

STATIC ANALYSIS OF A CANTILEVER BEAM

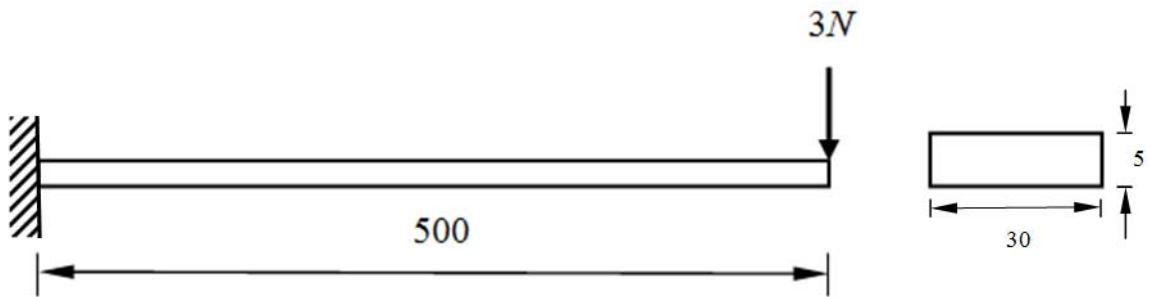


Figure 1

All dimensions are in *mm*

Objective: To find the deflection, stress, strain, shear force and bending moment diagram of cantilever beam shown in **Figure 1**.

Analysis Type : Static

Modulus of elasticity, $E = 200 \text{ GPa}$

Poisson's ratio, $\nu = 0.3$

Point load, $P = -3 \text{ N}$

PROCEDURE

STEP

1. Create two points at $(0, 0, 0)$ and $(500, 0, 0)$

Commands : POINT, ADD

Menu : Geometry \rightarrow Key point \rightarrow Create \rightarrow By X/Y/Z

Parameters : (To be filled by the user)

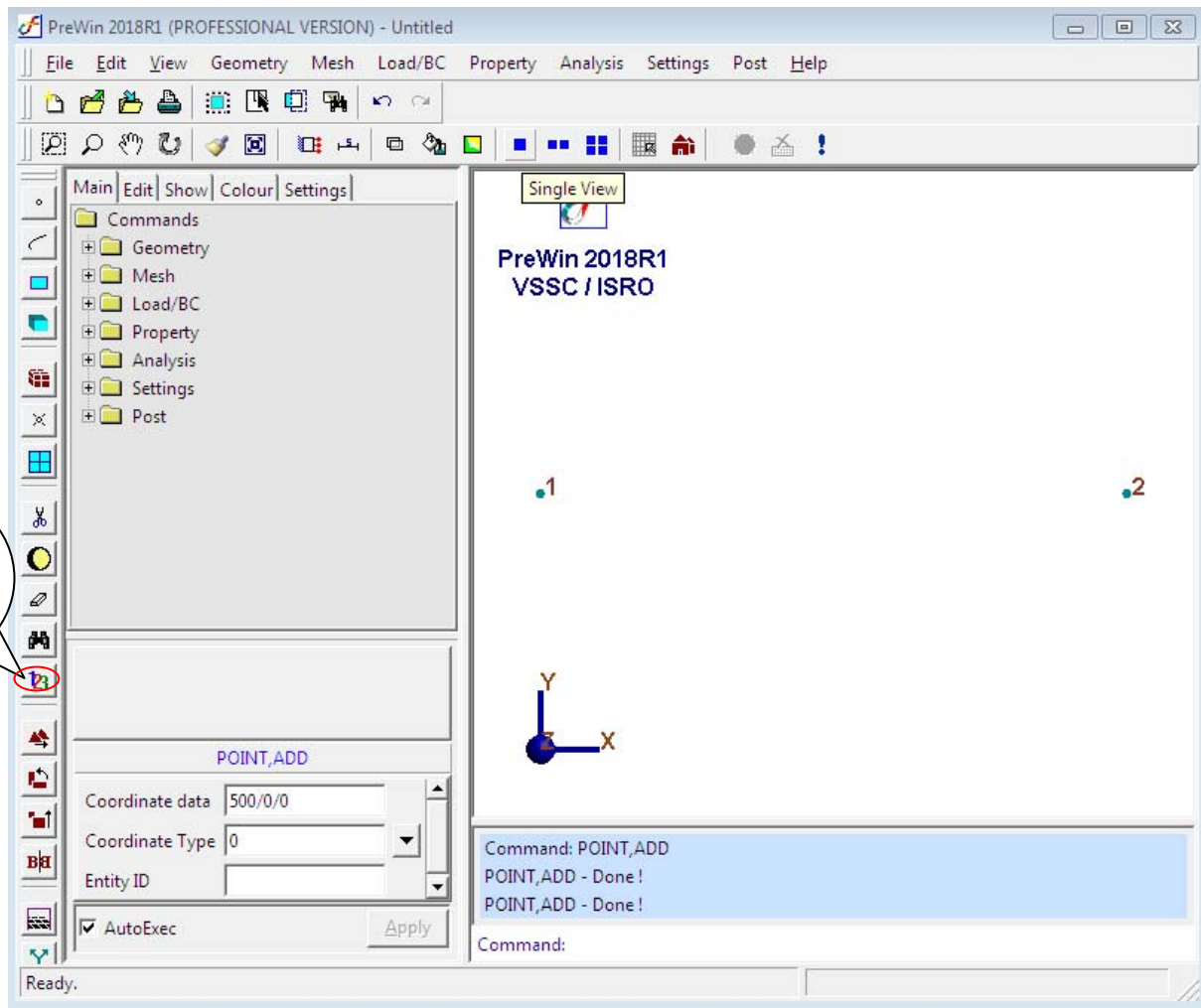
Coordinate Data	0/0/0
Coordinate Type	0
Entity ID	

Similarly create second point $(500/0/0)$

Note:

Click "**Apply**" button or press **ctrl+enter** key after completing each step. "Done" message appears on message box for every step executed successfully.

