

2. *Elastic Beam Analysis with Generic Section*

Applicable CivilFEM Product: All CivilFEM Products

Level of Difficulty: Easy

Interactive Time Required: 30-35 minutes

Discipline: Structural Steel

Analysis Type: Linear static

Element Type Used: BEAM4

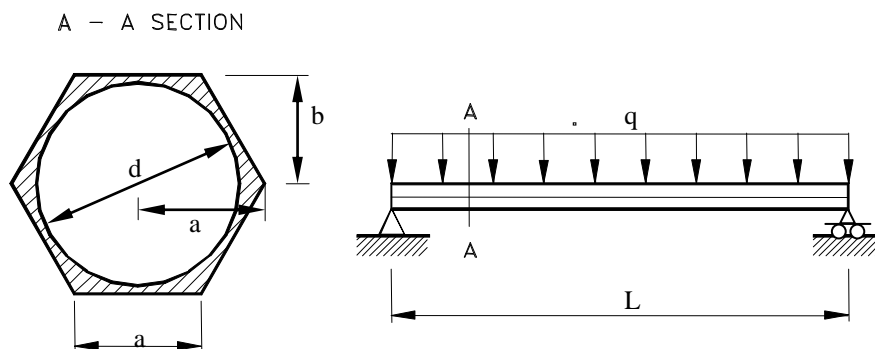
Active Code: Eurocode 3

Units System: lbf, ft, s

CivilFEM Features Demonstrated: Units selection, code selection, material definition, generic section definition, postprocessing of stresses

Problem Description

The objective is to analyze the behavior of a simply supported beam spanning 33 ft and loaded with a 900 lbf/ft distributed load. It has a hexagonal hollow cross-section.



■ Given

The geometry and load distribution of the simply supported beam are shown in the previous figure. The following is a list of all the input parameters:

Material	Generic material
Young modulus	4.2E9 lbf/ft ²
Length	L = 33 ft
Distributed load	q = 900 lbf/ft
Section geometric parameters	
	a = 2 ft
	b = a · $\sqrt{3}$ / 2 ft
	d = 3 ft

■ Approach and Assumptions

We are going to discretize the beam with a 3D model, using linear beam elements. Model geometry is defined with solid modeling and automatic meshing of elements and nodes.

■ Summary of Steps

Preprocessing

1. Specify title
2. Set code and units
3. Define material
4. Define element type
5. Capture section
6. Define Beam & Shell properties
7. Define solid modeling entities
8. 8. Mesh
9. Save the database

Solution

10. Apply displacement constraints
11. 11. Apply pressure load
12. Solve

Postprocessing

13. Enter the postprocessor and read results
14. Plot the deformed shape
15. Plot bending stress in Z top fiber
16. List bending stress in Z top fiber
17. Exit the ANSYS program

Interactive Step-by-Step Solution

■ Preprocessing

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

1. Specify title

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

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

- 1 Enter the title: "Elastic beam analysis with generic section"
- 2 OK to define the title and close the dialog box.



2. Set code and units

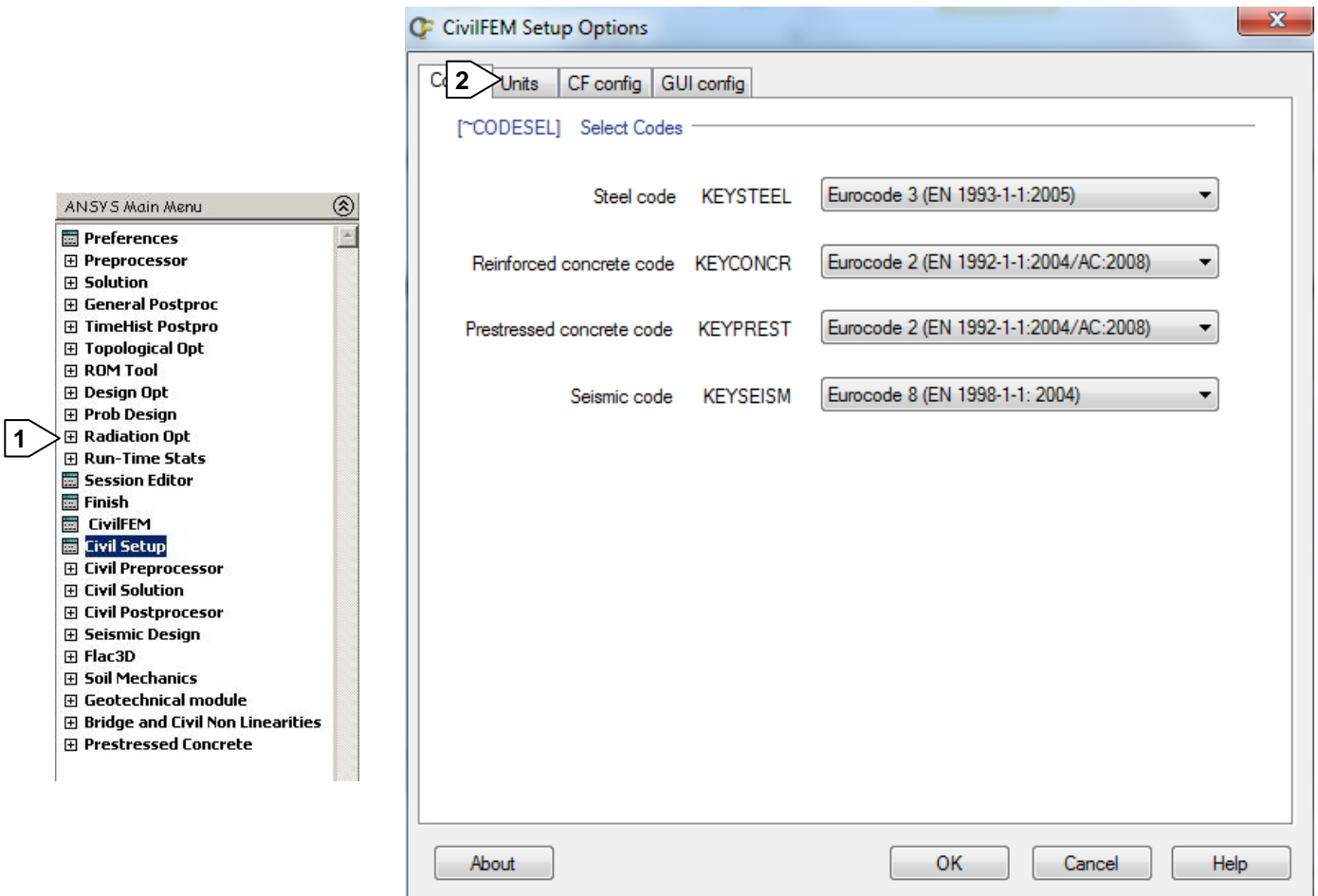
In CivilFEM you can choose between different codes for checking and designing. CivilFEM allows you to uphold different active codes simultaneously, one for concrete calculations another one for steel calculations and a third one for seismic design. In this example the active steel code is Eurocode No 3, it's the default code for steel so we don't need to change anything.

In CivilFEM you must also 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 American units, that is, feet, seconds and pounds-force.

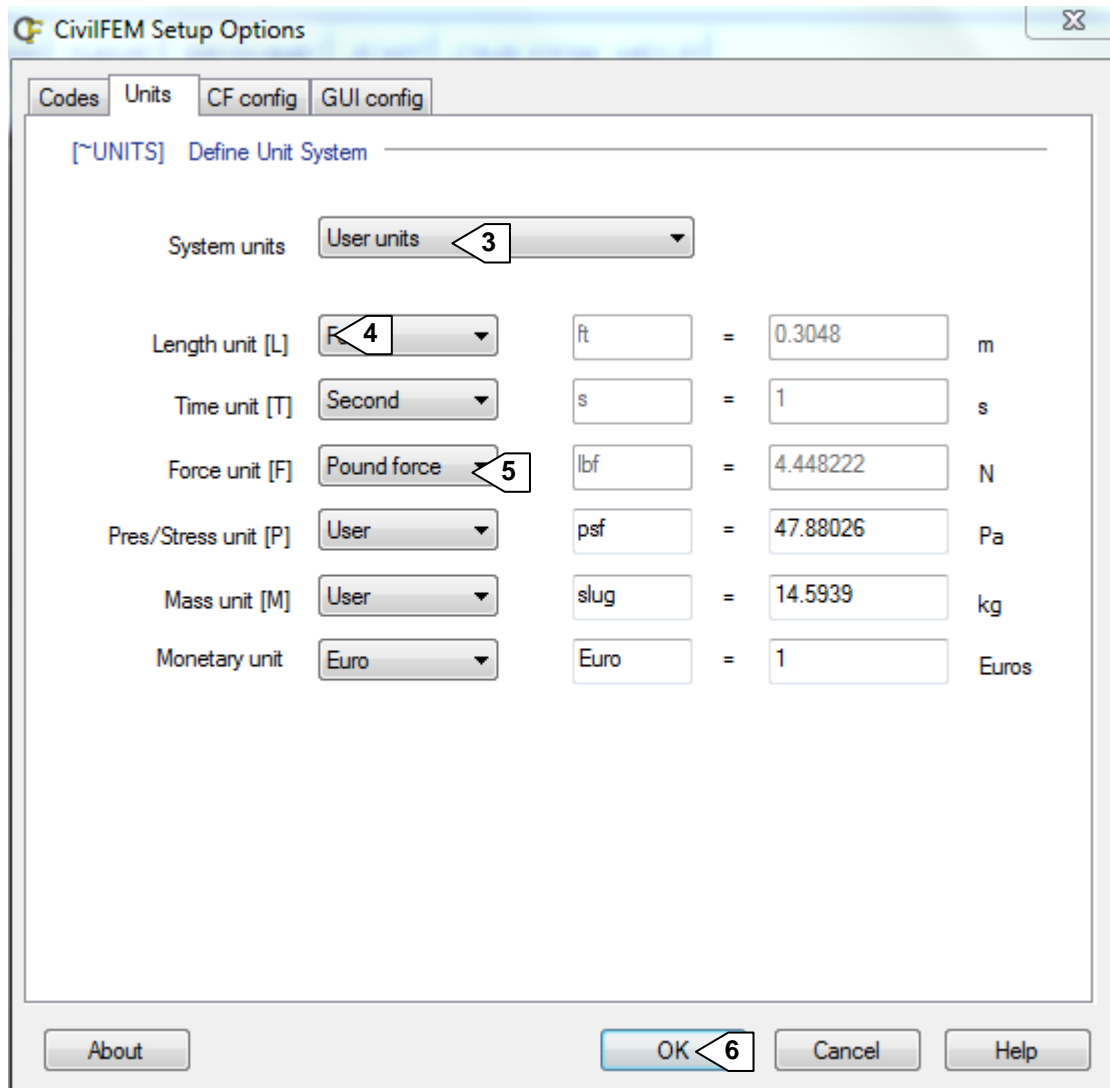
Main Menu: – CivilFEM – **Civil Setup**

- 1 Select CivilFEM Setup
- 2 Choose Units

2



- 3 Choose User units
- 4 Choose Length unit: Foot (ft)
- 5 Choose Force Unit: Pound Force (lbf)
- 6 OK to accept units and close the units dialog box



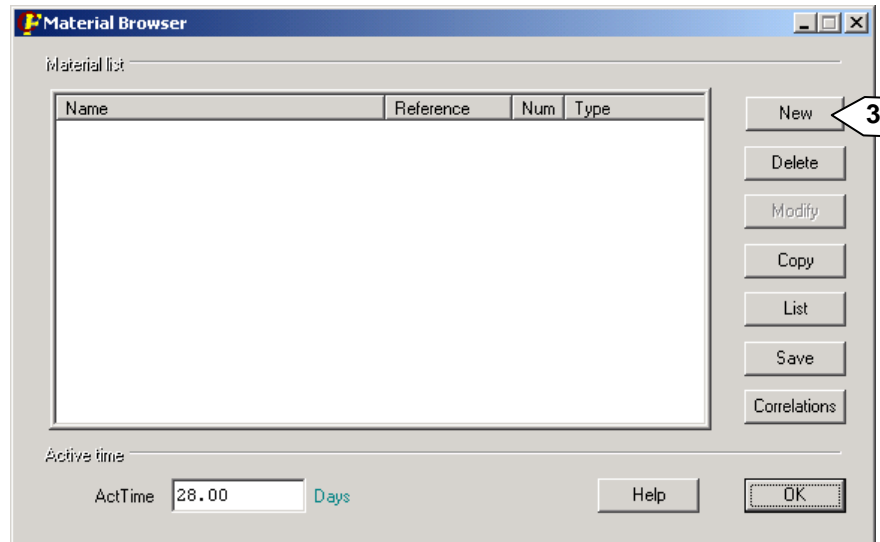
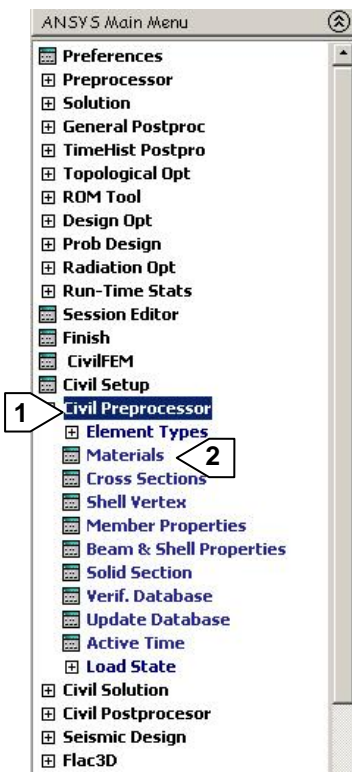
3. Define material

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 510 steel.

