

4. Frame Reinforcement by Eurocode 2

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: BEAM3

Active Code Eurocode 2

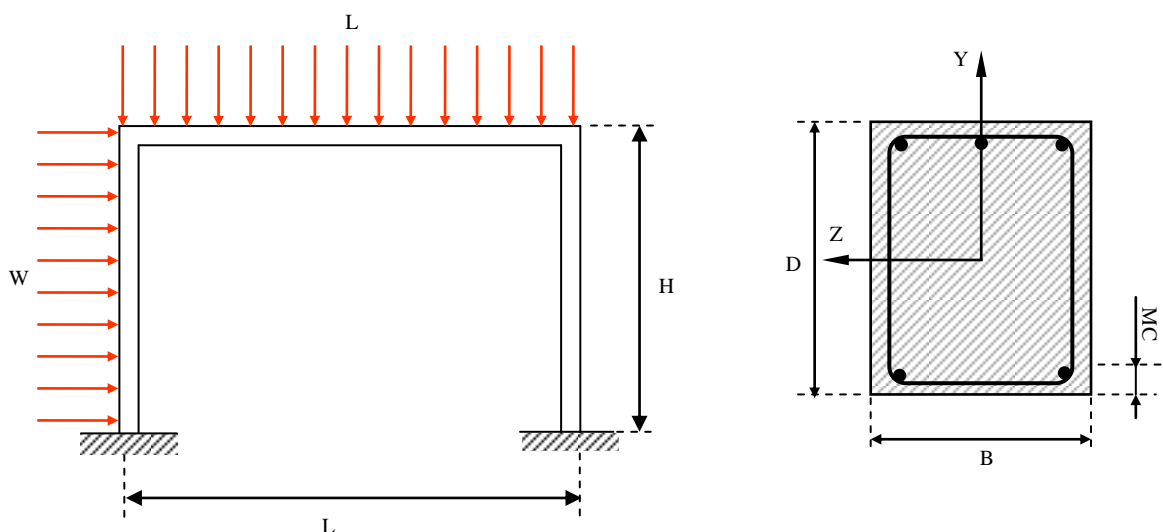
Units System N, m, s.

CivilFEM Features Demonstrated: Units selection, code selection, material definition, section definition by dimensions, axial and bending reinforcement checking, shear reinforcement checking, axial and bending reinforcement design, shear reinforcement design.

Problem Description

This example illustrates the Eurocode 2 designing criteria for a typical concrete constrained frame. We will use the program skills to check the frame reinforcement and to design the appropriate one if necessary. We will check for axial force, bending moment Z and shear in the Y direction.

We are not considering the self-weight of the reinforced frame. Nevertheless we will apply a distributed load along the left column.



Loads:	
W = 40 kN/m	
L = 60 kN/m	
Frame geometrical dimensions:	
Length	L = 6 m
Height	H = 5 m
Section properties:	
Dimensions:	
Width	B = 0.4 m
Depth	D = 0.5 m
Material:	
Concrete	C40/50
Steel	S400 $\gamma_c=1.5$ $\gamma_s=1.15$
Reinforcement:	
Bending reinforcement: Group 1	3 ϕ 16 mm at face number 2 (Scalable)
Bending reinforcement: Group 2	3 ϕ 16 mm at face number 4 (Scalable)
Shear reinforcement:	ϕ 10 mm / 30 cm

■ Given

The load distribution, the section geometrical dimensions, material properties and safety factors for concrete and steel reinforcement are shown in the previous table.

■ Approach and Assumptions

We will use 2D 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 units
3. Set code
4. Define material
5. Define element type
6. Define section
7. Define bending reinforcement properties
8. Define shear reinforcement properties
9. Define Beam & Shell properties
10. Define nodes and elements
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 under 2D bending moment and axial force
17. Review elements OK and No OK
18. Plot an interaction diagram
19. Checking under shear force in Y axis
20. Review elements OK and No OK
21. Design for bending moment and axial force
22. Review designed reinforcement for bending moment and axial force
23. Design for shear force in Y axis
24. Review designed reinforcement for shear force
25. Exit the ANSYS program

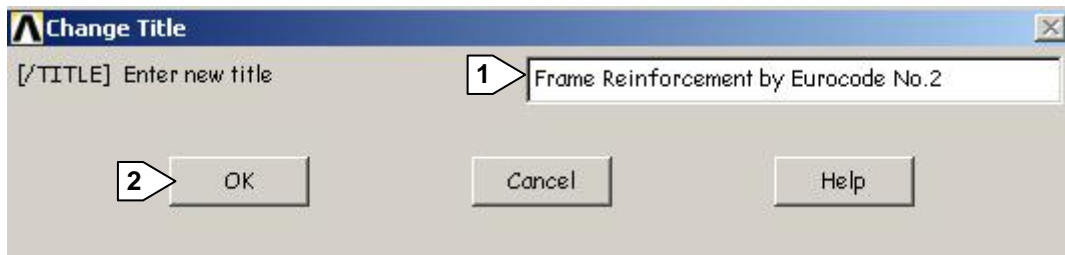
Interactive Step-by-Step Solution

■ Preprocessing

1. Specify title

Utility Menu: **File** → **Change title**

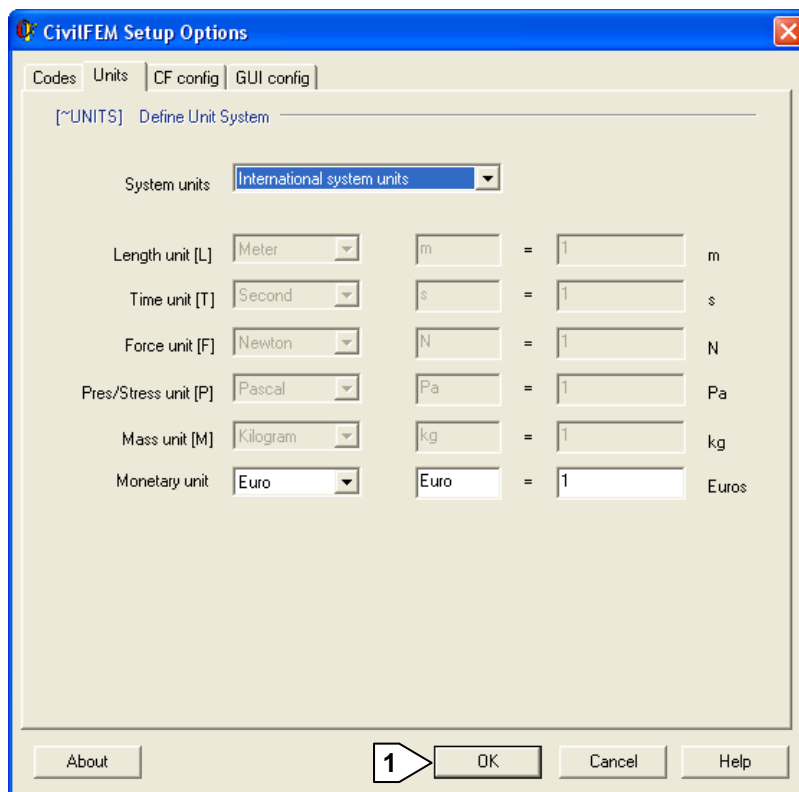
- 1 Enter the title: “Frame Reinforcement by Eurocode 2”
- 2 OK to define the title and close the dialog box.



2. Set units

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

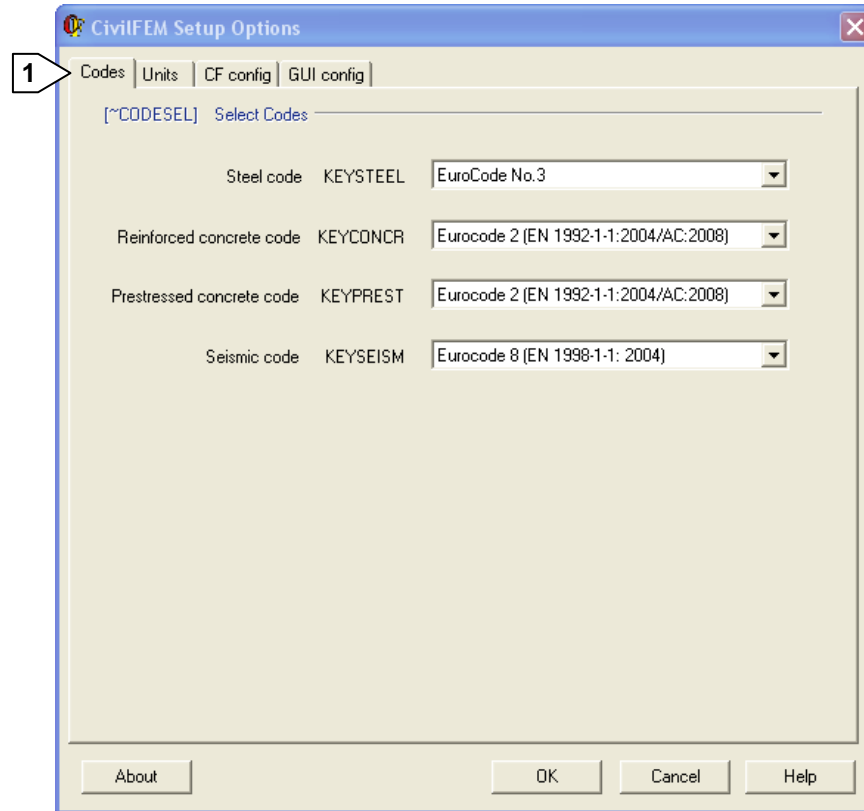
- 1 OK to accept meters, seconds and newtons and close the units dialog box



3. Set code

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

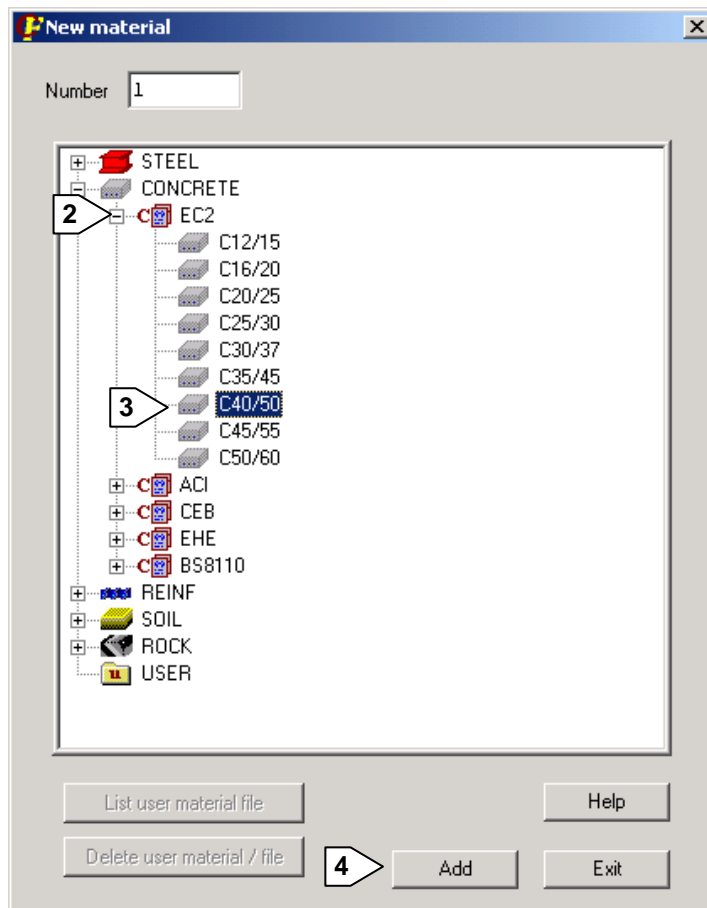
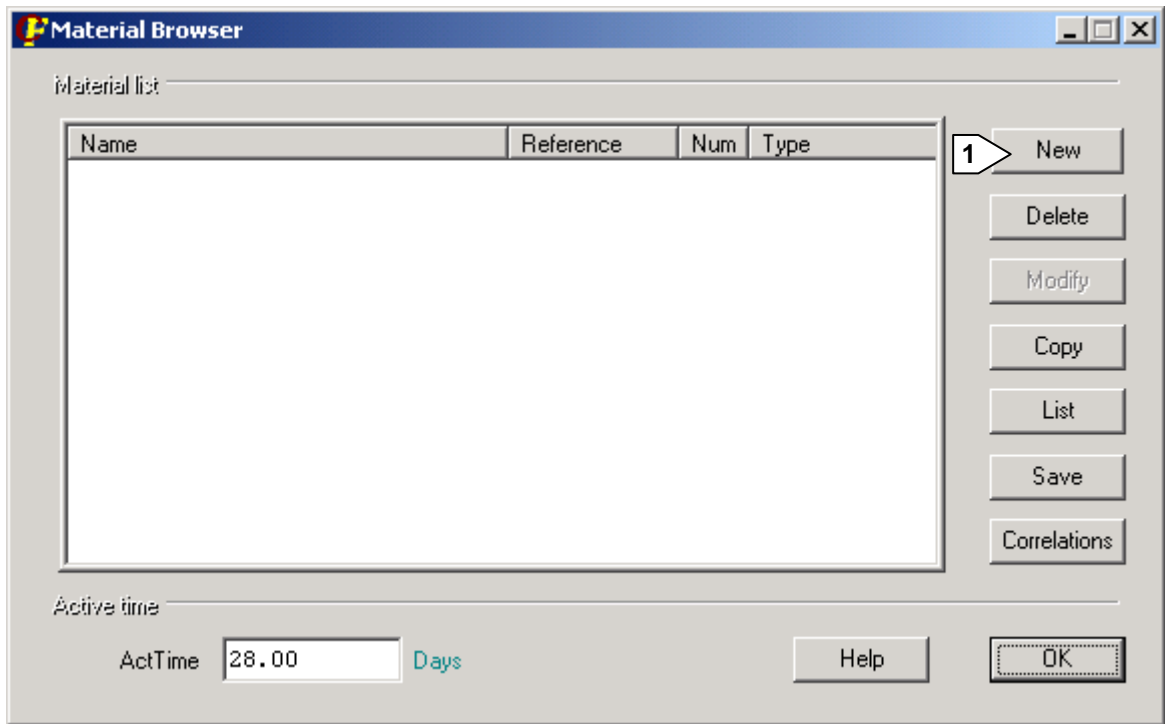
- 1 Pick on the Codes tab
- 2 OK to set active code and close the code dialog box.

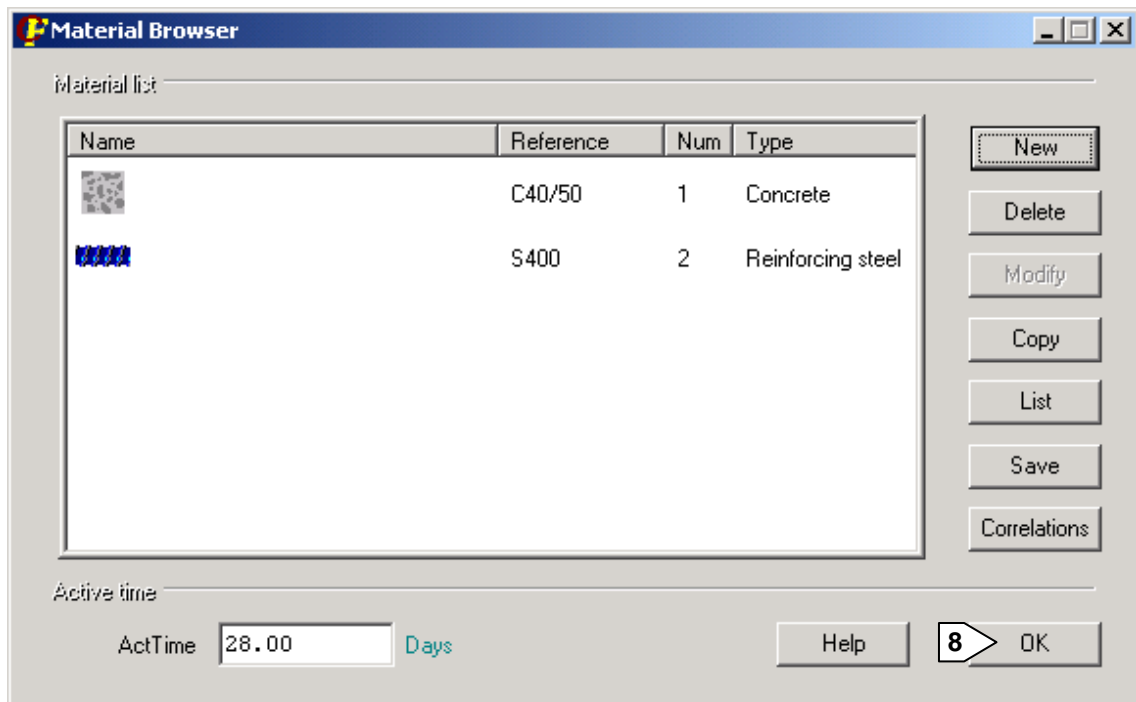
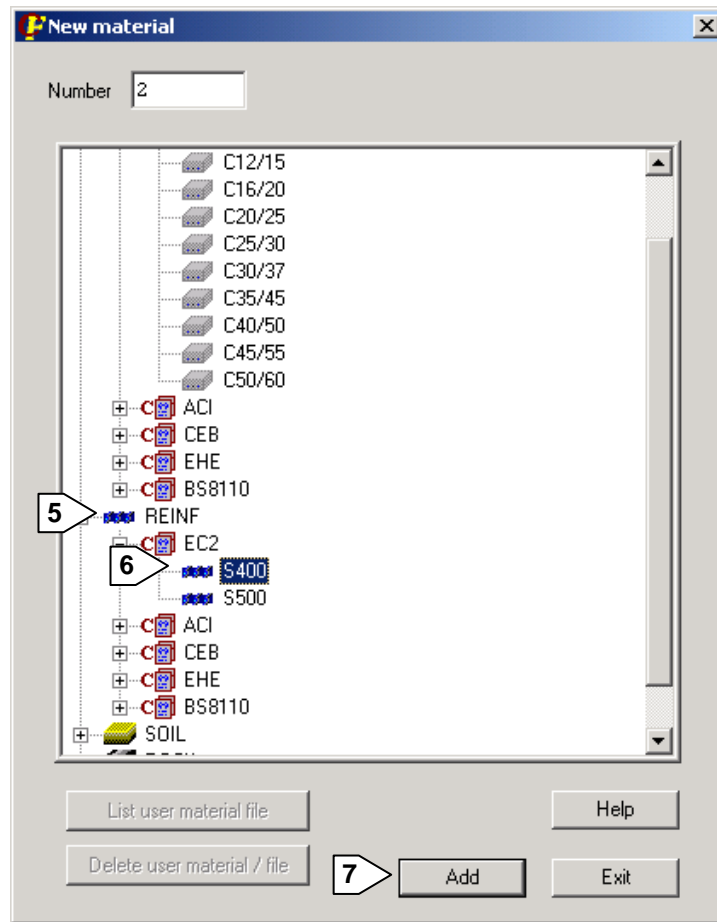


