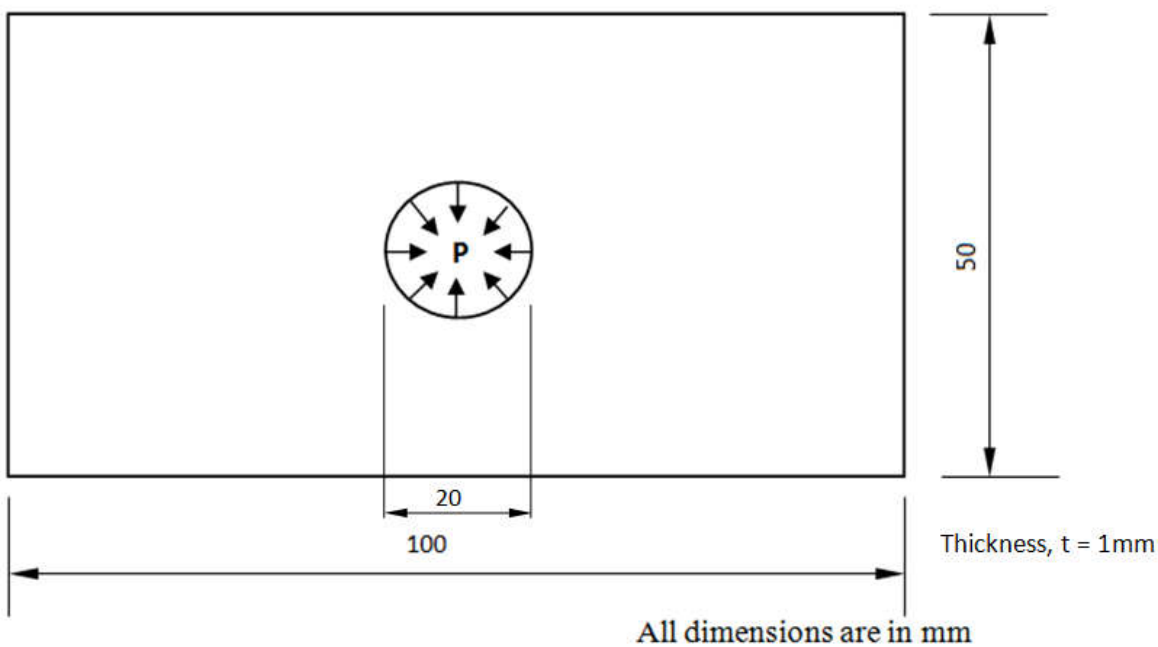
STATIC ANALYSIS OF FIXED PLATE WITH CRCULAR CUTOUT



Material property : *E = 200GPa, v = 0.3*

Load, *P = 100 N*

The plate is arrested at all four sides. Find the displacement and stress distribution

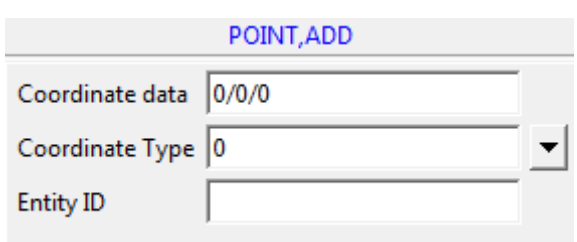
PROCEDURE

1. **Create Keypoints**

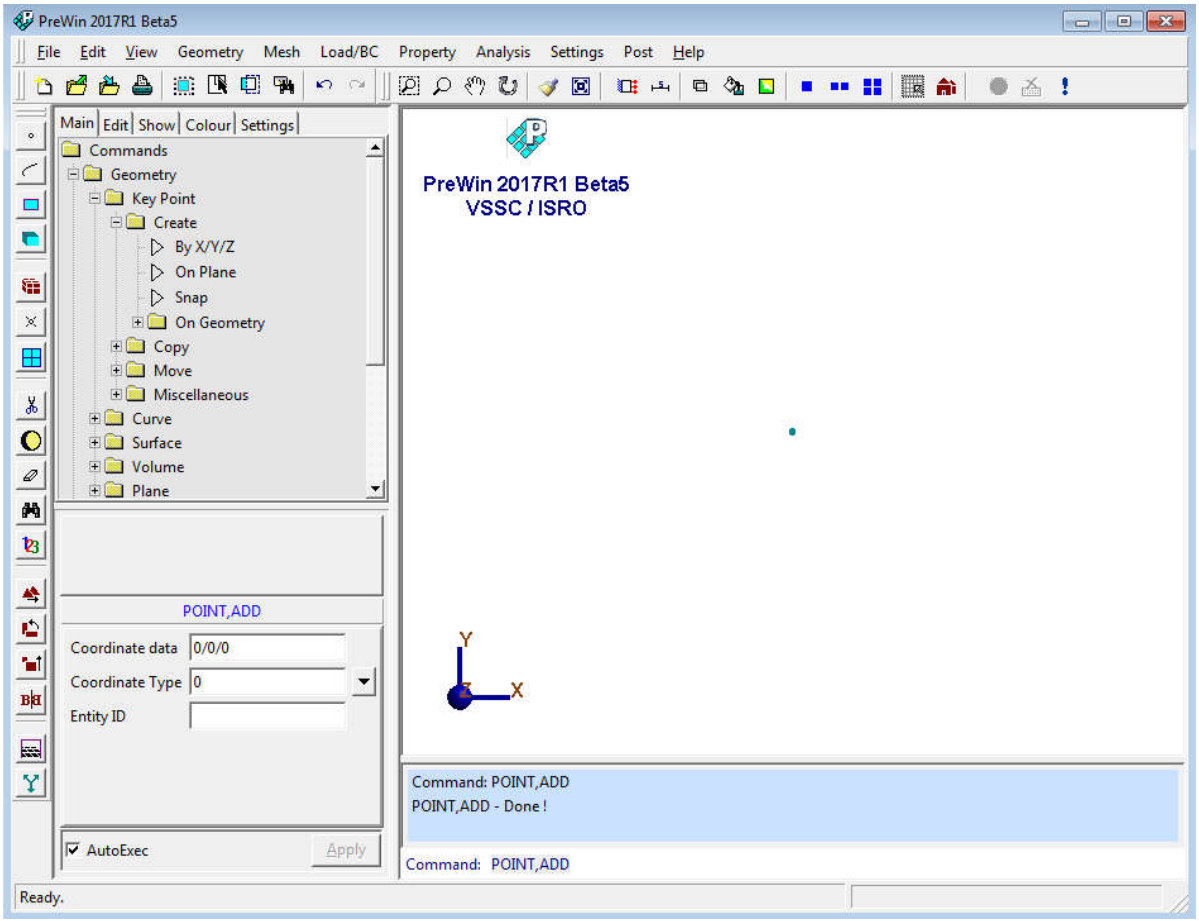
   Command : POINT,ADD

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

   Parameters :



