

## STATIC ANALYSIS OF A SIMPLY SUPPORTED BEAM WITH UNIFORMLY DISTRIBUTED LOAD

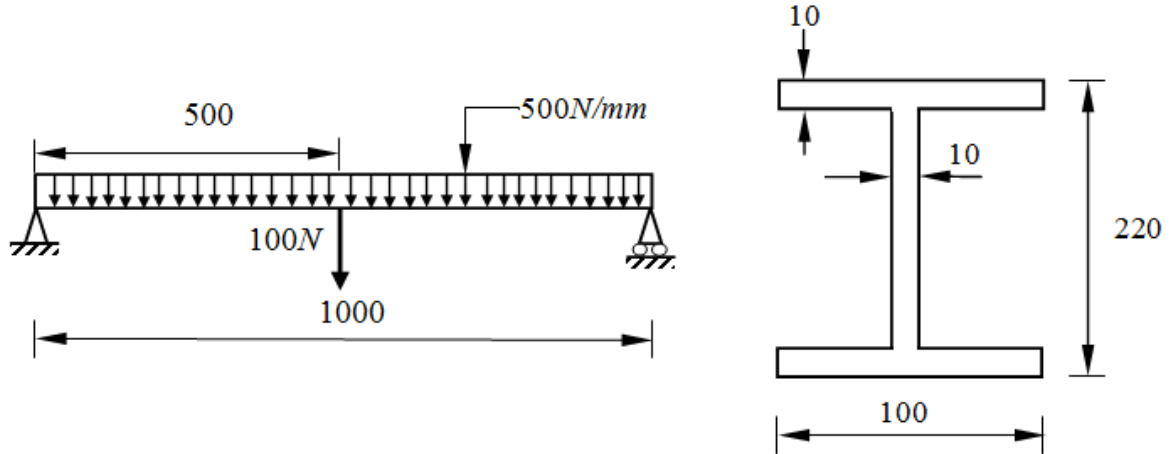


Figure 1

All dimensions are in *mm*

**Objective:** To find the deflection, stress, strain, shear force and bending moment diagram of simply supported beam with uniformly distributed load as shown in **Figure 1**.

**Analysis Type :** Static

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

**Poisson's ratio,  $\nu = 0.3$**

### PROCEDURE

#### STEP

1. Create three points at  $(0, 0, 0)$ ,  $(500, 0, 0)$  and  $(1000, 0, 0)$

Commands : POINT, ADD

Menu : Geometry → Key point → Create → By X/Y/Z

Parameters : (To be filled by the user)

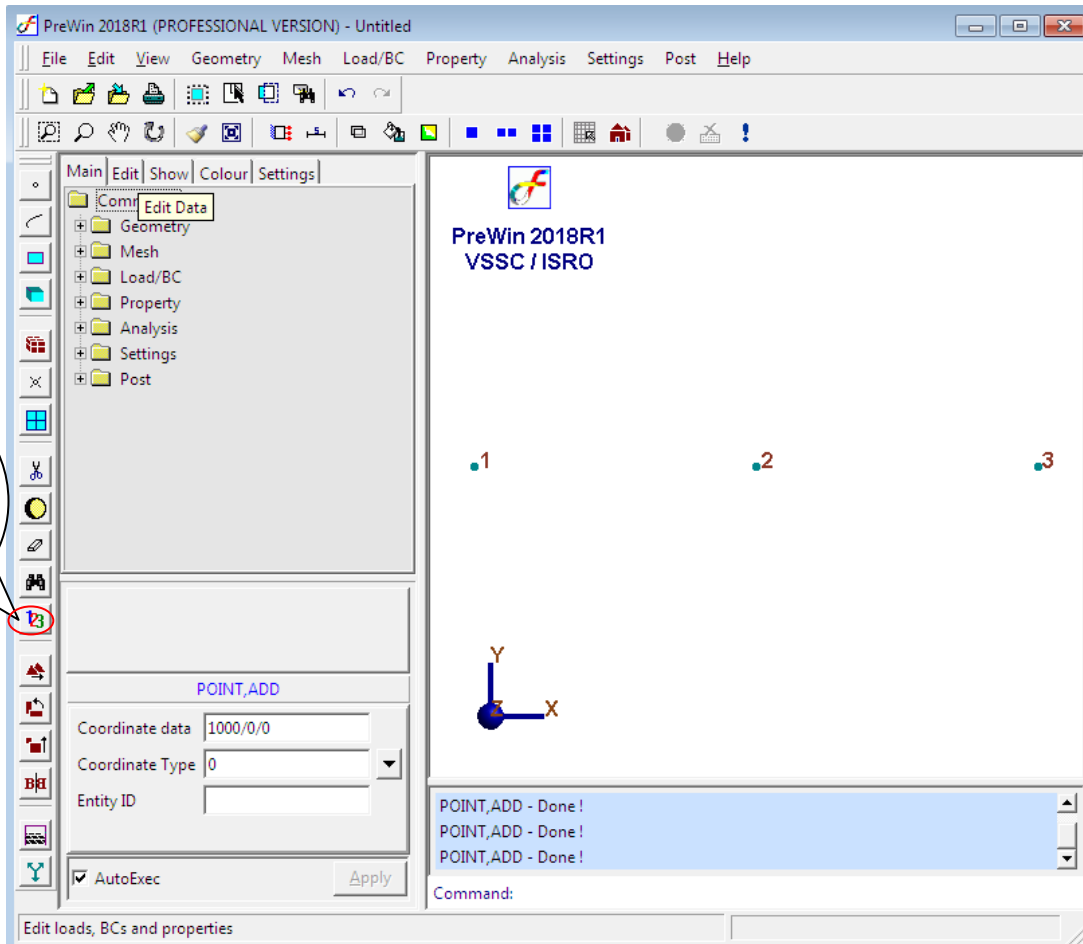
Coordinate Data	0/0/0
Coordinate Type	0
Entity ID	

