VISCOELASTIC ANALYSIS

Visco elastic analysis of axisymmetric element has been carried out with S139 solid motor propellant grain analysis. The motor consist of Propellant, Insulation and Casing. The analysis is carried out for 0 to 3.1536e+07sec time and 366 time steps. The finite element mesh as in Figure 1 is modeled.

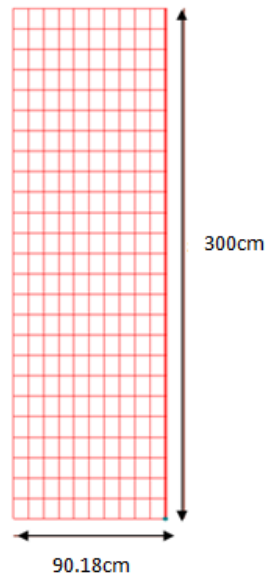


Figure 1: FE model

Procedure

1. **Create key points (50,0,0), (138.9,300,0), (138.9,0,0), (139.4,300,0), (139.4,0,0), (140.18,300,0)**

   Commands : POINT, ADD

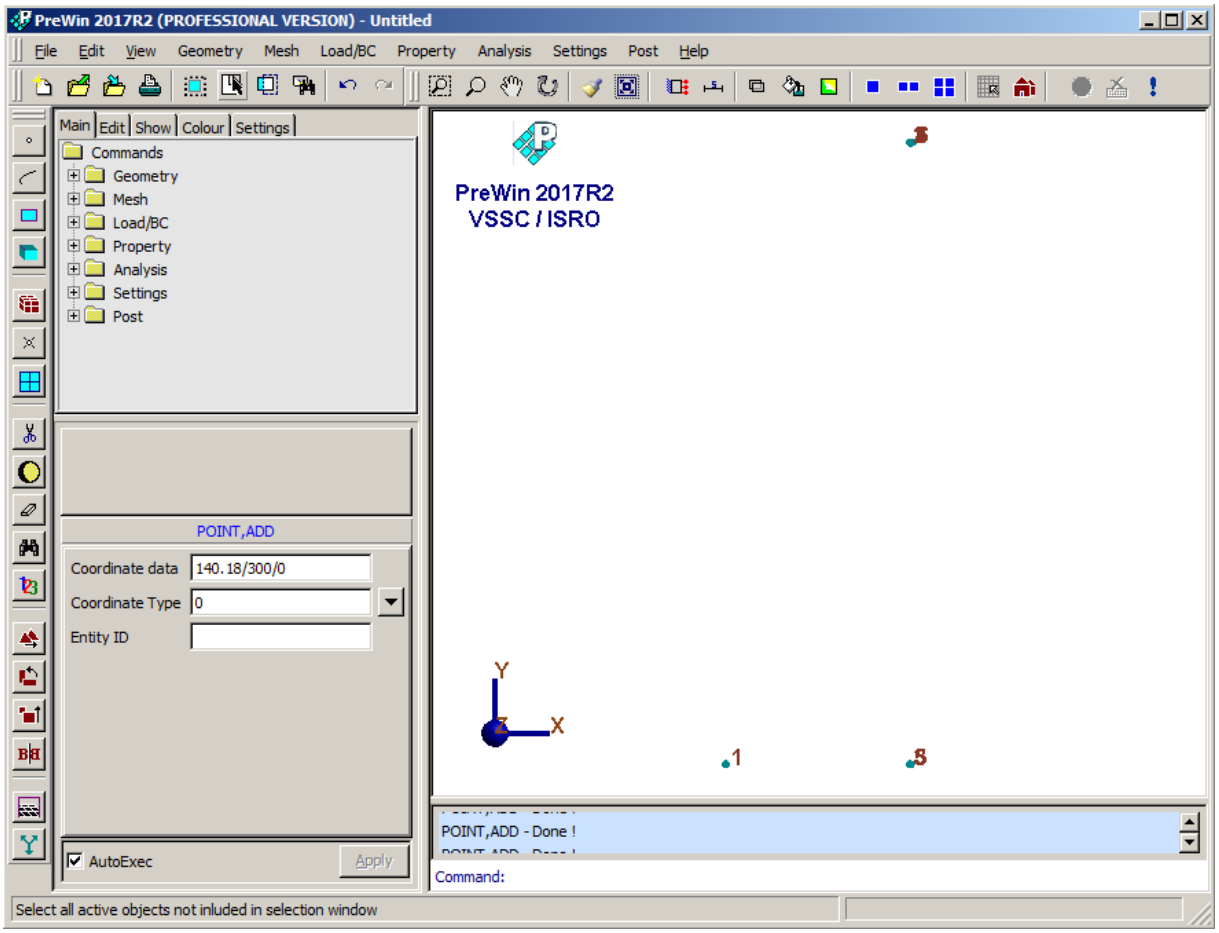
   Menu : Geometry → Key point → Create → By X/Y/Z

