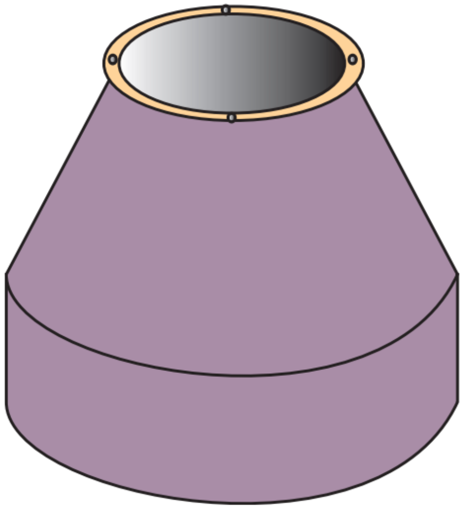
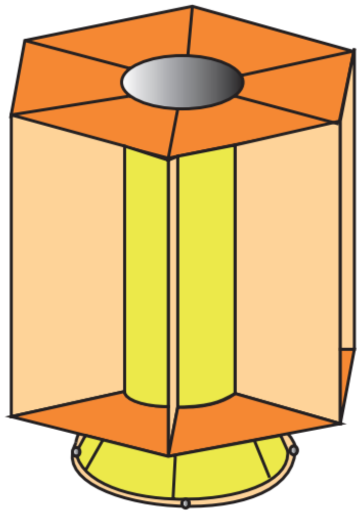
Curriculum Project in Structural Mechanics.

FINITE ELEMENT ANALYSIS OF A PAYLOAD ADAPTER AND SATELLITE FOR THE ARIANE 5 LAUNCHER

Professor, Walid LARBI



ACU



SATELLITE

IPSA
AERONAUTICS & SPACE ENGINEERING SCHOOL

Khady sarah SALL

CAE-A

22/01/2025

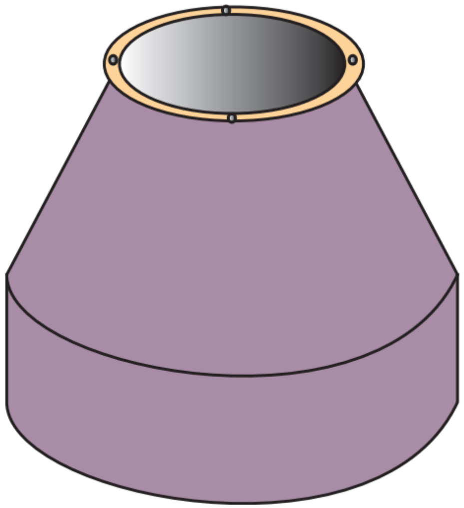
1. Payloads Adapters

Attachment: Securing the payload firmly to the launcher to prevent movement during transport.

Force Transmission: Transferring vibrations, axial, and lateral loads from the launcher to the payload.

Separation: Integrated separation systems release the payload in a controlled and damage-free manner.

Compatibility: Adapting the payload's shape to the launcher's fairing, which varies by configuration and provider.



Interface Rings: Connect the payload to the ACU and the ACU to the launcher.

Support Structures: Designed to transmit forces efficiently while being lightweight.

Separation Mechanisms: Including pyrotechnic devices, springs, or mechanical systems for precise separation.

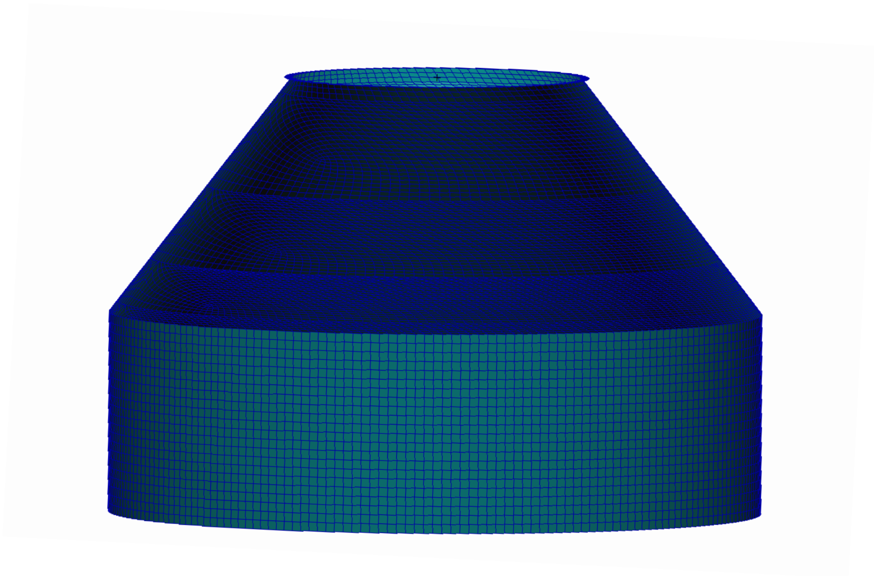
Polymers Matrix Composites: Such as high-strength carbon fibers for stiffness and low density.

Aluminum Alloys: Lightweight, corrosion-resistant, and easy to machine.

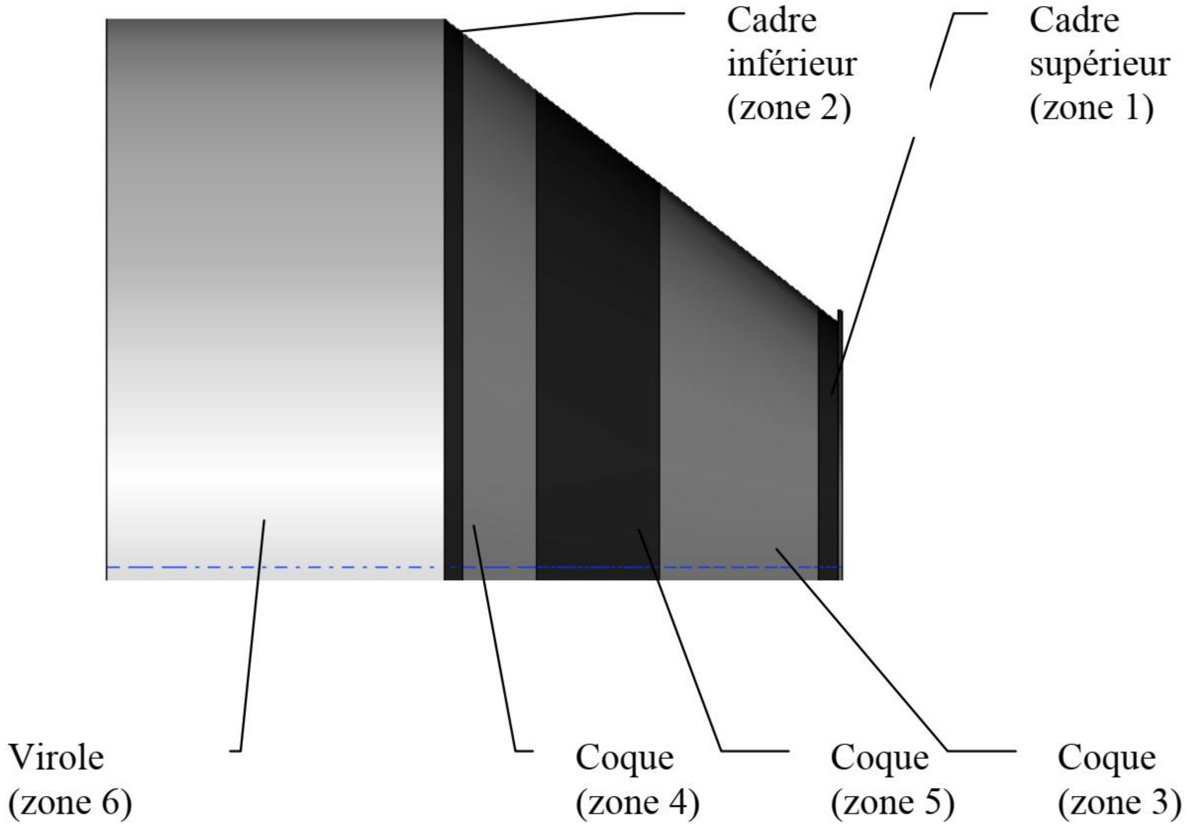
Titanium: High strength-to-weight ratio and compatibility with harsh conditions.

Stainless Steel: For components needing wear resistance and mechanical strength.

• Modal Analysis

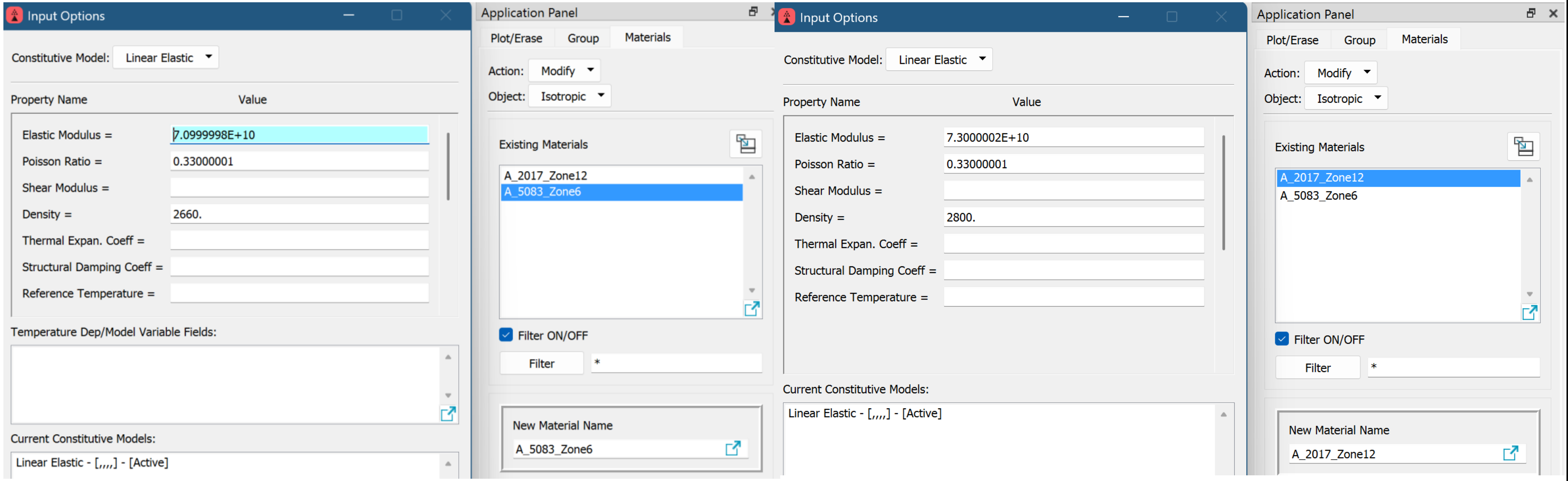


Creation of the materials



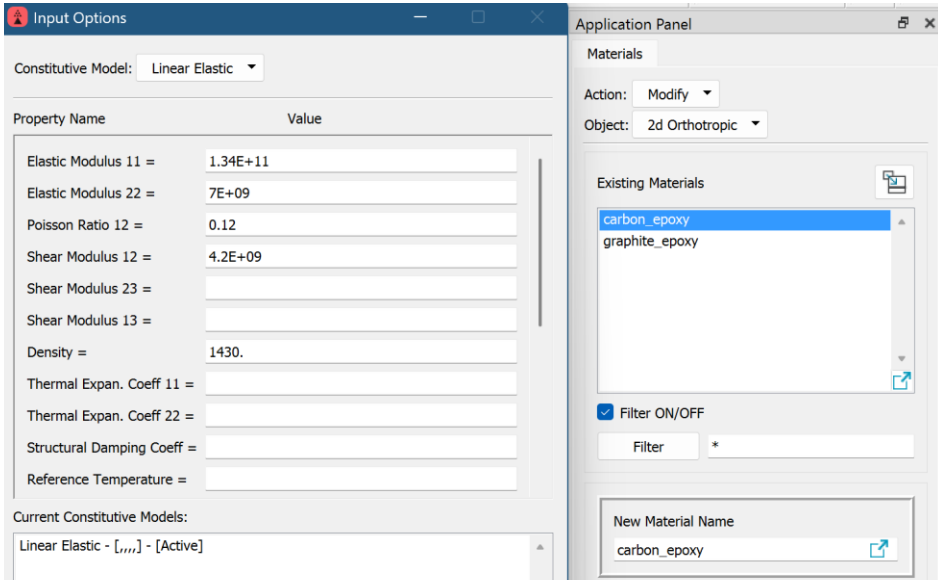
In this part we saw the required steps we used to create all the materials and how we assigned them using Patran.

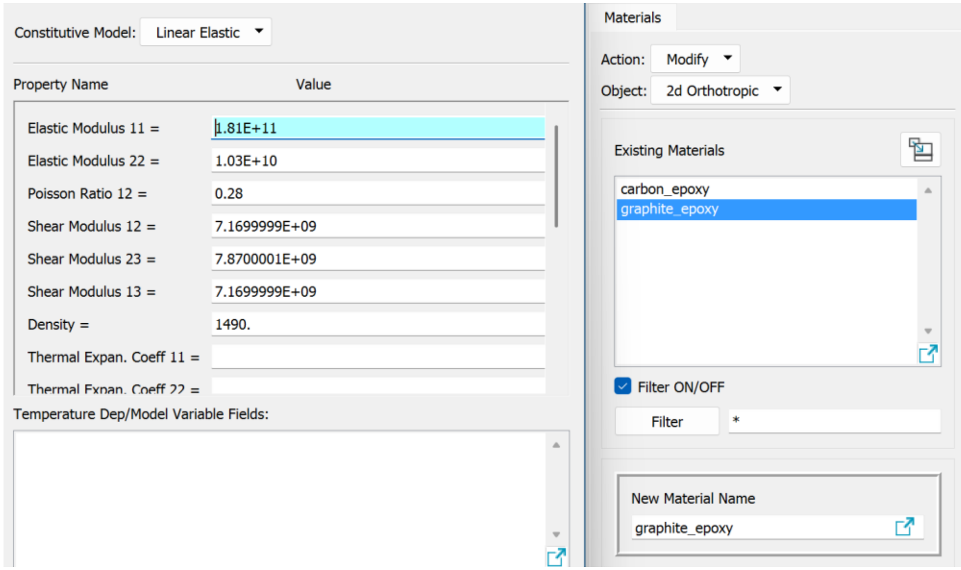
Isotropic



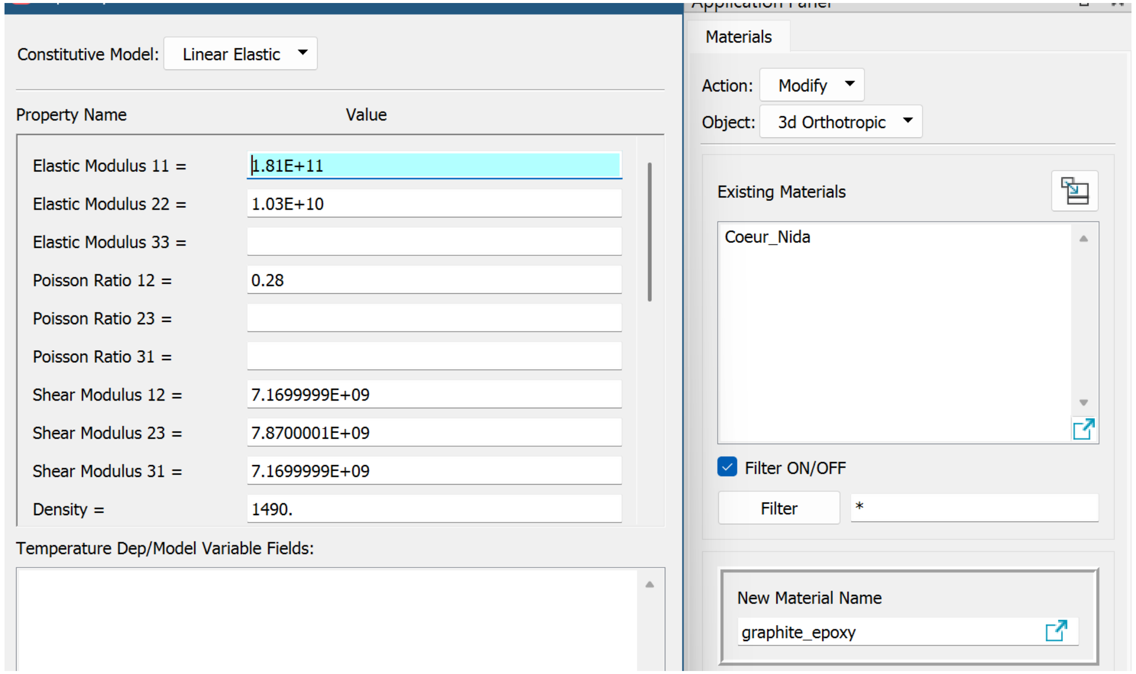
Input Options
Constitutive Model: Linear Elastic
Property Name
Value
Elastic Modulus = 7.0999998E+10
Poisson Ratio = 0.33000001
Shear Modulus =
Density = 2660.
Thermal Expan. Coeff =
Structural Damping Coeff =
Reference Temperature =
Temperature Dep/Model Variable Fields:
Current Constitutive Models:
Linear Elastic - [,,,,] - [Active]
Application Panel
Plot/Erase
Group
Materials
Action: Modify
Object: Isotropic
Existing Materials
A_2017_Zone12
A_5083_Zone6
Filter ON/OFF
Filter
*
New Material Name
A_5083_Zone6
Input Options
Constitutive Model: Linear Elastic
Property Name
Value
Elastic Modulus = 7.3000002E+10
Poisson Ratio = 0.33000001
Shear Modulus =
Density = 2800.
Thermal Expan. Coeff =
Structural Damping Coeff =
Reference Temperature =
Current Constitutive Models:
Linear Elastic - [,,,,] - [Active]
Application Panel
Plot/Erase
Group
Materials
Action: Modify
Object: Isotropic
Existing Materials
A_2017_Zone12
A_5083_Zone6
Filter ON/OFF
Filter
*
New Material Name
A_2017_Zone12

2D Orthotropic

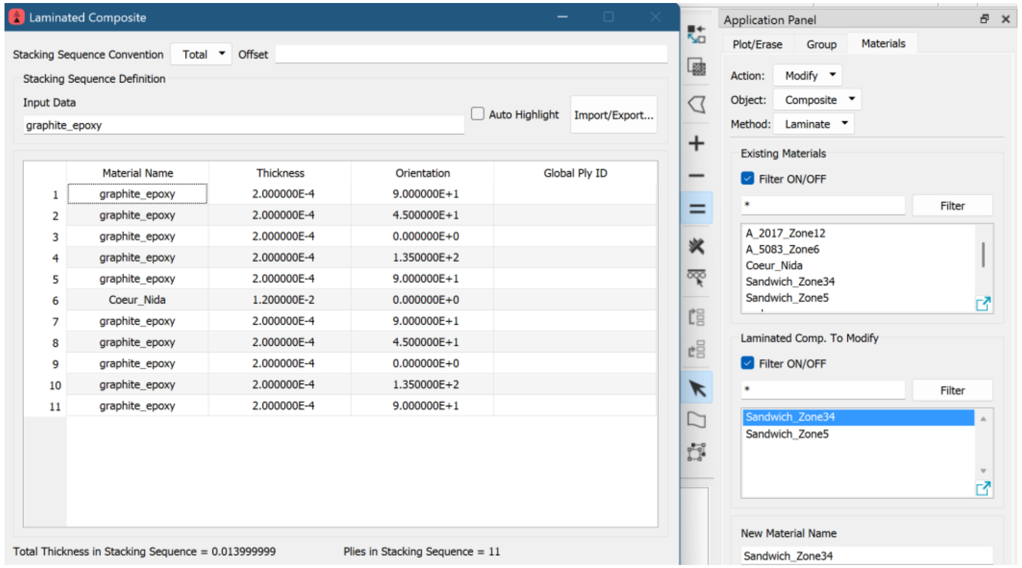


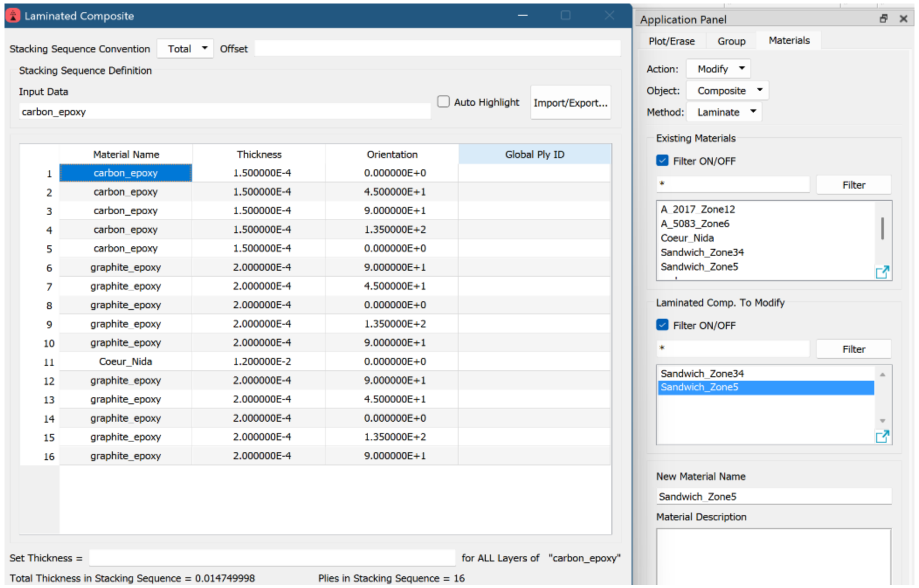


3D Orthotropic



Composite Laminate





Materials Properties Matrices

Zone 5

Composite Material Properties

Membrane, Bending, and Coupling Matrices

	Membrane			Bending		
Membrane	1.80E+08	4.78E+07	-6.02E-02	-3.25E+05	-5.46E+04	-8.29E+03
	4.78E+07	2.30E+08	-6.44E+00	-5.46E+04	-1.66E+05	-8.29E+03
	-6.02E-02	-6.44E+00	5.82E+07	-8.29E+03	-8.29E+03	-6.92E+04
Bending	-3.25E+05	-5.46E+04	-8.29E+03	7.98E+03	2.07E+03	1.49E+01
	-5.46E+04	-1.66E+05	-8.29E+03	2.07E+03	9.97E+03	1.49E+01
	-8.29E+03	-8.29E+03	-6.92E+04	1.49E+01	1.49E+01	2.55E+03

High Precision Value

Composite Property Display Options

- A, B, and D Matrices
- 3D Elasticity Matrix
- 3D Flexibility Matrix
- E's, NU's, G's, and Qij's
- Thermal: Kij, Ni, and Mi
- CTE's, CME's and Others

Cancel

Zone 3 and 4

Membrane, Bending, and Coupling Matrices

	Membrane			Bending		
Membrane	1.27E+08	3.79E+07	-6.02E-02	1.84E-01	1.46E-01	-6.86E+03
	3.79E+07	1.96E+08	-5.44E+00	1.46E-01	-9.38E-02	-6.86E+03
	-6.02E-02	-5.44E+00	4.59E+07	-6.86E+03	-6.86E+03	1.54E-01
Bending	1.84E-01	1.46E-01	-6.86E+03	5.35E+03	1.59E+03	-1.86E-04
	1.46E-01	-9.38E-02	-6.86E+03	1.59E+03	8.27E+03	-4.35E-04
	-6.86E+03	-6.86E+03	1.54E-01	-1.86E-04	-4.35E-04	1.94E+03

High Precision Value

Composite Property Display Options

- A, B, and D Matrices
- 3D Elasticity Matrix
- 3D Flexibility Matrix
- E's, NU's, G's, and Qij's
- Thermal: Kij, Ni, and Mi
- CTE's, CME's and Others

Cancel

Analytic Values of the matrices

```
Calculating ABD matrix for Zones 3 and 4...
ABD Matrix:
   6.5862e+07   2.2544e+07  -1.0928e-10   4.2803e+05   1.4654e+05      -7408.9
   2.2544e+07   1.0291e+08   8.3712e-09   1.4654e+05   6.6881e+05      -7408.9
  -1.0928e-10   8.3712e-09    8.484e+07      -7408.9      -7408.9   5.5138e+05
   4.2803e+05   1.4654e+05      -7408.9       2784.1       953.58      -96.315
   1.4654e+05   6.6881e+05      -7408.9       953.58       4361.2      -96.315
      -7408.9      -7408.9   5.5138e+05      -96.315      -96.315       3588.1
```

```
Calculating A, B, and D matrices for Zone 5...
Matrix A (Extensional stiffness) for Zone 5:
   1.4304e+08   3.9452e+07  -7.4615e-08
   3.9452e+07    1.523e+08  -7.3585e-08
    -7.834e-08   -6.986e-08   1.4846e+08

Matrix B (Coupling stiffness) for Zone 5:
   1.1839e+06   3.3041e+05       -11576
   3.3041e+05   1.2893e+06       -11576
       -11576       -11576   1.2433e+06

Matrix D (Bending stiffness) for Zone 5:
       9835.4       2776.1      -195.29
       2776.1        10956      -195.29
      -195.29      -195.29        10446
```

```
Calculating ABD matrix for Zone 5...
ABD Matrix:
   1.2174e+08   3.4629e+07  -1.0928e-10   8.3362e+05   2.4048e+05       -10457
   3.4629e+07   1.3846e+08   1.2096e-08   2.4048e+05   9.7265e+05       -10457
  -1.0928e-10   1.2096e-08   1.3073e+08       -10457       -10457   9.0755e+05
   8.3362e+05   2.4048e+05       -10457       5738.2       1677.5      -146.29
   2.4048e+05   9.7265e+05       -10457       1677.5       6867.8      -146.29
       -10457       -10457   9.0755e+05      -146.29      -146.29       6329.1
```

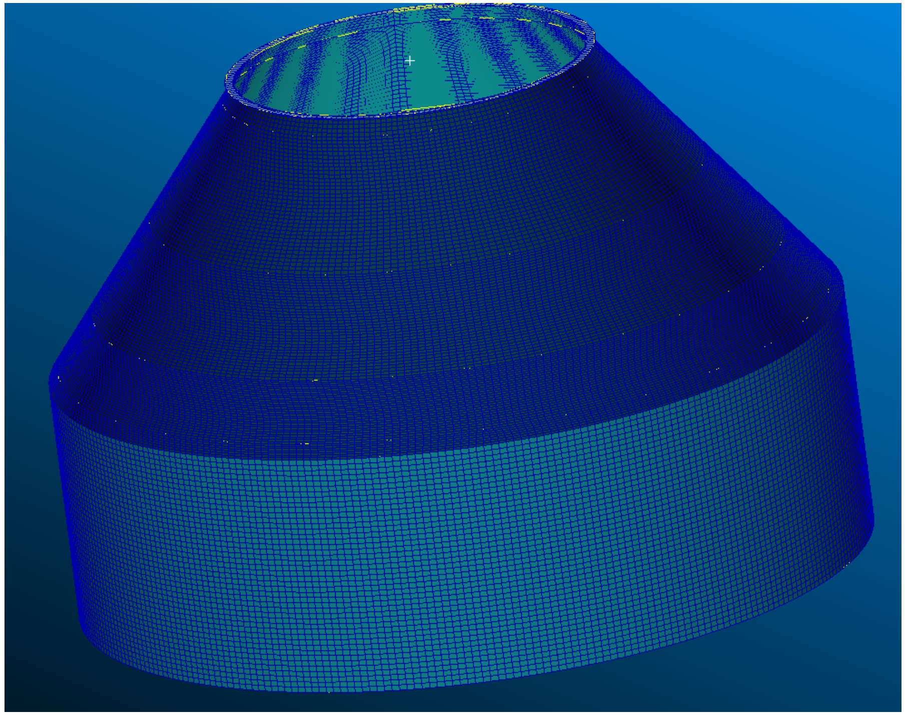
Comparison

For the comparison between the ABD matrices calculated in PATRAN and those calculated in MATLAB, we can observe that, in some cases, the results are identical up to a few thousandths. Sometimes, it is simply a matter of rounding to obtain the same result. I believe this is due to the numerical systems of the two platforms, which differ slightly. Additionally, PATRAN tends to apply decimal approximations at times, which may not necessarily affect the analytical part. However, we can see that, in terms of signs, the results are almost identical. Overall, we obtain approximately the same results, with minor discrepancies.

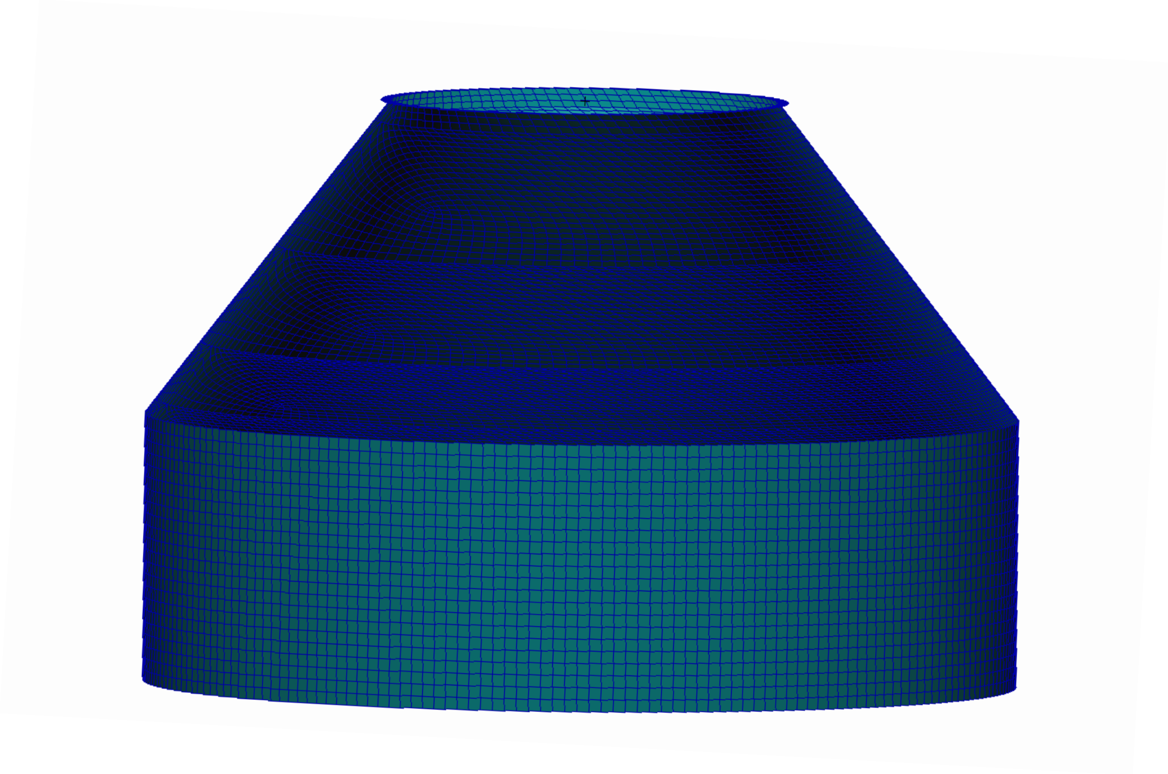
Analyse n°1: Convergence Study

Meshes

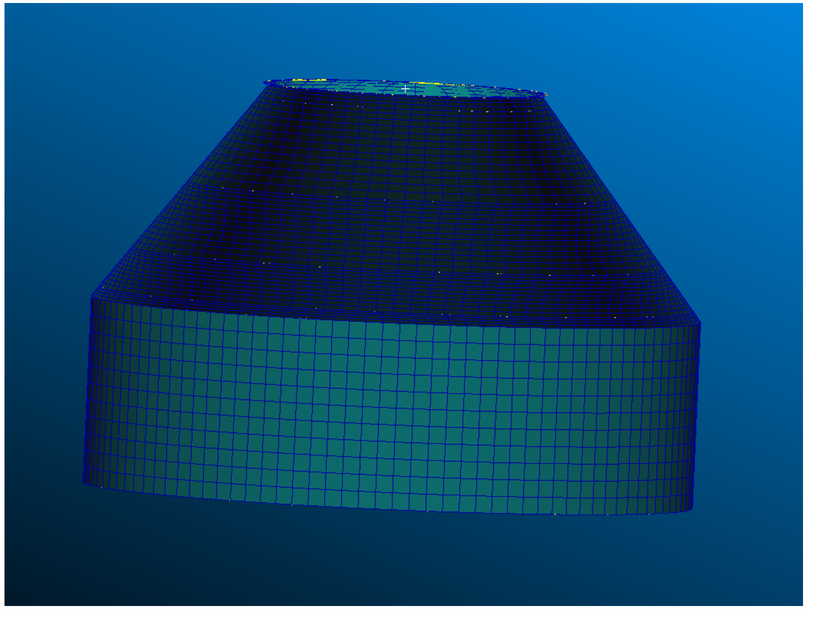
Fine Mesh

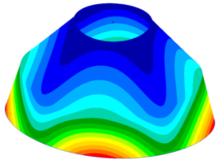
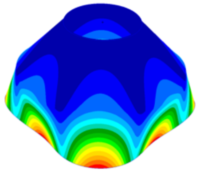
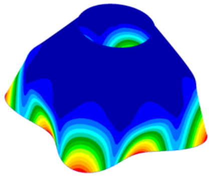
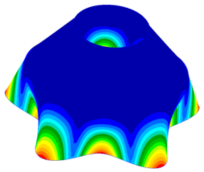
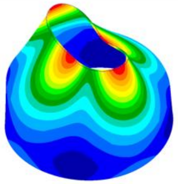
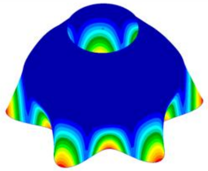
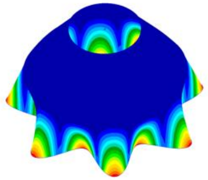
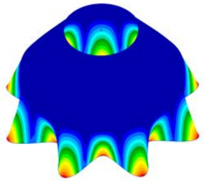
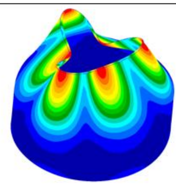
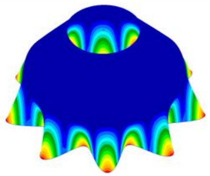


Medium Mesh



Coarse Mesh

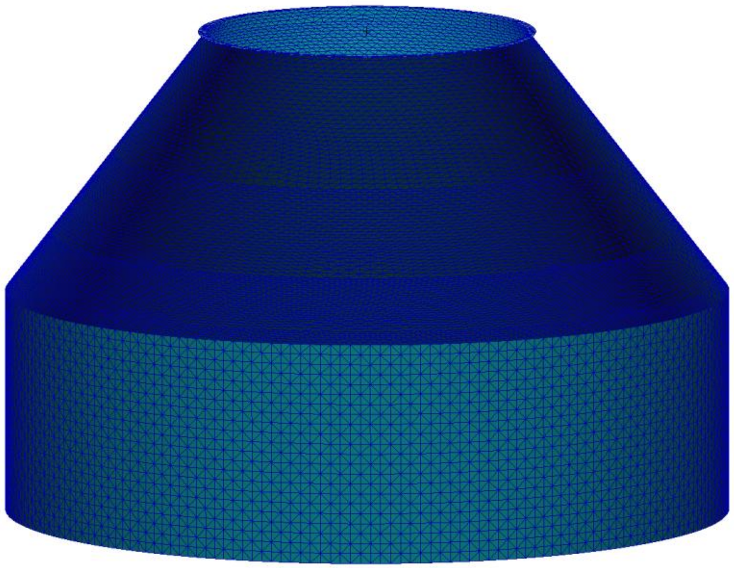


N° du Mode	***Fréquence Maillage gros*** *Nombre d'éléments :4821* *Nombre de nœuds :4903*	***Fréquence Maillage moyen*** *Nombre d'éléments :18305* *Nombre de nœuds :18469*	***Fréquence Maillage Fin*** *Nombre d'éléments :10000* *Nombre de nœuds :40*	***Déformée (Maillage moyen)***
7	*11.7309*	11.729	11.7262	
9	*27.3631*	27.4099	27.4187	
11	*46.2315*	46.3382	46.3599	
13	69.2025	69.3819	69.4138	
15	70.5493	70.4665	70.3751	
17	96.7094	96.9775	97.0257	
19	128.897	129.266	129.336	
21	165.841	166.322	166.414	
23	186.587	185.468	185.59	
25	207.594	208.191	208.308	

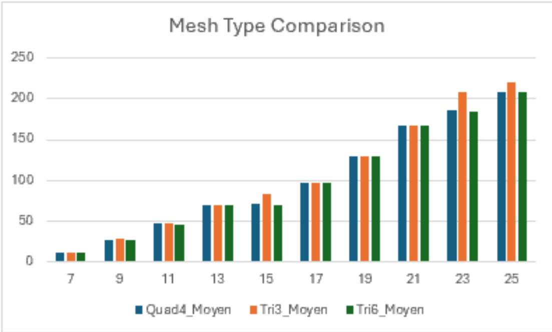
Analysis No. 2: QUAD4 – TRI3 Comparison

The objective here is to compare the Nastran QUAD4 element to the TRI3 element. We Create a TRI3 mesh of the structure with the same number of nodes as the medium QUAD4 mesh. We Compare with other types of elements.

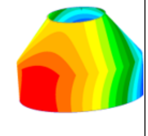
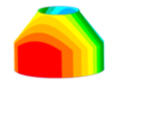
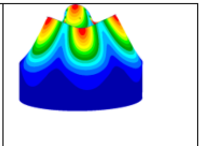
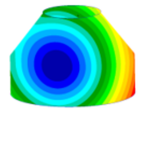
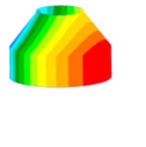
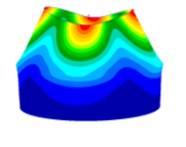
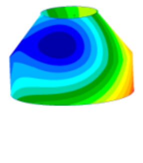
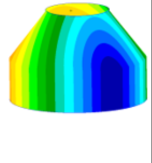
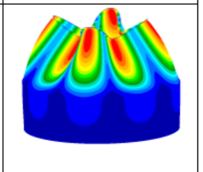
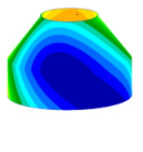
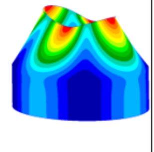
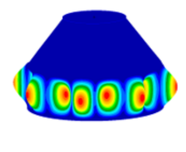
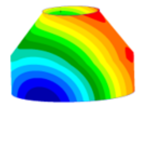
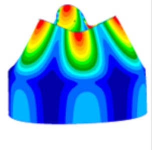
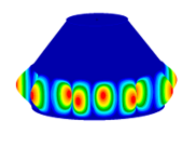
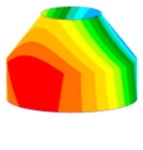
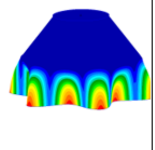
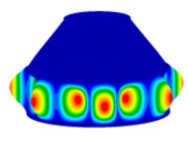
Mesh

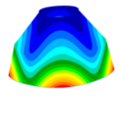
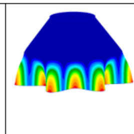
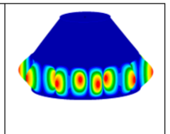
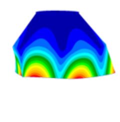
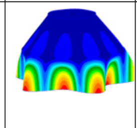
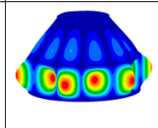
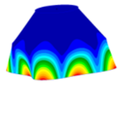
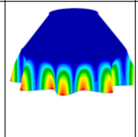
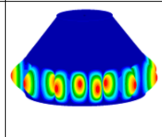
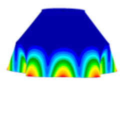
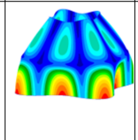
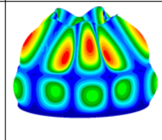
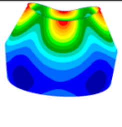
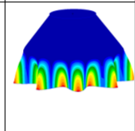
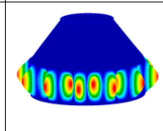
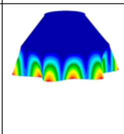
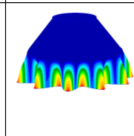
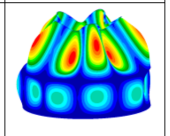
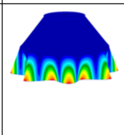
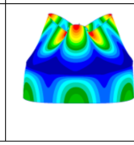
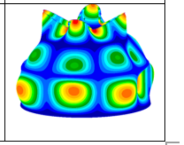


Comparaison QUAD – TRI *Conditions aux limites : libre*			
N° du mode	***Fréquence Maillage moyen QUAD*** *Nombre d'éléments : 18305* *Nombre de nœuds : 18469*	***Fréquence Maillage moyen TRI : Typology Tri3*** *Nombre d'éléments :36823* *Nombre de nœuds :18576*	***Fréquence Maillage moyen TRI : Typology Tri6*** *Nombre d'éléments :9307* *Nombre de nœuds :18779*
7	11.729	11.7984	11.653
9	27.4099	27.5396	27.3383
11	46.3382	46.4745	46.2907
13	69.3819	69.5189	69.323
15	70.4665	82.9792	69.3555
17	96.9775	97.1368	96.9792
19	129.266	129.47	129.3
21	166.322	166.583	166.379
23	185.468	208.526	183.5
25	208.191	220.29	208.254



Analysis No. 3: Influence of Boundary Conditions

1	0.00022647		8.12536e-5		204.032	
2	0.000176105		0.000109019		225.793	
3	9.25209e-05		0.000132119		324.813	
4	0.000127736		52.966		375.629	
5	0.000167945		159.537		378.878	
6	0.000209153		162.033		387.164	

7	11.7284		166.583		396.225	
9	27.4096		176.451		412.532	
11	46.3382		187.66		426.213	
13	69.3819		202.772		434.86	
15	70.4371		220.847		467.102	
17	96.9775		262.823		479.853	
19	129.266		287.983		498.278	

Transient Analysis

The structure is fixed. Only a time-varying load is taken into account. We use the loading represented by the sine curve below for a hammer impact in the direction normal to the structure. It is applied to the edge (from the outside toward the inside of the structure) in the direction normal to the surface, at a height of 50 mm, with a maximum force amplitude of 1000 N. The study duration is 1 ms. The figure below shows the force value as a function of time:

Calculation

$F(t)=F_{max}\cdot\sin(\omega t)$

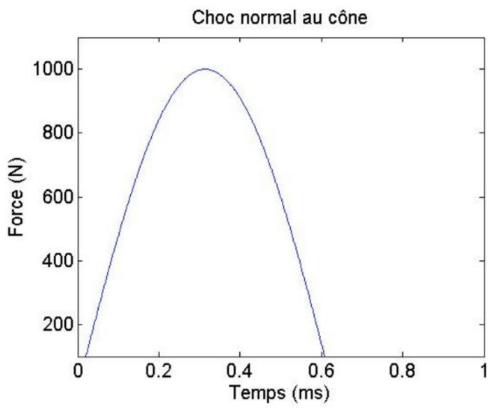
où :

- **F(t): l'effort en fonction du temps (en Newtons),**
- **Fmax=1000 : amplitude maximale,**
- **ω= =2πf : pulsation angulaire (f est la fréquence),**
- **ϕ : phase initiale (en radians),**
- **t: temps (en secondes).**

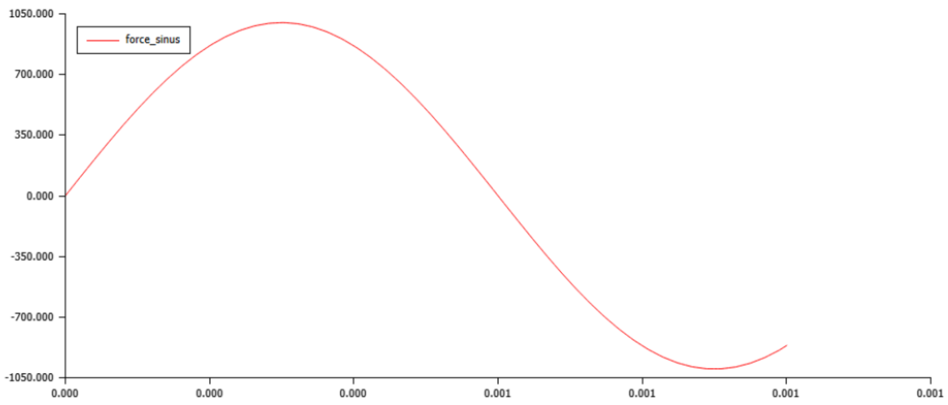
La durée totale de l'étude est 1ms=0.001s.
La courbe est sinusoïdale, donc un cycle ou une demi-période est probablement impliqué.

