STATIC ANALYSIS OF A SIMPLY SUPPORTED BEAM WITH VARYING LOADS

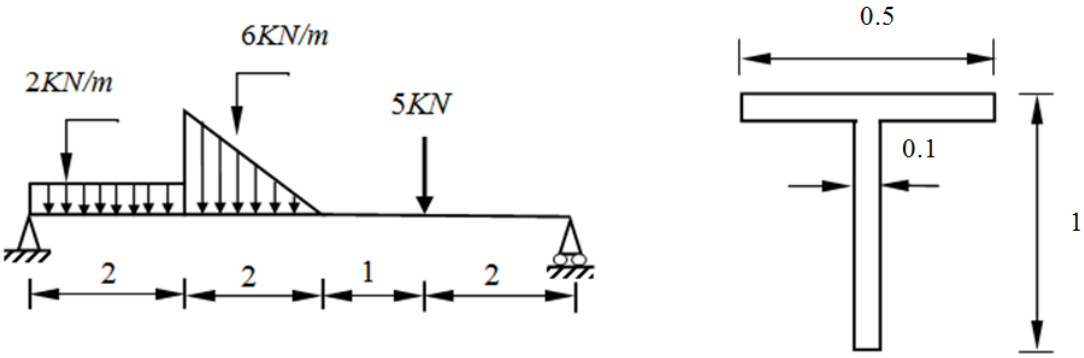


Figure 1

All dimensions are in *m*

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

Analysis Type : Static

Modulus of elasticity, *E* *=70 GPa*

Poisson's ratio, *v* *= 0.3*

PROCEDURE

STEP

1. Create key points at (0, 0, 0) (2, 0, 0) (4, 0, 0) (5, 0, 0) and (7, 0, 0)

Commands : POINT, ADD

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

Parameters : (To be filled by the user)

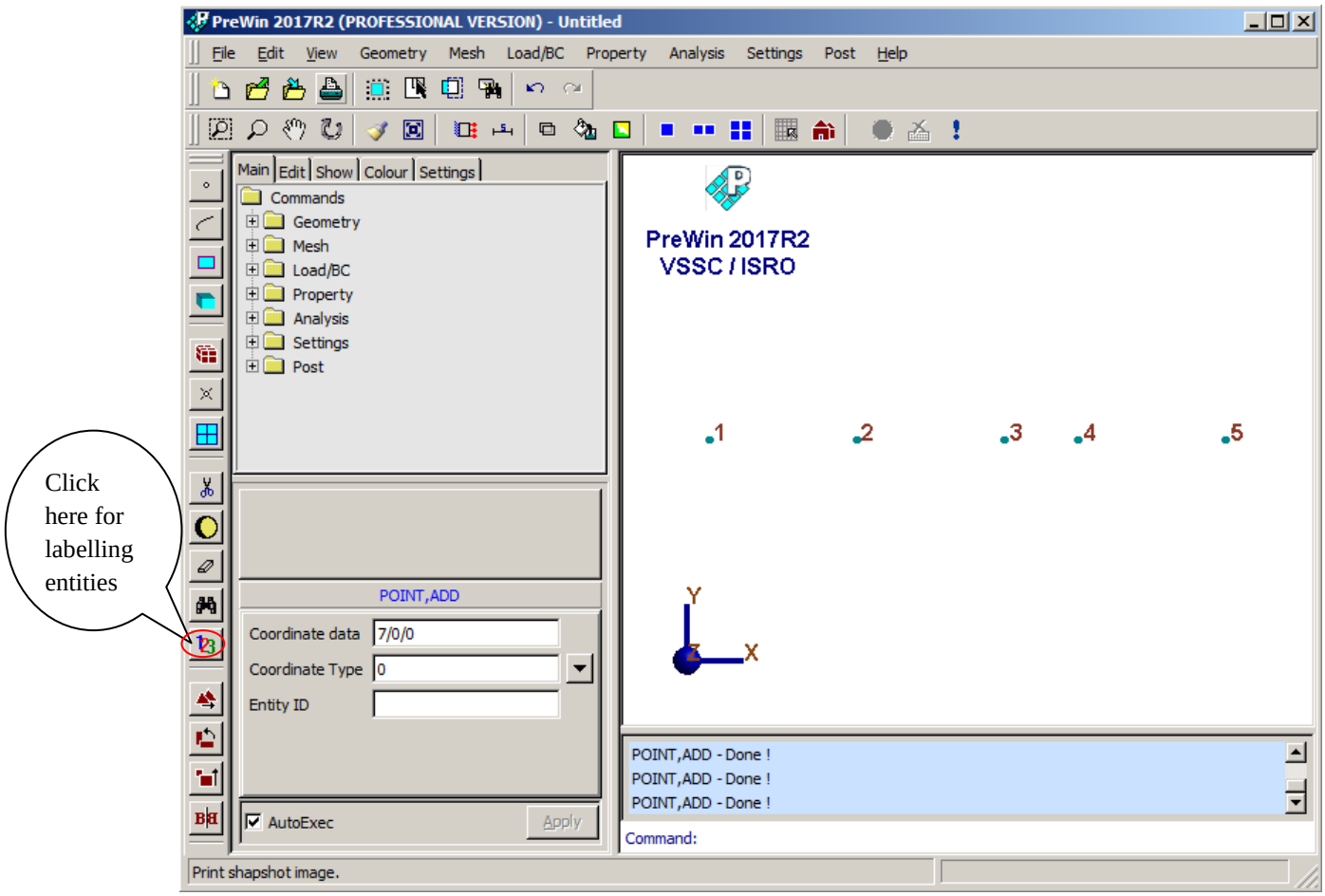
Coordinate Data	0/0/0
Coordinate Type	0
Entity ID	

Similarly create points (2/0/0) (4/0/0) (5/0/0) and (7/0/0)

Note:

Click "***Apply***" button or press ***cntrl+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 line

