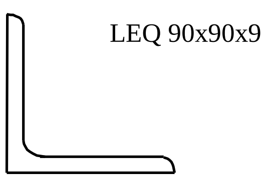
3. Check by Eurocode 3 a Steel Truss

Applicable CivilFEM Product:	All CivilFEM Products
Level of Difficulty:	Moderate
Interactive Time Required:	40 minutes
Discipline:	Structural Steel
Analysis Type:	Linear static
Element Type Used:	LINK 1
Active Code:	Eurocode 3
Units System:	N, m, s
CivilFEM Features Demonstrated:	Units selection, code selection, material definition, section and code properties definition, checking according Eurocode 3 and results postprocessing.

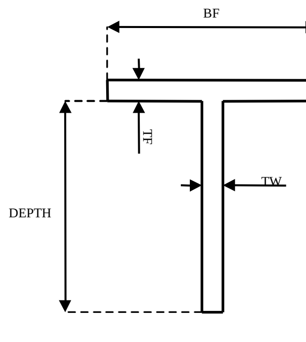
Problem Description

■ Given

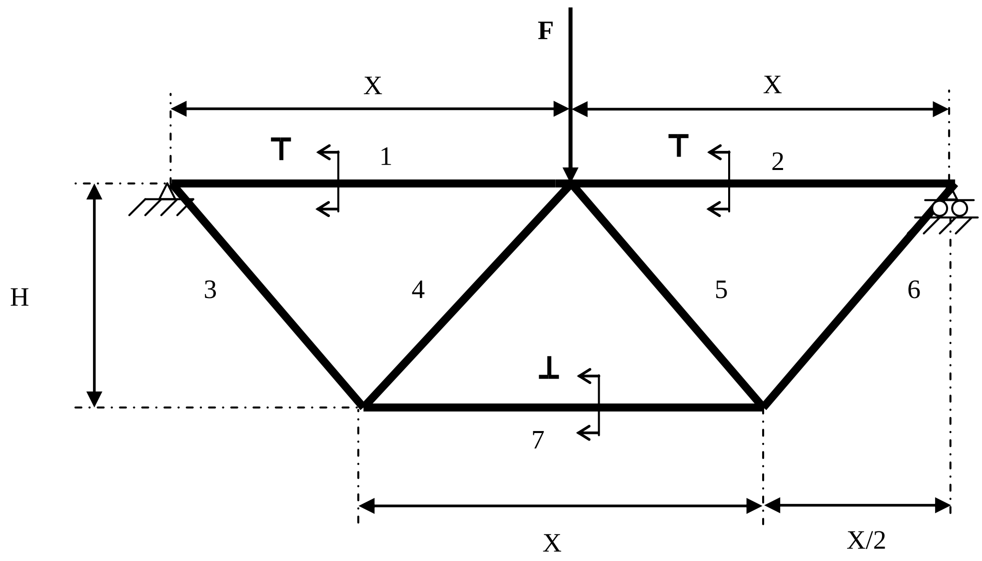
This is a typical ANSYS/CivilFEM analysis; a 2D steel truss subjected to a force applied at mid span. In this example we will introduce you to checking according to Eurocode 3. We will use two different steel sections, which are shown in the figure below, one defined from library and the other one by dimensions.



Section 1



Section 2



Section 1 (L) bars:

Bar 3
Bar 4
Bar 5
Bar 6

Section 2 (T) bars:

Bar 1 T
Bar 2 T
Bar 7 ⊥

Material	Steel Fe 430
Loads	F = 500 kN
Section 1	European L EQ 90x90x9
Section 2	Welded T section
Truss Geometric Dimensions:	
	X = 2.5 m
	H = 1.5 m

Section 2 (T) Dimensions:

DEPHT = 0.15 m

TW = 0.01 m

BF = 0.1 m

TF = 0.01 m

Approach and Assumptions

This is a static analysis with 2D elastic elements and elastic material properties. Model geometry is defined with nodes and elements.

Summary of Steps

Preprocessing

1. Preprocessing
2. Specify title
3. Set code
4. Set units
5. Define material
6. Define element type
7. Define Section
8. Define member properties
9. Define Beam & Shell properties
10. Define nodes and elements
11. Save the database

Solution

12. Apply displacement constrain
13. Apply force load
14. Solve

Postprocessing

15. Enter the postprocessor and read results
16. Plot Axial Force X
17. Checking for Tension according to Eurocode 3
18. Review Elements OK and Not OK
19. List Eurocode 3 Criterion Results
20. Check for Compression according to Eurocode 3
21. Review Elements OK and Not OK
22. List Eurocode 3 Criterion Results
23. Check for Buckling Compression
24. Review Elements OK and Not OK
25. List Eurocode 3 Criterion Results
26. 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.

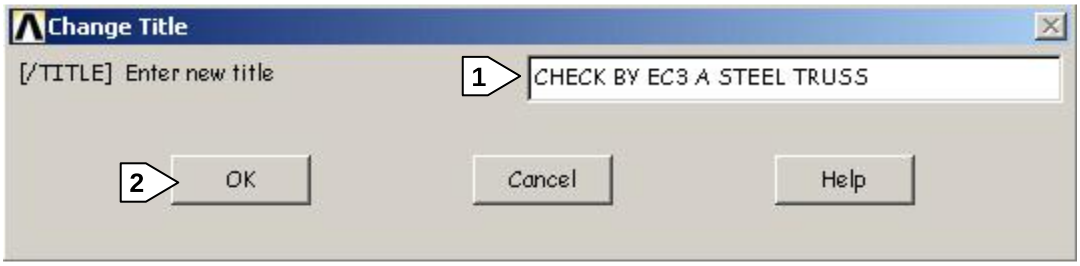
2. Specify title

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: CHECK BY EC3 A STEEL TRUSS

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



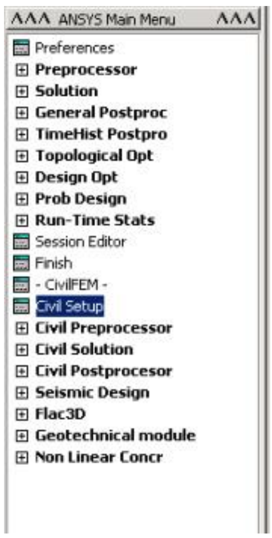
3. Set code

In CivilFEM you can choose between different codes for checking and designing. CivilFEM allows you to uphold different active codes simultaneously, one for concrete calculations another one for steel calculations and a third one for seismic design. In this example the active code is Eurocode 3, which is the default option.

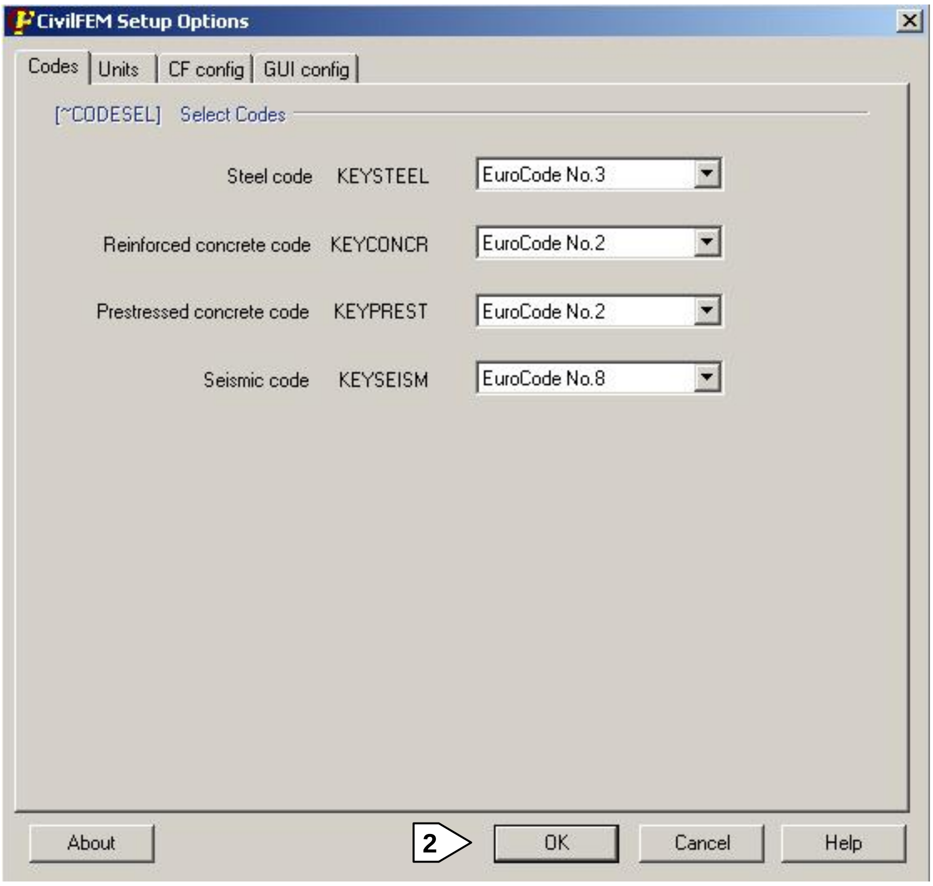
Main Menu: – CivilFEM – **Civil Setup**

1 Select Civil Setup

1



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

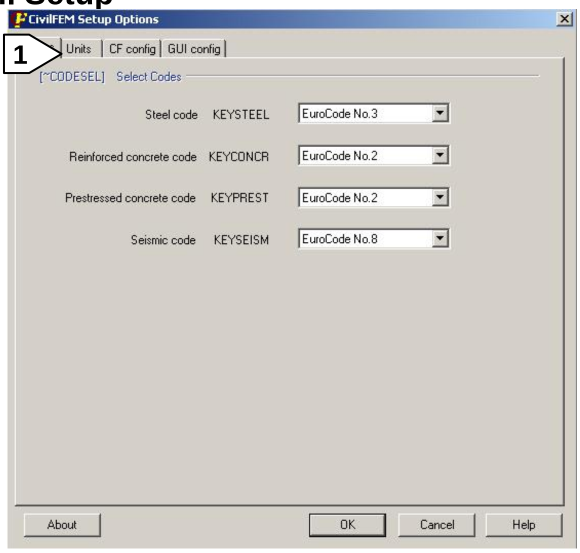


4. Set units

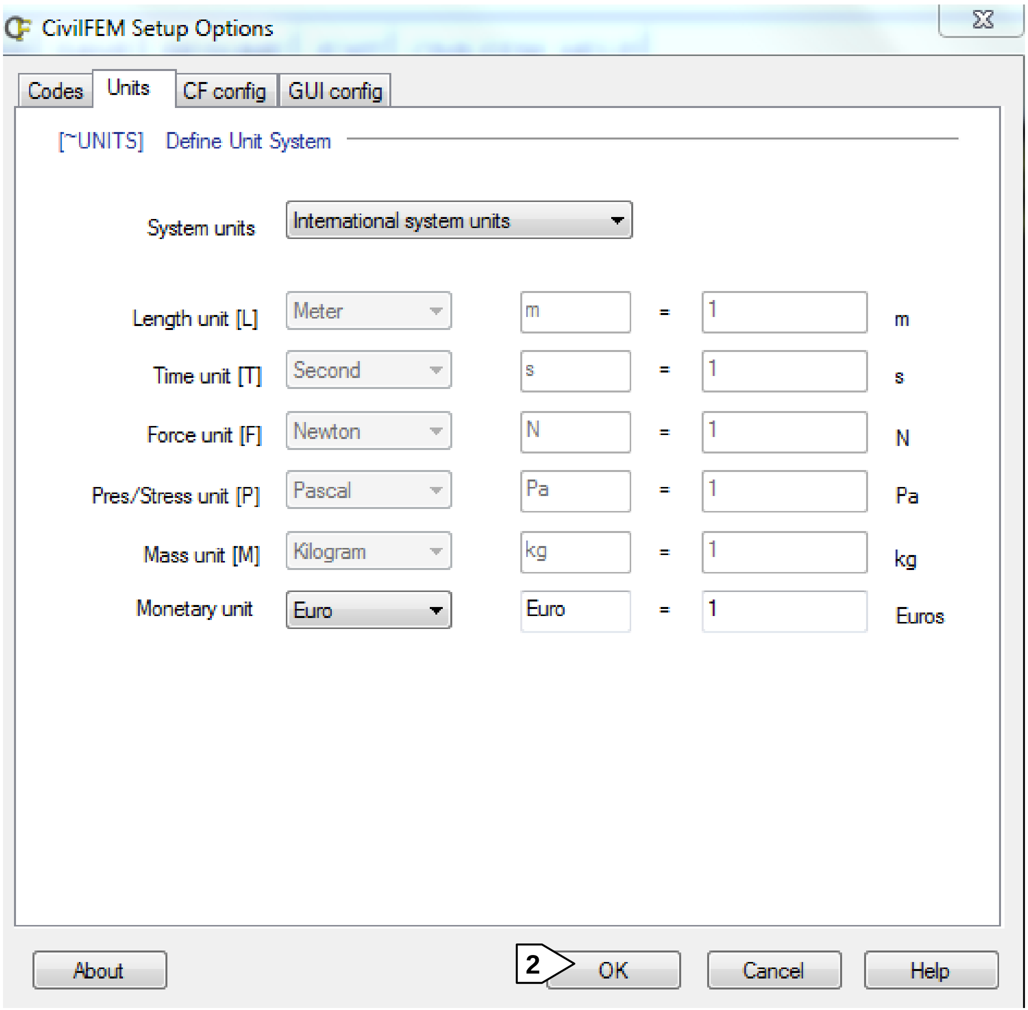
In CivilFEM you must define a unit system. CivilFEM will need such a system to perform calculations according to Code. You should maintain it during the entire design. In this analysis, we will select SI units, that is, meters, seconds and newtons.

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

1 Choose Units Library



2 OK to accept units and close the units dialog box



5. Define material

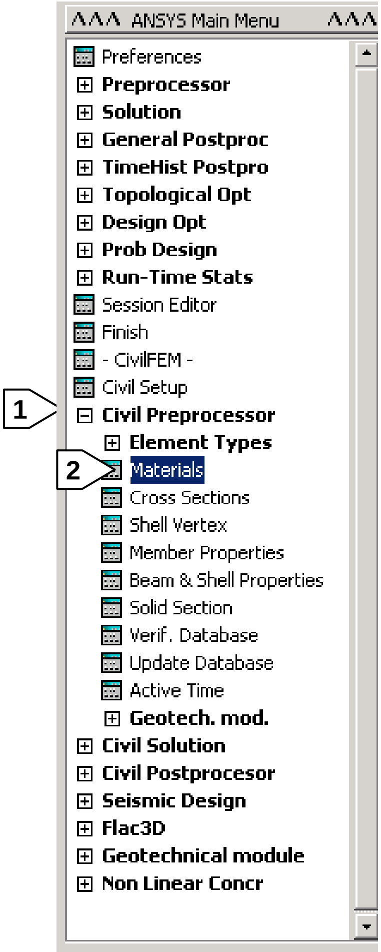
Material properties definition is performed with the CivilFEM **~CFMP** command. This command automatically defines the ANSYS material properties (density, Young's modulus, Poisson's ratio and thermal expansion coefficient) and the CivilFEM material properties necessary for code checking. In this case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

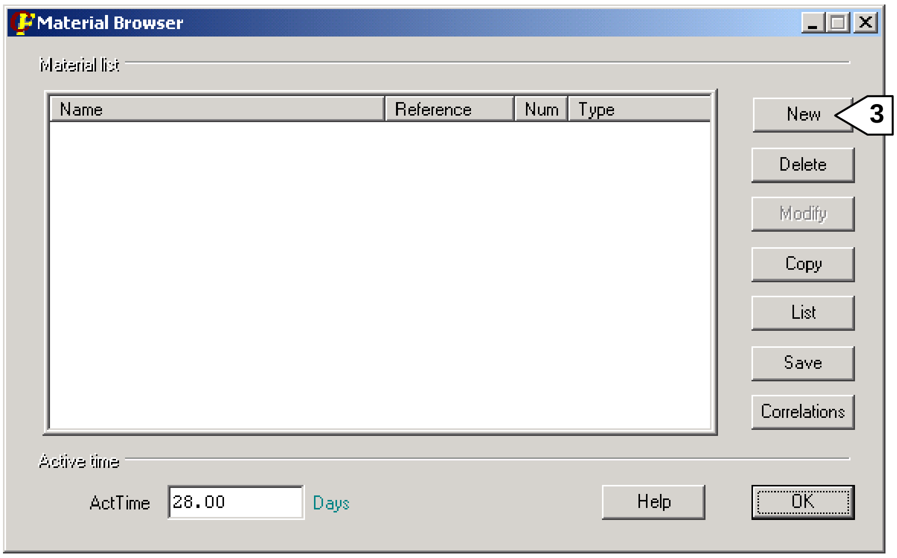
The CivilFEM **~CFMP** command allows us to define stress-strain diagrams, to define safety coefficients, to control the linear or non-linear behavior of the material and to select the activation time of the material.

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

1 Select Civil Preprocess

2 Choose Materials

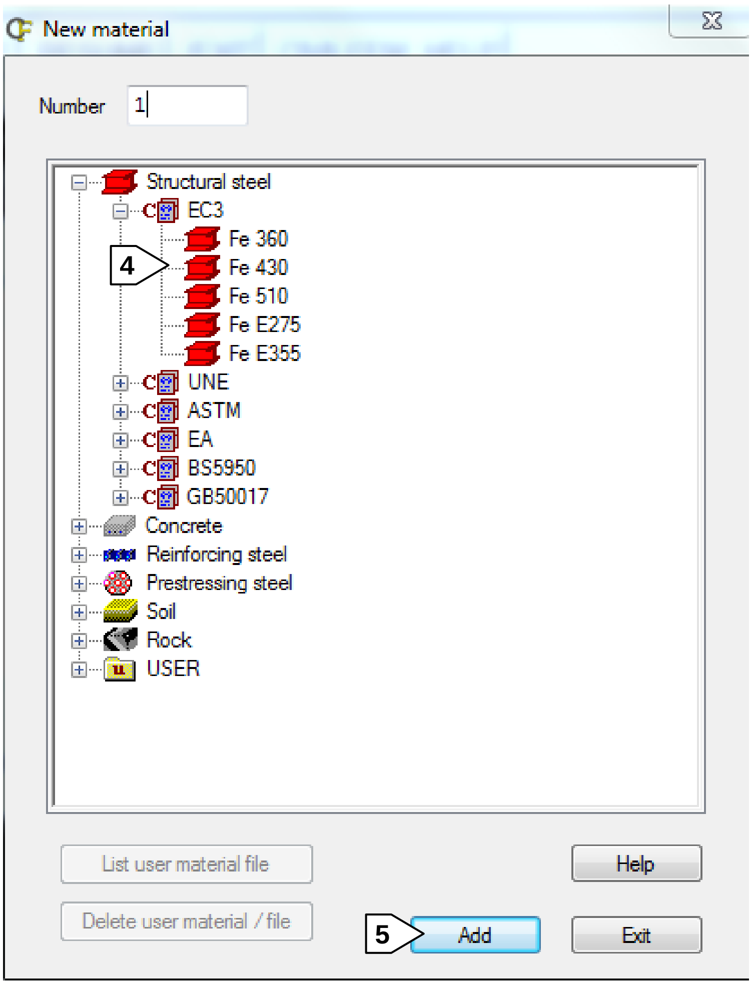


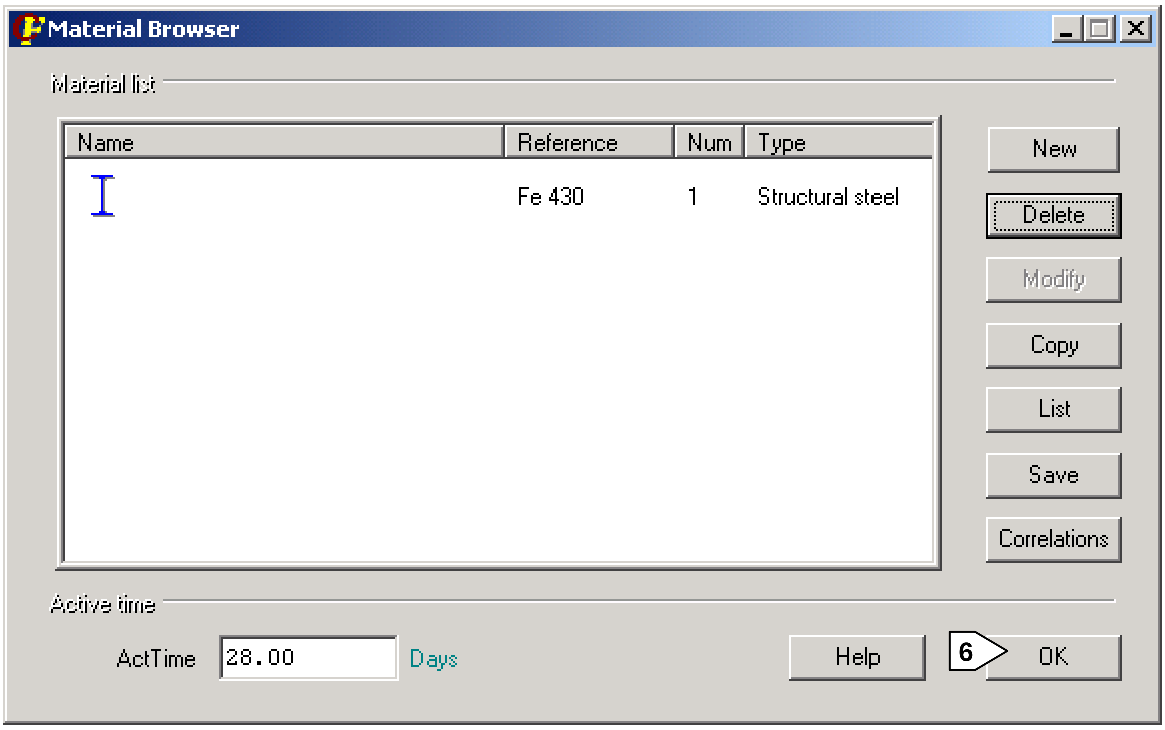


3 Pick new to define a new material

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

5 Add to define material properties set and close the dialog box


New material
Number
1
Structural steel
EC3
Fe 360
Fe 430
Fe 510
Fe E275
Fe E355
UNE
ASTM
EA
BS5950
GB50017
Concrete
Reinforcing steel
Prestressing steel
Soil
Rock
USER
List user material file
Help
Delete user material / file
Add
Exit
4
5