4. Define material

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

- 1 Choose New
- 2 Select concrete material type
- 3 Choose EC2: C40/50 concrete
- 4 Add to define material 1
- 5 Select Reinforcing steel material type
- 6 Choose EC2: S400 reinforcing steel
- 7 Add to define material 2 and Exit.
- 8 Ok to define material properties and close the dialog box





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



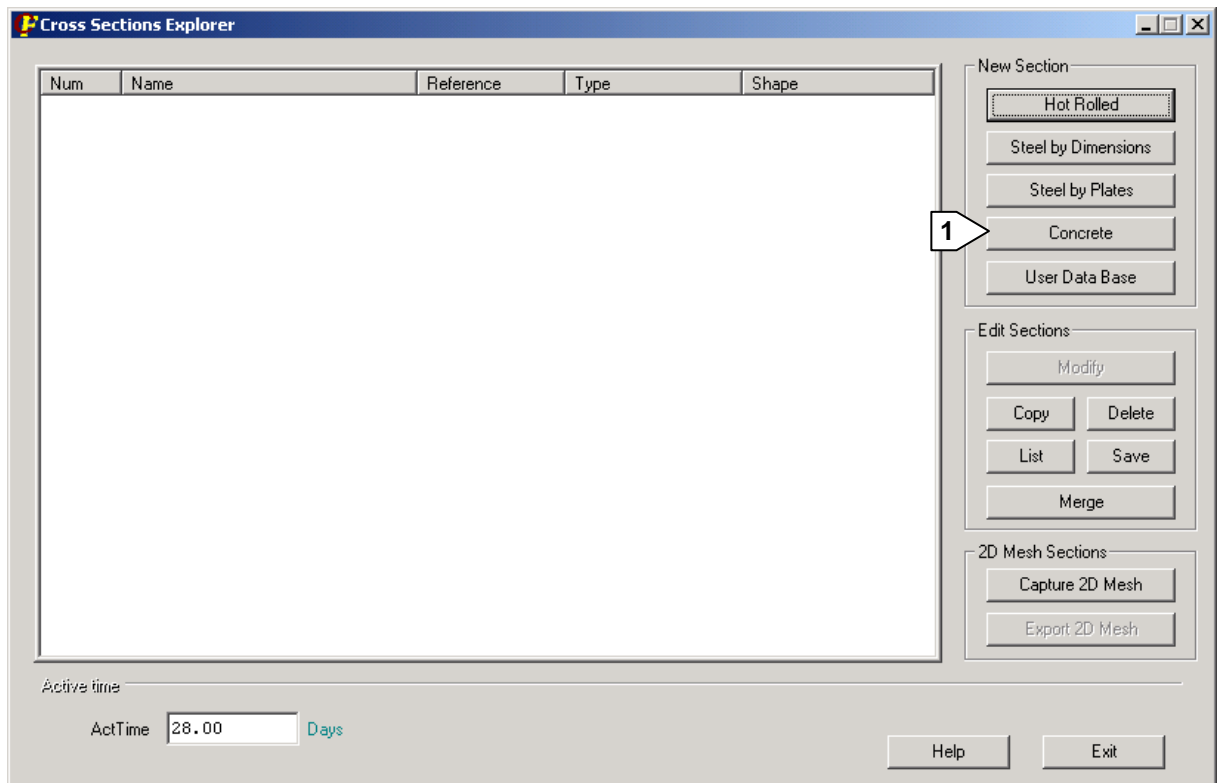
6. Define section

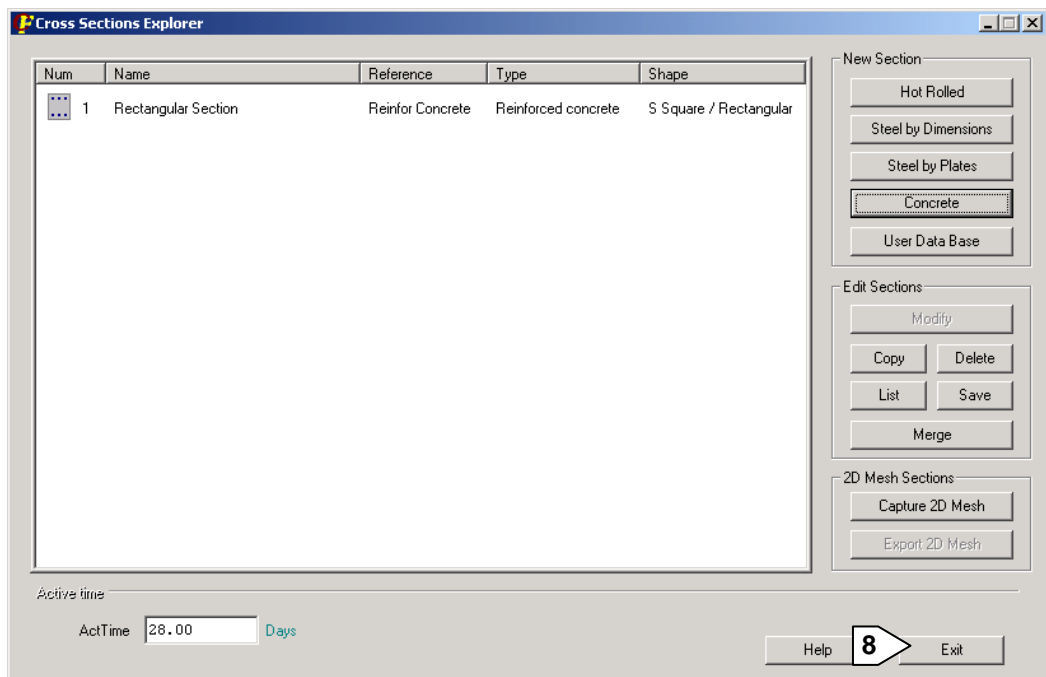
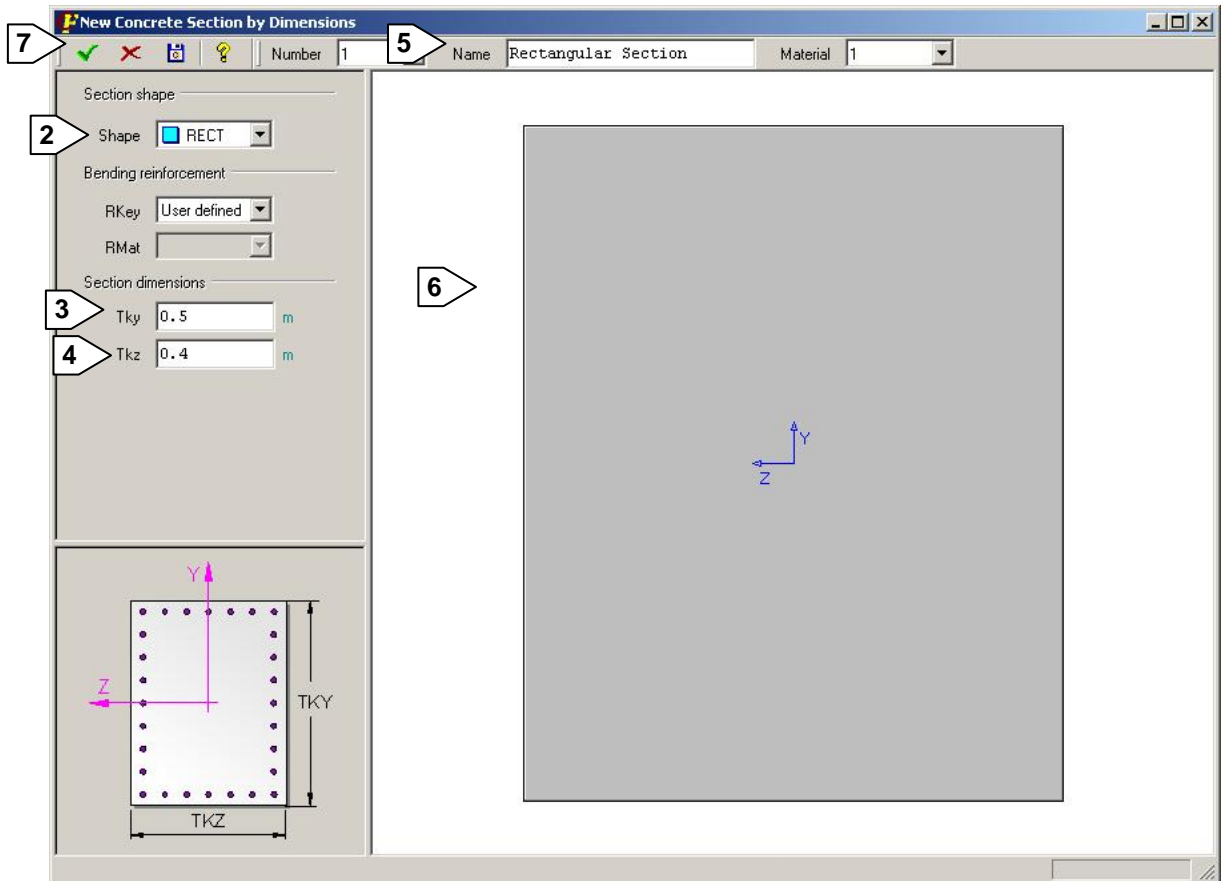
First, we will define the section geometry and then we will introduce the reinforcements.

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

- 1 Pick on the Concrete button to define a concrete section

- 2 Choose rectangular shape
- 3 Enter 0.5 as depth (TKY)
- 4 Enter 0.4 as width (TKZ)
- 5 Enter Rectangular Section as Name
- 6 To draw the rectangular section click with the right button on the window
- 7 Ok to define Concrete Section
- 8 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.

We will define two reinforcement groups as it was indicated in the section's graph.

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

Group	1
Reinforcement material	Material 2 (S 400)
Reinforcement class	Scalable
Diameter	16 mm
Number of bars	3
Face number	2
Geometrical cover	0.032 m

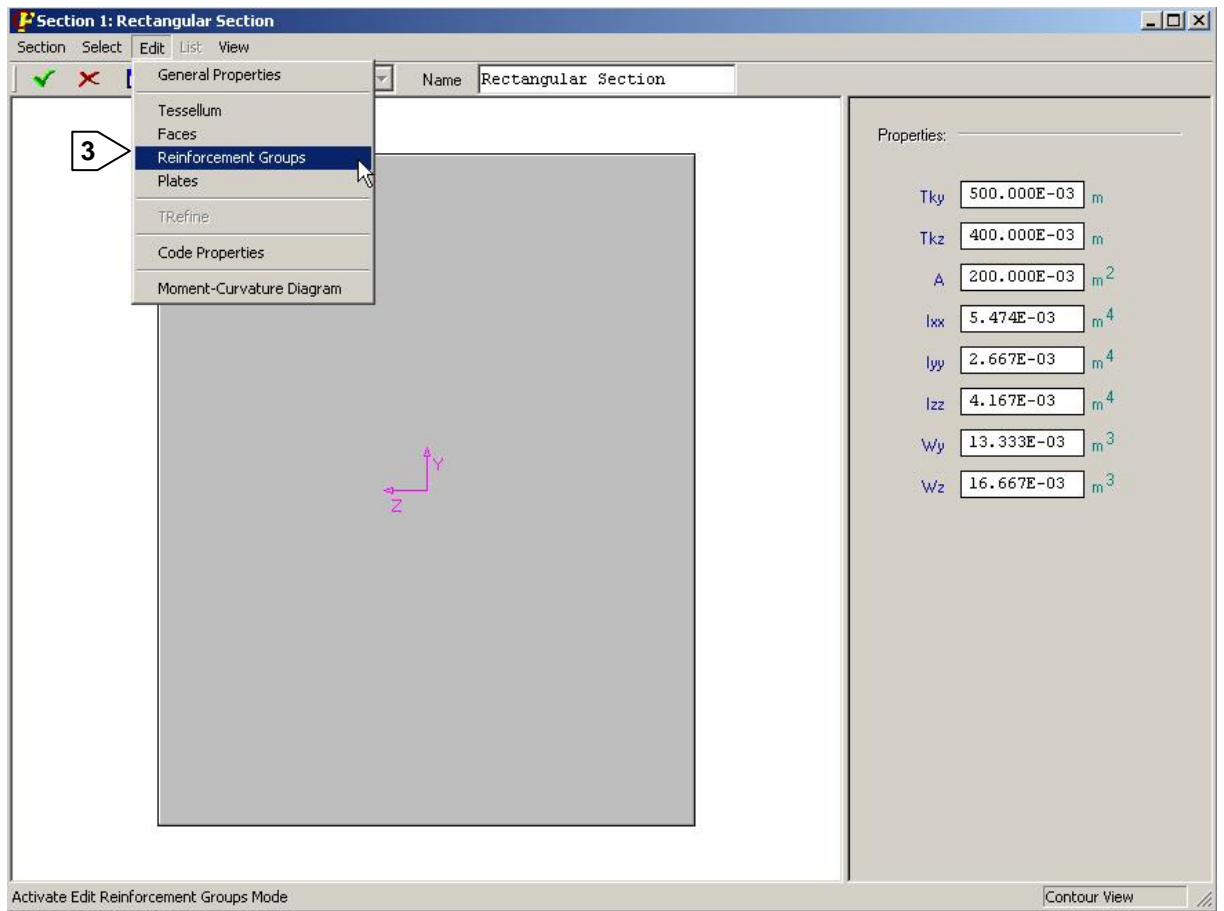
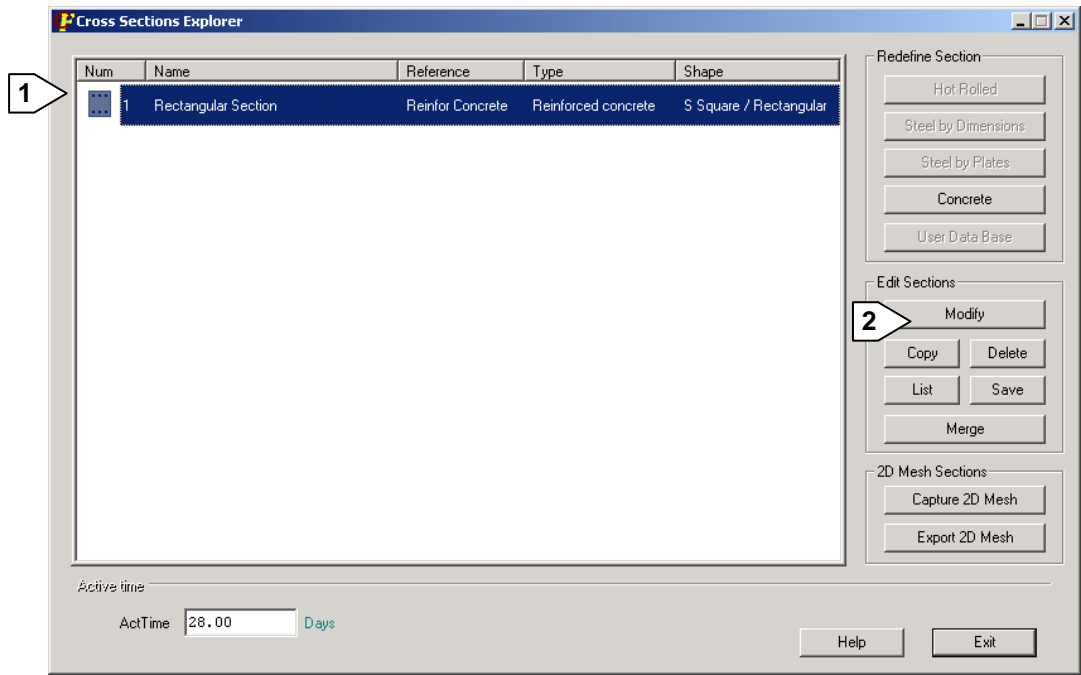
The characteristics of the reinforcement group 2 are shown in the following table:

Group	2
Reinforcement material	Material 2 (S 400)
Reinforcement class	Scalable
Diameter	16 mm
Number of bars	3
Face number	4
Geometrical cover	0.032 m