At the end of the above operation/s, your screen should look like this.

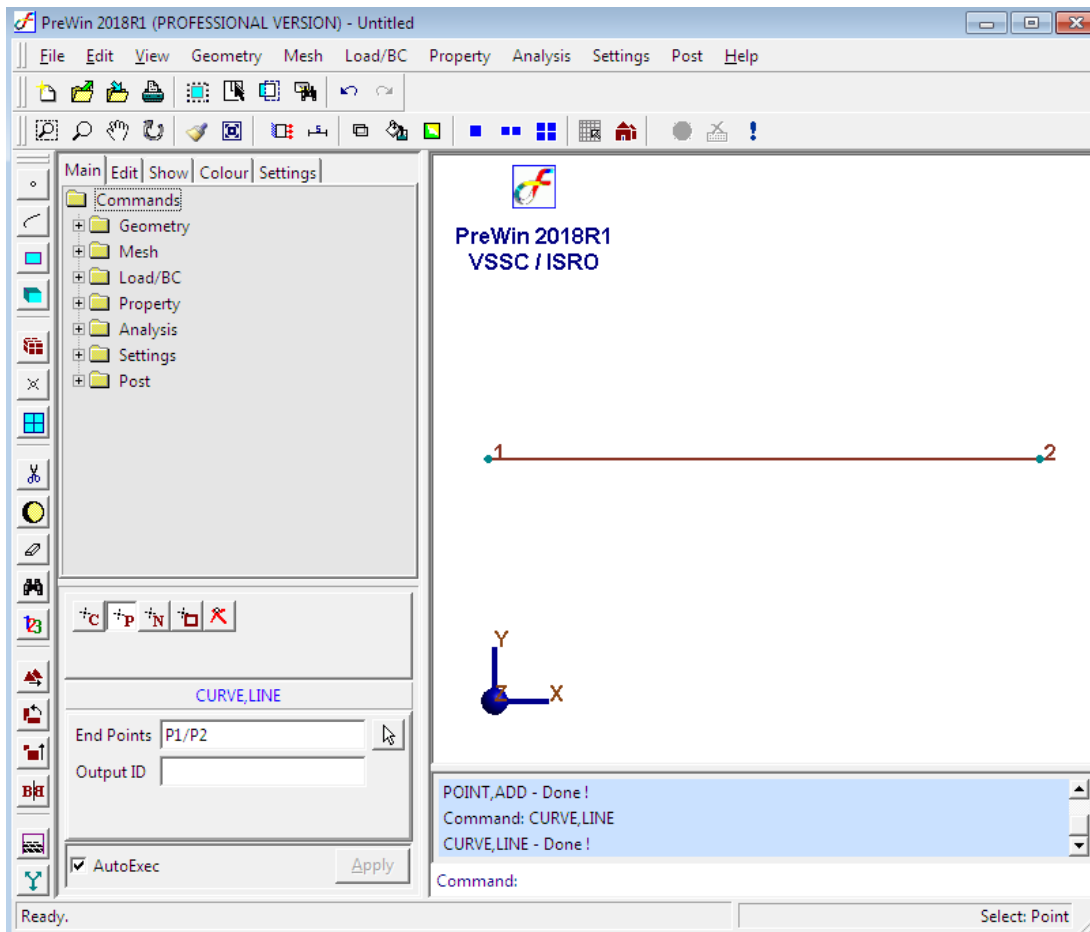


2. Create a line

- Commands : CURVE, LINE
- Menu : Geometry → Curve → Create → Line
- Parameters :

End points	Use Mouse to pick the points
Entity ID	1

At the end of the above operation/s, your screen should look like this.



3. Meshing using beam elements

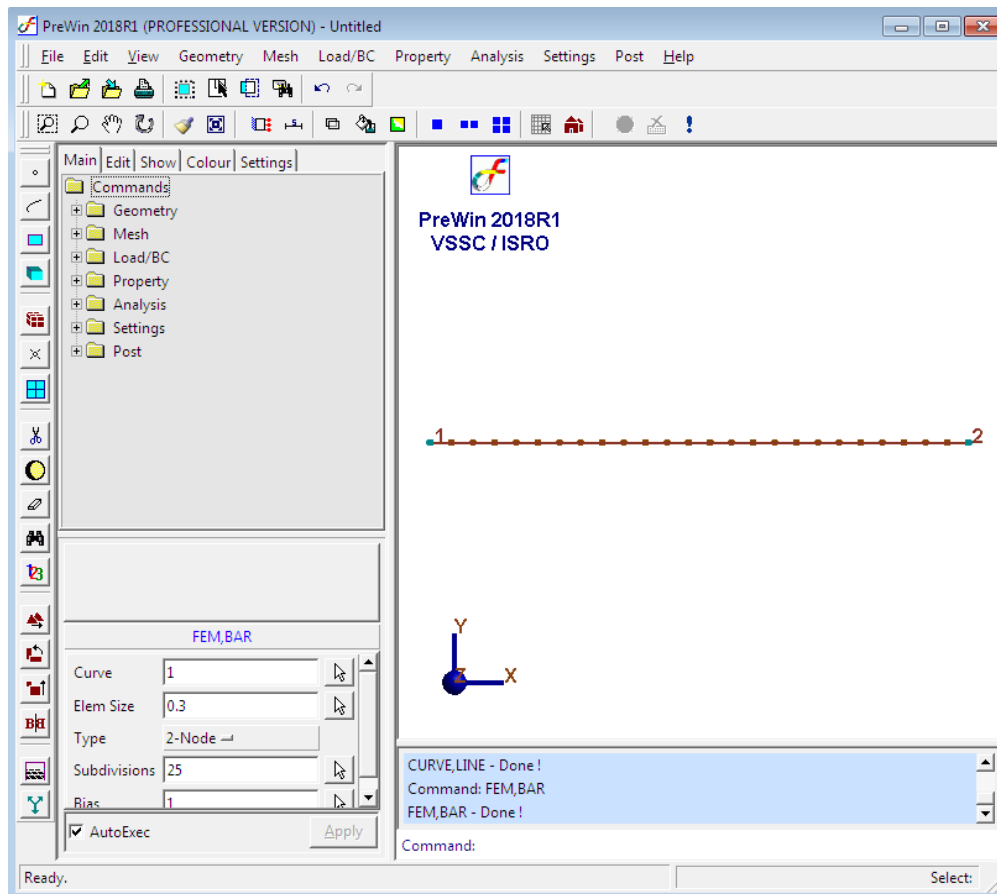
- Commands : FEM, BAR
- Menu : Mesh → FE Mesh → Bar
- Parameters :

