9 FREE VIBRATION AND EARTHQUAKE ANALYSIS OF A BUILDING

This example demonstrates the natural frequency of a long five-storey building when subjected to free vibration and earthquake loading. The two calculations employ different dynamic boundary conditions:

- In the free vibration, the *Viscous* boundary conditions are considered. This option is suitable for problems where the dynamic source is inside the mesh.
- For the earthquake loading, the *Free-field* and *Compliant base* boundary conditions are considered. This option is preferred for earthquake analysis, where the dynamic input is applied along the model boundary.

The building consists of 5 floors and a basement. It is 10 m wide and 17 m high including the basement. The total height from the ground level is 5 x 3 m = 15 m and the basement is 2 m deep. A value of 5 kN/m^2 is taken as the weight of the floors and the walls. The building is constructed on a clay layer of 15 m depth underlayed by a deep sand layer. In the model, 25 m of the sand layer will be considered.

Objectives:

- Performing a *Dynamic* calculation
- Defining dynamic boundary conditions (free-field and compliant base)
- Defining earthquakes by means of displacement multipliers
- Modelling of free vibration of structures
- Modelling of hysteretic behaviour by means of HS small model
- Calculating the natural frequency by means of Fourier spectrum

9.1 GEOMETRY

The length of the building is much larger than its width and the earthquake is supposed to have a dominant effect across the width of the building. Taking these facts into consideration, a representative section of 3 m will be considered in the model in order to decrease the model size. The geometry of the model is shown in Figure 9.1.

9.1.1 GEOMETRY MODEL

- Start the Input program and select *Start a new project* from the *Quick select* dialog box.
- In the *Project* tabsheet of the *Project properties* window, enter an appropriate title.
- Keep the default units and set the model dimensions to $X_{min} = -80$, $X_{max} = 80$, $Y_{min} = 0$ and $Y_{max} = 3$.

9.1.2 DEFINITION OF SOIL STRATIGRAPHY

The subsoil consists of two layers. The *Upper clayey layer* lies between the ground level (z = 0) and z = -15. The underlying *Lower sandy layer* lies to z = -40. Define the phreatic level by assigning a value of -15 to the *Head* in the borehole. Create the material data set

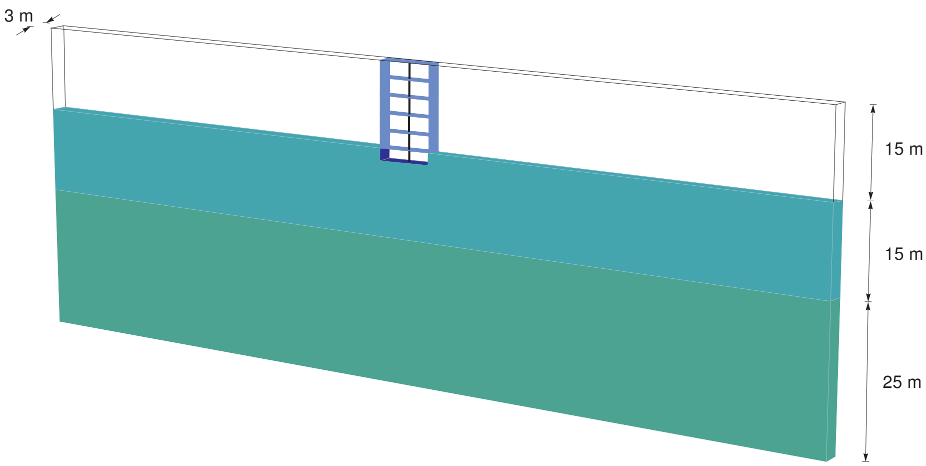

Figure 9.1 Geometry of the model

according to Table 9.1 and assign it to the corresponding soil layers. The upper layer consists of mostly clayey soil and the lower one consists of sandy soil.