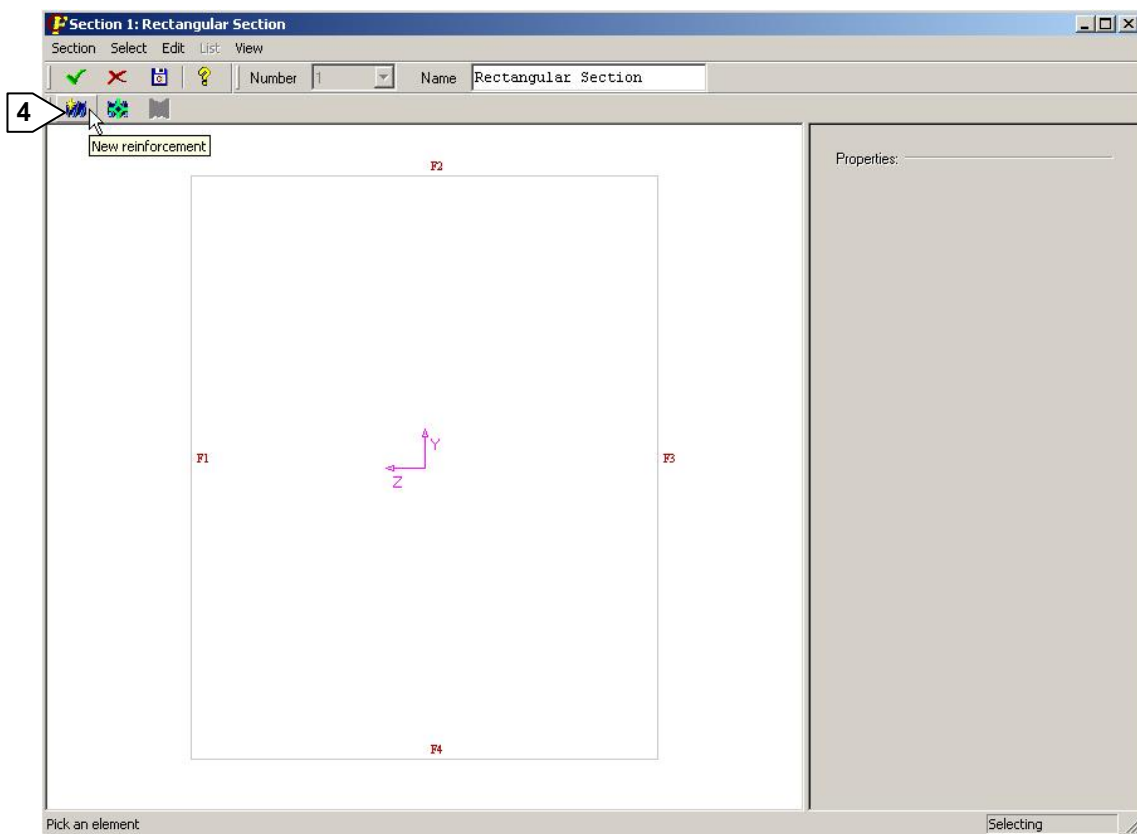
First, we will define the reinforcement group 1:

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 Select material 2 as material for bars
- 6 Select Reinforcement at Face 2
- 7 Enter 0.032 for Geometrical cover
- 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

Reinforcement properties

Bending | Shear | Torsion

Predefined bending reinforcement

RKey: User defined RMat: []

Bending reinforcement

Urf: 1 **5** Mat: 2

7 Gc: 32.000E-03 m Cls: 0: Scaleable

6 Ufc: 2 **10** End: 4: Bars at both ends at MC

Uniformly distributed reinforcement

AsT: 603.186E-06 m² Amount / Area

AsL: 1.508E-03 m²/m By Area

By Area / Length

Barc distribution reinforcement

11 N: 3 Amount / Bars

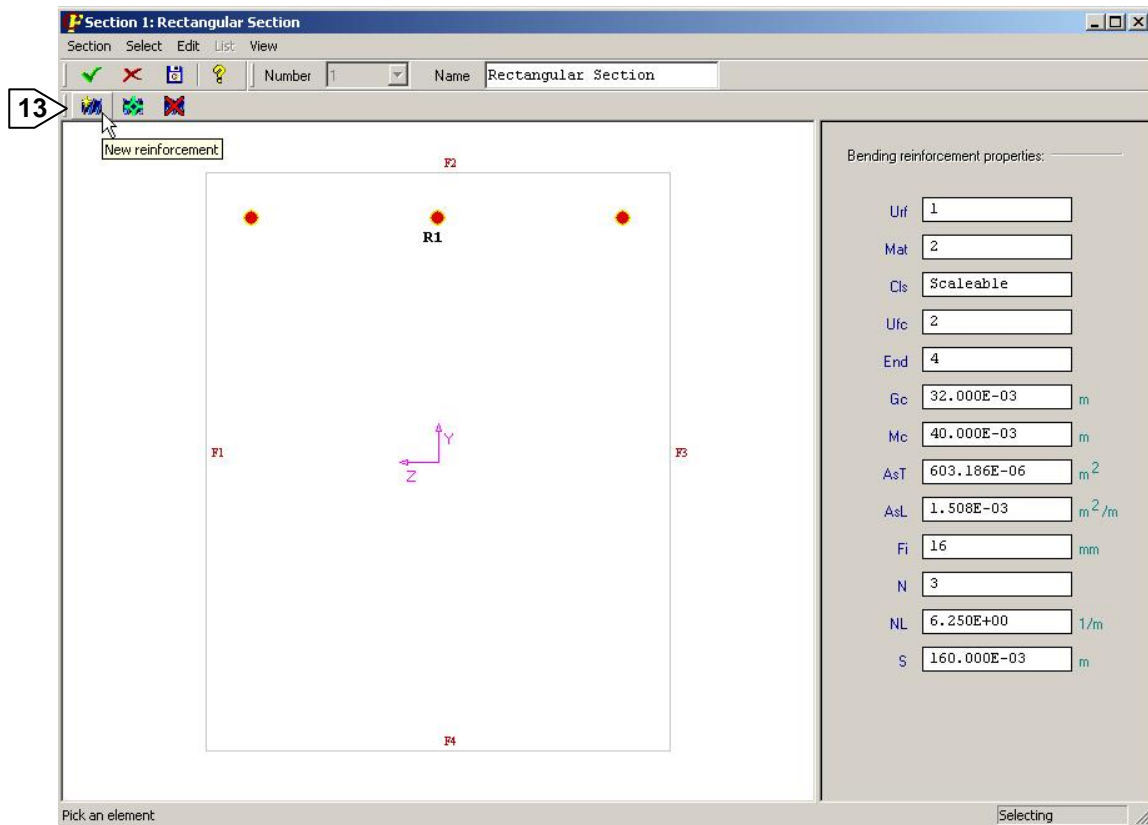
NL: 6.329E+00 **9** Fi: 16.00 mm **8** By Number of bars

S: 158.000E-03 m By Number of bars / Length

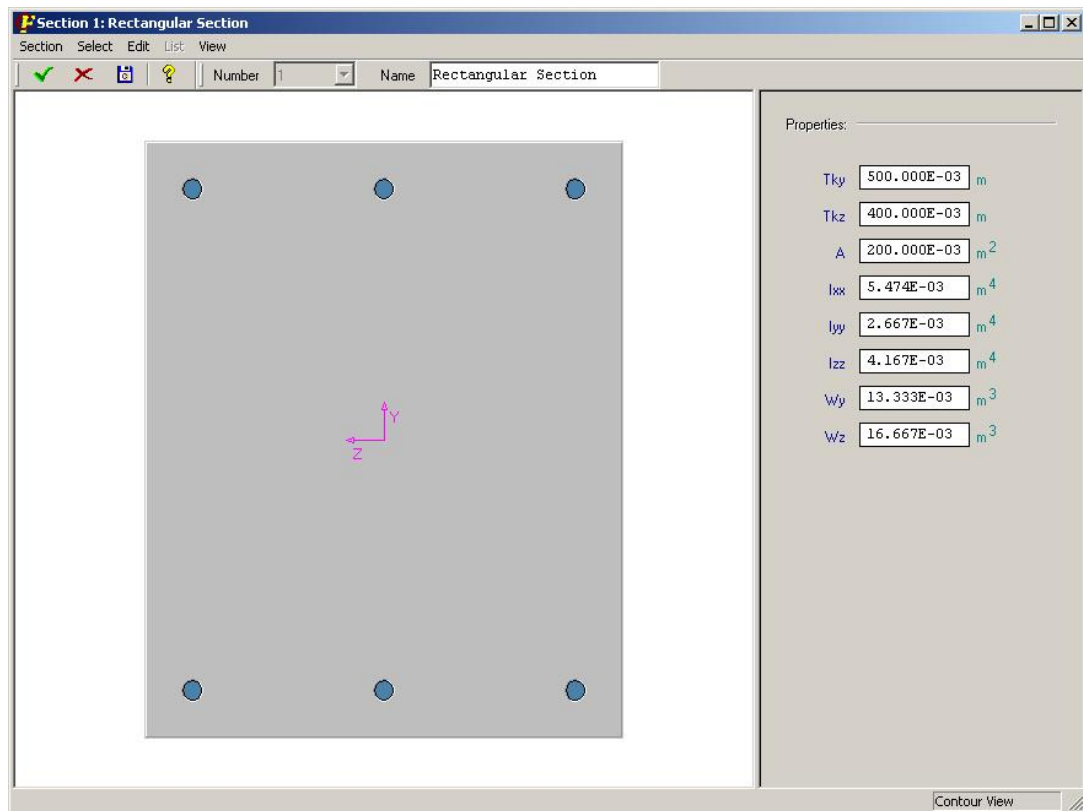
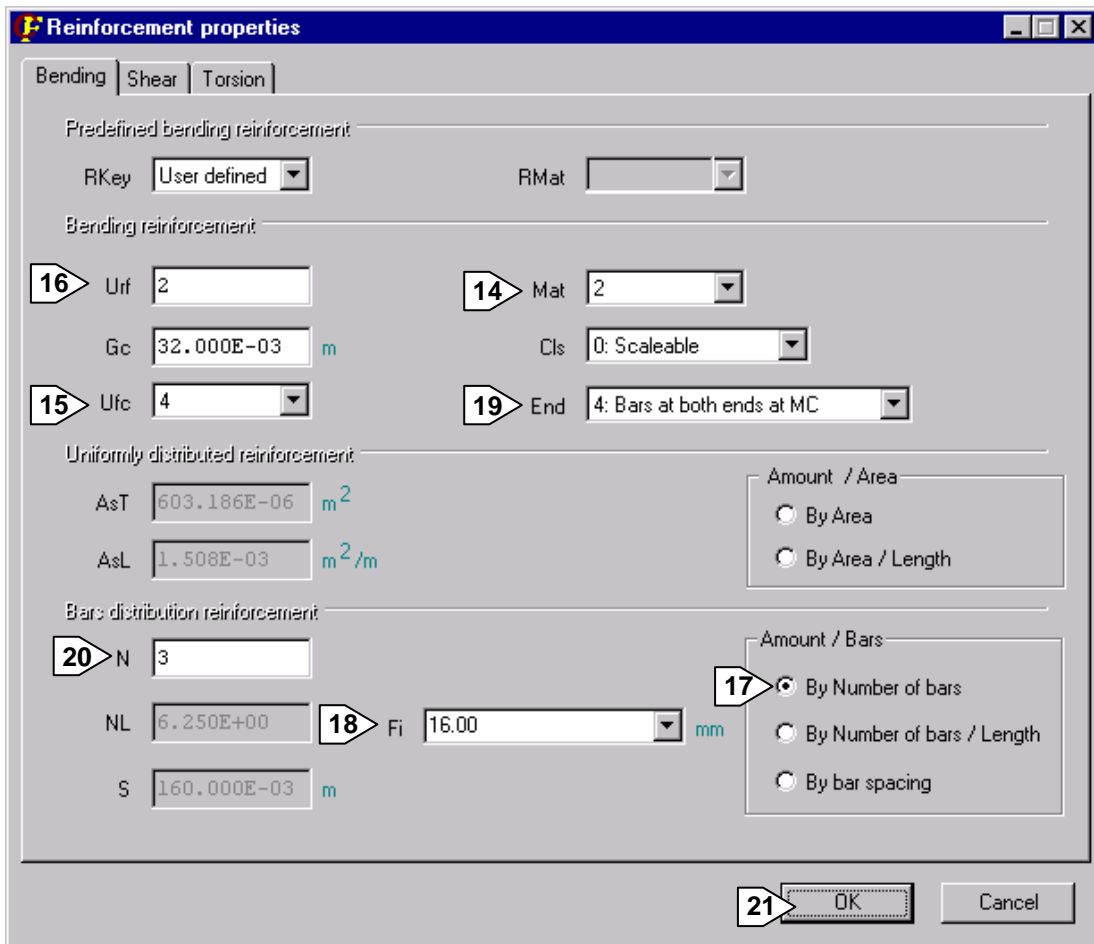
By bar spacing

12 OK Cancel

13 New Reinforcement



- 14 Select material 2 as material for bars
- 15 Select Reinforcement at Face 4
- 16 Enter 0.032 for Geometrical cover
- 17 Specify reinforcement distribution by number of bars
- 18 Select 16 mm as Diameter of Bars
- 19 Select Bars at both ends at MC
- 20 Enter 3 as number of bars
- 21 Ok



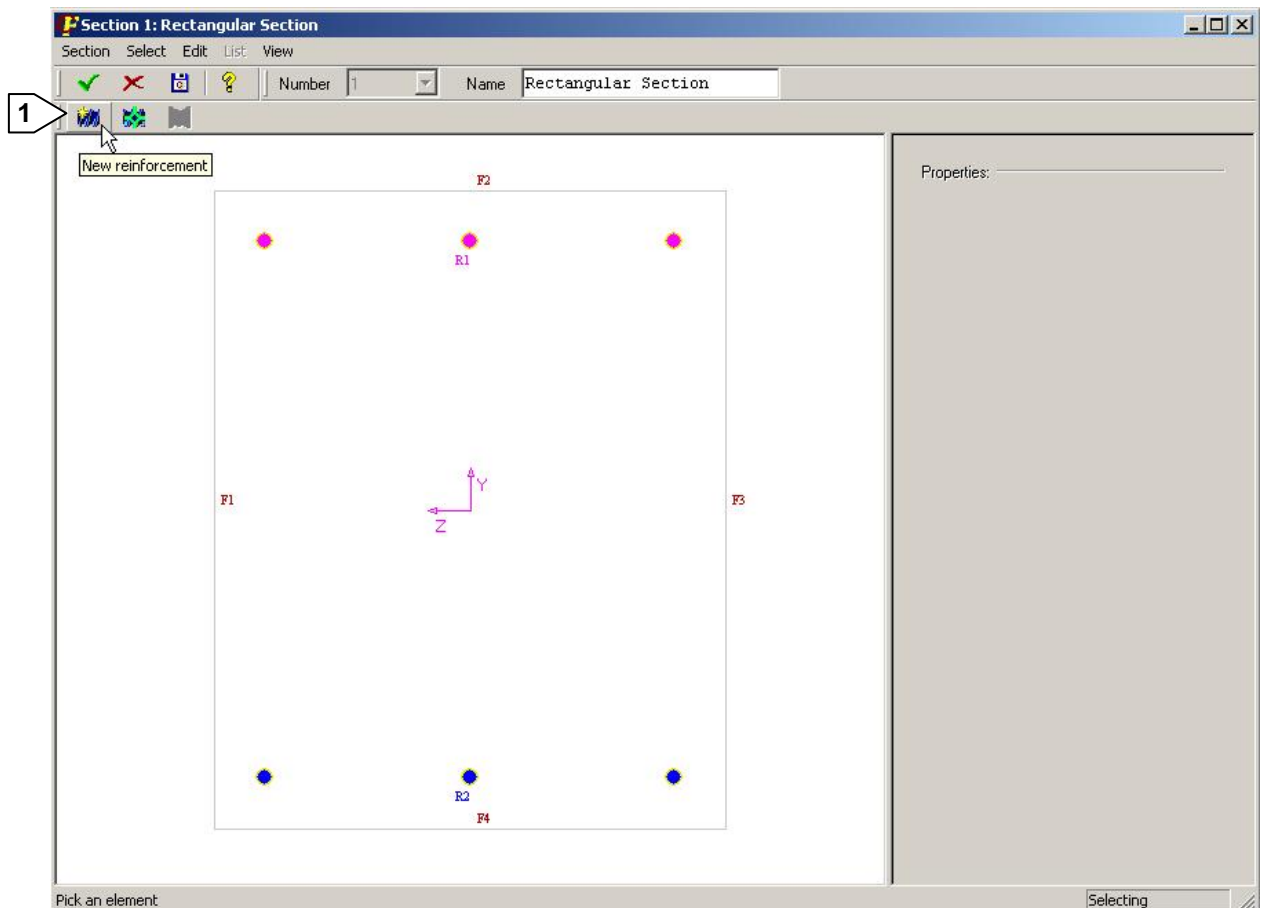
8. Define shear reinforcement properties

In order to define the shear reinforcement we have to take into account the following considerations:

- Angles between the stirrups and the element local axis X.
- Shear reinforcement amount.