Material Browser
Material list
Name
Reference
Num
Type
Fe 430
1
Structural steel
New
Delete
Modify
Copy
List
Save
Correlations
Active time
ActTime
28.00
Days
Help
OK
6

6 OK

6. Define element type

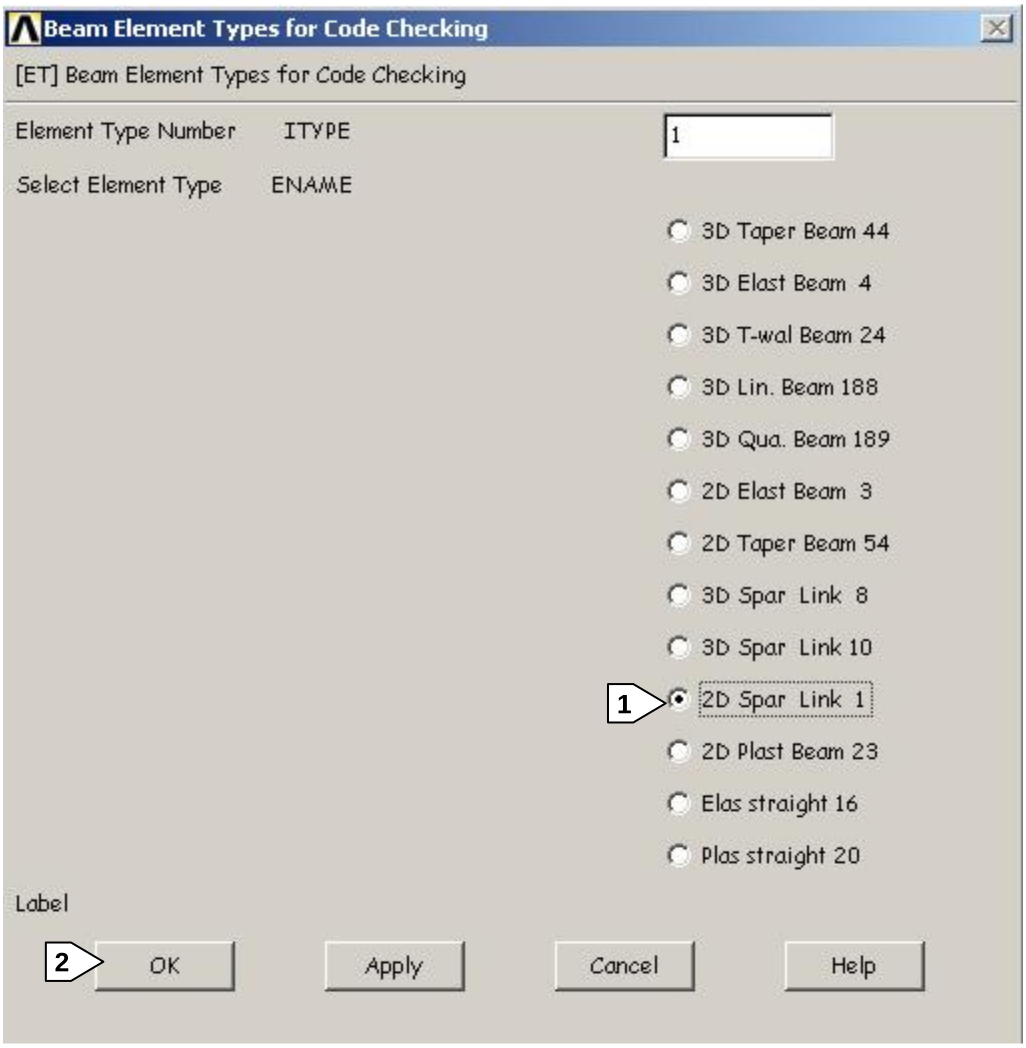
Checking and designing according to codes is only performed on CivilFEM supported element types, although you can use any ANSYS element to define your model, only the CivilFEM supported elements will be checked according to codes. In the element type menu you can see the CivilFEM supported beam elements.

We will use 2D Spar LINK 1 for this analysis.

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

1 Select Element Type 2D Spar LINK 1

2 Choose OK



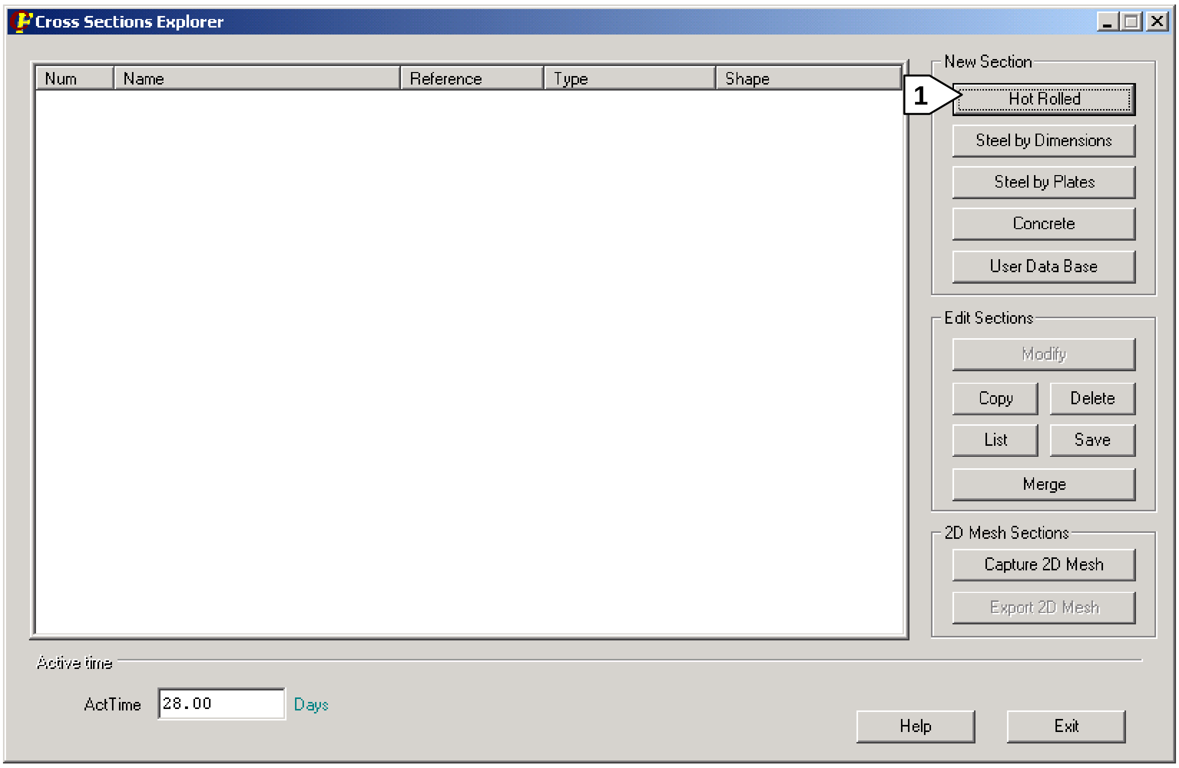
7.

7. Define section

We will use European L EQ 90x90x9 and it will be defined from the Library of *Hot Rolled Shapes Library*

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

1 Click the Hot Rolled Button

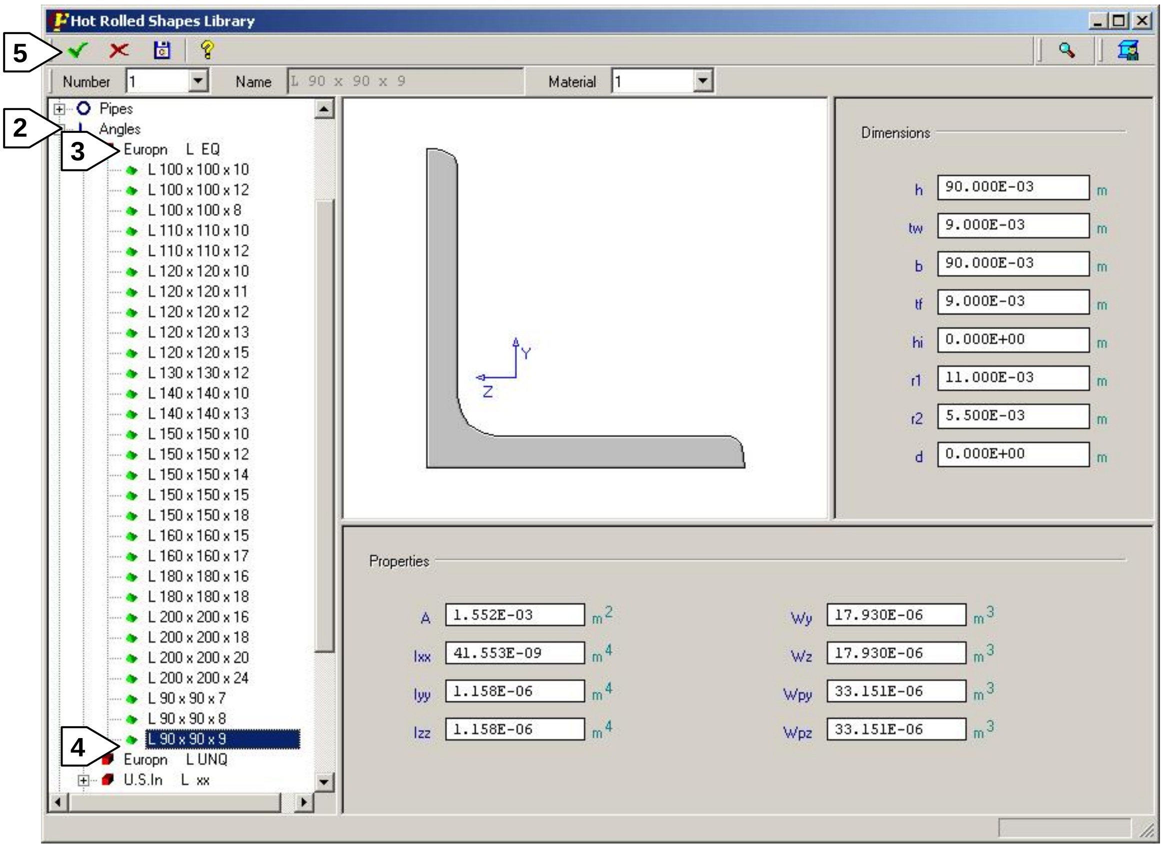


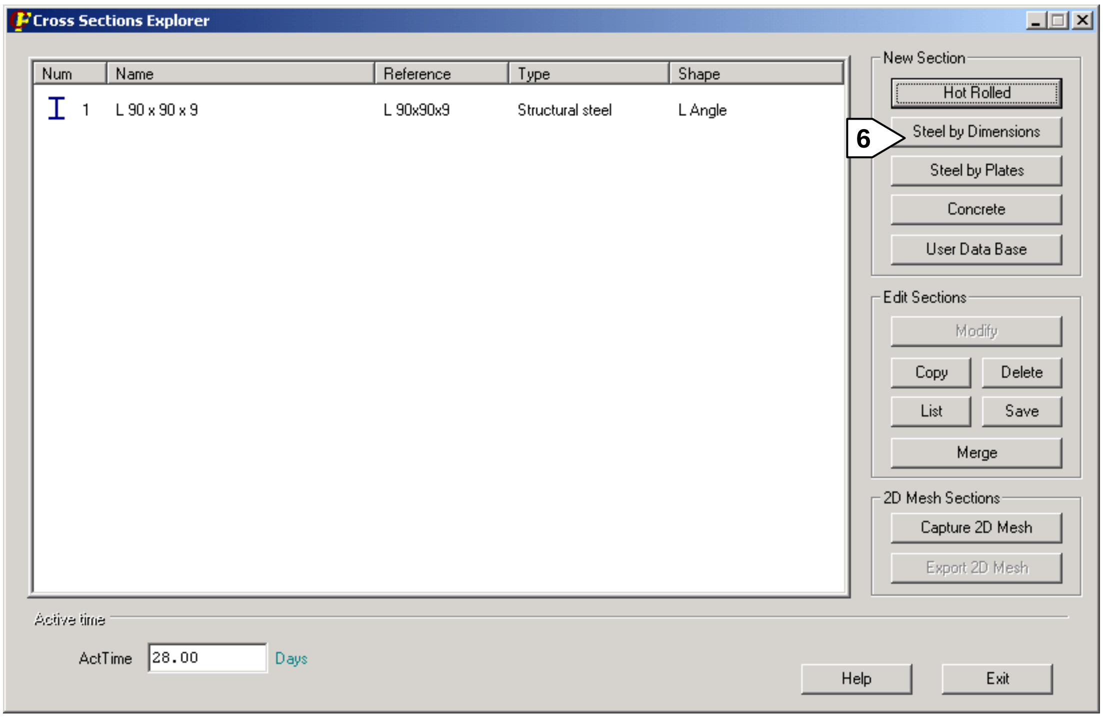
2 Select L Angles group.

3 Select European L EQ

4 Select L 90*90*9 Shape

5 OK to define Cross Section 1


Hot Rolled Shapes Library
5
2
3
4
Number
Name
Material
Pipes
Angles
Europn L EQ
L 90 x 90 x 9
Europn L UNQ
U.S.In L xx
Dimensions
Properties


Cross Sections Explorer
Num
Name
Reference
Type
Shape
1
L 90 x 90 x 9
L 90x90x9
Structural steel
L Angle
New Section
Hot Rolled
6
Steel by Dimensions
Steel by Plates
Concrete
User Data Base
Edit Sections
Modify
Copy
Delete
List
Save
Merge
2D Mesh Sections
Capture 2D Mesh
Export 2D Mesh
Active time
ActTime
28.00
Days
Help
Exit

Now, we are going to define section 2. It's a welded T section so the steps we must follow are:

6 Click the Steel by Dimensions button.

7 Select T shape

8 Enter Depth: 0.15

9 Enter Web Thickness: 0.01

10 Enter Flange Width: 0.1

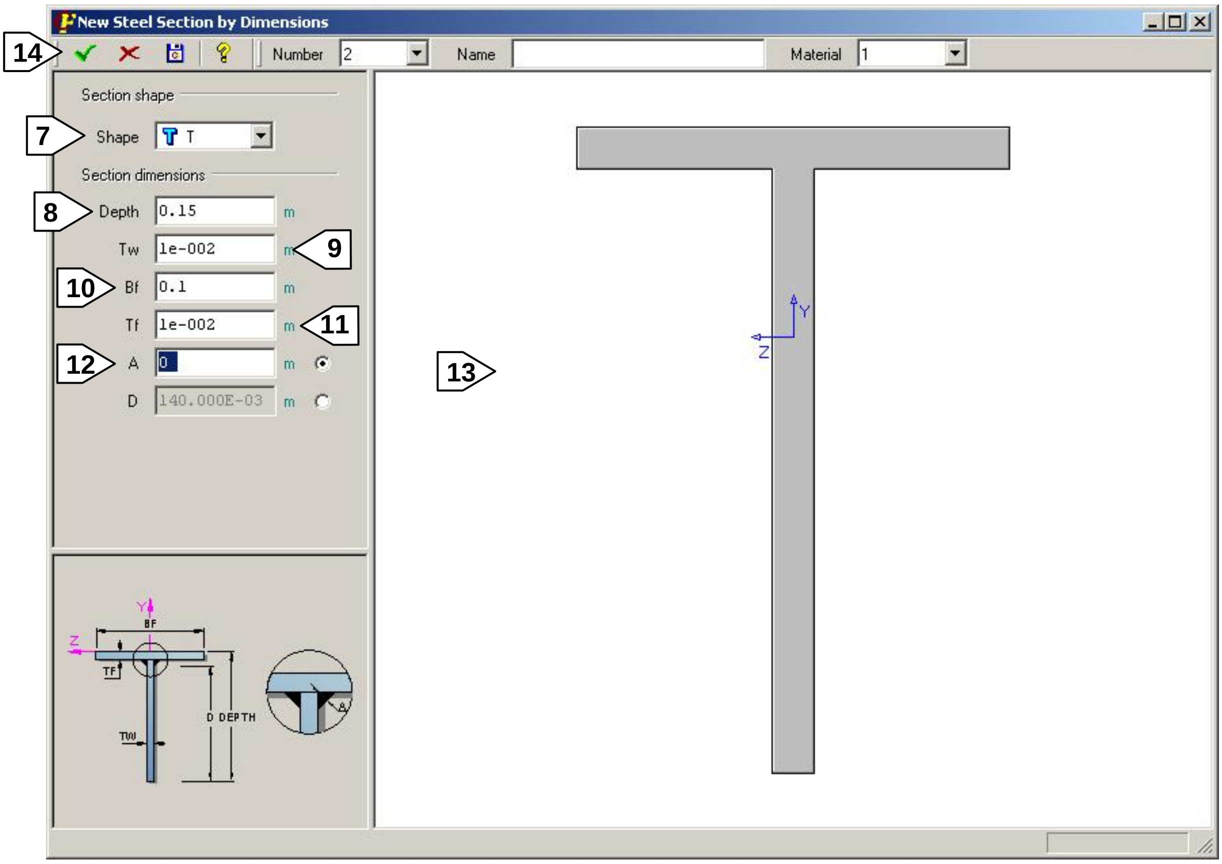
11 Enter Flange Thickness: 0.01

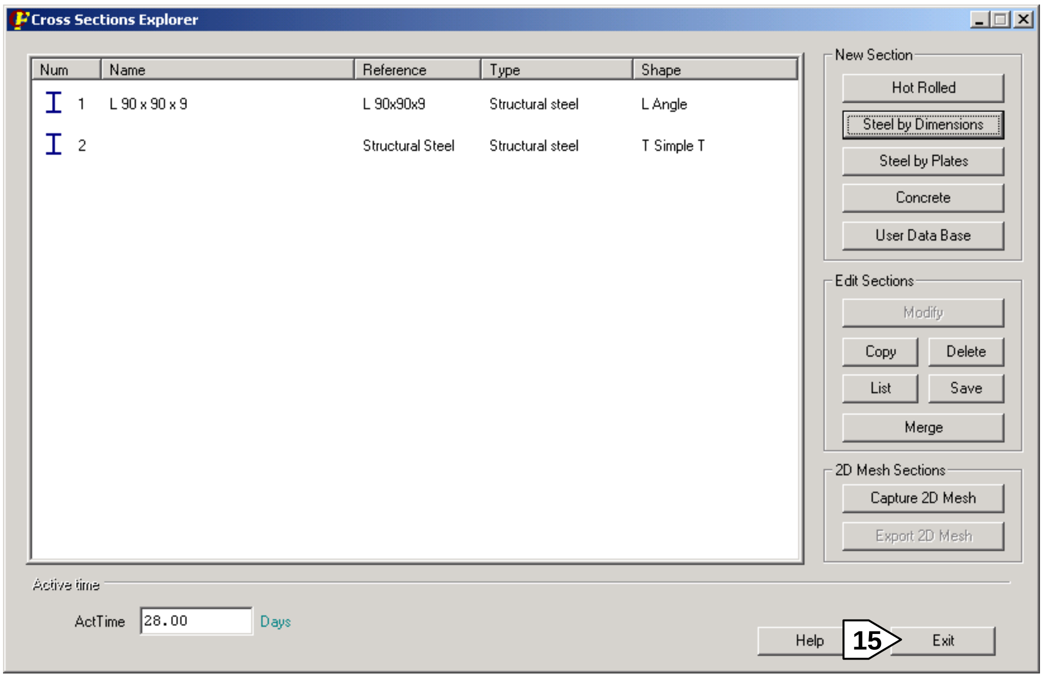
12 Enter Weld Throat Thickness: 0

13 Picking with the right button section shape will be drawn.

14 OK

15 Exit to close cross section explorer



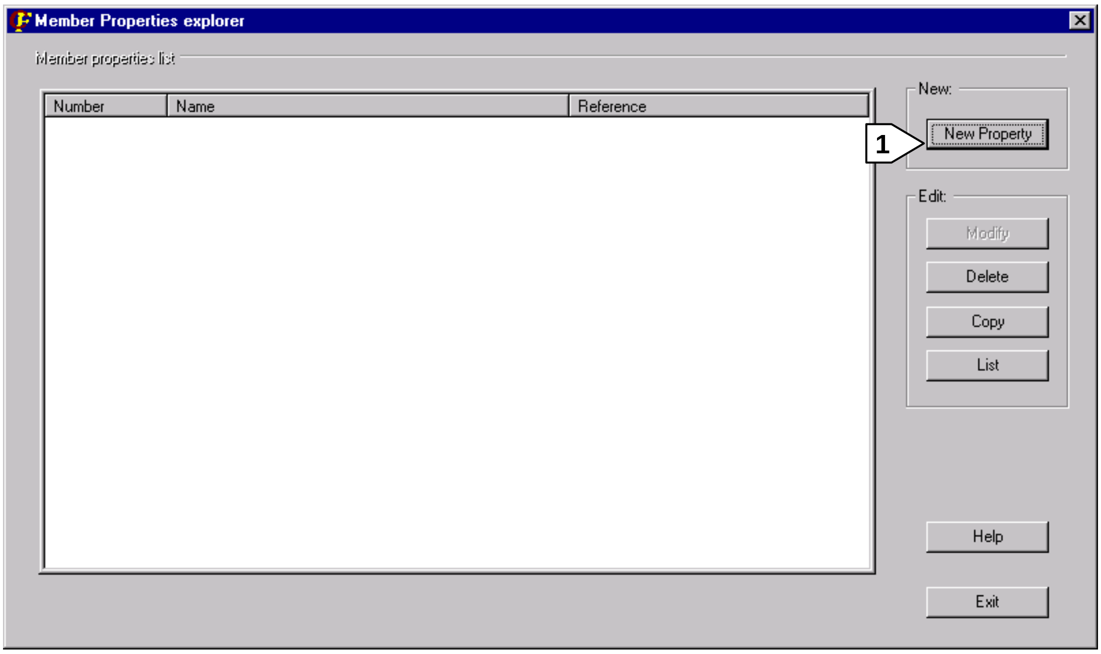


8. Define member properties

You only have to define the code properties needed for the checking you are going to accomplish. In this problem the member properties definition is necessary to check for compression buckling.

Main Menu: – CivilFEM – **Civil Preprocess →Member Propertie..**

1 Pick on New Property to define properties for diagonal elements.



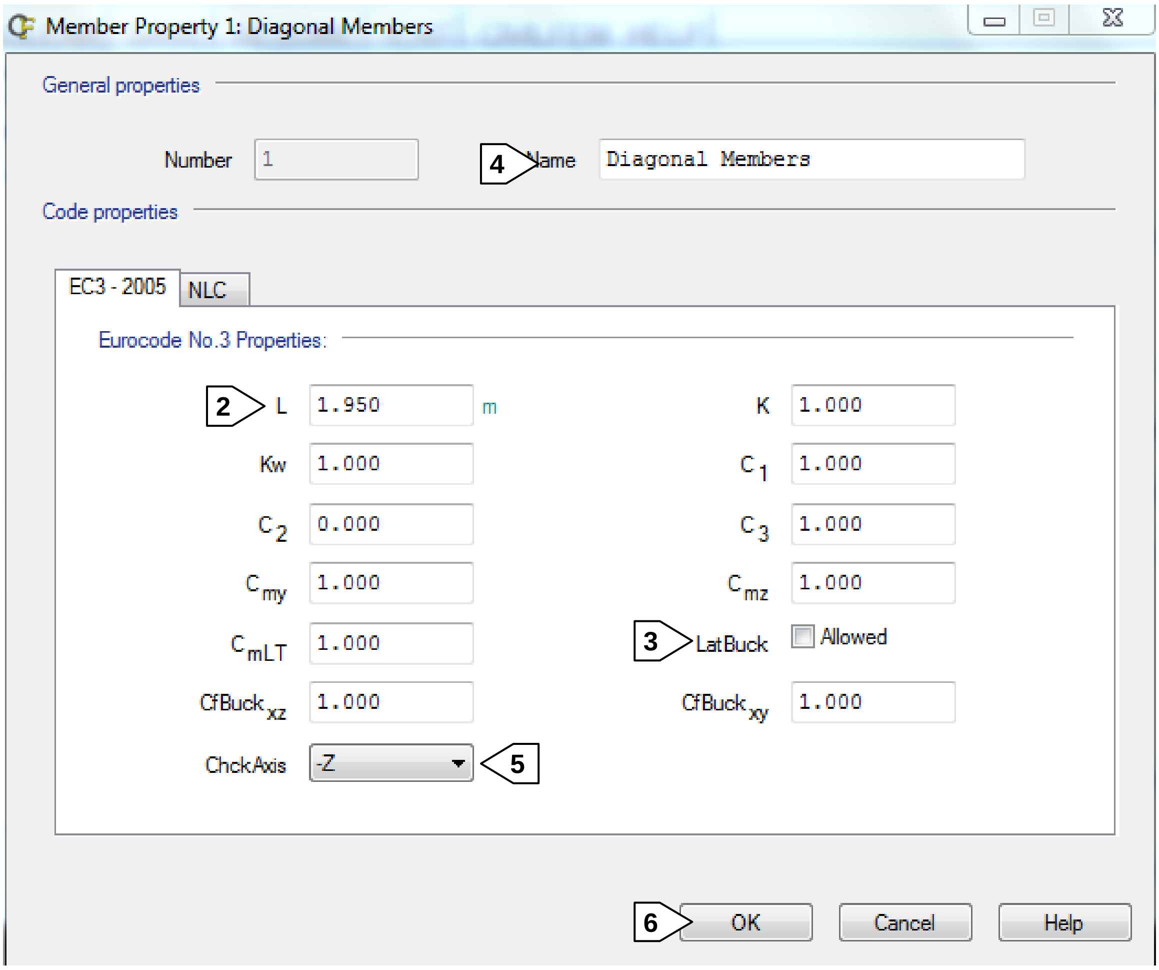
2 Enter 1.95 as Length Between Lateral Restraints

3 Do not consider Lateral Buckling as a Potential Failure

4 Enter “Diagonal Members” as Name

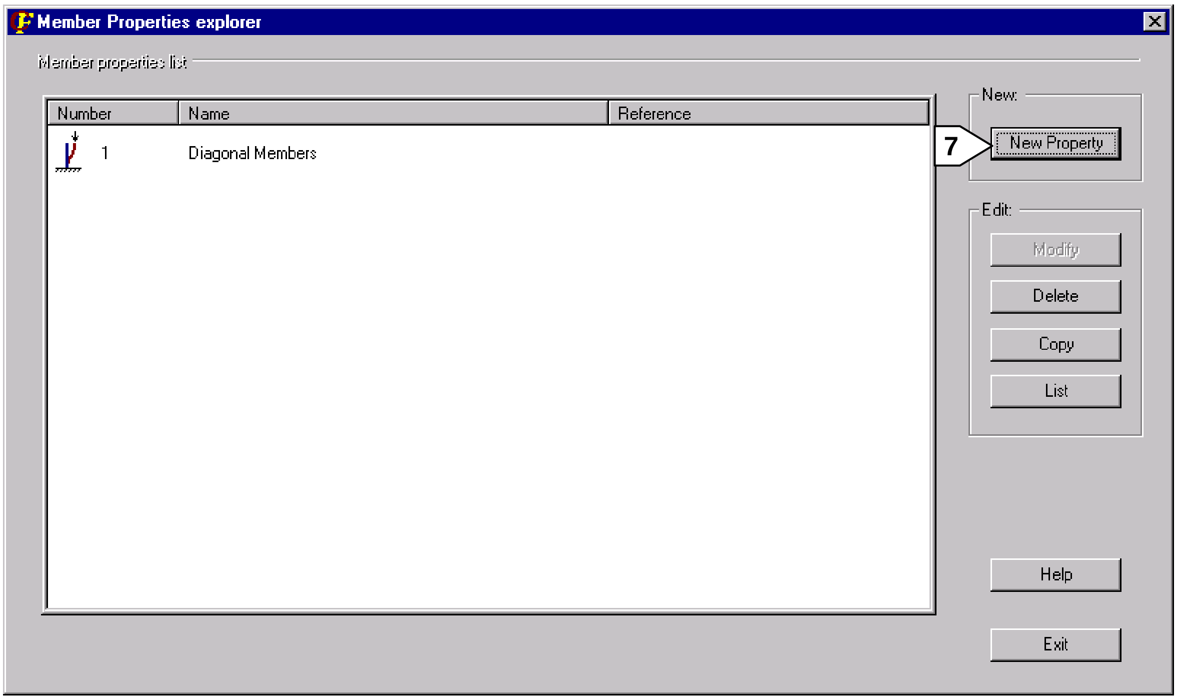
5 Consider Eurocode 3 Y Axis as CivilFEM –Z Axis

6 OK



Now we introduce code properties for section 2, (horizontal elements).

7 Click New property to define properties for beam elements



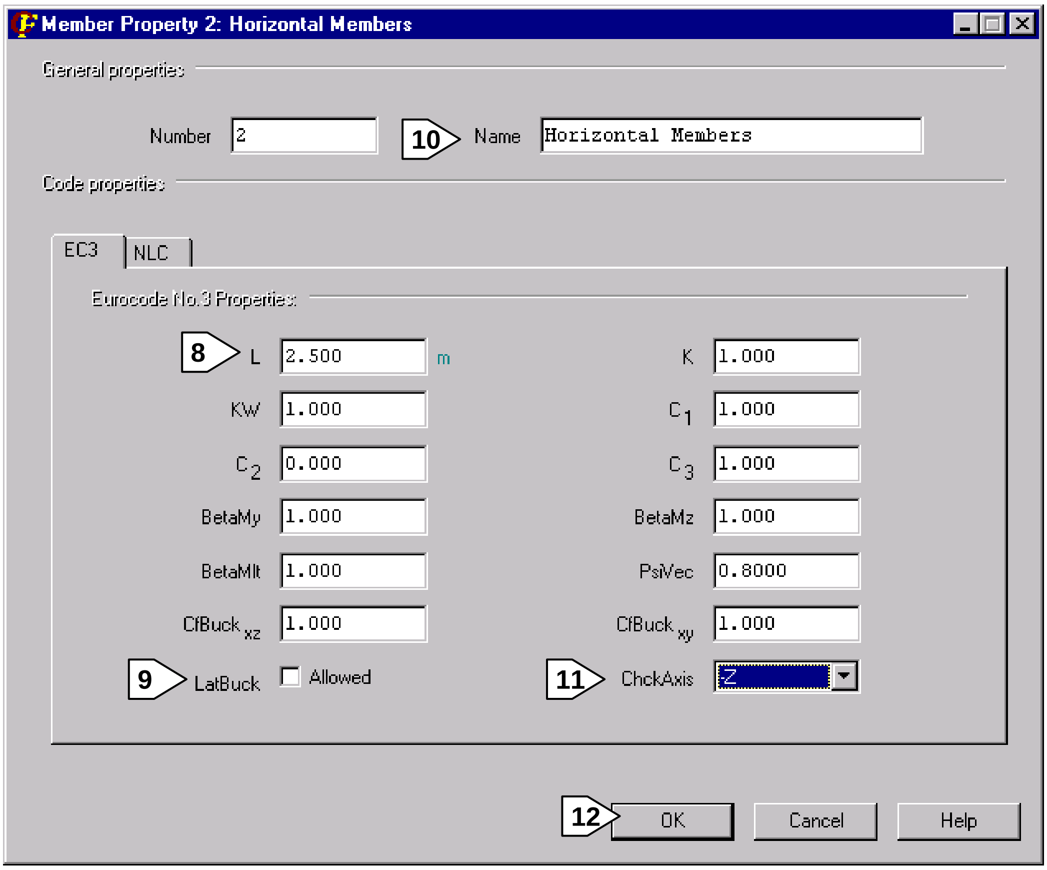
8 Enter 2.5 as Length Between Lateral Restraints