Curve	1
Element Size	0.3
Type	2-node
Subdivisions	25
Bias	1.0

Note:

Right click/left click mouse point to alter sub-divisions.

At the end of the above operation/s, your screen should look like this.

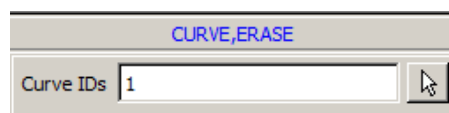


4. Erase curve

Commands : CURVE, ERASE

Menu : Geometry → Curve → Miscellaneous → Erase

Parameters :



Note:

Likewise erase key points by POINT,ERASE,ALL command.

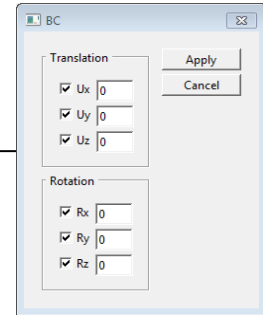
5. Specify displacement boundary conditions

Commands : BC, ADD

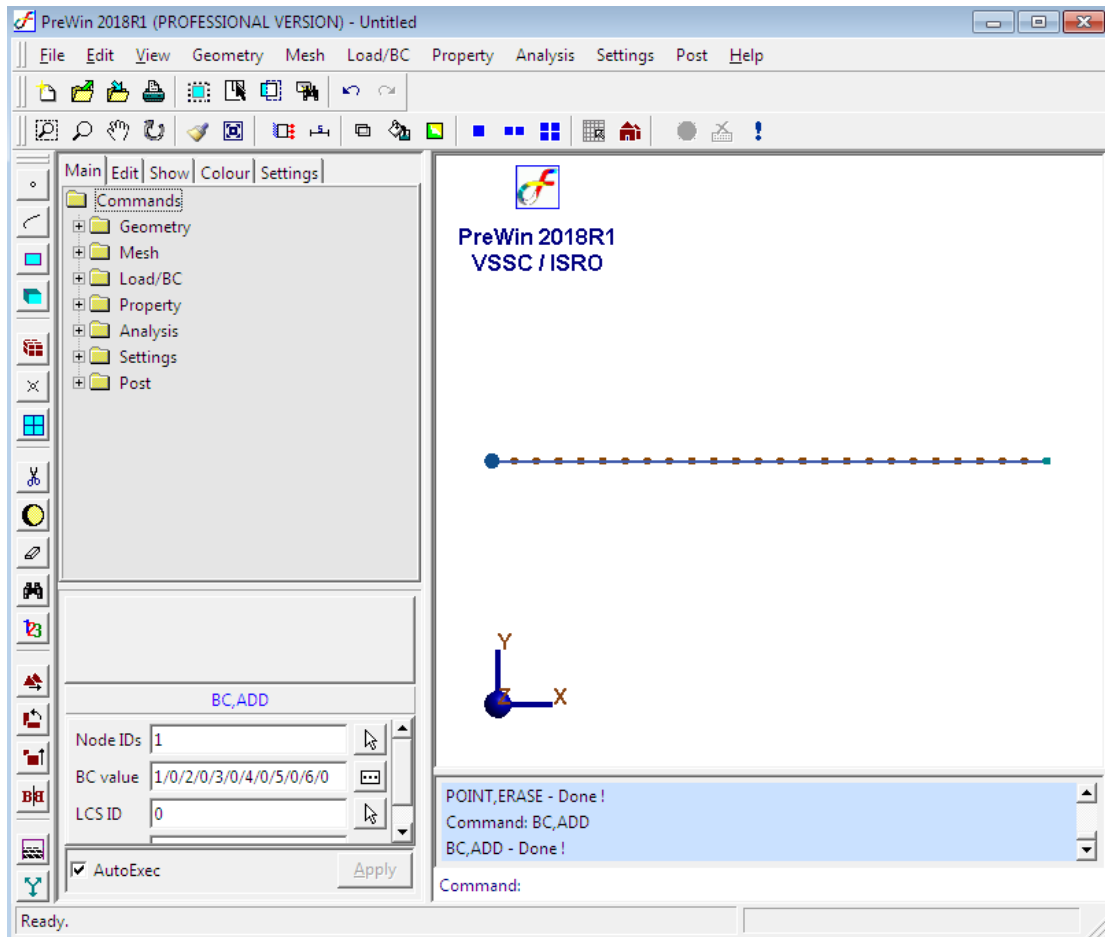
Menu : Load/BC → Displacement BC → Add

Parameters :

Node IDs	Select the node at the left boundary
BC Value	1/0/2/0/3/0/4/0/5/0/6/0
LCS ID	0



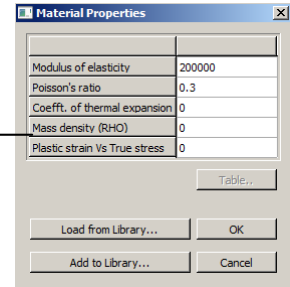
At the end of the above operation/s, your screen should look like this.



