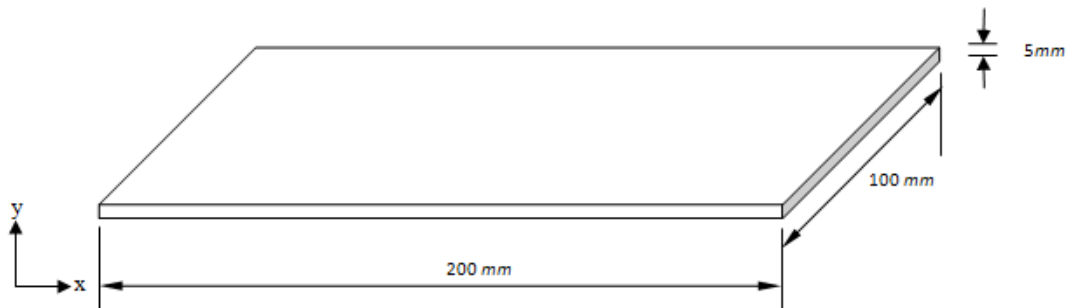


## MODAL FREQUENCY RESPONSE ANALYSIS OF A CANTILEVER PLATE SUBJECT TO HARMONIC PRESSURE LOAD

### Objective

- To model an isotropic rectangular plate with dimensions as in **Figure 1**.
- To define a harmonic pressure load of 0.1kPa normal to the plate
- To study the frequency response of the plate using FEAST<sup>SMT</sup>



**Figure 1: Isotropic rectangular plate**

### Problem definition

Plate dimensions are shown in the figure.

Analysis Type: Frequency response

Modulus of elasticity,  $E$  = 70000MPa

Poisson's Ratio,  $\nu$  = 0.3

Specific gravity = 2.8

Loads and boundary conditions:

- Pressure load .1kPa is applied normal to the plate.
- Left edge of the plate is assumed to be fixed.

## PROCEDURE

### 1. Creation of geometry

- Create 4 points at (0,0,0) (200,0,0) (0,100,0) and (200,100,0)

Commands : POINT, ADD

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

Parameters : (To be filled by the user)

