FIXED BEAM SUBJECTED TO AXIAL LOAD

Find the nodal displacement, stress in each material and reaction force at the support for the bar shown in Figure 1. Consider the self weight of the plate in addition to the load $P = 200 \times 10^3$ N. (Material property, Material 1(Aluminium): $E_1 = 70$ GPa, $v_1 = 0.3$; Material 2(Steel): $E_2 = 200$ GPa, $v_2 = 0.3$)

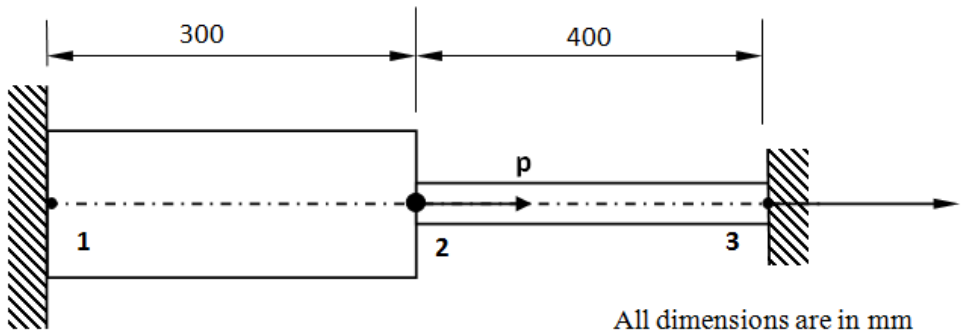


Figure 1

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

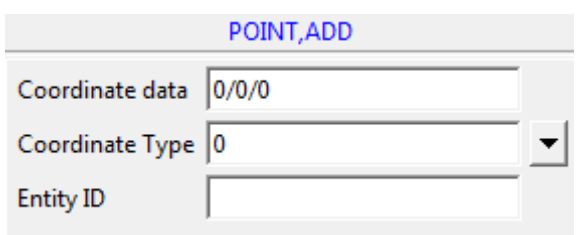
PROCEDURE

1. **Create key points**

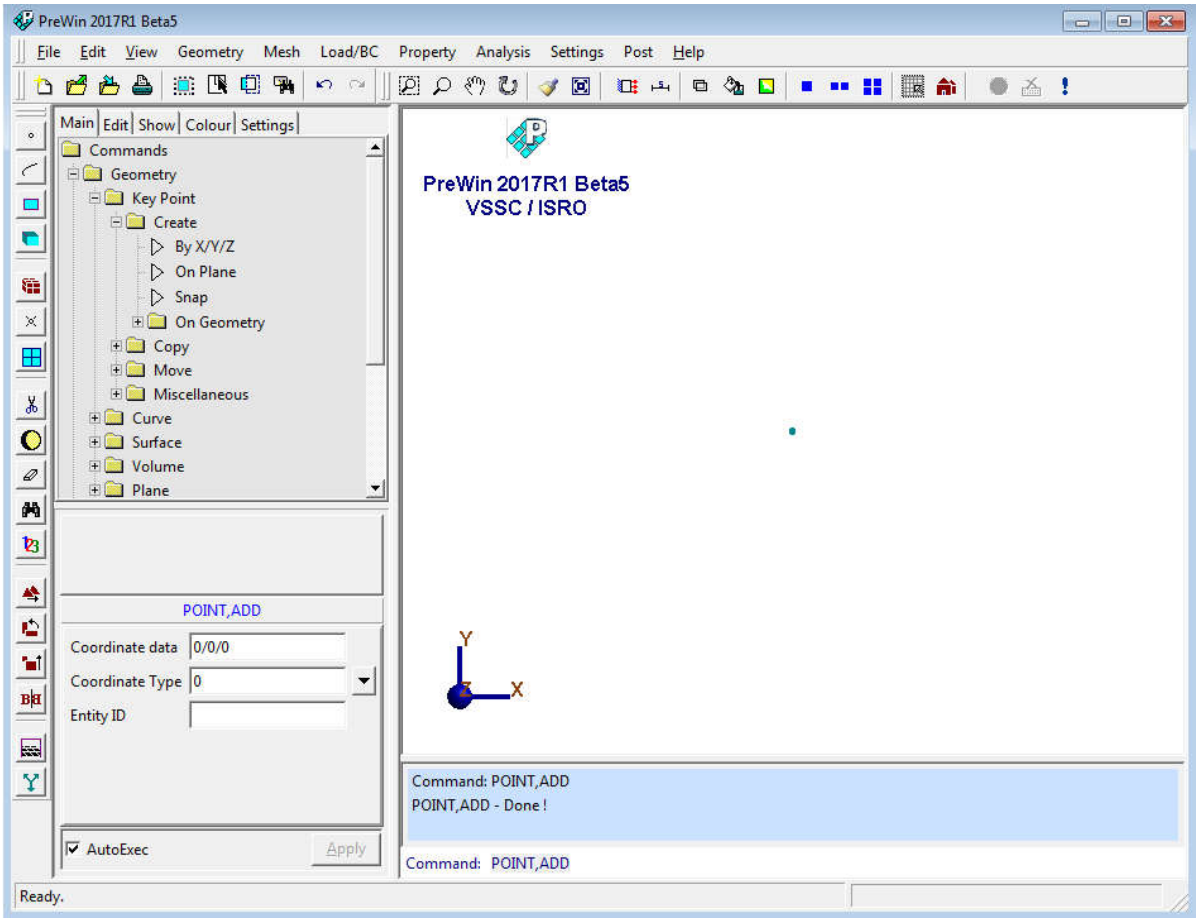
Command : POINT,ADD

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

Parameters :



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



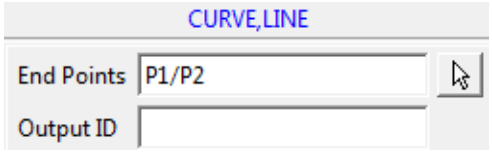
Similarly create key point at (300/0/0) & (700/0/0)

