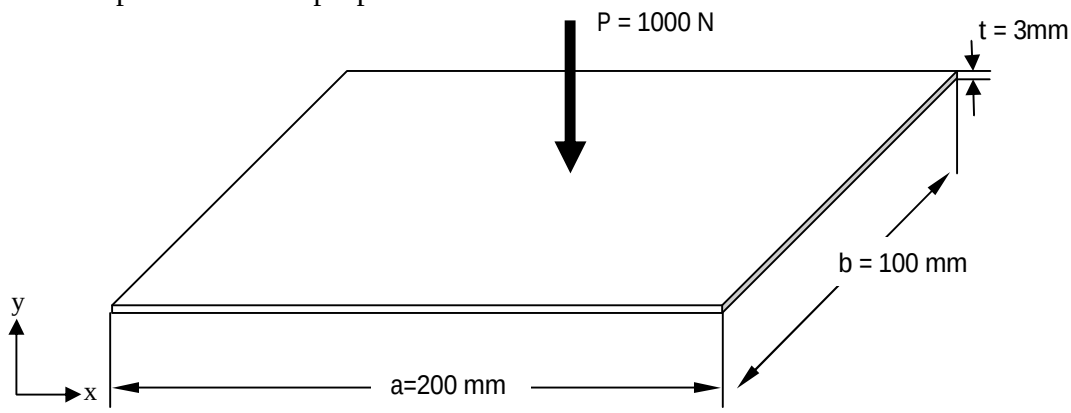
WORKSHOP PROBLEM – 1

STATIC ANALYSIS OF RECTANGULAR PLATE

The objective of this workshop problem is to determine the deflection of a rectangular plate subjected on all its edges. This can be extended to any type of load by approximate specifications. This familiarizes the user to the PreWin GUI for static analysis. This workshop problem contains the steps involved in specifying varying loads, thickness and fiber composite material properties.



Problem Definition

Plate dimensions are shown in the figure.

Analysis Type : Static

ISOTROPIC

Modulus of elasticity, E = 70000 MPa

Poisson's Ratio, υ = 0.3

Point Load 1000N is applied in z direction

PROCEDURE

STEP

1. Create 4 points at (0,0,0) (200,0,0) (0,100,0) and (200,100,0)

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

Commands : POINT, ADD

Parameters : (To be filled by the user)

Coordinate Data	0/0/0
Coordinate Type	0
Entity ID	1

After filling the parameters click *apply* button. If apply button is not active then you give *ctrl+enter* key.

2. Create quadrilateral surface on four points

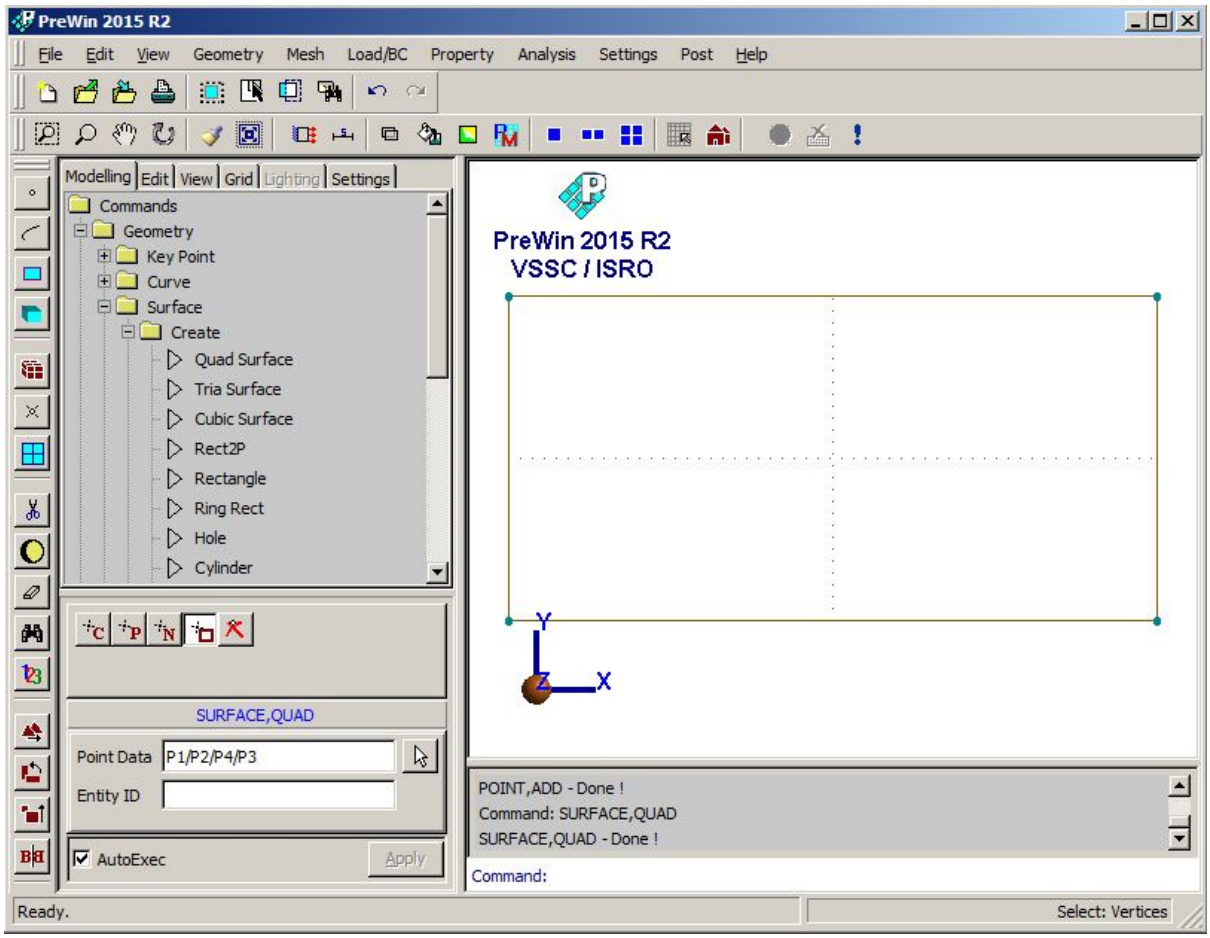
Menu : Geometry → Surface → Create → Quad Surface

