

MASTER OF SCIENCE IN COMPUTATIONAL MECHANICS

UNIVERSIDAD POLITÉCNICA DE CATALUÑA

Subject: Computational Structural Mechanics and dynamics

Student: ANTONIO SOLITO

Practice 5

Exercise 1: Plane Frame

As a problem type for the task we used the compass fem with:

- 3D simulation dimension;
- Element type: beams and shells (also for the 2° exercises);
- Dynamic modal analysis;
- Linear-elastic material model;
- Linear geometry model.

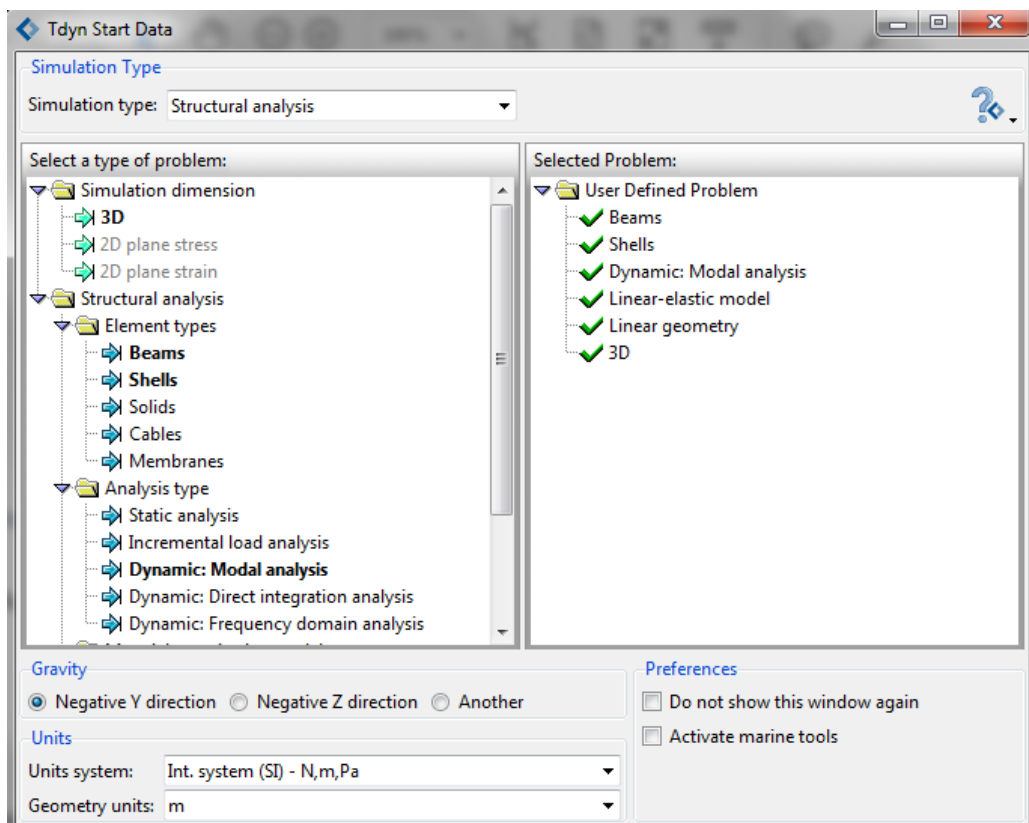


Figure 1 – Simulation Type

Afterwards we created the geometry in Tdyn. As constraints, we constrain the two nodes at the bottom in all directions and the rest of the nodes only in z-direction and the rotations around the x- and the y-axes because we have a plane frame.

The cross-section and the material of the beams are assigned according to the information given in next figure.