<b>Coordinate Data</b>	0/0/0
<b>Coordinate Type</b>	0
<b>Entity ID</b>	1

- Create quadrilateral surface on four points

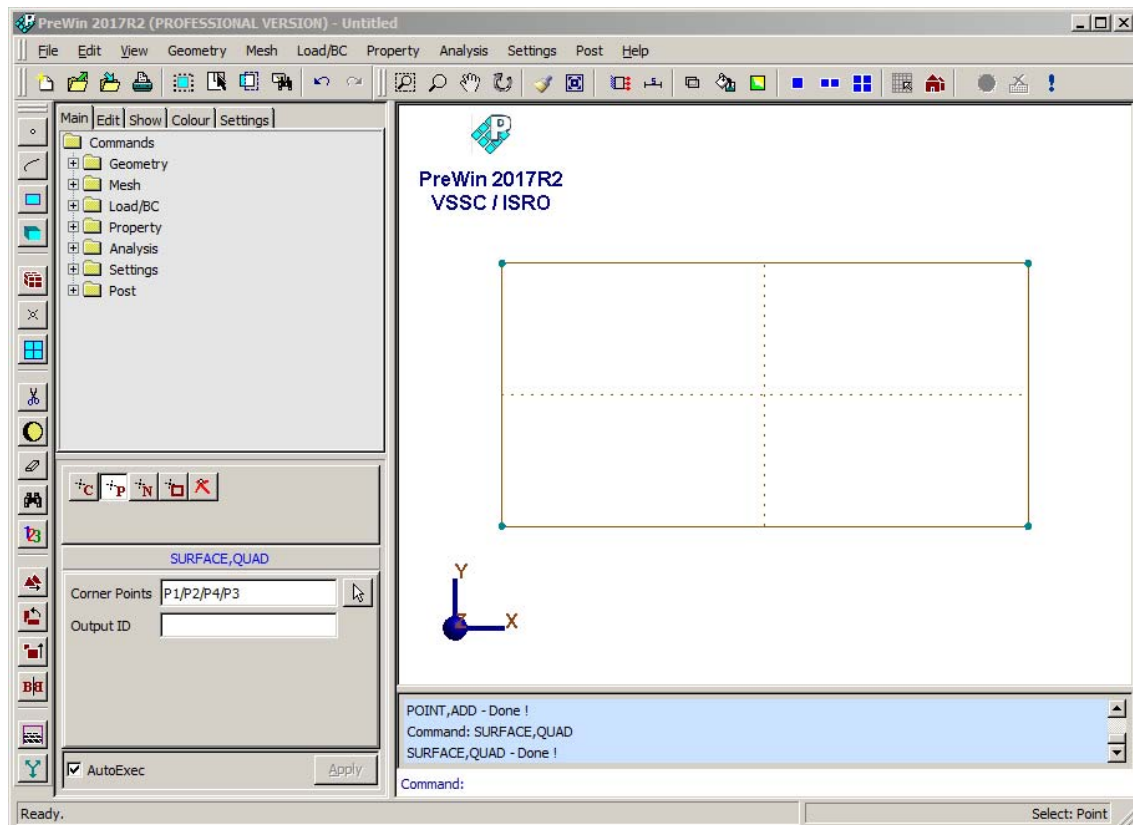
Command : SURFACE, QUAD

Menu : Geometry → Surface → Create → Quad Surface

Parameters : (To be filled by the users)