Command : SURFACE, QUAD

Parameters :

Point Data	Use mouse to pick the four points
Entity ID	1

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



3. Meshing using quadrilateral elements

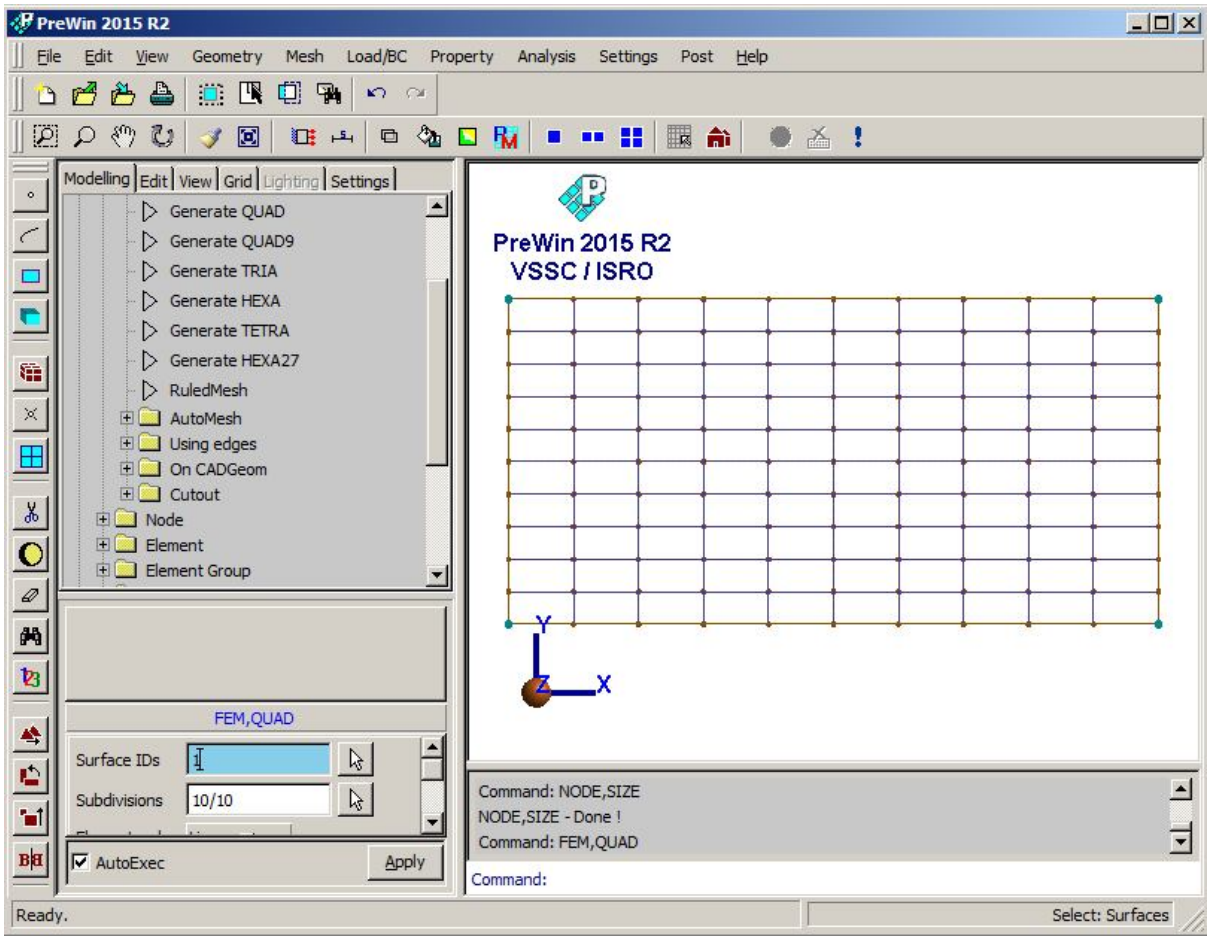
Menu : Mesh → FE Mesh → Generate QUAD

Command : FEM, QUAD

Parameters :

Surface IDs	Use mouse to select the surface
Subdivisions	10/10
Element order	Linear
Bias Factors	1/1
Material ID	1
Property ID	1

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



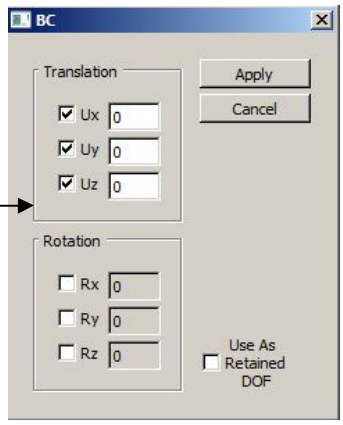
4. Specify displacement boundary conditions