T=0.0012s

$F(t)=1000\cdot\sin(5233.33t)$

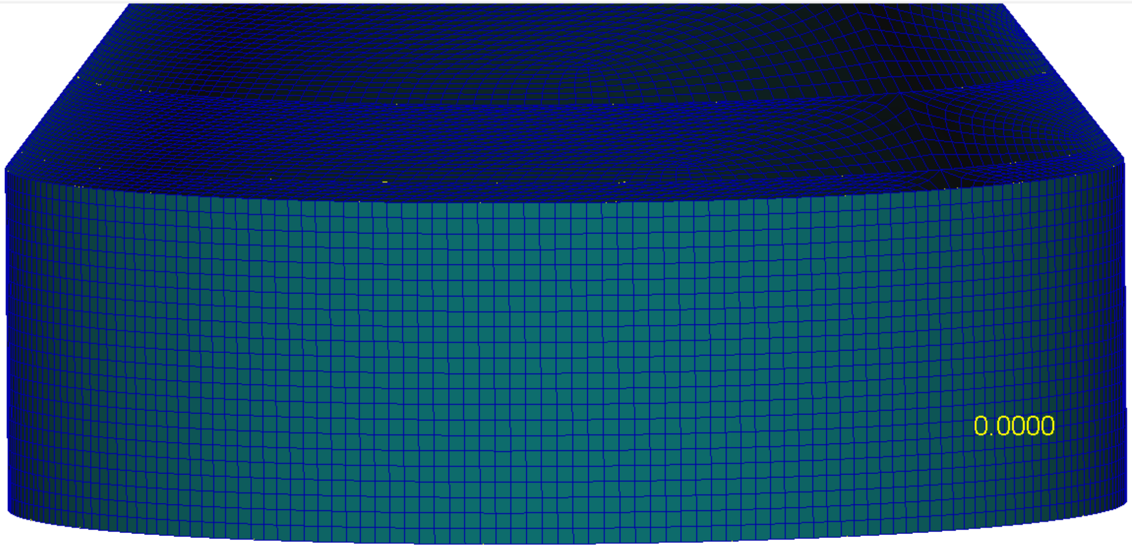


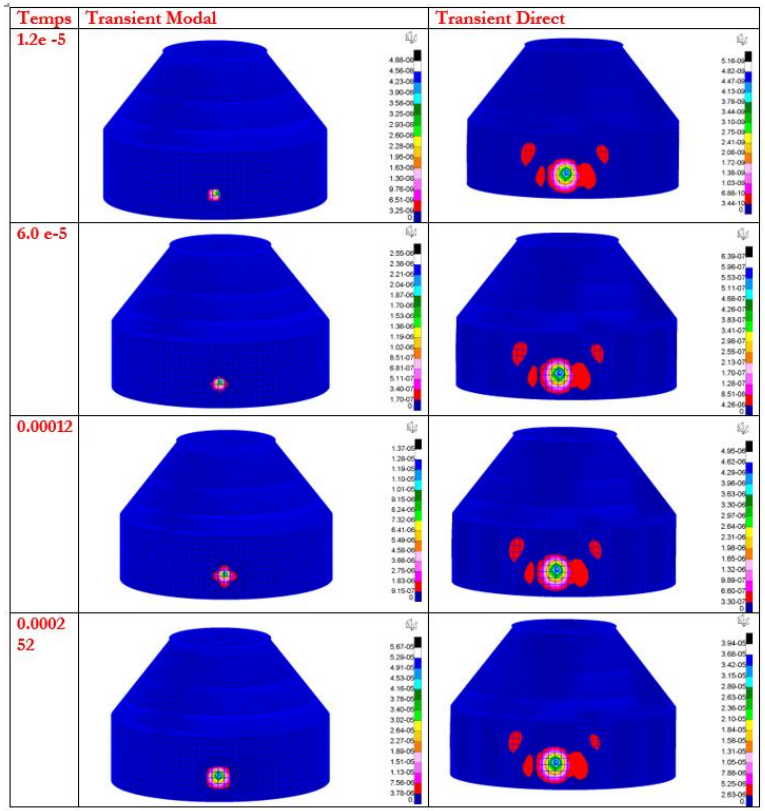
Force Display

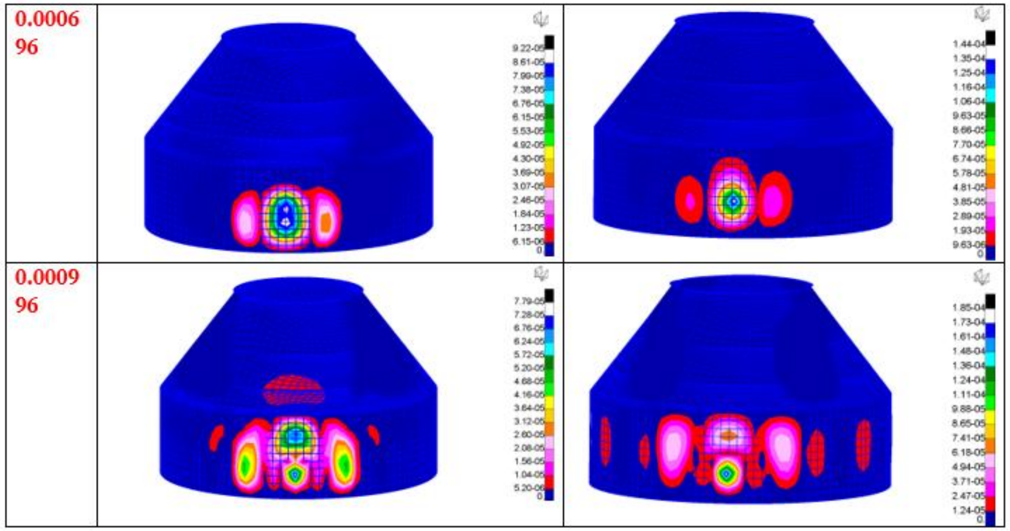


Force position

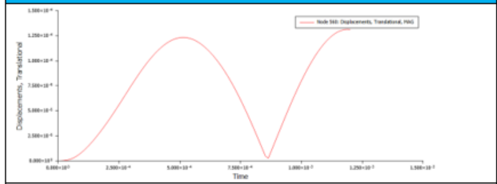
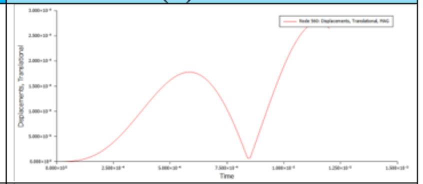
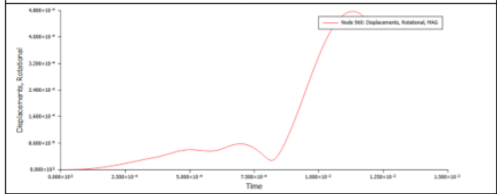
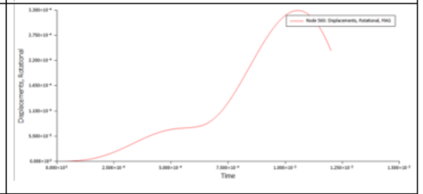
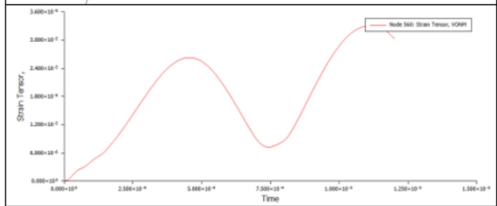
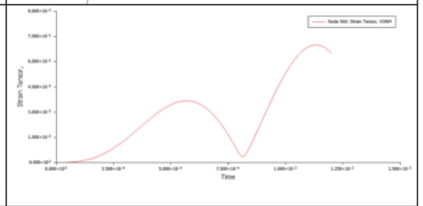
Node 560 is the node for a height of 50 mm

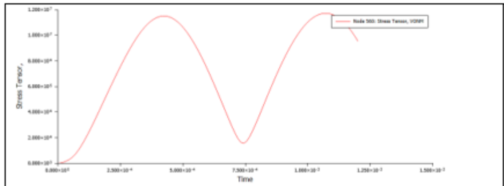
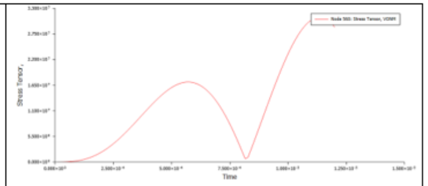
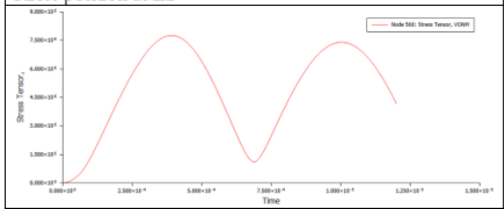
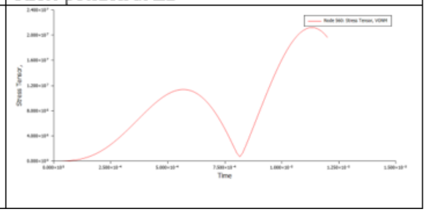






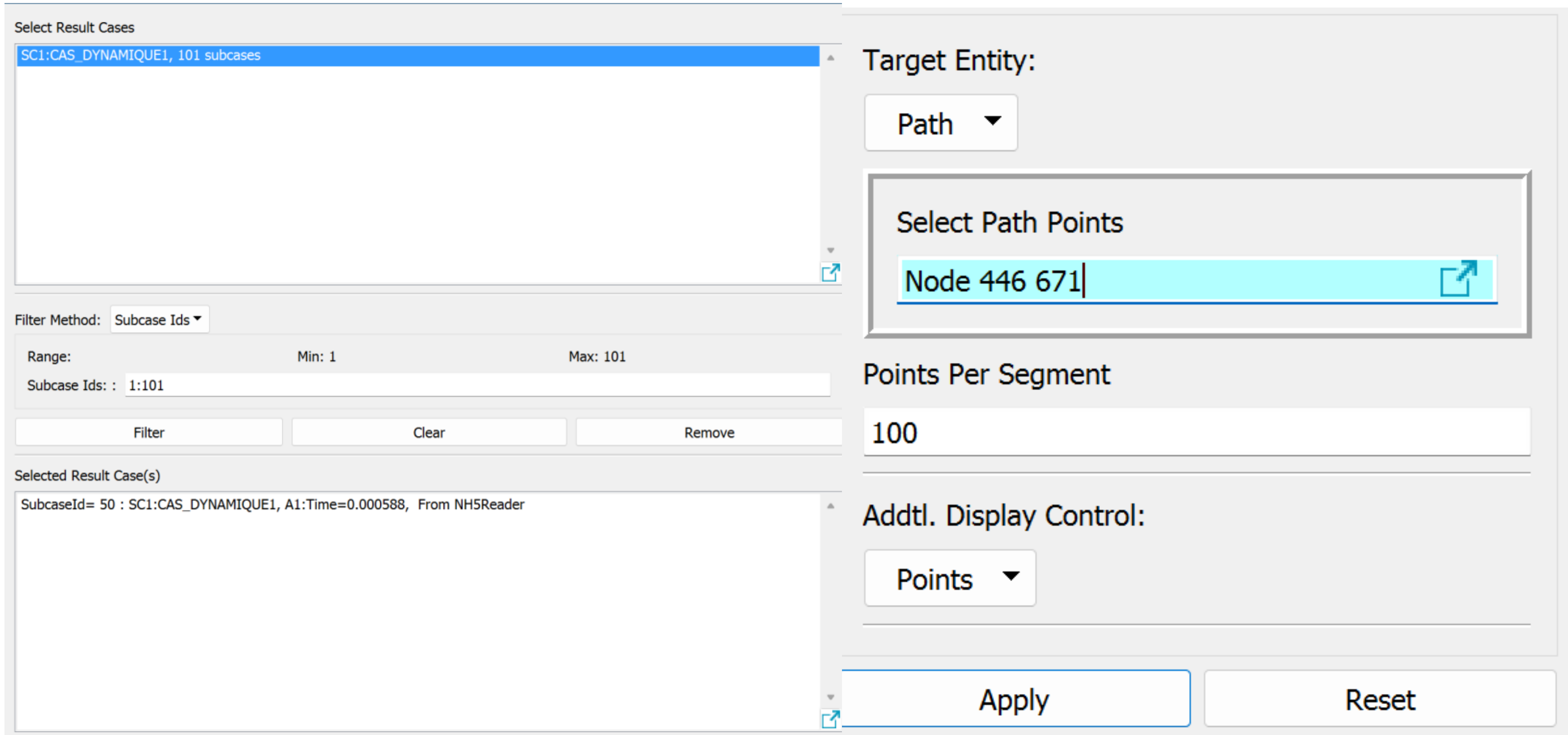
Time variation of strain, stress, and displacement at some most solicitated points
Direct and Modal Methods

Transient Direct	Transient Modal (30)
	
	
Strain Layer at the Middle	Strain Layer at the Middle
	
Stress Position at Z1	Stress Position at Z1

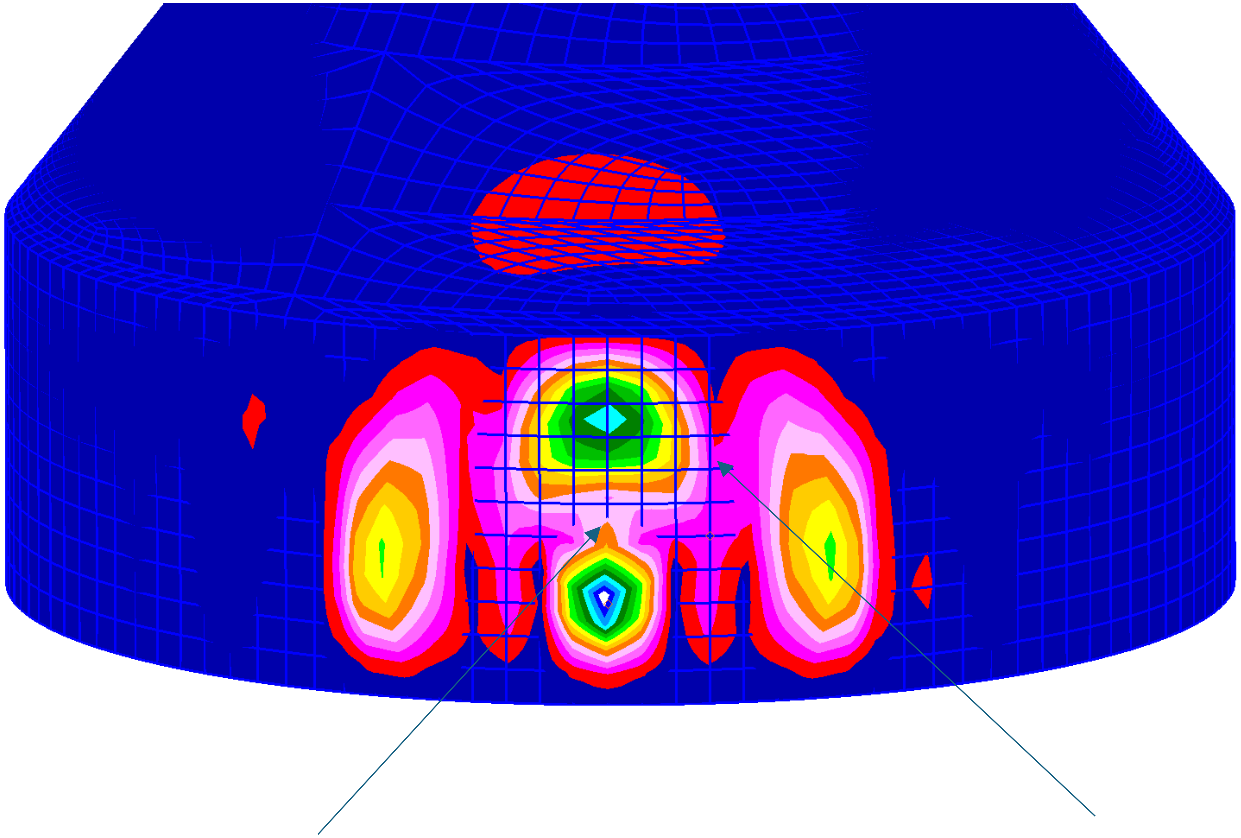
	
Stress position at Z2	Stress position at Z2
	

For a chosen time here t=0.000588 here is the plots of the strain, stress and displacement along a curvilinear abscise

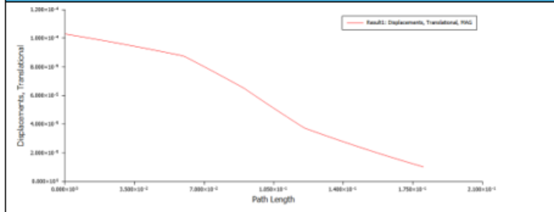
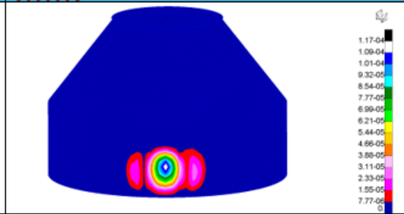
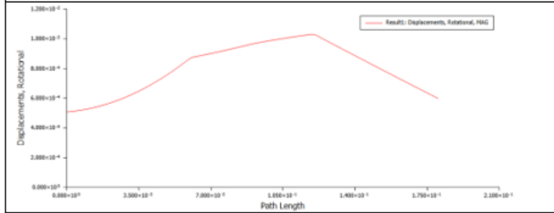
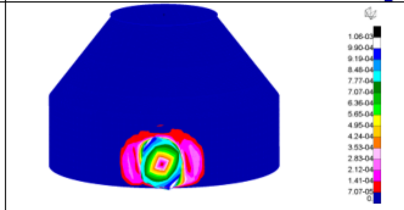
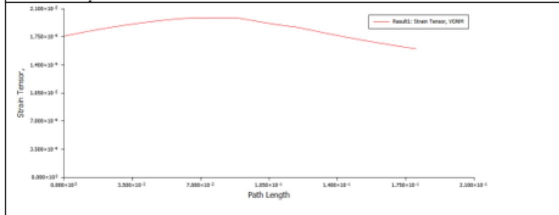
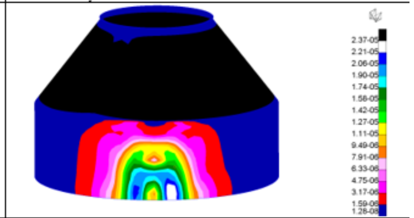
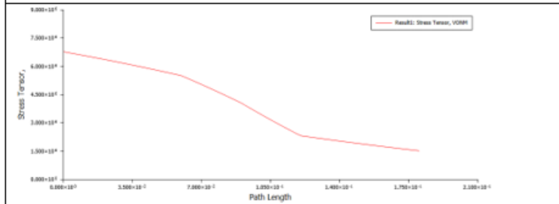
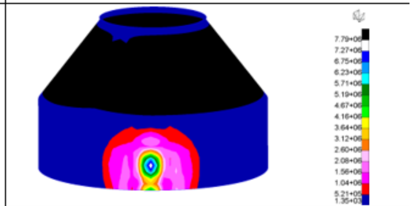
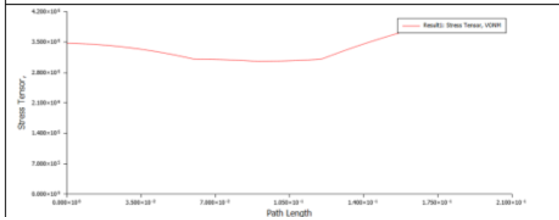
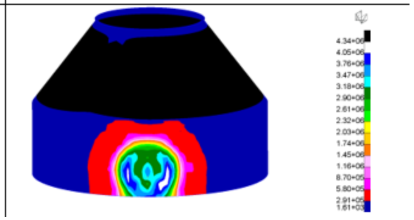
choosing time

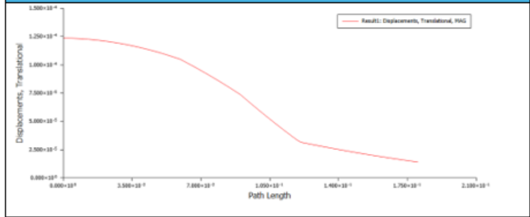
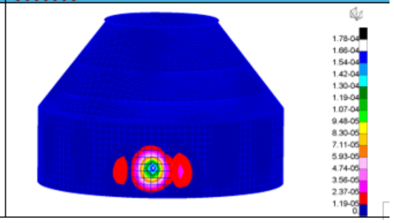


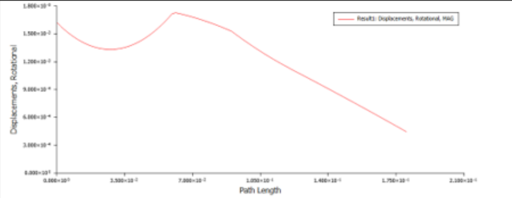
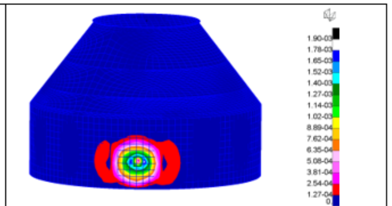
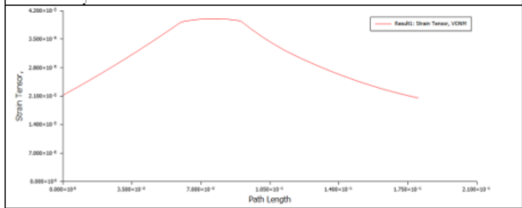
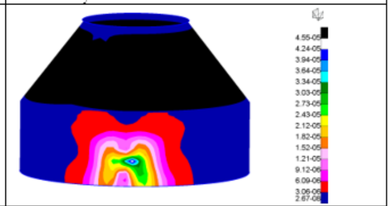
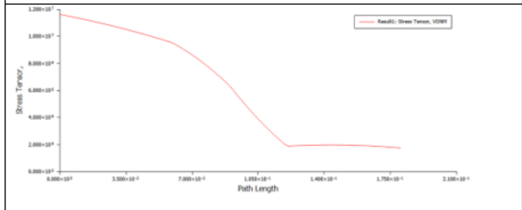
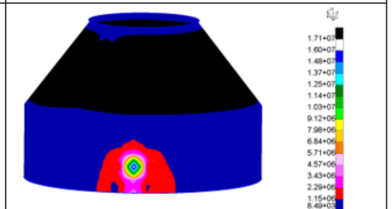
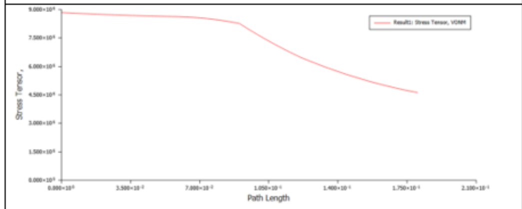
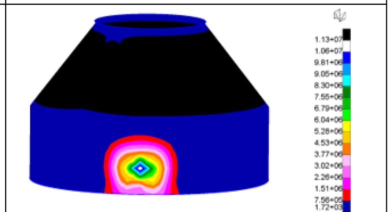
The curvilinear abscise



Plots and Visualizations For both methods: Direct and Modal

Transient Direct	Visuals of Direct Method
	
	
Strain Layer at the Middle	Strain Layer at the Middle
	
Stress at Z1	Stress at Z1
	
Stress at Z2	Stress at Z2
	

Transient Modal	Visuals of Modal Method
	

	
Strain Layer at the Middle	Strain Layer at the Middle
	
Stress at Z1	Stress at Z1
	
Stress at Z2	Stress at Z2
	

The structure responds to the excitation in the form of a natural vibration mode. When we observe the structure, initially, when it is fixed with no external force applied, the first frequency starts around 200 Hz. As we move through the modes, we notice that the frequency increases, reaching about 400 Hz from the 17th mode onward. This suggests that if we were to look further into higher modes, such as the 30th or 35th mode, the frequency would continue to rise. In fact, the frequency could eventually approach the values we see now.

This indicates that the structure is vibrating in a natural mode, responding to the applied sinusoidal force. Thankfully, since the force is not purely sinusoidal, it doesn't lead to continuous resonance. However, the structure is indeed resonating with the applied force at certain frequencies, which corresponds to the excitation of its natural vibration modes.

Studying the response to other types of loading (sinusoidal, step, Dirac).

Sinusoidal loading

Equation of the Sinusoidal Force

$F(t)=1000\sin(314.16\cdot t+\pi/4)$

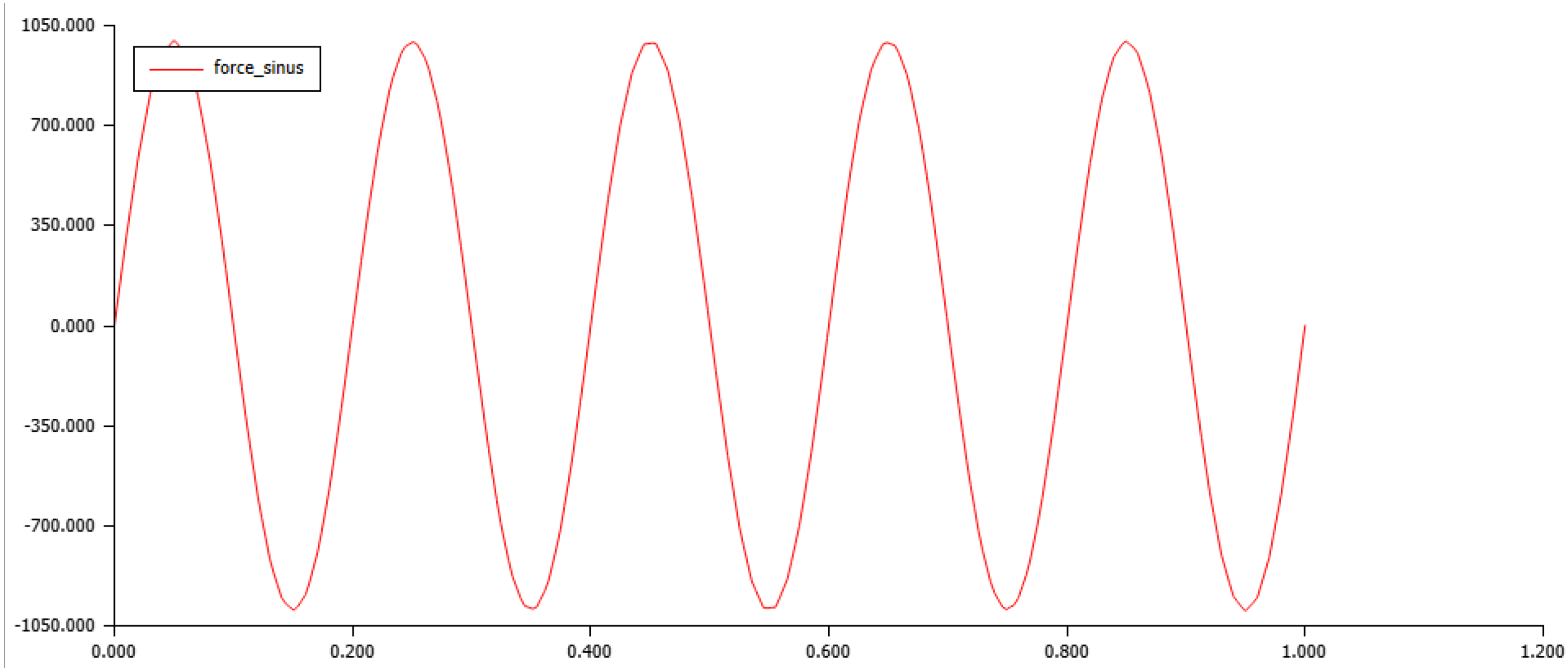
Parameters

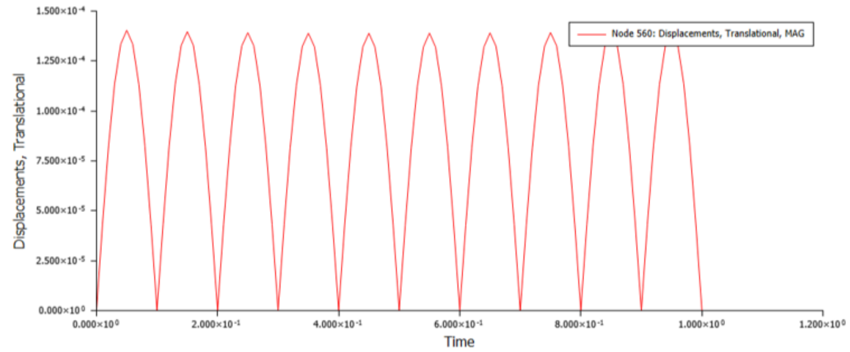
- **Amplitude** ($F0$) : =1000N
- **Angular frequency** (ω): 3141.59rad/s soit f=500Hz
- **Initial phase** (ϕ): $\pi/4$ rad
- **Period** (T): 0.002

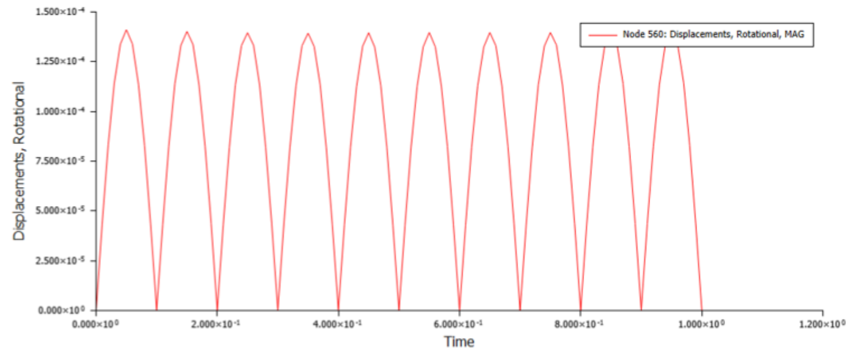
- **Simulation time**: 1s

Notes for Interpretation

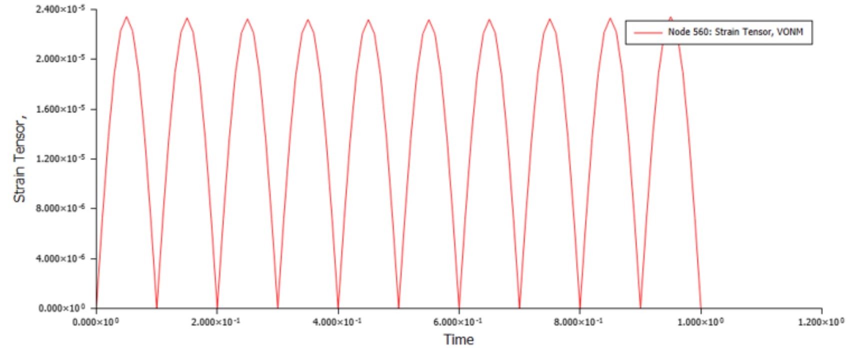
1. **Oscillations over 5 periods**: This simulation time will clearly illustrate the repeated effect of the force on our system.
2. **Initial phase**: With ϕ=π/4, the sinusoid does not start at zero but rather at a quarter-cycle before its maximum.



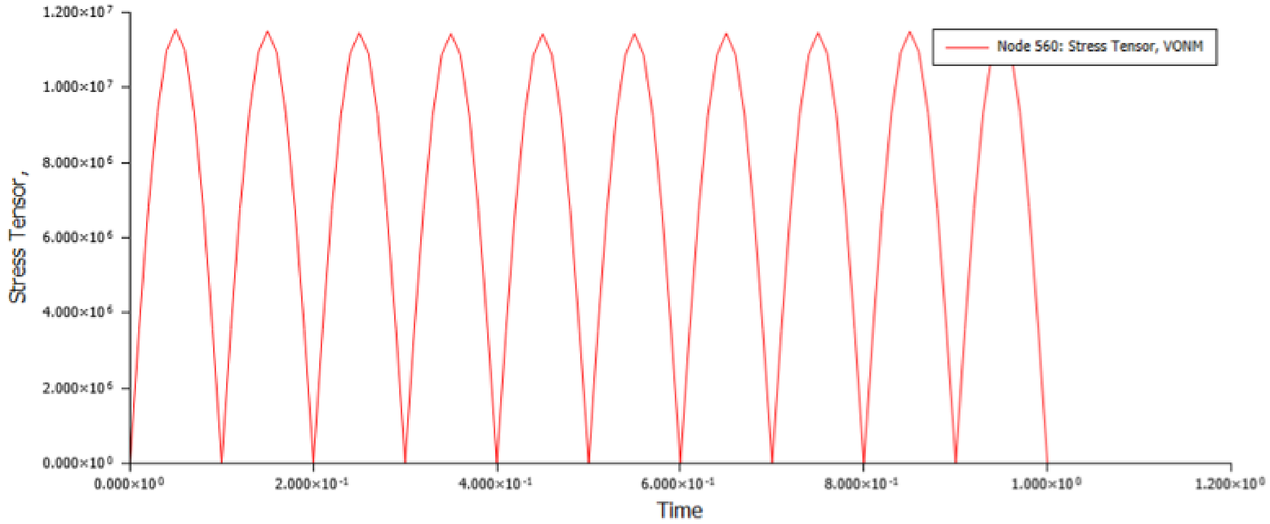




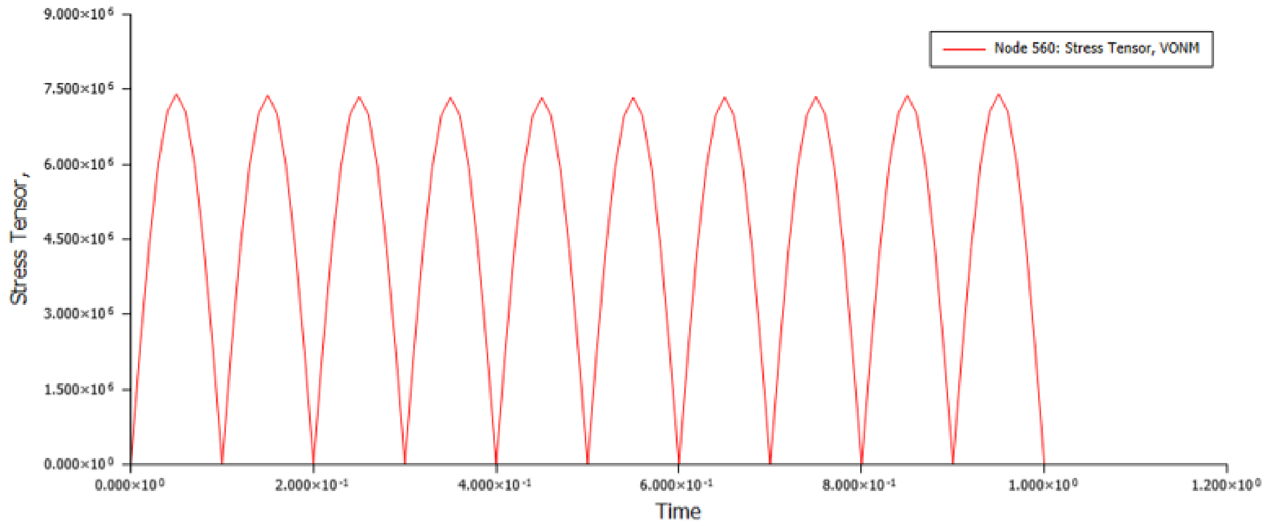
Strain at Middle

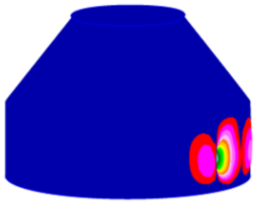
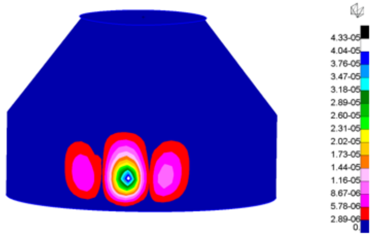
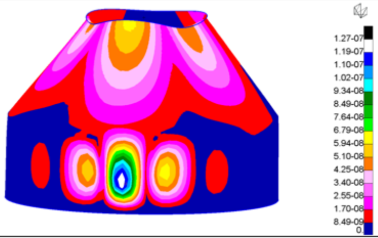
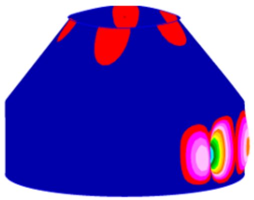
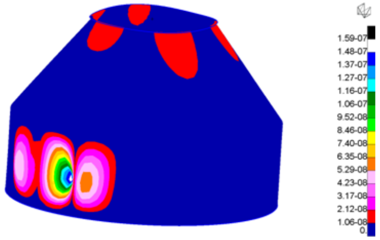
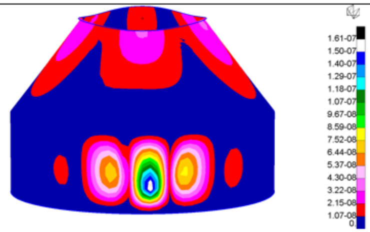


Stress at Z1

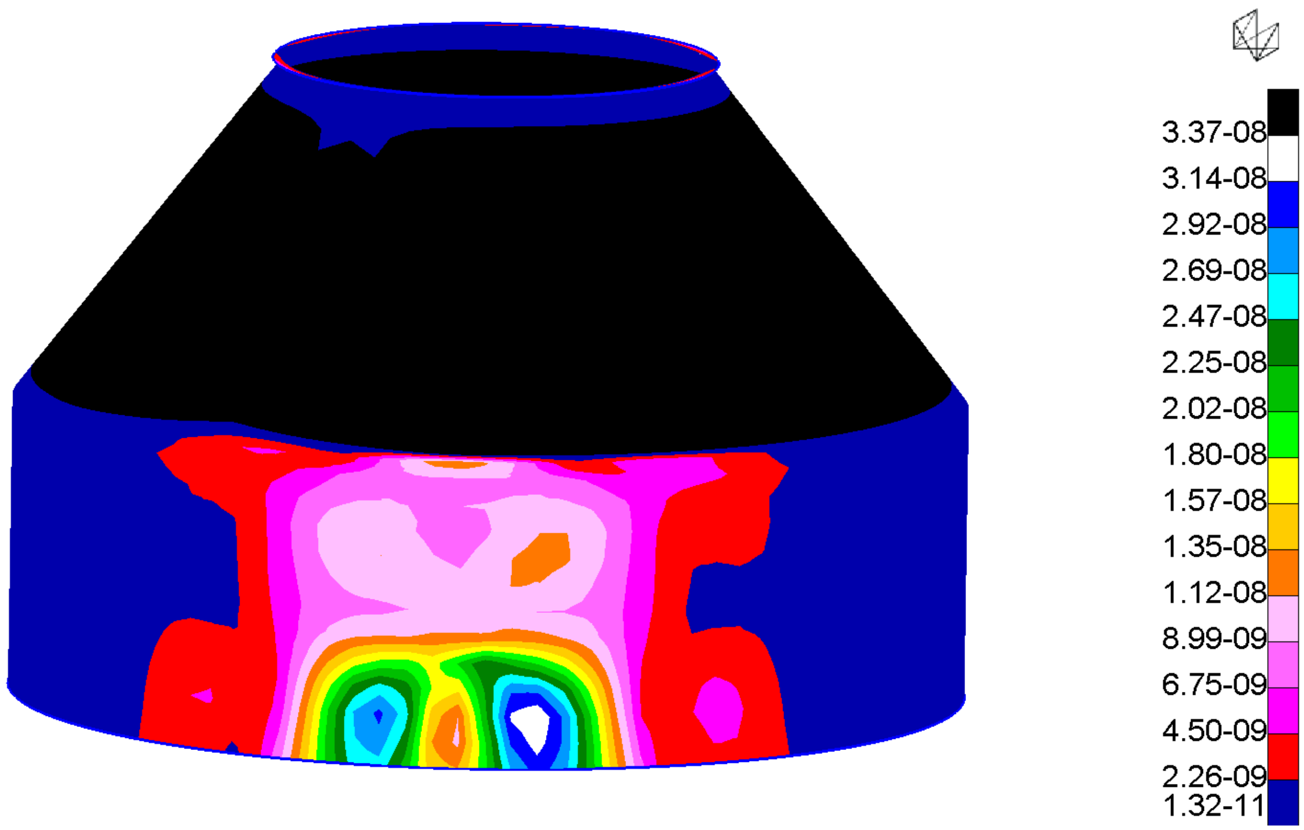


Stress at Z2

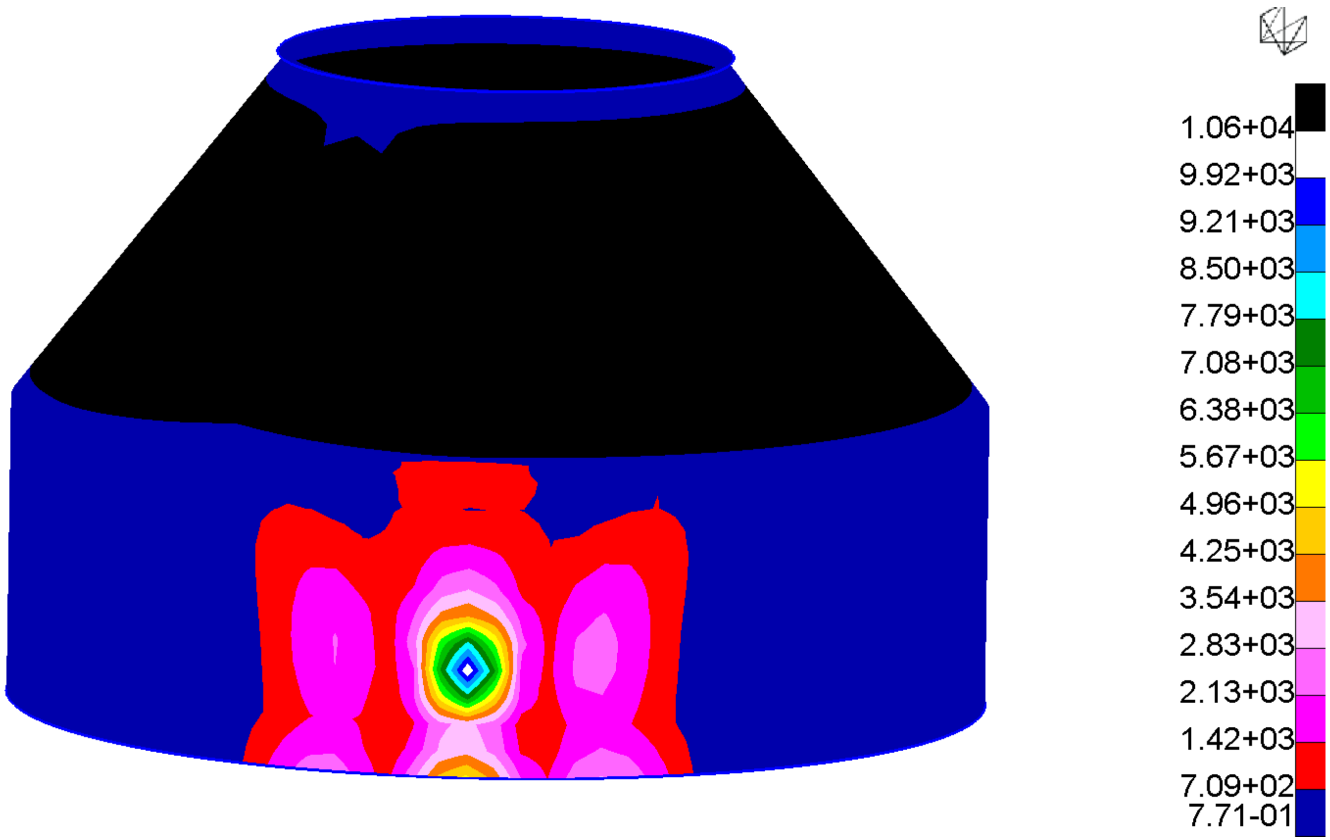


Time	Displacement Deformations
0.01 	
0.1 	
0.5 	
1 	

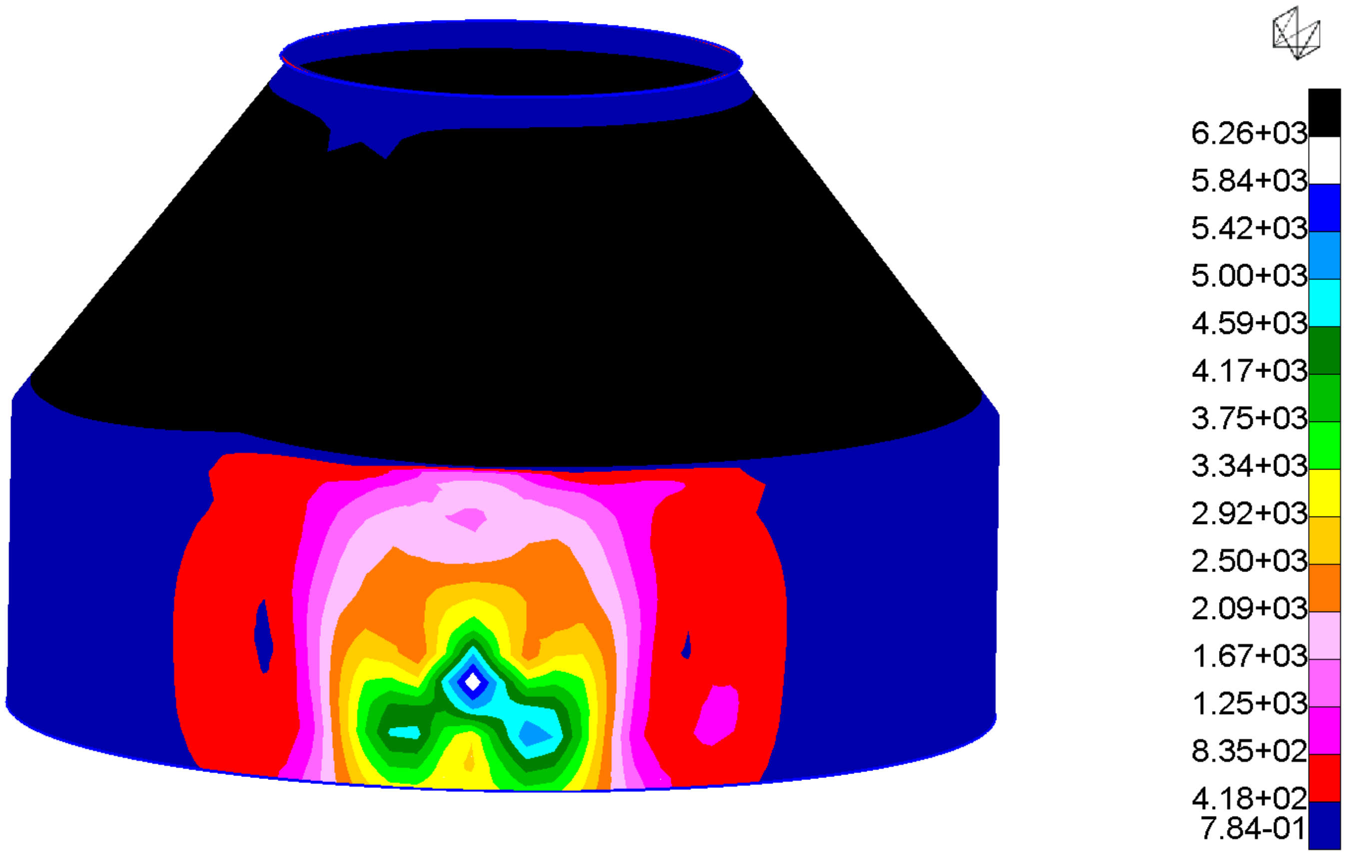
Strain at 0.5s at the middle



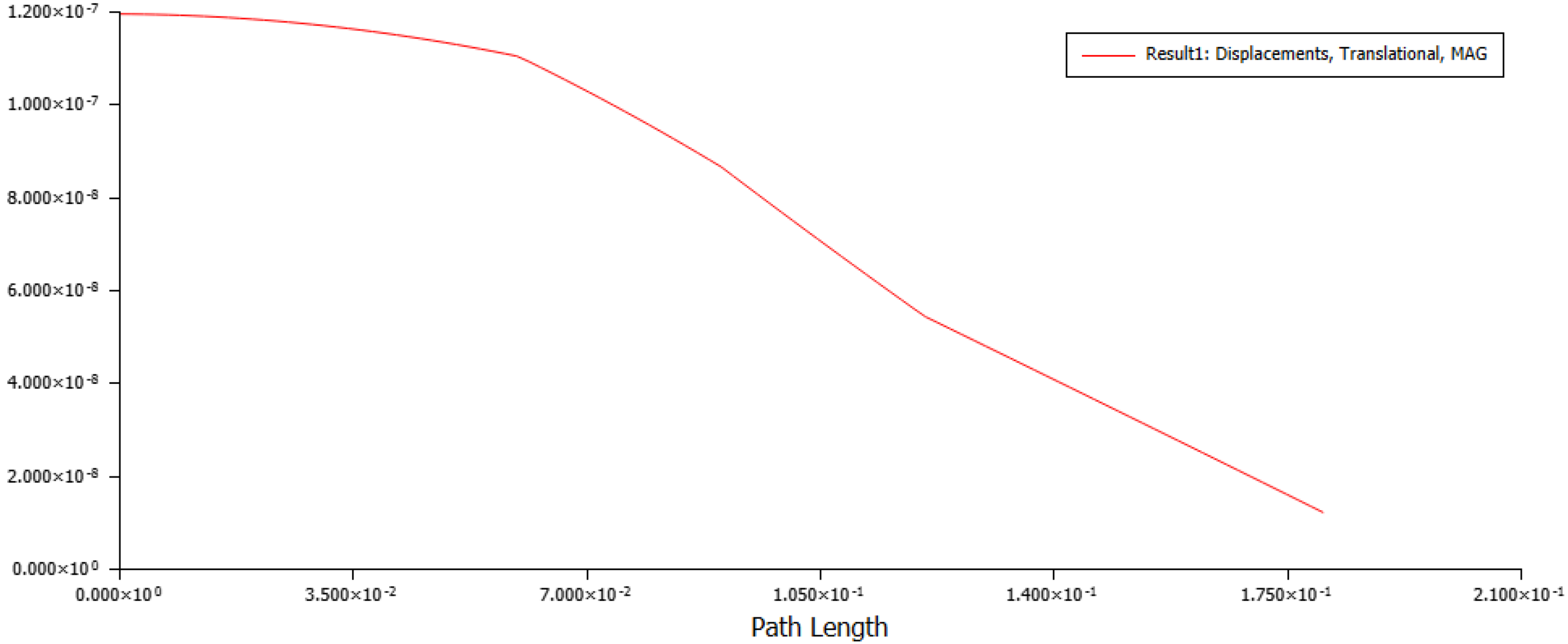
Stress at 0.5s at Z1

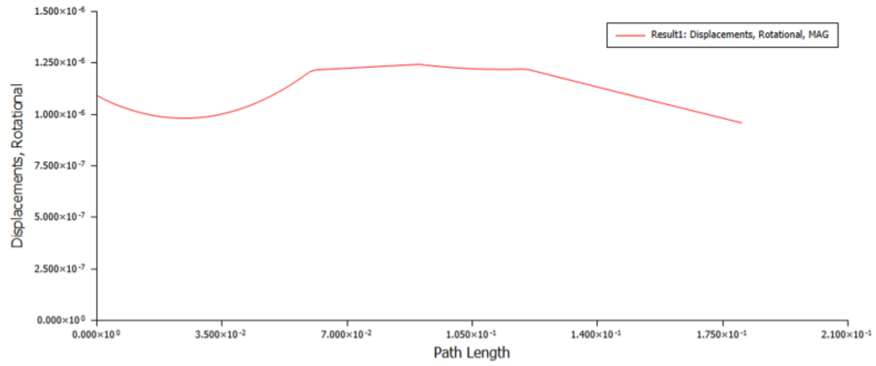


Stress at 0.5s at Z2

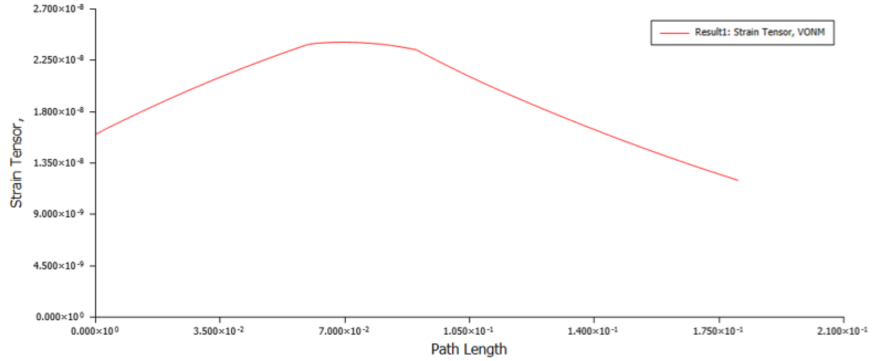


Displacement Path Length

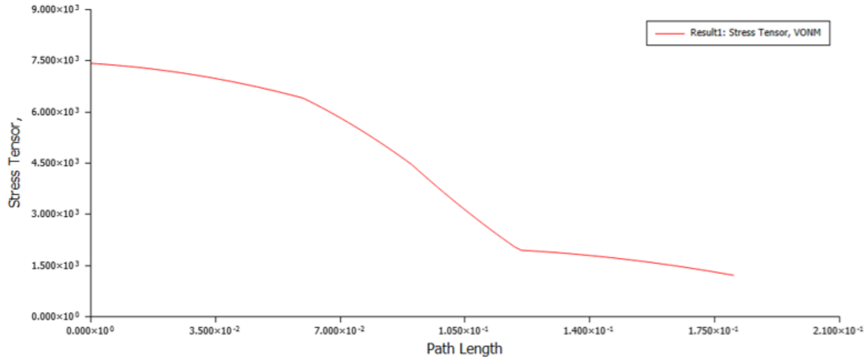




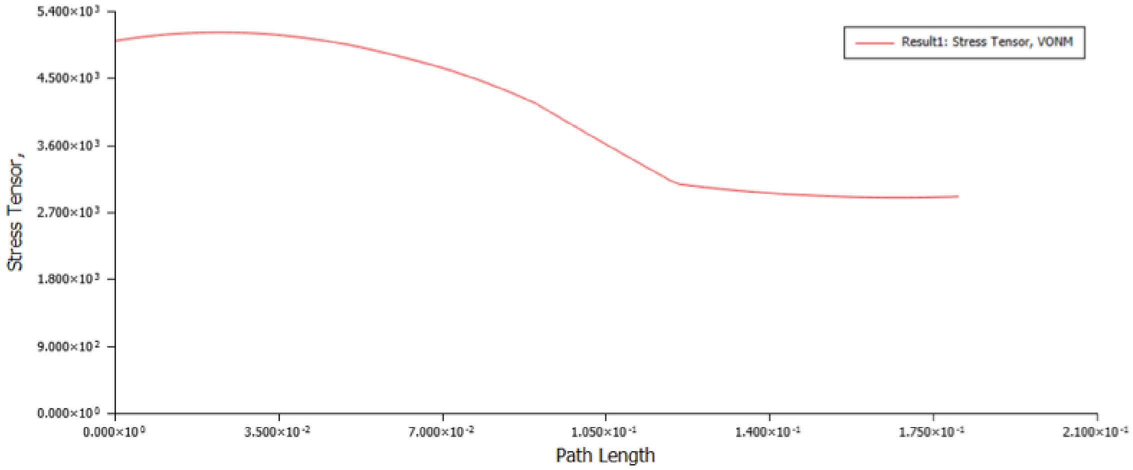
Strain At Middle Path Lenght



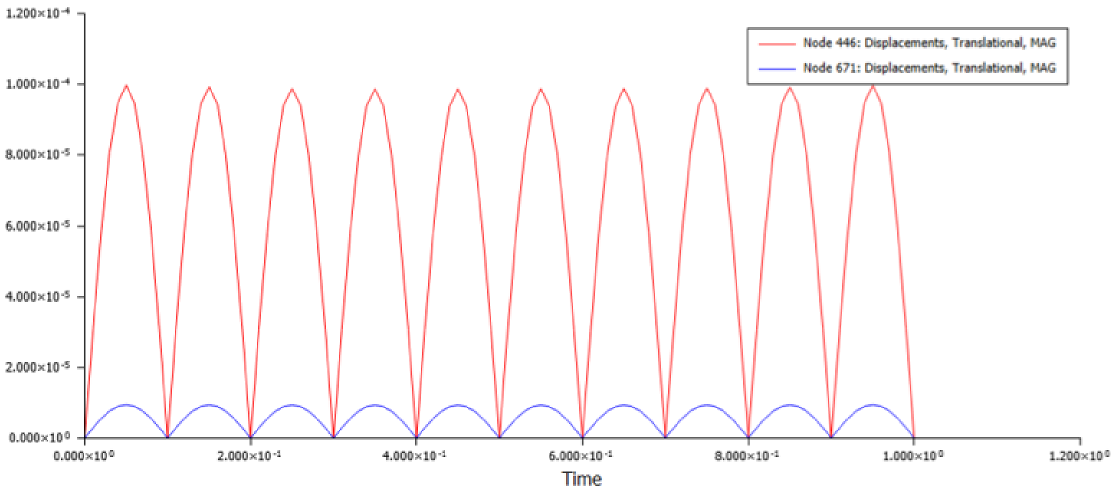
Stress at Z1 Path Length

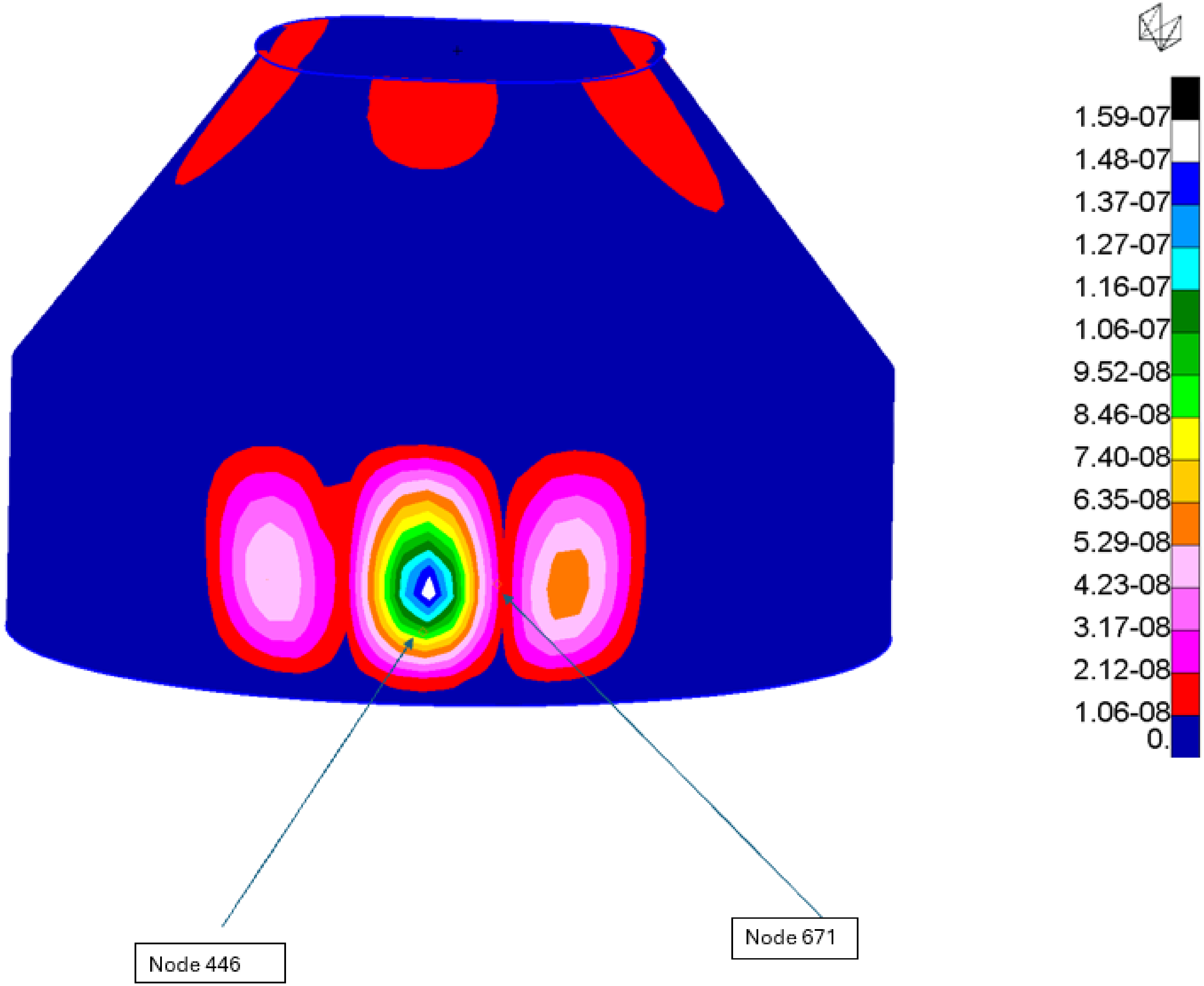


Stress At Z2 path Length

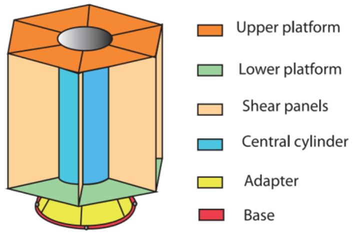


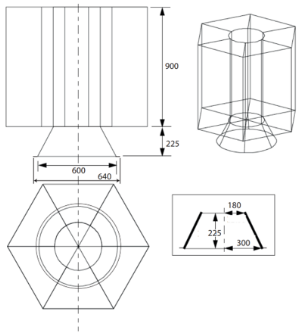
Displacement Intensity Comparison regarding to Node distance from the laoding

