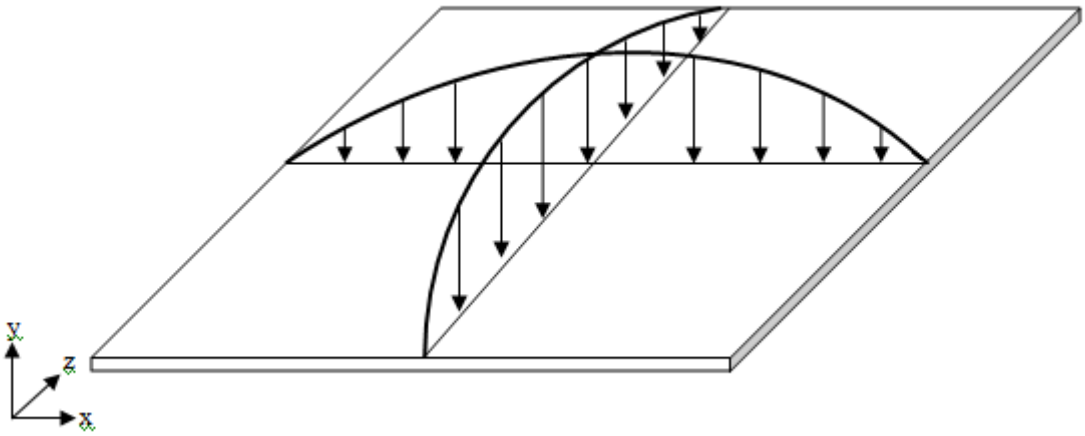
STATIC ANALYSIS OF RECTANGULAR PLATE WITH SINUSOIDAL LOAD



Problem Definition

Plate dimensions – 200mm x 100mm.

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

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

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

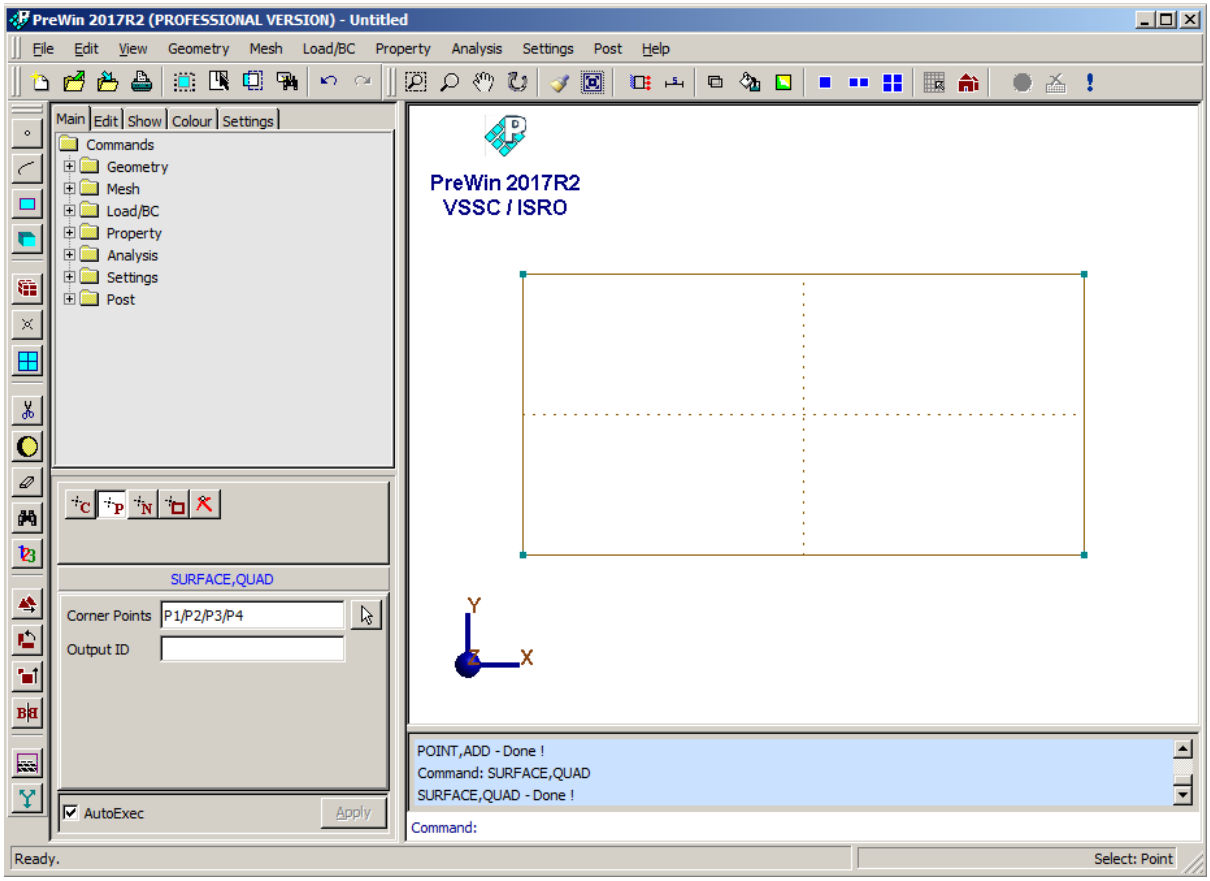
Menu : Geometry → Surface → Create → Quad Surface