Menu : Load/BC → Displacement BC → Add

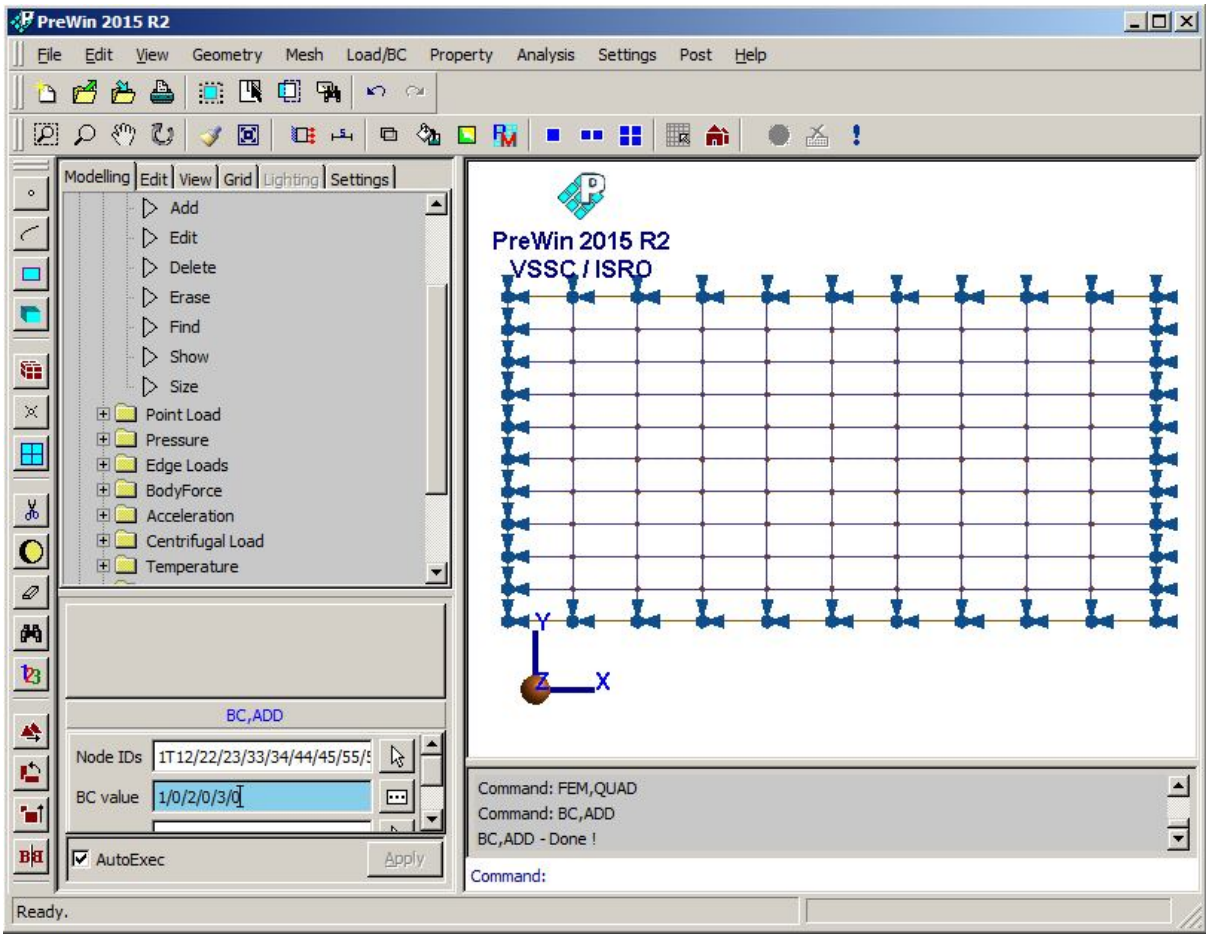
Command : BC,ADD

Parameters :

Node IDs	Select all nodes on the boundary edges Or Select *on boundary* button from the top tool bar
BC Value	1/0/2/0/3/0
LCS ID	0
Set ID	1



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



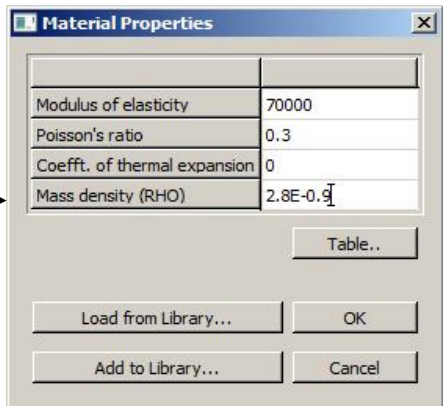
5. Specify material properties

Menu : Property → Material → Isotropic → Add

Command : MATERIAL, ISO

Parameters :

Element IDs	all
Material Data	70000/0.3/0/2.8E-09/0
Material ID	1



6. Specify shell thickness

Menu : Property → Physical → Thickness → Add

Command : THICKNESS, ADD

Parameters :

