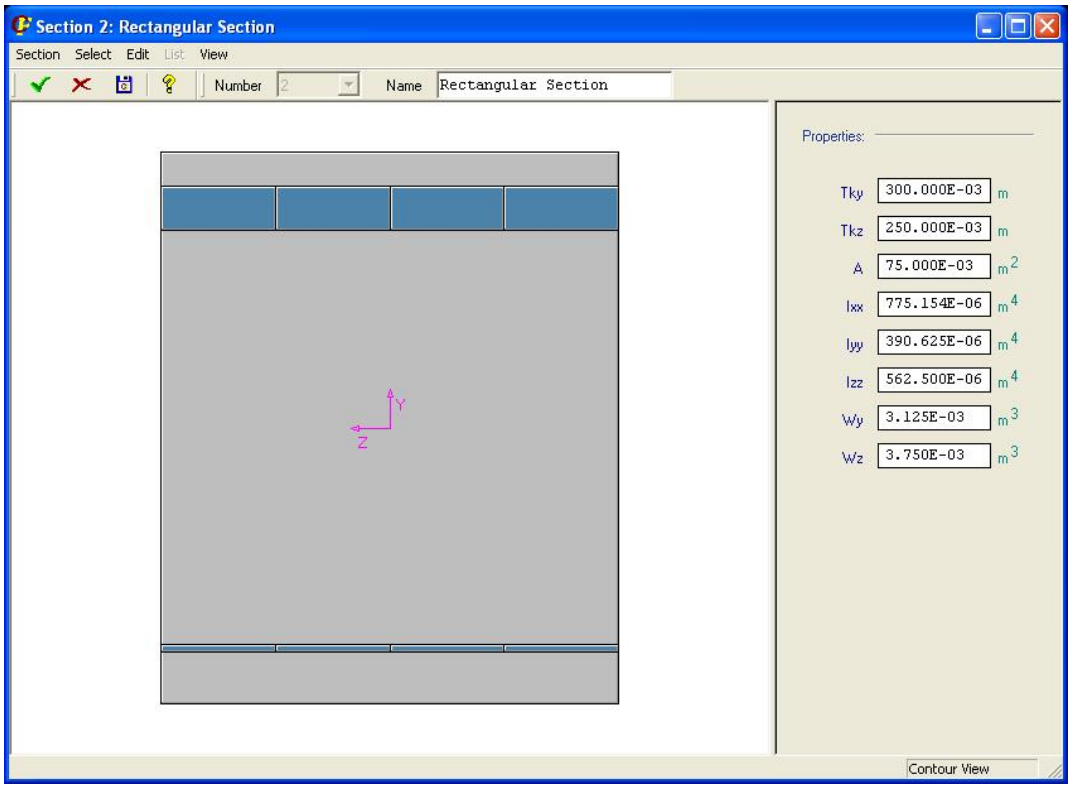
18 Select 6E-3 m^2 as total reinforcement area

19 Ok


Section 2: Rectangular Section
Section Select Edit List View
Number 2
Name Rectangular Section
13
New reinforcement
F2
F1
F3
F4
Y
Z
Properties:
Pick an element
Selecting


Reinforcement properties
Bending
Shear
Torsion
Predefined bending reinforcement
RKey User defined
RMat
Bending reinforcement
Urf 1
16 2
14 30.000E-03 m
Cls 0: Scaleable
15 2
End
Uniformly distributed reinforcement
18 6.000E-03 m2
AsL m2/m
Amount / Area
17 By Area
By Area / Length
Bars distribution reinforcement
N
NL 1/m
Fi mm
S m
Amount / Bars
By Number of bars
By Number of bars / Length
By bar spacing
19 OK
Cancel

The other reinforcement of this section is made follow the same procedure:

