Computational Structural Mechanics and Dynamics

GiD Homework 5

Prasad ADHAV

Contents
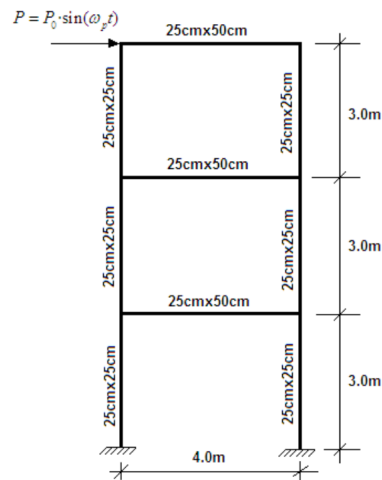
1 Exercise # 1: Plane frame

Figure 1.1: Exercise #1

1.1 Purpose of the exercise

In this first exercise we aim to calculate the natural frequencies and modes of the plane frame in Figure 1.1. Lately we will analyze the response of the frame when a point sinusoidal load is considered.

1.2 Free vibration: Natural frequencies.

1.2.1 Preprocessing

Geometry

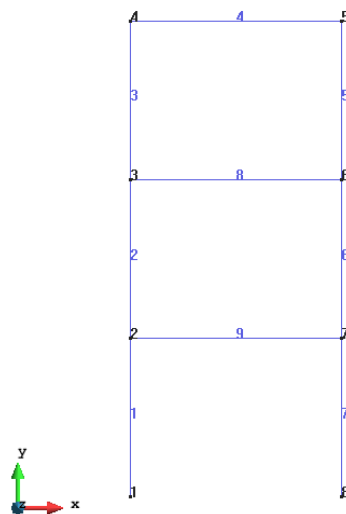First, we define the geometry using the RamSeries sketcher tool.

Figure 1.2: Geometry of plane frame for the analysis.

Data

Problem type

Once the geometry is defined, we can now choose the type of problem that must be solved using *RamSeries*. As we first want to obtain the natural frequencies of the structure, we select

Structural Analysis ⇒ *Analysis type: Dynamic analysis.*

Further, we first need to set

Dynamic Analysis data/ General / Type: modal analysis ⇒ *Only calculate natural freqs: 1*

Lately, we will change this condition to 0 so that we can simulate the action of the considered point load.

Boundary Conditions

The type of boundary conditions that are considered in this example are the following (for obtaining the natural frequencies):

- Constraints / Fixed constraints: points #1 and #8 in Figure 1.2 have their movement completely restricted. On the other hand, points # 2, 3, 4, 5, 6 and 7 cannot move in the z direction and can only rotate about the z axis, so that the problem occurs only in the x-y plane.

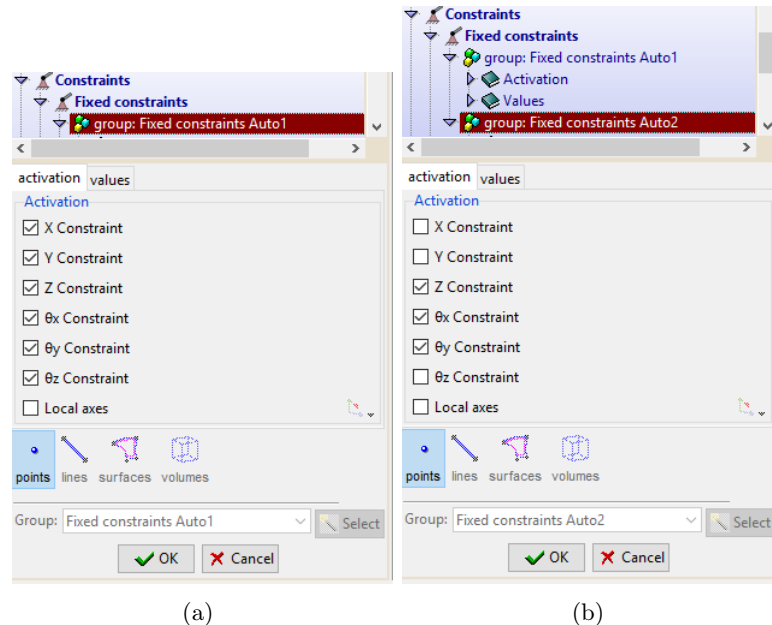(a)   (b)

Figure 1.3: (a) Fixed constraints on the base points of the geometry (b) Point constraints on the remaining nodes of the plane frame, to ensure a plane problem.

Material

The plane frame is made of a material with the following mechanical characteristics:

$$E = 3.0e10 \ N/m^2 \quad ; \quad \nu = 0.2 \quad ; \quad \gamma = 25000 \ N/m^3$$

3

Now it is important to assign the rectangular beams to the correspondent part of the frame. For the vertical ones we have that,
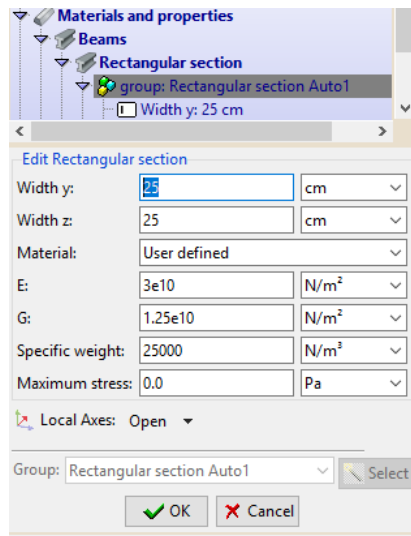
(a)

Figure 1.4: Definition of the beams considered for the vertical bars.
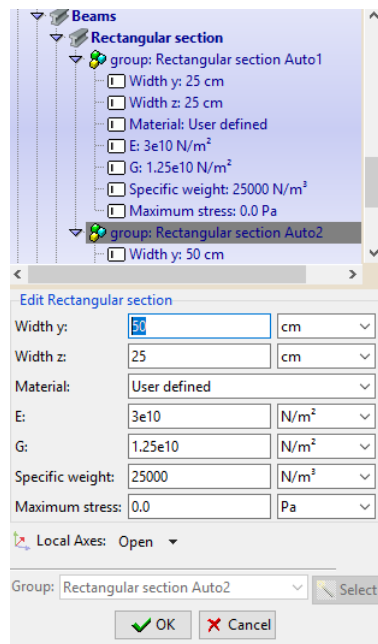
and for the horizontal ones,

Figure 1.5: Definition of the beams considered for the horizontal bars.