Element IDs	All
Thickness	3
Set ID	1

7. Specify point load

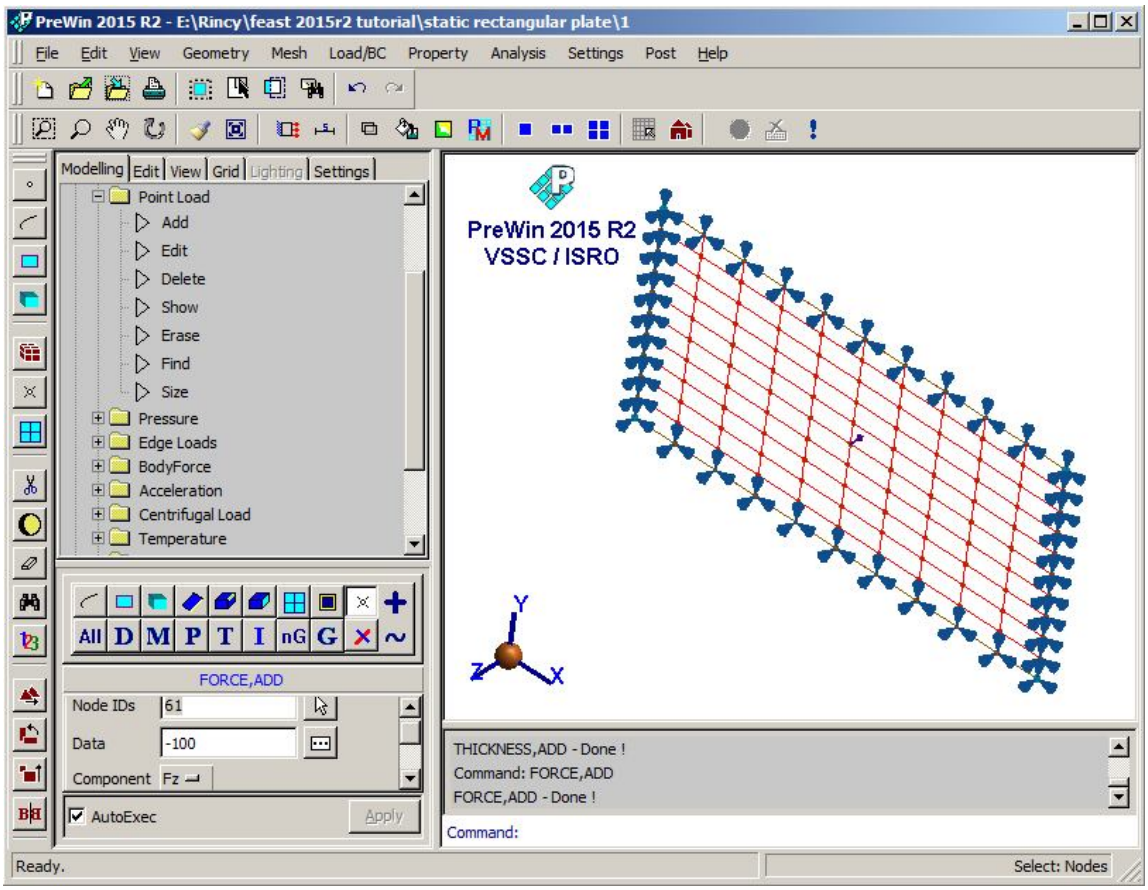
Menu : Load/BC → Point Load → Add

Command : FORCE, ADD

Parameters :

Node IDs	61
Data	-100
Component	Fz

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



8. Set the analysis type

Menu : Analysis → Analysis Type

Command : ANTYPE, SET

Analysis Type	Static

9. Set the analysis options

Menu : Analysis → Analysis Options