2. **Create curve**

Command : CURVE,LINE

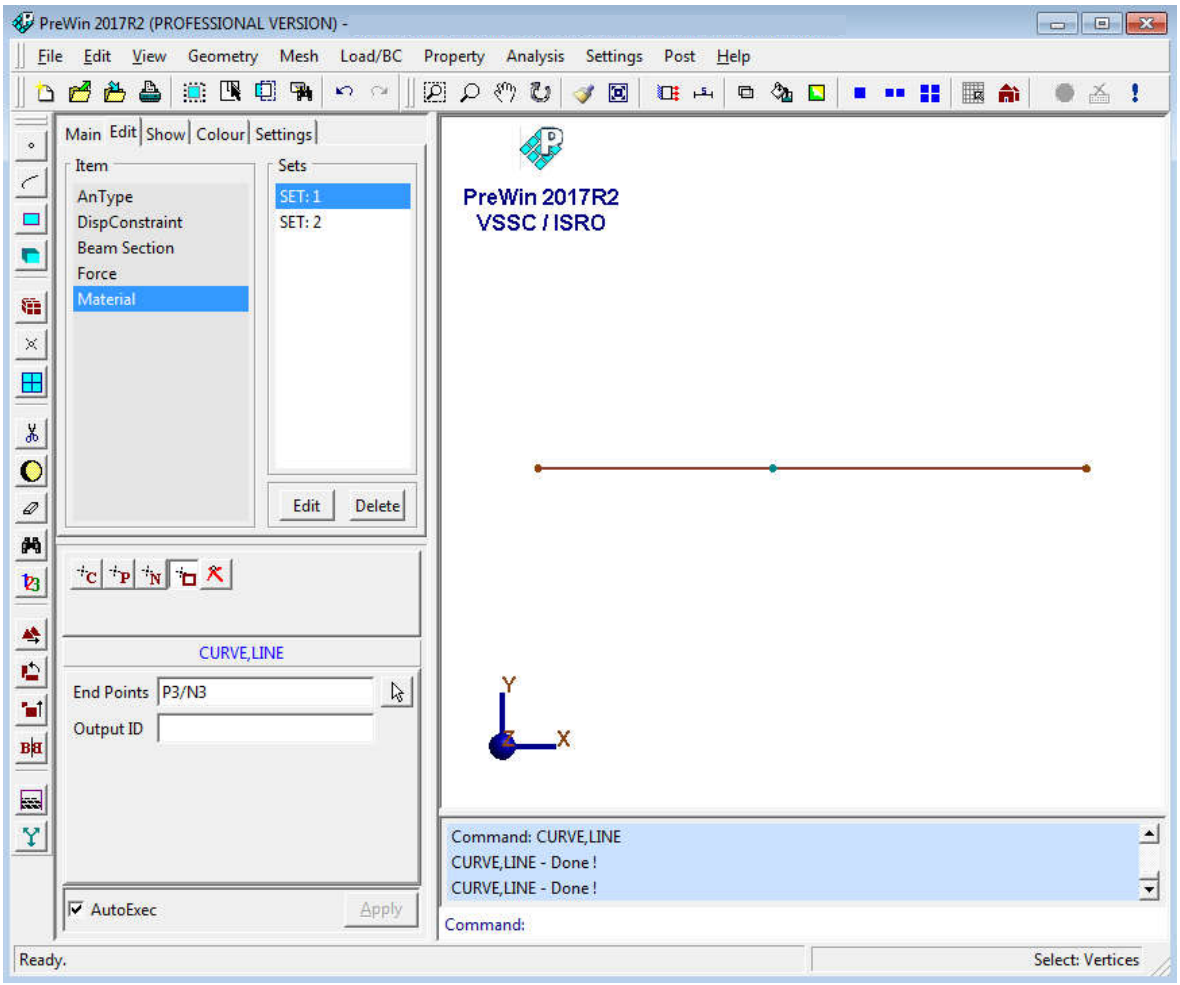
Menu : Geometry → Curve → Create → Line

Parameters :



Create line with key points (P2/P3) also.