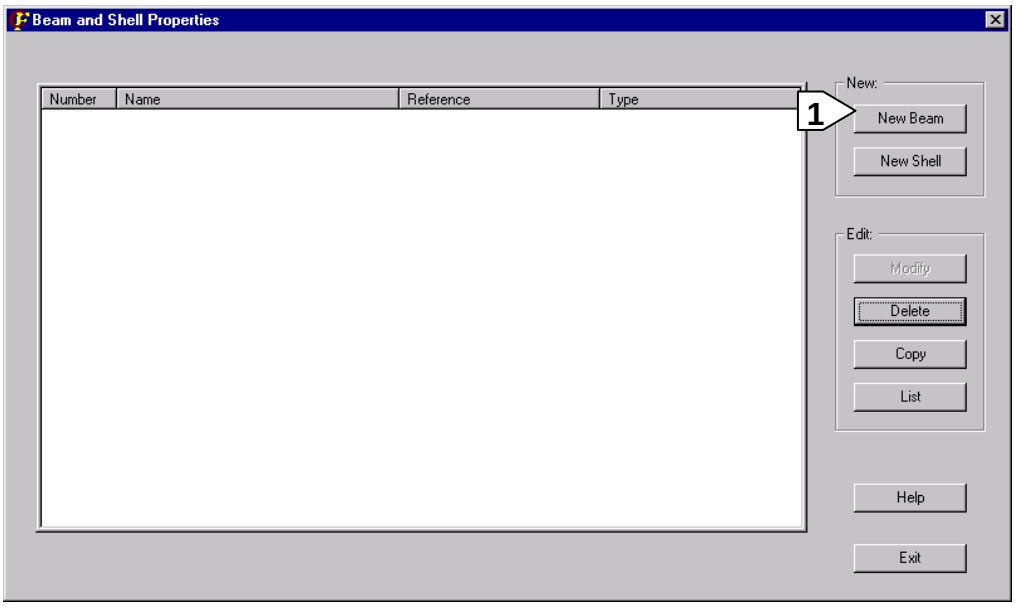


8. Define Beam & Shell properties

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

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

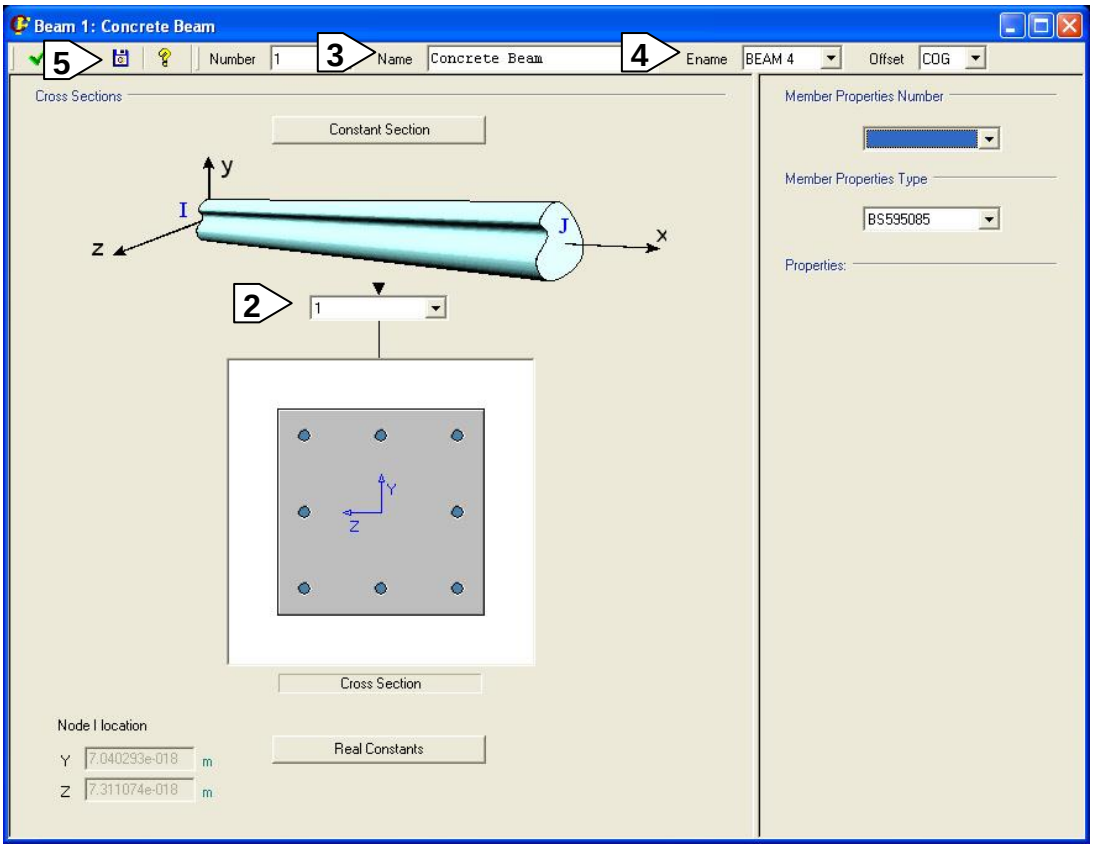
1 Pick on the New Beam button



2 Select cross section number 1

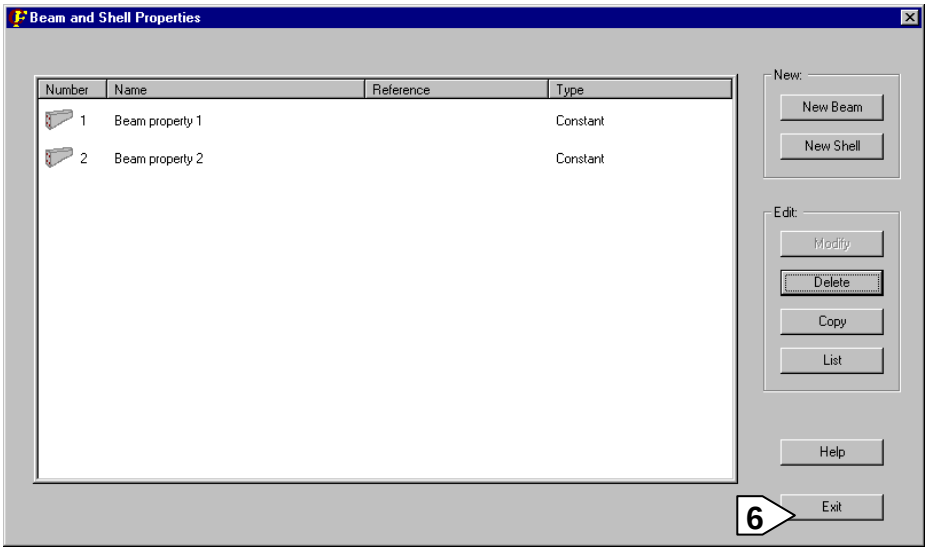
3 Enter “Concrete Beam” as Name for the Beam property

4 Select element type 4



5 Apply (Do the same procedure to define Beam&Shell property number 2 (cross section 2)

6 Exit



9. Define model geometry

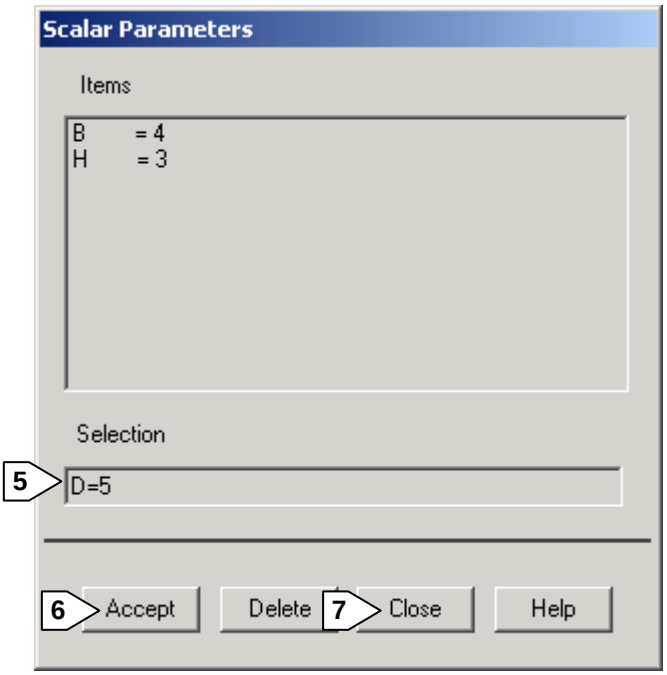
Geometry definition implies the generation of nodes and elements.

The steps that you must follow are:

a) Define parameters
b) Define keypoints
c) Define lines
d) Mesh lines

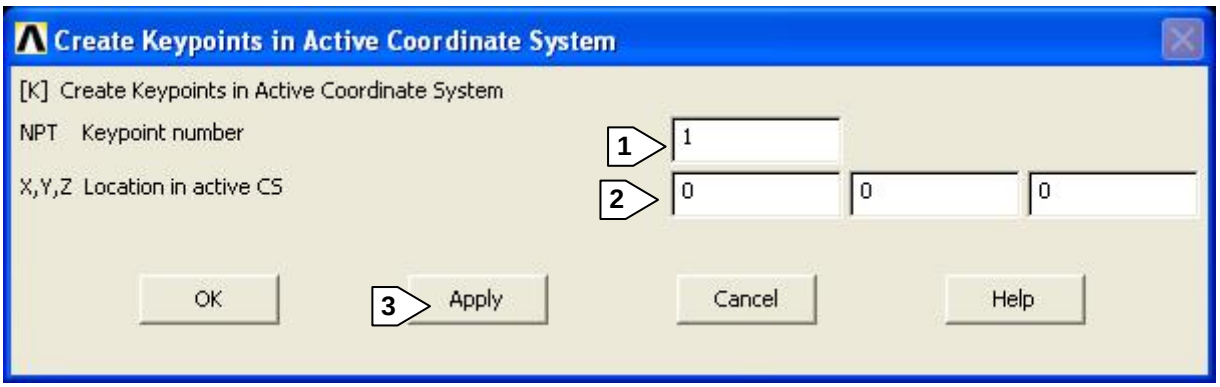
Utility Menu: **Parameters → Scalar Parameters**

1 Enter B=4
2 Pick on accept
3 Enter H=3
4 Pick on accept
5 Enter D=5
6 Pick on accept
7 Pick close



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

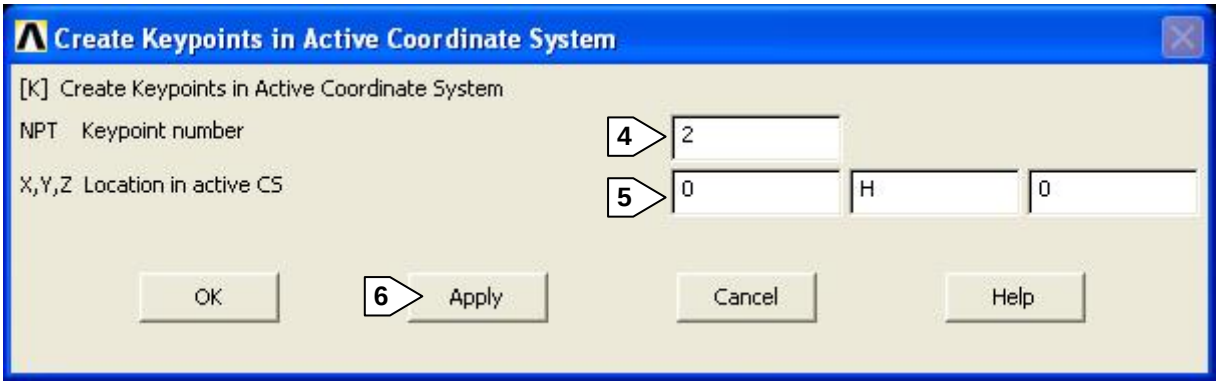
1 Enter 1 as keypoints 1
2 Introduce the coordinates of keypoints 1
3 Pick on apply



4 Enter 2 as keypoints 2

5 Introduce the coordinates of keypoints2

6 Pick on apply



7 Enter 3 as keypoints 3

8 Introduce the coordinates of keypoints 3 (X = 0; Y = H; Z = -B).

9 Pick on apply

10 Enter 4 as keypoints 4

11 Introduce the coordinates of keypoints 4 (X = D; Y = H; Z = -B).

12 Pick on apply

13 Enter 5 as keypoints 5

14 Introduce the coordinates of keypoints 5 (X = D; Y = H; Z = 0).

15 Pick on apply

16 Enter 6 as keypoints 6

17 Introduce the coordinates of keypoints 6 (X = 0; Y = 0; Z = -B).

18 Pick on apply

19 Enter 7 as keypoints 7

20 Introduce the coordinates of keypoints 7 (X = D; Y = 0; Z = -B).

21 OK

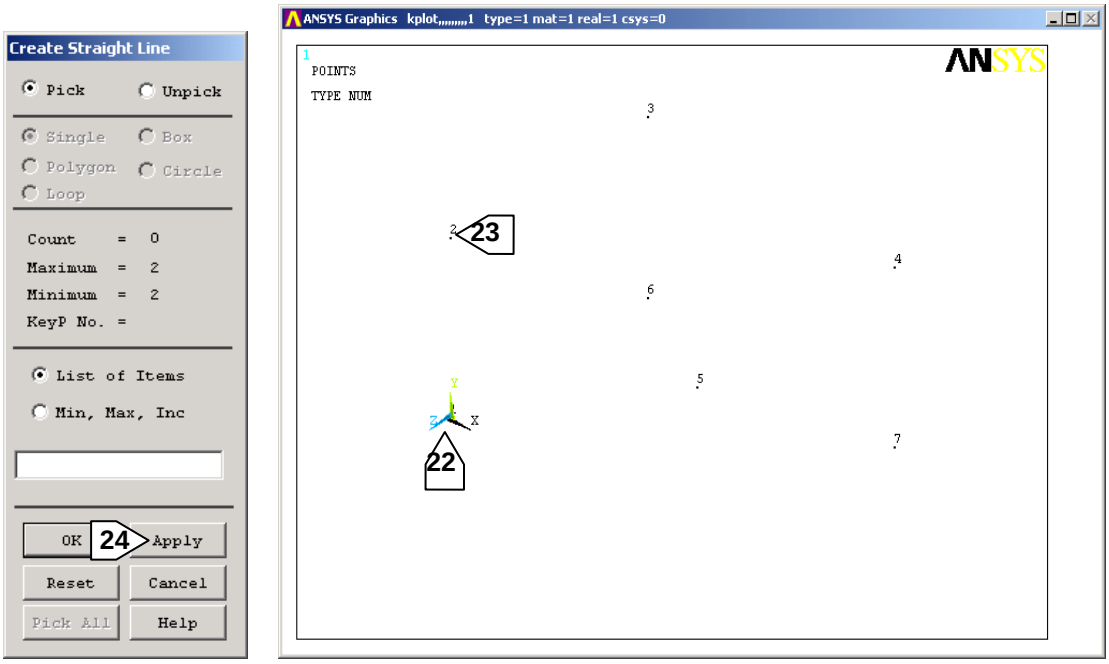
Now we are going to introduce the lines between keypoints.