Table 9.1 Material properties of the subsoil layers

Parameter	Name	Upper clayey layer	Lower sandy layer	Unit
General				
Material model	Model	HS small	HS small	-
Drainage type	Type	Drained	Drained	-
Soil unit weight above phreatic level	γ_{unsat}	16	20	kN/m^3
Soil unit weight above phreatic level	γ_{sat}	20	20	kN/m^3
Parameters				
Secant stiffness in standard drained triaxial test	E_{50}^{ref}	$2.0 \cdot 10^4$	$3.0 \cdot 10^4$	kN/m^2
Tangent stiffness for primary oedometer loading	E_{oed}^{ref}	$2.561 \cdot 10^4$	$3.601 \cdot 10^4$	kN/m^2
Unloading / reloading stiffness	E_{ur}^{ref}	$9.484 \cdot 10^4$	$1.108 \cdot 10^5$	kN/m^2
Power for stress-level dependency of stiffness	m	0.5	0.5	-
Cohesion	c'_{ref}	10	5	kN/m^2
Friction angle	φ'	18.0	28.0	°
Dilatancy angle	ψ	0.0	0.0	°
Shear strain at which $G_s = 0.722 G_0$	$\gamma_{0.7}$	$1.2 \cdot 10^{-4}$	$1.5 \cdot 10^{-4}$	-
Shear modulus at very small strains	G_0^{ref}	$2.7 \cdot 10^5$	$1.0 \cdot 10^5$	kN/m^2
Poisson's ratio	ν'_{ur}	0.2	0.2	-

When subjected to cyclic shear loading, the HS small model will show typical hysteretic behaviour. Starting from the small-strain shear stiffness, G_0^{ref}, the actual stiffness will decrease with increasing shear. Figures 9.2 and 9.3 display the Modulus reduction curves, i.e. the decay of the shear modulus with strain.

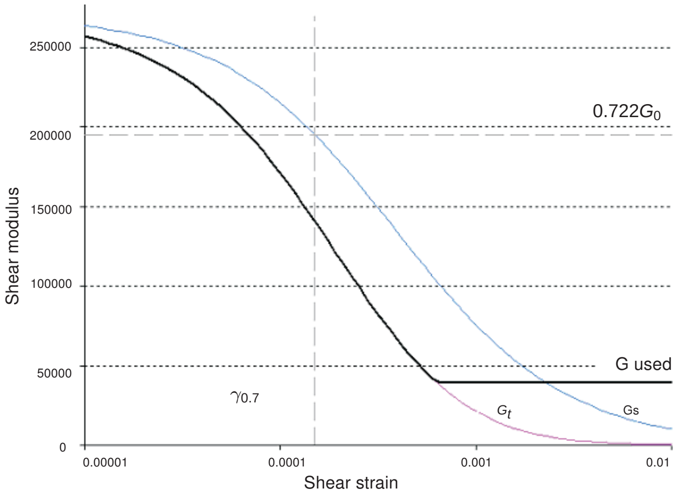

Figure 9.2 Modulus reduction curves for the upper clayey layer

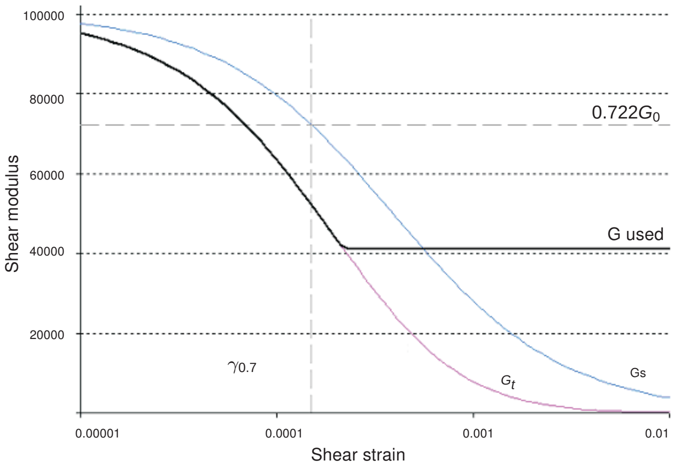

Figure 9.3 Modulus reduction curve for the lower sandy layer

In the HS small model, the tangent shear modulus is bounded by a lower limit, G_{ur}.

$$G_{ur} = \frac{E_{ur}}{2(1 + \nu_{ur})}$$

The values of G_{ur}^{ref} for the *Upper clayey layer* and *Lower sandy layer* and the ratio to G_0^{ref} are shown in Table 9.2. This ratio determines the maximum damping ratio that can be obtained.