6. Specify material properties

Command : MATERIAL, ISO
 Menu : Property → Material → Isotropic
 Parameters :

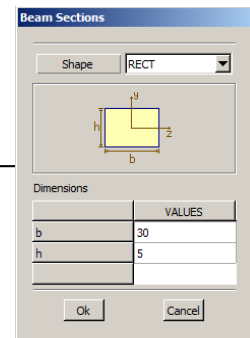
Element IDs	All
Material Data	200000/0.3/0/0/0
Material ID	1



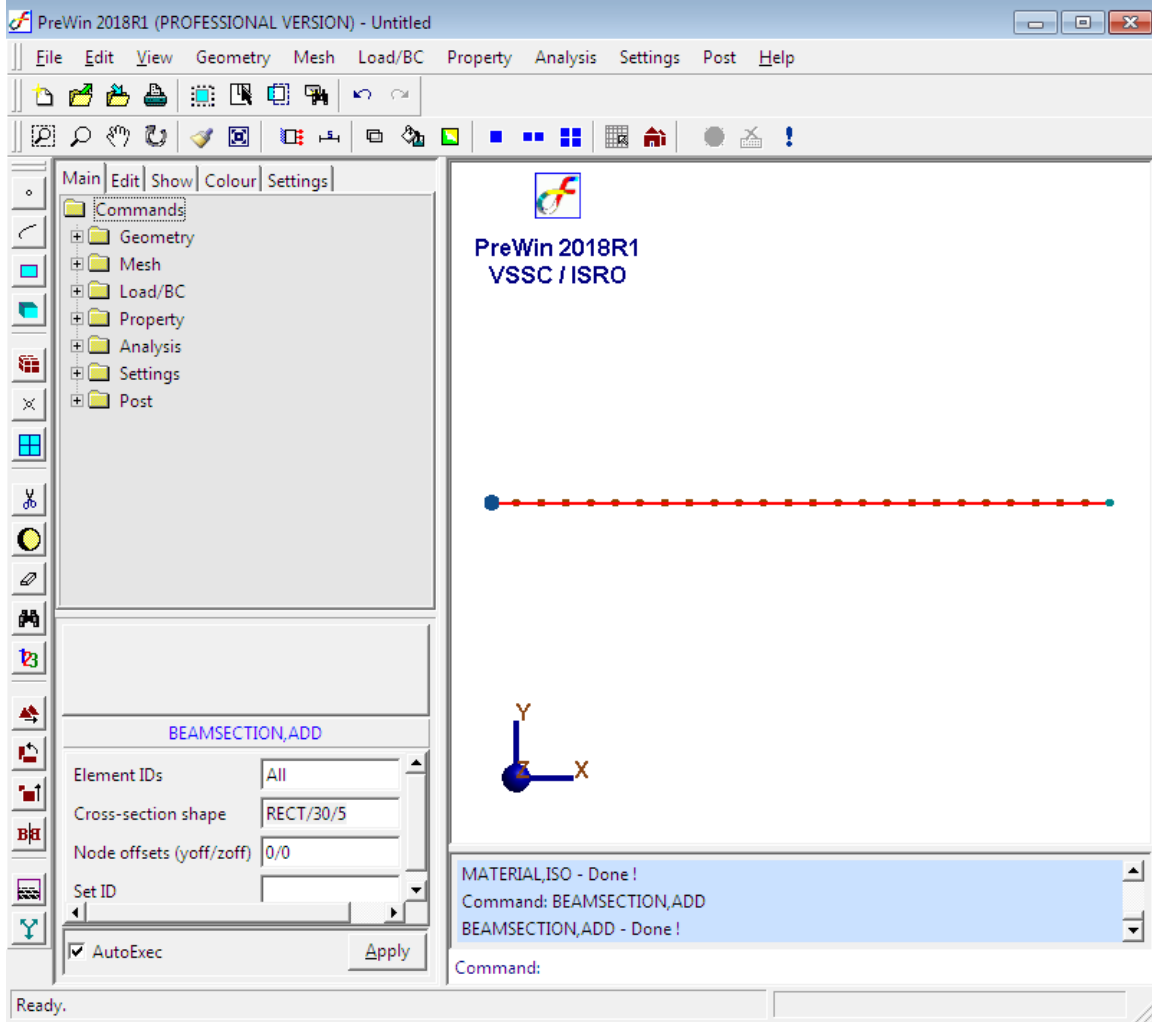
7. Specify Beam Properties

Command : BEAMSECTION, ADD
 Menu : Property → Physical → Beam Properties → Standard Section → Add
 Parameters :

Element IDs	All
Cross section Shape	RECT/30/5
Angle about axis	0
Node offsets (yoff/zoff)	0/0



At the end of the above operation/s, your screen should look like this.