Shear reinforcement	
Angle	90°
Diameter	10 mm
Spacing of stirrups	0.3 m
Number of legs	2

1 Click New Reinforcement



2 Pick on the Shear tab

Reinforcement properties

Bendi 2 Shear Torsion

Shear reinforcement

Amount

- By Ratio
- By Area / Distance
- By Number of bars

3 Mat 2

4 AlpY 90.000E+00 ° AlpZ 0.000E+00 ° 5

Ratio

AssY 523.599E-06 m²/m AssZ 0.000E+00 m²/m

Area

AsY 157.080E-06 m² AsZ 0.000E+00 m²

S 300.000E-03 m

Number

6 NY 2 NZ 0

7 S 300.000E-03 m

8 Fi 10.00 mm

9 OK Cancel

3 Enter 2 for reinforcement material

4 Enter 90 as angle with X axis (by default = 90°)

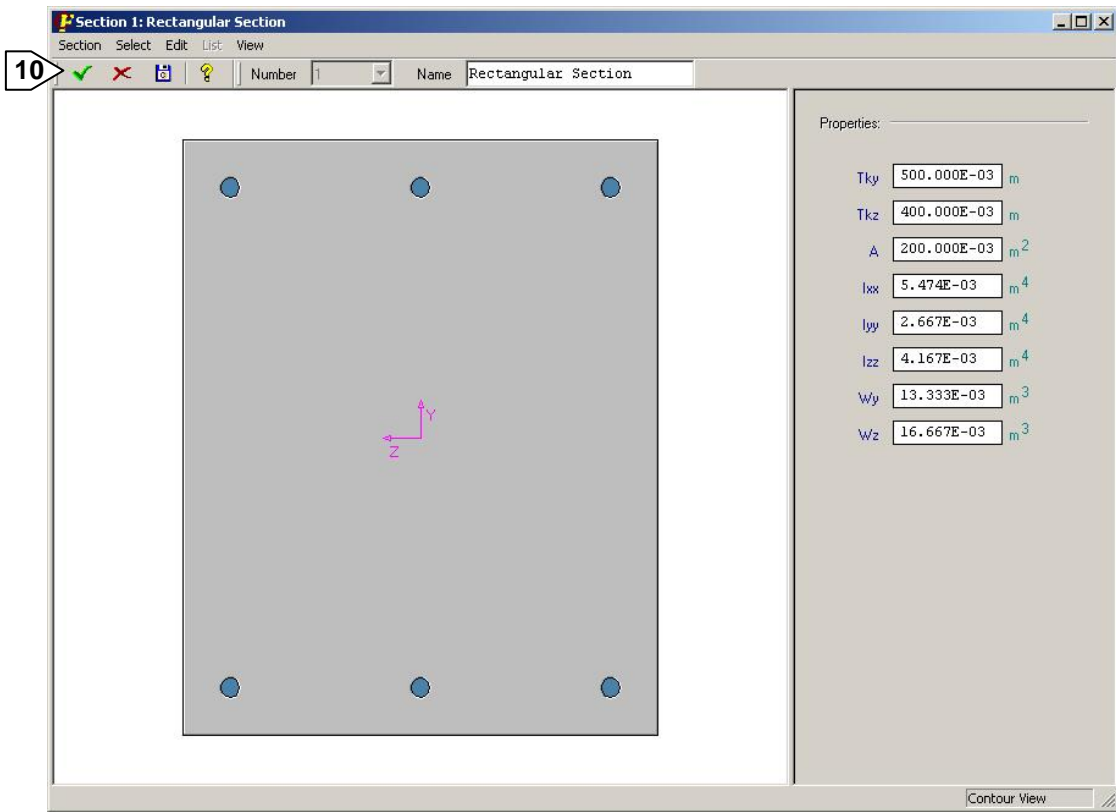
5 Select By number of bars

6 Enter 2 as number of legs in one stirrup.

7 Enter 0.3 as distance between stirrups

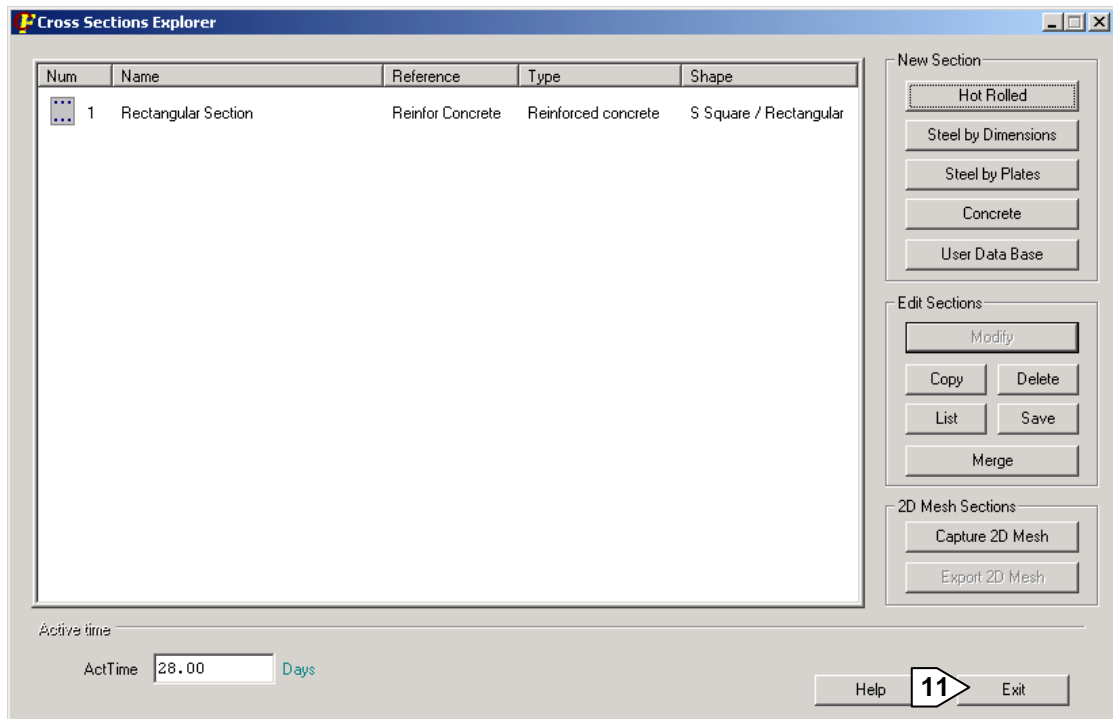
8 Select 10 mm as diameter of bars

9 OK



10 OK

11 Exit

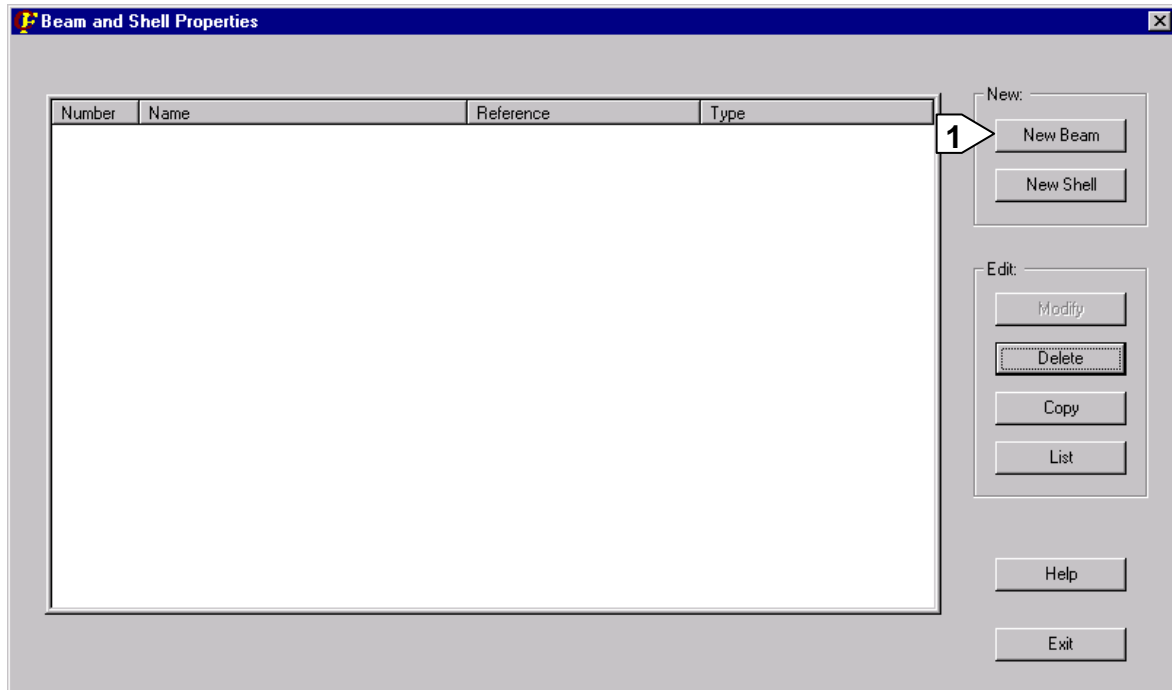


9. Define Beam & Shell properties

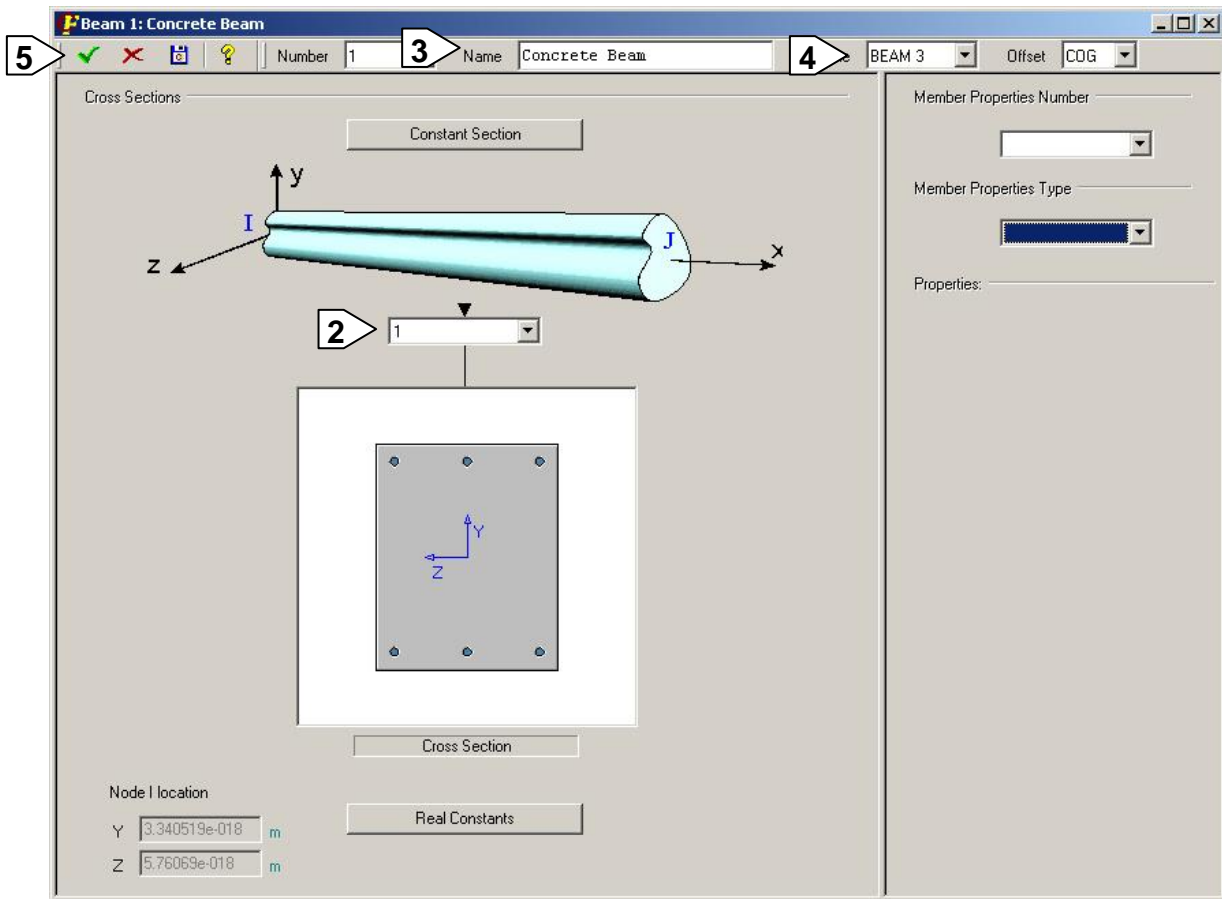
CivilFEM command **~BMSHPRO** will be used to define ANSYS real constants.

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

- 1 Pick on the New Beam button

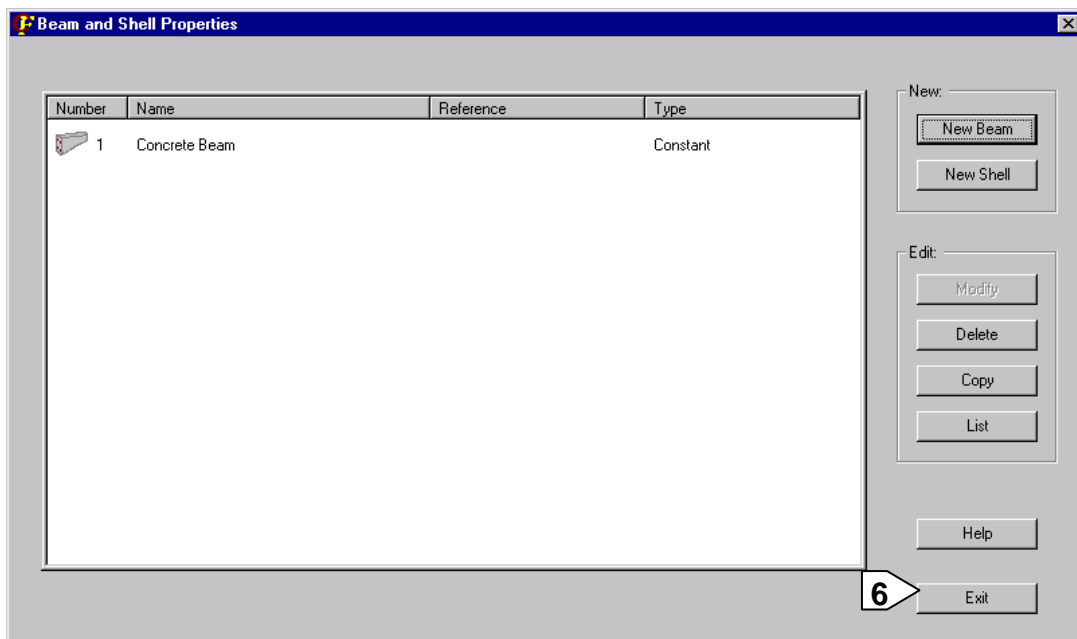


- 2 Select cross section number
- 3 Enter "Concrete Beam" as Name for the Beam property
- 4 Select element type BEAM 3



5 Ok

6 Exit



10. Define nodes and elements

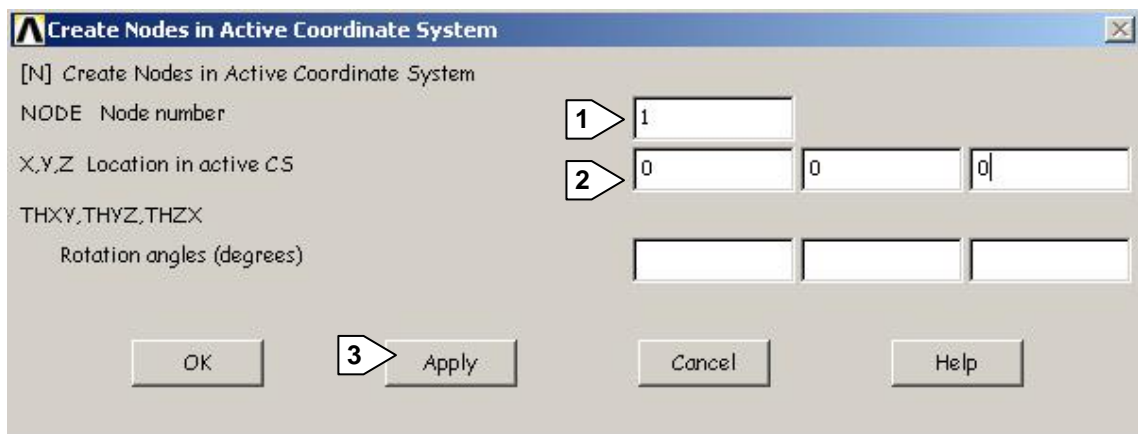
Frame geometry definition implies the creation of nodes and elements.

The steps that you must follow are:

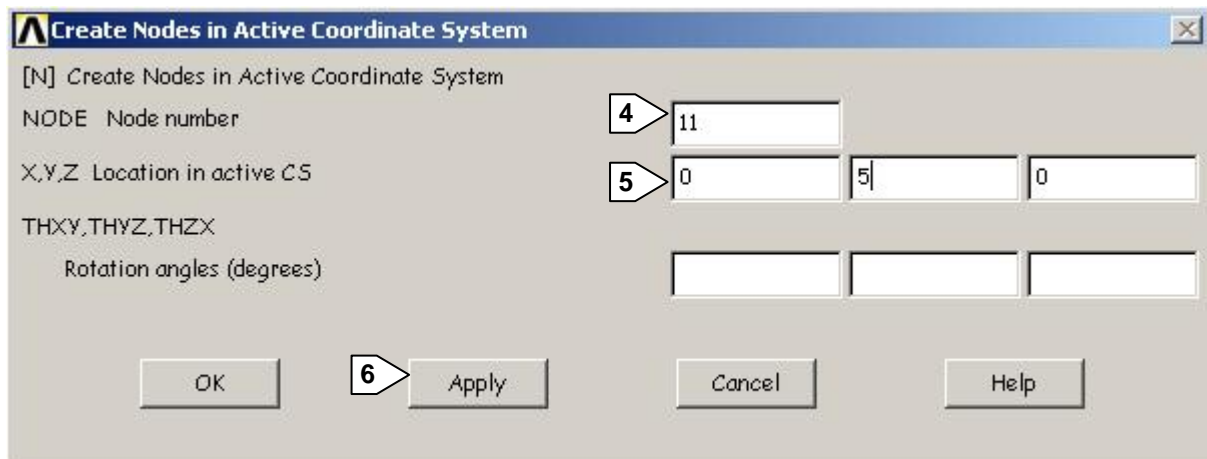
- a) Define nodes coordinates in active coordinate system
- b) Define elements

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

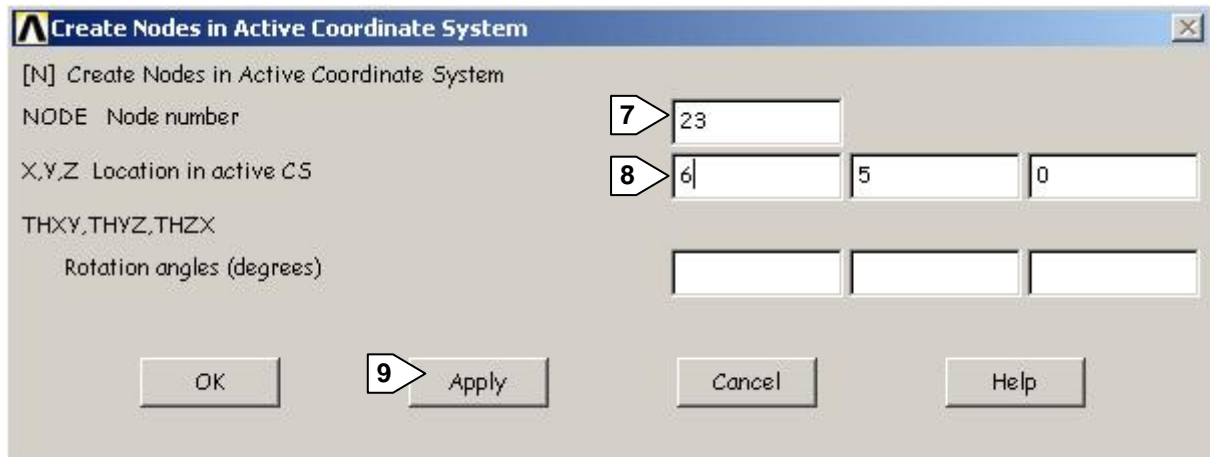
- 1 Enter 1 as node 1
- 2 Introduce the coordinates of node 1
- 3 Pick on apply