Similarly create points  $(500/0/0)$  and  $(1000/0/0)$

**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

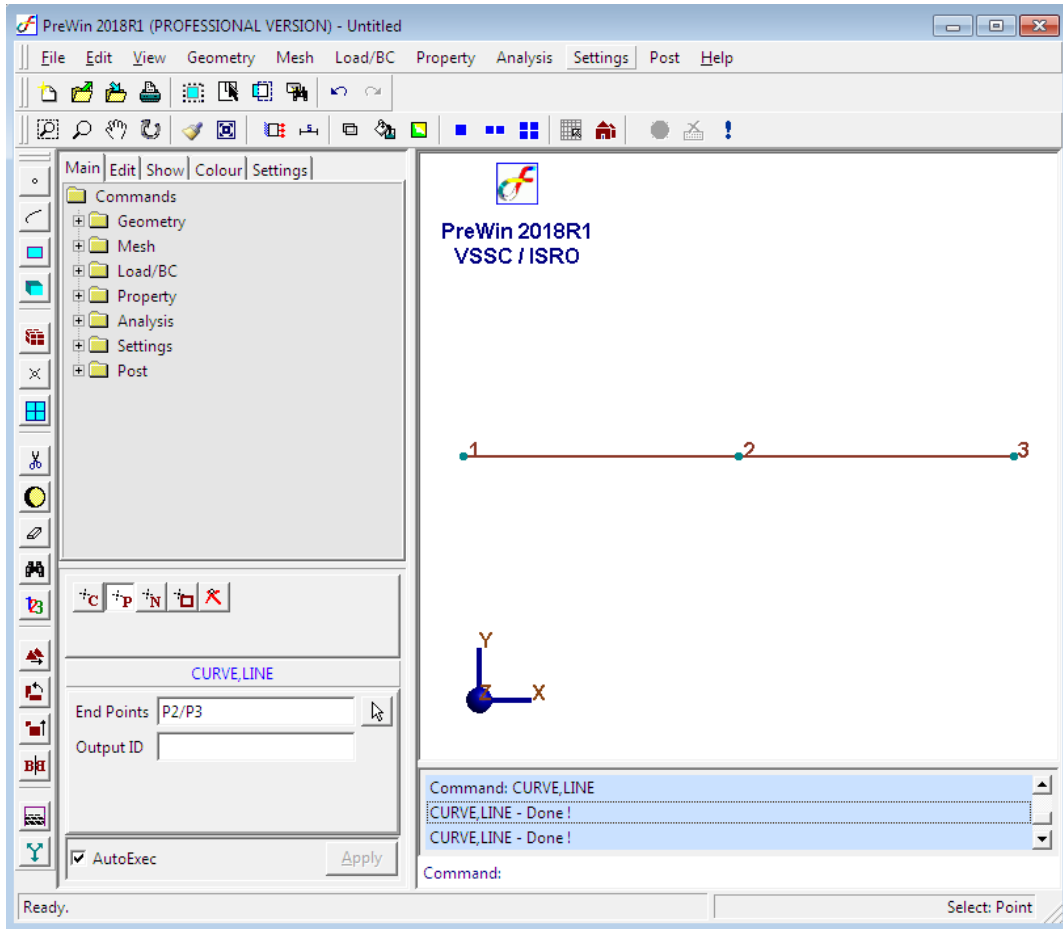
Menu : Geometry → Curve → Create → Line

Parameters :

<b>End points</b>	Use Mouse to pick the end points;P1/P2
<b>Entity ID</b>	1

Similarly create curve using points P2 and P3.

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



### 3. Meshing using beam elements

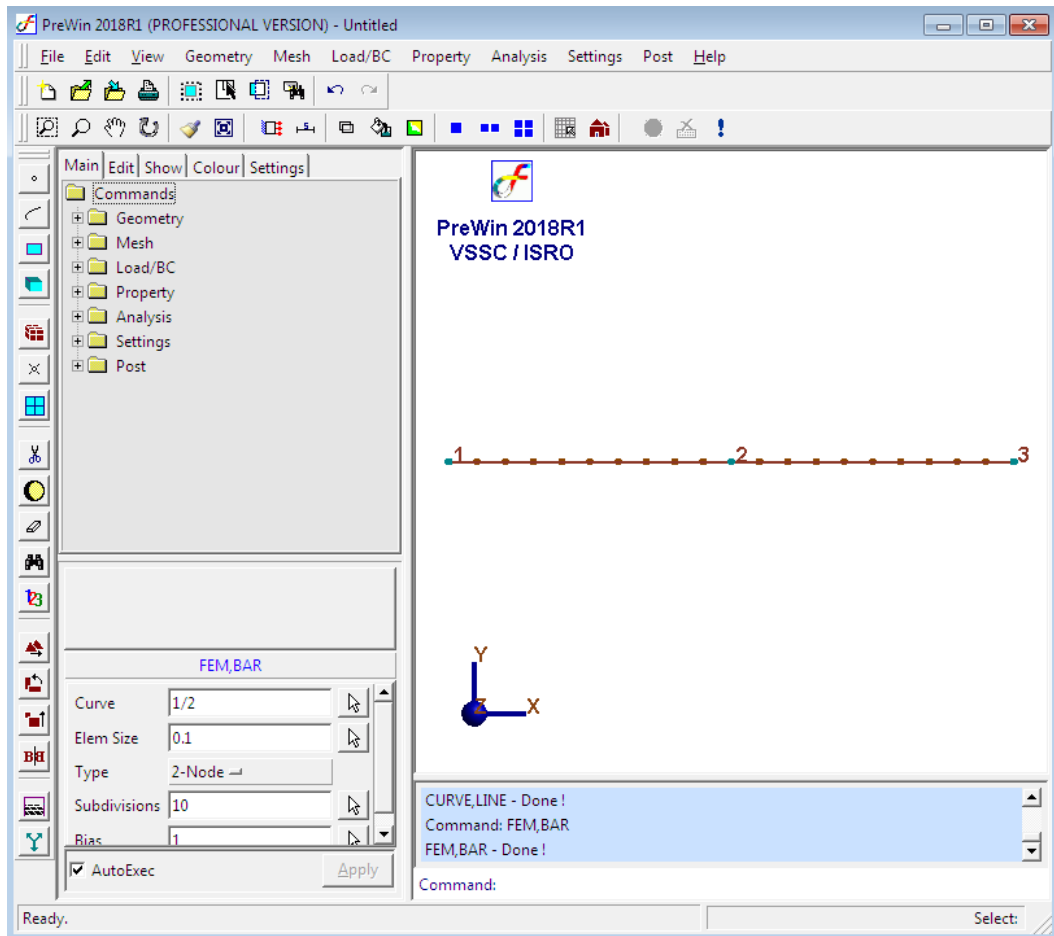
- Commands : FEM, BAR
- Menu : Mesh → FE Mesh → Bar
- Parameters :