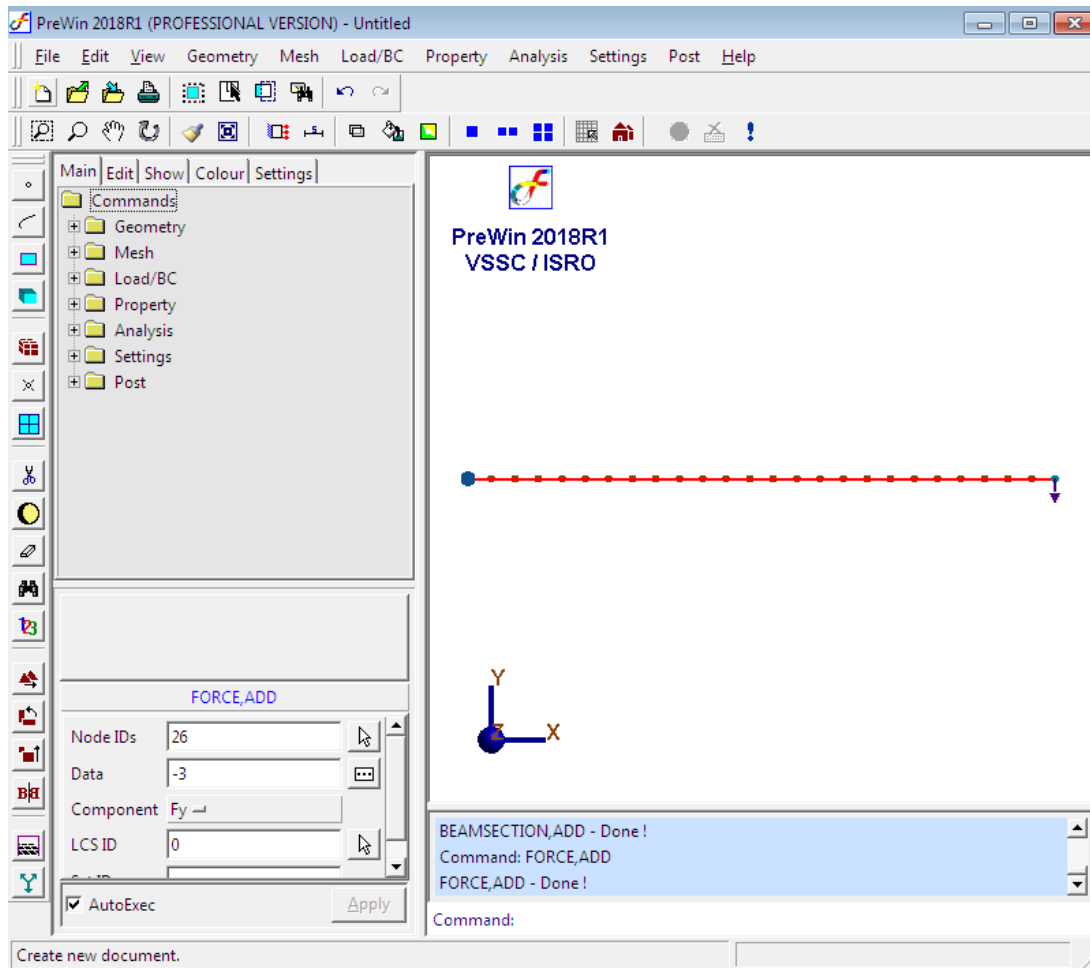


8. Specify point load

Command : FORCE, ADD
 Menu : Load/BC → Point Load → Add
 Parameters :

Node IDs	26
Data	-3
Component	Fy
LCS ID	0
Set ID	1

At the end of the above operation/s, your screen should look like this.



9. Set the analysis type

- Command : ANTYPE, SET
- Menu : Analysis → Analysis Type
- Parameters :

Analysis Type	Static
----------------------	--------

10. Set the analysis options

- Command : ANOPTION, SET
- Menu : Analysis → Analysis Options
- Parameters :

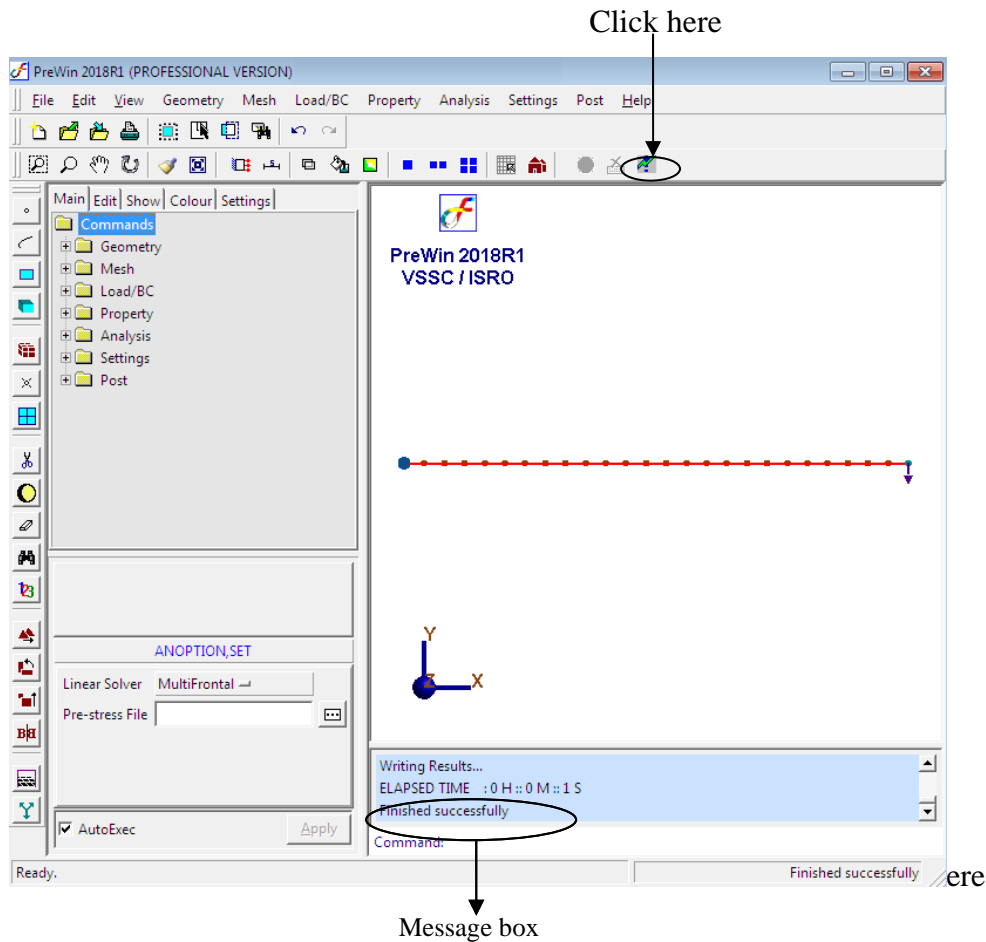
Linear Solver	Multi Frontal
----------------------	---------------

11. Save the project model

Menu : File → Save

12. Submit the job into FEAST

Menu : Analysis → Run solver



Note:

"Finished successfully" message appears on message box after executing is completed.