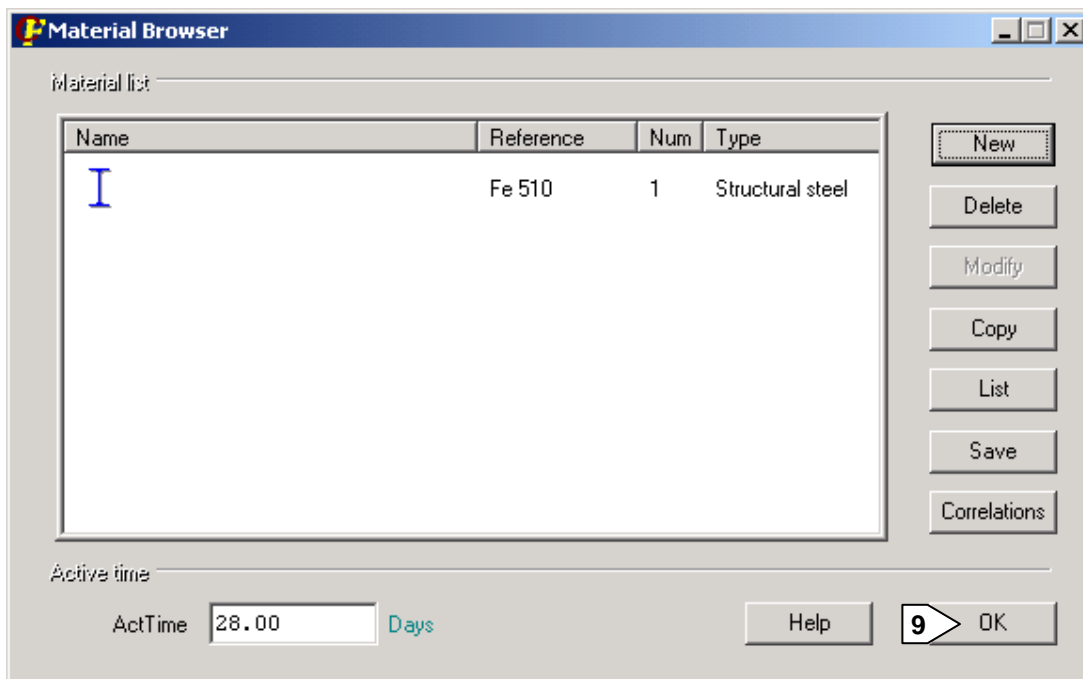
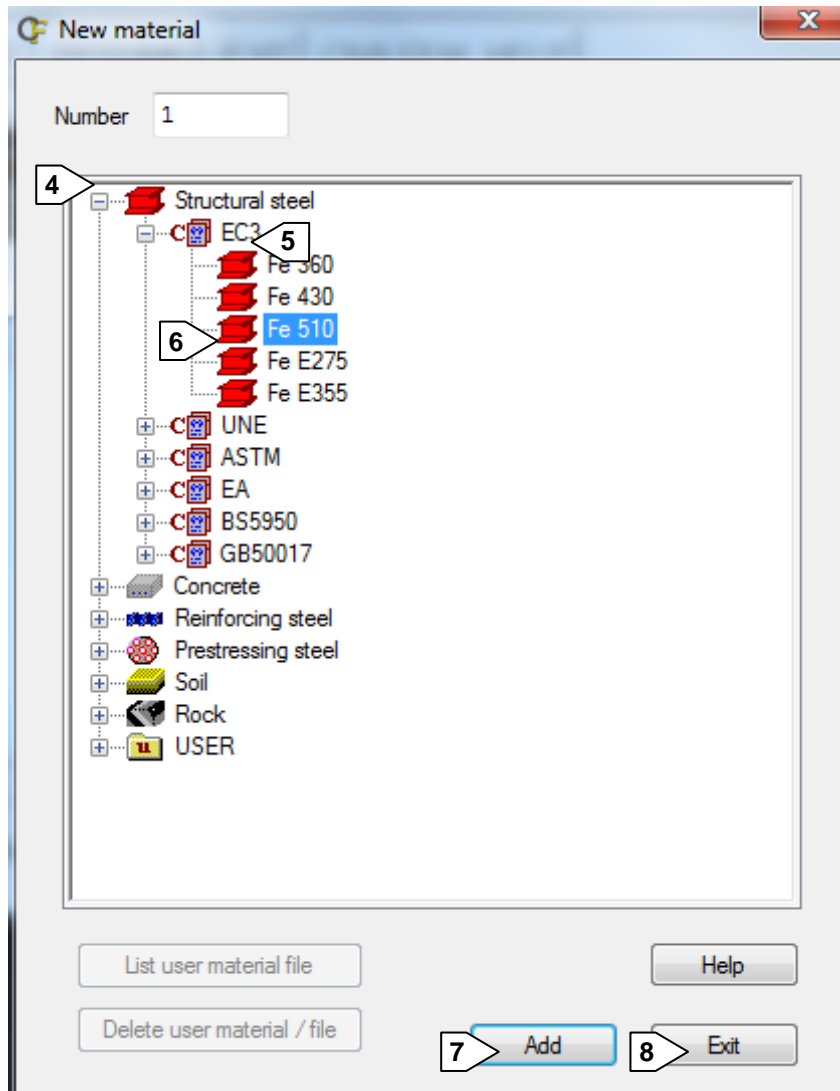
The CivilFEM **~CFMP** command allows us to define stress-strain diagrams, to define safety coefficients, to control the linear or non-linear behavior of the material and to select the activation time of the material.

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

- 1 Select Civil Preprocess
- 2 Choose Materials



- 3 Pick new to define a new material
- 4 Pick on the Steel icon for structural steel
- 5 Pick on the EC3 icon to choose steel from Eurocode 3 code
- 6 Choose Fe510 Steel and all the material properties corresponding to Fe510 steel are automatically calculated according to Eurocode 3 (active code)
- 7 Add to define the material properties set
- 8 Exit to close the dialog box
- 9 OK



4. Define element type

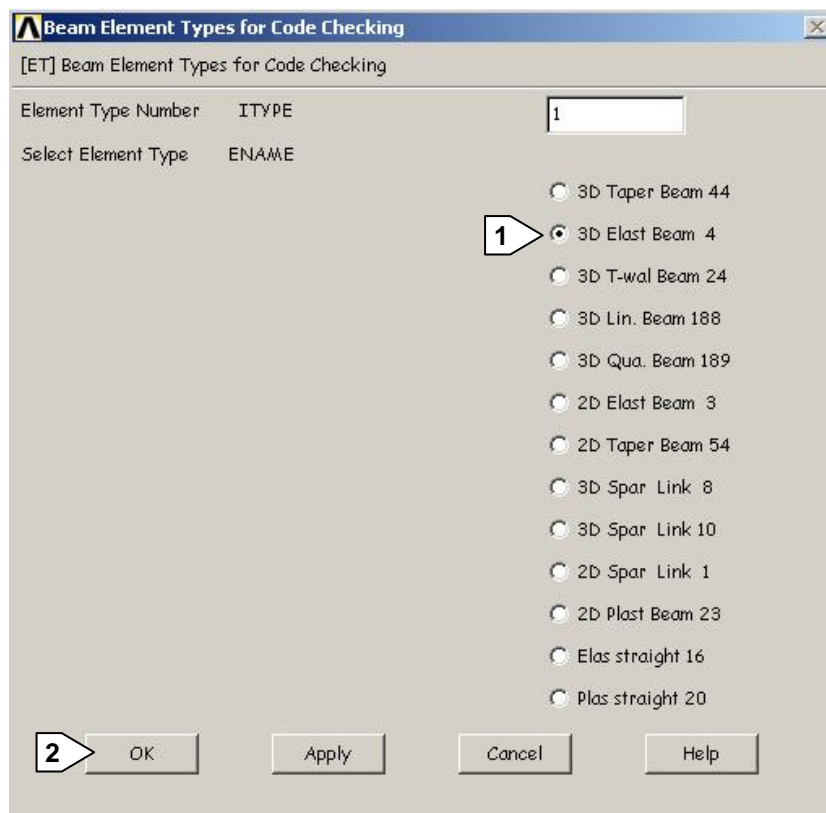
Checking and designing according to codes is performed only on CivilFEM supported element types. Although you can use any ANSYS element to define your model, only the CivilFEM supported elements will be checked according to codes. In the element type menu you can see the CivilFEM supported beam elements.

We will use a 3D elastic Beam 4 for this analysis.

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

1 Select 3D Elastic Beam 4

2 OK to define element type



5. Capture section

CivilFEM allows the use of ANSYS element MESH200 for automatic section definition, calculating its mechanical properties and defining its real constants.

First we define the element type MESH 200 with the 8 node quadrilateral option.

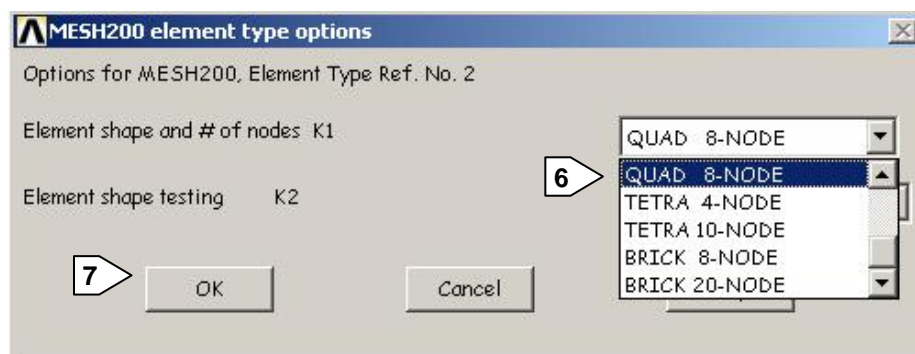
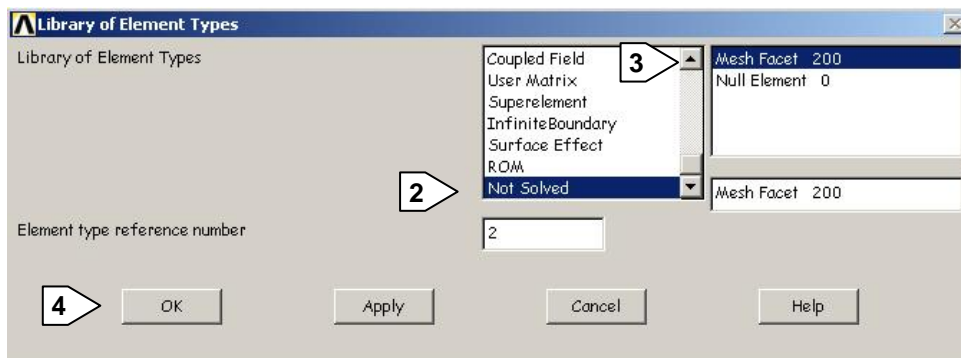
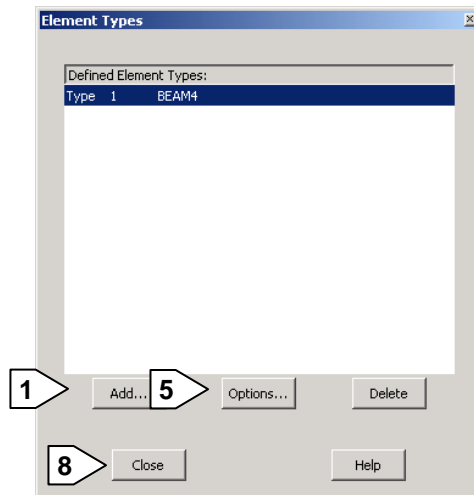
Main Menu: – CivilFEM – **Civil Preprocess** → **Element Types** → **Other Elements** → **Add/Edit/Delete**

1 Choose Add to add a new element type

2 Select Not Solved

3 Select Mesh Facet 200

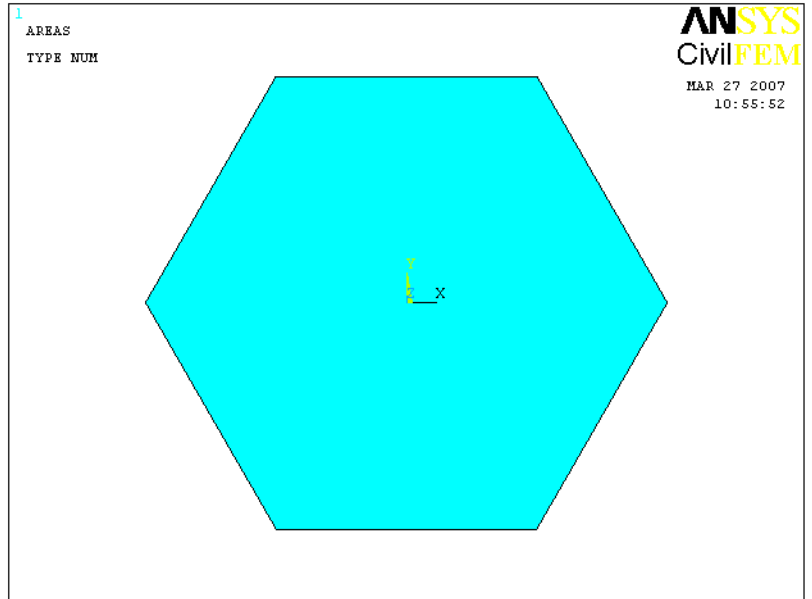
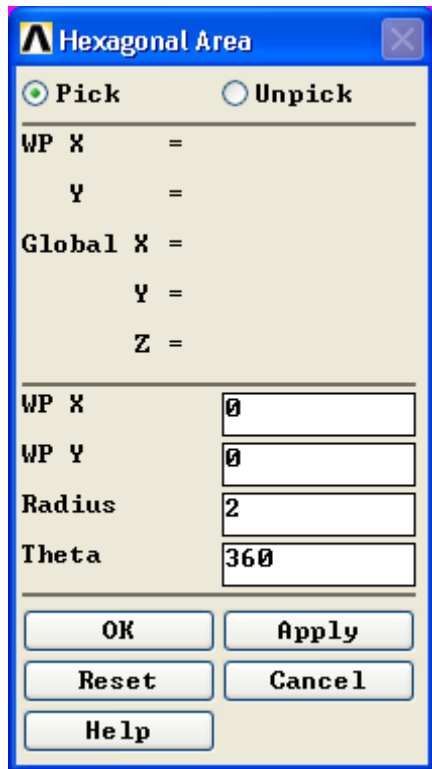
- 4 OK
- 5 Options
- 6 Select Quad 8 node
- 7 OK
- 8 Close



To define the hexagon:

Main Menu: – **Preprocessor** → – Modeling – **Create** → – Areas – **Polygon** → **Hexagon**

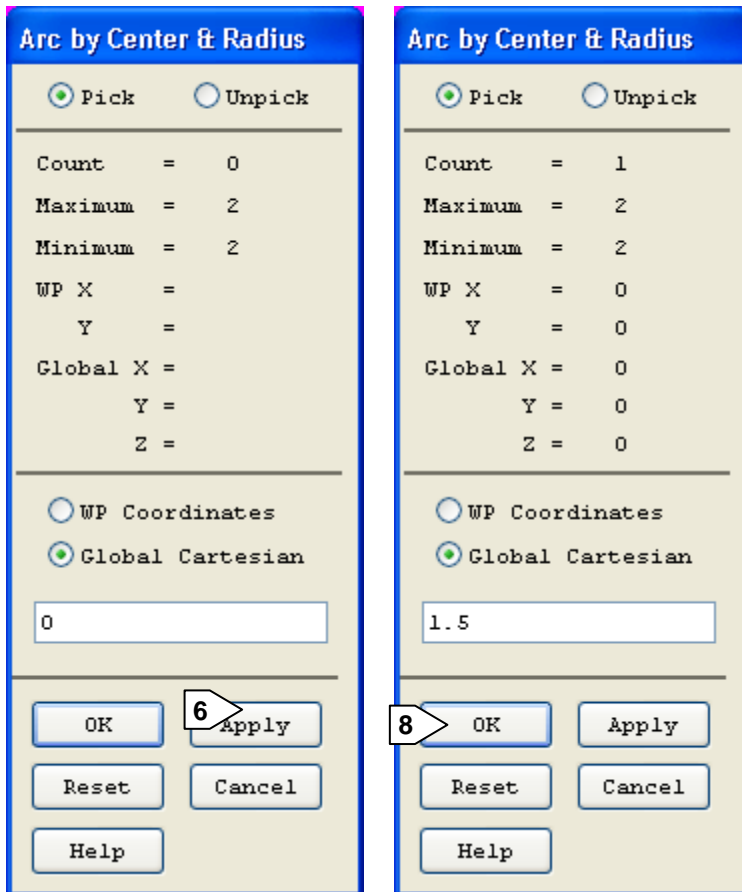
- 1 Enter 0 for Working Plane X coordinates
- 2 Enter 0 for Working Plane Y coordinates
- 3 Enter 2 for radius of the hexagon
- 4 OK to define hexagon



To define the central circle:

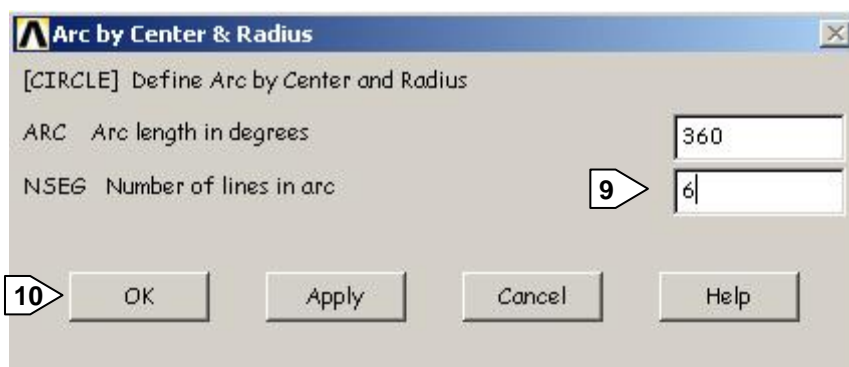
Main Menu: – **Preprocessor** → – Modeling – **Create** → – Lines – **Arcs** → **By Cent & Radius**