9 Do not consider Lateral Buckling as a Potential Failure

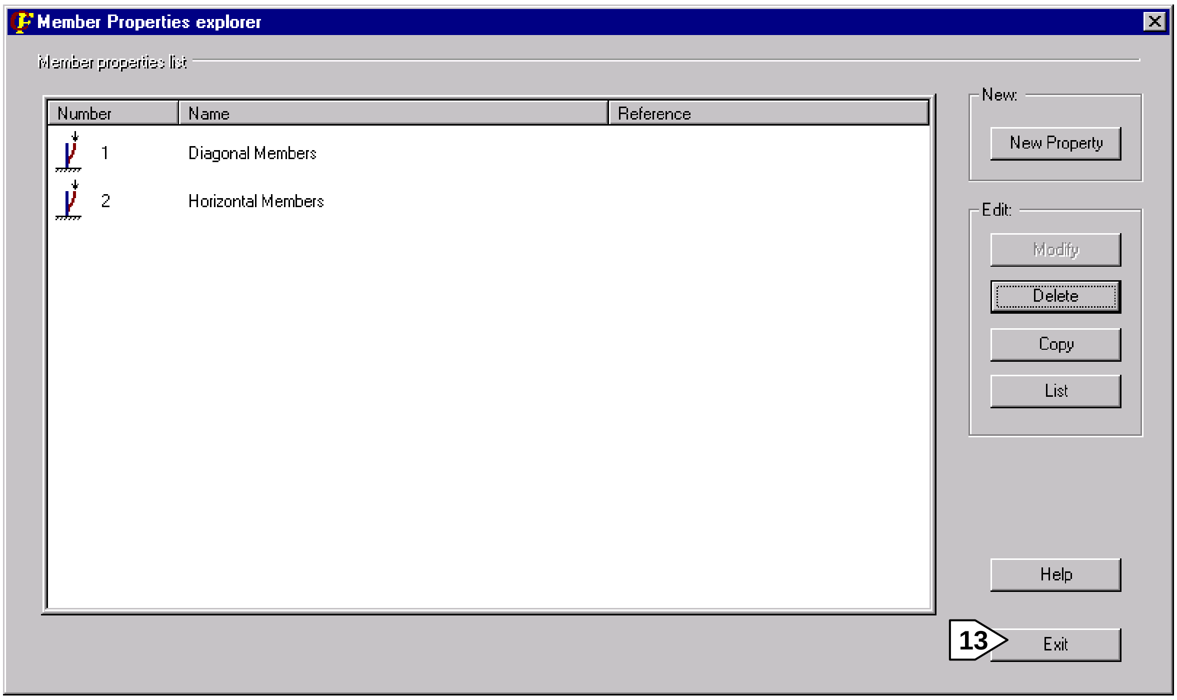
10 Enter “Horizontal Members” as Name

11 Consider Eurocode 3 Y Axis as CivilFEM –Z Axis

12 OK



13 Exit to close member properties explorer

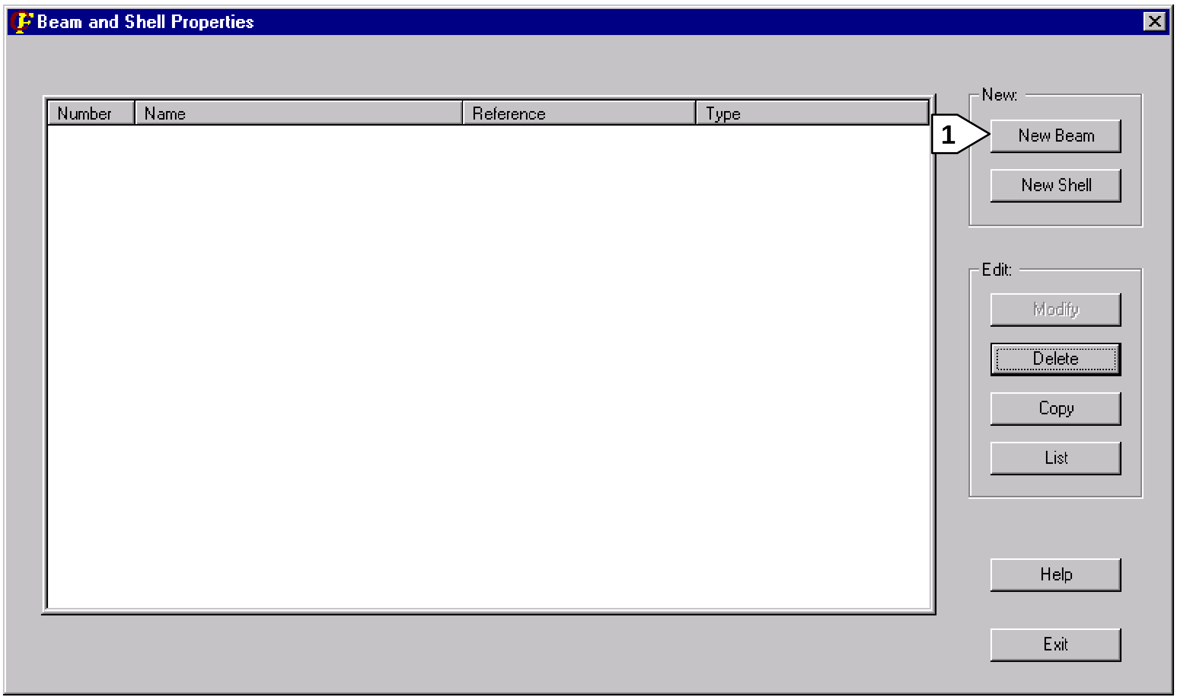


9. Define Beam & Shell properties

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

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

1 Click the New Beam button



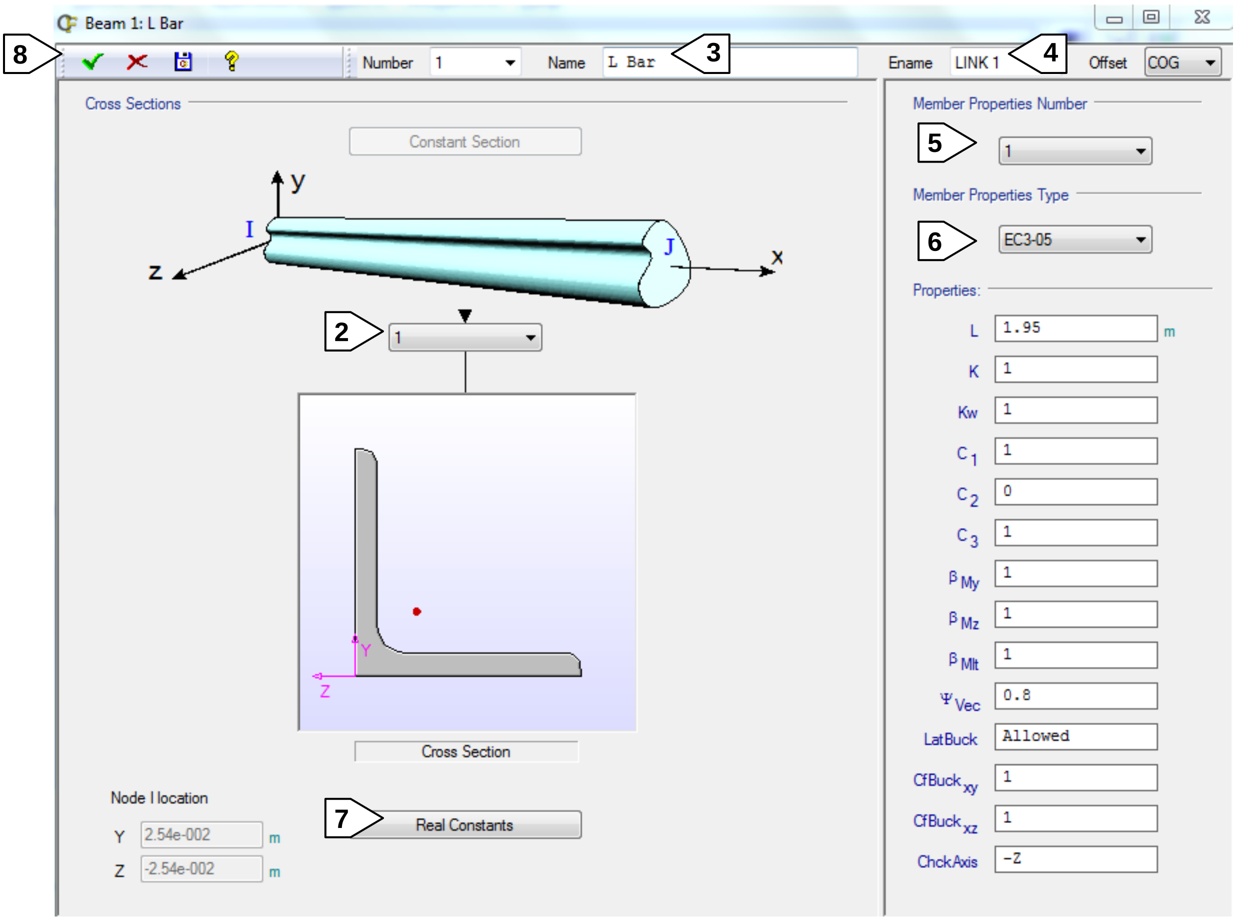
2 Select cross section number 1

3 Enter “L Bar” as name for the Beam property

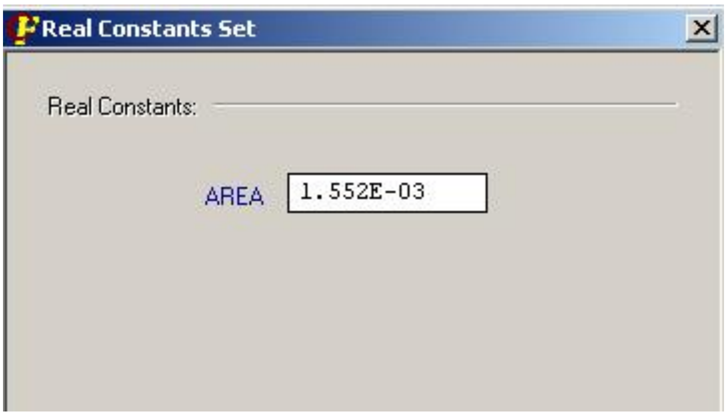
4 Select element type LINK1

5 Select Member Properties number 1

6 Select EC3 as Member Properties Type to visualize the defined properties



7 You can review the ANSYS real constants modified by CivilFEm by clicking on the Real Constants button



8 Apply to create new Beam Properties

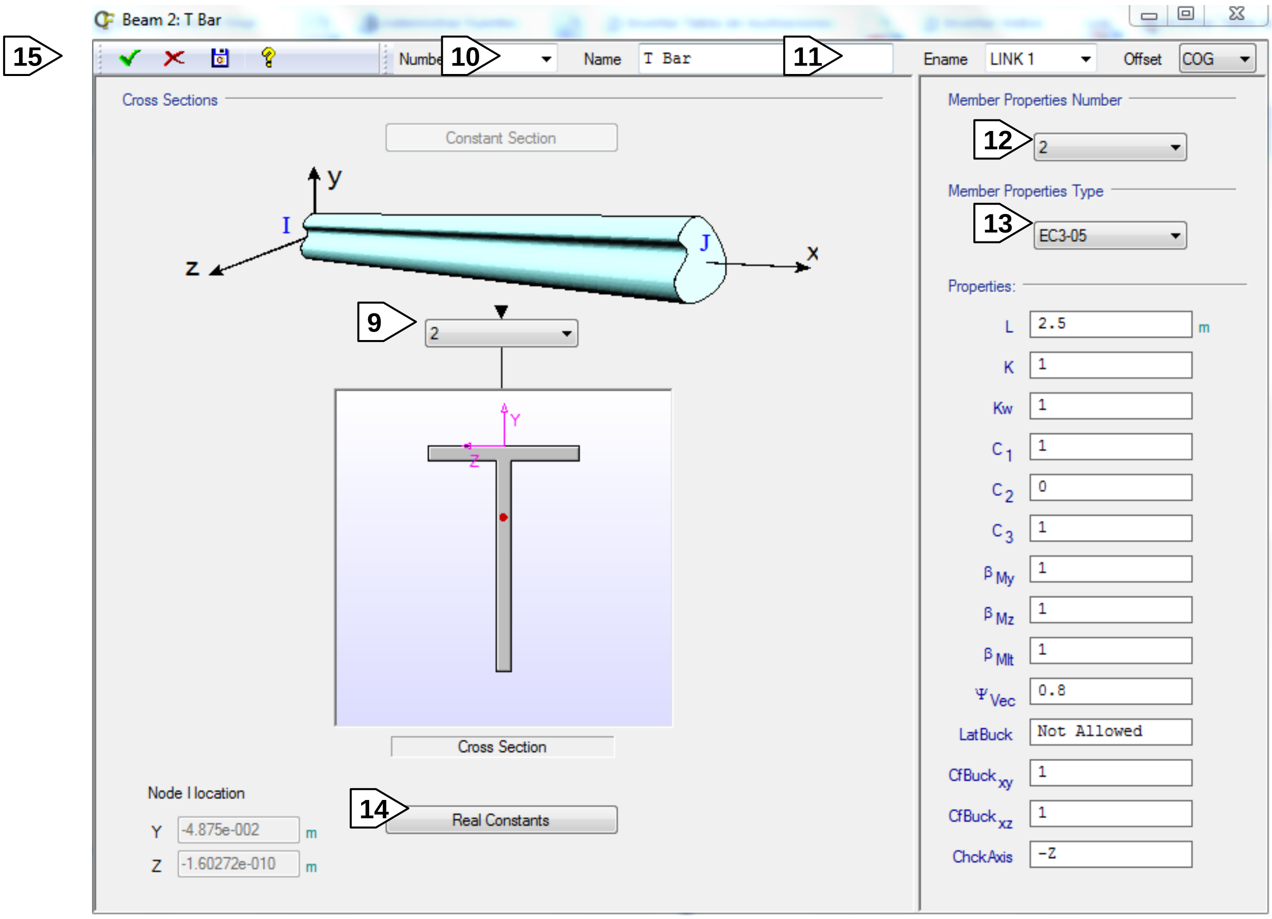
9 Select cross section number 2

10 Enter “T Bar” as name for the Beam property

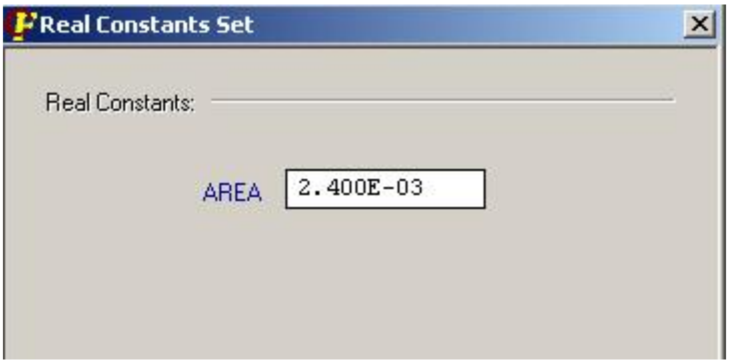
11 Select element type LINK1

12 Select Member Properties number 2

13 Select EC3 as Member Properties Type to visualize the defined properties

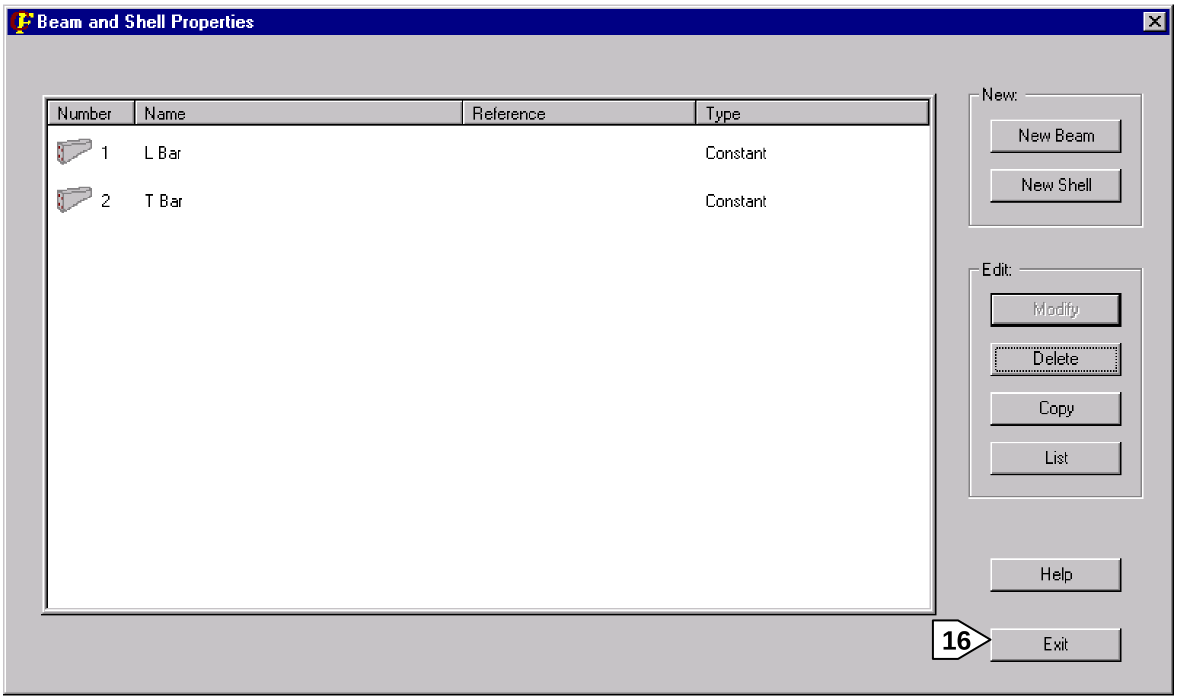


14 Again you can review the ANSYS real constants by clicking the Real Constants button



15 OK to define Beam & Shell properties number 2

16 Exit to close window



10. Define nodes and elements

The model geometry is defined by direct elements and node generation. We will use the working plane to introduce the node coordinates.

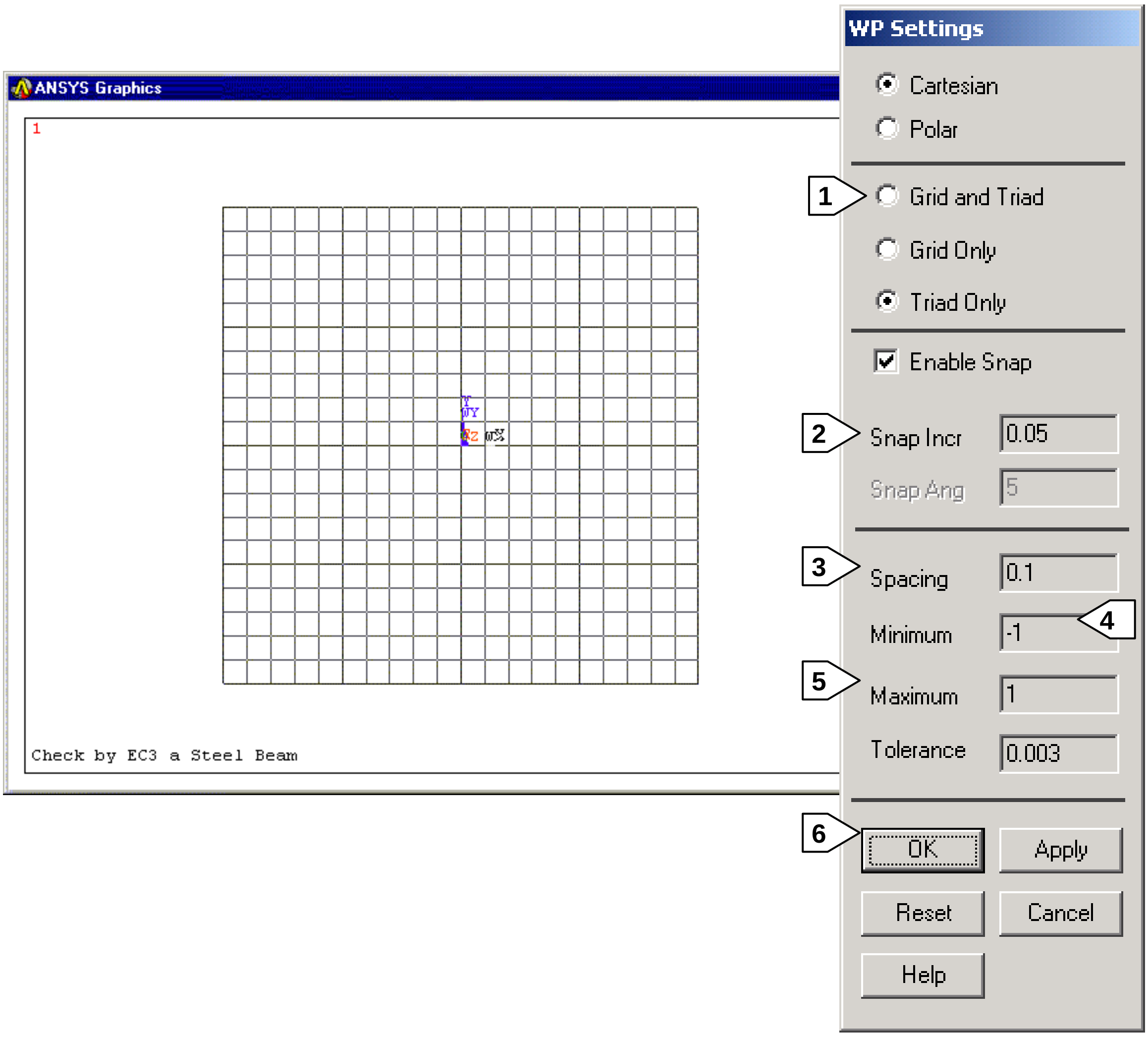
Once we have defined the nodes we will assign a section to each element.

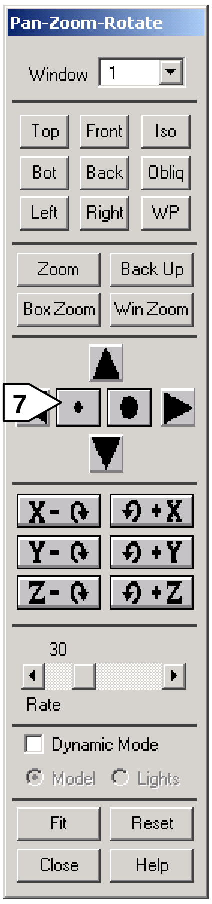
Utility Menu: **WorkPlane →Display WP**

Before we begin it is necessary to turn on the working plane and zoom out until we see all the created nodes.

Utility Menu: **WorkPlane →WP Settings**

1. Click on Grid and Triad
2. Enter 0.25 for snap increment
3. Enter 0.5 for Spacing
4. Enter –5 for Minimum
5. Enter 5 for Maximum
6. OK to define settings





Utility Menu: **PlotCtrls →Pan, Zoom, Rotate**

7 Zoom out till you see the entire Working Plane

Main Menu: **Preprocessor→ Modeling → Create →Nodes →On Working Plane**

8 Pick Node 1 on Working Plane: X=0, Y=0

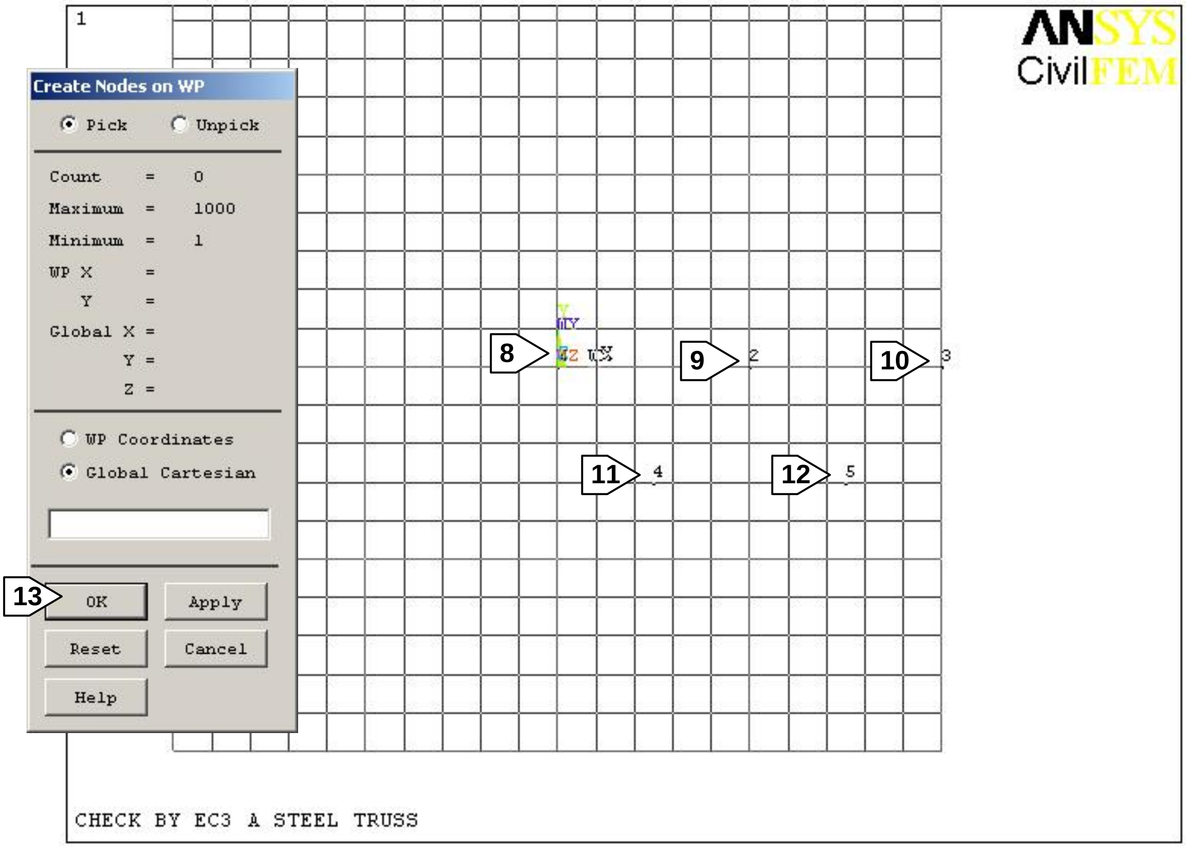
9 Pick Node 2 on Working Plane: X=2.5, Y=0

10 Pick Node 3 on Working Plane: X=5, Y=0

11 Pick Node 4 on Working Plane: X=1.25, Y=-1.5

12 Pick Node 5 on Working Plane: X=3.75, Y=-1.5

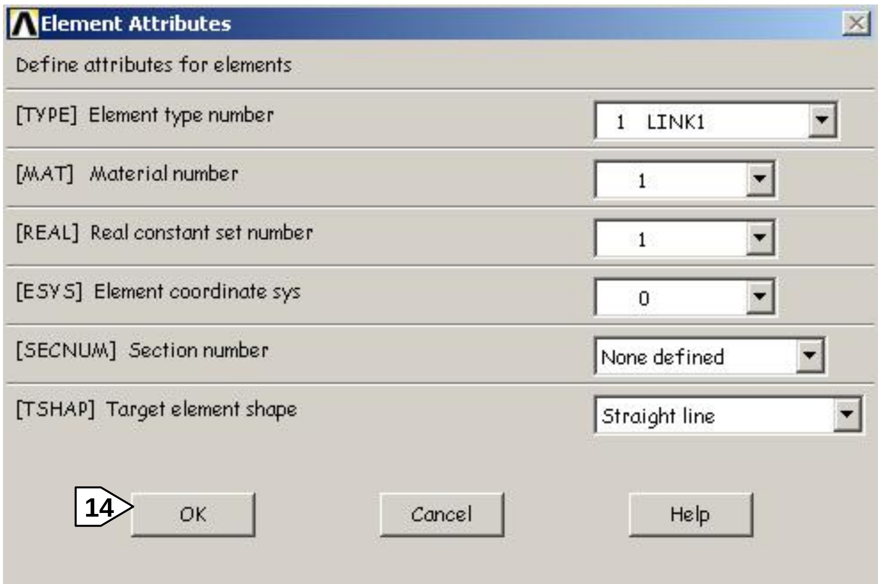
13 OK



First we activate the first section type, the, “L” section. CivilFEM considers section 1 data like a real constant set 1 so we need to assign the real constant 1 to the elements with section 1 through the **REAL** command.