Command : SURFACE, QUAD

Parameters :

Corner points	Use mouse to pick the points
Entity ID	1

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



3. Meshing using quadrilateral elements

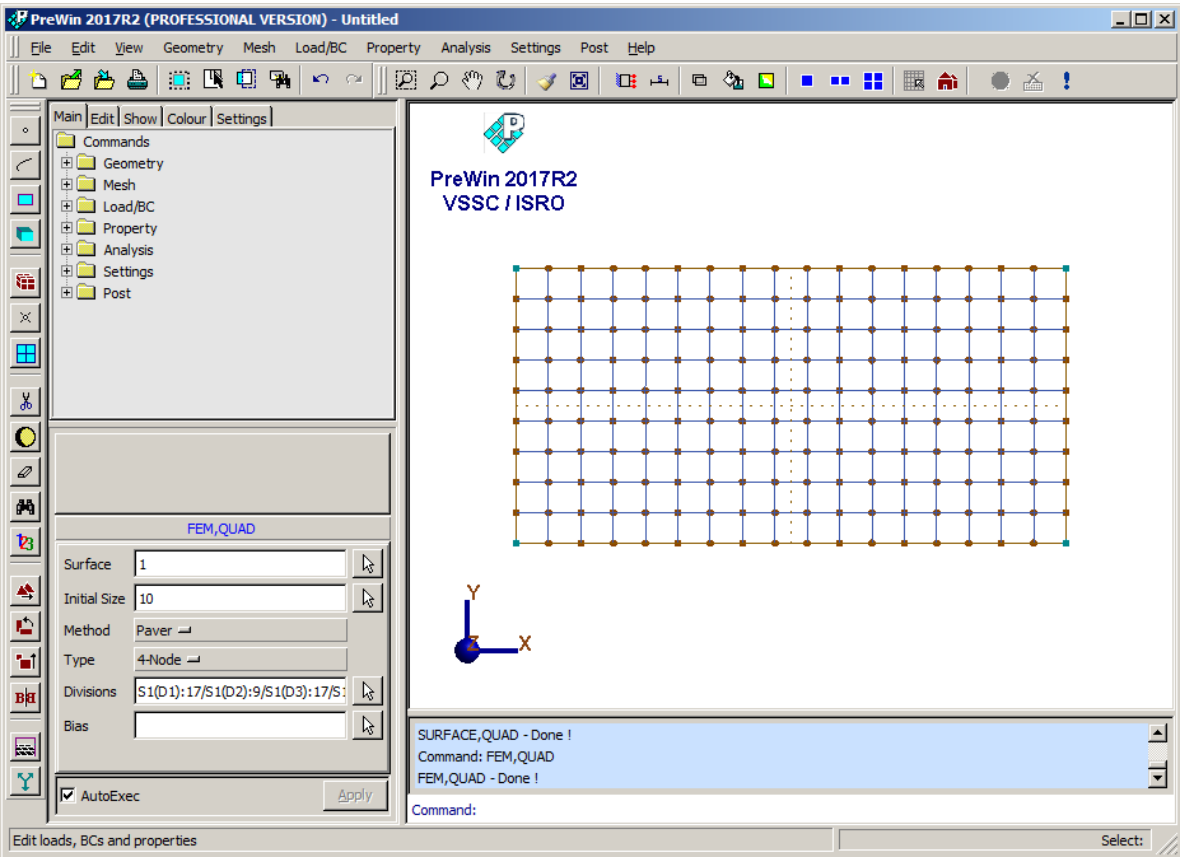
Menu : Mesh → FE Mesh →QUAD

Command : FEM, QUAD

Parameters :

Surface	Use mouse to select the surface
Initial Size	10 (An approximate initial size for laying out subdivisions)
Method	Paver
Type	4-node
Divisions	Left/Right click on edge to set subdivision S1(D1):17/S1(D2):9/S1(D3):17/S1(D4): 9

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



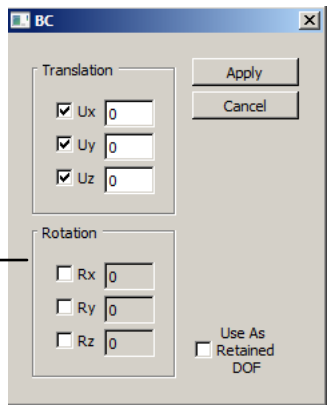
4. Specify displacement boundary conditions

