STATIC ANALYSIS OF A HANGING PLATE

Find the nodal displacement, stress in each material and reaction force at the support for the plate shown below. Consider the self weight of the plate in addition to the load $P = 444.82\ N$. (Material property: E = $206.842\ GPa$, $\nu = 0.3$, $\rho = 7850\ kg/m^3$)

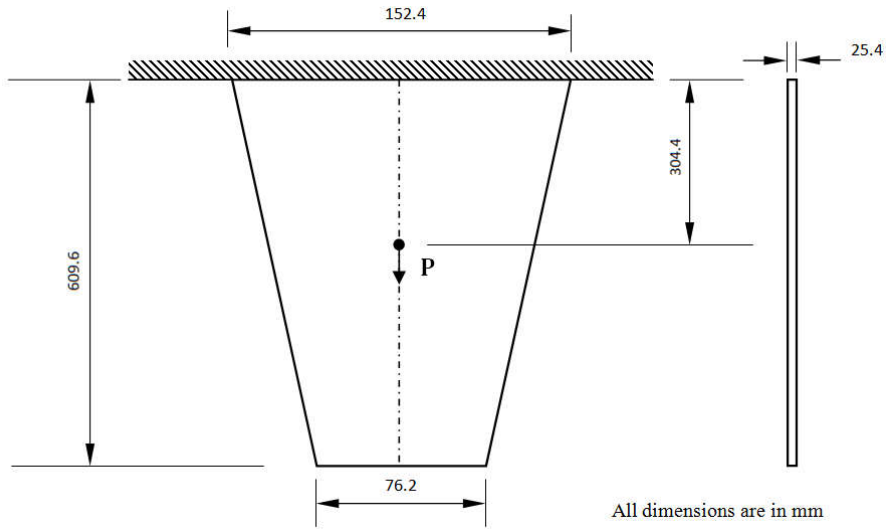


Figure 1

For solving the above problem using beam elements, finite element model with two elements can be considered. This will result in a finite element model as shown in figure 2.

From figure 1 the area at midpoint is $2903.22\ mm^2$, area at top is $3870.96\ mm^2$ and area at bottom is $1935.48\ mm^2$. Consequently the average area of element 1 in Figure 2 is A_1 = (387096+2903.22)/2 = $3387.09\ mm^2$. Hence the width of the first element will become (3387.09/25.4) = $133.35\ mm$. Similarly the width of the second element will be equal to $95.25mm$.

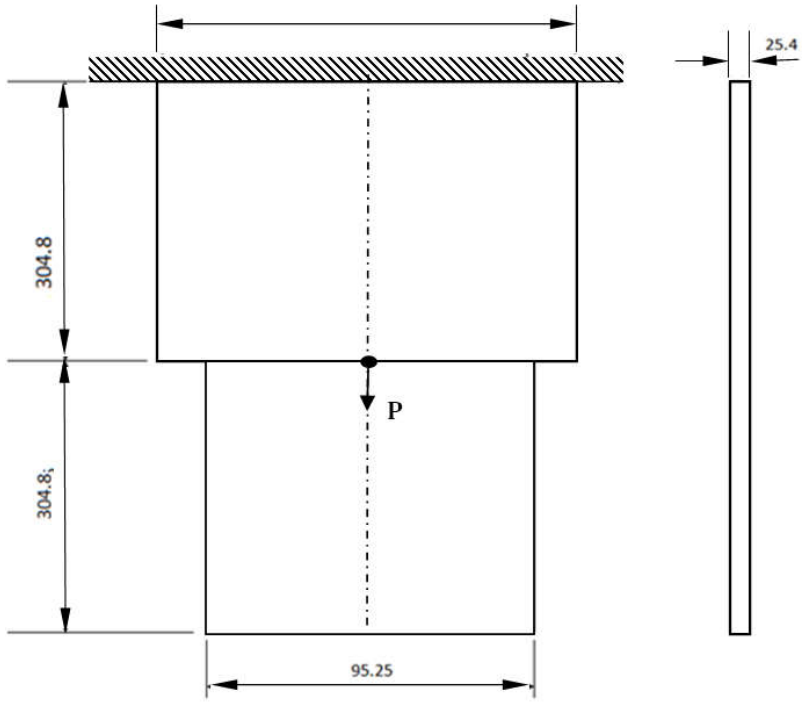


Figure 2

REFERENCE: Tirupathi R. Chandraupatla, Ashok D. Belegundu, Finite elements in engineering, Prentice-Hall of India, 1997, New Delhi.