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



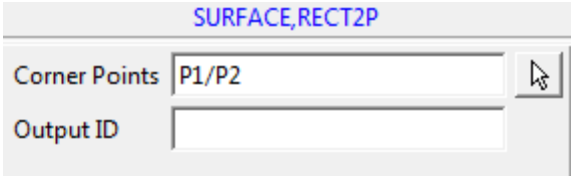
Similarly create key points at (100/50/0)

2. Create Surface

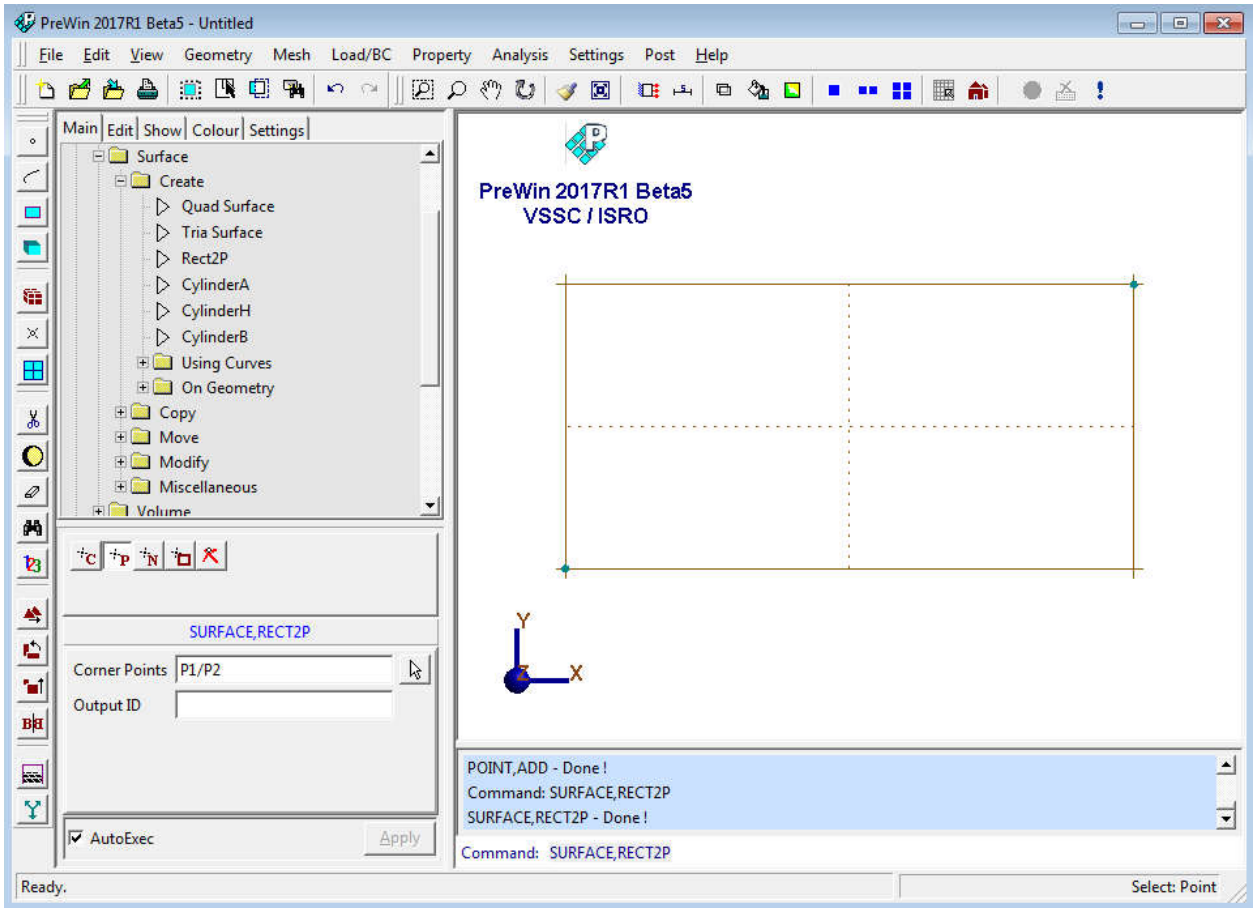
Command : SURFACE,RECT2P

Menu : Geometry → Surface → Create → Rect2P

Parameters :