Mesh

We consider the following mesh for the simulation, with default data. Note that the mesh is composed of "line" elements.
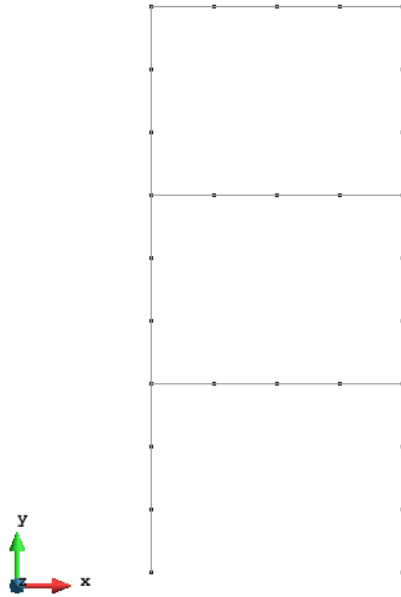
Figure 1.6: Mesh for problem # 1.

1.2.2 Results: modes of the plane frame

In this section we present now the first five modes of vibration of the plane frame.
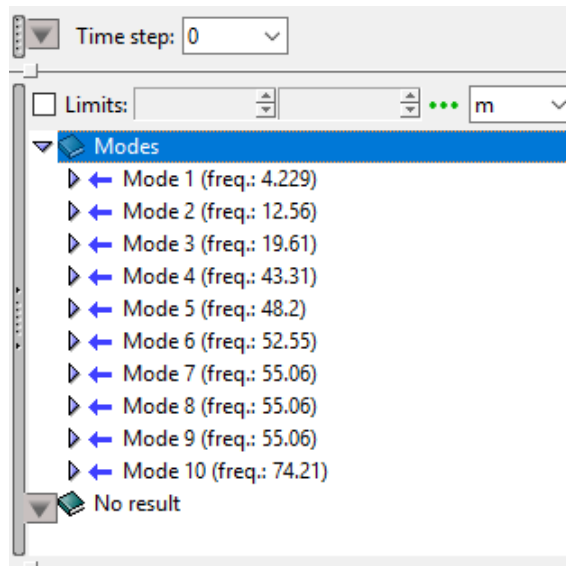

Figure 1.7: List containing the first ten modes of vibration with their natural frequencies.
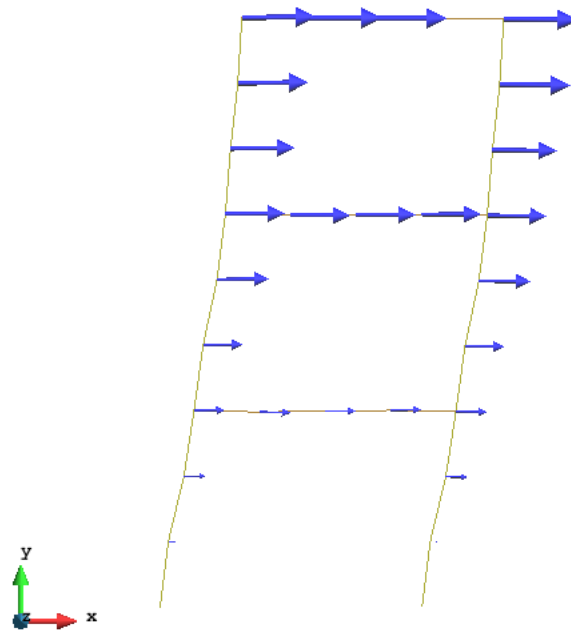
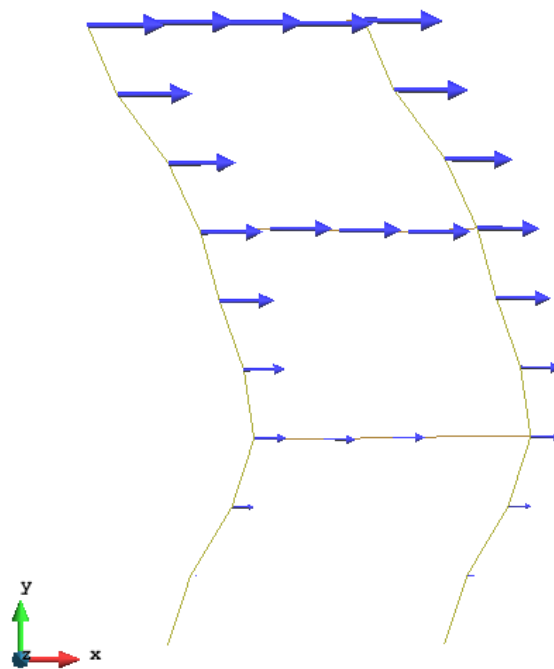Figure 1.8: Mode 1. Frequency = 4.229
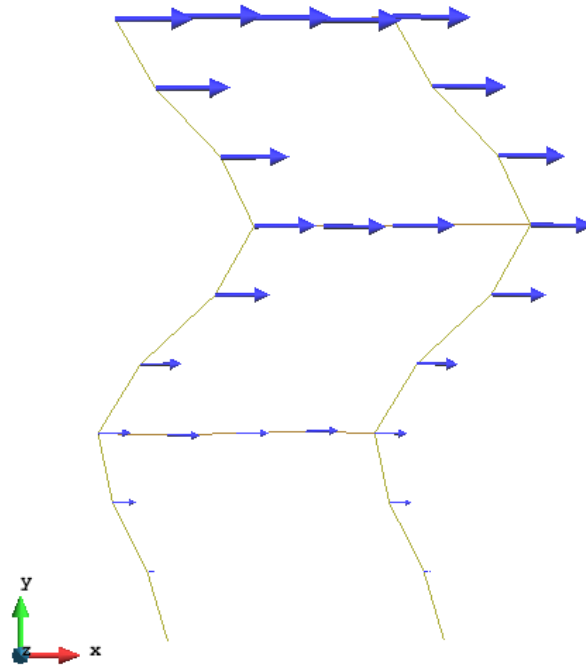