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

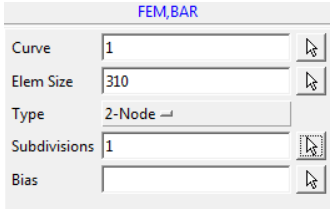


3. **Generate mesh**

Command : FEM,BAR

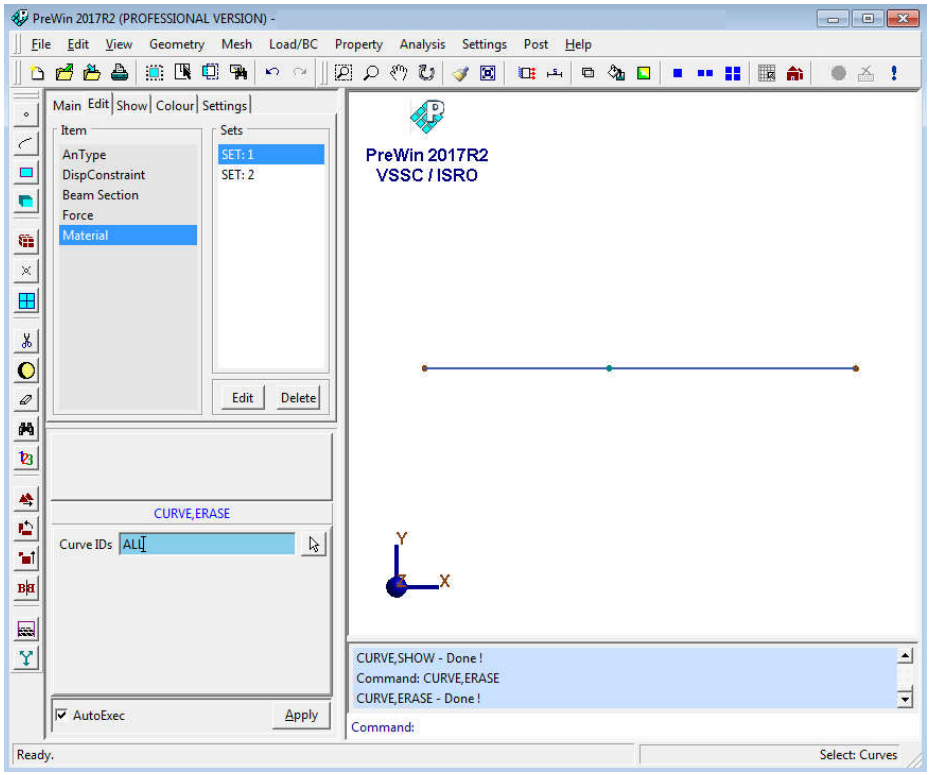
Menu : Mesh → FE Mesh → BAR

Parameters :



