

## 9. Check by EHE a Prestressed Slab Bridge

**Applicable CivilFEM Product:** All CivilFEM Products

**Level of Difficulty:** Moderate

**Interactive Time Required:** 90 minutes

**Discipline:** Prestressed Bridge

**Analysis Type:** Linear static

**Element Type Used:** Beam 44

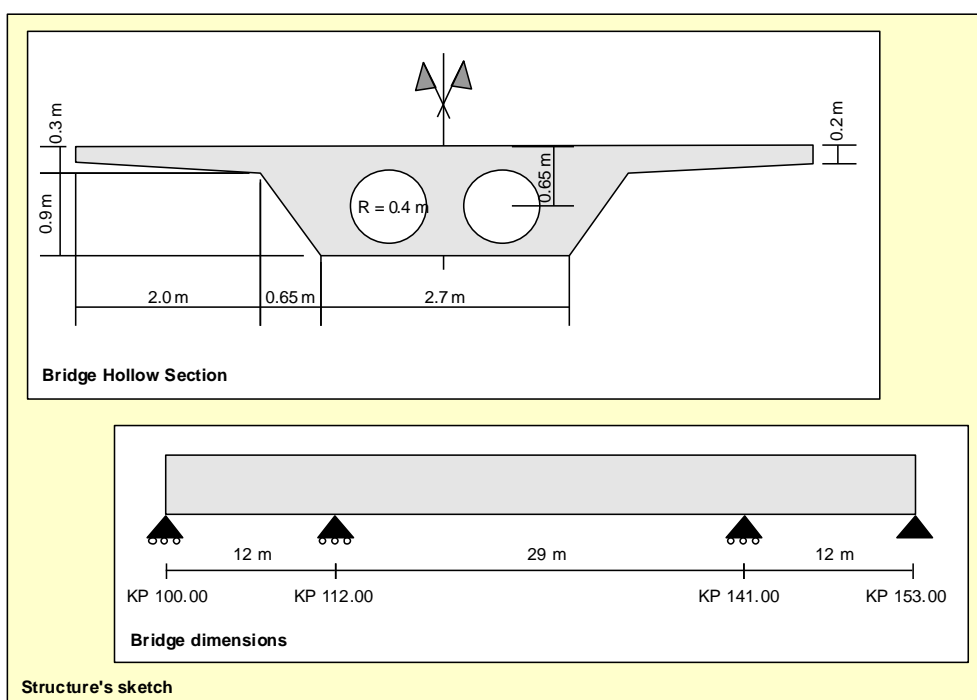
**Active Code:** EHE

**Units System:** N, m, s

**CivilFEM Features Demonstrated:** Bridge, Prestressed Concrete and Combination Modules, axial and bending reinforcement checking, cracking checking

### Problem Description

This is an advanced CivilFEM analysis, the basic idea consists in generating, from the cross section and main axis definition in plan and elevation views, the structure of the bridge. Tendon definition and all prestressing effects in the structure will be studied as well.



## ■ Given

The straight bridge given will have two cross sections: one solid for supports and the other with 2 holes for spans. The bridge is 53 m long divided in 3 spans, a central one of 29 m and the other two of 12 m and will be prestressed with 6 tendons.

The load distribution, material properties, cable definition, cracking properties and safety factors for concrete and steel reinforcement are the following:

- Mobile Load: One heavy vehicle in one line (Heavy Vehicle IAP)
- Self Weight
- Dead Load (along all deck): 33200 N/m<sup>2</sup>
- Surface load (along all deck): 40000 N/m<sup>2</sup>
- Initial prestressing (t = 0 days) and final (t = 10<sup>6</sup> days)
- Thermal gradient of 5°C

These loads will be combined to obtain the most unfavourable case according EHE criteria.

LOADS	γ COEFFICIENTS	
	Favourable effect	Unfavourable effect
Vehicle	0.90	1.10
Initial Prestressing	0.90	1.10
Final Prestressing	0.90	1.10
Self Weight	1.00	1.35
Dead Load	1.00	1.35
Surface Load	0.00	1.50
Thermal Gradient	1.00	1.35

Materials:

Concrete	HA-35 (EHE code)
Prestressing Steel	Y1860 S7 (EHE code)

There are two reinforcement groups

Group	1
Reinforcement material	Material 2 (S 500)
Reinforcement class	Scalable
Diameter	20 mm
Bar spacing	0.1992 m
Face	Bottom
Geometrical cover	0.050 m

Group	2
Reinforcement material	Material 2 (S 500)
Reinforcement class	Scalable
Diameter	20 mm
Bar spacing	0.201538 m
Face	Top
Geometrical cover	0.050 m

Cracking properties:

Bar spacing	20 mm
$\rho$ (effective reinforcement ratio)	104.2 (top fiber)
	212.4 (bottom fiber)

For prestressing losses calculation and prestress concrete checking:

Friction coefficient	$\mu = 0.21$
Unintentional friction	$K = 0.0126 \text{ m}^{-1}$
Anchorage slip	$A = 5 \text{ mm}$
Tendon area	$21 \text{ cm}^2$
Casing diameter	9 cm
Post-prestressing method	Initial end

Tendons in plan view have a straight trajectory, parallel to deck, and a distance from bridge axis of:

Tendon	Distance
1	0.09 m
2	-0.09 m
3	1.10 m
4	-1.10 m
5	1.30 m
6	-1.30 m

(Positive is down from the bridge axis)

Tendons trajectory in elevation view is the following (referred to section axes)

#### Tendons 1, 3 & 4

x (m)	0.0	4.5	12.5	27	41.5	49.5	53.0
y (m)	-0.613	-0.613	-0.135	-1.065	-0.135	-0.613	-0.613
Slope	0.0	0.0	0.0	0.0	0.0	0.0	0.0
Distance ratio to the next inflection point	0.5	0.8	0.2	0.8	0.2	0.5	0.5

#### Tendons 2, 5 & 6

x (m)	0.0	4.5	12.5	27	41.5	49.5	53.0
y (m)	-0.212	-0.212	-0.135	-1.065	-0.135	-0.212	-0.212
Slope	0.0	0.0	0.0	0.0	0.0	0.0	0.0
Distance ratio to the next inflection point	0.5	0.8	0.2	0.8	0.2	0.5	0.5

Initial tendon tensile force will be 3125 kN, then decreasing to 2930 kN at anchorage.

## ■ Approach and Assumptions

This is a static analysis with 3D elastic elements and elastic material properties. Model geometry is defined with nodes and elements. The finite element model will be automatically generated with the bridge utilities.

## ■ Summary of Steps

### Preprocessing

1. Preprocessing
2. Specify title
3. Define the viewing direction.
4. Set code and units
5. Define material
6. Define element
7. Define bridge section
8. Bridge Layout Design
9. Bridge Solid Modelling
10. Define bending reinforcement properties
11. Beam Support
12. Define tendons

### Solution

13. Define Boundary conditions
14. Families and Load States Definition

### Postprocessing

15. Conversion of families into combinations
16. Define targets
17. Define combination rules
18. Carry out combinations
19. Read & Plot results

20. Calculation of the load distribution to obtain the minimum deflection at a node (all loads)
21. Calculation of the load distribution to obtain the minimum deflection at a node (vehicle only)
22. Check by code
23. Plot Results: Axial+Bending checking for target 1 (Maximum MZ)
24. Plot Results: Axial+Bending checking for target 2 (Minimum MZ)
25. Plot Results: Crack checking for target 1 (Maximum MZ)
26. Plot Results: Crack checking for target 2 (Minimum MZ)
27. Interaction Diagram
28. Exit the ANSYS program

## Interactive Step-by-Step Solution

### 1. Preprocessing

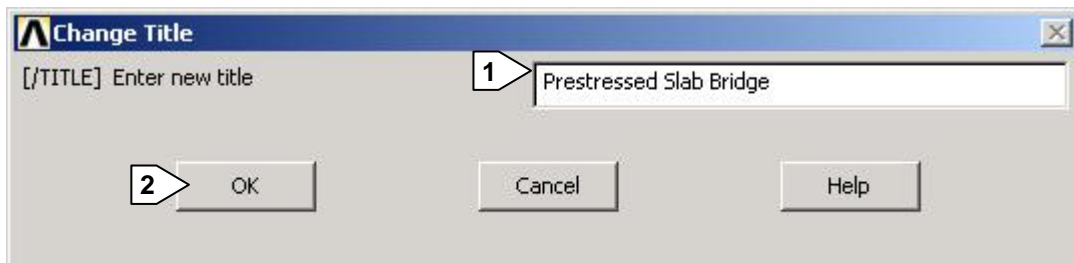
A typical CivilFEM analysis begins with providing data such as the units system, active code, materials, element types, section and model geometry definition.

### 2. Specify title

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

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

- 1 Enter the title: Prestressed Slab Bridge
- 2 OK to define the title and close the dialog box.

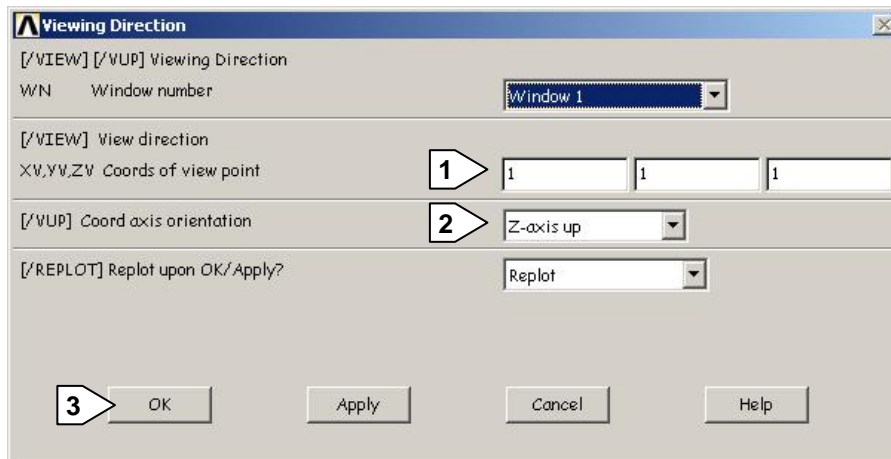


### 3. Define the viewing direction

We define the viewing direction for the display and specify the global Cartesian coordinate system reference orientation.

Utility Menu: **PlotCtrls** → **View Settings** → **Viewing Direction**

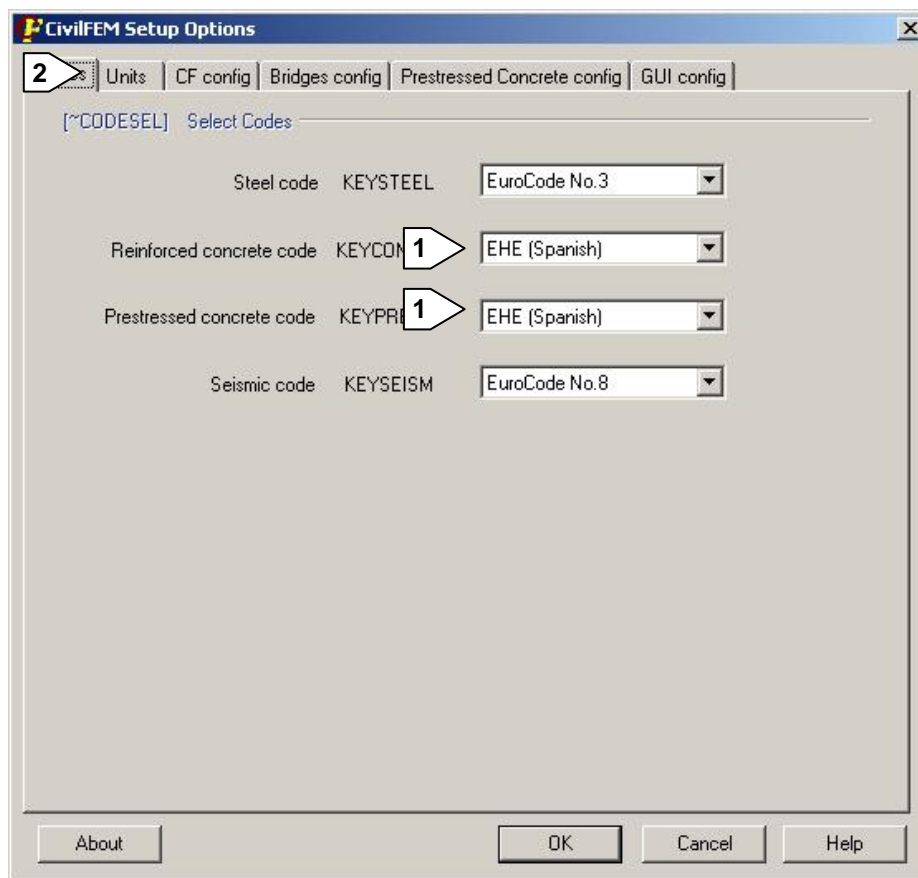
- 1 Set (1,1,1) as coordinates of point of view
- 2 Choose Z-axis up
- 3 Ok

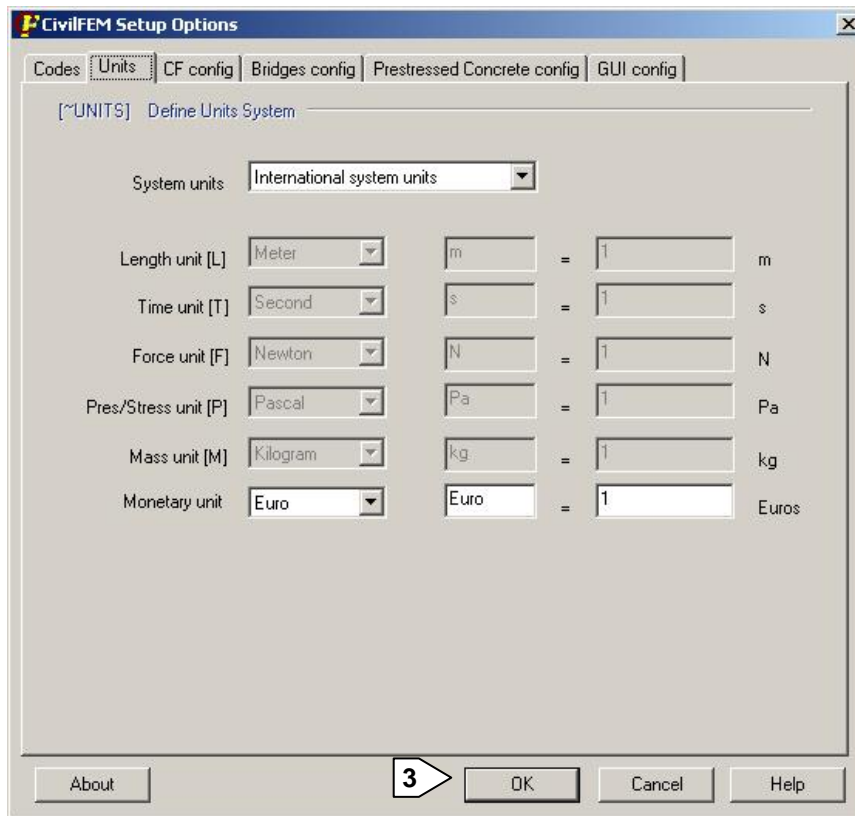


#### 4. Set code and units

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

- 1 Choose EHE for Reinforced and Prestressed Concrete Code
- 2 Pick on Units tab
- 3 OK to set active code and close the code dialog box

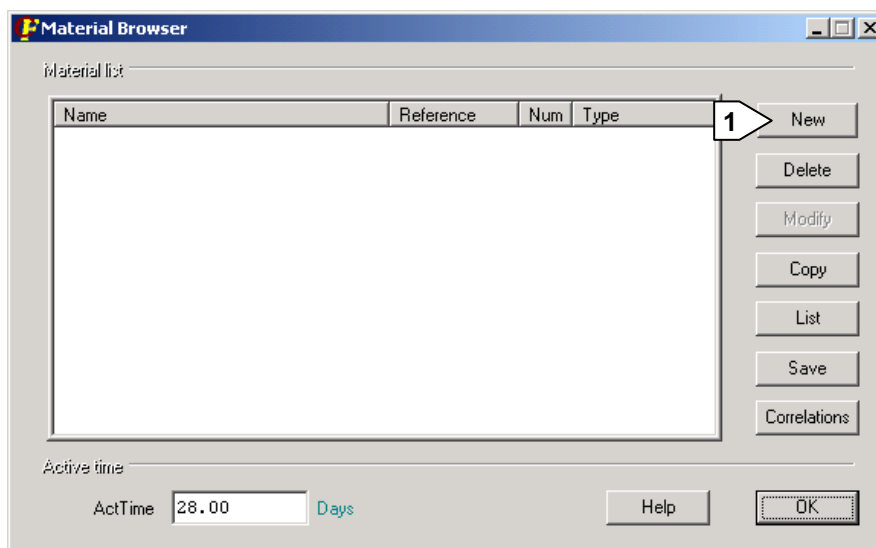


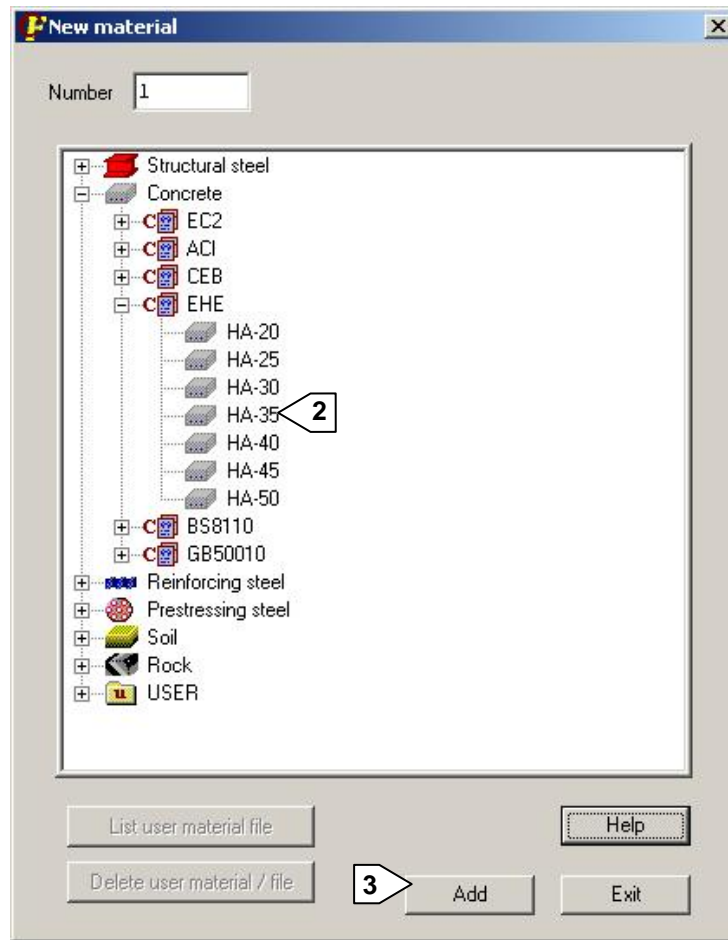


## 5. Define material

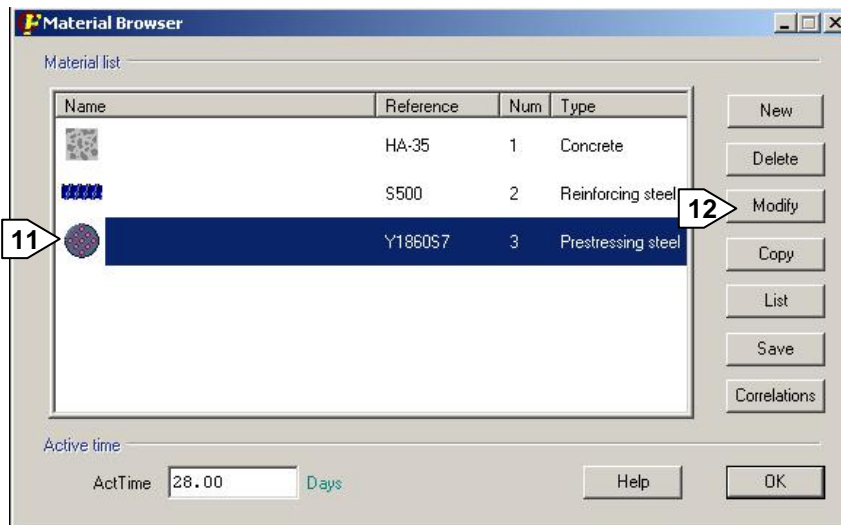
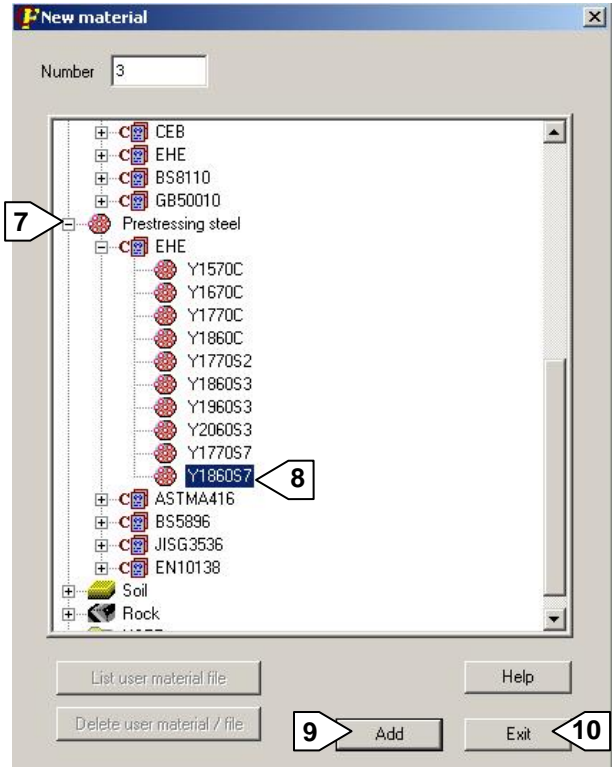
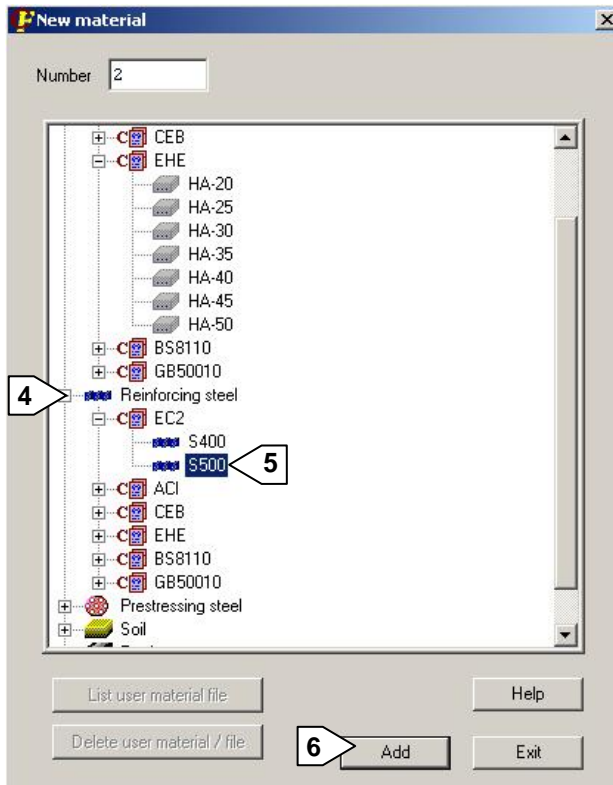
Main Menu: – CivilFEM – **Civil Preprocessor** → **Materials**

- 1 Pick New to define a new material
- 2 Choose EHE, then HA-35 concrete
- 3 Add to define material 1

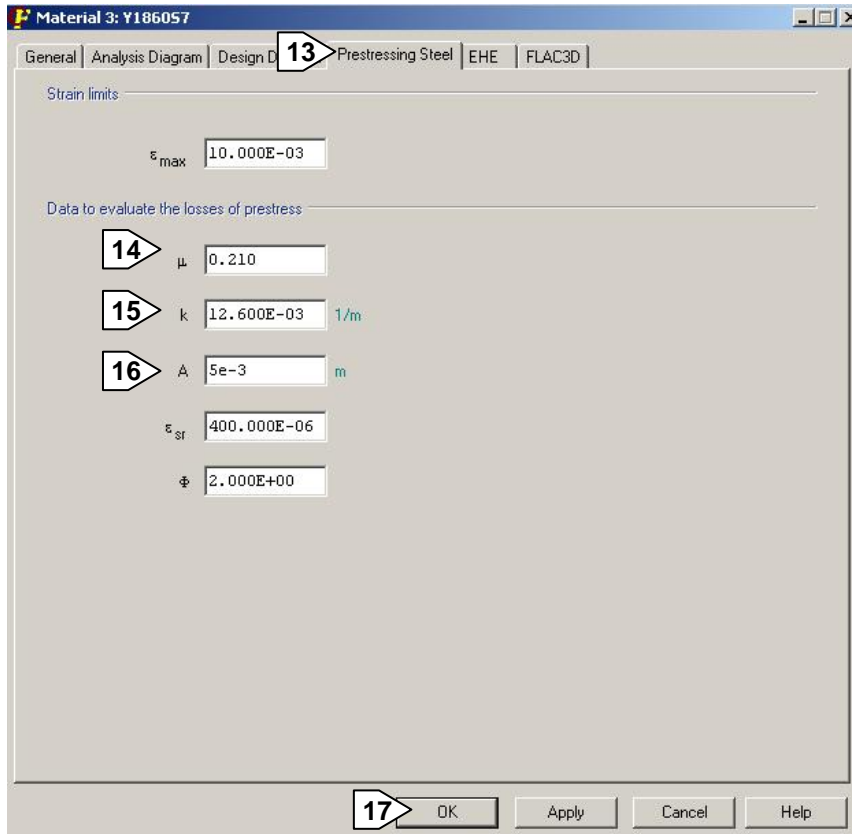




- 4 Select Reinforcing steel material type
- 5 Choose EC2: S500 reinforcing steel
- 6 Add to define material 2
- 7 Select Prestressing Steel
- 8 Choose EHE: Y1860S7
- 9 Add to define material 3
- 10 Exit
- 11 Select material 3
- 12 Modify



- 13 Pick on Prestressing Steel tab
- 14 Enter 0.21 as Friction coefficient
- 15 Enter 12.60e-03 as Unintentional friction
- 16 Enter 5e-3 as anchorage slip
- 17 OK

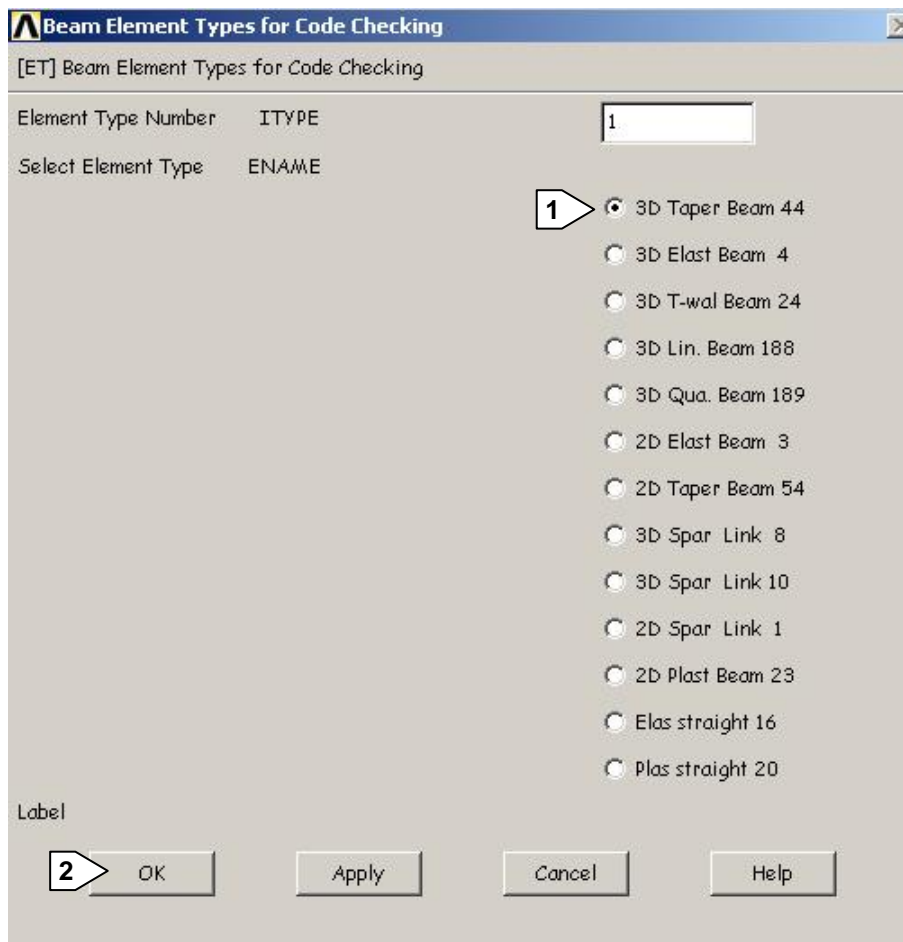


## 6. Define element type

We will use 3D Taper Beam 44 for this analysis.

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

- 1 Select Element Type 3D Taper Beam 44
- 2 Choose OK



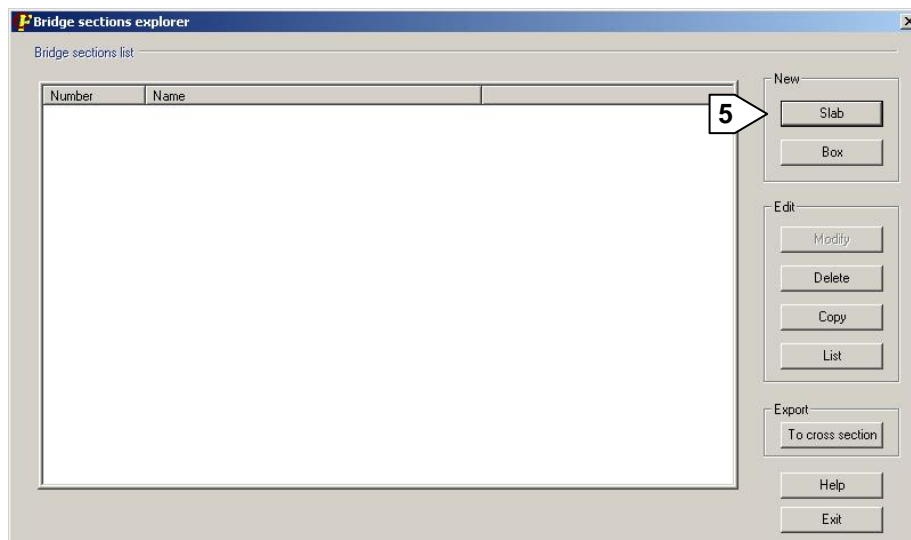
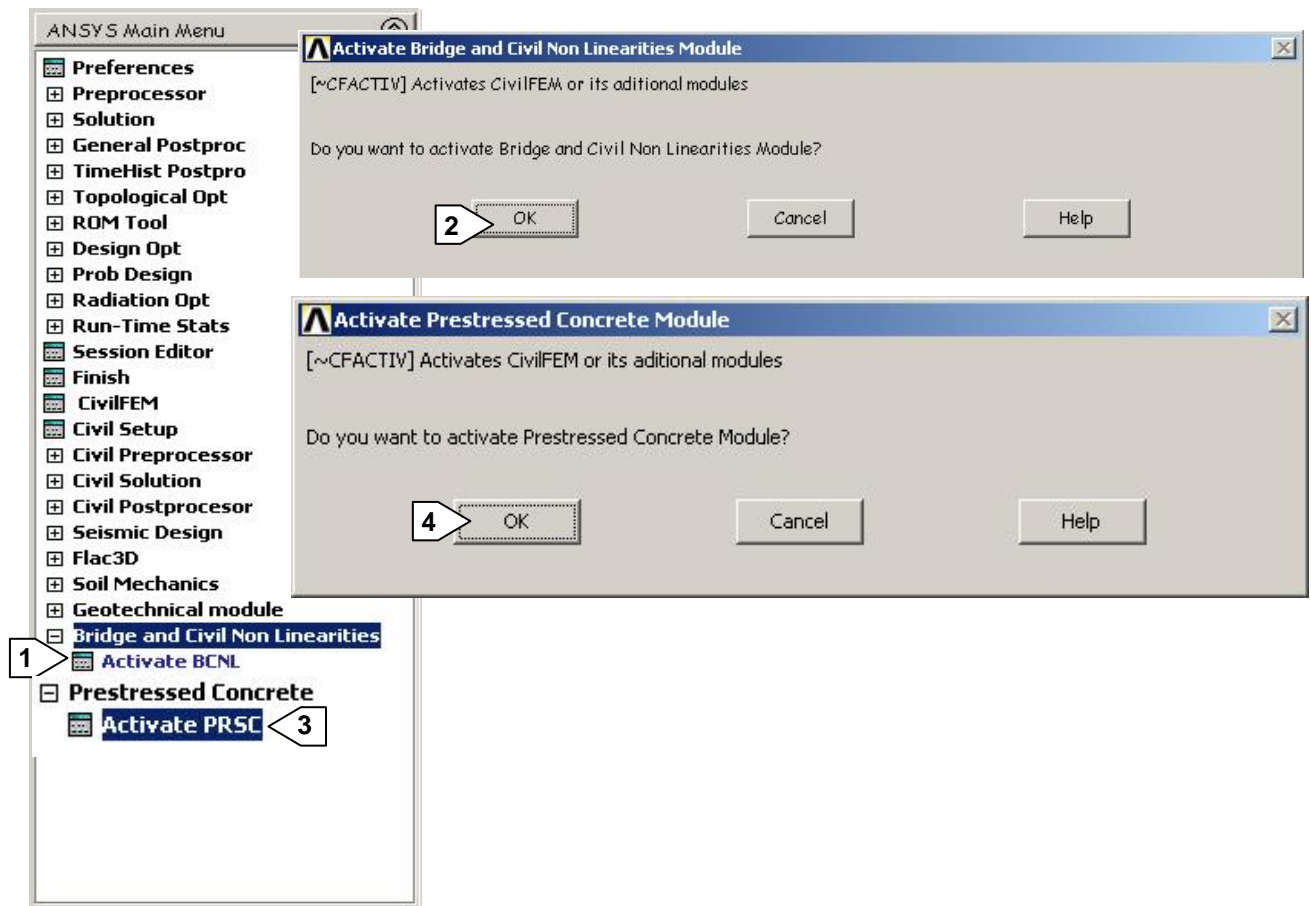
## 7. Define bridge section

