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 = $206.842\ GPa$, $\nu = 0.3$, $\rho = 7850\ kg/m^3$)

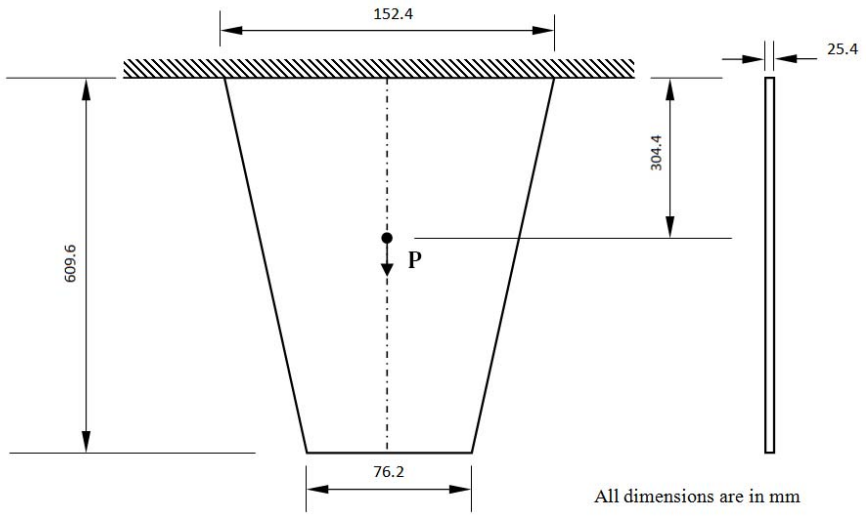


Figure 1

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 $95.25mm$.