Create mesh on curve 2 also with only 1 subdivision

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

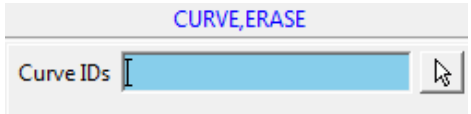


4. Erase curve

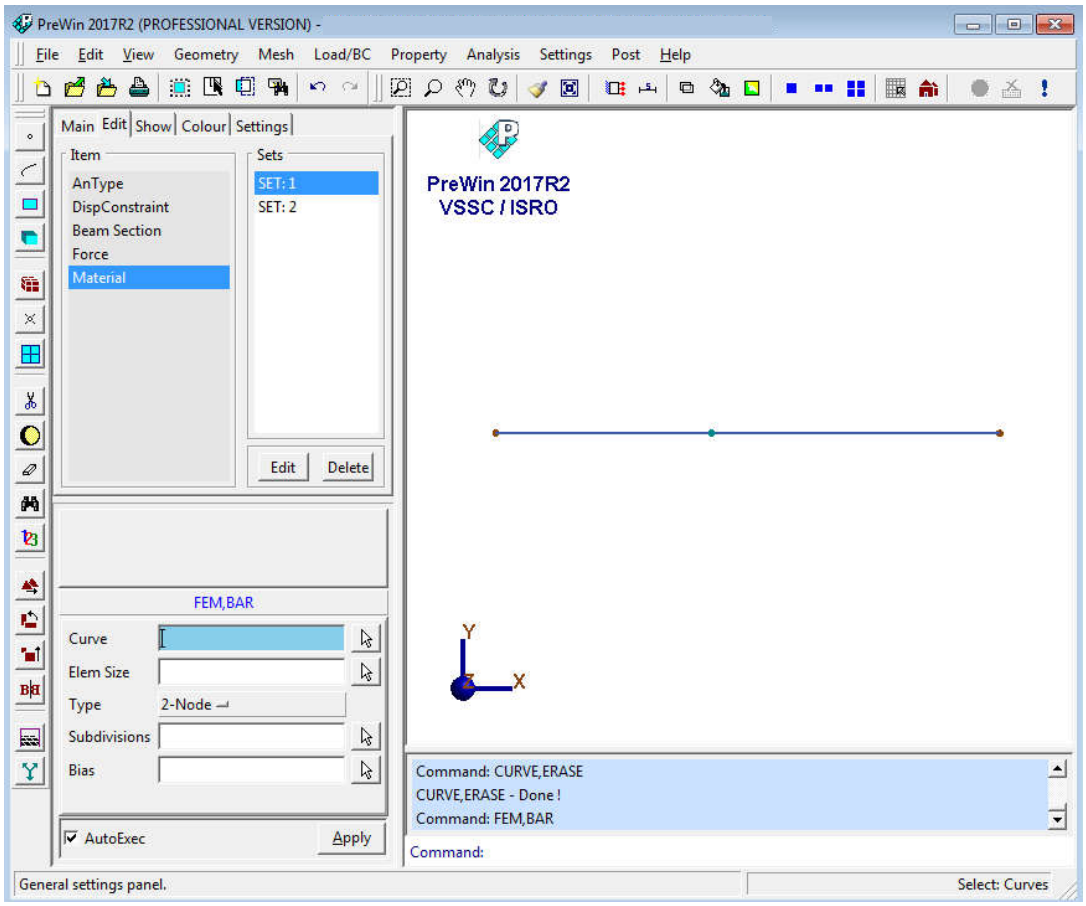
Command : CURVE,ERASE

Menu : Geometry → Curve → Miscellaneous → Erase

Parameters :



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



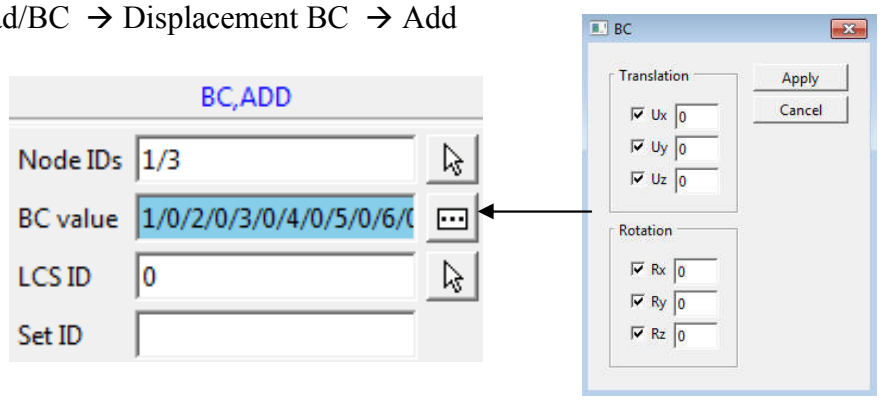
At the end of the operation/s your screen should look like this.