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

22 Select keypoint 1

23 Select keypoint 2

24 Apply



25 Create line between keypoint 2 and 3

26 Create line between keypoint 3 and 4

27 Create line between keypoint 2 and 5

28 Create line between keypoint 5 and 4

29 Create line between keypoint 6 and 3

30 Create line between keypoint 7 and 4

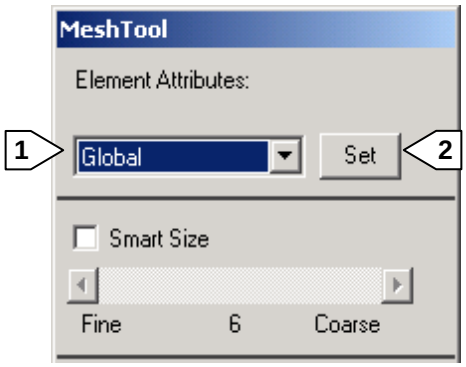
10. Mesh

We first specify the meshing control.

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

1 Choose Global

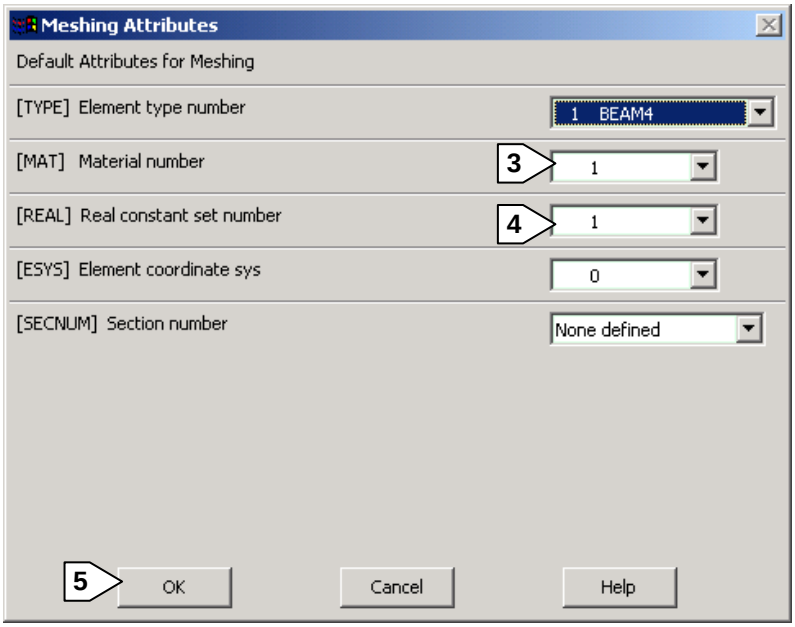
2 Set



3 Choose Material 1

4 Choose Real 1

5 OK

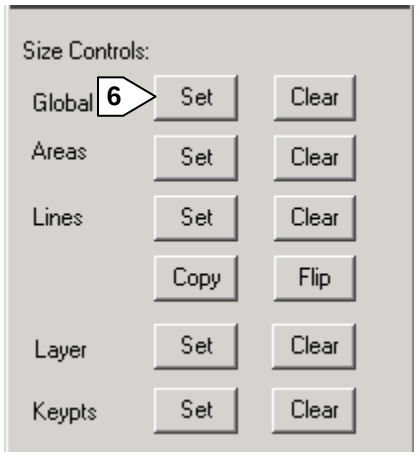


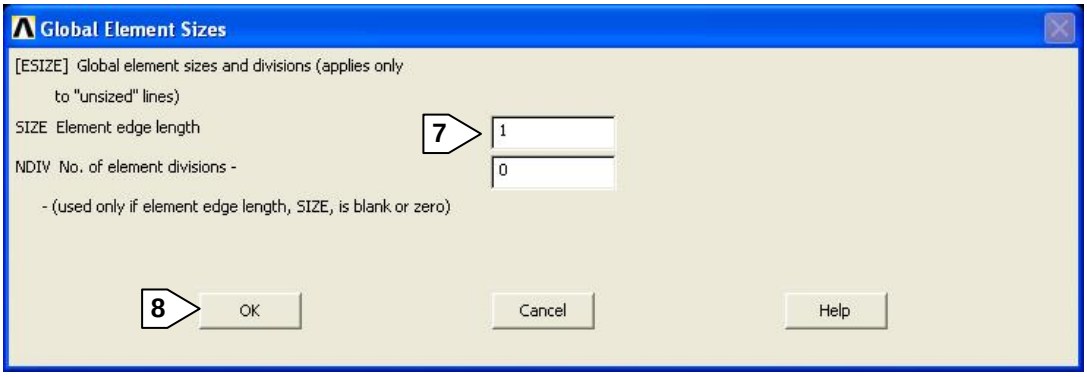
Main Menu: **Preprocessor → MeshTool**

6 Choose Global Set

7 Enter Size 1

8 OK





Utility Menu: **Select → Entities**

9 Choose Lines

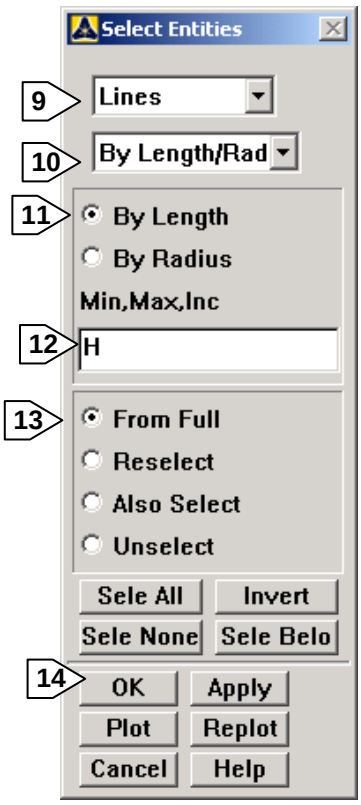
10 Choose By Length/Rad

11 Choose By Length

12 Enter H

13 Choose From Full

14 OK

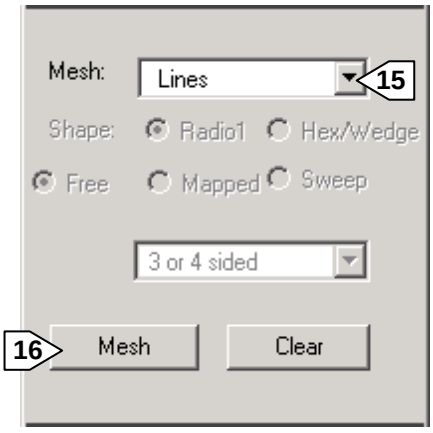


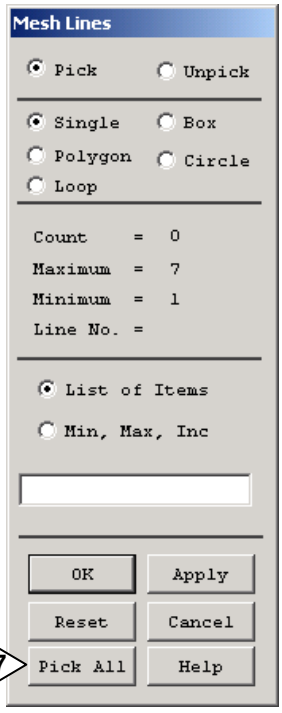
Main Menu: **Preprocessor → MeshTool**

15 Choose Lines

16 Pick on Mesh

17 Pick on Pick all