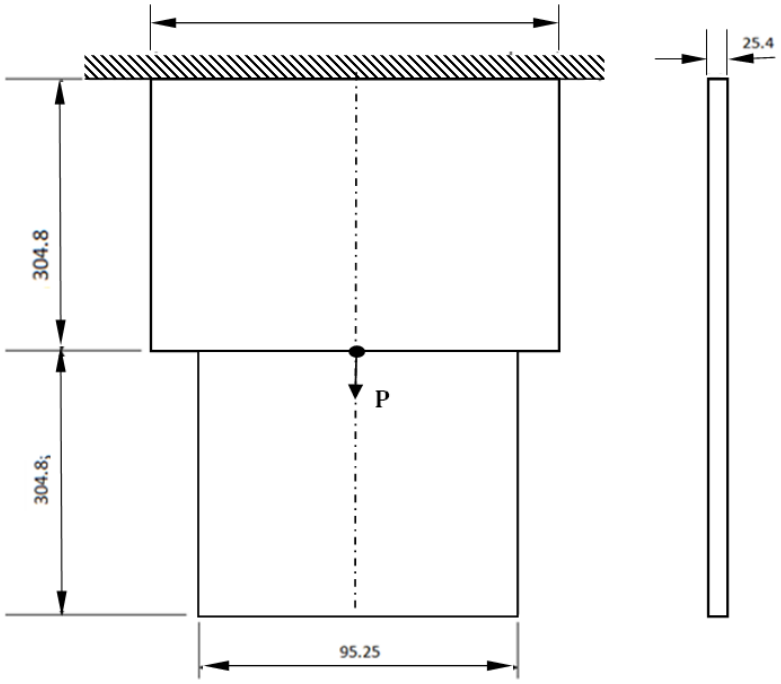


Figure 2

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

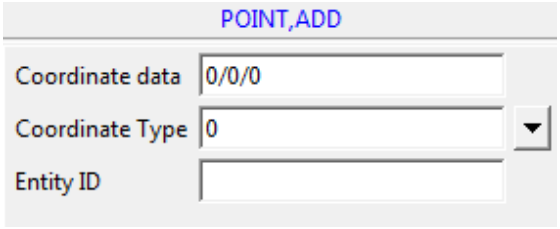
PROCEDURE

1. **Create key points**

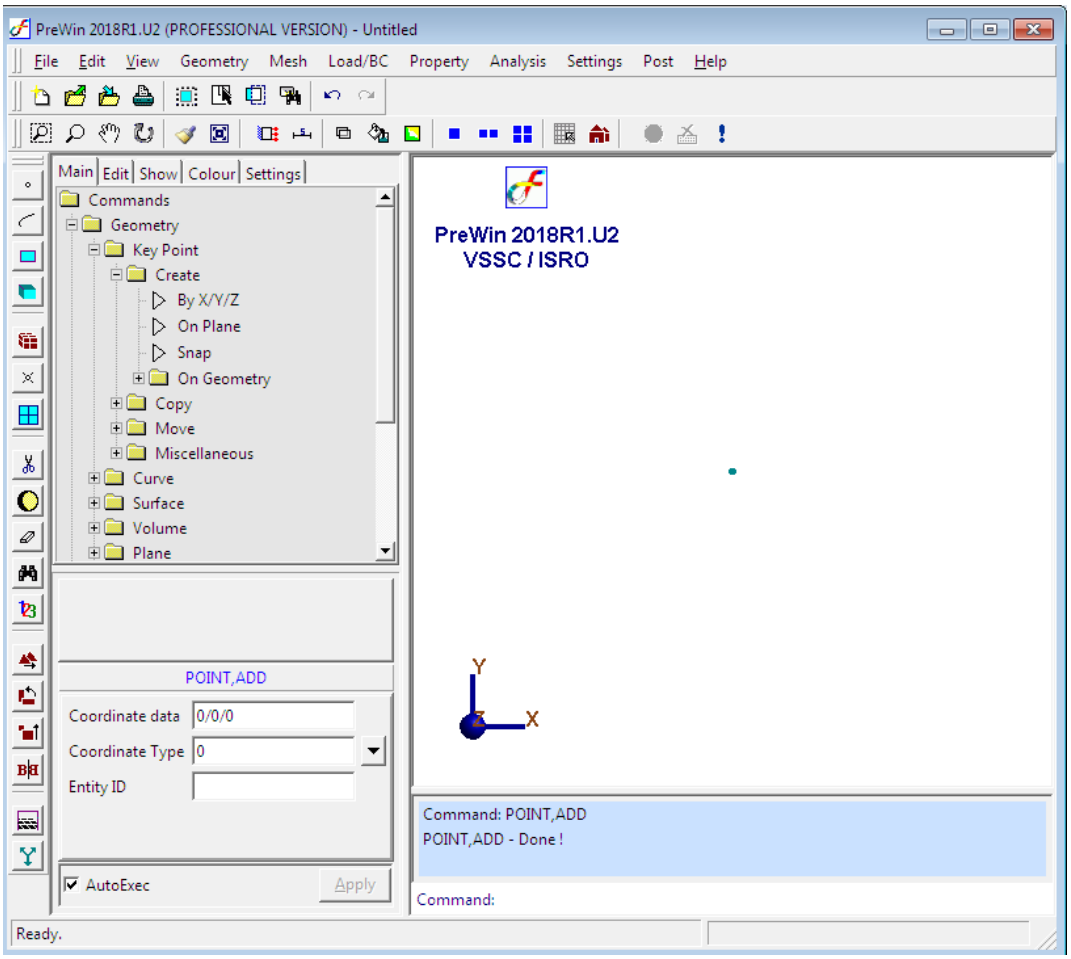
Command : POINT,ADD

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

Parameters :



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



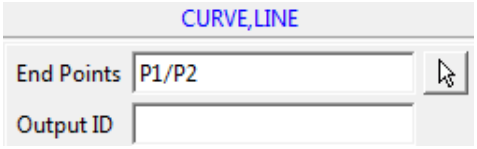
Similarly create key point at (0/609.6/0)

2. Create curve

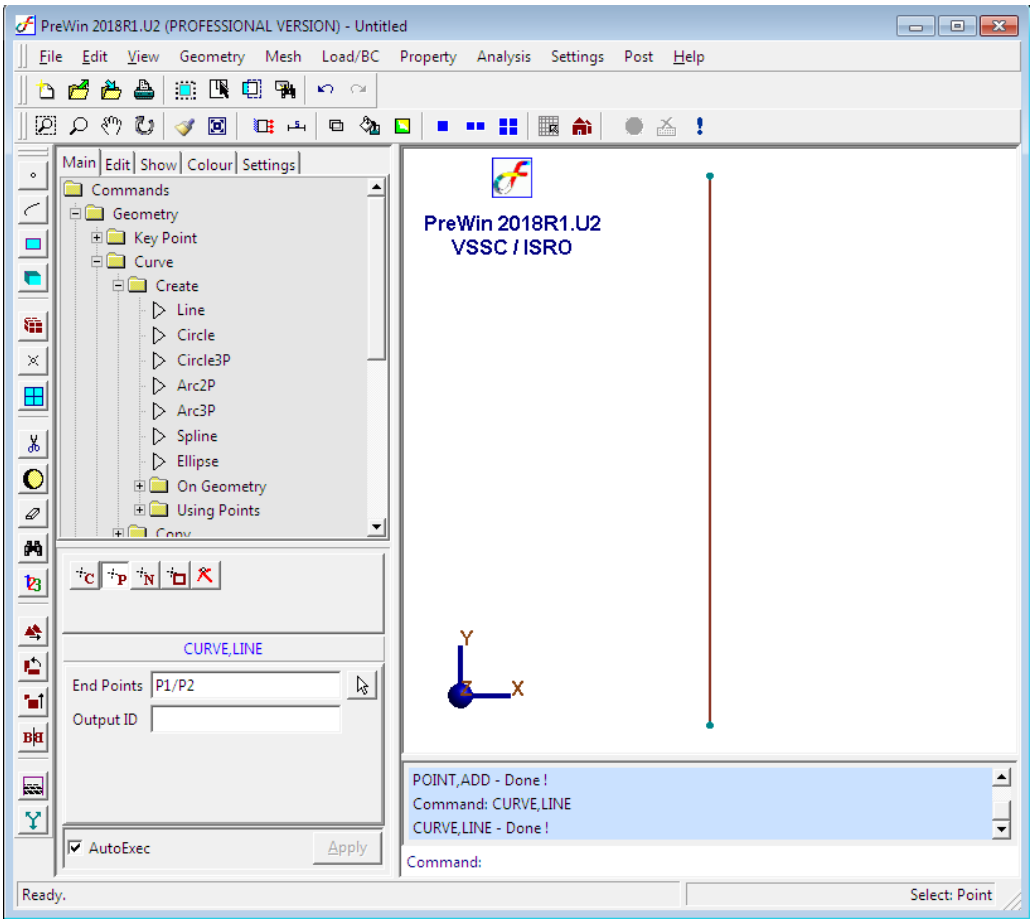
Command : CURVE,LINE

Menu : Geometry → Curve → Create → Line

Parameters :