Table 9.2 G_{ur} values and ratio to G_0^{ref}

Parameter	Unit	Upper clayey layer	Lower sandy layer
G_{ur}	kN/m^2	39517	41167
G_0^{ref}/G_{ur}	-	6.83	2.43

Figures 9.4 and 9.5 show the damping ratio as a function of the shear strain for the material used in the model. For a more detailed description and elaboration from the

modulus reduction curve to the damping curve can be found in the literature*.

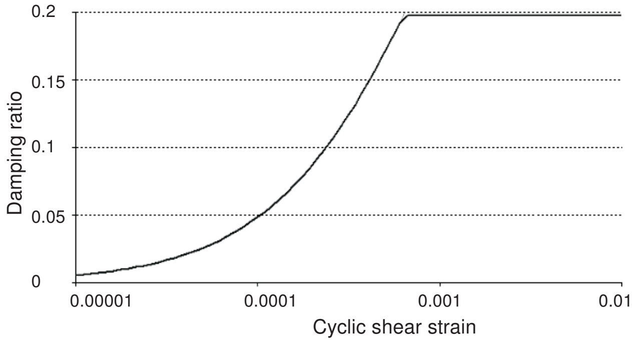

Figure 9.4 Damping curve for the upper clayey layer

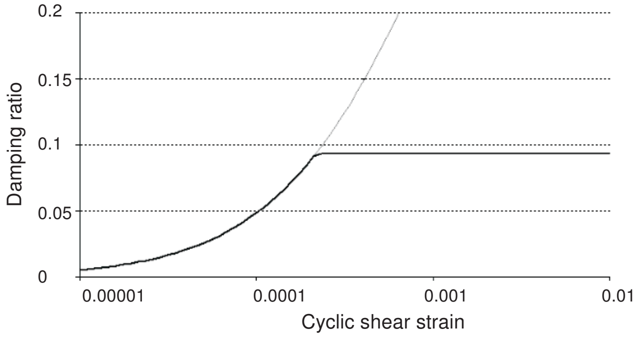

Figure 9.5 Damping curve for the lower sandy layer

9.1.3 DEFINITION OF STRUCTURAL ELEMENTS

The structural elements of the model are defined in the *Structures* mode.

Building

The building consists of 5 floors and a basement. It is 10 m wide and 17 m high including the basement. The total height from the ground level is 5 x 3 m = 15 m and the basement is 2 m deep. A value of 5 kN/m^2 is taken as the weight of the floors and the walls. To define the structure:

- Define a surface passing through the points (-5 0 -2), (5 0 -2), (5 3 -2) and (-5 3 -2).
- Create a copy of the surface by defining an 1D array in z-direction. Set the number of the columns to 2 and the distance between them to 2 m.
- Select the created surface at z = 0 and define a 1D array in the z-direction. Set the number of the columns to 6 and the distance between consecutive columns to 3 m.
- Define a surface passing through the points (5 0 -2), (5 3 -2), (5 3 15) and (5 0 15).

* Brinkgreve, R.B.J., Kappert, M.H., Bonnier, P.G. (2007). Hysteretic damping in small-strain stiffness model. In Proc. 10th Int. Conf. on Comp. Methods and Advances in Geomechanics. Rhodes, Greece, 737 – 742

- Create a copy of the vertical surface by defining an 1D array in x-direction. Set the number of the columns to 2 and the distance between them to -10 m.
- Multiselect the vertical surfaces and the horizontal surface located at z = 0.
- Right-click on the selection and select the *Intersect and recluster* option from the appearing menu. It is important to do the intersection in the *Structures mode* as different material data sets are to be assigned to the basement and the rest of the building.
- Select all the created surfaces representing the building (basement, floors and walls), right-click and select the *Create plate* option from the appearing menu.
- Define the material data set for the plates representing the structure according to Table 9.3. Note that two different material data sets are used for the basement and the rest of the building respectively.
- Assign the *Basement* material data set to the horizontal plate located at z = -2 and the vertical plates located under the ground level.
- Assign the corresponding material data set to the rest of the plates in the model.

Table 9.3 Material properties of the building (plate properties)

Parameter	Name	Rest of building	Basement	Unit
Type of behaviour	Type	Elastic; Isotropic	Elastic; Isotropic	-
Thickness	d	0.3	0.3	m
Material weight	γ	33.33	50	kN/m^3
Young's modulus	E_1	$3 \cdot 10^7$	$3 \cdot 10^7$	kN/m^2
Poisson's ratio	ν_{12}	0	0	-
Rayleigh damping	α	0.2320	0.2320	-
	β	$8 \cdot 10^{-3}$	$8 \cdot 10^{-3}$	-

In order to model the soil-structure intersection at the basement of the building assign interfaces to the outer side of the basement. Note that depending on the local coordinate system of the surfaces an interface either positive or negative is assigned.