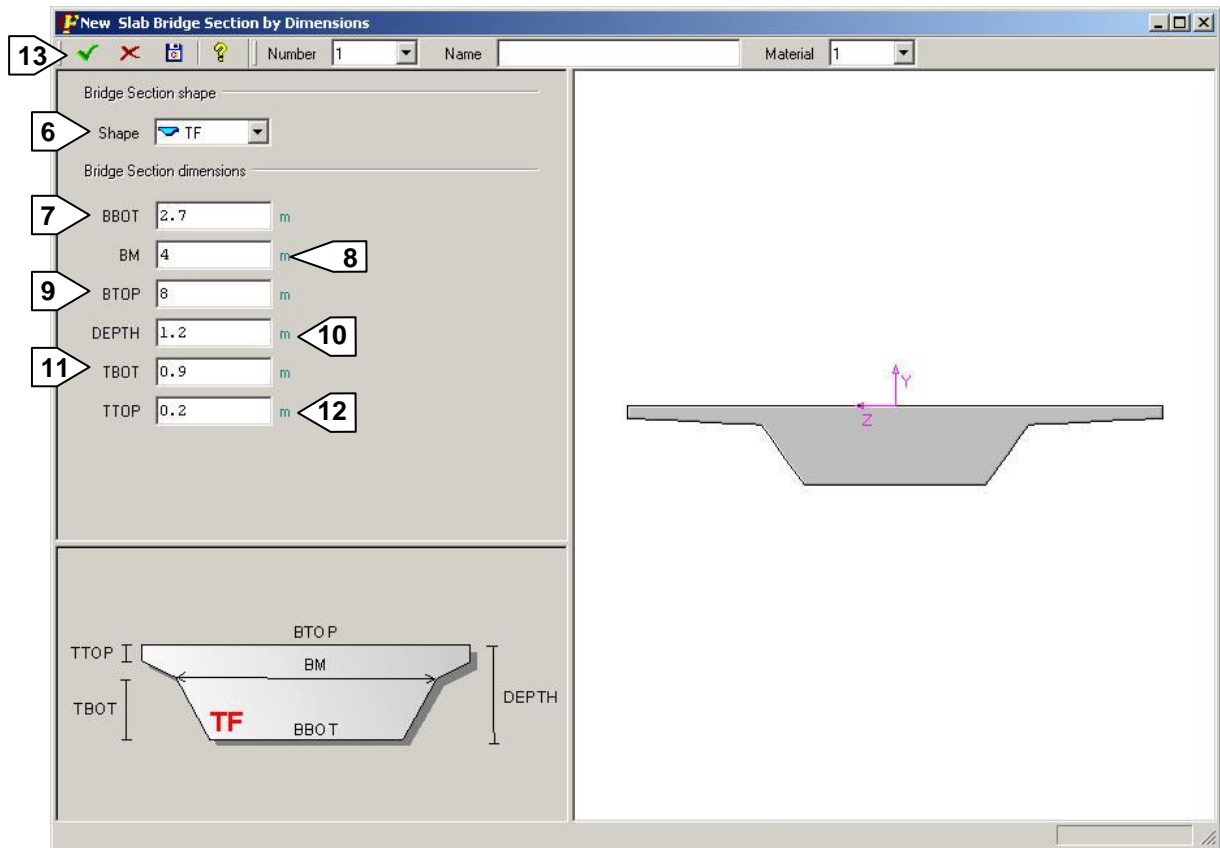
- 1 First we activate the Bridge and Civil Non Linearities Module
- 2 OK to activate
- 3 Then we activate the Prestressed Concrete Module
- 4 OK to activate

We will define a slab bridge cross section:

Main Menu: – CivilFEM – **Civil Preprocessor** → **Bridges Prep** → **Bridge Sections**

- 5 Choose New Slab
- 6 Select a TF shape
- 7 Enter Bottom Width: 2.7m
- 8 Enter Medium Width: 4m
- 9 Enter Top Width: 8m

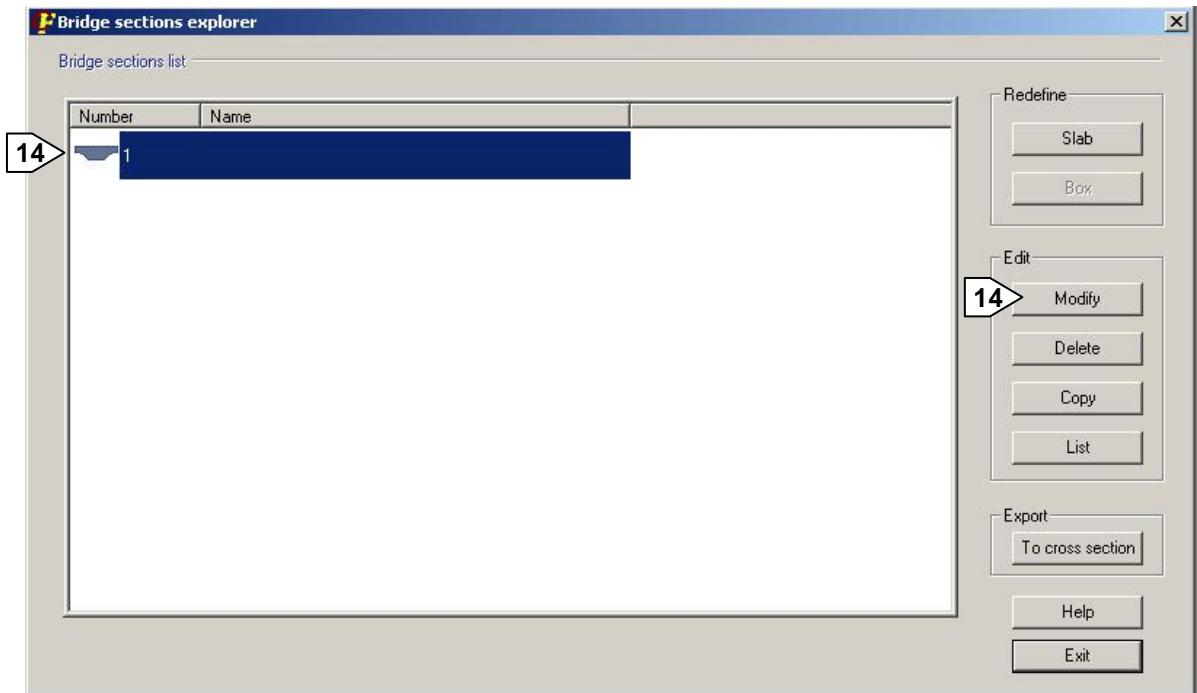




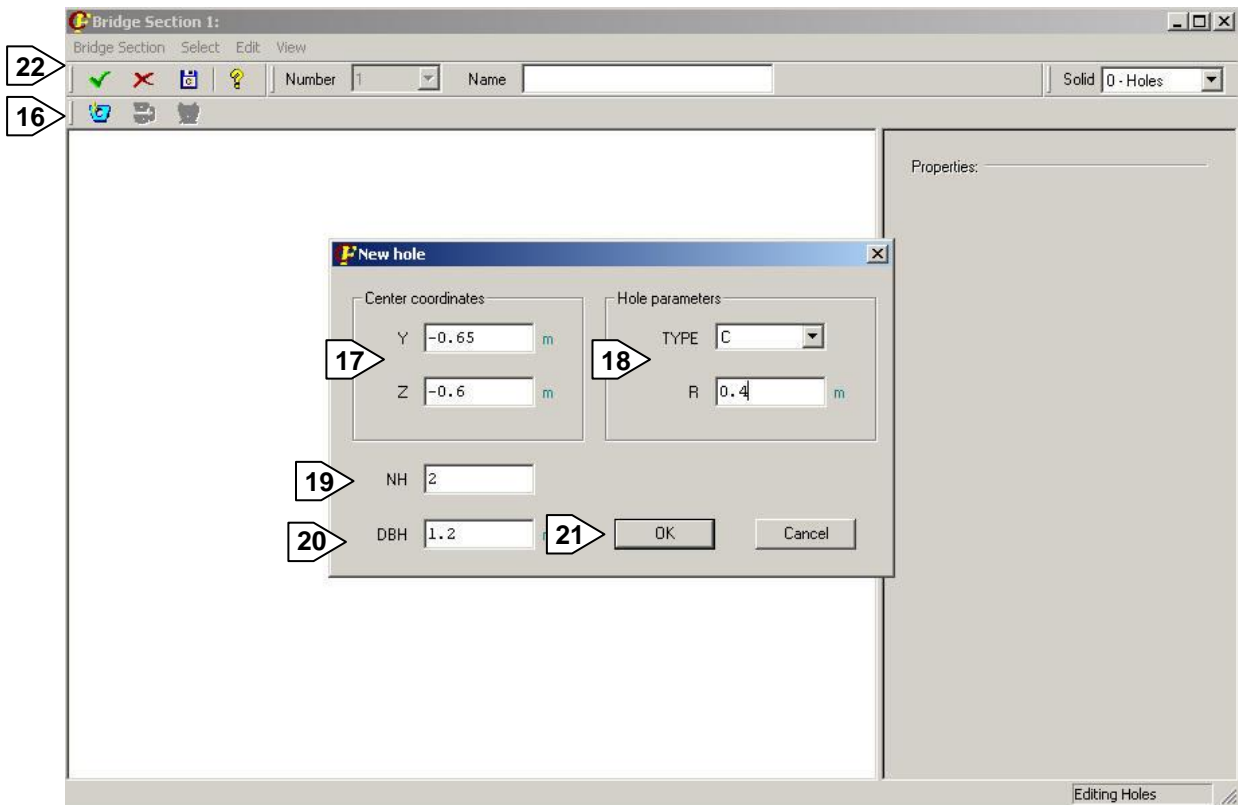
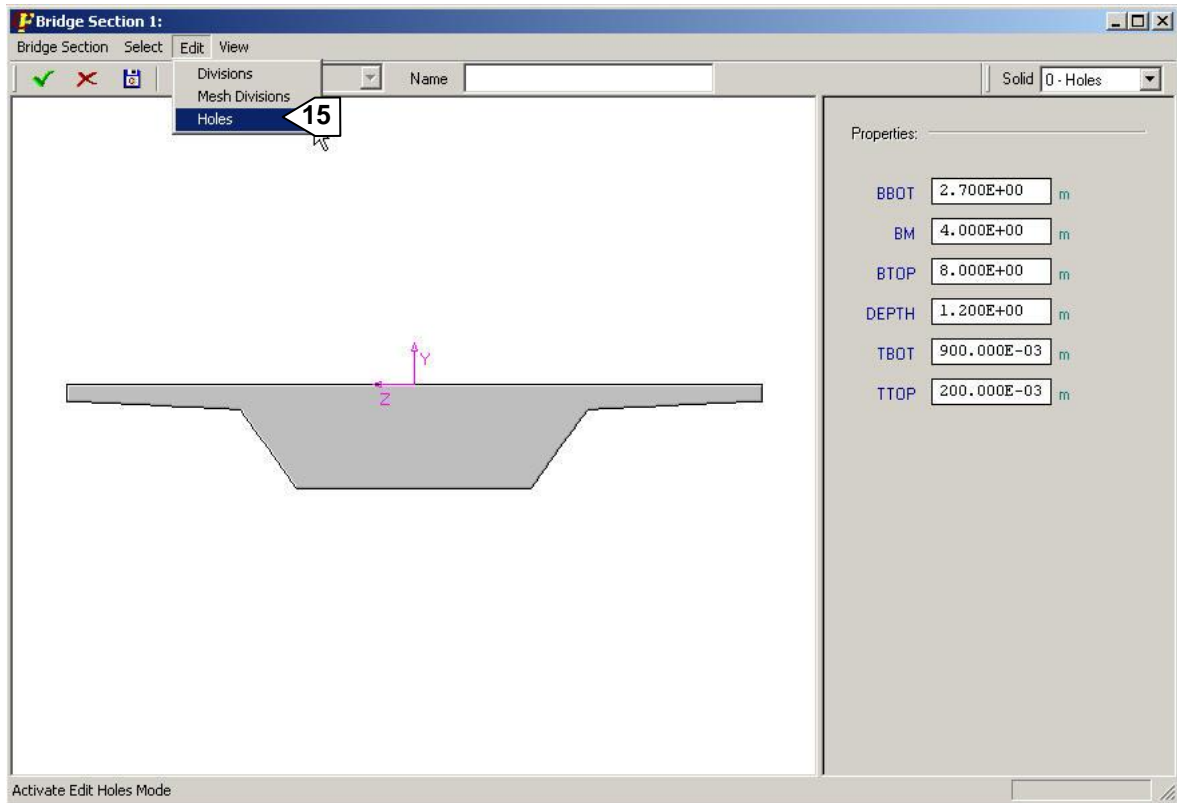
- 10 > Enter Total Depth: 1.2m
- 11 > Enter Bottom Thickness: 0.9m
- 12 > Enter Top Thickness: 0.2m
- 13 > Ok

Now we will modify the slab section to define 2 hollows:

- 14 > Pick Bridge Section 1 and Modify



- 15 Go to Edit Holes
- 16 Pick on New Hole
- 17 Enter Hole Center Coordinates:  $Y = -0.65\text{m}$ ,  $Z = -0.6\text{m}$
- 18 Choose Type C and Radius  $R = 0.4\text{m}$
- 19 Enter Number of Holes:  $NH = 2$
- 20 Enter Distance between Holes:  $DBH = 1.2\text{m}$
- 21 Ok to finish Holes definition
- 22 OK to finish th Bridge Section definition

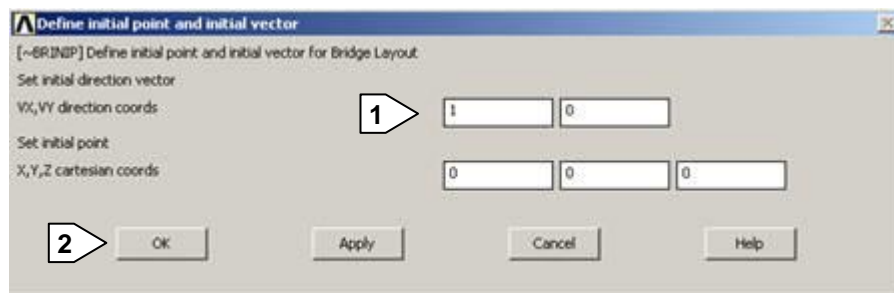


## 8. Bridge Layout Design

CivilFEM command **~BRINIP** will be used to define the Mileage point and Ansys direction from which the bridge model is generated:

Main Menu: – CivilFEM – **Civil Preprocessor** → **Bridges Prep** → **Layout**  
→ **Initial Point**

1 Enter (1,0) as global cartesian coordinates for the direction vector



2 Ok

We will define solid sections at both sides of each support, at a distance of 0.5m, therefore we define the bridge stretch starting at MP=99.5.

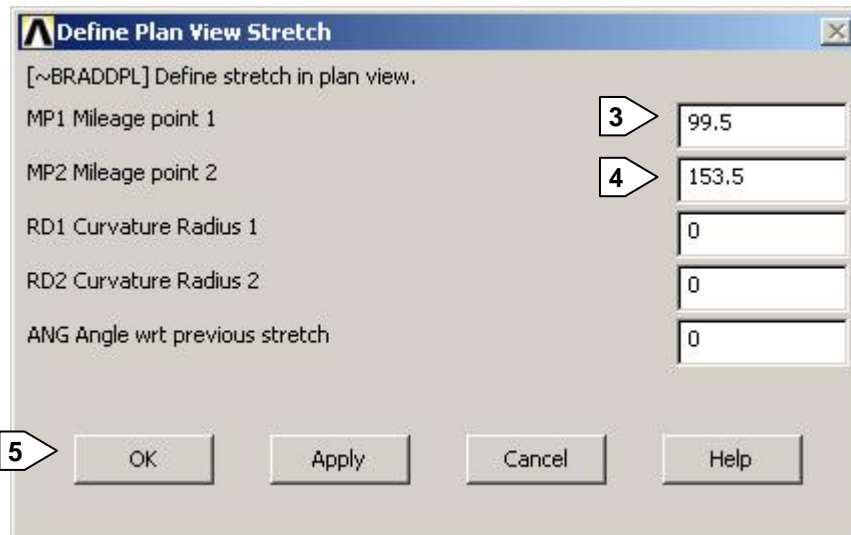
Now we define the bridge layout in plan view:

Main Menu: – CivilFEM – **Civil Preprocessor** → **Bridges Prep** → **P view generation** → **Define Plan View**

3 Enter MP1= 100 - 0.5

4 Enter MP2= 100 + 53 + 0.5

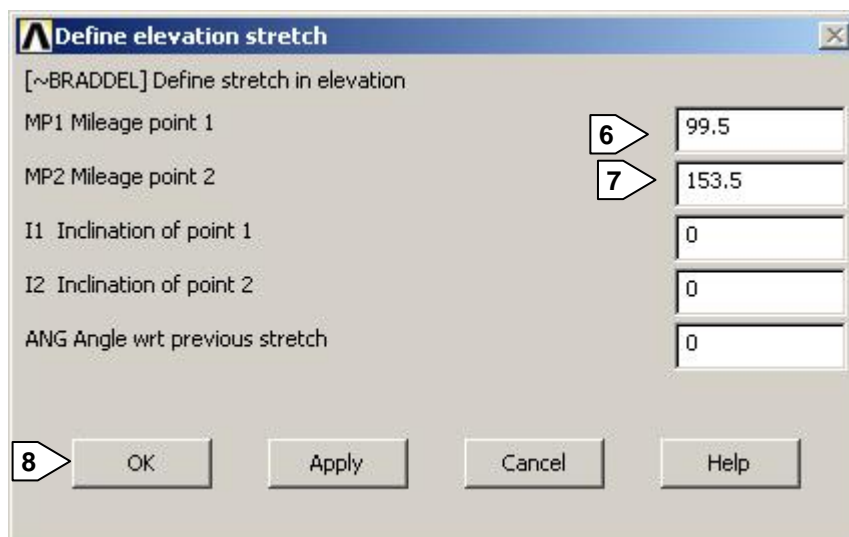
5 Ok



And now the same in elevation view:

**Main Menu: – CivilFEM – Civil Preprocessor → Bridges Prep → E view generation → Define Elevation View**

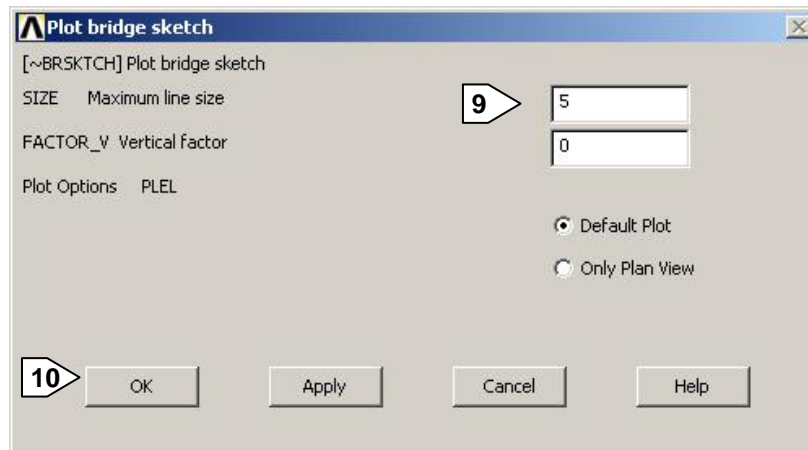
- 6 Enter MP1= 100 - 0.5
- 7 Enter MP2= 100 + 53 + 0.5
- 8 OK



And finally we plot the bridge axis:

**Main Menu: – CivilFEM – Civil Preprocessor → Bridges Prep → Layout → Plot sketch**

- 9 Enter 5 as maximum line size
- 10 Ok



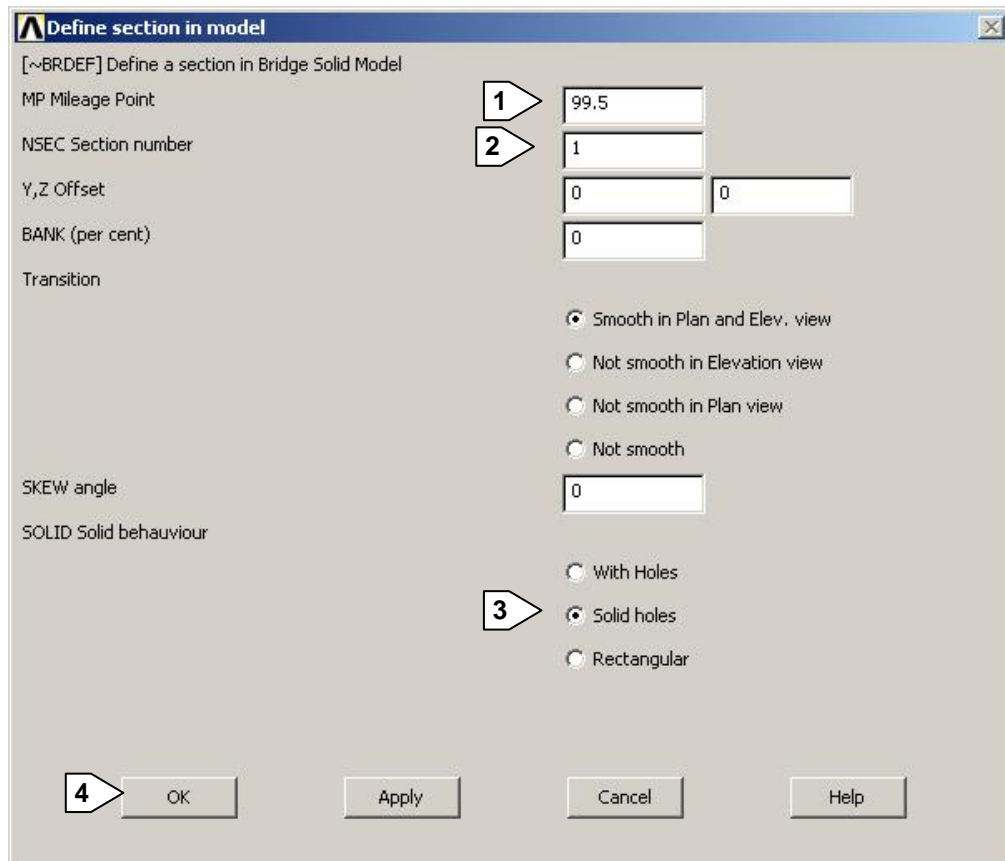
## 9. Bridge Solid Modelling

This utility allows generating the complete geometrical model of the structure as well as the finite element model from the cross sections definition (its location, “offsets”, banks, etc) by using either solid elements or beam elements. CivilFEM command **~BRDEF** will be used to define and assign attributes:

Main Menu: – CivilFEM – **Civil Preprocessor** → **Bridges Prep** → **Model Generation** → **Generate Section**

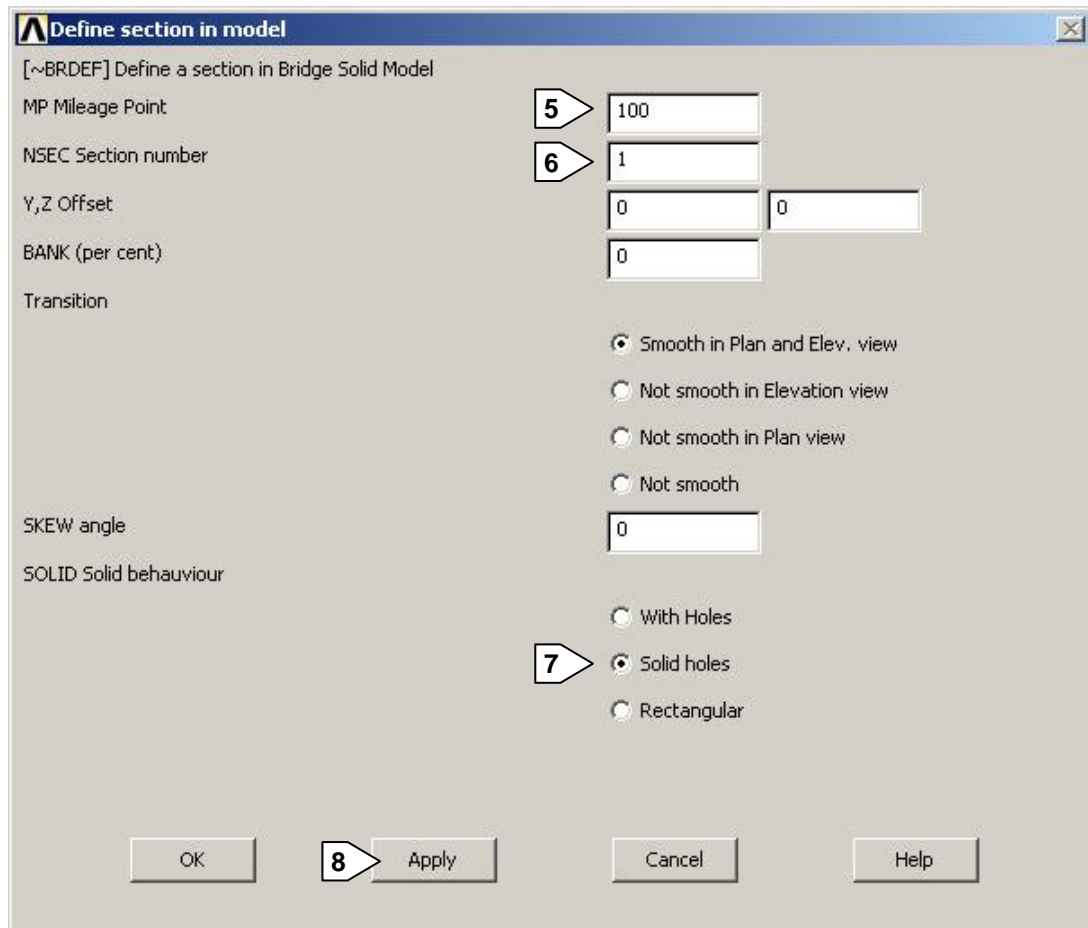
We start with the first support:

- 1 Enter Mileage Point: MP= 99.5
- 2 Enter Section number NSEC= 1



3 Specify Solid behaviour as Solid. This capability allows the user to define hollow or solid sections from a particular mileage point. It allows the user to consider hollow sections as solid section at particular points of the structure. In our case we will place solid sections at a distance of 0.5m of each support.

4 Apply to continue with the next cut



- 5 Enter Mileage Point: MP= 100
- 6 Enter Section number NSEC= 1
- 7 Specify Solid behaviour as Solid
- 8 Apply to continue with the next cut

We use the same procedure repeating steps 1 to 4 to define the rest of the cuts starting from MP= 100.5 of support 1 :

Support 1	
MP	SOLID
99.5	Solid
100	Solid
100.5	With Holes

Support 2	
MP	SOLID
111.5	Solid
112	Solid
112.5	With Holes

Center Span	
MP	SOLID
126.5	With holes

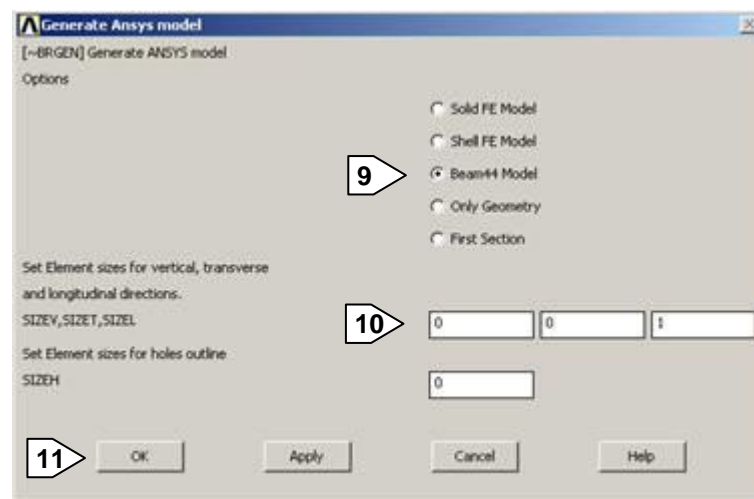
Support 3	
MP	SOLID
140.5	Solid
141	Solid
141.5	With holes

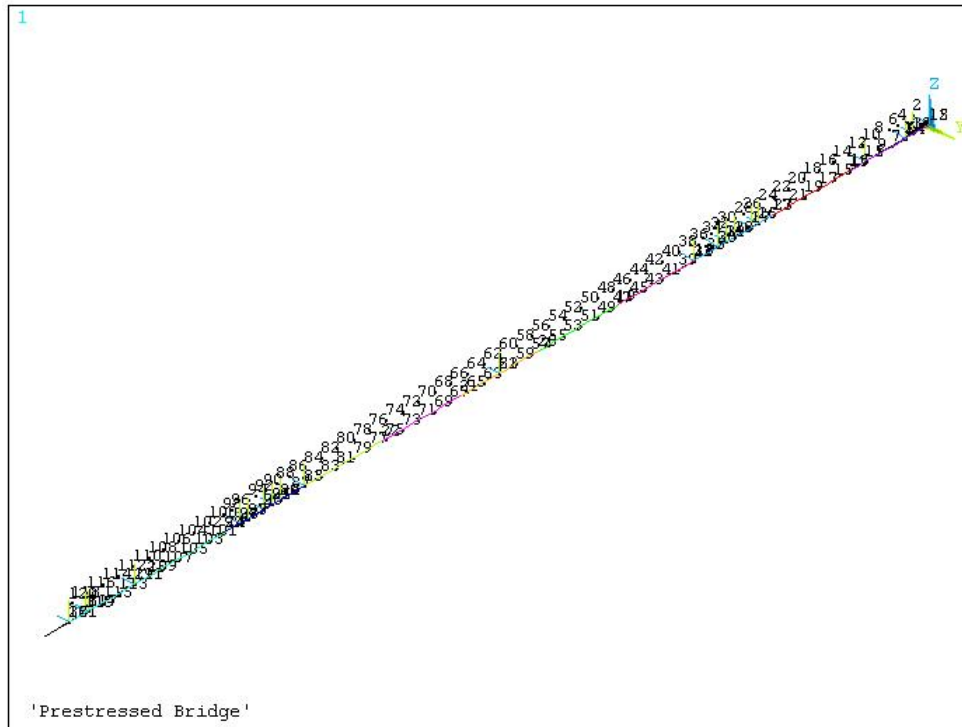
Support 4	
MP	SOLID
149	With Holes
152.5	Solid
153	Solid
153.5	Solid

The program creates a beam element model and automatically generates all cross sections and Beam&Shell properties needed.

Main Menu: – CivilFEM – **Civil Preprocessor** → **Bridges Prep** → **Model Generation** → **Generate Model**

- 9 Choose Beam44 Model
- 10 Enter size of longitudinal divisions SIZEL=1
- 11 Ok





## 10. Define bending reinforcement properties

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

- Number of reinforcement groups to be defined.
- Material of reinforcement group.
- Class of reinforcement group.
- Amount of reinforcement per group.
- Location of reinforcement group.

We will define two reinforcement groups for each of the 13 cross sections created after the bridge modeling.

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

Group	1
Reinforcement material	Material 2 (S 500)
Reinforcement class	Scalable
Diameter	20 mm
Bar spacing	0.1992 m
Face number	1 (bottom)
Geometrical cover	0.050 m

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

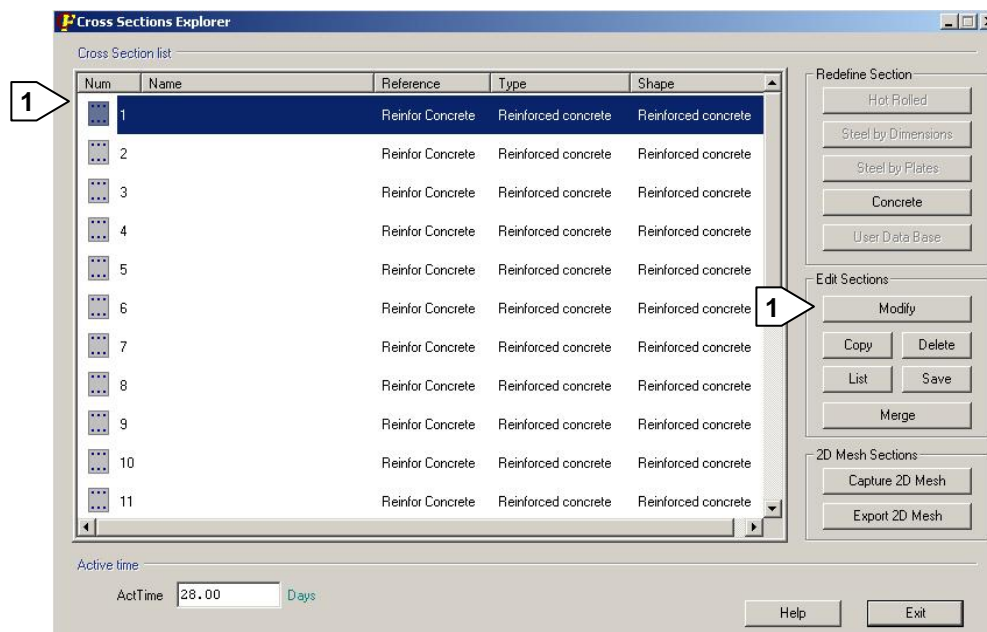
Group	2
Reinforcement material	Material 2 (S 500)
Reinforcement class	Scalable
Diameter	20 mm
Bar spacing	0.201538 m
Face number	2 (top)
Geometrical cover	0.050 m

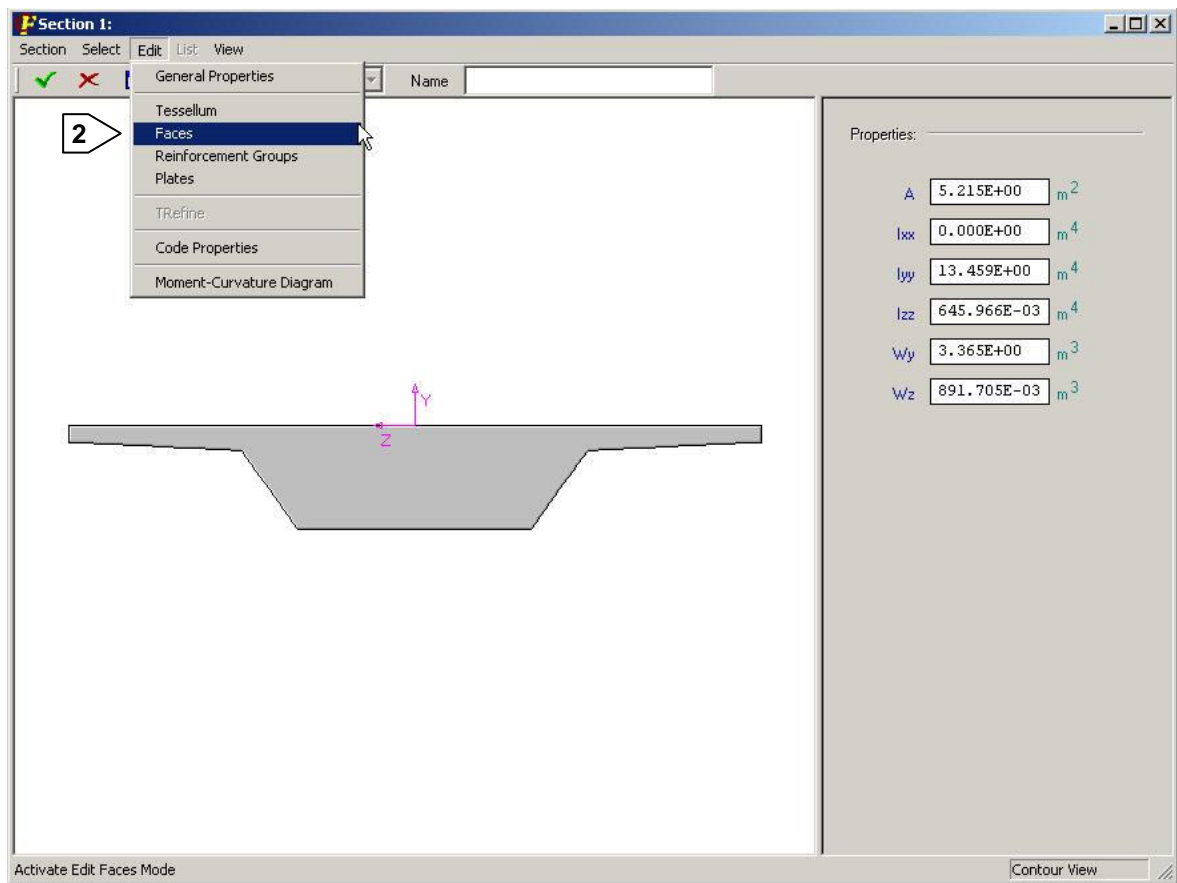
First we will define the faces and then the reinforcement groups for cross section 1:

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

1 Select section 1 and click on modify

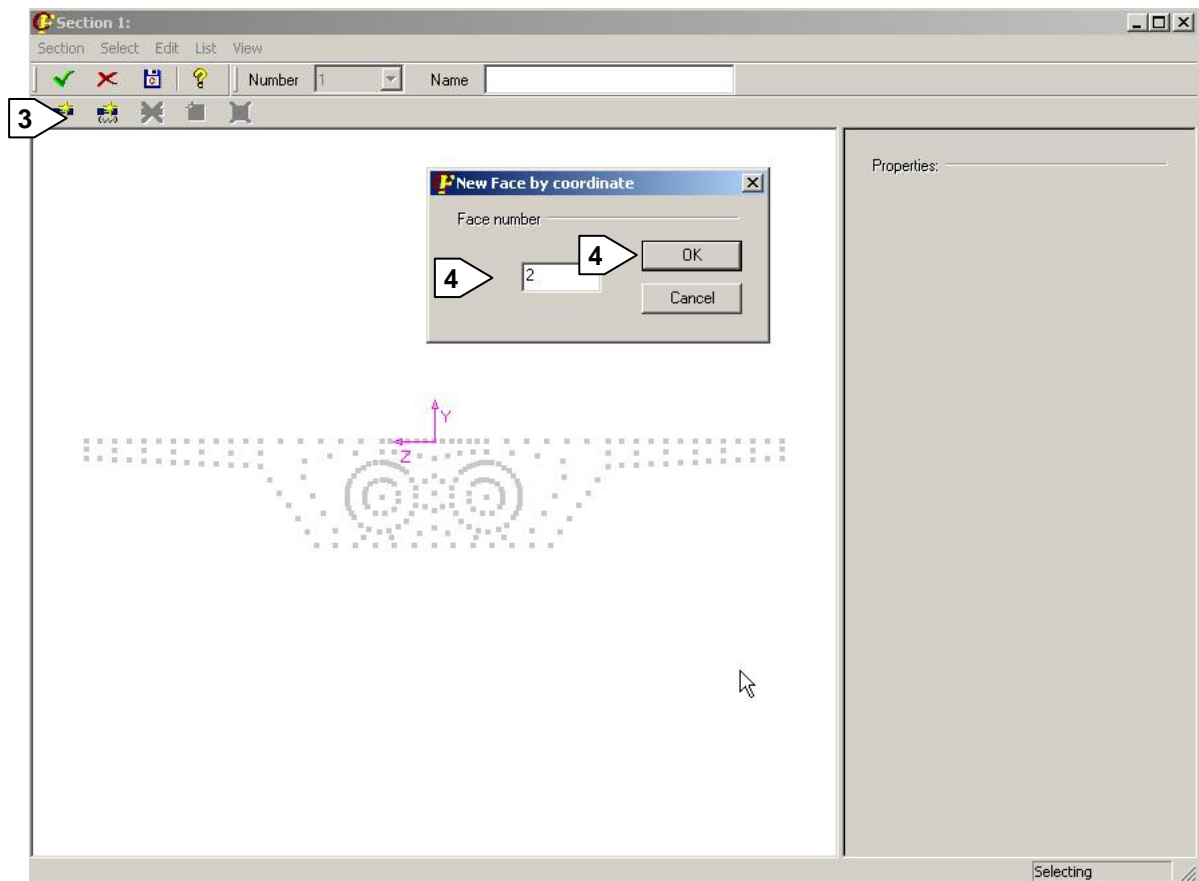
2 Edit Faces





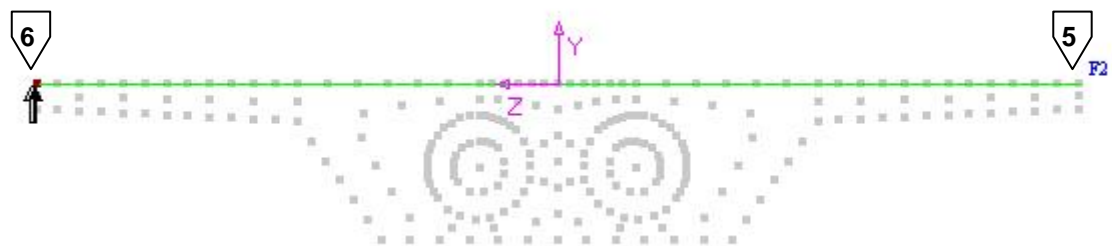
3 New Face by coordinates

4 Enter Face number 2 and Ok



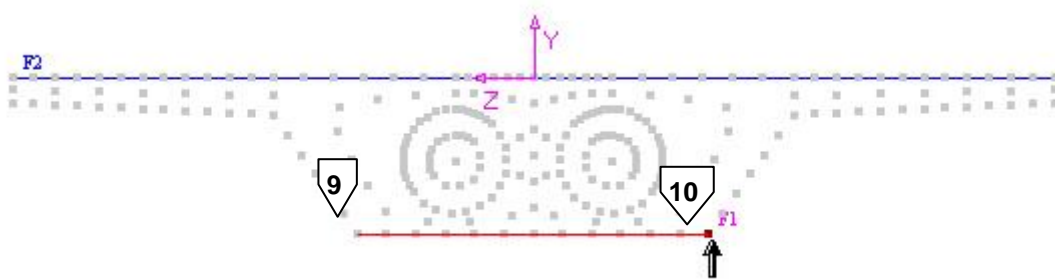
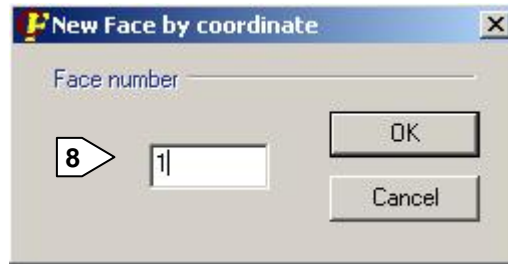
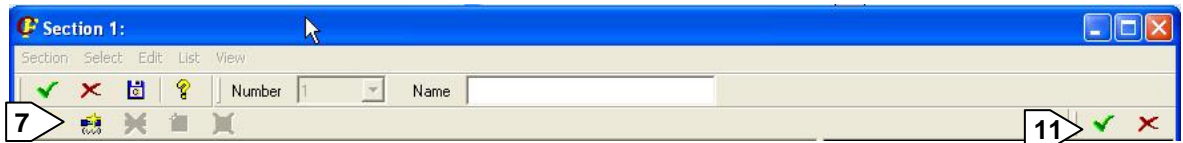
5 Pick on point 53

6 And then point 14 to define face 2

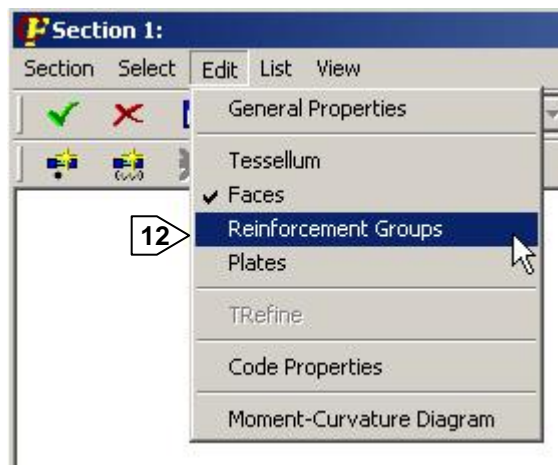


7 Now we define Face 1: New Face by coordinates

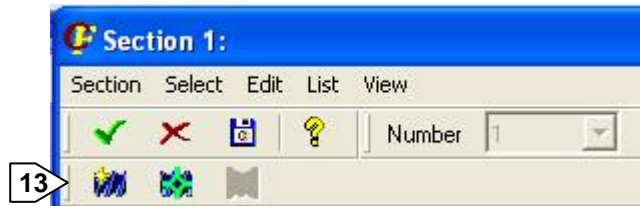
8 And enter number 1



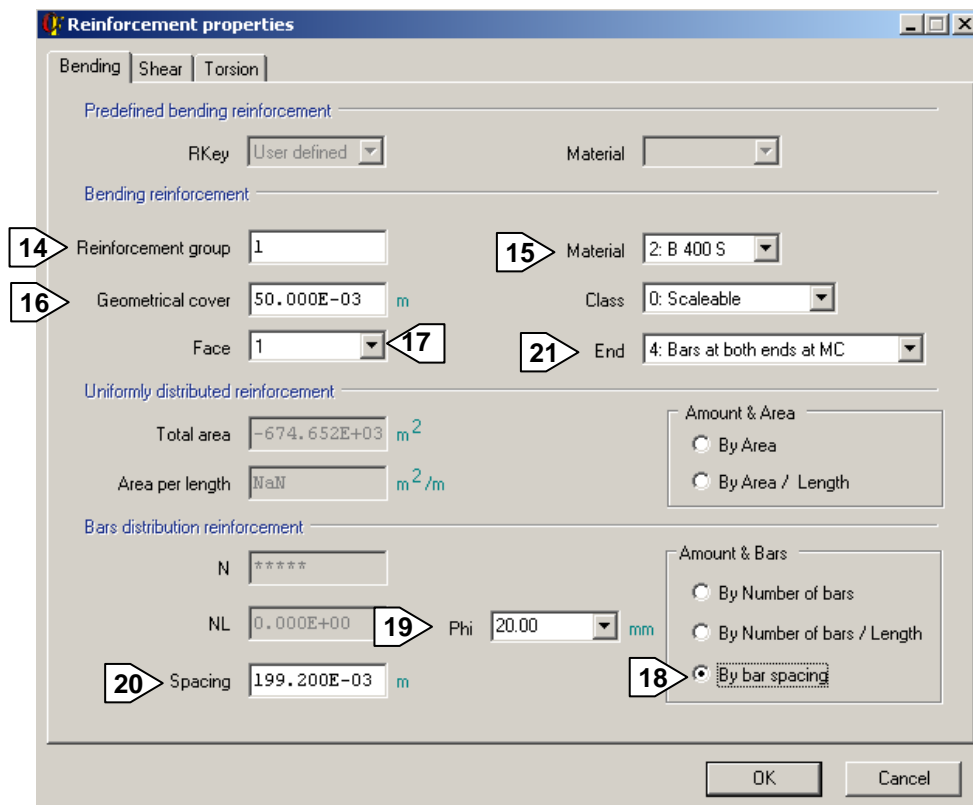
- 9 Pick on point 97
- 10 And then point 180 to define Face 1
- 11 Finish
- 12 Go now to Edit Reinforcement groups



- 13 New Reinforcement

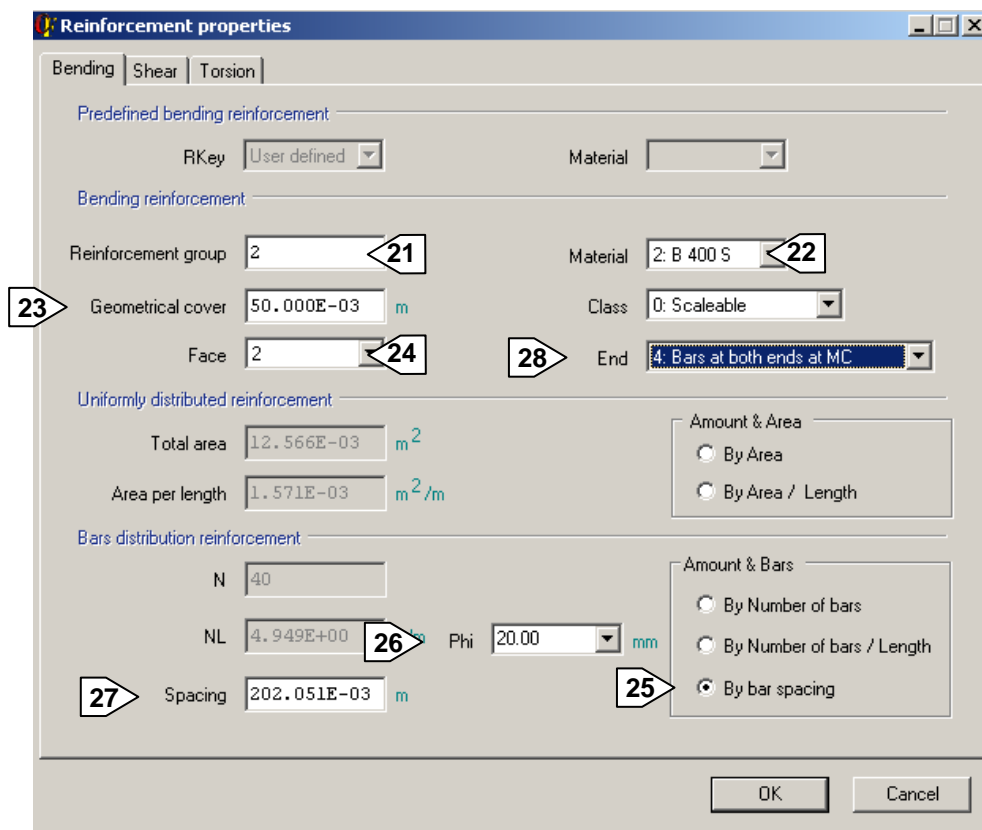


- 14 > Enter Bending Reinforcement Number 1
- 15 > Select Material 2
- 16 > Enter Geometrical cover  $G_c = 0.05$  m
- 17 > Choose Face 1
- 18 > Select Amount by bar spacing
- 19 > Enter diameter  $\Phi = 20$  mm
- 20 > Enter bar spacing  $S = 0.1992$  m
- 21 > Select 4:Bars at both ends at MC



Reinforcement group 1 created, now we define group 2 repeating the same process :

- 21 Enter Bending Reinforcement Number 2
- 22 Select Material 2
- 23 Enter Geometrical cover  $G_c = 0.05$  m
- 24 Choose Face 2
- 25 Select Amount by bar spacing
- 26 Enter diameter  $\Phi = 20$  mm
- 27 Enter bar spacing  $S = 0.201538$  m
- 28 Select 4:Bars at both ends at MC



Now we use the same procedure to define the same reinforcement groups to the rest of the 12 cross sections or we can use APDL commands:

Type this in Ansys Input Window:

**\*DO,I,2,13**

! Modifies the data of CivilFEM cross sections

**~SECMDF,I,FACES,PT\_ADD,LOC,,2,0,-4**

**~SECMDF,I,FACES,PT\_ADD,LOC,,2,0,4**

**~SECMDF,I,FACES,PT\_ADD,LOC,,1,-1.2,1.35**

~SECMDF,I,FACES,PT\_ADD,LOC,,1,-1.2,-1.35

! Defines a reinforcement group in cross sections

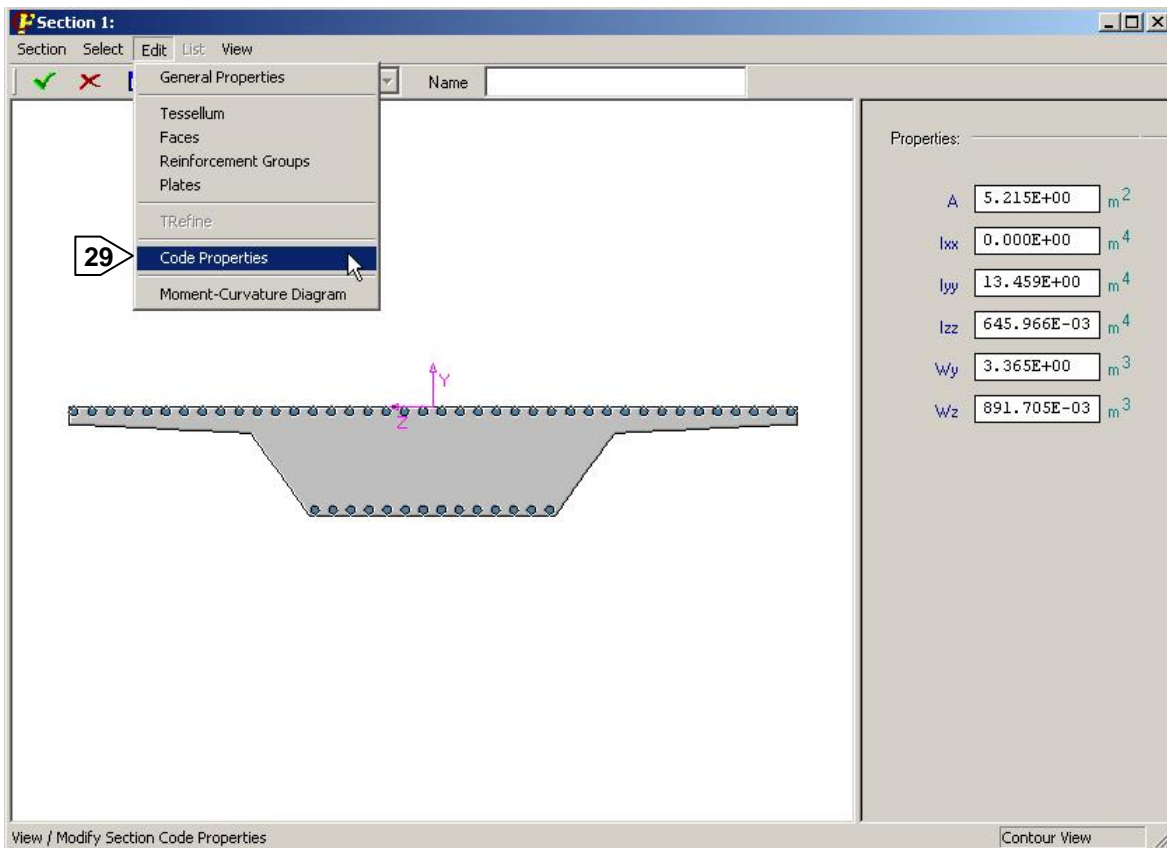
~RNFDEF,I,1,2,1,0,50.000E-03, , ,20.00, , ,199.200E-03,4

~RNFDEF,I,2,2,2,0,50.000E-03, , ,20.00, , ,201.538E-03,4

\*ENDDO

We will set the cracking properties by modifying again the cross section 1:

29 Edit Code Properties



- 30 Enter PHI YTOP= 20mm and PHI YBOT=20mm
- 31 Enter RHO YTOP=104.2 and RHO YBOT=212.4
- 32 Enter C YTOP=0.05 and C YBOT=0.05
- 33 Enter S YTOP=0.02 and S YBOT=0.02
- 34 Ok

	YTOP	YBOT	ZTOP	ZBOT	All
30 PHI (mm)	20.00	20.00	0.000	0.000	<input type="checkbox"/>
31 RHO	104.2	212.4	0.000	0.000	<input type="checkbox"/>
32 C	0.5000E-01	0.5000E-01	0.000	0.000	<input type="checkbox"/>
33 S	0.2000E-01	0.2000E-01	0.000	0.000	<input type="checkbox"/>

34 OK Cancel

Now we use the same procedure to define the same cracking properties to the rest of the 12 cross sections or we can use APDL commands (faster way).

Type this in Ansys Input Window:

```

*DO,1,2,13
  ~SECMDF, I,EHEPROP ,PHI    ,,20 ,1,0,0
  ~SECMDF, I,EHEPROP ,PHI    ,,20 ,2,0,0
  ~SECMDF, I,EHEPROP ,C      ,,5e-002,1,0,0
  ~SECMDF, I,EHEPROP ,C      ,,5e-002,2,0,0
  ~SECMDF, I,EHEPROP ,S      ,,0.2,1,0,0
  ~SECMDF, I,EHEPROP ,S      ,,0.2,2,0,0
  ~SECMDF, I,EHEPROP ,RHOR   ,,104.2,1,0,0
  ~SECMDF, I,EHEPROP ,RHOR   ,,212.4,2,0,0
*ENDDO

```

## 11. Support Beam

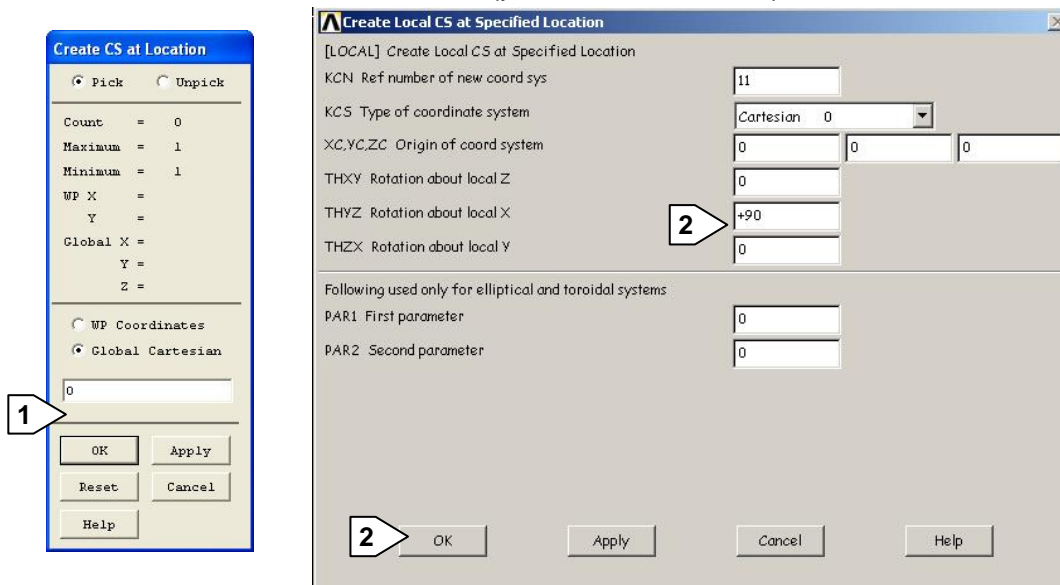
We must capture the support beam from the beam element model.

The support beam will be captured on the active coordinate system. The vertical direction must be oriented towards the Y axis. It is recommended to place the X axis along the main direction of the tendons, in order to avoid geometrical singularities (a tendon must not have any segment parallel to the Z axis). We first define a local coordinate system by a location and orientation:

Utility Menu: – WorkPlane – **Local Coordinate Systems** → **Create Local CS**  
→ **At Specified Loc +**

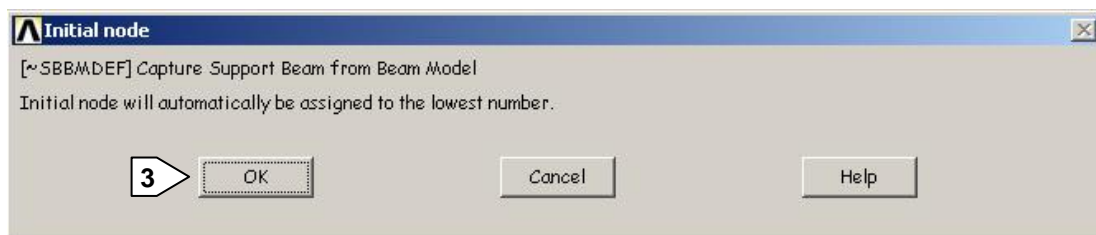
1 Enter 0 and Ok

2 Enter +90 rotation about local X (positive Y toward Z) and Ok



Main Menu: – CivilFEM – **Civil Preprocessor** → **Prestress Concrete Prep**  
→ **Beam model** → **Auto-Capture**

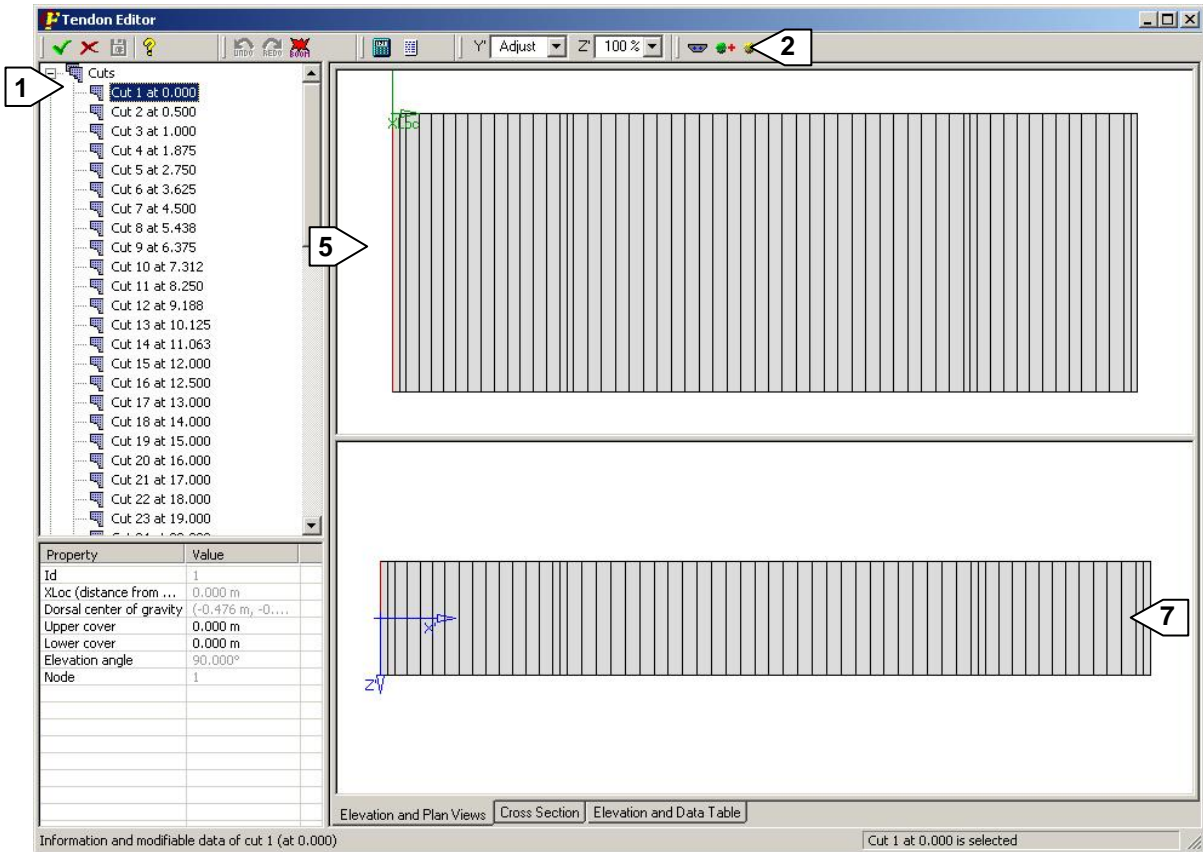
3 Ok to capture



## 12. Define tendons


To define a tendon we must define first all the control points in plan and elevation view. The Tendon Editor of CivilFEM will be very useful to us:

Main Menu: – CivilFEM – **Civil Preprocessor** → **Prestress Concrete Prep**  
→ **Tendons** → **Tendon Editor**



Point definition at the beginning (figure of problem description) is different from the one in tendon editor (cut position). This is because in tendon editor cut position begins in 0 but mileage points begin at -0.5.

We start with tendon points in plan view:

- 1 Pick on cut 1 (at 0.0)
- 2 Insert new plan view point 
- 3 Enter 0.09 as Zcoord for point 1
- 4 OK

Property	Value
Number ID	1
Cut Number	1
ZCoord	9e-002
Slope	0
Inflection distance ratio	0.500
Insert global plan point	
<input type="button" value="Ok"/> <input type="button" value="Cancel"/>	

5 Select again cut 1 and insert new plan viewpoint to define point 2

6 Enter -0.09 as Zcoord and OK

Property	Value
Number ID	2
Cut Number	1
ZCoord	-9e-002
Slope	0
Inflection distance ratio	0.500
Insert global plan point	Ok Cancel

6

We follow the same procedure to define the rest of points in cut 1:

Point 3: Cut 1, ZCoord= 1.1

Point 4: Cut 1, ZCoord= -1.1

Point 5: Cut 1, ZCoord= 1.3

Point 6: Cut 1, ZCoord= -1.3

7 Now we pick on cut 60 (last one) and we define the rest of plan view points:

Point 7: Cut 60, ZCoord= 0.09

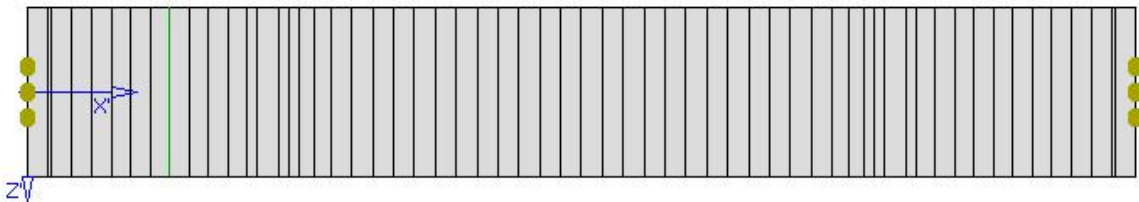
Point 9: Cut 60, ZCoord= -0.09

Point 11: Cut 60, ZCoord= 1.1

Point 13: Cut 60, ZCoord= -1.1

Point 15: Cut 60, ZCoord= 1.3

Point 17: Cut 60, ZCoord= -1.3

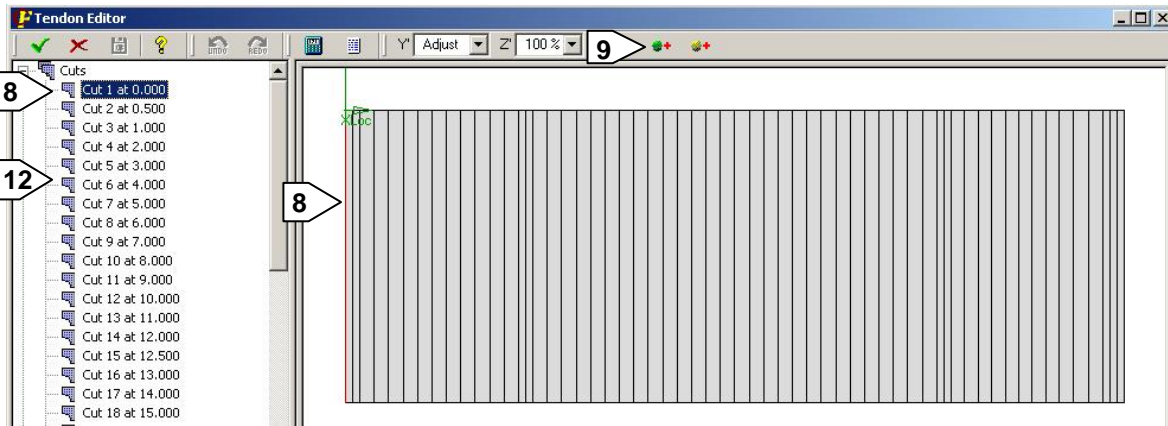


Now we have all plan view points defined, the next step is defining the tendon points in elevation view.

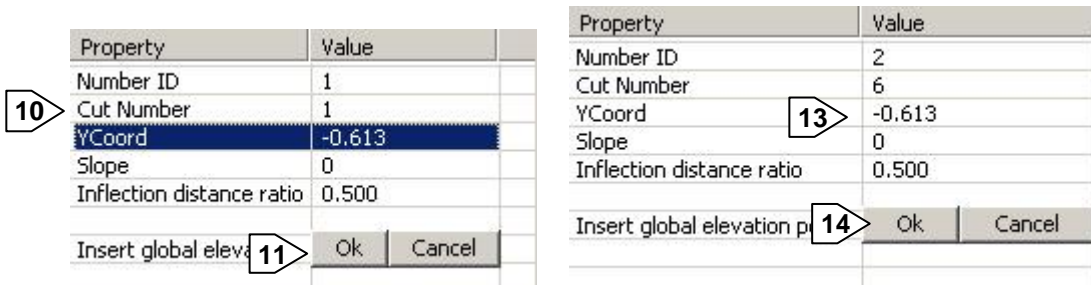
8 Pick again on cut 1(at 0.0) in elevation view

9 Insert new elevation point





- 10 Enter Ycoord= -0.613 m for point 1
- 11 OK
- 12 Select now cut 6 (at 4.0) and insert a new elevation view point to define point 2
- 13 Enter Ycoord= -0.613 m
- 14 OK

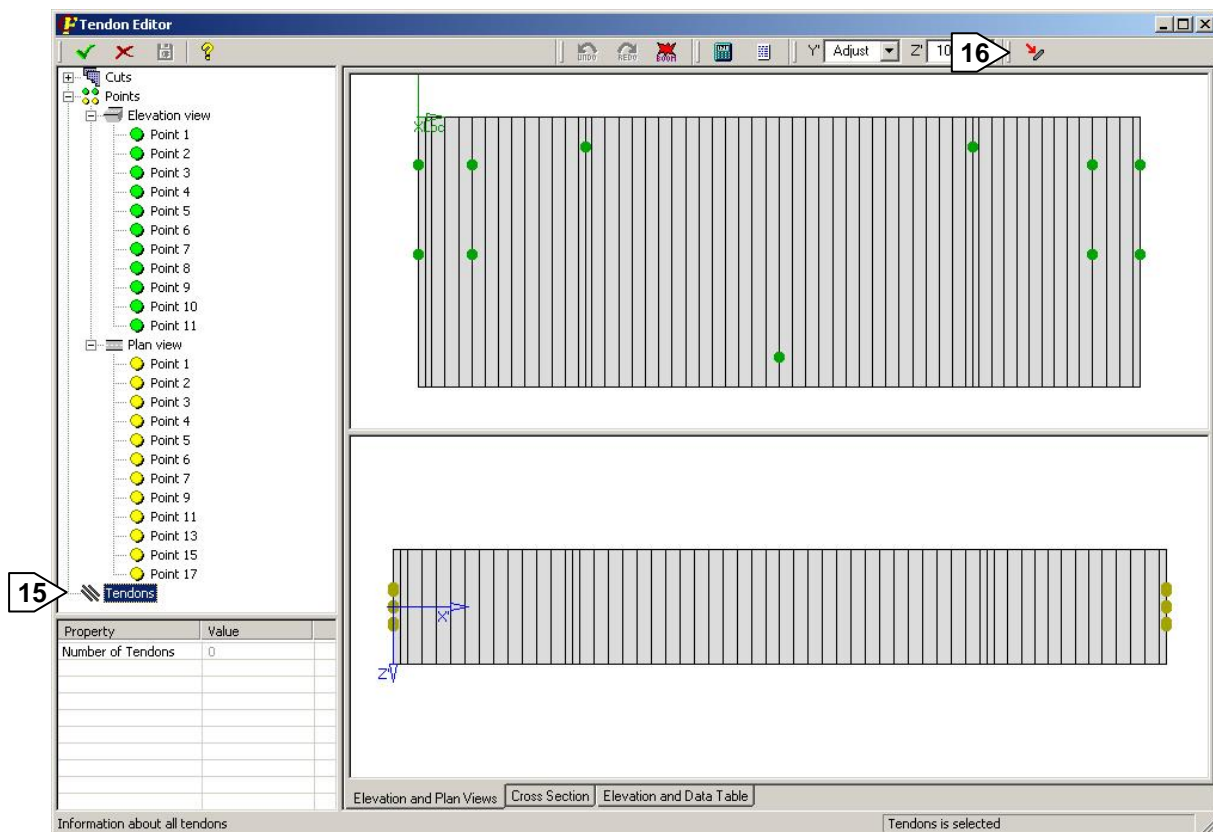


We follow the same procedure to define the other points:

Elevation View Point number	Cut	Ycoord	Slope	Inflection Distance Ratio
1	1 (x=0.0)	-0.613	0	0.5
2	6 (x=4.0)	-0.613	0	0.5
3	15 (x=12.5)	-0.135	0	0.5
4	30 (x=27)	-1.065	0	0.5
5	1 (x=0.0)	-0.212	0	0.5
6	6 (x=4.0)	-0.212	0	0.5

Elevation View Point number	Cut	Ycoord	Slope	Inflection Distance Ratio
7	55 (x=50.37)	-0.212	0	0.5
8	60 (x=54)	-0.212	0	0.5
9	45 (x=41.5)	-0.135	0	0.5
10	55 (x=50.37)	-0.613	0	0.5
11	60 (x=54)	-0.613	0	0.5

Once all points are defined in plan and elevation view we can trace the trajectory of all tendons:



15 Click on tendons

16 Insert tendon 

17 We start with tendon number 1 and enter 3 as material number


18 Enter 0.0021 m<sup>2</sup> as area of tendon

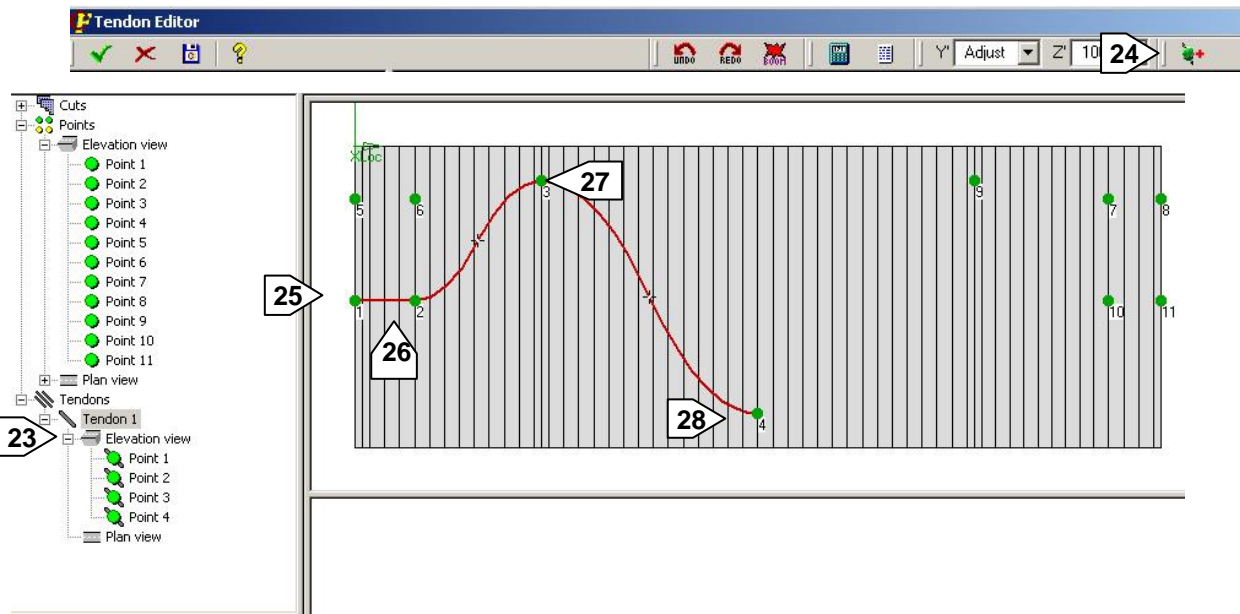
19 Enter 0.09 m as casing diameter

- 20 Enter 3,125,000 N as initial tendon tensile force
- 21 Enter 2,930,000 N as anchorage force
- 22 OK (Creep Stress will have default value of 5,000,000 N/m<sup>2</sup>)

Property	Value
Id	1
Prestressing steel	3
Area	2.1e-005
Casing diameter	9e-002
Prestressed to	3125000
Anchored at	2930000
Creep stress	N/m <sup>2</sup>
Insert Tendon	Ok Cancel

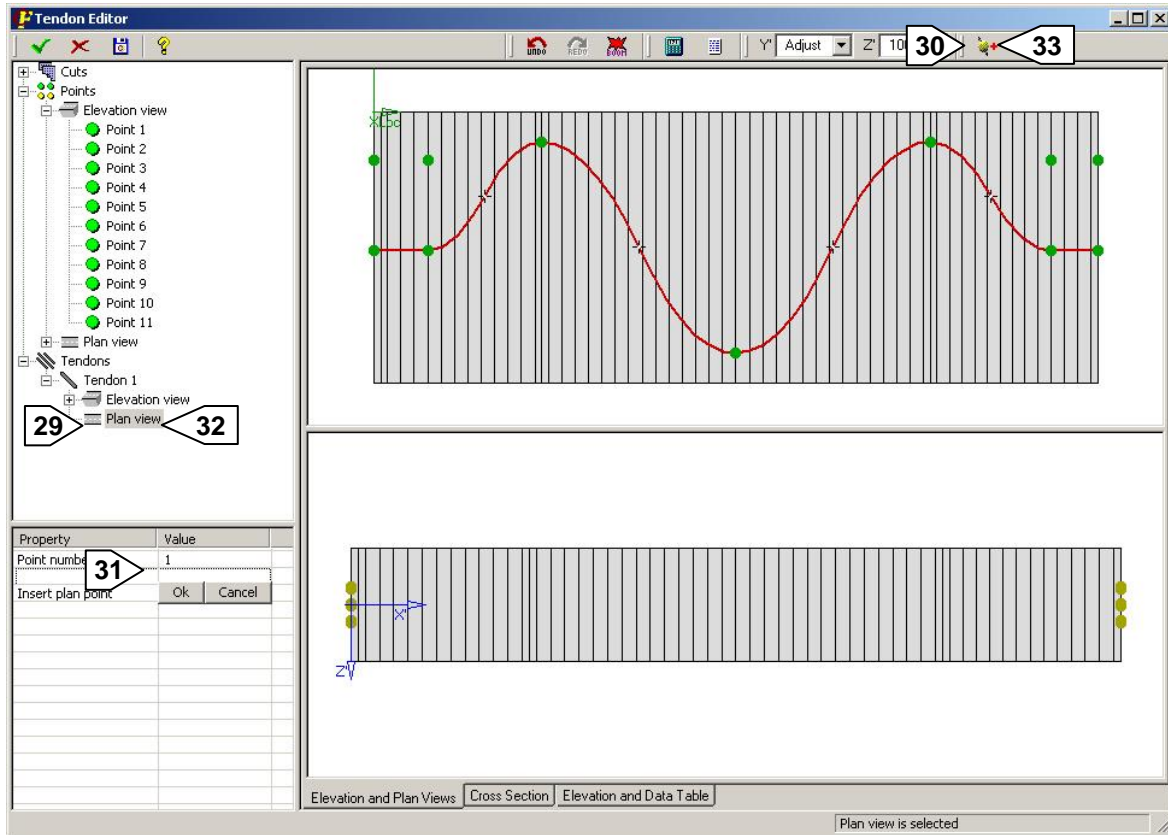
We define the tendon in elevation view:

- 23 Select Elevation view in tendon 1
- 24 Insert point in tendon 





- 25 Pick point 1
- 26 Pick point 2
- 27 Pick point 3
- 28 Pick point 4

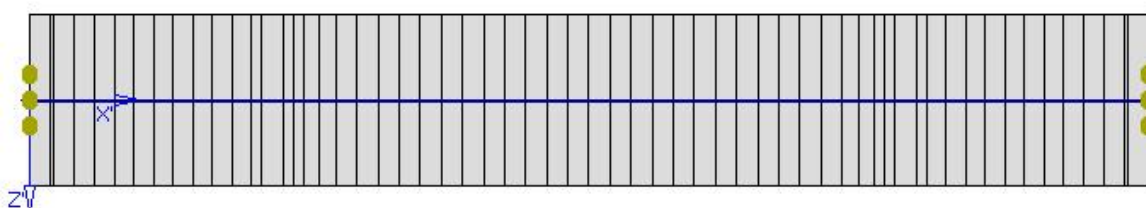
And continue to trace the trajectory in elevation view by picking points 9, 10 & 11.



To finish defining tendon 1 we create its plan view:

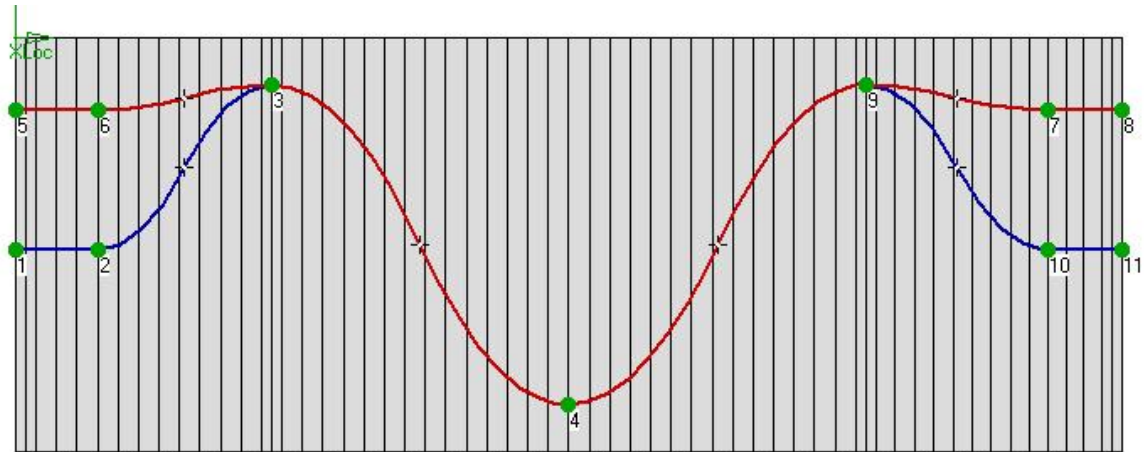
- 29 Select Plan view
- 30 Insert point in tendon 
- 31 Enter Point number 1 and OK
- 32 Select plan view again
- 33 Insert point in tendon 
- 34 Enter Point number 7 and OK

Property	Value
Point number	7
Insert plan point	Ok Cancel



To define tendon 2 we repeat the same steps from step 12 with these control points (same material, area and casing diameter):

<b>Elevation</b>	5	6	3	4	9	7	8
<b>View</b>							
<b>Plan view</b>	2	9					



**Tendon Editor**

35

Cuts  
Points  
Tendons  
Tendon 1  
Tendon 2

Property Value


Id	1
Number of points (Elev)	7
Number of points (Plan)	0
Active	Yes
Prestressing steel	3
Area	2.100E-03 m <sup>2</sup>
Casing diameter	0.090 m
Initial force	3.125E+06 N
Anchor force	2.930E+06 N
Initial stress	1.480E+09 N/m <sup>2</sup>
Anchor stress	1.395E+09 N/m <sup>2</sup>
Creep stress	5.000E+06 N/m <sup>2</sup>
Order	1
Prestressing order	Both sides

Elevation and Plan Views | Cross Section | Elevation and Data Table

Information and modifiable data of tendon 1


Tendon 1 is selected

Tendons 3 & 4 have the same trajectory in elevation view than tendon 1 so we will copy it:

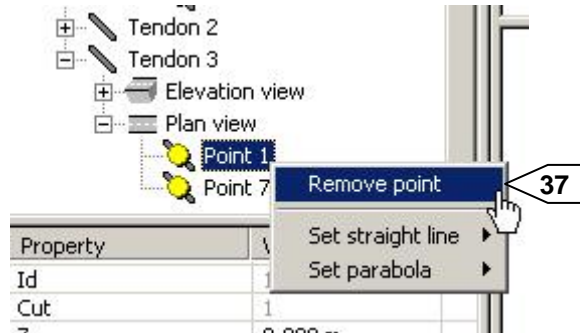
35 Select tendon 1 and then copy tendon 


36 Enter 3 and Ok

37 Remove points 1 & 7 in plan view

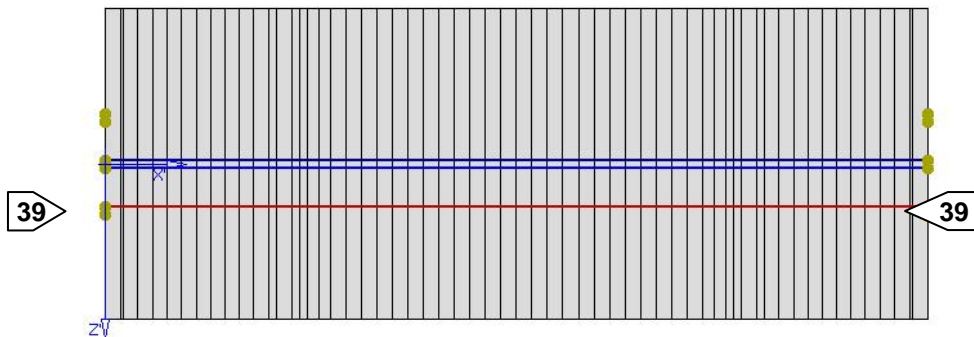
(By picking  or using right mouse button)

Property	Value
Copy to number	3
Copy tendon	Ok Cancel



38 Insert plan view point in tendon 

39 Enter plan point number 3 and number 11 to define tendon in plan view.



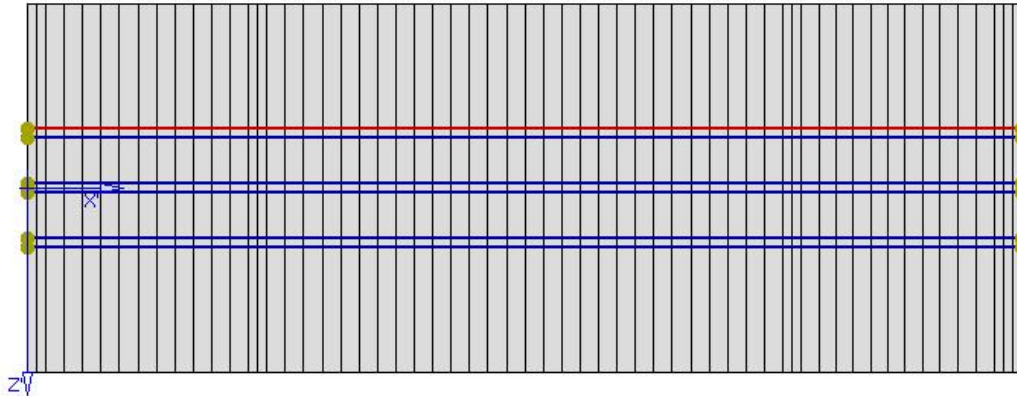
Z' Scale (Plan 250) %

Repeat the same process from step 35 to define tendon 4 and change the plan viewpoints by entering point number 4 and 13.

Tendons 5 & 6 have the same trajectory in elevation view than tendon 2 so we will do the same again as we did before with tendon 1, 3 & 4 but copying tendon 2:

Tendon 5	Plan view	
	5	15

Tendon 6	Plan view	
	6	17



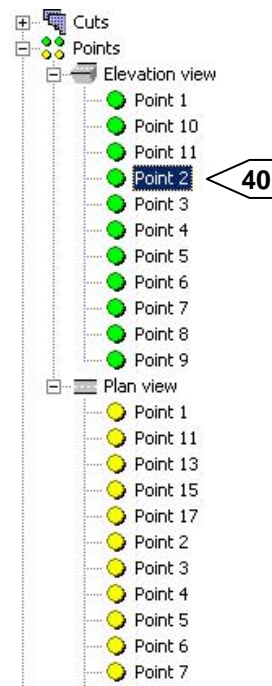
Z' Scale (Plan 250) %

Now we change the distance ratio of the inflection point (common tangent between both curves, default value 0.5).

40 Select Point 2 in elevation view

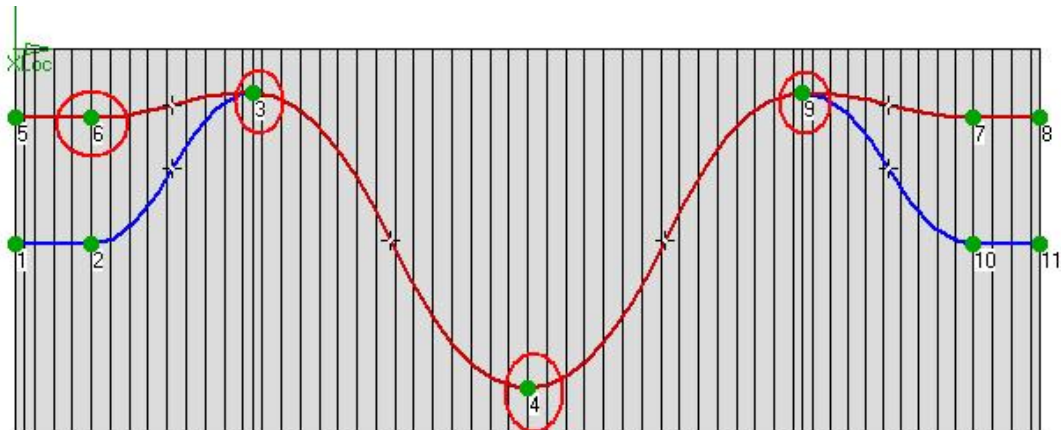
41 Enter 0.8 as Inflection distance ratio

Property	Value
Id	2
Cut	5
Y	-0.613
Slope	0.0000
Inflection distance ...	0.800



Do the same with these points:

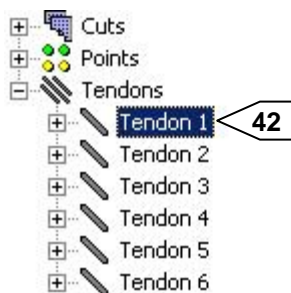
Point Number	Inflection distance ratio
6	0.8
3	0.2
4	0.8
9	0.2



Finally we set the Post-prestressed method: Prestressed from the initial end (cut=1):

42 Select tendon 1

43 Choose left side as prestressing order



Property	Value
Id	1
Number of points (Elev)	7
Number of points (Plan)	2
Active	Yes
Prestressing steel	3
Area	2.100E-03 m <sup>2</sup>
Casing diameter	0.090 m
Initial force	3.125E+06 N
Anchor force	2.930E+06 N
Initial stress	1.488E+09 N/m <sup>2</sup>
Anchor stress	1.395E+09 N/m <sup>2</sup>
Fluence stress	5.000E+06 N/m <sup>2</sup>
Order	1
Prestressing order	Left side

Repeat the same procedure with tendons 2, 3, 4, 5 & 6 and then

44 Ok to finish tendon definition



## ■ 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.

### 13. Define Boundary conditions

We will apply displacement constraints in all supports:

#### Support 1

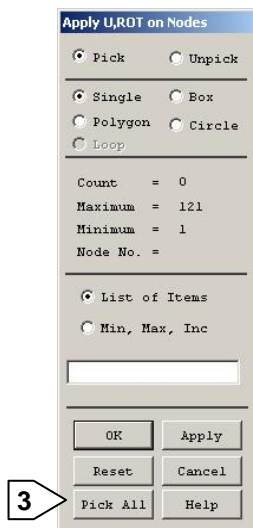
Utility Menu: **Select**→**Entities**

1 Select nodes by location

2 Enter X=0.5 and OK

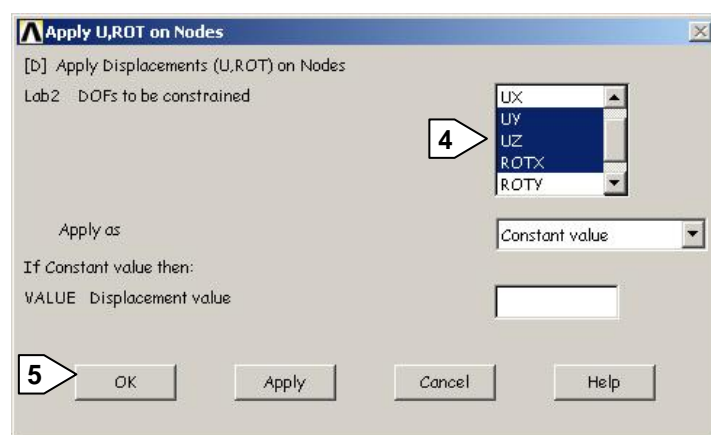


Main Menu: **Solution** → –Define Loads – **Apply** → – Structural – **Displacement** →**On Nodes**+



3 Pick All

4 Choose UY, UZ and ROTX



5 OK

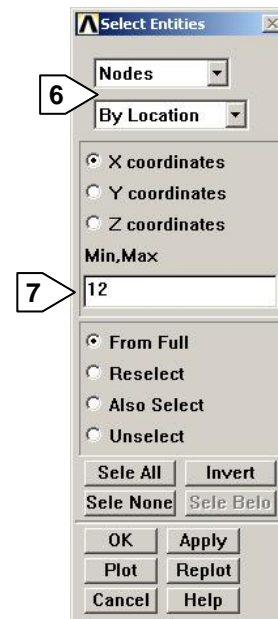
Utility Menu: **Select**→**Everything**

## Support 2

Utility Menu: **Select**→**Entities**

6 Select nodes by location

7 Enter X=12 and Ok

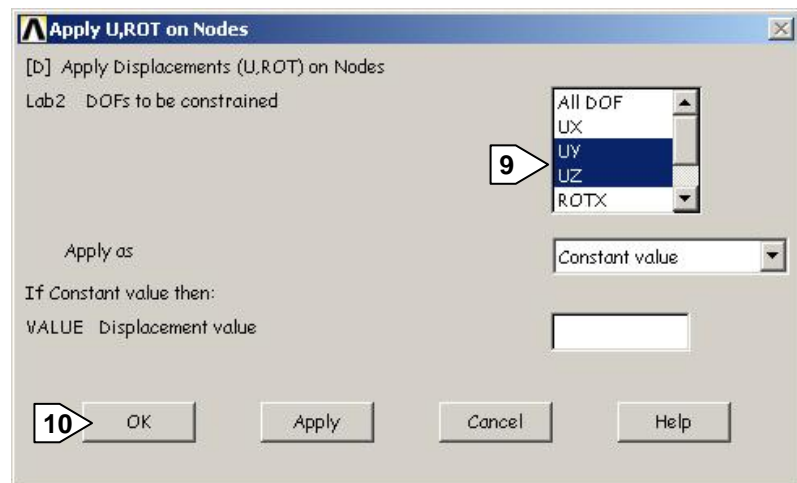


Main Menu: **Solution** → –Define Loads – **Apply** → – Structural – **Displacement** →**On Nodes**+

8 Pick all

9 Choose UY, UZ

10 Ok



Utility Menu: **Select**→**Everything**

We do the same with the rest of supports:

### Support 3

Utility Menu: **Select→Entities**

11 Select nodes by location

12 Enter X=41 and Ok

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

13 Pick all

14 Choose UY, UZ

15 Ok

Utility Menu: **Select→Everything**

### Support 4

Utility Menu: **Select→Entities**

11 Select nodes by location

12 Enter X=53 and Ok

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

13 Pick all

14 Choose UX, UY, UZ, ROTZ

15 Ok

Utility Menu: **Select→Everything**

## 14. Families and Load States Definition

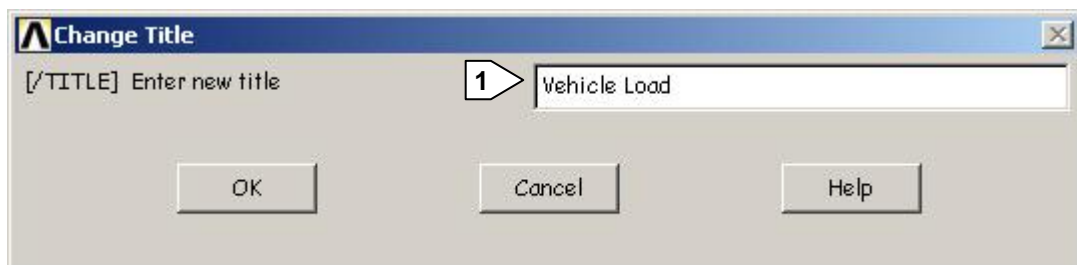
The user may define load families that can be later used in the combination module. A family is a group of load states, normally of the same topology. For example, the vehicles load would be a family formed by all loads created when the vehicles are placed in all the possible positions.

All the load steps belonging to a family are combined into one unique load step according to their nature

### a) Vehicle Load

Utility Menu: **File** → **Change Title**

1 Enter Vehicle Load



We load and modify properties of a vehicle from the library using the vehicle editor:

Main Menu: **Civil Solution** → **Bridges Solu** → **Traffic Loads** → **Vehicle**

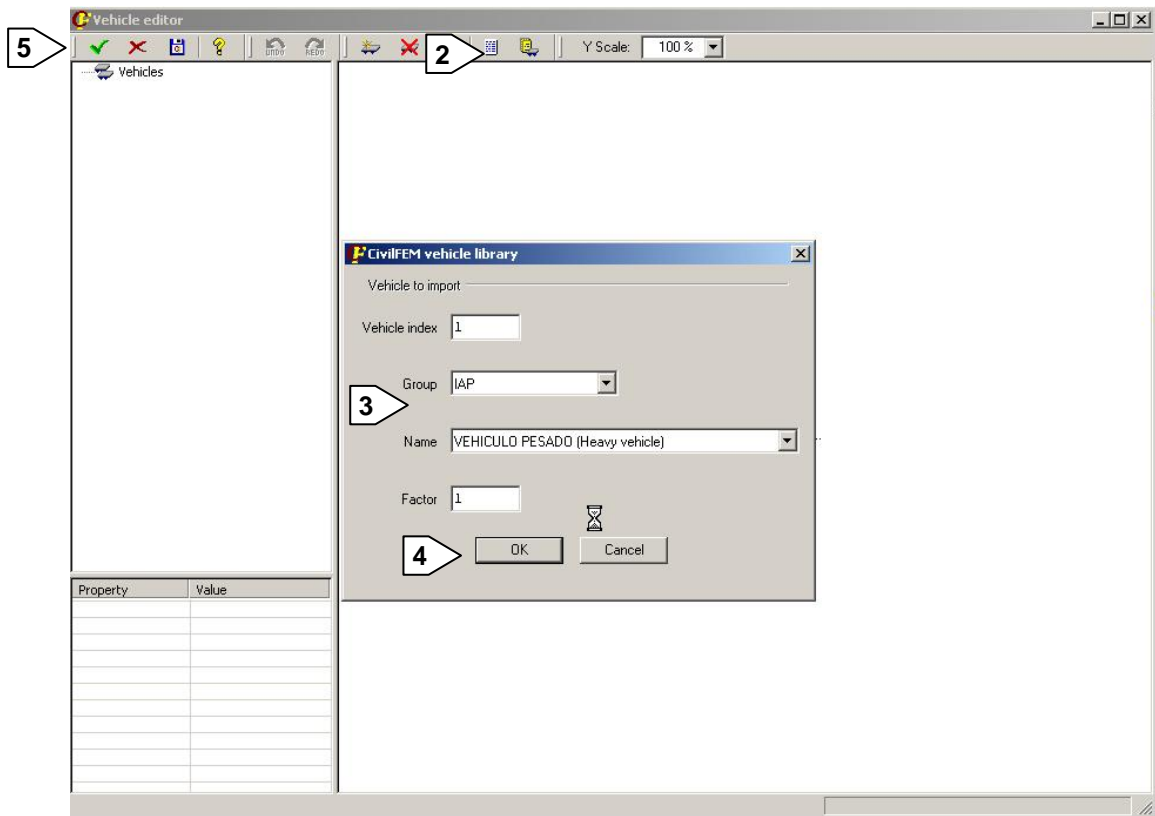
2 Vehicle Library



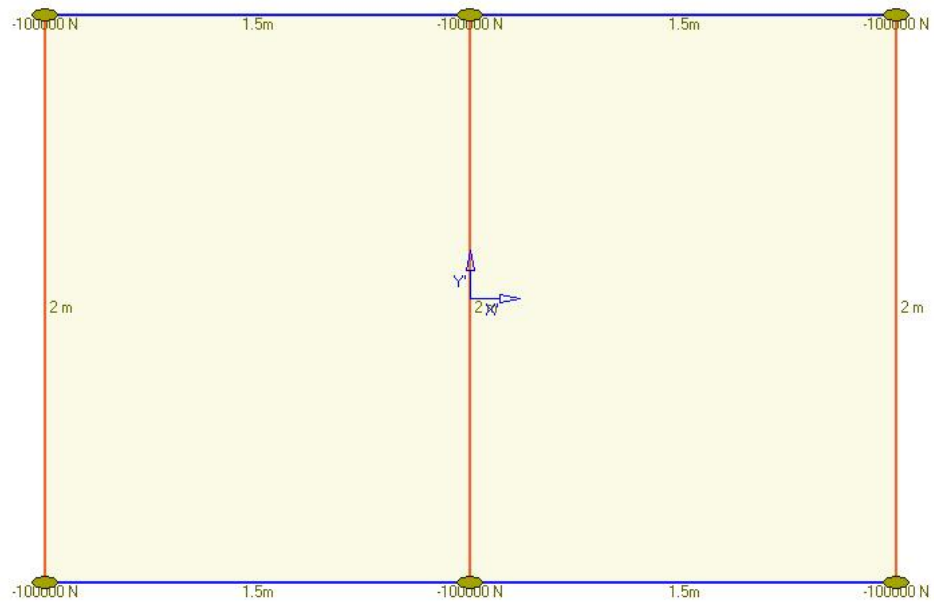
3 Select IAP Heavy Vehicle

4 Ok to define Vehicle 1

5 Ok to exit vehicle editor



Vehicle 1

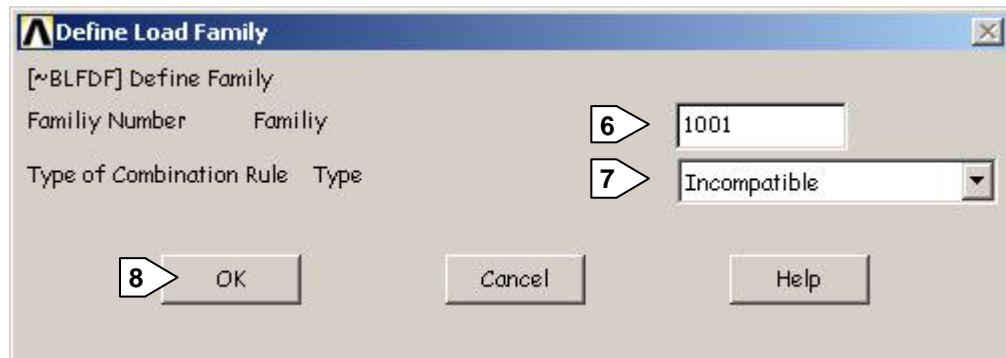


Main Menu: **Civil Solution** → **Bridges Solu** → **Family** → **Define Family**

One heavy vehicle in one line: Type INCOMPATIBLE

6 Family number 1001

7 Choose Incompatible



8 Ok

9 Ok to define family coefficients

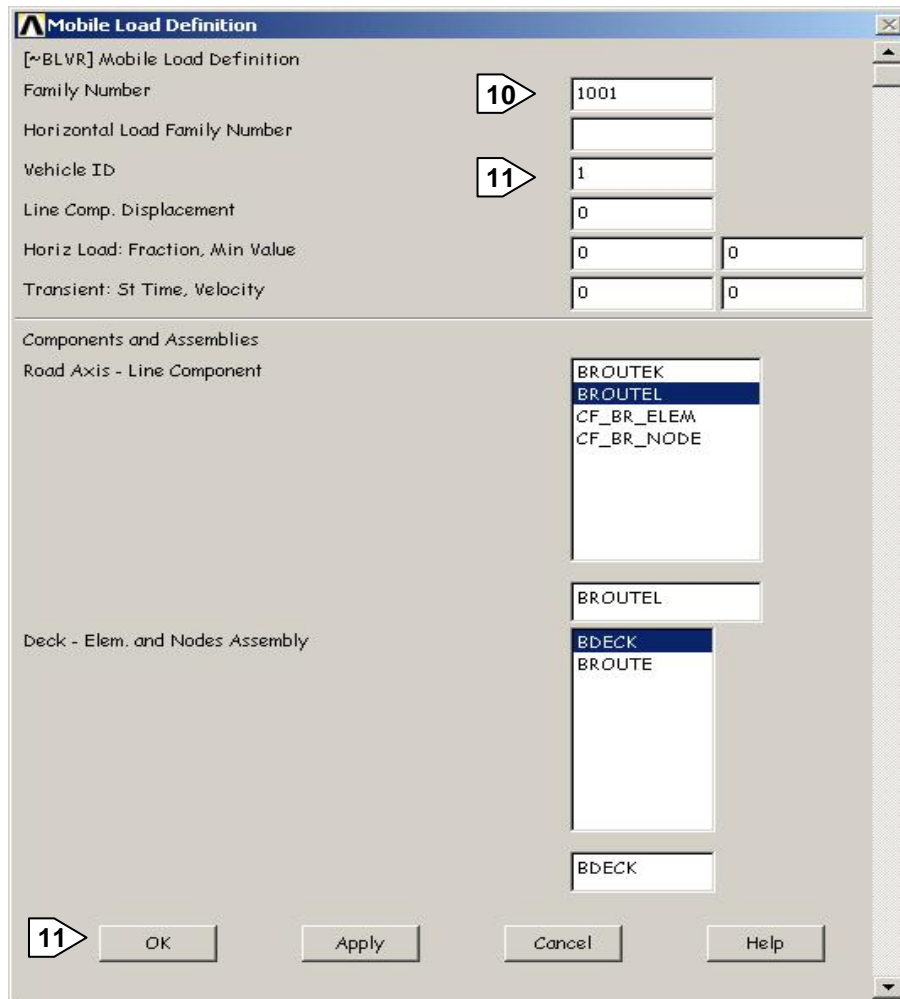


Now we define the family number to which the mobile loads will be assigned and the identification number of the vehicle that is going to move over the bridge:

Main Menu: **Civil Solution** → **Bridges Solu** → **Traffic Loads** → **Mobile Load**

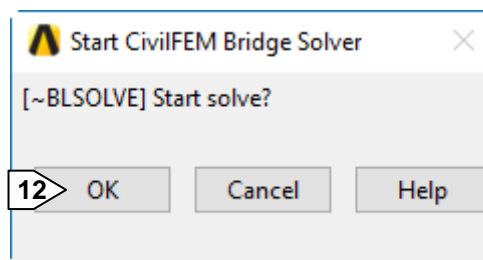
10 Enter Family number 1001

11 Enter Vehicle ID 1 and Ok



12 Solve

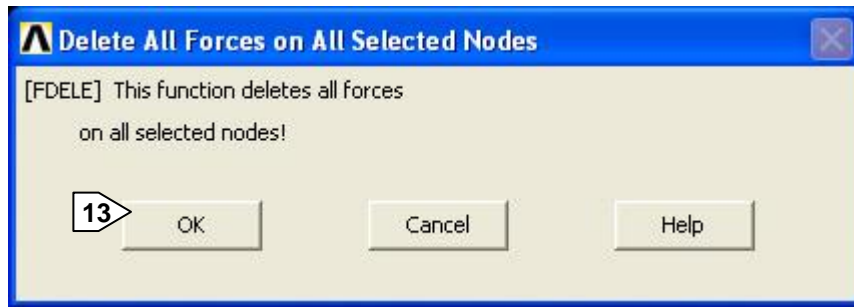
Main Menu: **Civil Solution** → **Bridges Solu** → **Solve**



Now we delete forces to continue with next load step:

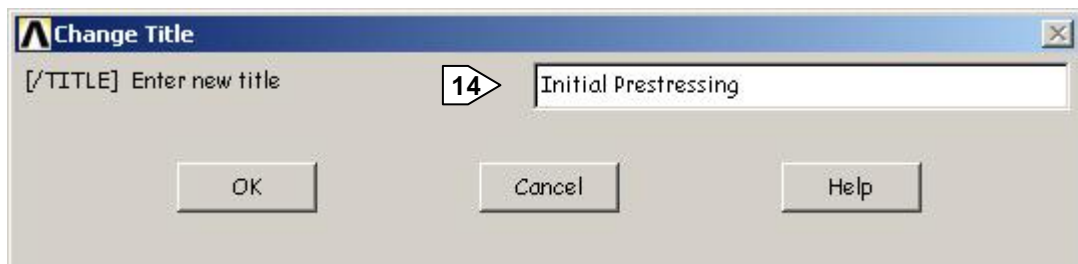
Main Menu: **Solution** → **Define Loads** → **Delete** → **All Load data** → **All Forces** → **On all nodes**

13 Ok



## b) Initial Prestressing

Utility Menu: **File** → **Change Title**



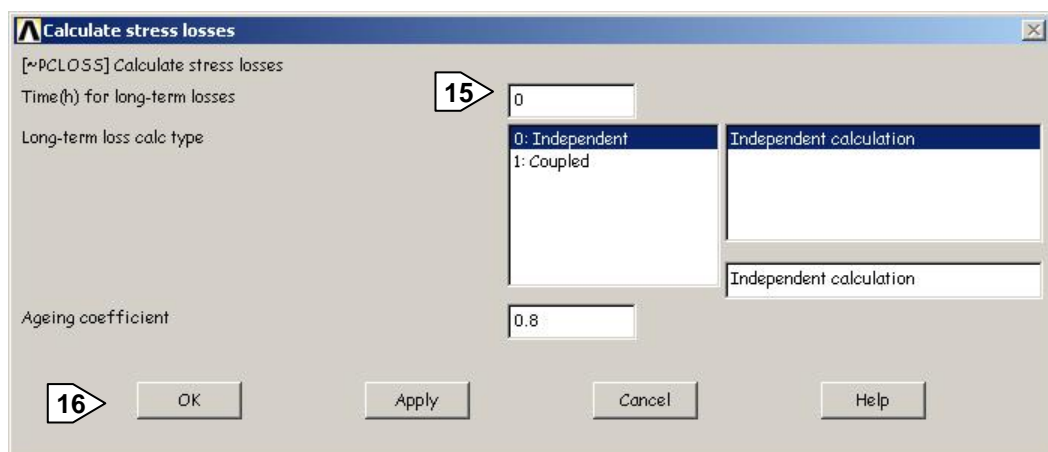
14 Enter "Initial Prestressing"

We will calculate the immediate losses, they occur once the prestressing force is applied, after the concrete has been placed and cured, and when the tendons are anchored.