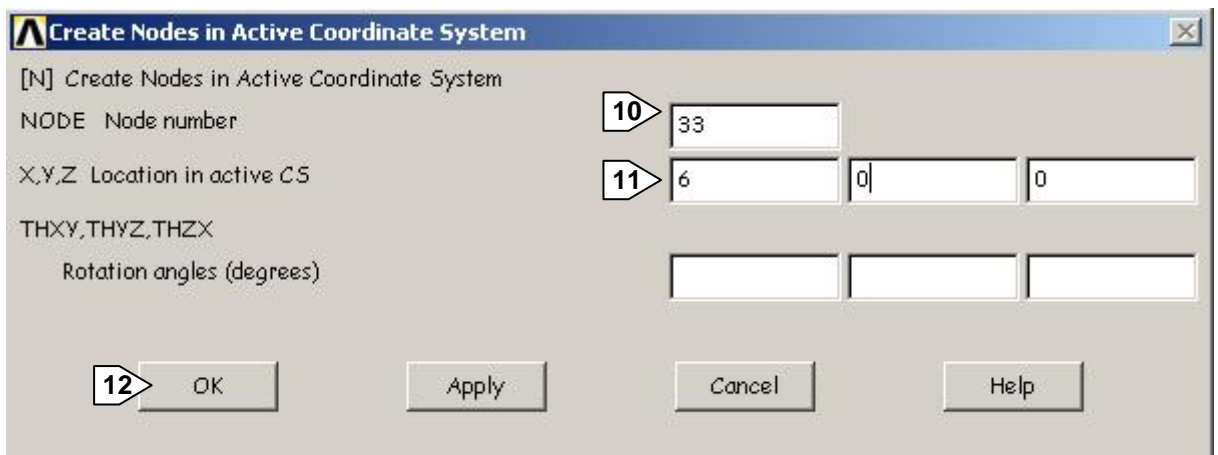
- 4 Enter 11 as node 11
- 5 Introduce the coordinates of node 11
- 6 Pick on apply



- 7 ➤ Enter 23 as node 23
- 8 ➤ Introduce the coordinates of node 23
- 9 ➤ Pick on apply



- 10 ➤ Enter 33 as node 33
- 11 ➤ Introduce the coordinates of node 33
- 12 ➤ OK

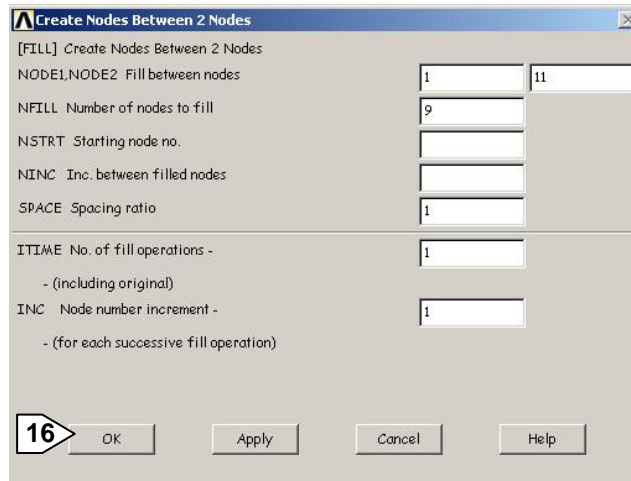
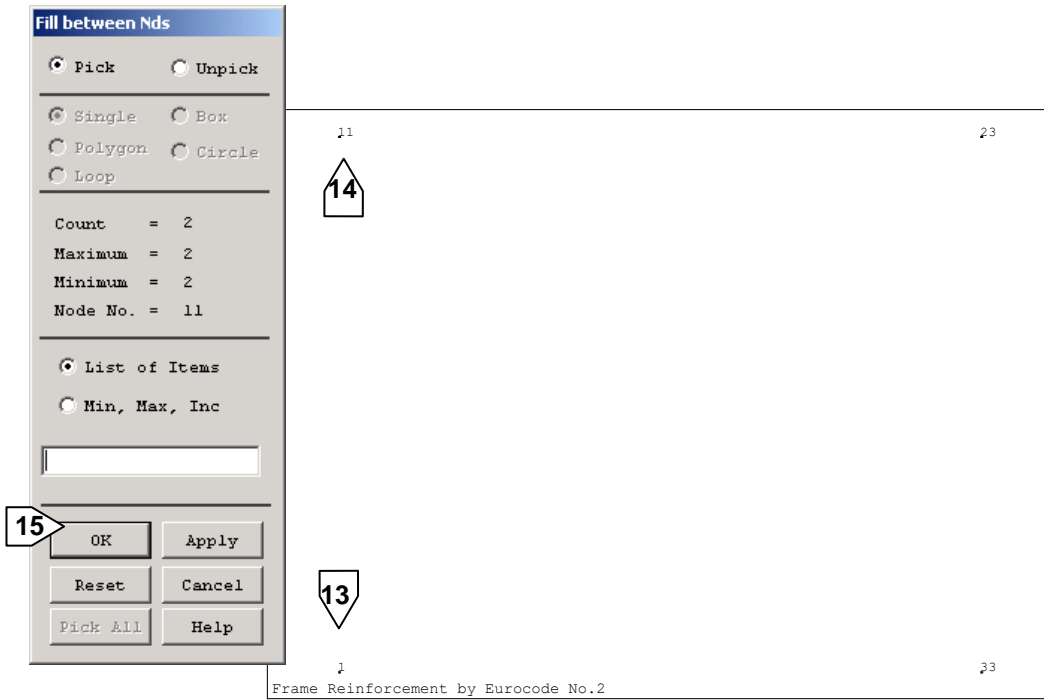


Now we are going to introduce the rest of the nodes by filling between nodes.

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

- 13 ➤ Select node 1
- 14 ➤ Select node 11
- 15 ➤ OK

16 OK



17 Select node 11

18 Select node 23

19 OK

20 OK

17

19 OK Apply
Reset Cancel
Pick All Help

1 11 23
10
9
8
7
6
5
4
3
2
1
Frame Reinforcement by Eurocode No.2 23

20 OK Apply Cancel Help

Create Nodes Between 2 Nodes

[FILL] Create Nodes Between 2 Nodes

NODE1,NODE2 Fill between nodes 11 23

NFILL Number of nodes to fill 11

NSTRT Starting node no.

NINC Inc. between filled nodes

SPACE Spacing ratio 1

ITIME No. of fill operations - 1
- (including original)

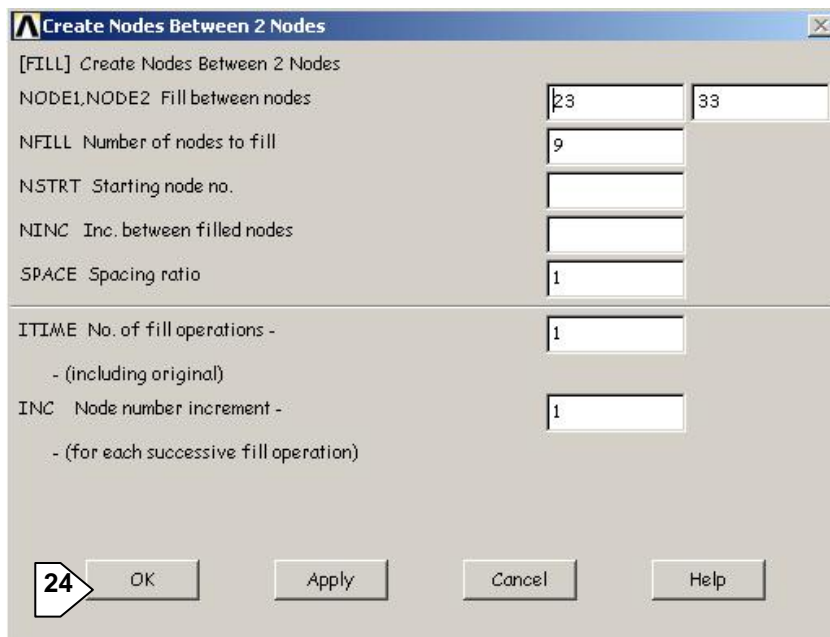
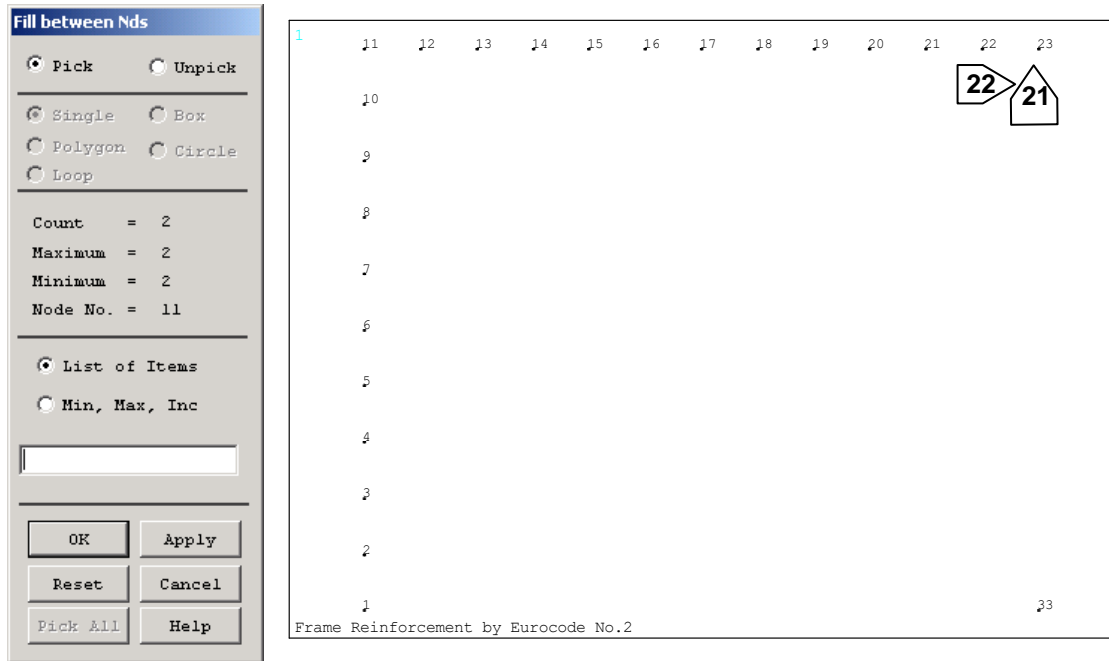
INC Node number increment - 1
- (for each successive fill operation)

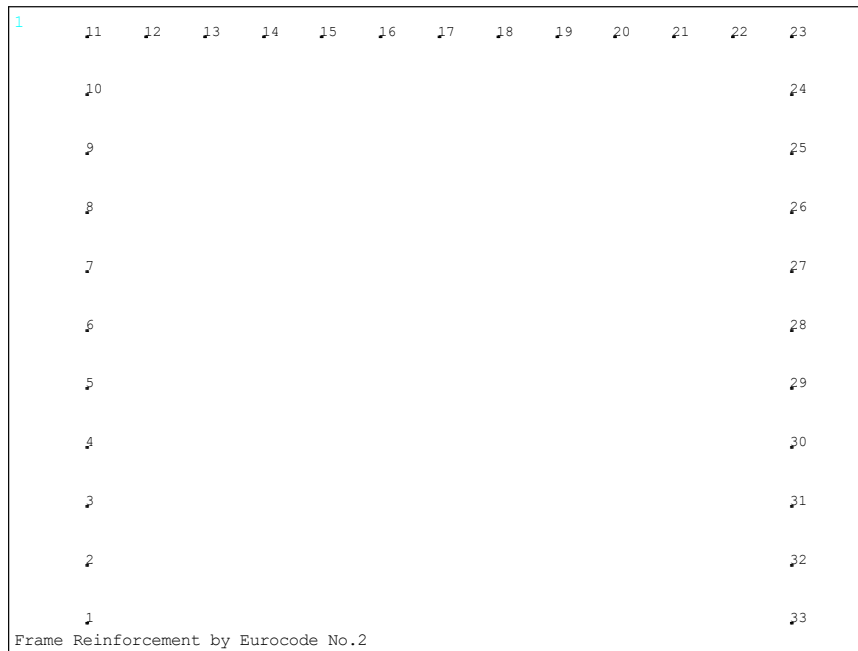
21 Select node 23

22 Select node 33

23 OK

24 OK





Now we are going to introduce the first element between nodes 1 and 2

Main Menu: **Preprocessor** → –Modeling– **Create** → **Elements** → –Auto Numbered–**Thru Nodes** +

25 Pick on nodes 1 and 2

26 OK

Copy the first defined element a total of 32 times, incrementing the nodes number by 1.

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

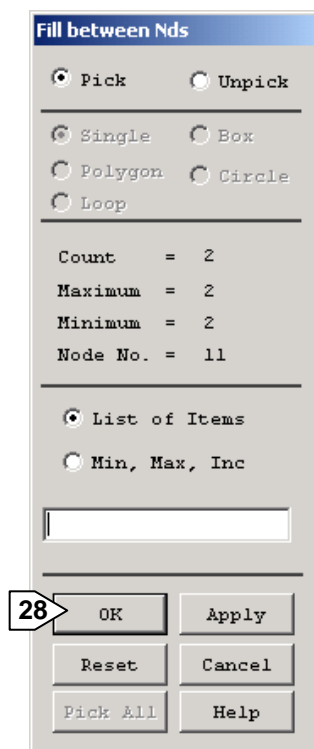
27 Pick on the existing element

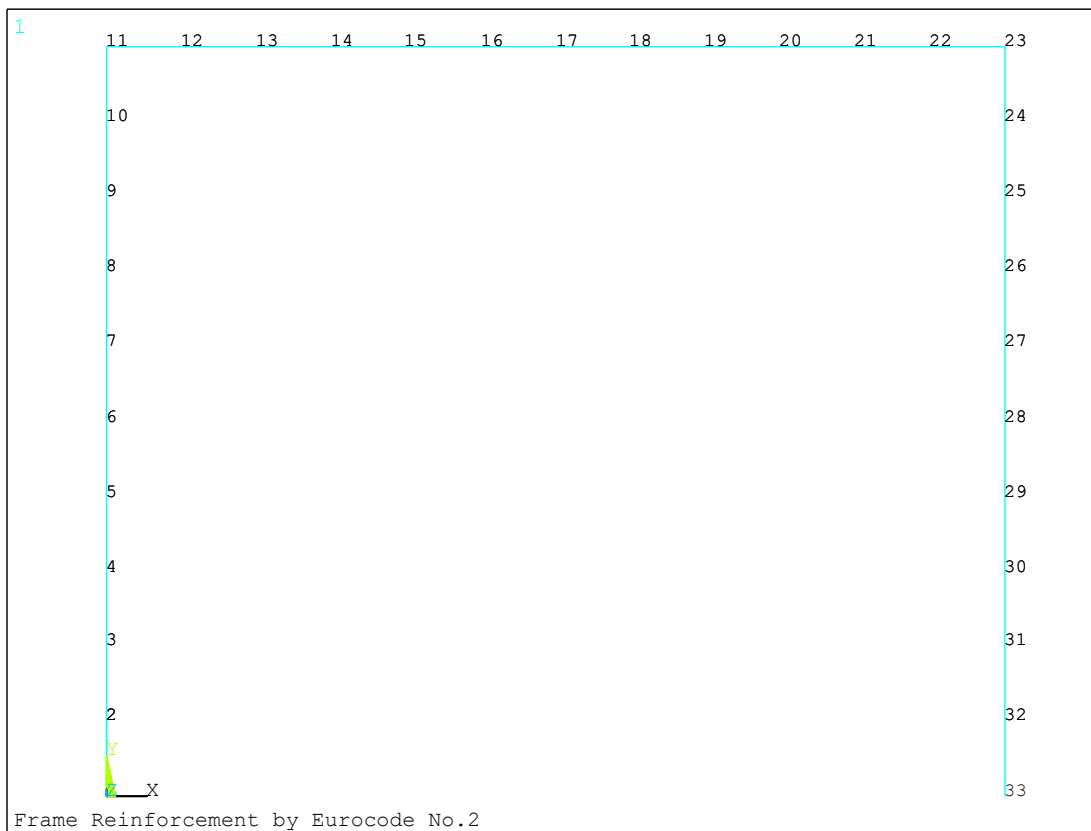
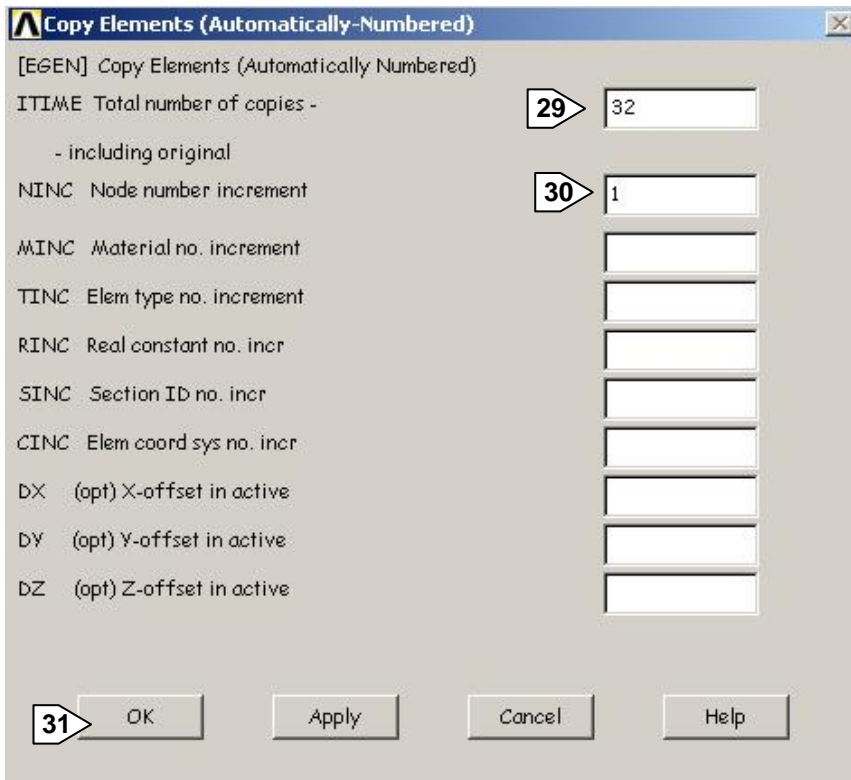
28 OK