5. Apply boundary condition

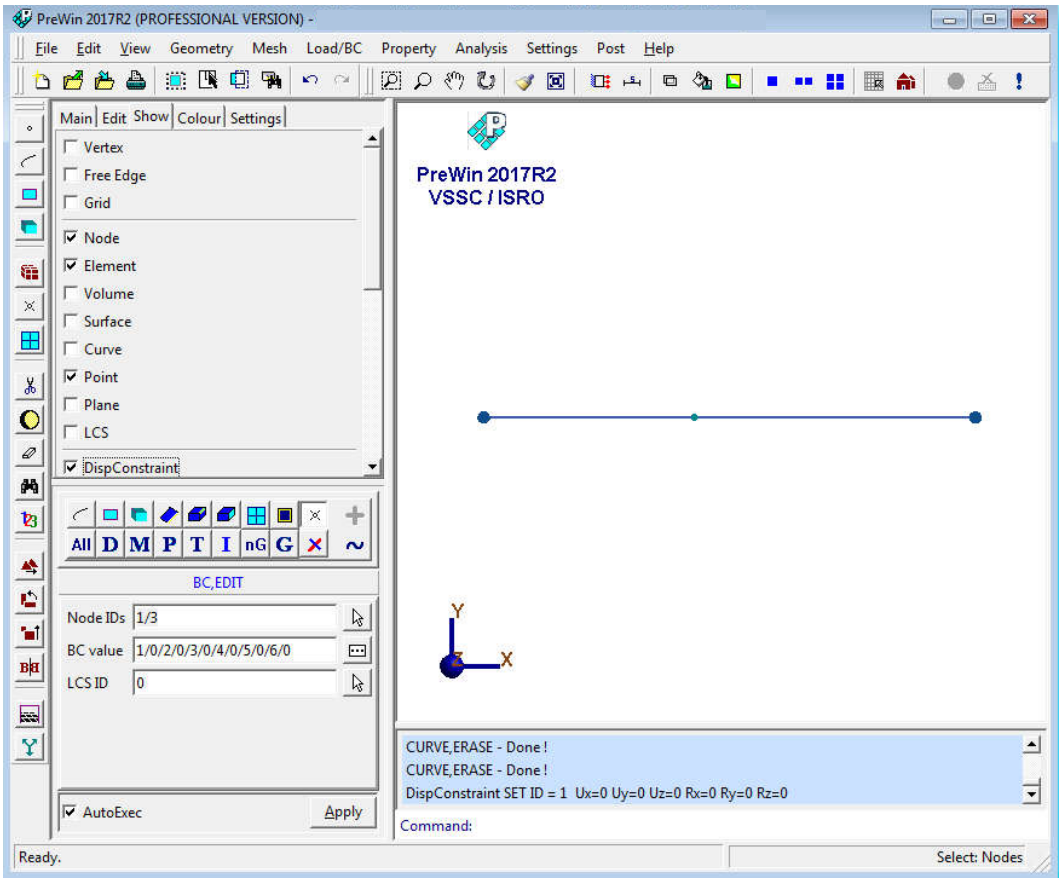
Command : BC,ADD

Menu : Load/BC → Displacement BC → Add

Parameters :