Figure 1.9: Mode 2. Frequency = 12.56

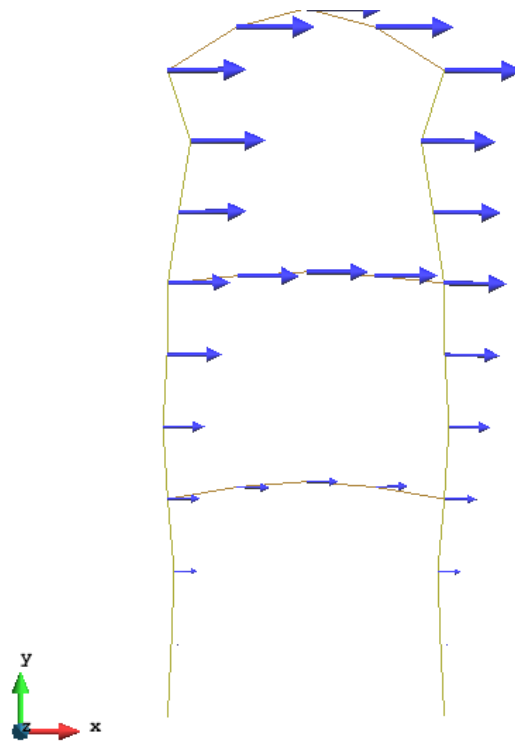Figure 1.10: Mode 3. Frequency = 19.61

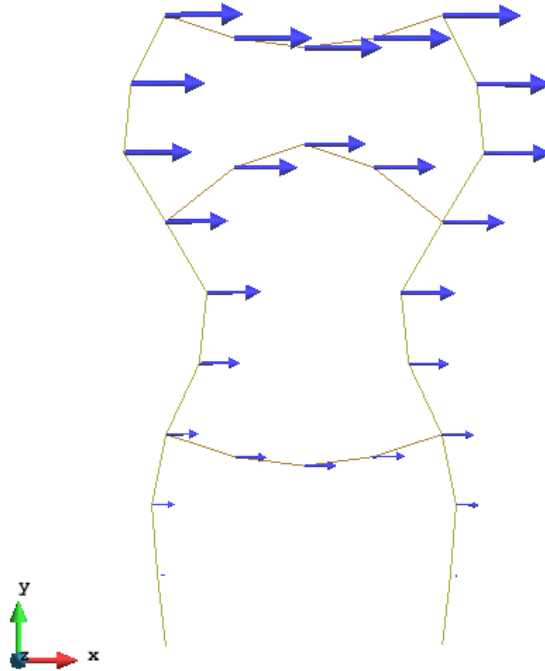Figure 1.11: Mode 4. Frequency = 43.31

Figure 1.12: Mode 5. Frequency $= 48.2$

1.3   Analysis with point load

Now we are going to analyze the dynamic response of the structure under the effect of a point load applied on the point $\# 4$ in Figure 1.2. Prior to that, we need to now change the analysis and set *Only calculate natural freqs* to 0.

The dynamic regime is defined by taking into account that

$$\omega_p^{max} > \frac{1}{4}\omega_0 \tag{1}$$

In our case of concern $\omega_p^{max} = 1.25\omega_1 = 5.28625$ where the value of ω_1 is taken from the list of natural frequencies presented above. Thus $\omega_0 < 21.145$. Therefore, having in mind Fig 1.7, we observe that the first three natural frequencies are the ones that need to be considered in the complete dynamic analysis. Further than this, the analysis is strictly static.

Now we also need to define a time step and a total time for the simulation. For this case, we choose to simulate $t = 1$ s as it is sufficient to obtain a descriptive result of the behavior of the structure. The value of the time step needs to be a proper one so that it enables to capture the oscillations but without increasing the computational cost of the analysis. Taking into account the method that *RamSeries* uses for the computations, the time step can be computed as

$$\Delta t = \frac{T}{20} = \frac{1}{20\omega_0} = 2.55E - 3 \ s$$

where $\omega_0 = 19.6$ is the highest value of the frequencies that satisfies the dynamic condition (1). Therefore we have that for this analysis,

- Dynamics analysis data

    - Only calculate natural freqs: 0

8

- $\Delta t = 2.55E - 3$
- Steps $= 392$

Figure 1.13 displays the definition of the load for this case, which is applied at point $\# 4$ in Figure 1.2.
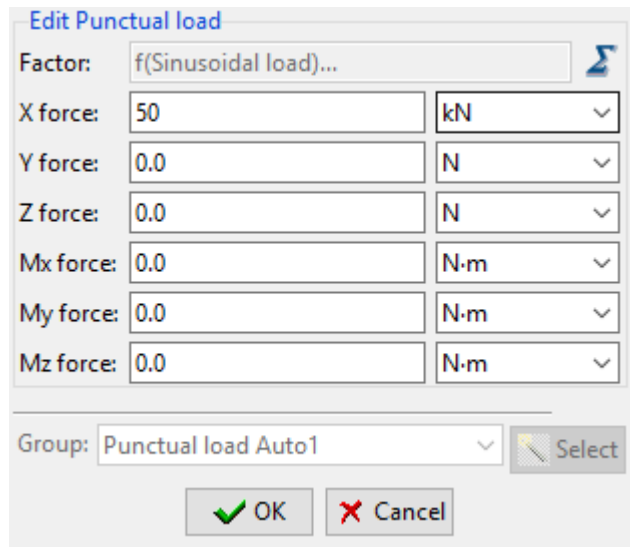

Figure 1.13: Definition of the point load for the dynamic analysis of the plane frame.

Once we click on the summation symbol a new windows pops-up, as appears in Figure 1.14 down below. Here we can choose a sinusoidal load an introduce the needed parameters. In the field frequency, we would be introducing the different values of the frequencies, say, $0.75\omega_1$, ω_1 and $1.25\omega_1$, respectively.
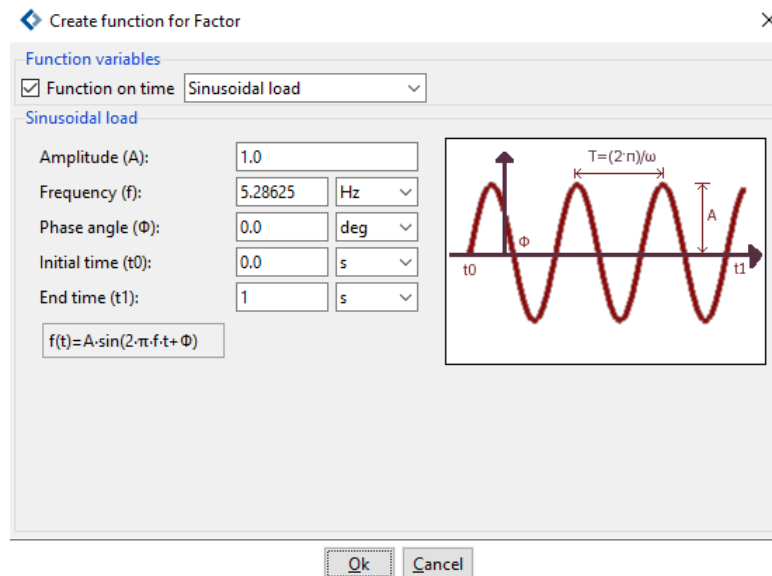

Figure 1.14: Sinusoidal load definition.