13. Perform post processing

i) Deformed shape

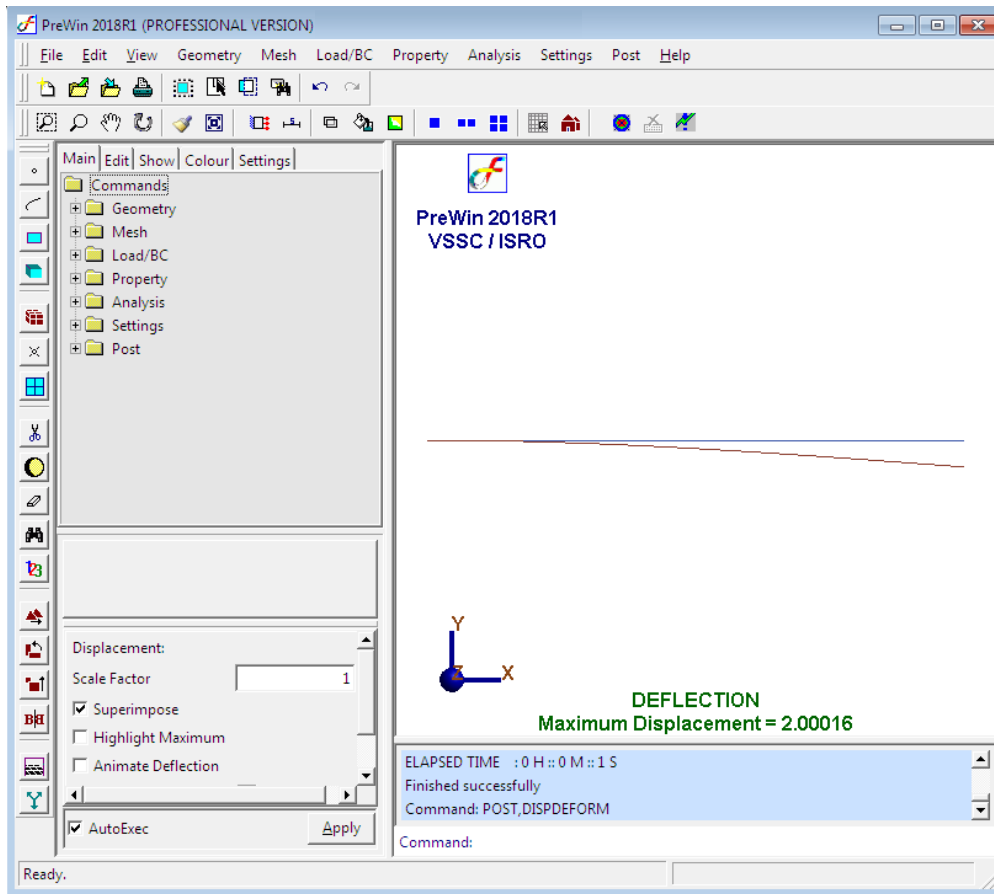
Command : POST, DISPDEFORM

Menu : Post → Deformed shape

Parameters :

Scale factor	1
---------------------	---

At the end of the above operation/s, your screen should look like this.