<b>Curve</b>	1/2
<b>Element Size</b>	0.1
<b>Type</b>	2-node
<b>Subdivisions</b>	10
<b>Bias</b>	1.0

**Note:**

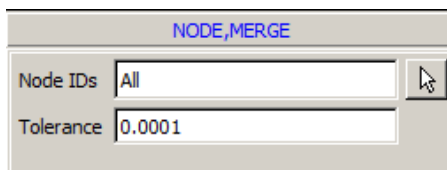
Right click/left click mouse point to alter sub-divisions.

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



**4. Merge duplicate nodes**

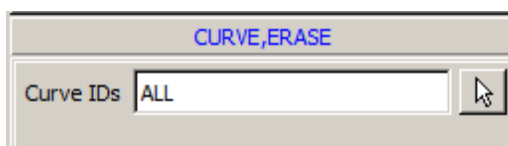
- Commands : NODE, MERGE
- Menu : Mesh → Node → Modify → Merge
- Parameters :



Duplicate nodes being merged will be highlighted.

**5. Erase curve**

- Commands : CURVE, ERASE
- Menu : Geometry → Curve → Miscellaneous → Erase
- Parameters :



**Note:**

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

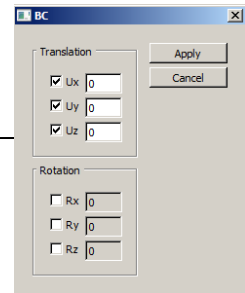
**6. Specify displacement boundary conditions**

Commands : BC, ADD

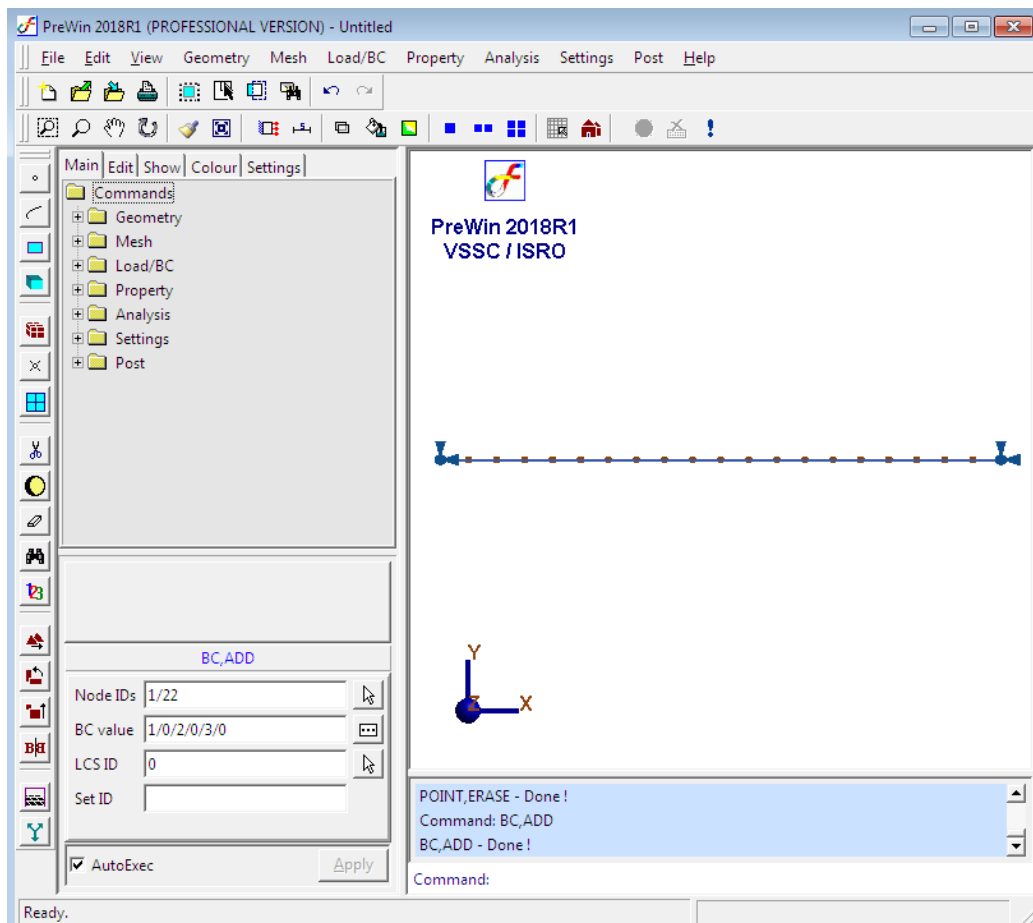
Menu : Load/BC → Displacement BC → Add

Parameters :