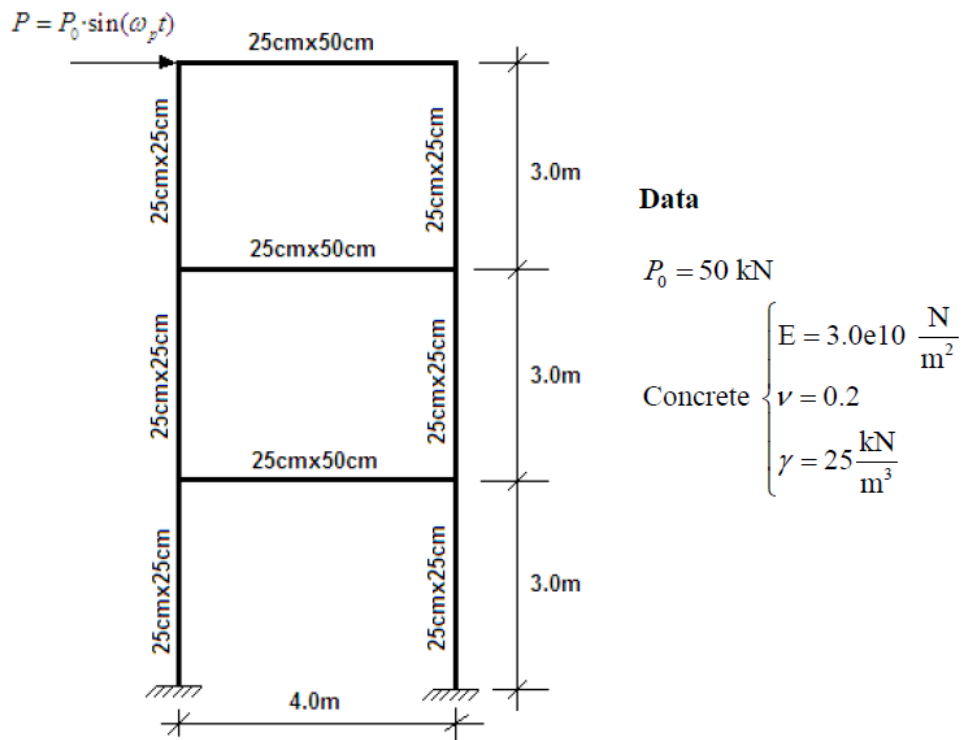


Figure 2 - Geometry and data

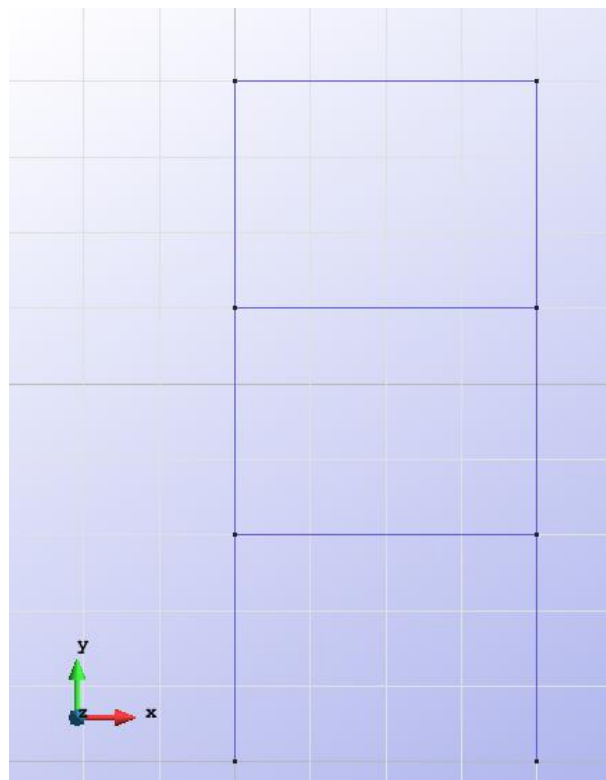


Figure 3 - Geometry structure in Tdyn

At first we calculate with the dynamic analysis only the natural frequencies, and decide afterwards which modal we must consider depending on the limiting equation where ω_p is

the frequency of the load and ω_0 is the maximum natural frequency which we must consider.

The condition to perform a dynamic analysis is:

$$\omega_p > \frac{1}{4} \omega_0$$

After creating the mesh we did the first calculation which gives us the natural frequencies of the structure.

The main natural frequency ω_1 is **4,292 Hz**.

The first 10 natural frequencies and modal mass are:

Mode	Freq [Hz]	Mass_x [Kg]	Mass_x [%]	Mass_y [Kg]	Mass_y [%]	Mass_z [Kg]	Mass_z [%]
1	4.292	5647	84.3840	1.557e-028	0.0000	3.664e-029	0.0000
2	12.74	586.3	8.7612	4.282e-029	0.0000	6.715e-028	0.0000
3	19.89	105.3	1.5731	2.155e-029	0.0000	1.575e-029	0.0000
4	43.93	1.591e-028	0.0000	4071	60.8379	5.529e-028	0.0000
5	48.89	1.395e-028	0.0000	353.1	5.2772	8.999e-028	0.0000
6	53.31	6.312e-029	0.0000	682.8	10.2035	2.134e-028	0.0000
7	55.8	3.906e-033	0.0000	1.508e-027	0.0000	385.8	5.7647
8	55.8	8.3e-032	0.0000	3.795e-027	0.0000	1158	17.3089
9	55.8	2.965e-033	0.0000	2.718e-027	0.0000	1085	16.2122
10	75.26	0.3601	0.0054	4.05e-027	0.0000	4.045e-028	0.0000