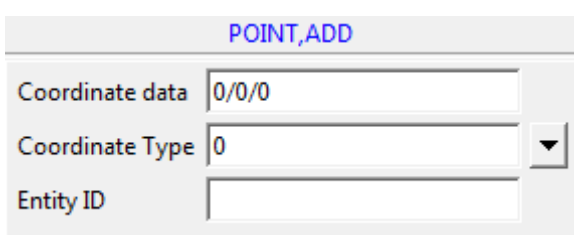
PROCEDURE

1. **Create key points**

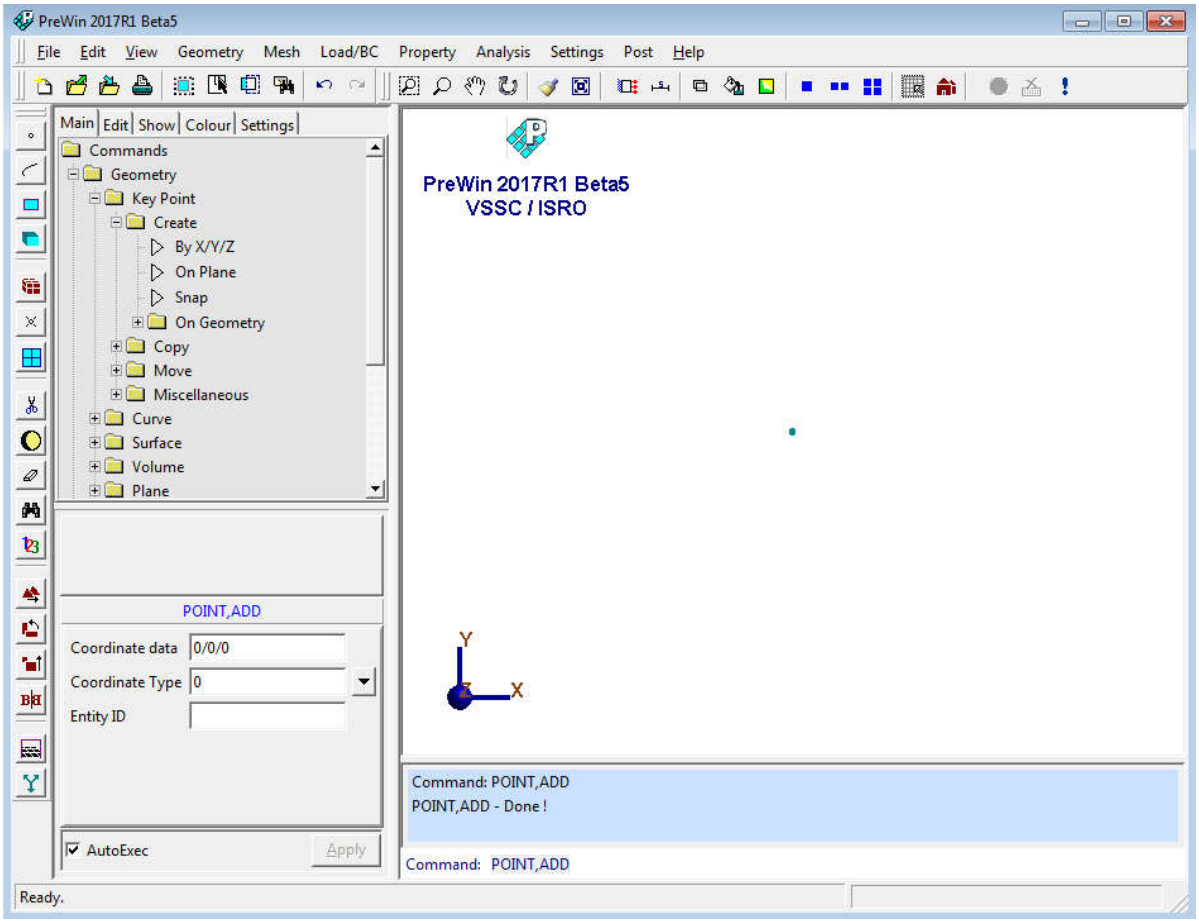
Command : POINT,ADD

Menu : Geometry → Keypoint → Create → By X/Y/Z

Parameters :



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



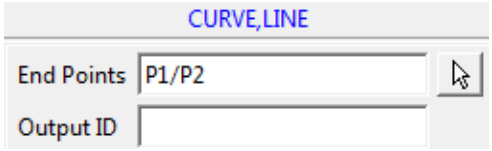
Similarly create key point at (0/609.6/0)