Main Menu: **Civil Solution** → **Prestress Concrete Solu** → **Tendons** → **Calculate Losses**

15 Enter 0 as time

16 OK

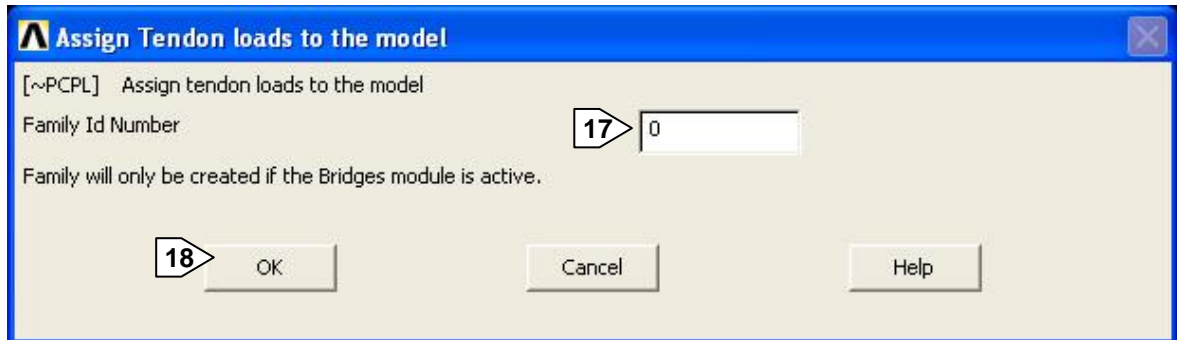


Now we transfer the tendons' actions over the finite elements model:

Main Menu: **Civil Solution** → **Prestress Concrete Solu** → **Tendons** → **Load Tendon**

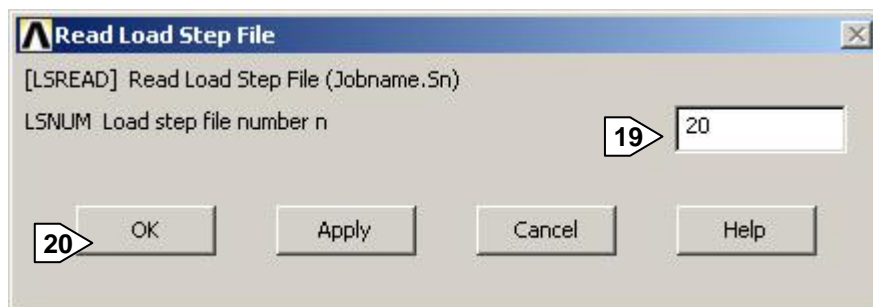
17 Enter 0

18 OK



**~CFLSWRT** command writes all load and load step option data for the selected model to a load step file for later use.

Main Menu: **Civil Solution** → **Load State** → **Write LS file**

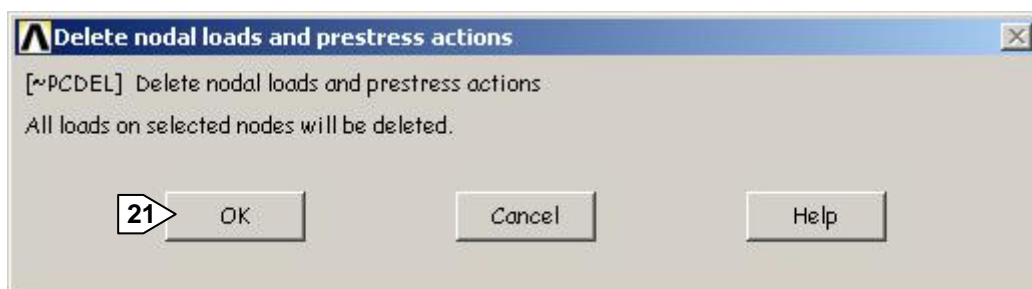


19 Enter Load step number 20

20 OK

In order for CivilFEM to process the results correctly, it is necessary to use this command to delete the prestressing loads.

Main Menu: **Civil Solution** → **Prestress Concrete Solu** → **Tendons** → **Delete Loads**



21 > Ok to delete all loads

### c) Final Prestressing

Utility Menu: **File** → **Change Title**

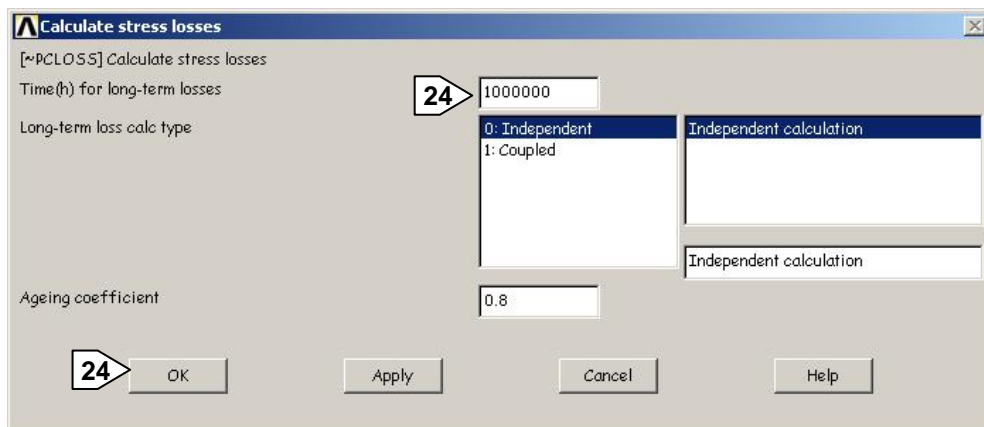


22 > Enter Final Prestressing

We will calculate the long-term losses, they occur after the end of the last prestressing operation.

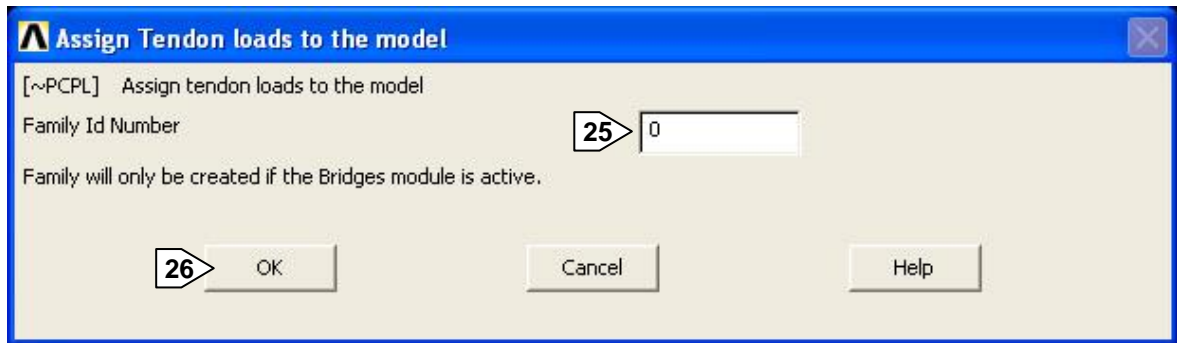
Main Menu: **Civil Solution** → **Prestress Concrete Solu** → **Tendons** → **Calculate Losses**

23 > Enter time 1,000,000 and OK



Now we transfer the tendons' actions over the finite elements model:

Main Menu: **Civil Solution** → **Prestress Concrete Solu** → **Tendons** → **Load Tendon**

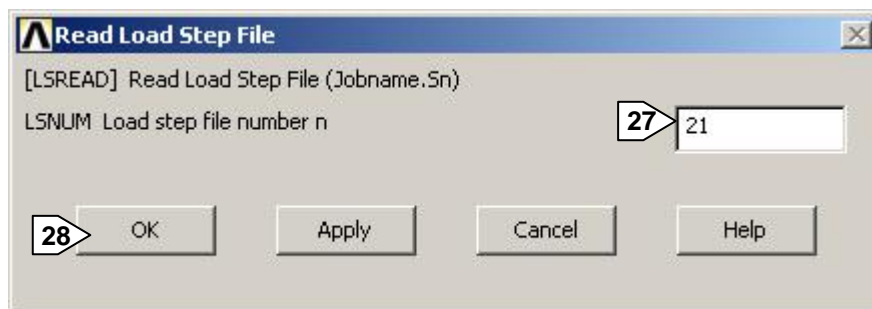


25 Enter 0

26 OK

**~CFLSWRT** command writes all load and load step option data for the selected model to a load step file for later use.

Main Menu: **Civil Solution** → **Load State** → **Write LS file**

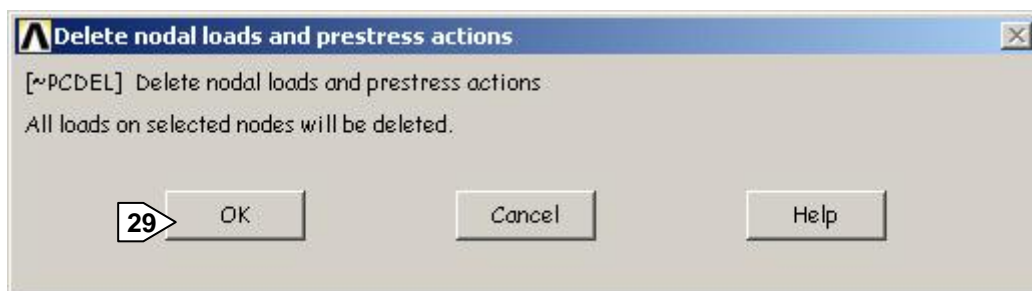


27 Enter Load step number 21

28 Ok

In order for CivilFEM to process the results correctly, it is necessary to use this command to delete the prestressing loads.

Main Menu: **Civil Solution** → **Prestress Concrete Solu** → **Tendons** → **Delete Loads**



29 Ok to delete all loads

#### d) Self Weight

Utility Menu: **File** → **Change Title**

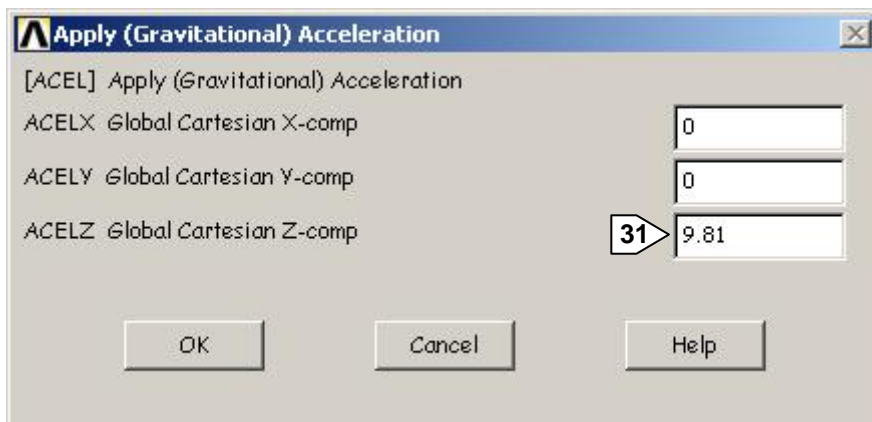


30 Enter Self Weight

We specify the linear acceleration of the structure:

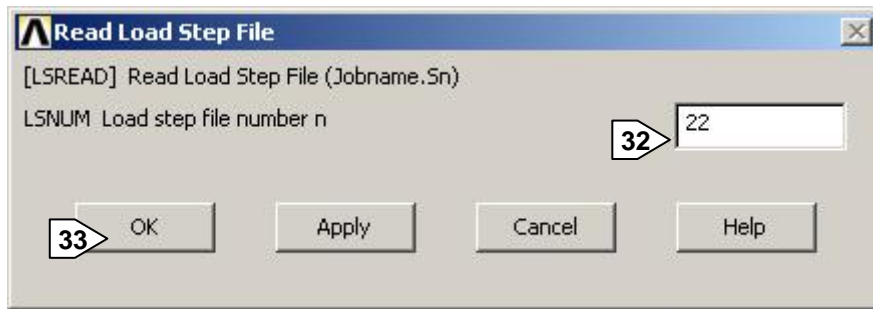
Main Menu: **Solution** → **Define Loads** → **Apply**  
→ **Structural** → **Inertia** → **Gravity** → **Global**

31 Enter 9.81 as ACELZ and OK



~**CFLSWRT** command writes all load and load step option data for the selected model to a load step file for later use.

Main Menu: **Civil Solution** → **Load State** → **Write LS file**



32 Enter Load step number 22

33 Ok

Now we delete this load by entering zero gravity:

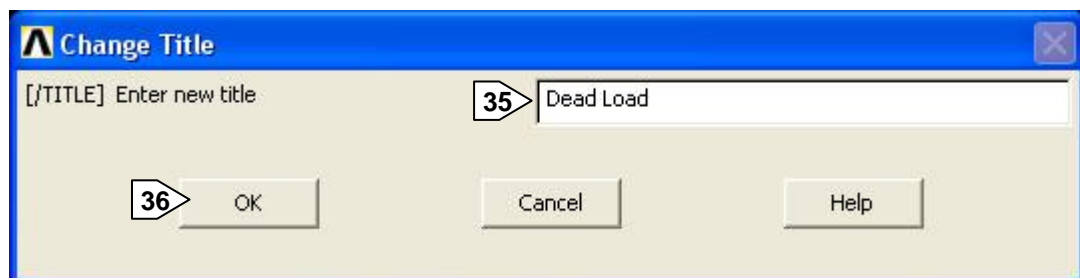
Main Menu: **Solution** → **Define Loads** → **Apply**  
 → **Structural** → **Inertia** → **Gravity**



34 Enter 0 as ACELZ and Ok

### e) Dead Load

Utility Menu: **File** → **Change Title**

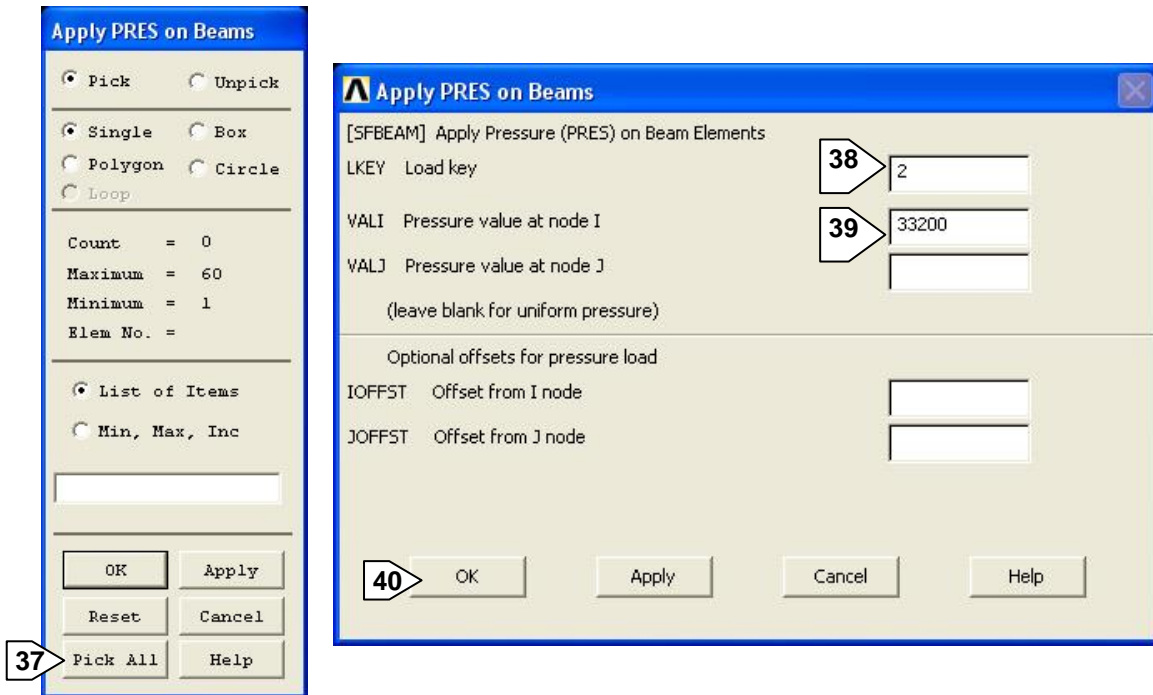


35 Enter Dead Load

36 OK

We specify a surface load on all beam elements:

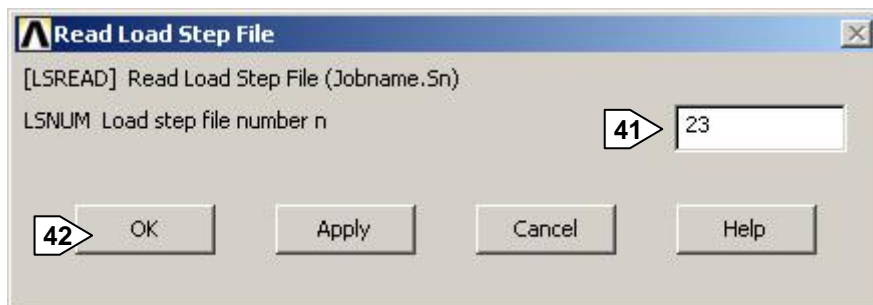
Main Menu: **Solution** → **Define Loads** → **Apply** → **Structural** → **Pressure** → **On Beams+**



- 37 Pick all
- 38 Enter 2 as load key
- 39 Enter pressure value= 33200
- 40 OK

~**CFLSWRT** command writes all load and load step option data for the selected model to a load step file for later use.

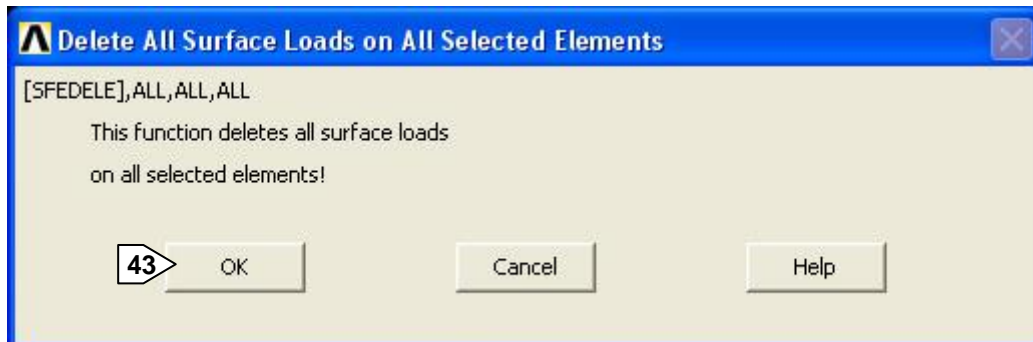
Main Menu: **Civil Solution** → **Load State** → **Write LS file**



- 41 Enter Load step number 23
- 42 Ok

Now we delete the surface loads from all elements:

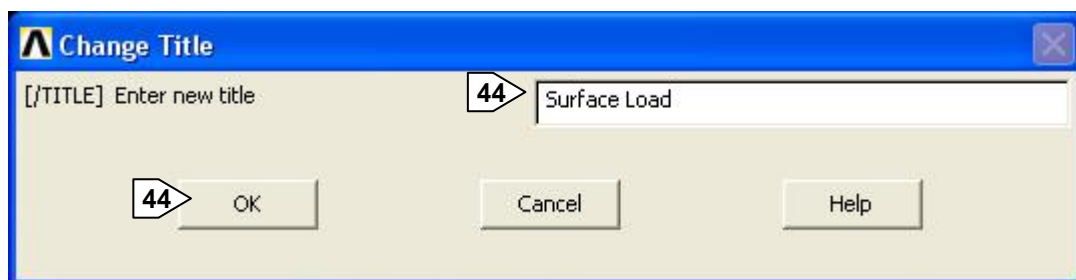
Main Menu: **Solution** → **Define Loads – Delete** → **All Load data** → **All Surface Ld** → **On all Elems**



43 > Ok

#### f) Surface Load

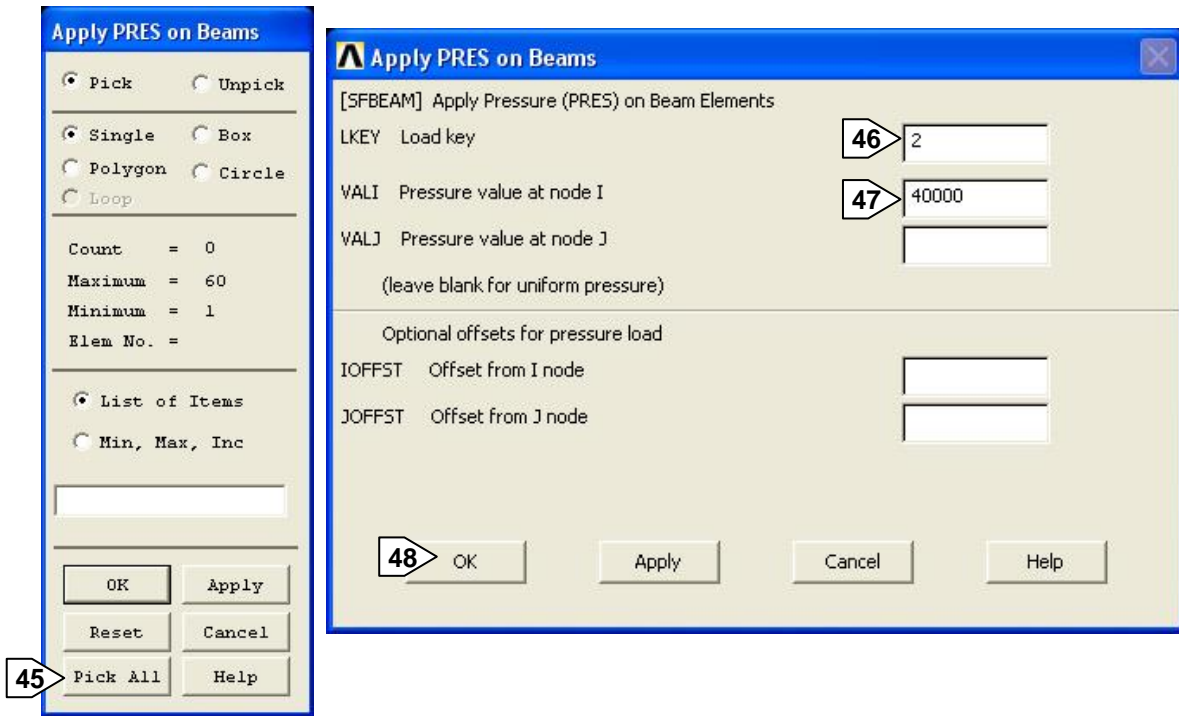
Utility Menu: **File** → **Change Title**



44 > Enter Surface Load an Ok

We specify a surface load on all beam elements:

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



45 Pick all

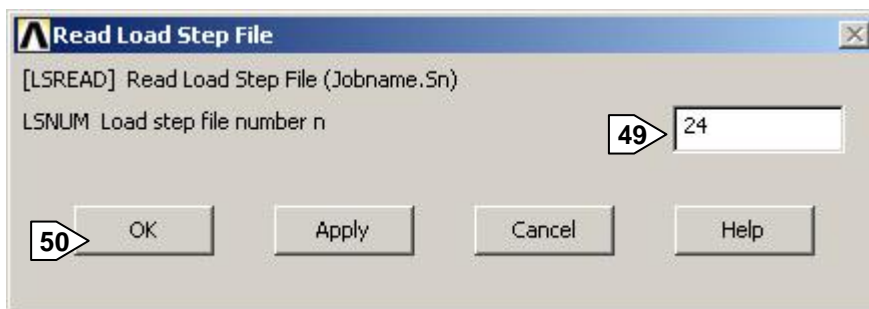
46 Enter 2 as Load key

47 Enter pressure value = 40000

48 Ok

~CFLSWRT command writes all load and load step option data for the selected model to a load step file for later use.

Main Menu: **Civil Solution** → **Load State** → **Write LS file**



49 Enter 24

50 Ok

Now we delete the surface loads from all elements:

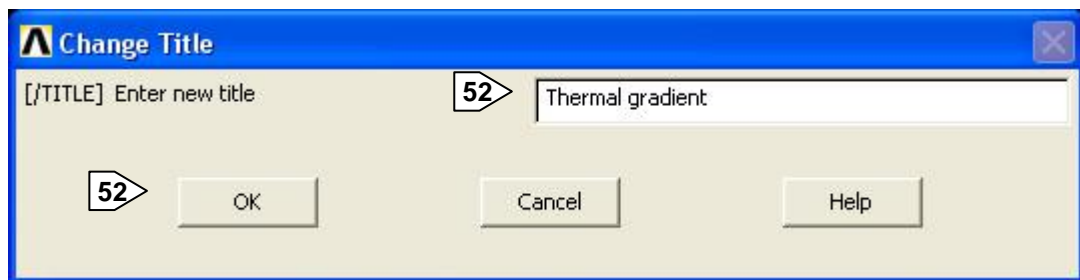
Main Menu: **Solution** → **Define Loads – Delete** → **All Load data** → **All Surface Ld** → **On all Elems**



51 > Ok

### g) Thermal Gradient

Utility Menu: **File** → **Change Title**



52 > Enter Thermal gradient and Ok

Now we apply temperature on all elements:

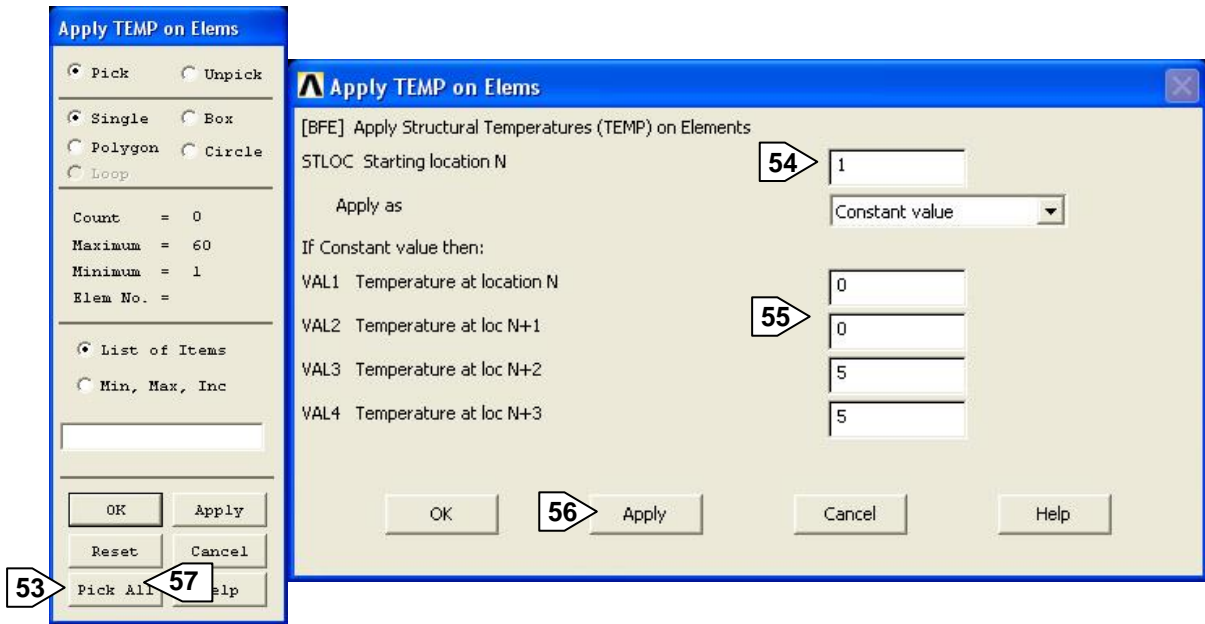
Main Menu: **Solution** → **Loads – Apply** → **Structural – Temperature** → **On Elements**

53 > Pick all

54 > Starting location at location 1

55 > VAL1=0, VAL2=0, VAL3=5, VAL4=5

56 > Apply

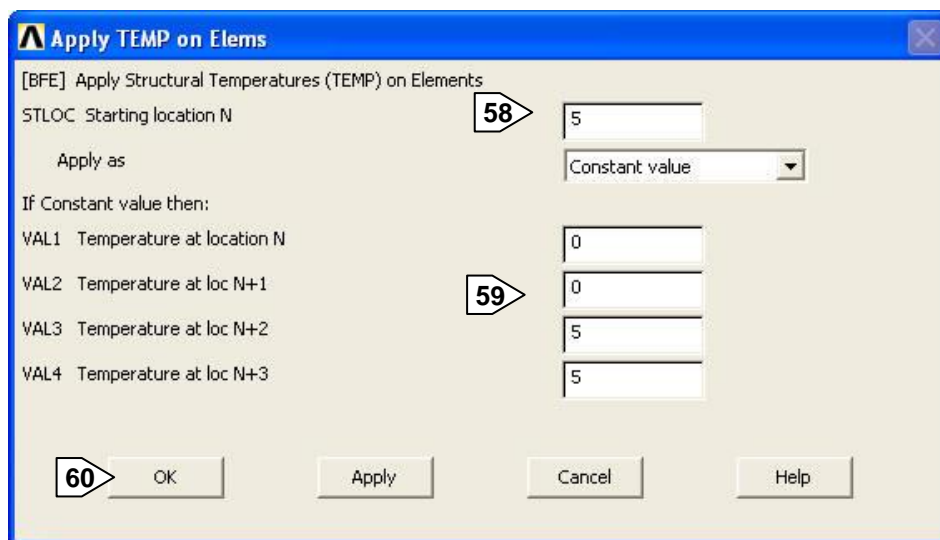


57 Pick all again

58 Starting location at node 5

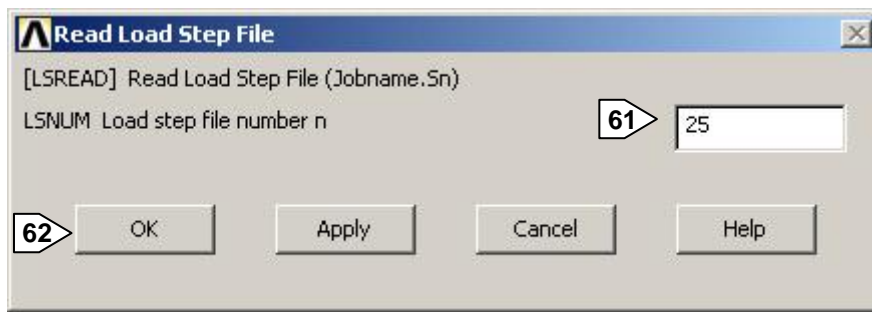
59 VAL1=0, VAL2=0, VAL3=5, VAL4=5

60 Ok



**~CFLSWRT** command writes all load and load step option data for the selected model to a load step file for later use.

Main Menu: **Civil Solution** → **Load State** → **Write LS file**

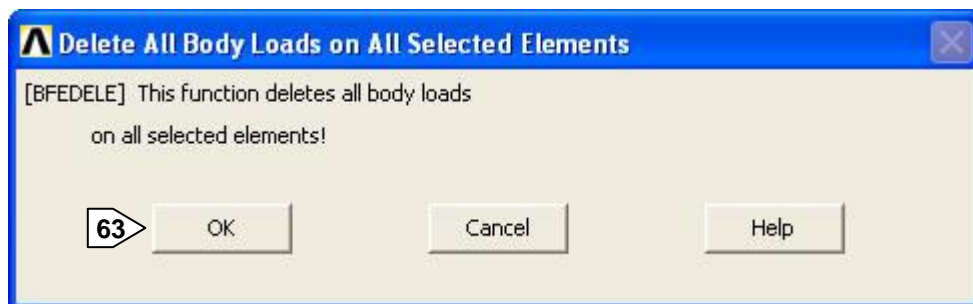


61 Enter 25

62 Ok

We finally delete element body force loads:

Main Menu: **Solution** → **Define Loads** – **Delete** → **All Load data** → **All Body loads** → **On all Elems**



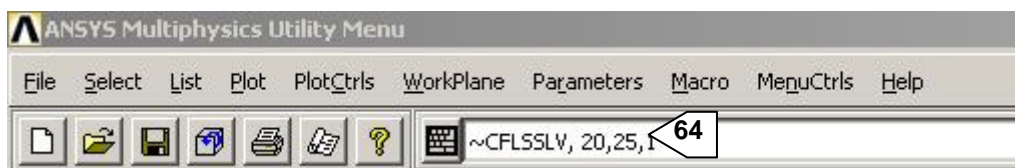
63 Ok

To finish we must read and solve multiple load steps taking into account the construction process:

64 We type this in the ANSYS command window:

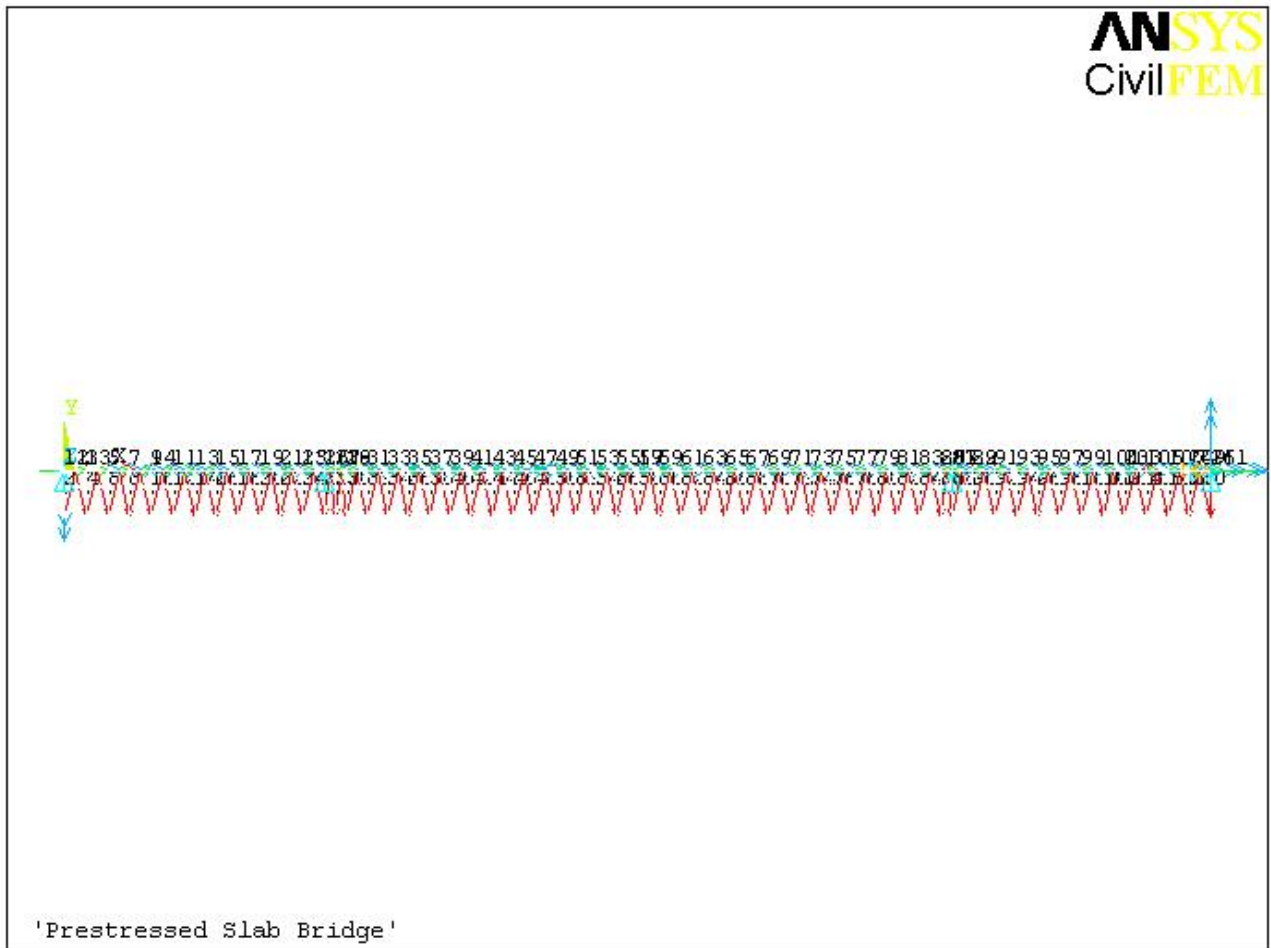
~CFLSSLV, 1,25,1

Range of load step files to be read and solved, from LS20 to LS25 in steps of 1



These are all the load steps and start states created :

Load Steps	Start States
Vehicle Load	LS1:19
Initial Prestressing	LS20
Final Prestressing	LS21
Self Weight	LS22
Dead Load	LS23
Surface Load	LS24
Thermal Gradient	LS25



## ■ Postprocessing

### 15. Conversion of families into combinations

Constructing the combinations from the families previously defined, is carried out through command **~BLF2CMB**. This command converts families in CivilFEM load combinations. The combination will always have the same numbering as the family from which it has been generated.