Menu : Load/BC → Displacement BC → Add

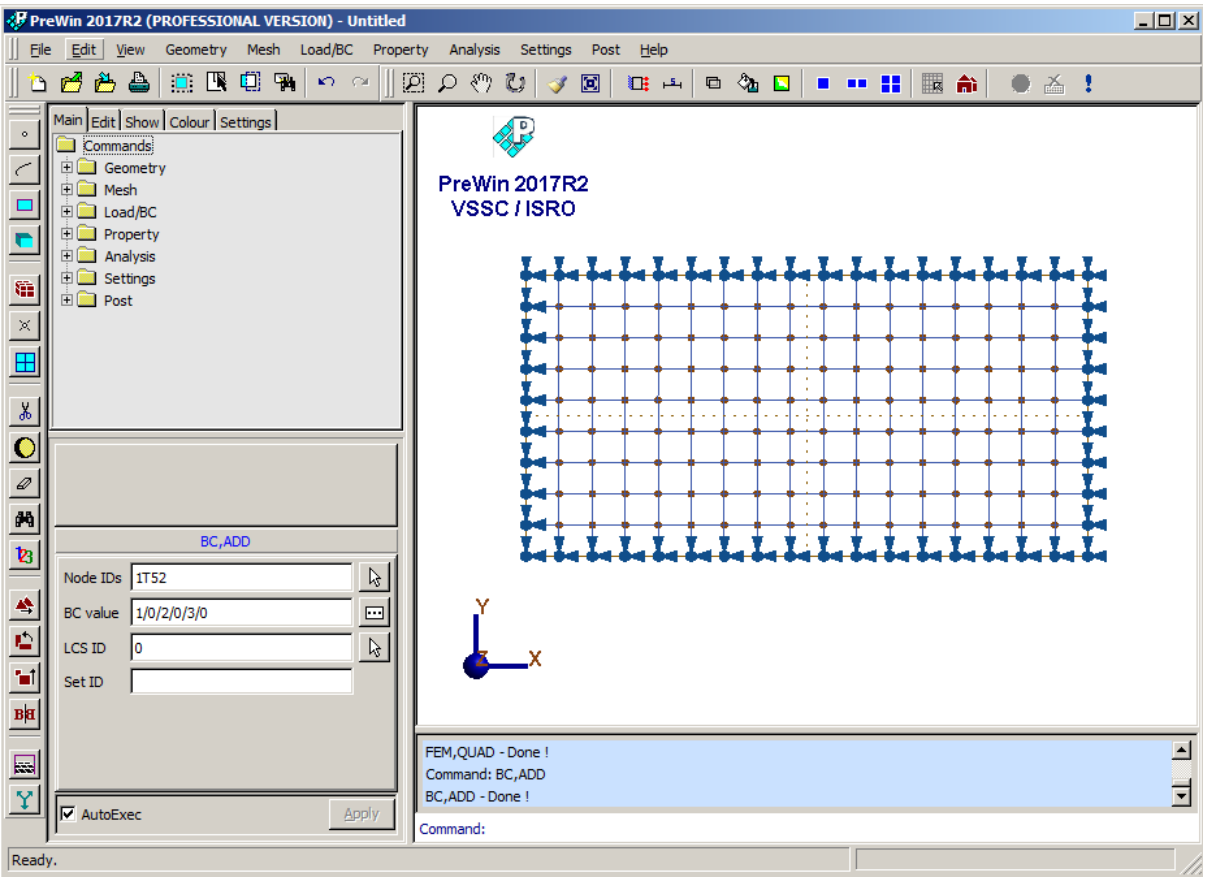
Command : BC,ADD

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.



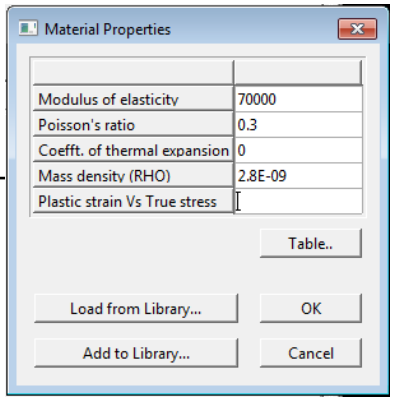
5. Specify material properties

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. Define the expression

Menu : Property → Function → Expression

Command : FUNCTION, EXPRESSION

Parameters :

Set ID	1
Data	-(.05*sin(3.14*X/200)*sin(3.14*Y/100))