29 Enter 32 as Total number of the copies –including the original-

30 Enter 1 as Node number increment

31 OK





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

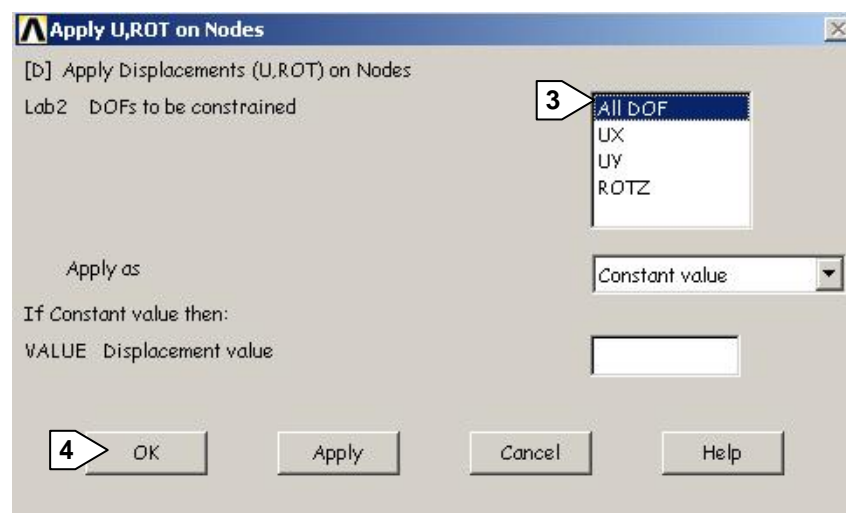
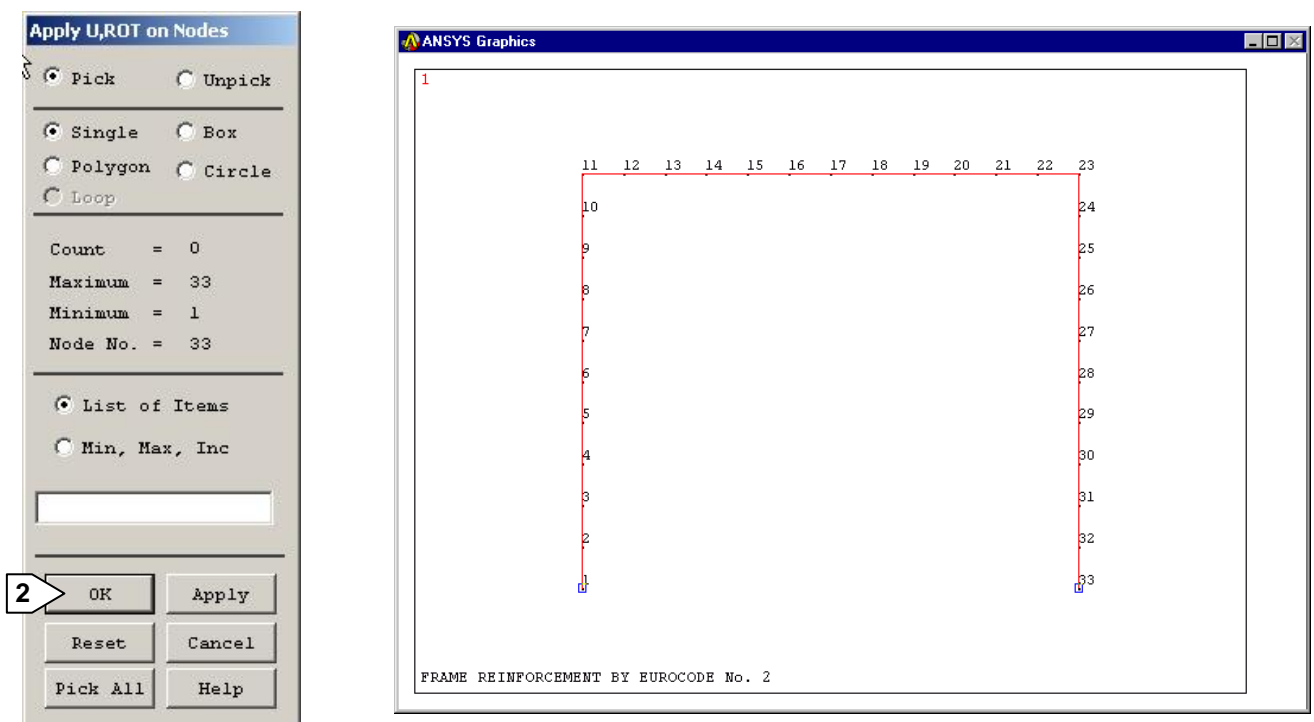
Toolbar: **CFSAVE**

■ SOLUTION

12. Apply displacement constraints

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

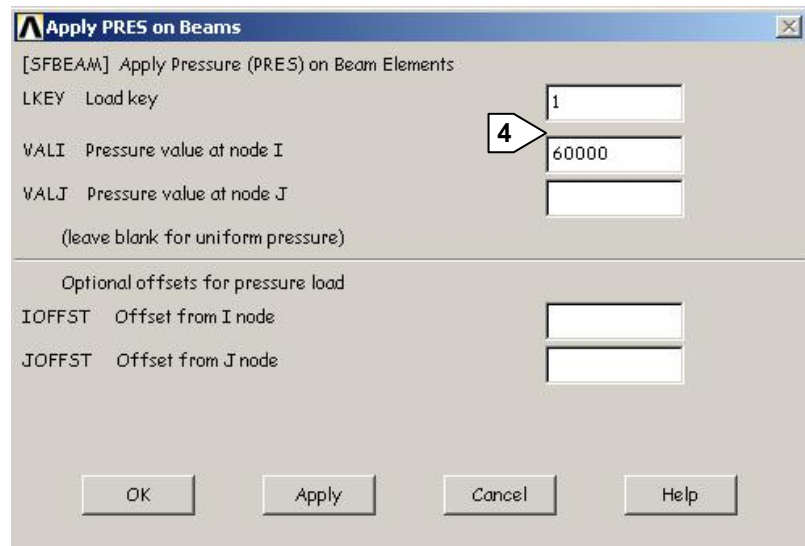
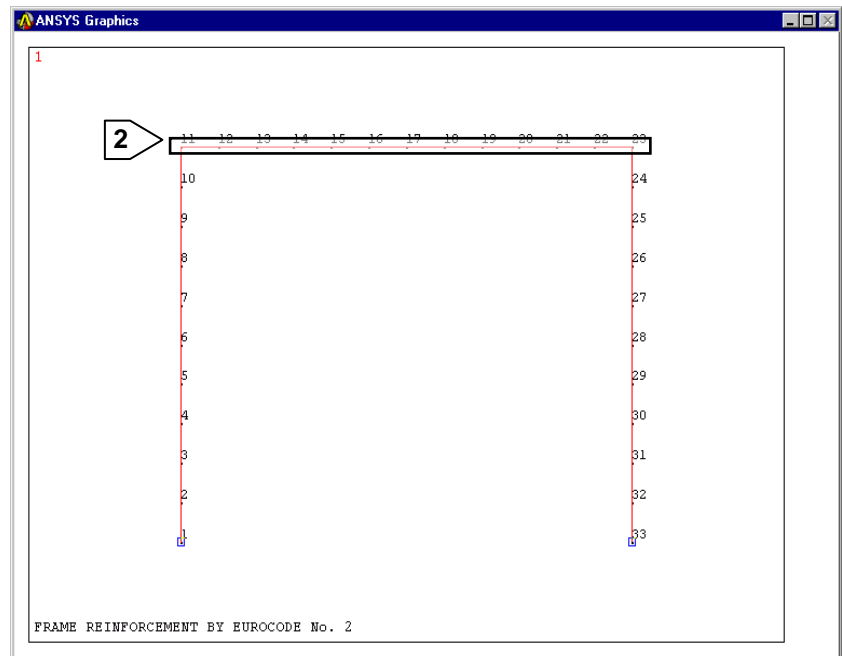
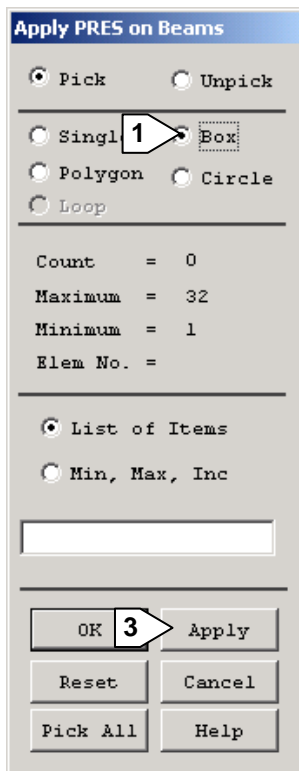
- 1 ▷ Pick on the nodes 1 and 33
- 2 ▷ OK
- 3 ▷ Choose All DOF
- 4 ▷ Click OK to close dialog box

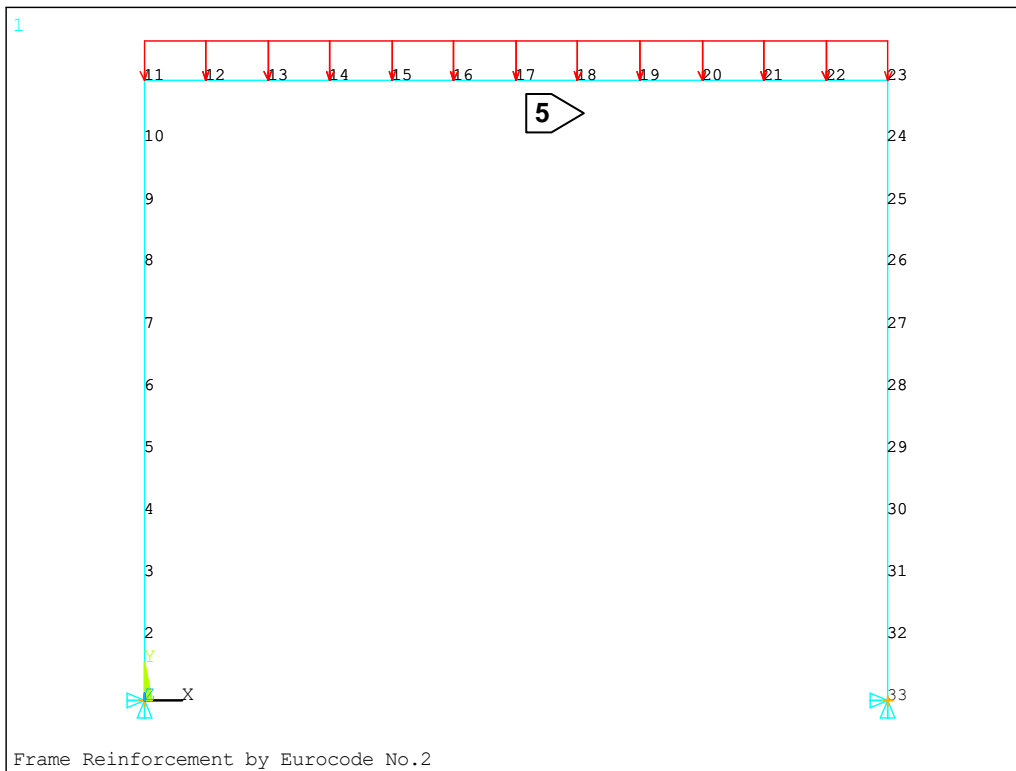


13. Apply load

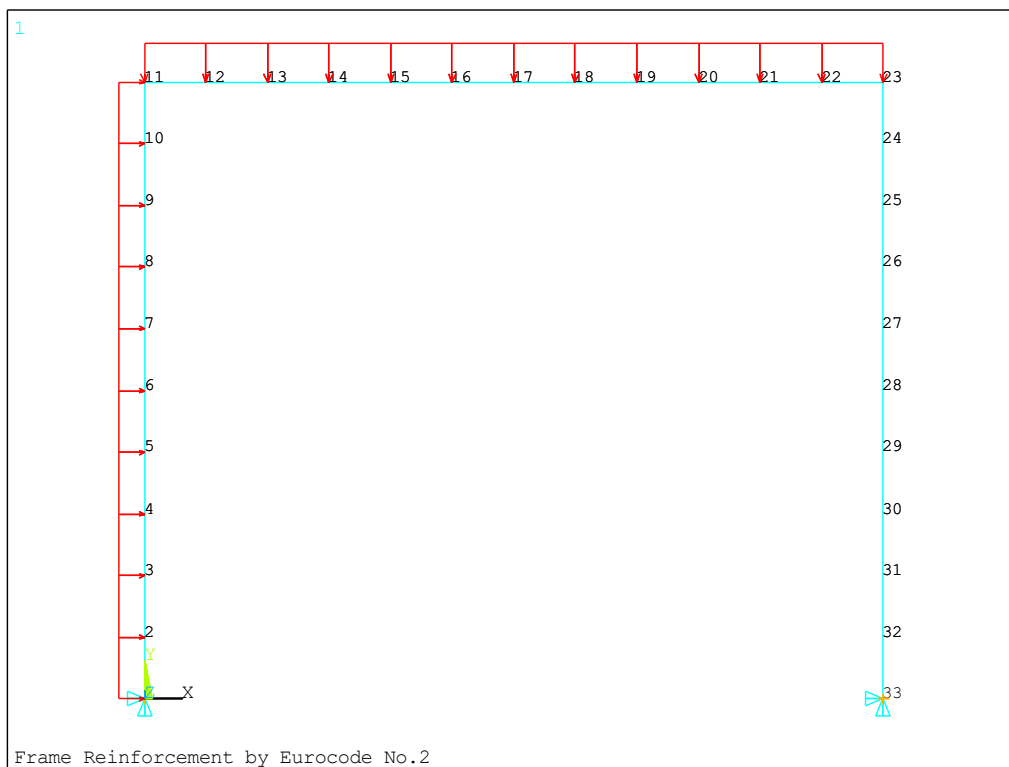
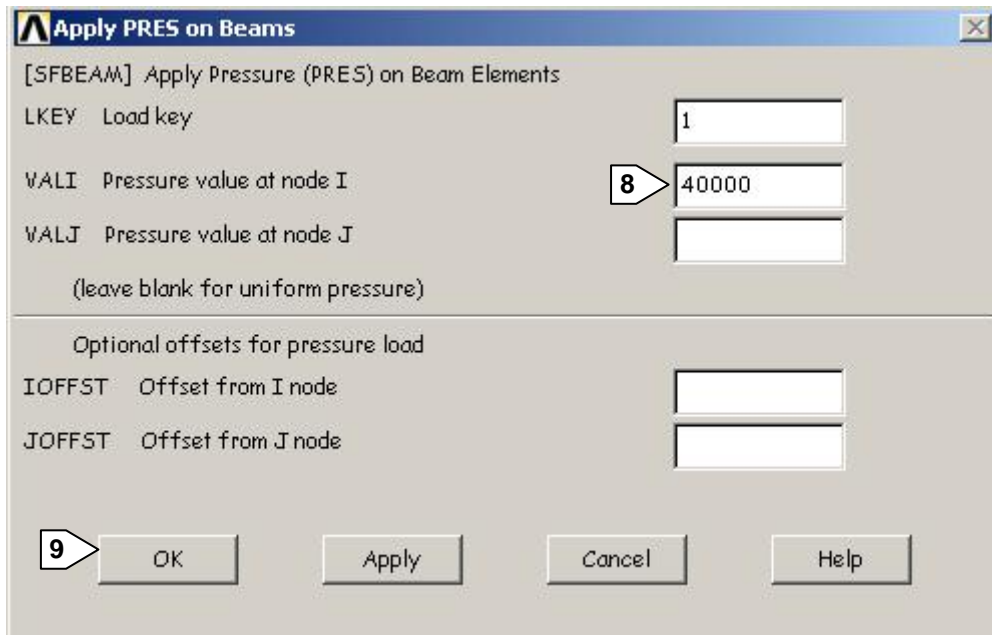
Main Menu: **Solution** → – Loads – **Apply** → – Structural – **Pressure** → **On Beams**

- 1 Pick Box
- 2 Press and drag the left mouse button to form a box around the top beam elements
- 3 Apply
- 4 Enter 60e3 as pressure value
- 5 Choose Apply



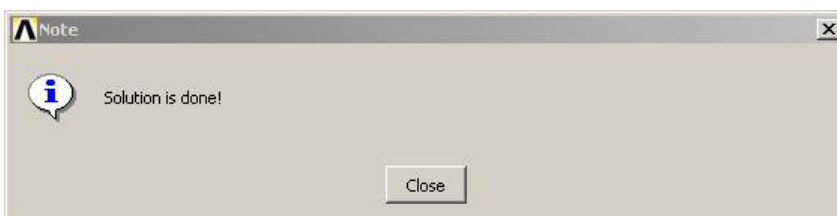
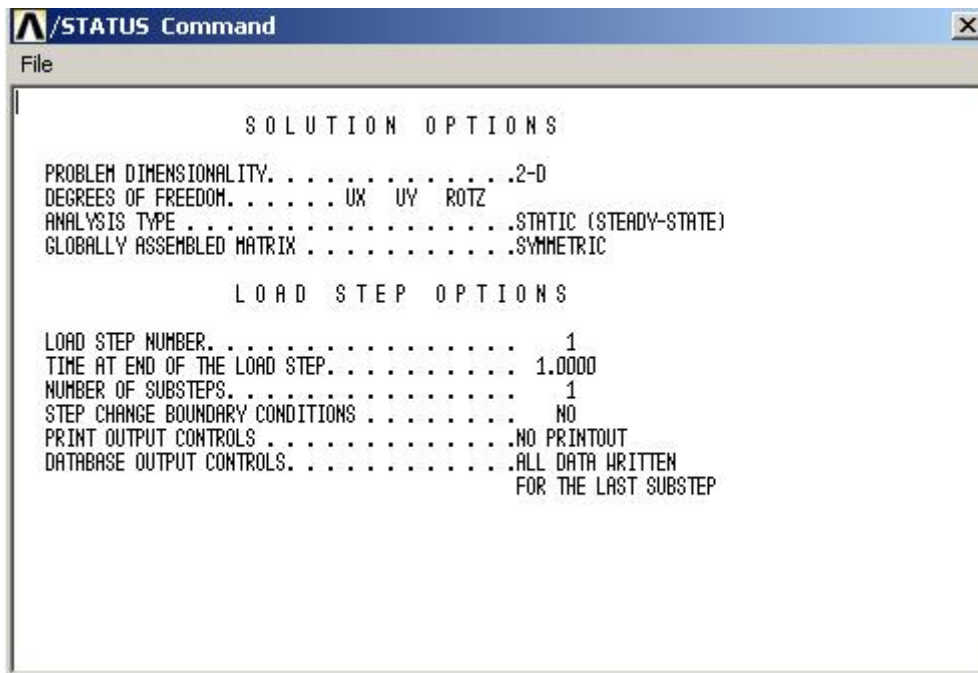