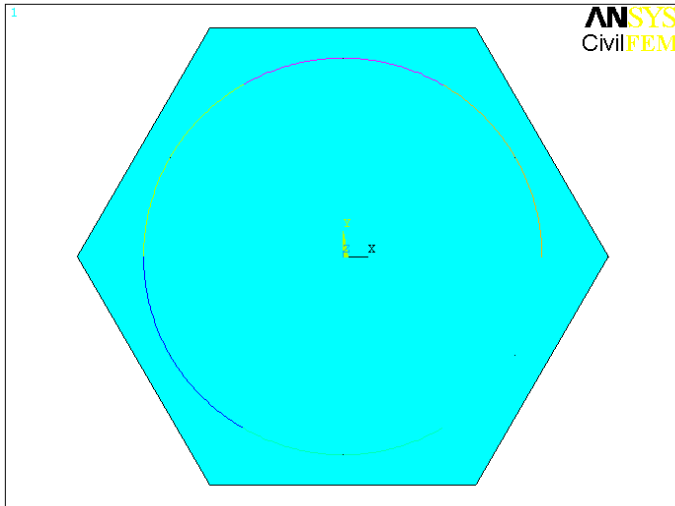
- 5 Enter 0
- 6 Apply
- 7 Enter 1.5
- 8 OK



9 Enter 6 lines in arc

10 Ok to define circle



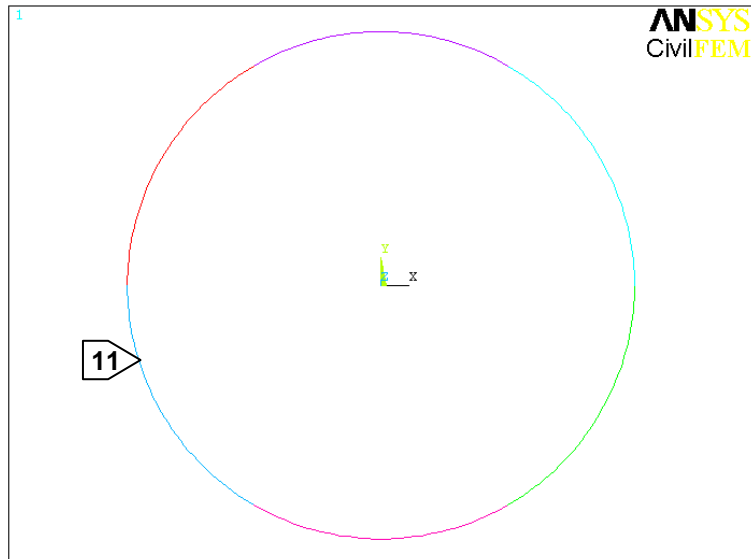
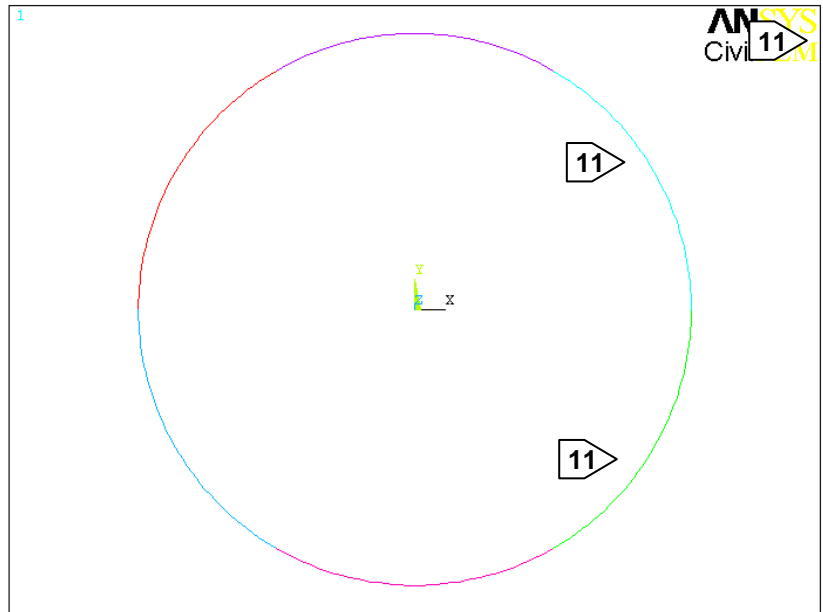
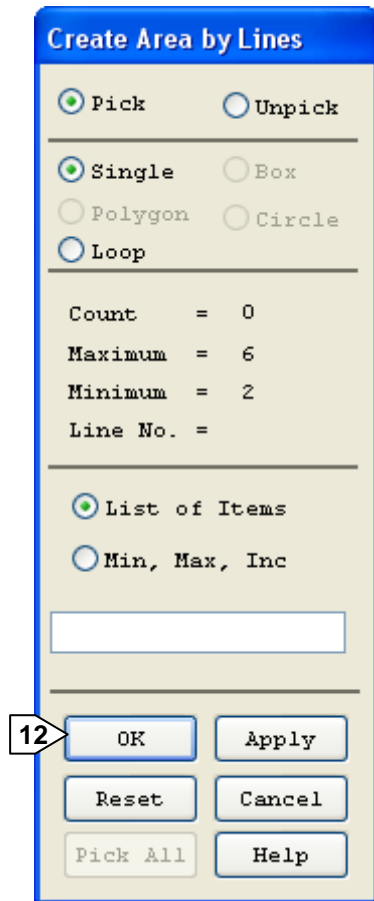


To define the central circles area:

Main Menu: – **Preprocessor** → – Modeling – **Create** → – Areas – **Arbitrary** →
By Lines

11 Pick lines

12 OK



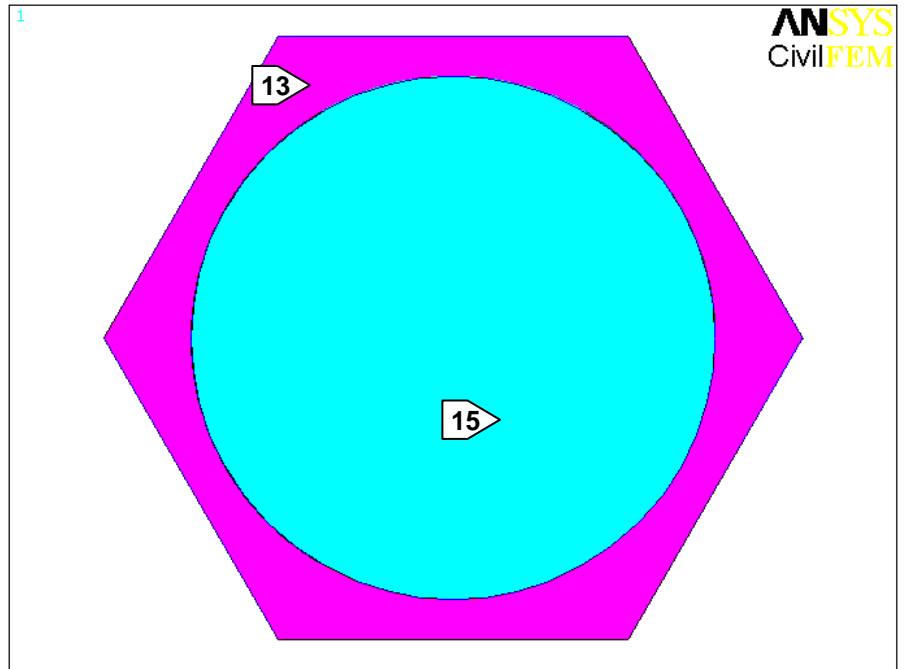
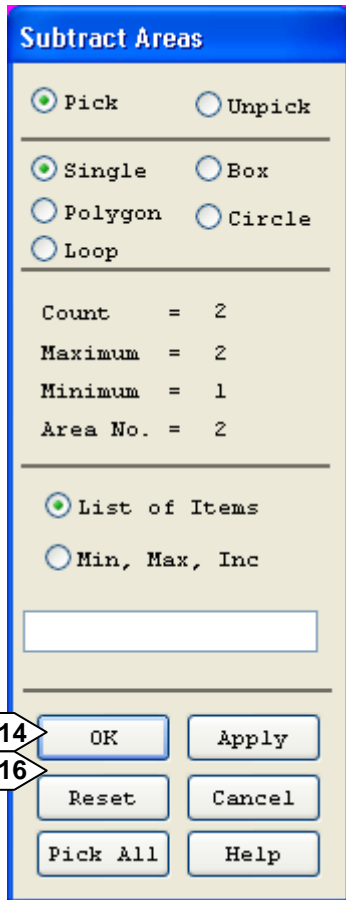
Main Menu: – **Preprocessor** → – Modeling – **Operate** → – Booleans – **Subtract**
→ **Areas**

13 Pick on the hexagon as area from which to subtract

14 OK

15 Pick on the circle as area to subtract

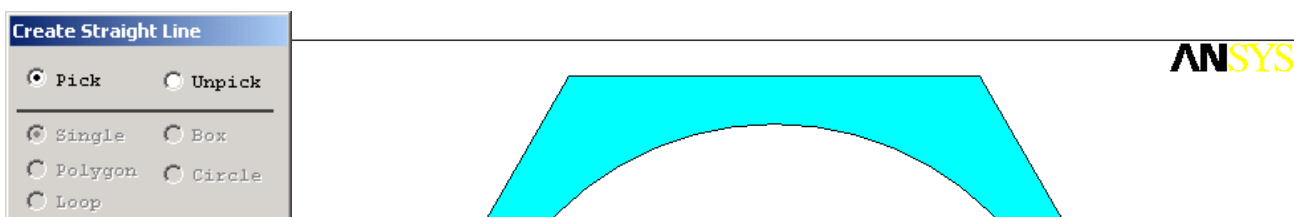
16 OK



Now we divide the area by lines into six to allow mapped meshing.

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

- 17 Pick on the first keypoint
- 18 Pick on the second keypoint
- 19 OK to define line



Repeating with the remaining keypoints we define the lines that we are going to use to divide de area of the section.

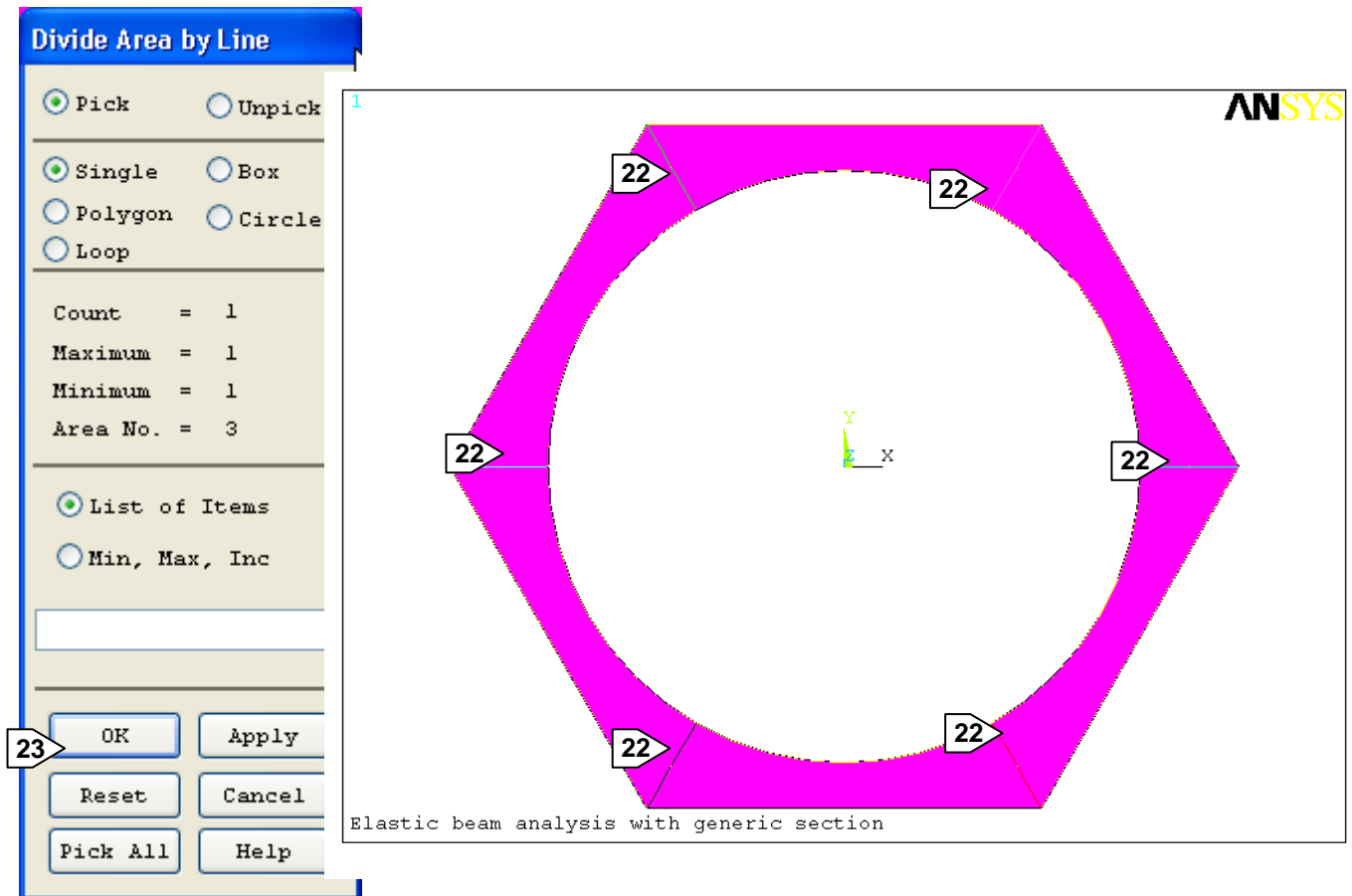
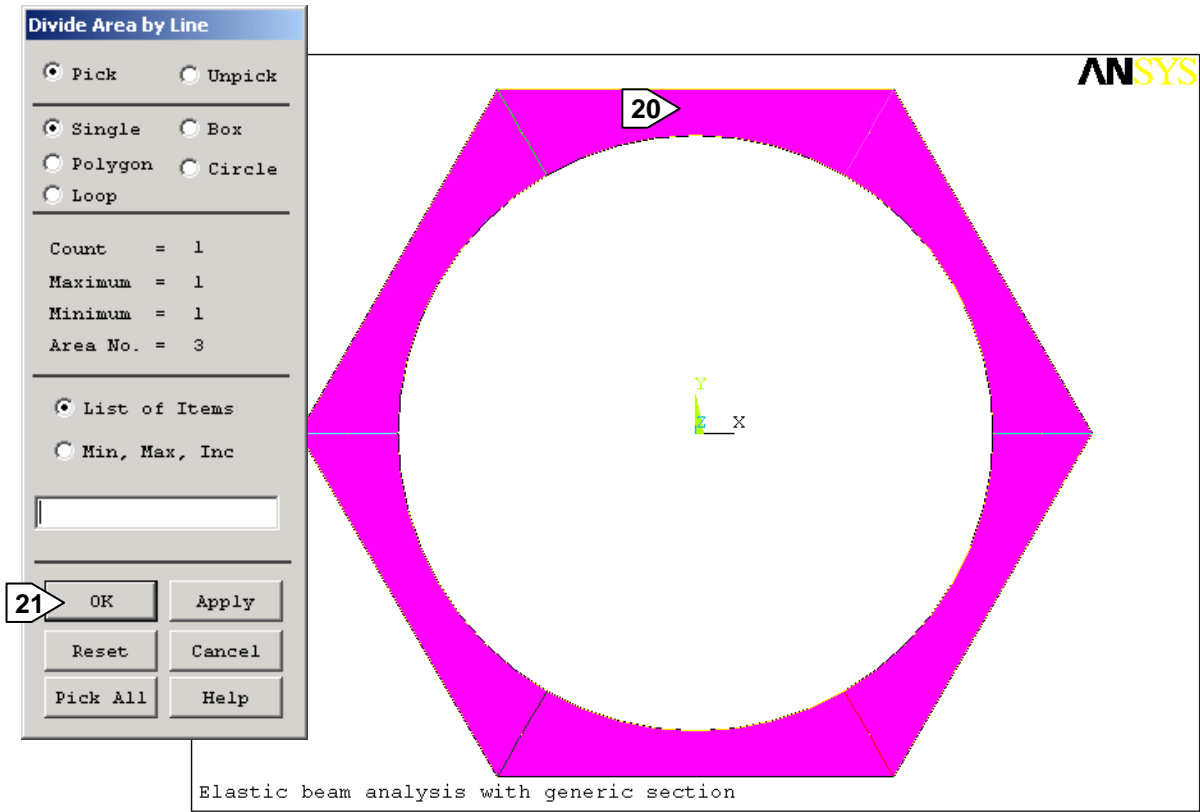
Main Menu: – **Preprocessor** → – Modeling – **Operate** → – Booleans – **Divide**
→ **Area by line**