8. Add pressure load

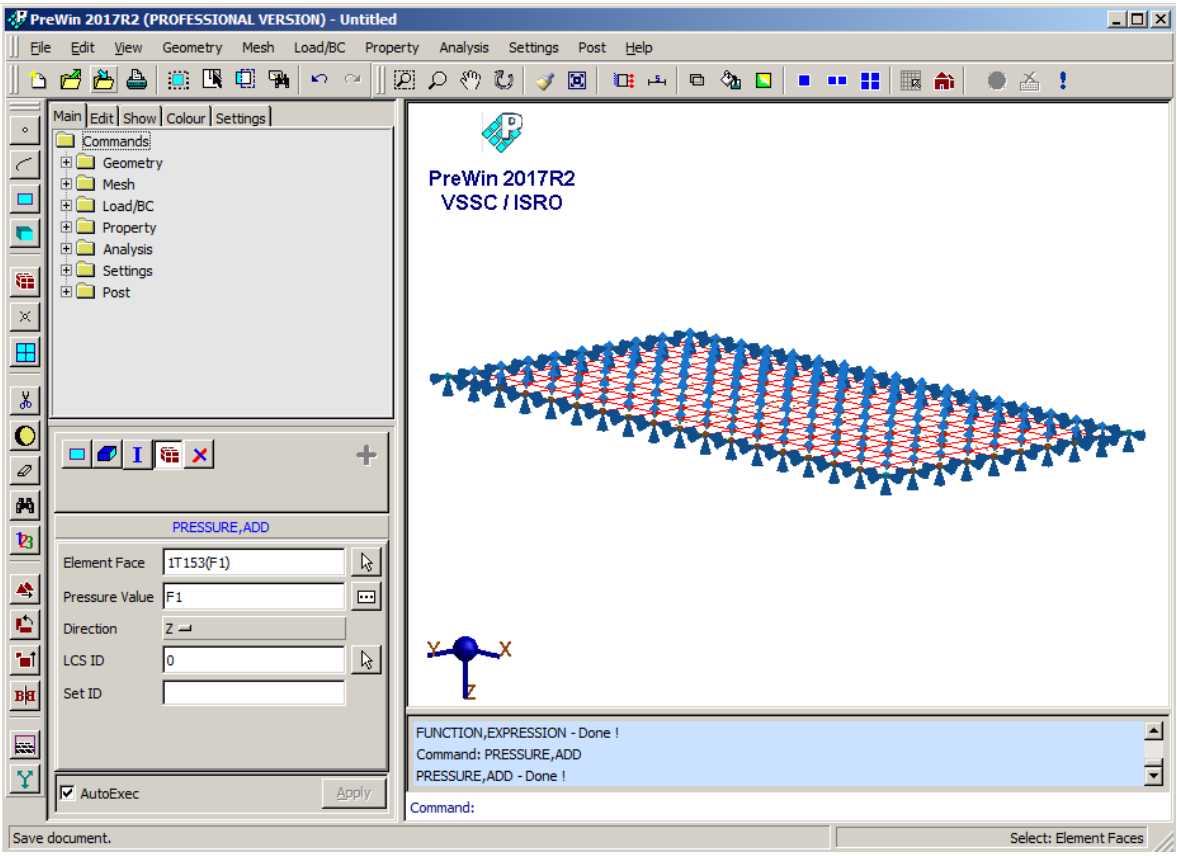
Menu : Load/BC → Pressure → Add

Command : PRESSURE, ADD

Parameters :

Element Face	ALL
Pressure Value	F1 (Select Expression1)
Direction	Z
LCS ID	0
Set ID	1

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



Note : - The varying parameters (such as pressure value/s thickness value/s) is specified in the dialog box must contain a parenthesis. The trigonometric