Command : ANOPTION,SET

Linear Solver	MultiFrontal

10. Save the project model

Menu : File → Save

11. Submit the job into FEAST

Menu : Analysis → Run Solver

After the solution is completed the message *"successfully completed"* appears in the *message box.*

12. Perform post processing

a) Deformed Shape

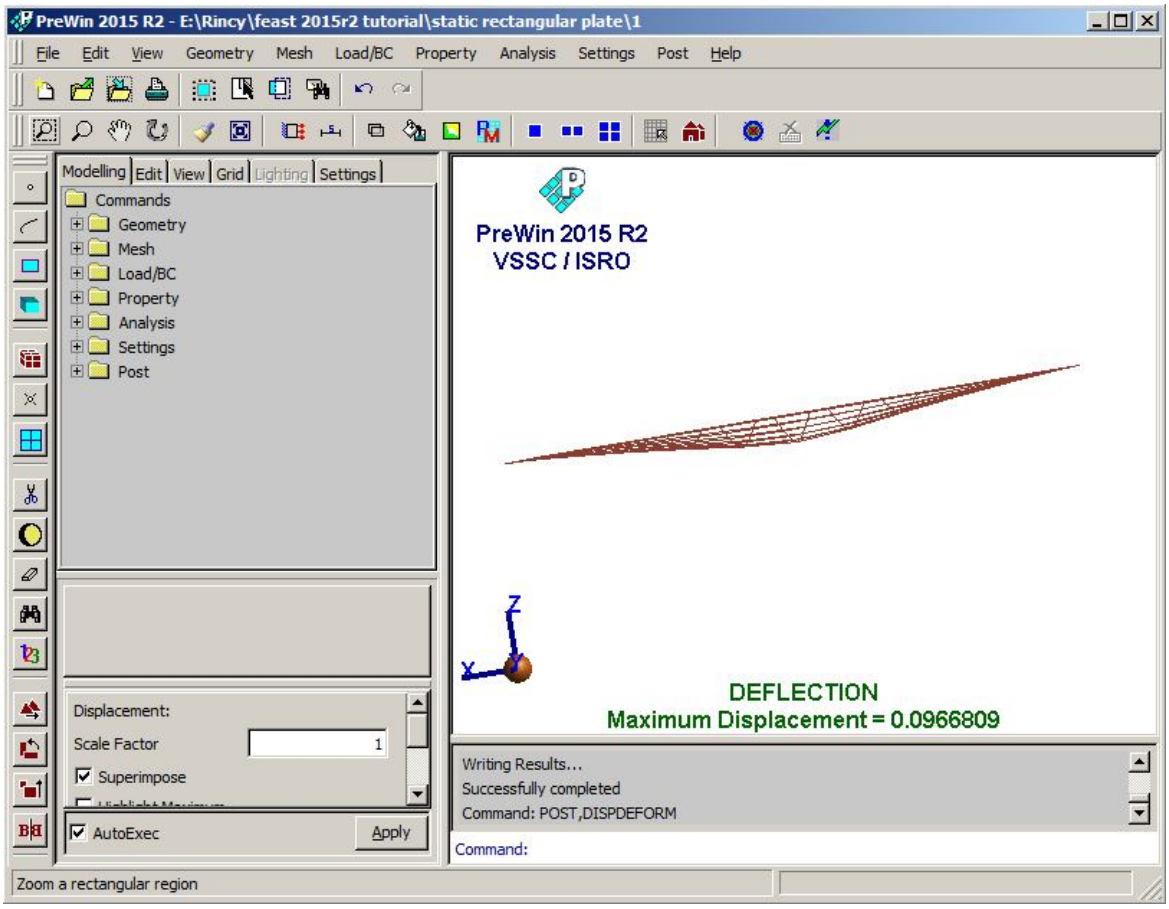
Menu : Post → Deformed Shape

Command : POST, DISPDEFORM

Parameters :