20 Pick the area to divide

21 OK

22 Pick on the lines dividing the area

23 OK

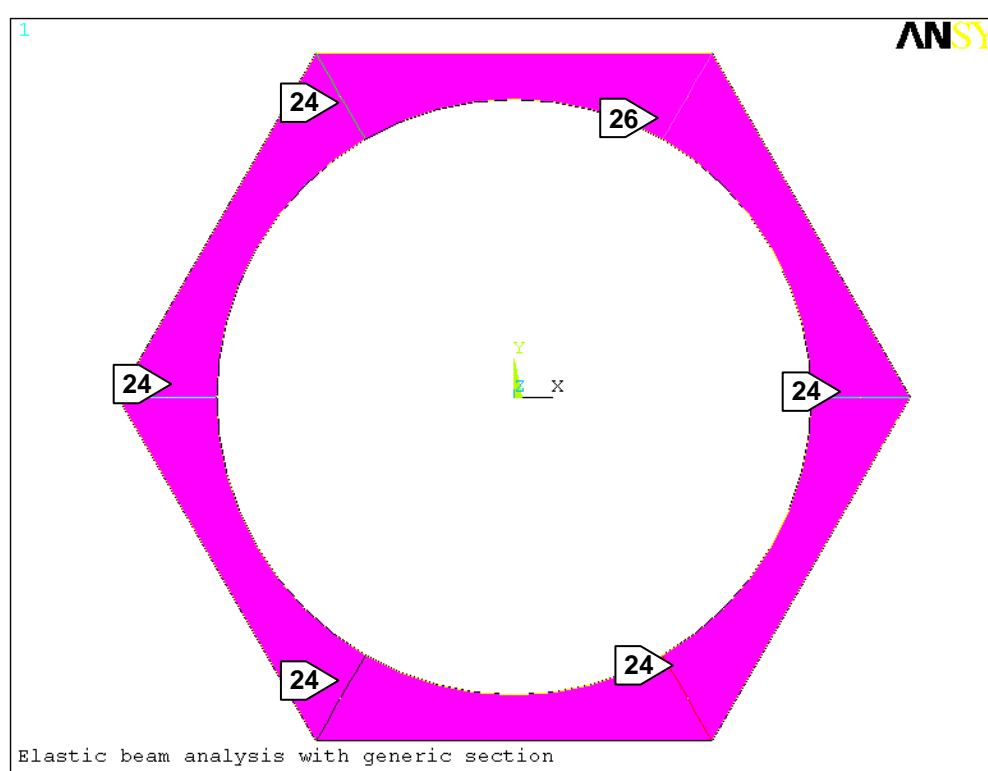
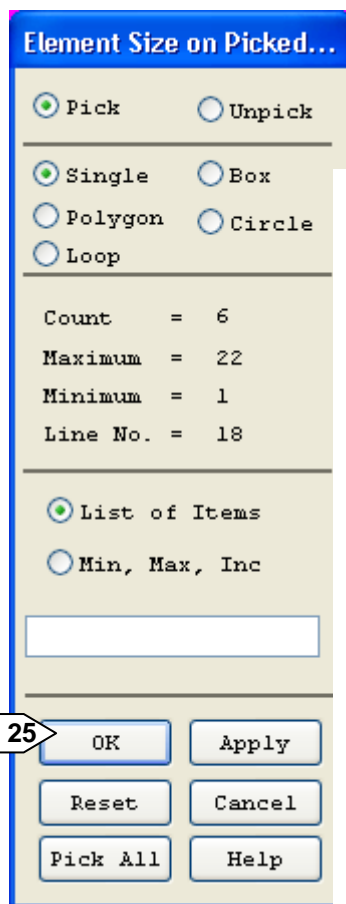


To control the mesh process we define the number of element divisions on each line.

Main Menu: – **Preprocessor** → – Meshing – **Size controls** → – Manual size
– – Lines – **Picked Lines**

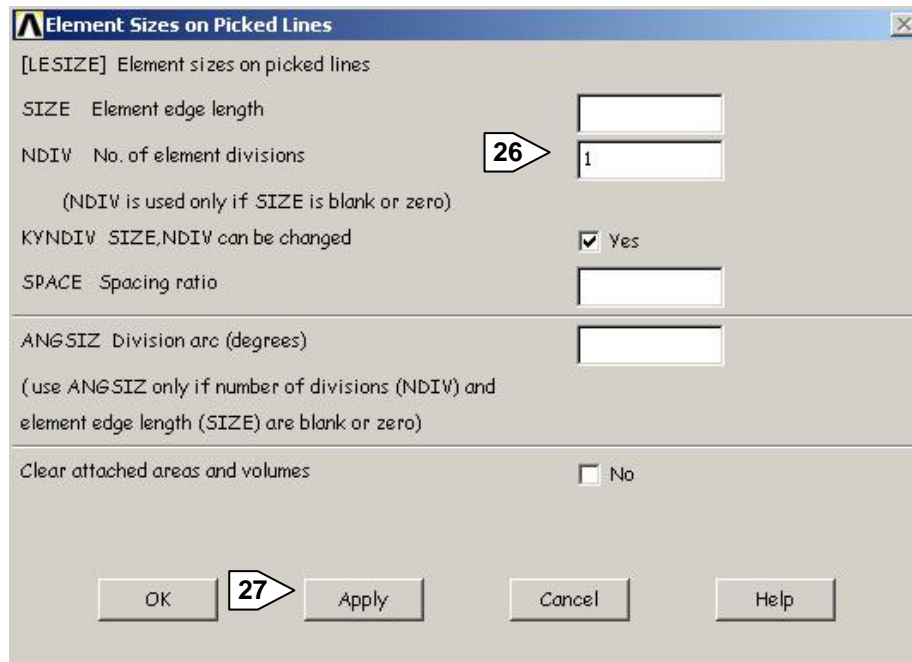
24 Pick the lines dividing the section

25 OK

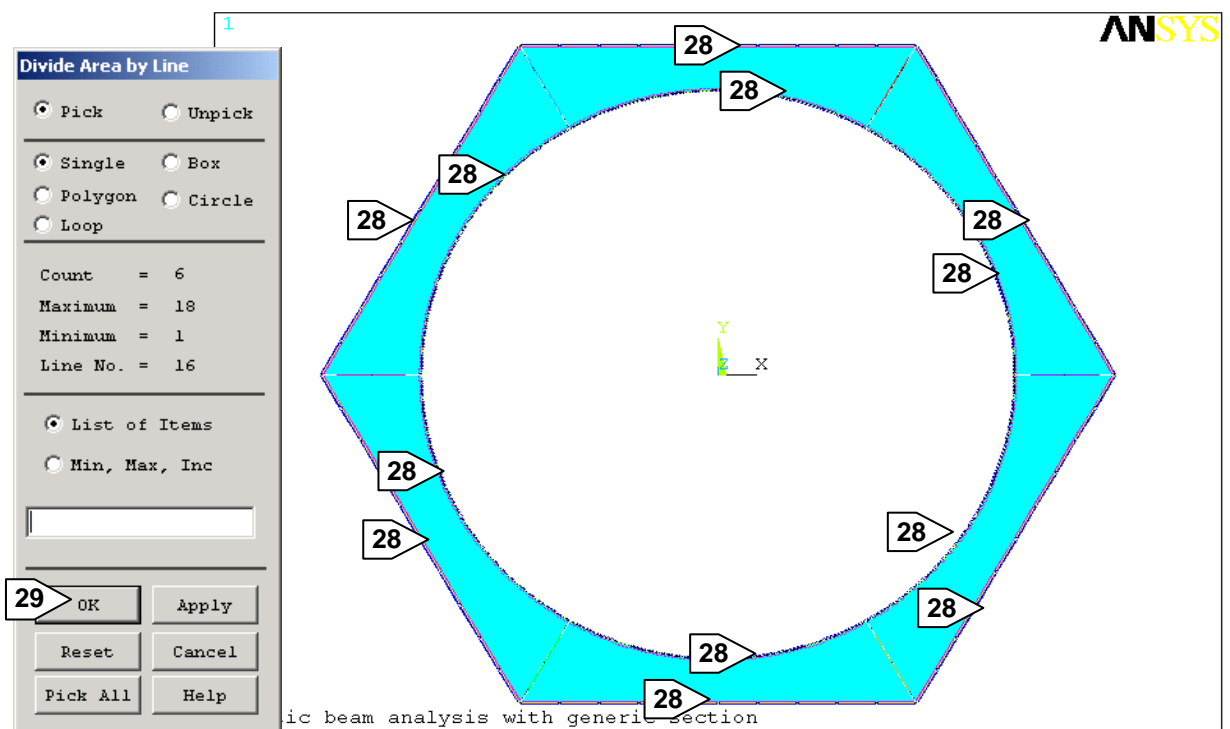


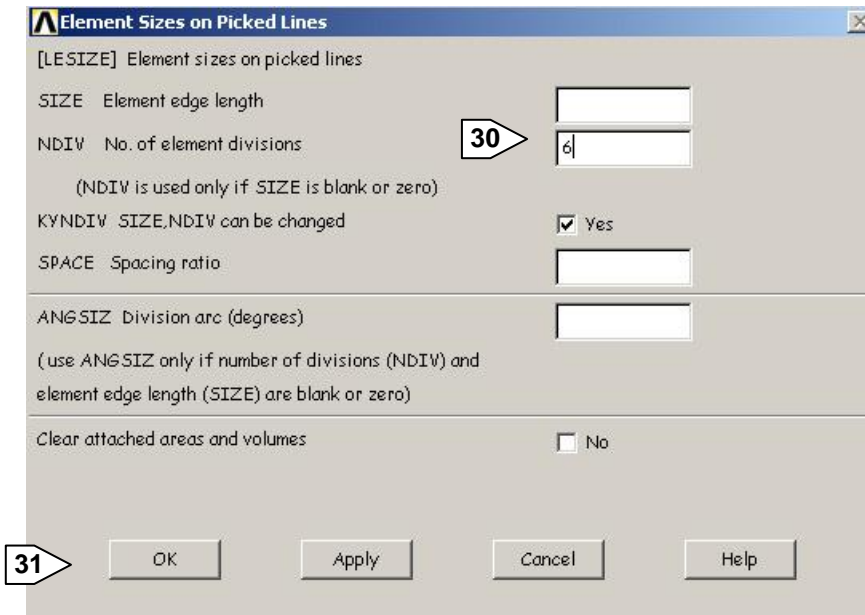
26 Enter 1 as number of element divisions

27 Apply to continue selecting lines



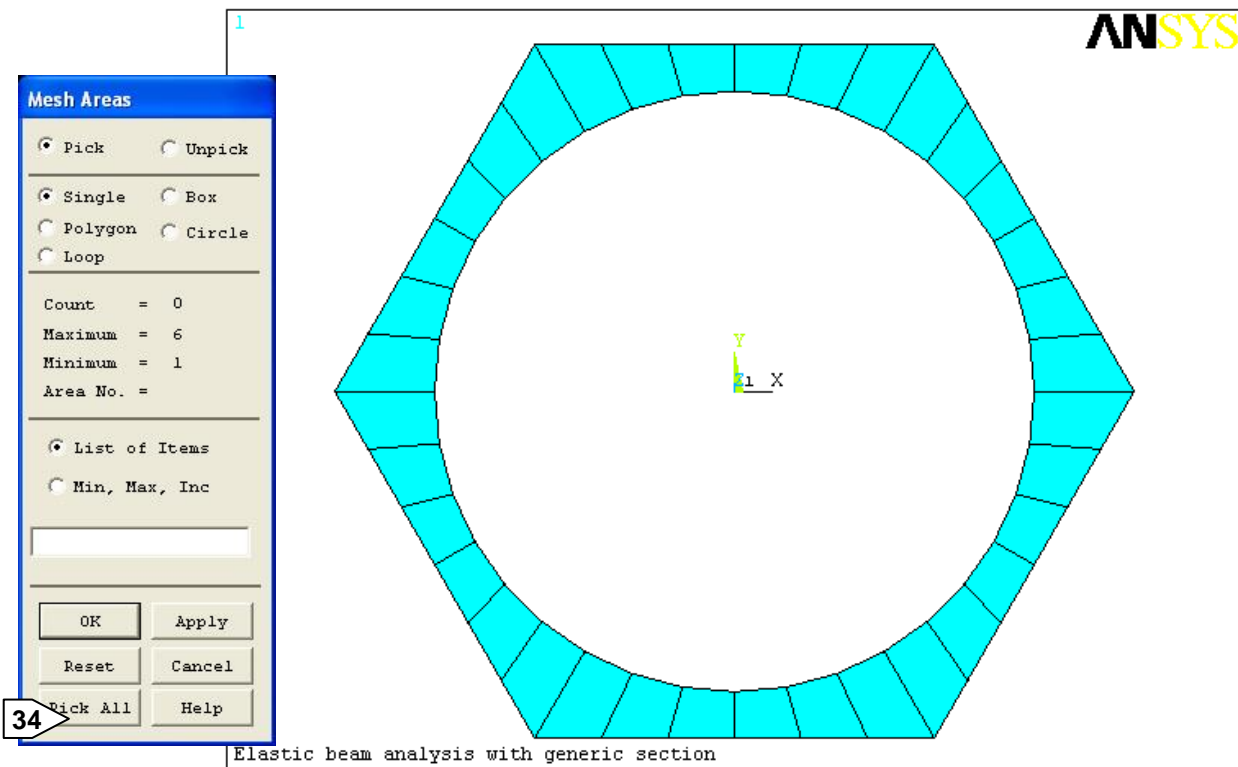
- 28 Pick on the rest of the lines
- 29 OK
- 30 Enter 6 as number of element divisions
- 31 OK





Main Menu: – **Preprocessor** → – Meshing – **Mesh** → – Areas – **Mapped** → **3 or 4 sided**

34 Pick All



Once we have created the section with ANSYS we use the CivilFEM command ~**SEC2DIN** to import the section. We need a new local coordinate system to capture the section. It must contain the section on the YZ plane.