functions need to specified in lower case letters (eg: sin( )). The arguments of these functions must be in upper case letters. If these functions are described with respect to a local coordinate system, then it must be specified at the end of the expressions in square brackets.

9. Set the analysis type

Menu : Analysis → Analysis Options

Command : ANTYPE, SET

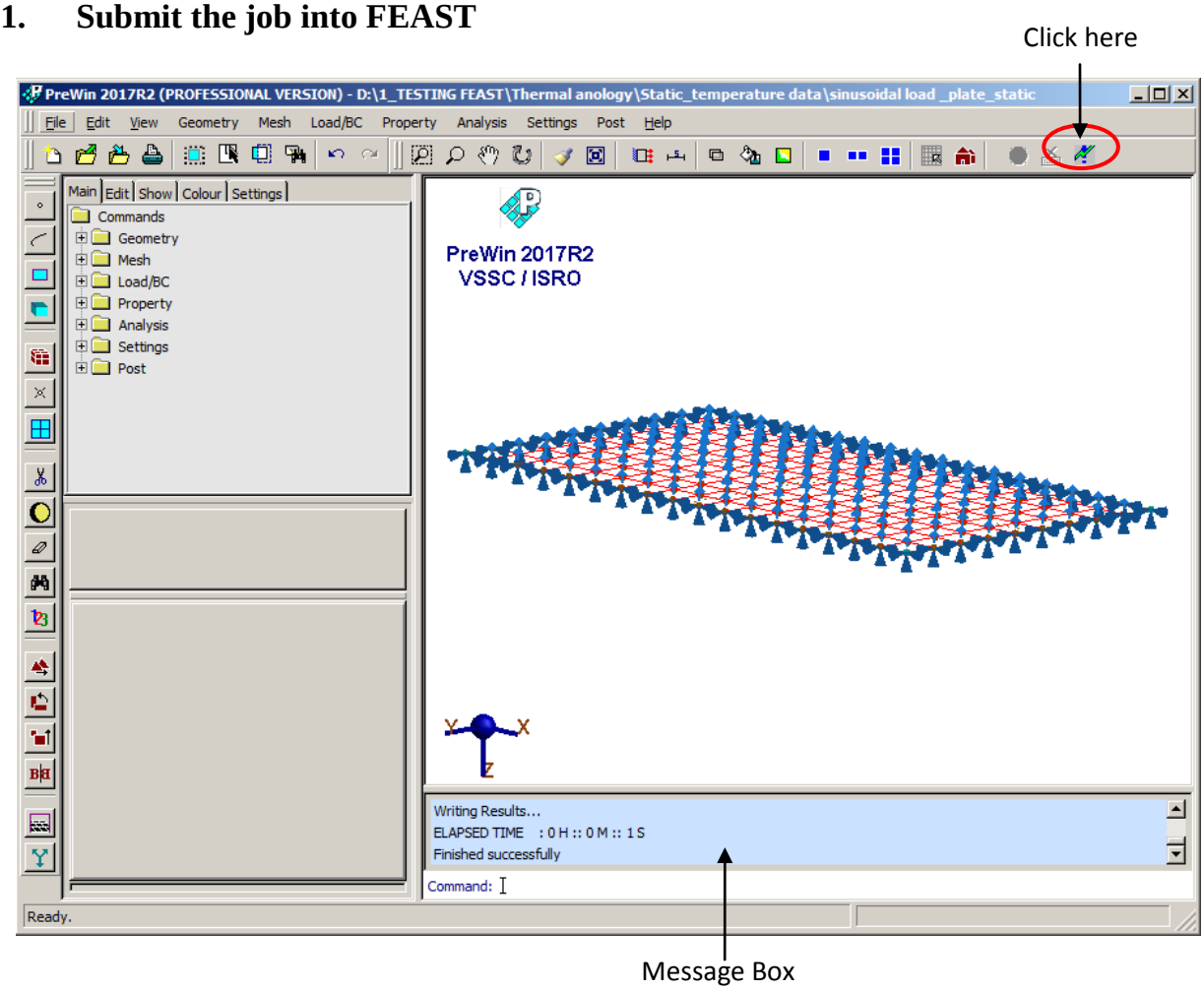
Parameters :