Displacement	
Scale Factor	0

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



b) Animation of deformed model

Menu : Post → Deformed Shape

Command : POST, DISPDEFORM

Parameters :

Animate	ON

Animation continues until either click stop from post menu or Esc key is pressed

c) Displacement contours

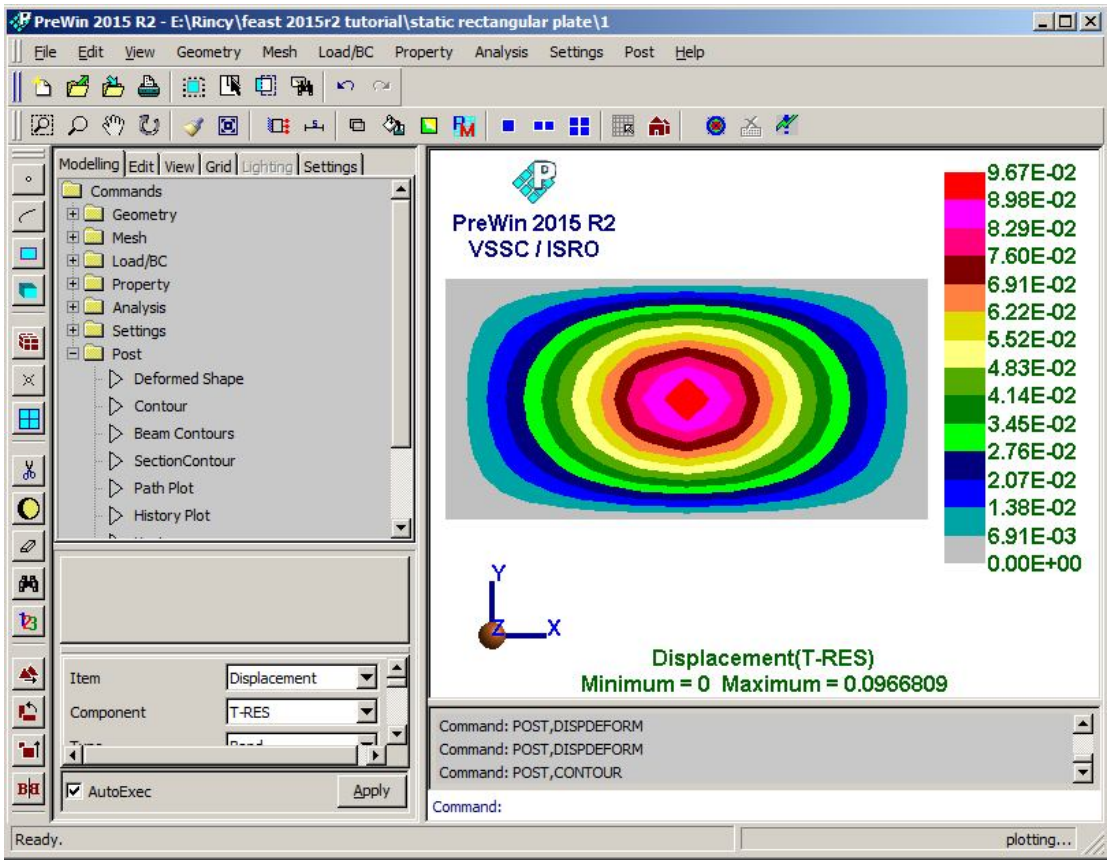
Menu : Post → Contour

Command : POST, CONTOUR

Parameters :

Item	Displacement
Component	T-RES
Type	Band
Decimal Places	2
LCS	Global Cartesian
No. of contours	14