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

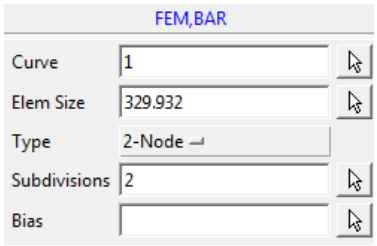


3. Generate mesh

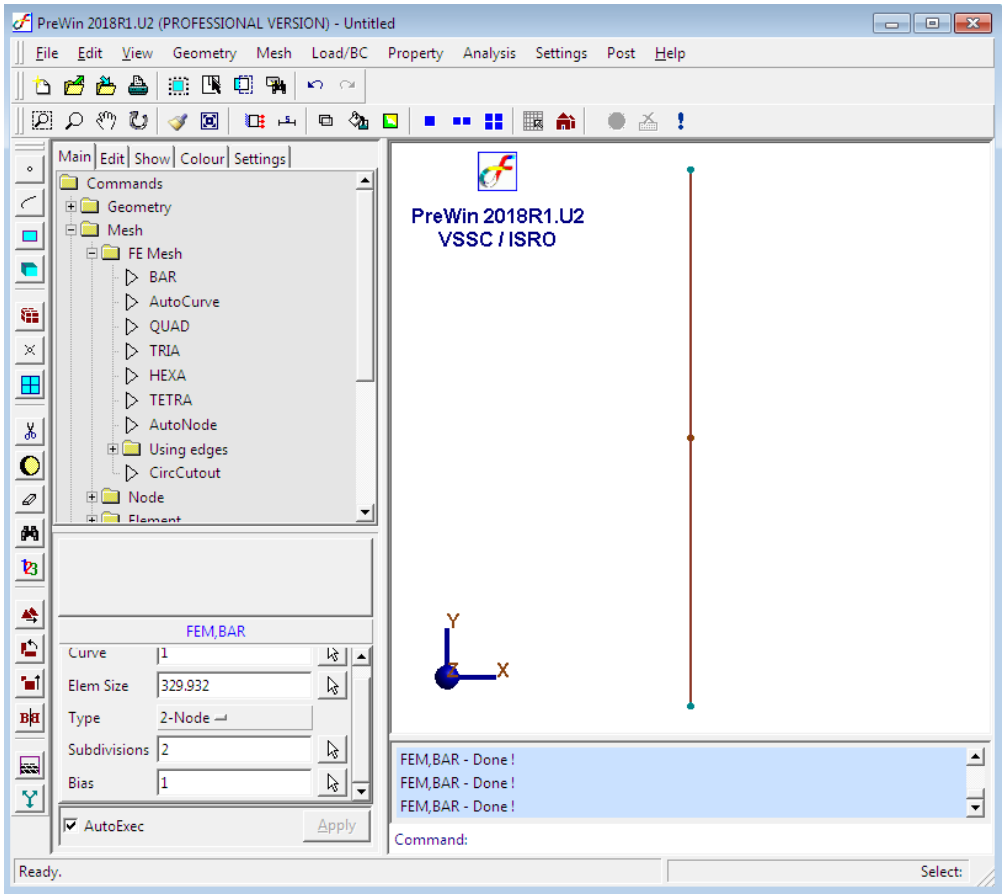
Command : FEM,BAR

Menu : Mesh → FE Mesh → BAR

Parameters :



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

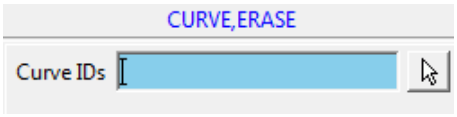


4. Erase curve

Command : CURVE,ERASE

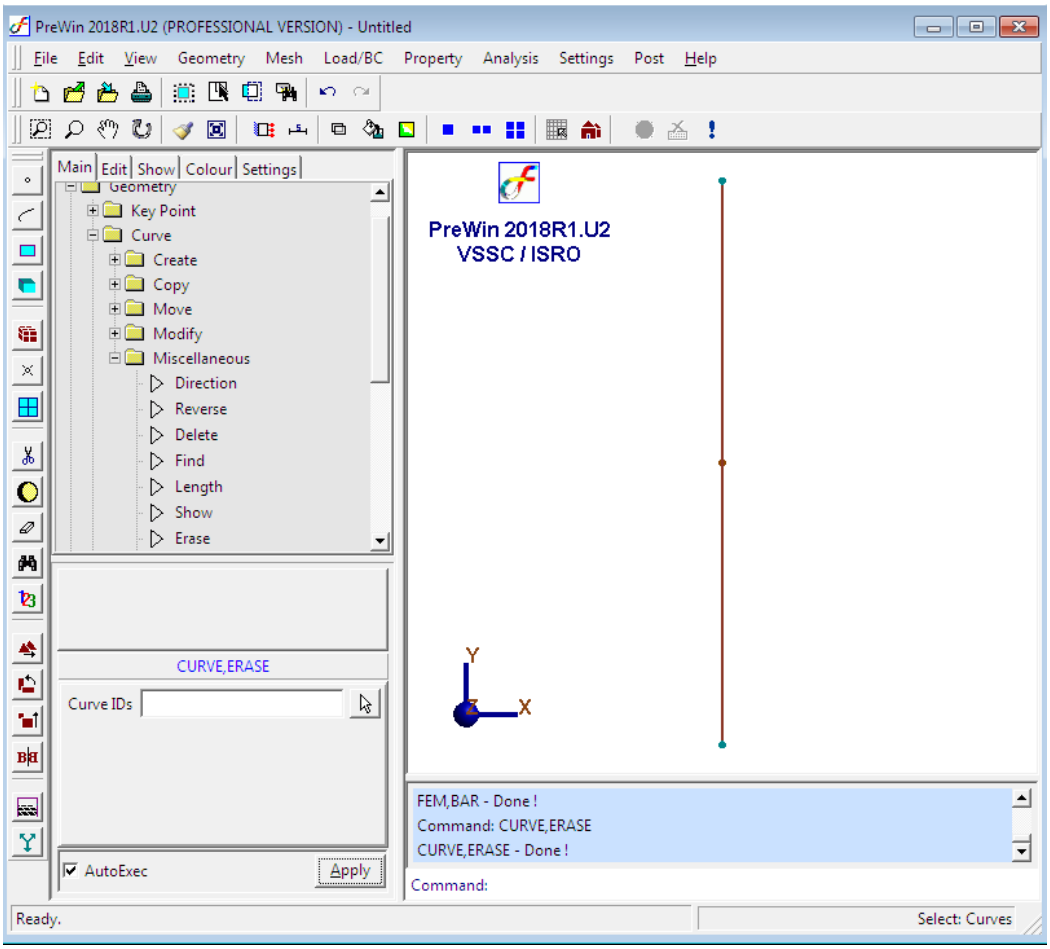
Menu : Geometry → Curve → Miscellaneous → Erase

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.