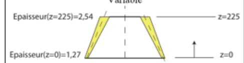



1.59-07
1.48-07
1.37-07
1.27-07
1.16-07
1.06-07
9.52-08
8.46-08
7.40-08
6.35-08
5.29-08
4.23-08
3.17-08
2.12-08
1.06-08
0.
Node 446
Node 671

2.Vibratory Study of a Satellite



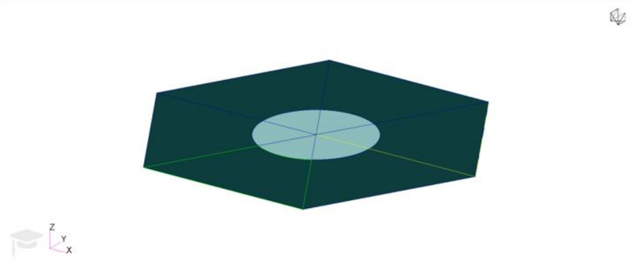


Partie du satellite	Matériau	Epaisseur (mm)
Upper platform	Aluminium	2,54
Lower platform	Aluminium	8,89
Shear panels	Aluminium	3,2
Central cylinder	Aluminium	6,35
Adapter	Titane	Variable 
Base	Aluminium	7

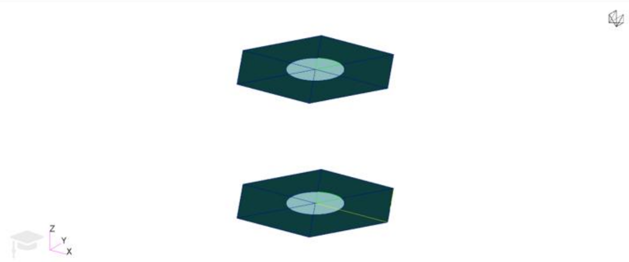
Aluminium : E=70 GPa ; γ=0,33 ; ρ=2700 kgm^{-3} ; $R_{e0,2}$=70 MPa
Titane : E=103 GPa ; γ=0,27 ; ρ=4510 kgm^{-3} ; $R_{e0,2}$=170 MPa

• The Design

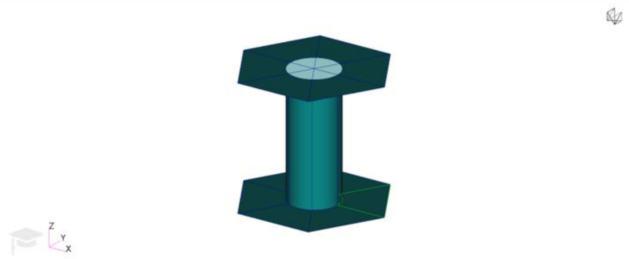
Lower Platform



Upper Platform

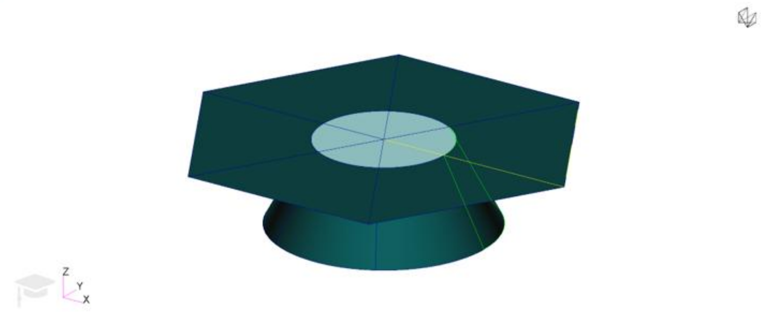


Central cylinder

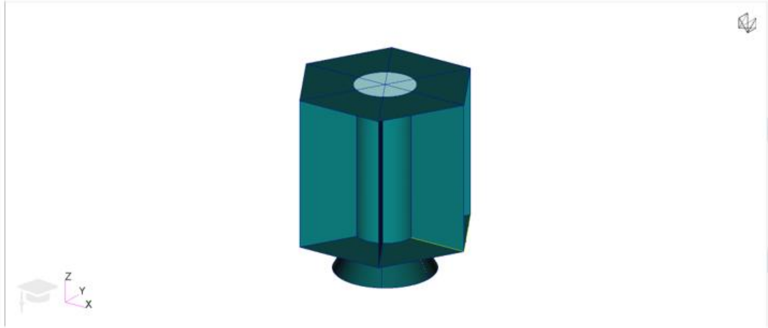


Shear Panels

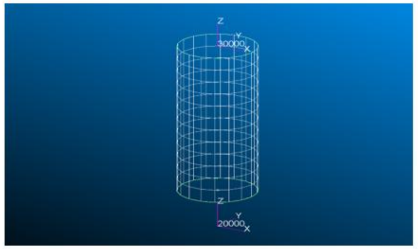
Adapter

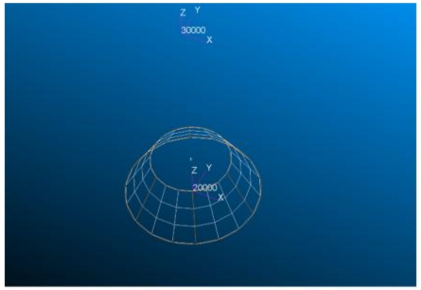


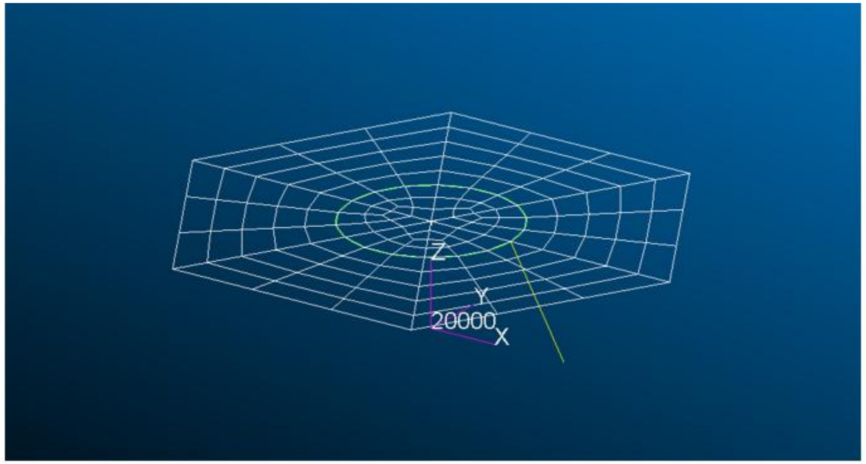
The Satellite Model Geometry

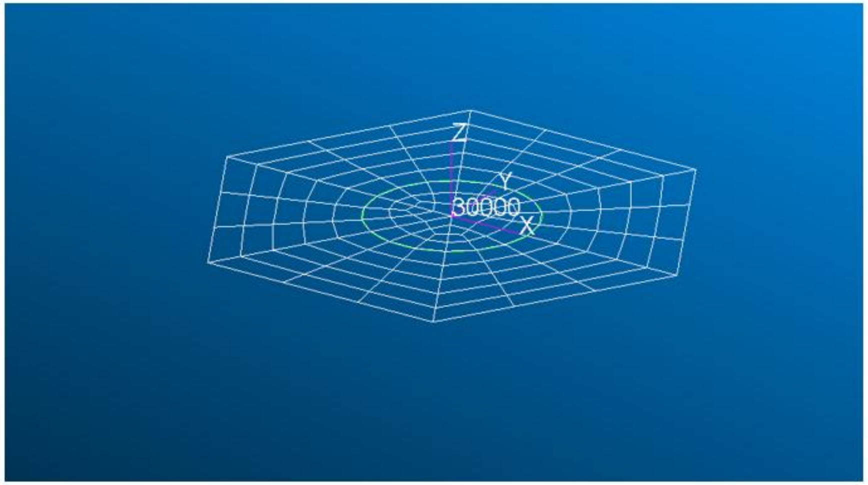


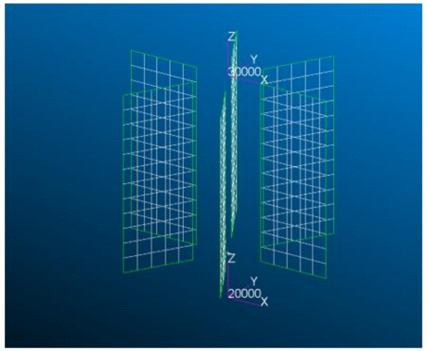
Meshing

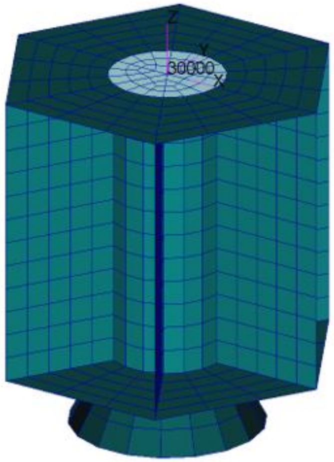

Z
Y
30000
X
Z
Y
20000
X

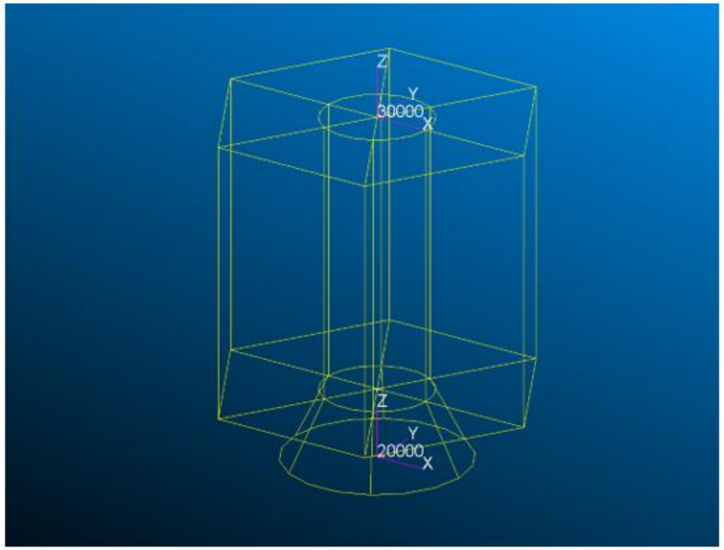

Z
Y
30000
X
Z
Y
20000
X


Z
Y
20000
X

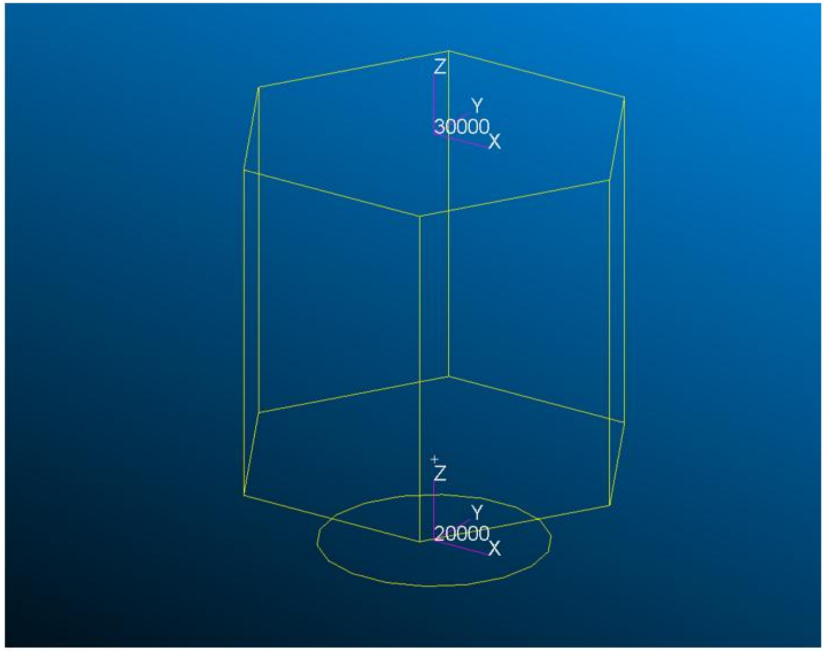

Z
Y
30000
X


Z
Y
30000
X
Z
Y
20000
X

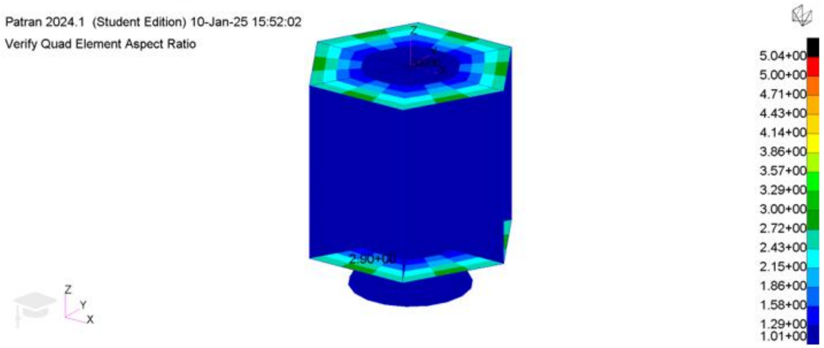

Z
Y
30000
X


Z
Y
30000
X
Z
Y
20000
X

After Equivalence

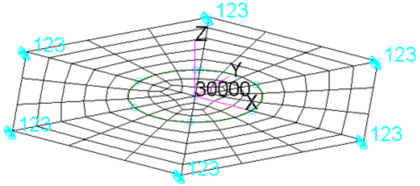


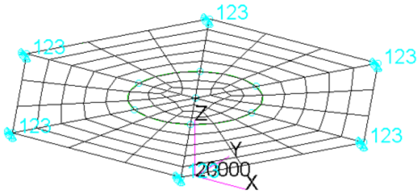
Final Verification of QUAD elements



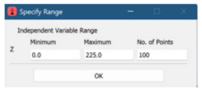
Satellite Properties and Boundaries Conditions

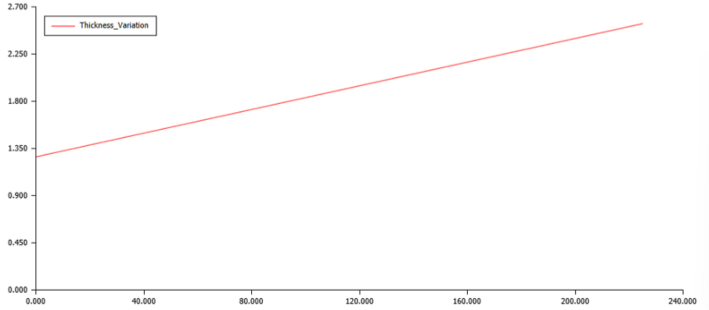
Lower and Upper Platform Launch Constraint



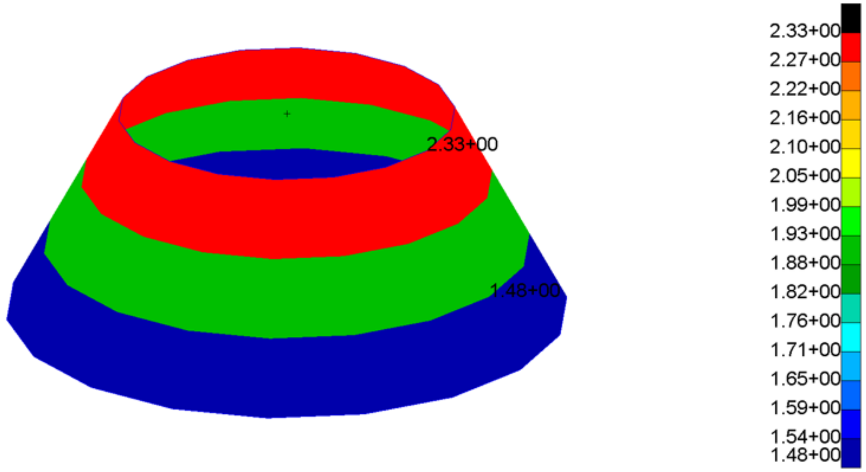


Thickness Variation

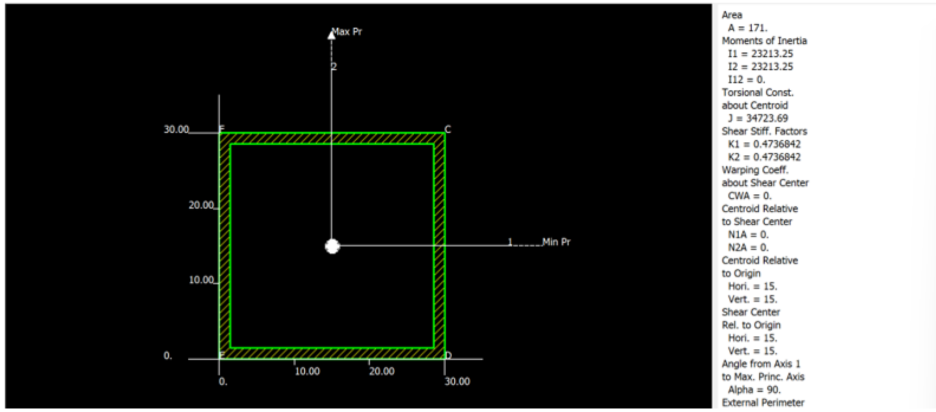


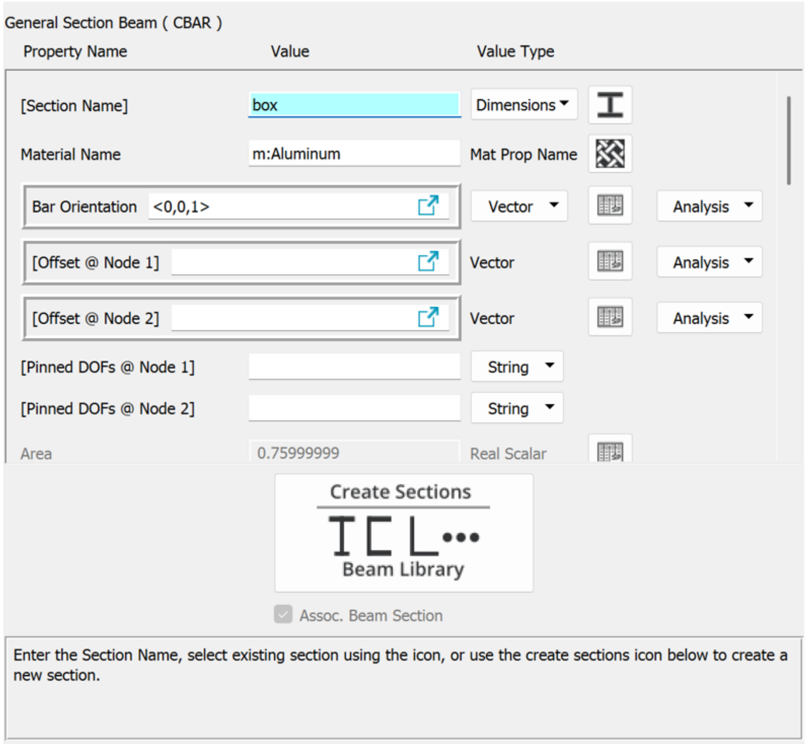


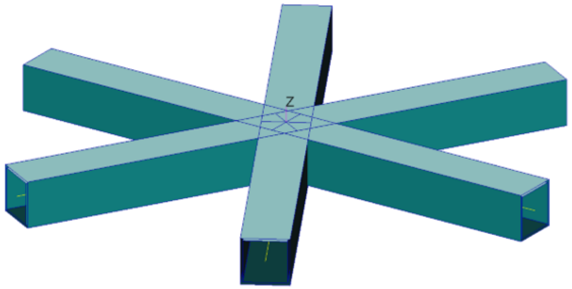
Thickness Variation as Applied to the Adapter Cone



Navigational Platform properties




General Section Beam (CBAR)
Property Name
Value
Value Type
[Section Name]
box
Dimensions
Material Name
m:Aluminum
Mat Prop Name
Bar Orientation
<0,0,1>
Vector
Analysis
[Offset @ Node 1]
Vector
Analysis
[Offset @ Node 2]
Vector
Analysis
[Pinned DOFs @ Node 1]
String
[Pinned DOFs @ Node 2]
String
Area
0.75999999
Real Scalar
Create Sections
Beam Library
Assoc. Beam Section
Enter the Section Name, select existing section using the icon, or use the create sections icon below to create a new section.



Mass Calculation

61.22 kg

Mass Properties display

Summary Display of Center of Gravity, Principal Inertias, Radii of Gyration, Mass, and Volume

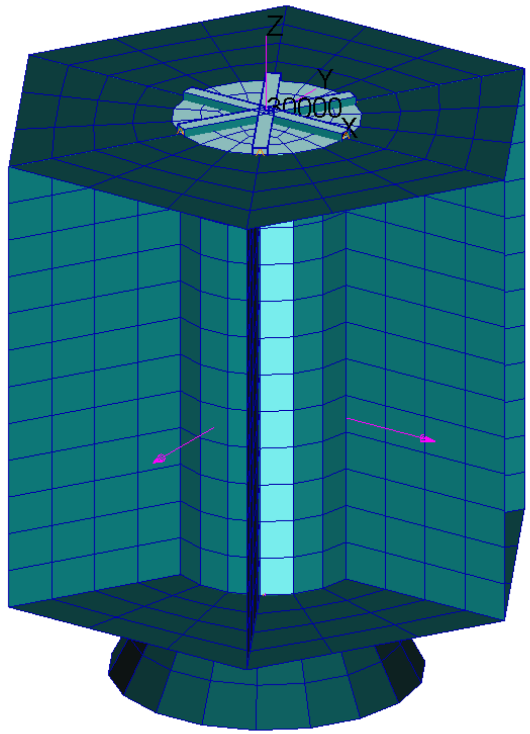
	CG(CID 0)	CG(CID 0)	I-Principal	Radii of Gyr.	Mass	Volume
1	-1.615E-06	-1.615E-06	5.666E+04	9.620E+02	6.122E-02	2.220E+07
2	-6.371E-07	-6.371E-07	5.666E+04	9.620E+02		
3	3.290E+02	3.290E+02	5.938E+03	3.114E+02		

Expanded Cell Value

Mass Property Display Option

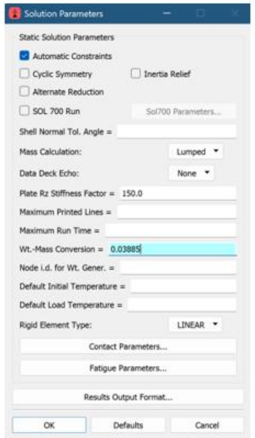
- Mass, CG, Principal Inertias, and Others
- Inertia Tensor
- Inertia Tensor at CG
- Principal Directions in User-Specified Frame
- Principal Directions in Ref. Cartesian Frame

Cancel

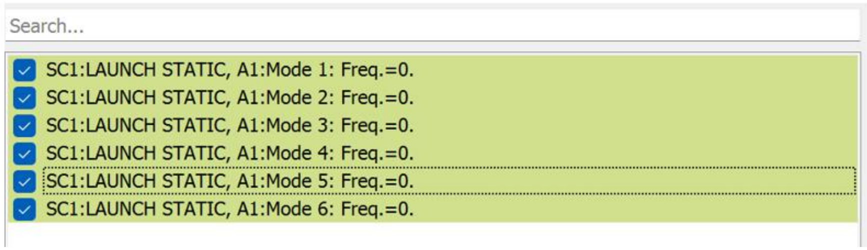

Z
Y
30000
X

Linear static Study

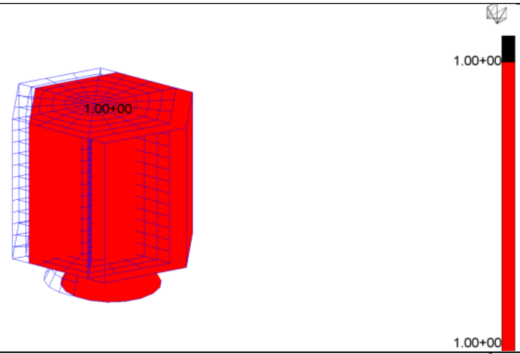
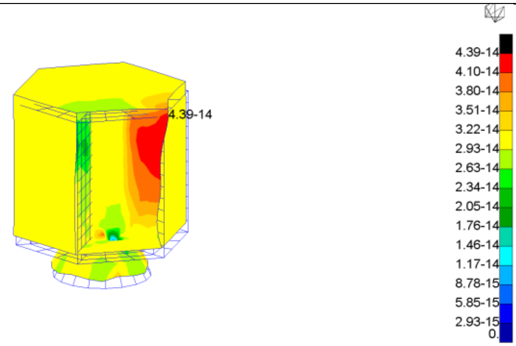
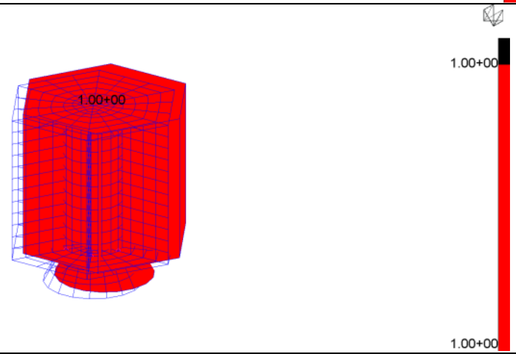
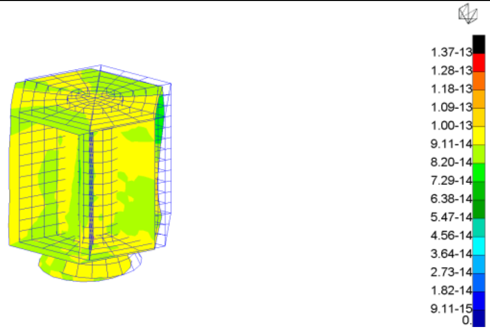
For our linear study, we tried the default number of modes, which is 6. We observed the stresses at the bars, particularly at the navigation platform, and also examined the stress tensors. All of this was, of course, done in the free state.



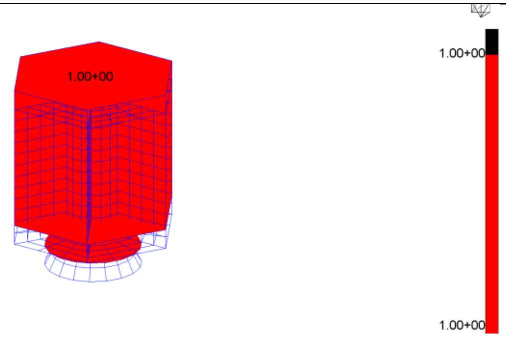
It is crucial to perform a static analysis of a structure before any other type of analysis, as it allows for an understanding of the structure's behaviour when it is free, meaning without constraints or fixations. This provides insights into how the structure can move in space and highlights the associated degrees of freedom. In this specific case, no external forces are applied to the structure, except for gravity, and it is modelled as being in equilibrium.

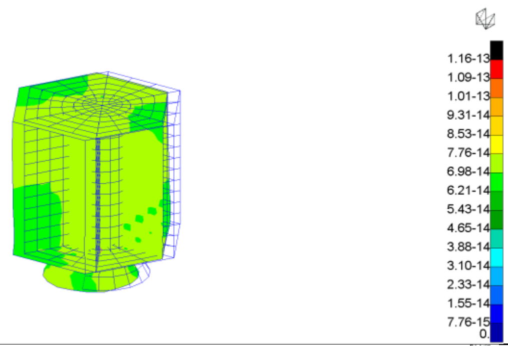


Displacement Eigenvectors

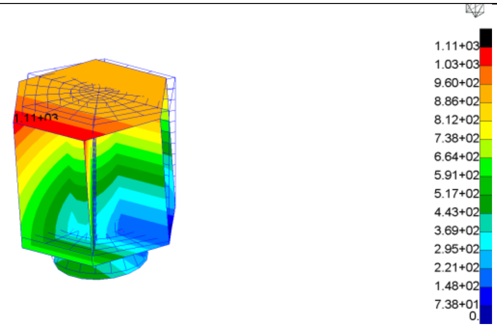
Frequencies	Eigenvectors Translational	Eigenvectors Rotational
Mode 1		
Mode 2		

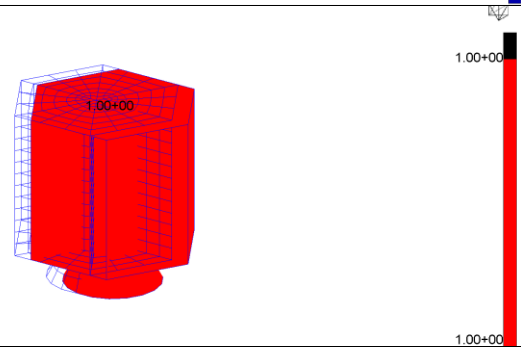
Mode 3



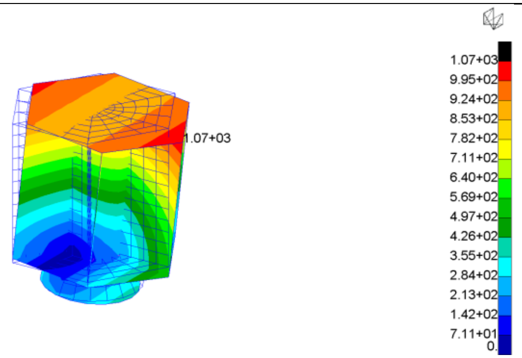


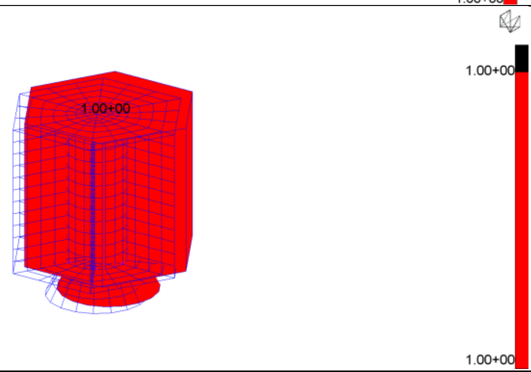
Mode 4

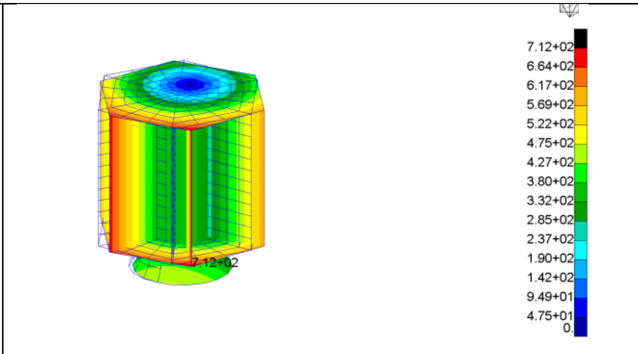
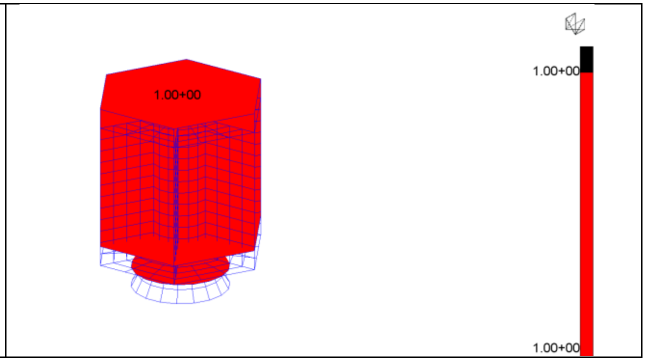




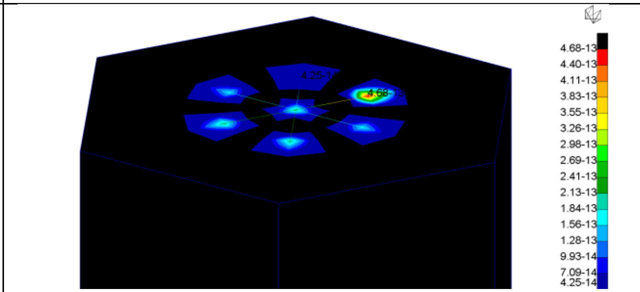
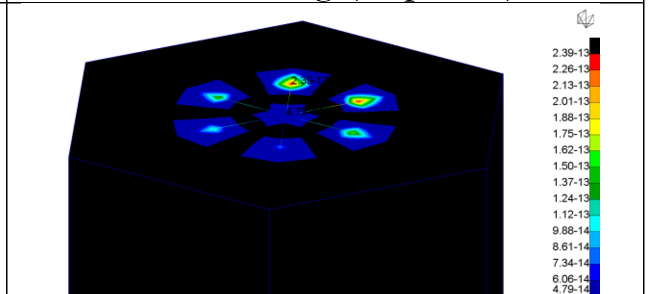
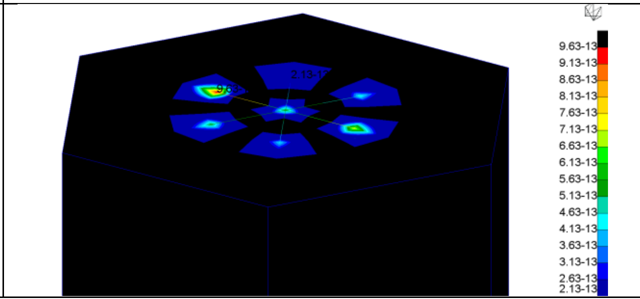
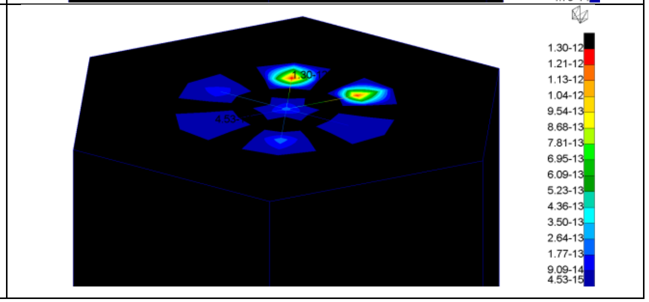
Mode 5

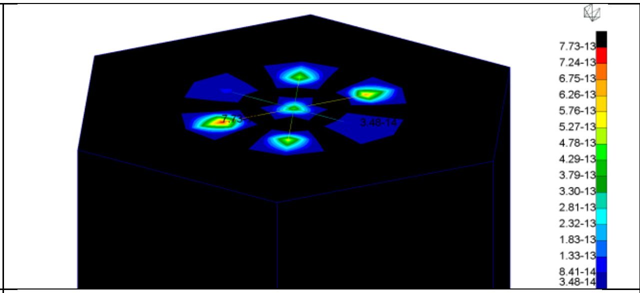
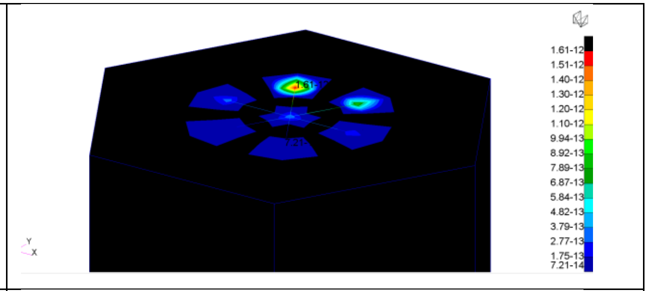
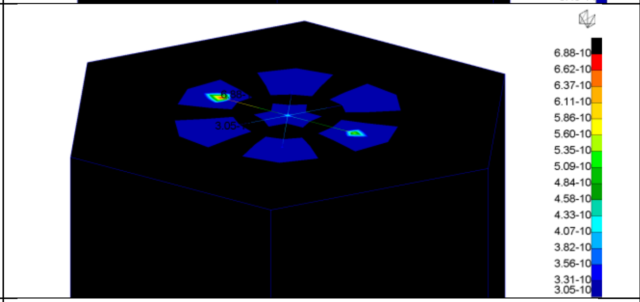
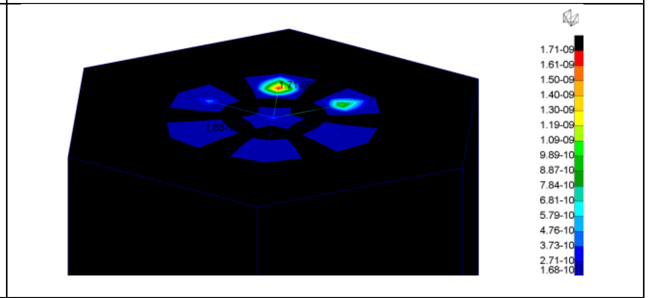
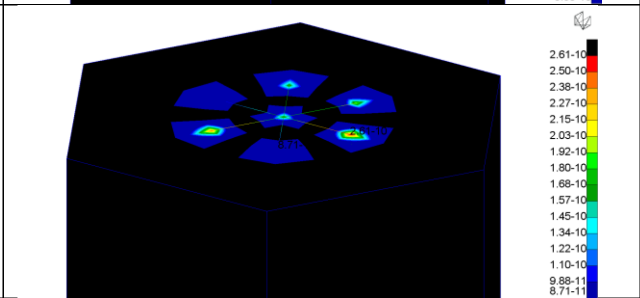
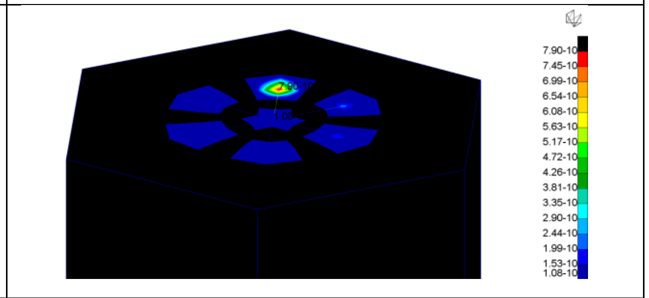
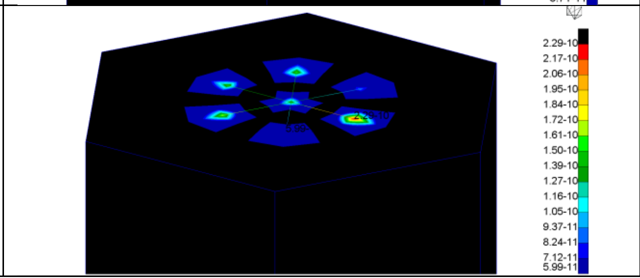
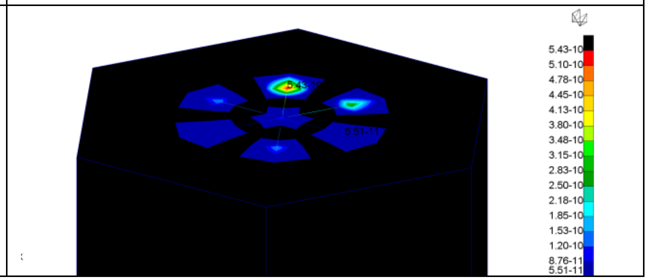




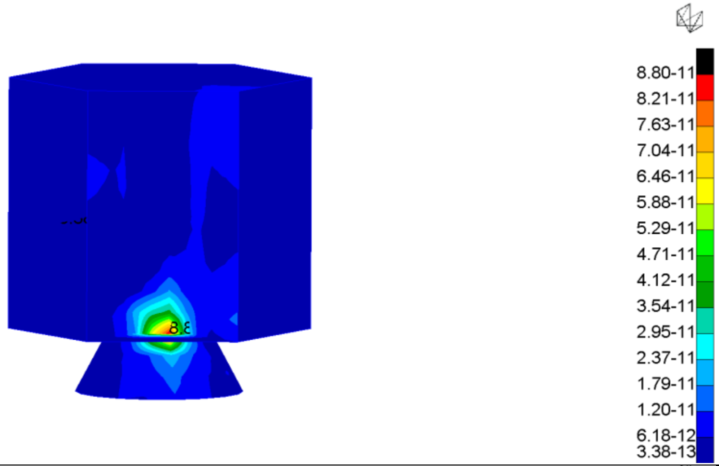
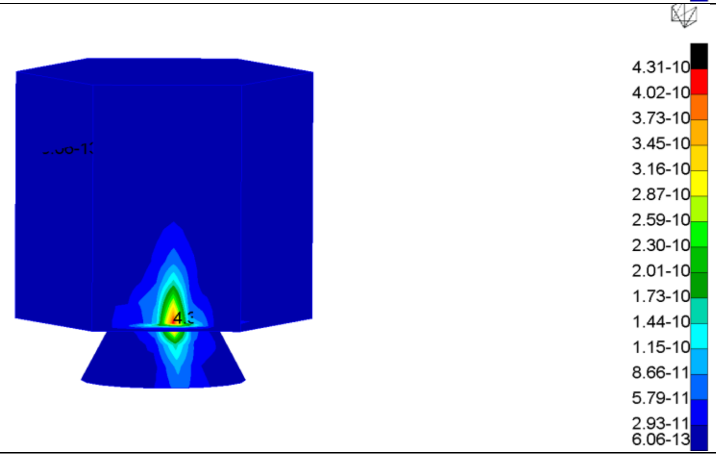
Mode 6		

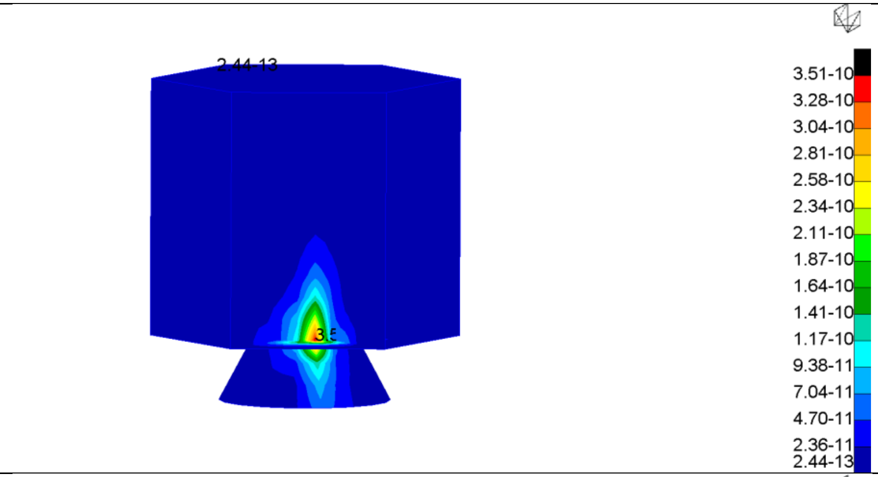
Bars Stresses

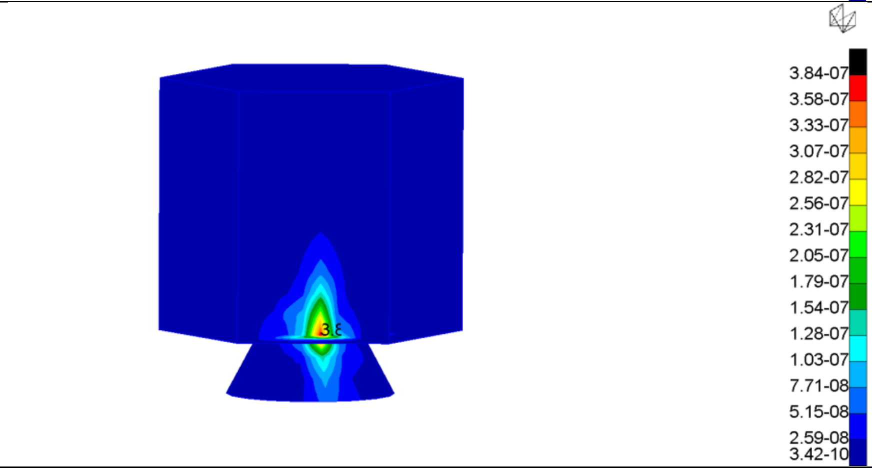
Modes	Bar Stress Axial Von Mises	Bar Stress Bending (all points)
1		
2		

3		
4		
5		
6		

Stress Tensors

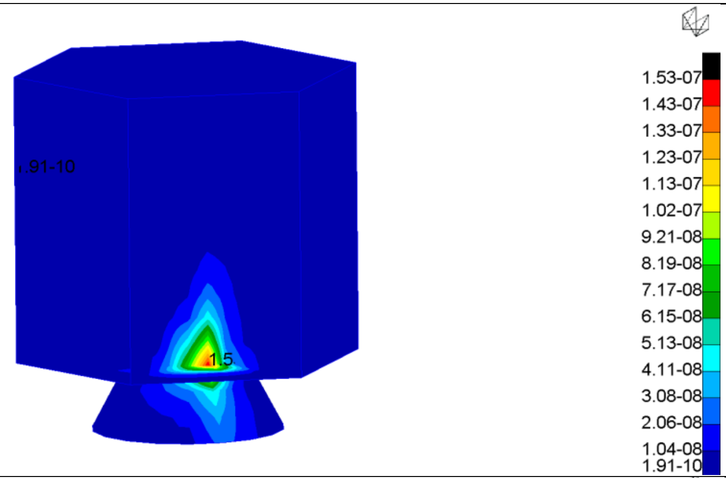
Modes	Stress Tensor
1	
2	

<table>
<tr>
<td>3</td>
<td>


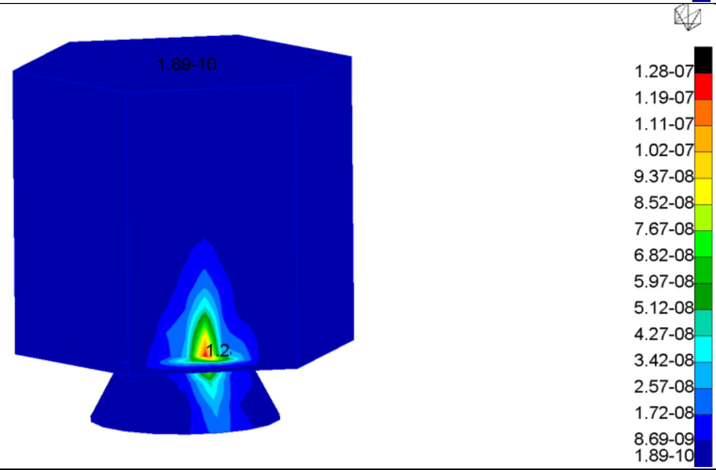
</td>
</tr>
<tr>
<td>4</td>
<td>


</td>
</tr>
</table>

5



6



The results of this static analysis show that the structure’s natural frequencies are close to zero. These frequencies correspond to the rigid modes of the structure, which are global movements of translation and rotation in space. These modes do not represent internal vibrations or elastic deformations but only global movements. This indicates that the structure is completely free, with no imposed constraints or fixations.