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



d) Graph plots

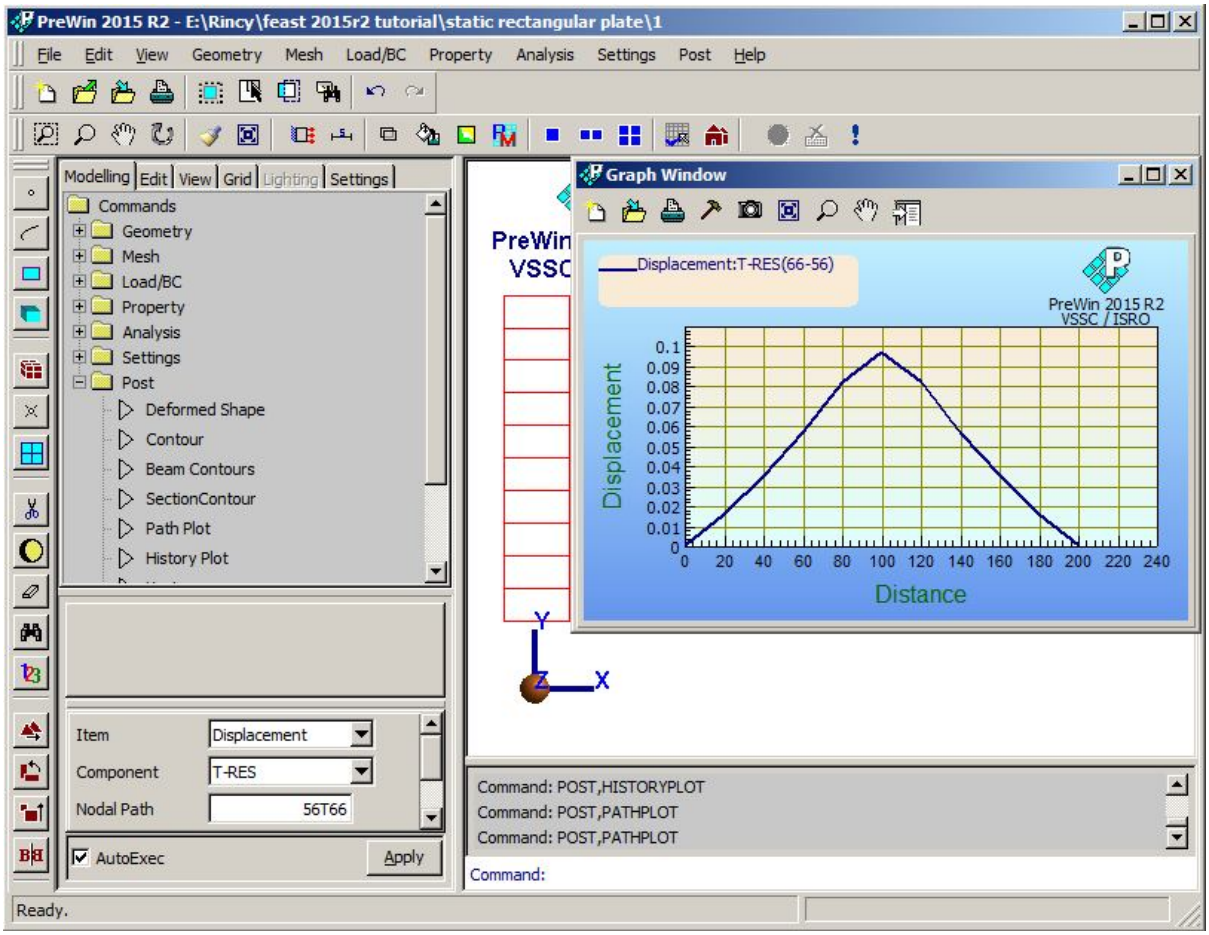
Menu : Post → Path Plot

Command : POST, PATHPLOT

Parameters :

Item	Displacement
Component	T-RES
Nodal Path	56T66

At the end of the above operations a graph as shown appears in the viewport



e) View results

Nodal displacement

Menu : Post → View Results

Command : POST, VIEWRESULTS

Parameters :

Item	Displacement
Nodes	All
LCS	Global Cartesian

At the end of the above mentioned operations a table as shown below.

Displacement

	Node ID	TX	TY	TZ	RX	RY	RZ	T-RES	R-RES
1	1	0	0	0	-1.3652e-05	2.66694e-05	0	0	2.99606e-05
2	2	0	0	0	-0.000509835	2.98906e-05	0	0	0.00051071
3	3	0	0	0	-0.00110601	3.64252e-05	0	0	0.00110661
4	4	0	0	0	-0.00178379	3.96483e-05	0	0	0.00178423
5	5	0	0	0	-0.00238726	2.79693e-05	0	0	0.00238742
6	6	0	0	0	-0.00263532	-2.60209e-18	0	0	0.00263532
7	7	0	0	0	-0.00238726	-2.79693e-05	0	0	0.00238742
8	8	0	0	0	-0.00178379	-3.96483e-05	0	0	0.00178423
9	9	0	0	0	-0.00110601	-3.64252e-05	0	0	0.00110661
10	10	0	0	0	-0.000509835	-2.98906e-05	0	0	0.00051071
11	11	0	0	0	-1.3652e-05	-2.66694e-05	0	0	2.99606e-05
12	12	0	0	0	-1.41943e-05	0.000236	0	0	0.000236427
13	13	0	0	-0.00502804	-0.000485734	0.000252714	0	0.00502804	0.000547541
14	14	0	0	-0.0108126	-0.00105004	0.000308933	0	0.0108126	0.00109454
15	15	0	0	-0.0175427	-0.00170768	0.000342624	0	0.0175427	0.00174172