<b>Node IDs</b>	Select the end nodes ;1/22
<b>BC Value</b>	1/0/2/0/3/0
<b>LCS ID</b>	0



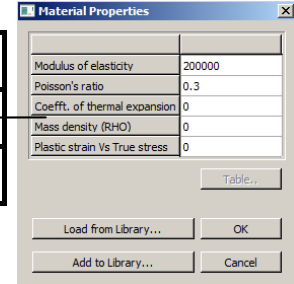
At the end of the above operation/s, your screen should look like this.



### 7. Specify material properties

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

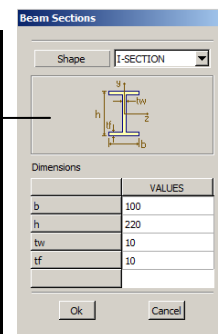
<b>Element IDs</b>	All
<b>Material Data</b>	200000/0.3/0/0/0
<b>Material ID</b>	1



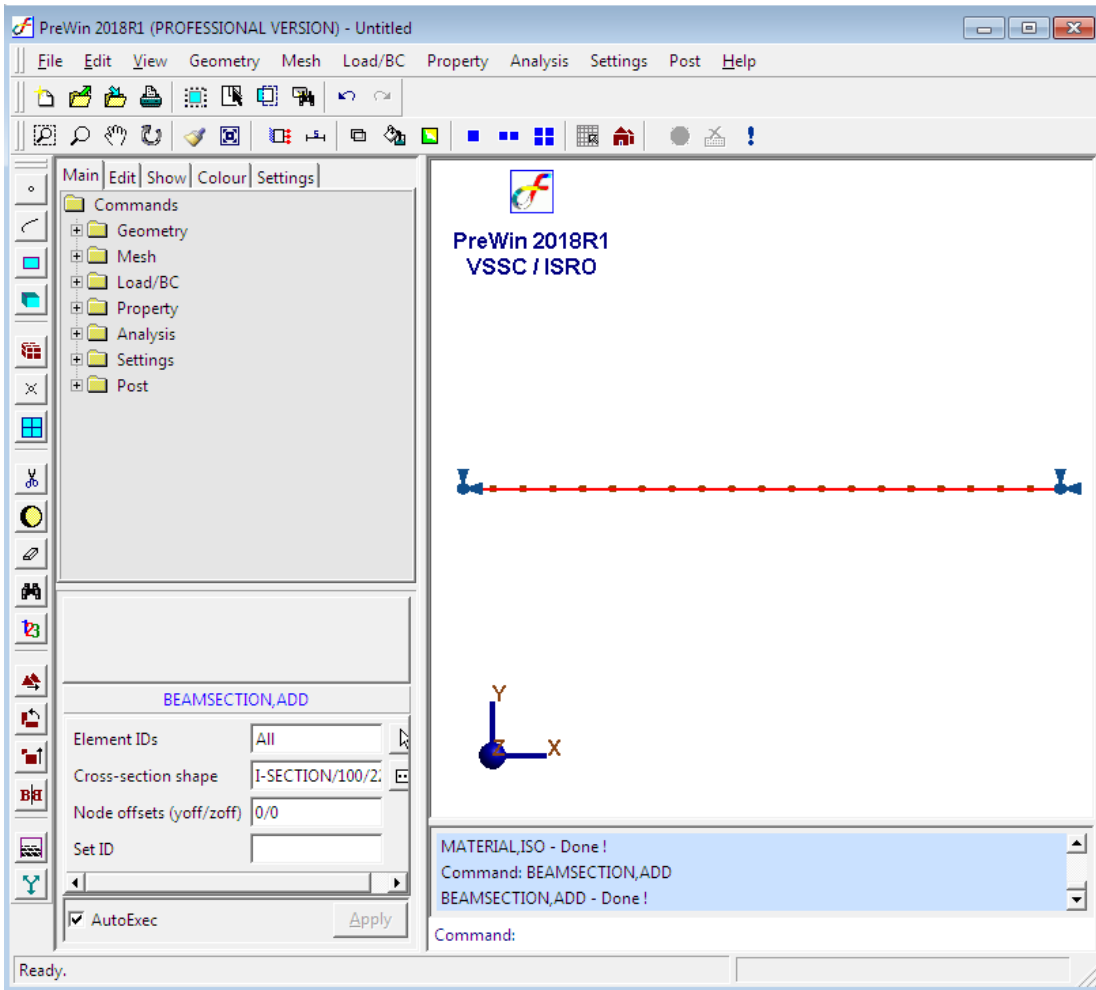
### 8. Specify Beam Properties

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

<b>Element IDs</b>	All
<b>Cross section Shape</b>	I-SECTION/100/220/10/10
<b>Angle about axis</b>	0
<b>Node offsets (yoff/zoff)</b>	0/0



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



## 9. Specify load

### 1) Specify point load

Command : FORCE, ADD

Menu : Load/BC → Point Load → Add

Parameters :

<b>Node IDs</b>	11
<b>Data</b>	-100
<b>Component</b>	Fy
<b>LCS ID</b>	0
<b>Set ID</b>	1