- 5 Pick Box
- 6 Press and drag the left mouse button to form a box around the left beam elements
- 7 OK
- 8 Enter 40e3 as pressure value
- 9 OK



14. Solve

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



■ Postprocessing

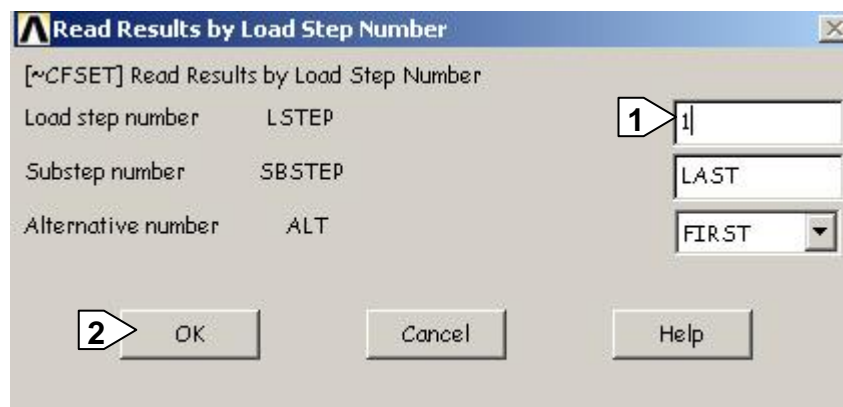
15. Enter the postprocessor and read in results

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 Postproces** → **Read Results** → **By Load Step**

1 Enter 1 in the Load Step number box

2 OK to read load step 1



16. Checking under 2D bending moment and axial force

Now we are going to check the frame under bending moment and axial force according to Eurocode 2 provisions. We will check bending on element local plane XY (bending M_z) which is the default value.

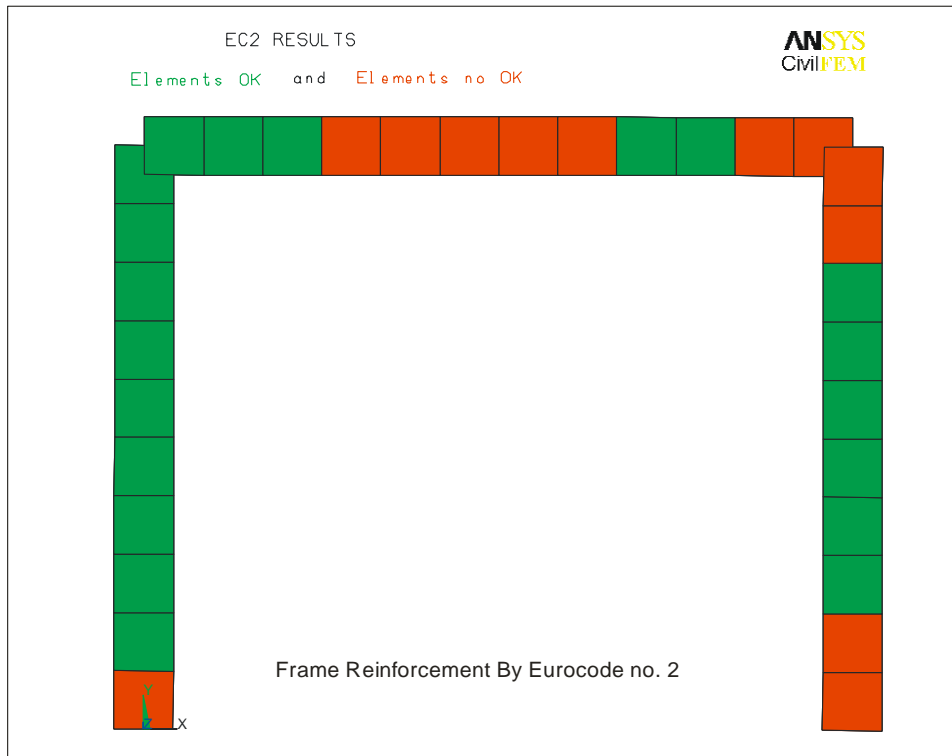
Main Menu: CivilFEM – **Civil Postprocessor** → **Code Checking** → **Eurocode 2** → **CHECK BY CODE** → **Beams & Solid** → **2D Axial+Bend**

1 OK to check bending moment M_z according to Eurocode 2



17. Review elements OK and No OK

In order to review the checked results according to Eurocode 2, 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 Eurocode 2 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.

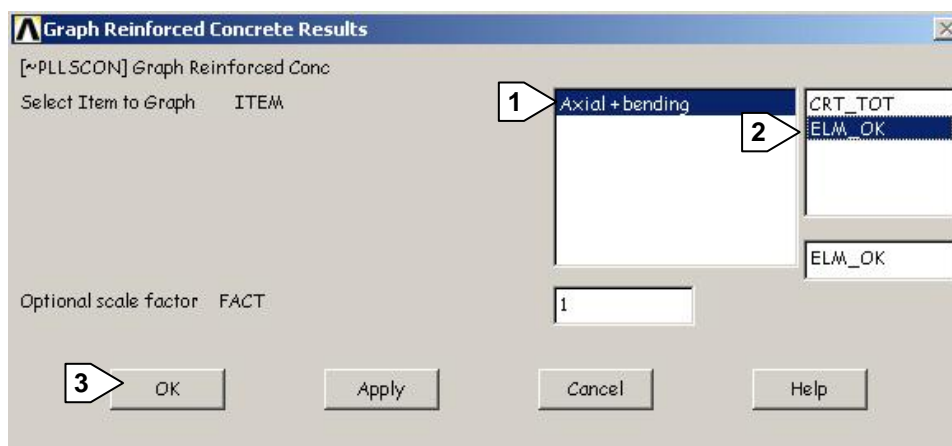


Main Menu: - CivilFEM – **Civil Postprocessor** → **Code Checking** → **Eurocode 2** → **BEAM RESULTS: Plot Results...**

1 Choose Axial+Bending

2 Elements OK/NoOK

3 OK

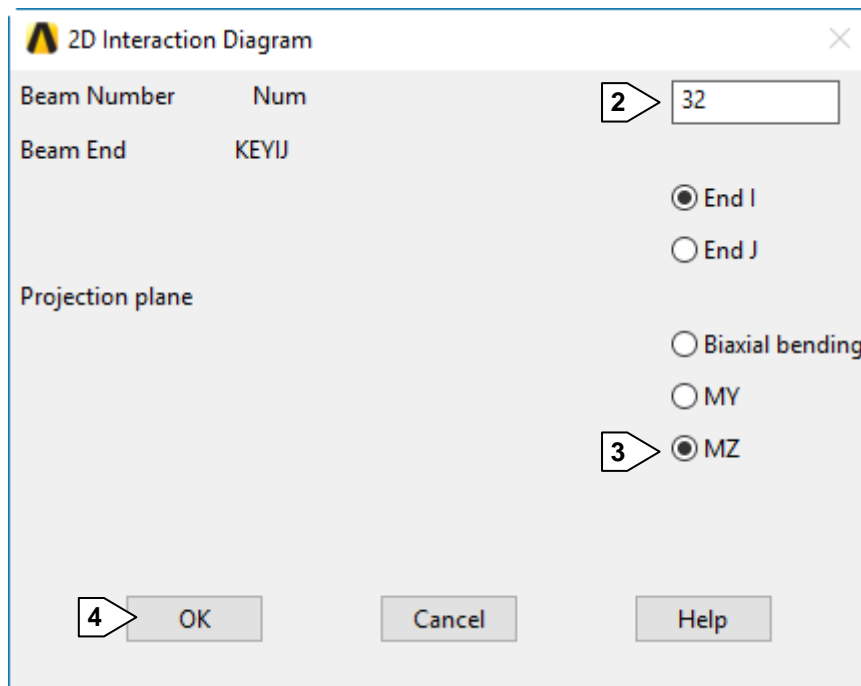
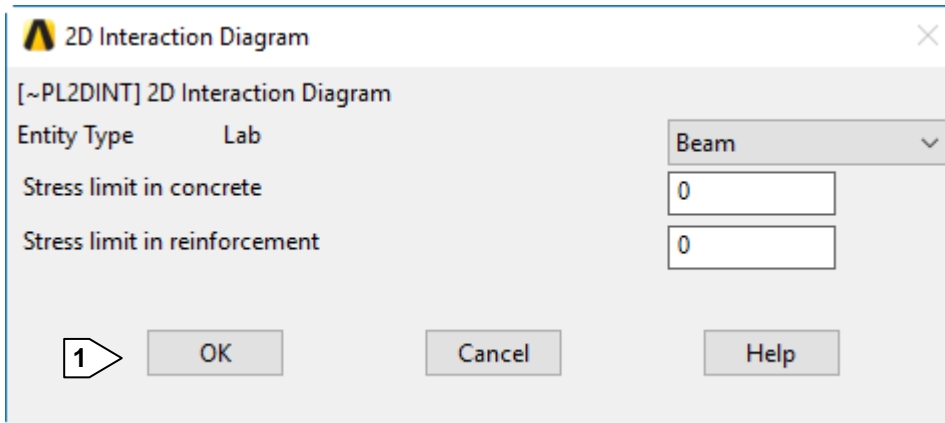


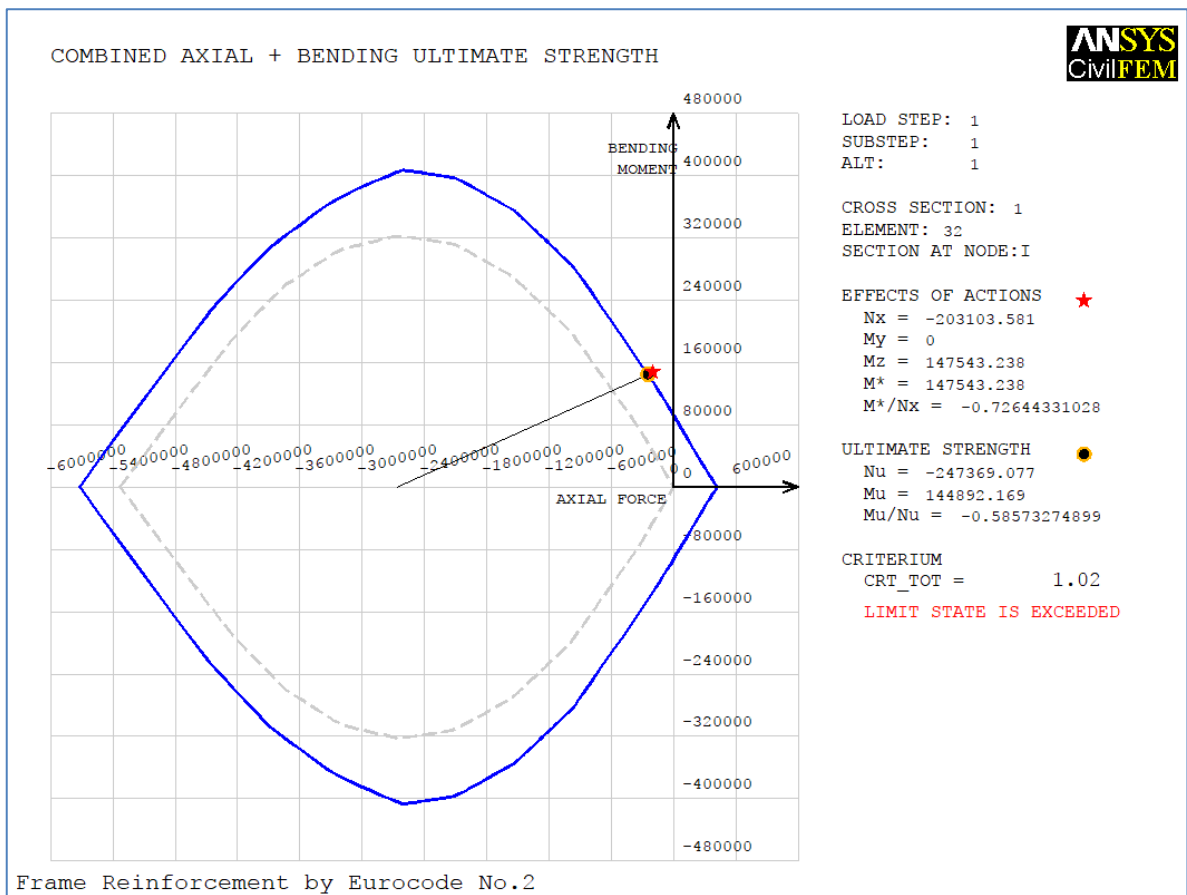
18. Plot an interaction diagram

We are going to select a non-valid element, for example element 32, and we will get the Ultimate Axial Force – Ultimate Bending Moment Interaction Diagram of the section, for bending on the element local plane XY, (moment Mz).

Main Menu: - CivilFEM – **Civil Postproces** → **Code Checking**→ **Eurocode 2**
→ **BEAM RESULTS: Interac Diag** → **2D Plot**

- 1 Click OK
- 2 Enter 32 (beam element number)
- 3 Choose MZ plane
- 4 Choose MZ plane





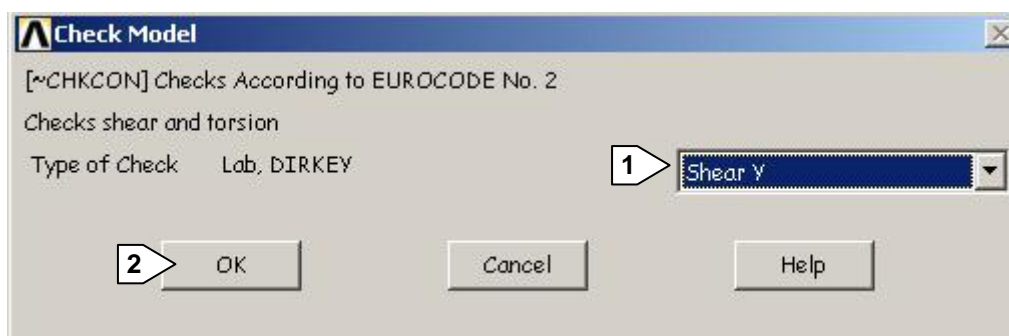
19. Check under shear force in Y axis

Now we are going to check the column elements under shear force in the Y direction, following Eurocode 2 provisions.

Main Menu: - CivilFEM – **Civil Postproces Code Checking** → **Eurocode 2** → **CHECK BY CODE: Beams & Solid** → **Shear & Torsion**

1 Choose Shear Y

2 OK



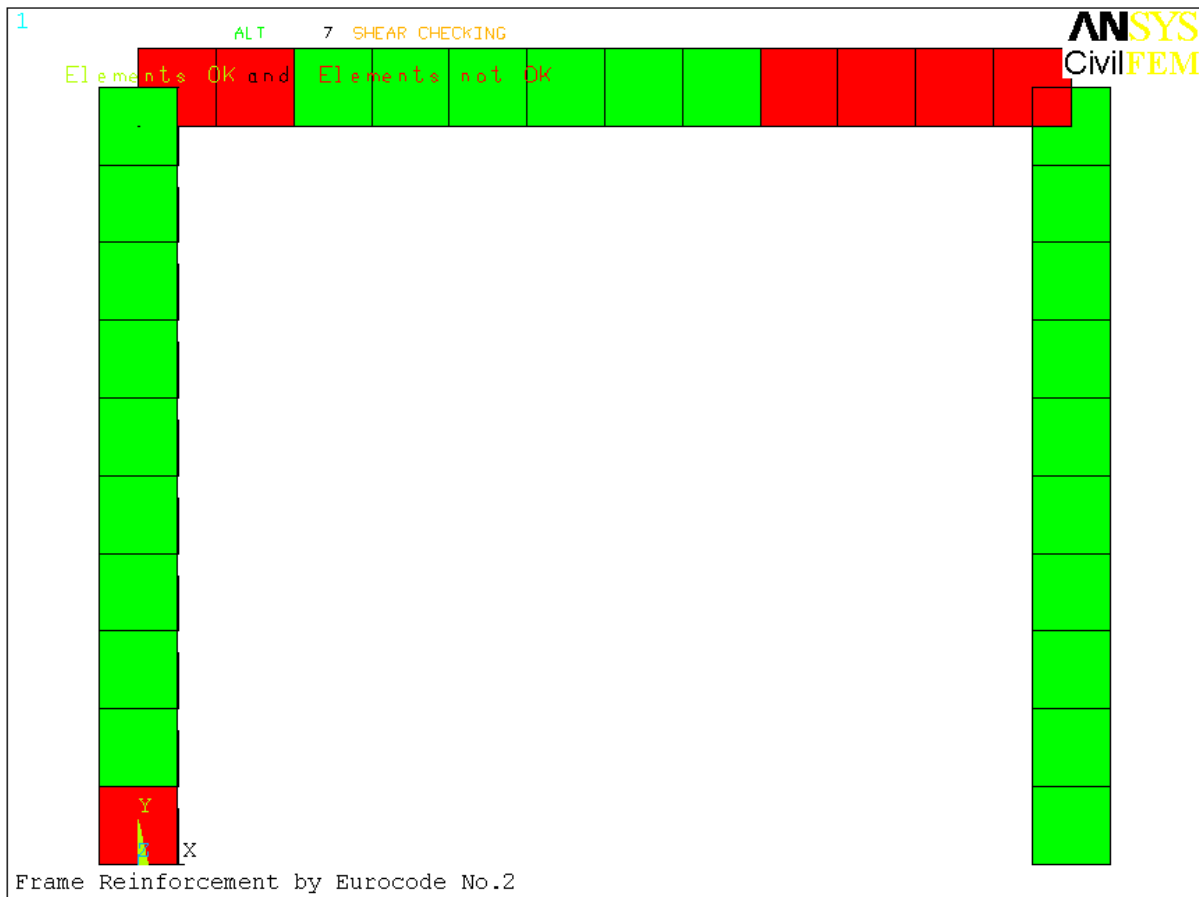
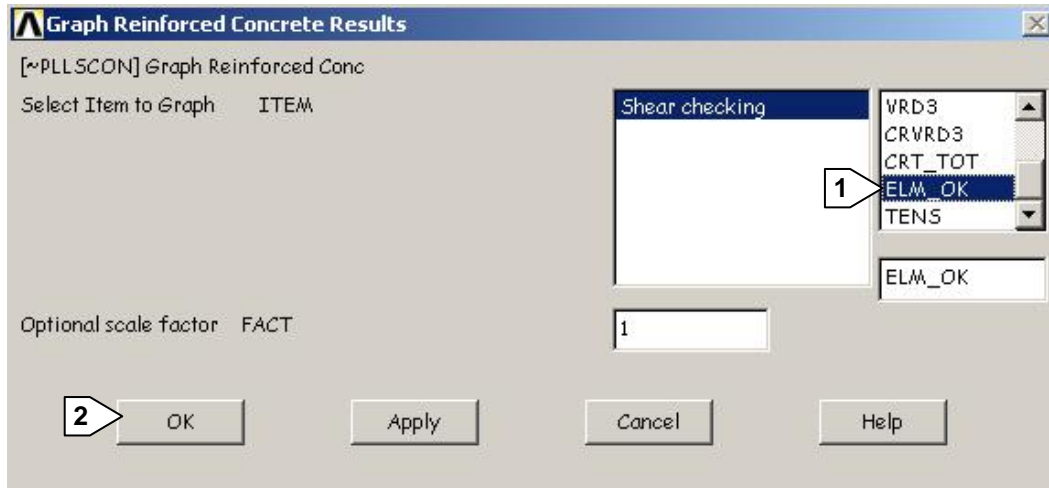
20. Review elements Ok and No OK

As we did for checking under bending moment and axial force, we are going to review the valid and non-valid elements according to Eurocode 2 criteria for checking under shear force.