2. **Create curve**

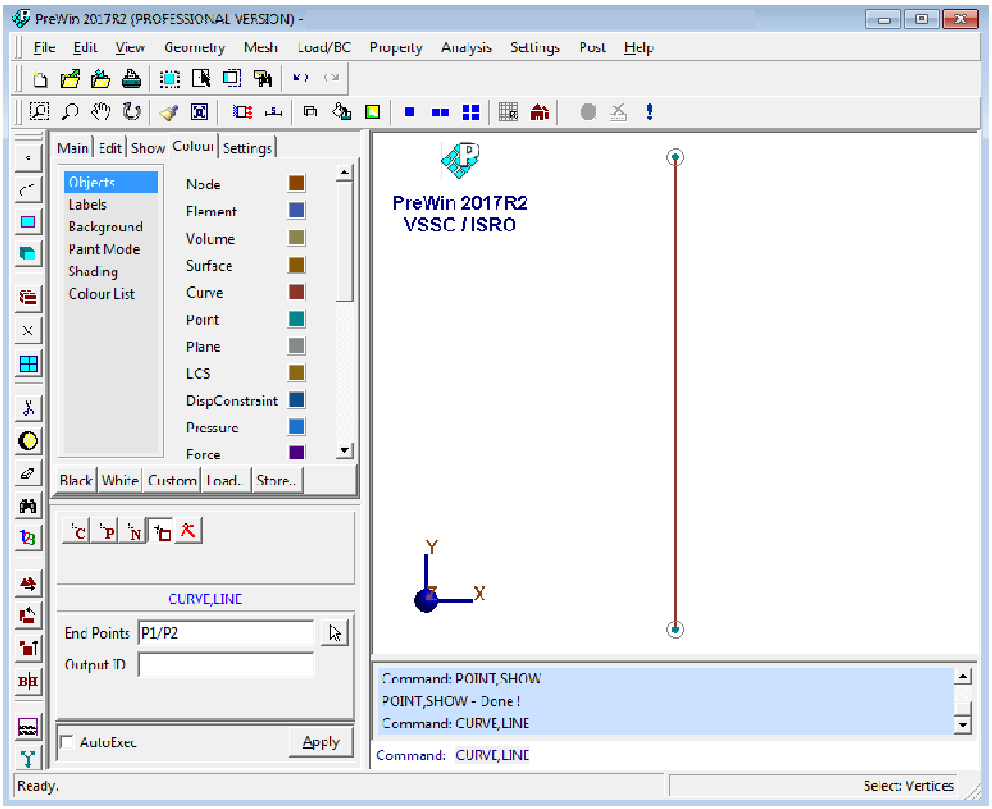
Command : CURVE,LINE

Menu : Geometry → Curve → Create → Line

Parameters :



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

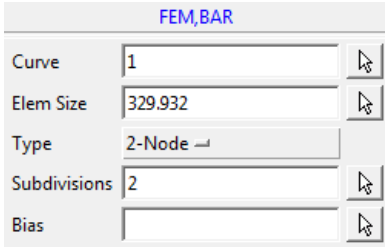


3. **Generate mesh**

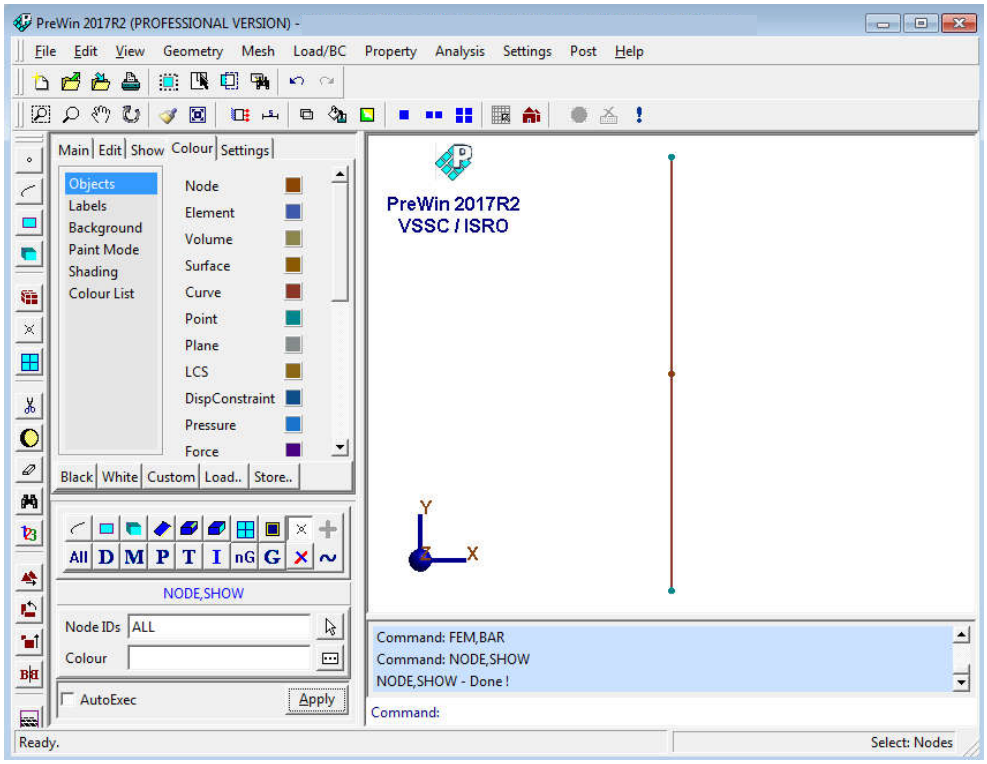
Command : FEM,BAR

Menu : Mesh → FE Mesh → BAR

Parameters :