## (2) Specify uniformly distributed load

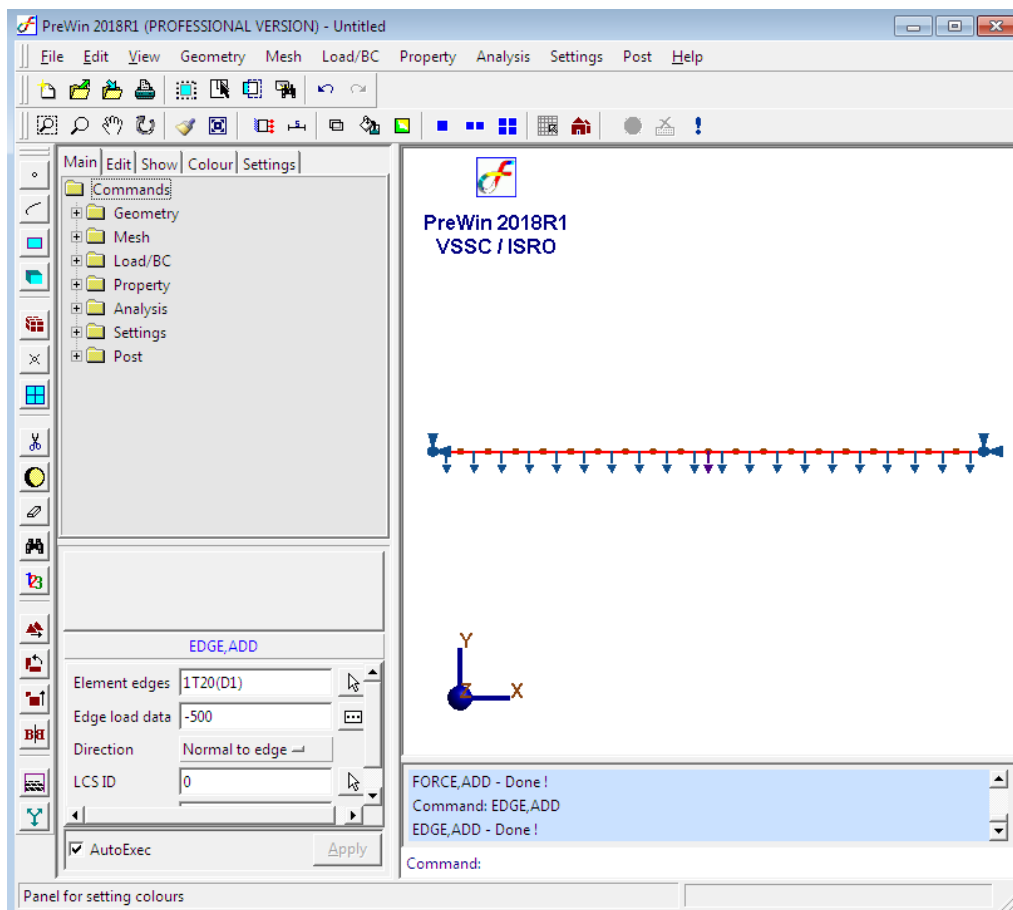
Command : EDGE, ADD

Menu : Load/BC → Edge Loads → Add

Parameters :

<b>Element Edges</b>	1T20(D1)
<b>Edge load data</b>	-500
<b>Direction</b>	Normal to edge
<b>LCS ID</b>	0
<b>Set ID</b>	1