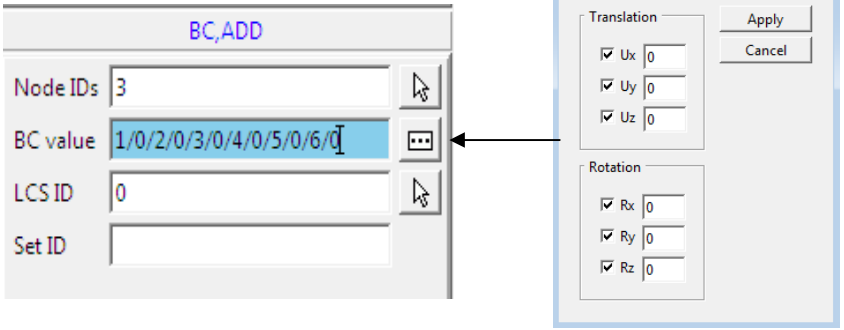


5. Apply boundary condition

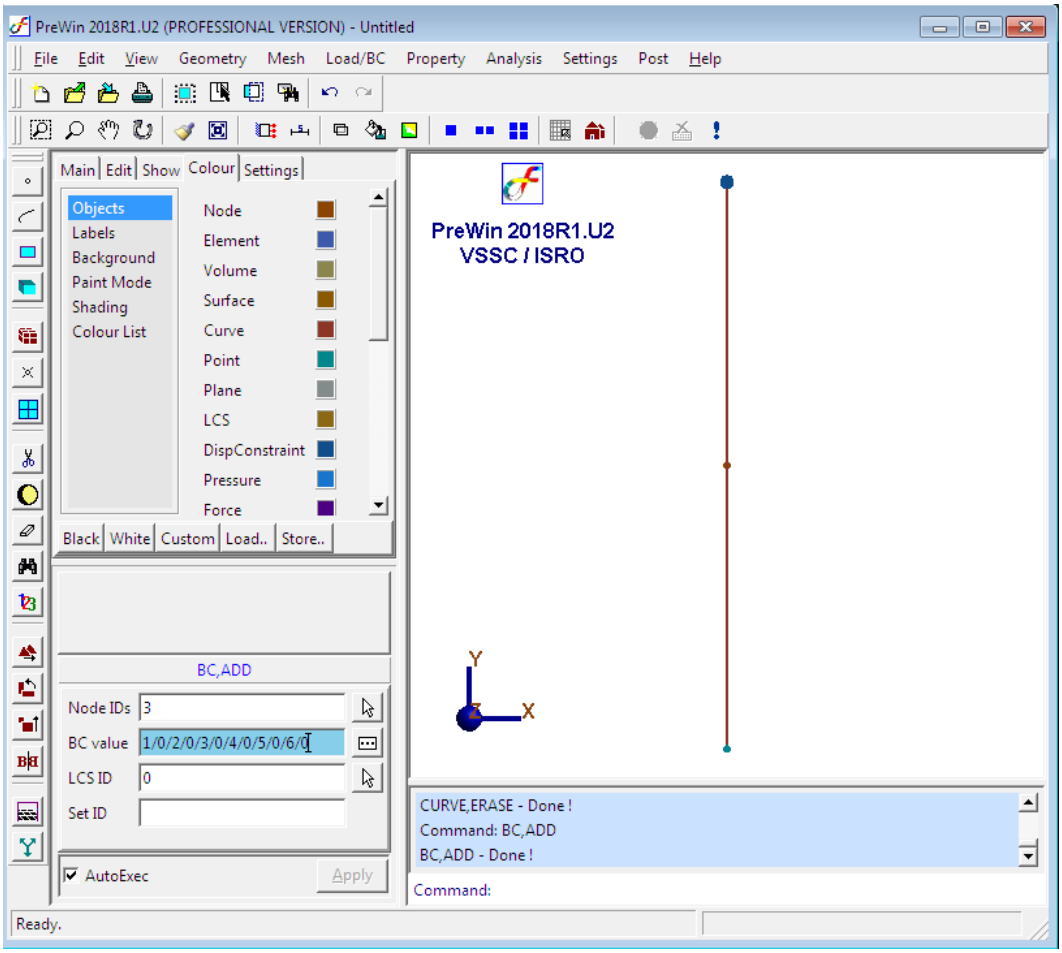
Command : BC,ADD

Menu : Load/BC → Displacement BC → Add

Parameters :



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

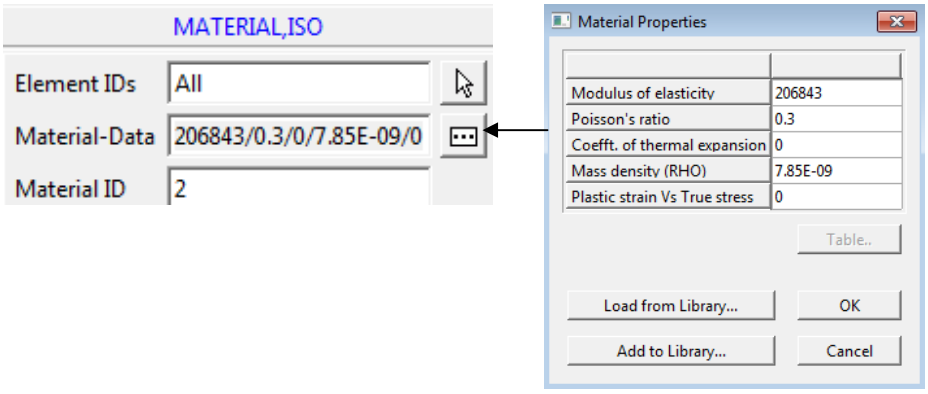


6. Specify material property

Command : MATERIAL,ISO

Menu : Property → Material → Isotropic

Parameters :