   Parameters : (To be filled by the user)

Coordinate Data	50,0,0
Coordinate Type	0
Entity ID	1

Similarly create all the six key points mentioned above.

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



Note:

'Zoom in' the points created using icon on top menu tips to visible all the six points. Label the points using icon on left menu tips to visible the point id's.

2. Create quadrilateral surface on two diagonal points

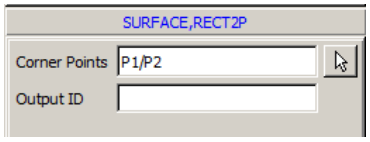
Command : SURFACE,RECT2P

Menu : Geometry → Surface → Create → Rect2P

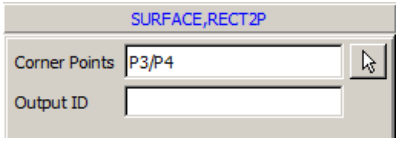
Parameters :

Point Data	Use mouse to pick the two diagonal points. P1/P2
Entity ID	1

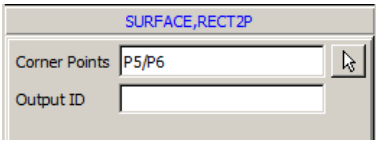
**Likewise create surfaces by changing point data in the first parameter box. For the 2nd surface i.e. for insulation, the point data is P3/P4 and that for casing surface the point data is P5/P6.Now total three surfaces are created for propellant, insulation and casing. Your parameter box while creating surfaces are as shown below.