1.3.1 Results

In this section we include several figures which show a plot of horizontal displacement versus time of the node in which the load was applied.

Figure 1.15 displays the result when considering $0.75\omega_1$. As we can see, this image shows the sinusoidal behavior of the point due to the load. As the value of the frequency is not coincident with that of the natural frequency of the structure, the displacements are kept within acceptable values. This could be expected a priori since the value of the frequency of the force for this case does not coincide (or is sufficiently close) to the natural frequency of the structure.
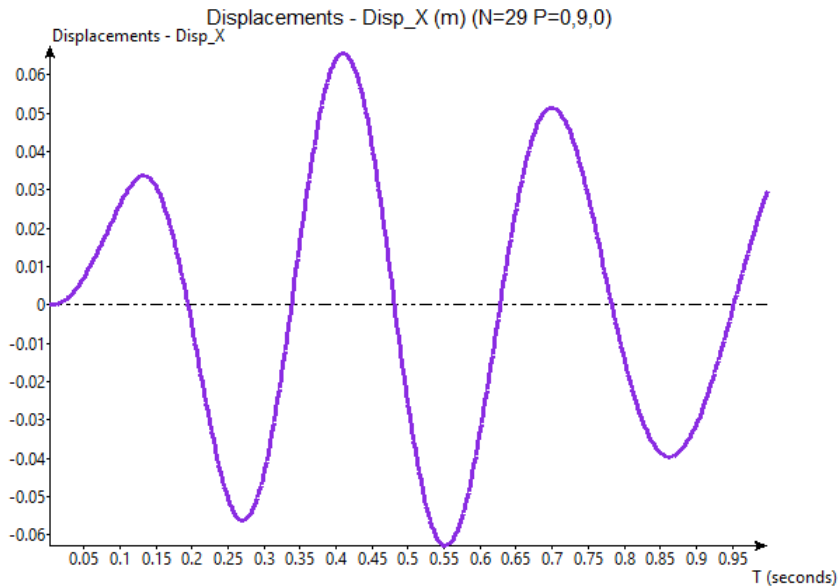

Figure 1.15: Horizontal displacements vs. time for $0.75\omega_1 = 3.171$.

Figure down below shows the result when the frequency of the load is exactly the first natural frequency of the plane frame. As expected, the displacements start to dramatically increase without control. This behavior is the so-called resonance phenomena, during which the frame's oscillations are amplified. This could lead to the collapse of the structure.
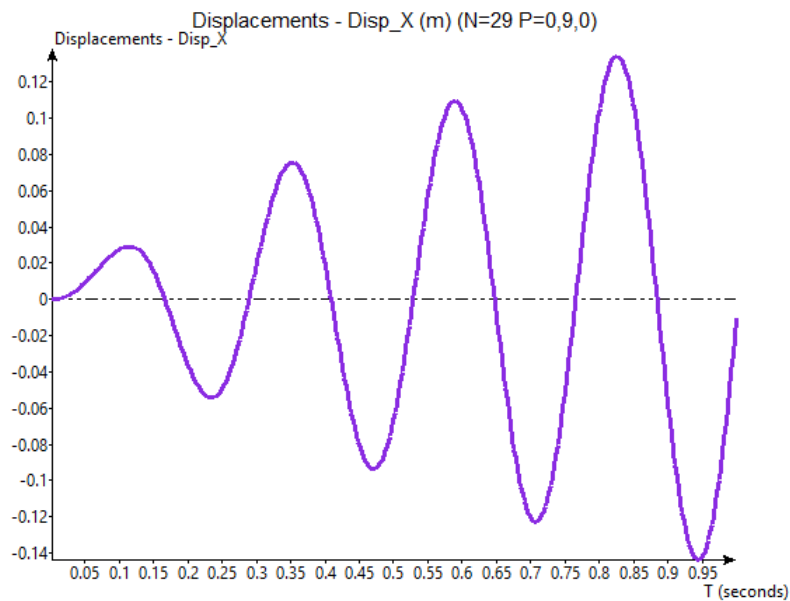

Figure 1.16: Horizontal displacements vs. time for $\omega_1 = 4.229$.

Finally, for the case of $1.25\omega_1$ the graph is somewhat similar to that obtained for the first

case. The frequency of the load is not sufficiently close to the natural frequency of the structure and therefore the displacements are damped and do not increase dramatically.
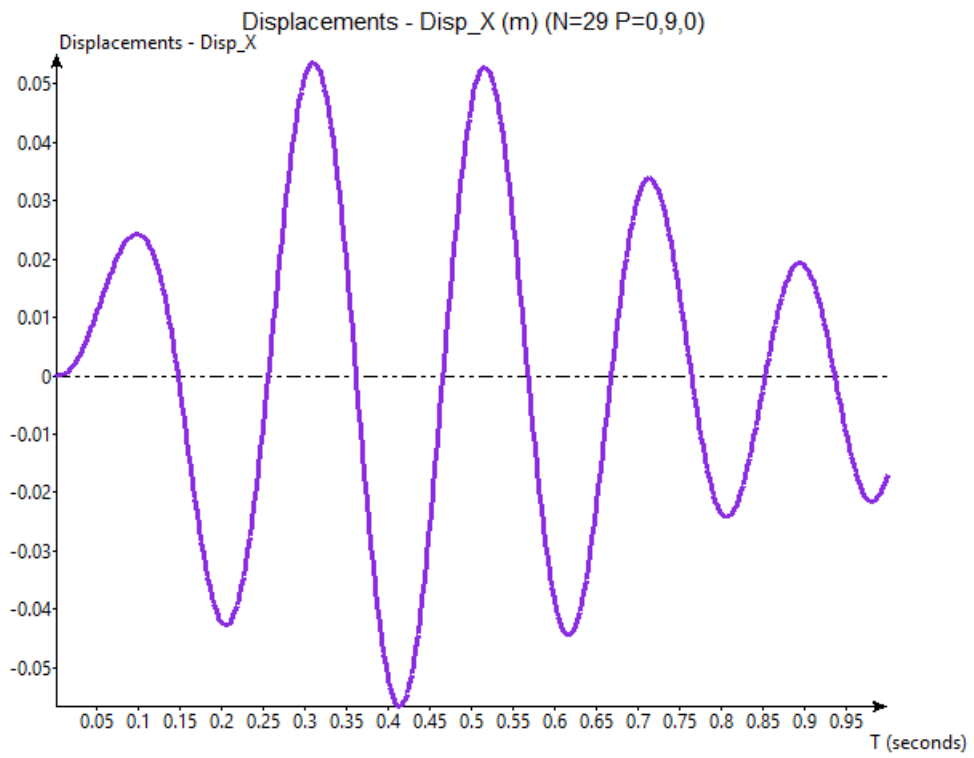