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

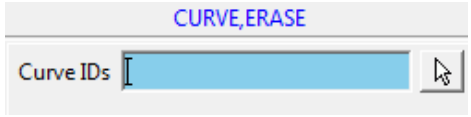


4. **Erase curve**

Command : CURVE,ERASE

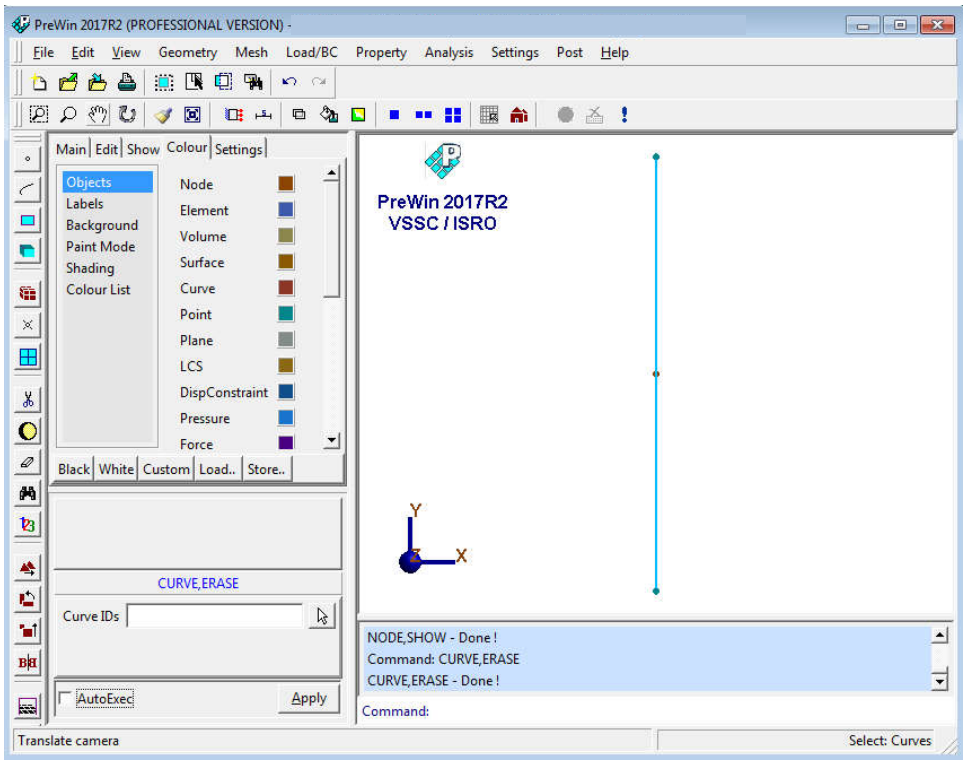
Menu : Geometry → Curve → Miscellaneous → Erase

Parameters :



Type in the curve ID or pick the curve after clicking the arrow in the surface ID box