Commands : CURVE, LINE

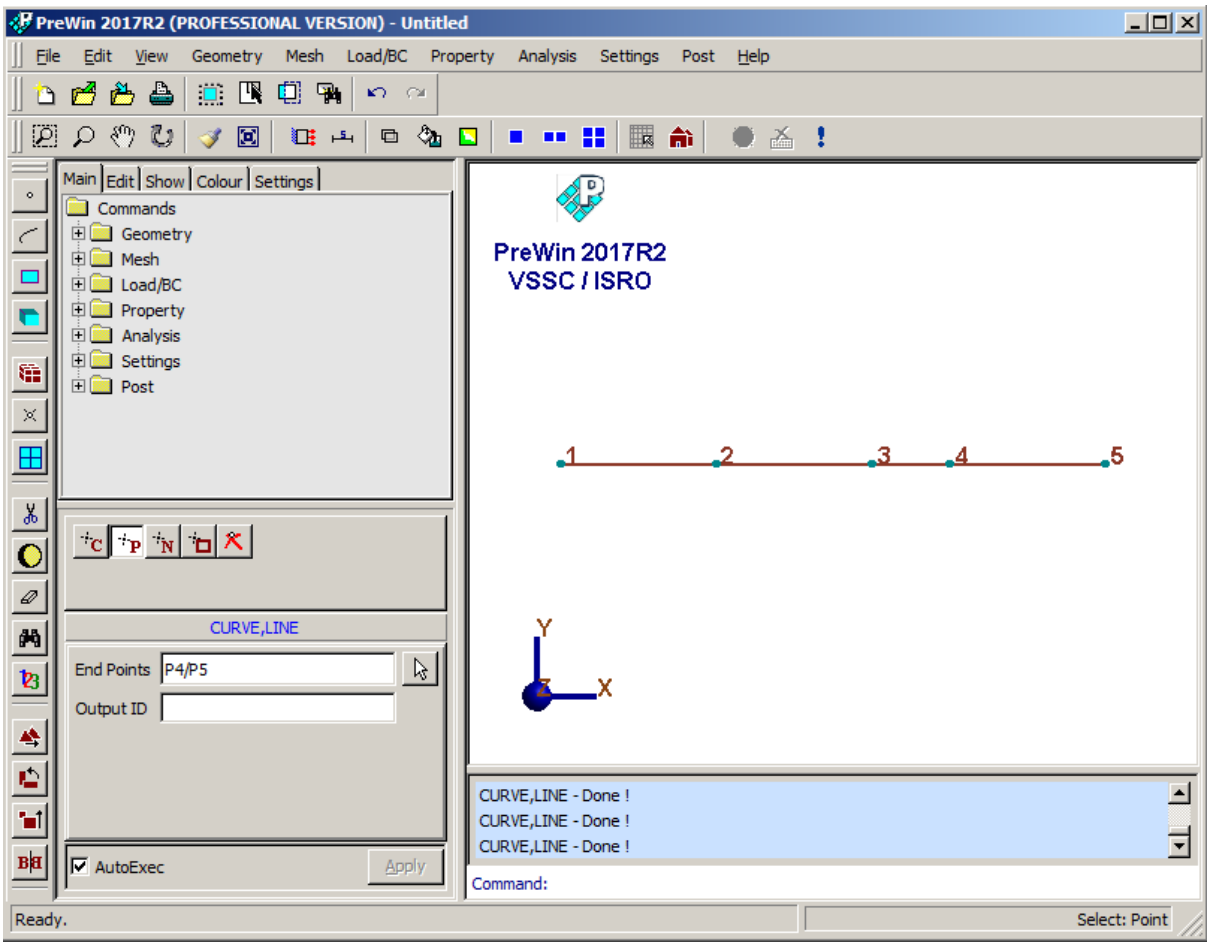
Menu : Geometry → Curve → Create →Line

Parameters :

End points	Use Mouse to pick the end points;P1/P2
Output ID	

Similarly create curves using points P2/P3, P3/P4, and P4/P5.

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



3. Meshing using beam elements

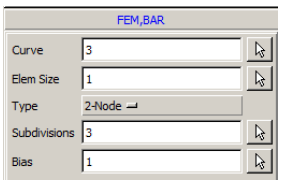
Commands : FEM, BAR

Menu : Mesh → FE Mesh → Bar

Parameters :

Curve	1/2/4
Element Size	1
Type	2-node
Subdivisions	6
Bias	1.0

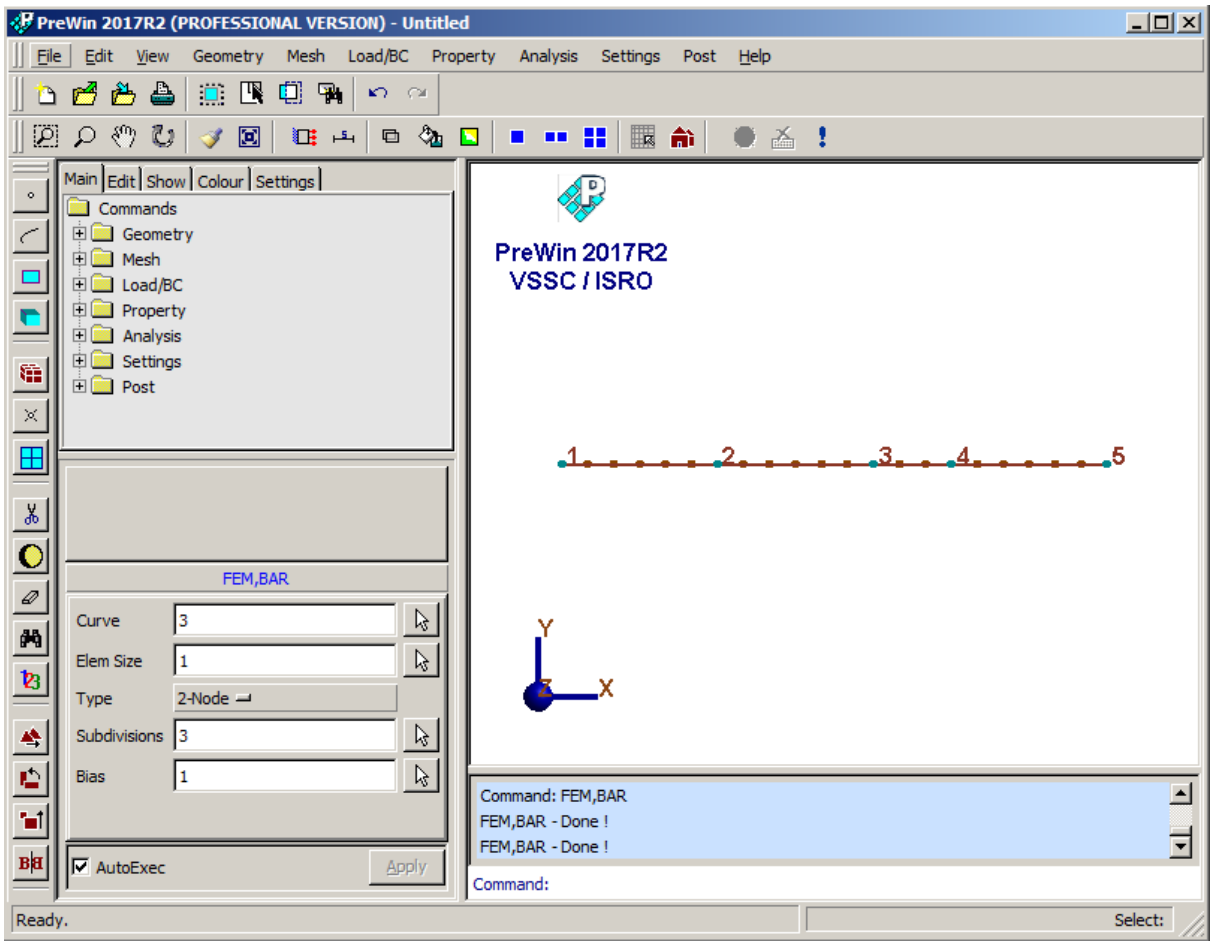
Similarly mesh curve 3 with 3 sub-divisions as shown below.