Surface1, Propellant

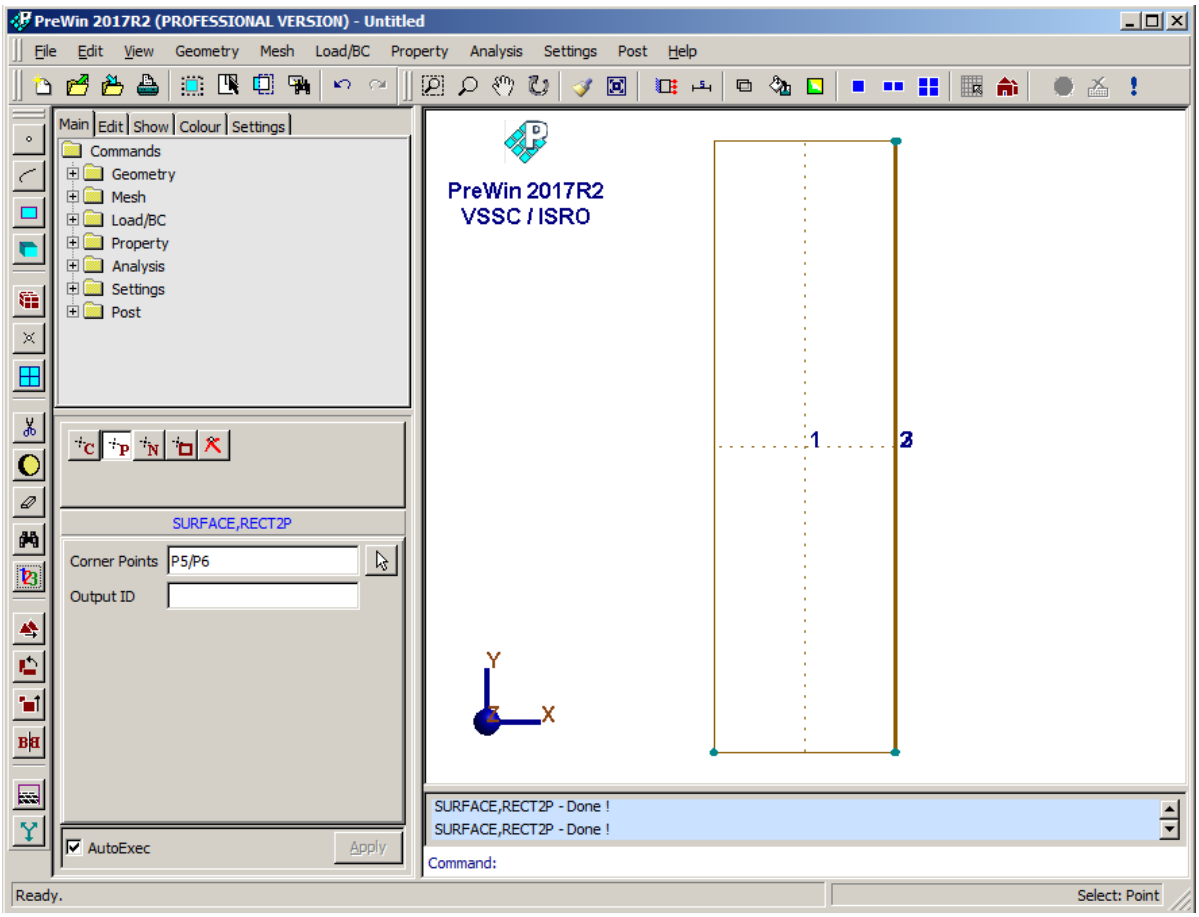


Surface2, Insulation



Surface3, Casing

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



3. Meshing using quadrilateral elements

Command : FEM,QUAD

Menu : Mesh → FE Mesh → QUAD

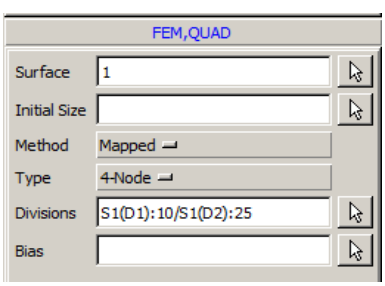
Parameters :

Surface	Use mouse to select the surface
Initial size	
Method	Mapped
Type	4-node
Divisions	S1(D1)/10

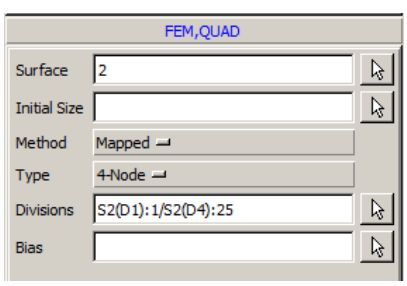
Note:

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):10 shows 10 sub divisions along edge1 & S1(D2):25 shows 25 sub divisions along edge2 of surface1.

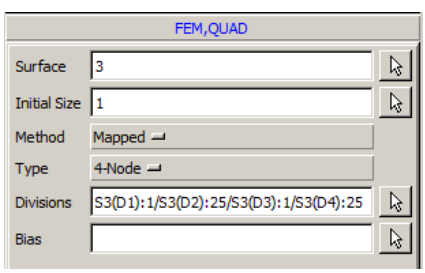
Likewise the surfaces 2 &3 are also meshed. For surface 2 divisions are given as S2(D1):1 for edge1 and S2(D2):25 for edge2. For surface 3 divisions are given as S3(D1):1 for edge1 and S3(D2):25 for edge2. Your parameter box while meshing surfaces are as shown below.