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

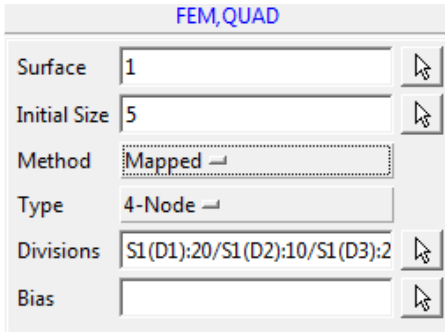


3. **Generate mesh**

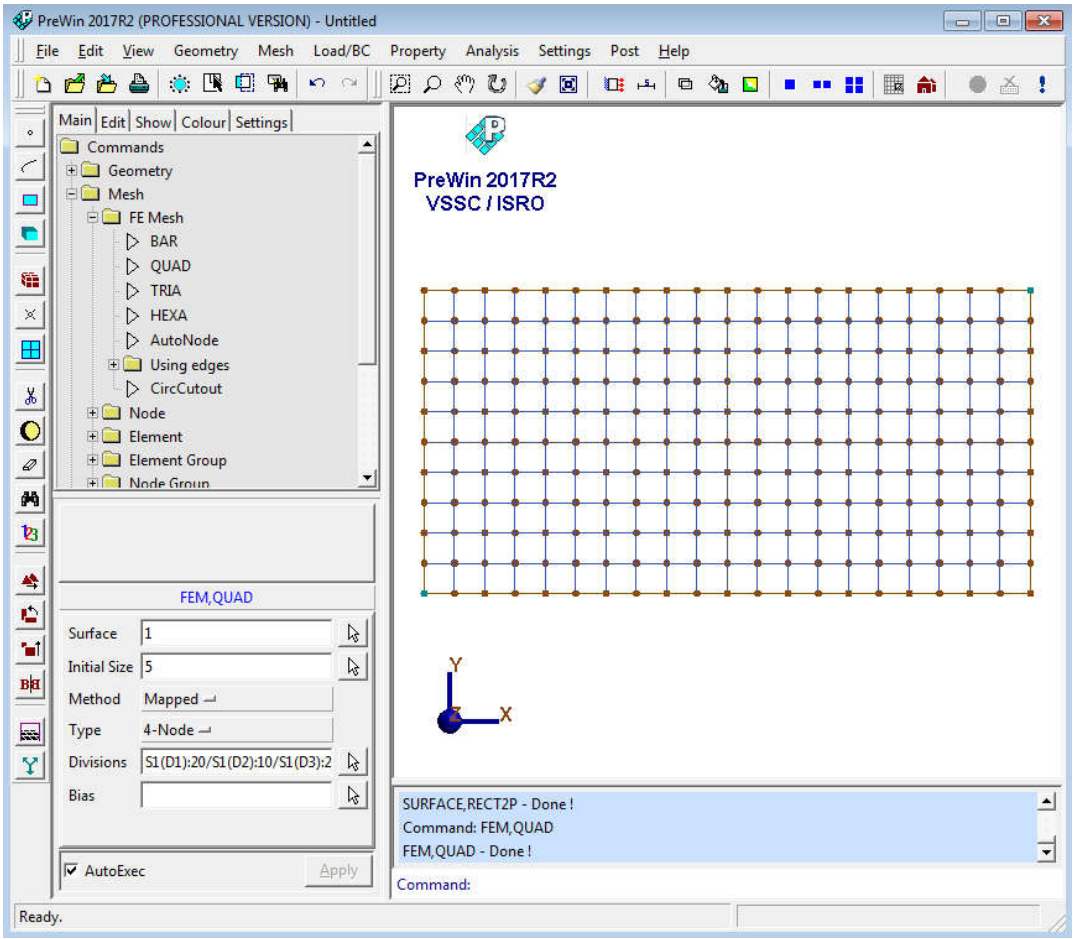
Command : FEM,QUADAUTO

Menu : Mesh → FE Mesh → QUADAuto

Parameters :



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

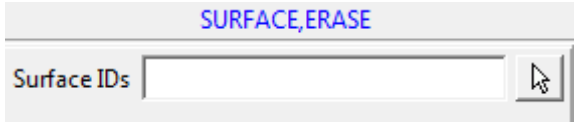


4. Erase Surface

Command : SURFACE,ERASE

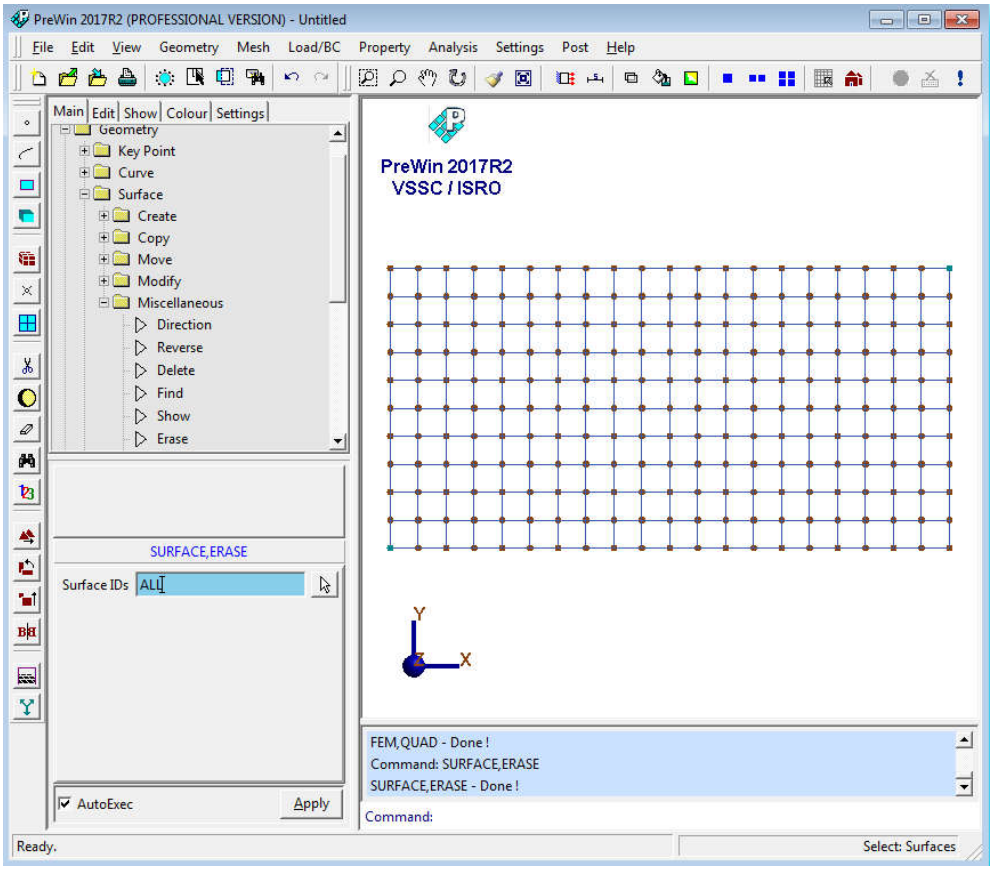
Menu : Geometry → Curve → Miscellaneous → Erase