Note:

1. Right click/left click mouse point on each curve to alter sub-divisions.
2. Make sure nodes are present at locations of applied point load.

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

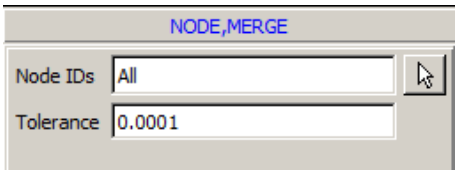


4. Merge duplicate nodes

Commands : NODE,MERGE

Menu : Mesh → Node → Modify → Merge

Parameters :



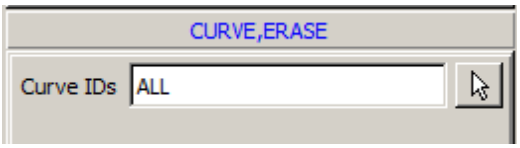
Duplicate nodes being merged will be highlighted.

5. Erase curve

Commands : CURVE,ERASE

Menu : Geometry → Curve → Miscellaneous → Erase

Parameters :



Note:

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

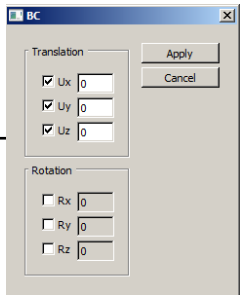
6. Specify displacement boundary conditions

Commands : BC,ADD

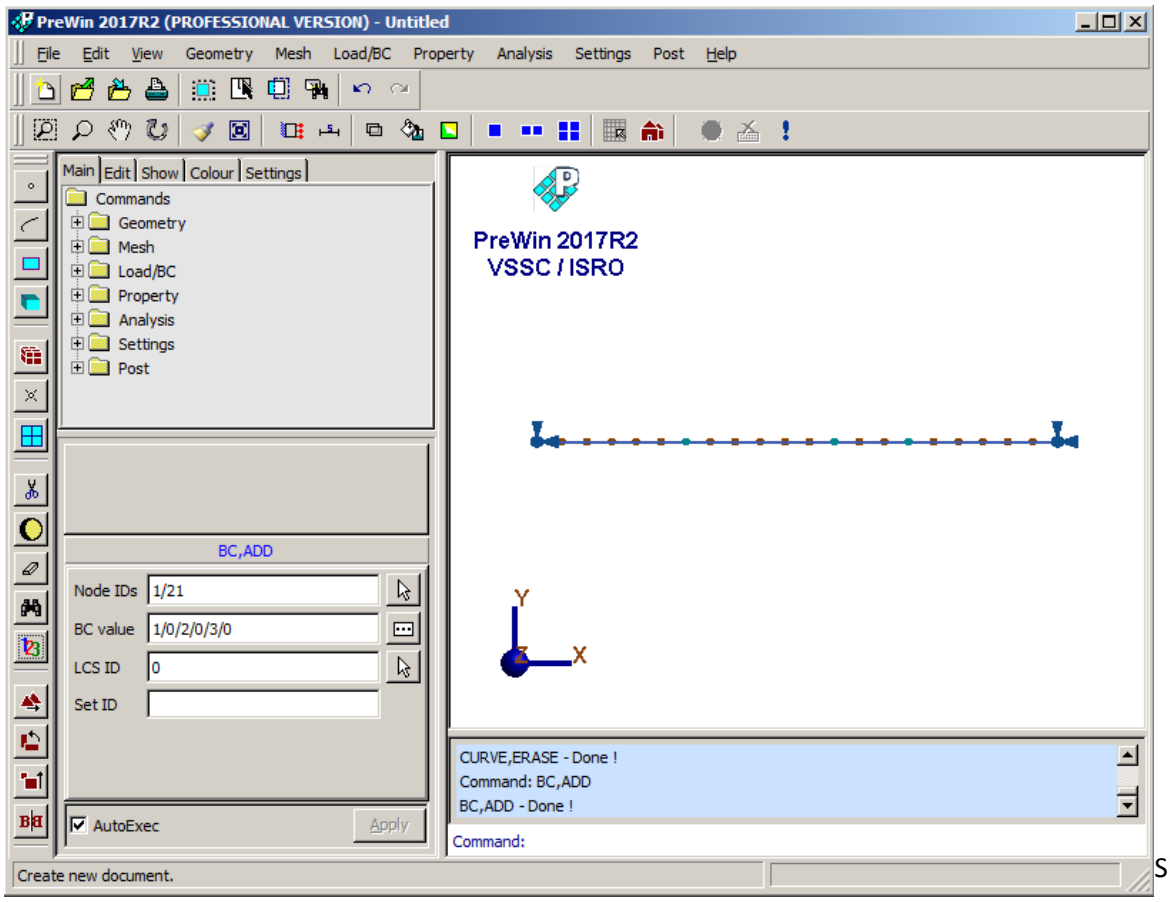
Menu : Load/BC → Displacement BC → Add

Parameters :

Node IDs	Select the end nodes ;1/21
BC Value	1/0/2/0/3/0
LCS ID	0



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



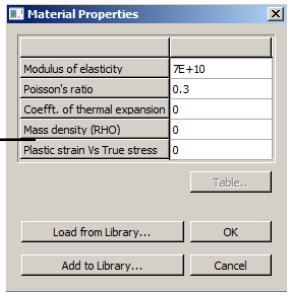
7. Specify material properties

Command : MATERIAL, ISO

Menu : Property → Material → Isotropic → Add

Parameters :

Element IDs	All
Material Data	7E+10/0.3/0/0/0
Material ID	1



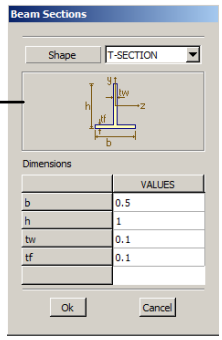
8. Specify beam properties

Command : BEAMSECTION, ADD

Menu : Property → Physical → Beam Properties →Standard Section →Add

Parameters :

Element IDs	All
Cross section Shape	T-SECTION/0. 5/1/0.1/0.1
Angle about axis	0
Node offsets (yoff/zoff)	0/0