Figure 1.17: Horizontal displacements vs. time for $1.25\omega_1 = 5.2863$.
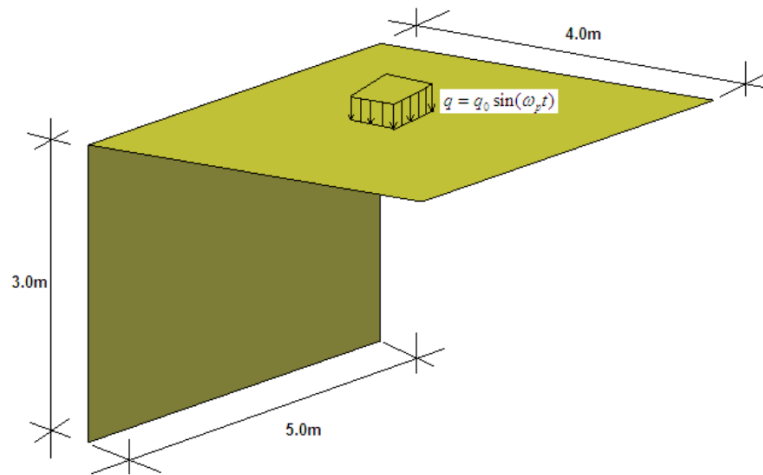
2 Exercise # 2: Spatial shell

Figure 2.1: Exercise #1

2.1 Purpose of the exercise

As in the first problem we aim to calculate the natural frequencies and modes of the structure in Figure 2.1 and also analyze its response after applying a sinusoidal distributed force.

2.2 Free vibration: Natural frequencies.

2.2.1 Preprocessing

Geometry

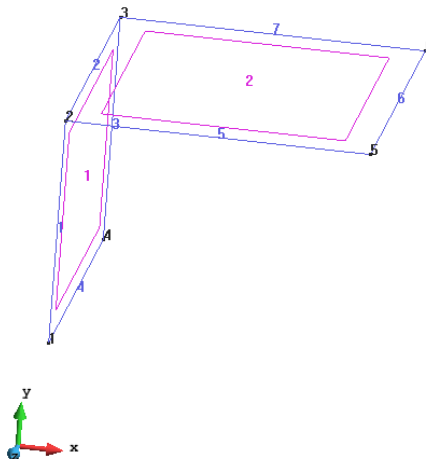First, we define the geometry using the *RamSeries* sketcher tool.

Figure 2.2: Geometry of the problem.

Data

Problem type

The procedure followed for obtaining the natural frequencies of this shell structure is the same as before.

$$Structural\ Analysis \Rightarrow Analysis\ type:\ Dynamic\ analysis.$$

Further, we first need to set

Dynamic Analysis data/ General / Type: modal analysis $\Rightarrow$ Only calculate natural freqs: 1

Boundary Conditions

The type of boundary conditions that are considered in this example are the following (for obtaining the natural frequencies):

- Constraints / Fixed constraints: line #4 in Figure 2.2 is completely fixed. No further restrictions have been applied.
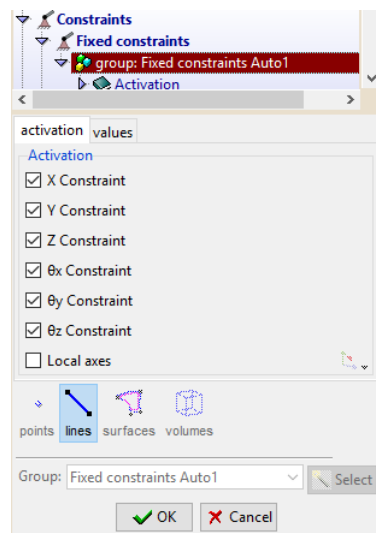

Figure 2.3: Boundary conditions in displacements

Material

The plane frame is made of a material with the following mechanical characteristics:

$$E = 3.0e10\ N/m^2 \quad ; \quad \nu = 0.2 \quad ; \quad \gamma = 25000\ N/m^3 \quad ; \quad t = 0.3\ m$$

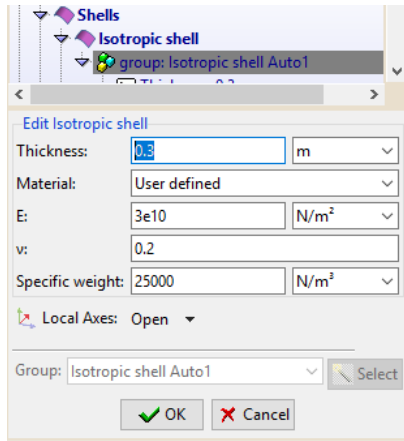Now we assign as material an isotropic shell, which also allows to give the shell its thickness.

Figure 2.4: Definition of the shell considered for the problem.

Mesh

We consider the following structured mesh for the simulation, with linear quadrilateral elements of size = 0.25
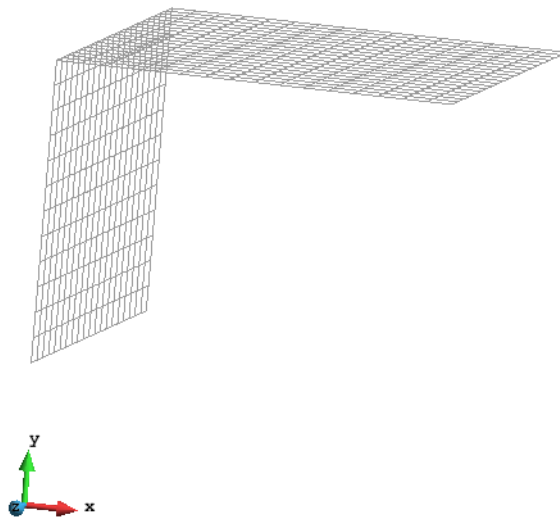

Figure 2.5: Mesh for problem # 2.