Figure 4 -The first 10 natural frequencies of the structure

All this implies that the load frequency varies between:

- $\omega_p = 3,219 \text{ Hz} (=0,75 \omega_0)$;
- $\omega_p = 4,292 \text{ Hz} (=1,00 \omega_0)$;
- $\omega_p = 5,365 \text{ Hz} (=1,25 \omega_0)$.

To check the number of eigenmodes we have to consider that we use $\omega_p = 5,365 \text{ Hz}$ which allows us to use all natural frequencies which are smaller than 21,46 Hz. This means we will consider the first three eigenmodes in our calculation.

To calculate the structure, we define the self-weight load and a dynamic punctual load of 1 second at the top left corner of the frame in x-direction.

The parameters of the load are shown in the next figure.

Figure 5 – Apply Punctual load

Function variables

Function on time Sinusoidal load

Sinusoidal load

Amplitude (A):

Frequency (f): Hz

Phase angle (Φ): deg

Initial time (t0): s

End time (t1): s

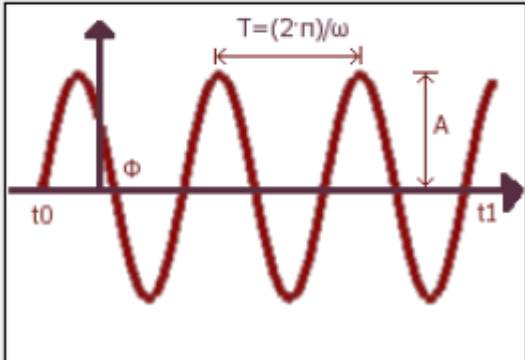


Figure 6 – Function variables for the first $\omega_p = 3,219$

To choose a time-step for the calculation we divide the period time of the maximum frequency by 20:

$$\Delta t = \frac{T}{20} = \frac{1/\omega_{max}}{20} = \frac{1/5,365}{20} = 0,0093s$$

To get the required number of time step to show 1 s in total we divide 1s by the time step:

$$\frac{1s}{\Delta t} = \frac{1}{0,0093} = 107,5 \approx 108$$

MODAL ANALYSIS

Type: Modal analysis

Only calculate natural freqs.

Δt: s

Number of steps:

Type of modal analysis: Number of modes

Number of modes:

Compute all modes

Range of modes, min. value (Hz):

Range of modes, max. value (Hz):

Figure 7 - Dynamic analysis data for the modal analysis method

In the nexts figures there are the results with the differents load frequency useds.