These results indicate that the structure can move or rotate freely without opposition. This is typical for structures that are neither fixed nor tightly constrained. The six rigid degrees of freedom (three translations + three rotations) reflect the structure's complete freedom of movement in space.

Role of Eigenvectors

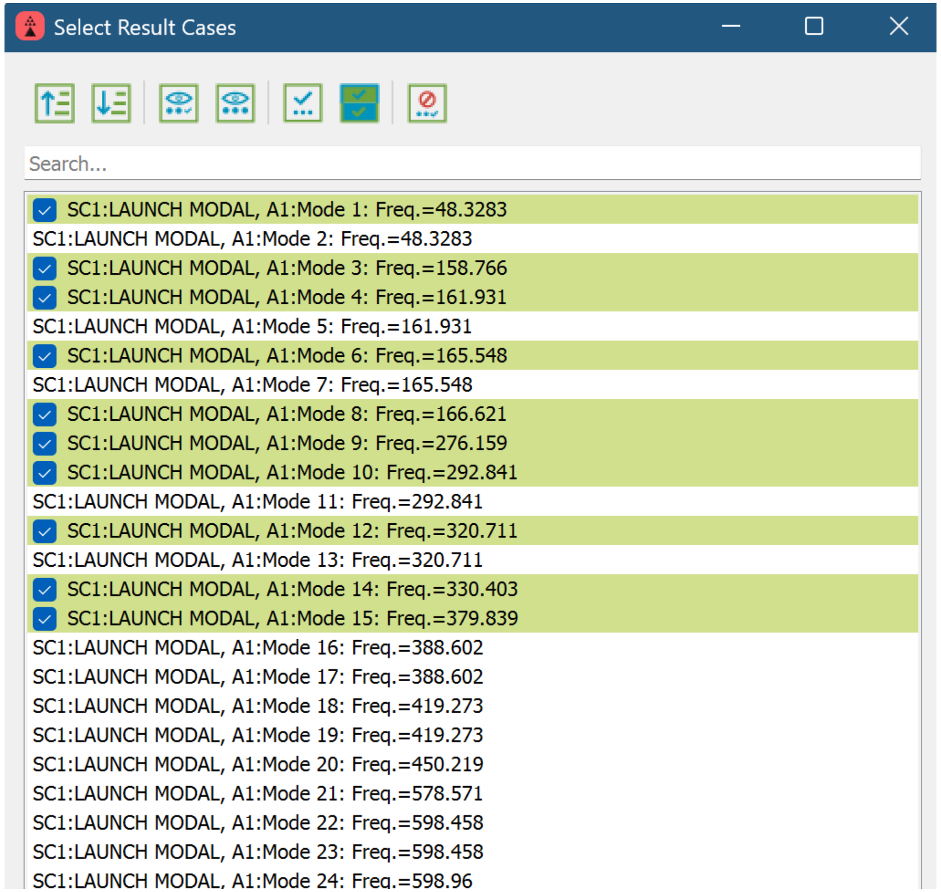
The eigenvectors associated with these frequencies indicate the directions of the rigid movements:

- **Translational modes** show linear displacements in the x, y, and z directions.
- **Rotational modes** illustrate pivots around the axes of rotation (x, y, and z).

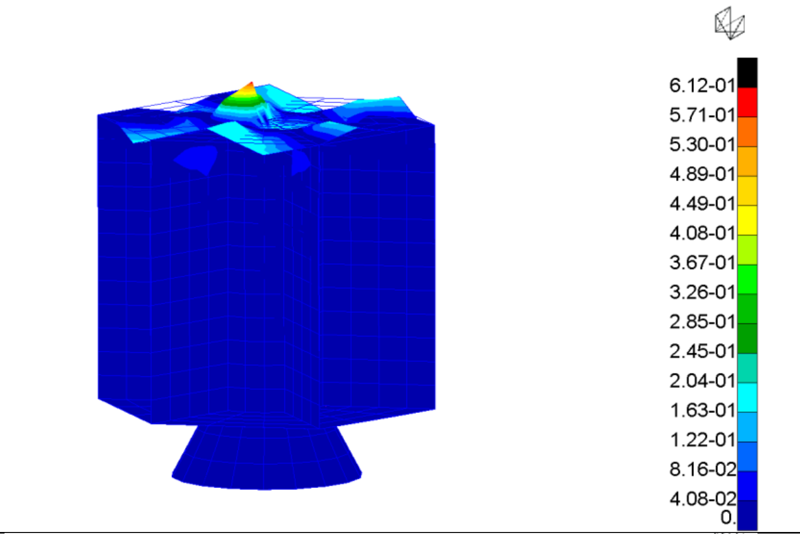
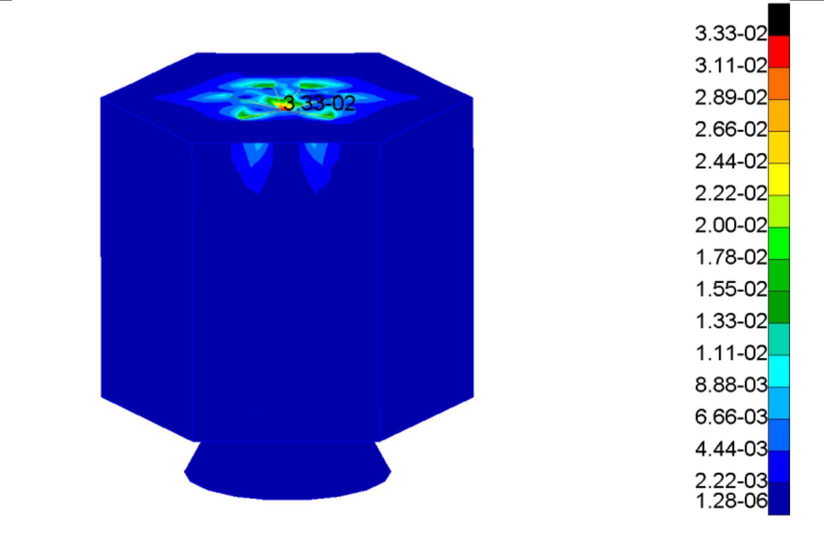
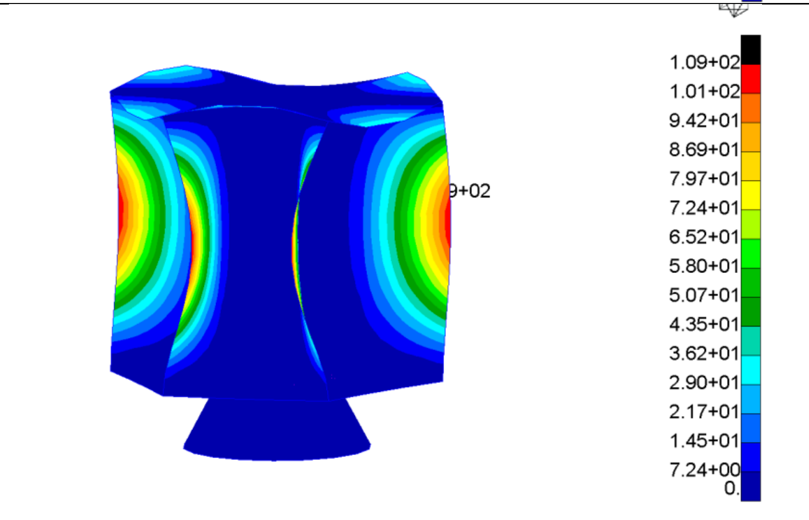
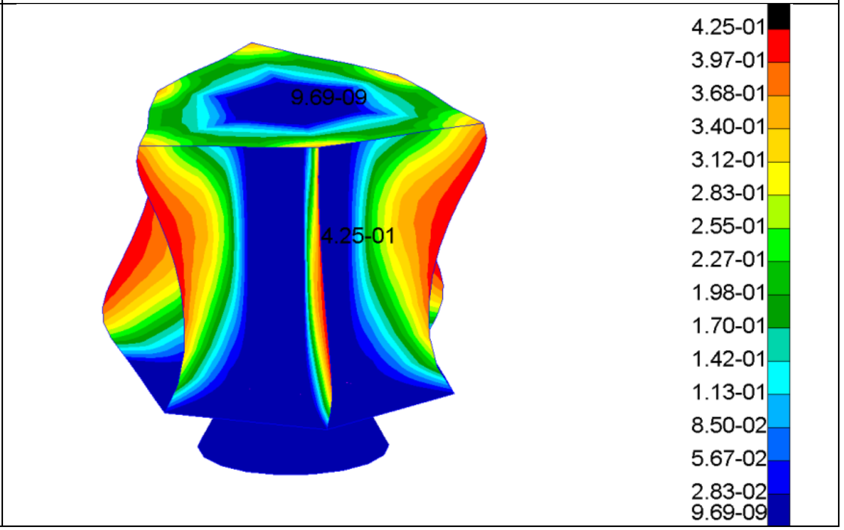
For example, an eigenvector can describe pure translation along the x-axis or a rotation around the z-axis. These vectors are useful for visualizing the global movements of the structure.

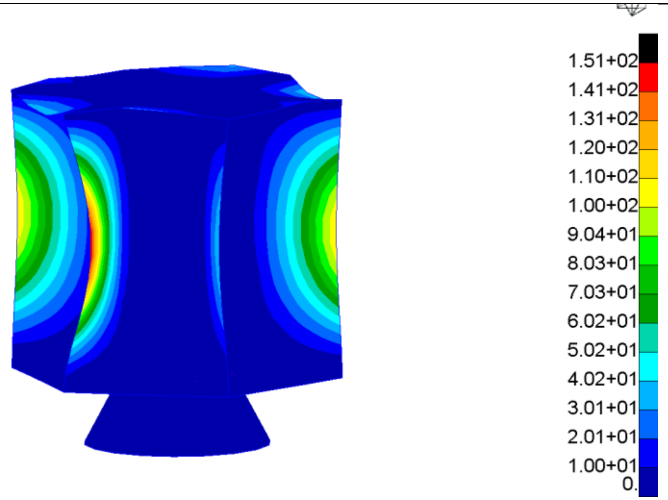
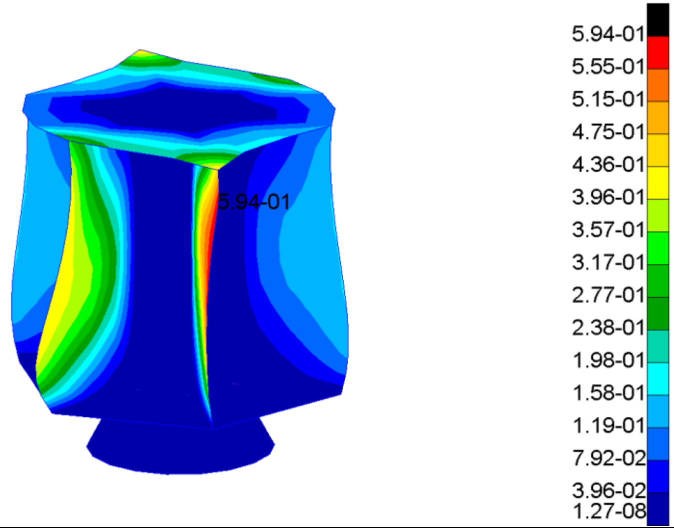
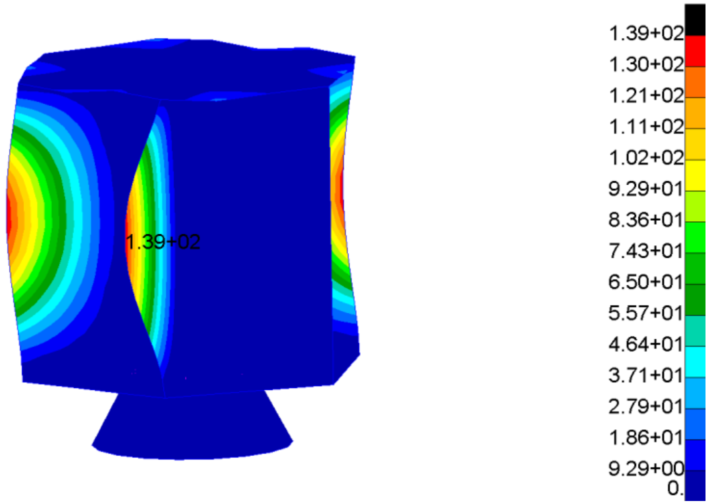
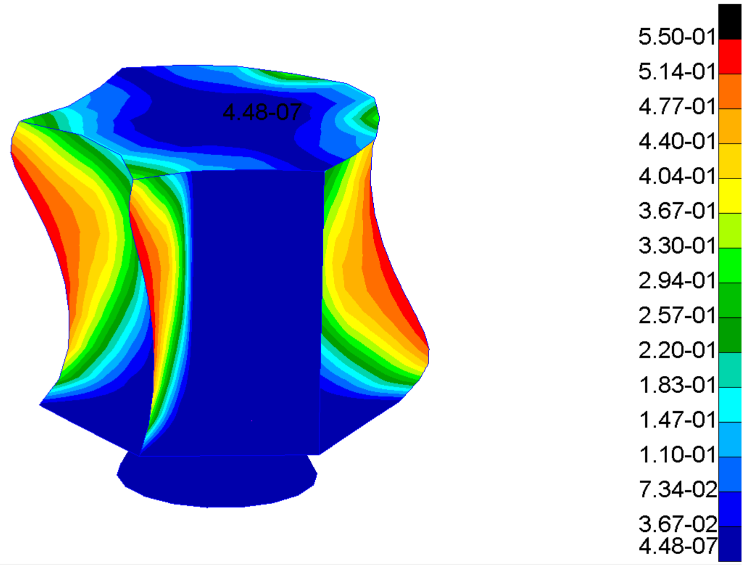
Modal analysis

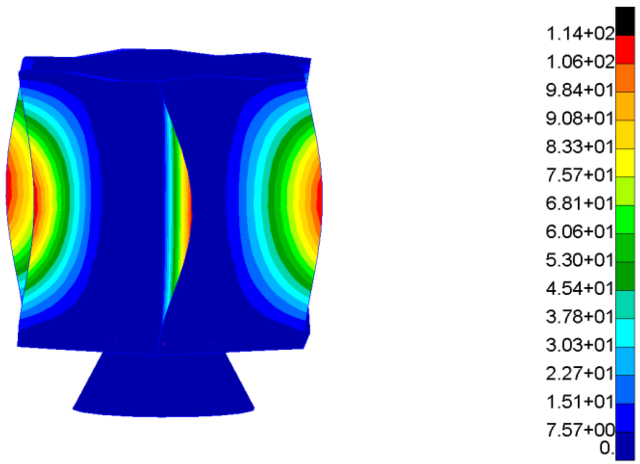
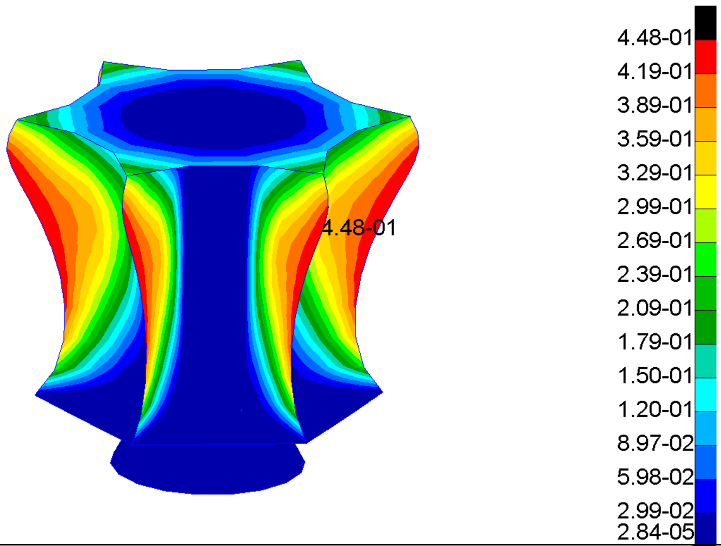
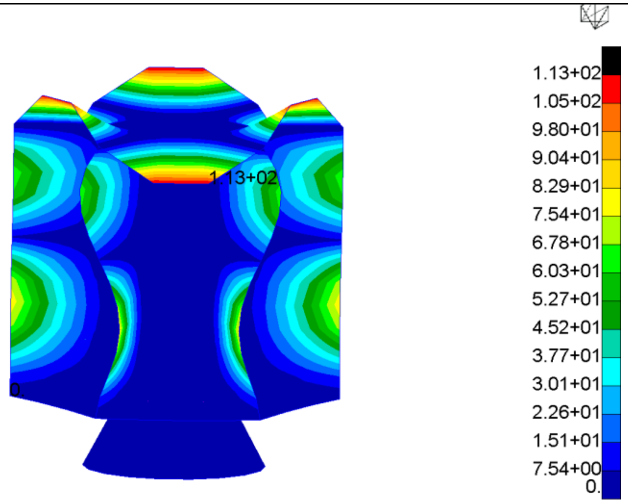
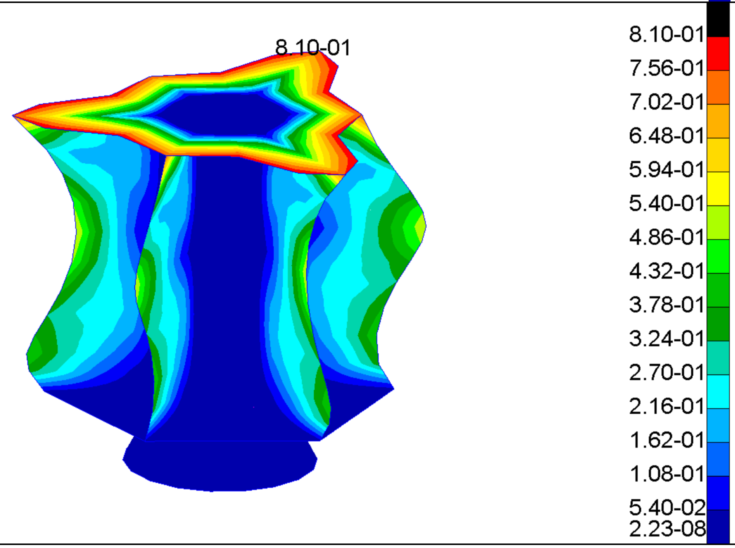
Calculation of natural vibration modes for free boundary conditions free



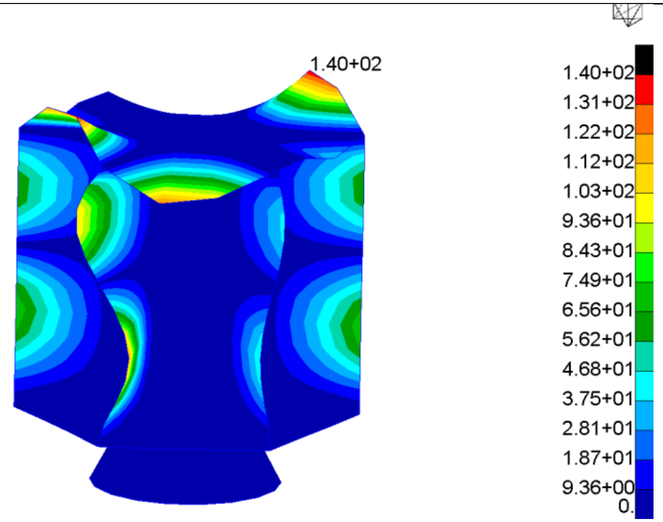
Modes	Eigenvectors Translational	Eigenvectors Rotational

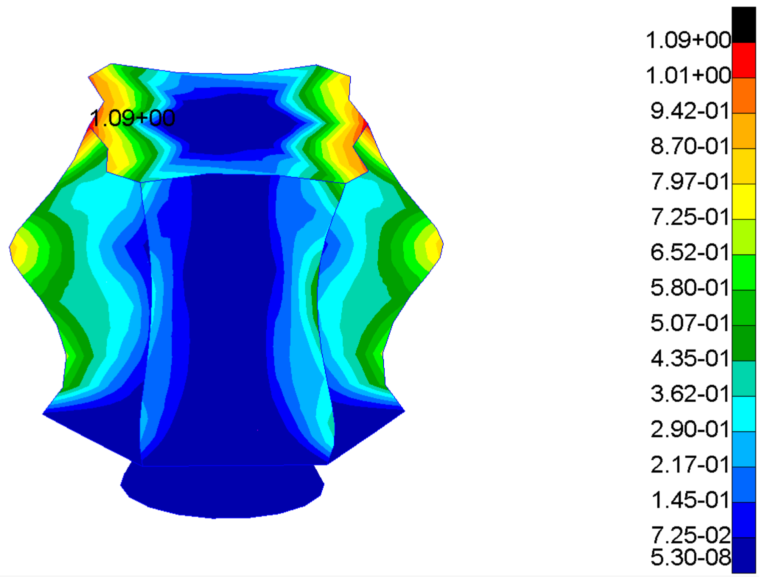
1	6.12-01 5.71-01 5.30-01 4.89-01 4.49-01 4.08-01 3.67-01 3.26-01 2.85-01 2.45-01 2.04-01 1.63-01 1.22-01 8.16-02 4.08-02 0.	3.33-02 3.33-02 3.11-02 2.89-02 2.66-02 2.44-02 2.22-02 2.00-02 1.78-02 1.55-02 1.33-02 1.11-02 8.88-03 6.66-03 4.44-03 2.22-03 1.28-06
3	9+02 1.09+02 1.01+02 9.42+01 8.69+01 7.97+01 7.24+01 6.52+01 5.80+01 5.07+01 4.35+01 3.62+01 2.90+01 2.17+01 1.45+01 7.24+00 0.	9.69-09 4.25-01 4.25-01 3.97-01 3.68-01 3.40-01 3.12-01 2.83-01 2.55-01 2.27-01 1.98-01 1.70-01 1.42-01 1.13-01 8.50-02 5.67-02 2.83-02 9.69-09

<table>
<tr>
<td>6</td>
<td>
</td>
<td>
</td>
</tr>
<tr>
<td>6</td>
<td>
</td>
<td>
</td>
</tr>
</table>

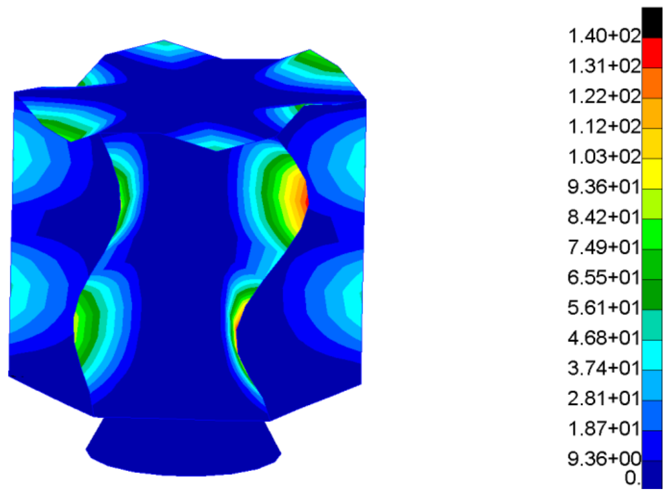
8	1.14+02 1.06+02 9.84+01 9.08+01 8.33+01 7.57+01 6.81+01 6.06+01 5.30+01 4.54+01 3.78+01 3.03+01 2.27+01 1.51+01 7.57+00 0.	4.48-01 4.48-01 4.19-01 3.89-01 3.59-01 3.29-01 2.99-01 2.69-01 2.39-01 2.09-01 1.79-01 1.50-01 1.20-01 8.97-02 5.98-02 2.99-02 2.84-05
9	1.13+02 0 1.13+02 1.05+02 9.80+01 9.04+01 8.29+01 7.54+01 6.78+01 6.03+01 5.27+01 4.52+01 3.77+01 3.01+01 2.26+01 1.51+01 7.54+00 0.	8.10-01 8.10-01 7.56-01 7.02-01 6.48-01 5.94-01 5.40-01 4.86-01 4.32-01 3.78-01 3.24-01 2.70-01 2.16-01 1.62-01 1.08-01 5.40-02 2.23-08

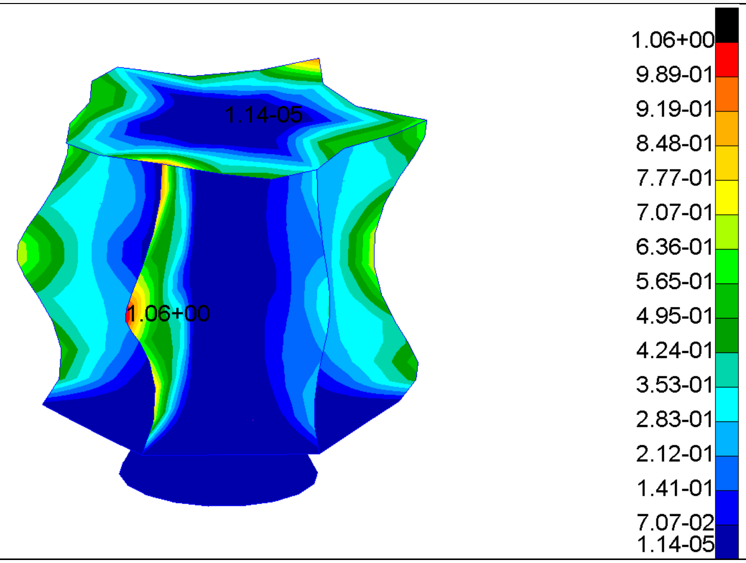
10


1.40+02
1.40+02
1.31+02
1.22+02
1.12+02
1.03+02
9.36+01
8.43+01
7.49+01
6.56+01
5.62+01
4.68+01
3.75+01
2.81+01
1.87+01
9.36+00
0.

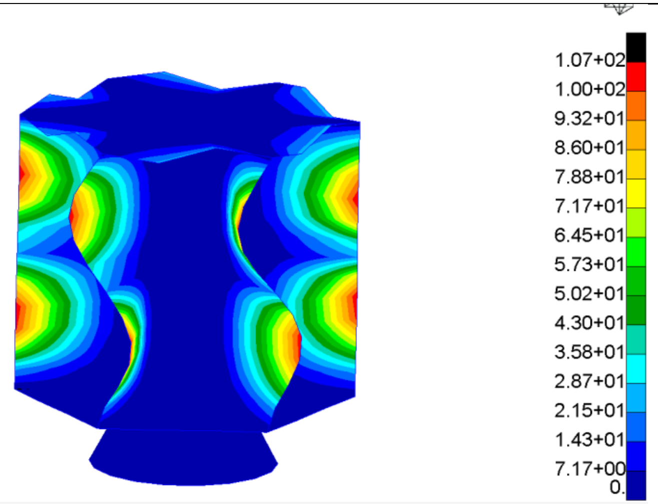

1.09+00
1.09+00
1.01+00
9.42-01
8.70-01
7.97-01
7.25-01
6.52-01
5.80-01
5.07-01
4.35-01
3.62-01
2.90-01
2.17-01
1.45-01
7.25-02
5.30-08

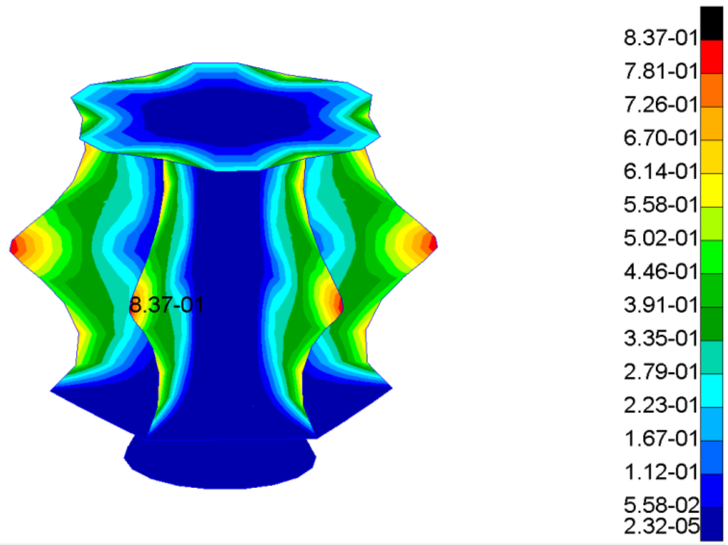
12


1.40+02
1.31+02
1.22+02
1.12+02
1.03+02
9.36+01
8.42+01
7.49+01
6.55+01
5.61+01
4.68+01
3.74+01
2.81+01
1.87+01
9.36+00
0.

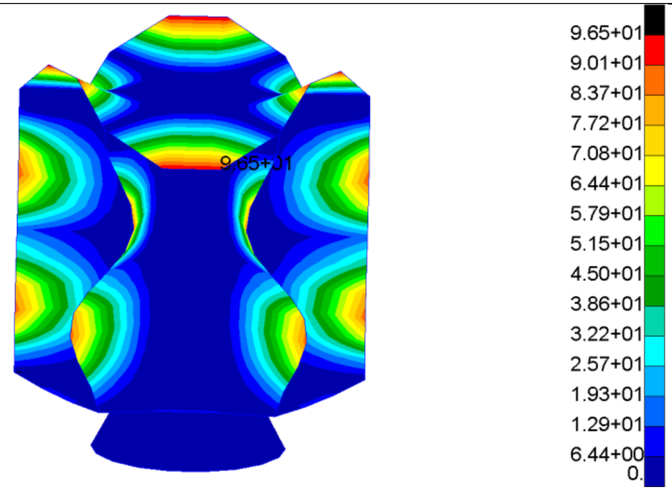

1.14-05
1.06+00
1.06+00
9.89-01
9.19-01
8.48-01
7.77-01
7.07-01
6.36-01
5.65-01
4.95-01
4.24-01
3.53-01
2.83-01
2.12-01
1.41-01
7.07-02
1.14-05

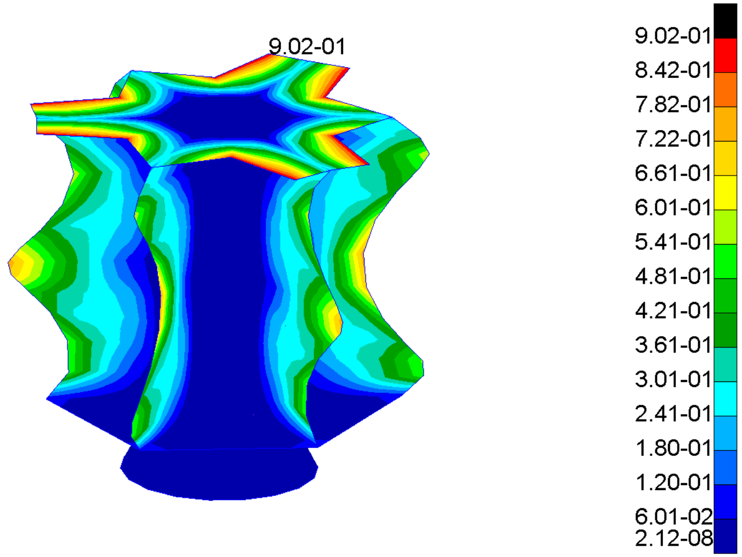
14


1.07+02
1.00+02
9.32+01
8.60+01
7.88+01
7.17+01
6.45+01
5.73+01
5.02+01
4.30+01
3.58+01
2.87+01
2.15+01
1.43+01
7.17+00
0.

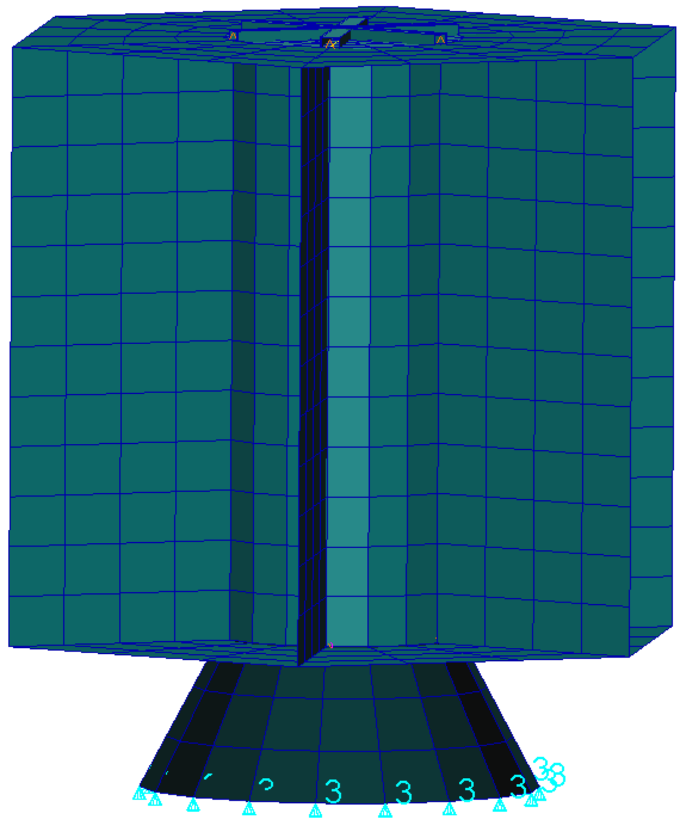

8.37-01
7.81-01
7.26-01
6.70-01
6.14-01
5.58-01
5.02-01
4.46-01
3.91-01
3.35-01
2.79-01
2.23-01
1.67-01
1.12-01
5.58-02
2.32-05
8.37-01

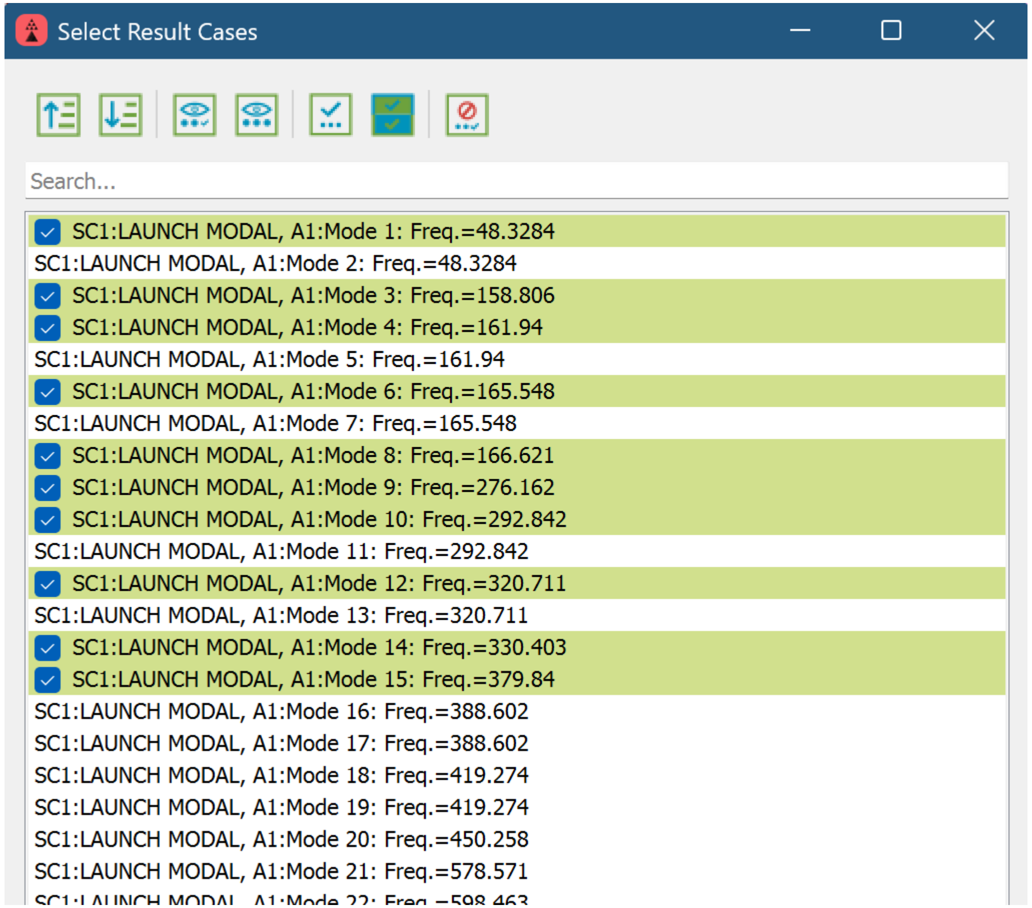
15


9.65+01
9.01+01
8.37+01
7.72+01
7.08+01
6.44+01
5.79+01
5.15+01
4.50+01
3.86+01
3.22+01
2.57+01
1.93+01
1.29+01
6.44+00
0.

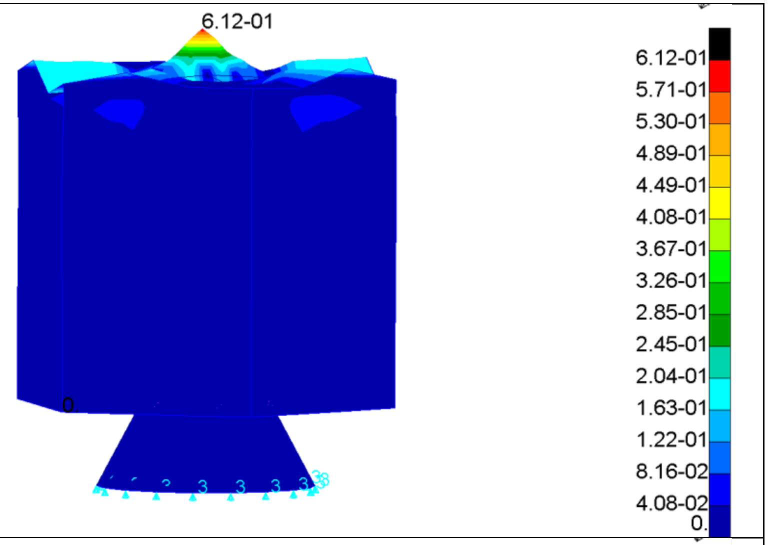
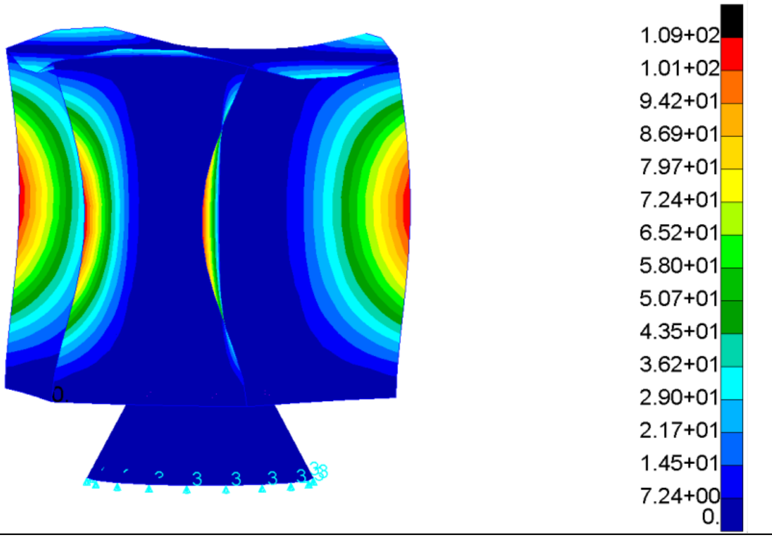

9.02-01
9.02-01
8.42-01
7.82-01
7.22-01
6.61-01
6.01-01
5.41-01
4.81-01
4.21-01
3.61-01
3.01-01
2.41-01
1.80-01
1.20-01
6.01-02
2.12-08

Calculation of natural vibration modes for simply supported boundary conditions

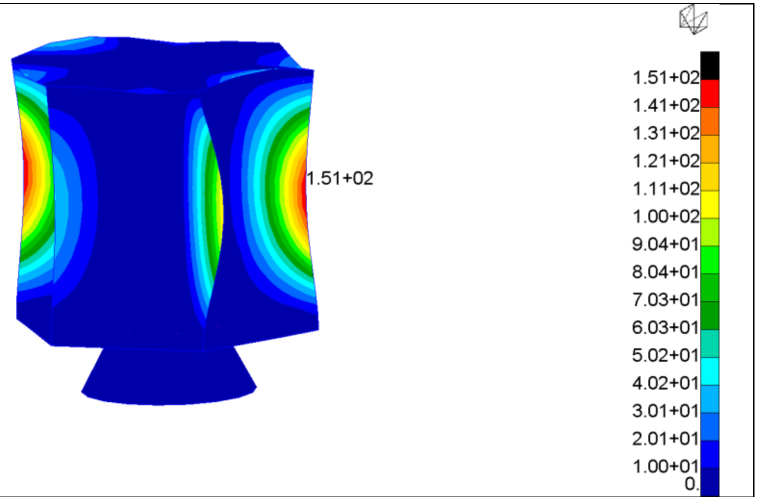




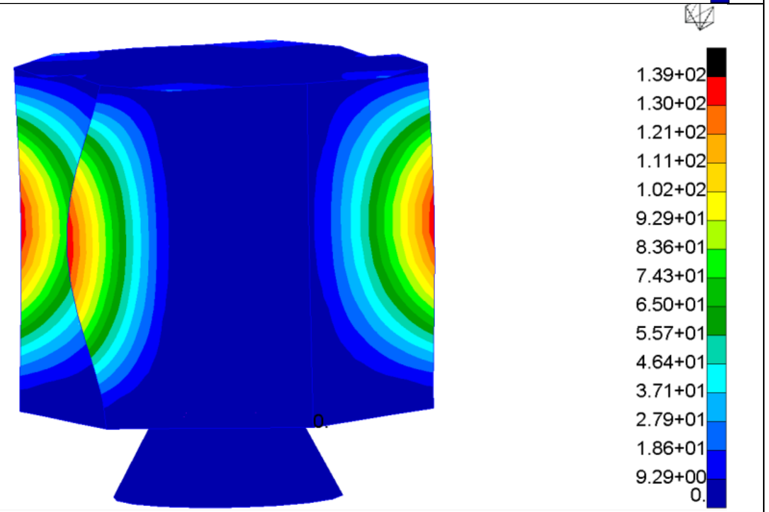
Modes	Eigenvectors Translational

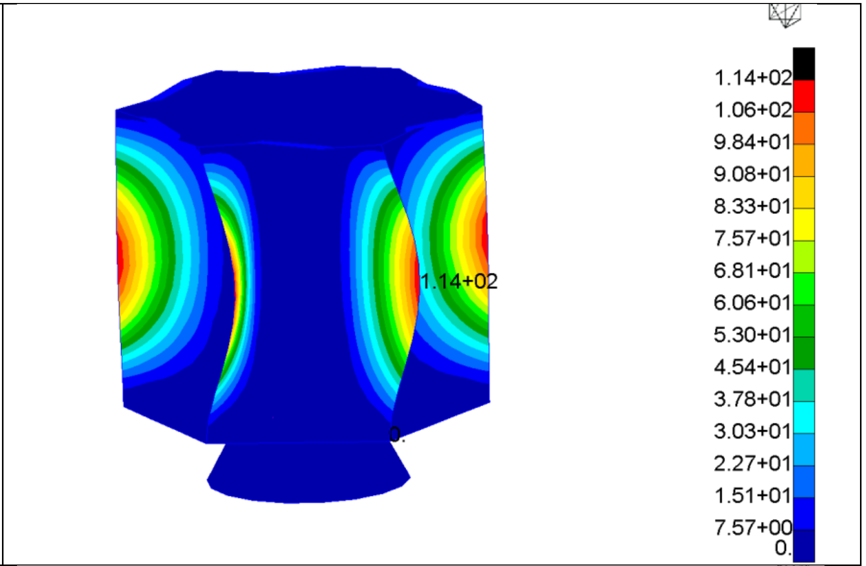
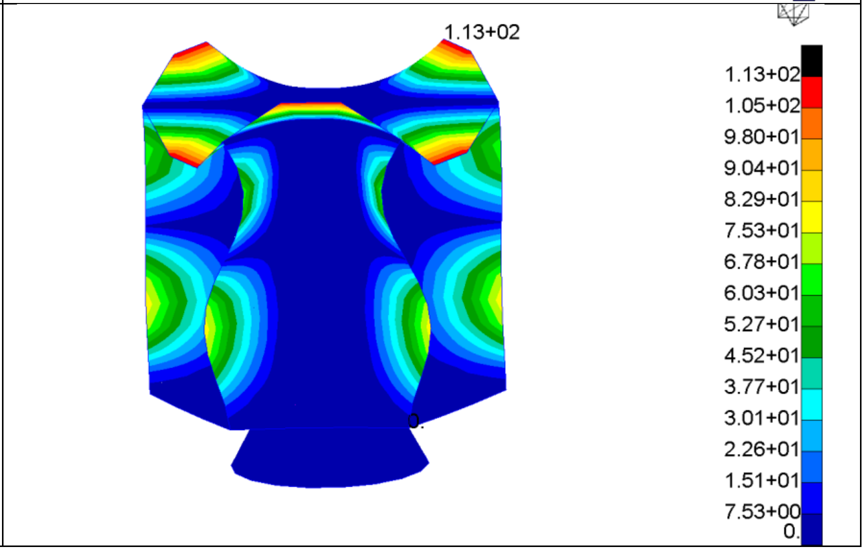
1	6.12-01 6.12-01 5.71-01 5.30-01 4.89-01 4.49-01 4.08-01 3.67-01 3.26-01 2.85-01 2.45-01 2.04-01 1.63-01 1.22-01 8.16-02 4.08-02 0.
3	1.09+02 1.01+02 9.42+01 8.69+01 7.97+01 7.24+01 6.52+01 5.80+01 5.07+01 4.35+01 3.62+01 2.90+01 2.17+01 1.45+01 7.24+00 0.