Surface1, Propellant

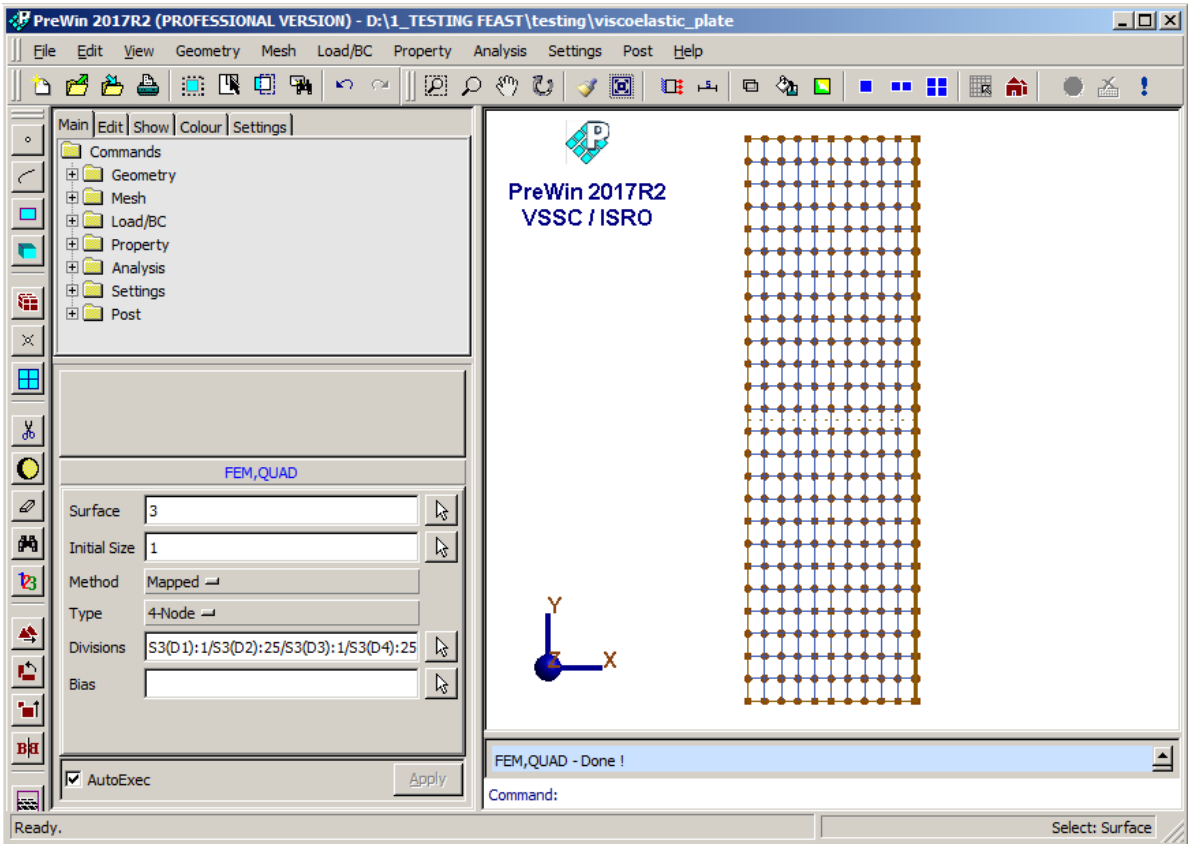


Surface2, Insulation



Surface3, Casing

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

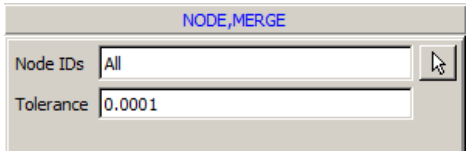


4. Merge the duplicate nodes

Command : NODE,MERGE

Menu : Mesh → Node → Modify→ Merge

Parameters :



Click 'Apply'.

5. Convert the QUAD4 element to QUAD8 element

Command : ELEMENT,CONVERT

Menu : Mesh → Element → Modify→ Convert

Parameters :

Element IDs	All
Convert to	QUAD8

6. Change the element type

Command : ELEMENT,TYPE

Menu : Mesh → Element → Modify→ Type

Parameters :

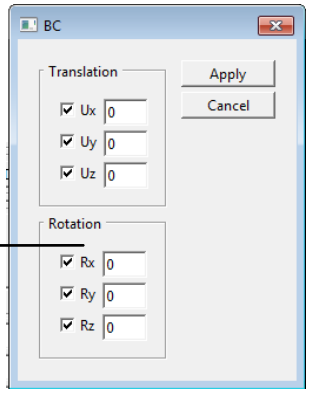
Element IDs	All
Element Type	Axisymmetric