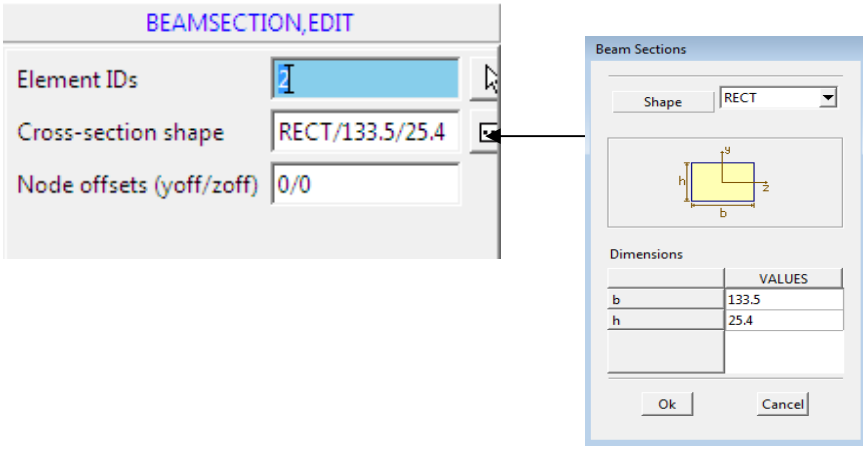


7. Specify beam section

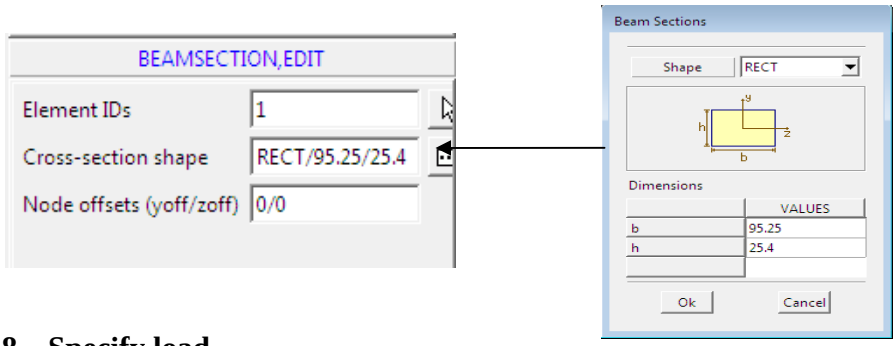
Command : BEAMSECTION,ADD

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

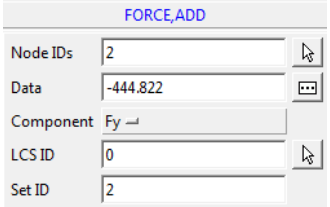


8. Specify load

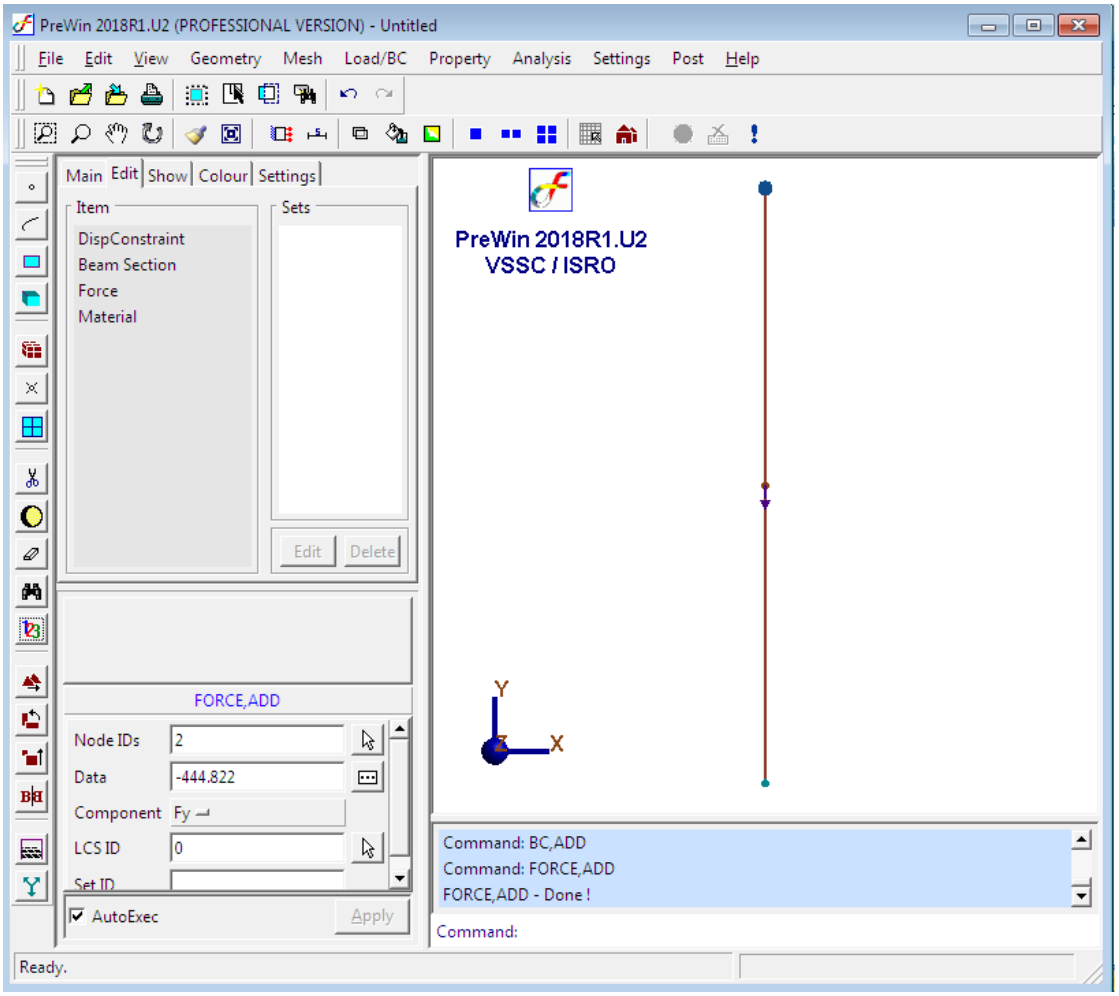
Command : FORCE,ADD

Menu : Load/BC → Point Load → Add

Parameters :