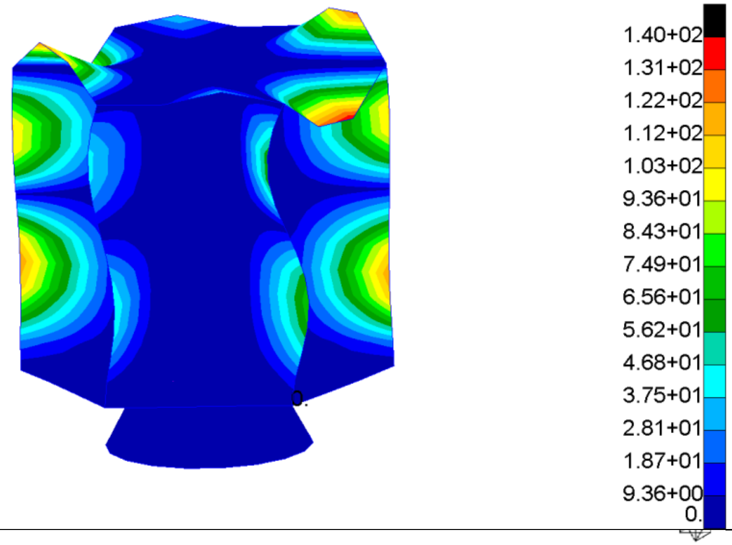
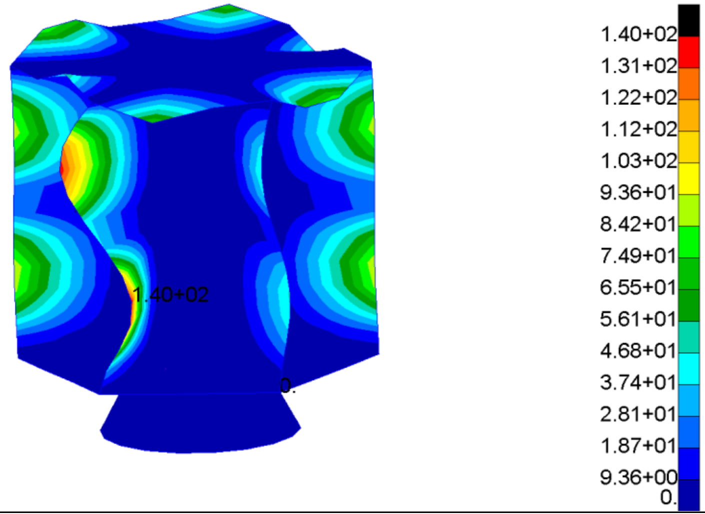
4

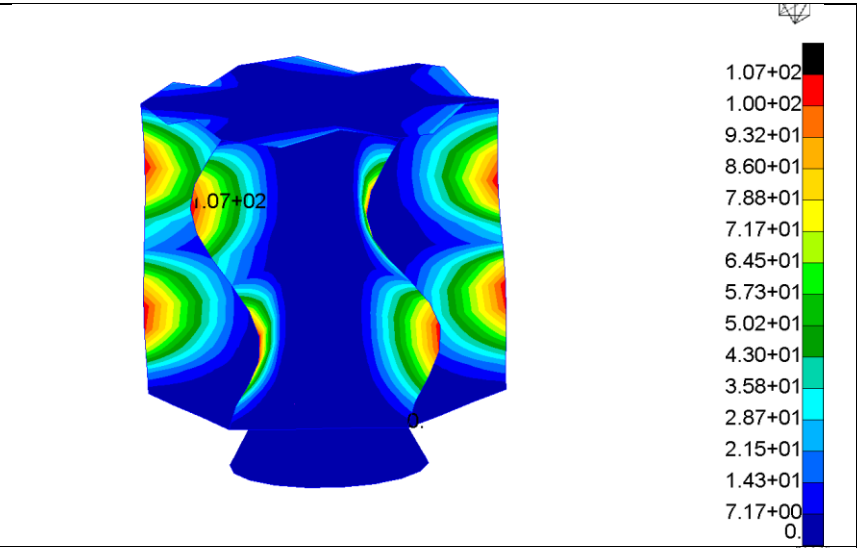
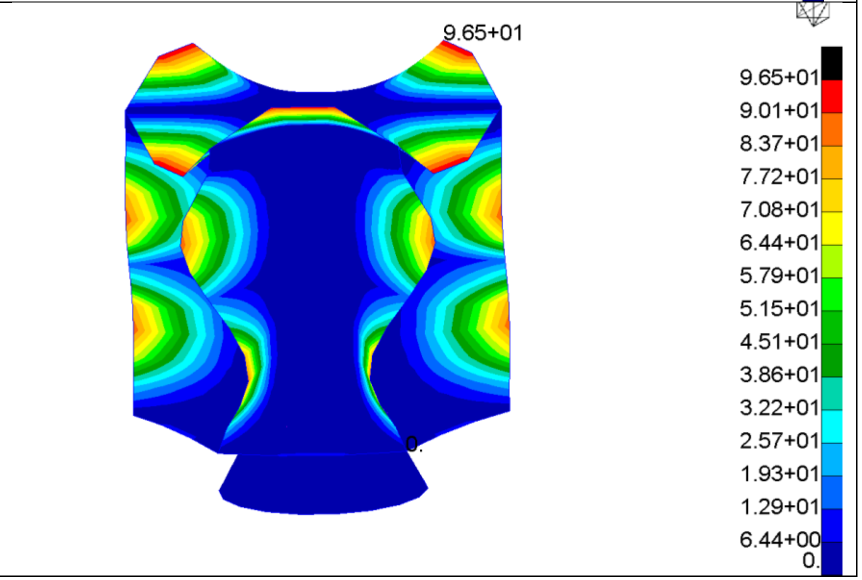

1.51+02
1.41+02
1.31+02
1.21+02
1.11+02
1.00+02
9.04+01
8.04+01
7.03+01
6.03+01
5.02+01
4.02+01
3.01+01
2.01+01
1.00+01
0.
1.51+02

6

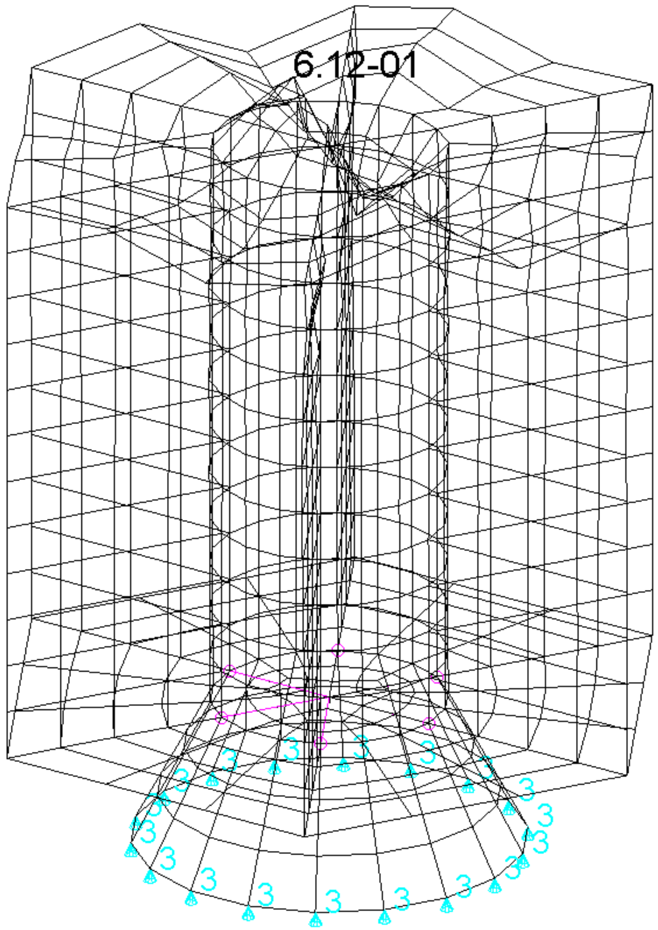

1.39+02
1.30+02
1.21+02
1.11+02
1.02+02
9.29+01
8.36+01
7.43+01
6.50+01
5.57+01
4.64+01
3.71+01
2.79+01
1.86+01
9.29+00
0.
0.

8	 1.14+02 1.06+02 9.84+01 9.08+01 8.33+01 7.57+01 6.81+01 6.06+01 5.30+01 4.54+01 3.78+01 3.03+01 2.27+01 1.51+01 7.57+00 0. 1.14+02 0.
9	 1.13+02 1.13+02 1.05+02 9.80+01 9.04+01 8.29+01 7.53+01 6.78+01 6.03+01 5.27+01 4.52+01 3.77+01 3.01+01 2.26+01 1.51+01 7.53+00 0. 0.

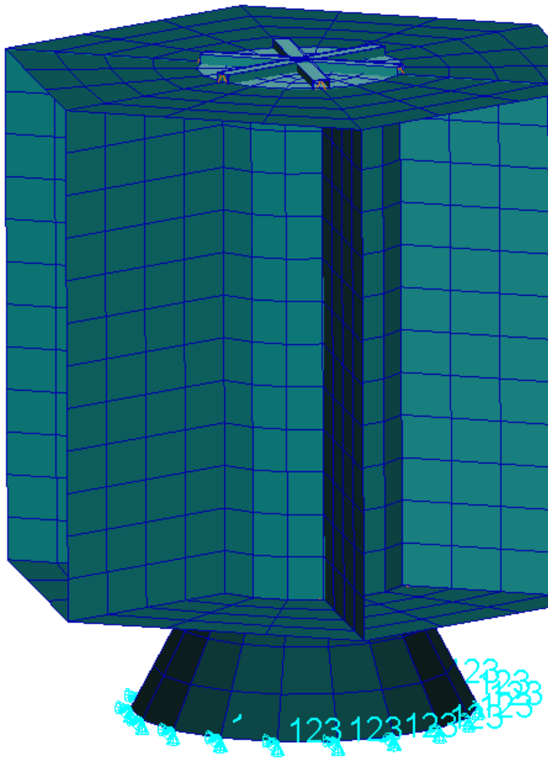
<table>
<tr><td>10</td><td>
</td></tr>
<tr><td>12</td><td>
</td></tr>
</table>

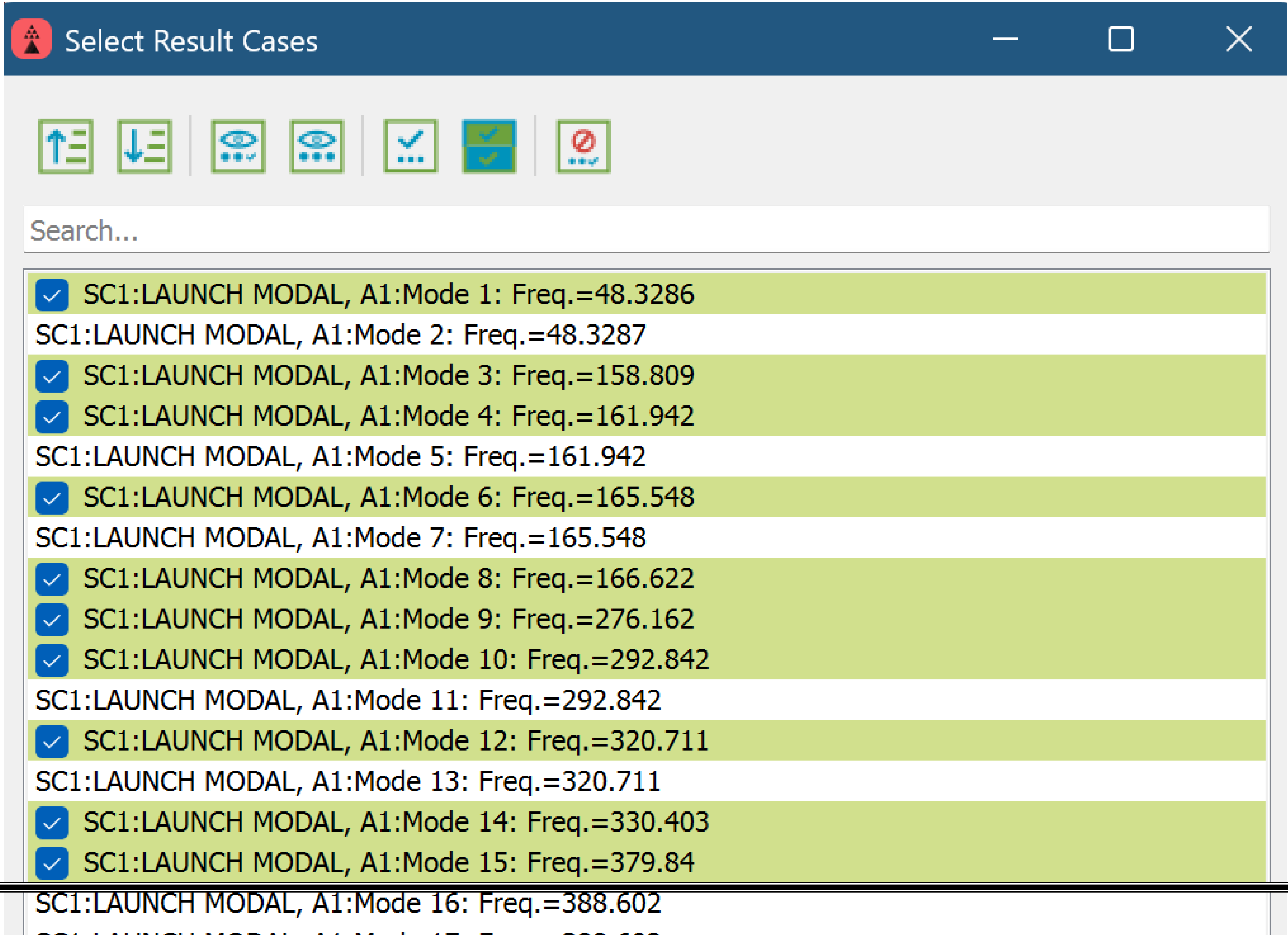
14	1.07+02 0. 1.07+02 1.00+02 9.32+01 8.60+01 7.88+01 7.17+01 6.45+01 5.73+01 5.02+01 4.30+01 3.58+01 2.87+01 2.15+01 1.43+01 7.17+00 0.
15	9.65+01 0. 9.65+01 9.01+01 8.37+01 7.72+01 7.08+01 6.44+01 5.79+01 5.15+01 4.51+01 3.86+01 3.22+01 2.57+01 1.93+01 1.29+01 6.44+00 0.

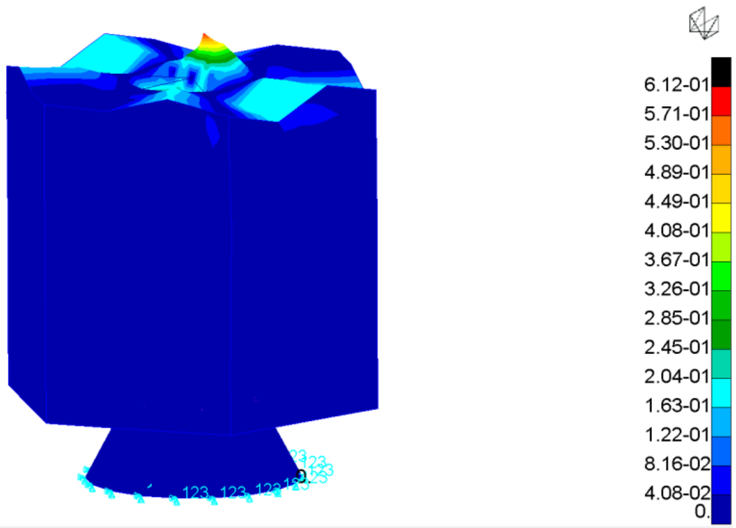
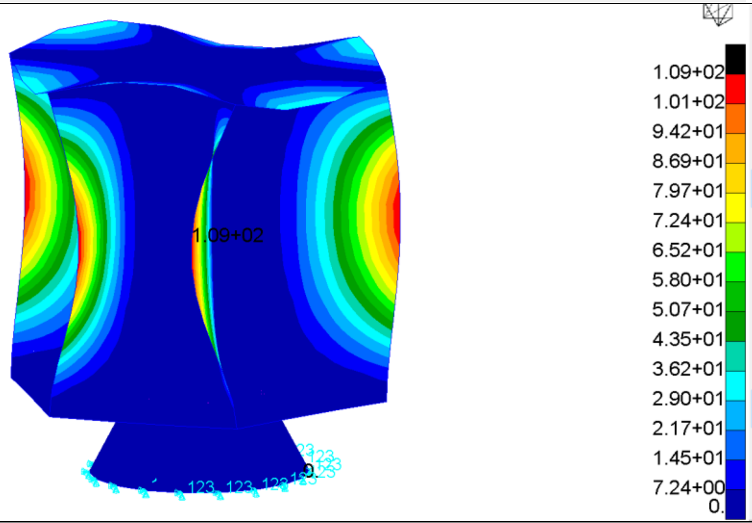
At the first frequency

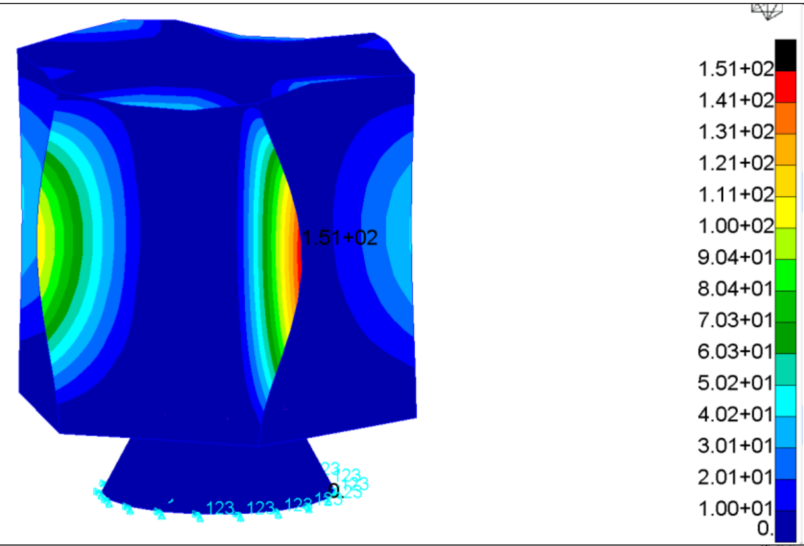
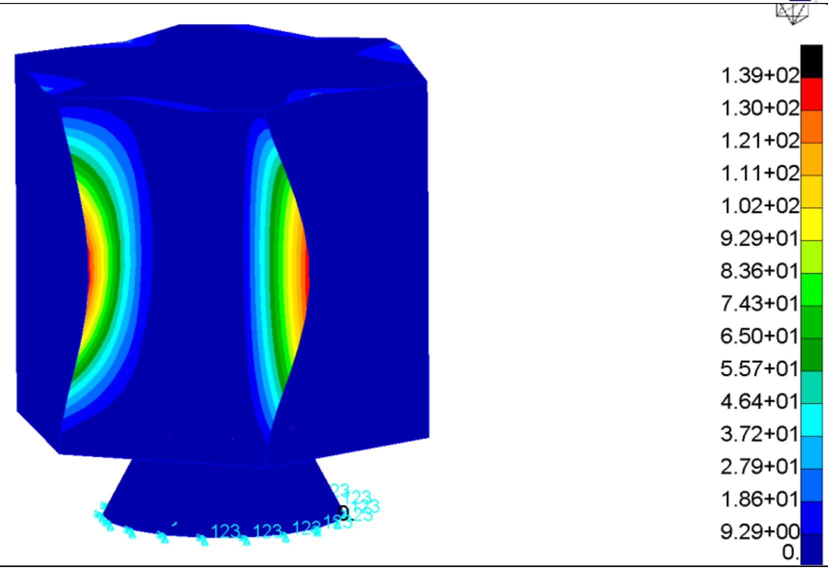


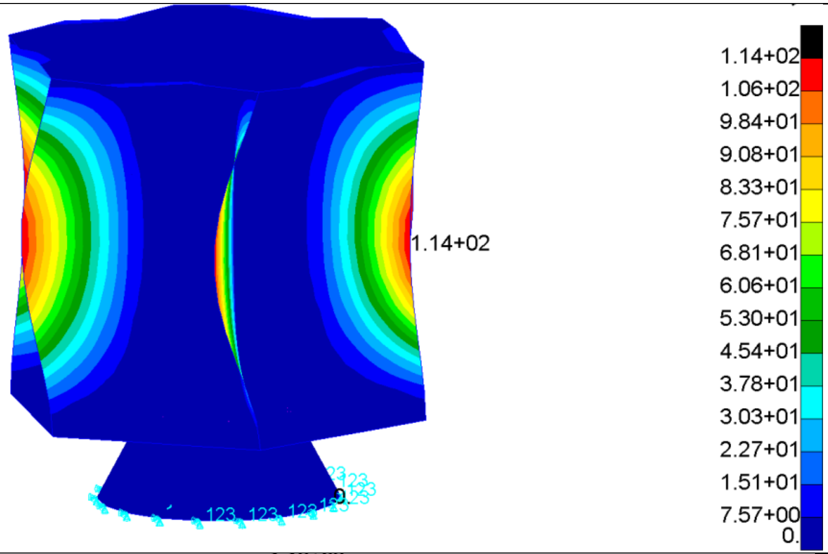
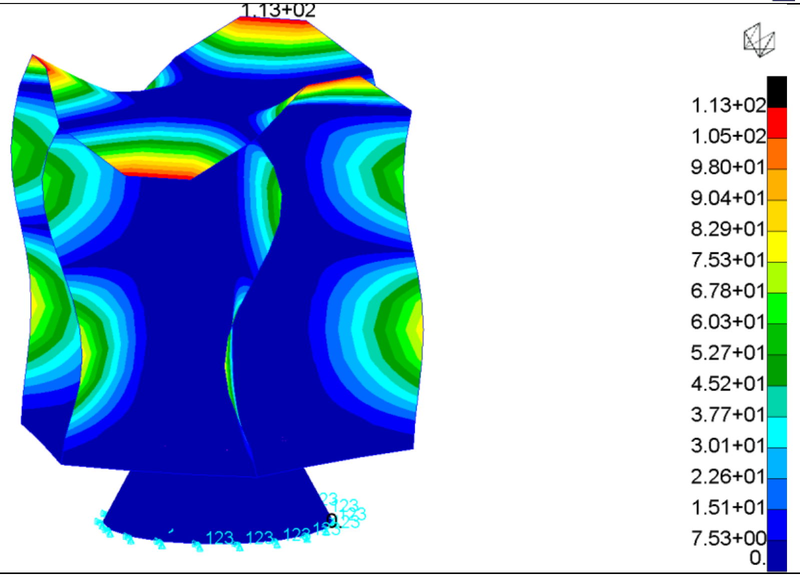
Calculation of natural vibration modes for fixed boundary conditions



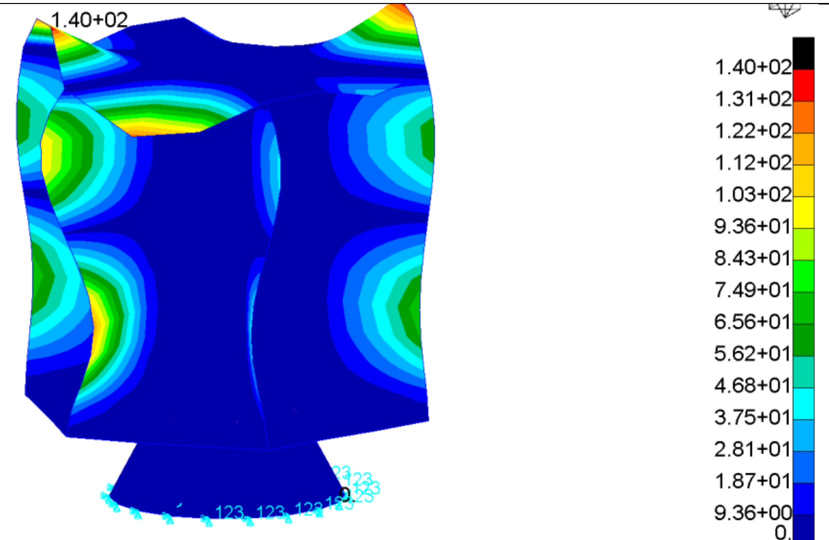

Select Result Cases
Search...
SC1:LAUNCH MODAL, A1:Mode 1: Freq.=48.3286
SC1:LAUNCH MODAL, A1:Mode 2: Freq.=48.3287
SC1:LAUNCH MODAL, A1:Mode 3: Freq.=158.809
SC1:LAUNCH MODAL, A1:Mode 4: Freq.=161.942
SC1:LAUNCH MODAL, A1:Mode 5: Freq.=161.942
SC1:LAUNCH MODAL, A1:Mode 6: Freq.=165.548
SC1:LAUNCH MODAL, A1:Mode 7: Freq.=165.548
SC1:LAUNCH MODAL, A1:Mode 8: Freq.=166.622
SC1:LAUNCH MODAL, A1:Mode 9: Freq.=276.162
SC1:LAUNCH MODAL, A1:Mode 10: Freq.=292.842
SC1:LAUNCH MODAL, A1:Mode 11: Freq.=292.842
SC1:LAUNCH MODAL, A1:Mode 12: Freq.=320.711
SC1:LAUNCH MODAL, A1:Mode 13: Freq.=320.711
SC1:LAUNCH MODAL, A1:Mode 14: Freq.=330.403
SC1:LAUNCH MODAL, A1:Mode 15: Freq.=379.84
SC1:LAUNCH MODAL, A1:Mode 16: Freq.=388.602

Modes	Eigenvectors Translational
1	
3	

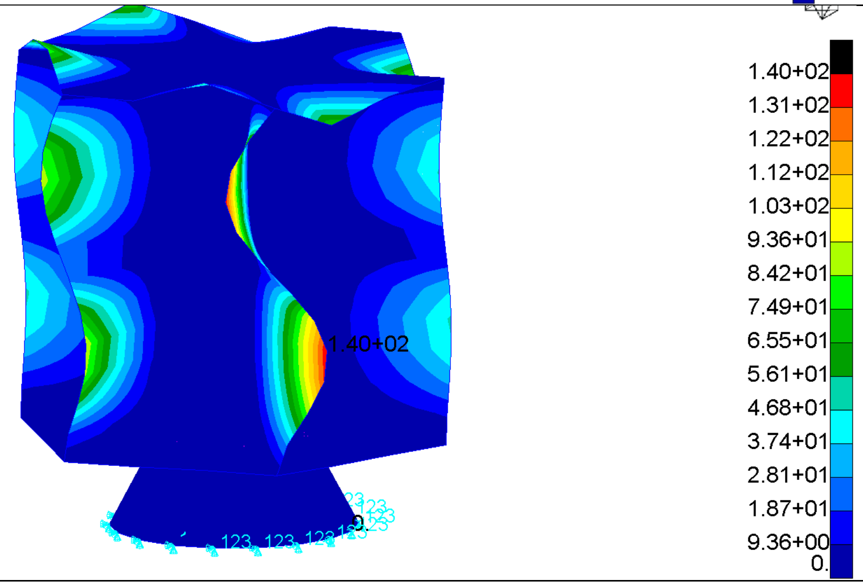
4	1.51+02 1.41+02 1.31+02 1.21+02 1.11+02 1.00+02 9.04+01 8.04+01 7.03+01 6.03+01 5.02+01 4.02+01 3.01+01 2.01+01 1.00+01 0. 1.51+02
6	1.39+02 1.30+02 1.21+02 1.11+02 1.02+02 9.29+01 8.36+01 7.43+01 6.50+01 5.57+01 4.64+01 3.72+01 2.79+01 1.86+01 9.29+00 0.

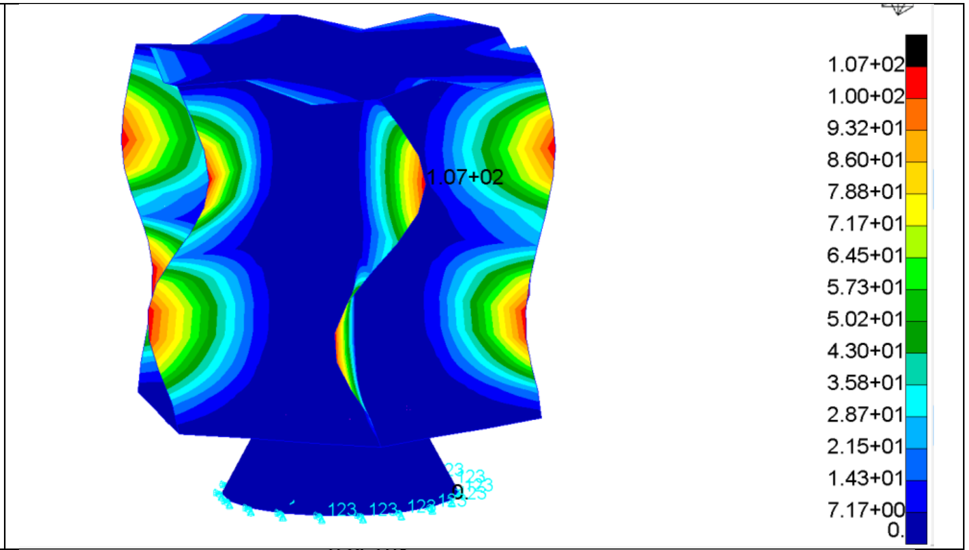
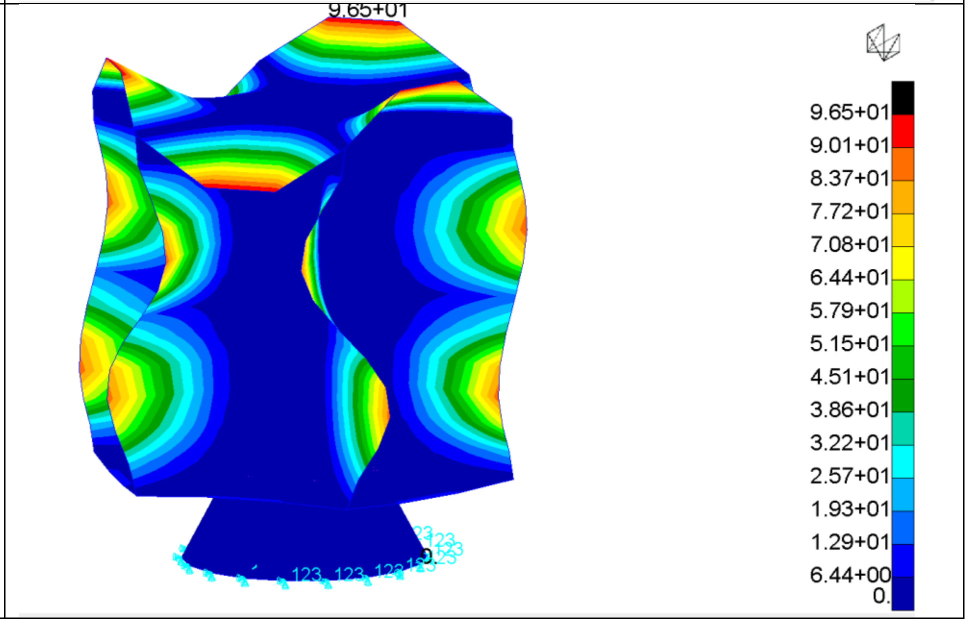
8	1.14+02 1.06+02 9.84+01 9.08+01 8.33+01 7.57+01 6.81+01 6.06+01 5.30+01 4.54+01 3.78+01 3.03+01 2.27+01 1.51+01 7.57+00 0. 1.14+02
9	1.13+02 1.13+02 1.05+02 9.80+01 9.04+01 8.29+01 7.53+01 6.78+01 6.03+01 5.27+01 4.52+01 3.77+01 3.01+01 2.26+01 1.51+01 7.53+00 0.

10


1.40+02
1.40+02
1.31+02
1.22+02
1.12+02
1.03+02
9.36+01
8.43+01
7.49+01
6.56+01
5.62+01
4.68+01
3.75+01
2.81+01
1.87+01
9.36+00
0.

12


1.40+02
1.40+02
1.31+02
1.22+02
1.12+02
1.03+02
9.36+01
8.42+01
7.49+01
6.55+01
5.61+01
4.68+01
3.74+01
2.81+01
1.87+01
9.36+00
0.

14	1.07+02 1.07+02 1.00+02 9.32+01 8.60+01 7.88+01 7.17+01 6.45+01 5.73+01 5.02+01 4.30+01 3.58+01 2.87+01 2.15+01 1.43+01 7.17+00 0.
15	9.65+01 9.65+01 9.01+01 8.37+01 7.72+01 7.08+01 6.44+01 5.79+01 5.15+01 4.51+01 3.86+01 3.22+01 2.57+01 1.93+01 1.29+01 6.44+00 0.

For this part of the satellite, the studies were lengthy and complex. They began with the analysis of our satellite, specifically studying the natural modes of the satellite under different conditions: free, fixed, and clamped. We observed that there were no significant differences in the boundary conditions, as the frequencies are essentially the same, with only minor differences on the order of a thousandth, as shown in the tables above. Whether fixed, free, or simply supported, the frequencies are roughly the same. This consistency might be due to the satellite being designed to be upright and fixed. The construction, with its dimensions and distributions, may explain why there is no noticeable difference

However, we can clearly see that there are natural modes of vibration. The satellite can indeed resonate regardless of the boundary conditions, and this is something to be cautious about.

Satellite sizing

Calculate the maximum mass of equipment that the satellite can carry.

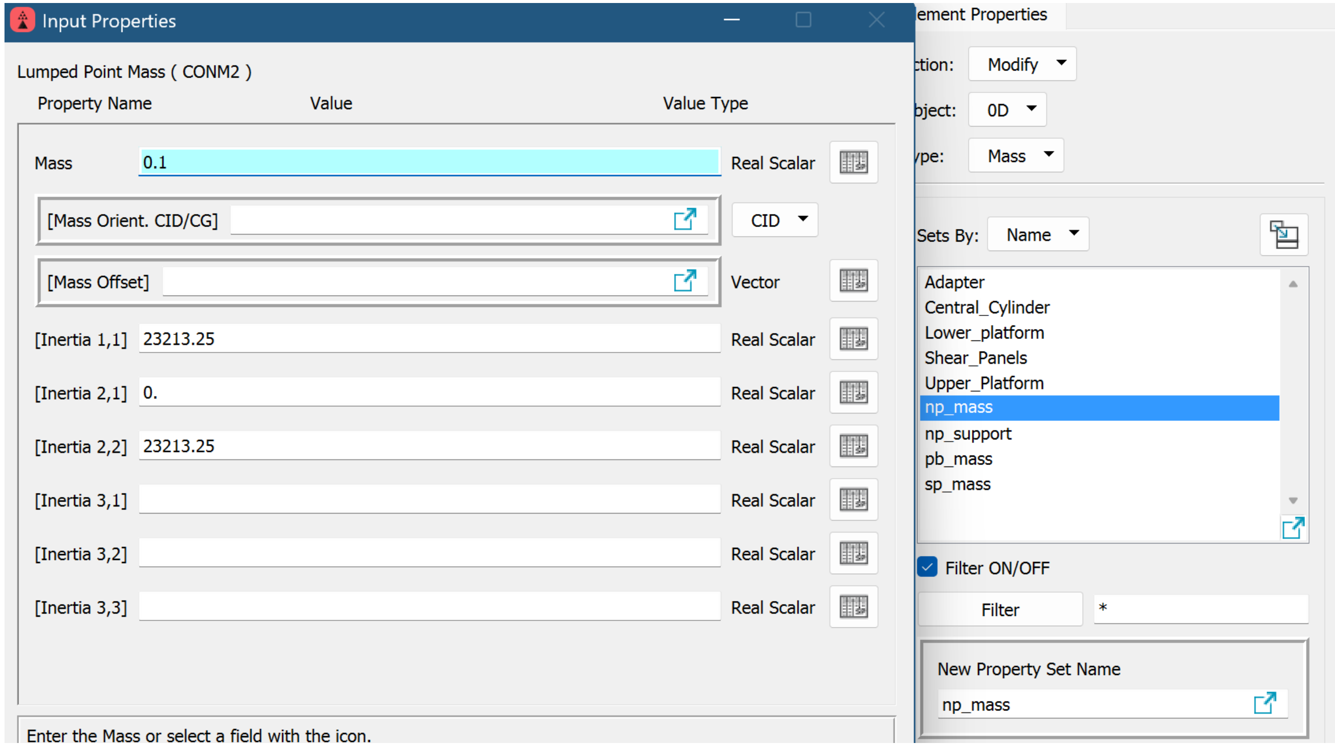
The total mass will be discretized into point masses placed at specific defined nodes

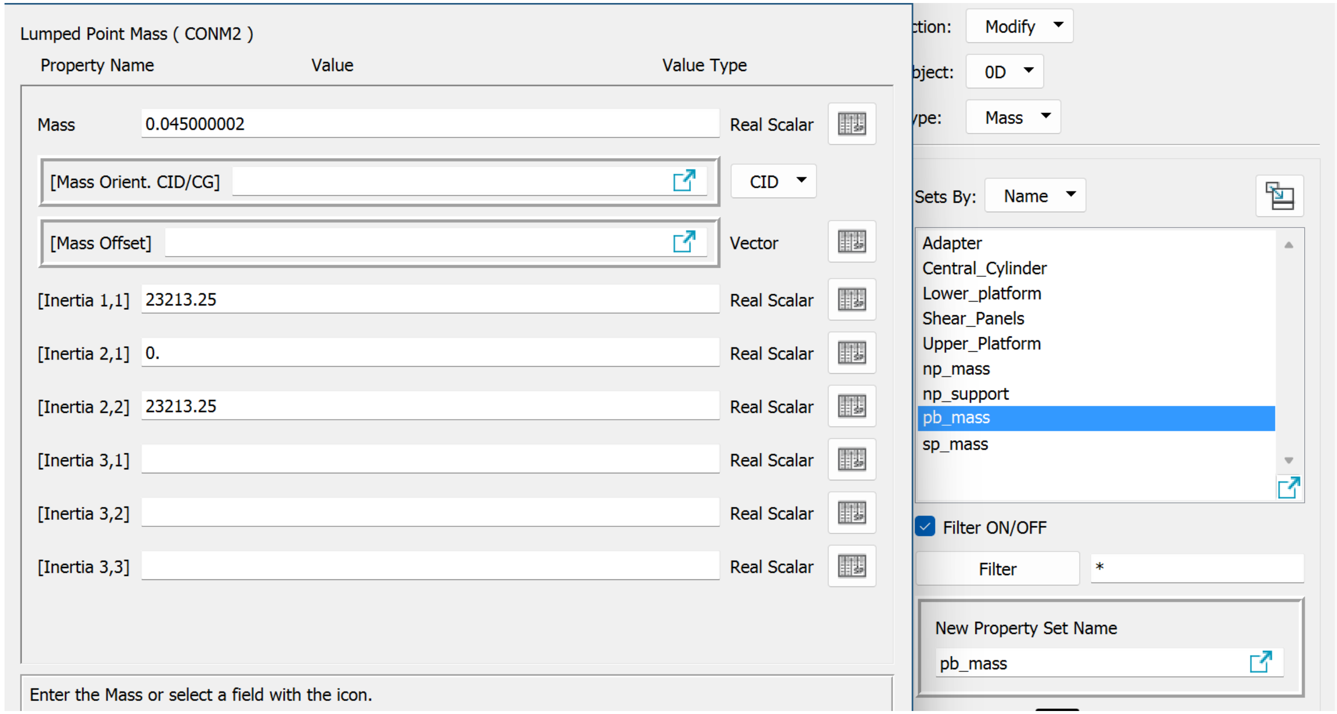
Np_mass=0.1 Tons

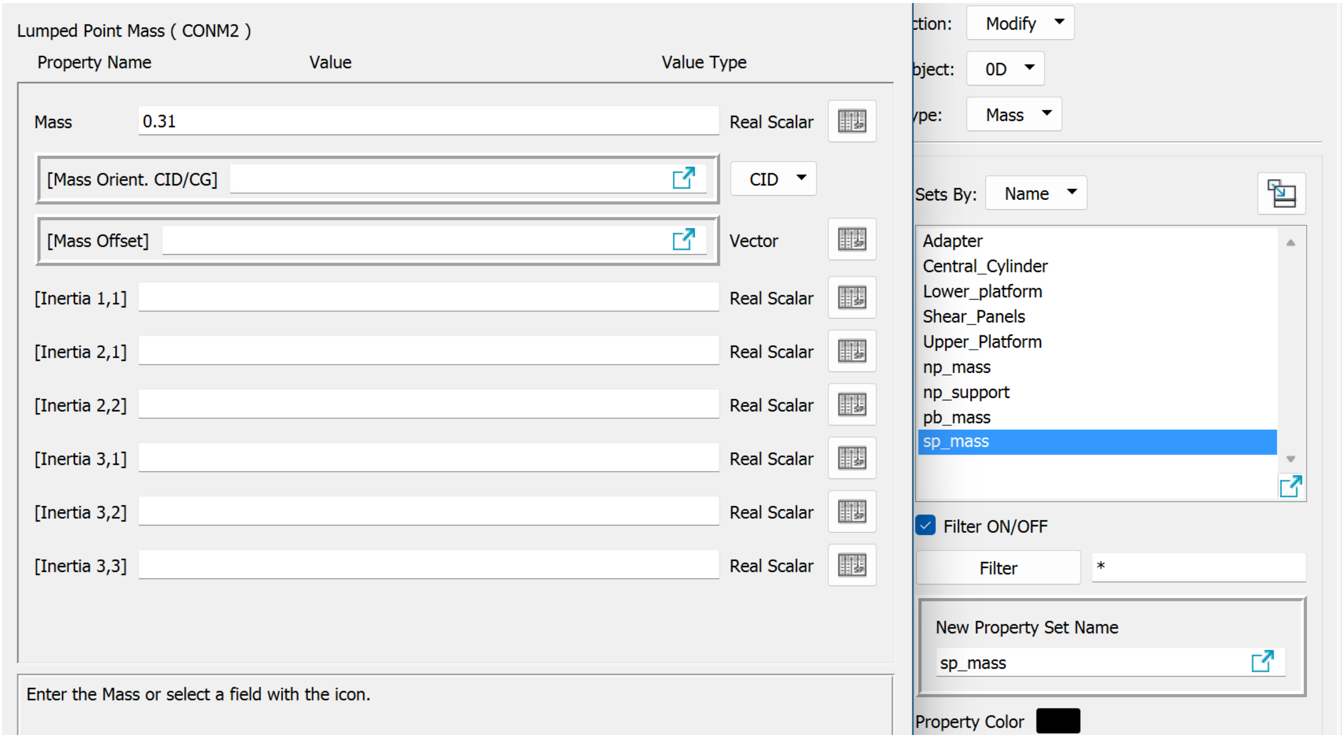
Pb_mass=0.0450 Tons

Sc_mass=0.61 Tons

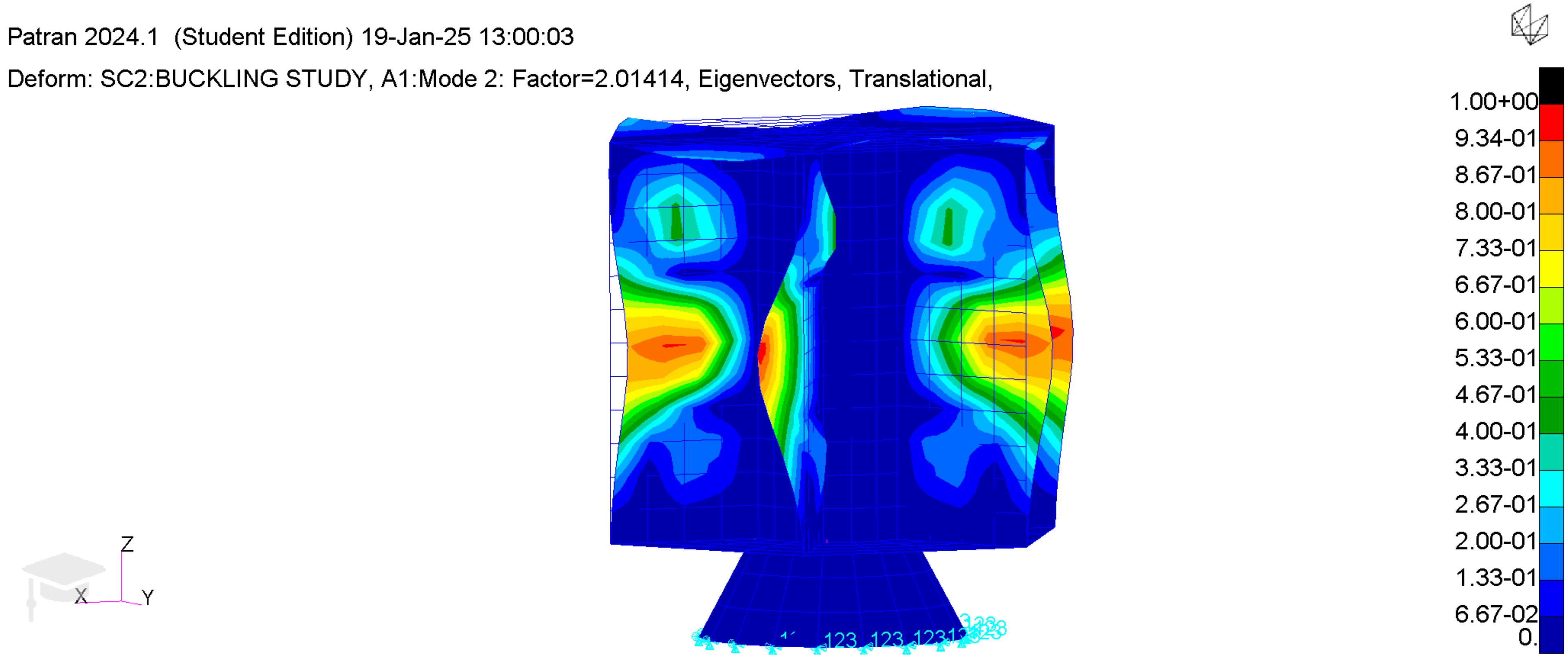
Maximal Mass= 450 kg

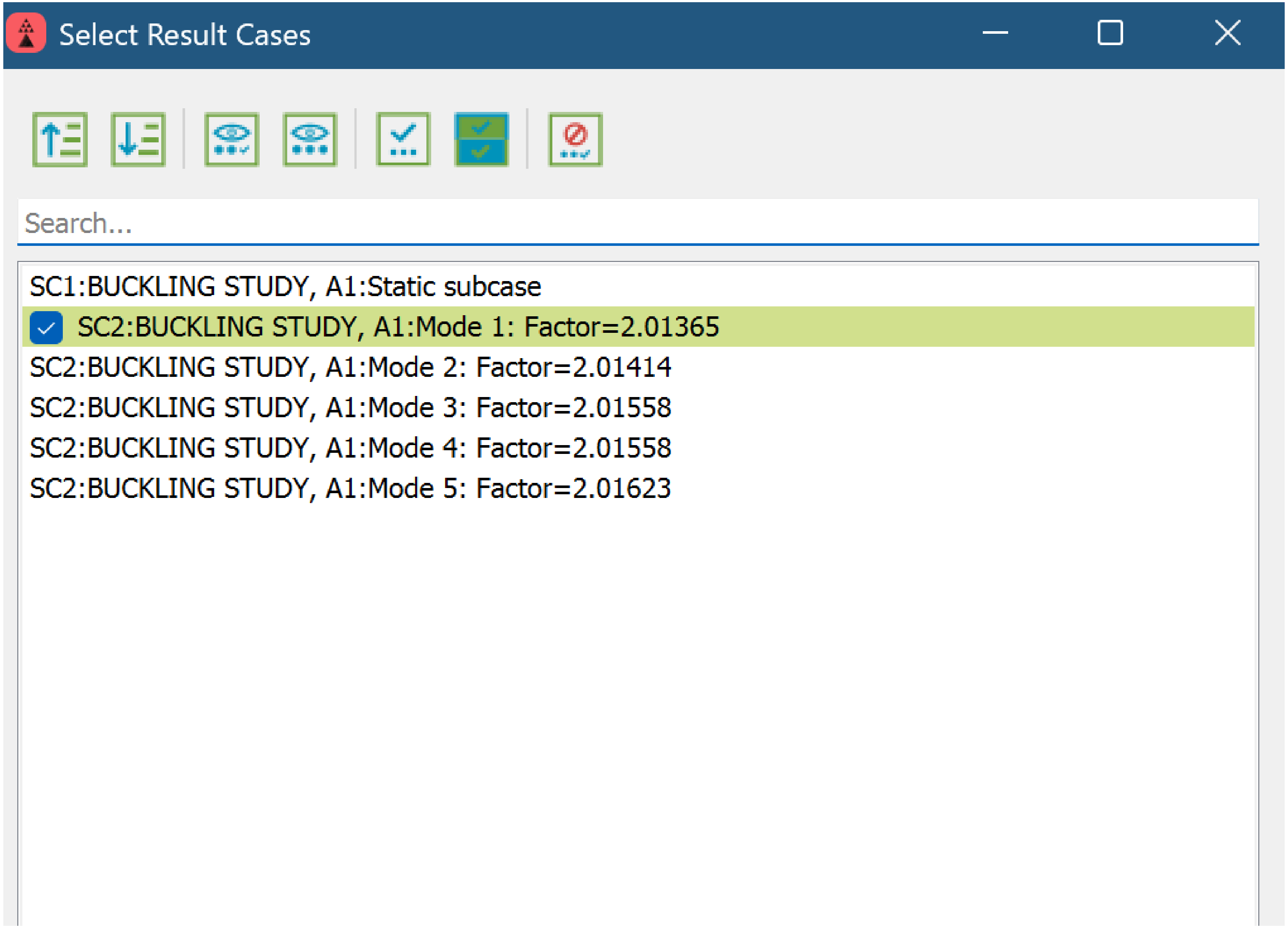

Input Properties
Lumped Point Mass (CONM2)
Property Name
Value
Value Type
Mass
0.1
Real Scalar
[Mass Orient. CID/CG]
CID
[Mass Offset]
Vector
[Inertia 1,1]
23213.25
Real Scalar
[Inertia 2,1]
0.
Real Scalar
[Inertia 2,2]
23213.25
Real Scalar
[Inertia 3,1]
Real Scalar
[Inertia 3,2]
Real Scalar
[Inertia 3,3]
Real Scalar
Enter the Mass or select a field with the icon.
ement Properties
ction:
Modify
bject:
0D
ype:
Mass
Sets By:
Name
Adapter
Central_Cylinder
Lower_platform
Shear_Panels
Upper_Platform
np_mass
np_support
pb_mass
sp_mass
Filter ON/OFF
Filter
*
New Property Set Name
np_mass


Lumped Point Mass (CONM2)
Property Name
Value
Value Type
Mass
0.045000002
Real Scalar
[Mass Orient. CID/CG]
CID
[Mass Offset]
Vector
[Inertia 1,1]
23213.25
Real Scalar
[Inertia 2,1]
0.
Real Scalar
[Inertia 2,2]
23213.25
Real Scalar
[Inertia 3,1]
Real Scalar
[Inertia 3,2]
Real Scalar
[Inertia 3,3]
Real Scalar
Enter the Mass or select a field with the icon.
tion:
Modify
bject:
0D
ype:
Mass
Sets By:
Name
Adapter
Central_Cylinder
Lower_platform
Shear_Panels
Upper_Platform
np_mass
np_support
pb_mass
sp_mass
Filter ON/OFF
Filter
*
New Property Set Name
pb_mass

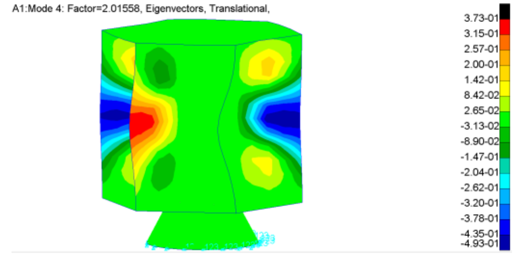
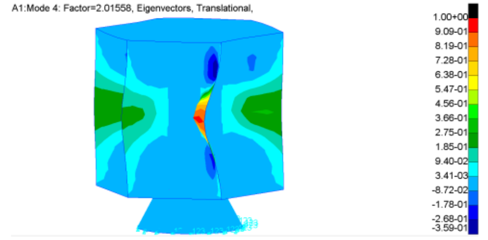
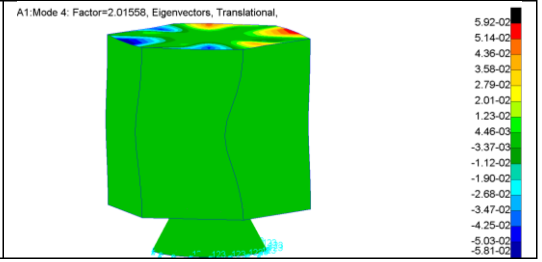

Lumped Point Mass (CONM2)
Property Name
Value
Value Type
Mass
0.31
Real Scalar
[Mass Orient. CID/CG]
CID
[Mass Offset]
Vector
[Inertia 1,1]
Real Scalar
[Inertia 2,1]
Real Scalar
[Inertia 2,2]
Real Scalar
[Inertia 3,1]
Real Scalar
[Inertia 3,2]
Real Scalar
[Inertia 3,3]
Real Scalar
Enter the Mass or select a field with the icon.
ction:
Modify
bject:
0D
ype:
Mass
Sets By:
Name
Adapter
Central_Cylinder
Lower_platform
Shear_Panels
Upper_Platform
np_mass
np_support
pb_mass
sp_mass
Filter ON/OFF
Filter
*
New Property Set Name
sp_mass
Property Color

Critical Charge observation




Select Result Cases
Search...
SC1:BUCKLING STUDY, A1:Static subcase
SC2:BUCKLING STUDY, A1:Mode 1: Factor=2.01365
SC2:BUCKLING STUDY, A1:Mode 2: Factor=2.01414
SC2:BUCKLING STUDY, A1:Mode 3: Factor=2.01558
SC2:BUCKLING STUDY, A1:Mode 4: Factor=2.01558
SC2:BUCKLING STUDY, A1:Mode 5: Factor=2.01623

Reaction along axis with critical mass

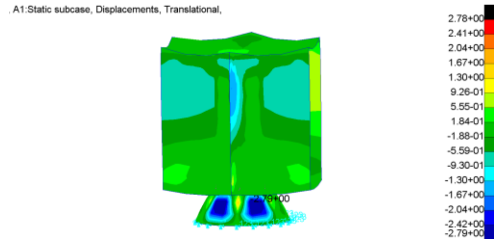
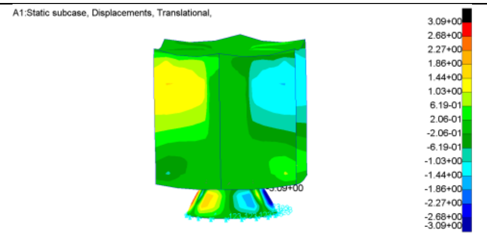
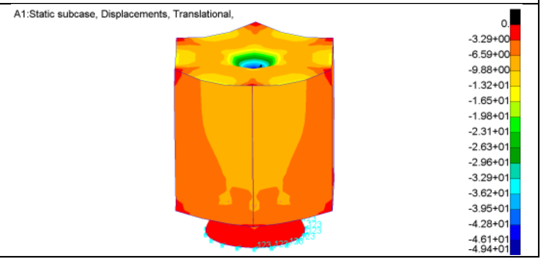
Modes	X	Y	Z
4			

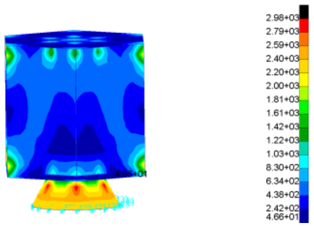
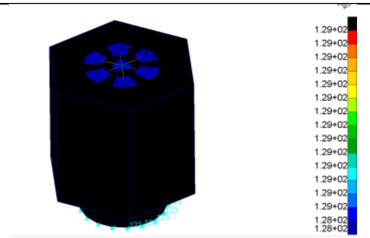
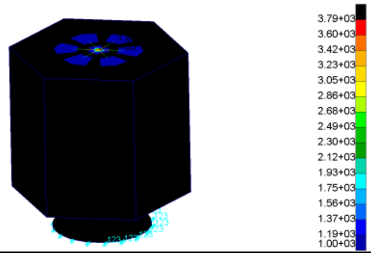
When we talk about the critical mass, the first phenomenon we encounter in structural mechanics is buckling. This is why, when I was asked to calculate the critical mass, I immediately turned to a buckling simulation, specifically a buckling and steady-state analysis. Since my satellite, in its upright structure, is somewhat like a beam. It's not a uniform beam, as there are varying thicknesses and shear panels in the satellite, but it can still be approximated as such. With Hurler's method, we use the coefficient K when dealing with the buckling phenomenon, and this coefficient K can be displayed on the diagram of a strand. My method is based on the following approach: I have a satellite fixed at one end, so the coefficient K should be equal to 2. I first ran a simulation with no load, but the coefficient K was not equal to 2. So, what did I do? I gradually added masses, considering that the scientific platform is the most evenly distributed in mass. I made sure to place the maximum mass on this platform, with less mass on the propulsion and navigation platforms. I did this gradually, adding masses and redistributing them until I found the correct balance. I continued adding masses until I achieved a K value of 2 across five modes. This K value of 2 corresponds to a structure that is fixed at one end. That is how I determined the maximum mass my satellite can support, which is 450 kg, distributed as shown.