Parameters :



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

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

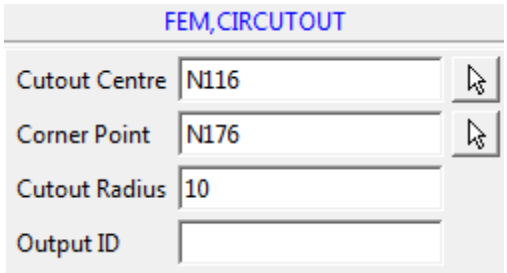


5. Create cutout at the centre

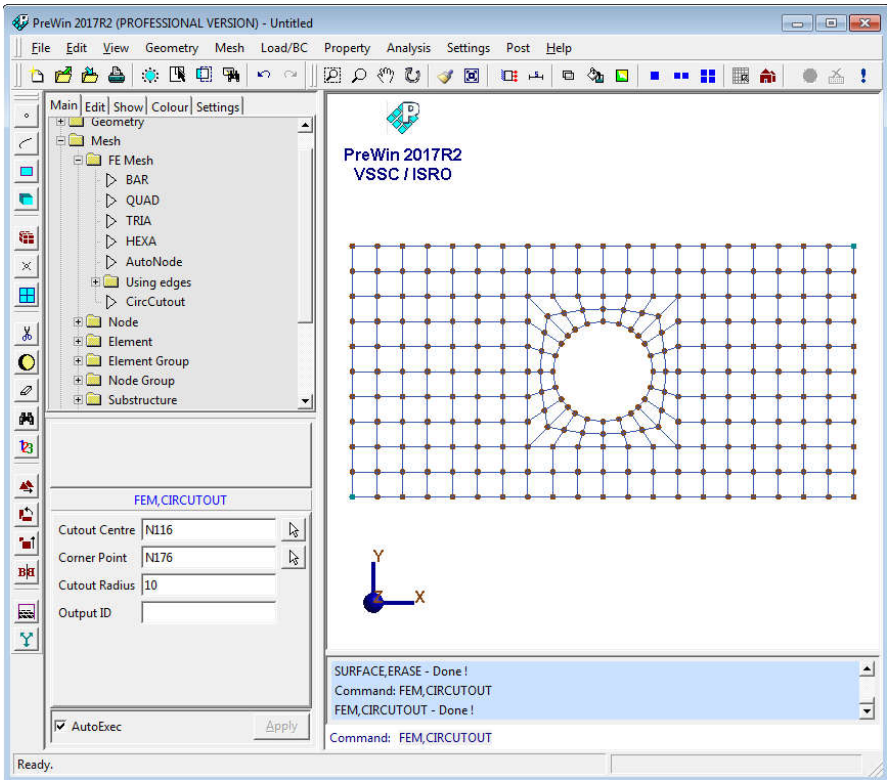
Command : FEM,CIRCUTOUT

Menu : Mesh → FE Mesh → CircCutout

Parameters :