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

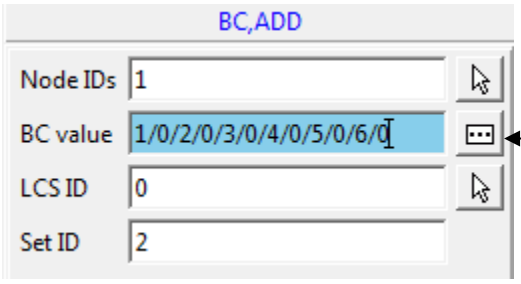


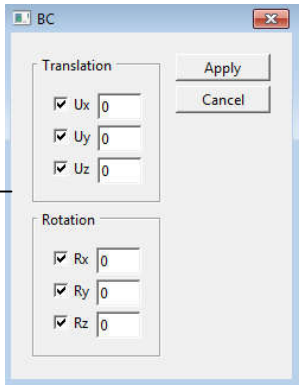
5. Apply boundary condition

Command : BC,ADD

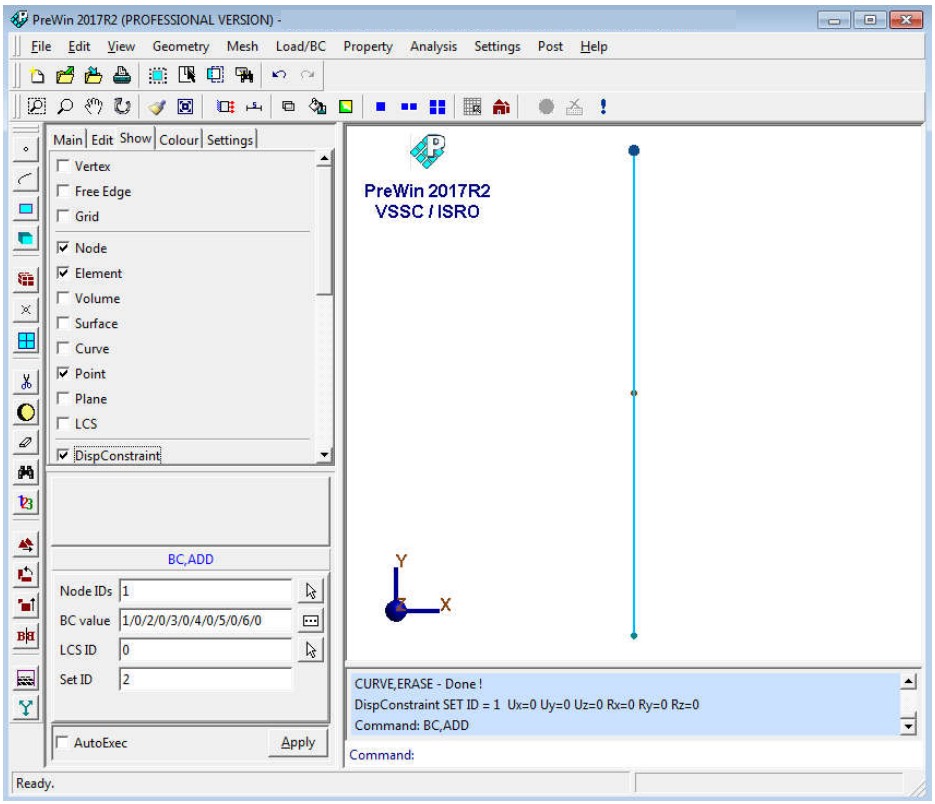
Menu : Load/BC → Displacement BC → Add

Parameters :





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

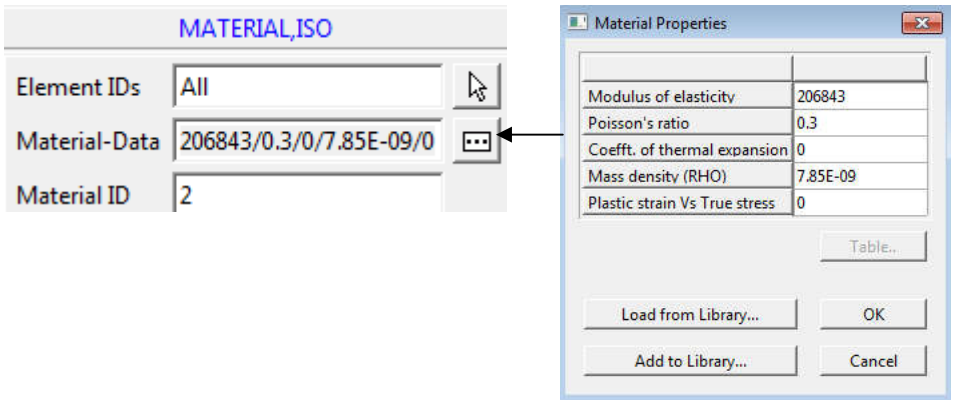


6. Specify material property

Command : MATERIAL,ISO

Menu : Property → Material → Isotropic → Add

Parameters :

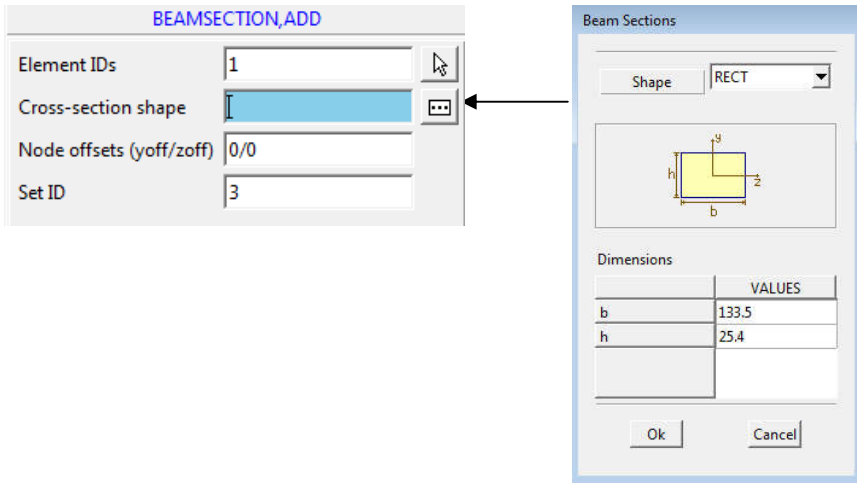


7. Specify beam section

Command : BEAMSECTION,ADD

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

Parameters :



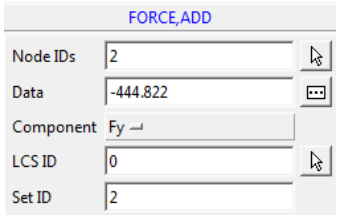
Similarly specify beam property for the second element with b = 95.25 mm and h = 25.4 mm