<b>Point Data</b>	Use mouse to pick the points
<b>Entity ID</b>	1

At the end of this operation, the screen looks like this.



## 2. Meshing using quadrilateral elements

Command : FEM, QUAD

Menu : Mesh → FE Mesh → QUAD

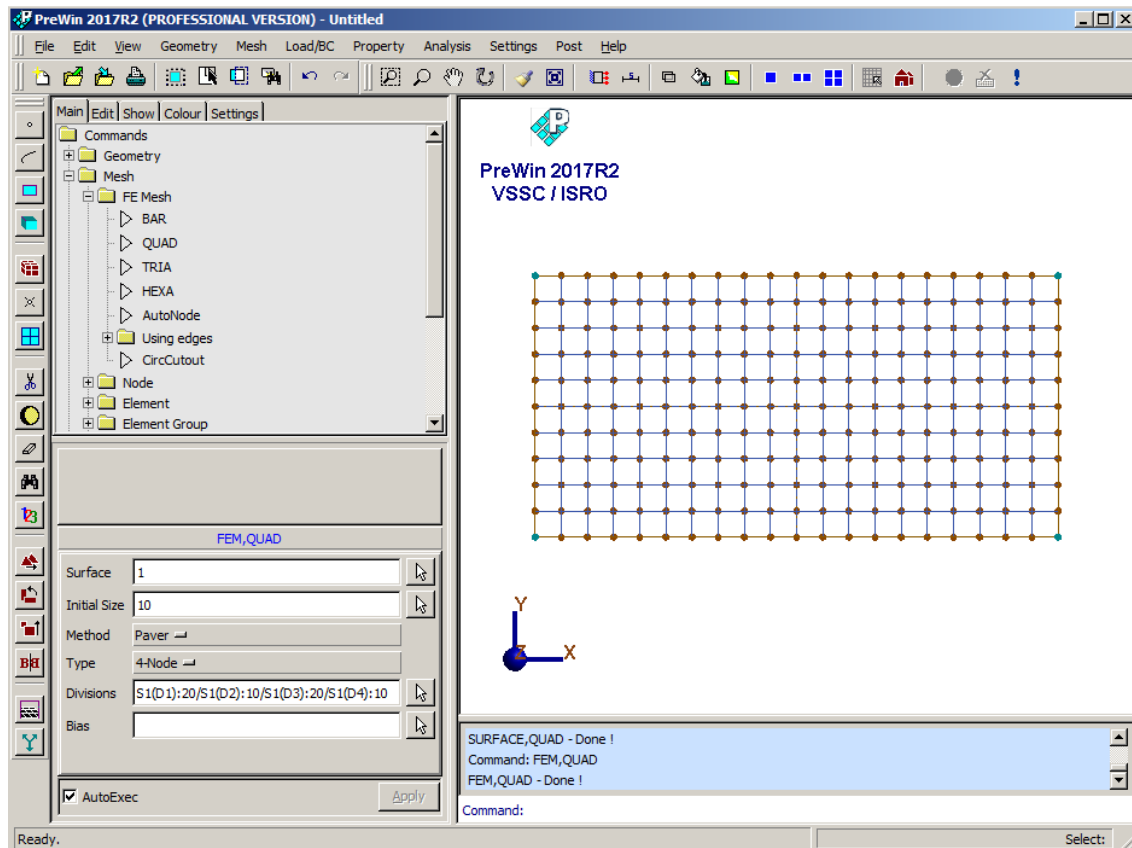
Parameters :