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

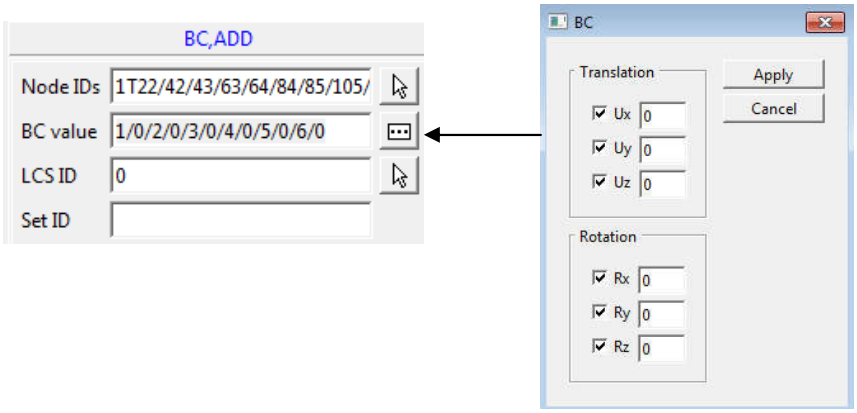


6. Apply Boundary Condition

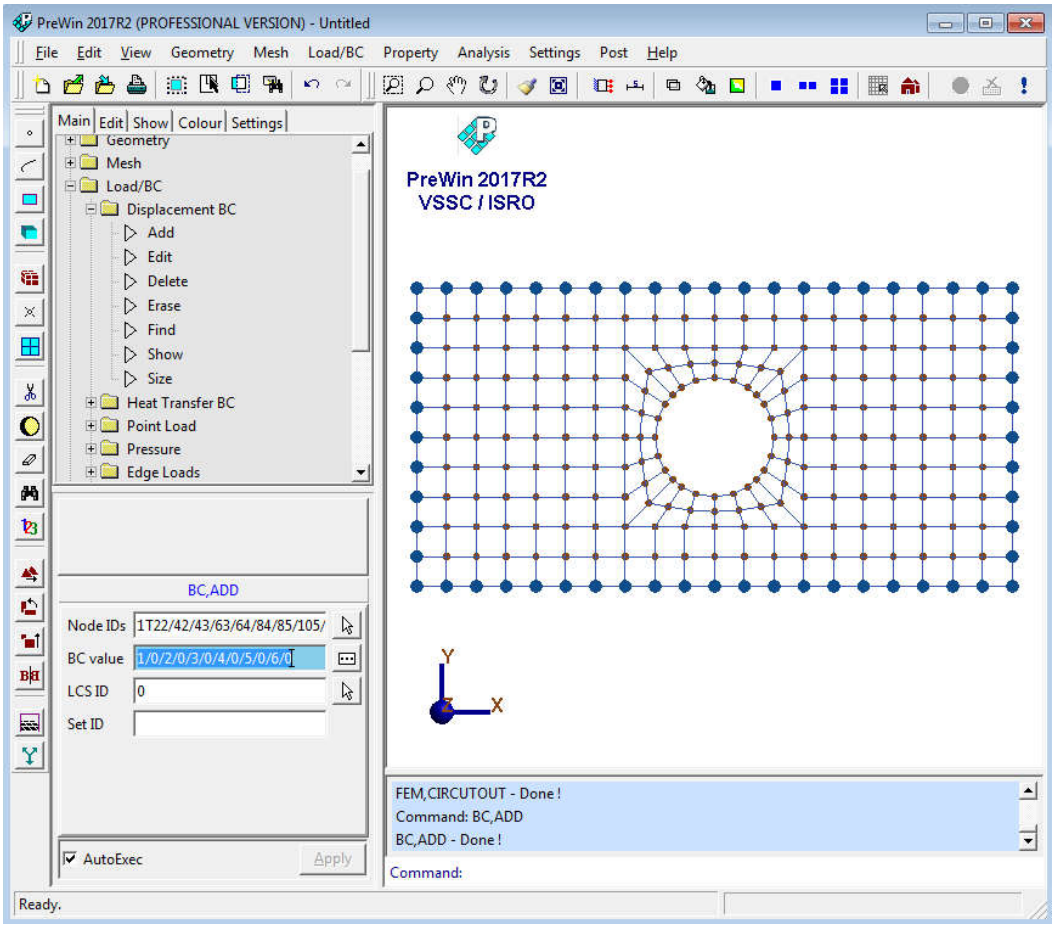
Command : BC,ADD

Menu : Load/BC → Displacement BC → Add

Parameters :



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

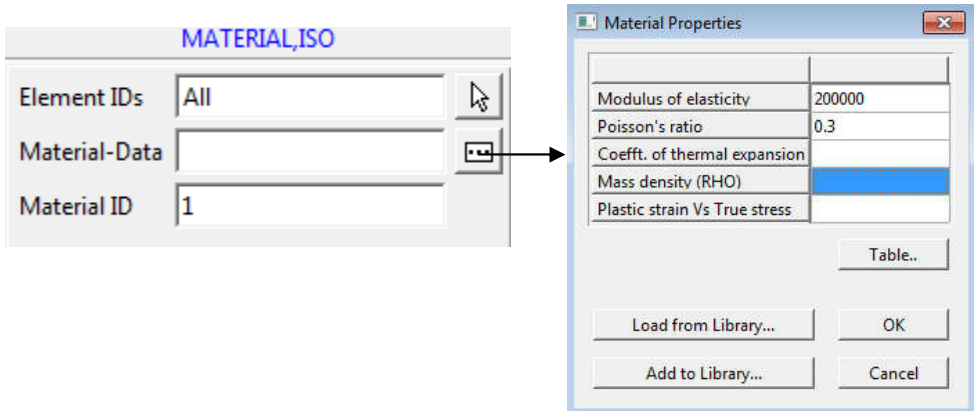


7. Apply Material Property

Command : MATERIAL,ISO

Menu : Property → Material → Isotropic → Add

Parameters :

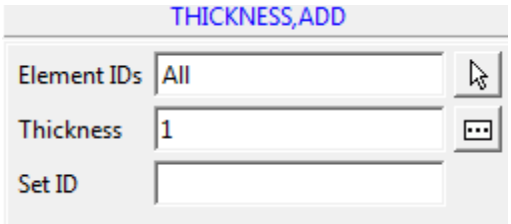


8. Apply Thickness

Command : THICKNESS,ADD

Menu : Property → Physical → Thickness → Add

Parameters :



9. Specify load

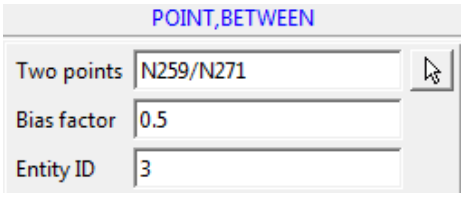
In order create radial distributed load a local cylindrical co-ordinate system is necessary. For that follow the procedure given below.

i) A key point is required at the cutout centre.

Command : POINT,BETWEEN

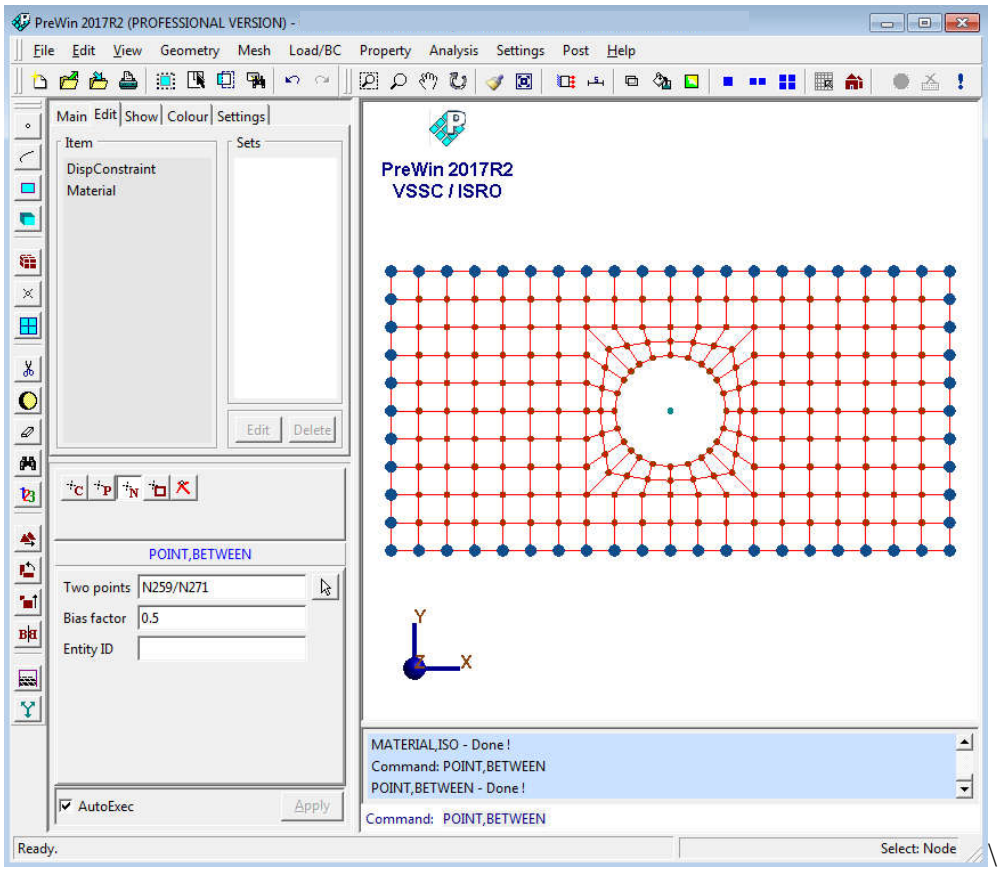
Menu : Geometry → Key Point → Create → On Geometry → Between

Parameters :



In the 'Two Points' command box pick the two points/nodes between which the key point has to be created.