ii) Stress contour

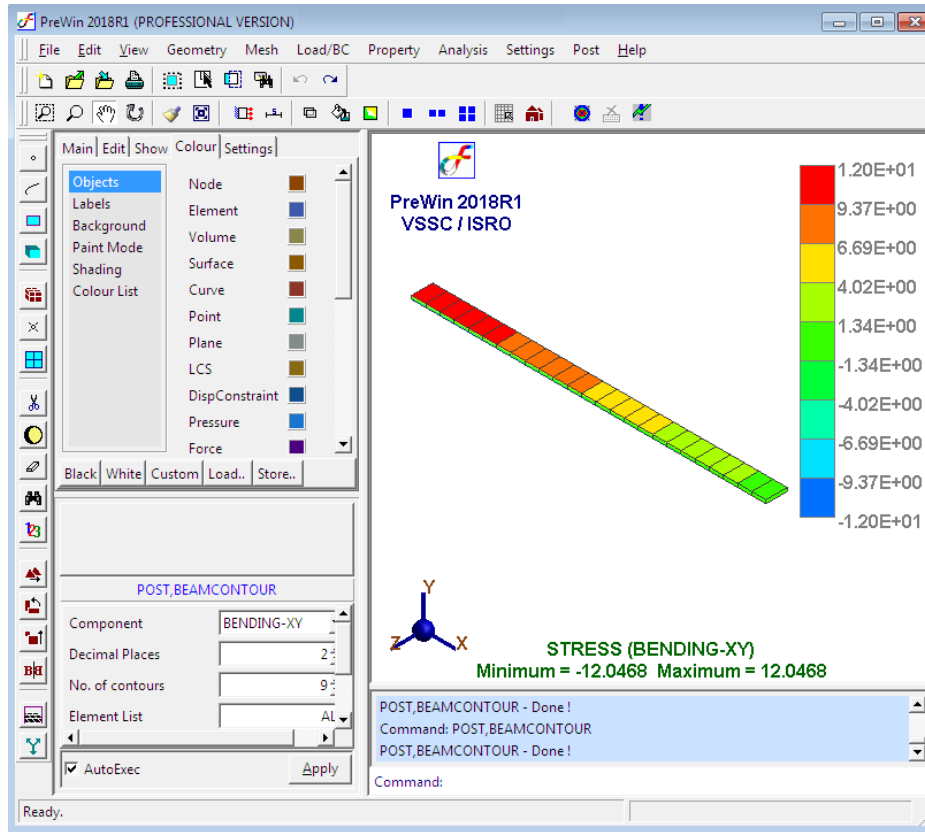
Command : POST, BEAMCONTOUR

Menu : Post → Beam Plots → Stress Contour

Parameters :

Component	BENDING-XY
Decimal Places	2
No. of contours	9
Element List	All

At the end of the above operation/s, your screen should look like this.



iii) Strain contour

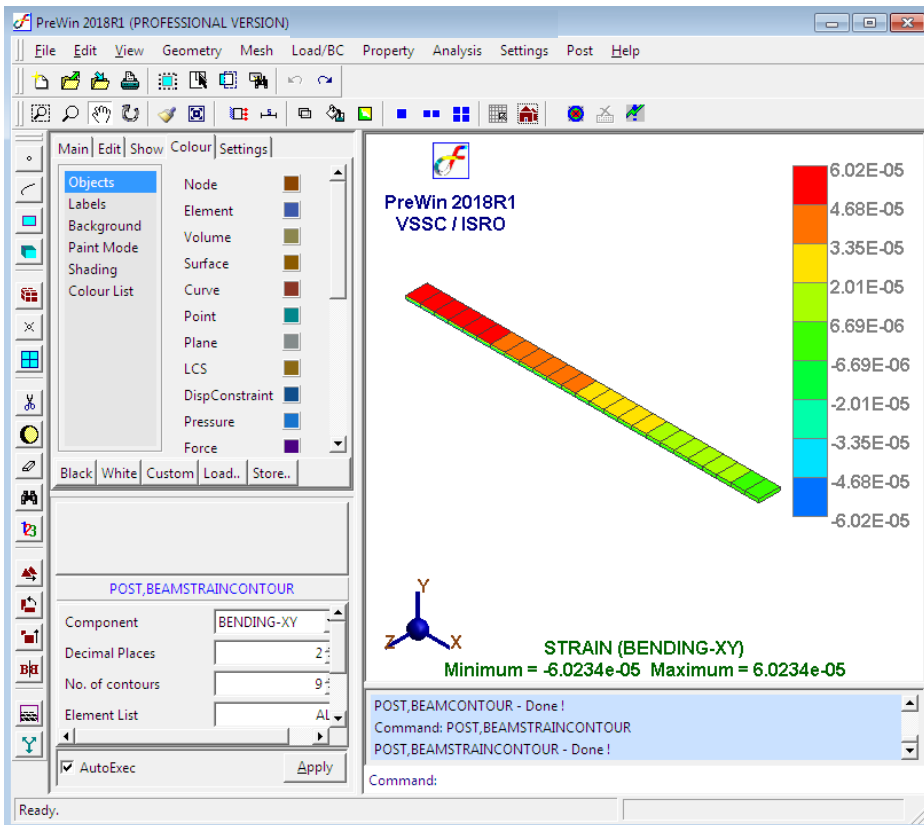
Command : POST, BEAMSTRAINCONTOUR

Menu : Post → Beam Plots → Strain Contour

Parameters :

Component	BENDING-XY
Decimal Places	2
No. of contours	9
Element List	All

At the end of the above operation/s, your screen should look like this.