The central column of the structure is modelled using the *Node-to-node anchor* feature. To create the central column of the structure:

- Create a *Line* through points (0 1.5 -2) and (0 1.5 0) corresponding to the column in the basement floor.
- Create a *Line* through points (0 1.5 0) and (0 1.5 3) corresponding to the column in the first floor.
- Create a copy of the last defined line by defining an 1D array in z-direction. Set the number of the columns to 5 and the distance between them to 3 m.
- Select the created lines, right-click and select the *Create node-to-node anchor* option from the appearing menu.
- Create the material data set according to the Table 9.4 and assign it to the anchors.

Table 9.4 Material properties of the node-to-node anchor

Parameter	Name	Column	Unit
Material type	Type	Elastic	-
Normal stiffness	*EA*	$2.5 \cdot 10^6$	kN

Loads

A static lateral force of 10 kN/m is applied laterally at the top left corner of the building. To create the load:

- Create a line load passing through (-5 0 15) and (-5 3 15).
- Specify the components of the load as (10 0 0).

The earthquake is modelled by imposing a prescribed displacement at the bottom boundary. To define the prescribed displacement:

- Create a surface prescribed displacement passing through (-80 0 -40), (80 0 -40), (80 3 -40) and (-80 3 -40).
- Specify the x-component of the prescribed displacement as *Prescribed* and assign a value of 1.0. The y and z components of the prescribed displacement are *Fixed*. The default distribution (*Uniform*) is valid.

To define the dynamic multipliers for the prescribed displacement:

- In the *Model explorer* expand the *Attributes library* subtree. Right-click on *Dynamic multipliers* and select the *Edit* option from the appearing menu. The *Multipliers* window pops up displaying the *Displacement multipliers* tabsheet.
- To add a multiplier click the corresponding button in the *Multipliers* window.
- From the *Signal* drop-down menu select the *Table* option.
- The file containing the earthquake data is available in the PLAXIS knowledge base (http://kb.plaxis.nl/search/site/smc).
- Open the page in a web browser, copy all the data to a text editor (e.g. Notepad) and save the file in your computer with the extension *.smc. Alternatively this file can also be found in the *Importables* folder in the PLAXIS directory.
- In the *Multipliers* window click the *Open* button and select the saved file. In the *Import data* window select the *Strong motion CD-ROM files* option from the *Parsing method* drop-down menu and press *OK* to close the window.
- Select the *Acceleration* option in the *Data type* drop-down menu.
- Select the *Drift correction* options and click *OK* to finalize the definition of the multiplier.
- In the *Dynamic multipliers* window the table and the plot of the data is displayed (Figure 9.6).
- In the *Model explorer* expand the *Surface displacements* subtree and assign the *Multiplier*$_x$ to the x- component by selecting the option in the drop-down menu.

Create interfaces on the boundary

Free-field and *Compliant base* require the manual creation of interface elements along the vertical and bottom boundaries of the model in the *Structures mode*. The interface elements must be added inside the model, else the *Free-field* and *Compliant base* boundary conditions are ignored. To define the interfaces:

- Create a surface passing through (-80 3 0), (-80 0 0), (-80 0 -40) and (-80 3 -40). Right-click the created surface and and click *Create positive interface* to add an