Copy Close

f) Arrow plot

Displacement

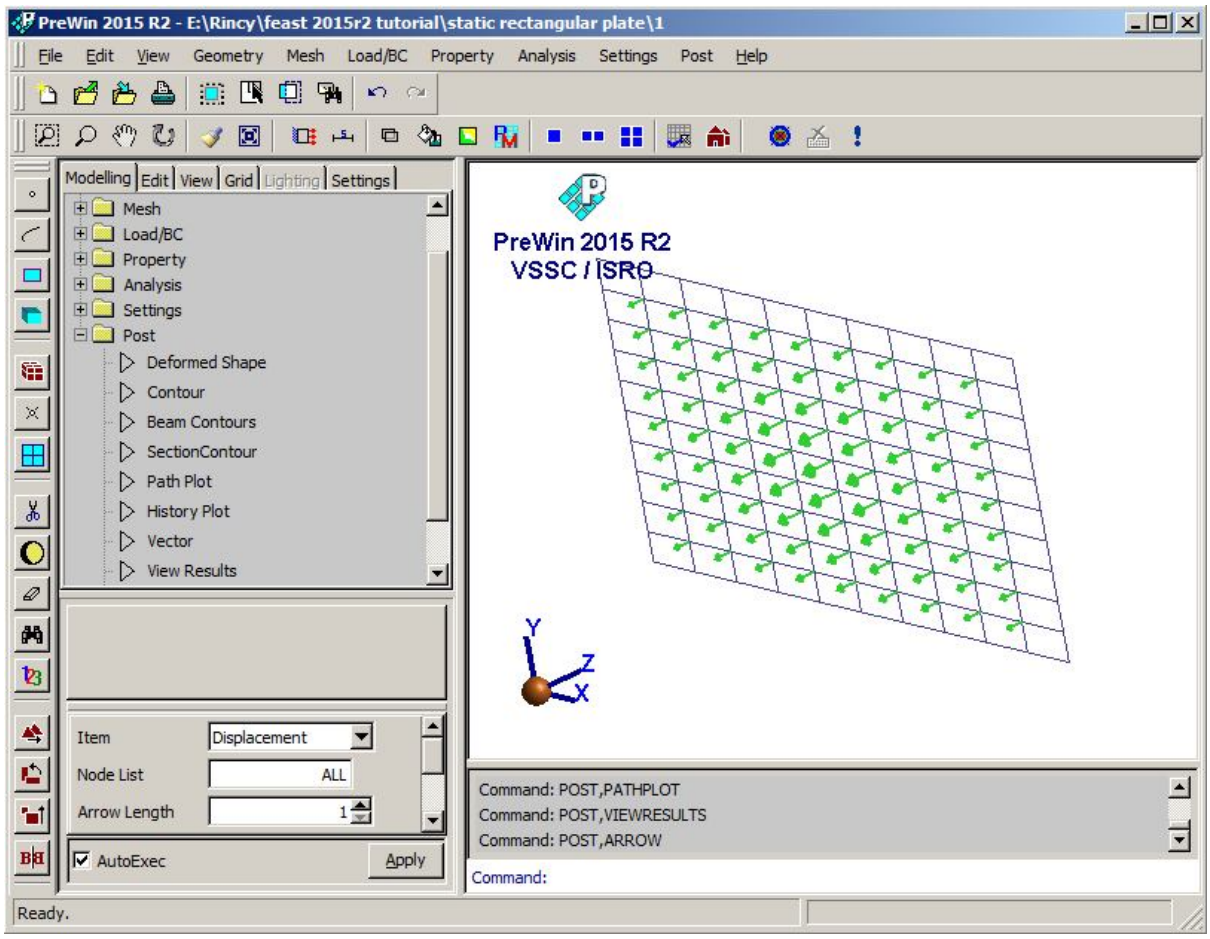
Menu : Post → Vector

Command : POST, ARROW

Parameters :

Item	Displacement
Node List	All
Arrow Length	1

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



g) Output file can be seen in *.OUT