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

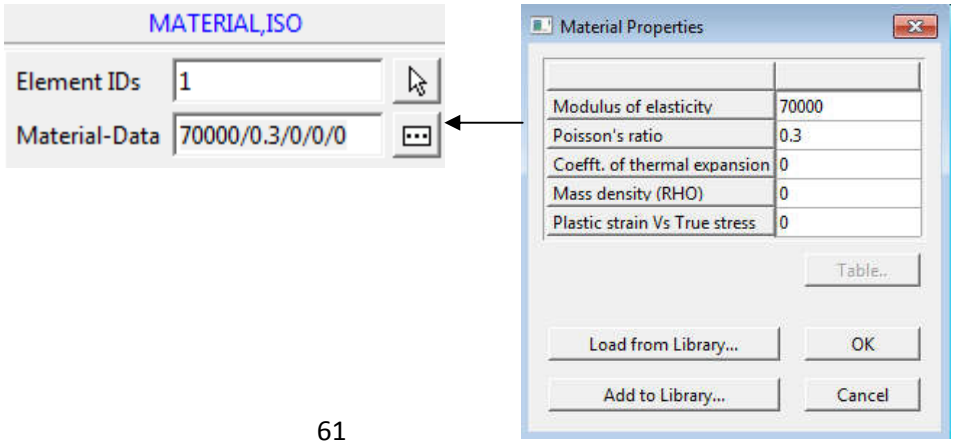


6. Specify material property

Command : MATERIAL,ISO

Menu : Property → Material → Isotropic → Add

Parameters :



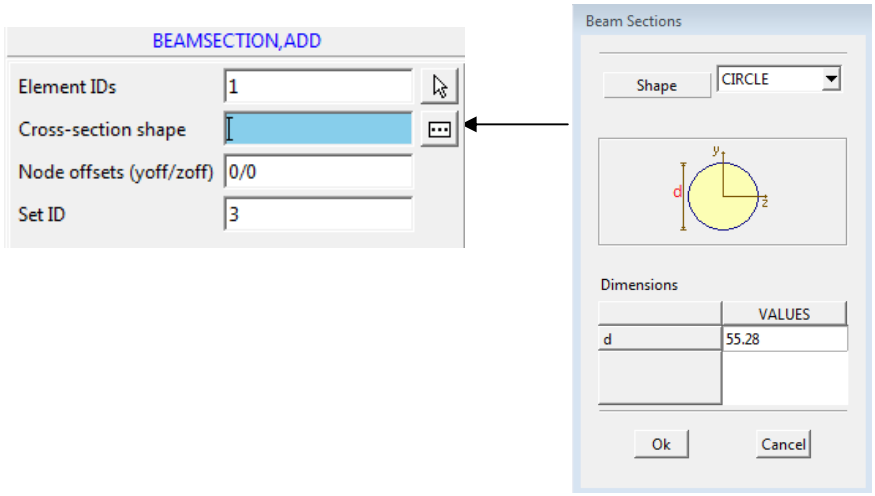
Similarly specify material property for the second element with input data as E = 200 GPa, ν = 0.3

7. Specify beam section

Command : BEAMSECTION,ADD

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

Parameters :



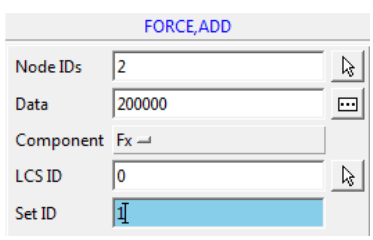
Similarly specify beam property for the second element with d = 27.64 mm