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



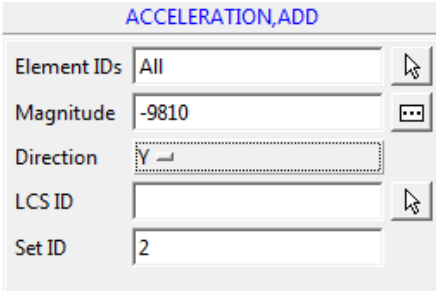
9. Specify self weight

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

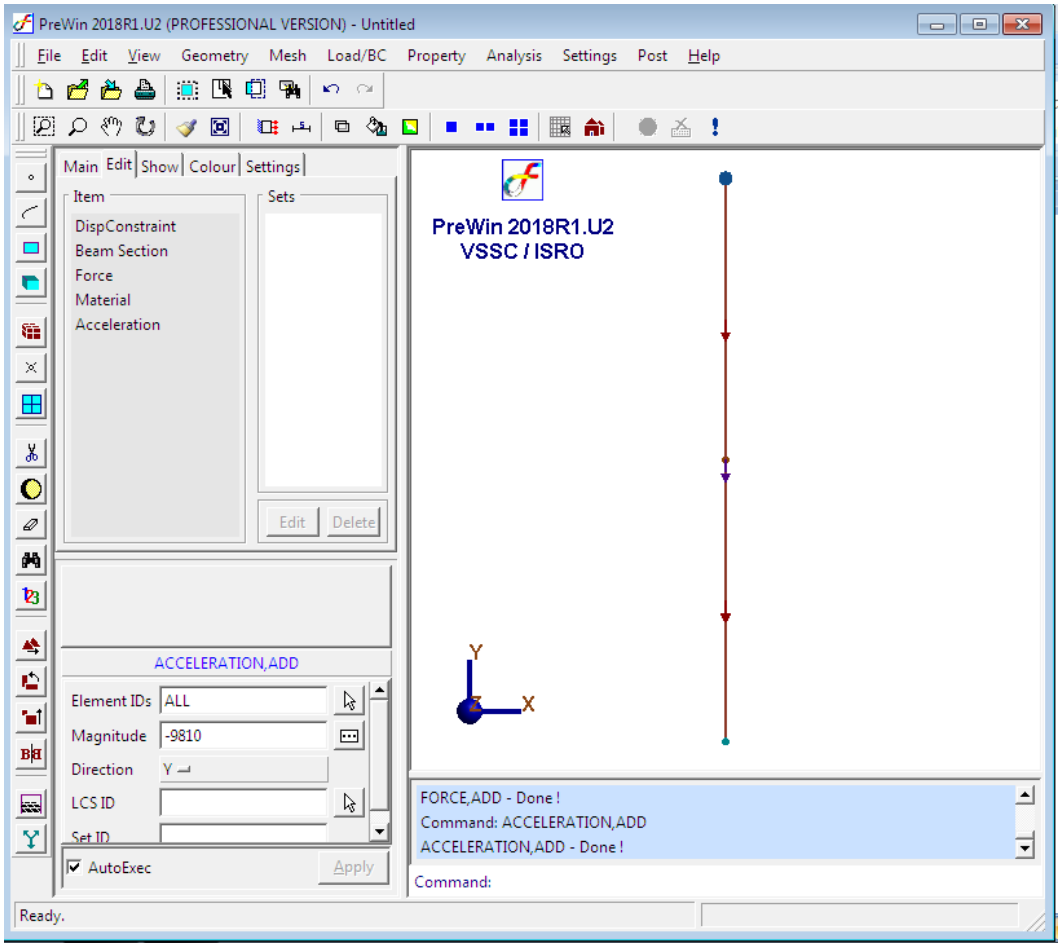
Command : ACCELERATION,ADD

Menu : Load/BC → Acceleration → Add

Parameter :



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

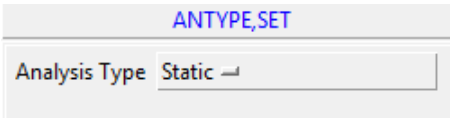


10. Set analysis type

Command : ANTYPE,SET

Menu : Analysis → Analysis Type

Parameter :

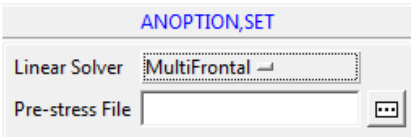


11. Set analysis option

Command : ANOPTION,SET

Menu : Analysis → Analysis Option

Parameters :