<b>Surface</b>	Use mouse to select the surface
<b>Initial size</b>	10
<b>Method</b>	Paver
<b>Type</b>	4 node
<b>Divisions</b>	S1(D1):20/S1(D2):10/S1(D3):20/S1(D4):10

Note:

Divisions along each edge can be varied by selecting the corresponding edges and right/left clicking the mouse point to adjust the subdivisions. Eg: S1(D1)/20 shows 20 sub divisions along edge1 & S1(D2)/10 shows 10 sub divisions along edge2 of surface1.

At the end of this operation, the screen looks like this.

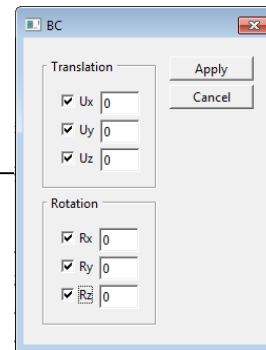


### 3. Specify displacement boundary conditions

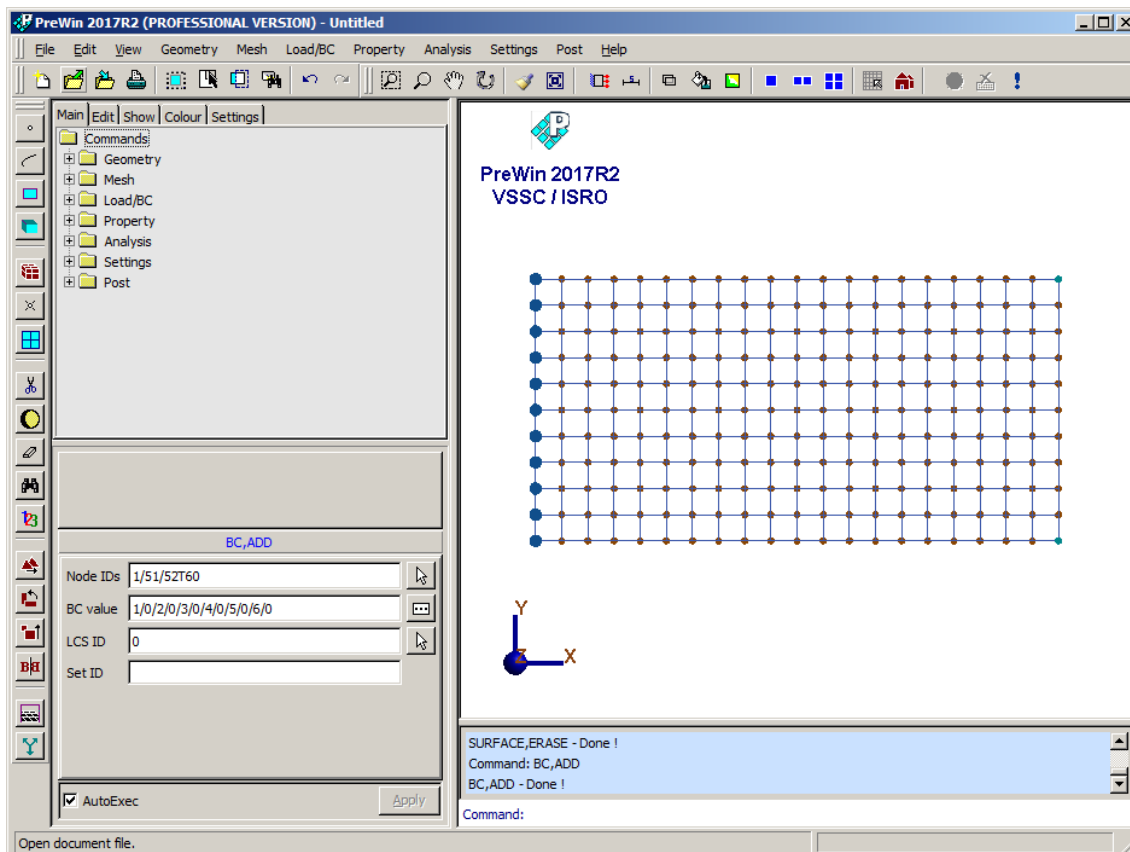
Command : BC, ADD

Menu : Load/BC → Displacement BC → Add

<b>Node ID</b>	Select the nodes on the left side boundary edges
<b>BC Value</b>	1/0/2/0/3/0/4/0/5/0/6/0
<b>LCS ID</b>	0
<b>Set ID</b>	1



At the end of this operation, the screen looks like this.




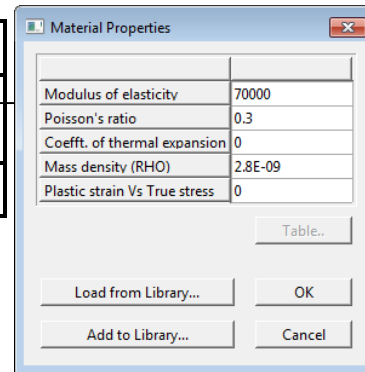
#### 4. Specify material properties

Command : MATERIAL, ISO

Menu : Property → Material → Isotropic → Add

Parameters :

<b>Element IDs</b>	All
<b>Material Data</b>	70000/0.3/0/2.8E-09/0 
<b>Material ID</b>	1