Main Menu: - CivilFEM – **Civil Postprocessor** → **Code Checking**→ **Eurocode 2** → **BEAM RESULTS: Plot Results**

1 Choose Elem OK

2 OK

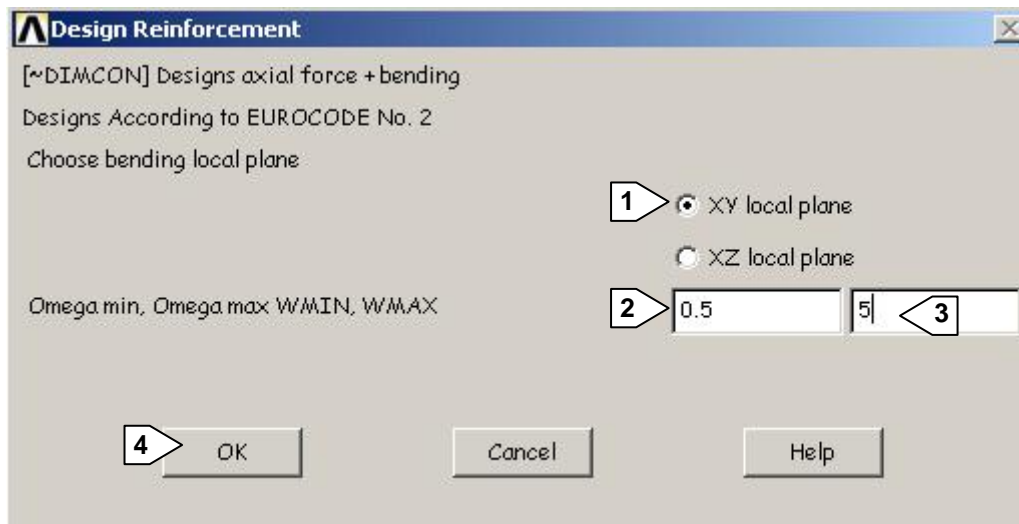


21. Design for bending moment and axial force

In this step, we are going to design the reinforcements for bending moment and axial force, to check if we satisfy the active code criteria. For this calculation, we will define the minimum and maximum allowable reinforcement ratios. During the design process, the program will always consider a reinforcement ratio equal or higher than W_{MIN} times the reinforcement initially placed in the section.

Main Menu: - CivilFEM – **Civil Postprocessor** → **Code Checking** → **Eurocode 2** → DESIGN BY CODE: -Beams & Solid → **2D Axial + Bend**

- 1 Select plane XY (Mz) as Elem. Local Plane Bend
- 2 Enter 0.5 as Omega Min.
- 3 Enter 5 as Omega Max.
- 4 OK

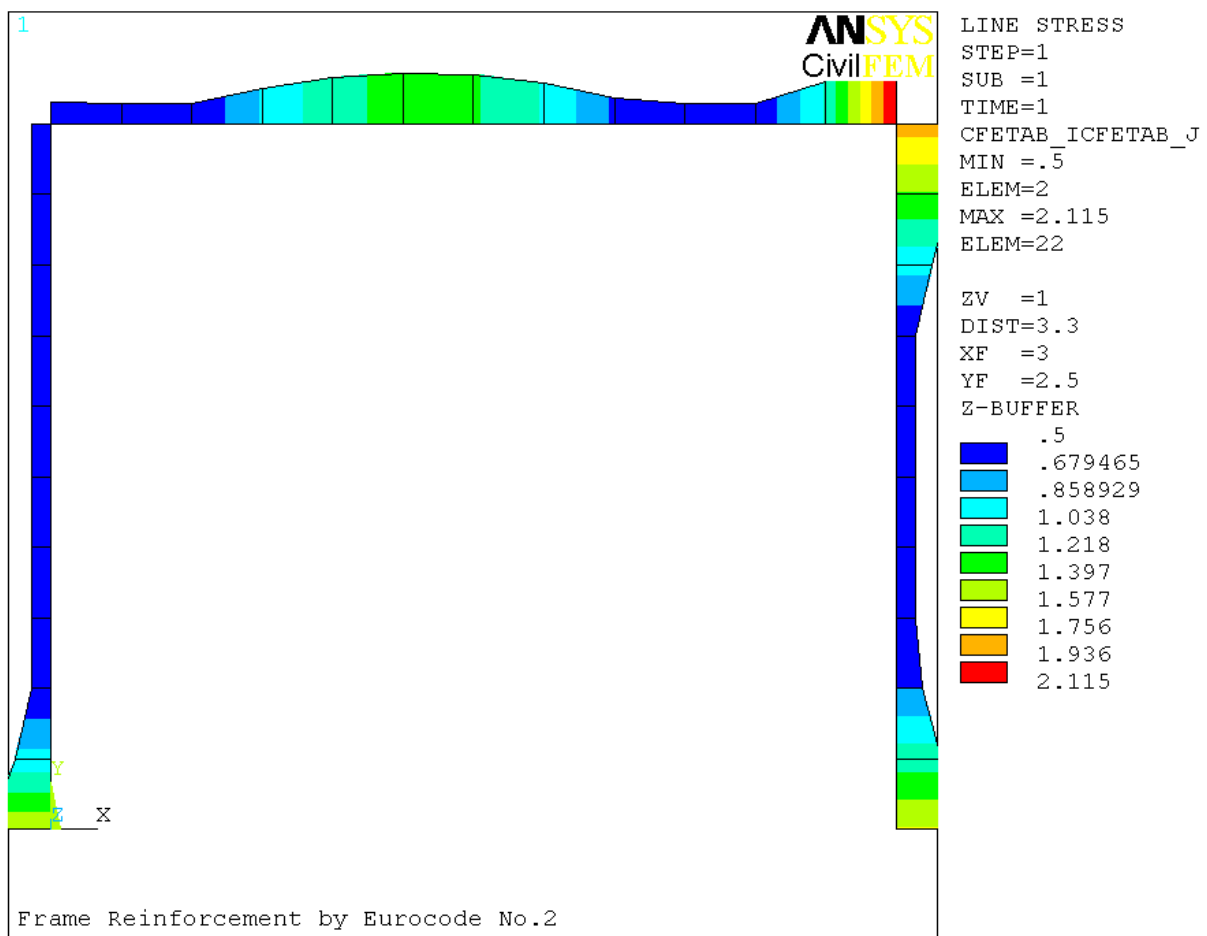
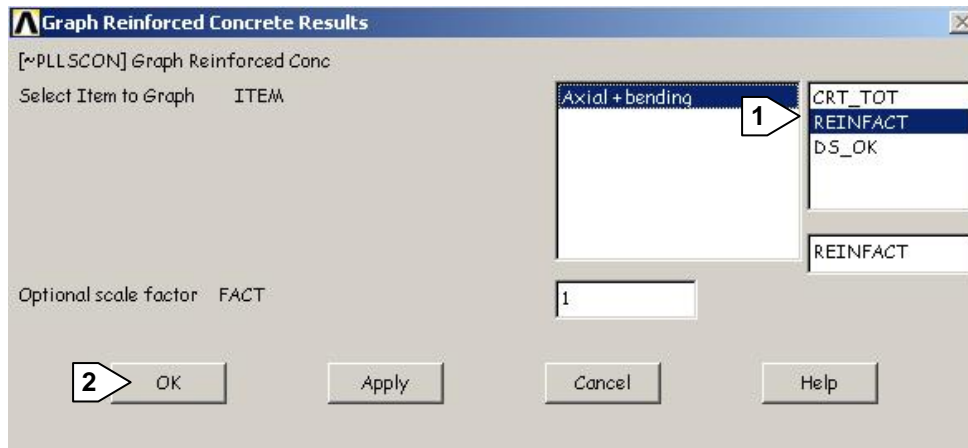


22. Review designed reinforcement for bending moment and axial force

First we are going to plot the designed reinforcement factor.

Main Menu: - CivilFEM – **Civil Postprocessor** → **Code Checking** → **Eurocode 2** → BEAM RESULTS: **Plot Results**

- 1 Choose REINFACT
- 2 OK



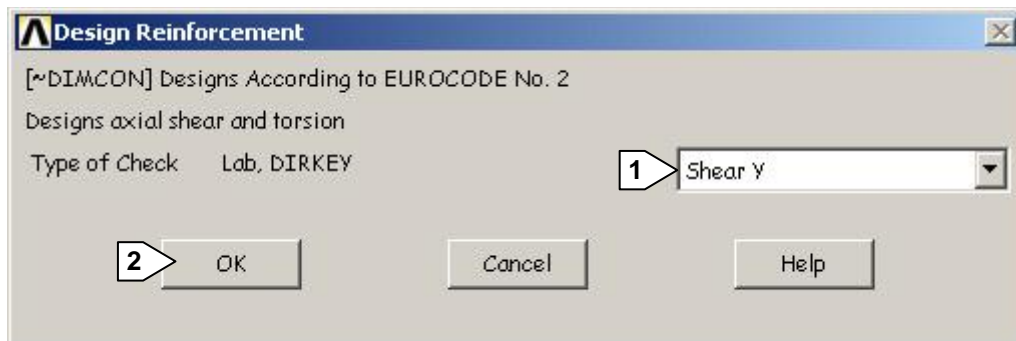
23. Design for shear force in Y axis

Now, we are going to design for shear force in Y to check if all elements satisfy the active code criteria.

Main Menu: - CivilFEM – **Civil Postprocessor** → **Code Checking** → **Eurocode 2** → **DESIGN BY CODE**: - Beams & Solid → **Shear & Torsion**

1 Select shear Y as type of check

2 OK



24. Review designed reinforcement for shear force

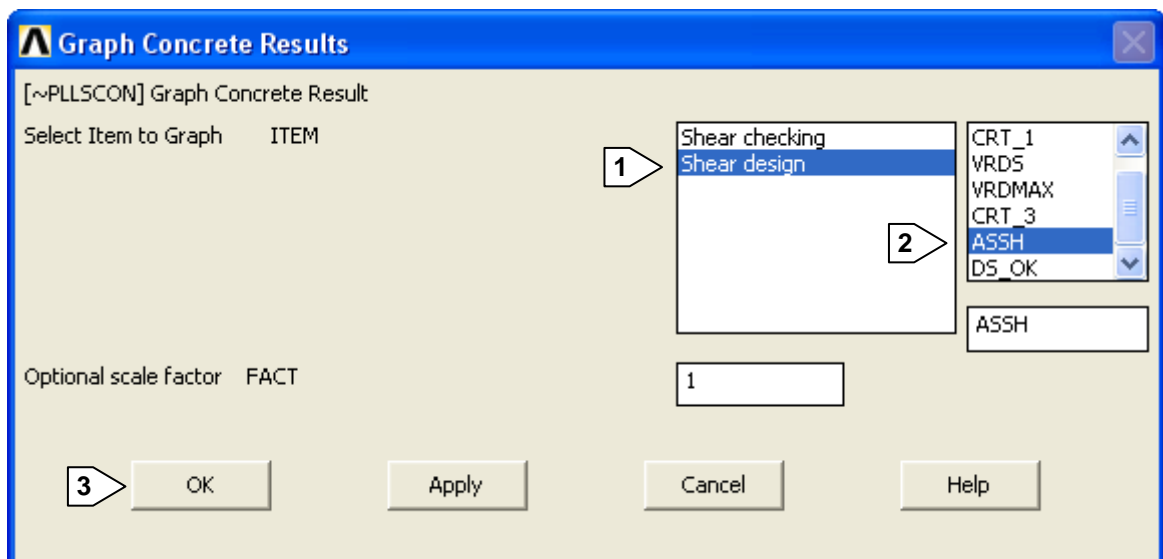
We are going to plot the designed reinforcement for shear force.

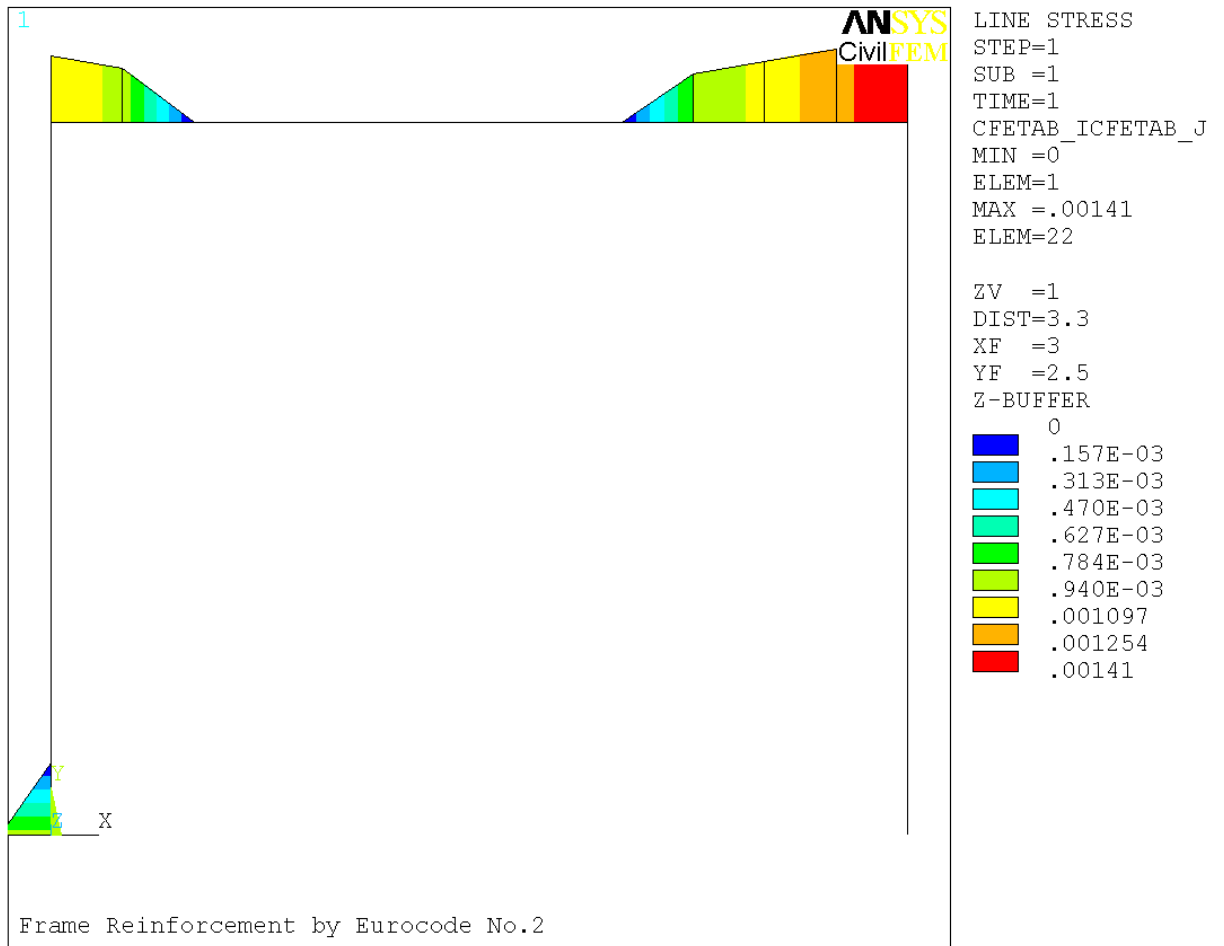
Main Menu: - CivilFEM – **Civil Postprocessor** → **Code Checking** → **Eurocode 2** → **BEAM RESULTS: Plot Results**

1 Choose Shear Design

2 Choose Area Rnf/Length

3 OK





25. Exit the ANSYS program

ANSYS Toolbar: **QUIT**

1 Pick on Save Everything option

2 OK