Analysis Type	Static

10. Save the project model

Menu : File → Save

11. Submit the job into FEAST



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

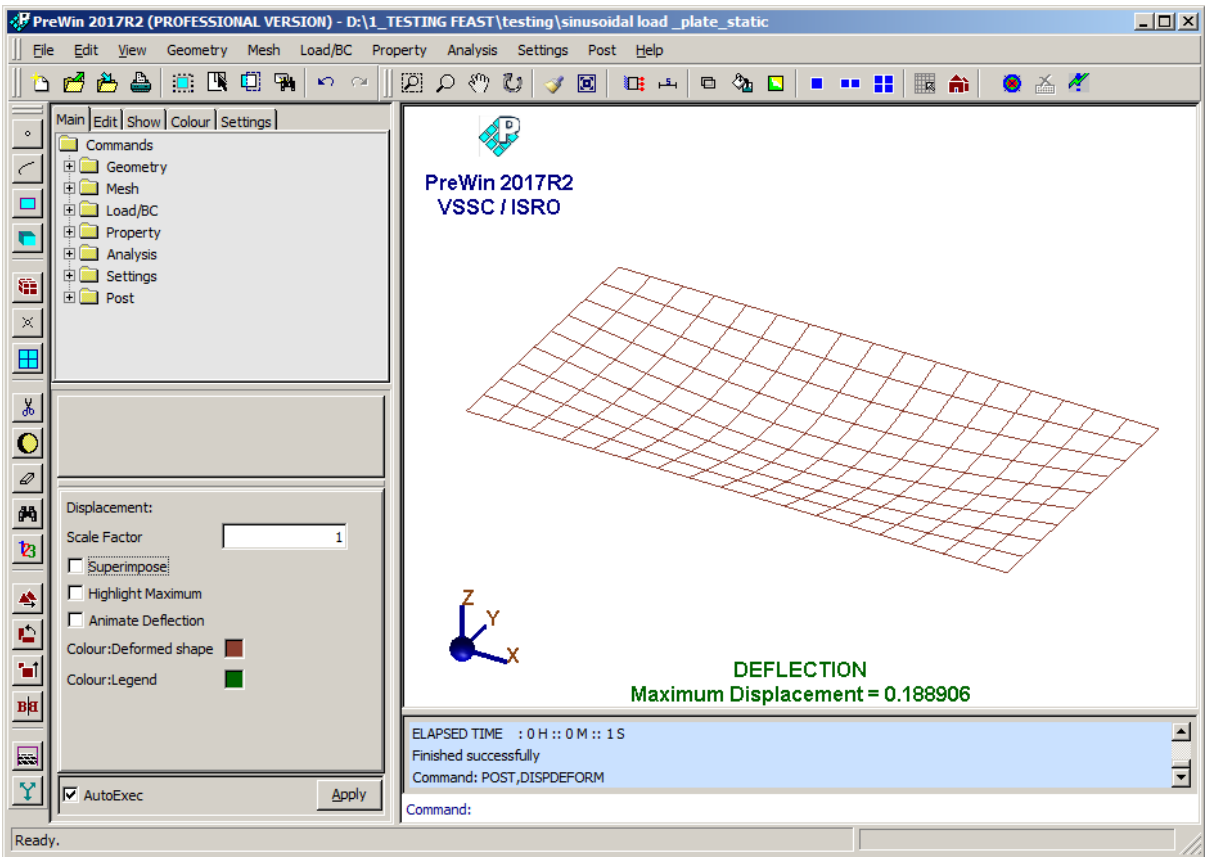
12. Perform post processing

a) Deformed shape

Menu : Post → Deformed Shape

Command : POST, DISPDEFORM

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



b) Displacement contour

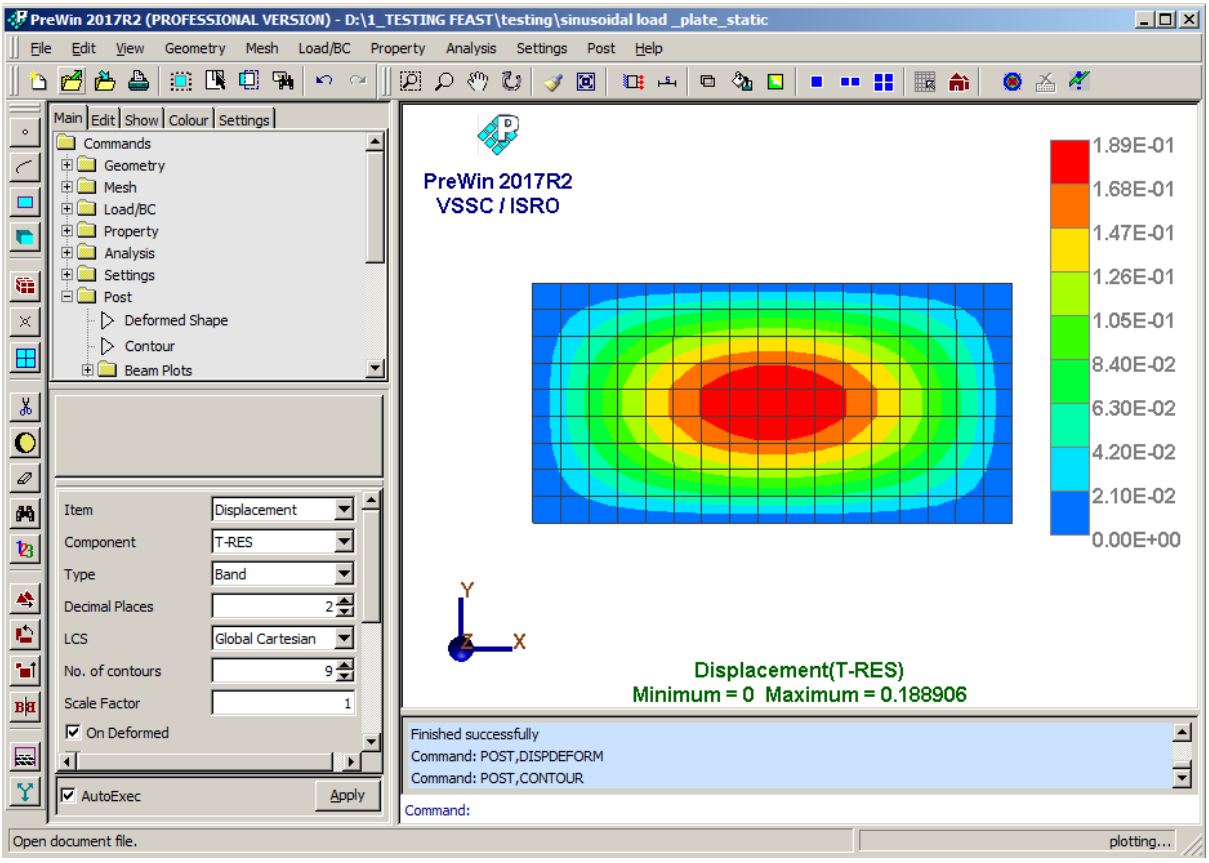
Menu : Post → Contour

Command : POST, CONTOUR

Parameters :

Item	Displacement
Component	T-RES
Type	Band
Decimal Places	3
LCS	Global Cartesian
No. of contours	9