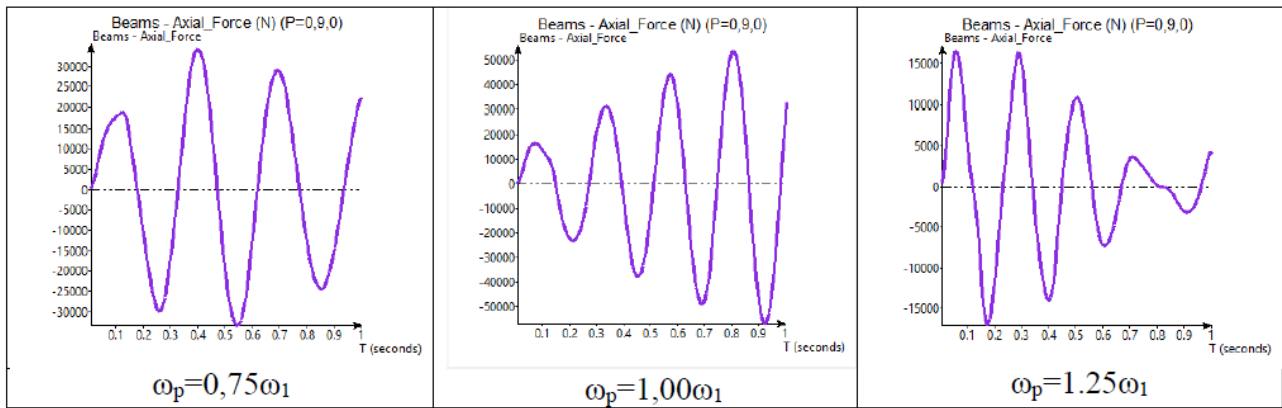


Figure 8 – Axial Force

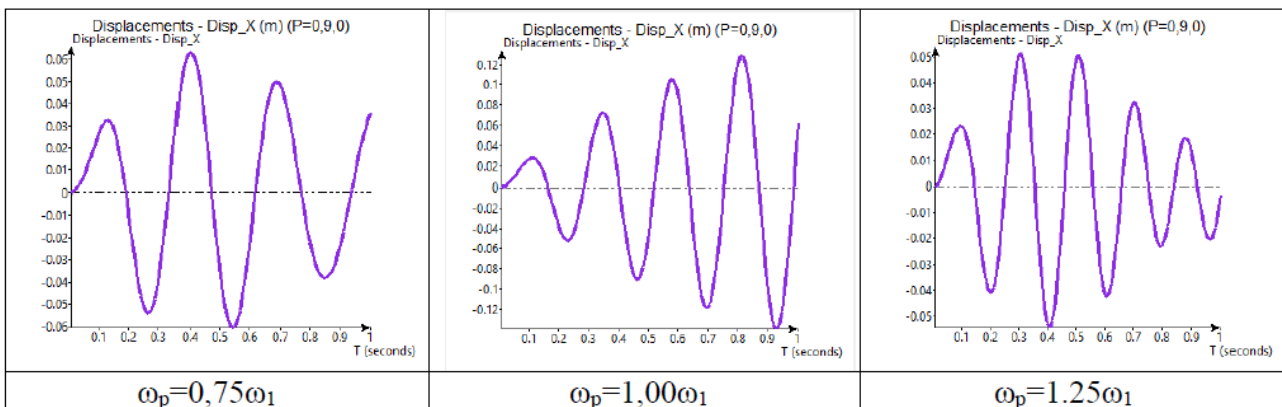


Figure 9 – Displacements in x direction

In figure 8 and figure 9 it can be observed that the maximum forces and displacements occur in the case $\omega_p = 1,00\omega_1$.

This is the resonance case because the frequency of the load is equal to the main natural frequency of the structure. This is the worse cause from a structural point of view because the forces and displacements are statically growing along the time. In the other two cases where the loading frequency is smaller or bigger than the natural frequency the structure has smaller forces and displacements.

The best case for the structure is a loading frequency which is bigger than the natural frequency because in this case the forces and displacements reduce more with the time than when the frequency is smaller than the natural frequency.

DIRECT INTEGRATION

With the direct integration method, the dynamic analysis is changed as following:

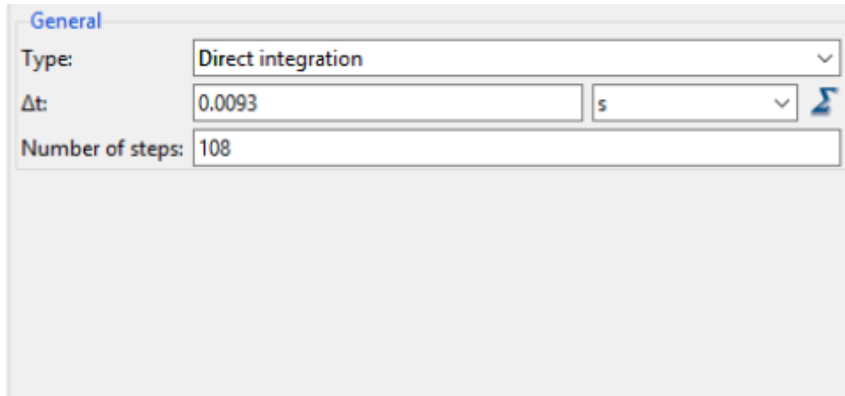


Figure 10 - Dynamic analysis data for the direct integration method

In the nexts figures there are the results with the differents load frequency useds.