8. Specify load

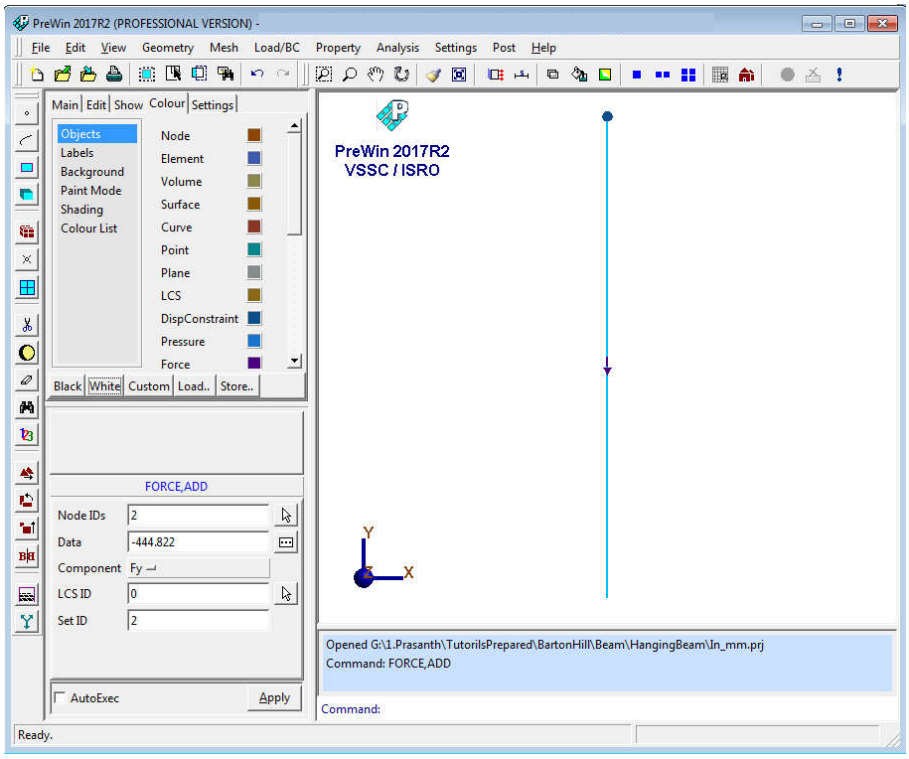
Command : FORCE,ADD

Menu : Load/BC → Point Load → Add

Parameters :



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



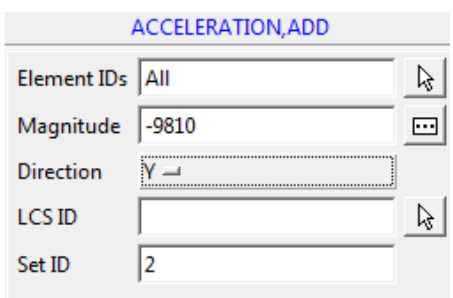
9. Specify self weight

In order to specify the body force an acceleration of '1g' is applied to the model.