9. Specify load

1) Specify point load

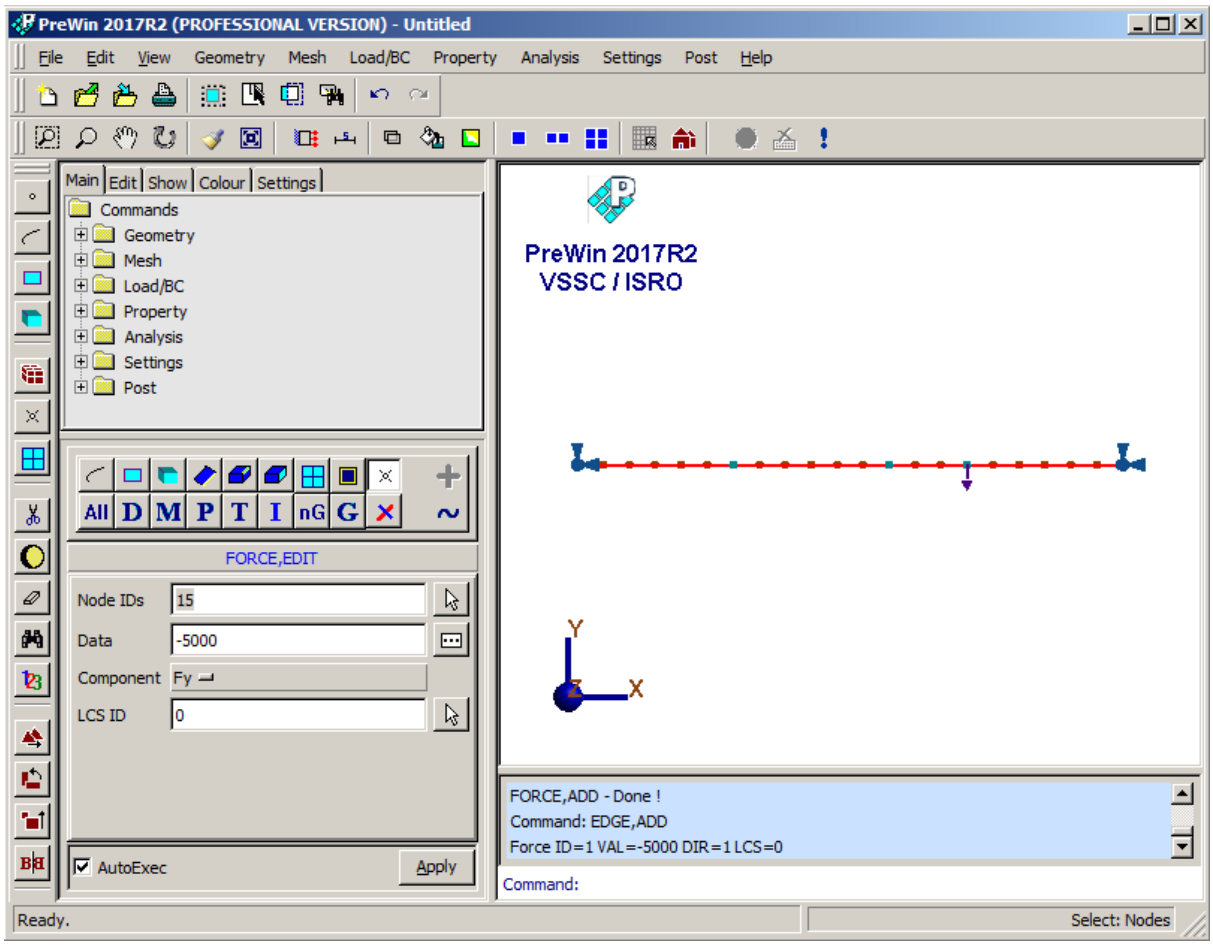
Command : FORCE, ADD

Menu : Load/BC → Point Load → Add

Parameters :

Node IDs	15
Data	-5000
Component	Fy
LCS ID	0
Set ID	1

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



(2) Specify uniformly distributed load

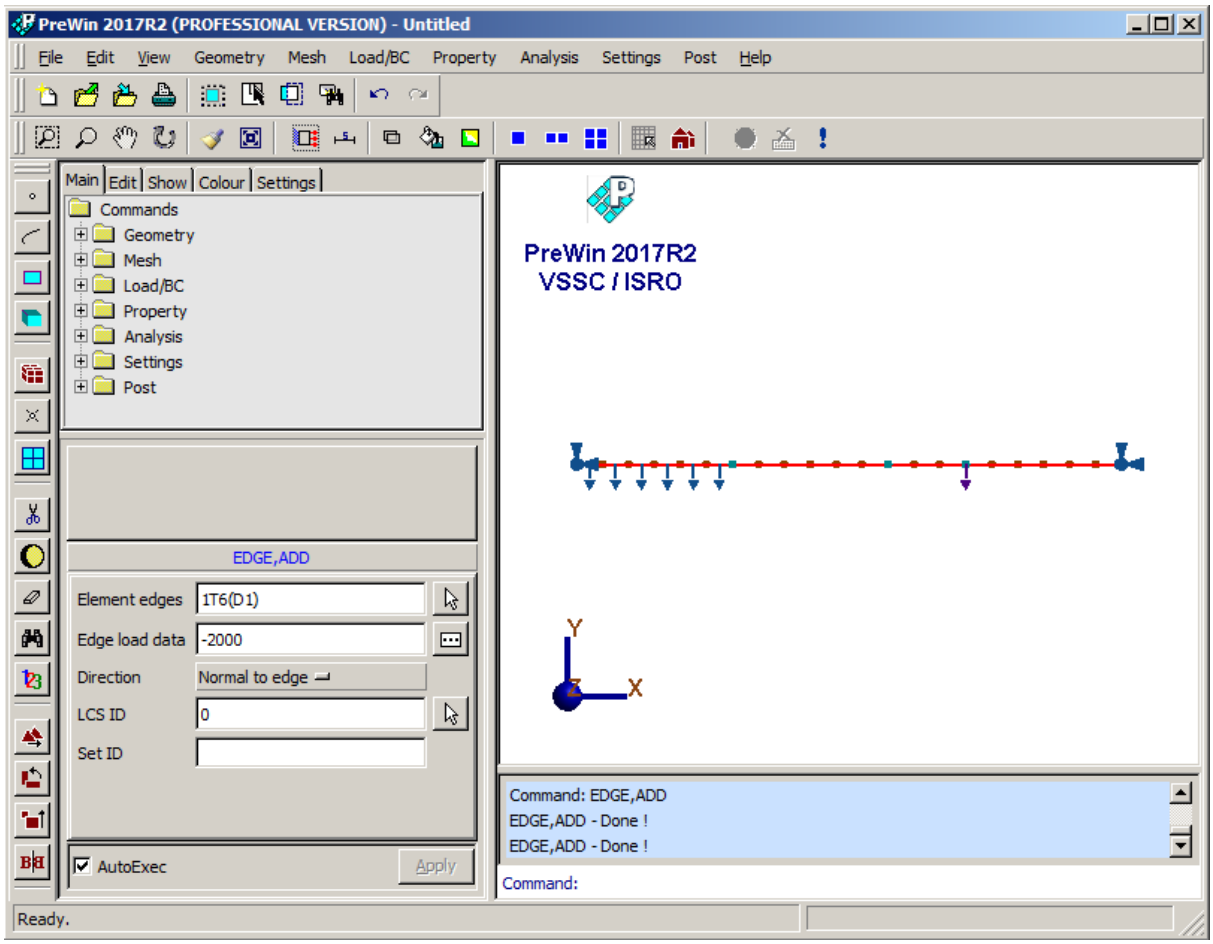
Command : EDGE,ADD

Menu : Load/BC → Edge Loads → Add

Parameters :