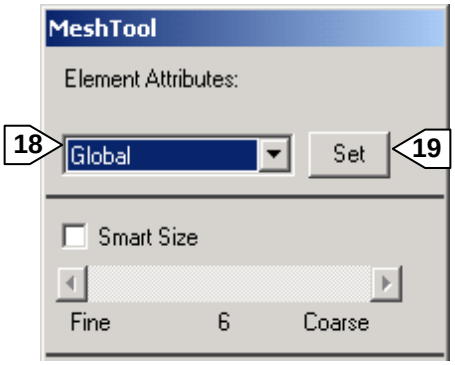


Utility Menu: **Select** → Everything

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

18 Choose Global

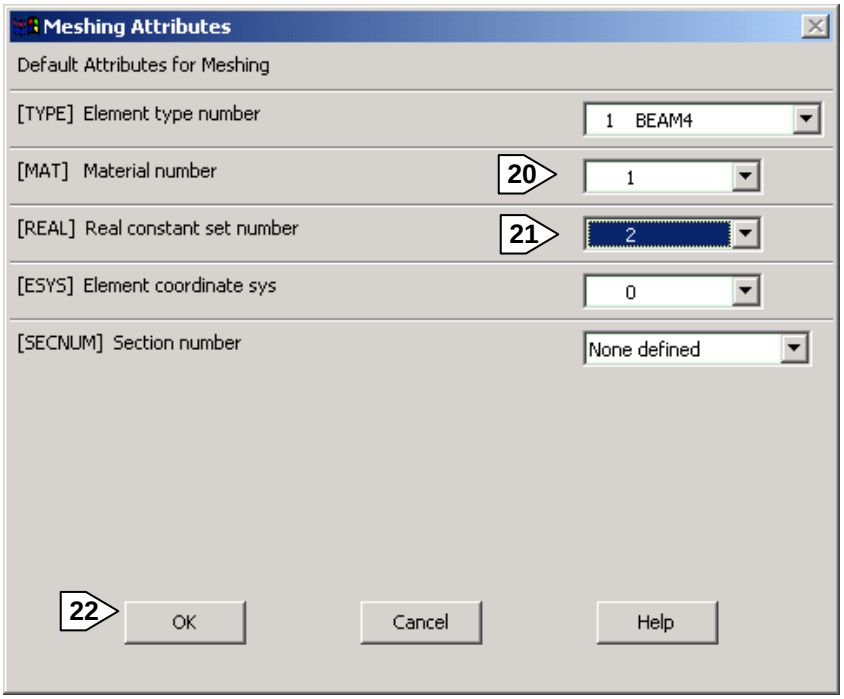
19 Set



20 Choose Material 1

21 Choose Real 2

22 OK

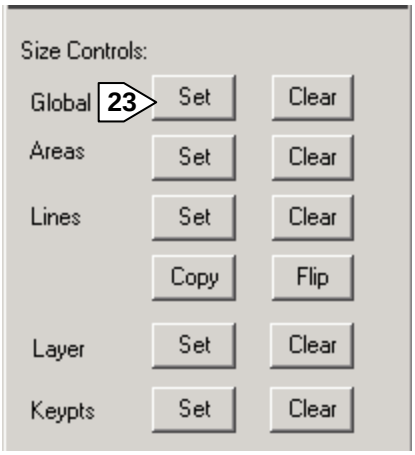


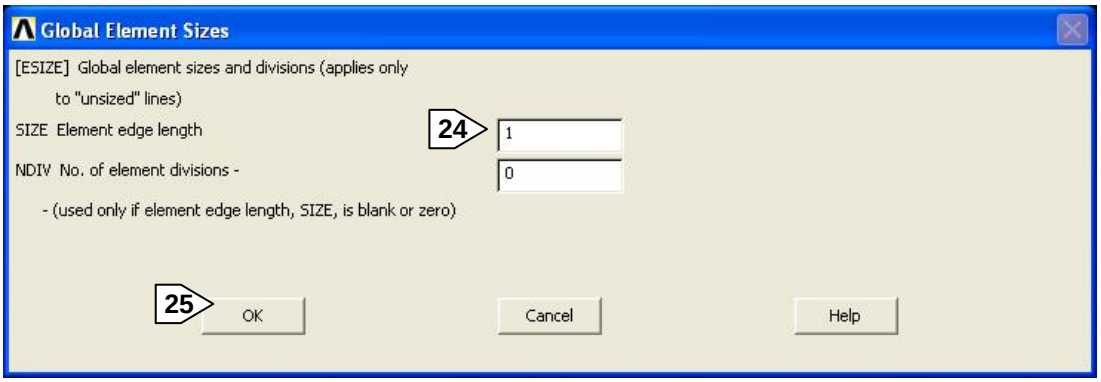
Main Menu: **Preprocessor** → **MeshTool**

23 Choose Global Set

24 Enter Size 1

25 OK





Utility Menu: **Select → Entities**

26 Choose Lines

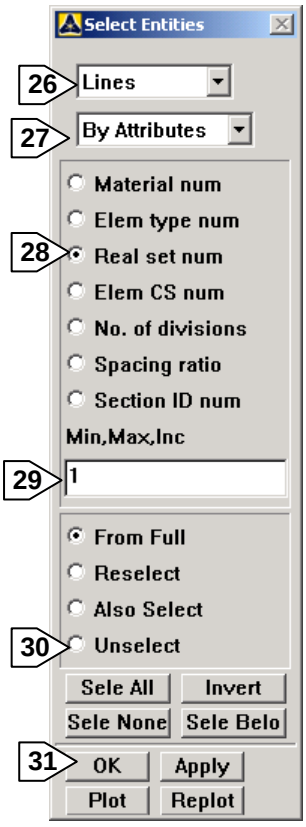
27 Choose By Attributes

28 Choose Real set num

29 Enter 1

30 Choose Unselect

31 OK

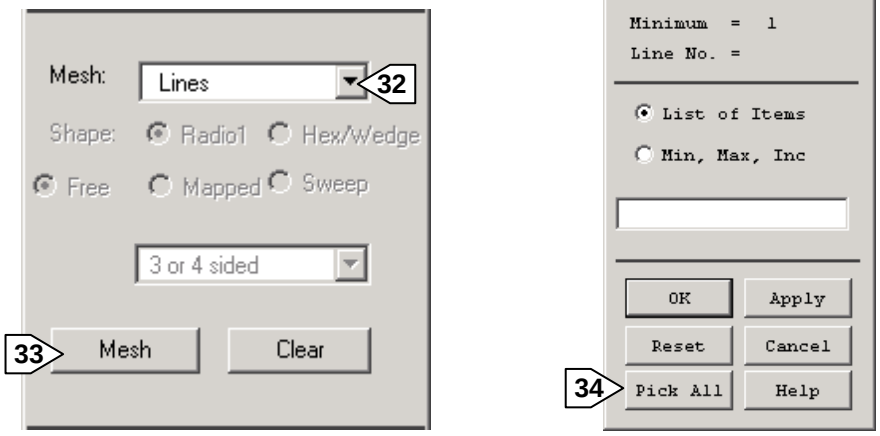


Main Menu: **Preprocessor** → **MeshTool**

32 Choose Lines

33 Pick on Mesh

34 Pick on Pick all



Utility Menu: **Select** → Everything

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

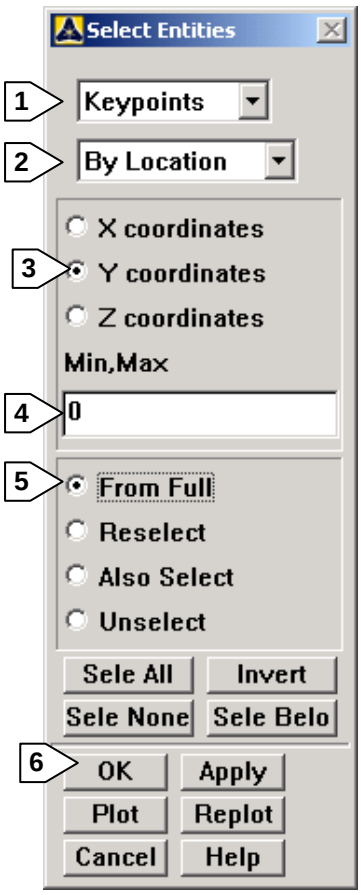
Toolbar: **CFSAVE**

■ SOLUTION

12. Apply displacement constraints

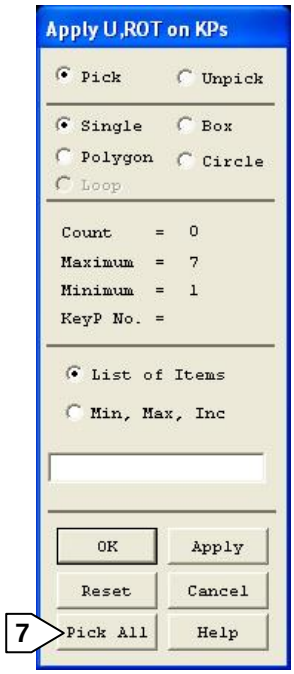
Utility Menu: **Select → Entities**

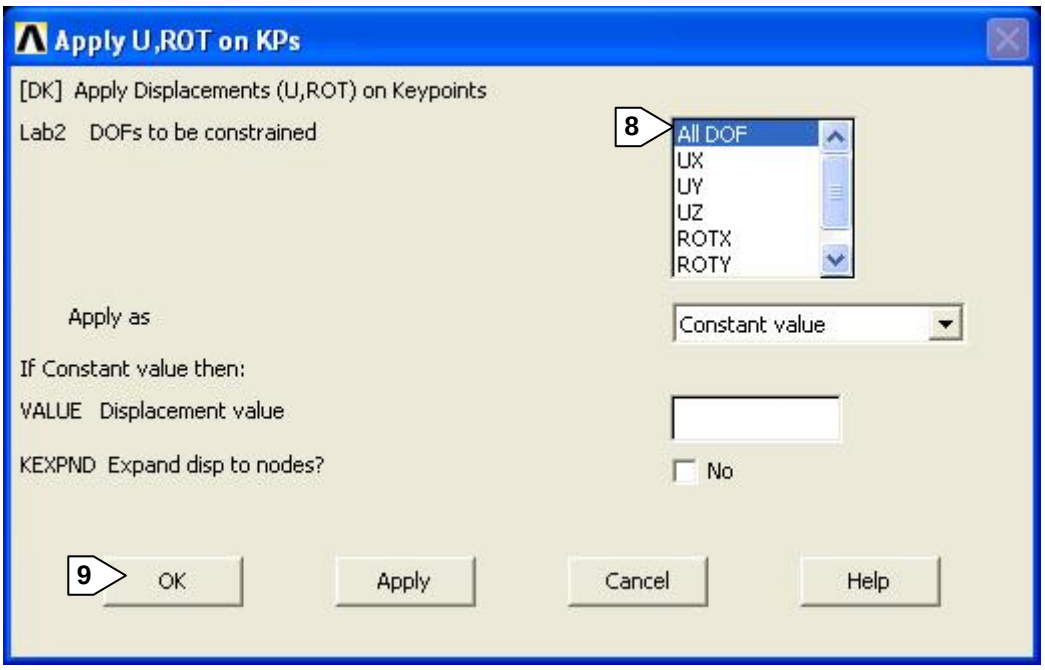
1. Choose Keypoints
2. Choose By Location
3. Choose Y coordinates
4. Enter 0
5. Choose From Full
6. OK