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





**10. Set the analysis type**

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

<b>Analysis Type</b>	Static
----------------------	--------

**11. Set the analysis options**

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

<b>Linear Solver</b>	Multi Frontal
----------------------	---------------

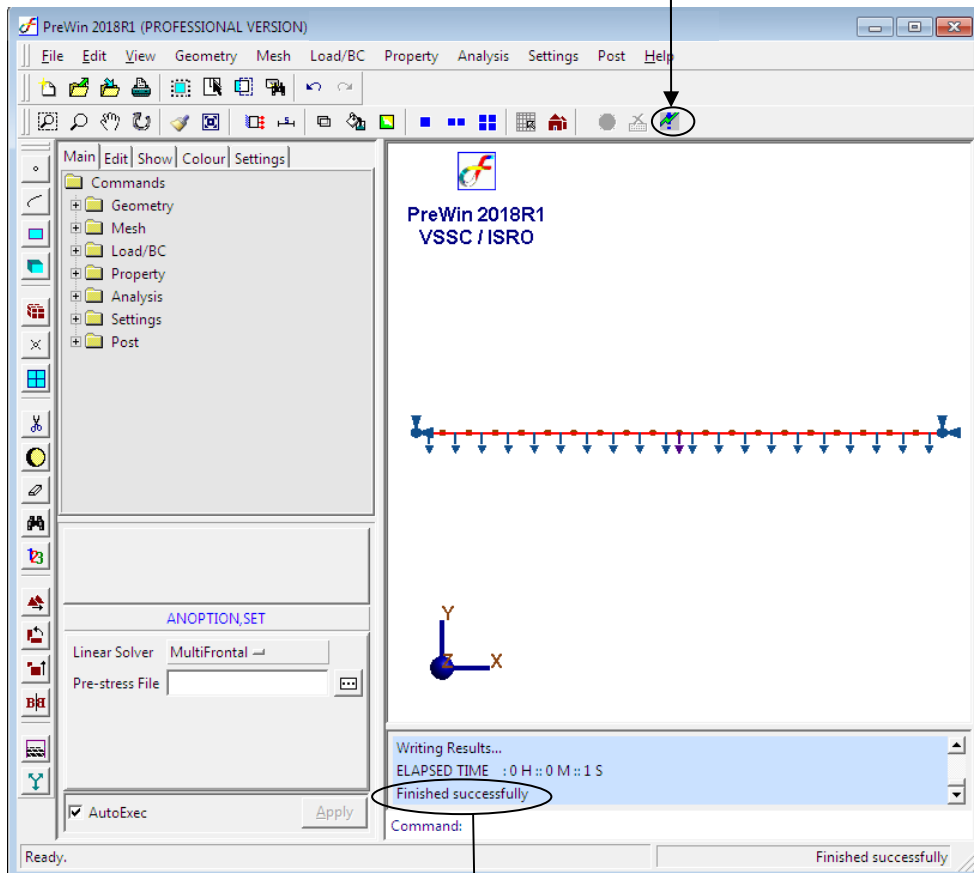
**12. Save the project model**

Menu : File → Save

**13. Submit the job into FEAST**

Menu : Analysis → Run solver

Click here



Message box

**Note:**

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

**14. Perform post processing****i) Deformed shape**

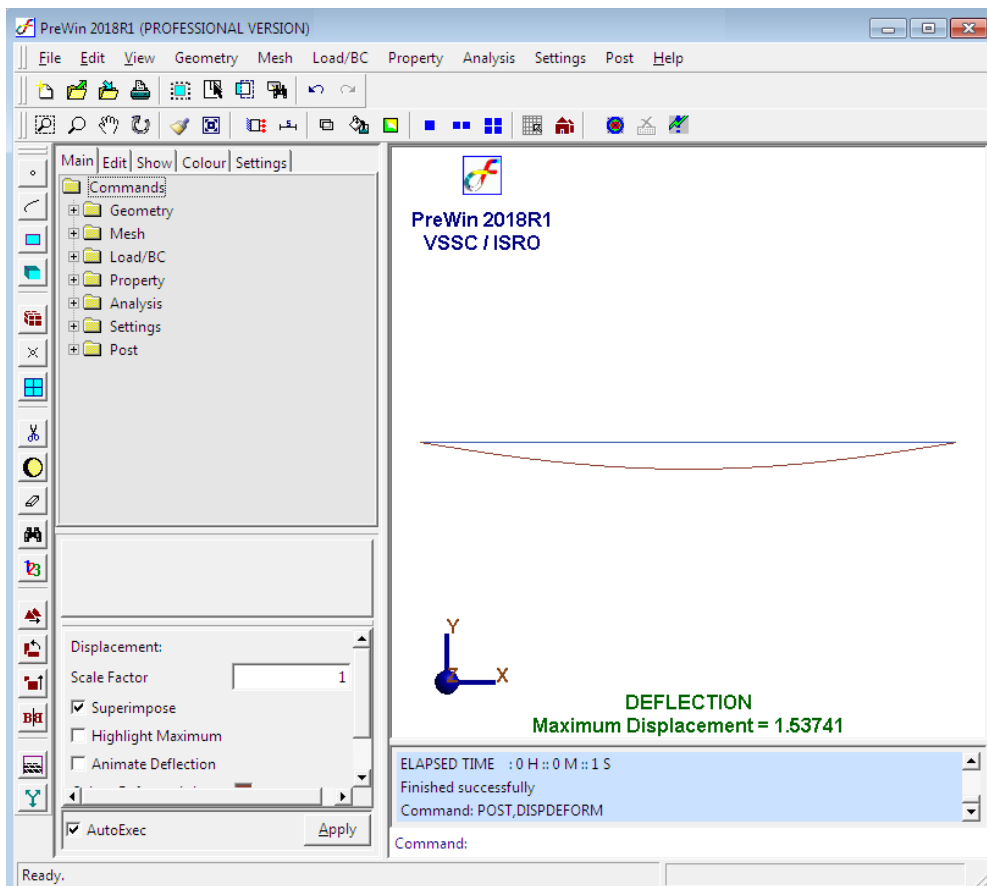
Command : POST, DISPDEFORM

Menu : Post → Deformed shape

Parameters :