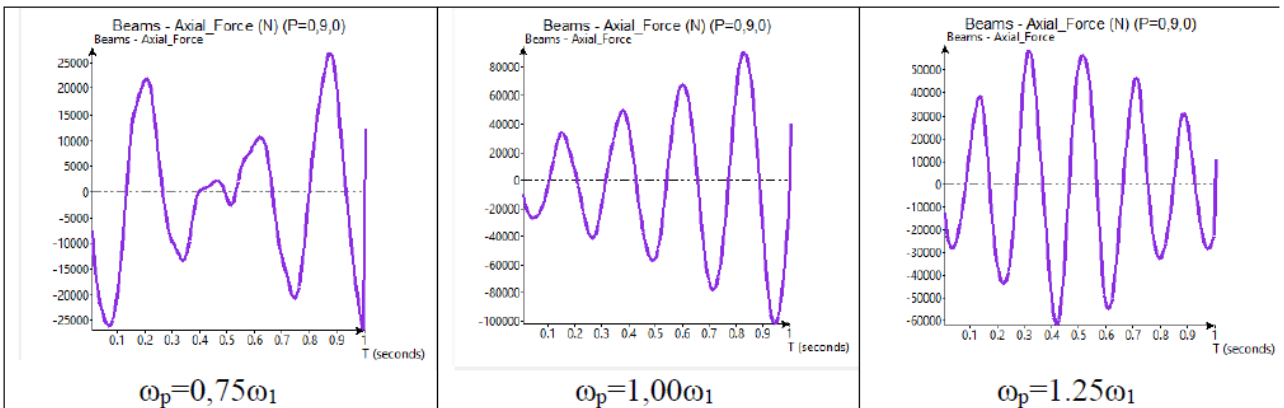


Figure 11 – Axial Force

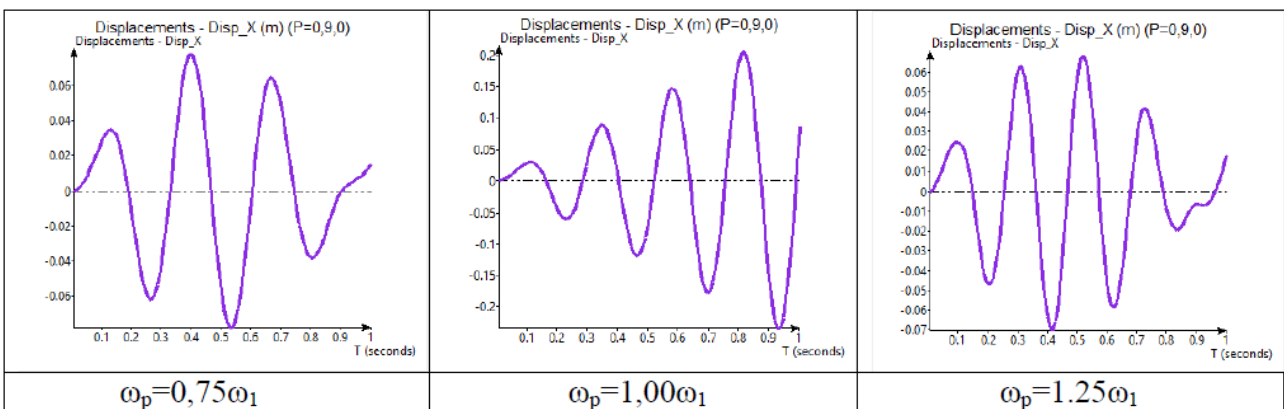


Figure 12 – Displacements in x direction

The principal features that were observed for the modal analysis can be seen for the direct integration as well (worst case=resonance case, best case=bigger frequency). But the results are quite different for the values.

The results of the direct integration have bigger values for the same calculation even if the form of the graphs are similar. But an advantage of the direct method is much calculation time in contrast to the modal analysis.

The only huge difference is for axial force with the lowest frequency (also changing the calculation with a smaller time step does not change the result).

Exercise 2: Spatial shell

To model the geometry we choose isotropic shells in Tdyn. As a problem type for the task we used the compass fem with:

- 3D simulation dimension;
- Element type: beams and shells (also for the 1° exercises);
- Dynamic modal analysis;
- Linear-elastic material model;
- Linear geometry model.

We assumed that the structure is clamped at its base line and does not have other constraint. For the mesh we used quadrilateral elements with 4 nodes. The geometry and the mesh can be seen in nexts figures.