It's important to note that even a lightweight structure can support a substantial mass, but this depends on the distribution. It really depends on how the mass is distributed. As seen here, the navigation platform is somewhat at risk. The shear panels are particularly sensitive to this mass, as we observe some deformation in these panels, which are the least robust and rigid part of the structure. This is the method I used to determine my critical mass. This method will be supported by additional analyses that I will perform, including static and modal analysis, keeping the same maximum satellite mass and the same conditions.

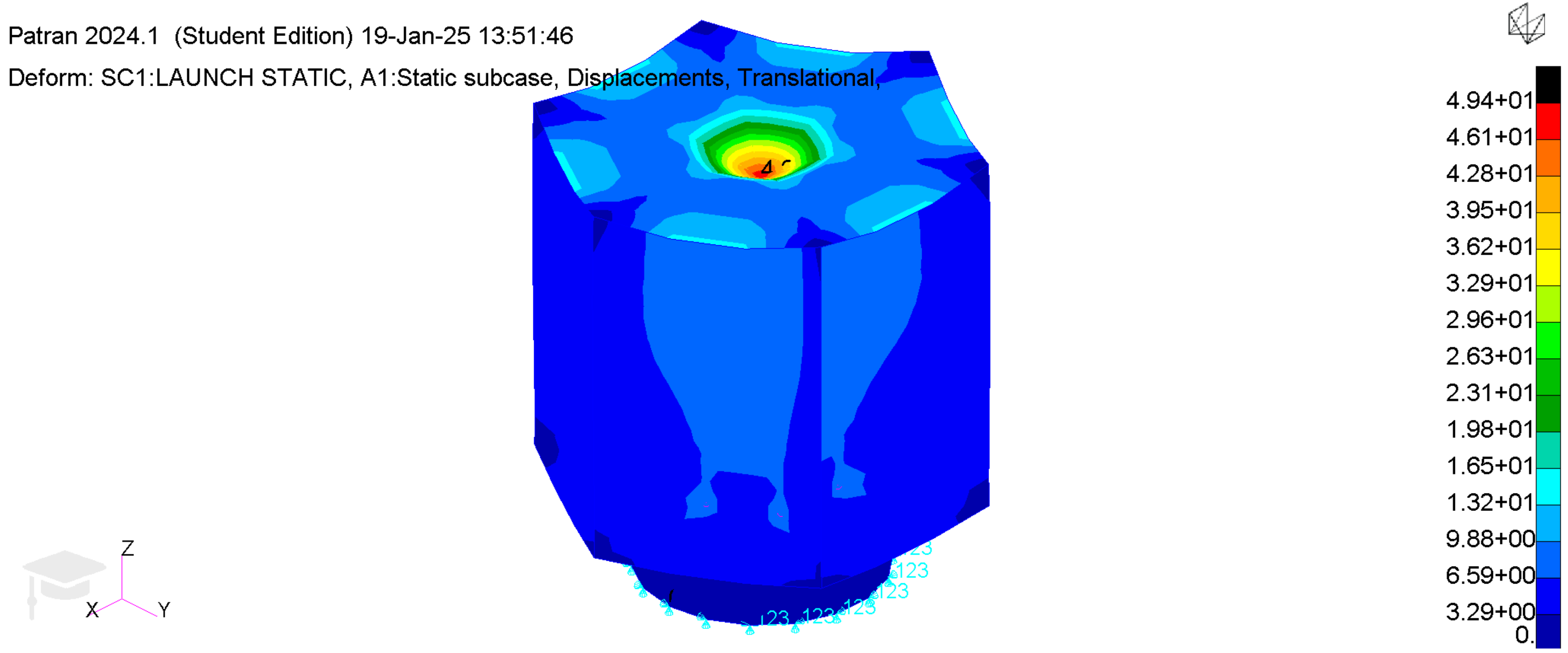
Linear Static verification of the precedent Mass

<u>Observation along axis</u>

<table>
<tr><th>Linear Static</th><th>X</th><th>Y</th><th>Z</th></tr>
<tr><td></td><td></td><td></td><td></td></tr>
</table>

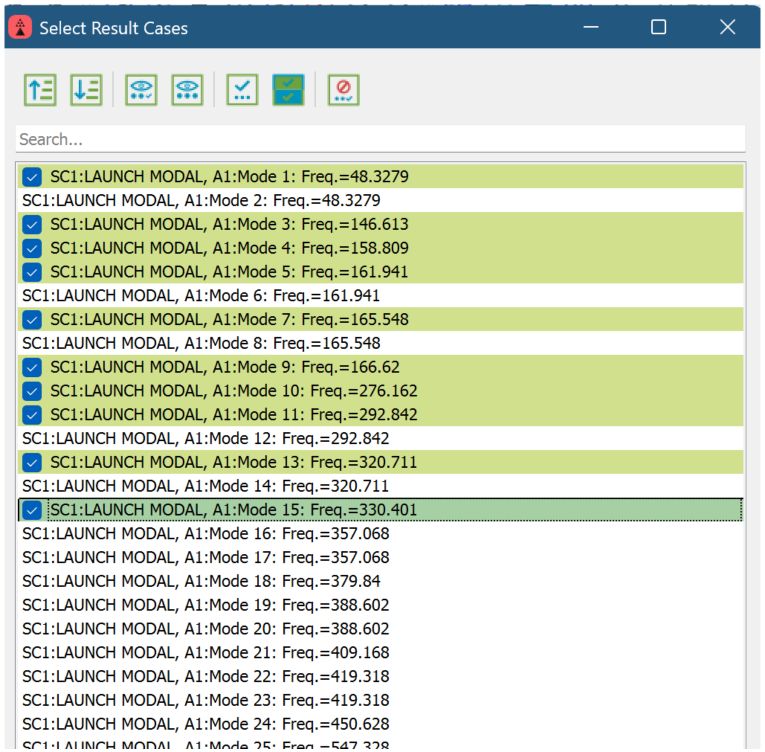
<table>
<tr><th>Stress Tensor at Z1 and Z2</th><th>Bar Stresses axial</th><th>Bar Stress Bending at all Points</th></tr>
<tr><td></td><td></td><td></td></tr>
</table>

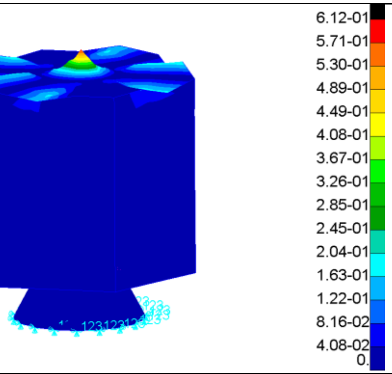
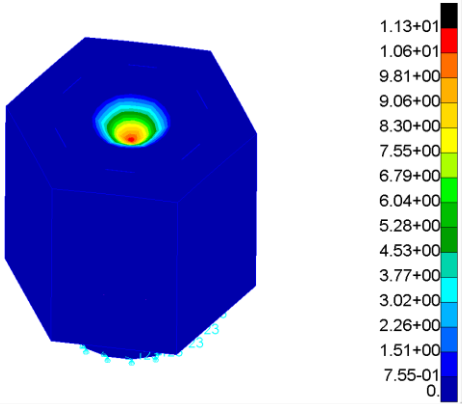
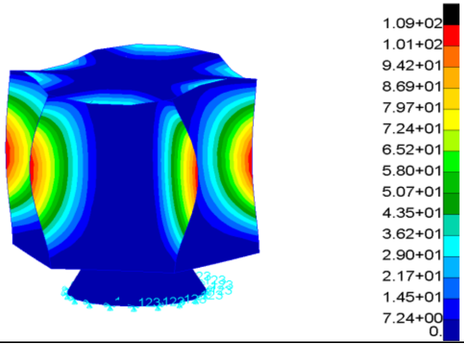
<u>Magnitude Displacement Deformation</u>

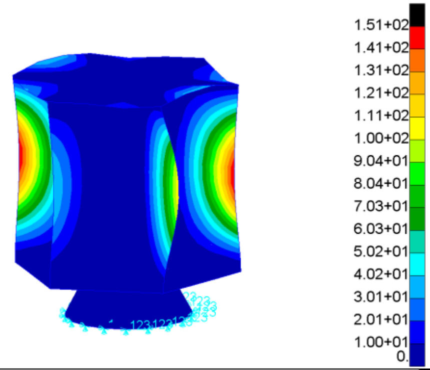
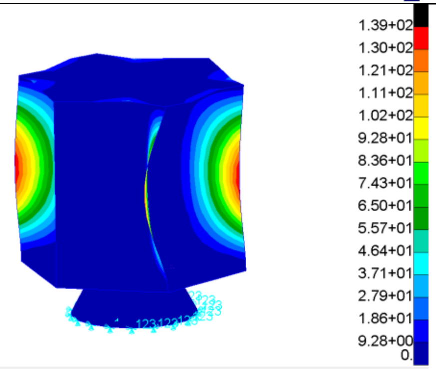
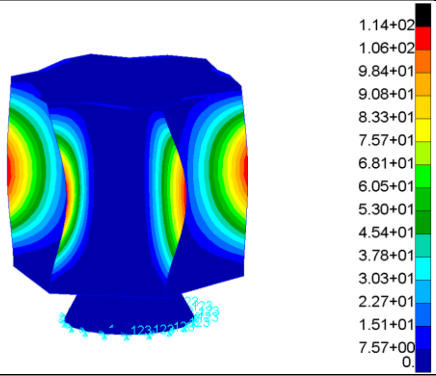


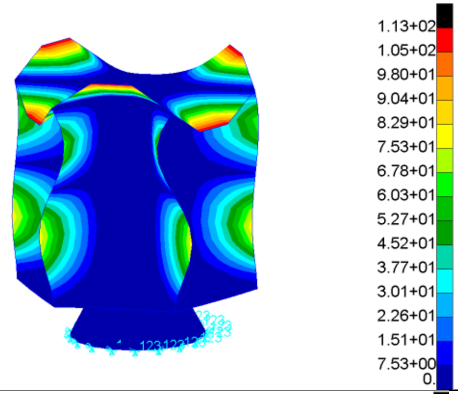
I tried to verify my mass by keeping the same 450 kg mass for my fixed satellite. I ran linear and static simulations again to check. I can clearly see from the axes that there are indeed displacements and some sagging at the navigation platform. It's nothing major, just some slight deformations. I also observed the bars, the stress on them, and looked at the overall magnitude of the displacement. It was very difficult to confirm with the linear and static mode because I didn't fully understand the results. I then ran another modal analysis to further verify my maximum mass. You will see the results below.

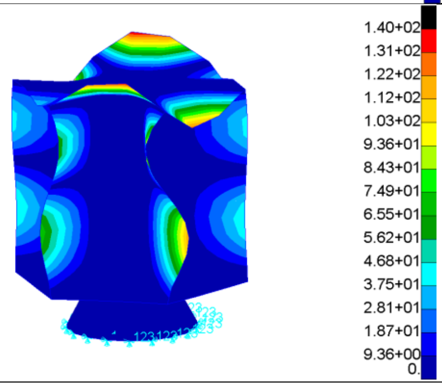
Modal Analysis with added mass for verification

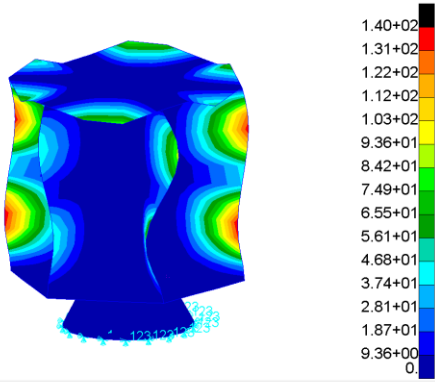


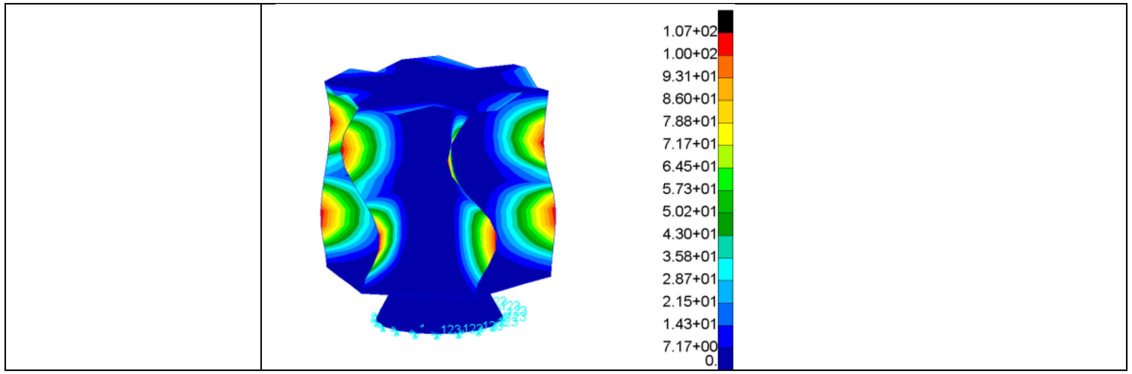
Frequencies	Deformation
	
	
	

<table>
<tr><td></td><td>
</td></tr>
<tr><td></td><td>
</td></tr>
<tr><td></td><td>
</td></tr>
</table>


1.13+02
1.05+02
9.80+01
9.04+01
8.29+01
7.53+01
6.78+01
6.03+01
5.27+01
4.52+01
3.77+01
3.01+01
2.26+01
1.51+01
7.53+00
0.


1.40+02
1.31+02
1.22+02
1.12+02
1.03+02
9.36+01
8.43+01
7.49+01
6.55+01
5.62+01
4.68+01
3.75+01
2.81+01
1.87+01
9.36+00
0.


1.40+02
1.31+02
1.22+02
1.12+02
1.03+02
9.36+01
8.42+01
7.49+01
6.55+01
5.61+01
4.68+01
3.74+01
2.81+01
1.87+01
9.36+00
0.



So, I ran a modal analysis of our structure again, this time with the mass, which is very important to check the frequencies. Since we added the mass, we should observe a decrease in frequency to see if our previous calculations are consistent. Indeed, we see a decrease in frequencies. Let's take a closer look. Our 15th frequency, when the structure was empty, was around 379 Hz. With the mass, our frequency is now around 330 Hz. That means there is a 50 Hz difference. This shows that the structure is now more sensitive because of the added mass. It also shows that the structure's rigidity decreases when loaded, especially in a certain way, and this also depends on how the mass is distributed.

We also observed the deformations at these frequencies. As we can see, there is significant deformation at the navigation platform, which is the most sensitive area of our structure. The shear panel, which is the plate above the navigation platform, is also important because there are noticeable vibrations at this platform. These vibrations are more intense because the platform is relatively free. The only part that is fixed are the edges of this plate, which causes the vibration to concentrate in the centre. That's why we reinforced the canter with bars earlier, but we still observe some vibrations.

So, this is the result of the mass determination. The results seem relevant for the mass we have. Personally, as a structural mechanics student, I wouldn't be surprised if we were told that the structure could support even more mass, because, as I said, it really depends on the mass distribution. However, for a small satellite like this one, with an empty mass of 61.22 kg, I would be surprised, because the mass distribution is specific and depends on the satellite's objectives and missions, such as the science, navigation, or propulsion platforms. But I do think that 450 kg is reasonable for this mass, because let's not forget, in space, the satellite will navigate itself to orbit. The satellite contains some fuel that should not be overlooked, especially since there are propulsion systems. The propulsion area is seen at the adapter, which is why we talk about the propulsion area—because the satellite will also navigate to its orbit. So, 450 kg doesn't shock me. It makes sense given all the platforms. So, this is my reasoning and my comment as a structural mechanics student.

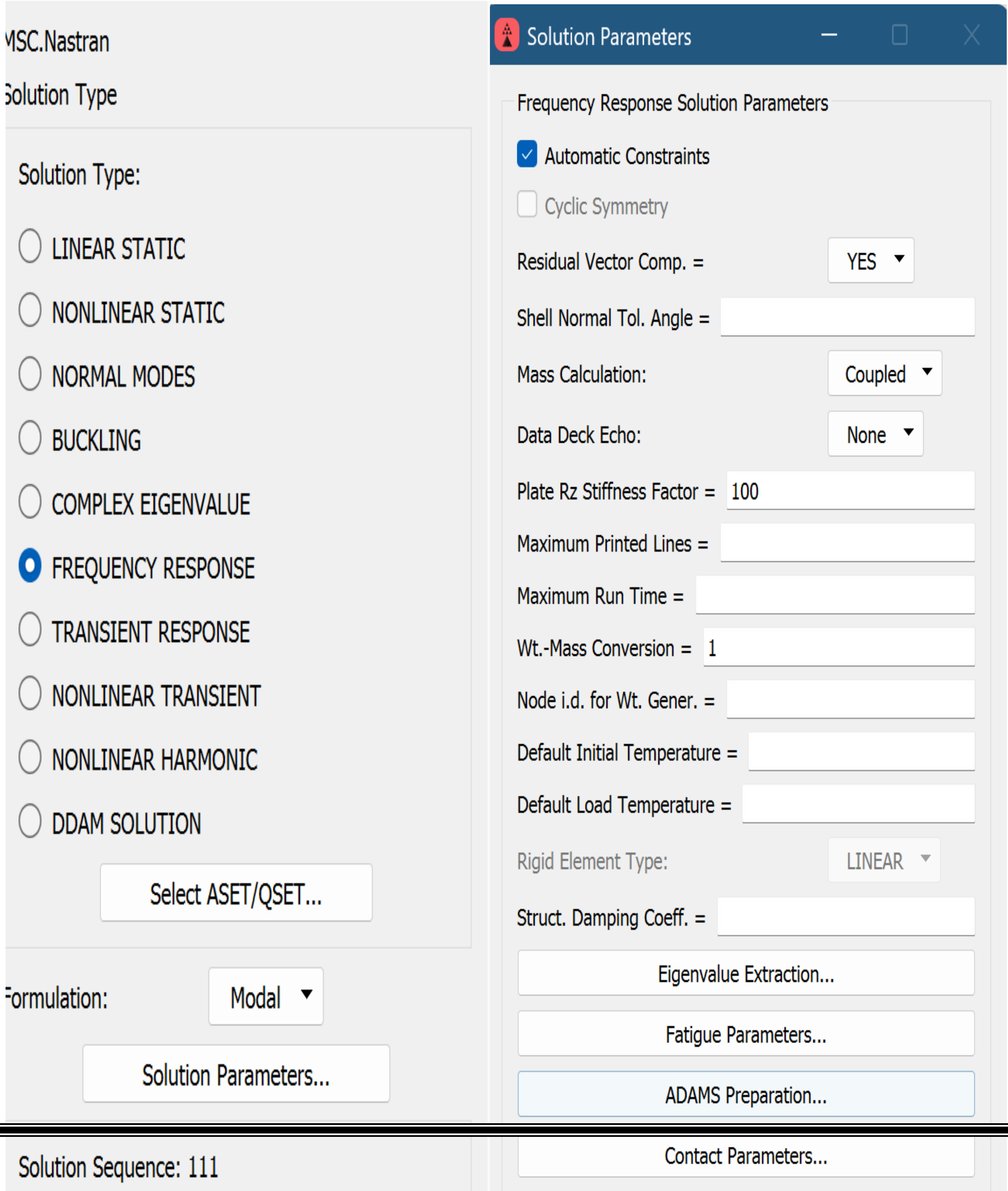
Frequency analysis

Out design is subjected to dynamic environments, static studies cannot be used to evaluate the response. Frequency studies can help avoid resonance and design vibration isolation systems. They also form the basis for evaluating the response of linear dynamic systems, where the system's response to a dynamic environment is assumed to be the sum of the contributions of the modes considered in the analysis.

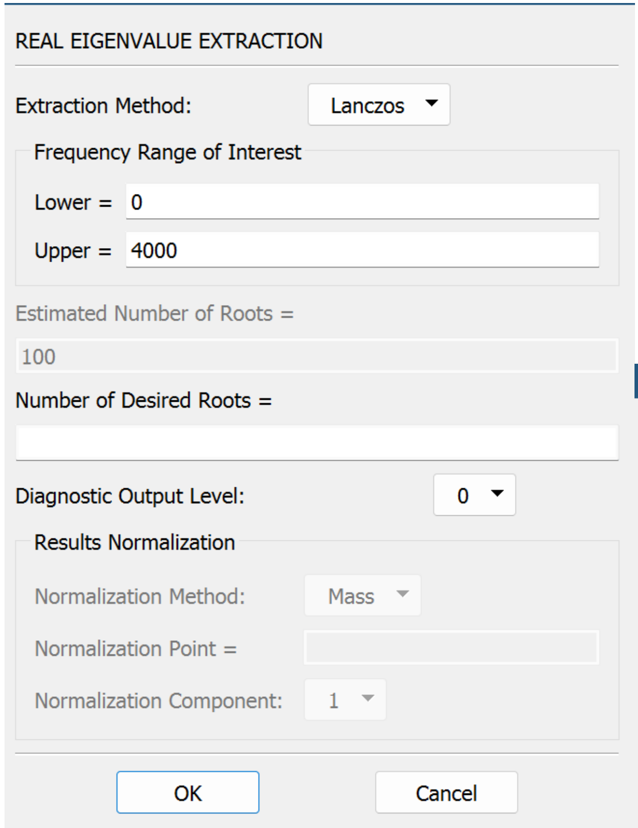
So now we going to study the frequency response of the structure within a defined frequency range and excitation. And interpret the results.

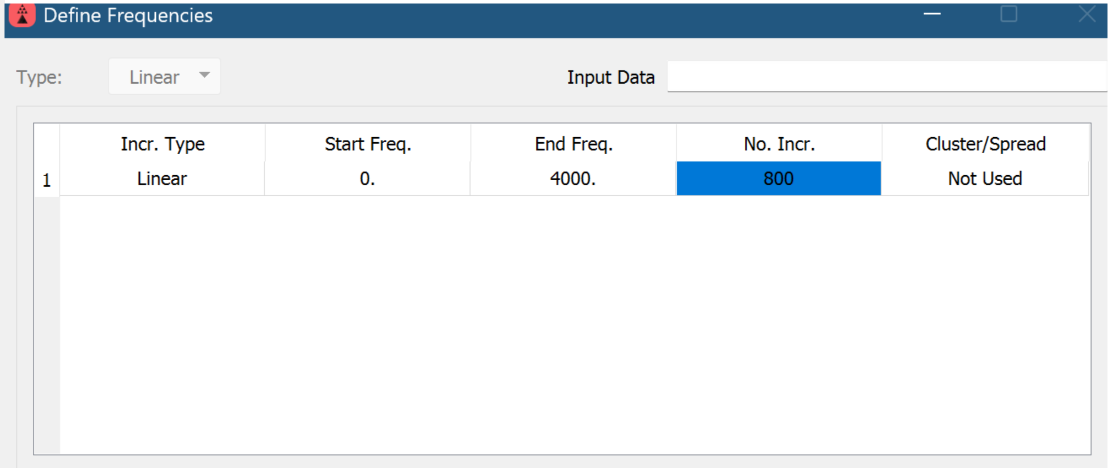
Frequency Analysis Modal without excitation with the maximum Mass found

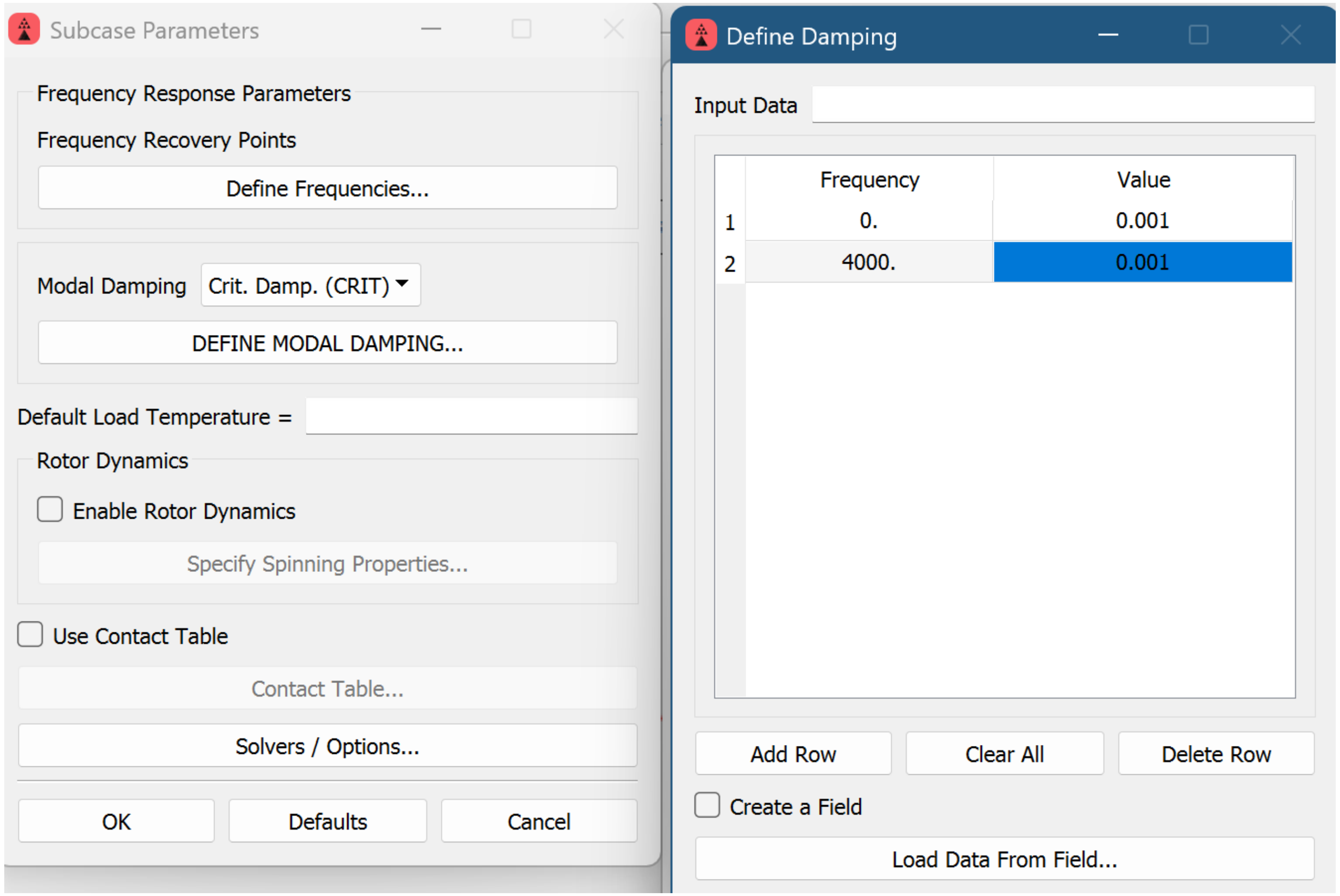
Steps and chosen options

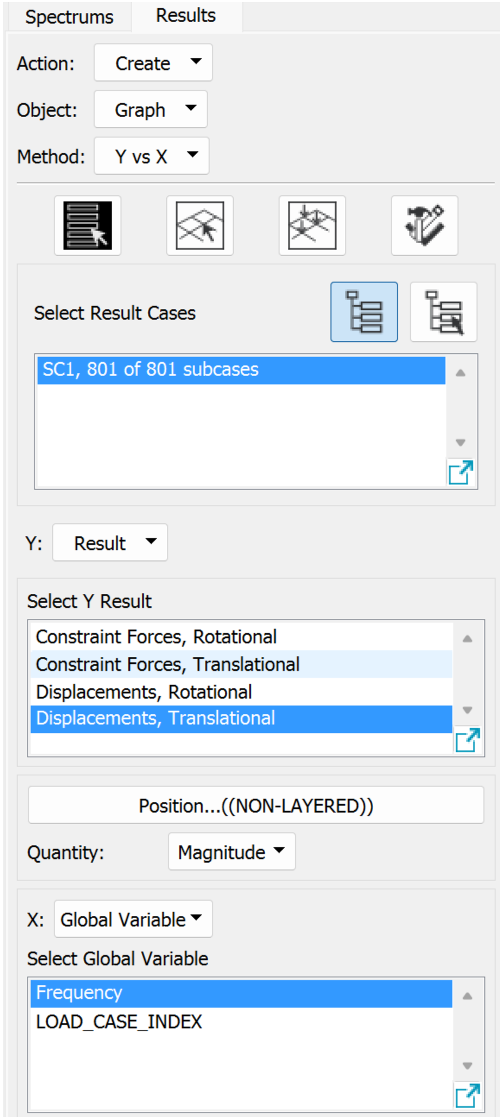


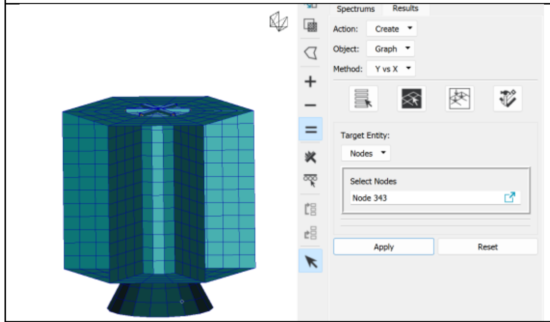
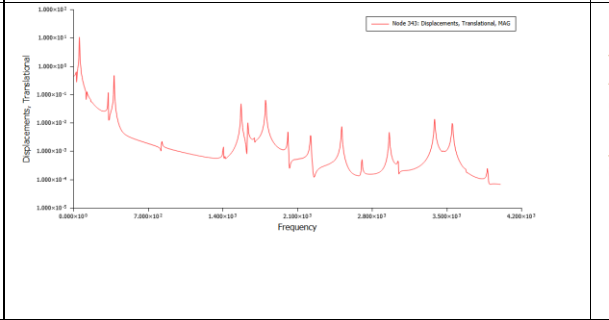
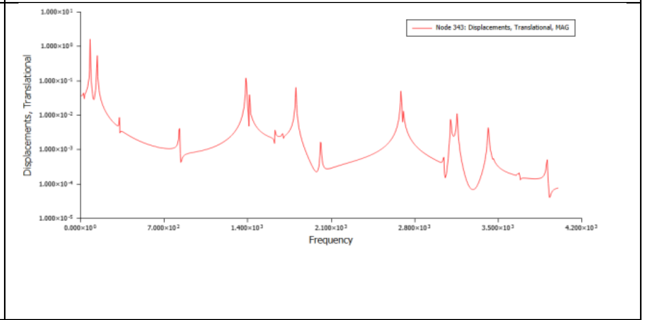
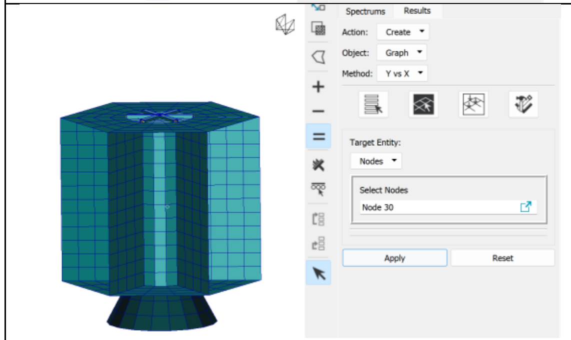
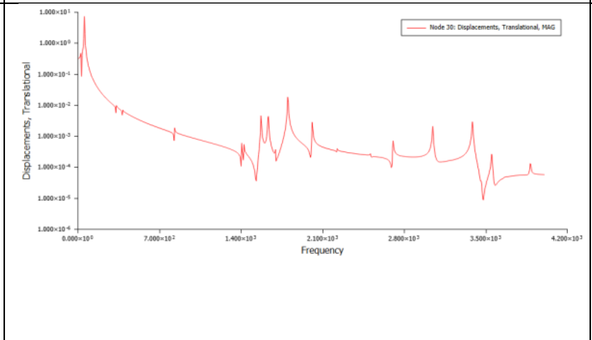
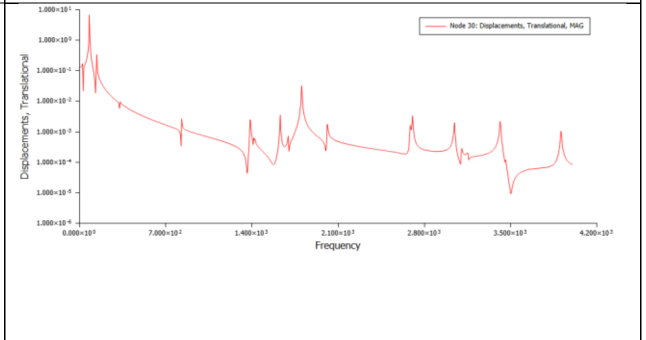
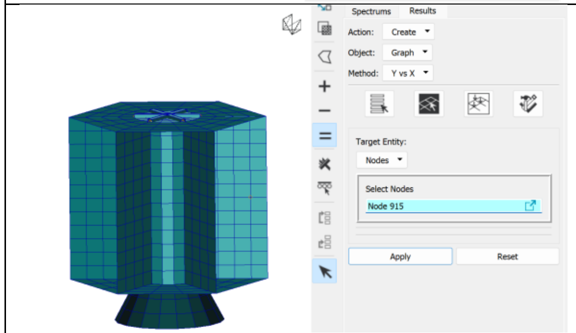
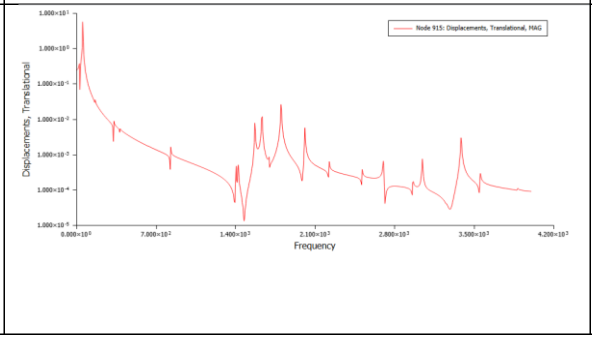
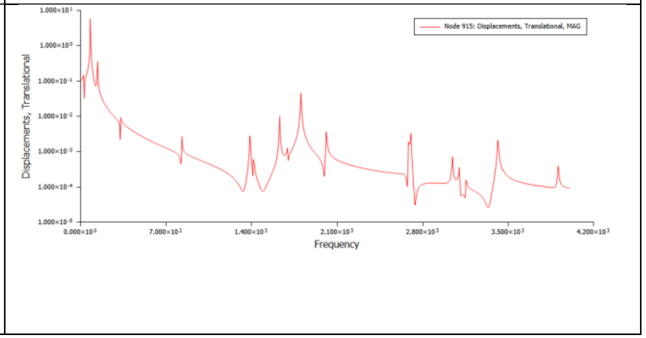
MSC.Nastran
Solution Type
Solution Type:
LINEAR STATIC
NONLINEAR STATIC
NORMAL MODES
BUCKLING
COMPLEX EIGENVALUE
FREQUENCY RESPONSE
TRANSIENT RESPONSE
NONLINEAR TRANSIENT
NONLINEAR HARMONIC
DDAM SOLUTION
Select ASET/QSET...
Formulation:
Modal
Solution Parameters...
Solution Sequence: 111
Solution Parameters
Frequency Response Solution Parameters
Automatic Constraints
Cyclic Symmetry
Residual Vector Comp. =
YES
Shell Normal Tol. Angle =
Mass Calculation:
Coupled
Data Deck Echo:
None
Plate Rz Stiffness Factor =
100
Maximum Printed Lines =
Maximum Run Time =
Wt.-Mass Conversion =
1
Node i.d. for Wt. Gener. =
Default Initial Temperature =
Default Load Temperature =
Rigid Element Type:
LINEAR
Struct. Damping Coeff. =
Eigenvalue Extraction...
Fatigue Parameters...
ADAMS Preparation...
Contact Parameters...

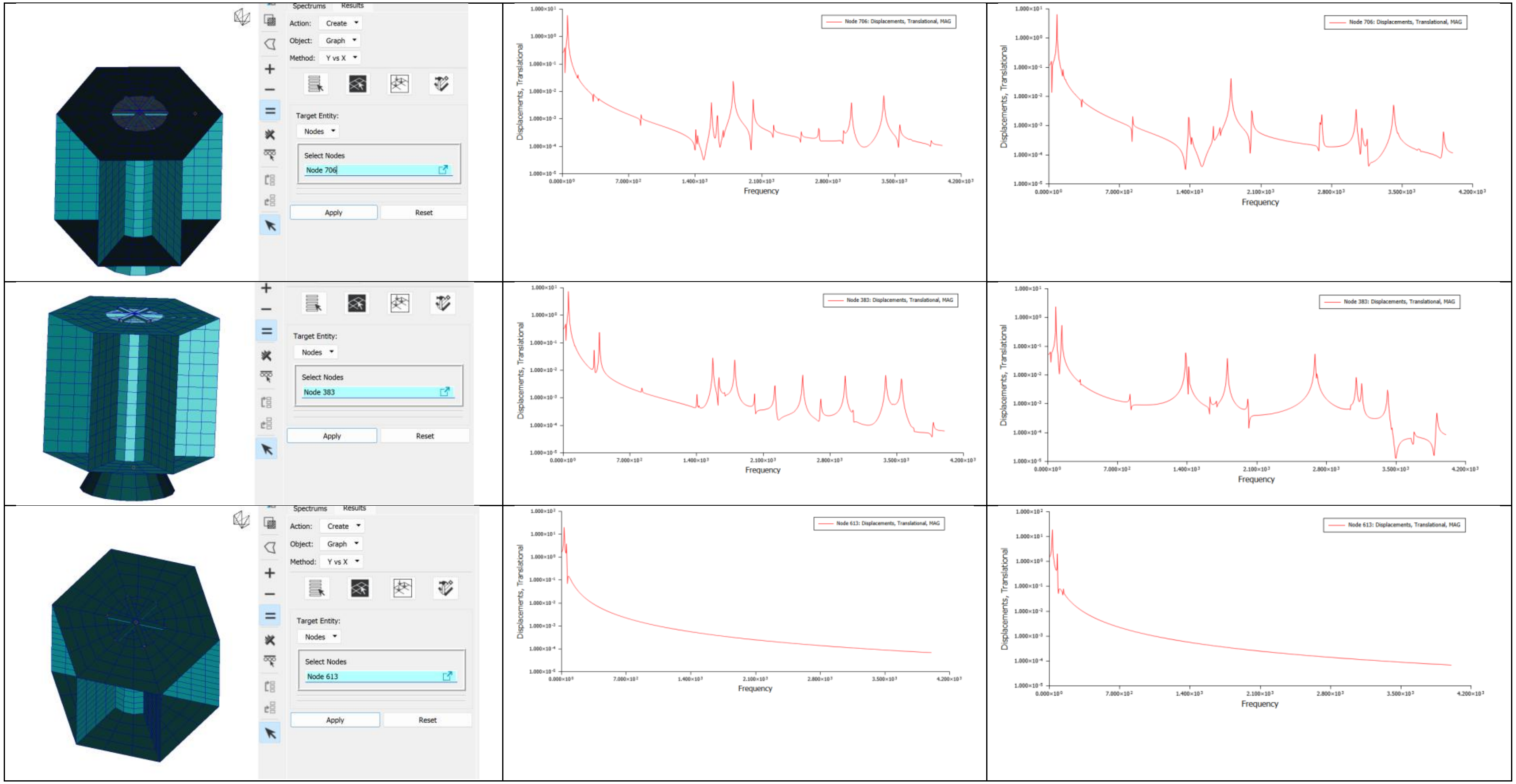

REAL EIGENVALUE EXTRACTION
Extraction Method:
Lanczos
Frequency Range of Interest
Lower =
0
Upper =
4000
Estimated Number of Roots =
100
Number of Desired Roots =
Diagnostic Output Level:
0
Results Normalization
Normalization Method:
Mass
Normalization Point =
Normalization Component:
1
OK
Cancel

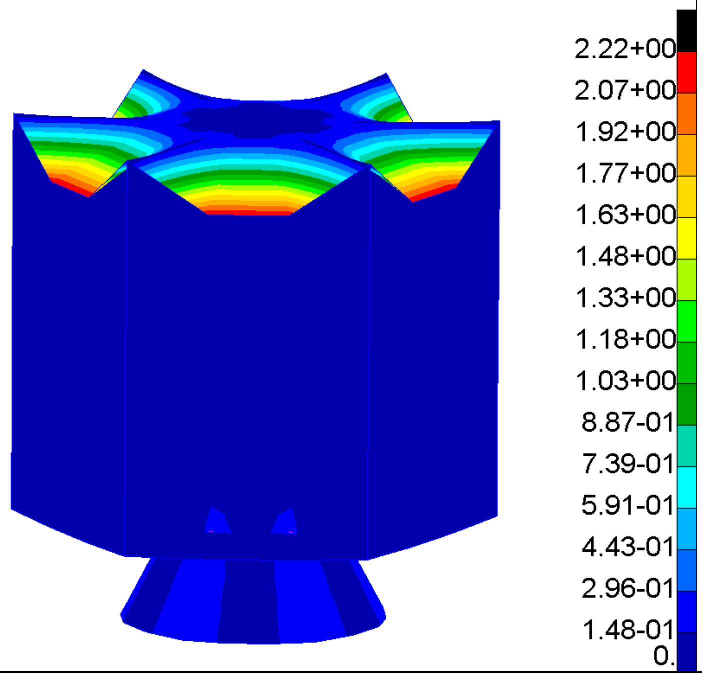
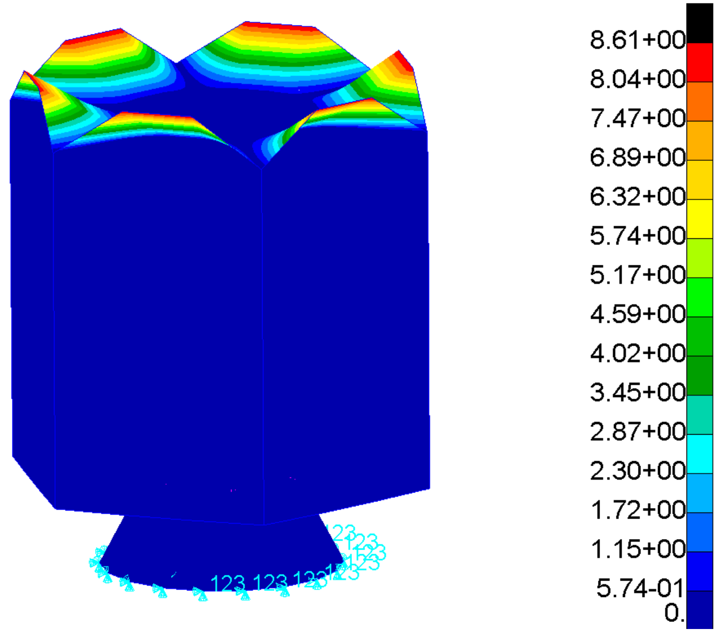

Define Frequencies
Type:
Linear
Input Data
Incr. Type
Start Freq.
End Freq.
No. Incr.
Cluster/Spread
1
Linear
0.
4000.
800
Not Used


Subcase Parameters
Frequency Response Parameters
Frequency Recovery Points
Define Frequencies...
Modal Damping
Crit. Damp. (CRIT)
DEFINE MODAL DAMPING...
Default Load Temperature =
Rotor Dynamics
Enable Rotor Dynamics
Specify Spinning Properties...
Use Contact Table
Contact Table...
Solvers / Options...
OK
Defaults
Cancel
Define Damping
Input Data
Frequency
Value
1
0.
0.001
2
4000.
0.001
Add Row
Clear All
Delete Row
Create a Field
Load Data From Field...