2.2.2 Results: modes of the plane frame

In this section we present now the first five modes of vibration of the plane frame.
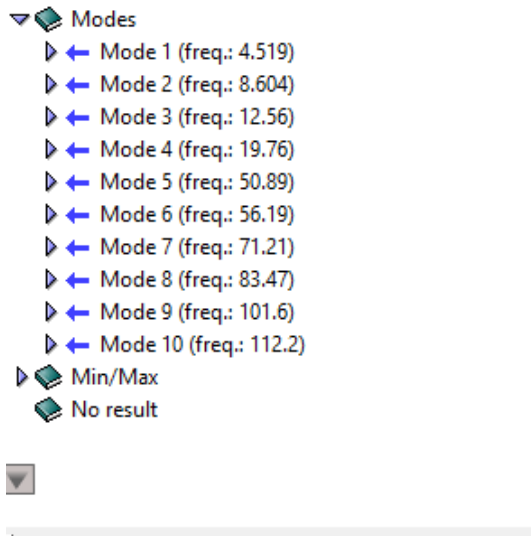
Figure 2.6: List containing the first ten modes of vibration with their natural frequencies.
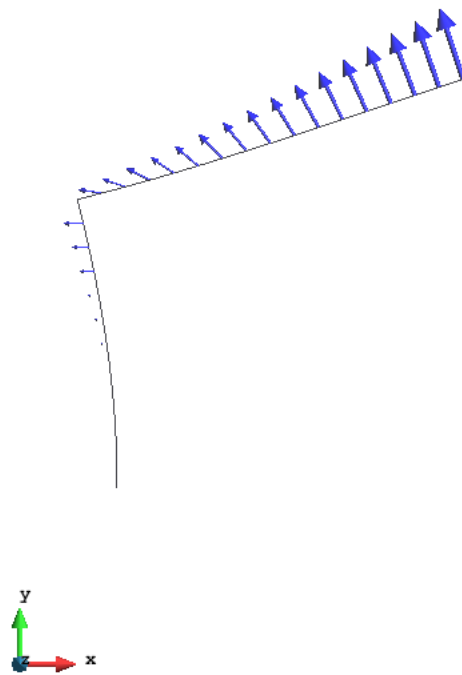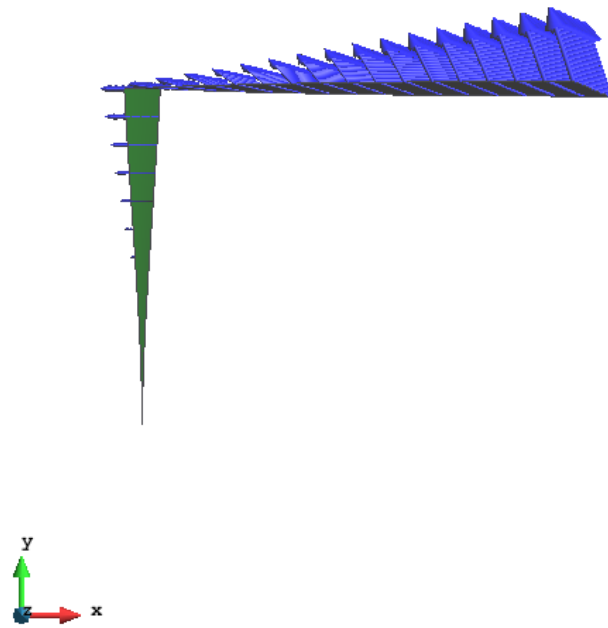

Figure 2.7: Mode 1. Frequency = 4.519

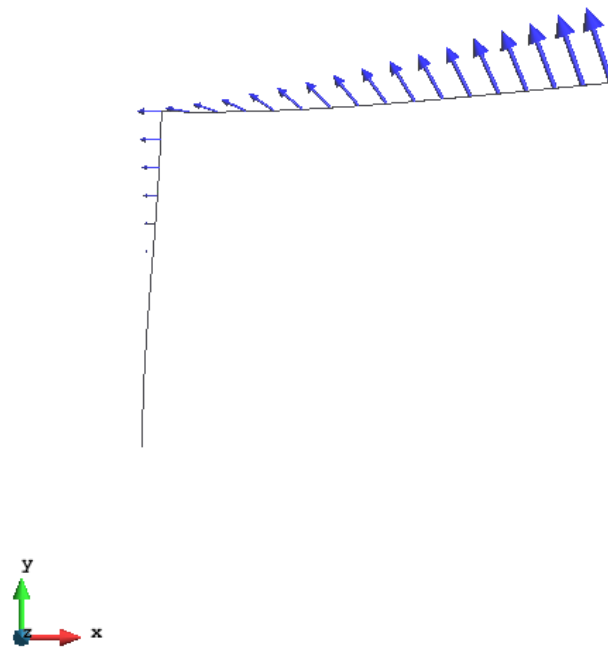Figure 2.8: Mode 2. Frequency = 8.604
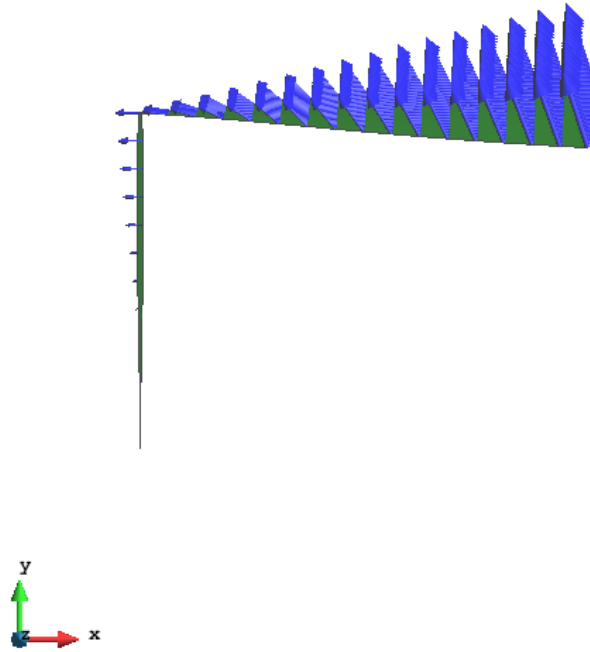