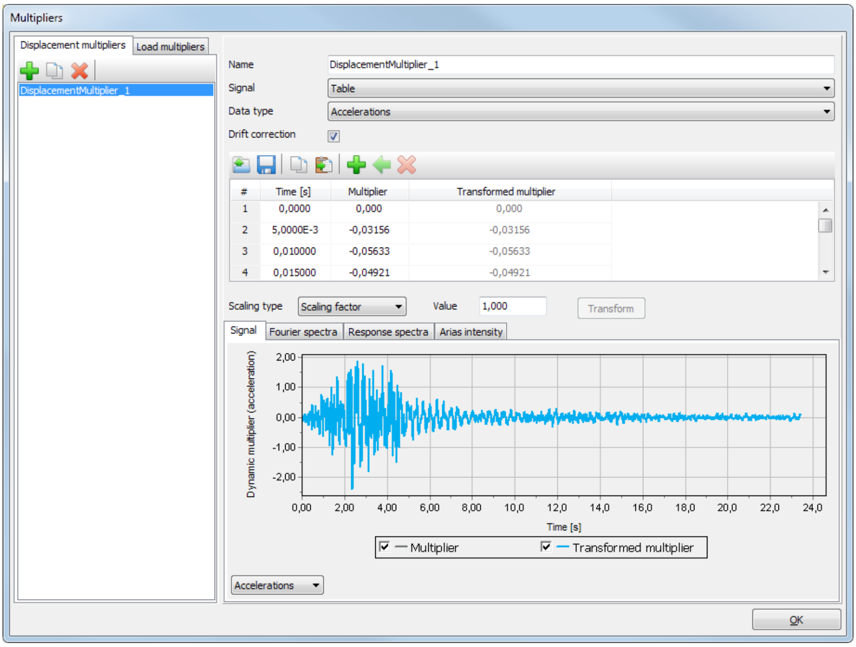

Figure 9.6 *Dynamic multipliers* window

interface inside the model.

Create a surface passing through (80 3 0), (80 0 0), (80 0 -40) and (80 3 -40). Right-click the created surface and and click *Create negative interface* to add an interface inside the model.

- The surface at the bottom of the model is already created by the prescribed displacement. Right-click the surface at the bottom of the model and click *Create positive interface* to add an interface inside the model.

9.2 MESH GENERATION

- Proceed to the *Mesh* mode.
- Click the *Generate mesh* button. Set the element distribution to *Fine*.
- View the generated mesh (Figure 9.7).

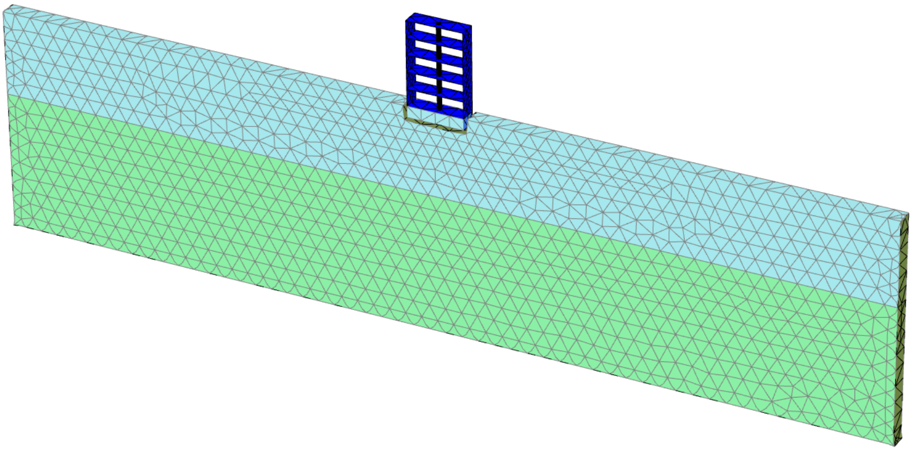

Figure 9.7 Geometry and mesh

9.3 PERFORMING CALCULATIONS

The calculation process consists of the initial conditions phase, simulation of the construction of the building, loading, free vibration analysis and earthquake analysis.

Initial phase

- Click on the *Staged construction* tab to proceed with definition of the calculation phases.
- The initial phase has already been introduced. The default settings of the initial phase will be used in this tutorial.
- In the *Staged construction* mode check that the building and load are inactive.

Phase 1

Add a new phase (Phase_1). The default settings of the added phase will be used for this calculation phase.

- In the *Staged construction* mode construct the building (activate all the plates, the anchors and only the interfaces of the basement) and deactivate the basement volume (Figure 9.8).

Phase 2

Add a new phase (Phase_2).

- In the *Phases* window select the *Reset displacement to zero* in the *Deformation control parameters* subtree. The default values of the remaining parameters will be used in this calculation phase.
- In the *Staged construction* mode activate the line load. The value of the load is already defined in the *Structures* mode.