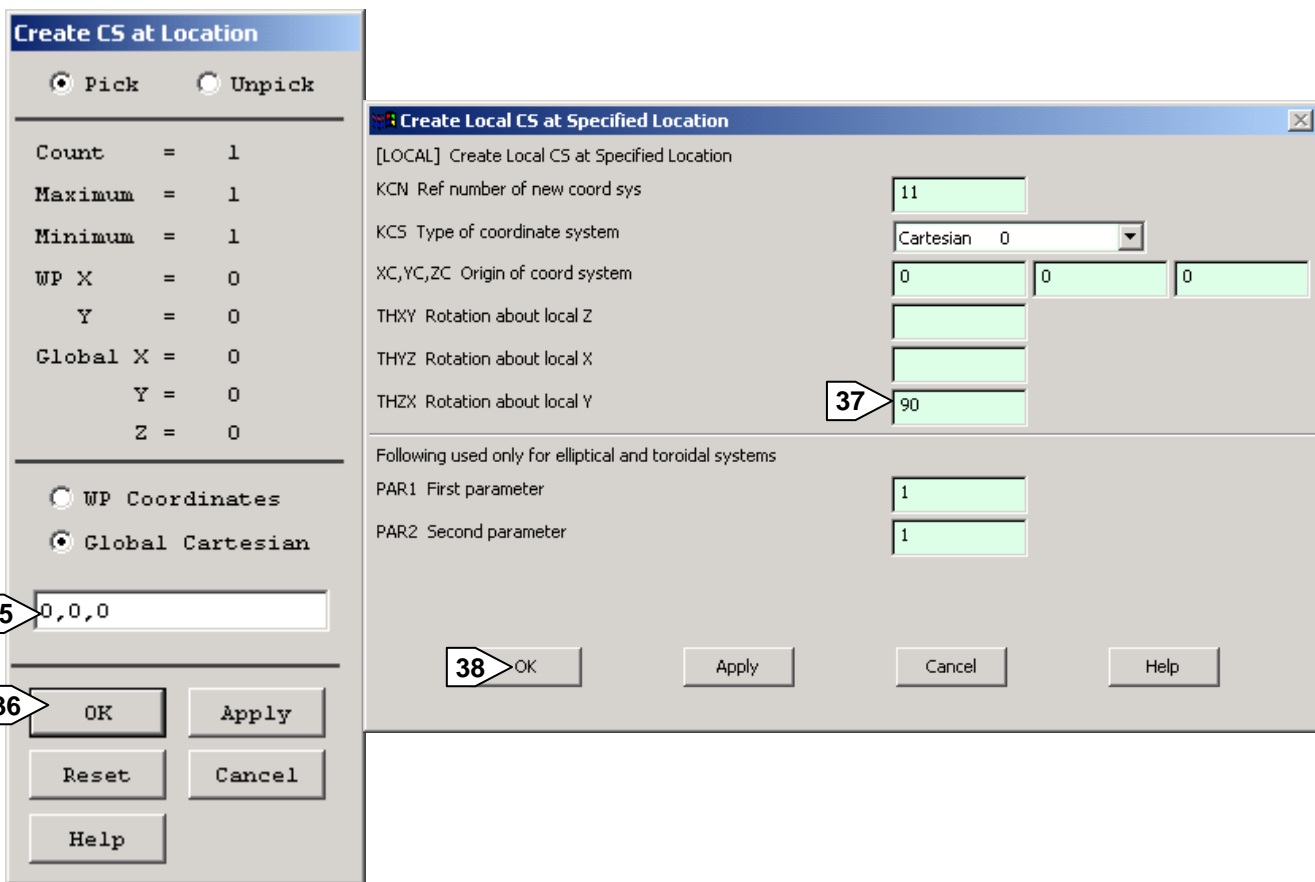
Utility Menu: **WorkPlane** → **Local Coordinate Systems** → **Create Local CS** → **At specified Loc**

35 Enter 0,0,0 as origin of coordinate system

36 OK

37 Enter 90 for angle of rotation about Y

38 OK

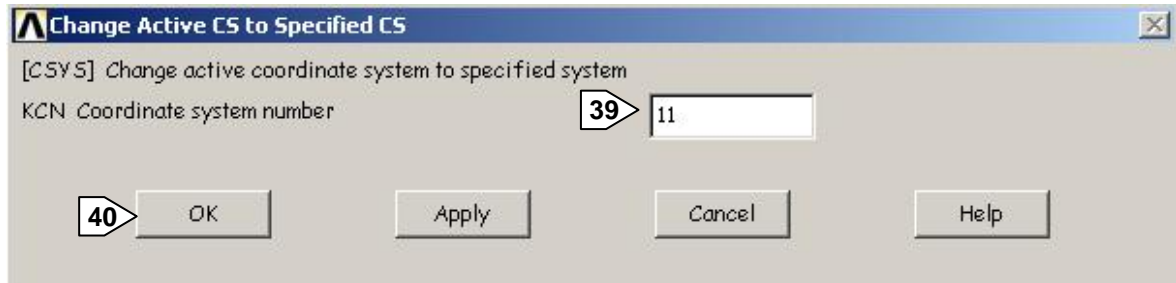


To activate this coordinate system:

Utility Menu: **WorkPlane** → **Change Active Cs to** → **Specified coord sys...**

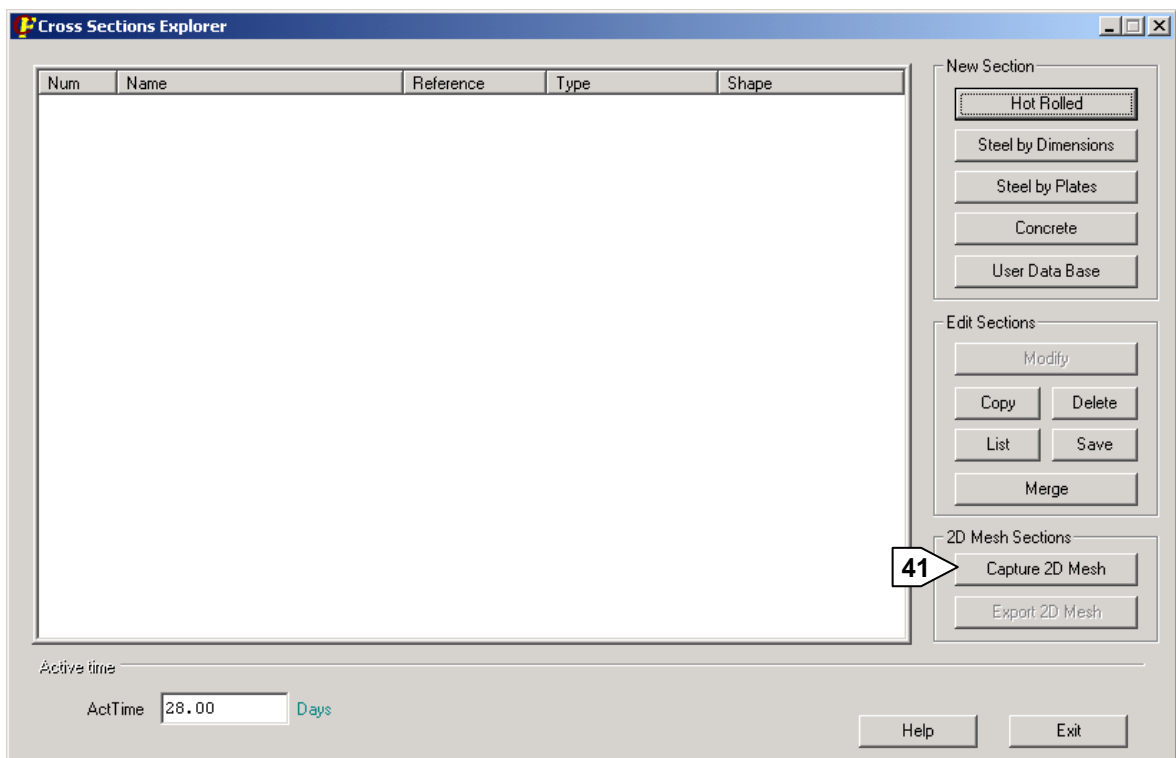
39 Enter Coordinate system 11

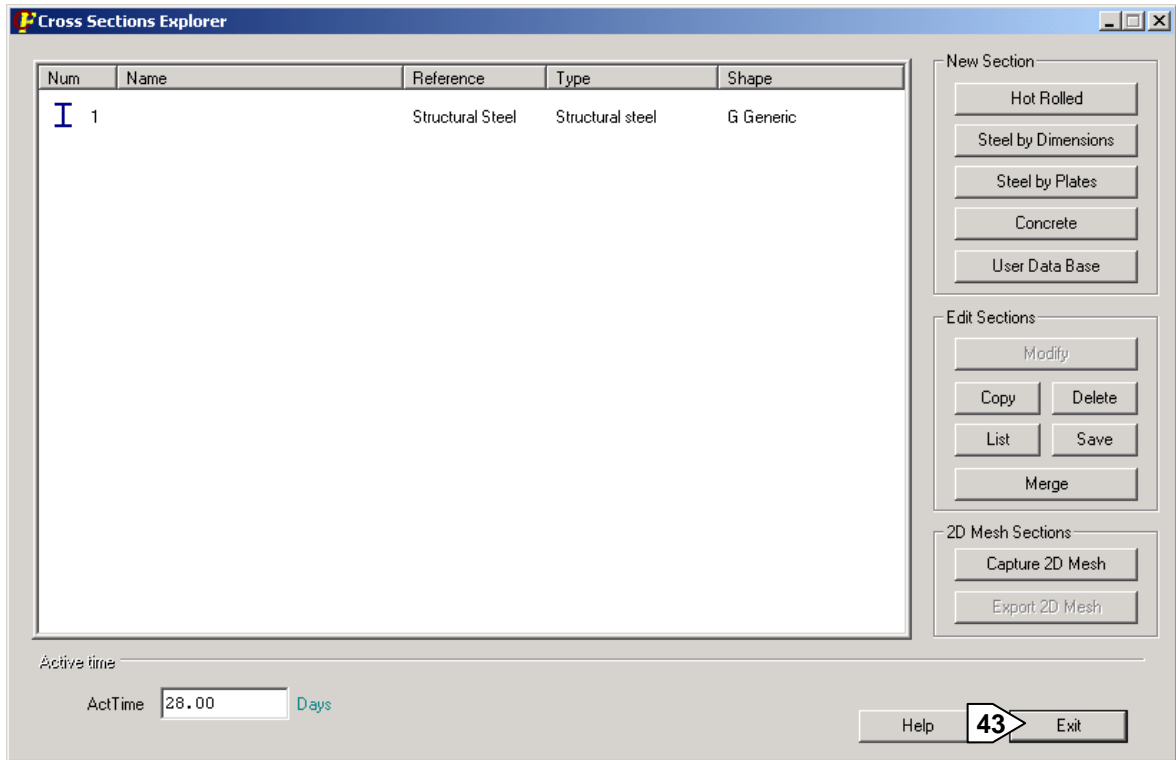
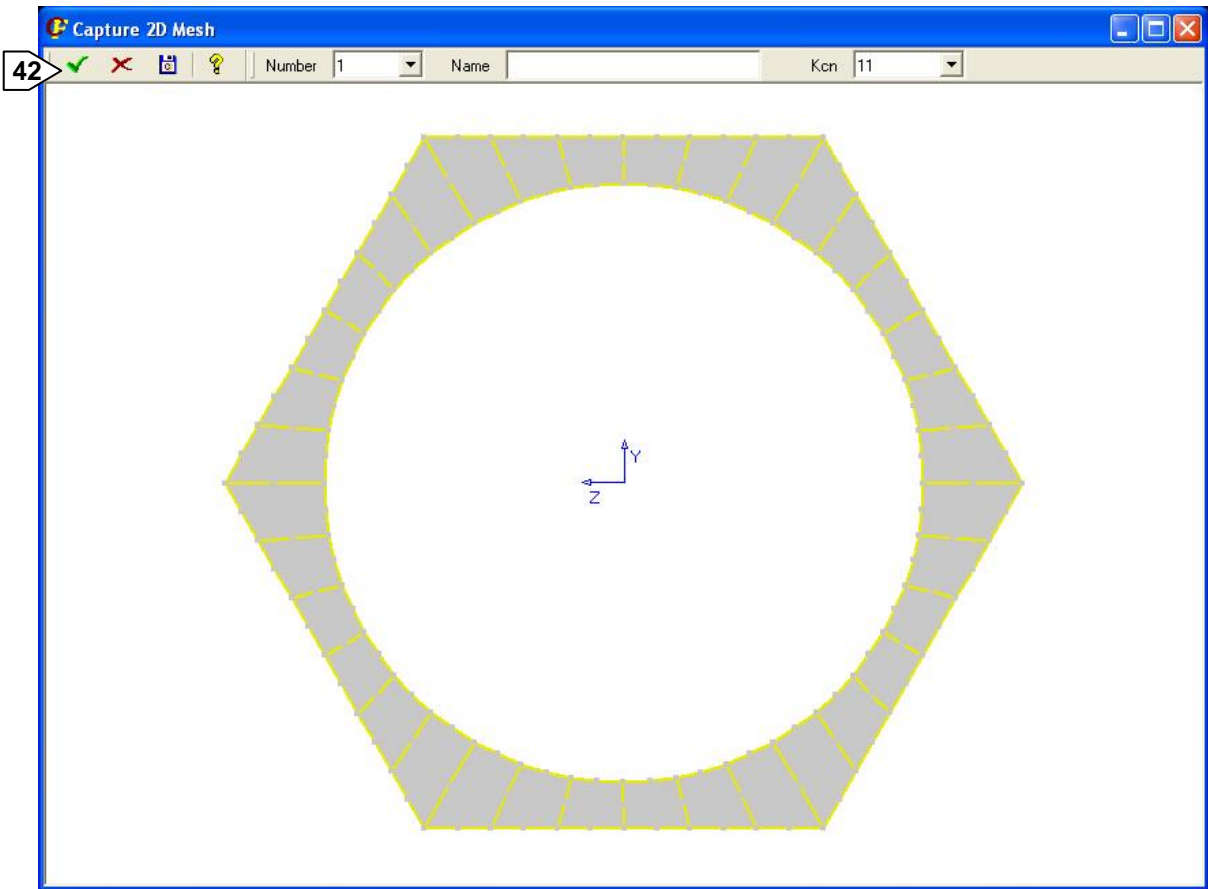
40 OK



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

- 41 Click the Capture 2D Mesh button
- 42 OK to accept tessellation and define cross section
- 43 Exit to close cross section explorer

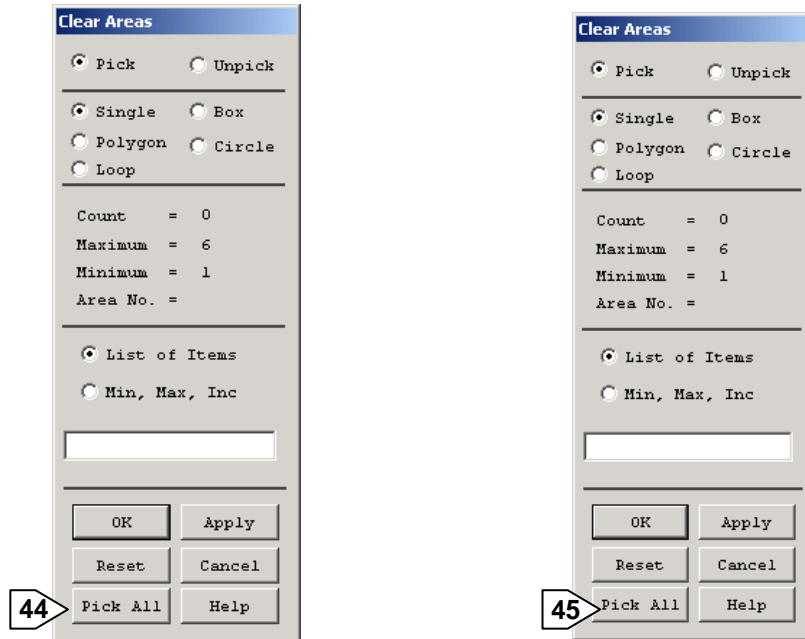




To finish the process we delete all the geometric entities and elements that we have created to build the beam element model. First we clear the meshed areas.