Figure 2.9: Mode 3. Frequency = 12.56

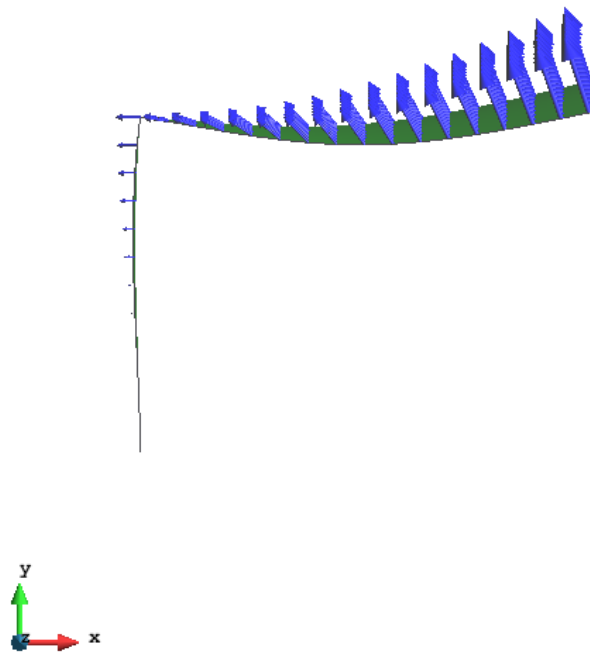Figure 2.10: Mode 4. Frequency = 19.76

Figure 2.11: Mode 5. Frequency = 50.89

2.3   Analysis with distributed load

Now we are going to analyze the dynamic response of the structure under the effect of a distributed sinusoidal pressure applied on the surface #2 in Figure 2.2. Prior to that, we need to now change the analysis and set *Only calculate natural freqs* to 0.

Again, the dynamic regime is defined by taking into account that

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

In this case $\omega_p^{max} = 1.25\omega_1 = 5.64875$ and thus $\omega_0 < 22.595$. Therefore, having in mind the table of natural frequencies of the shell provided previously, Fig 2.6, we observe that the frequencies corresponding to modes 1, 2, 3 and 4 are the ones needed to be considered in the complete dynamic analysis.

Now we also need to define a time step and a total time for the simulation. For this case, we choose to simulate $t = 1$ s as it is sufficient to obtain a descriptive result of the behavior of the structure. For the time step, we can calculate with exactly the same procedure as for the first exercise and we obtain,

$$\Delta t = \frac{T}{20} = \frac{1}{20\omega_0} = 2.53E - 3 \ s$$

where $\omega_0 = 19.76$. Therefore we have that for this analysis,

- Dynamics analysis data

  - Only calculate natural freqs: 0
  - $\Delta t = 2.53E - 3$
  - Steps $= 396$

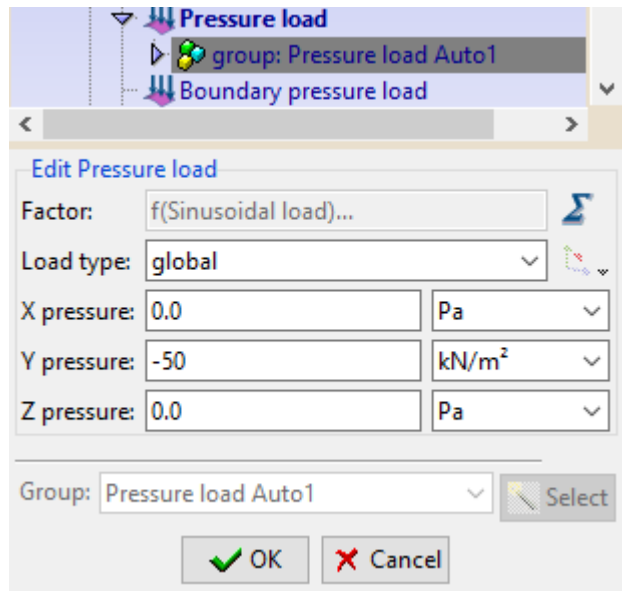Figure 2.12 displays the definition of the load for this case.

Figure 2.12: Definition of the distributed load for the dynamic analysis of the structure.

Once we click on the summation symbol a new windows pops-up, as appears in Figure 2.13 down below. Here we can choose a sinusoidal load an introduce the needed parameters. In the field frequency, we will introduce the different values of the frequencies, that is, 0.75ω, ω and 1.25ω, respectively.
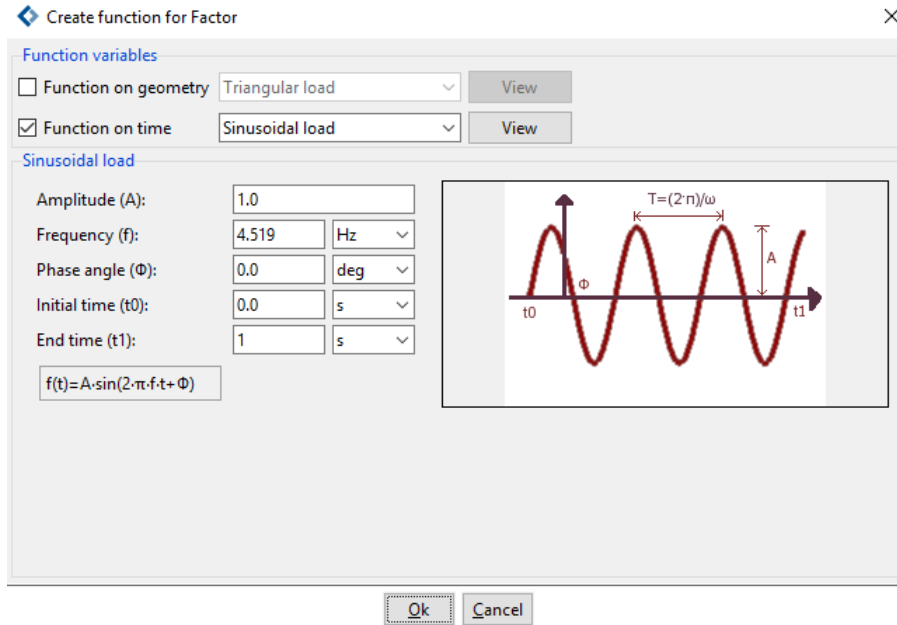
Figure 2.13: Sinusoidal load definition.

2.3.1   Results

In this section we include several figures that show the horizontal displacement versus time of the node corresponding to point # 5 in Figure 2.2. This has been considered as representative point for the analysis.

Figure 2.14 displays the result when considering $0.75\omega_1$. As one can tell, the values of the displacements in this point are kept within an admissible range and they do not increase without control. This results was expected a priori as the frequency of the load does not coincide with the natural frequency of the shell.

The situation in Figure 2.15 is completely different. Here, we now see that the resonance phenomena appears as the value of the displacement start to increase with no control. This is due to the fact that now the frequency of the loading conditions is coincident with one of the natural frequencies of the shell.