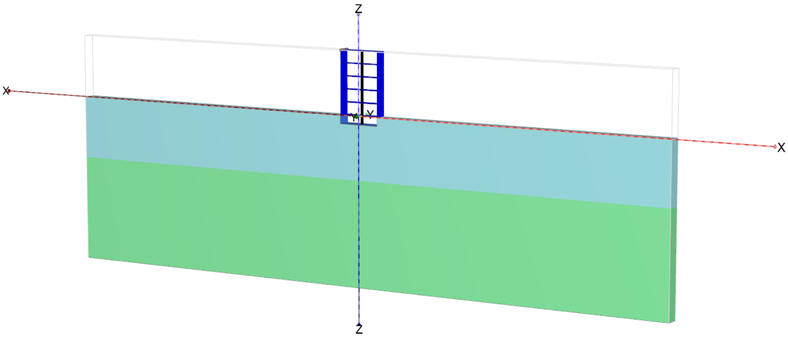

Figure 9.8 Construction of the building

Phase 3

Add a new phase (Phase_3).

In the *Phases* window select the *Dynamic* option as *Calculation type*.

- Set the *Time interval* parameter to 5 sec.
- In the *Staged construction* mode deactivate the line load.
- In the *Model explorer* expand the *Model conditions* subtree.
- Expand the *Dynamics* subtree. By default the boundary conditions in the x and y directions are set to viscous. Select the *None* option for the boundaries in the y direction. Set the boundary Zmin to viscous (Figure 9.9).

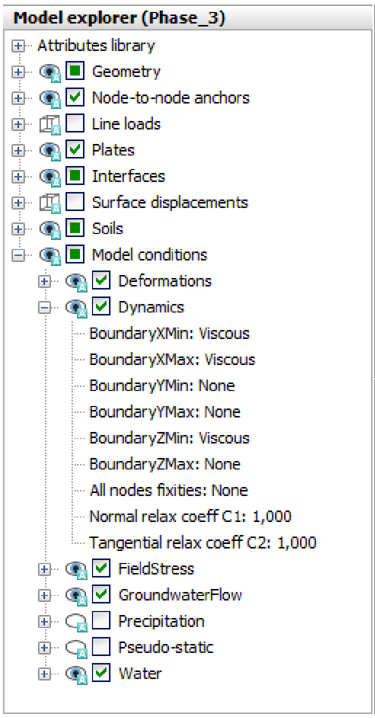

Figure 9.9 Boundary conditions for *Dynamic calculations* (Phase_3)

Hint: For a better visualisation of the results, animations of the free vibration and earthquake can be created. If animations are to be created, it is advised to increase the number of the saved steps by assigning a proper value to the *Max steps saved* parameter in the *Parameters* tabsheet of the *Phases* window.

Phase 4

Add a new phase (Phase_4).

- In the *Phases* window set the *Start from phase* option to Phase 1 (construction of building).

Select the *Dynamic* option as *Calculation type*.

- Set the *Dynamic time interval* parameter to 20 sec.
- Select the *Reset displacement to zero* in the *Deformation control parameters* subtree. The default values of the remaining parameters will be used in this calculation phase.
- In the *Numerical control parameters* subtree uncheck the *Use default iter parameters* checkbox, which allows you to change advanced settings and set the *Time step determination* to *Manual*.
- Set the *Max steps* to 1000 and the *Max number of sub steps* to 4.
- In the *Model explorer* expand the *Model conditions* subtree.
- Expand the *Dynamics* subtree. Set the *Free-field* option for the boundaries in the x direction. The boundaries in the y direction are already set to *None*. Set the boundary Zmin to *Compliant base* (Figure 9.10).
- Make sure that the interfaces on the boundary of the model are not activated in the *Model explorer*.
- In the *Model explorer* activate the *Surface displacement* and its dynamic component. Set the value of u_x to 0.5 m. Considering that the boundary condition at the base of the model will be defined using a *Compliant base*, the input signal has to be taken as half of the outcropping motion.

Select points for load displacement curves at (0 1.5 15), (0 1.5 6), (0 1.5 3) and (0 1.5 -2). The calculation may now be started.