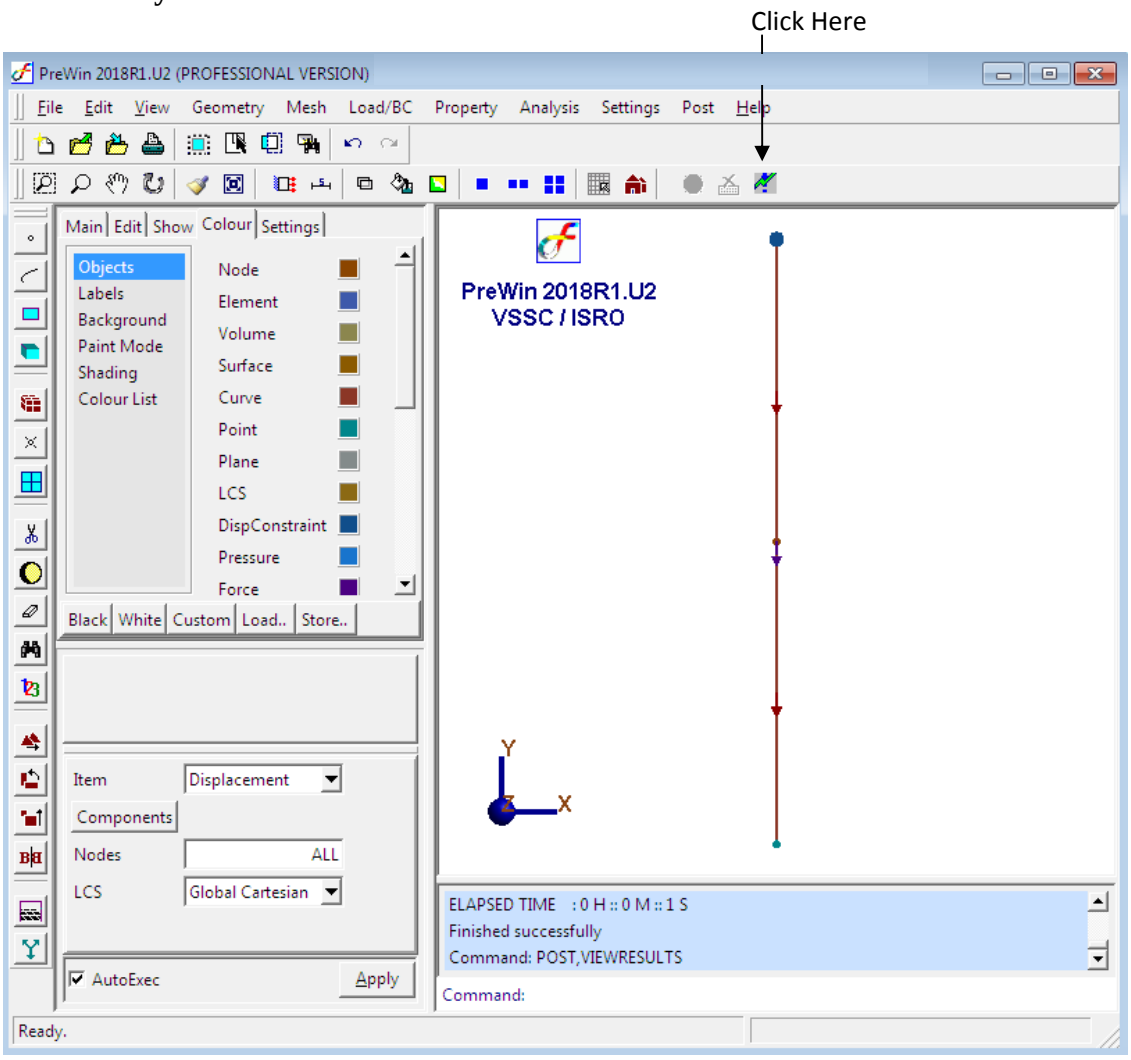


12. Save the project model

Menu : File → Save

13. Submit the job in to FEAST

Menu**: Analysis** → Run Solver



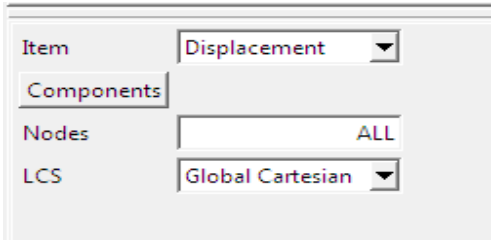
14. Perform Post Processing

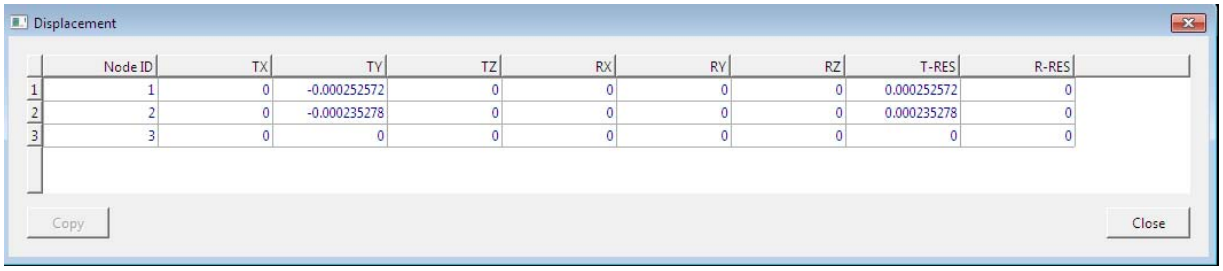
i) Displacement

Command : POST,VIEWRESULTS

Menu : Post → View Results

Parameters :



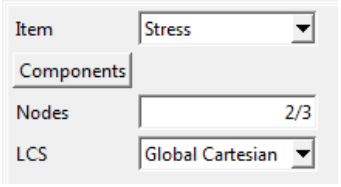


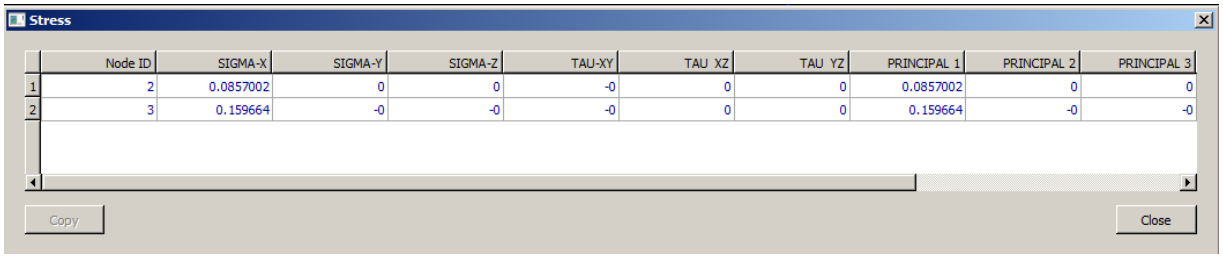
ii) Stress

Command : POST,VIEWRESULTS

Menu : Post → View Results

Parameters :



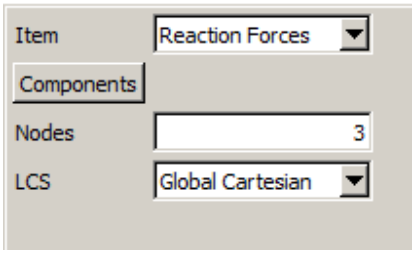


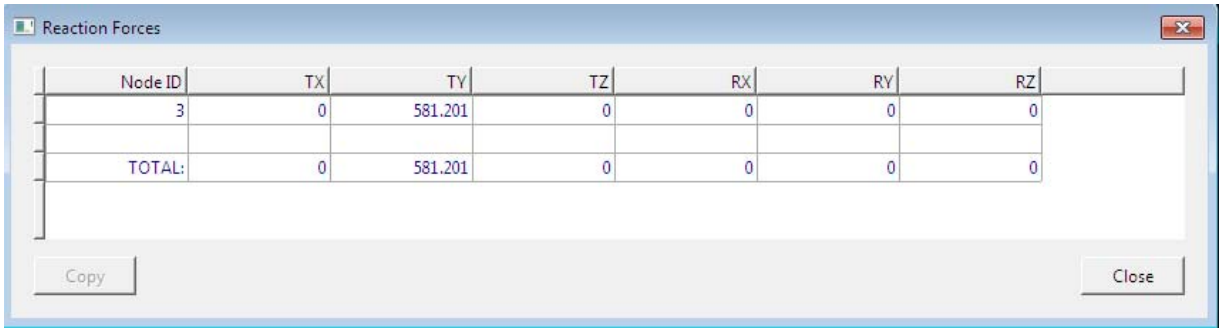
iii) Reaction force.