## 5. Specify shell thickness

Command : THICKNESS, ADD

Menu : Property → Physical → Thickness → Add

Parameters

<b>Element IDs</b>	All
<b>Thickness</b>	5
<b>Set ID</b>	1

## 6. Specify pressure load

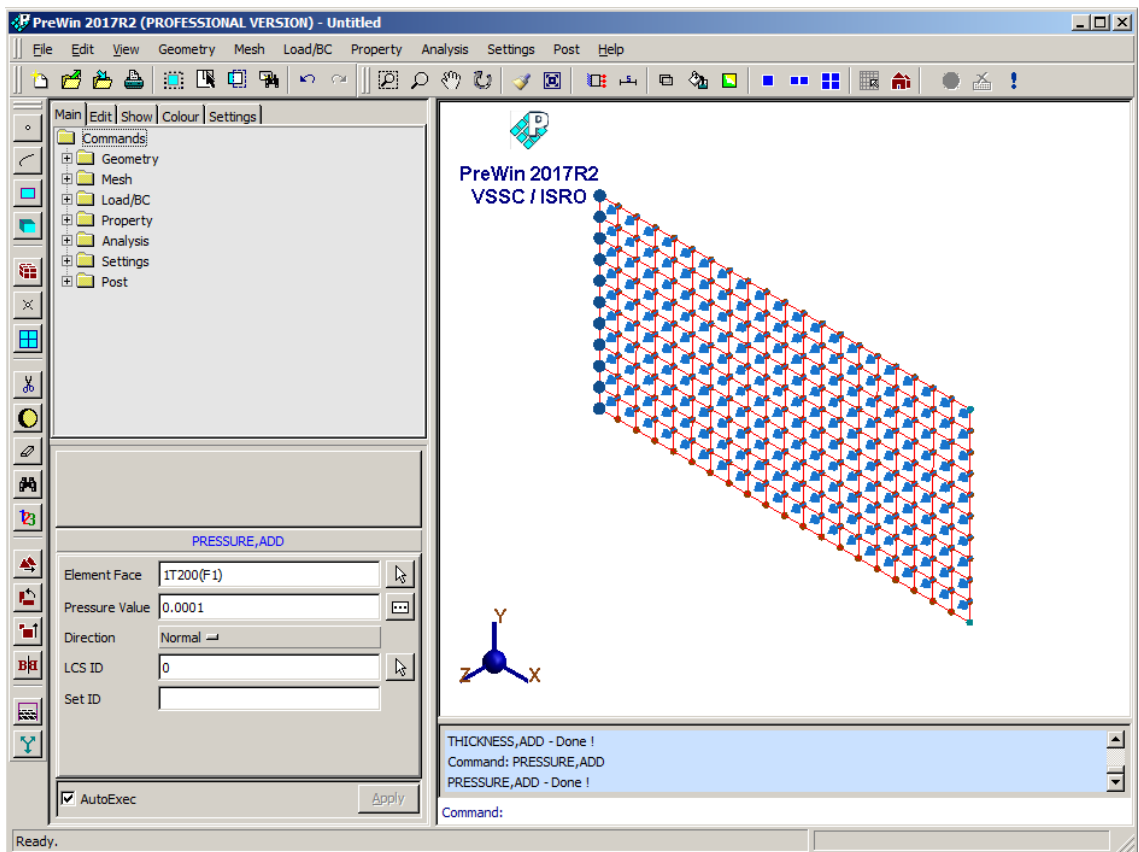
Command : PRESSURE, ADD

Menu : Load/BC → Pressure → Add

Parameters :

<b>Element face</b>	All
<b>Pressure value</b>	.0001
<b>Direction</b>	Normal
<b>LCS ID</b>	0
<b>Set ID</b>	1

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



## 7. Specify analysis type

Commands : ANTYPE, SET  
 Menu : Analysis → Analysis Type  
 Parameters : (To be filled by the user)

<b>Analysis Type</b>	Frequency Response
----------------------	--------------------

## 8. Specify analysis options

Commands : ANOPTION, SET  
 Menu : Analysis → Analysis Options  
 Parameters : (To be filled by the user)

<b>Linear Solver</b>	Multi Frontal
<b>Eigen Solver</b>	Lanczos

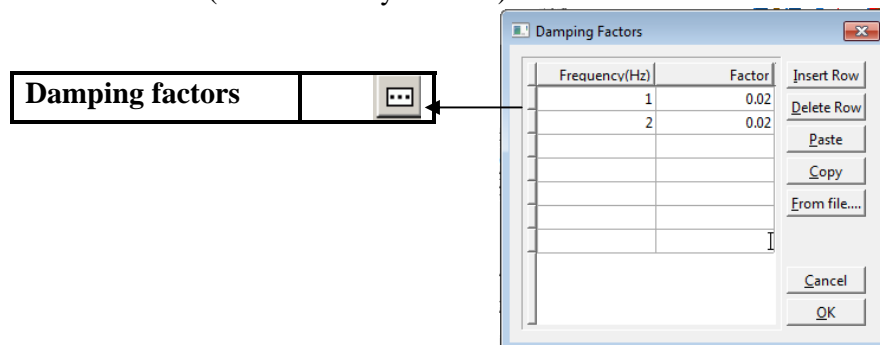
## 9. Specify frequency response general properties

Commands : FREQGEN, ADD  
 Menu : Analysis → Frequency Response → General → Add  
 Parameters : (To be filled by the user)

<b>Response Extraction</b>	Auto
<b>No. of modes</b>	10
<b>Node list</b>	All
<b>Start Frequency</b>	10
<b>Maximum Frequency</b>	1000
<b>Finer Increment</b>	1
<b>Coarser Increment</b>	10

## 10. Specify damping factor

Commands : EXPFRDAMP, ADD  
 Menu : Analysis → Frequency Response → Damping → Add  
 Parameters : (To be filled by the user)