8. Specify load

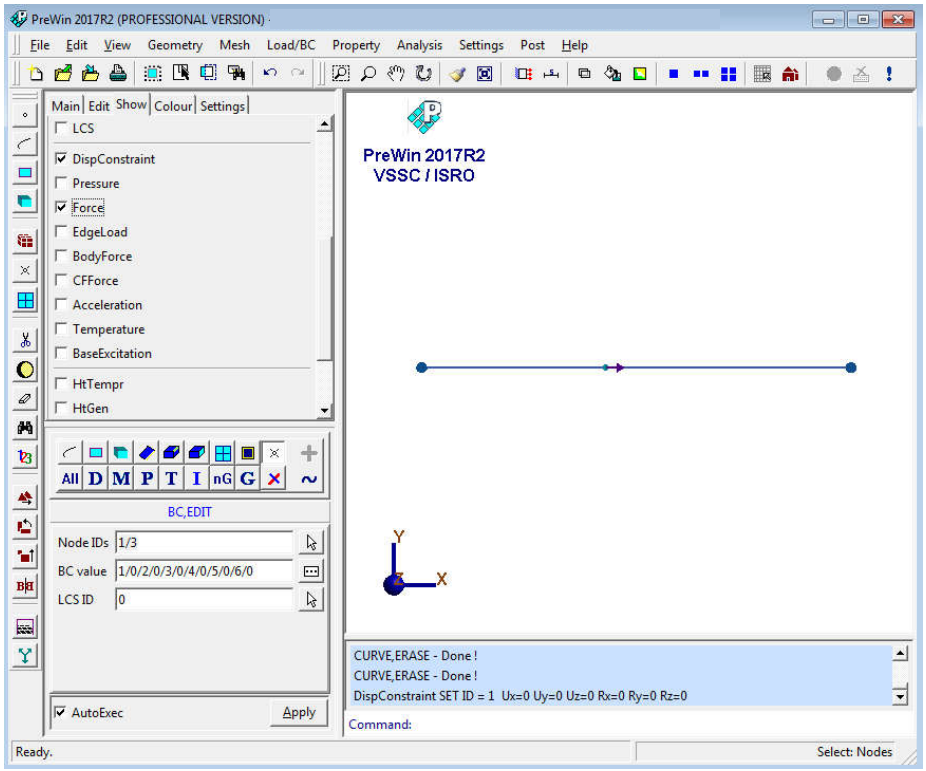
Command : FORCE,ADD

Menu : Load/BC → Point Load → Add

Parameters :



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

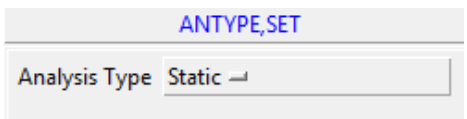


9. Set analysis type

Command : ANTYPE,SET

Menu : Analysis → Analysis Type

Parameter :

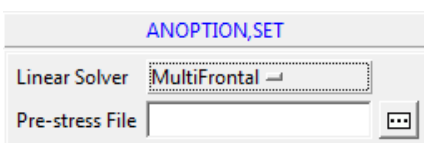


10. Set analysis option

Command : ANOPTION,SET

Menu : Analysis → Analysis Option

Parameters :



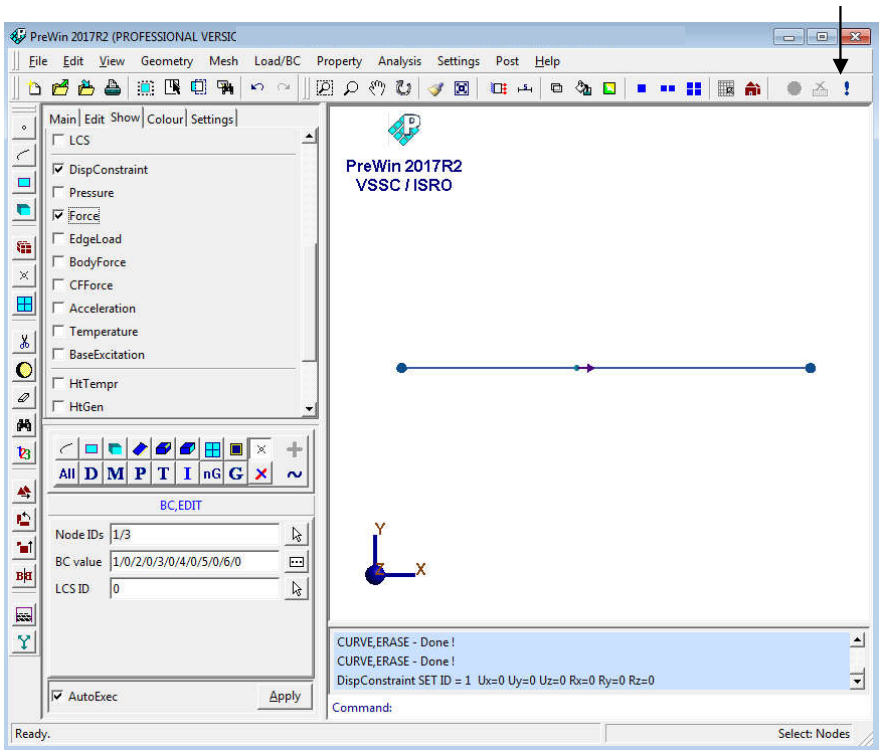
11. Save the project model

Menu : File → Save

12. Submit the job in to FEAST

Menu: **Analysis** → Run Solver

Click Here



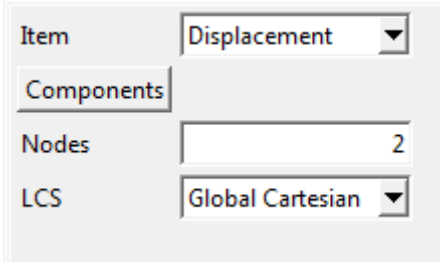
13. Perform Post Processing

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	0.23255	0	0	0	0	0	0.23255	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	1	54.2617	0	0	-0	0	0	54.2617	0	0
2	2	-31.0067	0	0	-0	0	0	0	0	-31.0067
3	3	-116.275	-0	-0	-0	0	0	-0	-0	-116.275

Copy Close

iii) Reaction force.