iv) Beam force

a. Shear force diagram

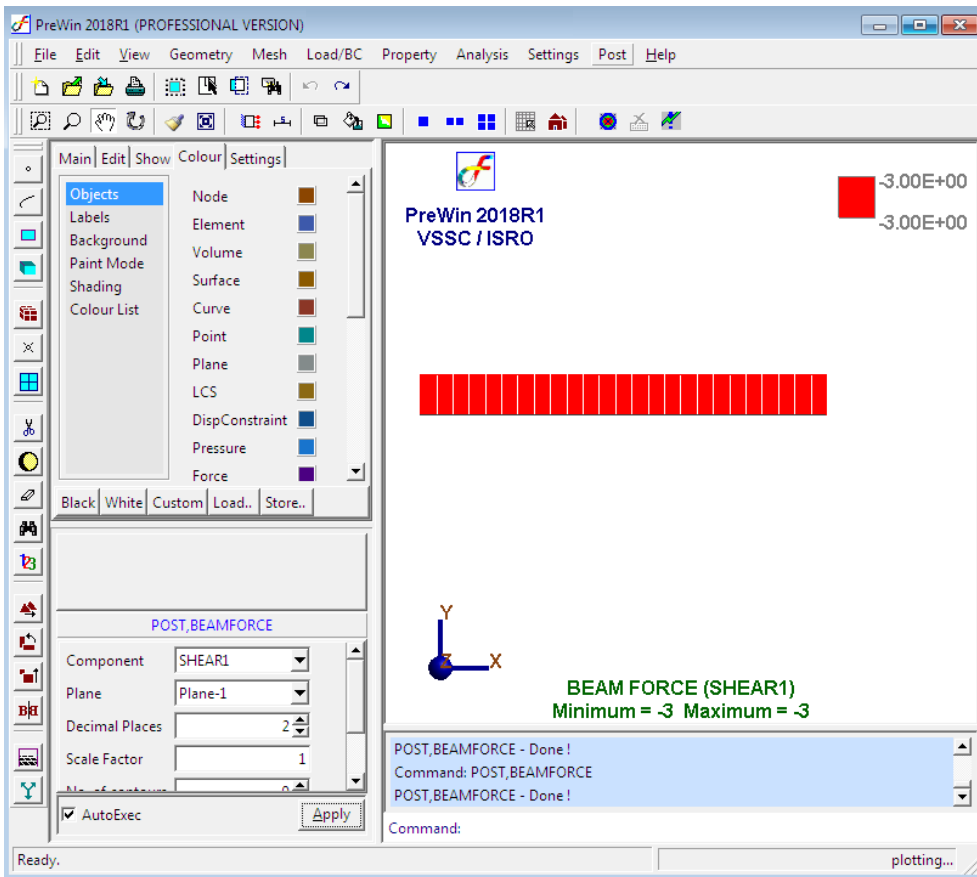
Command : POST, BEAMFORCE

Menu : Post → Beam Plots → Force Diagram

Parameters :

Component	SHEAR 1
Plane	Plane-1
Decimal Places	2
Scale Factor	1
No. of contours	9

At the end of the above operation/s, your screen should look like this.

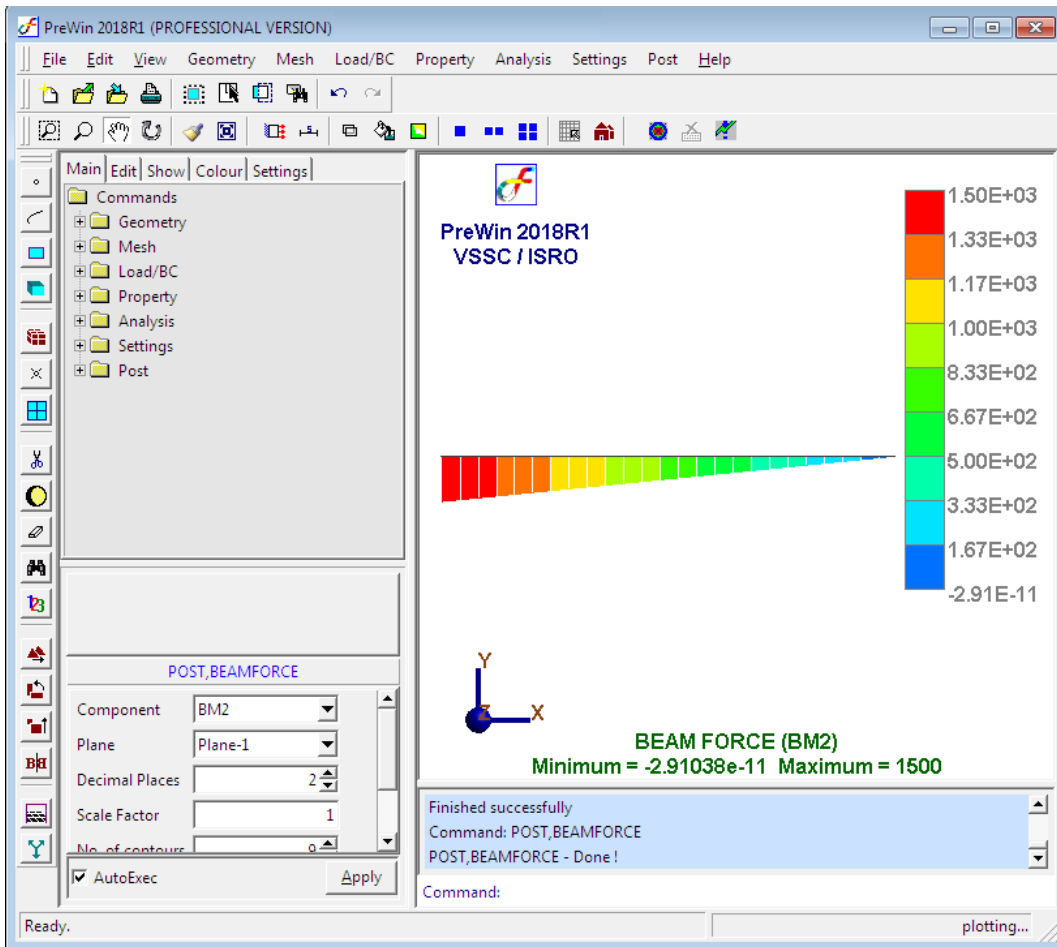


b. Bending moment diagram

- Command : POST, BEAMFORCE
- Menu : Post → Beam Plots → Force Diagram
- Parameters :

Component	BM2
Plane	Plane-1
Decimal Places	2
Scale Factor	1
No. of contours	9

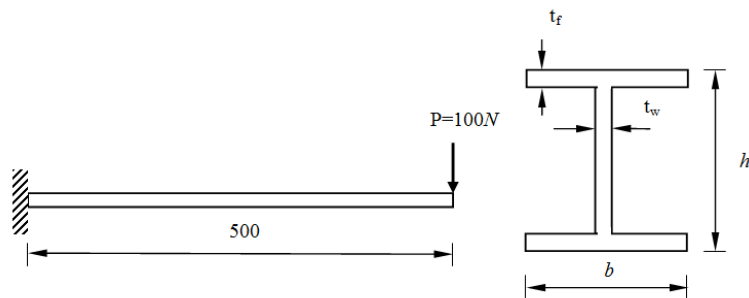
At the end of the above operation/s, your screen should look like this.



c. *.DAT file shows the input data and *.OUT file shows the output file.

Note:

Now try for an i-section beam as shown below.



$E=200GPa, \nu=0.3, b=50mm, h=100mm, t_w=7mm, t_f=5mm$

Ans: Deflection, $\delta=0.0144mm$