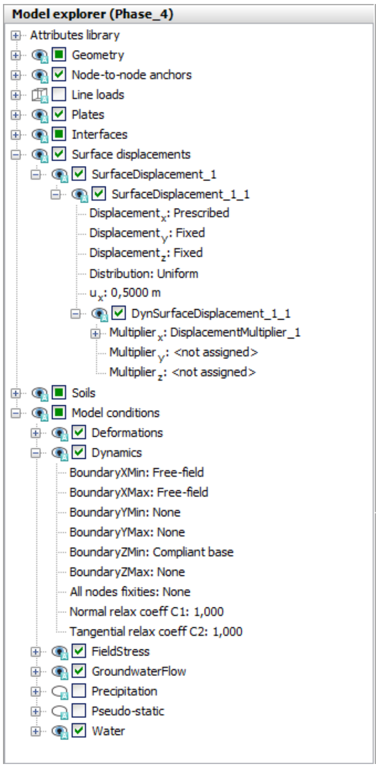

Figure 9.10 Boundary conditions for *Dynamic calculations* (Phase_4)

9.4 VIEWING THE RESULTS

Figure 9.11 shows the deformed structure at the end of the Phase 2 (application of horizontal load). Figure 9.12 shows the time history of displacements of the selected points A (0 1.5 15), B (0 1.5 6), C (0 1.5 3) and D (0 1.5 -2) for the free vibration phase. It may be seen from the figure that the vibration slowly decays with time due to damping in the soil and in the building.

In the *Chart* tabsheet of the *Settings* window select the *Use frequency representation (spectrum)* and *Use standard frequency (Hz)* options in the *Dynamics* box. The plot is shown in Figure 9.13. From this figure it can be evaluated that the dominant building frequency is around 1 Hz. For a better visualisation of the results animations of the free vibration and earthquake can be created.

Figure 9.14 shows the time history of displacements of the point A (0 1.5 15) for the earthquake phase. It may be seen from the figure that the vibration slowly decays with time due to damping in the soil and in the building.

The time history signature of the point A (0 1.5 15) of the earthquake phase has been transformed to normalised power spectra through Fast Fourier transform for Phase 4 and is plotted in Figure 9.15.