Element Edges	1T6(D1)
Edge load data	-2000
Direction	Normal to edge
LCS ID	0
Set ID	1

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



(3) Specify uniformly varying load

<u>Note:</u>

For uniformly varying load, function varying with respect to x-direction has to be defined and this expression is attached to edge load.

i. Define function

Menu : Property → Function → Expression

Command : FUNCTION, EXPRESSION

Parameters :

Set ID	1
Data	(6-(3*X))

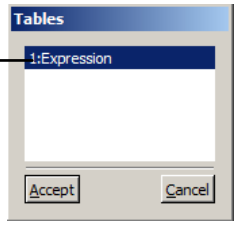
ii. Define varying load

Command : EDGE,ADD

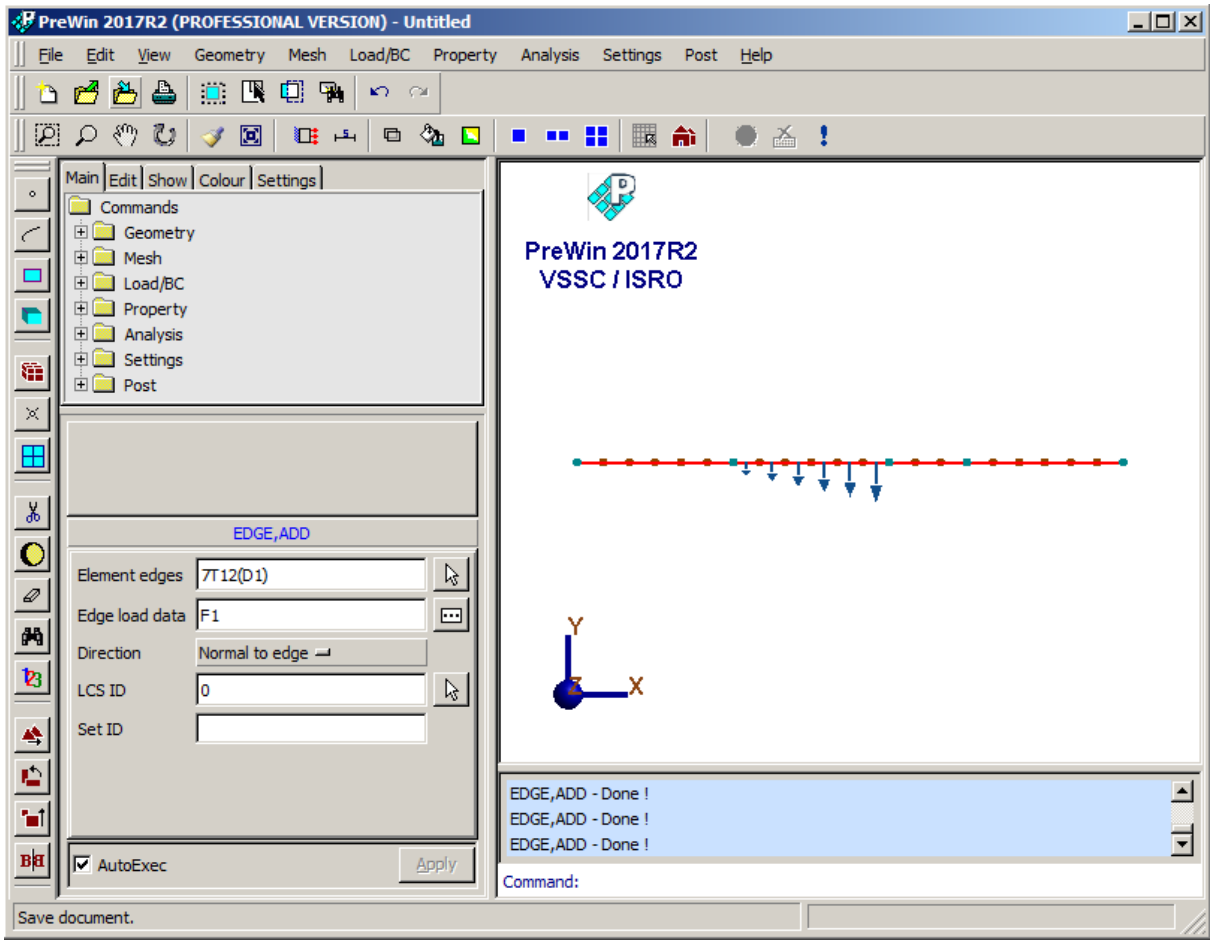
Menu : Load/BC → Edge Loads → Add

Parameters :

Element Edges	7T12(D1)
Edge load data	F1(select the function)
Direction	Normal to edge
LCS ID	0
Set ID	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 :

Analysis Type	Static

11. Set the analysis options

Command : ANOPTION, SET

Menu : Analysis → Analysis Options

Parameters :