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

7. Pick on Pick all
8. Choose All DOF
9. OK





Utility Menu: **Select** → Everything

13. Apply load

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

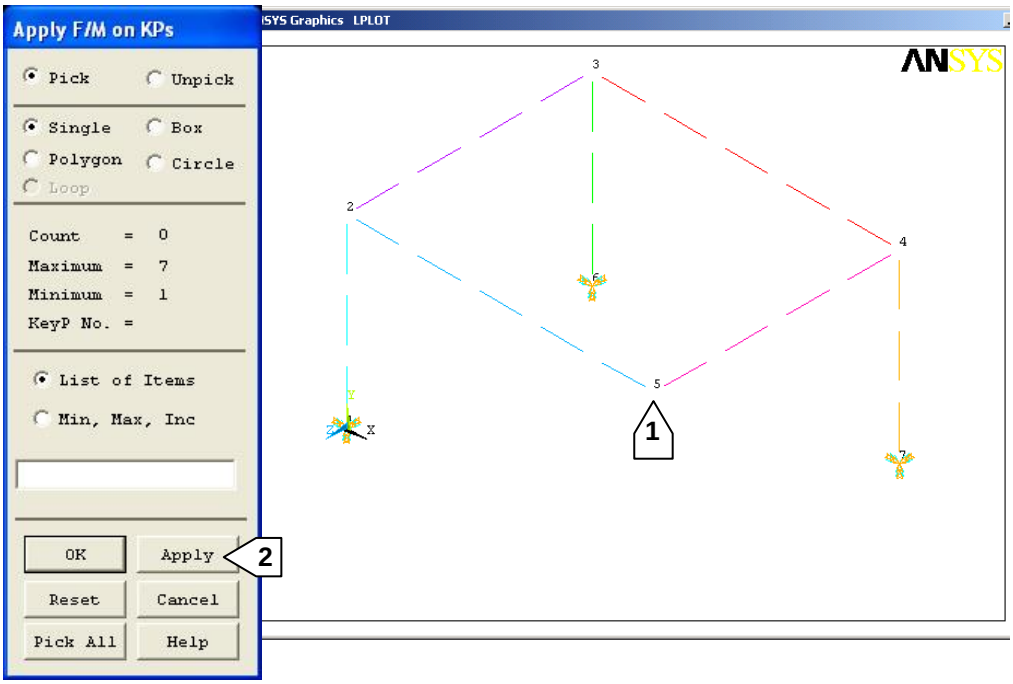
1 Pick on the keypoint 5

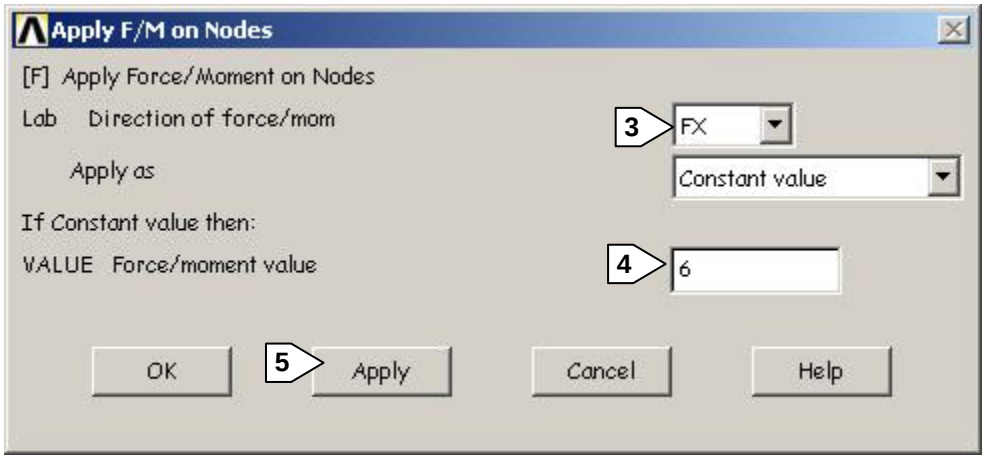
2 Apply

3 Choose FX

4 Enter 6 like force

5 Apply





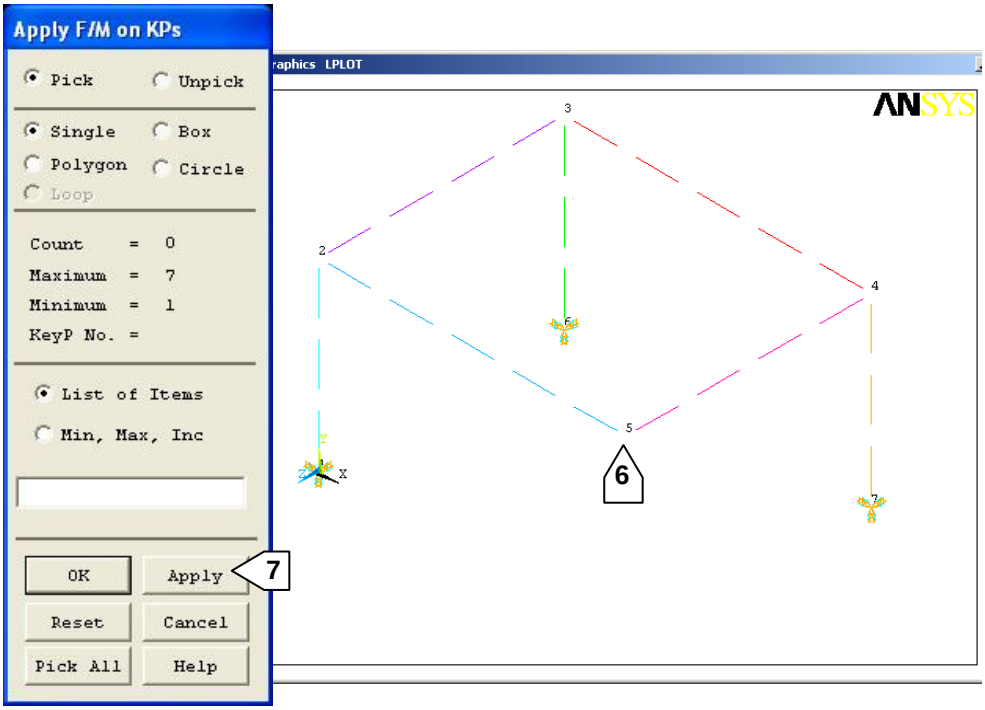
6 Pick on the keypoint 5

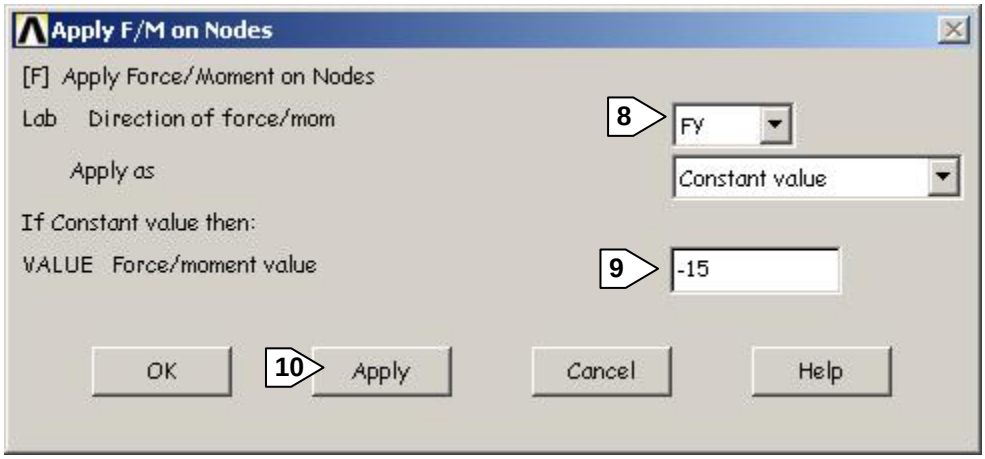
7 Apply

8 Choose FY

9 Enter -15 like force