7. Specify boundary conditions

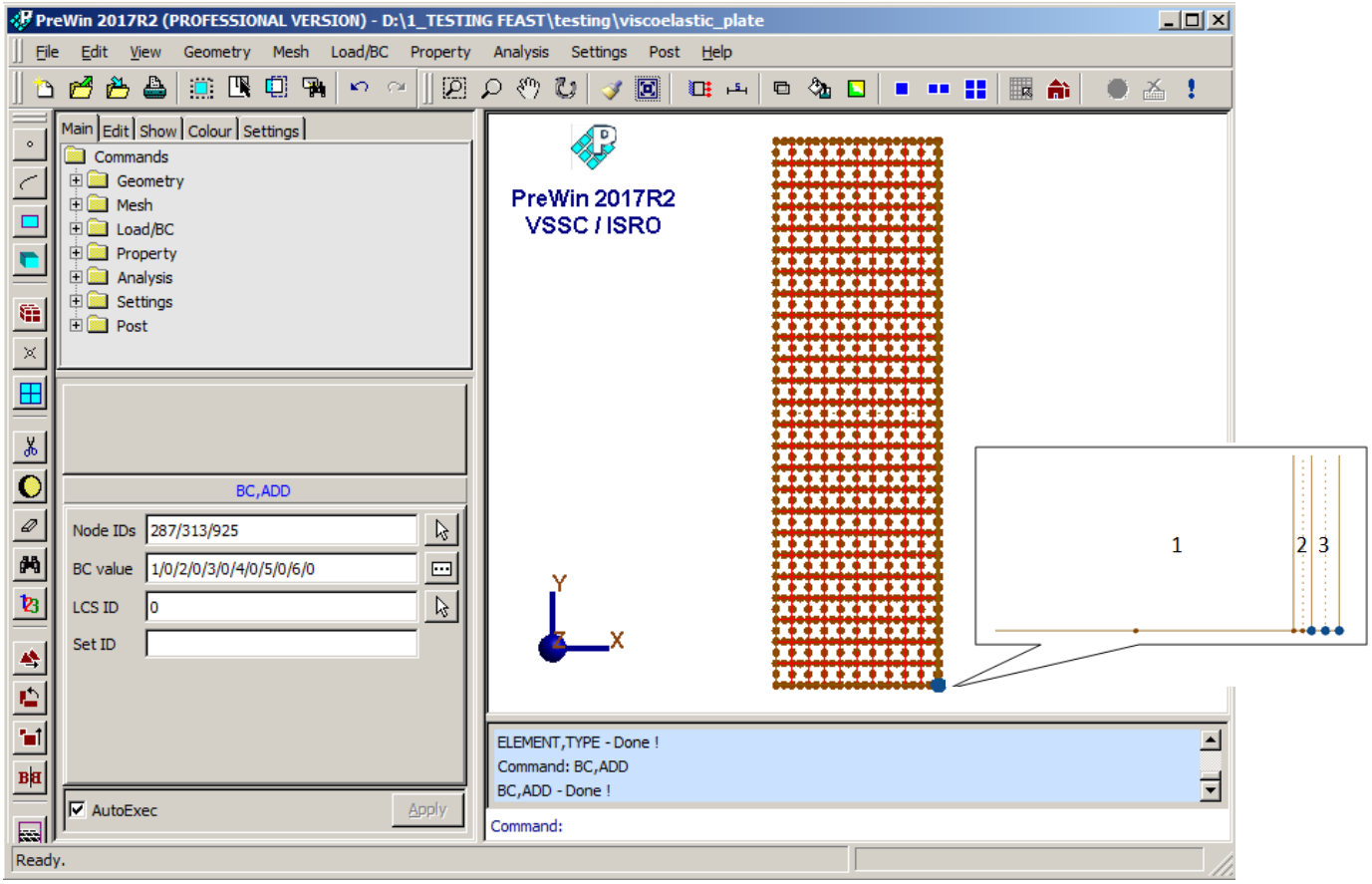
Command : BC, ADD

Menu : Load/BC → Displacement BC → Add

Node ID	Fix the bottom boundary nodes along casing. N287/N313/N925
BC Value	1/0/2/0/3/0/4/0/5/0/6/0
LCS ID	0
Set ID	1



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



8. Specify load case

Here, gravity loading of 1g load is considered.