Once this command is executed, it deletes the existing families and the user must start using at this stage the corresponding commands of the CivilFEM load combination module (command **~TRGDEF** for target definition and command **~COMBINE** to carry out the combinations).

Main Menu: **Civil Postprocessor** → **Bridge Post** → **Family to Cmb**

1 ▷ Ok

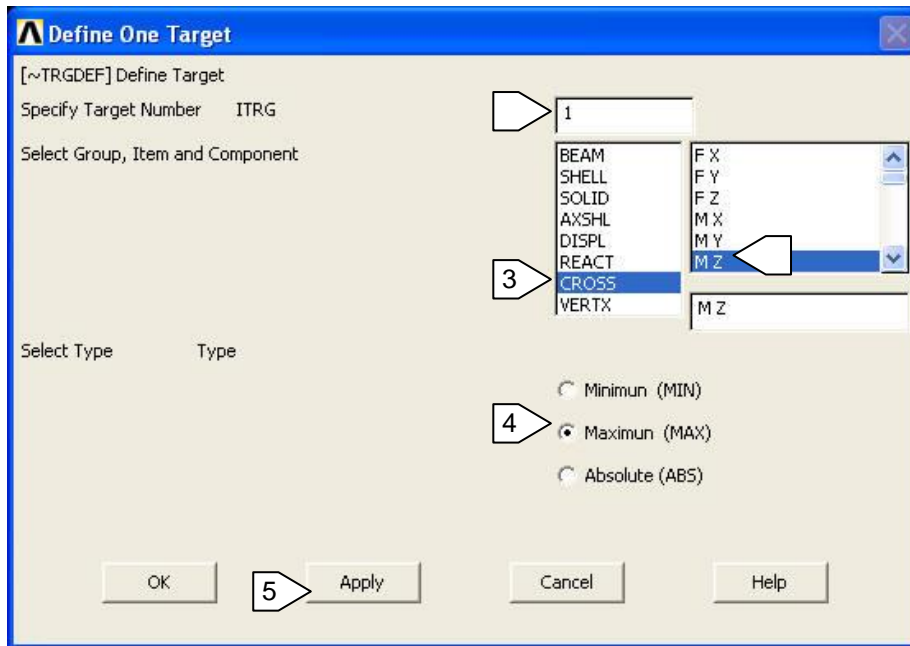


### 16. Define targets

The targets are the objectives to reach; in this case we will define two, the maximum and minimum bending moments.

Main Menu: **Civil Postprocessor** → **Combine Results** → **Def One Target**

- 2 ▷ Specify target number 1
- 3 ▷ Select CROSS and MZ
- 4 ▷ Select Maximum
- 5 ▷ Apply to define next target

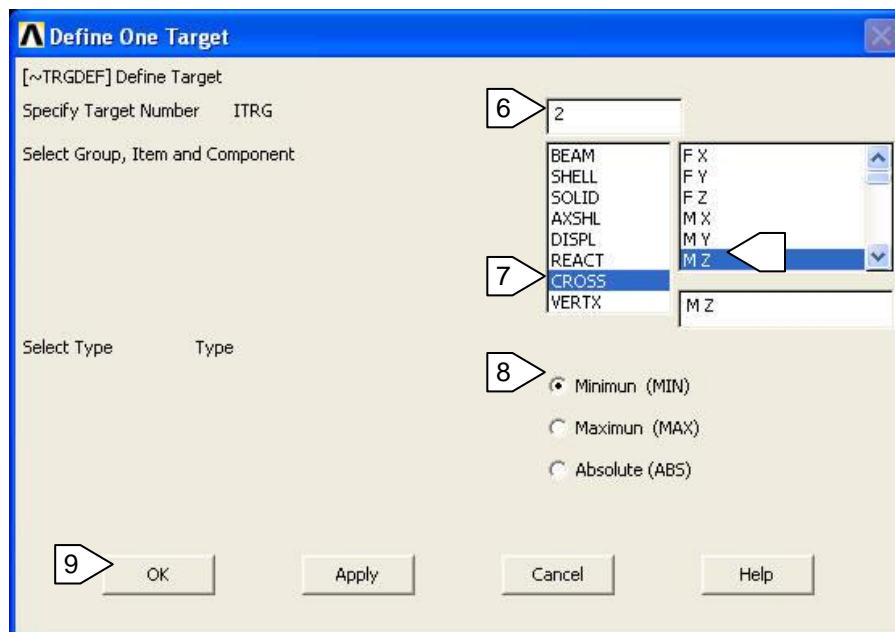


6 Specify target number 2

7 Select CROSS and MZ

8 Select Minimum

9 Ok



## 17. Define combination rules

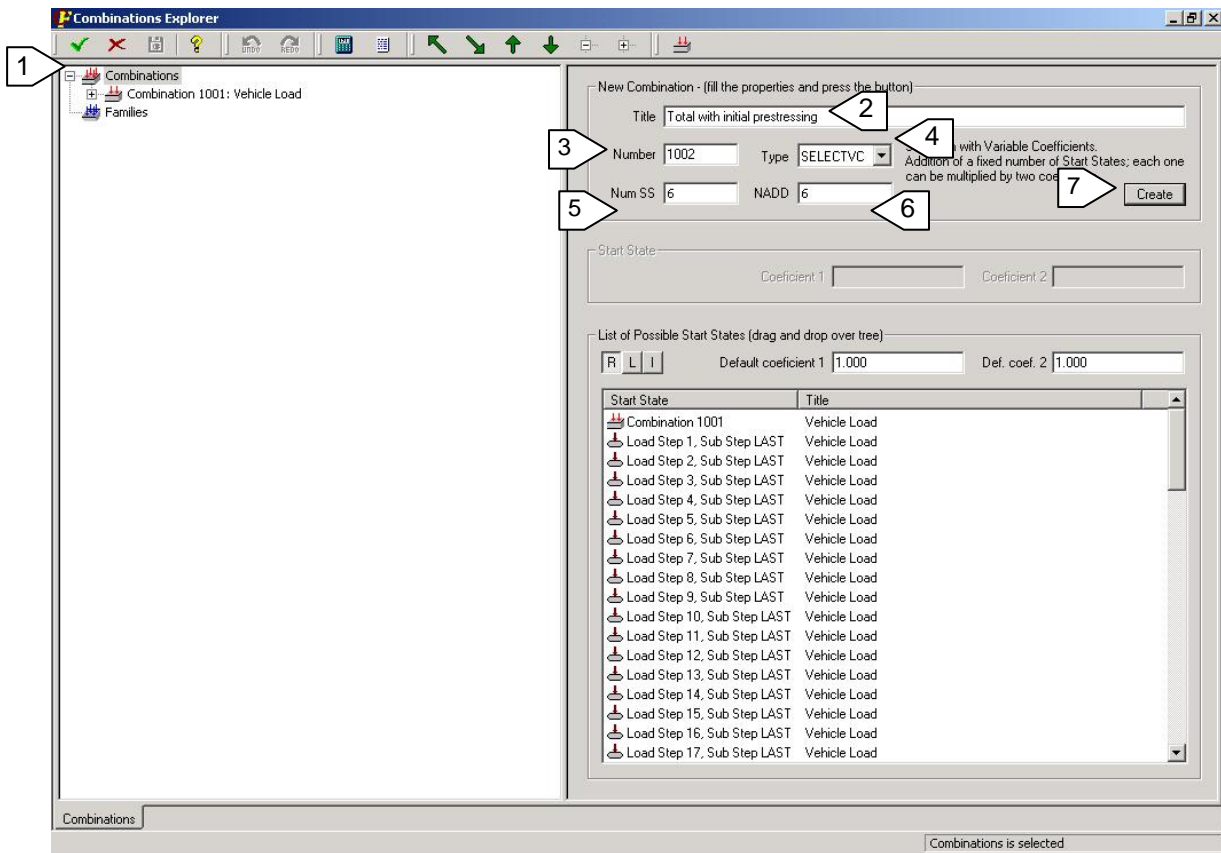
CivilFEM provides, by the combinations module, the possibility of operating with a results sets and combine them in such a way that a given targets are achieved. Therefore, the results combination is based on the search of the combination among certain data set that, following certain rules, fulfills the given targets in each node of the structure.

In our case we will define these combination rules using the start states created before:

Combination Rule	Start States	Type	Nº Start States	Cf1	Cf2
Total with Initial Prestressing 1002	CMB1001, LS20, 22, 23, 24, 25	SELECTVC	6	1.10 1.10 1.35 1.35 1.5 1.35	0.90 0.90 1.00 1.00 0.00 1.00
Total with Final Prestressing 1003	CMB1001, LS21,22, 23, 24, 25	SELECTVC	6	1.10 1.10 1.35 1.35 1.5 1.35	0.90 0.90 1.00 1.00 0.00 1.00
Total with Initial or Final Prestressing 1004	CMB1002, 1003	OPTION	2	1	1

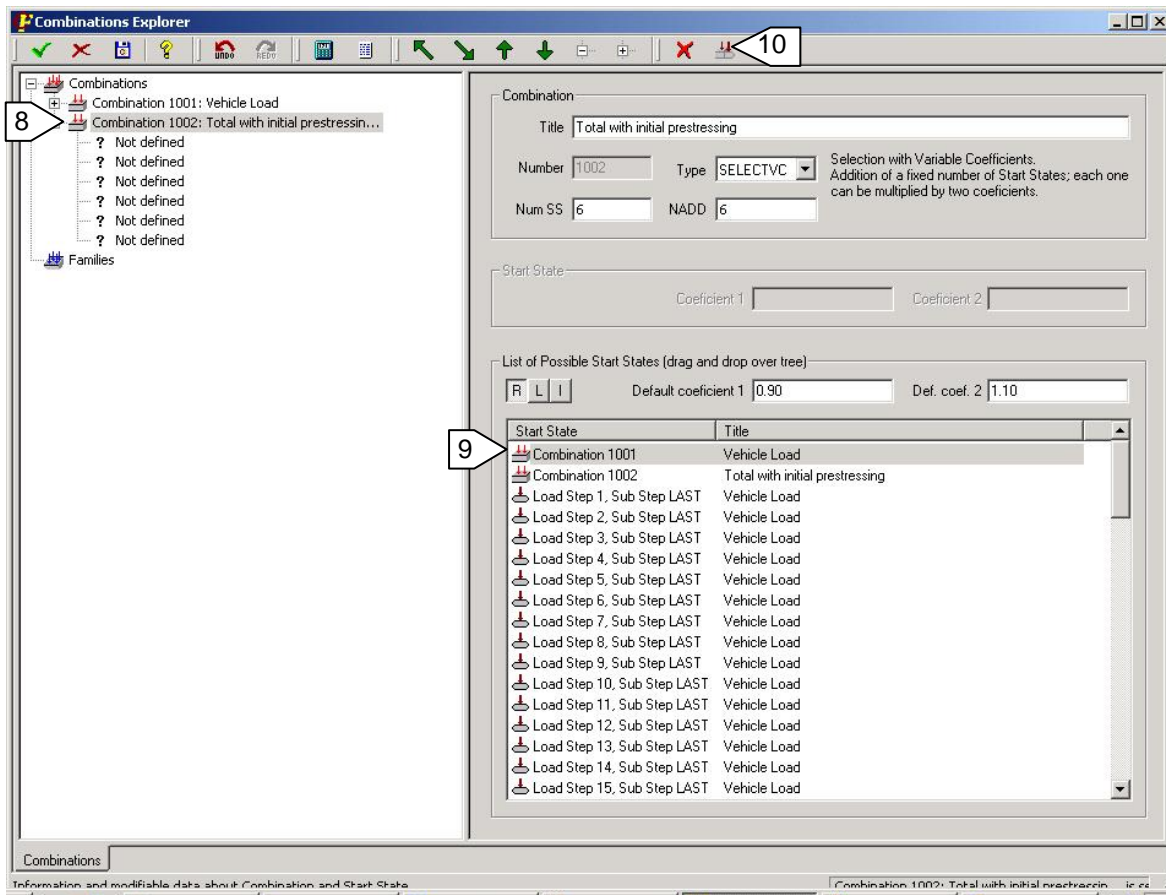
Main Menu: **Civil Postprocessor** → **Combine Results** → **Combination**

The combinations explorer window opens:



- 1 Pick on Combinations
- 2 Enter title "Total with initial prestressing"
- 3 Enter combination number 1002
- 4 Choose type SELECTVC
- 5 Enter 6 as number of start states
- 6 Enter 6 as NADD
- 7 Click on Create

Combination 1002 is created, if we select the combination from the tree (it still does not have its start states defined):



So, now we select in the start states list, the ones desired to be included in the combination (more than one can be selected maintaining the control key [CTRL] pressed).

8 Select combination 1002

9 Select from List of Possible Start States: *Combination 1001: Vehicle Load*

10 Press the contextual button in toolbar 

to define a new start state (user can drag with the mouse the selected start states selected from the list, and drop them on the combination)

Start state 1 is defined, now we must define the rest:

11 Select combination 1002 again

12 Select from *List of Possible Start States* ( hold control key [CTRL] )

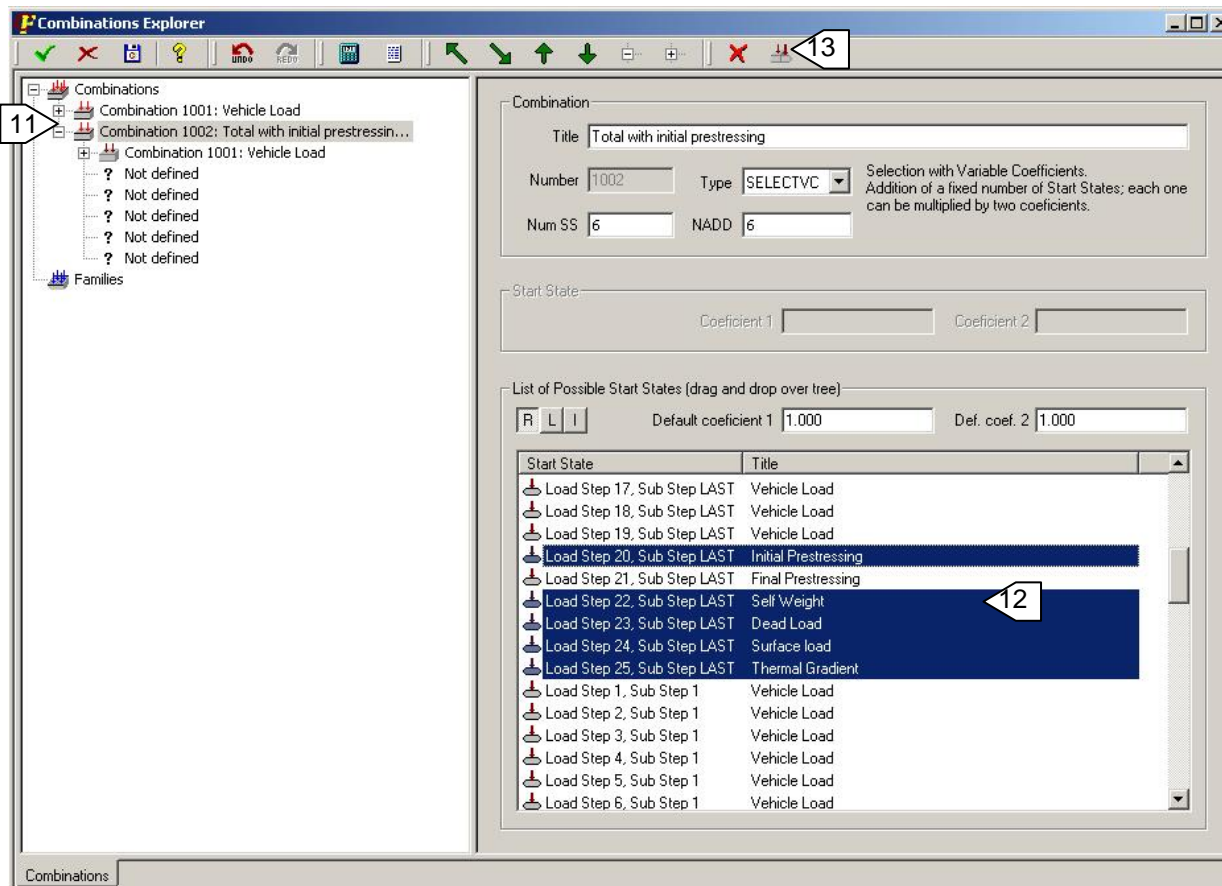
*Load Step 20 Initial Prestressing*


*Load Step 22 Self Weight*

*Load Step 23 Dead Load*

*Load Step 24 Surface Load*

*Load Step 25 Thermal Gradient*



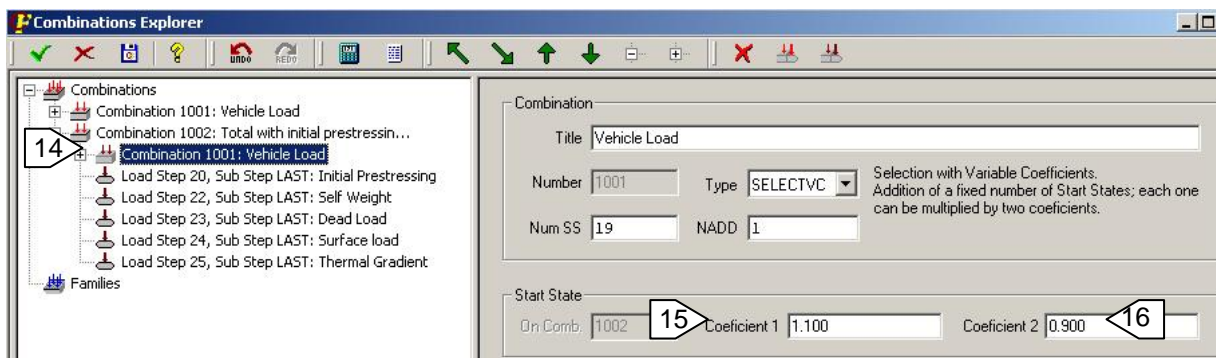
13 Press the contextual button in toolbar  to define a new start state

The default coefficients that these start states will have in the combination must also be defined:

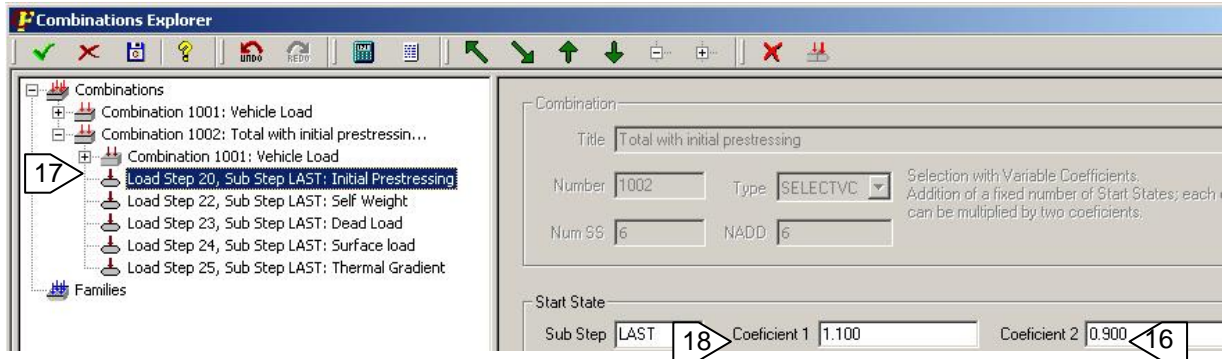
14 Select Combination 1001: Vehicle Load

15 Enter 1.10 as coefficient 1

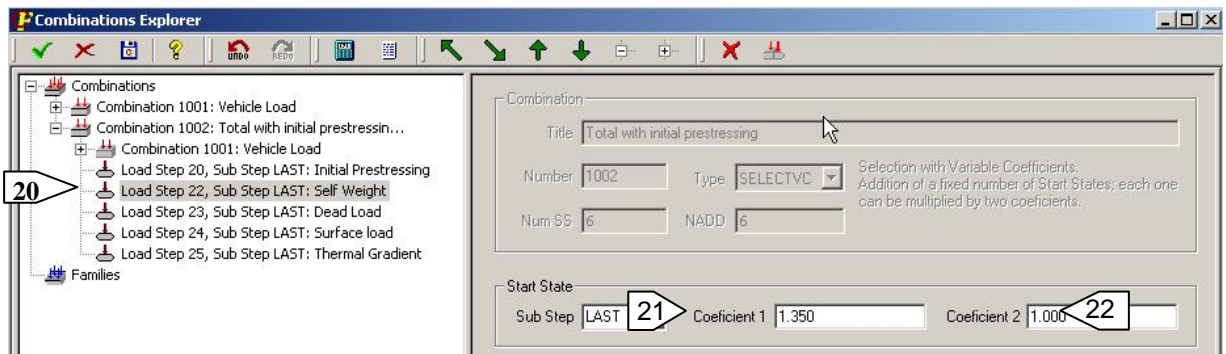
16 Enter 0.90 as coefficient 2



- 17 Select Load Step 20: Initial Prestressing
- 18 Enter 1.10 as coefficient 1
- 19 Enter 0.90 as coefficient 2



- 20 Select Load Step 22: Self Weight
- 21 Enter 1.35 as coefficient 1
- 22 Enter 1.00 as coefficient 2

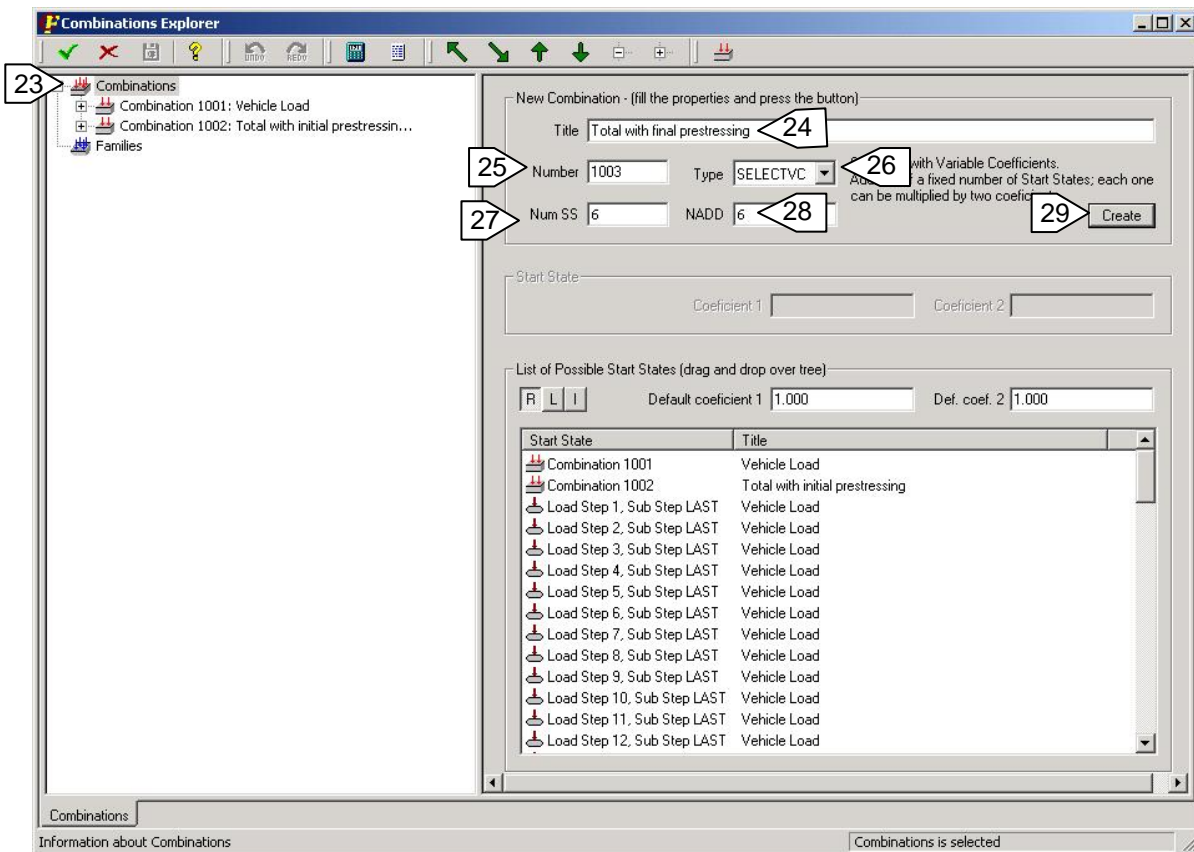


We will complete defining all coefficients by following this table:

Combination 1002 Total with Initial Prestressing		
Start State	Coefficient 1	Coefficient 2
CMB 1001 Vehicle Load	1.10	0.90
LS 20 Initial Prestressing	1.10	0.90
LS 22 Self Weight	1.35	1.00
LS 23 Dead Load	1.35	1.00
LS 24 Surface Load	1.50	0.00
LS 25 Thermal Gradient	1.35	1.00

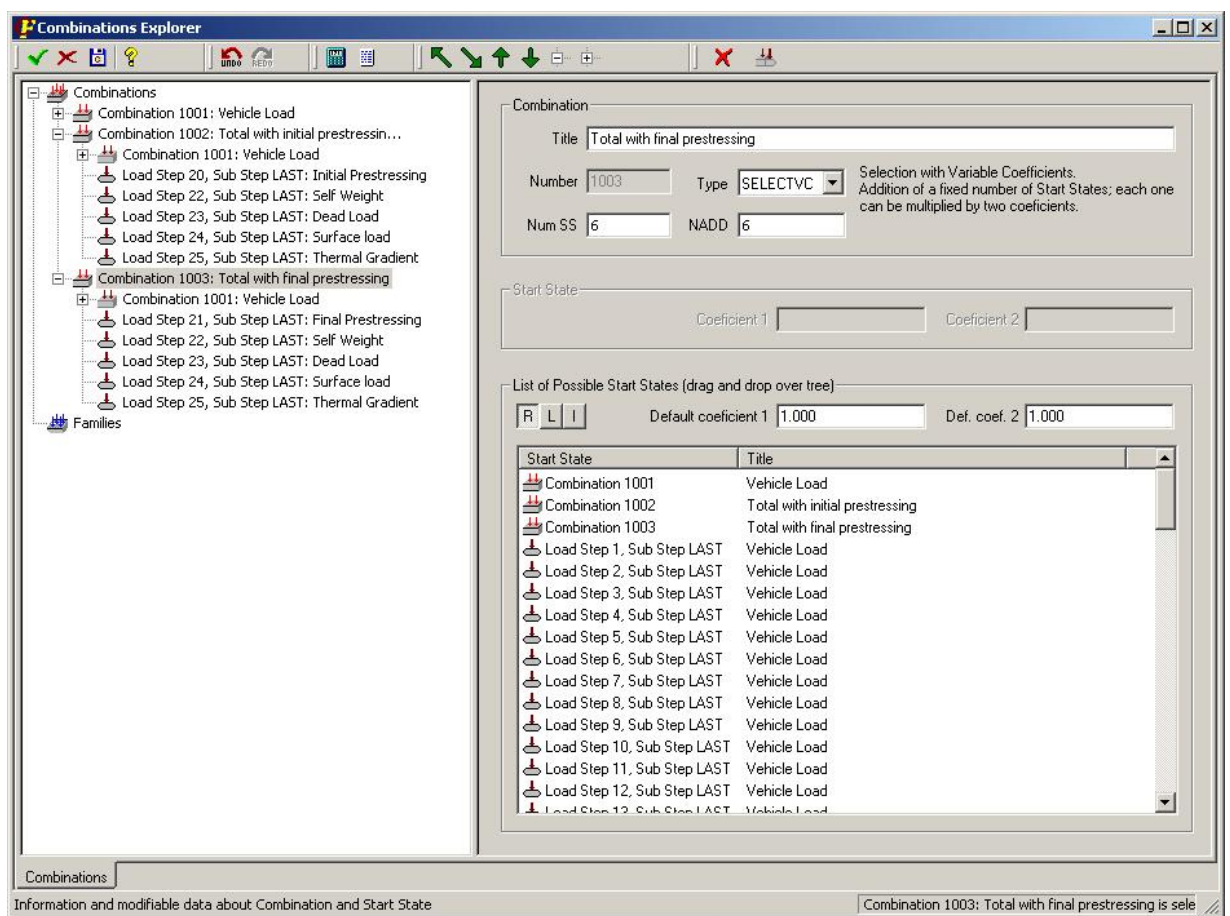
Now we define a new combination:

- 23 Pick on Combinations
- 24 Enter title "Total with final prestressing"
- 25 Enter combination number 1003
- 26 Choose type SELECTVC
- 27 Enter 6 as number of start states
- 28 Enter 6 as NADD
- 29 Click on Create



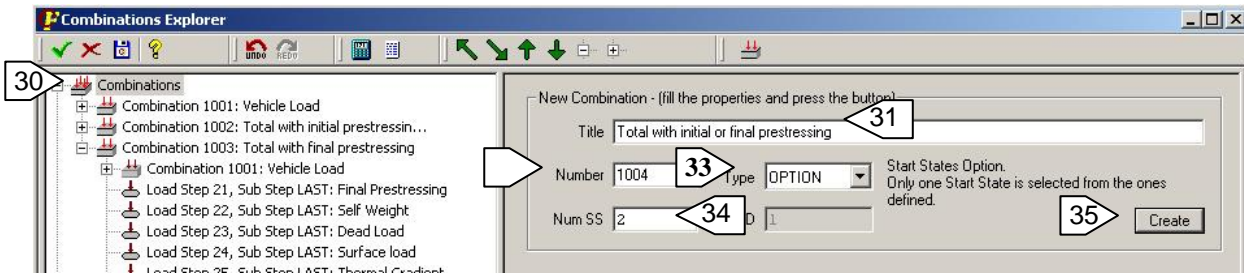
We follow the same procedure (as we did with combination rule 1002) to define all start states and coefficients repeating the same steps with this data:

Combination 1003 Total with Final Prestressing		
Start State	Coefficient 1	Coefficient 2
CMB 1001 Vehicle Load	1.10	0.90
LS 21 Final Prestressing	1.10	0.90
LS 22 Self Weight	1.35	1.00
LS 23 Dead Load	1.35	1.00
LS 24 Surface Load	1.50	0.00
LS 25 Thermal Gradient	1.35	1.00



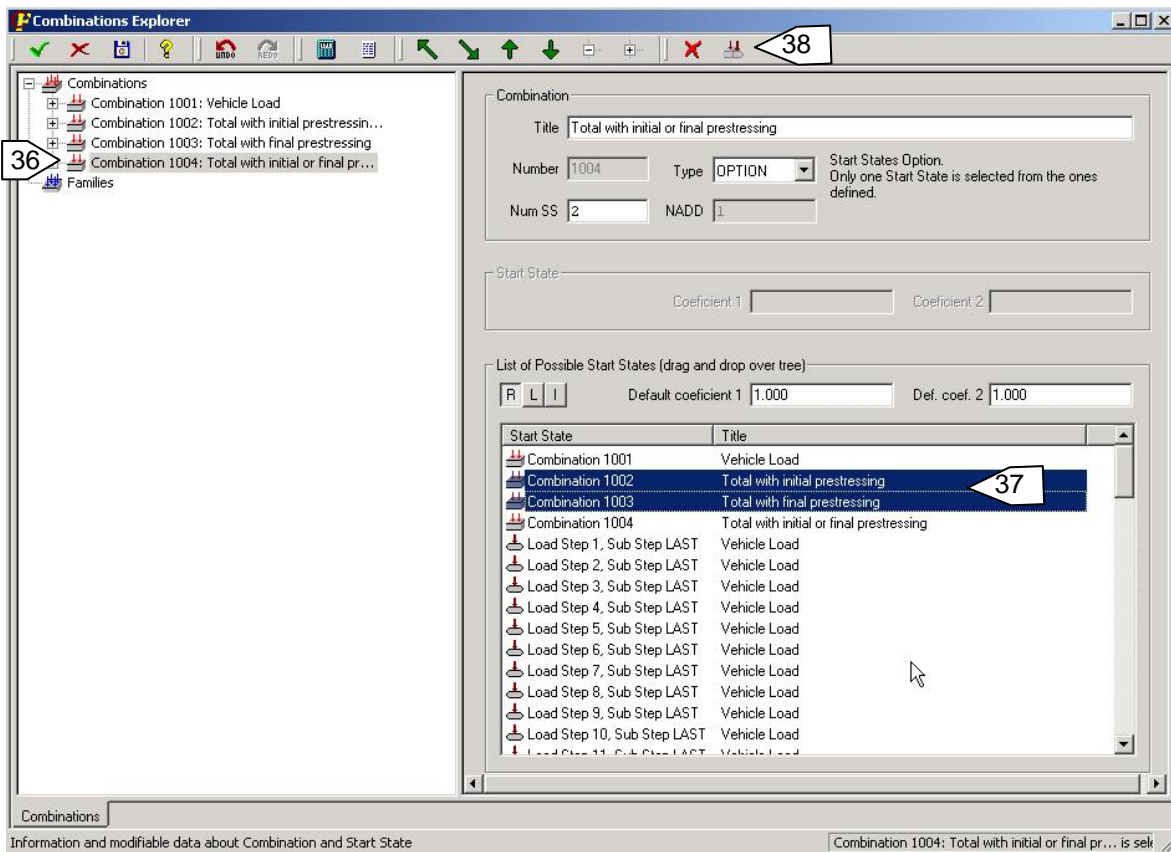
The final combination will be a combination of 1002 and 1003:


- 30 Pick on Combinations
- 31 Enter title "Total with initial or final prestressing"
- 32 Enter combination number 1004
- 33 Choose type OPTION
- 34 Enter 2 as number of start states
- 35 Click on Create



36 Select combination 1004

37 Select from *List of Possible Start States* ( hold control key [CTRL] ):  
Combination 1002 and Combination 1003



38 Press the contextual button in toolbar  to define the 2 start states.

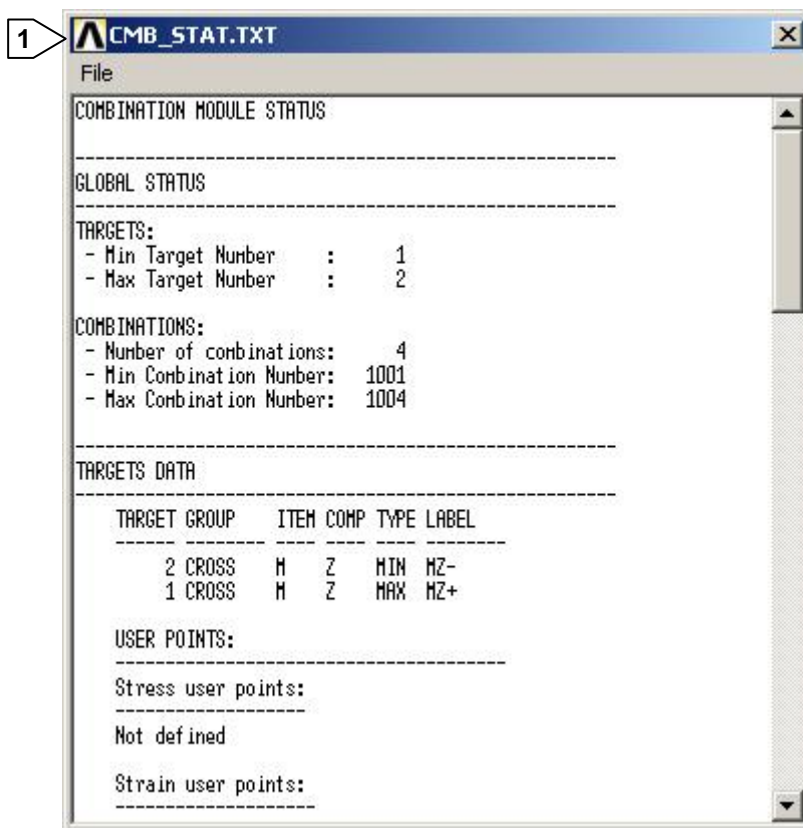
You can drag with the mouse the selected start states selected from the list, and drop them on the combination tree instead of clicking the button.