Finally, for Figure 2.16, the situation is similar to the first case, in which the value of the frequency of the load is not close enough to any of the natural frequencies of the shell. Therefore, no resonance is expected and the displacements are kept in a safe range.
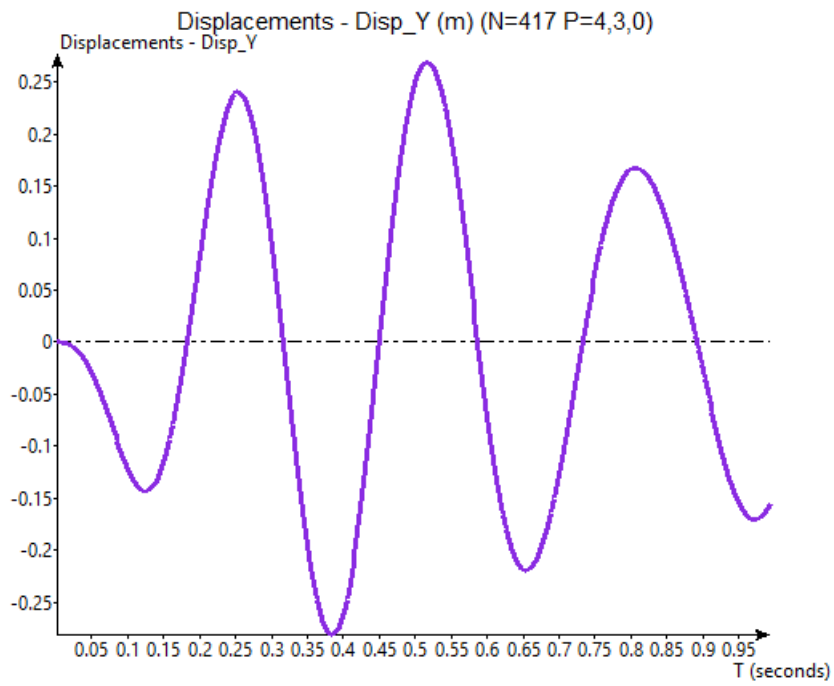
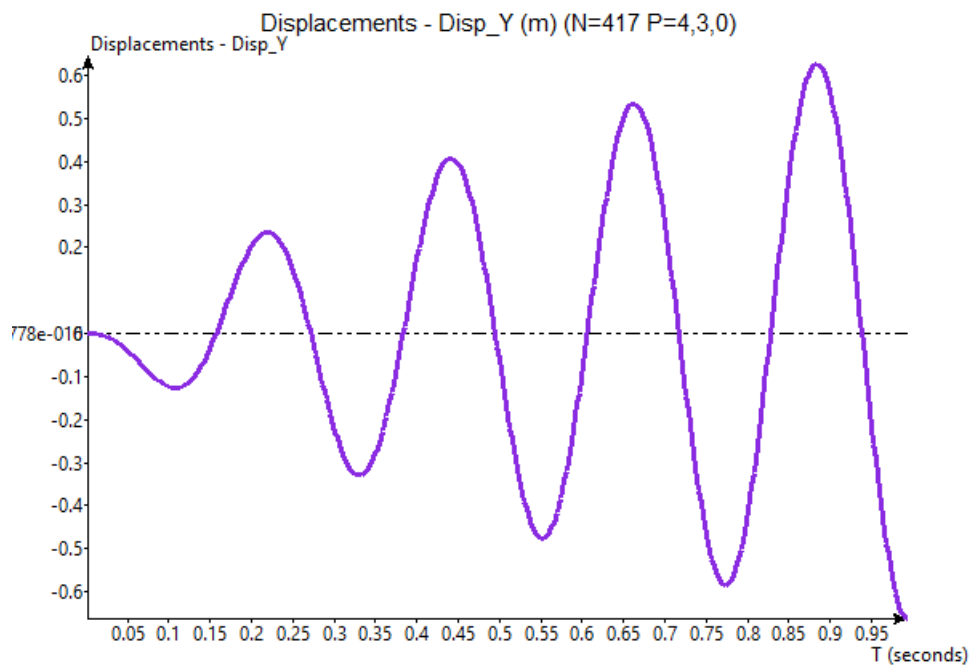Figure 2.14: y component of the displacements vs. time for $0.75\omega_1 = 3.3893$.

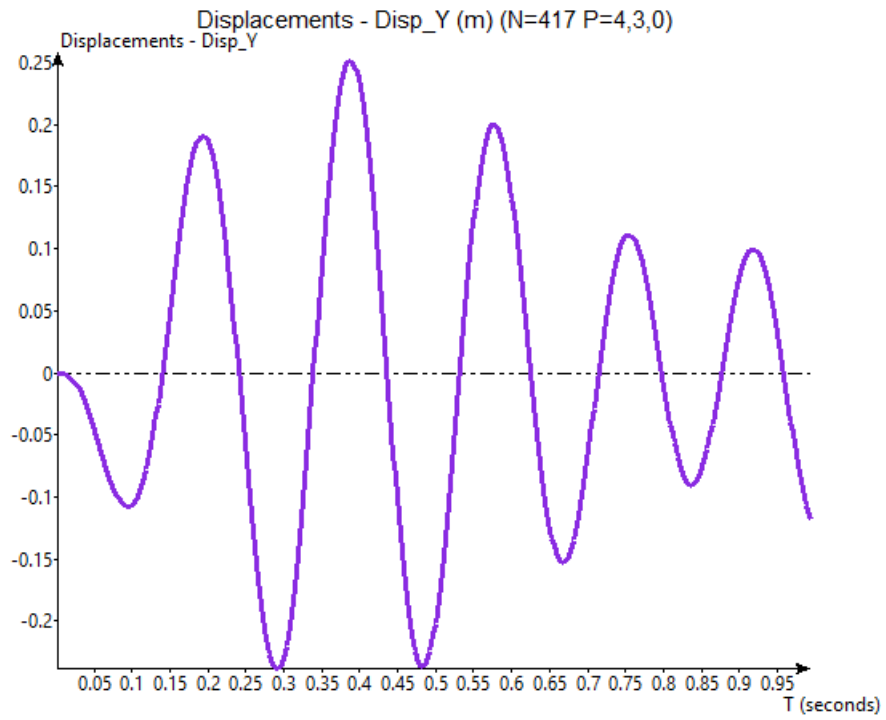Figure 2.15: y component of the displacements vs. time for $\omega_1 = 4.519$.

Figure 2.16: y component of the displacements vs. time for $1.25\omega_1 = 5.6488$.