Main Menu: – **Preprocessor** → – Meshing – **Clear** → **Areas**

44 Pick All



Now delete the areas, lines and keypoints

Main Menu: – **Preprocessor** → – Modeling – **Delete** → **Areas and below**

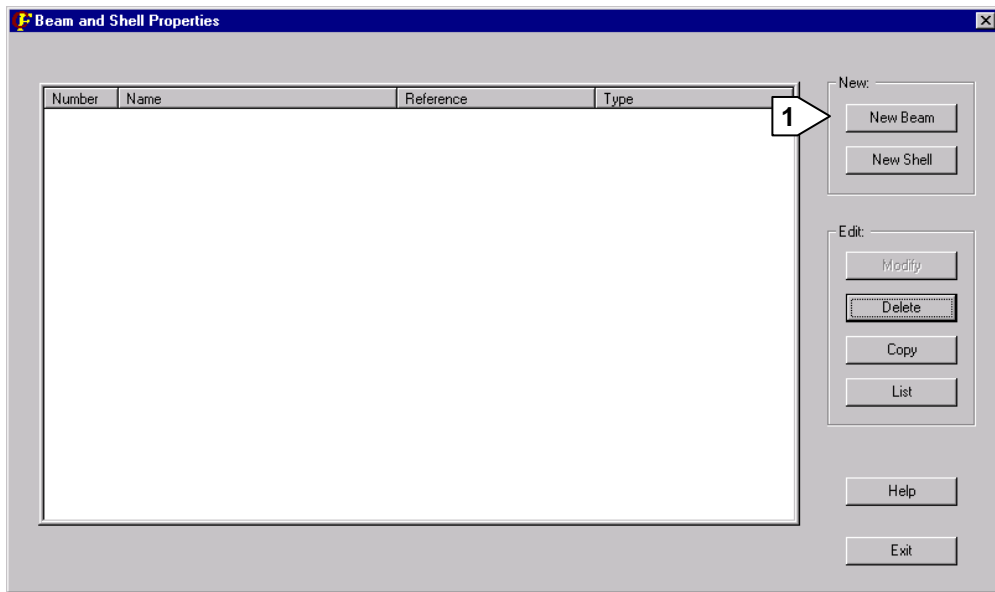
45 Pick All

6. Define Beam & Shell properties

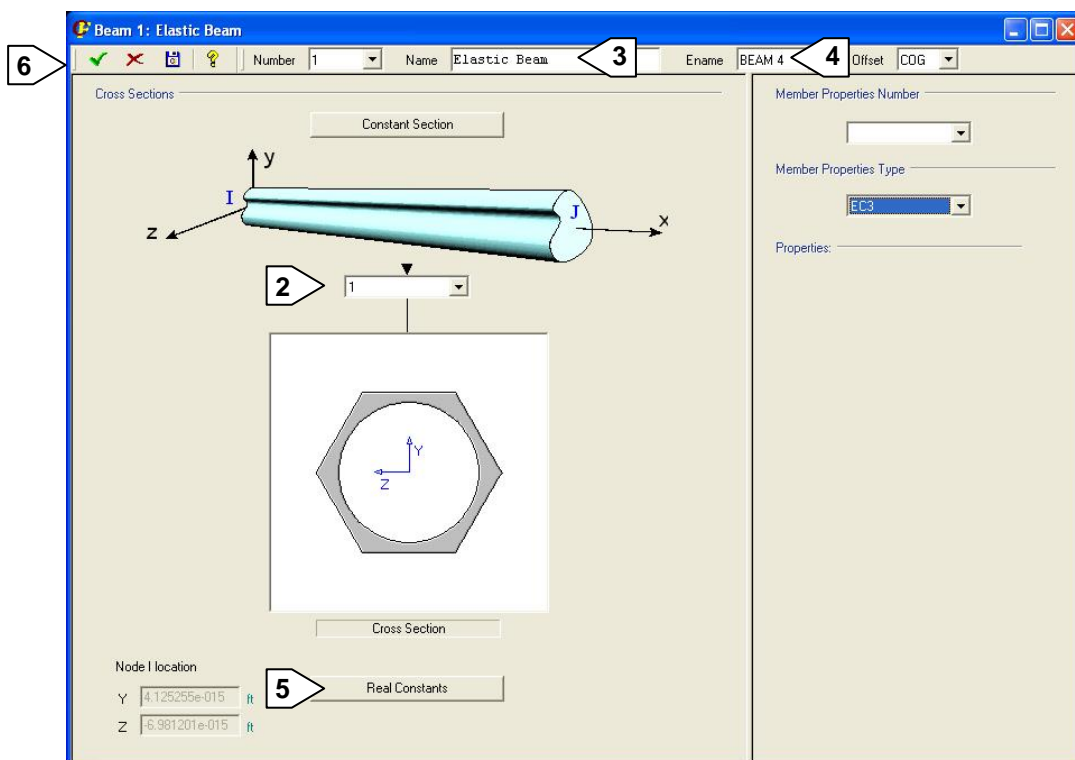
The CivilFEM command **~BMSHPRO** will be used to define ANSYS real constants.

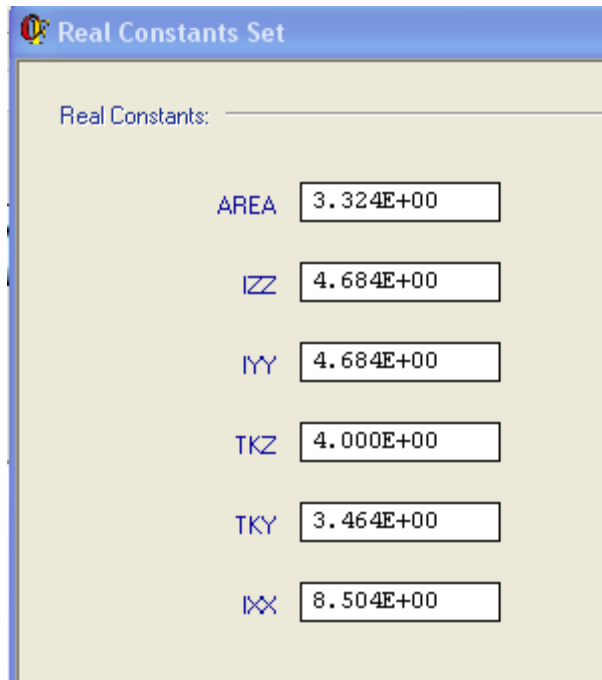
Main Menu: – CivilFEM – **Civil Preprocessor** → **Beam & Shell pro**

1 Click the New Beam button



- 2 Select cross section number 1
- 3 Enter Elastic Beam as Name
- 4 Select element type BEAM 4

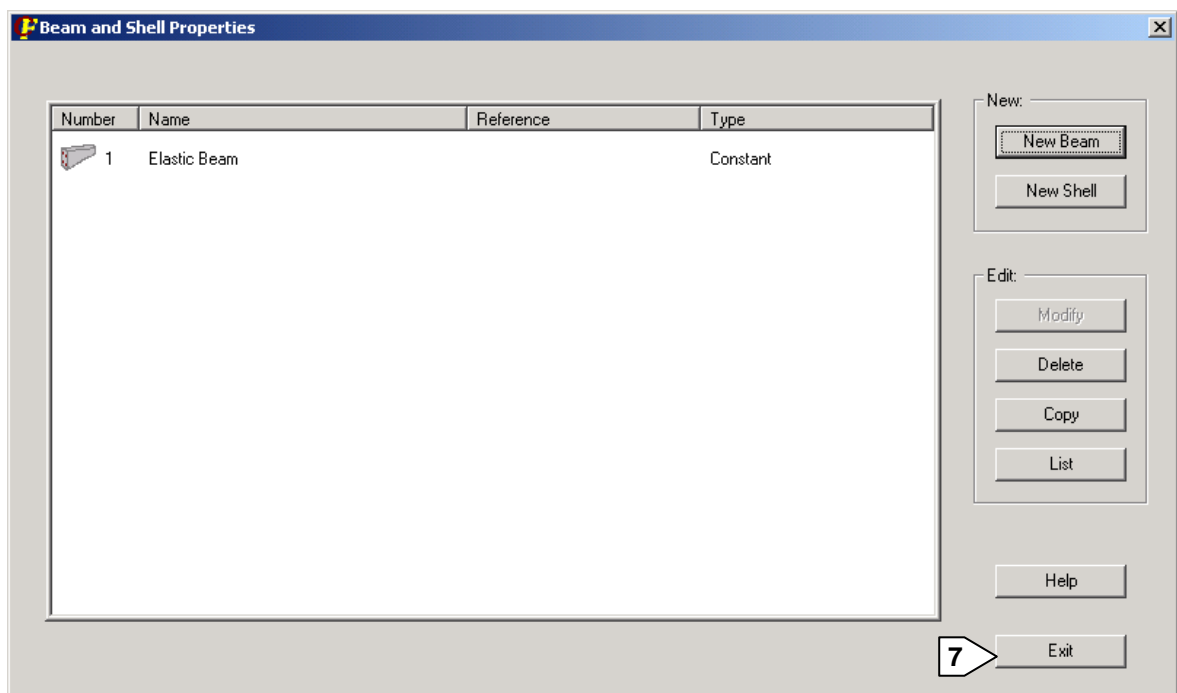




5 You can review ANSYS real constants modified by CivilFEM by clicking the Real Constants button

6 OK to define Beam & Shell properties

7 EXIT to close window



7. Define solid modeling entities

To create the model geometry we will use the ANSYS solid modeling tools.

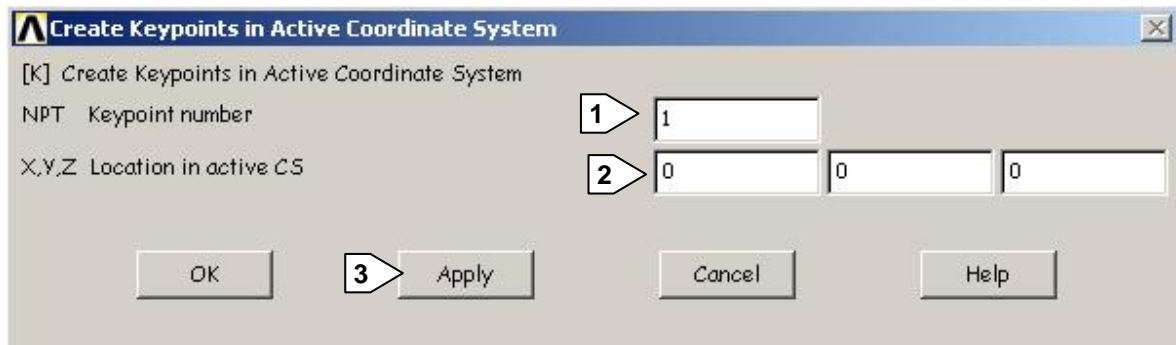
First, change the coordinate system

Utility Menu: **WorkPlane** → **Change Active Cs to** → **Global Cartesian**

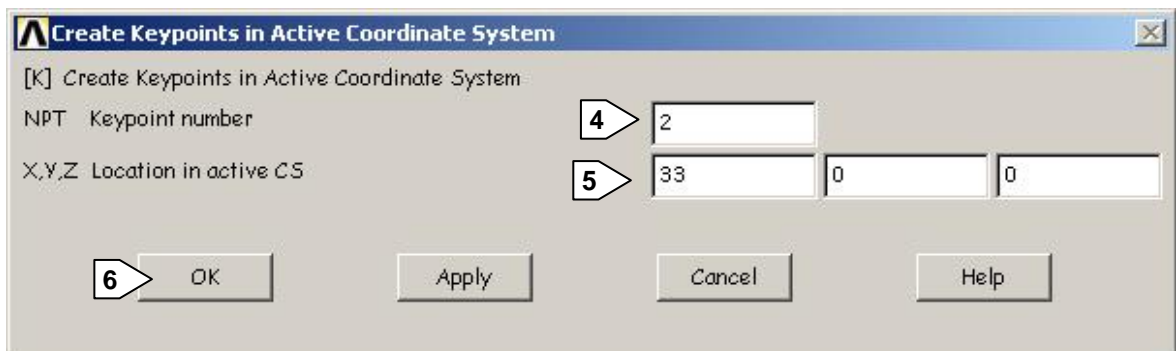
Now we create keypoints.

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

- 1 Enter 1 for first keypoint
- 2 Enter $x=0$, $y=0$, $z=0$ for coordinates of keypoint 1
- 3 Apply to create the first keypoint



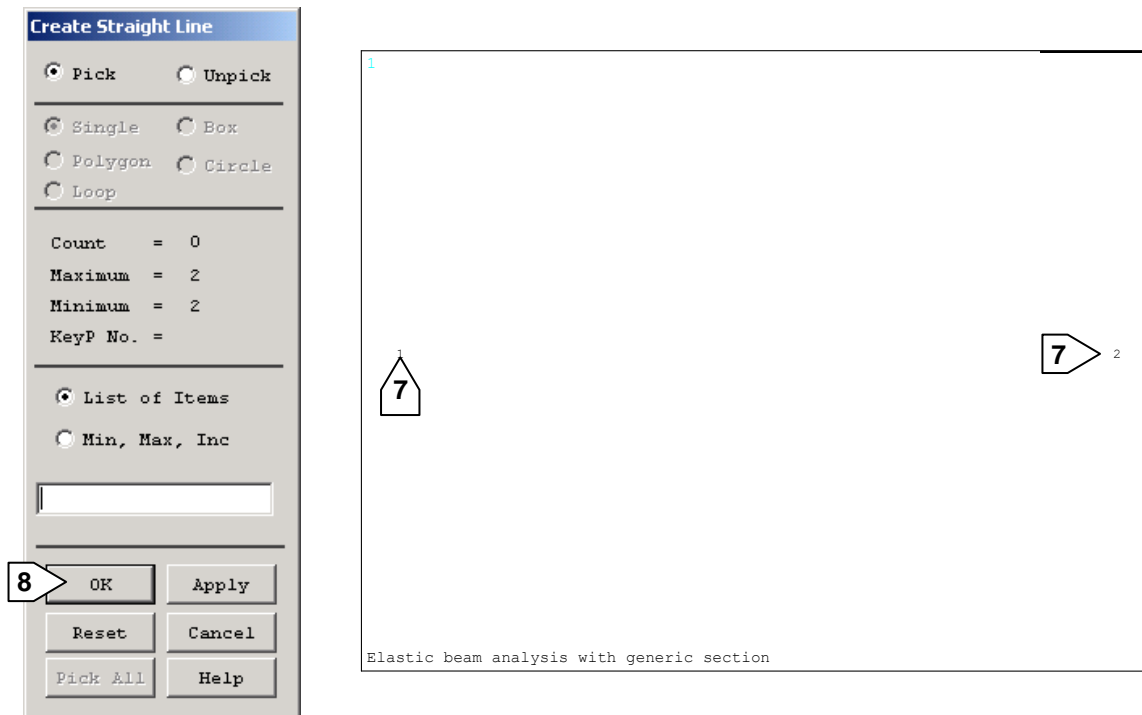
- 4 Enter 2 for second keypoint
- 5 Enter $x=33$, $y=0$, $z=0$ for coordinates of keypoint
- 6 Ok



Use these two keypoints to generate a line.