Command : POST,VIEWRESULTS

Menu : Post → View Results

Parameters :



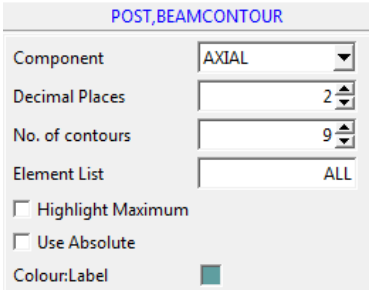


iv) Stress contour

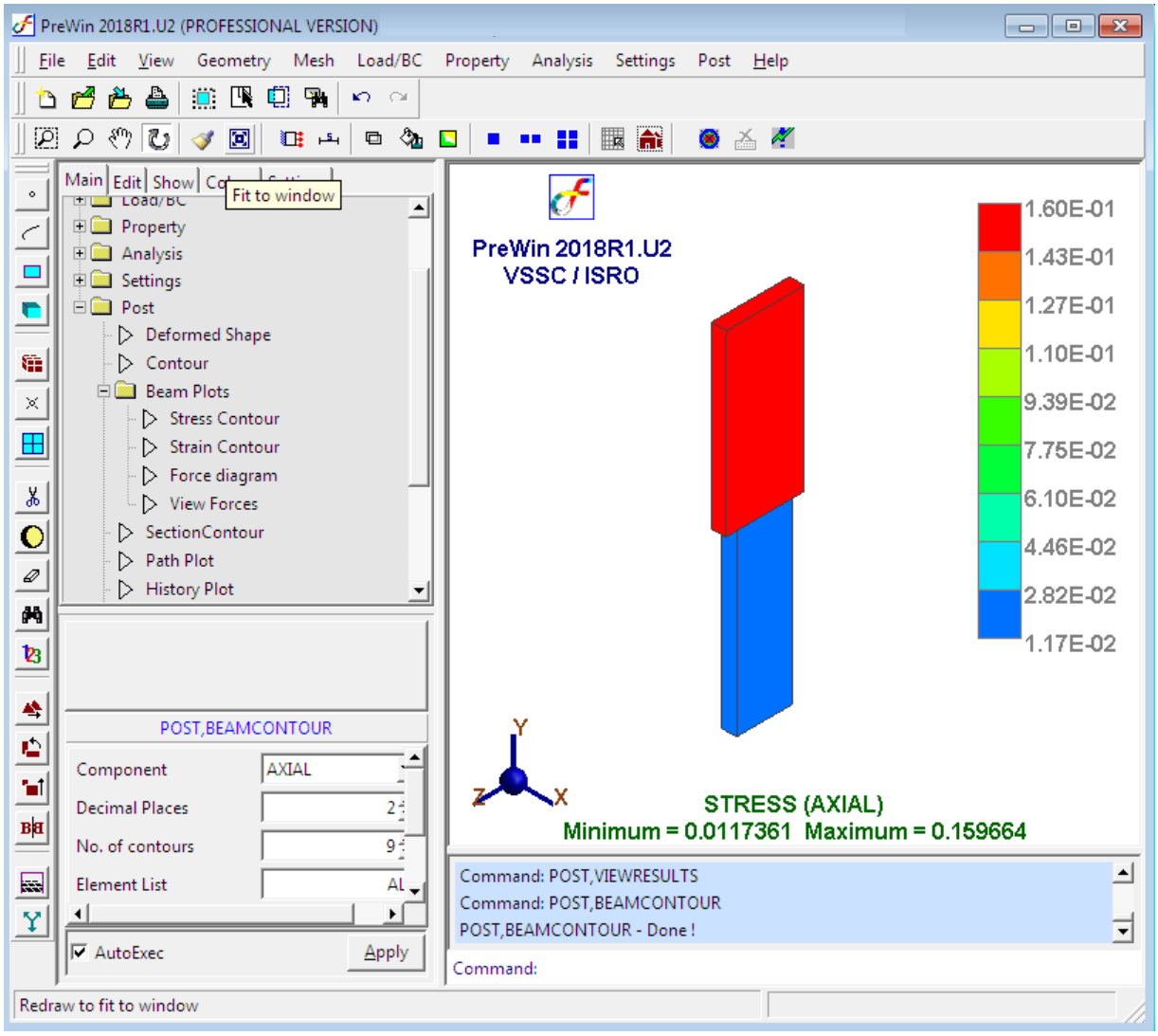
Command : POST,BEAMCONTOUR

Menu : Post → Beam plots → Stress Contour

Parameters :



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


PreWin 2018R1.U2 (PROFESSIONAL VERSION)
File Edit View Geometry Mesh Load/BC Property Analysis Settings Post Help
Main Edit Show
Fit to window
Property
Analysis
Settings
Post
Deformed Shape
Contour
Beam Plots
Stress Contour
Strain Contour
Force diagram
View Forces
SectionContour
Path Plot
History Plot
POST,BEAMCONTOUR
Component AXIAL
Decimal Places
No. of contours
Element List
AutoExec
Apply
PreWin 2018R1.U2
VSSC / ISRO
1.60E-01
1.43E-01
1.27E-01
1.10E-01
9.39E-02
7.75E-02
6.10E-02
4.46E-02
2.82E-02
1.17E-02
Y
Z
X
STRESS (AXIAL)
Minimum = 0.0117361 Maximum = 0.159664
Command: POST,VIEWRESULTS
Command: POST,BEAMCONTOUR
POST,BEAMCONTOUR - Done !
Command:
Redraw to fit to window