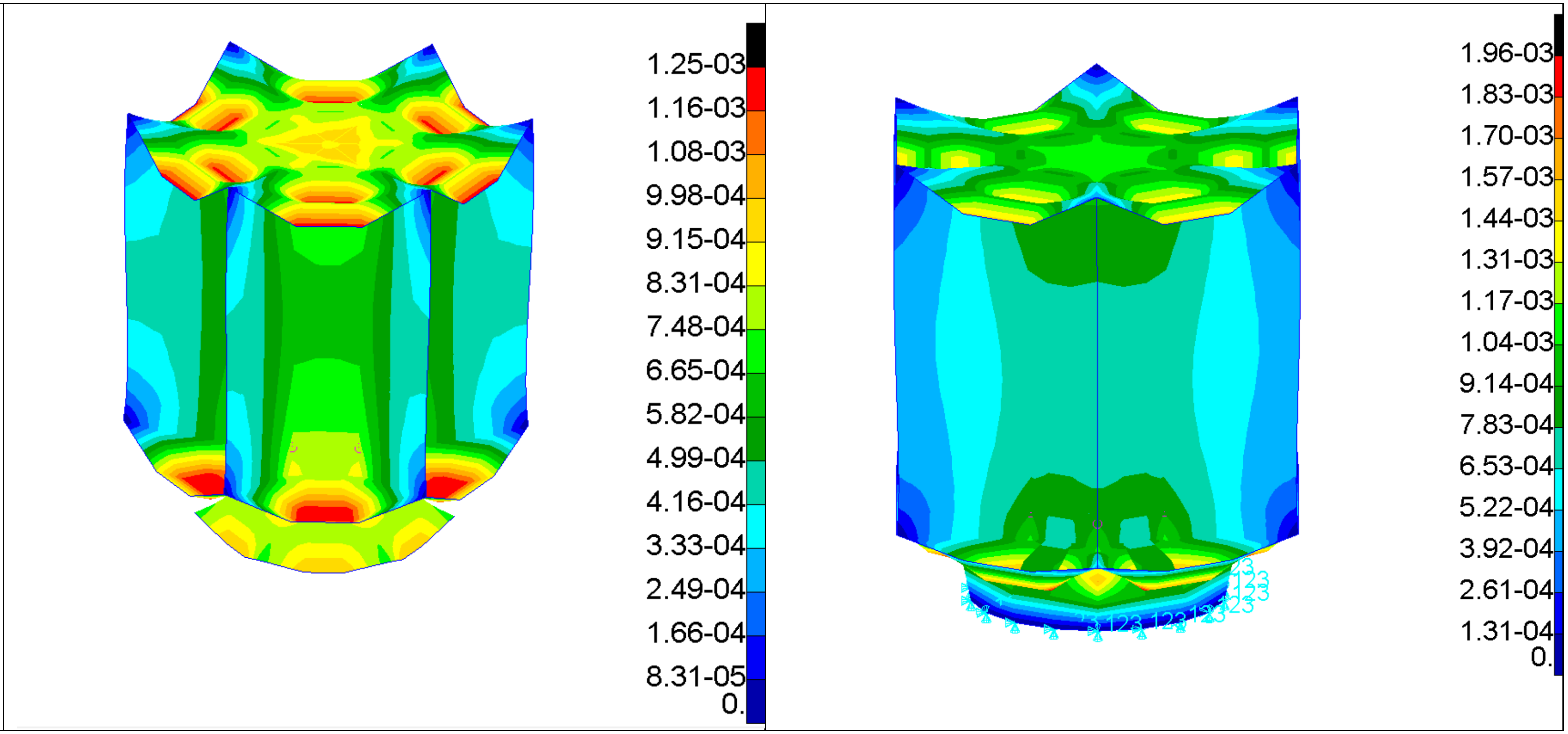

Spectrums
Results
Action: Create
Object: Graph
Method: Y vs X
Select Result Cases
SC1, 801 of 801 subcases
Y: Result
Select Y Result
Constraint Forces, Rotational
Constraint Forces, Translational
Displacements, Rotational
Displacements, Translational
Position...((NON-LAYERED))
Quantity: Magnitude
X: Global Variable
Select Global Variable
Frequency
LOAD_CASE_INDEX

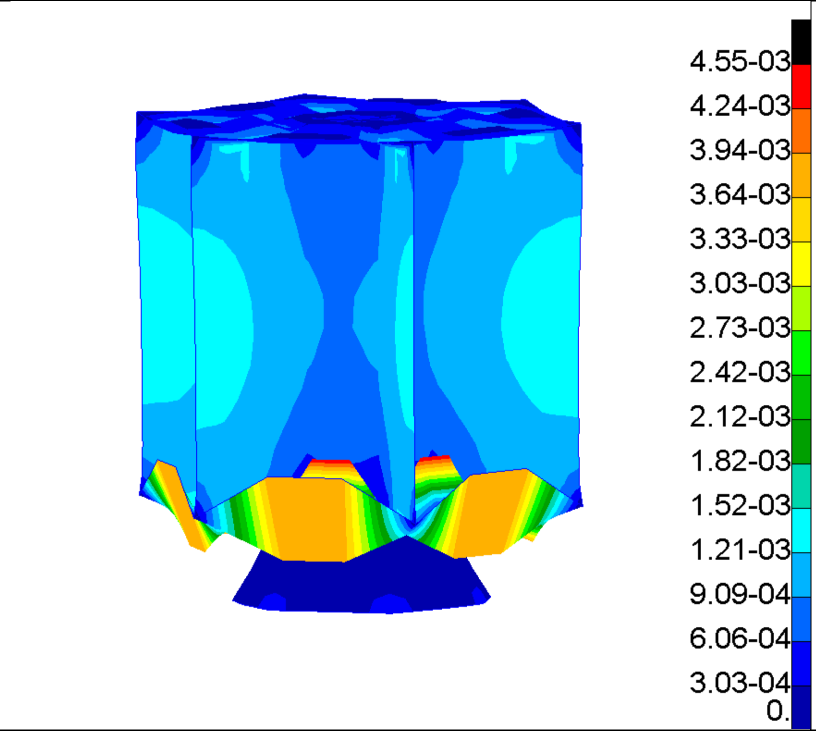
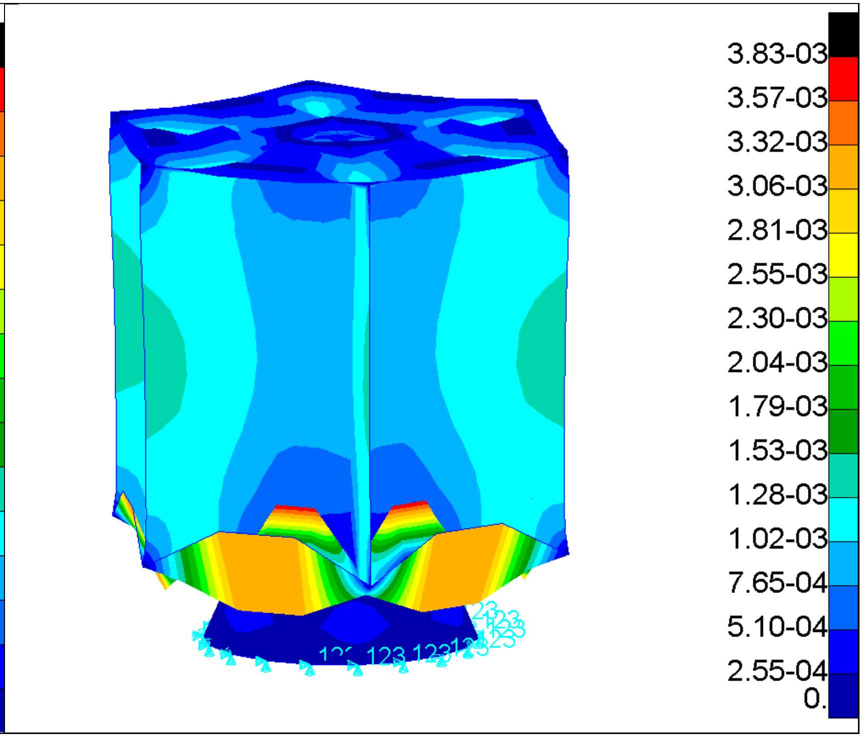
Node Localisation	Frequency Response (free)	Frequency Response (fixed)
		
		
		

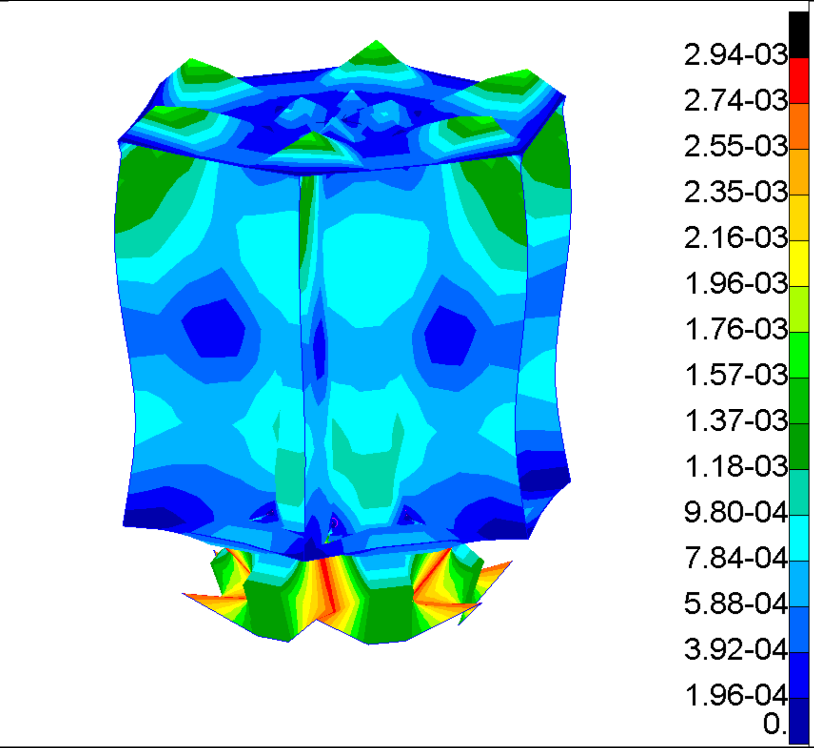
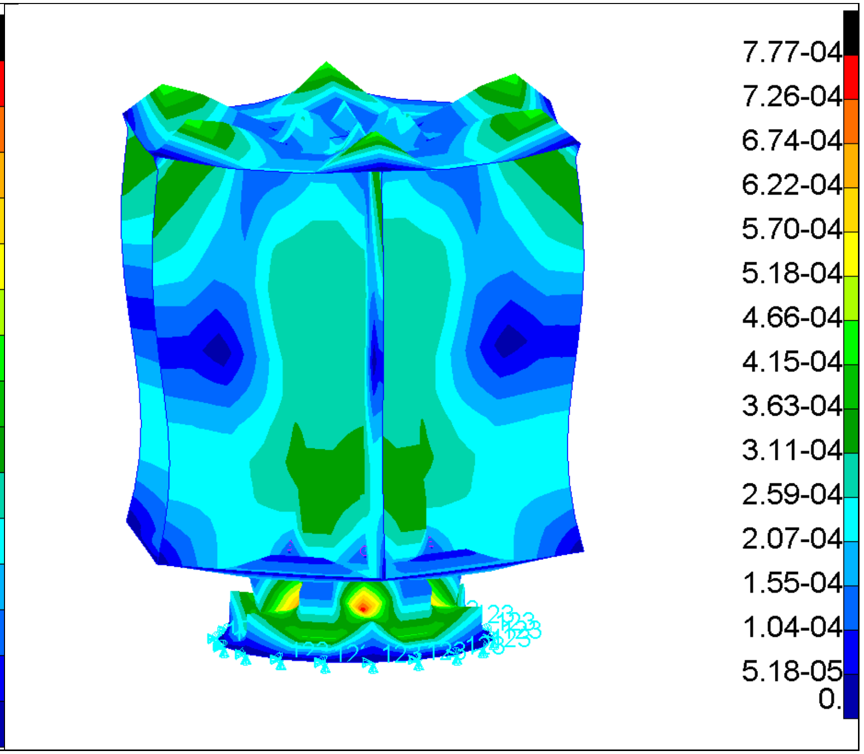

Spectrums
Results
Action: Create
Object: Graph
Method: Y vs X
Target Entity:
Nodes
Select Nodes
Node 706
Apply
Reset
Node 383
Node 613
Displacements, Translational
Frequency

Frequencies	Deformation (Free Structure)	Deformation (Fixed)
100		

1010

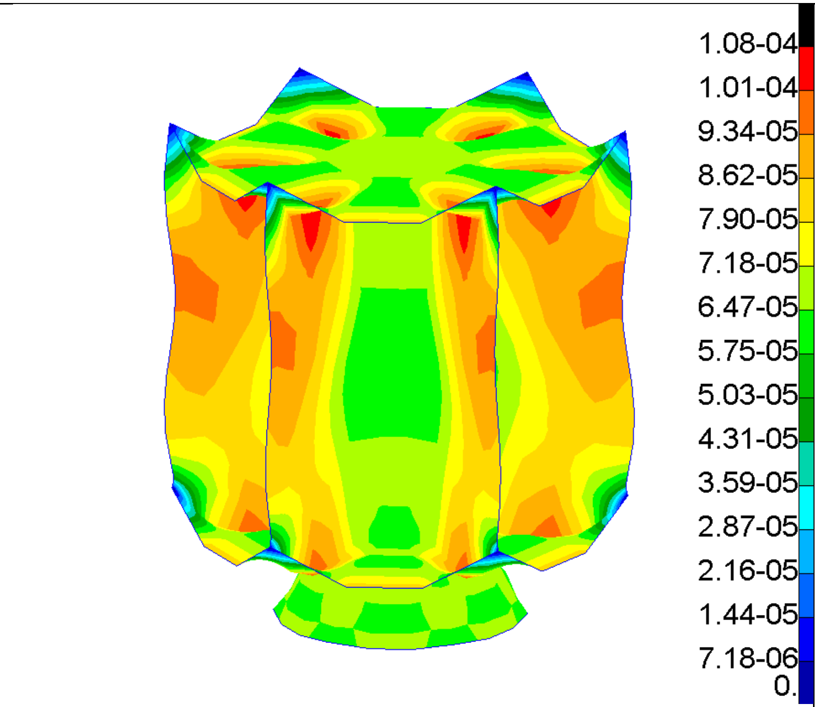
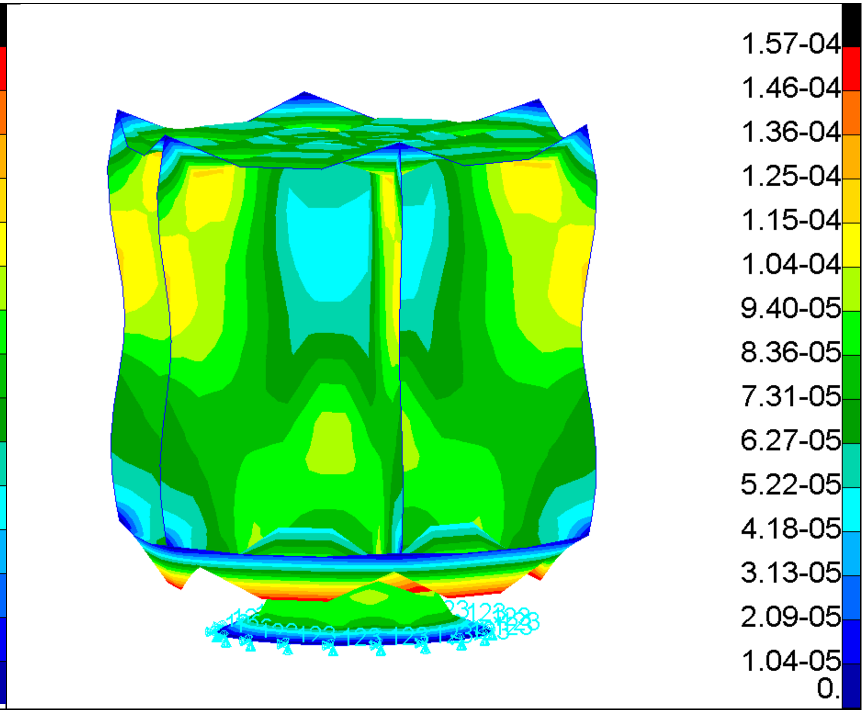

1.25-03
1.16-03
1.08-03
9.98-04
9.15-04
8.31-04
7.48-04
6.65-04
5.82-04
4.99-04
4.16-04
3.33-04
2.49-04
1.66-04
8.31-05
0.
1.96-03
1.83-03
1.70-03
1.57-03
1.44-03
1.31-03
1.17-03
1.04-03
9.14-04
7.83-04
6.53-04
5.22-04
3.92-04
2.61-04
1.31-04
0.

2025	 4.55-03 4.24-03 3.94-03 3.64-03 3.33-03 3.03-03 2.73-03 2.42-03 2.12-03 1.82-03 1.52-03 1.21-03 9.09-04 6.06-04 3.03-04 0.	 3.83-03 3.57-03 3.32-03 3.06-03 2.81-03 2.55-03 2.30-03 2.04-03 1.79-03 1.53-03 1.28-03 1.02-03 7.65-04 5.10-04 2.55-04 0.

3370		

2.94-03
2.74-03
2.55-03
2.35-03
2.16-03
1.96-03
1.76-03
1.57-03
1.37-03
1.18-03
9.80-04
7.84-04
5.88-04
3.92-04
1.96-04
0.

7.77-04
7.26-04
6.74-04
6.22-04
5.70-04
5.18-04
4.66-04
4.15-04
3.63-04
3.11-04
2.59-04
2.07-04
1.55-04
1.04-04
5.18-05
0.

4000		

Before conducting a frequency analysis with excitation, I first performed a frequency analysis without excitation, using two boundary conditions. The first condition considered the structure as free, and the second condition treated the structure as fixed. For the frequency analysis method, I chose the modal analysis approach. Although this part wasn't required, I thought it was relevant to include.

Here's what I observed:

Since no force was applied, I selected nodes at different parts of the satellite—at the shear panels, the upper platform, the lower platform, the adapter, and the cylinder. I observed how the frequencies at these points responded when the structure was free versus when it was fixed.

- When the structure is fixed, the frequencies are much lower. For example, the amplitude drops by a factor of 10 compared to when the structure is free.

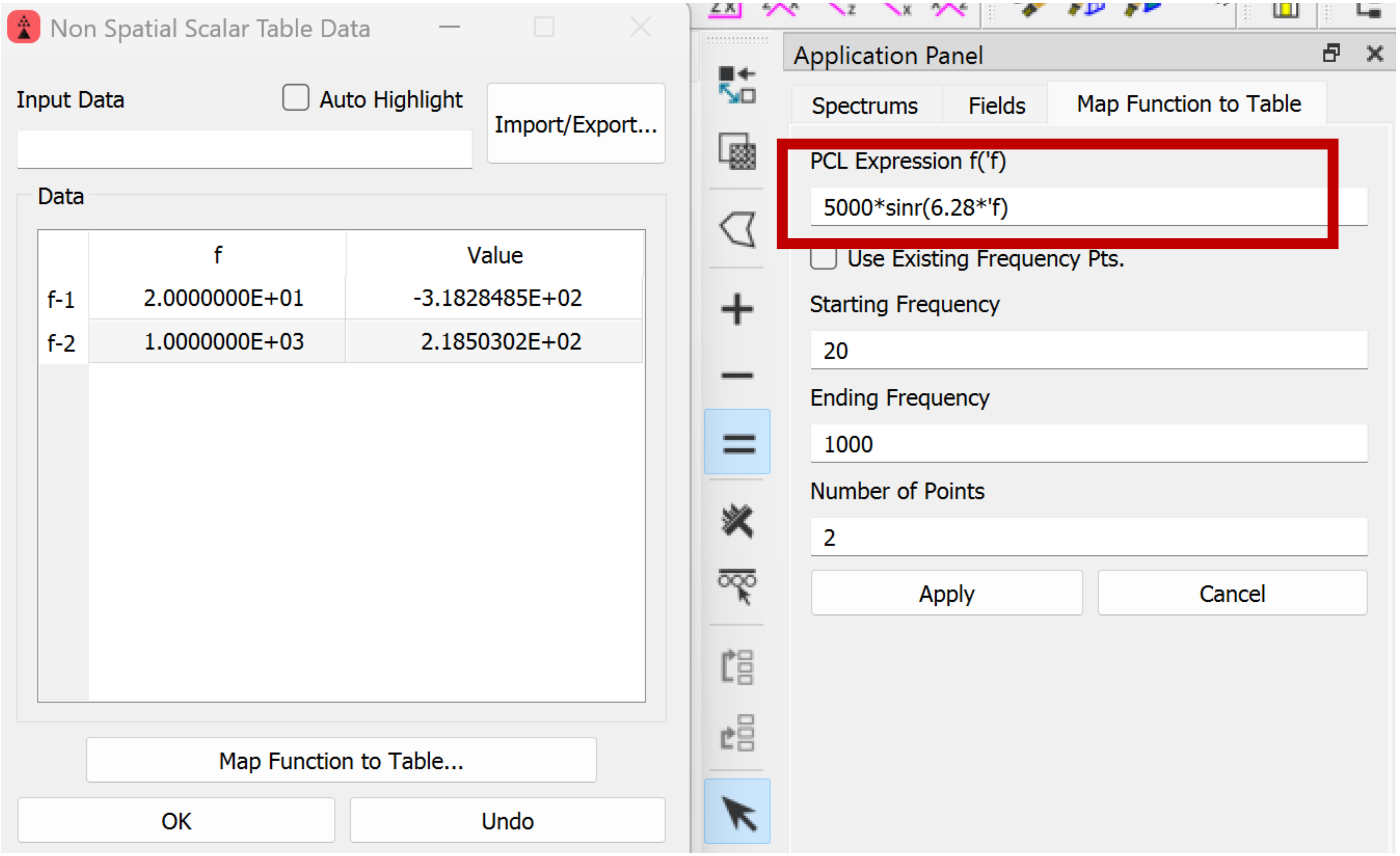
- In the free condition, the amplitude is around 10, but when fixed, the amplitude decreases to about 1.

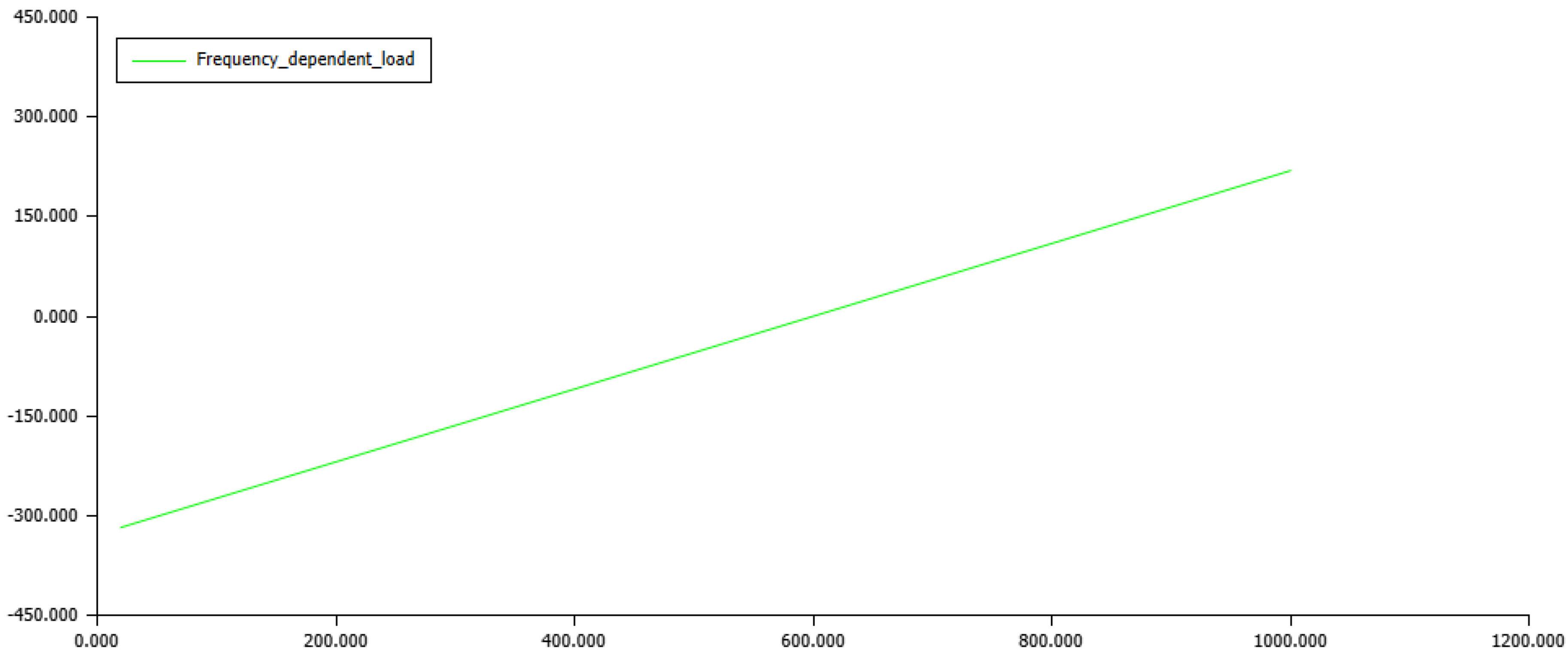
This indicates that the frequency response changes significantly depending on whether the structure is fixed or free. However, I'm unsure about the exact interpretation of this result, so if you have any insights, please share.

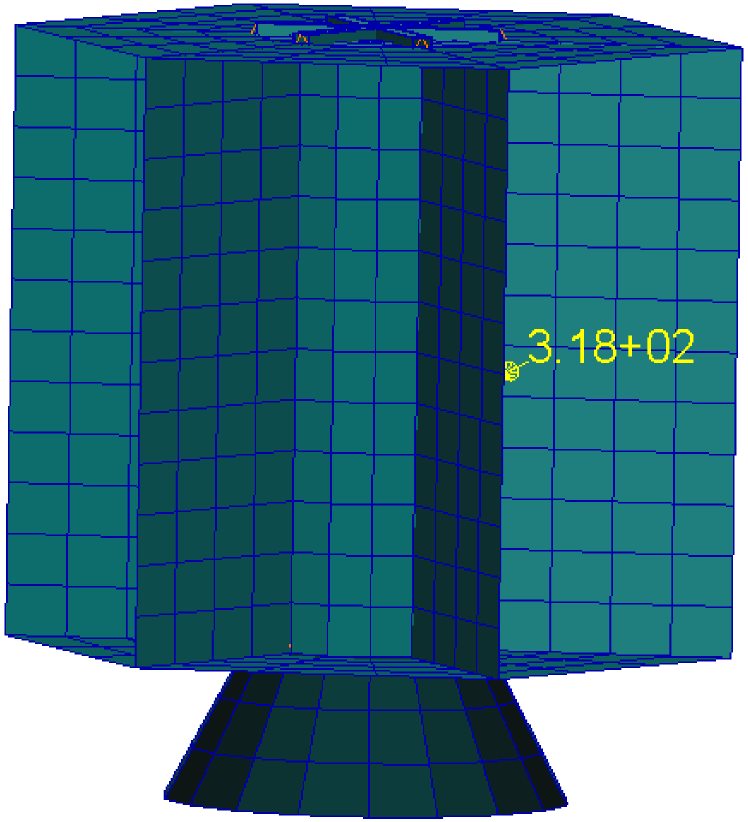
- The frequency response varies based on the position of the nodes.
- For example, areas such as the lower platform and the upper platform show more peaks, greater dynamism, and significant variations in the response.
- Deformations are more concentrated at the upper plate, which appears to be highly sensitive.
- This sensitivity is likely because the upper plate is exposed and serves as the navigation platform, making it more prone to vibrations and fragility.
- As the frequency increases, the entire structure becomes more affected.
- When the structure is free, there are significant deformations, especially in the adapter.
- When the structure is fixed, these deformations are more limited.
- Overall, higher frequencies lead to more pronounced involvement of all structural parts.
- When the structure is fixed, the amplitudes decrease significantly compared to when the structure is free.

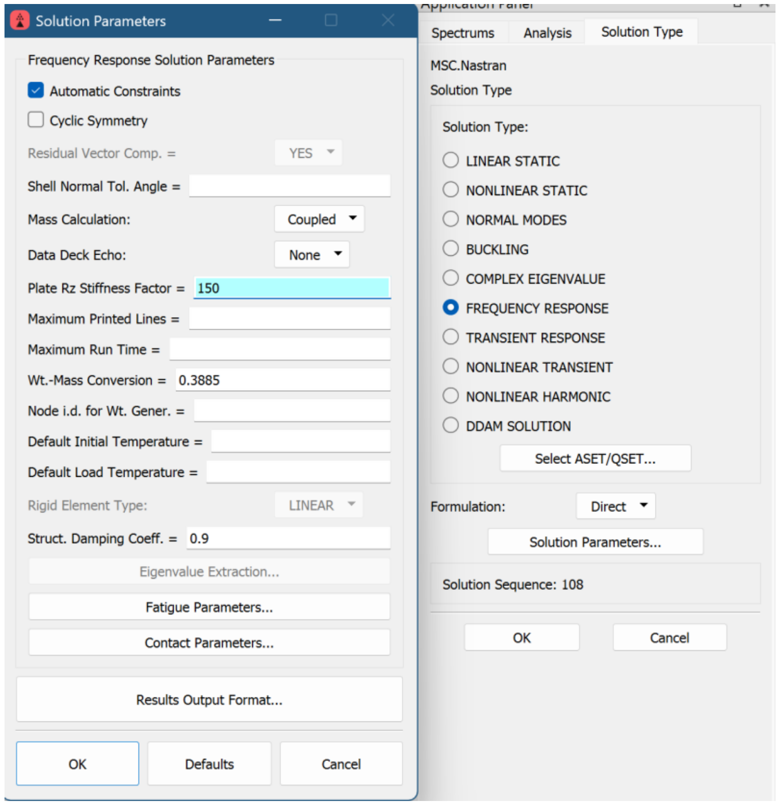
This preliminary study highlights how the structure's frequency response varies with boundary conditions, node positions, and increasing frequencies. I found it particularly important to conduct this analysis before moving on to the study involving excitation. It provides a baseline understanding of the structure's sensitivity and deformation under different conditions.

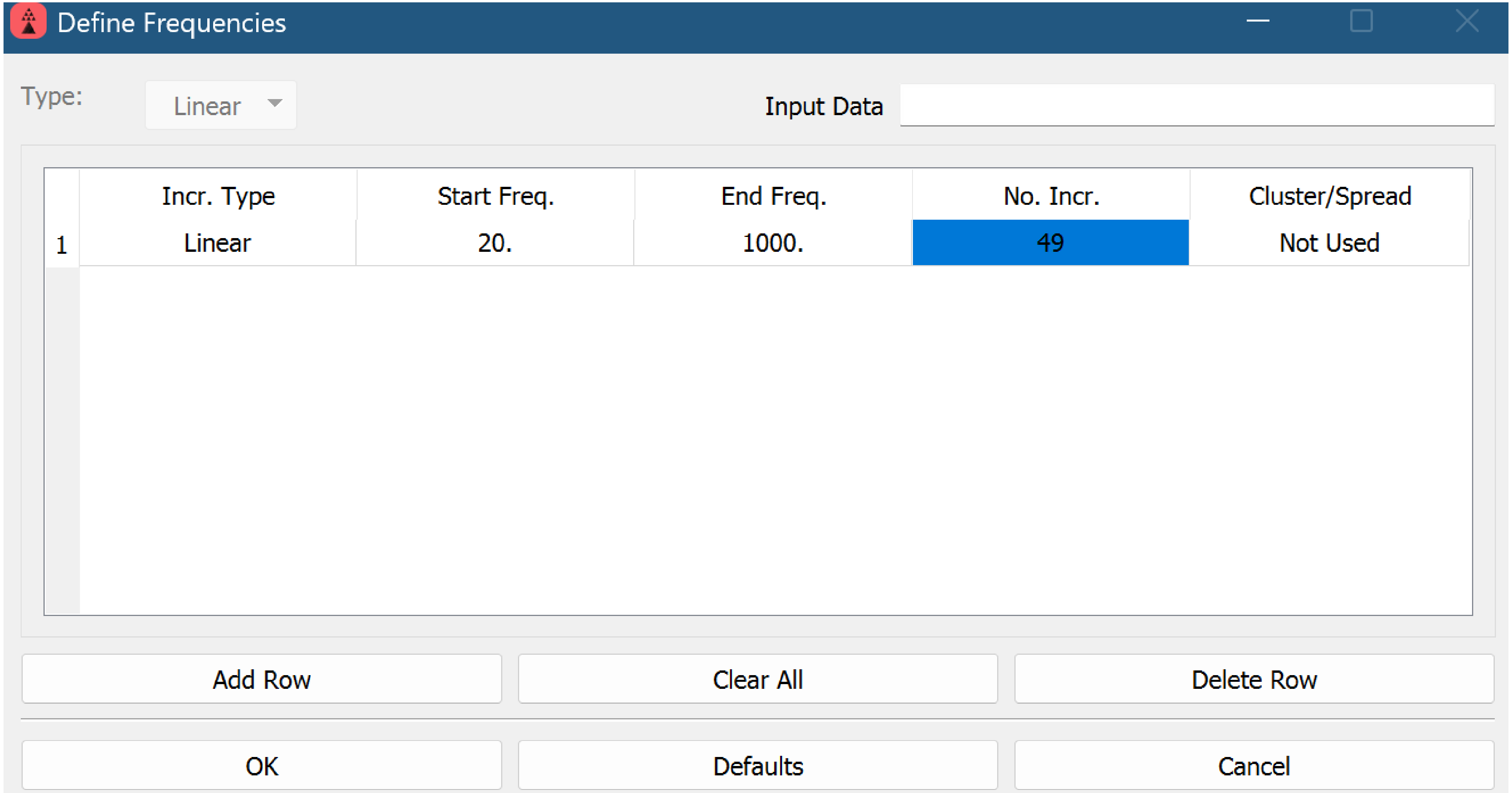
Frequency Analysis Direct with excitation with the maximum Mass found added



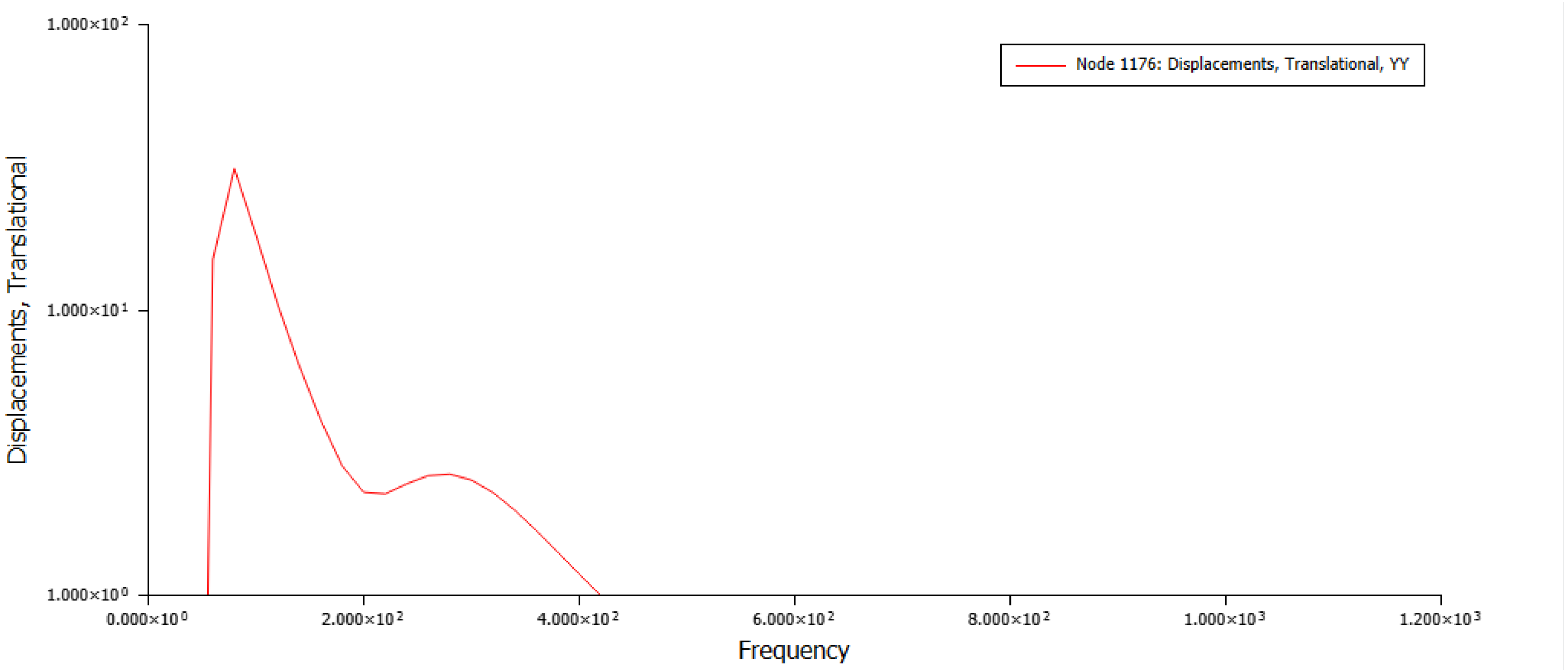

450.000
300.000
150.000
0.000
-150.000
-300.000
-450.000
0.000
200.000
400.000
600.000
800.000
1000.000
1200.000
Frequency_dependent_load

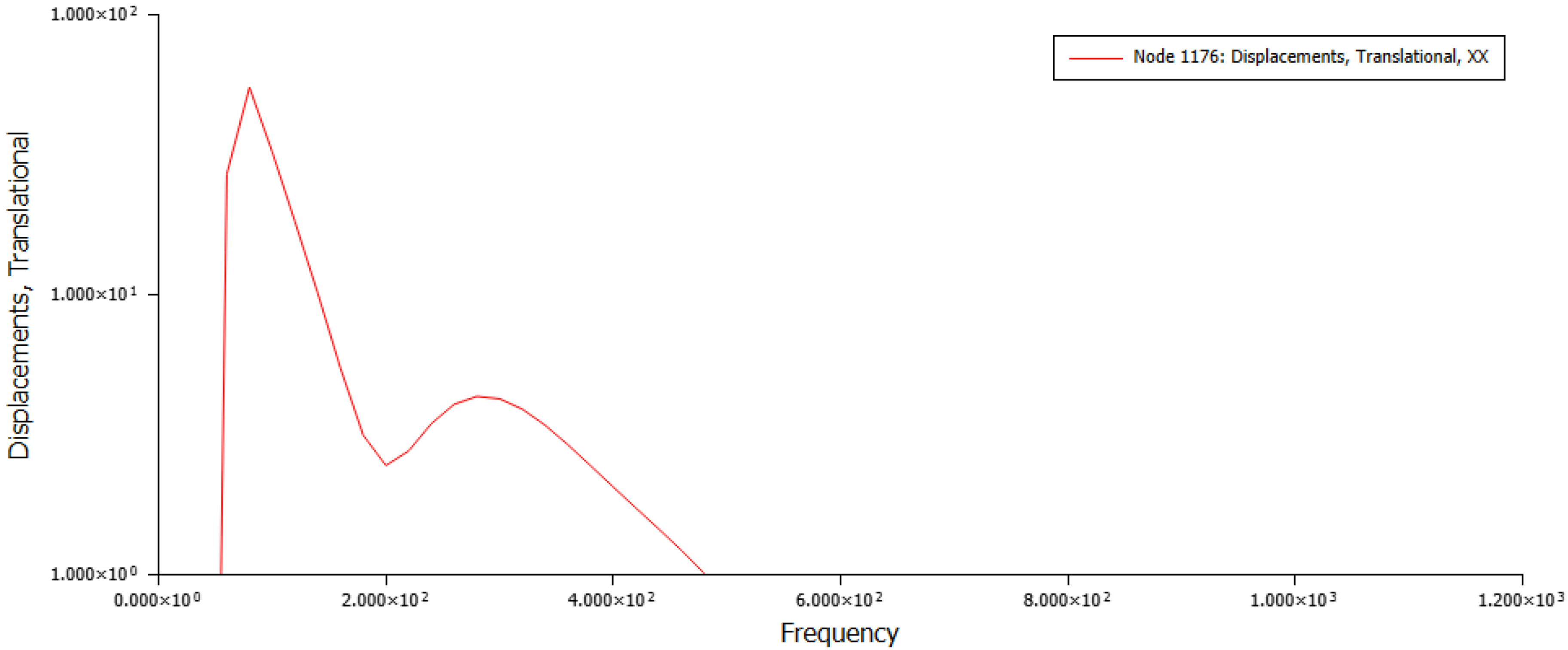

3.18+02


Solution Parameters
Frequency Response Solution Parameters
Automatic Constraints
Cyclic Symmetry
Residual Vector Comp. = YES
Shell Normal Tol. Angle =
Mass Calculation: Coupled
Data Deck Echo: None
Plate Rz Stiffness Factor = 150
Maximum Printed Lines =
Maximum Run Time =
Wt.-Mass Conversion = 0.3885
Node i.d. for Wt. Gener. =
Default Initial Temperature =
Default Load Temperature =
Rigid Element Type: LINEAR
Struct. Damping Coeff. = 0.9
Eigenvalue Extraction...
Fatigue Parameters...
Contact Parameters...
Results Output Format...
OK
Defaults
Cancel
Spectrums
Analysis
Solution Type
MSC.Nastran
Solution Type
Solution Type:
LINEAR STATIC
NONLINEAR STATIC
NORMAL MODES
BUCKLING
COMPLEX EIGENVALUE
FREQUENCY RESPONSE
TRANSIENT RESPONSE
NONLINEAR TRANSIENT
NONLINEAR HARMONIC
DDAM SOLUTION
Select ASET/QSET...
Formulation: Direct
Solution Parameters...
Solution Sequence: 108
OK
Cancel

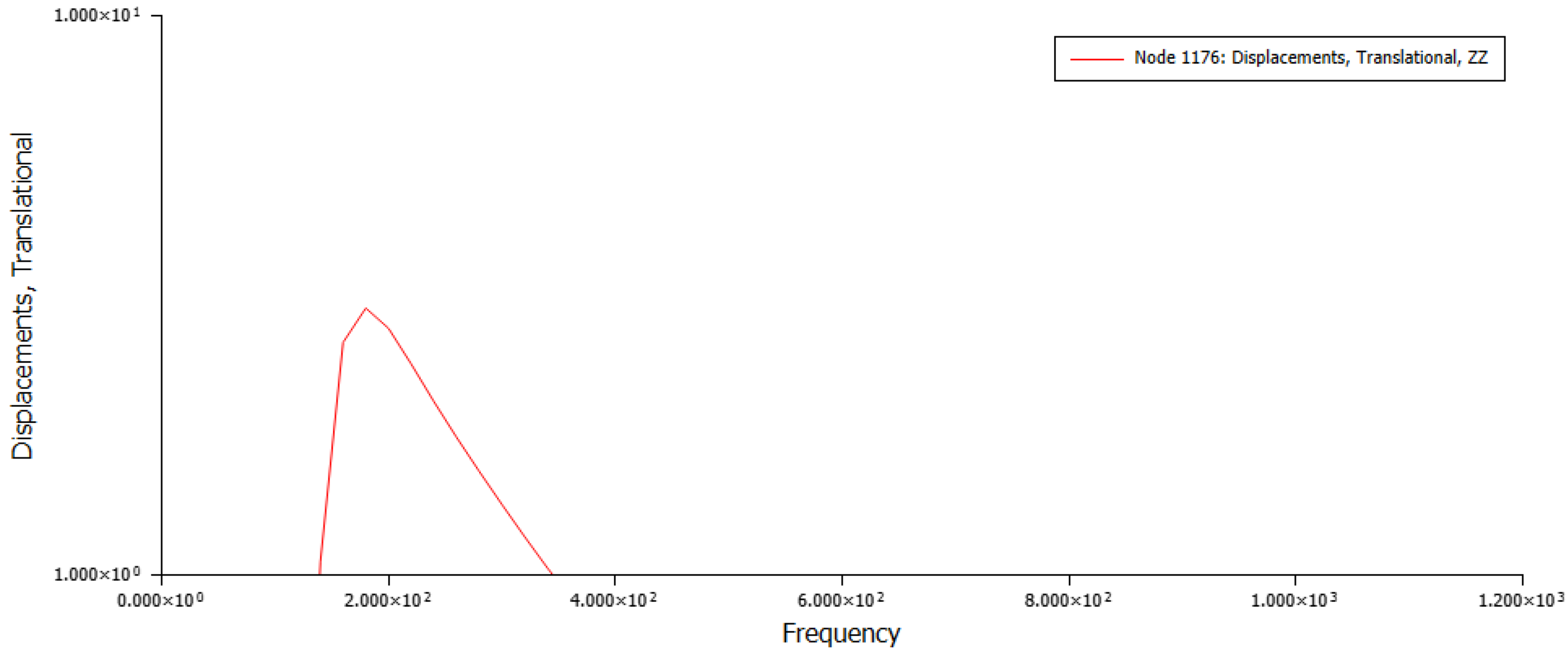

Define Frequencies
Type:
Linear
Input Data
Incr. Type
Start Freq.
End Freq.
No. Incr.
Cluster/Spread
1
Linear
20.
1000.
49
Not Used
Add Row
Clear All
Delete Row
OK
Defaults
Cancel

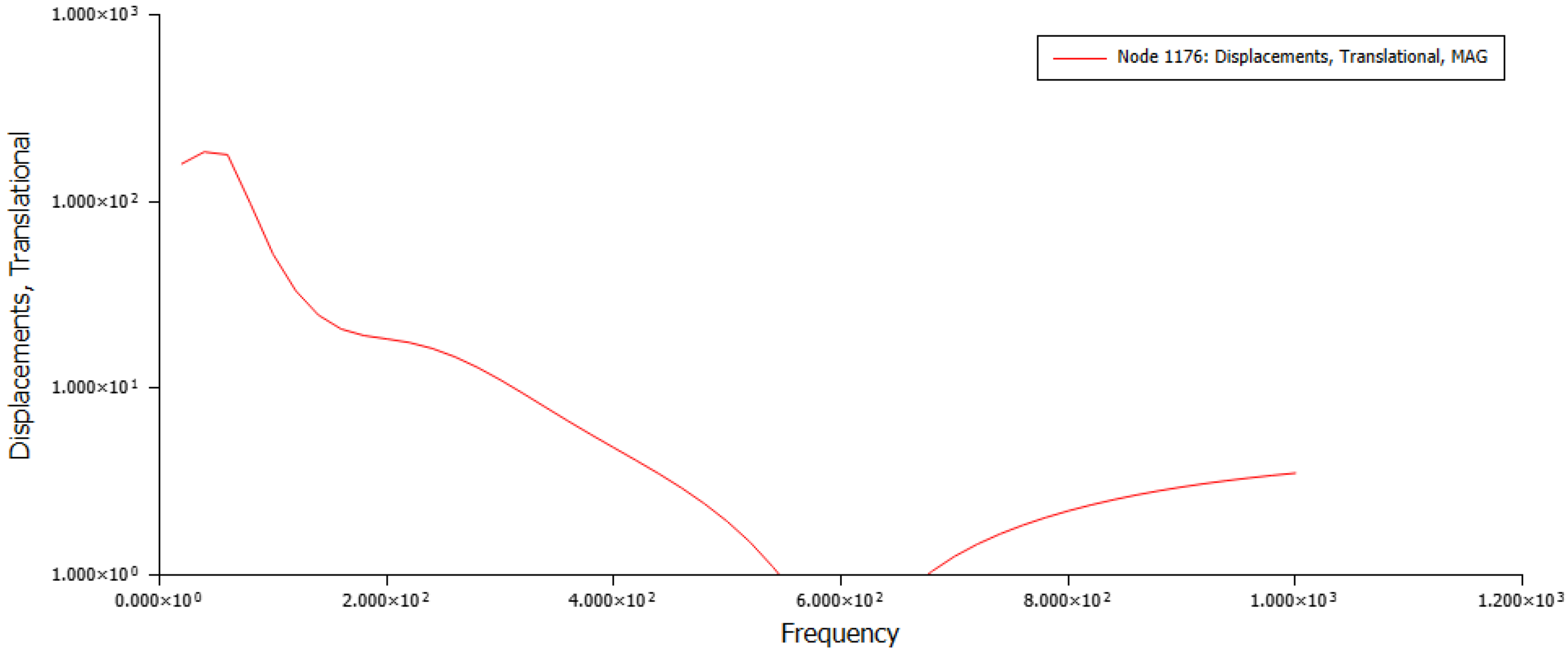
For fixed Boundary Conditions





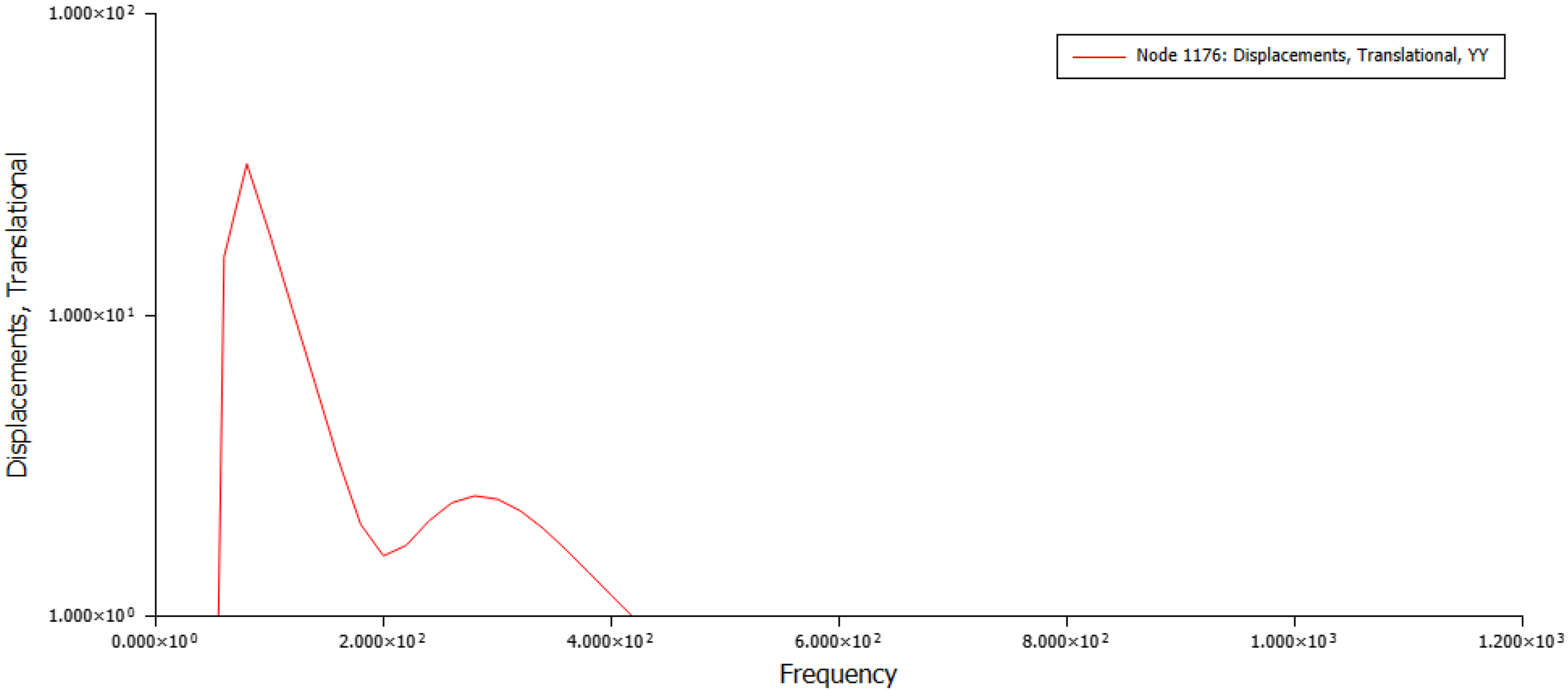
1.000×10²
1.000×10¹
1.000×10⁰
Node 1176: Displacements, Translational, XX
Displacements, Translational
0.000×10⁰
2.000×10²
4.000×10²
6.000×10²
8.000×10²
1.000×10³
1.200×10³
Frequency