## 18. Carry out combinations

Prior to completing the combinations CivilFEM shows a window with the global status in order to review all data before carrying out the combinations.

Main Menu: **Civil Postprocessor** → **Combine Results** → **Combine** → **Combine for targets**

- 1 Review the information in the list window
- 2 OK to start the combination process

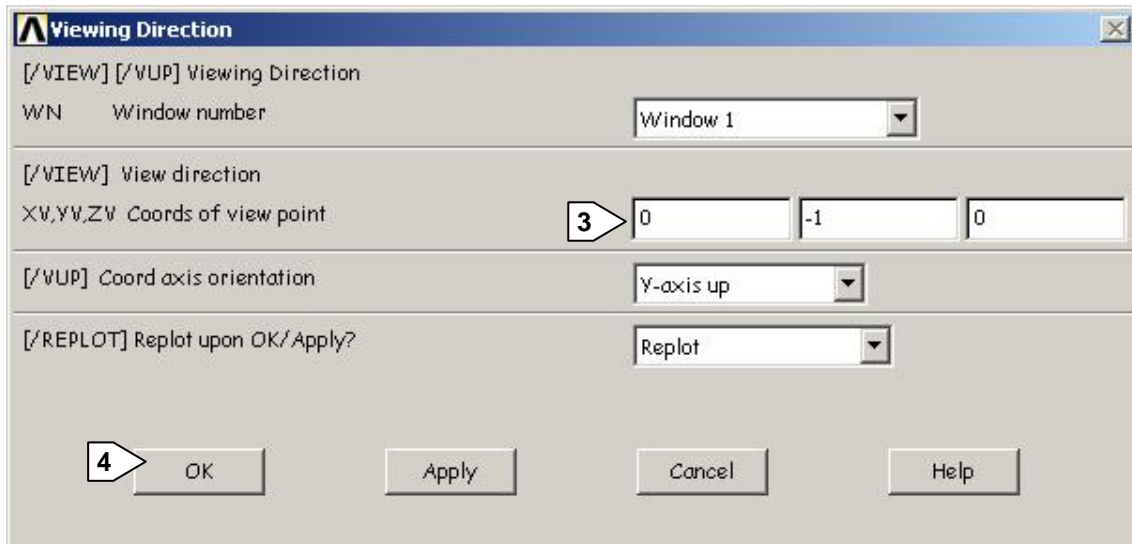


Before plotting results we define the viewing direction for the display:

Utility Menu: **PlotCtrls** → **View Settings** → **Viewing Direction**

3 Enter (0,-1,0) as coordinates of view point

4 Ok



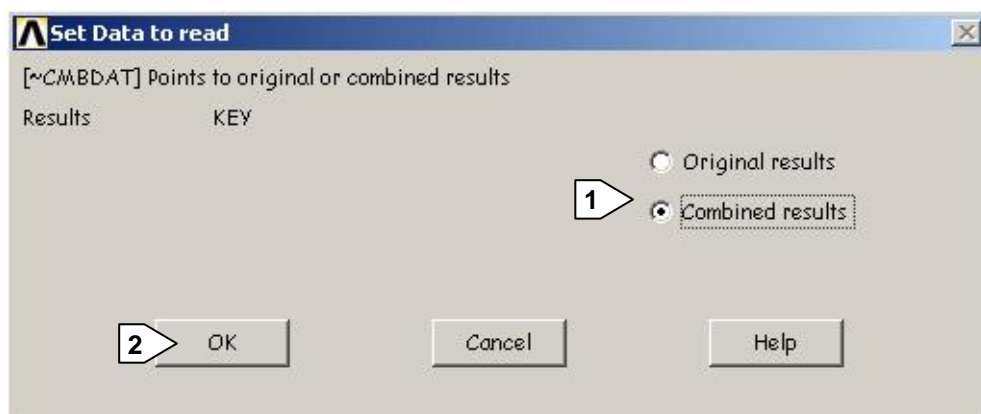
## 19. Read & Plot results

First of all we need to point to the combined results.

Main Menu: – CivilFEM – **Civil Postprocessor** → **Combine Results** → **Set data to read**

1 Select combined results

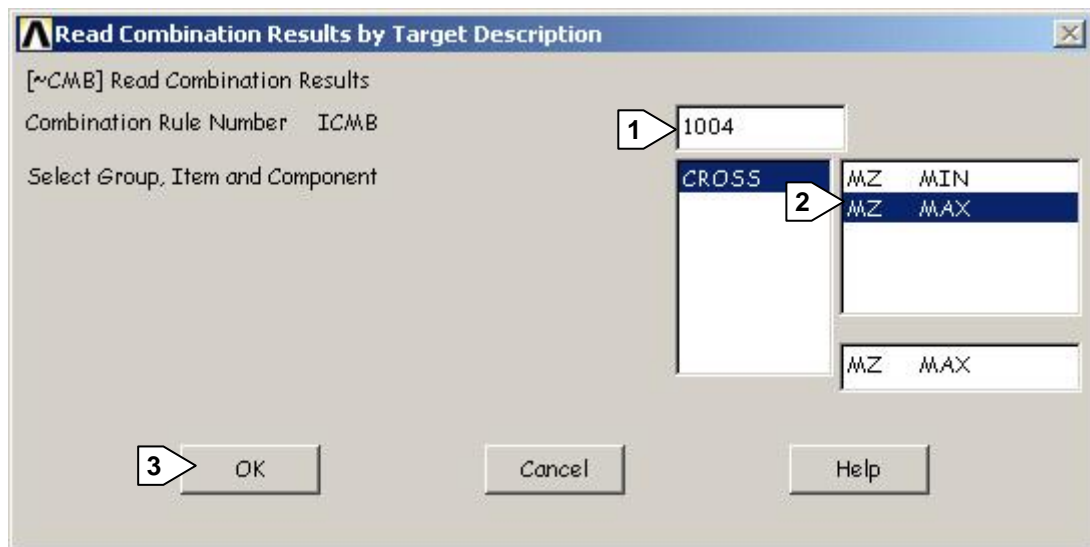
2 OK



We read the results of combination rule 1004 that satisfy the defined target.

Main Menu: – CivilFEM – **Civil Postprocess** → **Combine Results** → **By description...**

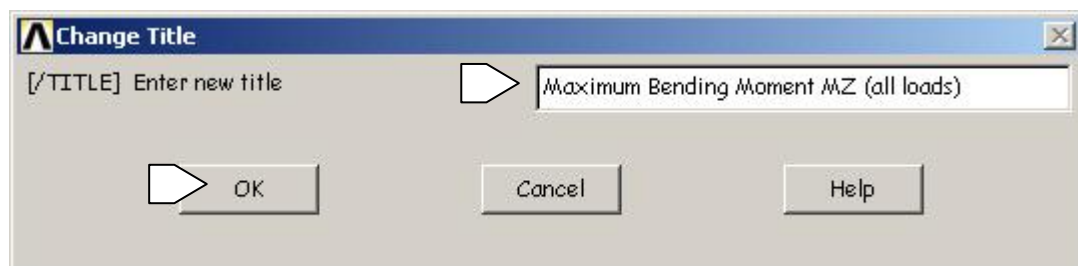
- 1 Enter 1004 to read result of combination rule 1004
- 2 Select the target MZ MAX
- 3 Ok



We change title:

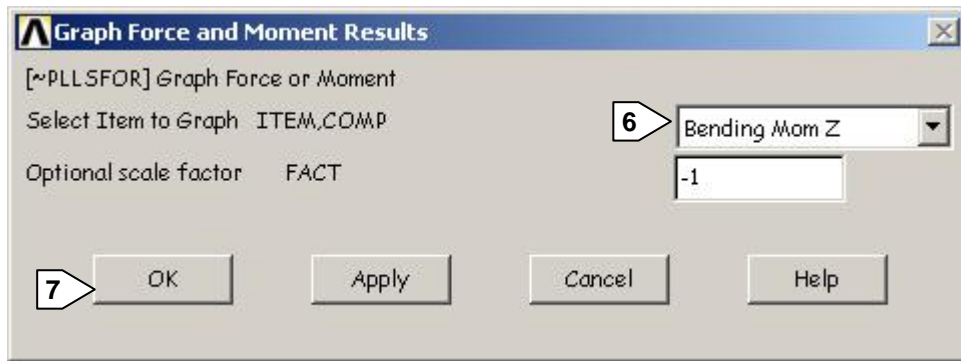
Utility Menu: **File** → **Change Title**

- Enter Maximum Bending Moment MZ (All loads)
- Ok



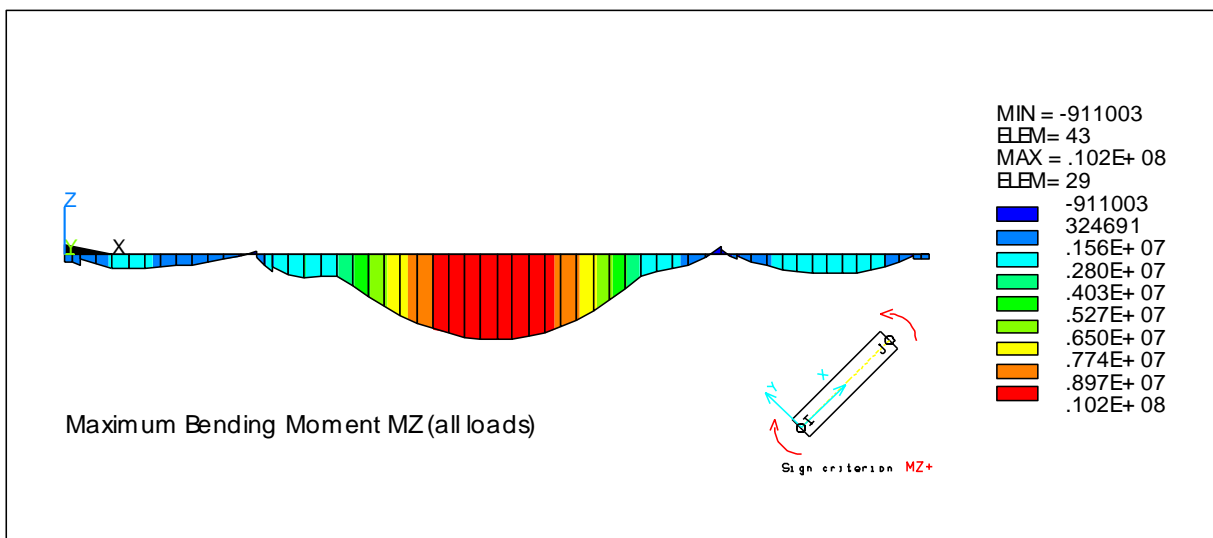
In the following step we plot the maximum bending moment MZ in each point of the bridge:

Main Menu: **Civil Postprocessor** → **Beam Utilities** → **Graph results** → **Forces & Moments**



6 Choose Bending Mom Z

7 Ok



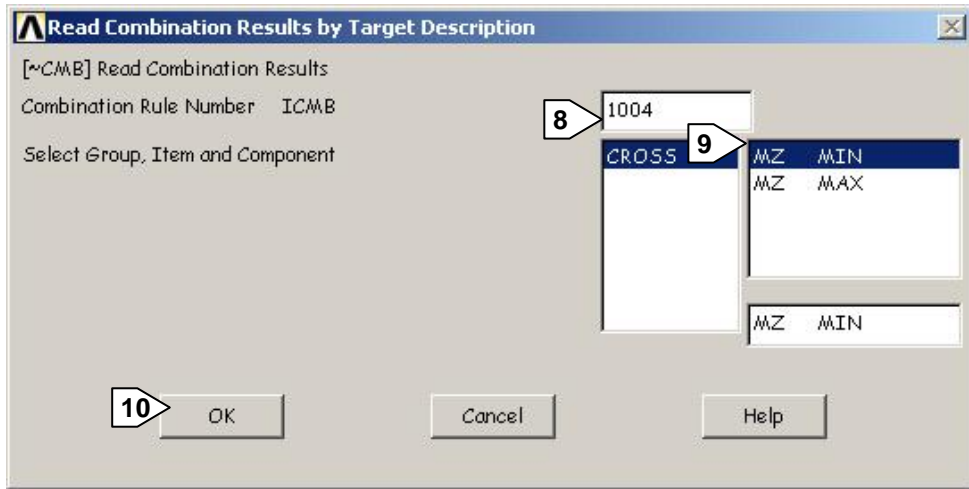
Now we point to target 2: minimum bending moment MZ:

Main Menu: – CivilFEM – **Civil Postprocess** → **Combine Results** → **By description...**

8 Enter 1004 to read result of combination rule 1004

9 Select the target MZ MIN

10 Ok

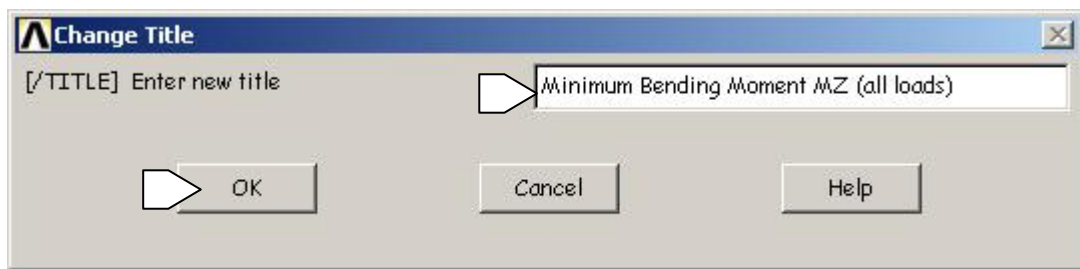


We change title:

Utility Menu: **File** → **Change Title**

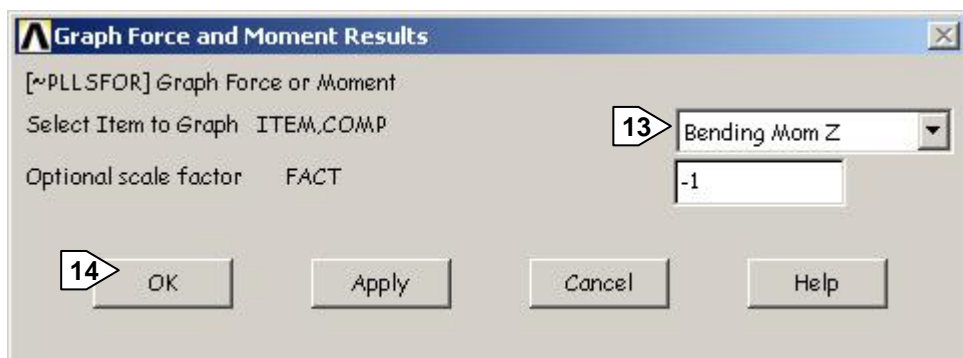
➤ Enter Minimum Bending Moment MZ (all loads)

➤ Ok



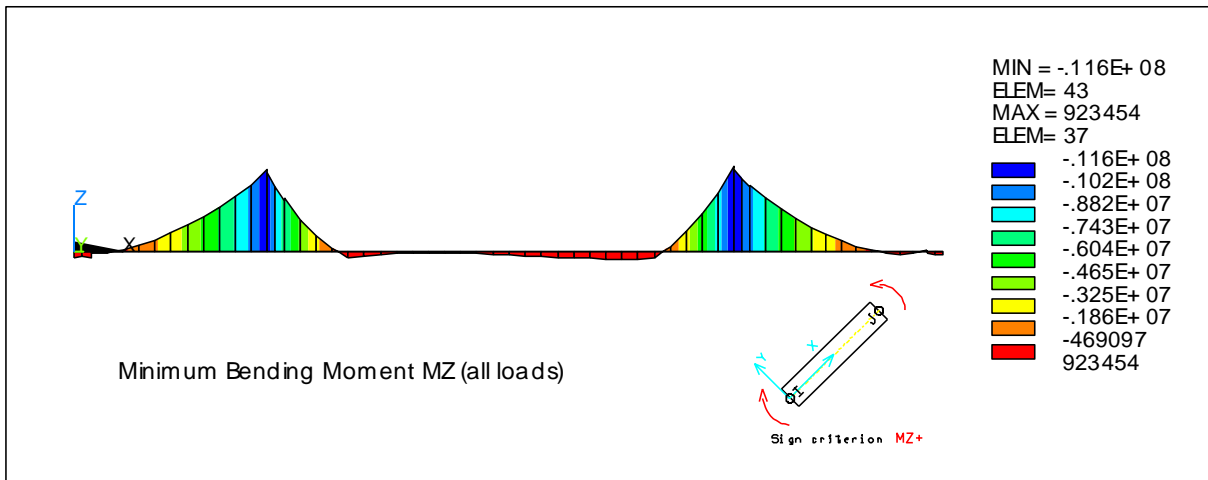
In the following step we plot the minimum bending moment MZ in each point of the bridge:

Main Menu: **Civil Postprocessor** → **Beam Utilities** → **Graph results** → **Forces & Moments**



13 > Choose Bending Mom Z

14 > Ok



## 20. Calculation of the load distribution to obtain the minimum deflection at a node (all loads)

We can know which are the coefficients that multiply the Start States of a combination and which of them can be used to achieve a Target at a determined node of the structure. The **~CMBINQ** command shows a list with the coefficients applied over each one of the Start States that forms the combination rule.

We change title:

Utility Menu: **File** → **Change Title**

> Type Maximum negative displacement and load location (all loads)

> Ok



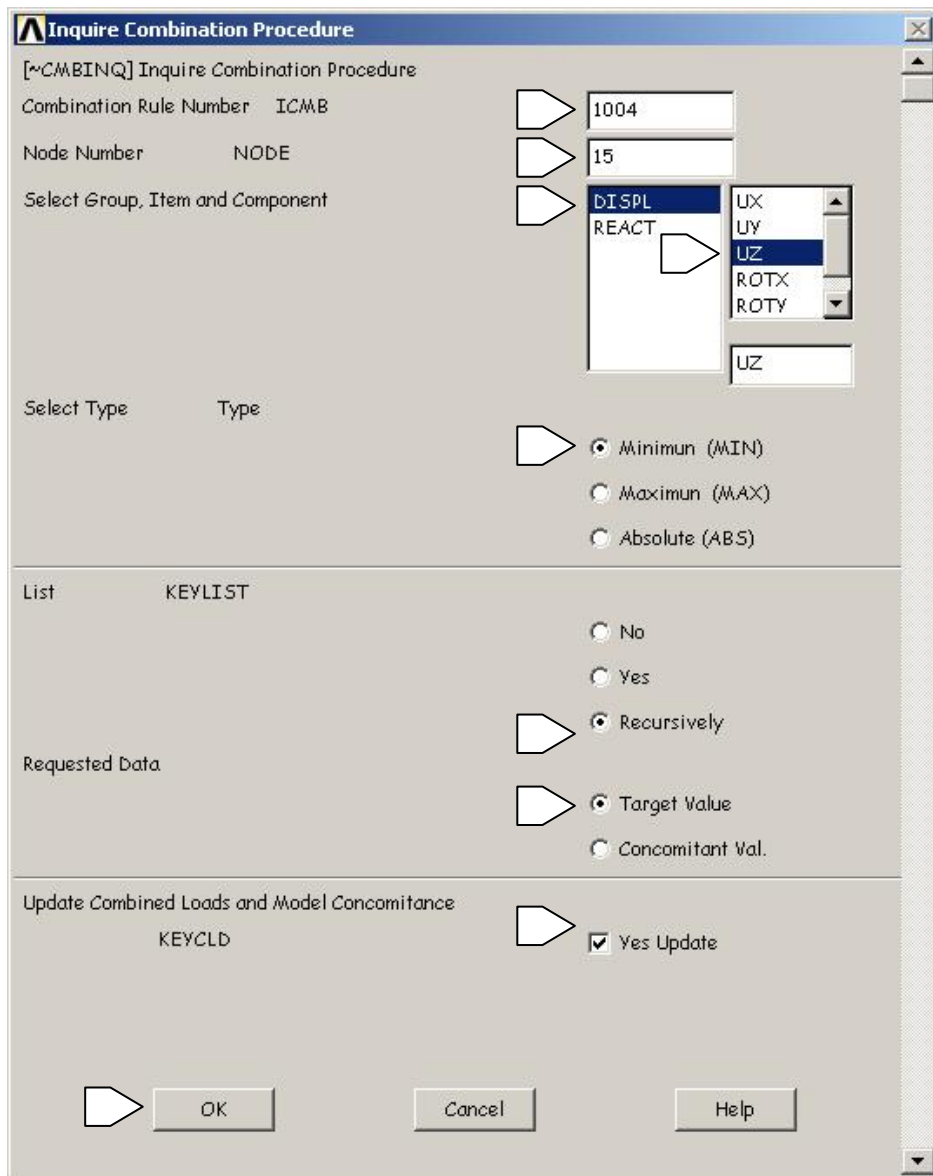
Main Menu: – CivilFEM – **Civil Postprocessor** → **Combine Results** → **INQUIRE: Nodal Results**

3 > Enter 1004 to select combination rule 1004 (all loads)

> Enter node 15

> Select group DISPL

- Select item and component UZ
- Pick Minimum
- Pick Recursively
- Pick Target Value
- Select Yes Update
- Ok



In the following window we can read a list of starting states that satisfy the defined target:

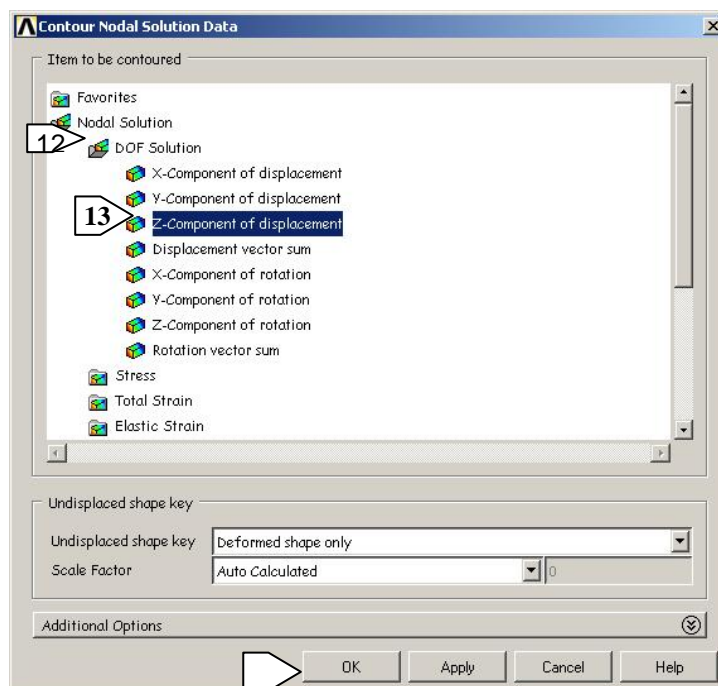
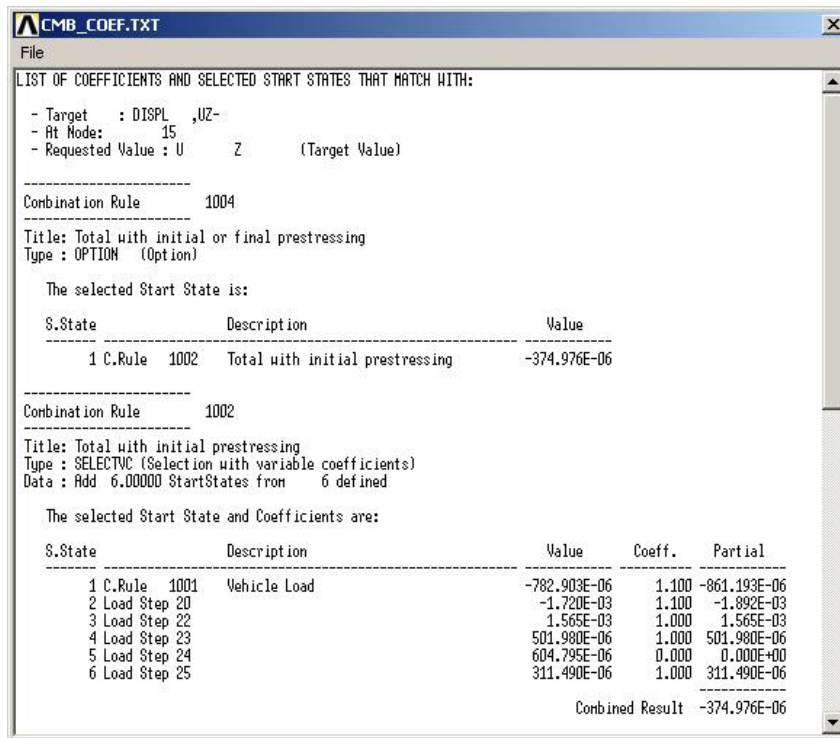
To obtain the deflections distribution for the start states combination that satisfy the requested targets we must proceed as follows:

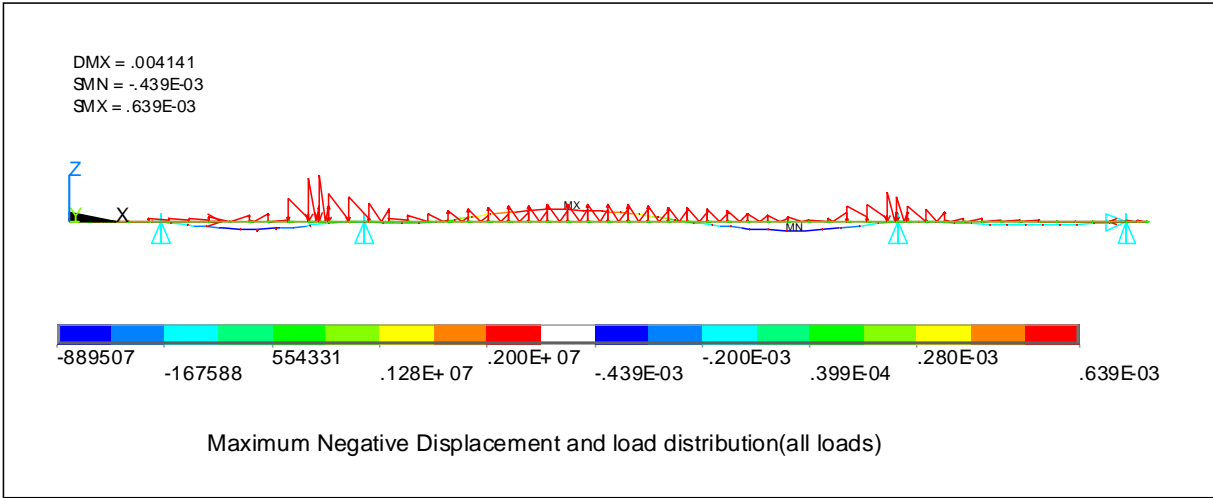
Main Menu: **General Postproc** → **Plot Results** → **-Contour Plot-Nodal Solu**

12 Select DOF solution

13 Select Z-Component of displacement

OK

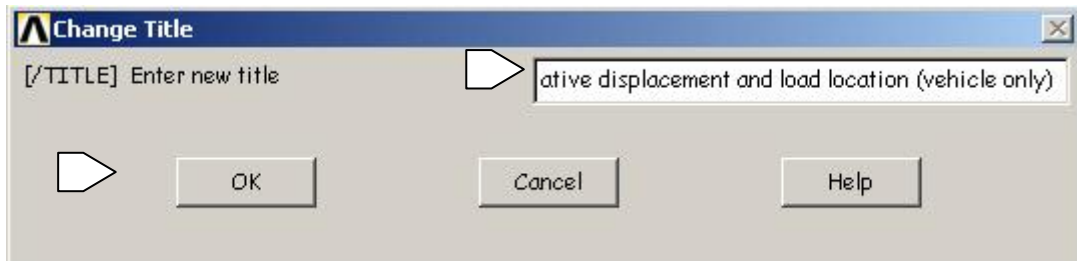




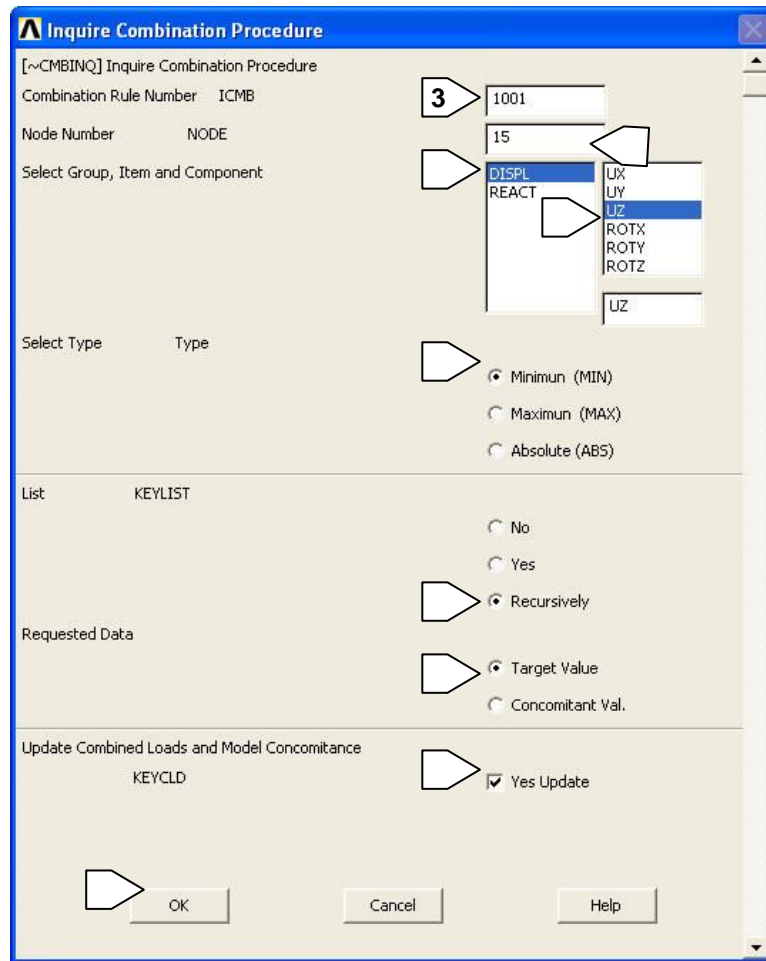
**21. Calculation of the load distribution to obtain the minimum deflection at a node (vehicle only)**

Utility Menu: **File** → **Change Title**

- ☞ Enter Maximum negative displacement and load location (vehicle only)
- ☞ Ok

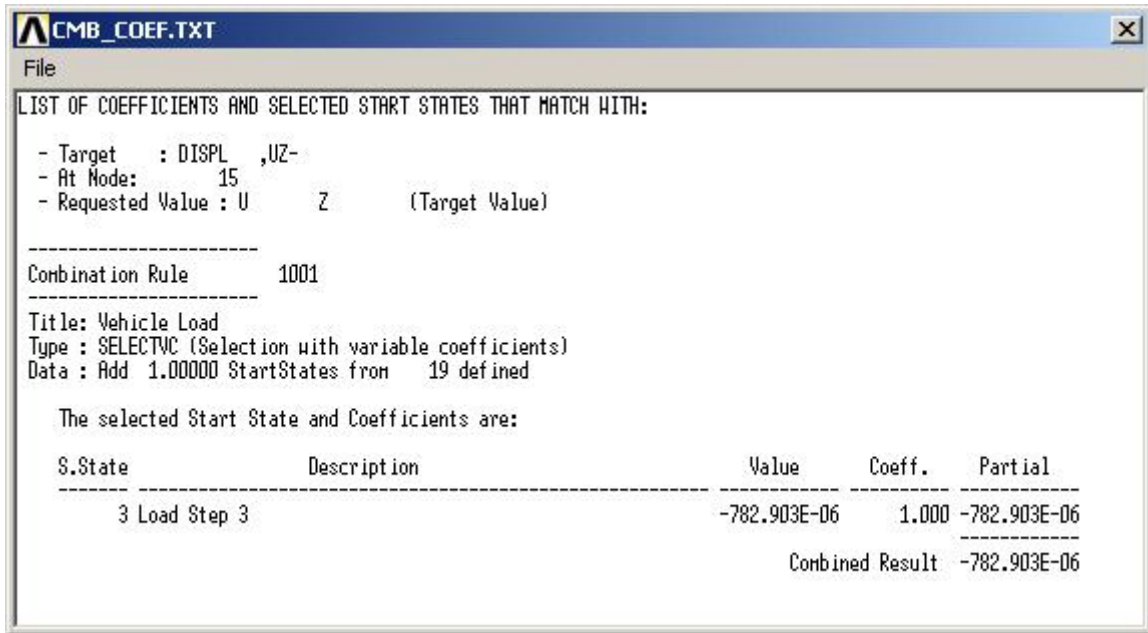


Main Menu: – CivilFEM – **Civil Postprocessor** → **Combine Results** → INQUIRE: **Nodal Results**



- 3 Enter 1001 to select combination rule 1001 (vehicle only)
- Enter node 15
- Select group DISPL
- Select item and component UZ
- Pick Minimum
- Pick Recursively
- Pick Target Value
- Select Yes Update
- Ok

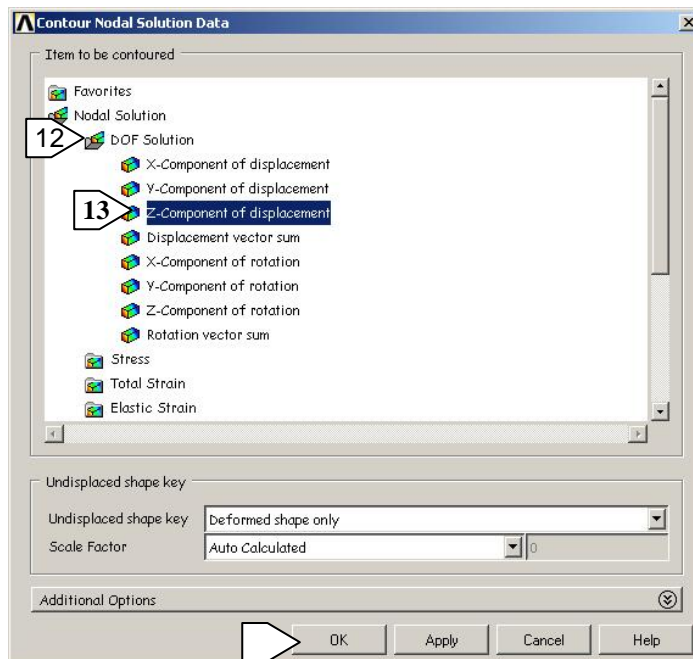
In the following window we can read a list of starting states that satisfy the defined target :