10 Apply





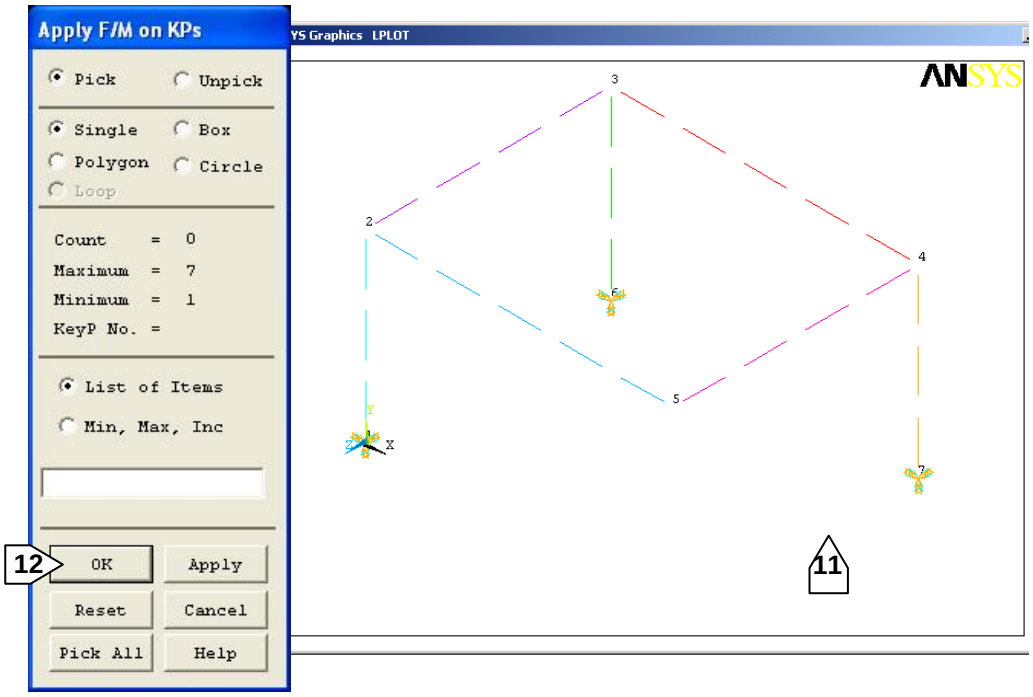
11 Pick on the keypoint 5

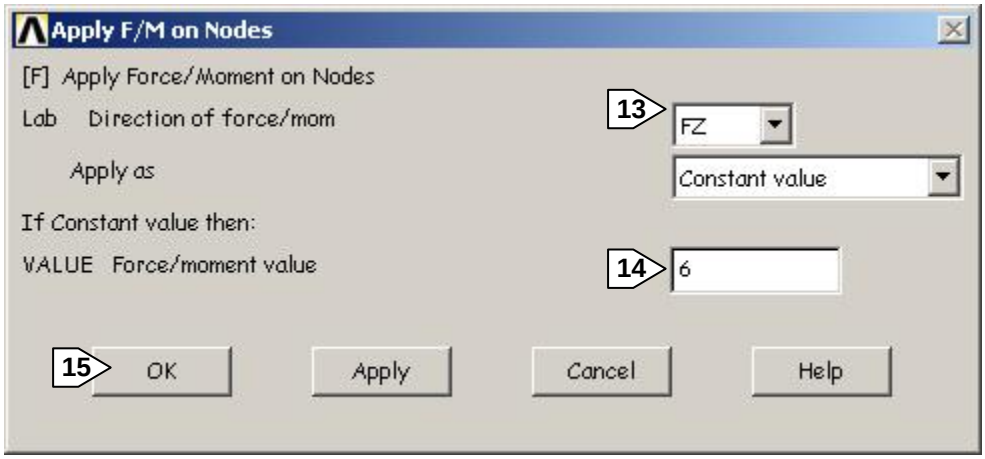
12 OK

13 Choose FZ

14 Enter 6 like force

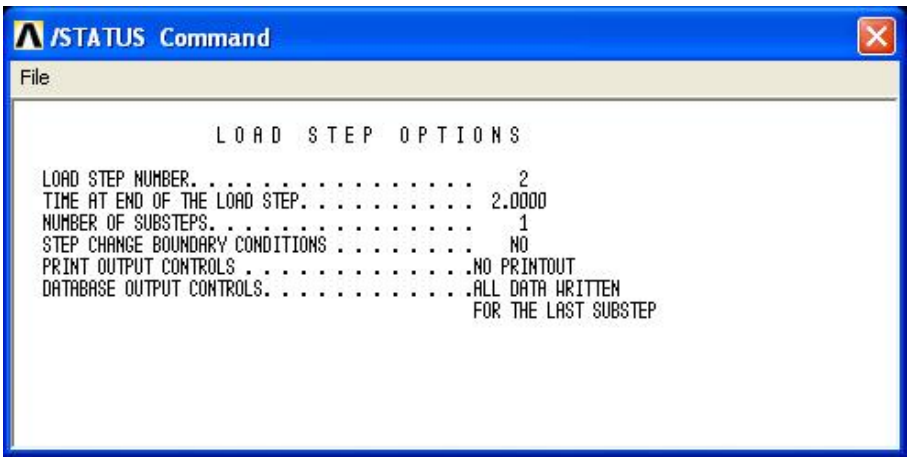
15 OK





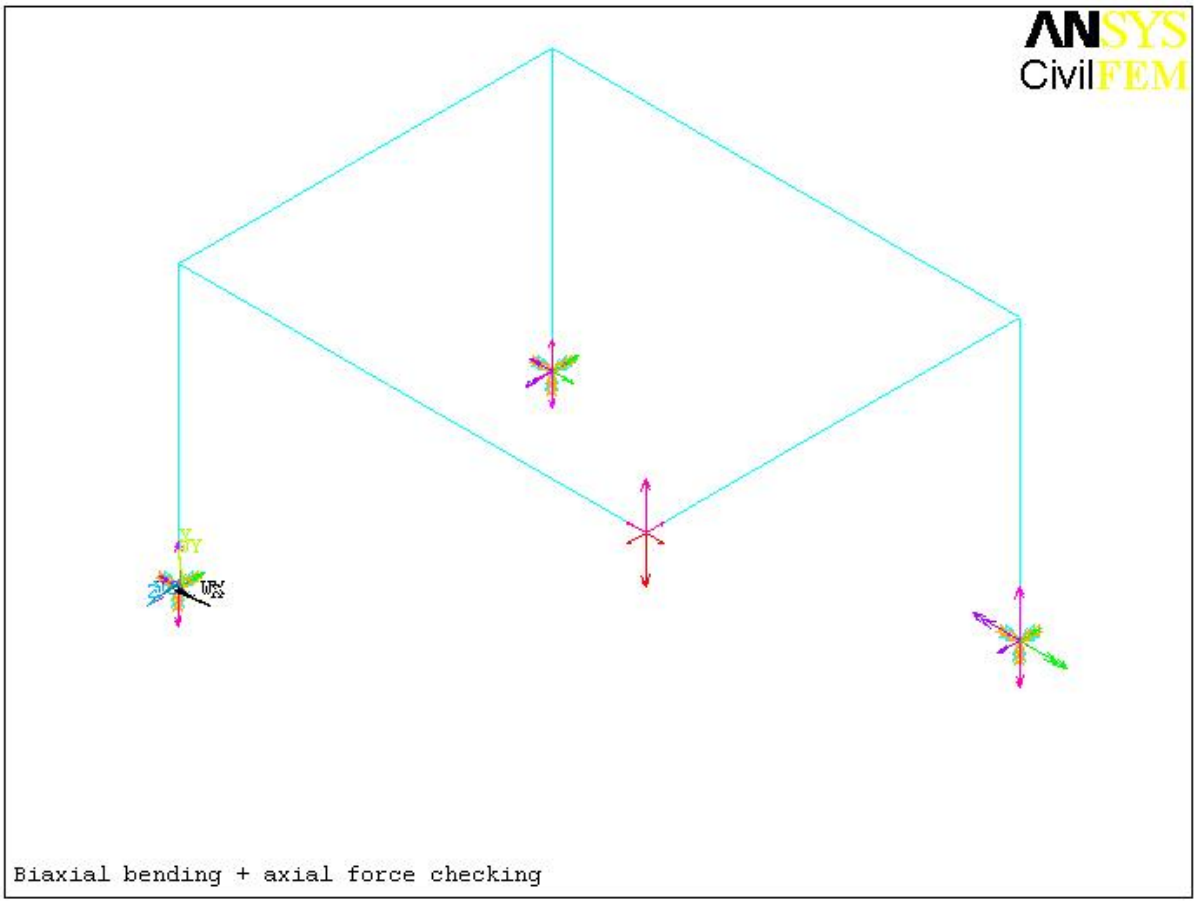
14. Solve

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




Solve Current Load Step
[SOLVE] Begin Solution of Current Load Step
Review the summary information in the lister window (entitled "/STATUS Command"), then press OK to start the solution.
OK
Cancel
Help