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



c) Stress contour

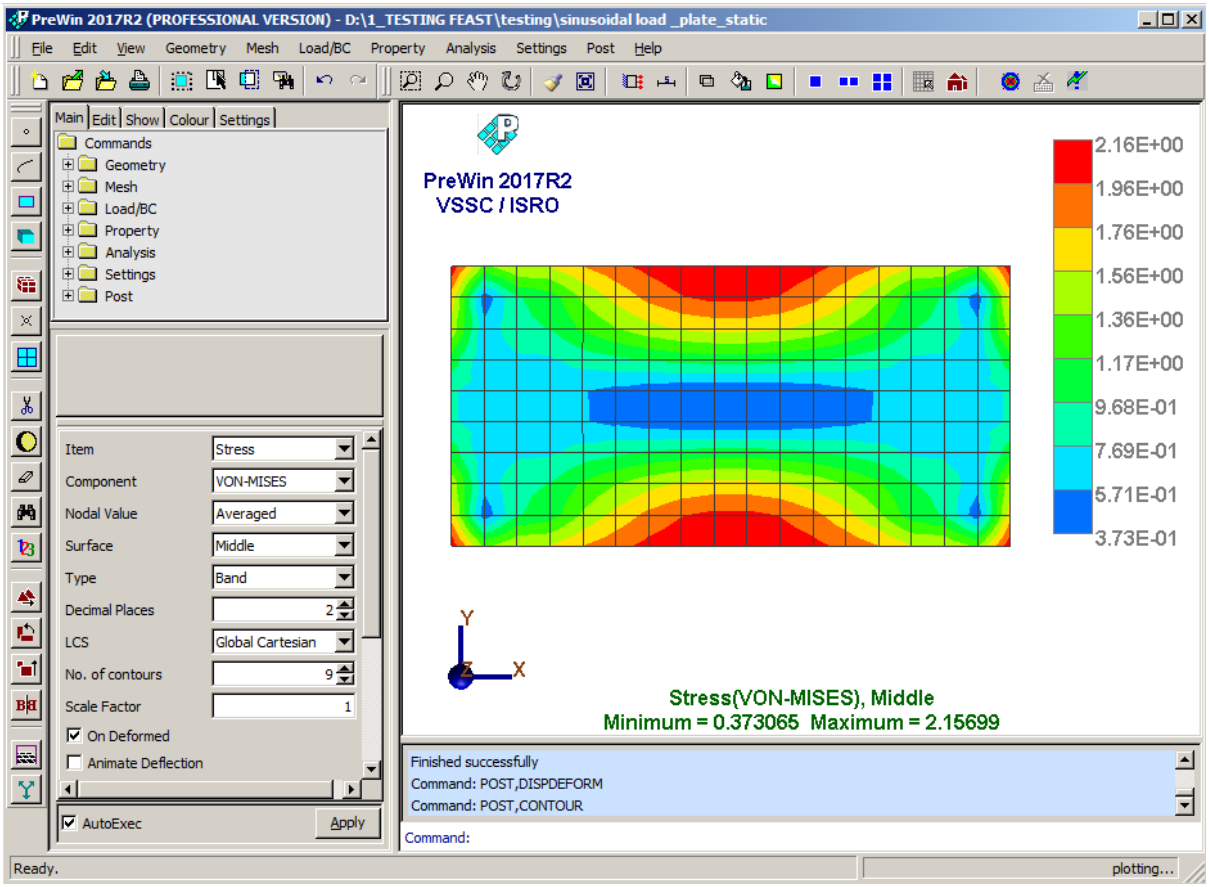
Menu : Post → Contours

Command : POST, CONTOUR

Parameters :

Item	Stress
Component	Von mises
Nodal Value	Averaged
Surface	Middle
Type	Band
Decimal places	3

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



d) Graph plot

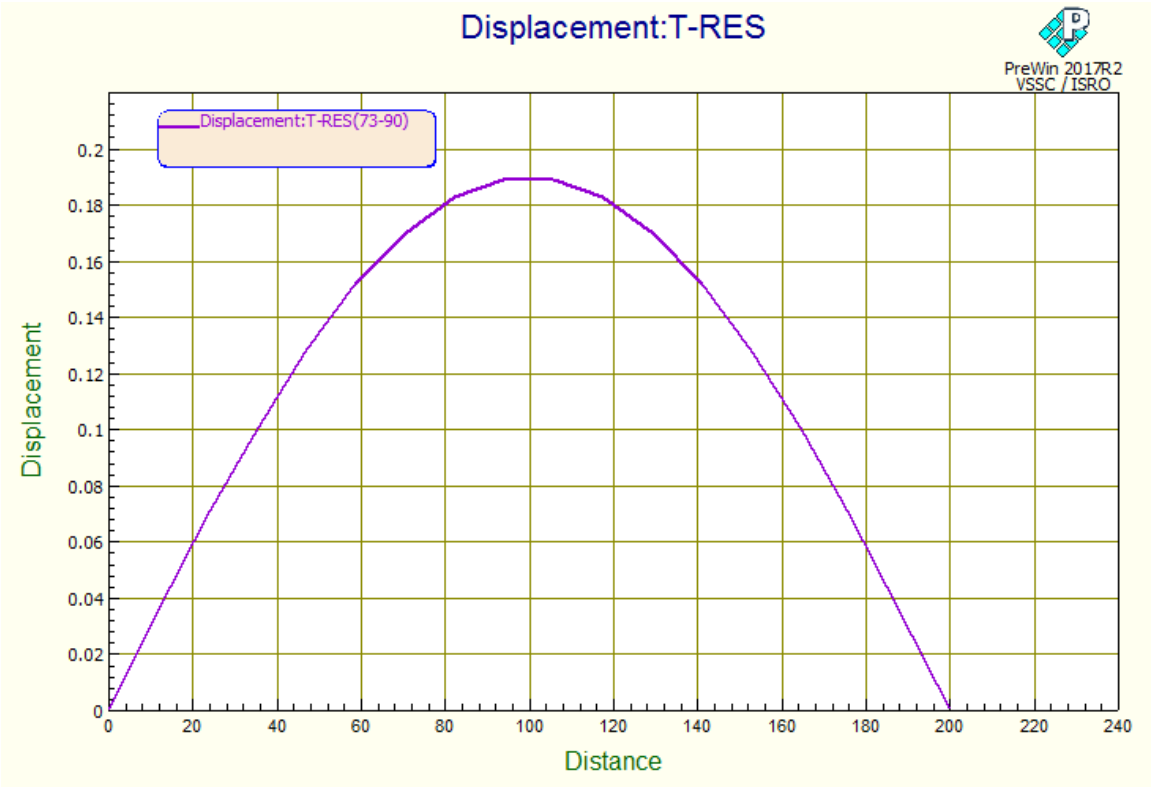
Menu : Post → Path Plot

Command : POST, PATHPLOT

Parameters :

Item	Displacement
Component	T-RES
Nodal Path	73T90

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



e) View results

Stress

Menu : Post → View Results

Command : POST, VIEWRESULTS

Parameters :