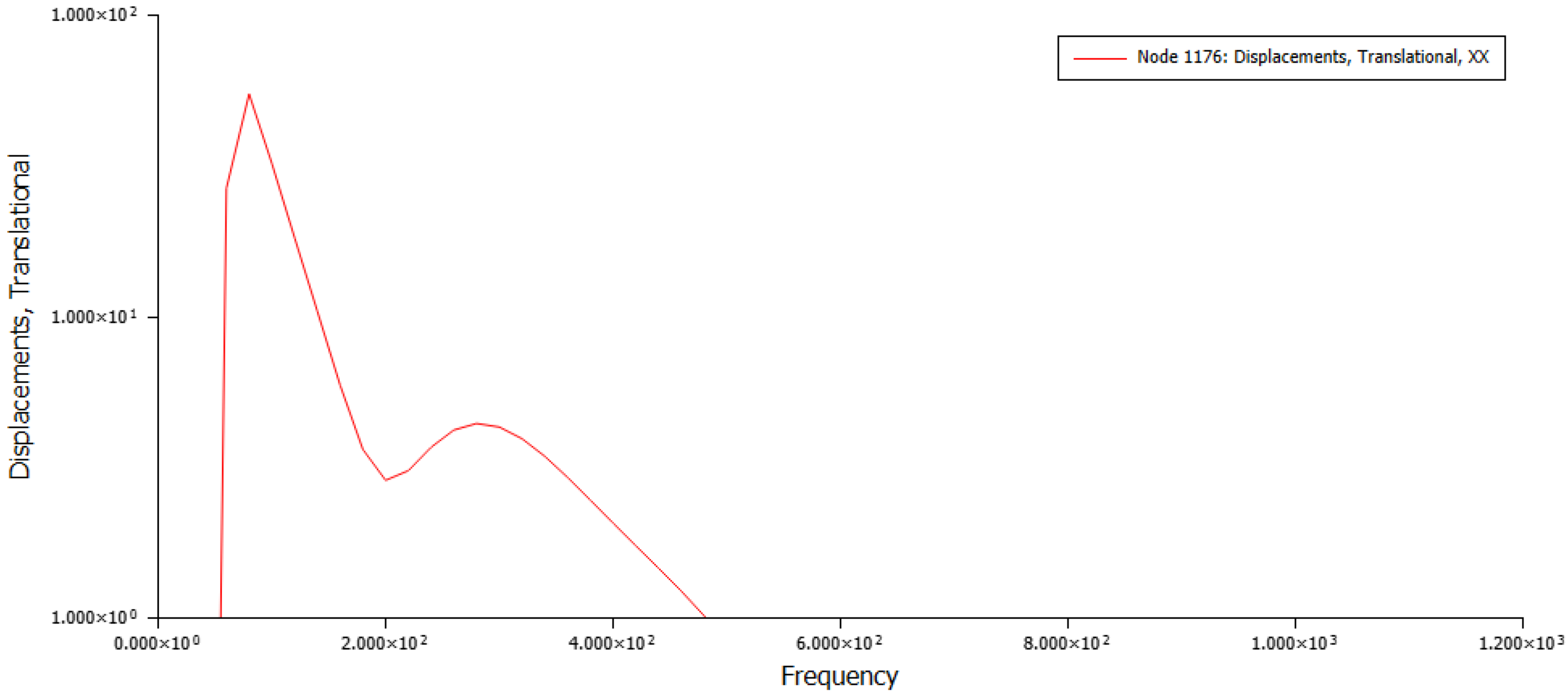

Node 1176: Displacements, Translational, ZZ
Displacements, Translational
1.000×10¹
1.000×10⁰
0.000×10⁰
2.000×10²
4.000×10²
6.000×10²
8.000×10²
1.000×10³
1.200×10³
Frequency



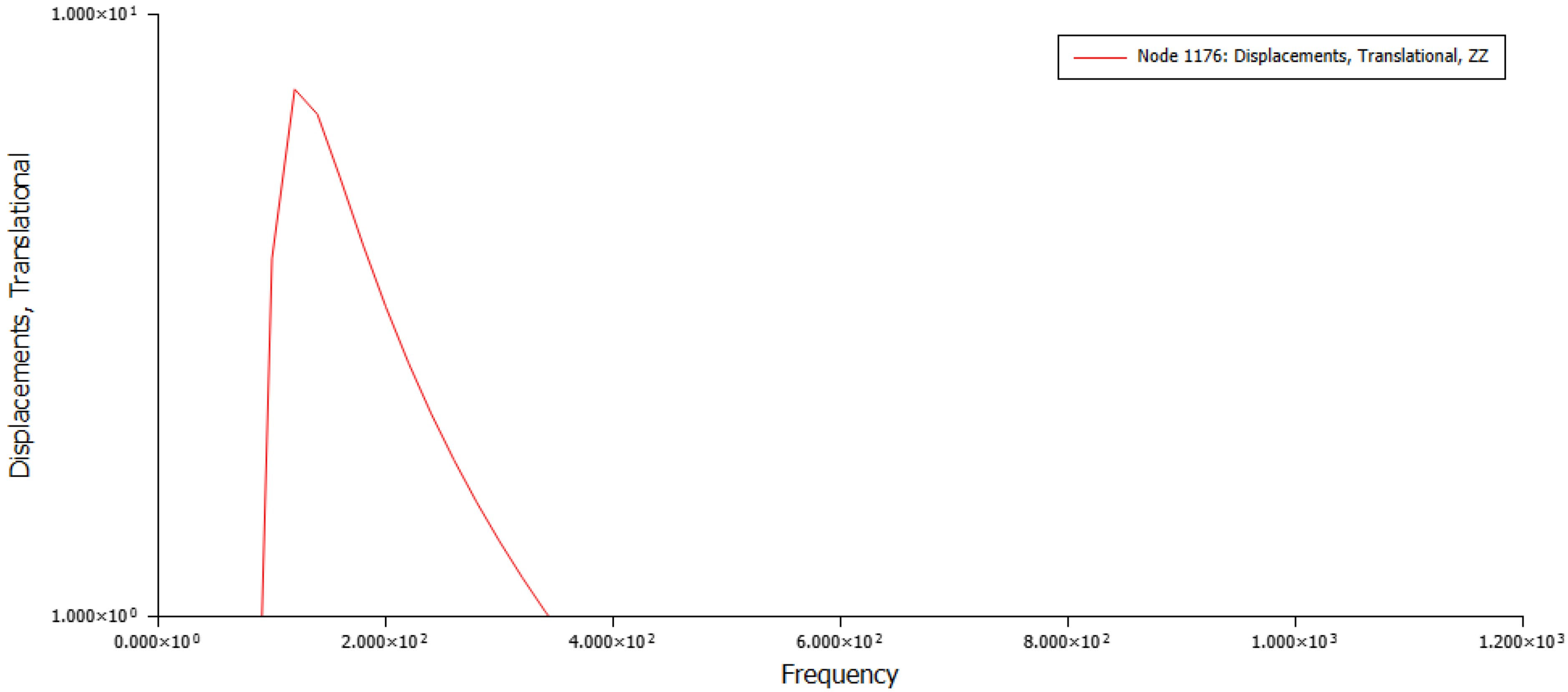
1.000×10³
1.000×10²
1.000×10¹
1.000×10⁰
Displacements, Translational
Node 1176: Displacements, Translational, MAG
0.000×10⁰
2.000×10²
4.000×10²
6.000×10²
8.000×10²
1.000×10³
1.200×10³
Frequency

For free Boundary Conditions

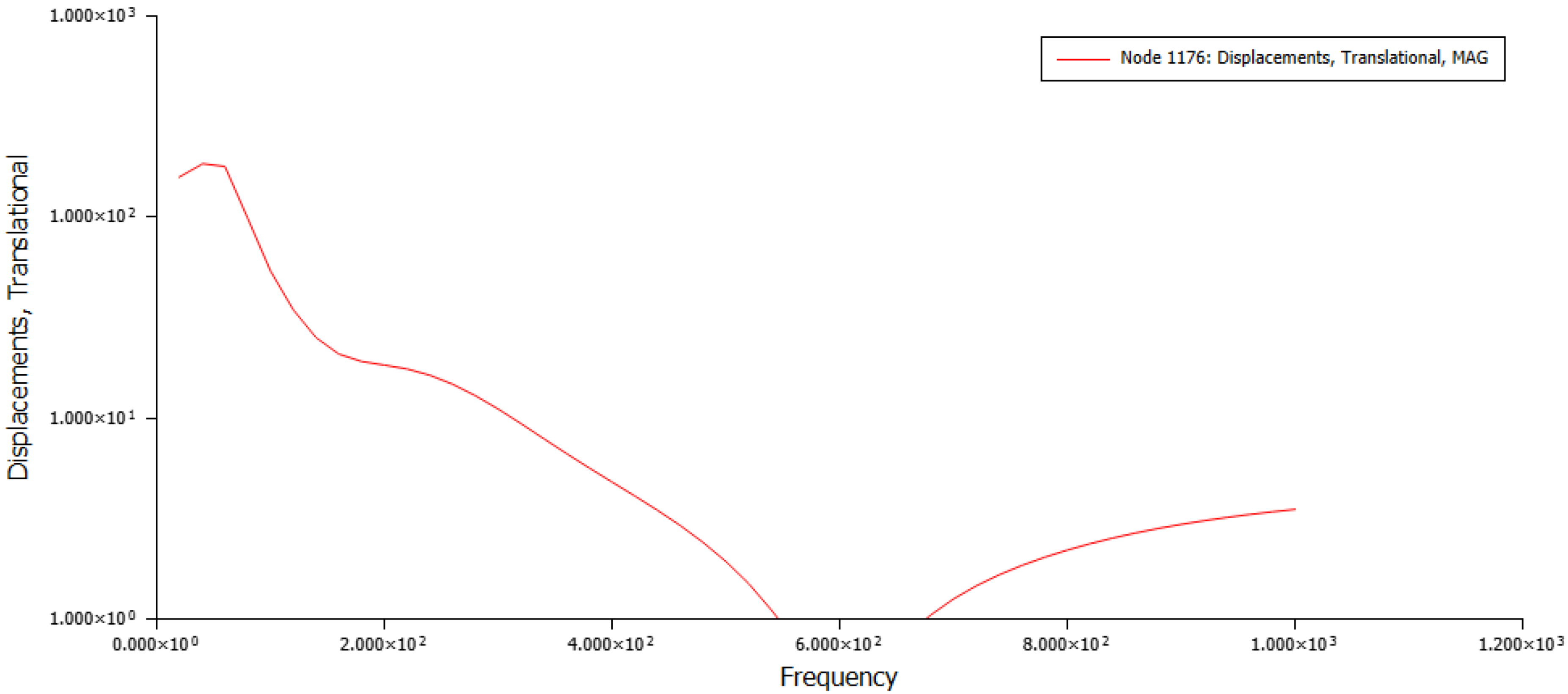




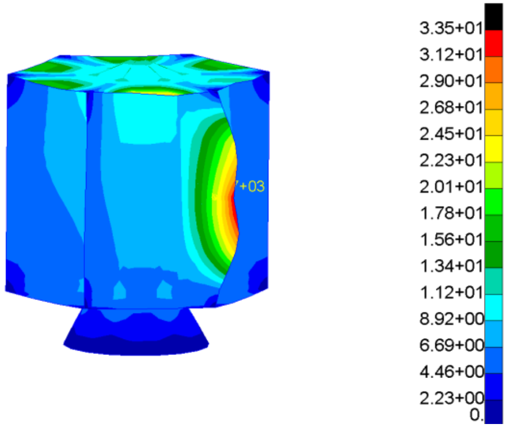
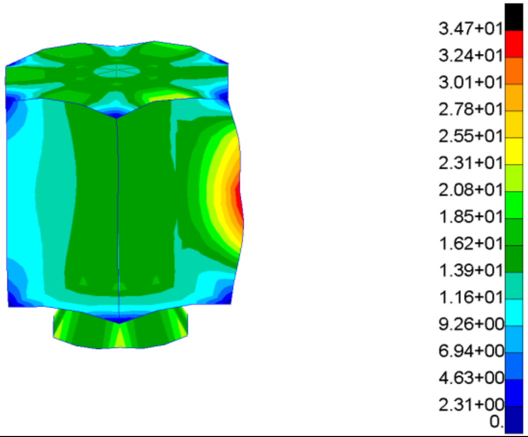
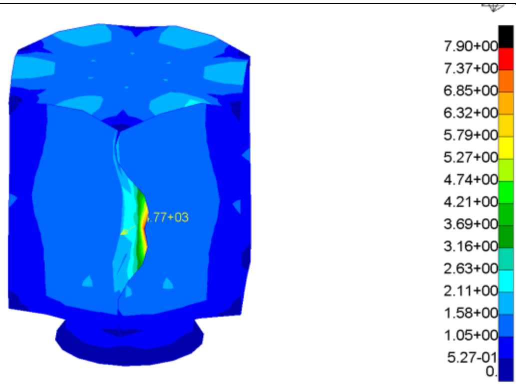
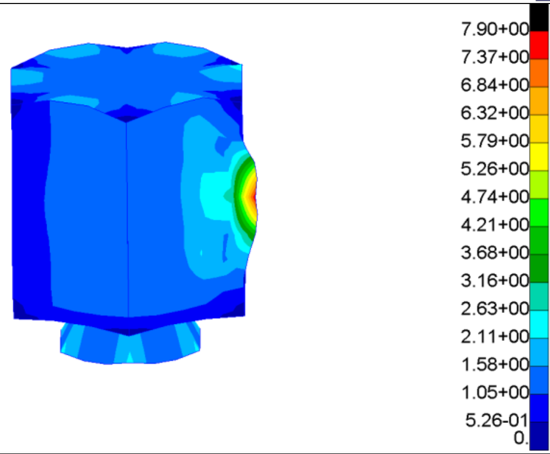
Node 1176: Displacements, Translational, XX
1.000×10^2
1.000×10^1
1.000×10^0
Displacements, Translational
0.000×10^0
2.000×10^2
4.000×10^2
6.000×10^2
8.000×10^2
1.000×10^3
1.200×10^3
Frequency

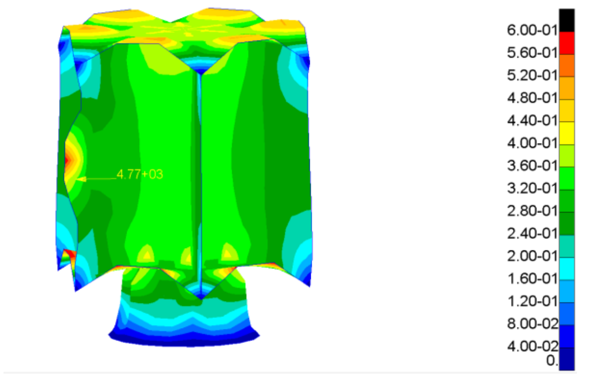
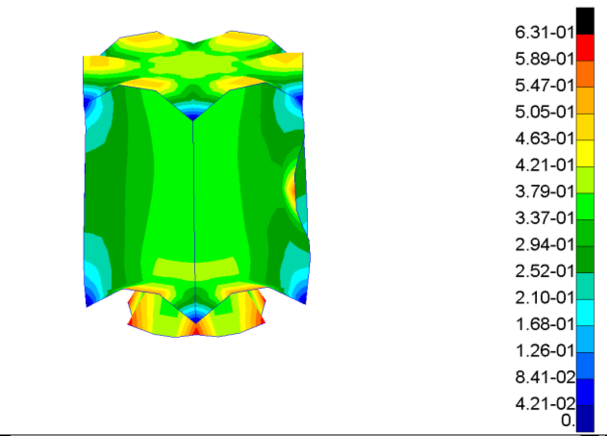
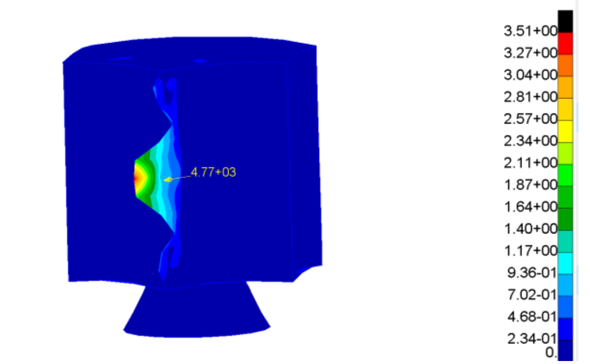
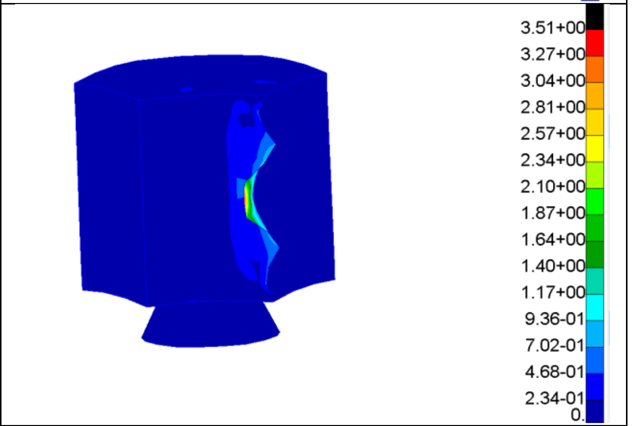


1.000×10¹
1.000×10⁰
Node 1176: Displacements, Translational, ZZ
Displacements, Translational
0.000×10⁰
2.000×10²
4.000×10²
6.000×10²
8.000×10²
1.000×10³
1.200×10³
Frequency



Frequencies	Deformations (Fixed)	Deformations (Free)

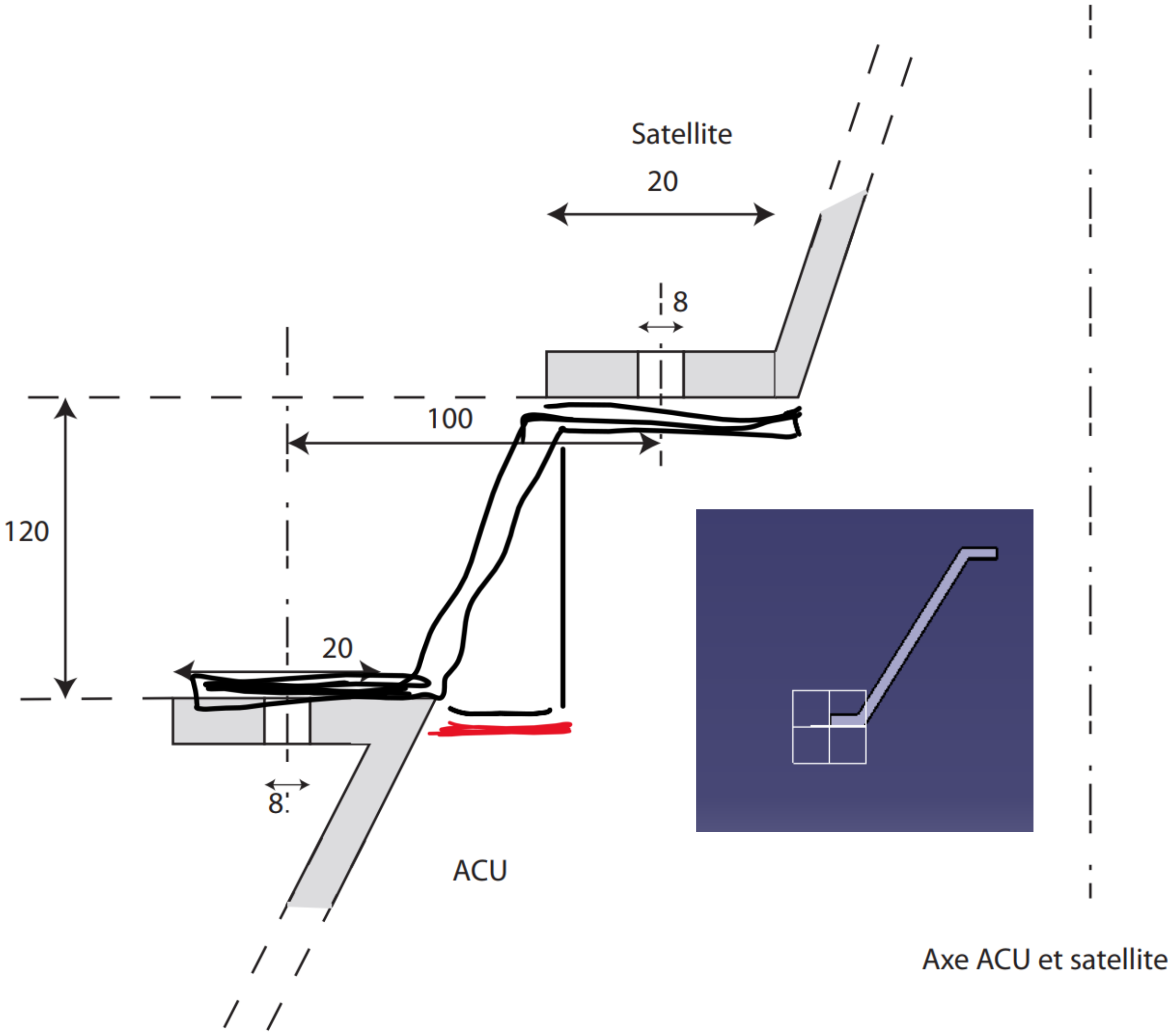
<table>
<tr>
<td>120</td>
<td>
</td>
<td>
</td>
</tr>
<tr>
<td>340</td>
<td>
</td>
<td>
</td>
</tr>
</table>

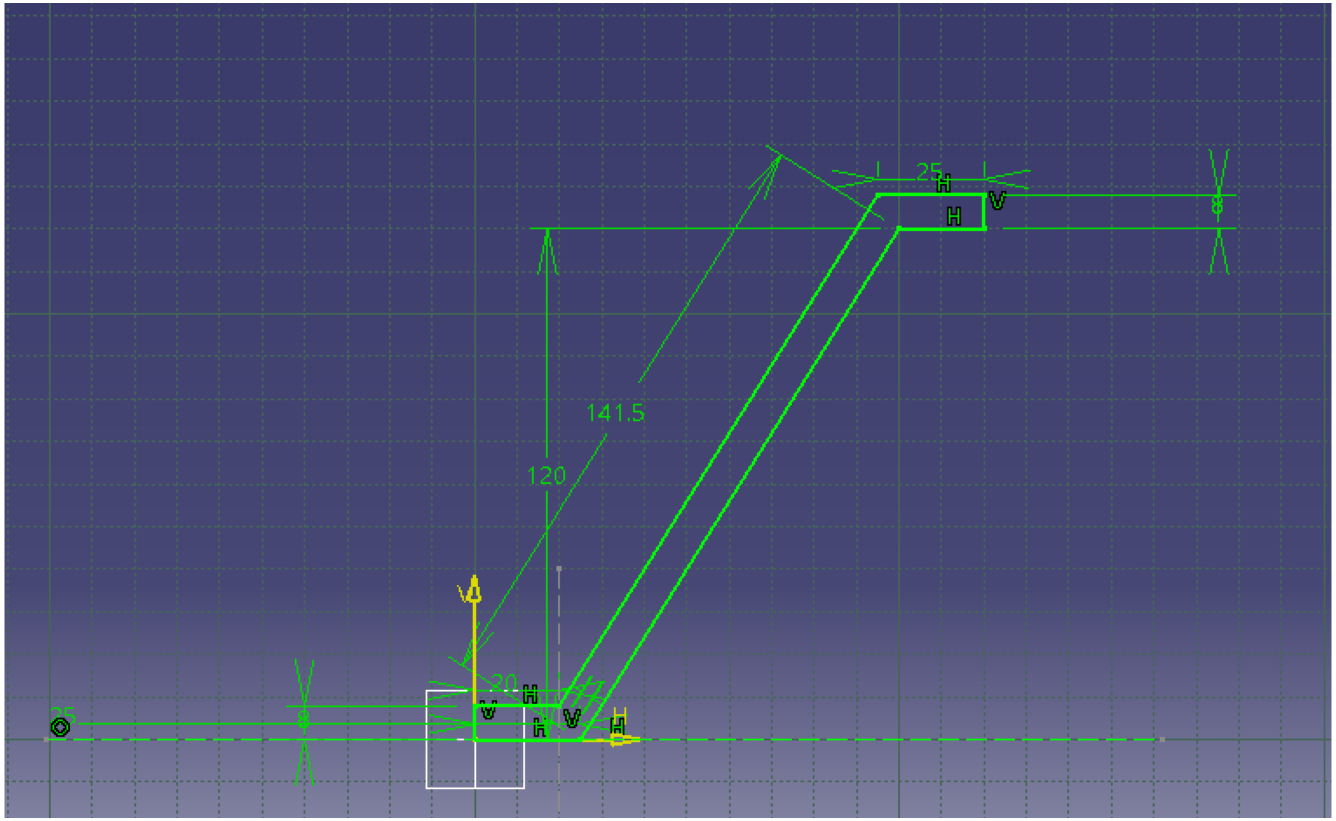
640		
1000		

We applied a frequency-dependent excitation to one of the shear panels and used the direct frequency response method for the analysis. We also tested the modal method but observed no significant difference between the two approaches, likely due to the way the analysis was configured. For this reason, the results of the modal method are not presented, as they add no new insights. Observations show frequency peaks with variations depending on the direction (X, Y, or Z). When examining displacements as a function of frequency, we noticed that displacement decreases as frequency increases. This behaviour can be explained by the dynamics of vibrational systems: as frequency increases, the dynamic stiffness (proportional to the square of the frequency) rises, reducing the system's ability to respond to excitation. Additionally, at higher frequencies, the inertia of the structure becomes dominant, resisting rapid changes in motion and thus further reducing displacements. Energy dissipation due to material damping and structural losses also plays a role in limiting displacement amplitudes at higher frequencies. Deformations were observed, and as the frequency increased, more parts of the structure were affected, with the deformations most pronounced in the area where the excitation was applied. Since the central cylinder is highly rigid, the excitation did not deeply penetrate the structure but remained concentrated around the shear panel. This indicates that changing the point of application of the excitation would alter its effects on the structure, which is an important consideration for future analyses. These results demonstrate the expected behaviour of vibrational systems, where higher frequencies reduce displacements due to increased stiffness, inertia effects, and energy dissipation, while also highlighting the localized effect of excitation in the shear panel due to the rigidity of the central cylinder.

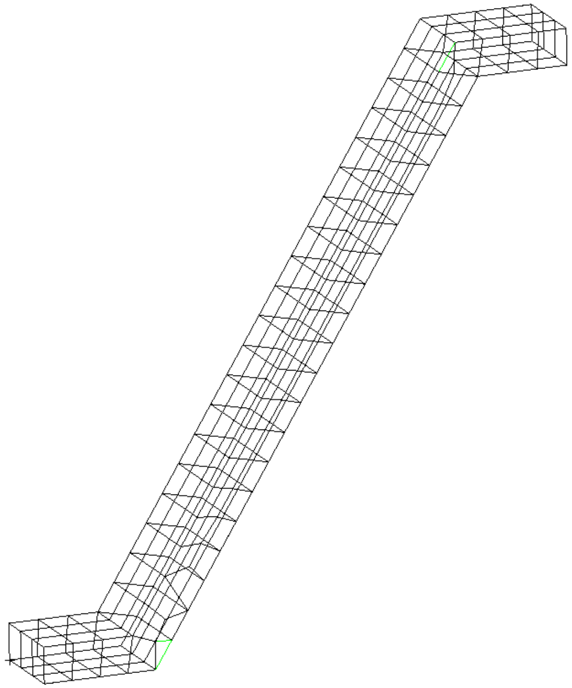
3. Conception des attaches du satellite à l'ACU

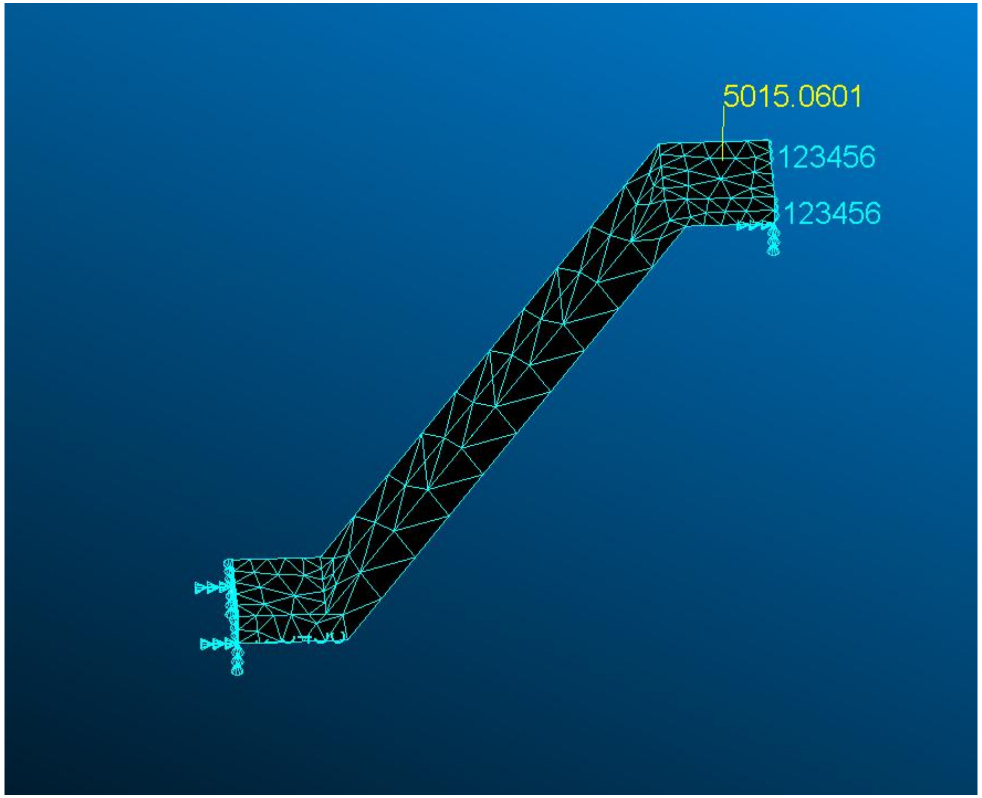
The satellite is attached to the ACU by four evenly distributed fasteners on the upper aluminium frame. The task is to design and size a fastener based on the dimensional constraints provided in the following figure (using the previously calculated satellite mass with equipment, and the static acceleration given in Annex 2). The material used is aluminium: E=70,000 MPa, γ=0.33, and Re=220 MPa.

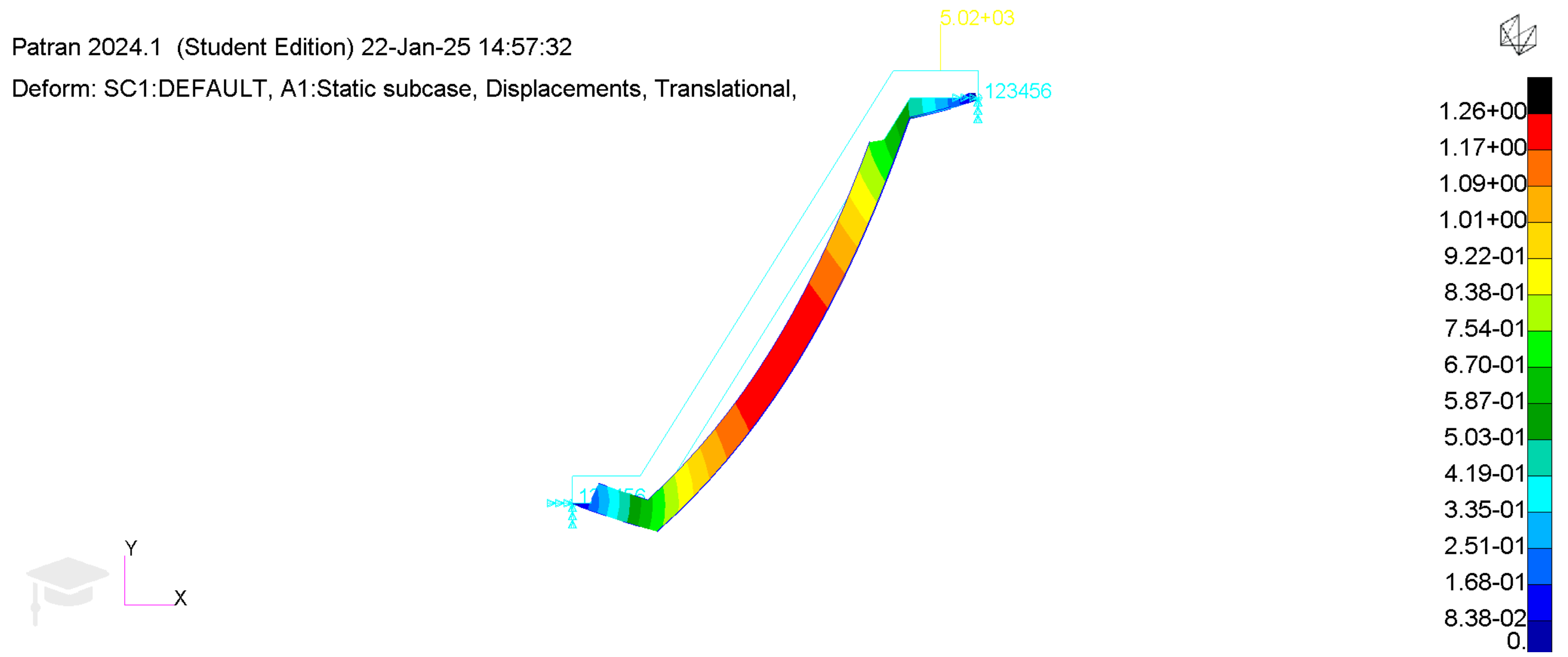

Satellite
20
8
100
120
20
8
ACU
Axe ACU et satellite



Width: 20 mm



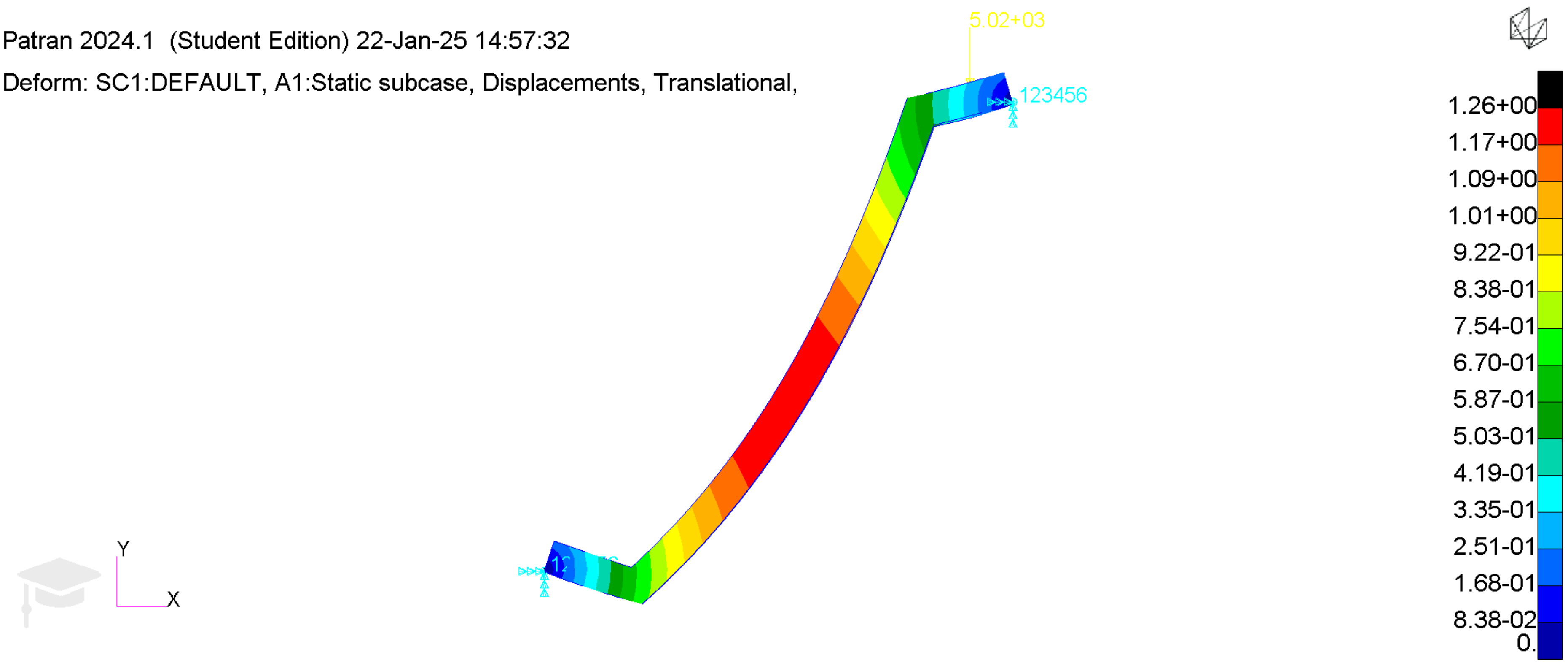

5015.0601
123456
123456

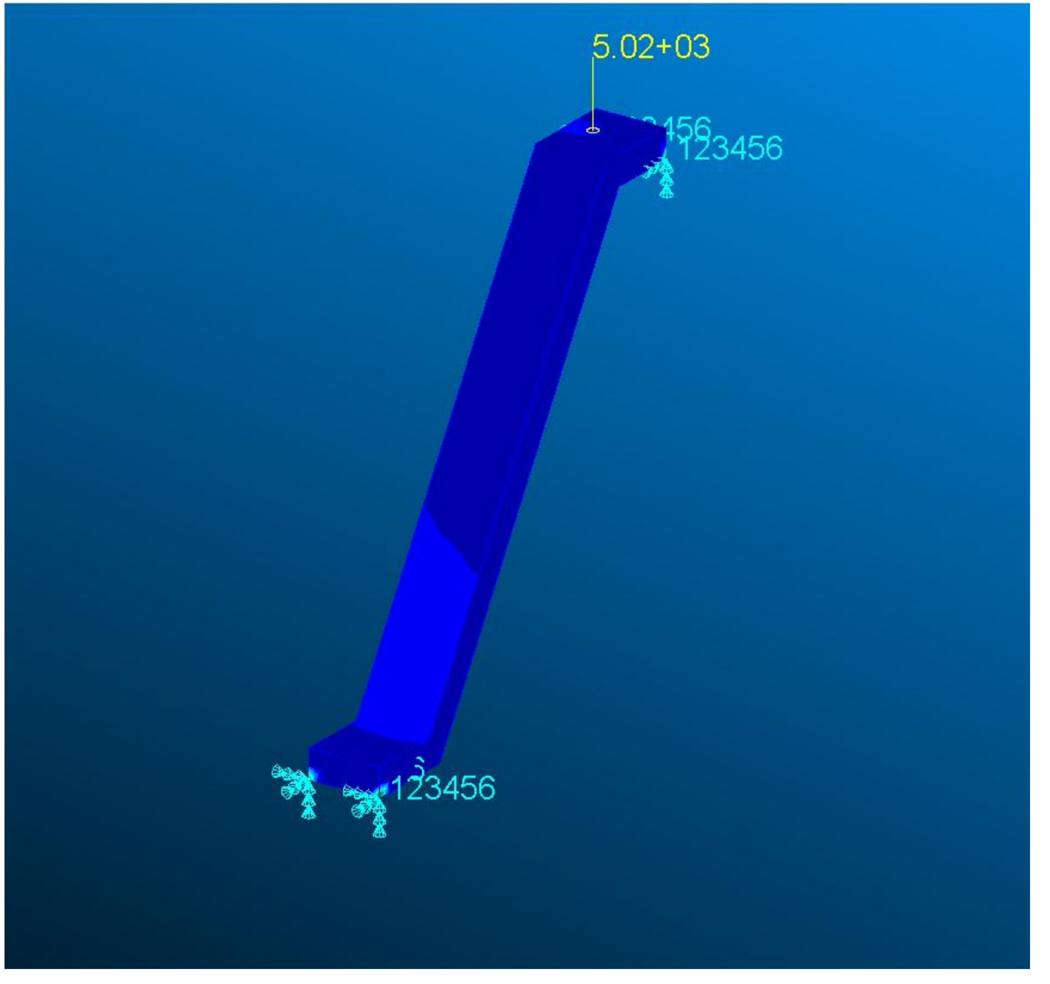


Patran 2024.1 (Student Edition) 22-Jan-25 14:57:32
Deform: SC1:DEFAULT, A1:Static subcase, Displacements, Translational,
5.02+03
123456
1.26+00
1.17+00
1.09+00
1.01+00
9.22-01
8.38-01
7.54-01
6.70-01
5.87-01
5.03-01
4.19-01
3.35-01
2.51-01
1.68-01
8.38-02
0.
Y
X

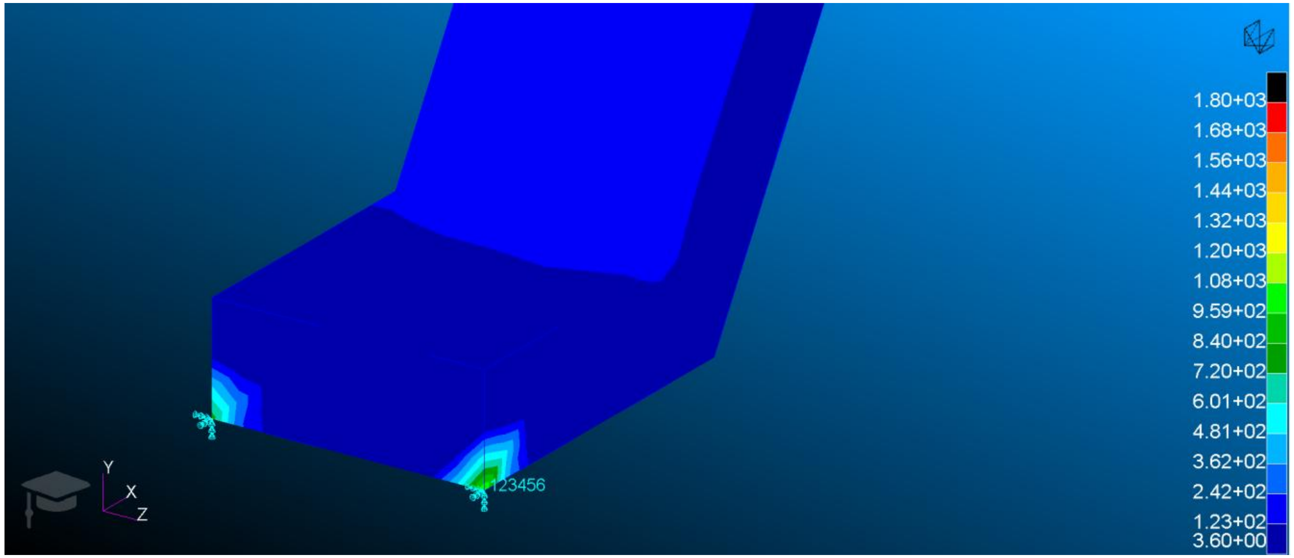
Patran 2024.1 (Student Edition) 22-Jan-25 14:57:32

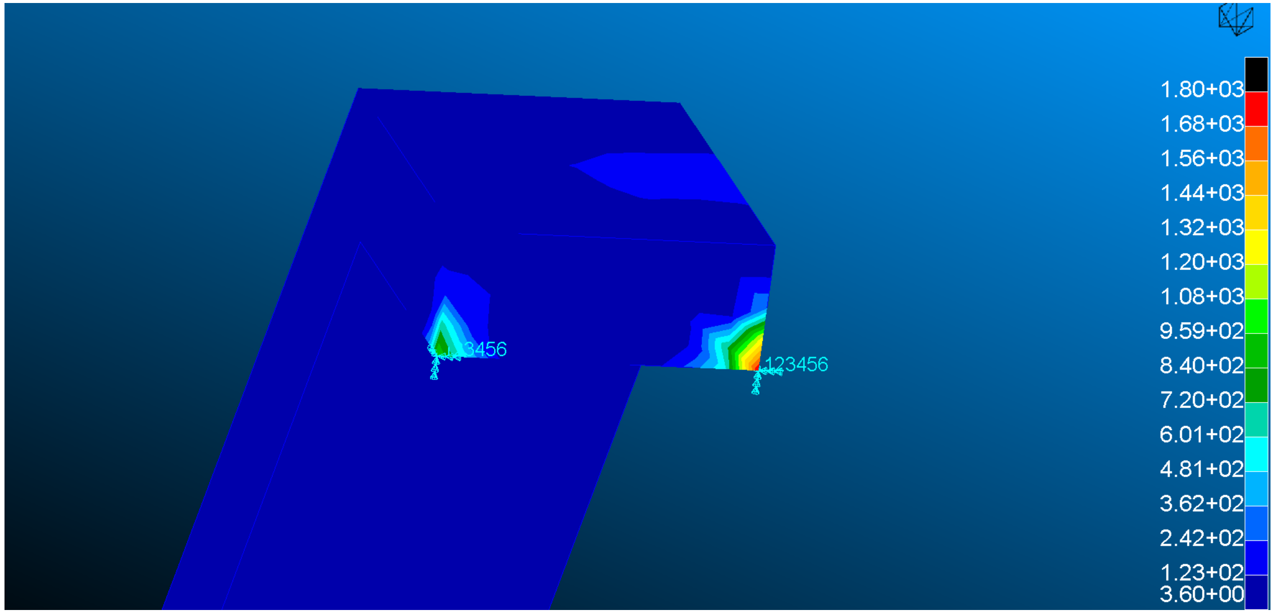
Deform: SC1:DEFAULT, A1:Static subcase, Displacements, Translational,





Bar Stress


1.80+03
1.68+03
1.56+03
1.44+03
1.32+03
1.20+03
1.08+03
9.59+02
8.40+02
7.20+02
6.01+02
4.81+02
3.62+02
2.42+02
1.23+02
3.60+00
123456
Y
X
Z


1.80+03
1.68+03
1.56+03
1.44+03
1.32+03
1.20+03
1.08+03
9.59+02
8.40+02
7.20+02
6.01+02
4.81+02
3.62+02
2.42+02
1.23+02
3.60+00
3456
123456

Conclusion

This project focused on studying the payload adapter and the Ariane-type satellite, and I have shared all the work I completed throughout the project. Although I could not finish the third part, which involved studying the payload task of the satellite, I firmly believe that with more time, I would have been able to complete it. The scope of this project was vast, and as the sole contributor, I had to manage both the analysis and the report writing, which were further complicated by some simulation errors that required me to redo several parts of the work. Nevertheless, I can confidently state that I have fully understood everything needed for this project. This experience allowed me to master the required software and thoroughly grasp the principles underlying the analyses performed.

The primary goal of the project was to study the satellite's frequencies rather than directly calculate the maximum load it can support. *My professor Walid Larbi raised an intriguing question: if we know the maximum load a satellite can bear, why do we need to conduct all these detailed studies instead of simply applying the critical mass and launching the structure?*

My answer lies in the critical importance of understanding the satellite's dynamic behaviour under various conditions. Determining the maximum load without considering factors such as frequency response, vibrations, deformation patterns, and the impact of different load distributions could lead to structural failures during the satellite's operational lifespan. For example, resonances at certain frequencies could cause severe damage, even if the satellite's load remains below the critical threshold. Additionally, factors like the satellite's operating environment, mission duration, and the need to ensure structural integrity under dynamic conditions make these studies indispensable. It is not only about ensuring the satellite can bear the weight but also about guaranteeing its performance, reliability, and safety throughout its mission.

In conclusion, this project was a challenging yet enriching experience. Despite its complexity and the setbacks, I encountered, I am proud of what I achieved and the knowledge I gained. It reinforced my understanding of structural dynamics, frequency analysis, and deformation behaviour while also sharpening my problem-solving skills and proficiency with simulation tools. Although I could not complete the entire project, the work I did represents a significant accomplishment, particularly considering it was carried out entirely on my own. This process has been a valuable learning journey that has equipped me with deeper technical expertise and a strong foundation for future endeavours.