## 11. Save the data file

Menu : File → Save

## 12. Submit the job into FEAST

Menu : Analysis → Run Solver





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

### 13. Post processing

#### a) Displacement Plot

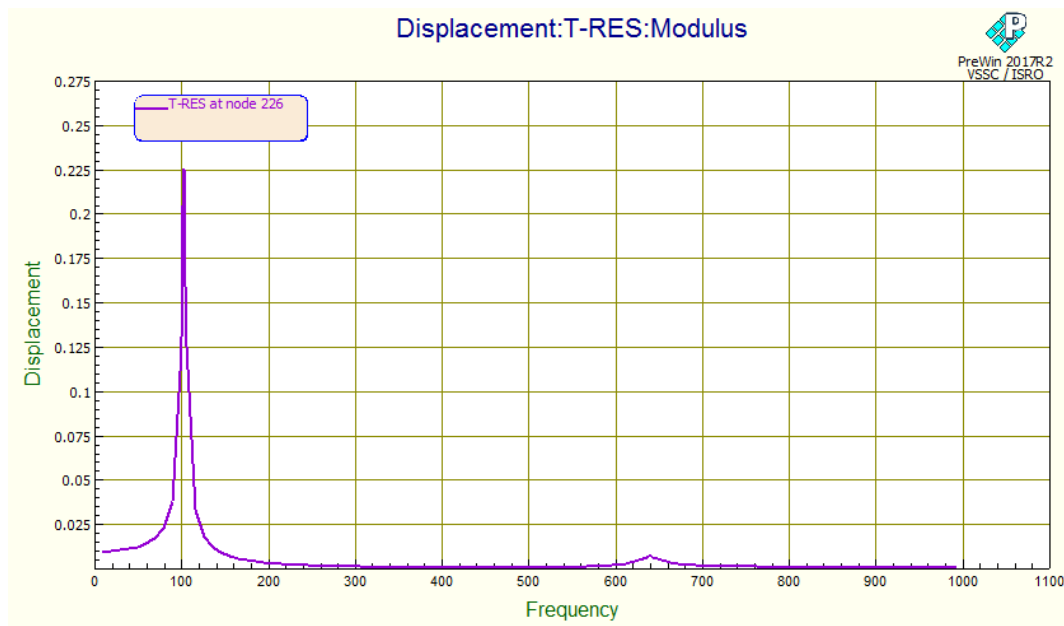
Command : POST, HISTORYPLOT

Menu : Post → History plot

Parameters :

<b>Item</b>	Displacement
<b>Component</b>	T-RES
<b>Nodes</b>	226
<b>Complex As</b>	Modulus

At the end of this operation, the screen looks like this.



Velocities and accelerations at desired nodes and degrees of freedom can be plotted using similar procedure.

## b) View Results

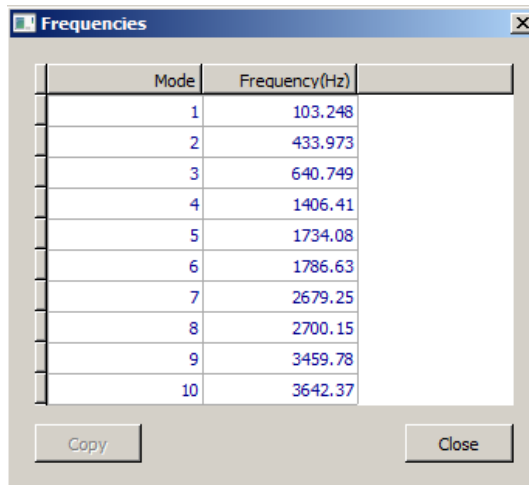
Command : POST, VIEWRESULTS

Menu : Post → View results

Parameters :

<b>Item</b>	Frequencies
-------------	-------------

At the end of this operation, an output window like this is obtained.



Mode	Frequency(Hz)
1	103.248
2	433.973
3	640.749
4	1406.41
5	1734.08
6	1786.63
7	2679.25
8	2700.15
9	3459.78
10	3642.37

Copy Close

c) Output file can be seen in \*.OUT