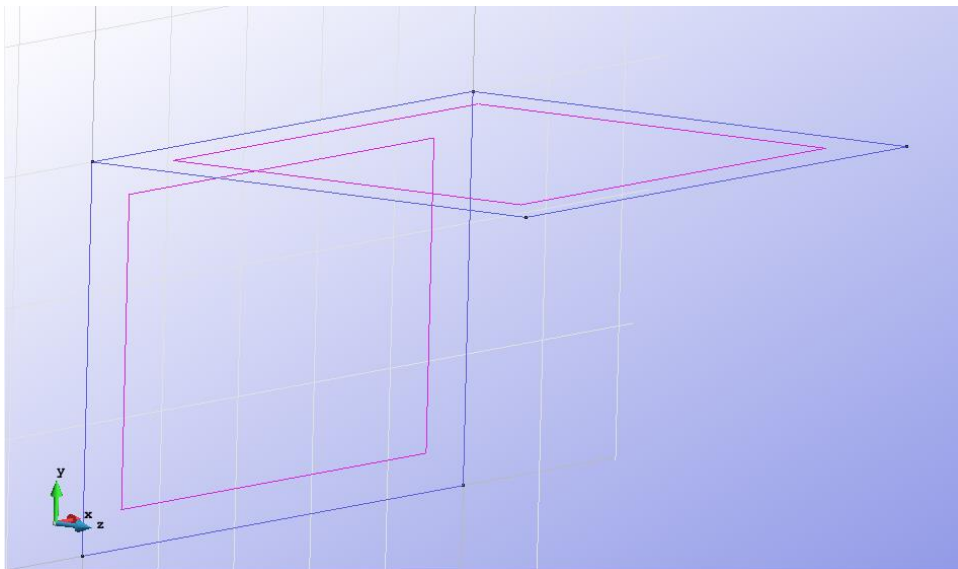


Figure 13 – Geometry structure in Tdyn

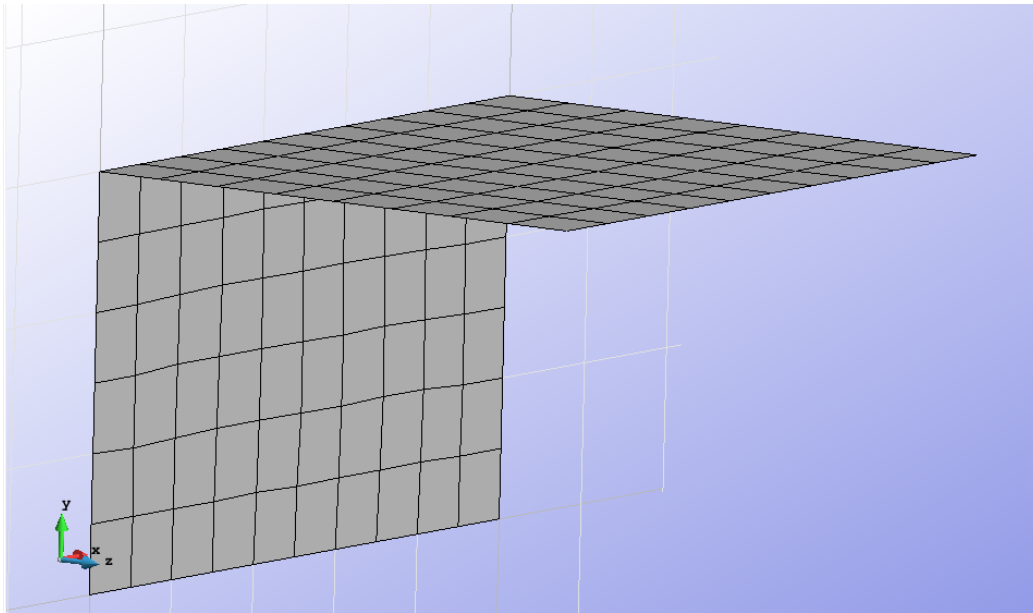


Figure 14 – Meshed structure in Tdyn

After creating the mesh we did the first calculation which gives us the natural frequencies of the structure.

The main natural frequency ω_1 is **4,522 Hz**.