Note
Solution is done!
Close


ANSYS
CivilFEM
Biaxial bending + axial force checking

Postprocessing

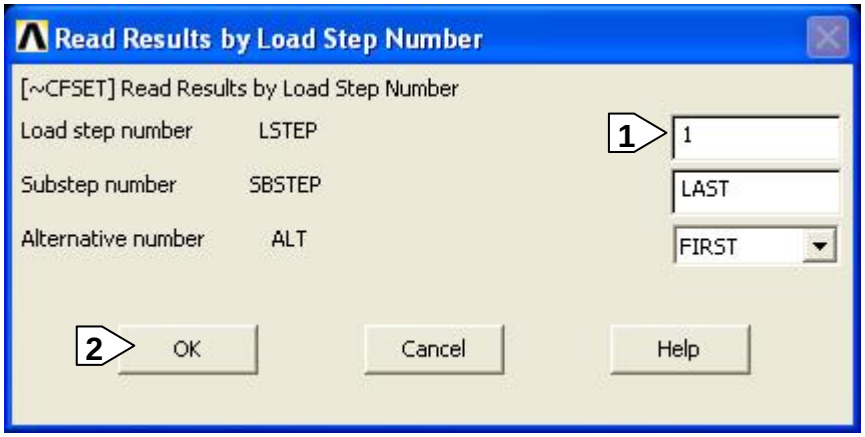
15. Enter the postprocessor and read in results

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

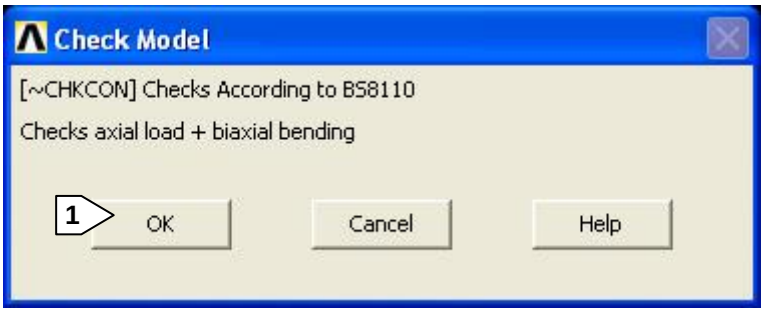


16. Checking under 3D biaxial bending moment and axial force

Now we are going to check the frame under bending moment and axial force according to British Standard provisions.

Main Menu: -CivilFEM – **Civil Postprocessor → Code Checking→ BS 8110→** CHECK BY CODE: **Beams & Solid →3D Axial+Bend**

1 OK to check axial load + biaxial bending according to BS8110



17. Review elements OK and No OK

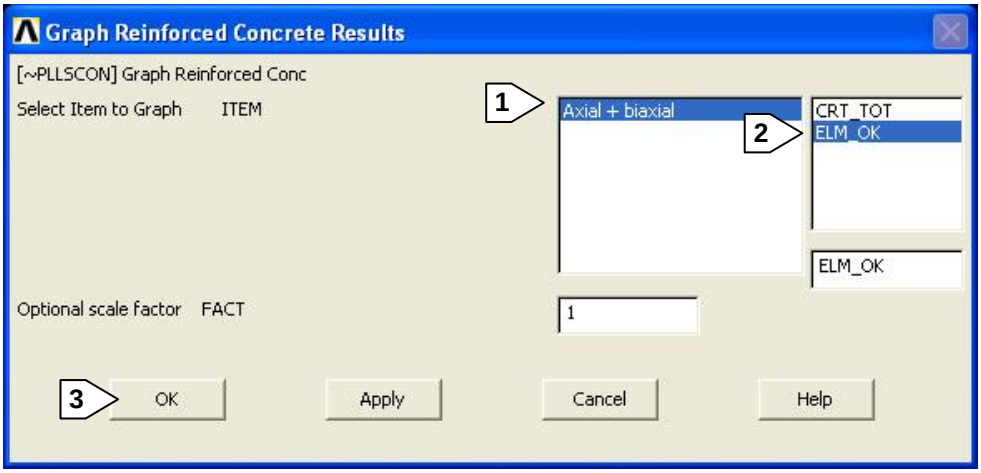
In order to review the checked results according to British Standard, we are going to plot the elements OK and No OK in accordance with the code criteria for checking under bending moment and axial force. In this graph, elements that satisfy British Standard specifications for the requested check are plotted in green, while elements that do not satisfy the code provisions are plotted in red. Elements plotted in grey are elements that have not been checked.

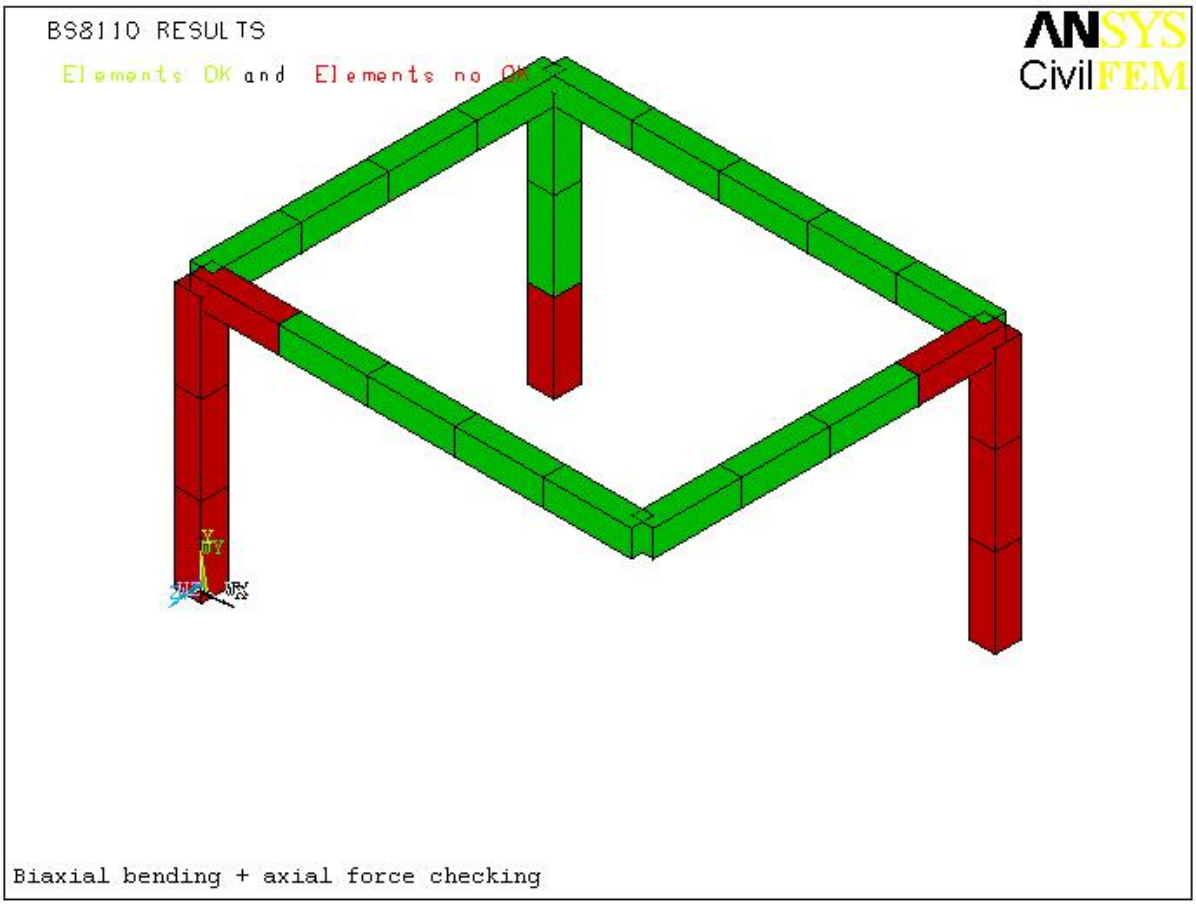
Main Menu: - CivilFEM – **Civil Postprocessor → Code Checking→ BS 8110 →** BEAM RESULTS: **Plot Results…**

1 Choose Axial+Biaxial

2 Elements OK/NoOK

3 OK





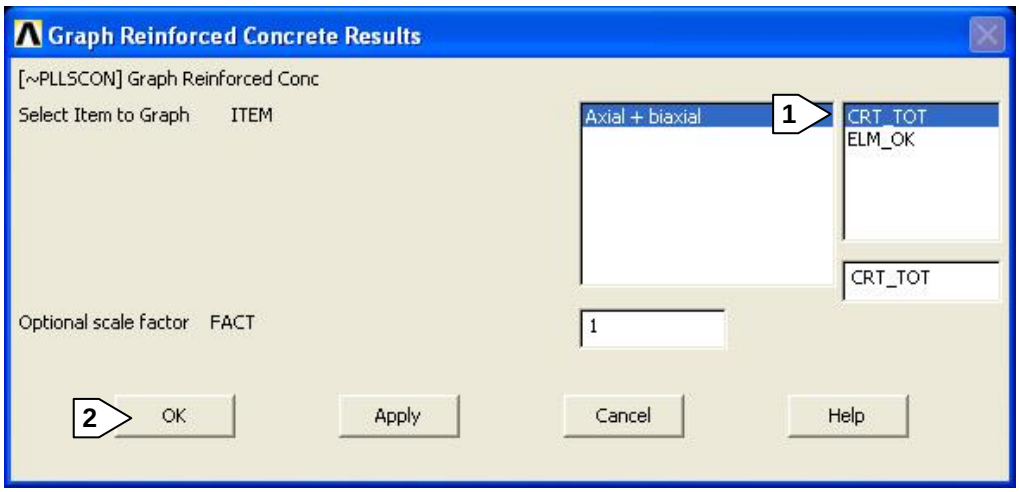
18. Review axial + biaxial bending total criterion