Main Menu: **Preprocessor→ Modeling →Create →Elements → Elem Attributes**

14 OK to assign section 1 to the diagonal beams

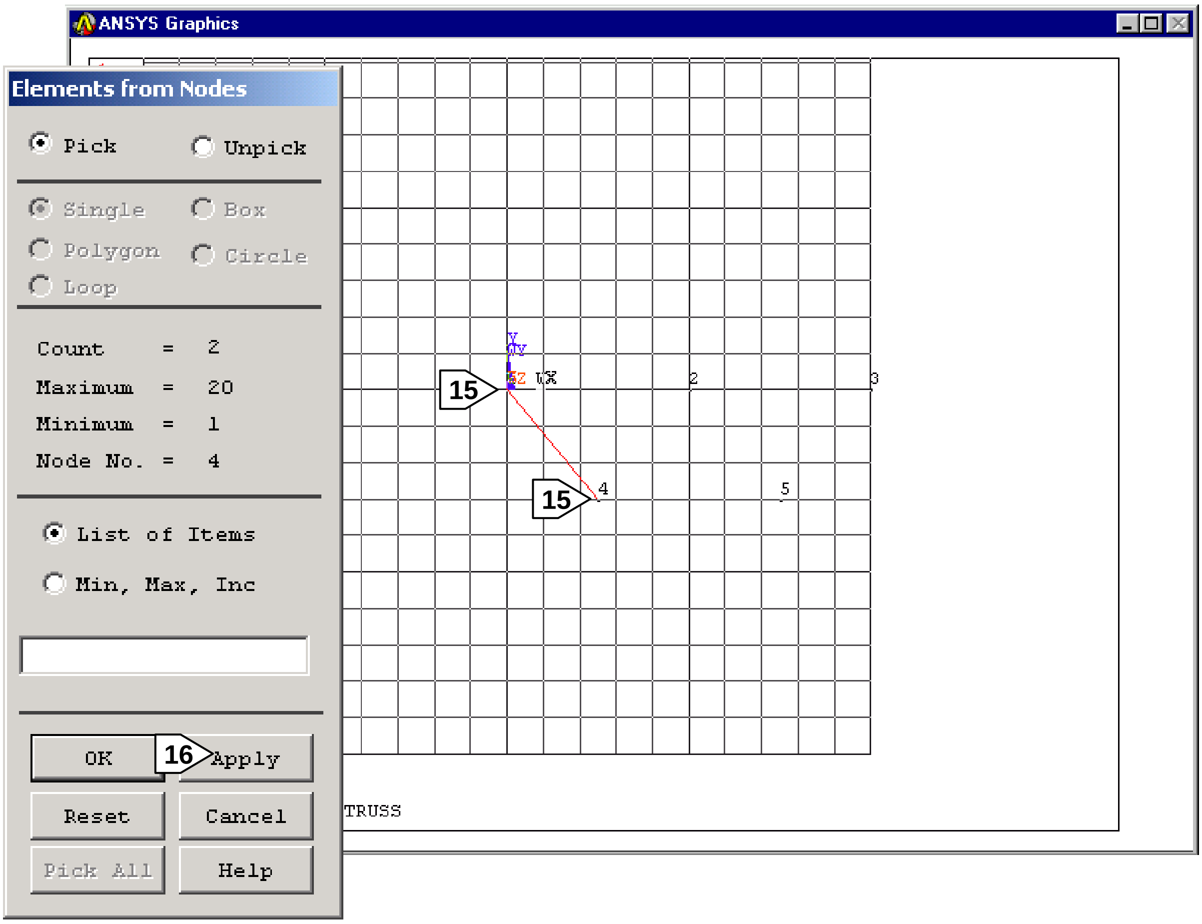


We define the elements with section 1, (bars 3, 4, 5 and 6)

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

15 Pick the first two nodes like in the figure below

16 Apply

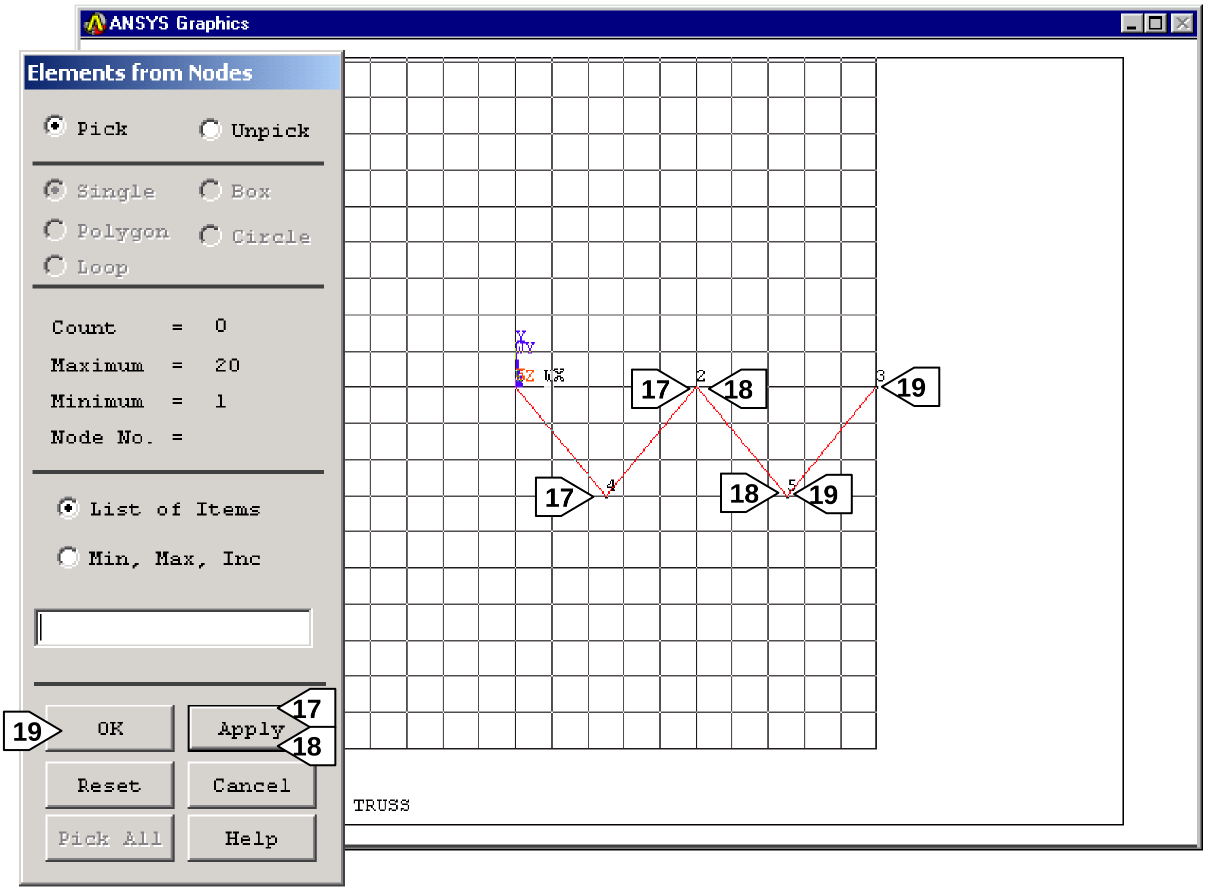


Follow this process with nodes 4-2, 2-5, and 5-3 to create the rest of the elements.

17 Pick nodes 4-2 and click Apply

18 Pick nodes 2-5 and click Apply

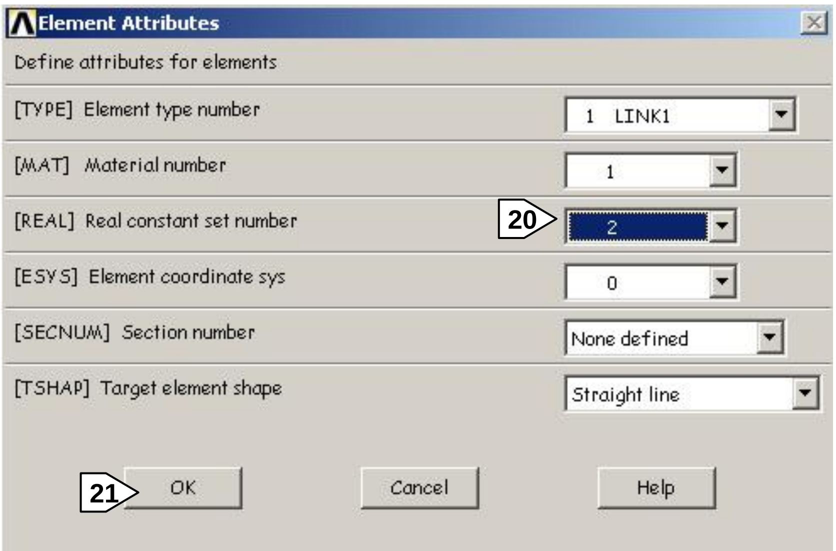
19 Pick nodes 5-3 and click OK



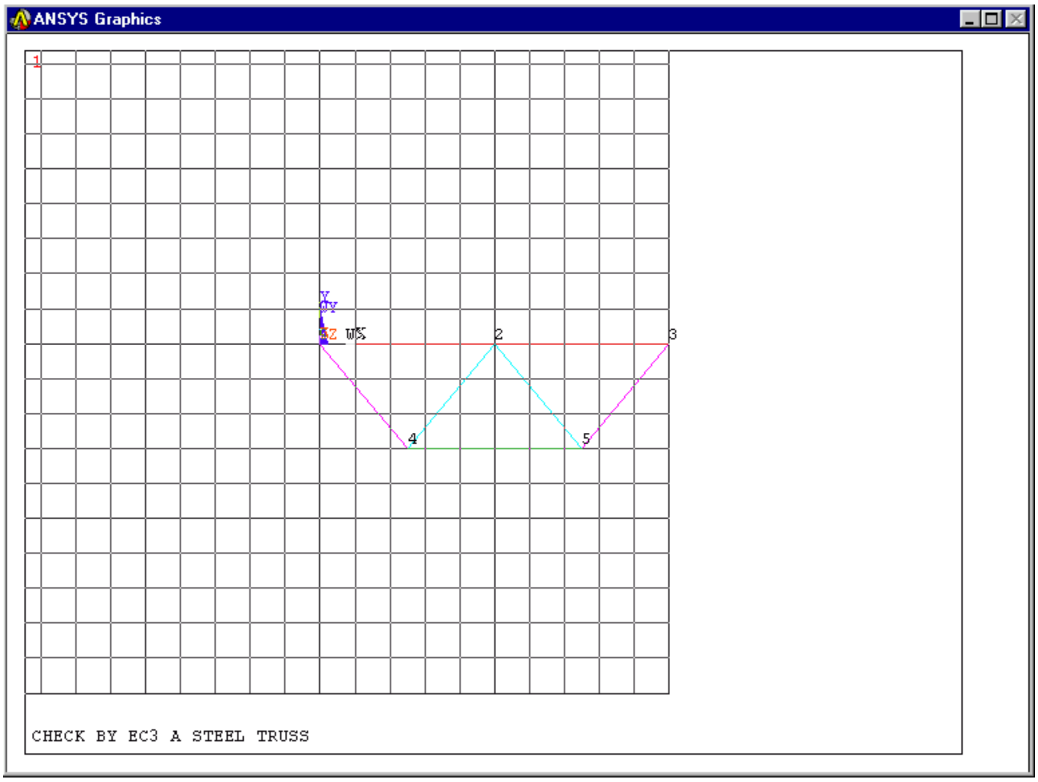
We use the same procedure to assign section 2 to the rest of the elements, (bars 1,2 and 7).

20 Choose real constants set 2

21 OK



Repeat this process with nodes 1-2, 2-3, and 5-4 to create the rest of the elements. Bar 7 needs to be introduced as 5-4 to invert section position (flanges at bottom).



Now we adjust the graph.

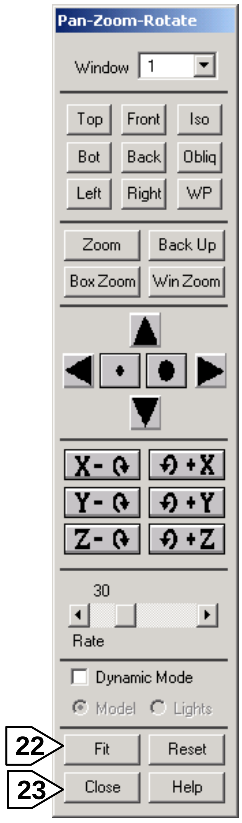
Utility Menu: **WorkPlane →Display WP**

Utility Menu: **PlotCtrls →Pan, Zoom, Rotate**

22 Fit

23 Close

Utility Menu: **Plot →Elements**

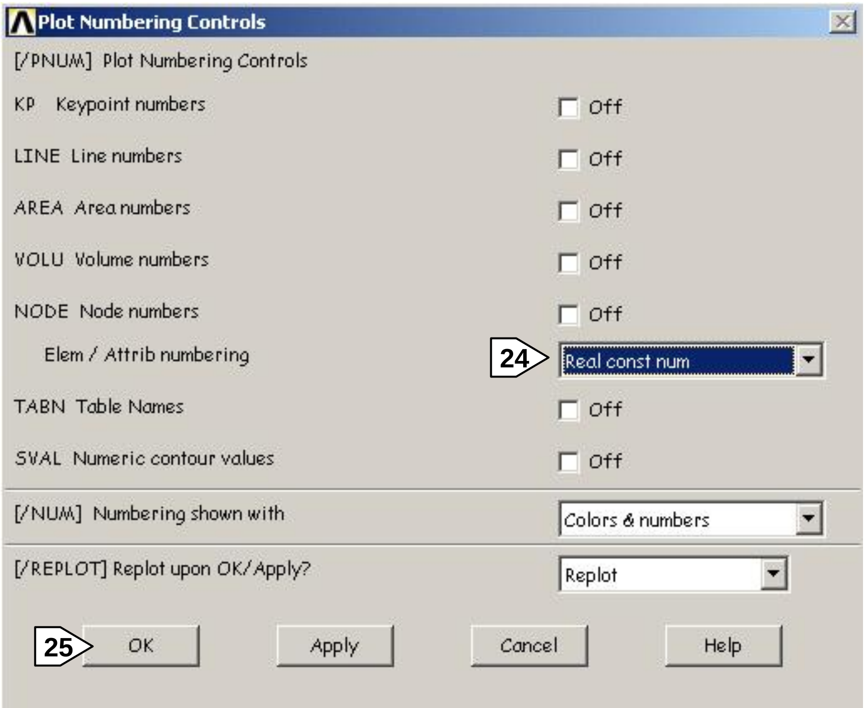


Finally, we can check that all elements have their corresponding section type by numbering the real constants set:

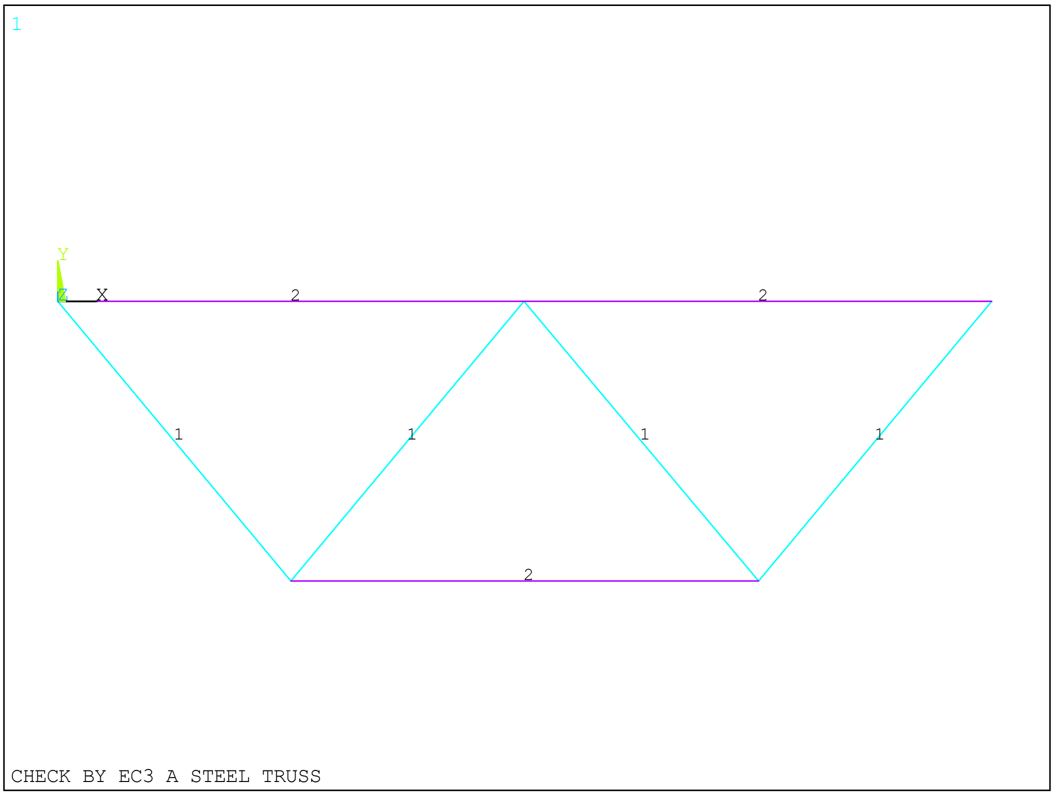
Utility Menu: **PlotCtrls →Numbering**

24 Choose Real const num

25 OK



1



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

Toolbar: **CFSAVE**

■ Solution

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

12. Apply displacement constraints

We will constrain all DOF at the left end of the truss

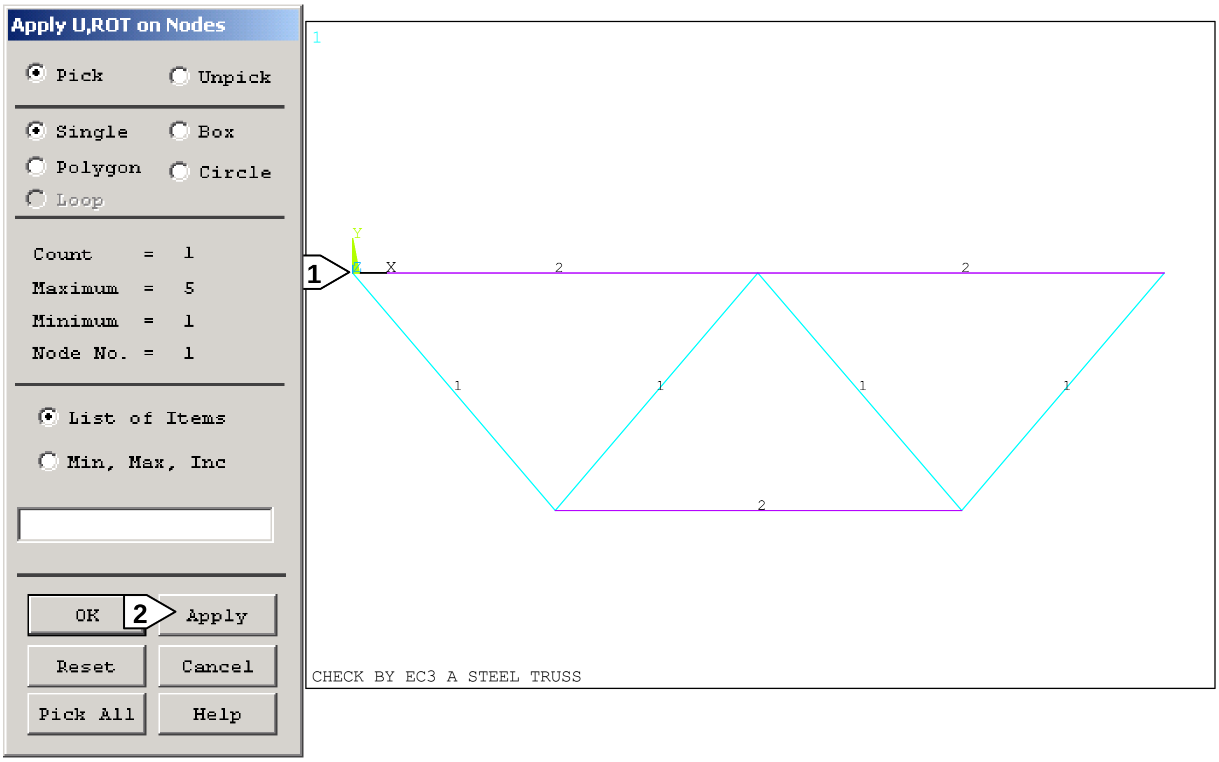
Main Menu: **Solution →** – Define Loads – **Apply →** – Structural – **Displacement → On Nodes**