The first 10 natural frequencies and modal mass are:

Mode	Freq [Hz]	Mass_x [Kg]	Mass_x [%]	Mass_y [Kg]	Mass_y [%]	Mass_z [Kg]	Mass_z [%]
1	4.522	6.229e-023	0.0000	7942	29.6708	6035	22.5443
2	8.681	7742	28.9234	1.5e-022	0.0000	3.303e-023	0.0000
3	12.65	7.013e-023	0.0000	2479	9.2629	1.581e+004	59.0465
4	19.94	231.1	0.8633	1.404e-023	0.0000	1.305e-023	0.0000
5	51.89	1.855e-026	0.0000	2088	7.8000	473.9	1.7703
6	57.14	1.238e-026	0.0000	715.6	2.6734	150.6	0.5624
7	72.77	322.6	1.2054	2.99e-025	0.0000	1.298e-028	0.0000
8	84.27	1.268e+004	47.3585	1.478e-026	0.0000	2.475e-026	0.0000
9	106.8	4.176e-026	0.0000	91.6	0.3422	1869	6.9834
10	116.9	844.1	3.1536	2.537e-027	0.0000	1.74e-025	0.0000

Figure 15 – The first 10 natural frequencies of the structure

All this implies that the load frequency varies between:

- $\omega_p = 3,392$ Hz (=0,75 ω_0);
- $\omega_p = 4,522$ Hz (=1,00 ω_0);
- $\omega_p = 5,653$ Hz (=1,25 ω_0).

This is the condition to perform a dynamic analysis:

$$\omega_p > \frac{1}{4} \omega_0$$

To check the number of eigenmodes we have to consider that we use $\omega_p = 5,653$ Hz which allows us to use all natural frequencies which are smaller than 22,612 Hz. This means we will consider the first four eigenmodes in our calculation.

To choose a time-step for the calculation we divide the period time of the maximum frequency by 20:

$$\Delta t = \frac{T}{20} = \frac{1/\omega_{max}}{20} = \frac{1/5,653}{20} = 0,0088s$$

To get the required number of time step to show 1 s in total we divide 1s by the time step:

$$\frac{1s}{\Delta t} = \frac{1}{0,0088} = 113,6 \approx 114$$

We apply the self-weight load and the dynamical sinus pressure load on the top surface. The frequency varies for the 3 different loads we apply. The pressure load has an amplitude of 50 kN/m² and ends at 1 second.

Apply Pressure load

Factor: 1.0

Load type: global

X pressure: 0.0 Pa

Y pressure: 50 kN/m²

Z pressure: 0.0 Pa

Group: [] Select

Figure 16 – Apply Pressure load

Function on time Sinusoidal load View

Sinusoidal load

Amplitude (A): 1.0

Frequency (f): 3.392 Hz

Phase angle (Φ): 0.0 deg

Initial time (t0): 0.0 s

End time (t1): 1 s

$f(t) = A \cdot \sin(2 \cdot \pi \cdot f \cdot t + \Phi)$

Graph showing a sinusoidal wave with amplitude A, phase angle Φ, and period T = (2·π)/ω.

Figure 17 – Function variables for the first $\omega_p = 3,392$

RESULTS

The results for the different load frequencies are shown in the following figures. We decided to compare the displacement at the outer edge of the shell cantilever (green circle in the next figure).