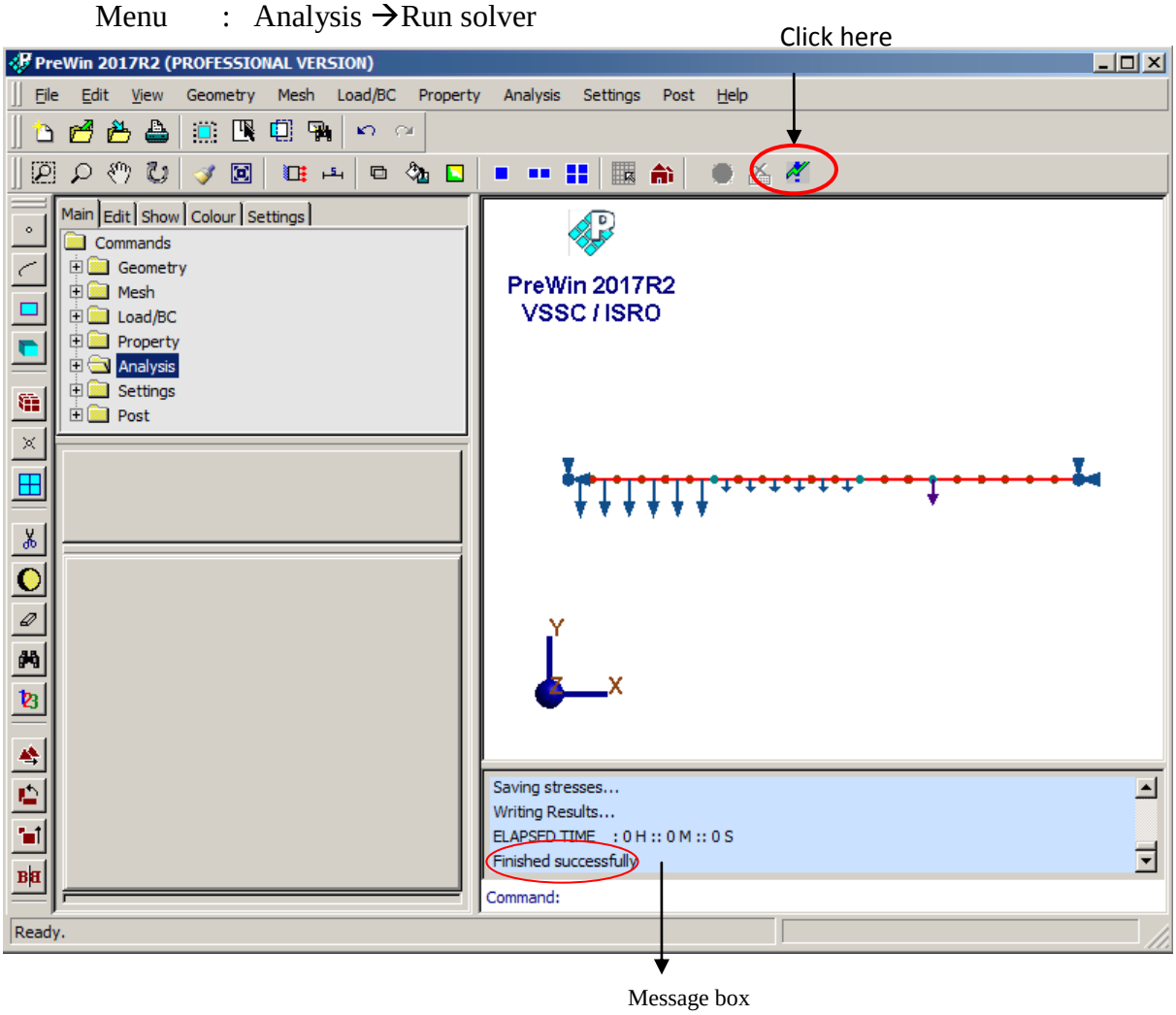
Linear Solver	Cholesky

12. Save the project model

Menu : File → Save

13. Submit the job into FEAST

Menu : Analysis →Run solver



Note:

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

14. Perform post processing

i) Deformed shape

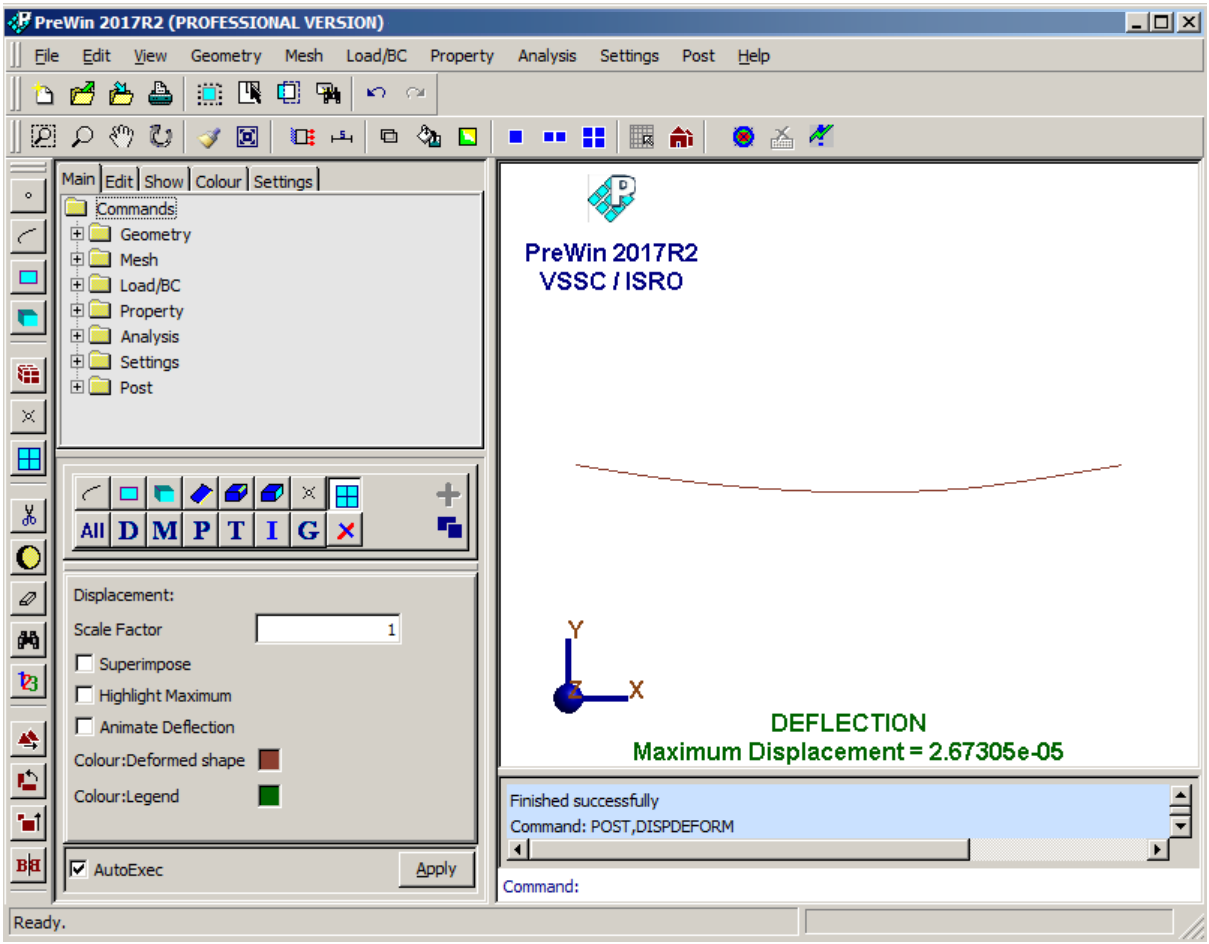
Command : POST, DISPDEFORM

Menu : Post → Deformed shape

Parameters :