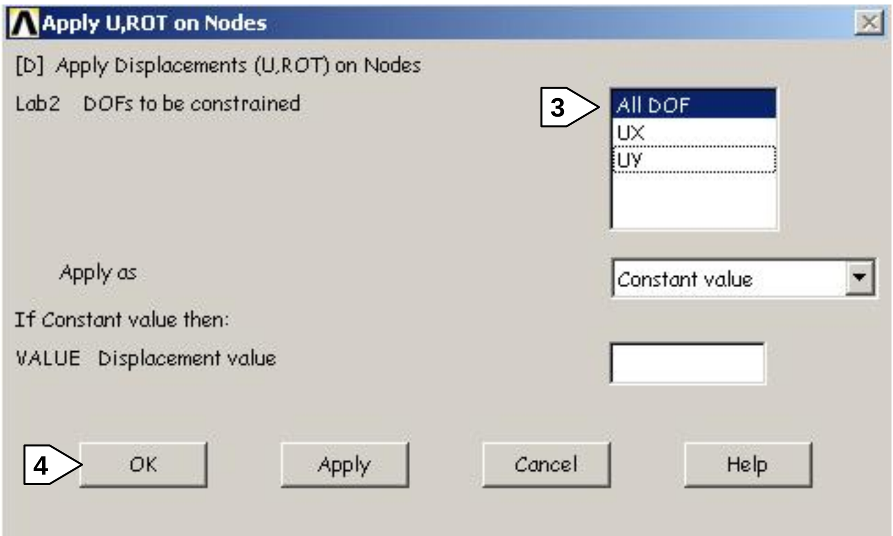
1 Pick the left end of the truss

2 Apply

3 Choose All DOF to be constrained

4 OK





We will constrain Uy displacement at the right end of the truss

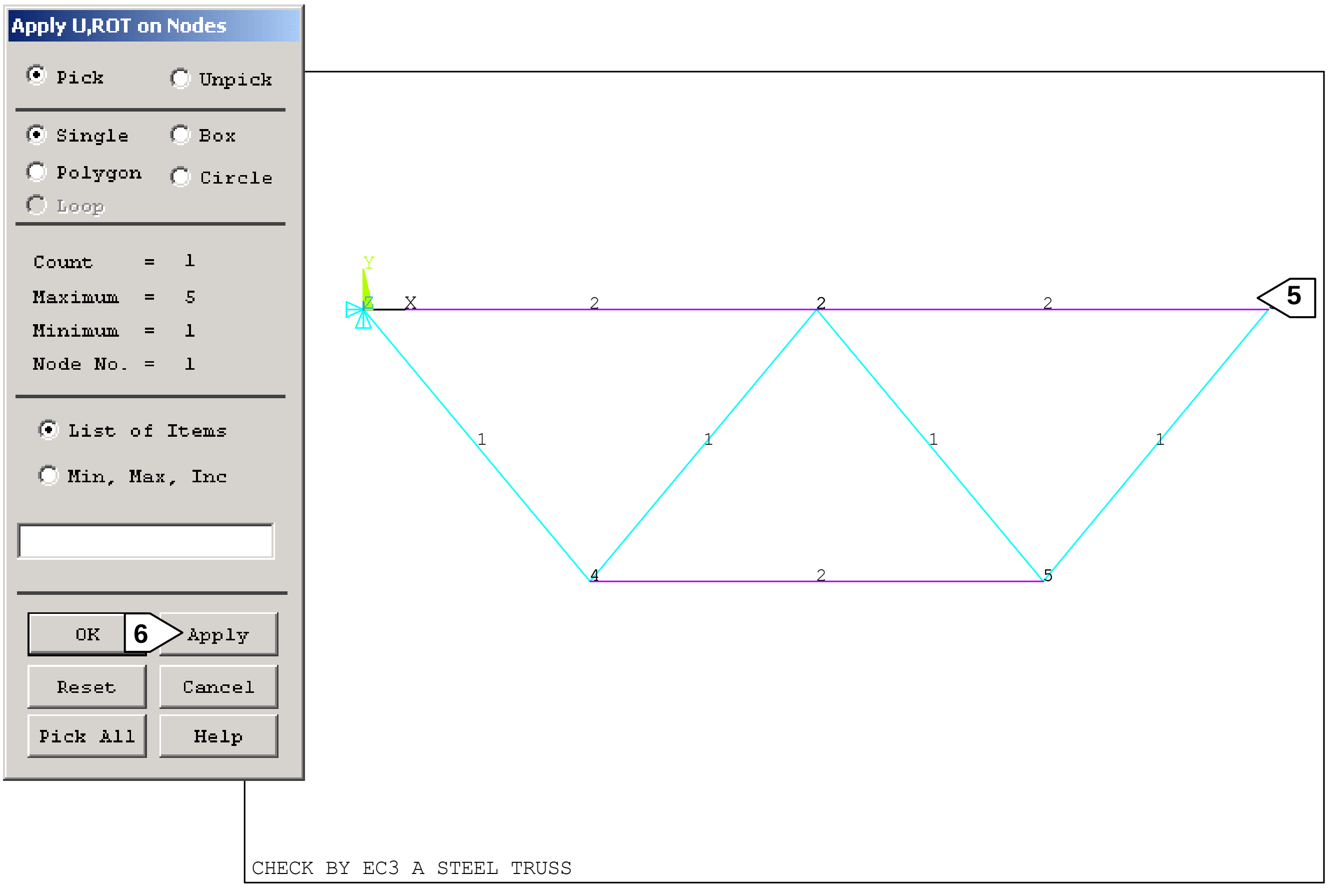
Main Menu: **Solution** → – Loads – **Apply** → – Structural – **Displacement** → **On Nodes**

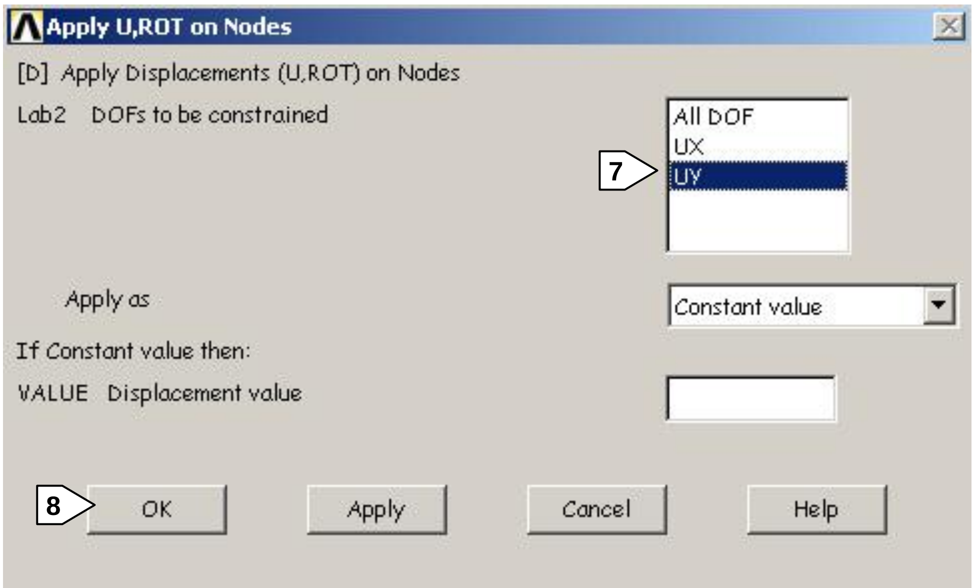
5 Pick the right end of the truss

6 Apply

7 Choose UY to be constrained

8 OK



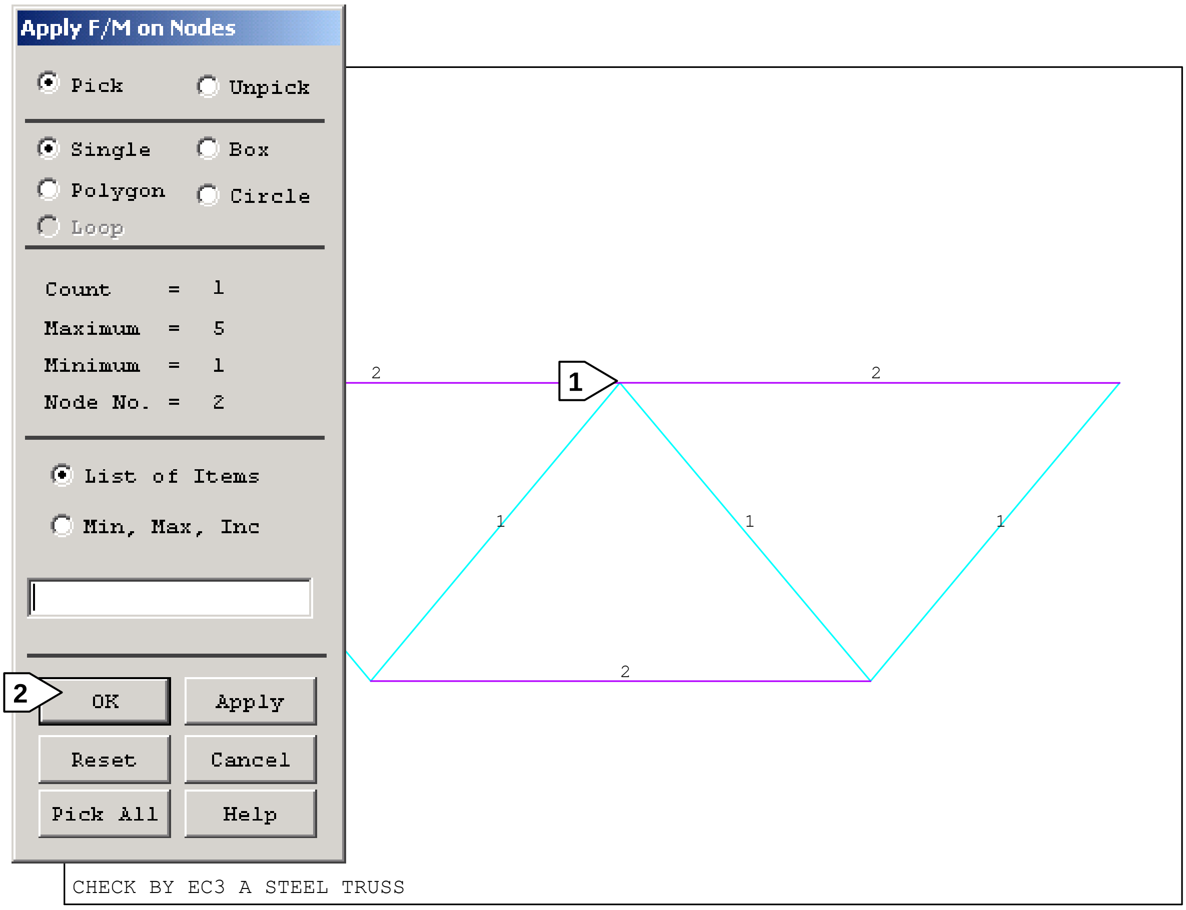


13. Apply force load

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

1 Pick on the mid span of the truss

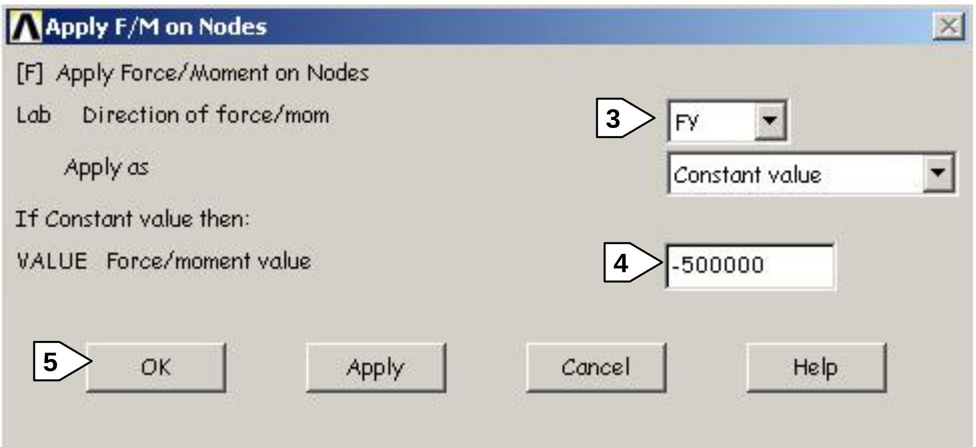
2 OK

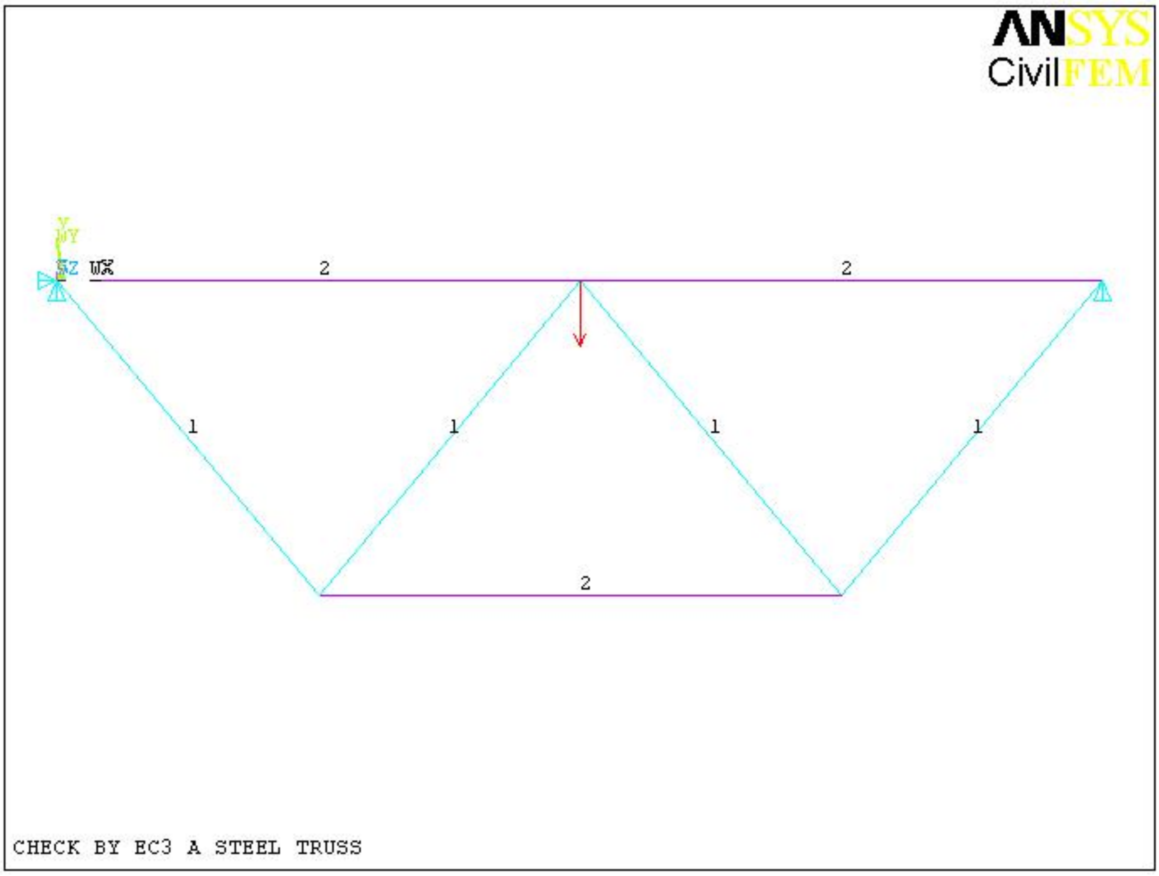


3 Choose FY as direction of force

4 Enter force value: -500000

5 OK

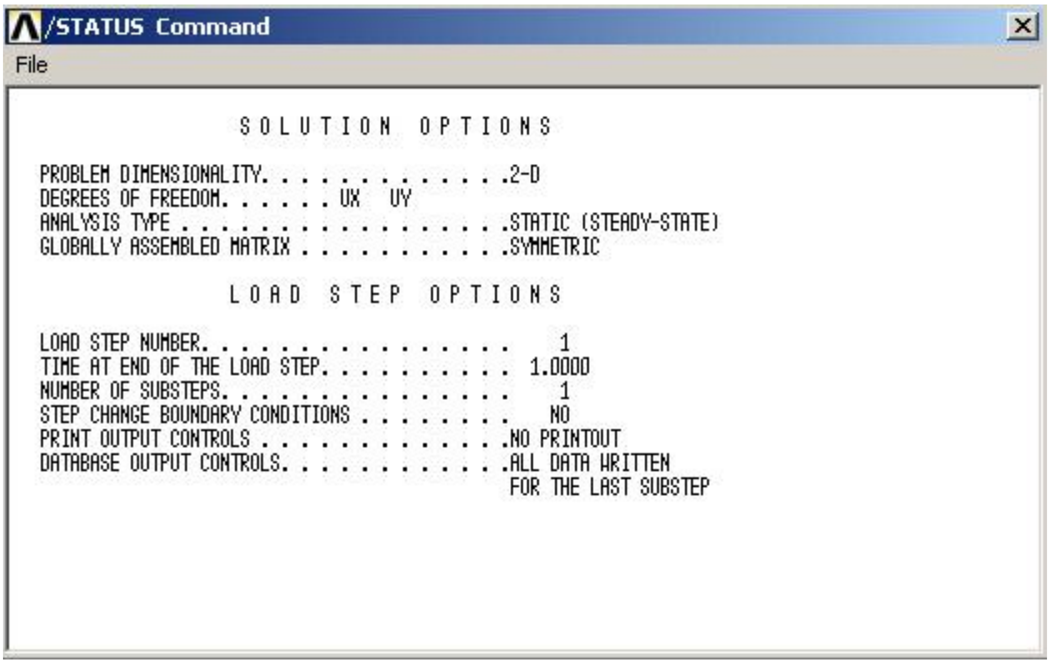


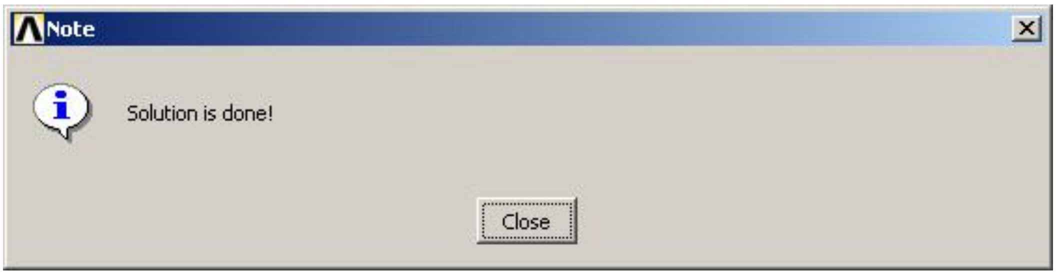


14. Solve

Utility Menu: **ANSYS Main Menu →Solution →** – Solution – **Solve current LS**

1 OK





Postprocessing

Postprocessing is where you review the analysis results through graphic displays and tabular listings.