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

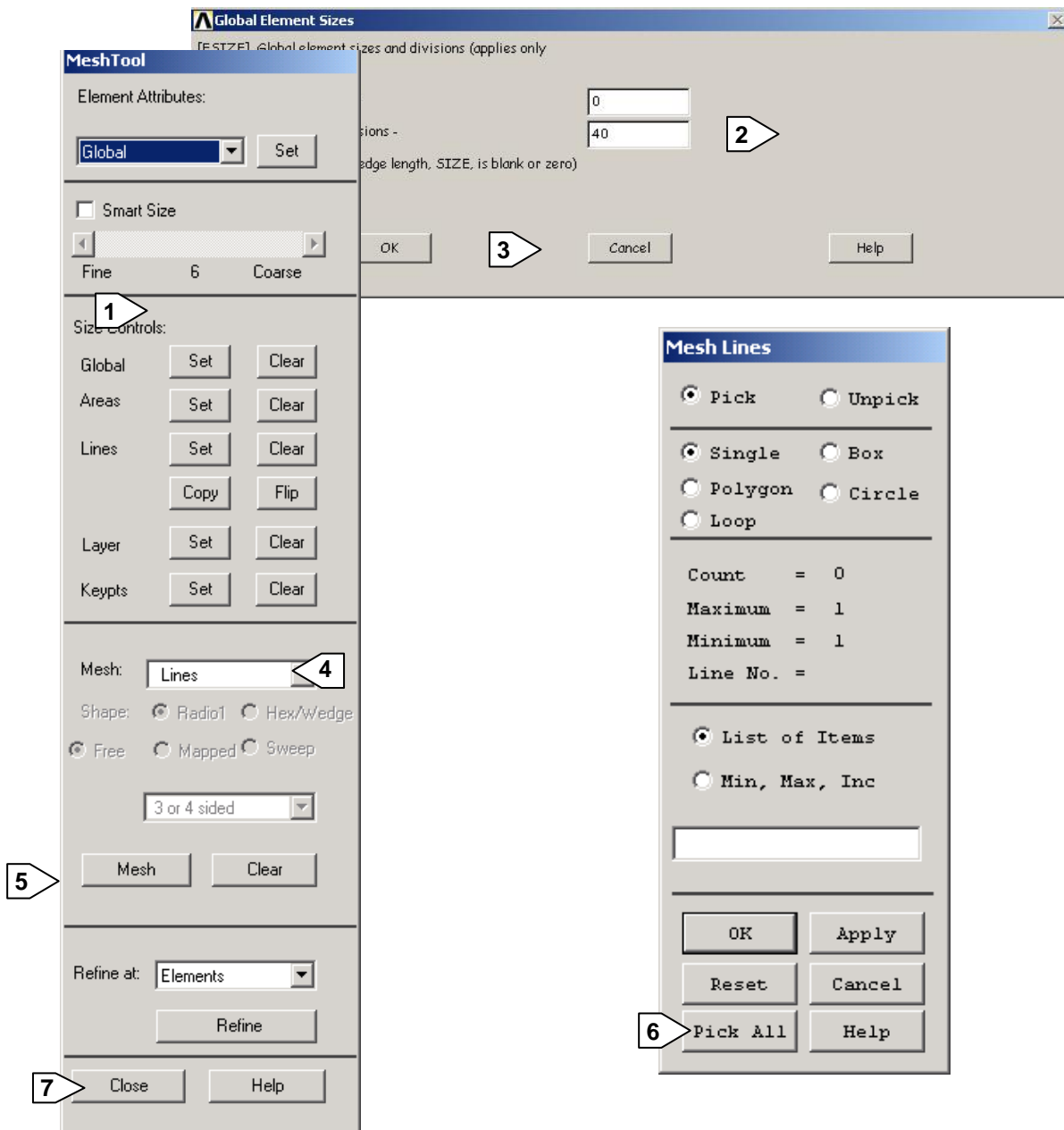
- 7 Pick keypoint 1 and keypoint 2
- 8 OK



8. Mesh

Main Menu: **Preprocessor** → **Meshing** → **Mesh Tool**

- 1 > Set global size control
- 2 > Set number of element divisions to 40
- 3 > OK
- 4 > Choose line meshing
- 5 > Click on mesh
- 6 > Pick All
- 7 > Close the mesh tool



9. Save the database

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.

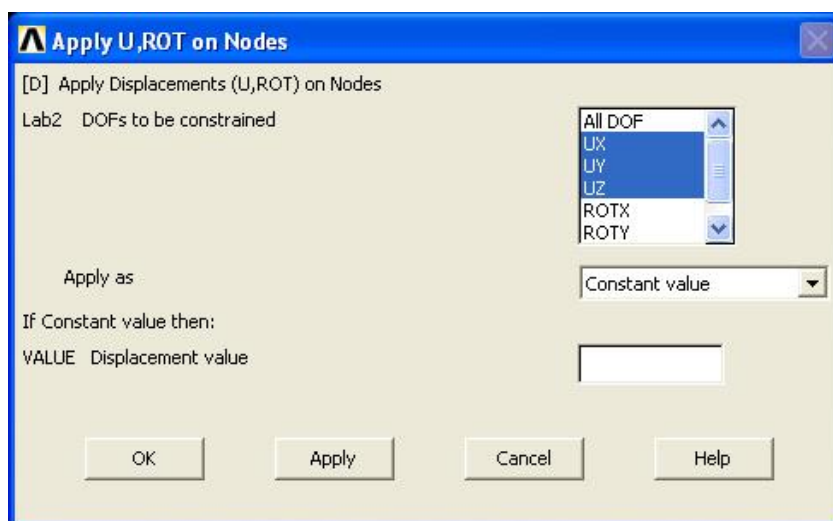
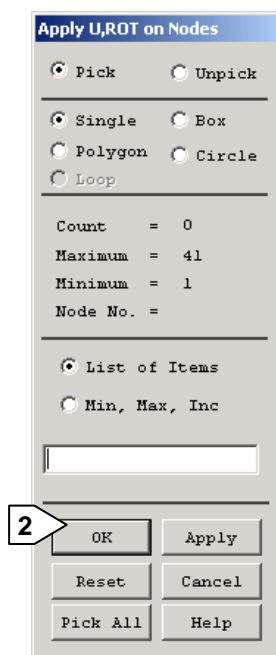
Toolbar: **CFSAVE**

■ Solution

10. Apply displacement constraints

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

- 1 ▷ Pick node 1 at beam left end
- 2 ▷ OK to finish picking nodes
- 3 ▷ Choose UX, UY, UZ and ROTX
- 4 ▷ Apply



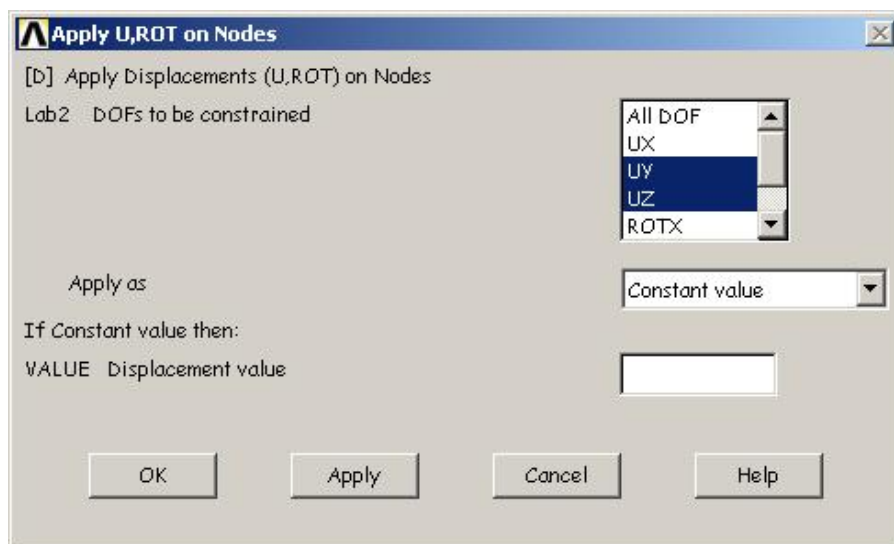
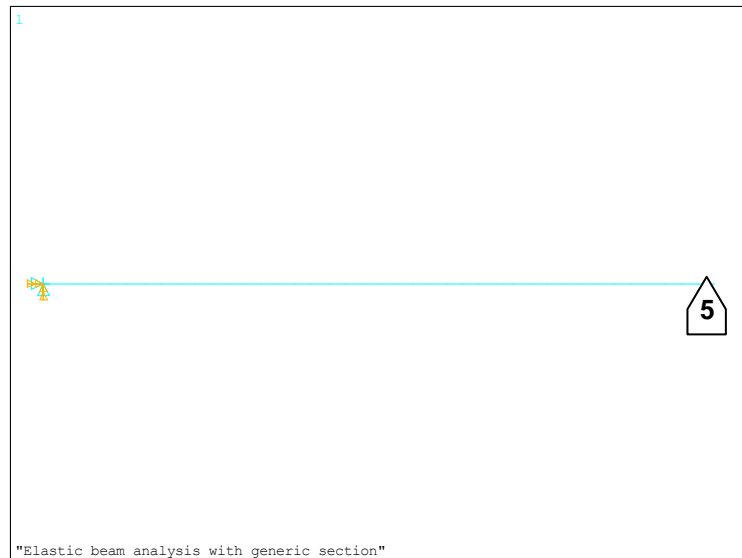
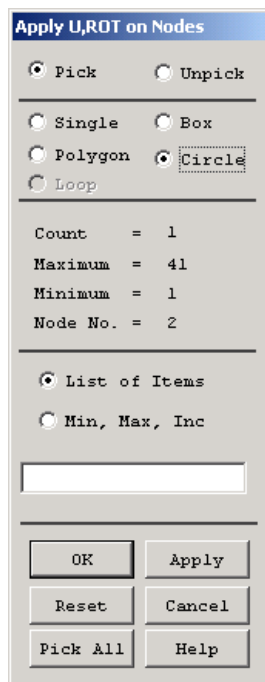
5 Pick node 2 at beam right end

6 OK to finish picking nodes

7 Choose UY and UZ

8 OK

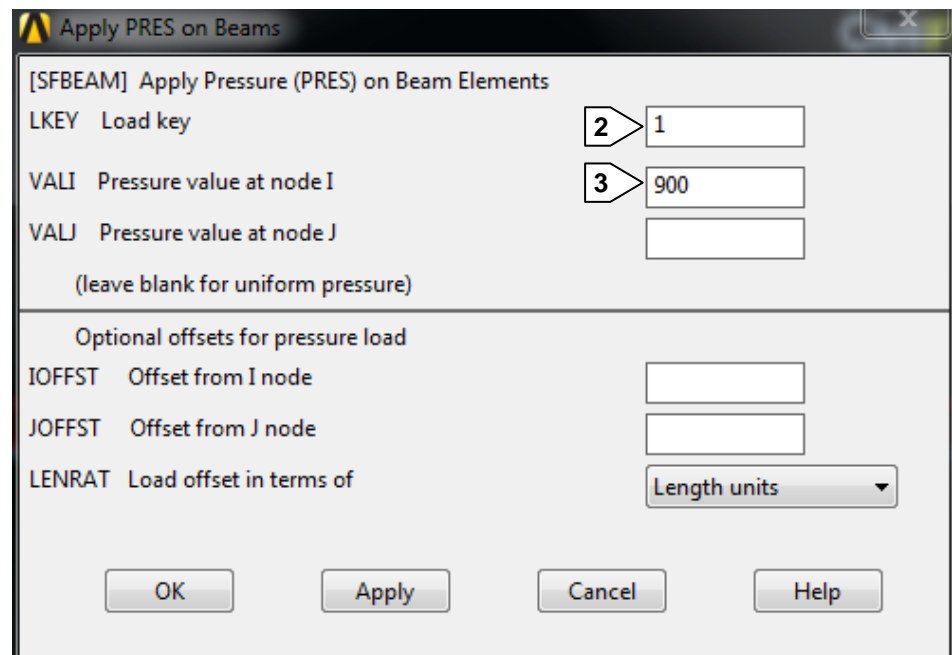
12.



11. Apply pressure load

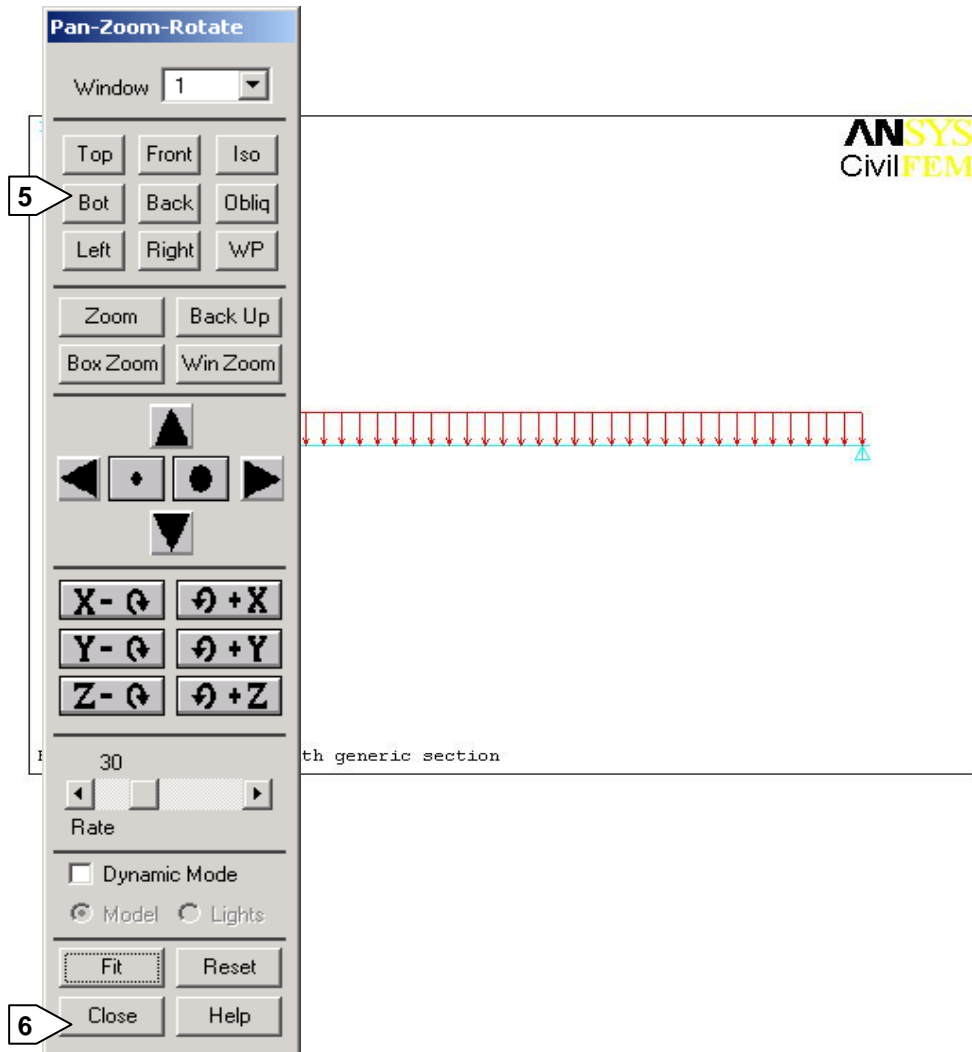
Main Menu: **Solution** → – Define Loads – **Apply**→ – Structural – **Pressure**
→**On Beams**

- 1 Pick All to select all elements
- 2 Enter 1 for Load key
- 3 Enter 900 for VAL I
- 4 OK to apply pressure and close dialog box