<b>Scale factor</b>	1
---------------------	---

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

**ii) Stress contour**

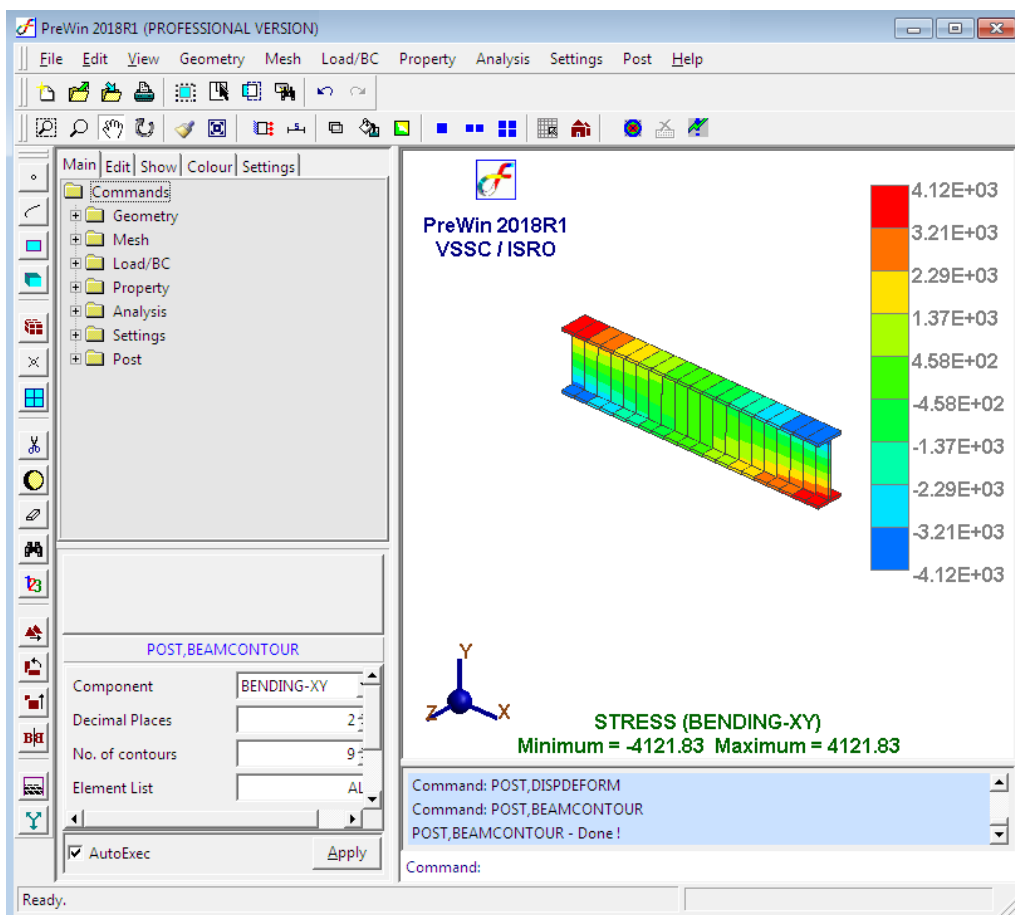
Command : POST, BEAMCONTOUR

Menu : Post → Beam Plots → Stress Contour

Parameters :

<b>Component</b>	BENDING-XY
<b>Decimal Places</b>	2
<b>No. of contours</b>	9
<b>Element List</b>	All

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



### iii) Strain contour

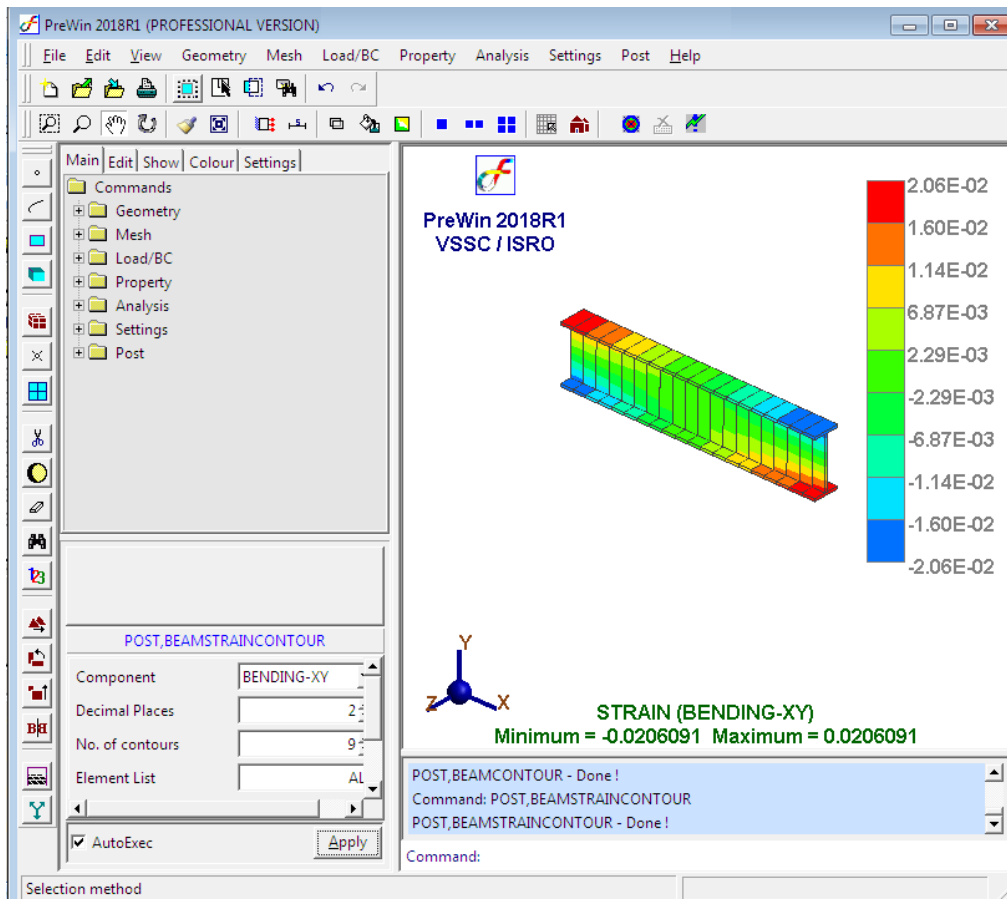
Command : POST, BEAMSTRAINCONTOUR

Menu : Post → Beam Plots → Strain Contour

Parameters :

<b>Component</b>	BENDING-XY
<b>Decimal Places</b>	2
<b>No. of contours</b>	9
<b>Element List</b>	All