15. Enter the postprocessor and read results

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

1 Enter 1 in the Load Step number box

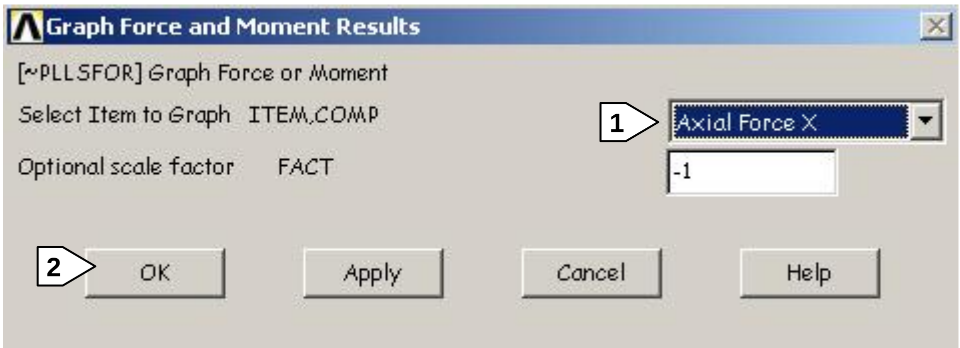
2 OK to read load step 1

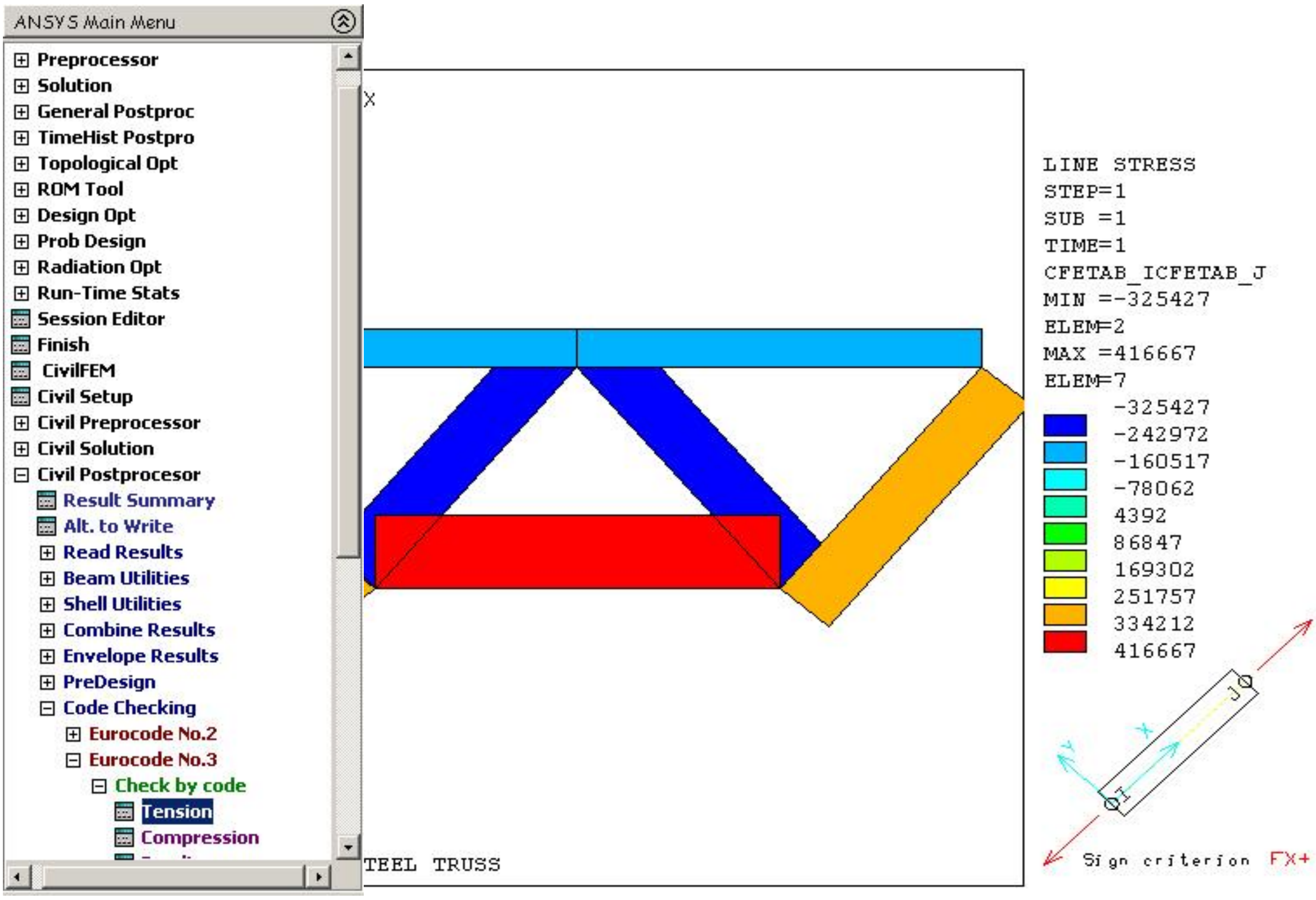
16. Plot Axial Force X

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

1 Choose Axial Force X

2 OK



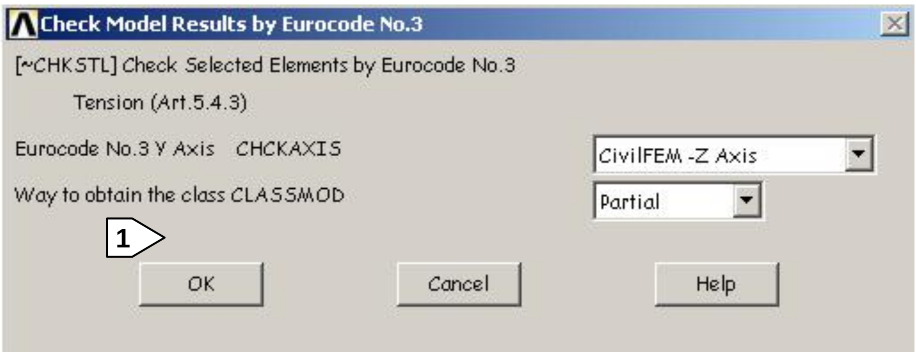


17. Checking for Tension according to Eurocode 3

We are going to check the elements for tension according to Eurocode 3 provisions. We will check considering the CivilFEM -Z axis as the principal axis or Y axis for Eurocode 3.

Main Menu: – CivilFEM – **Civil postprocess →Code checking →Eurocode 3** →CHECK BY CODE –**Tension**

1 OK for Eurocode 3 checking



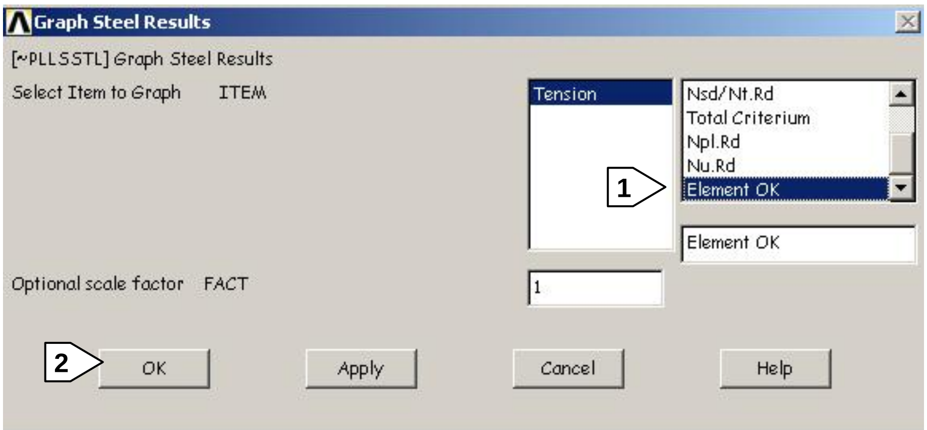
18. Review Elements OK and Not OK

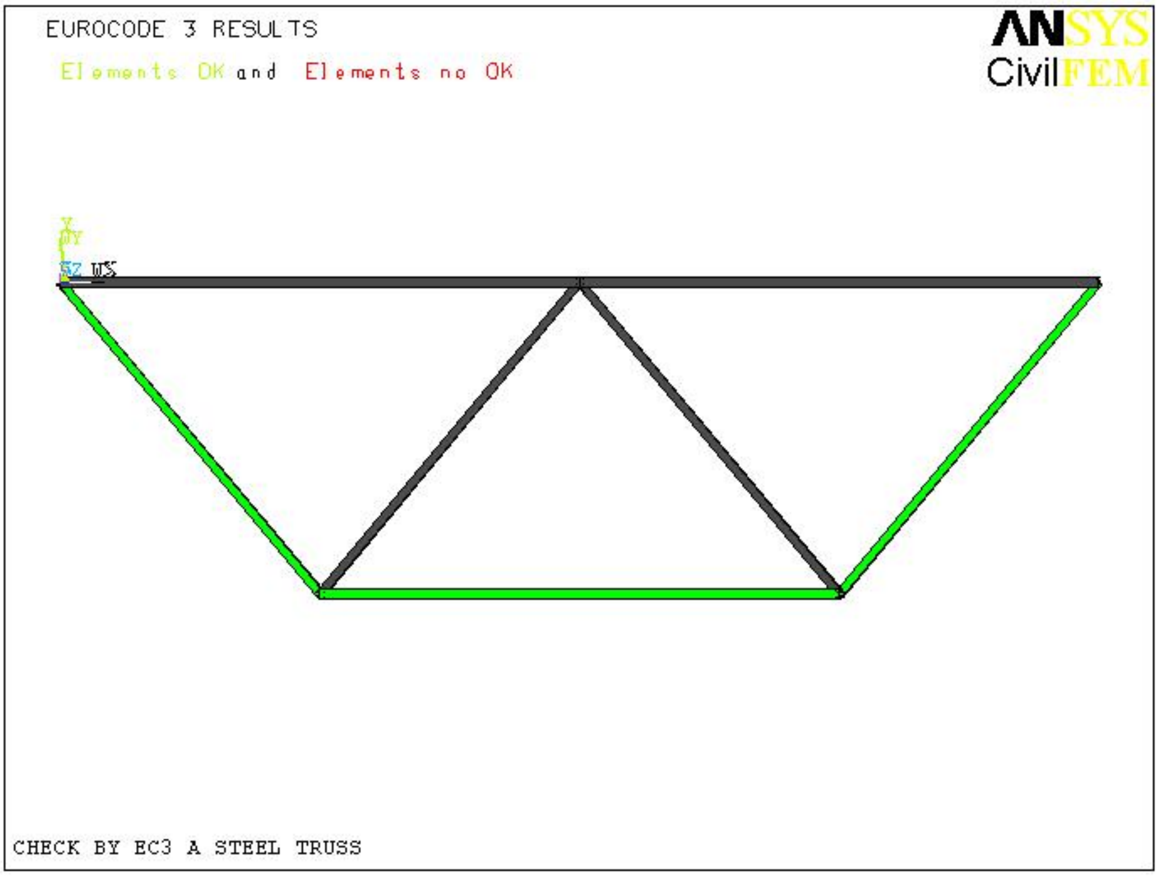
In the OK/Not OK graphs, elements that satisfy the code specifications for the requested check are plotted in green, while those that do not satisfy are plotted in red. Elements plotted in gray are elements that have not been checked.

Main Menu: – CivilFEM – **Civil postprocess →Code checking →Eurocode 3** →BEAM RESULTS: **Plot results**

1 Choose Elements OK/ Not OK

2 OK



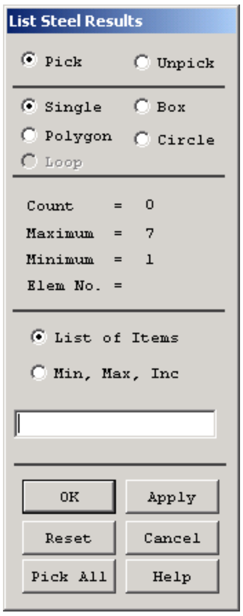


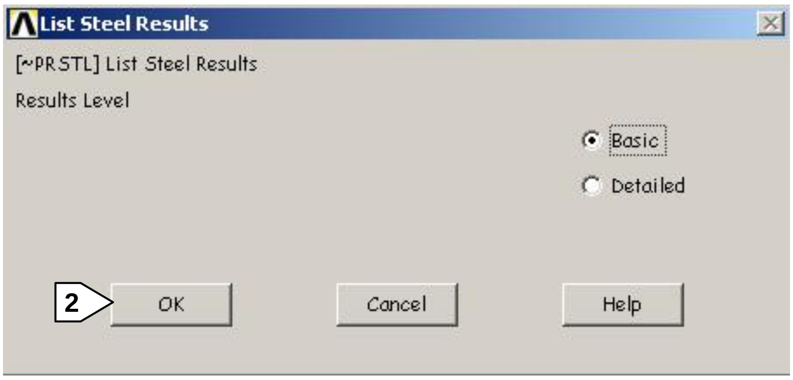
19. List Eurocode 3 Criterion Results

Main Menu: – CivilFEM – **Civil postprocess →Code checking →Eurocode 3** →BEAM RESULTS: **List results +**

1 Pick all

2 OK

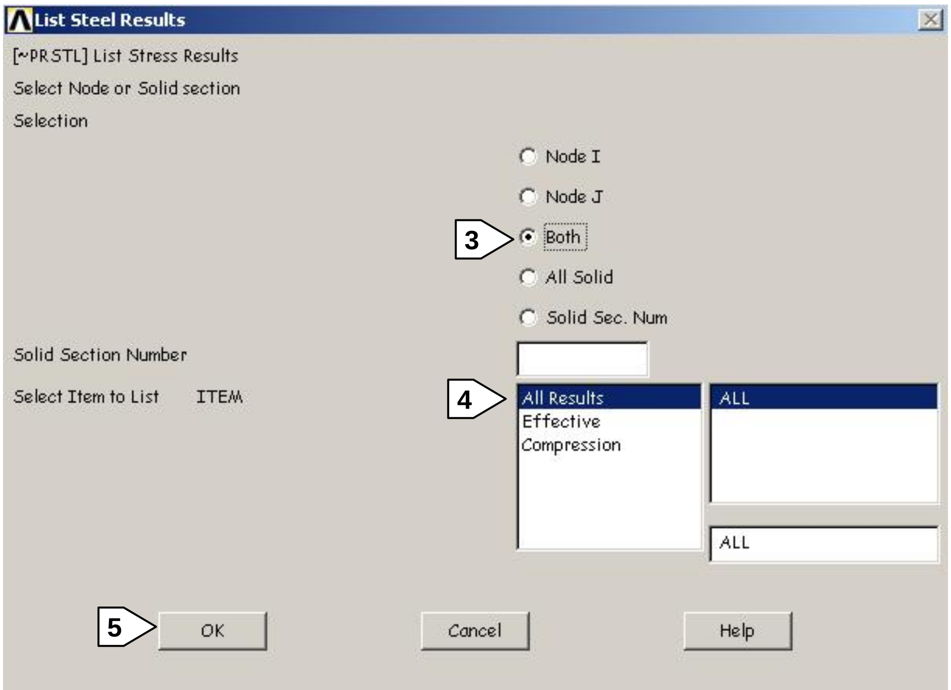




3 To list both nodes pick Both

4 Select ALL Results to list all the available results

5 OK



ANSYS **CivilFEM**

STRUCTURAL STEEL CHECKING RESULTS

Alternative 1

EC3 - TENSION (5.4.3)

Element	Node	CLASS	NSD	NTRD	CRT_N	CRT_TOT	NPLRD	NURD
1	I	1.000E+00	325.427E+03	388.000E+03	838.730E-03	838.730E-03	388.000E+03	480.499E+03
1	J	1.000E+00	325.427E+03	388.000E+03	838.730E-03	838.730E-03	388.000E+03	480.499E+03
4	I	1.000E+00	325.427E+03	388.000E+03	838.730E-03	838.730E-03	388.000E+03	480.499E+03
4	J	1.000E+00	325.427E+03	388.000E+03	838.730E-03	838.730E-03	388.000E+03	480.499E+03
7	I	1.000E+00	416.667E+03	600.000E+03	694.444E-03	694.444E-03	600.000E+03	743.040E+03
7	J	1.000E+00	416.667E+03	600.000E+03	694.444E-03	694.444E-03	600.000E+03	743.040E+03

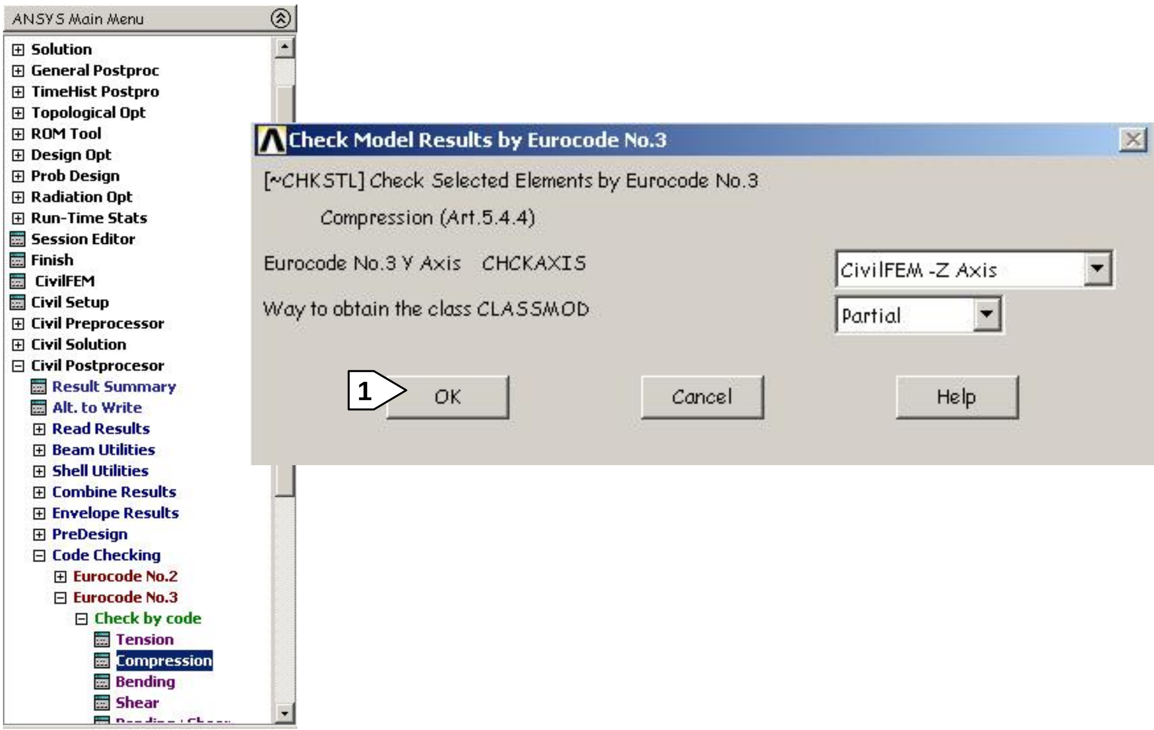
In the previous list we can see the Eurocode 3 Criterion for checking in accordance with the provisions of Article 5.4.3 (tension). Elements with a Eurocode 3 Criterion (CRT_TOT) greater than 1 are not OK elements, plotted in red. On the other hand, elements having a Eurocode 3 Criterion less than 1 are valid elements and are plotted in green. Elements 2, 3, 5 and 6 are not checked because they are in compression

20. Check for Compression according to Eurocode 3

We are going to check the elements for compression according to the Eurocode 3 provisions. We will check considering the CivilFEM -Z axis as the principal axis or Y axis for Eurocode 3.