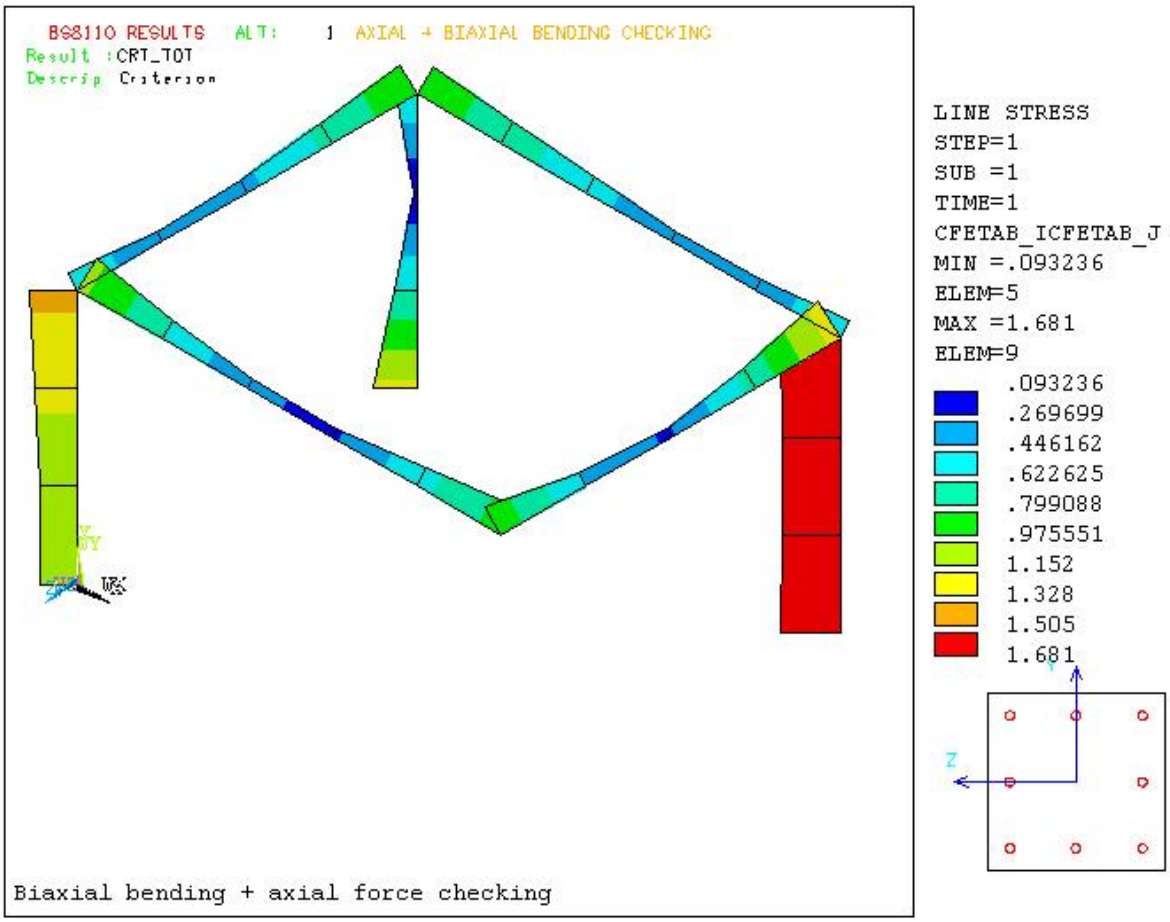
We are going to plot the axial + biaxial bending total criterion

Main Menu: - CivilFEM – **Civil Postprocessor → Code Checking→ BS8110 →** BEAM RESULTS: **Plot Results**

1 Choose CRT_TOT

2 OK



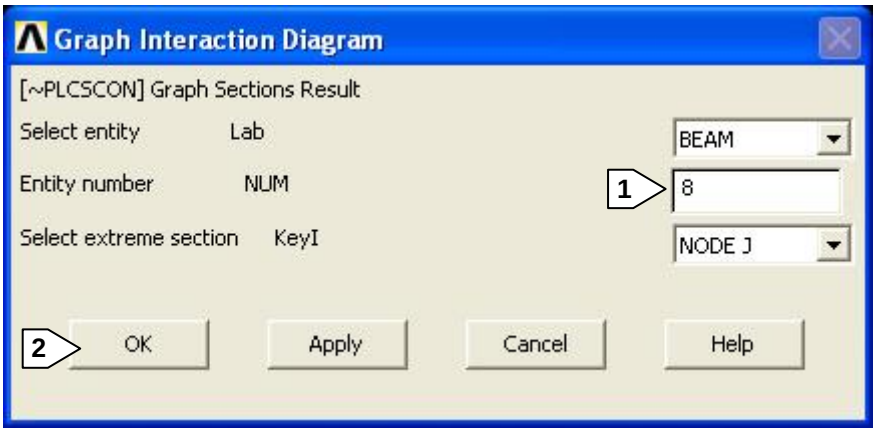


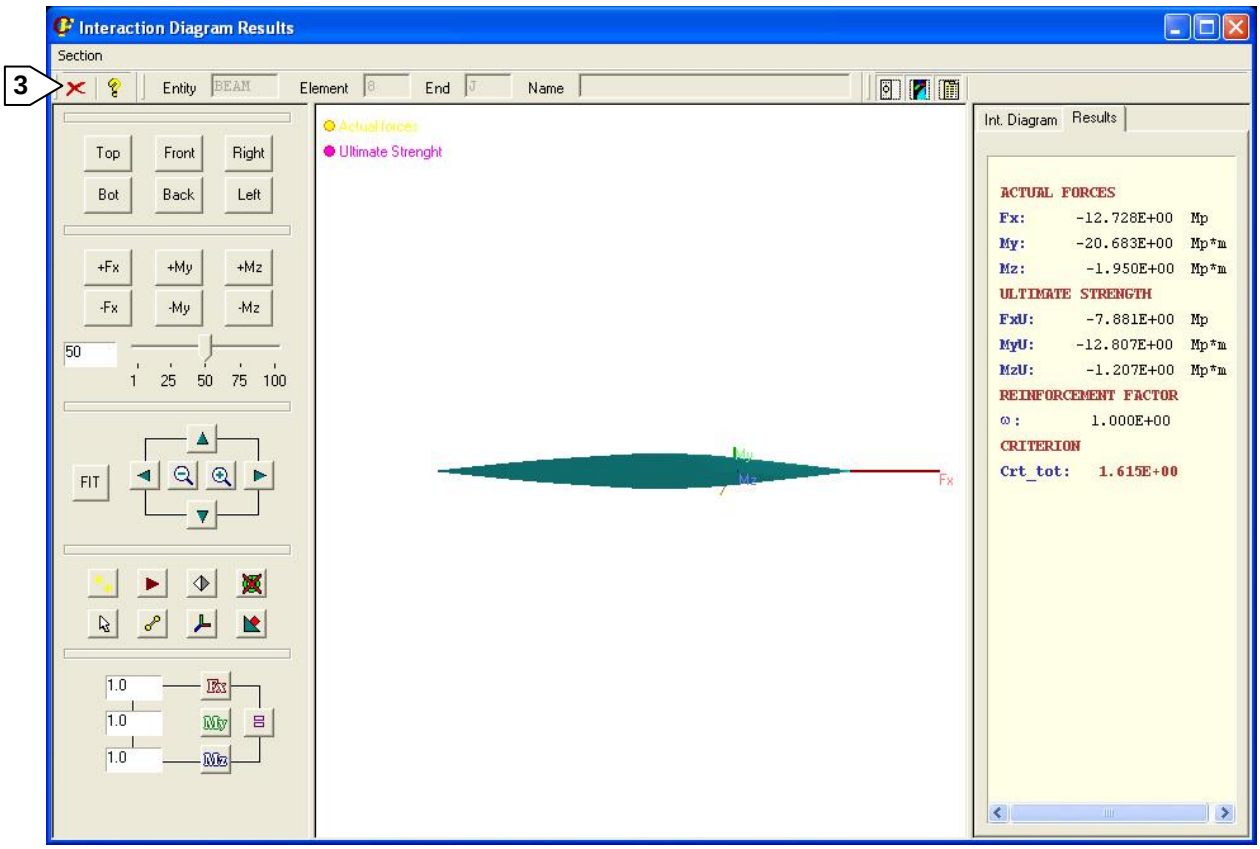
19. Plot an interaction diagram

Main Menu: - CivilFEM – **Civil Postprocess → Code Checking→ BS 8110 →** BEAM RESULTS: **Plot Sec Result…**

1 Select element 8 node J

2 OK





3 To close this window pick on the exit button

20. Exit the ANSYS program

ANSYS Toolbar: **QUIT**

1 Pick on Save Everything option

2 OK