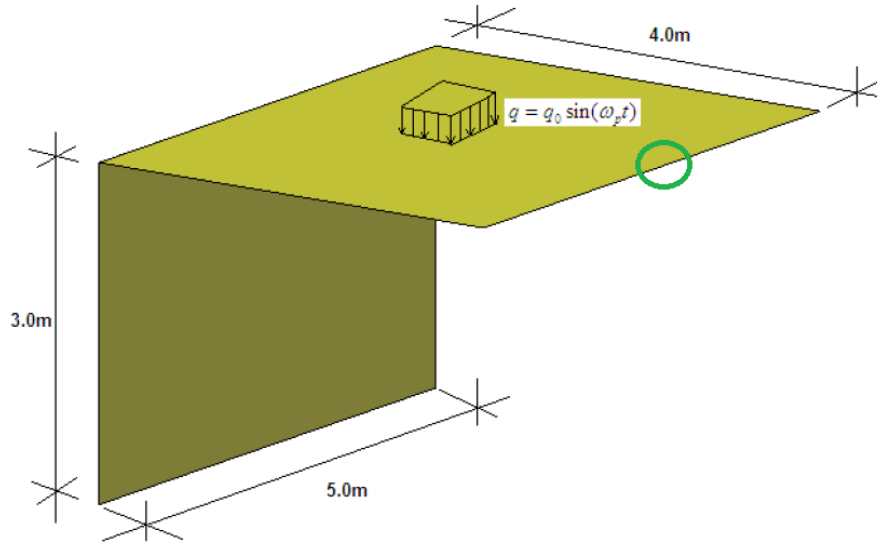


Figure 18 – Structure made

MODAL ANALYSIS

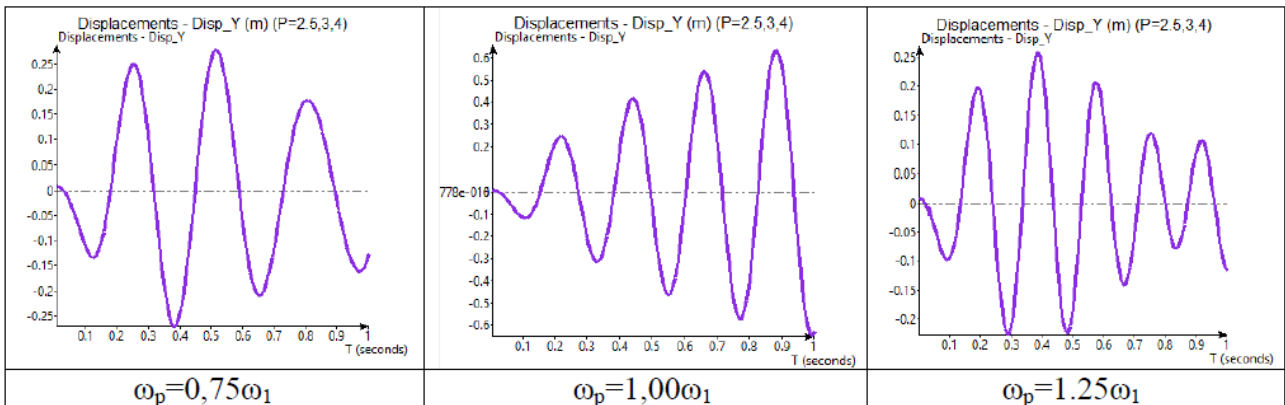


Figure 19 – Displacements in y direction

DIRECT INTEGRATION

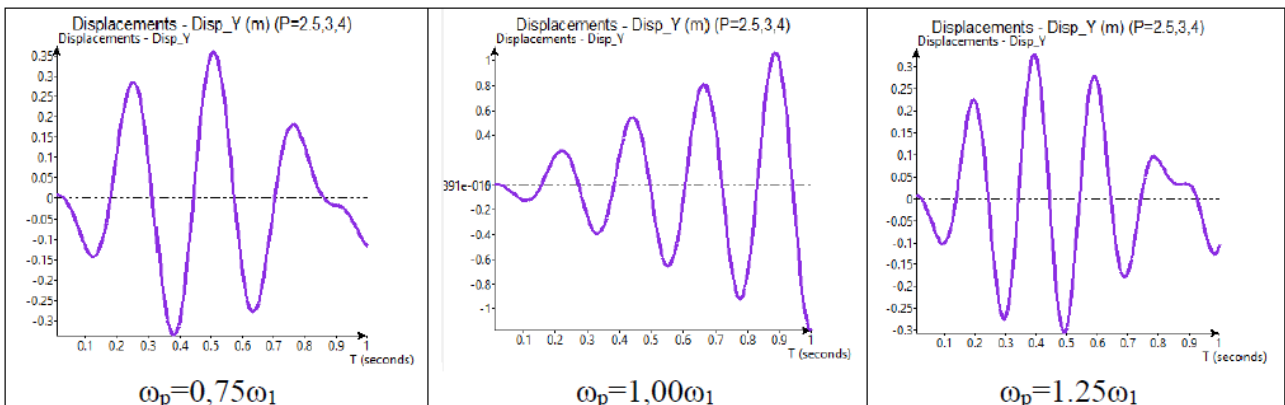


Figure 20 – Displacements in y direction

Like in the first part of the task, that the displacement grows the most when the load frequency is equal to the first eigenfrequency of the structure ($\omega_p=1.0\omega_1$). It is called the resonance case. And the best case for the structure is when the loading frequency is higher than the first eigenfrequency of the structure ($\omega_p=1.25\omega_1$), because in this case the displacement will decrease by time.

The results of the modal analysis and the direct method are quite different again, regarding to their values, while their appearance is still looking similar. The direct method is again leading to higher values on the one hand, but on the other hand it is beneficial for the computational time.

We can state that it is important to study the frequencies of the dynamic loads of a structure and compare them to the eigenfrequencies. Because when they are getting close or even the same it will result to very high and increasing stresses and displacements of the structure. It is always better to choose or make the frequencies different to avoid this case. The best case for the structural behavior of the structure is a higher loading frequency than the eigenfrequency.