Main Menu: – CivilFEM – **Civil postproc →Code →Eurocode 3** →CHECK BY CODE – **Compression**

1 OK



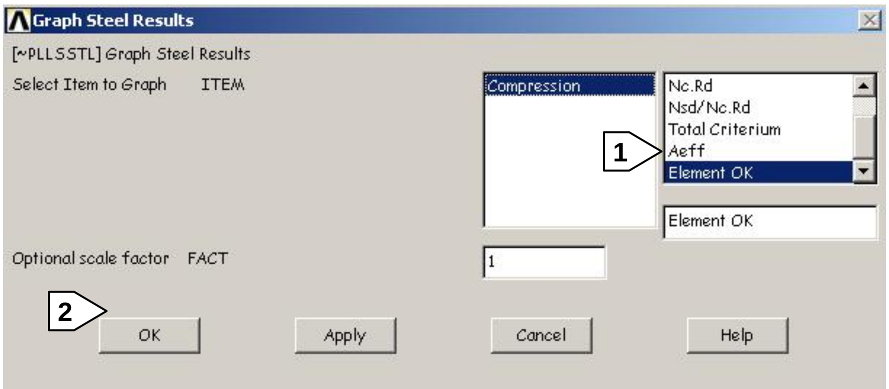
21. Review Elements OK and Not OK

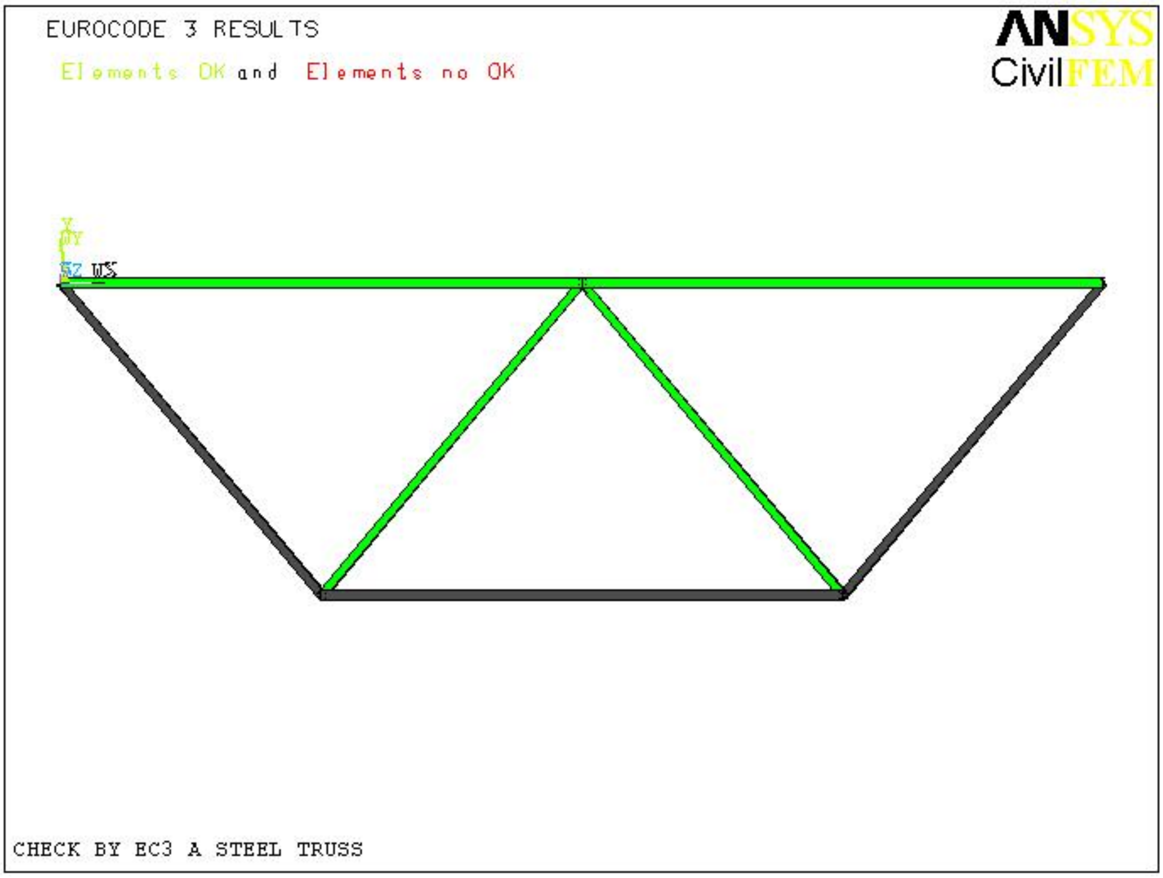
Valid elements are plotted in green while non-valid ones are plotted in red.

Main Menu: – CivilFEM – **Civil postproces →Code checking →Eurocode 3** →BEAM RESULTS: **Plot results**

1 Choose Element OK

2 OK



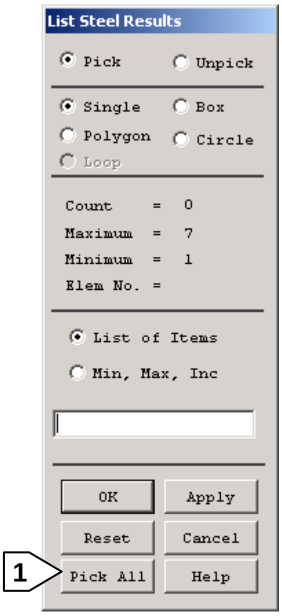


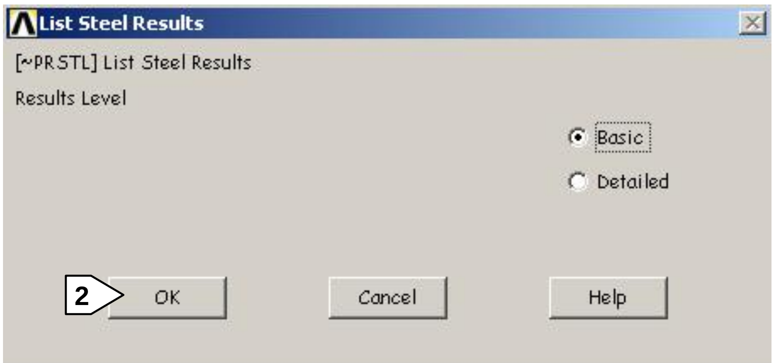
22. List Eurocode 3 Criterion Results

Main Menu: – CivilFEM – **Civil postproces →Code checking →Eurocode 3** →BEAM RESULTS: **List Results+**

1 Pick all

2 OK

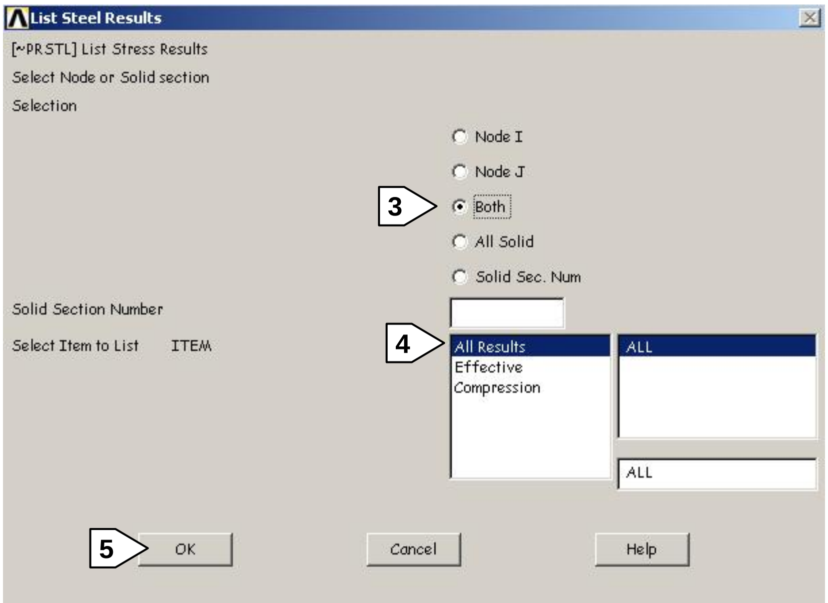




3 ist both nodes pick Both

4 Select ALL Results to list all the available results

5 OK



STRUCTURAL STEEL CHECKING RESULTS

Alternative 2

EC3-05 - COMPRESSION (6.2.4)

Element	Node	CLASS	NED	NCRD	CRT_N	CRT_TOT	AREA
2	I	3.000E+00	325.427E+03	388.000E+03	838.730E-03	838.730E-03	1.552E-03
2	J	3.000E+00	325.427E+03	388.000E+03	838.730E-03	838.730E-03	1.552E-03
3	I	3.000E+00	325.427E+03	388.000E+03	838.730E-03	838.730E-03	1.552E-03
3	J	3.000E+00	325.427E+03	388.000E+03	838.730E-03	838.730E-03	1.552E-03
5	I	4.000E+00	208.333E+03	580.891E+03	358.644E-03	358.644E-03	2.324E-03
5	J	4.000E+00	208.333E+03	580.891E+03	358.644E-03	358.644E-03	2.324E-03
6	I	4.000E+00	208.333E+03	580.891E+03	358.644E-03	358.644E-03	2.324E-03
6	J	4.000E+00	208.333E+03	580.891E+03	358.644E-03	358.644E-03	2.324E-03

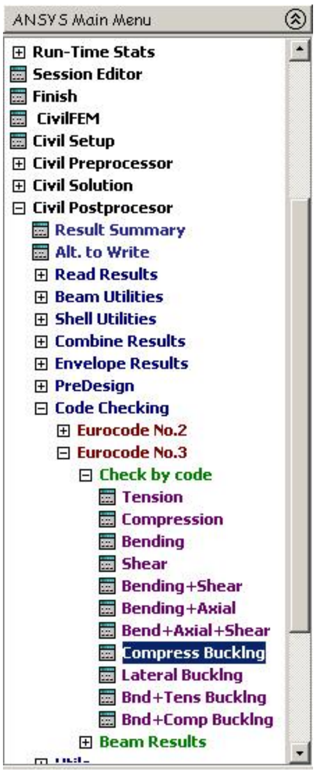
As we explained before, the elements with a criterion greater than 1 are not valid in accordance with Eurocode 3.

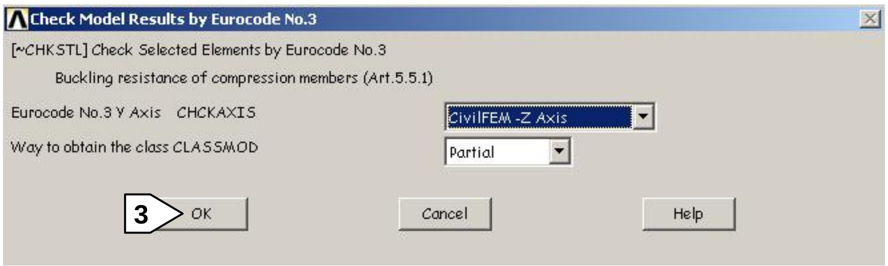
23. Check for Buckling Compression

We are going to check the elements for buckling compression according to the Article 5.5.1. provisions of Eurocode 3. We will check considering the CivilFEM -Z axis as the principal axis or Y axis for Eurocode 3.

Main Menu: – CivilFEM – **Civil postproc** →**Code checking** →**Eurocode 3** →CHECK BY CODE **Compression Buckling**

3 > OK





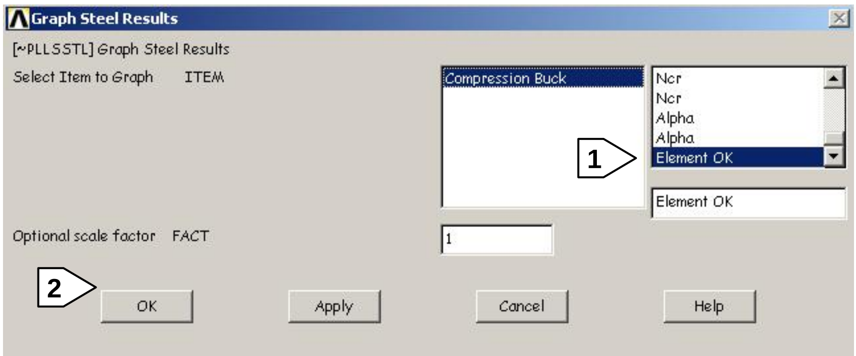
24. Review Elements OK and Not OK

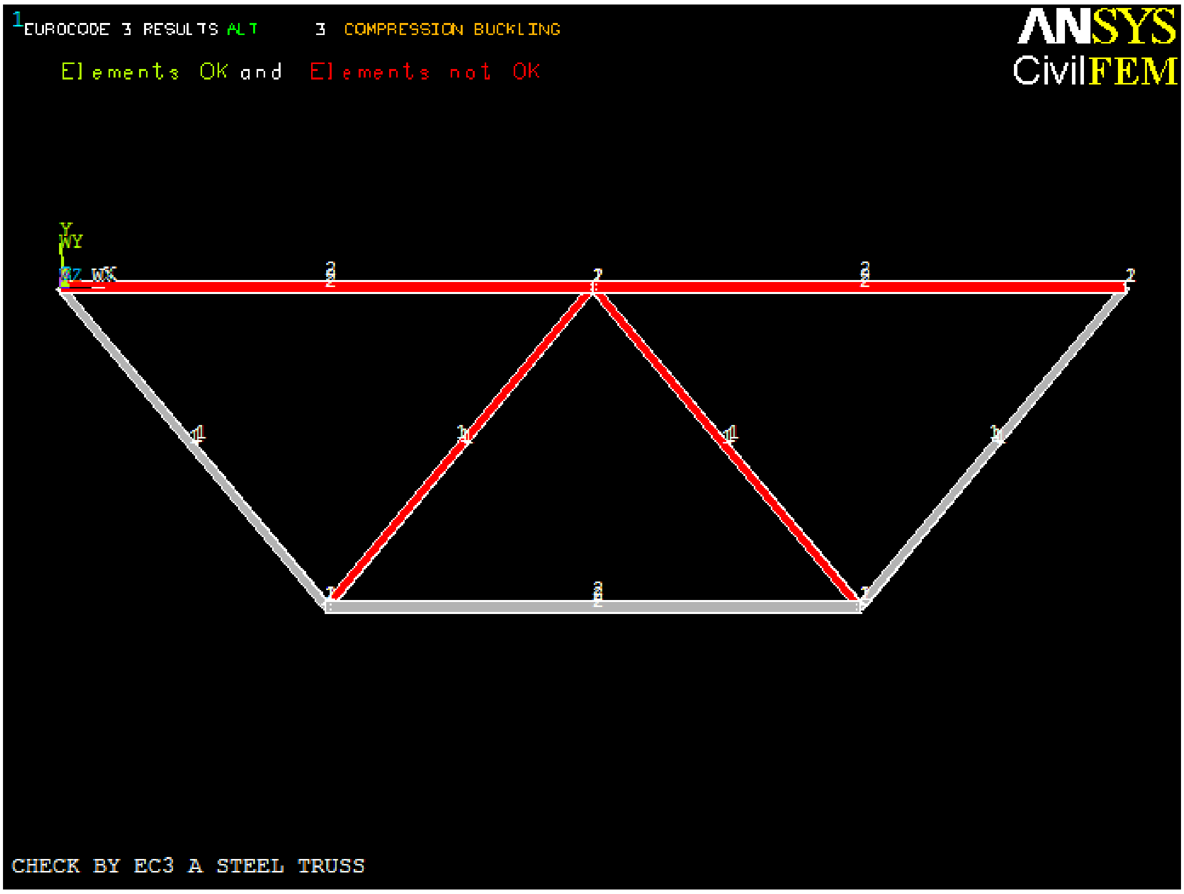
Valid elements are plotted in green and non-valid ones are plotted in red.

Main Menu: – CivilFEM – **Civil postproces →Code checking →Eurocode 3 →**BEAM RESULTS: **Plot results+**

1 > Choose Elements OK

2 > Choose basic level results and click OK



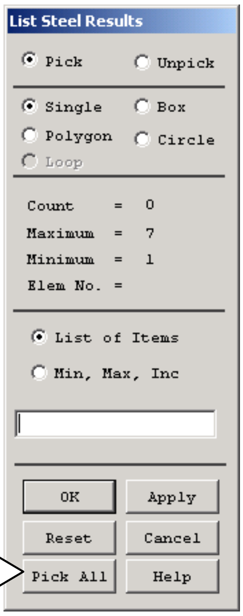


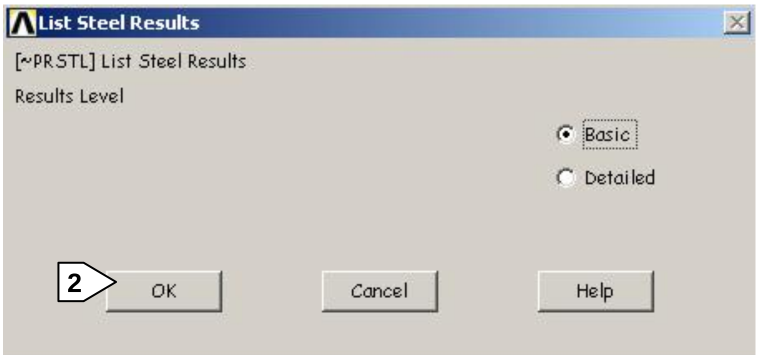
25. List Eurocode 3 Criterion Results

Main Menu: – CivilFEM – **Civil postproces →Code checking →Eurocode 3** →BEAM RESULTS: **List results+**

1 Pick all

2 Basic

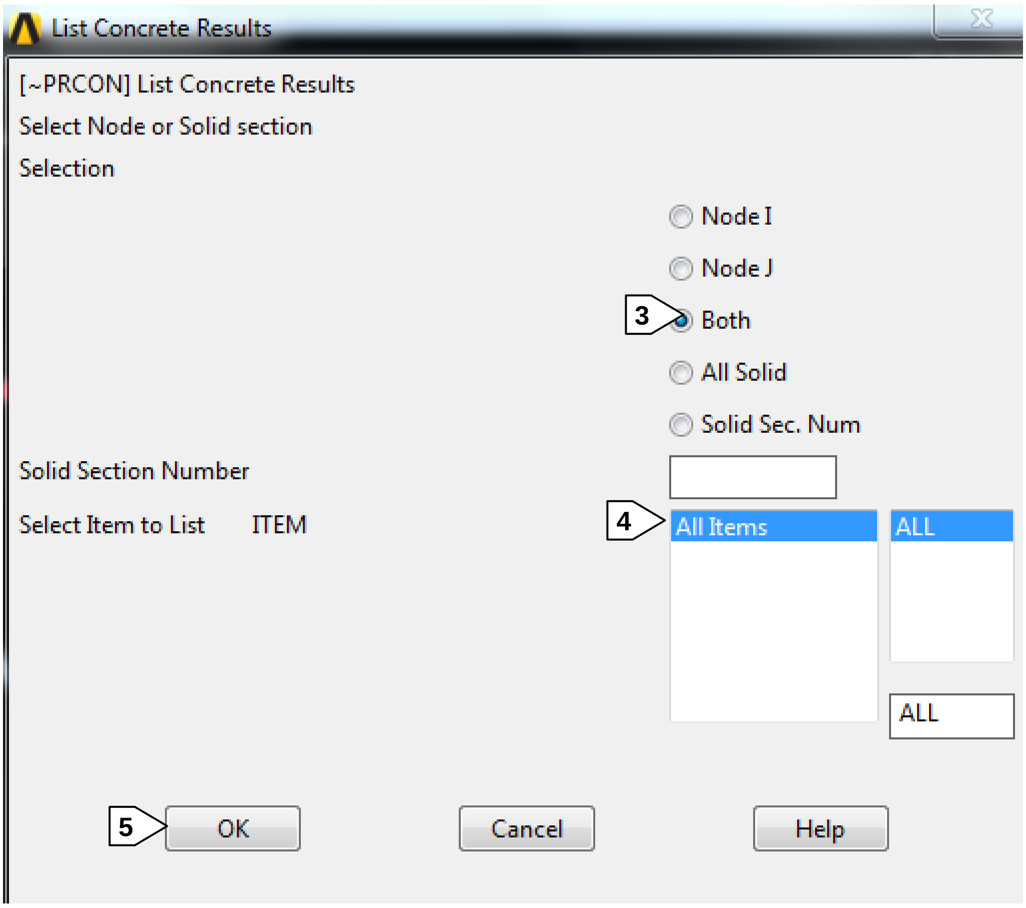




3 List both nodes pick Both

4 Select ALL

5 OK



STRUCTURAL STEEL CHECKING RESULTS

Alternative 3

EC3-05 - COMPRESSION BUCKLING (6.3.1)

Element	Node	CRT_CB
2	I	1.181E+00
2	J	1.181E+00
3	I	1.181E+00
3	J	1.181E+00
5	I	1.152E+00
5	J	1.152E+00
6	I	1.152E+00
6	J	1.152E+00

In the previous list we can see the Eurocode 3 Criterion for buckling of compressed members. In that list, elements are only checked in compression, that is, elements 2, 3, 5 and 6. All these elements exceed that criterion. Elements 1, 4 and 7 are in tension.

26. Exit the ANSYS program

Utility Menu: **File →Exit**

1 Pick on Save Everything

2 OK