Item	Stress
Surface	Middle
Nodes	All
LCS	Global Cartesian

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

Stress

	Node ID	SIGMA-X	SIGMA-Y	SIGMA-Z	TAU-XY	TAU XZ	TAU YZ	PRINCIPAL 1	PRINCIPAL 2	PRINCIPAL
1	1	0	0	0	0	0.727699	0.664552	1.33096	-0.265508	-1.065
2	2	0	0	0	0	0.733803	0.127003	0.769105	-0.0242041	-0.7449
3	3	0	0	0	0	0.713566	-0.375508	0.968065	-0.148595	-0.8194
4	4	0	0	0	0	0.647412	-0.431978	0.983977	-0.178604	-0.8053
5	5	0	0	0	0	0.561316	-0.57522	1.10897	-0.220956	-0.8880
6	6	0	0	0	0	0.458053	-0.675363	1.18603	-0.211846	-0.9741
7	7	0	0	0	0	0.338451	-0.758311	1.24363	-0.170916	-1.072
8	8	0	0	0	0	0.20805	-0.814331	1.26913	-0.109863	-1.159
9	9	0	0	0	0	0.0703113	-0.84306	1.26283	-0.0378472	-1.224
10	10	0	0	0	0	-0.069629	-0.843163	1.22529	0.0374815	-1.262
11	11	0	0	0	0	-0.207442	-0.814633	1.15978	0.109557	-1.269
12	12	0	0	0	0	-0.337987	-0.75879	1.07338	0.170727	-1.244
13	13	0	0	0	0	-0.457799	-0.675977	0.97493	0.211824	-1.186
14	14	0	0	0	0	-0.561321	-0.57592	0.888785	0.221118	-1.10

Copy Close

f) Arrow plot

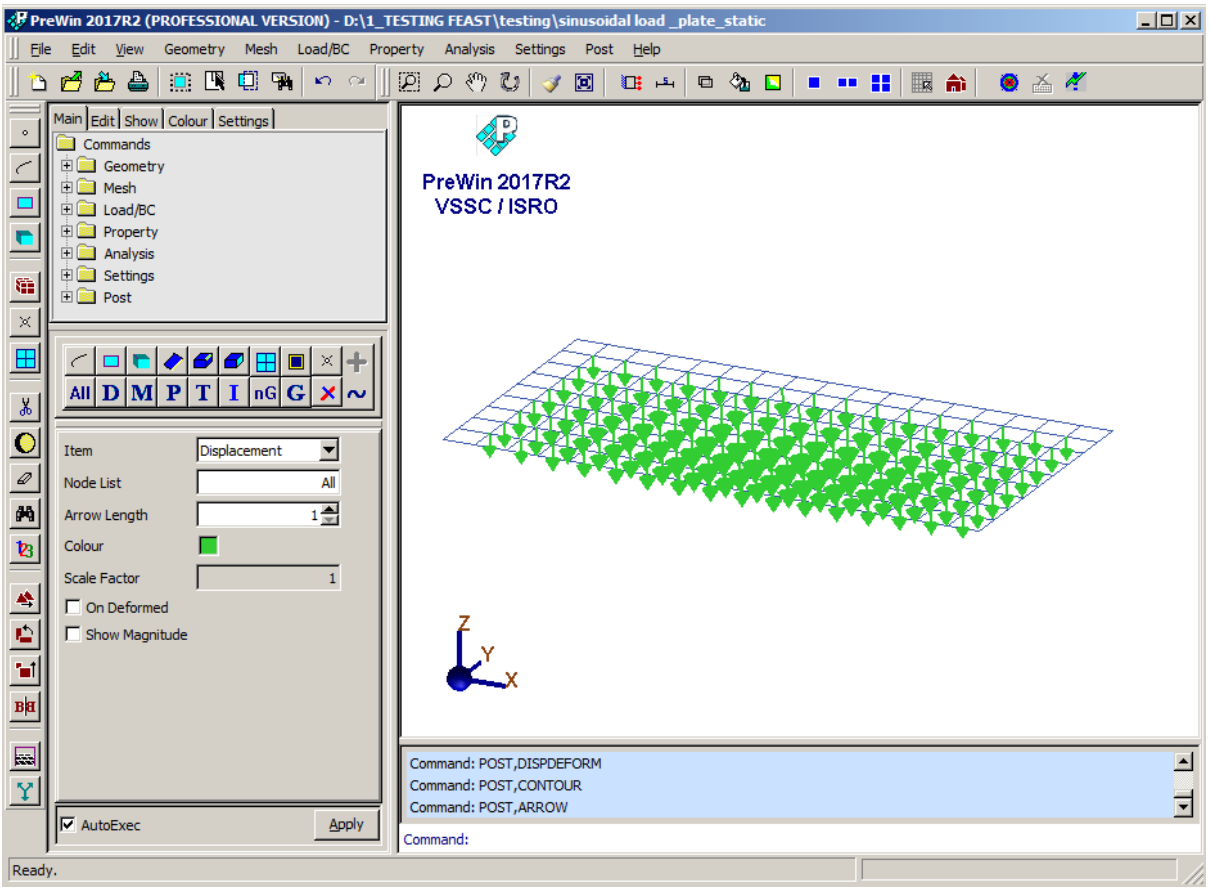
Menu : Post → Vector

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 can be seen in *.OUT