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

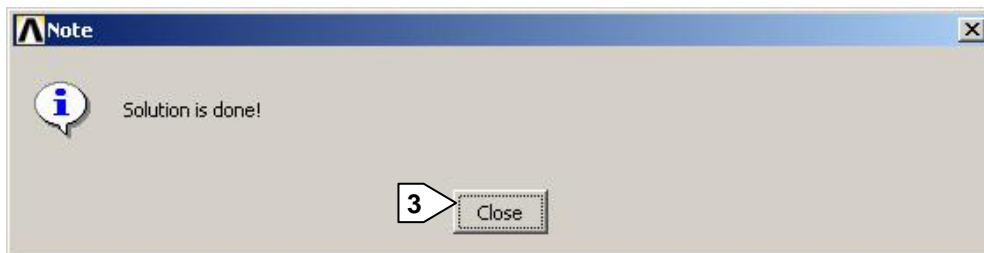
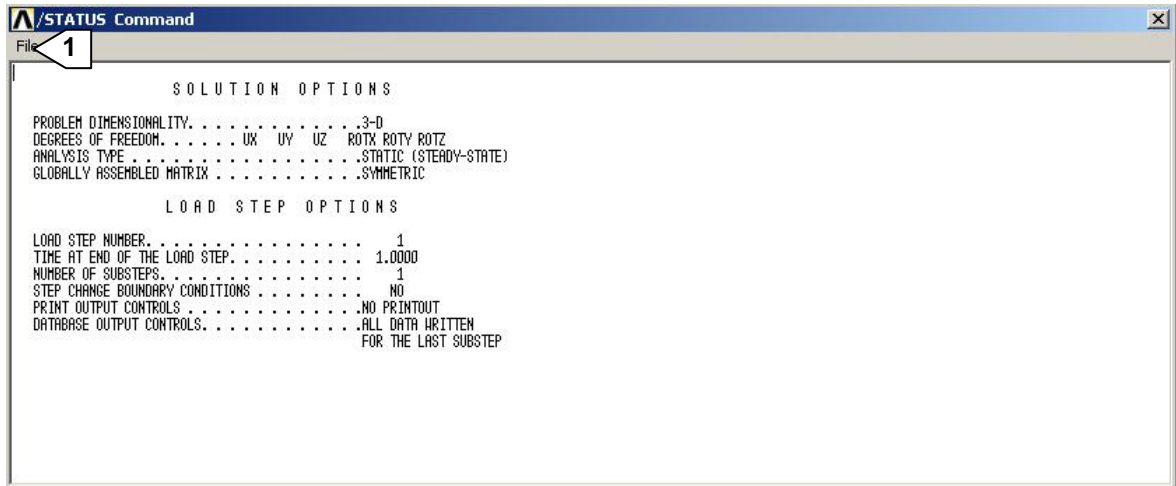
- 5 Choose Bot
- 6 Close



12. Solve

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

- 1 Review information in the status window, and then pick File →Close to close the window
- 2 OK to begin the solution
- 3 Close the information window when solution is done



■ Postprocessing

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

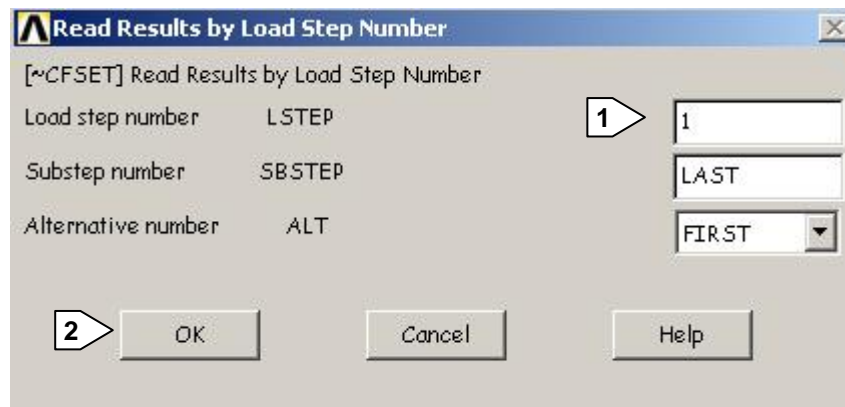
13. Enter the postprocessor and read results

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

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

1 Enter 1 in the Load Step number box

2 OK to read load step 1

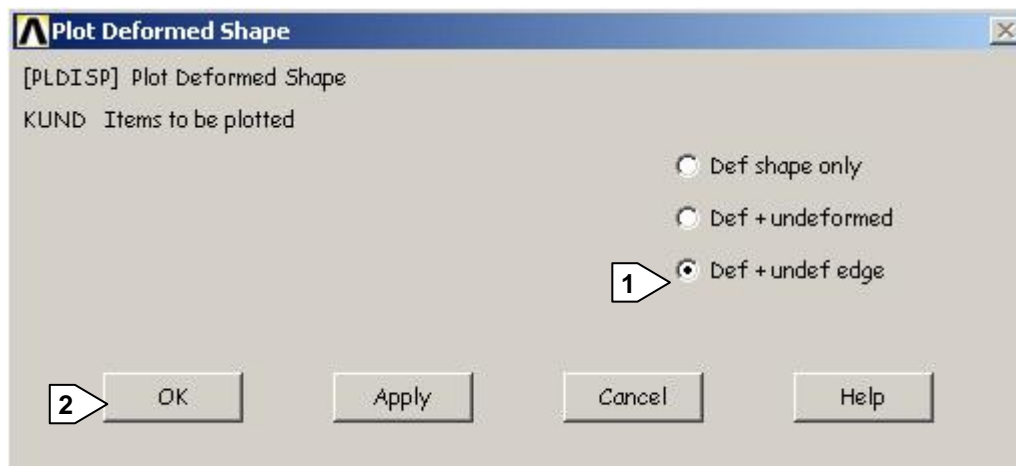


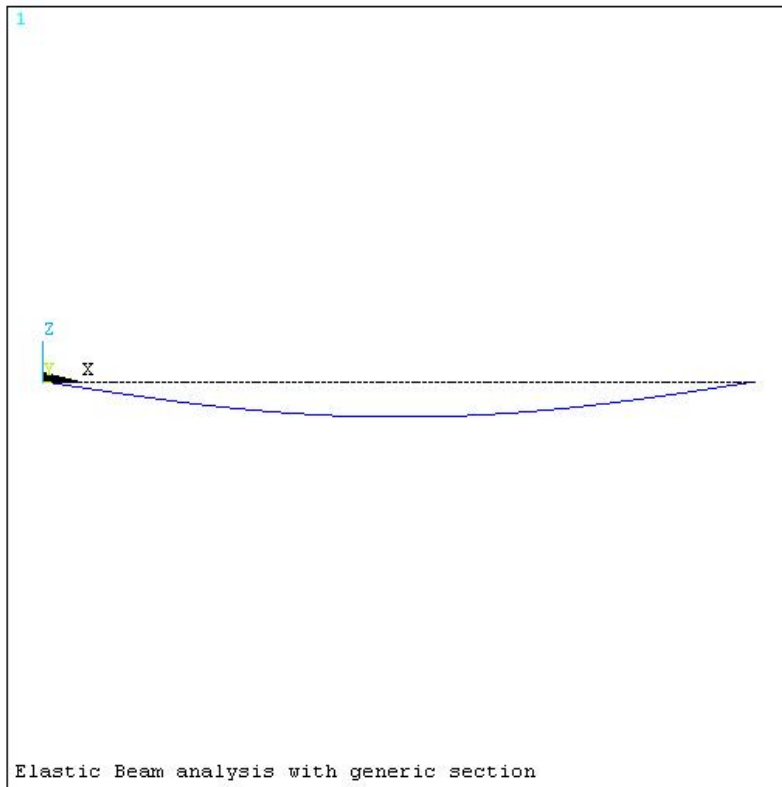
14. Plot the deformed shape

Main Menu: **General Postproc** → **Plot Results** → **Deformed Shape**

1 Choose Def + undef edge

2 OK





```
DISPLACEMENT
STEP=1
SUB =1
TIME=1
PowerGraphics
EFACET=1
AVRES=Mat
DMX =.676E-03

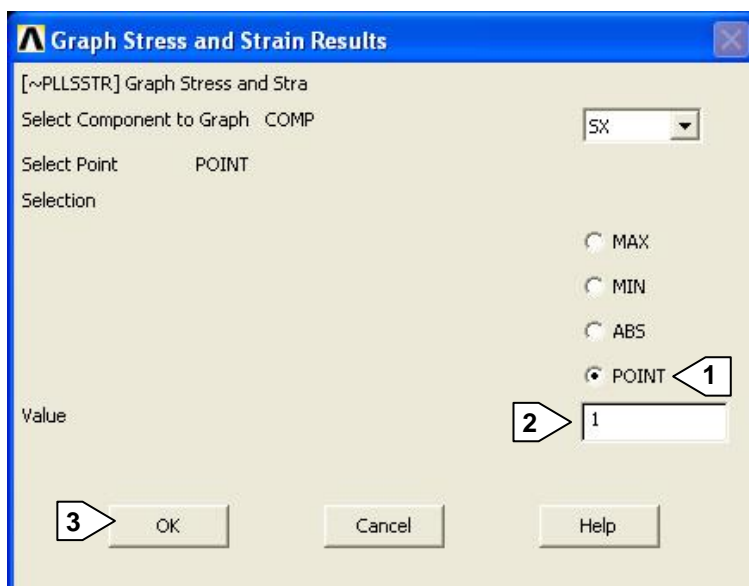
DSCA=2439
YV  =-1
DIST=18.15
XF  =16.5
ZF  =-.825
Z-BUFFER
```

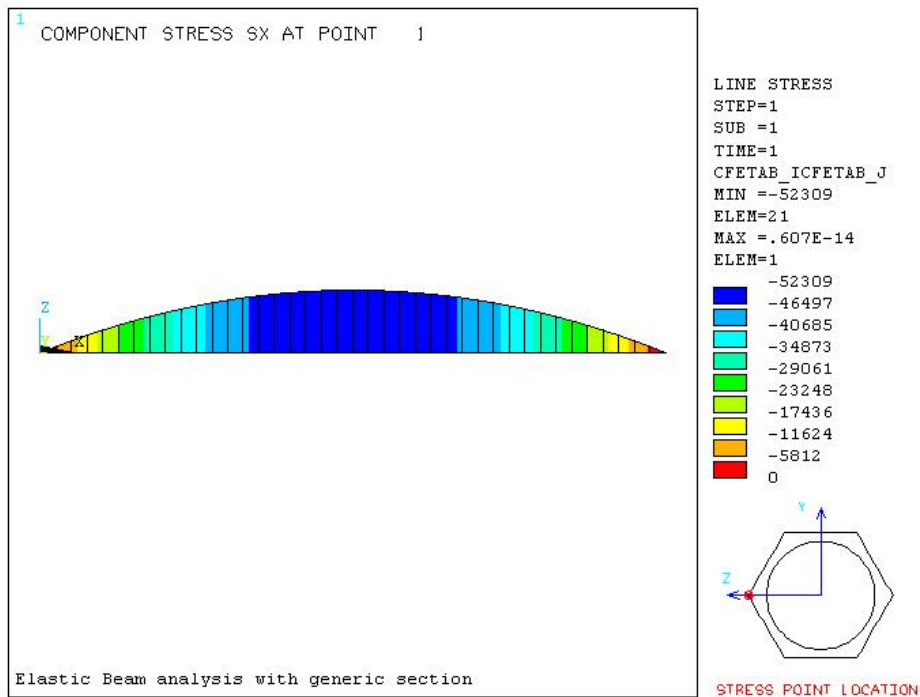
15. Plot bending stress in Z top fiber

The CivilFEM **~PLLSSTR** command draws the stress distribution in the beam. We are interested in the bending stress in the section Z top fiber.

Main Menu: – CivilFEM – **Civil Postprocess** → **Beam Utilities** → **GRAPH RESULTS: Stress & Strain**

- 1 Select Point to plot stresses on a section point
- 2 Enter point number
- 3 OK to plot stress results

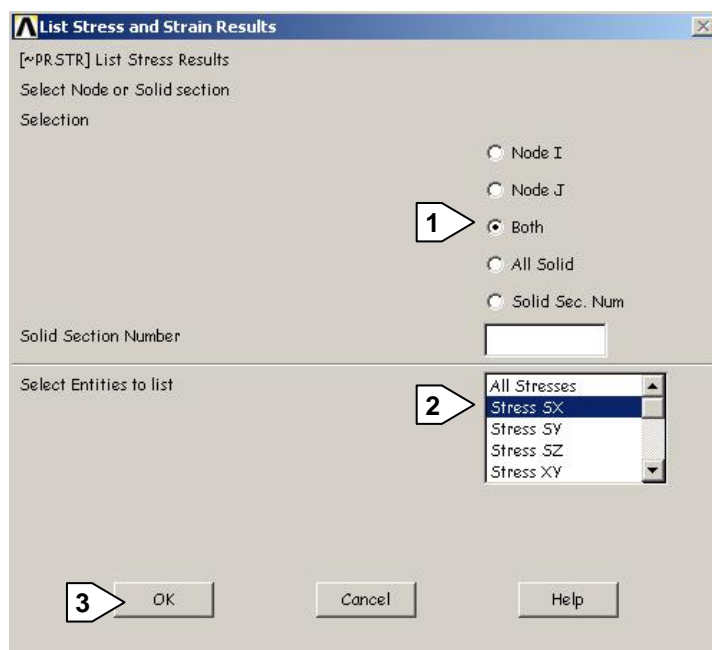




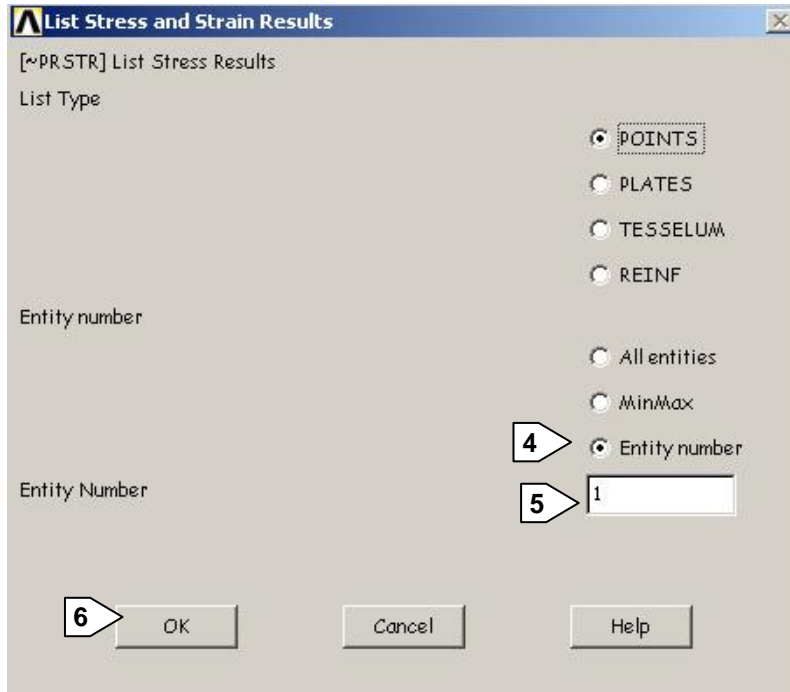
16. List bending stress in Z top fiber

Main Menu: – CivilFEM – **Civil Postprocess** → **Beam Utilities** → **LIST RESULTS: Stress & Strain**

- 1 Choose Both in order to list the stress at node I and J
- 2 Pick on Stress SX to list section stresses
- 3 OK



- 4 Select on Entity Number
- 5 Enter Point 1
- 6 OK



STRESSES AND STRAINS LIST

BEAM ELEMENT 1, END I, SECTION 1 - STRESSES AND STRAINS

POINT	SX
1	6.068E-15

BEAM ELEMENT 1, END J, SECTION 1 - STRESSES AND STRAINS

POINT	SX
1	-5.100E+03

BEAM ELEMENT 2, END I, SECTION 1 - STRESSES AND STRAINS

POINT	SX
-------	----

17. Exit the ANSYS program

ANSYS Toolbar: **Quit**

1 Choose to save everything

2 OK