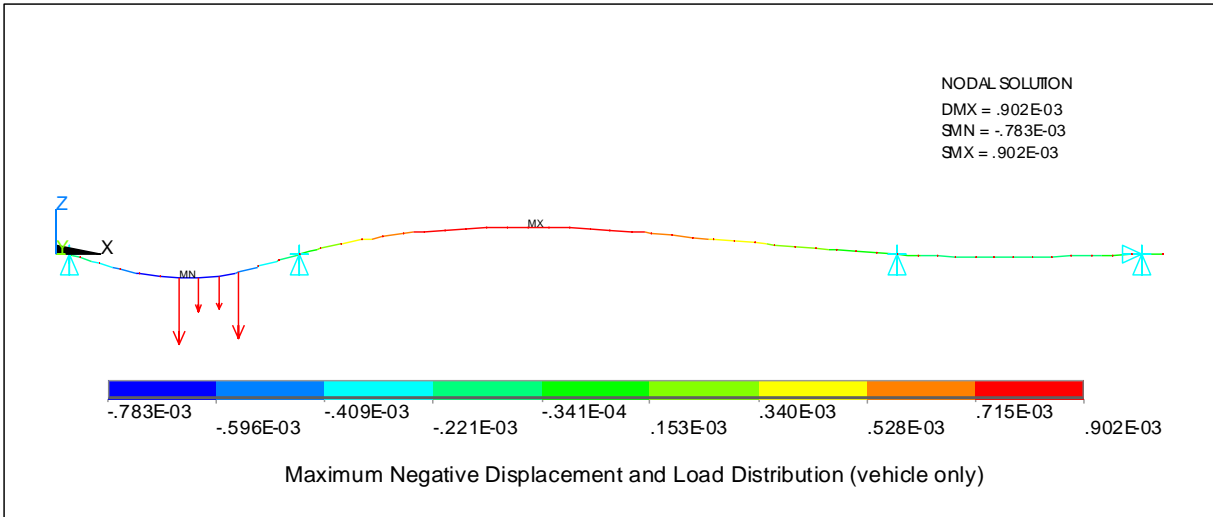


To obtain the deflections distribution for the start states combination that satisfy the requested targets we must proceed as follows:

Main Menu: **General Postproc** → **Plot Results** → **-Contour Plot-Nodal Solu**

- 12 Select DOF solution
- 13 Select Z-Component of displacement
- Ok





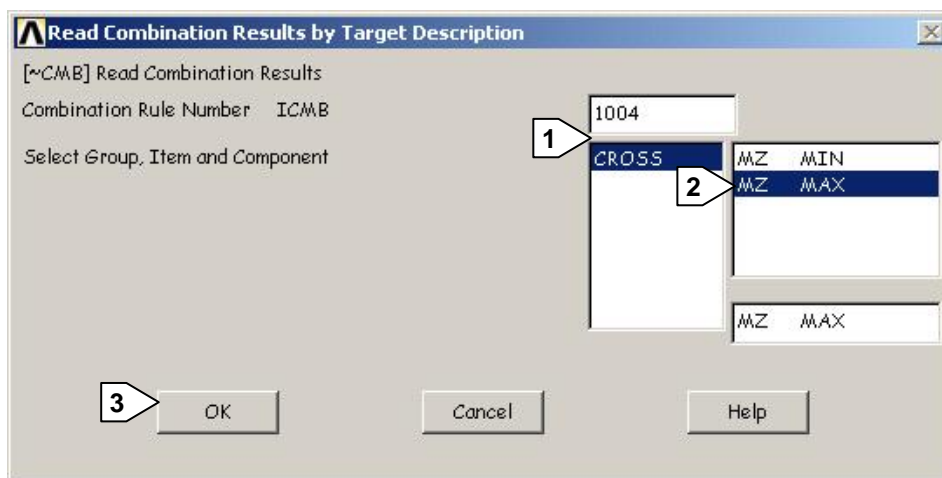
## 22. Check by code

### TARGET 1: MAX MZ

We first read the results of combination rule 1004, target 1 (maximum MZ)

Main Menu: – CivilFEM – **Civil Postprocess** → **Combine Results** → READ RESULTS: **By description...**

- 1 Enter 1004 to read results of combination rule 1004
- 2 Select the target MZ MAX
- 3 Ok



## 1) Cracking checking

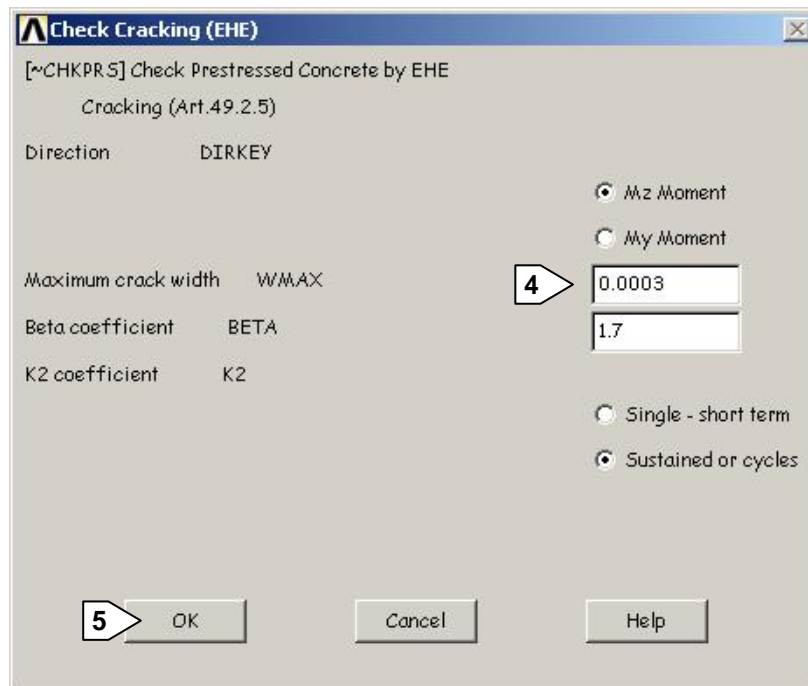
We will check cracking in prestressed cross sections according to EHE code. The checking consists of the verification of the decompression state or the crack width calculation.

Checking is made taking into account the bending according to the local cross section axis, as a user request, and the bending moment in the perpendicular direction.

Main Menu: – CivilFEM – **Civil Postprocessor** → **Prestress Concrete Post** → **EHE** → **Code Checking** → **EHE** → **Check by Code** → **Beams & Solid** → **Prestress Crack**

4 Enter maximum crack width 0.0003 m

5 Ok

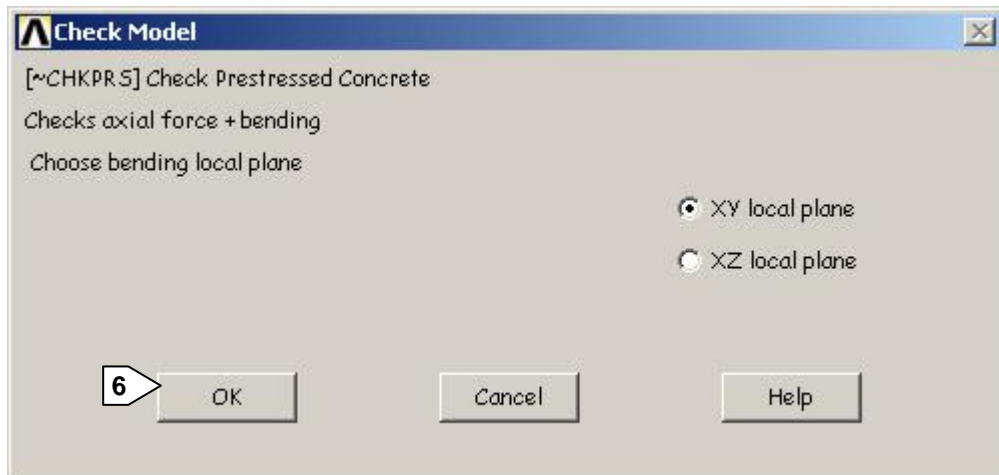


Results will be stored in Alternative 1

## 2) Axial force + bending checking

Main Menu: – CivilFEM – **Civil Postprocessor** → **Prestress Concrete Post** → **EHE** → **Check by Code** → **Beams & Solid** → **2D Axial + Bend**

6 Ok to check model



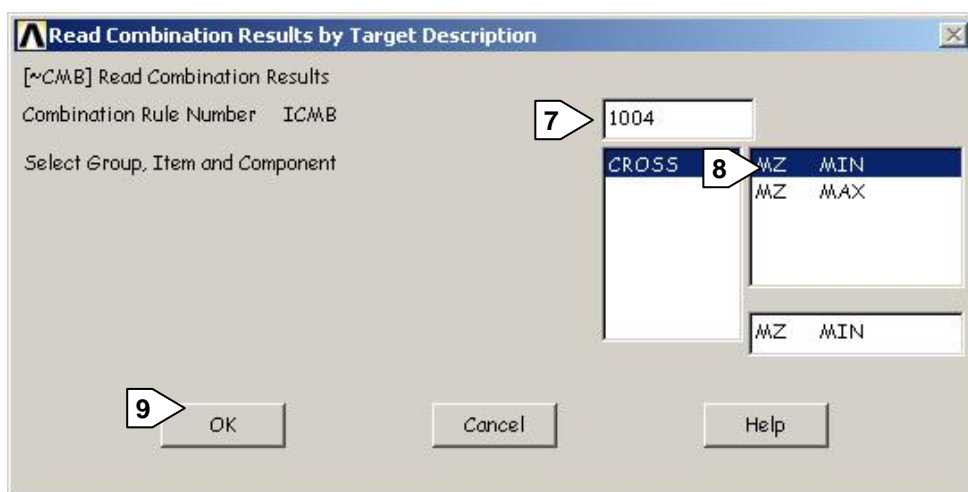
Results will be stored in Alternative 2

## TARGET 2: MIN MZ

Now we point to target 2: minimum bending moment MZ

Main Menu: – CivilFEM – **Civil Postprocess** → **Combine Results** → READ RESULTS: **By description...**

- 7 Enter 1004 to read results of combination rule 1004
- 8 Select the target MZ MIN
- 9 Ok

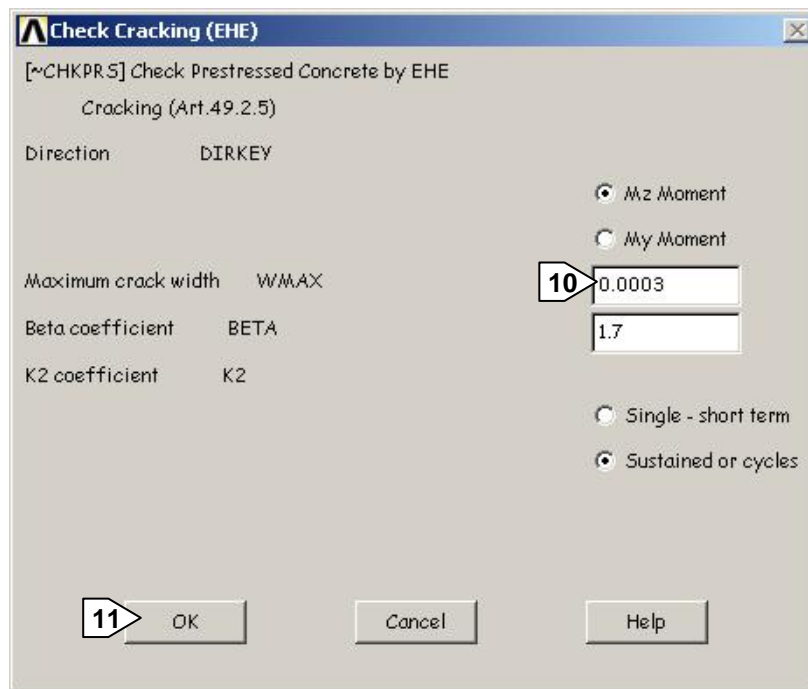


### 3) Cracking checking

Main Menu: – CivilFEM – **Civil Postprocessor** → **Prestress Concrete Post** → **EHE** → **Check by Code** → **Beams & Solid** → **Prestress Crack**

10 Enter maximum crack width 0.0003 m

11 Ok

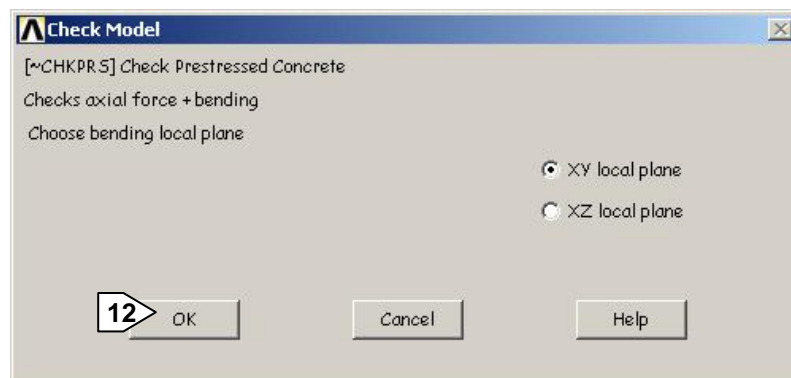


Results will be stored in Alternative 3

### 4) Axial force + bending checking

Main Menu: – CivilFEM – **Civil Postprocessor** → **Prestress Concrete Post** → **EHE** → **Check by Code** → **Beams & Solid** → **2D Axial + Bend**

12 Ok to check model



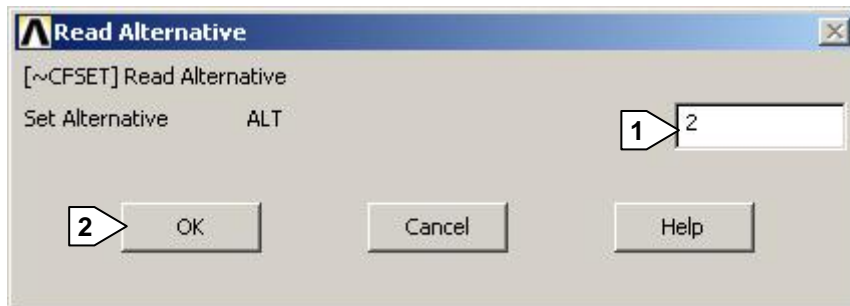
Results will be stored in Alternative 4

### 23. Plot Results: Axial+Bending checking for target 1 (Maximum MZ)

User must select the alternative of results calculated before:

Main Menu: – CivilFEM – **Civil Postprocess** → **Read Results** → **By Num. Alt.**

- 1 Enter alternative number 2
- 2 Ok

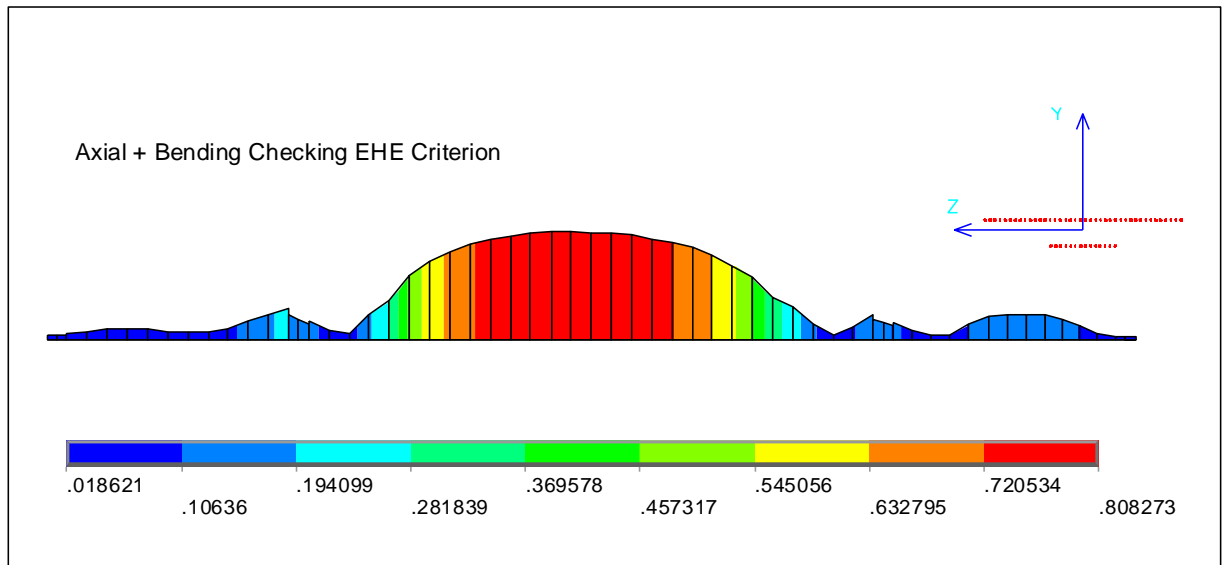


To plots crack checking results in prestressed concrete beams:

Main Menu: – CivilFEM – **Civil Postprocessor** → **Prestress Concrete Post** → **Code Checking** → **EHE** → **Beam Results** → **Plot Results**

- 3 Select Axial + bending
- 4 Select CRT\_TOT
- 5 Ok





The represented result of checking is a criterion that indicates the proximity or distance to the limits established by the code. An equal to one criterion indicates that the forces and moments are equal to the limits specified by the code. A criterion greater than one indicates that the code is not being fulfilled and a criterion smaller than one indicates that the code requirements are satisfied.

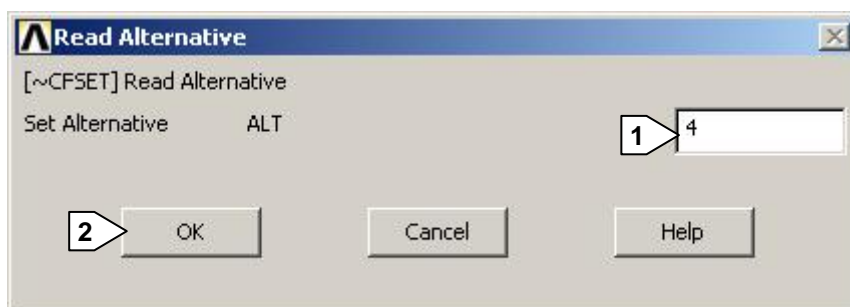
## 24. Plot Results: Axial+Bending checking for target 2 (Minimum MZ)

User must select the alternative of results calculated before:

Main Menu: – CivilFEM – **Civil Postprocess** → **Read Results** → **By Num. Alt.**

1 Enter alternative number 4

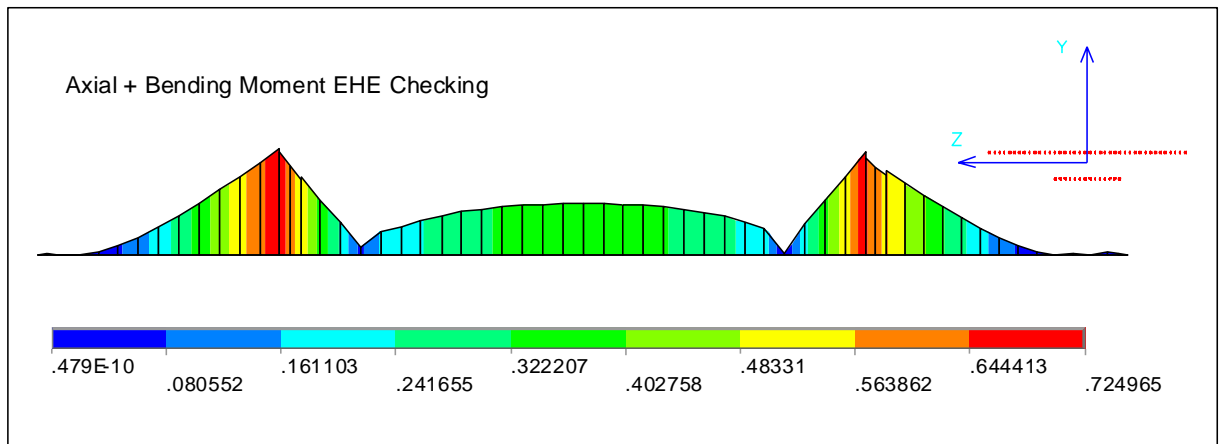
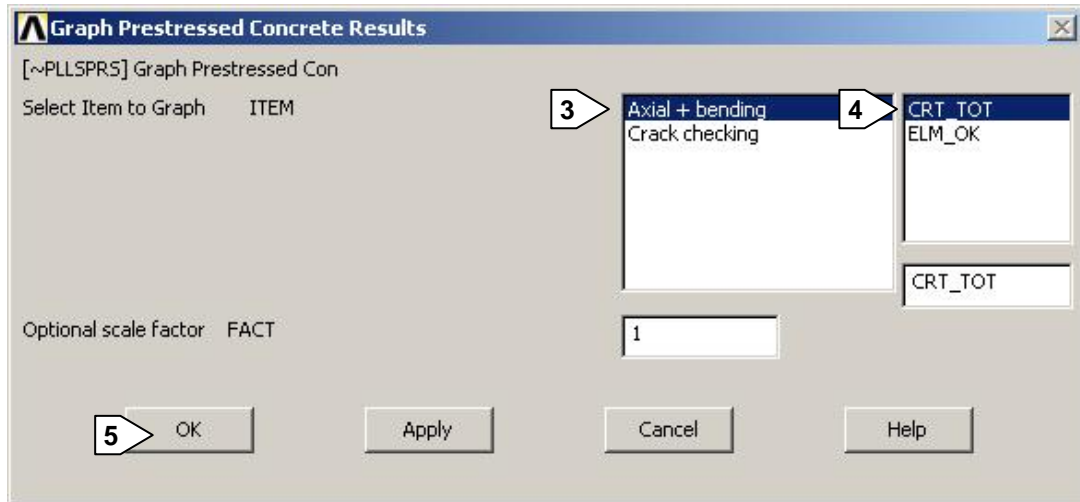
2 Ok



To plots crack checking results in prestressed concrete beams:

Main Menu: – CivilFEM – **Civil Postprocessor** → **Prestress Concrete Post** → **Code Checking** → **EHE** → **Beam Results** → **Plot Results**

- 3 Select Axial + bending
- 4 Select CRT\_TOT
- 5 Ok



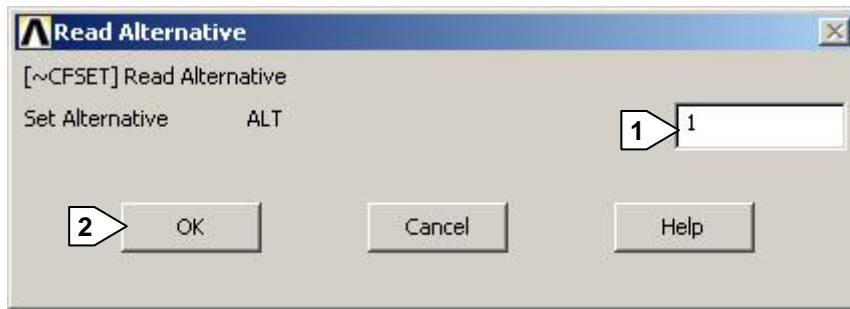
The represented result of checking is a criterion that indicates the proximity or distance to the limits established by the code. An equal to one criterion indicates that the forces and moments are equal to the limits specified by the code. A criterion greater than one indicates that the code is not being fulfilled and a criterion smaller than one indicates that the code requirements are satisfied.

## 25. Plot Results: Crack checking for target 1 (Maximum MZ)

User must select the alternative of results calculated before:

Main Menu: – CivilFEM – **Civil Postprocess** → **Read Results** → **By Num. Alt.**

- 1 Enter alternative number 1
- 2 Ok



To plots crack checking results in prestressed concrete beams:

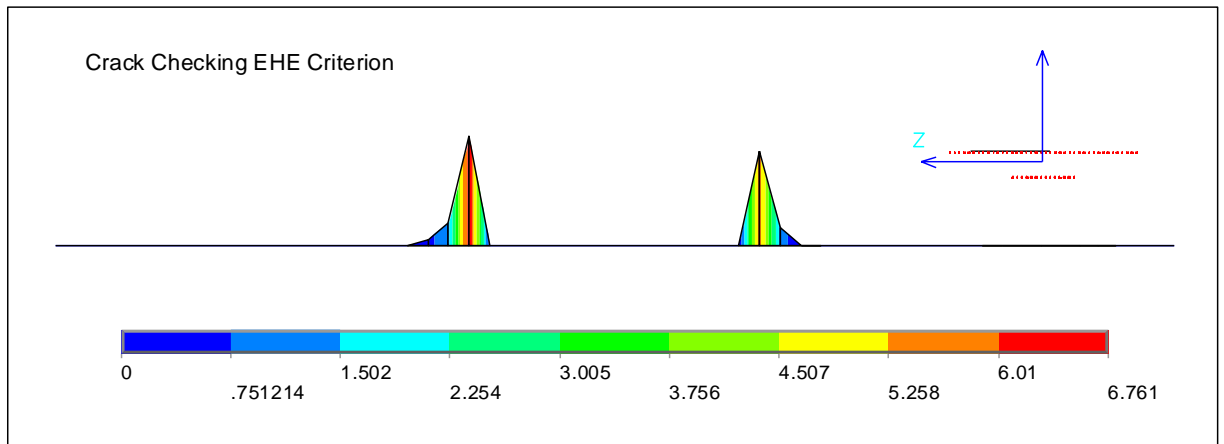
Main Menu: – CivilFEM – **Civil Postprocessor** → **Prestress Concrete Post** → **Code Checking** → **EHE** → **Beam Results** → **Plot Results**

3 Select Crack Checking

4 Select CRT\_TOT

5 Ok





In previous graph the EHE Criterion for crack checking is plotted. Elements with a criterion (CRT\_TOT) greater than 1 do not satisfy the code provisions.

This means that the reinforcement indicated in the statement would not be valid in order to fulfil the established cracking criterium. In order to improve the results it is not necessary to increase the reinforcement area, because the bending checking was correct. The best solution, in this case, would be to decrease the bars spacing.

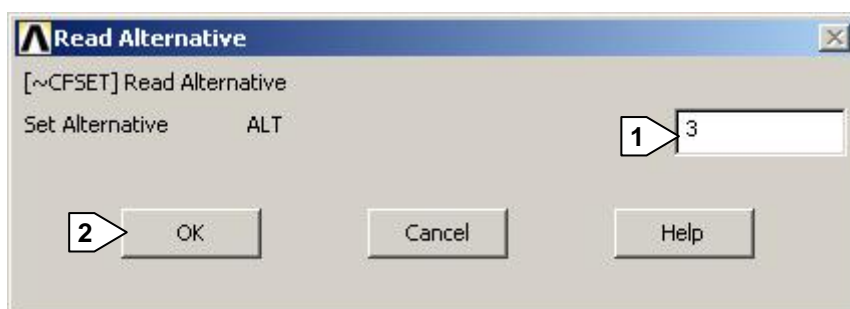
## 26. Plot Results: Crack checking for target 2 (Minimum MZ)

User must select the alternative of results calculated before:

Main Menu: – CivilFEM – **Civil Postprocess** → **Read Results** → **By Num. Alt.**

1 Enter alternative number 3

2 Ok



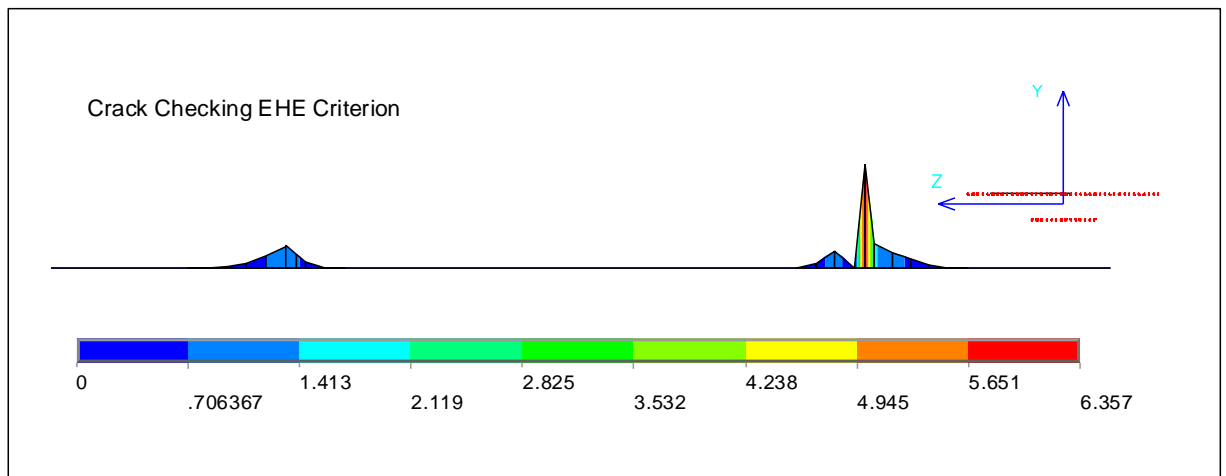
To plots crack checking results in prestressed concrete beams:

Main Menu: – CivilFEM – **Civil Postprocessor** → **Prestress Concrete Post** → **Code Checking** → **EHE** → **Beam Results** → **Plot Results**

3 Select Crack Checking

4 Select CRT\_TOT

5 Ok



In previous graph the EHE Criterion for crack checking is plotted. Elements with a criterion (CRT\_TOT) greater than 1 do not satisfy the code provisions.

This means that the reinforcement indicated in the statement would not be valid in order to fulfil the established cracking criterium. In order to improve the results it is not necessary to increase the reinforcement area, because the bending checking was correct. The best solution, in this case, would be to decrease the bars spacing.

## 27. Interaction Diagram

The interaction diagram is a surface in space (F,M) that contains the forces and moments corresponding to the section's ultimate strength states. In CivilFEM the ultimate strength states are determined through the pivots diagram.

To plot the 2D interaction diagram:

Main Menu: – CivilFEM – **Civil Postprocessor** → **Prestress Concrete Post** →EHE  
→**Beam Results**→ **Prestress 2D Int Diag**

13 Select Beam as entity type

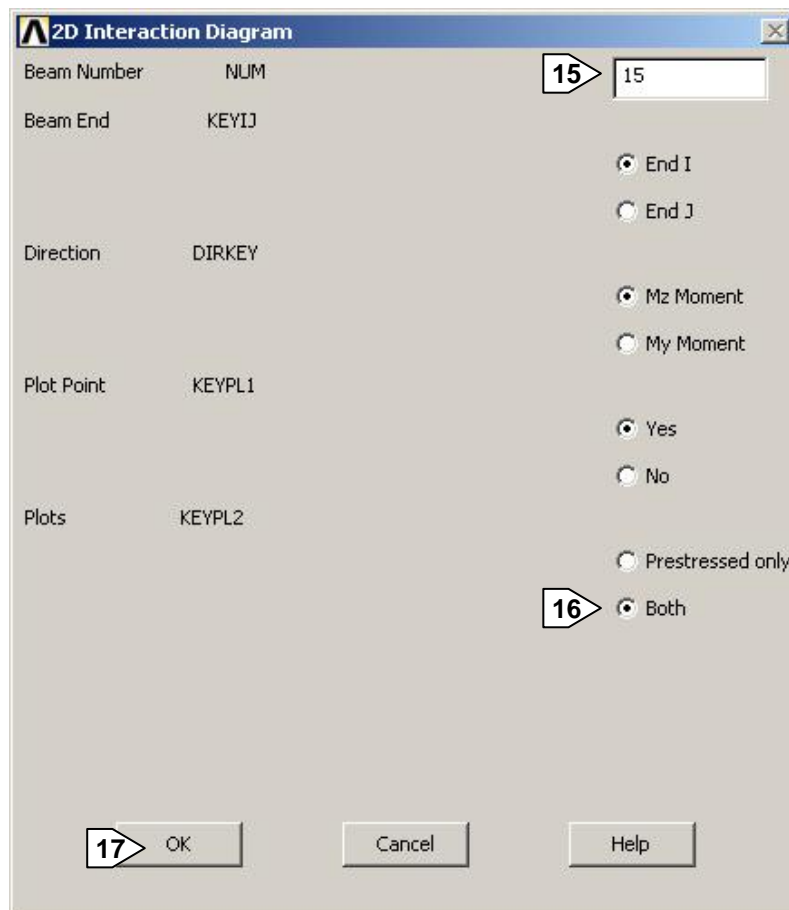
14 Ok



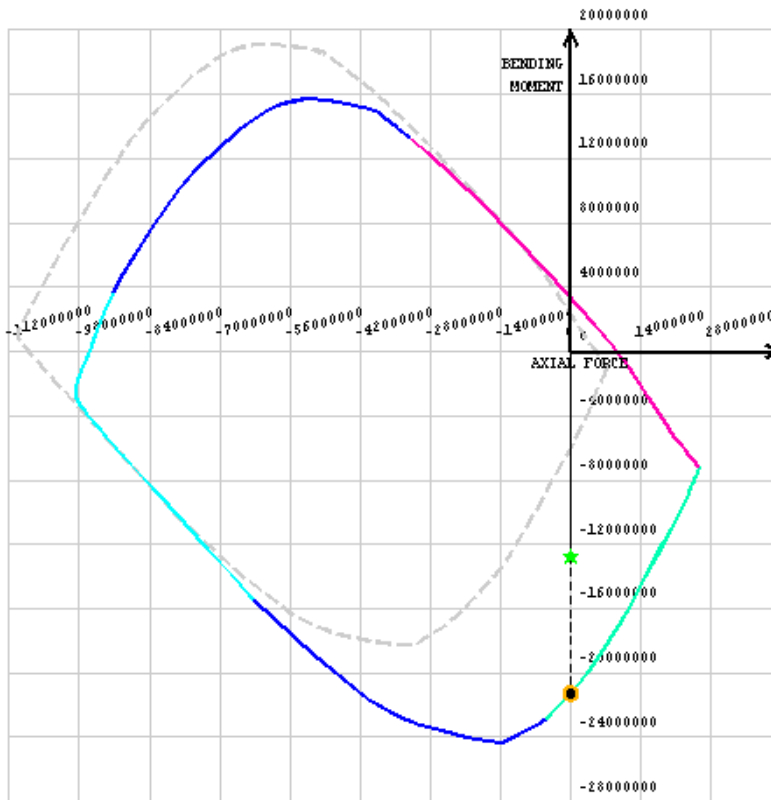
15 Enter element number 15

16 Select Both to add reinforced concrete interaction diagram and Ok

17 Ok



COMBINED AXIAL + BENDING ULTIMATE STRENGTH (PRSC)



LOAD STEP: 1004  
SUBSTEP: 2  
ALT: 2

CROSS SECTION: 5

EFFECTS OF ACTIONS ★  
 $N_x = 0.0379955098$   
 $M_y = 0$   
 $M_z = -12810074.6$   
 $M^* = -12810074.6$   
 $M^*/N_x = -237147066$

ULTIMATE STRENGTH ●  
 $N_u = 0.06307295756$   
 $M_u = -21264862.6$   
 $M_u/N_u = -237147066$

CRITERIUM  
 CRT\_TOT = 0.6  
CROSS SECTION IS OK

FAILURE CRITERIA	
---	WITHOUT PRESTRESSING TENDONS
---	STEEL YIELD
---	PRESTRESSING STEEL YIELD
---	EXTERNAL PIVOT CONCR CRUSH
---	INTERNAL PIVOT CONCR CRUSH

Vehicle Load

## 28. Exit the ANSYS program

We save everything before exiting the ANSYS program.

Utility Menu: **File** → **Exit**

- 1 Choose Save Everything
- 2 Choose OK