Command : POST,VIEWRESULTS

Menu : Post → View Results

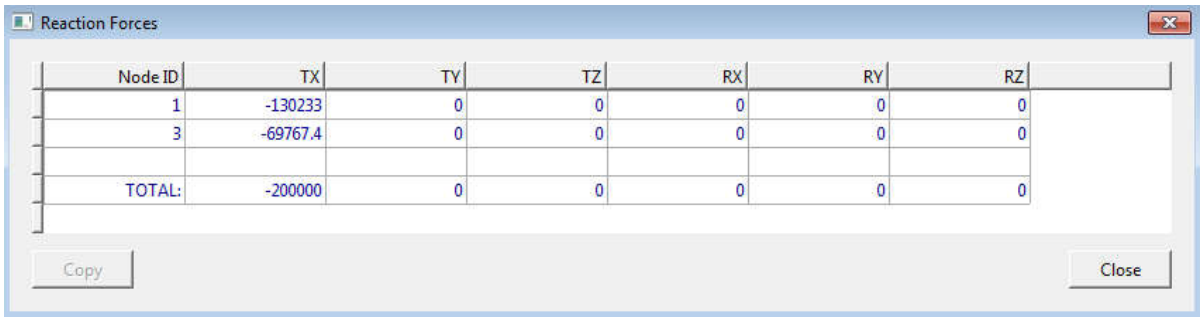
Parameters :

Item: Reaction Forces

Components

Nodes: ALL

LCS: Global Cartesian



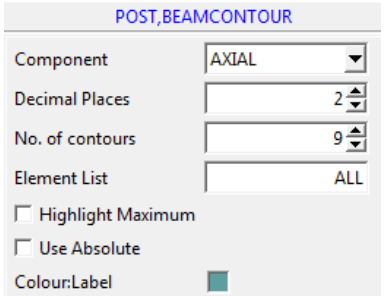
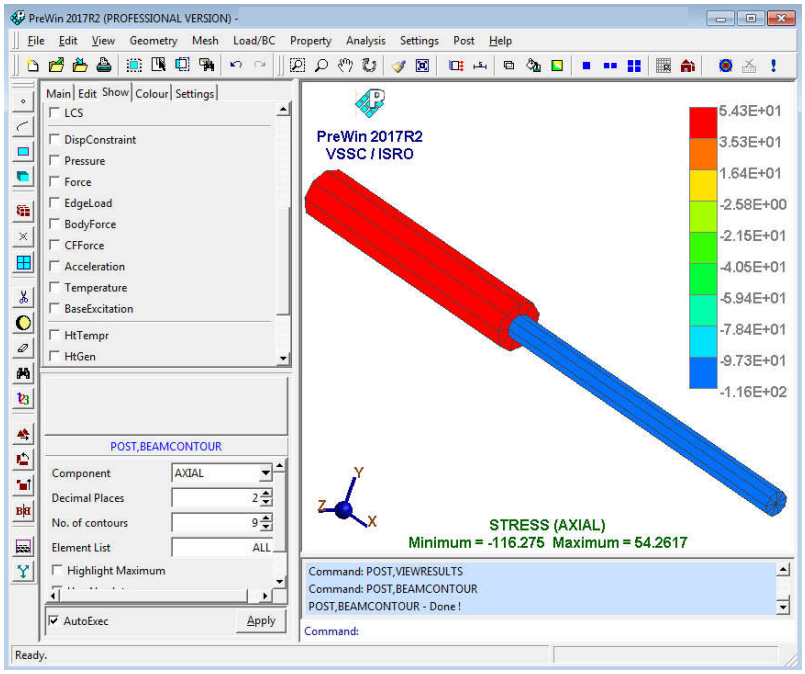
Node ID	TX	TY	TZ	RX	RY	RZ
1	-130233	0	0	0	0	0
3	-69767.4	0	0	0	0	0
TOTAL:	-200000	0	0	0	0	0

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