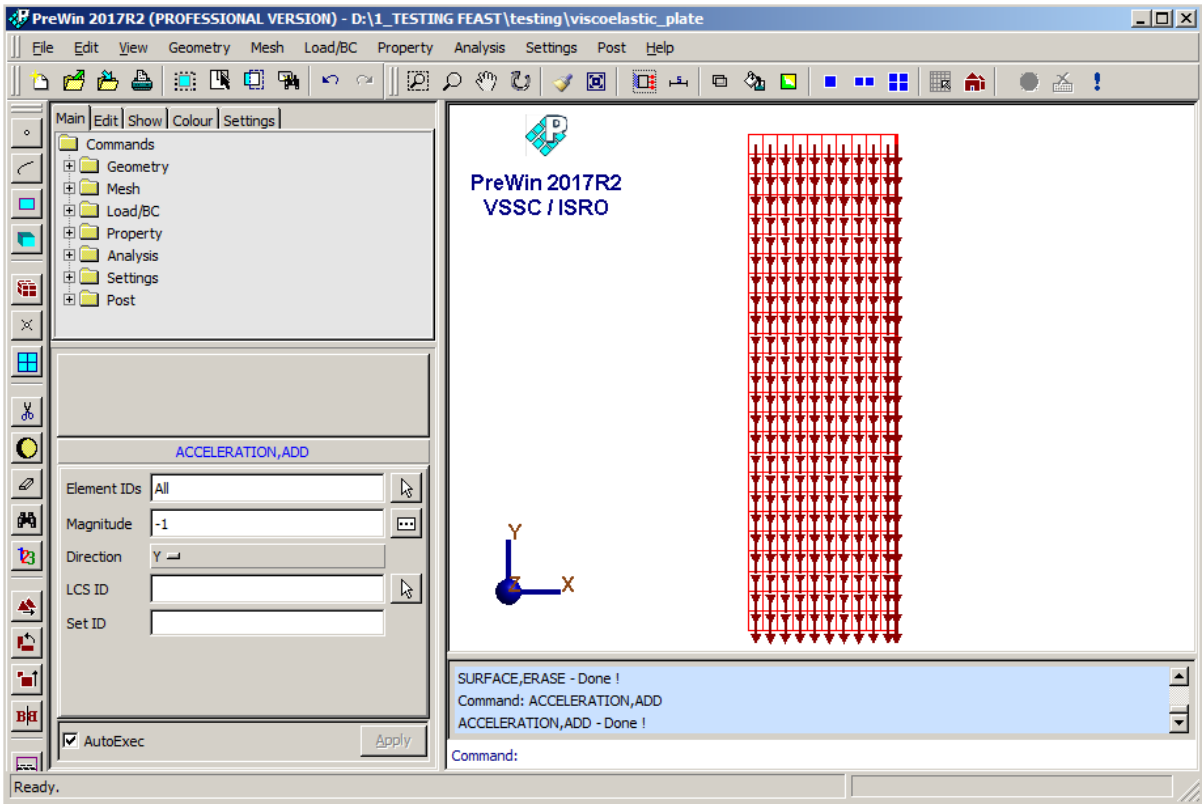
Command : ACCELERATION,ADD

Menu : Load/BC → Acceleration → Add

Parameters :

Element IDs	All
Magnitude	-1
Direction	Y
LCS ID	0
Set ID	1

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



9. Specify material properties

Note:

Visco elastic material property is considered for Propellant and elastic material property is considered for insulation and casing.

Propellant:

Instantaneous modulus, E_0 = 1594.11 ksc

Poisson's ratio = 0.4924

Coefficient of thermal expansion (α) = 0.00011 / K

Density = 1780 kg/m^3

16 term Prony series is used for the shear relaxation behavior of the propellant.

Table 2 Time Vs Modulus (E)

Time (sec)	Modulus (E) ksc
1.00E-08	1587.89
1.00E-07	1513.56
1.00E-06	1258.93
1.00E-05	831.76
0.0001	426.58
0.001	237.14
0.01	131.83
0.1	83.18
1	50.12
10	37.15
100	28.18
1000	23.99
10000	21.38
100000	21.14
1.00E+06	20.89
1.00E+07	20.4

Table 1 Prony coefficients

A_i	τ_i
-0.39	1.00E-08
52.9267	1.00E-07
129.113	1.00E-06
176.743	1.00E-05
75.19	0.0001
46.44	0.001
17.4067	0.01
15.19	0.1
4.63333	1
3.96667	10
1.48667	100
1.38	1000
0.086667	10000
0.033333	100000
0.146667	1.00E+06
0.216667	1.00E+07

Above Prony series shown in Table 1 is inserted in FEAST as a tabular function.

Command : FUNCTION,REAL

Menu : Property → Physical → Function→ Real Table.