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



#### iv) Beam force

##### a. Shear force diagram

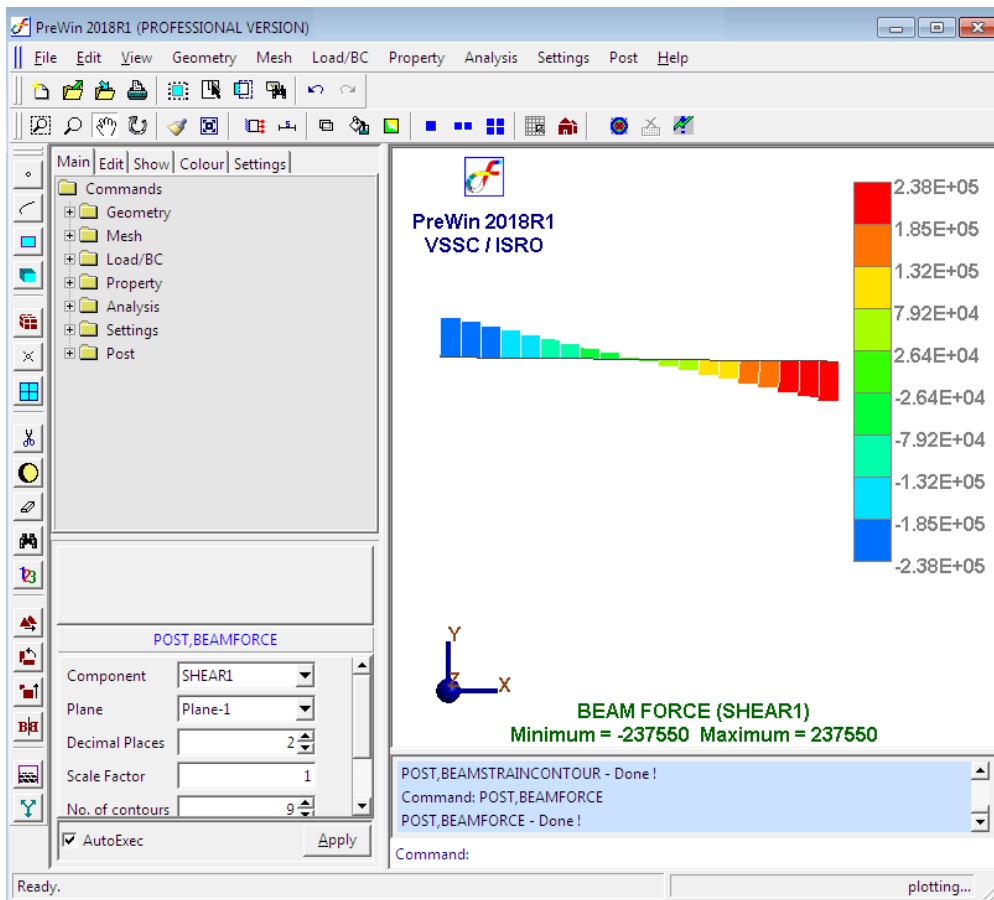
Command : POST, BEAMFORCE

Menu : Post → Beam Plots → Force Diagram

Parameters :

<b>Component</b>	SHEAR 1
<b>Plane</b>	Plane-1
<b>Decimal Places</b>	2
<b>Scale Factor</b>	1
<b>No. of contours</b>	9

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



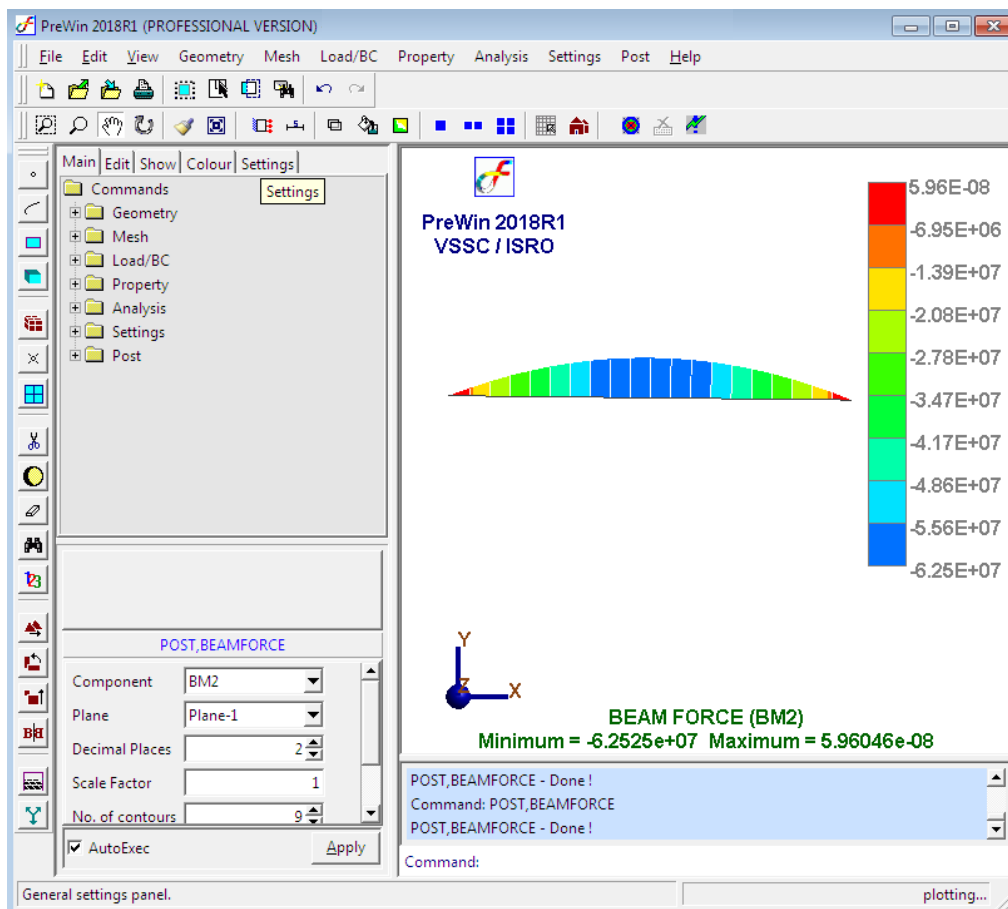
**b. Bending moment diagram**

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

Parameters :

<b>Component</b>	BM2
<b>Plane</b>	Plane-1
<b>Decimal Places</b>	2
<b>Scale Factor</b>	1
<b>No. of contours</b>	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.

**Hint:** Deflection,  $\delta = \frac{PL^3}{48EI}$

$$\text{Deflection, } \delta = \frac{5WL^3}{384EI}; W = w \times L$$