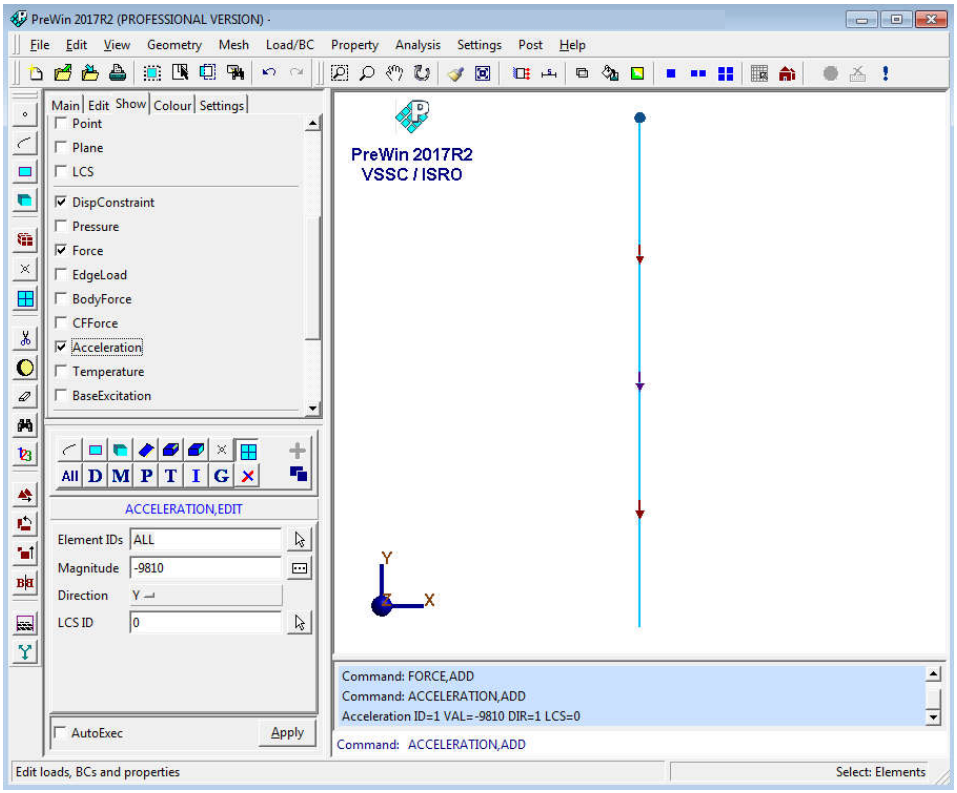
Command : ACCELERATION,ADD

Menu : Load/BC → Acceleration → Add

Parameter :



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

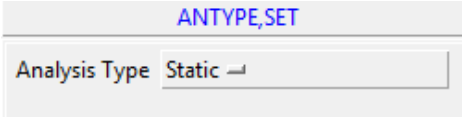


10. Set analysis type

Command : ANTYPE,SET

Menu : Analysis → Analysis Type

Parameter :

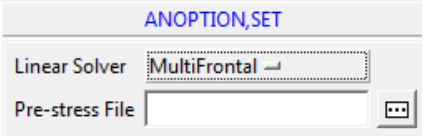


11. Set analysis option

Command : ANOPTION,SET

Menu : Analysis → Analysis Option

Parameters :

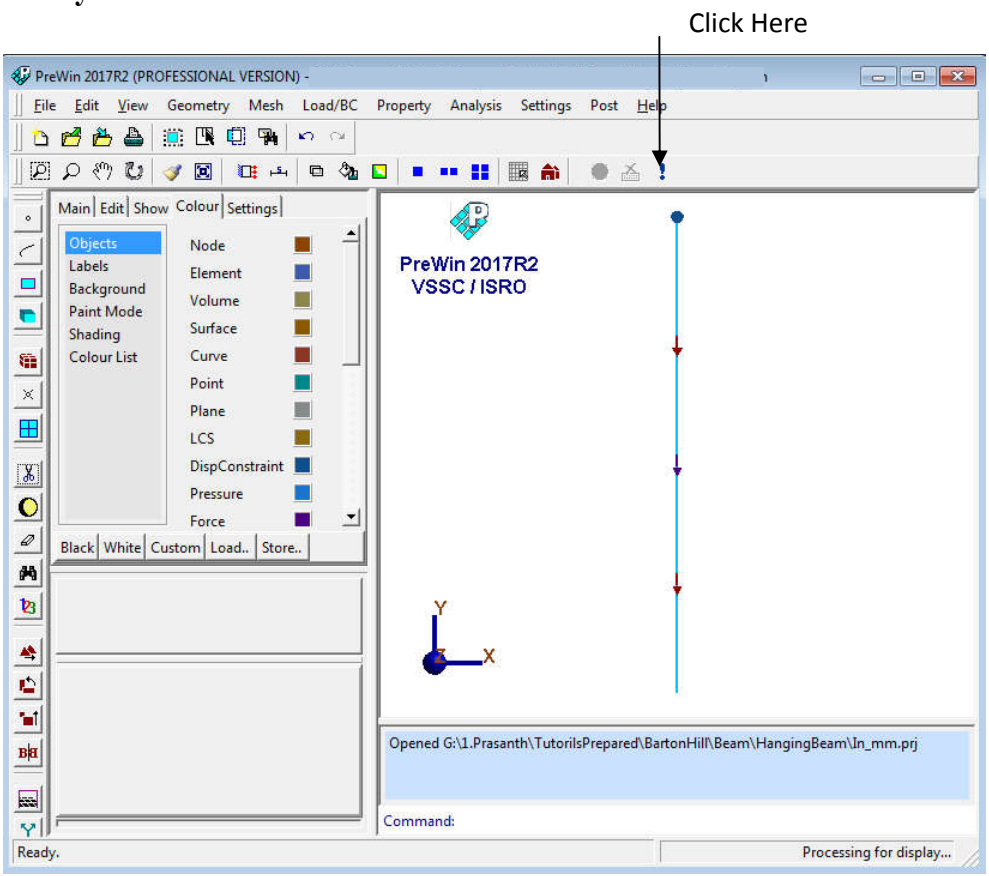


12. Save the project model

Menu : File → Save

13. Submit the job in to FEAST

Menu**: Analysis** → Run Solver



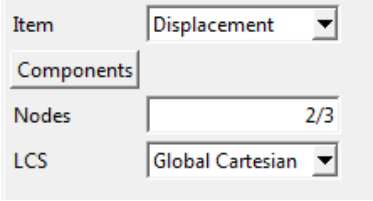
14. Perform Post Processing

i) Displacement

Command : POST,VIEWRESULTS

Menu : Post → View Results

Parameters :