Scale factor	1

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



ii) Stress contour

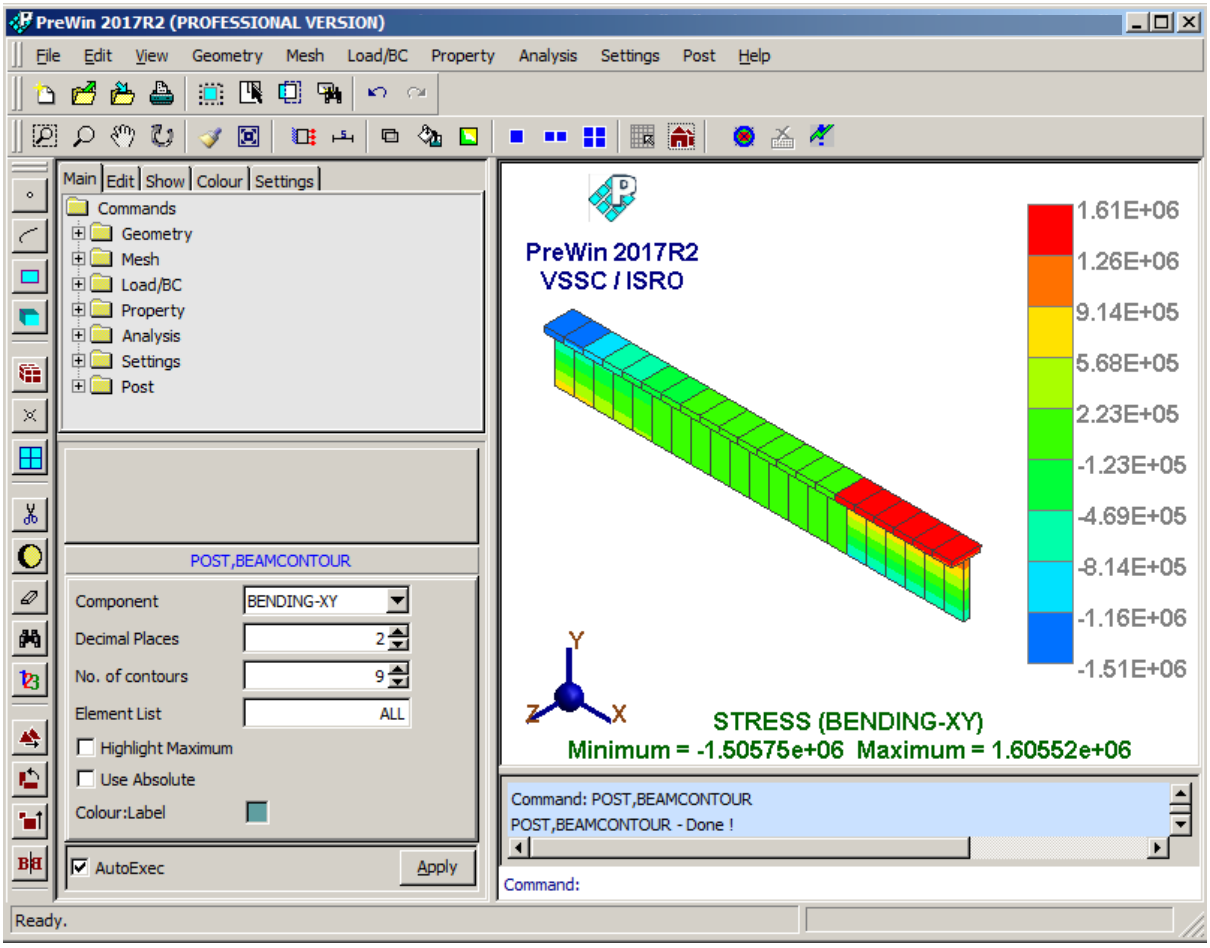
Command : POST, BEAMCONTOUR

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

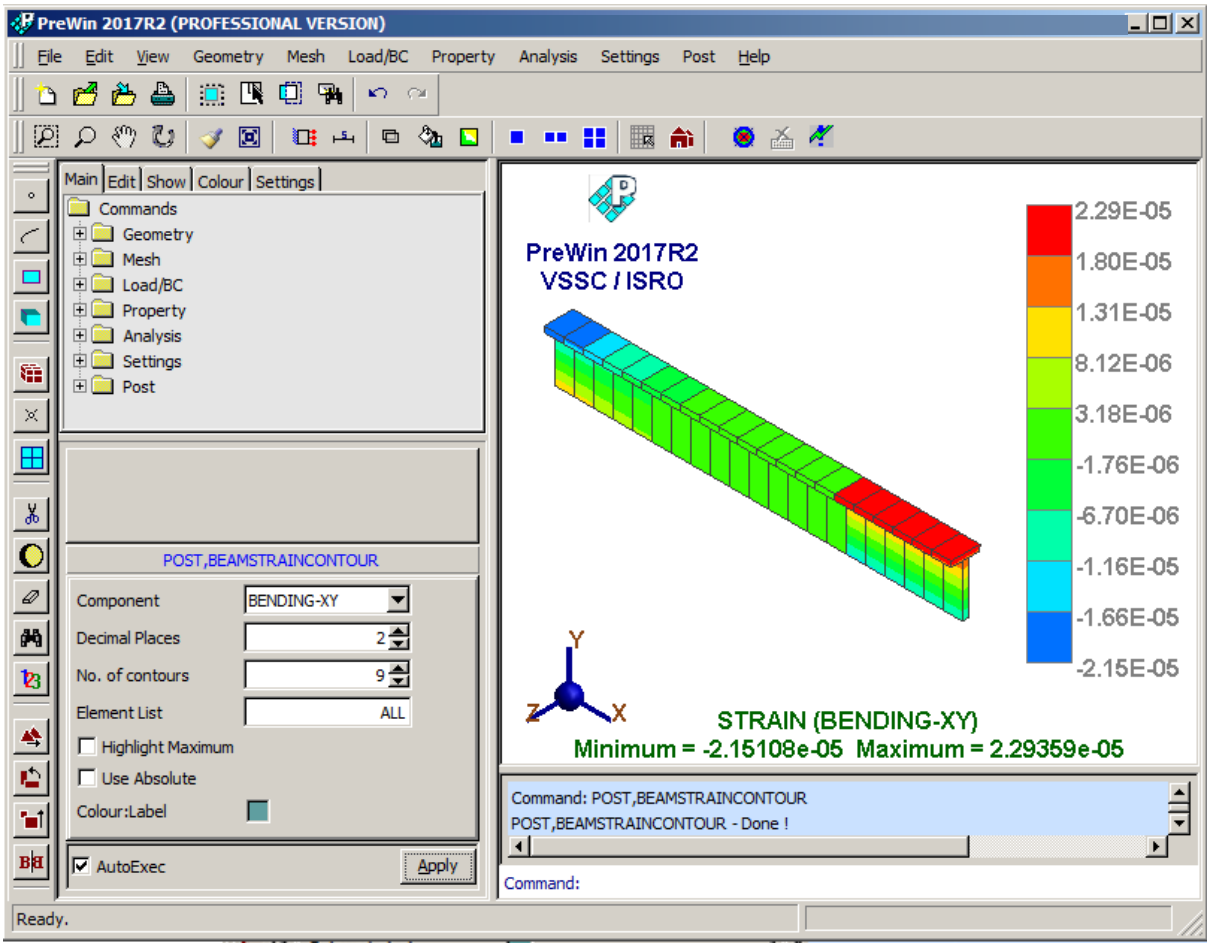
Command : POST, BEAMSTRAINCONTOUR