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

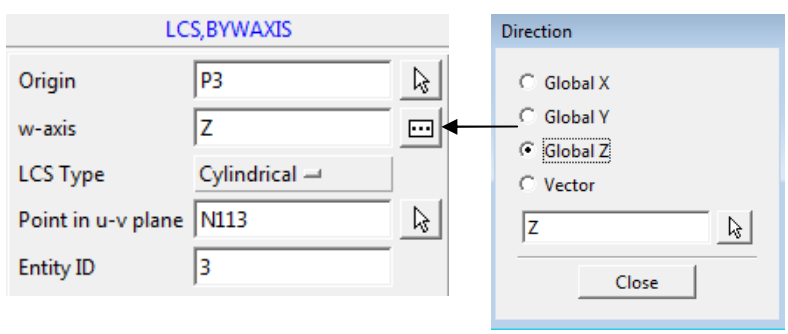


ii) Create local co-ordinate system

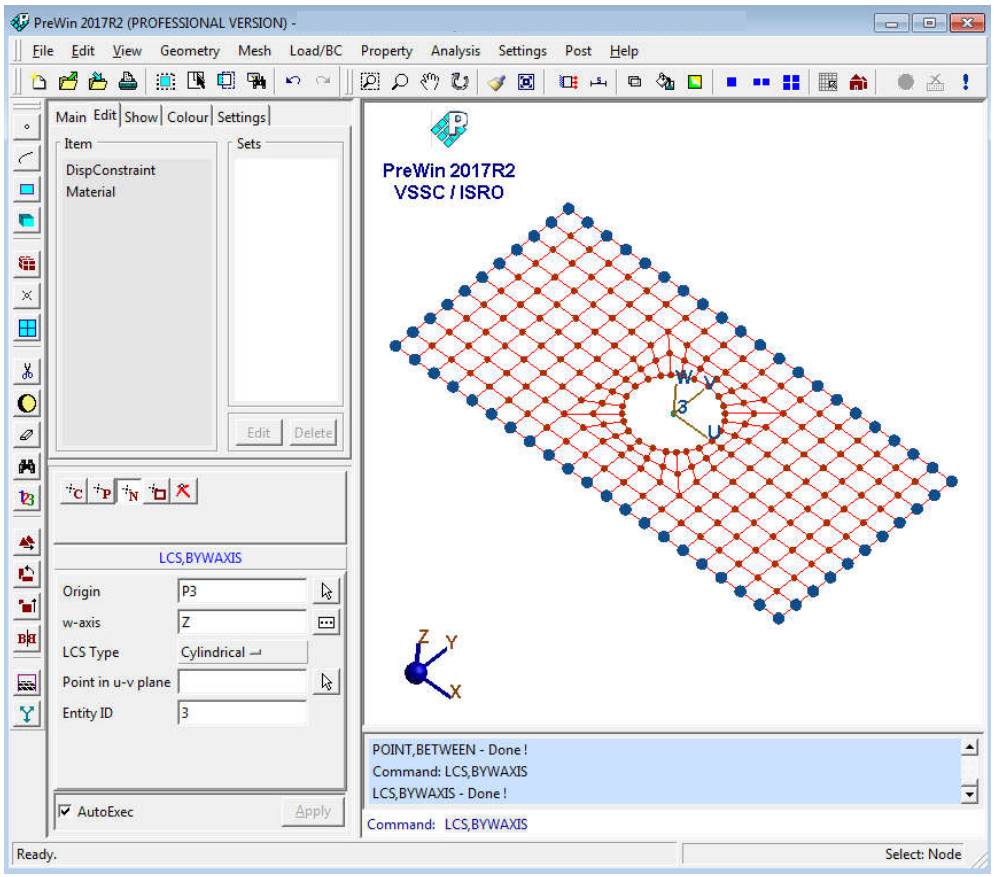
Command : LCS,BYWAXIS

Menu : Geometry → LCS → Create → By w-axis

Parameters :