Displacement

	Node ID	TX	TY	TZ	RX	RY	RZ	T-RES	R-RES
1	2	0	-0.000235523	0	0	0	0	0.000235523	0
2	3	0	-0.000252817	0	0	0	0	0.000252817	0

Copy Close

ii) Stress

Command : POST,VIEWRESULTS

Menu : Post → View Results

Parameters :

Item: Stress
Components
Nodes: 2/3
LCS: Global Cartesian

Stress

	Node ID	SIGMA-X	SIGMA-Y	SIGMA-Z	TAU-XY	TAU XZ	TAU YZ	PRINCIPAL 1	PRINCIPAL 2	PRINCIPAL 3
1	2	0.0857834	0	0	-0	0	0	0.0857834	0	0
2	3	0.0117361	-0	-0	-0	0	0	0.0117361	-0	-0

Copy Close

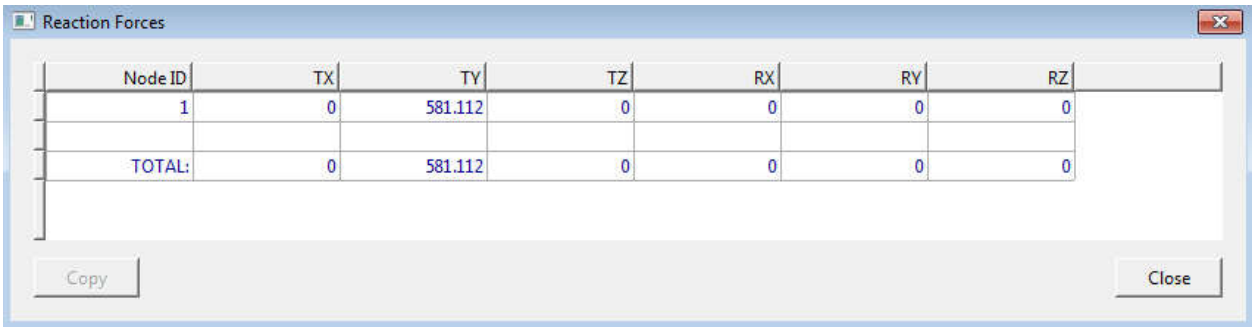
iii) Reaction force.

Command : POST,VIEWRESULTS

Menu : Post → View Results

Parameters :

Item: Reaction Forces
Components
Nodes: 1
LCS: Global Cartesian



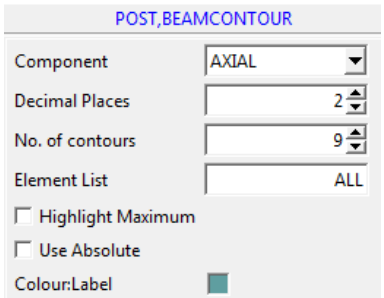
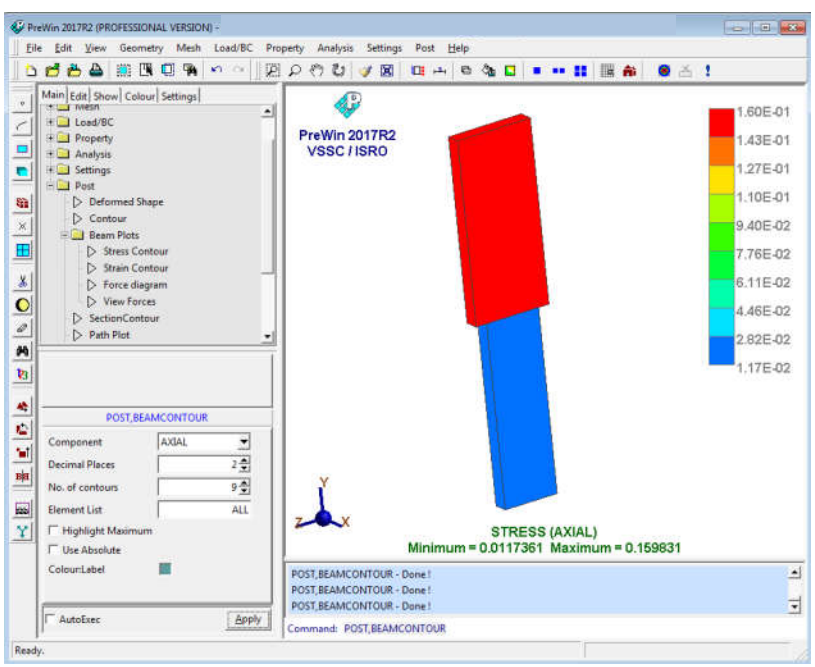
Node ID	TX	TY	TZ	RX	RY	RZ
1	0	581.112	0	0	0	0
TOTAL:	0	581.112	0	0	0	0

iv) Stress contour

Command : POST,BEAMCONTOUR

Menu : Post → Beam plots → Stress Contour

Parameters :



At the end of above operation your screen should look like this