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

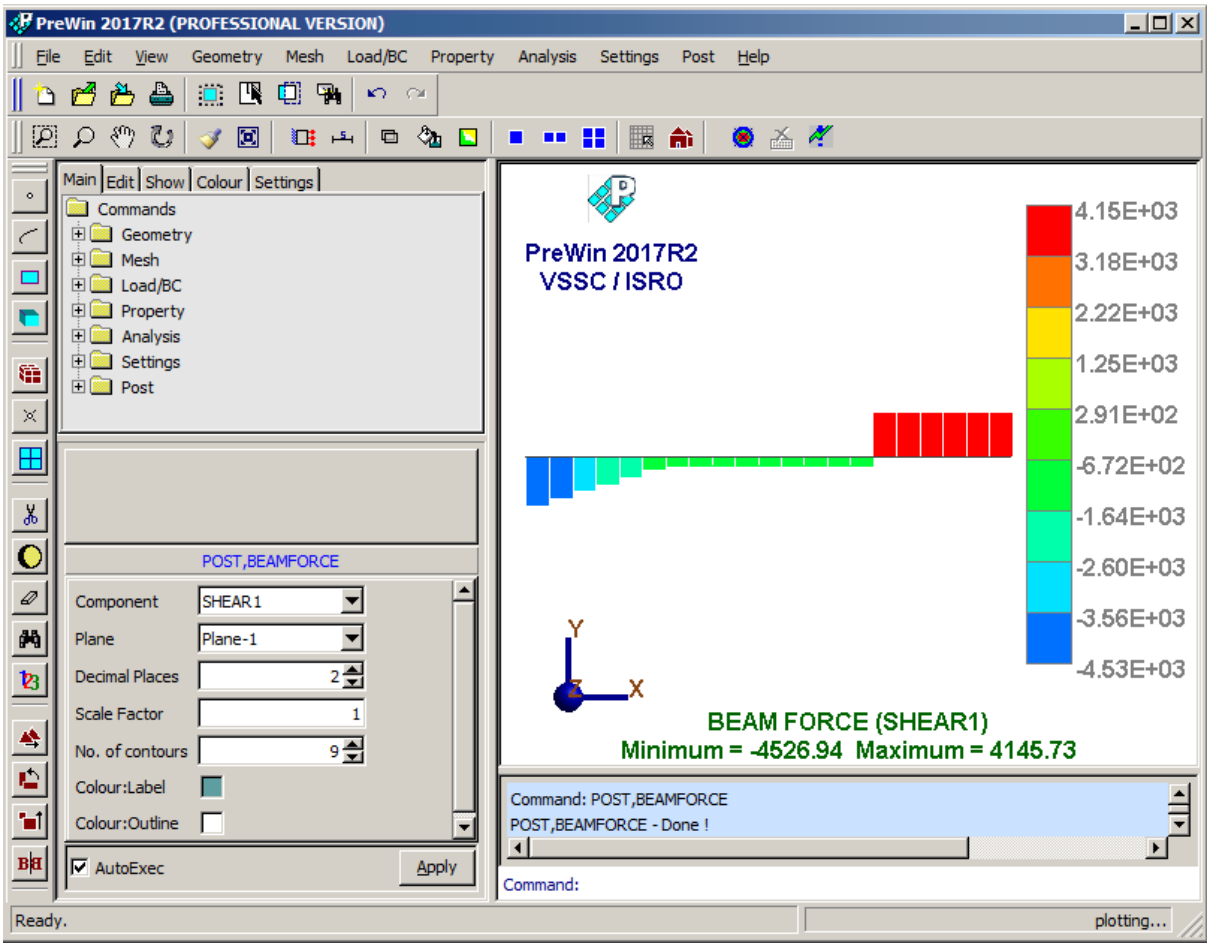
Command : POST, BEAMFORCE

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

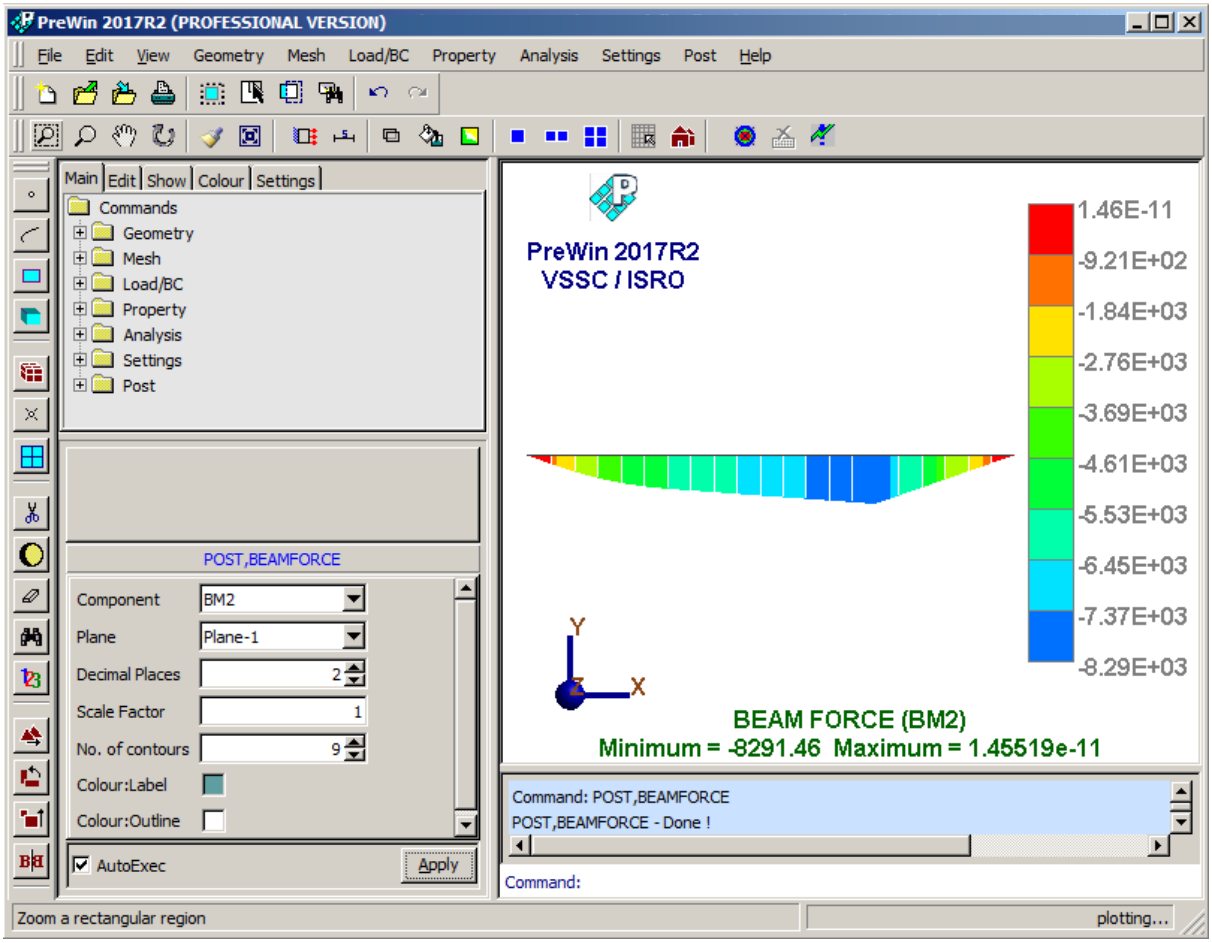
Command : POST, BEAMFORCE

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.