This function F1, is added in visco elastic material property of propellant. Visco elastic material property of propellant is given as:

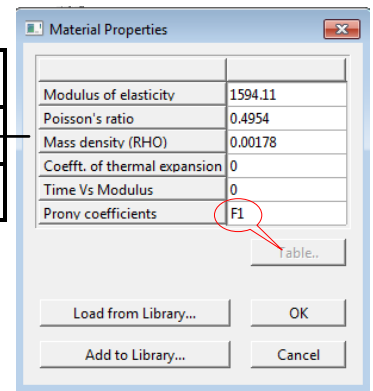
a.) Specify visco elastic material properties

Command : MATERIAL,VISCOMAT

Menu : Property → Material → Visco Elastic → Add

Parameters :

Element IDs	1T250
Material Data	1594.11/0.4954/0.00178/0/0/F1
Material ID	1



b.) Specify elastic material properties

<u>Insulation:</u>

Young's modulus, E = 100 ksc

Poisson's ratio = 0.5

Coefficient of thermal expansion (α) = 0.00021 / K

Density = 2000 kg/m^3

Command : MATERIAL, ISO

Menu : Property → Material → Isotropic → Add

Parameters :

Element IDs	Insulation elements. 251T275
Material Data	100/0.489/0/0.002
Material ID	1

Casing:

Young's modulus, E = 1.9e6 ksc

Poisson's ratio = 0.3

Coefficient of thermal expansion (α) = 0.000011 / K

Density = 7800 kg/m^3

Command : MATERIAL, ISO

Menu : Property → Material → Isotropic → Add

Parameters :

Element IDs	Casing elements. 276T300
Material Data	1.9E+06/0.3/0/0.0078
Material ID	2

10.Specify analysis type

Commands : ANTYPE, SET

Menu : Analysis → Analysis Type

Parameters : (To be filled by the user)

Analysis Type	Visco elastic

11.Specify analysis options

Commands : ANOPTION, SET

Menu : Analysis → Analysis Options

Parameters : (To be filled by the user)

Linear Solver	Multi Frontal

12.Specify viscoelastic general properties

Commands : VISCOELASTIC,ADD

Menu : Analysis →Visco elastic→General →Add

Parameters : (To be filled by the user)

Start time for analysis	0
End time for analysis	3.1536E+07
No. of time steps	366
ReferenceTemperature	1

13.Save the data file

Menu : File→ Save

14.Submit the job into FEAST

Menu : Analysis →Run Solver

```
Writing Results...
ELAPSED TIME  : 0 H :: 0 M :: 1 S
Finished successfully
```

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

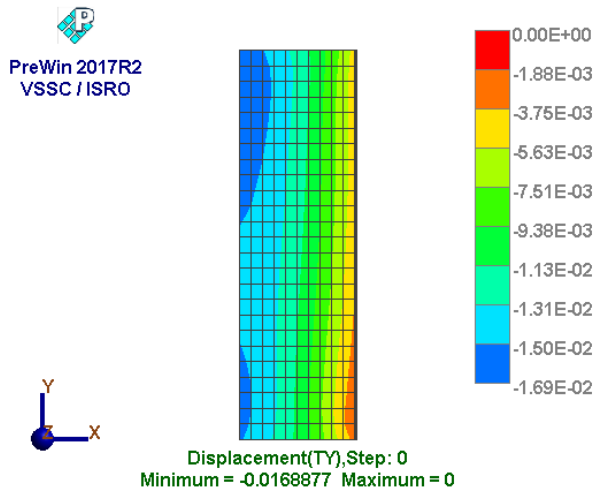
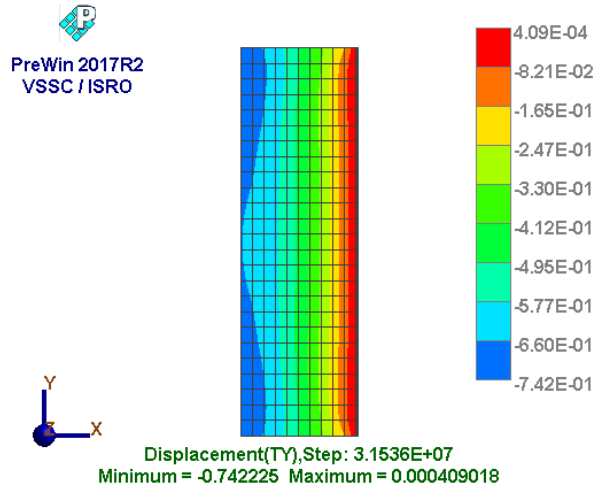
15.Post processing

a) Displacement Contour

Command : POST,CONTOUR

Menu : Post → Contour

Parameters :

Item	Displacement
Component	TY
Time step	3.1536E+07

The displacement contours, Ty component obtained for start and end time analysis is shown below.