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

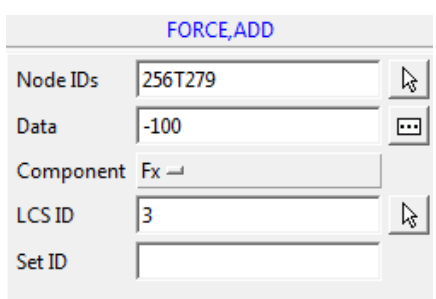


iii) Apply load

Command : FORCE,ADD

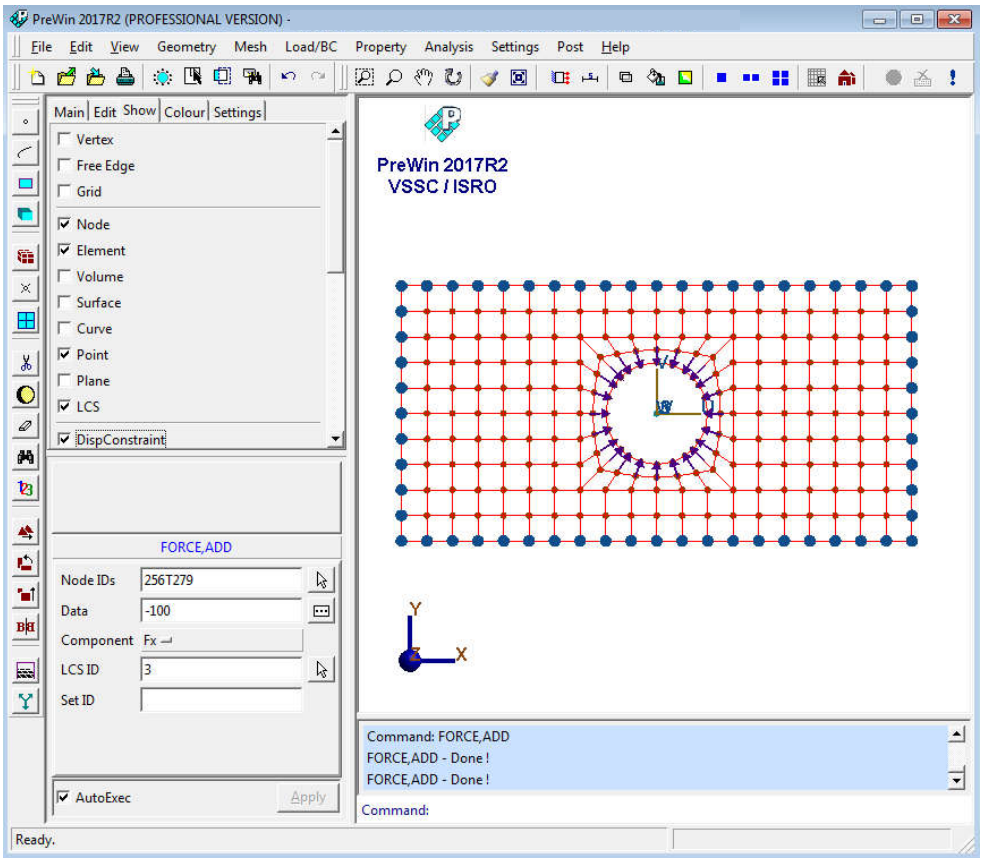
Menu : Load/BC → Point Load → Add

Parameters :



Select the user defined LC ID in 'LCS ID' command box.For selecting the nodes around the circular boundary circular selection can be used from selection method drop list.

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

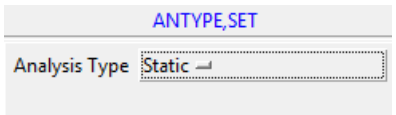


10. Set Analysis Type

Command : ANTYPE,SET

Menu : Analysis → Analysis Type

Parameters :

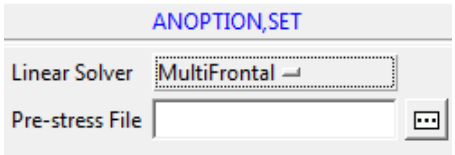


11. Set analysis Option

Command : ANOPTION,SET

Menu : Analysis → Analysis Option

Parameters :

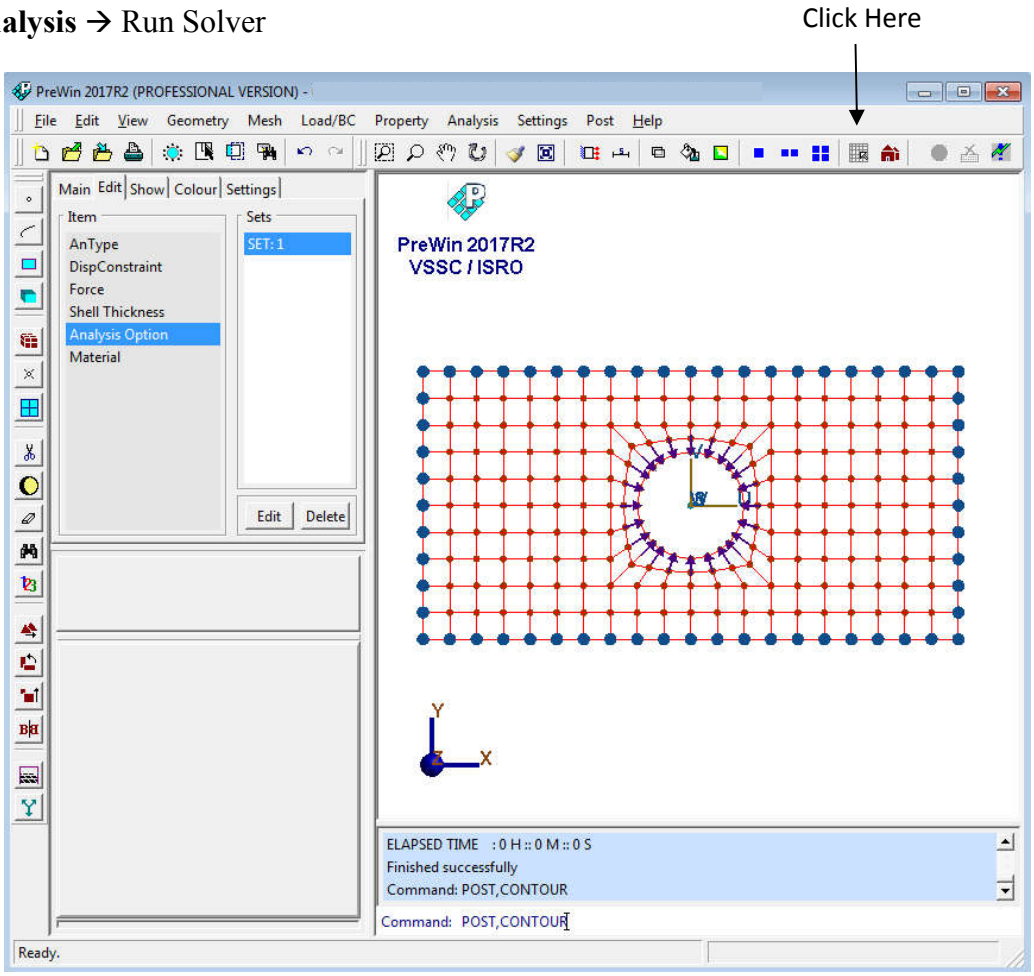


12. Save the project model

Menu : File → Save

13. Submit the job in to FEAST

Menu: **Analysis** → Run Solver



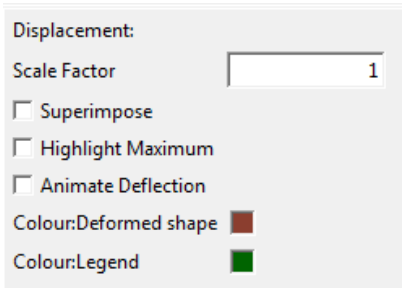
14. Perform Post Processing

i) Deformation

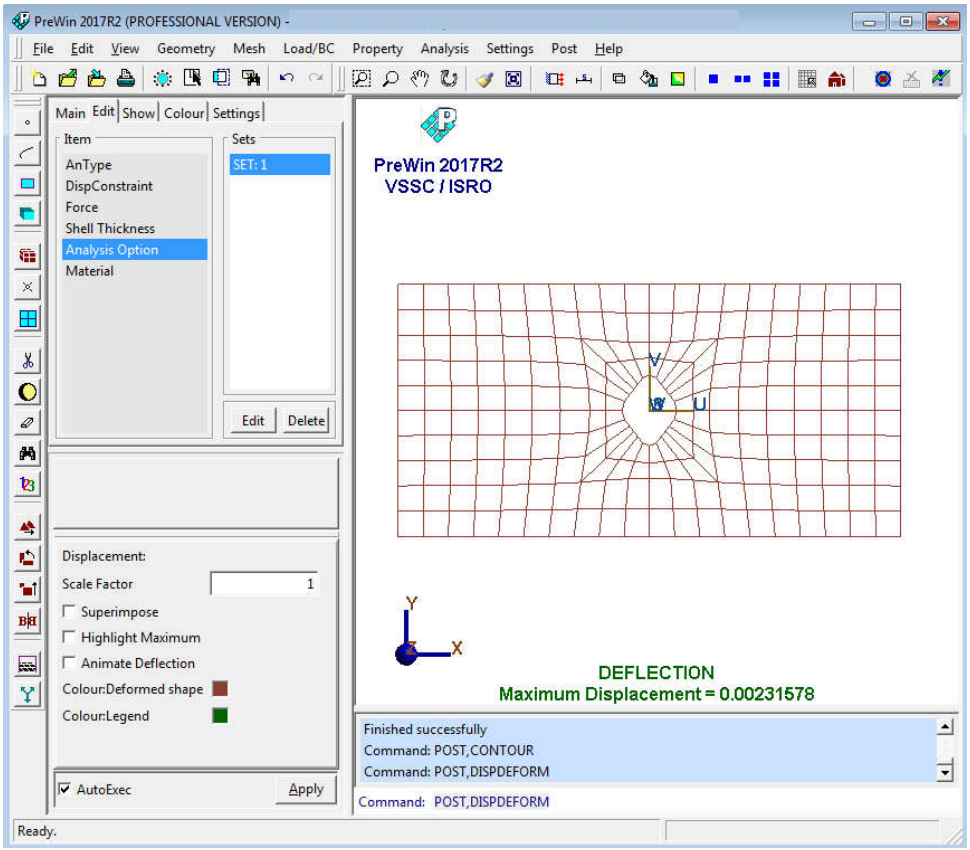
Command : POST,DISPDEFORM

Menu : Post → Deformed Shape

Parameters:



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

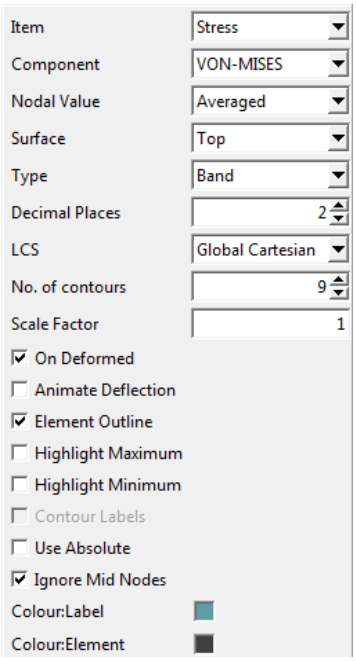


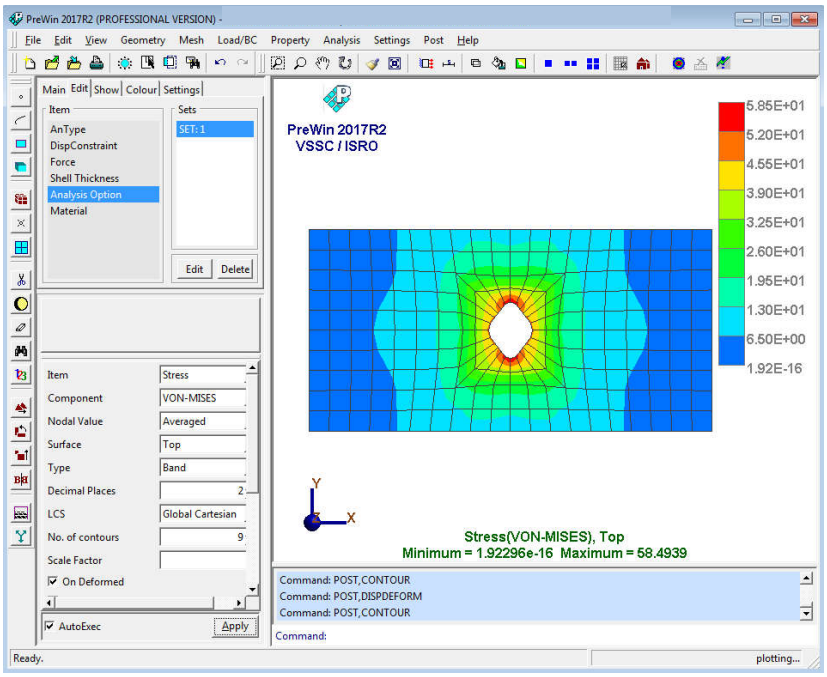
ii) Contour

Command : POST,CONTOUR

Menu : Post → Contour

Parameters:





iii) Output file can be seen in *.OUT