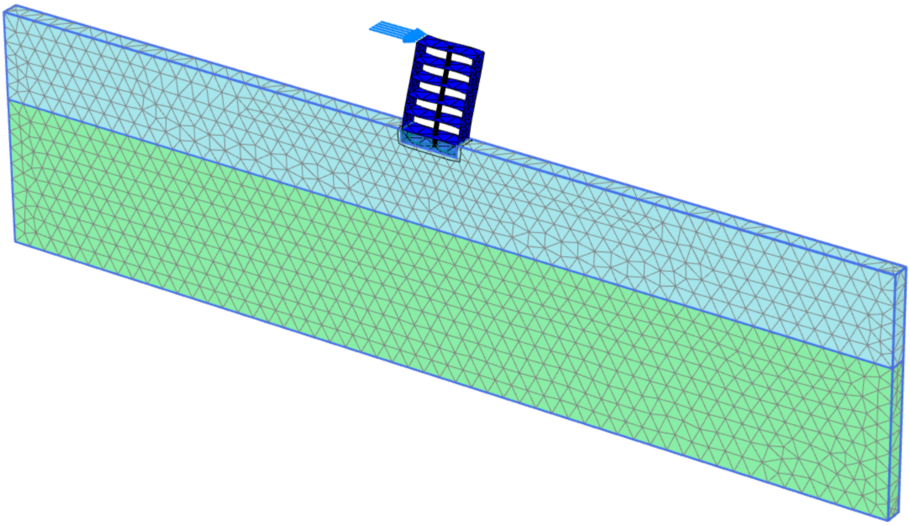

Figure 9.11 Deformed mesh of the system at the end of Phase_2

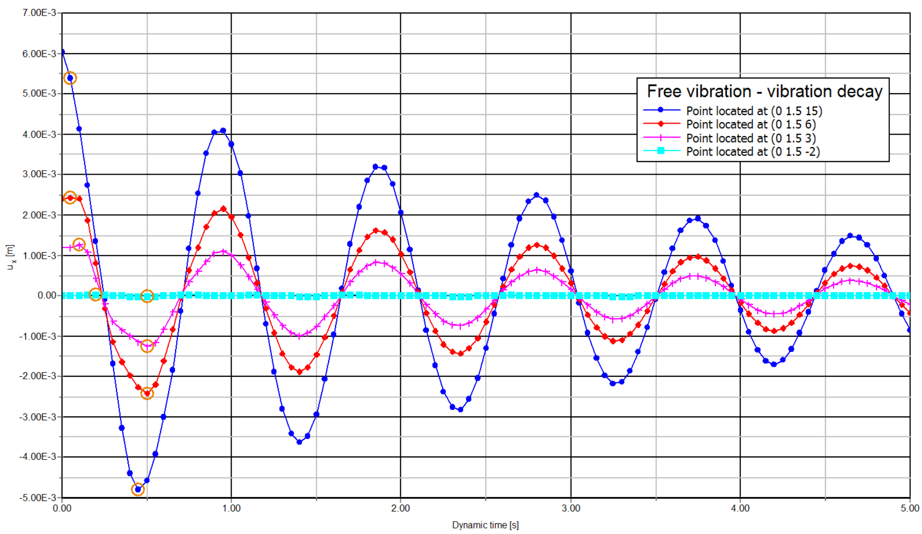

Figure 9.12 Time history of displacements (Free vibration)

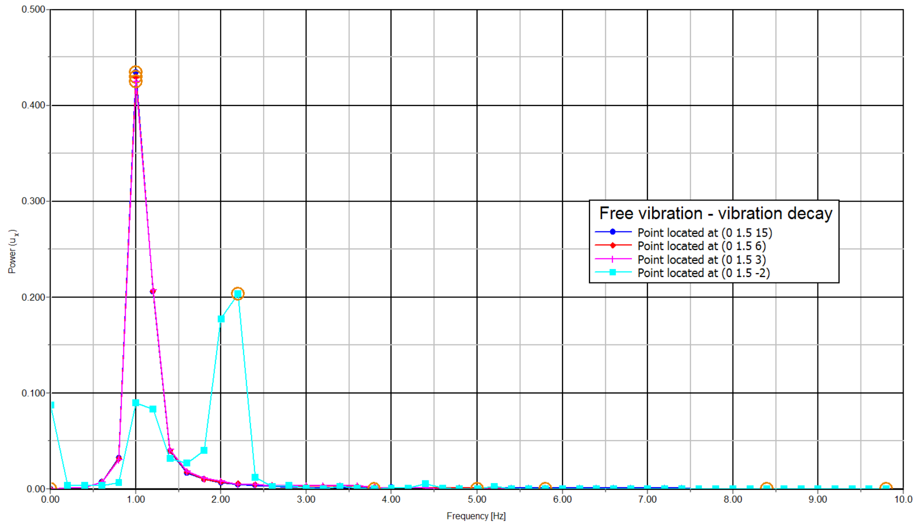

Figure 9.13 Frequency representation (spectrum - Free vibration)

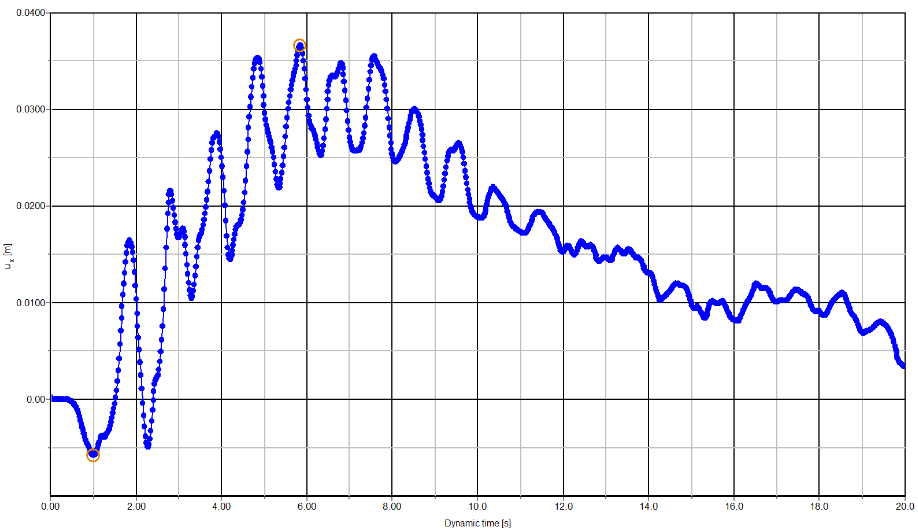

Figure 9.14 Time history of displacements of the top of the building (Earthquake)

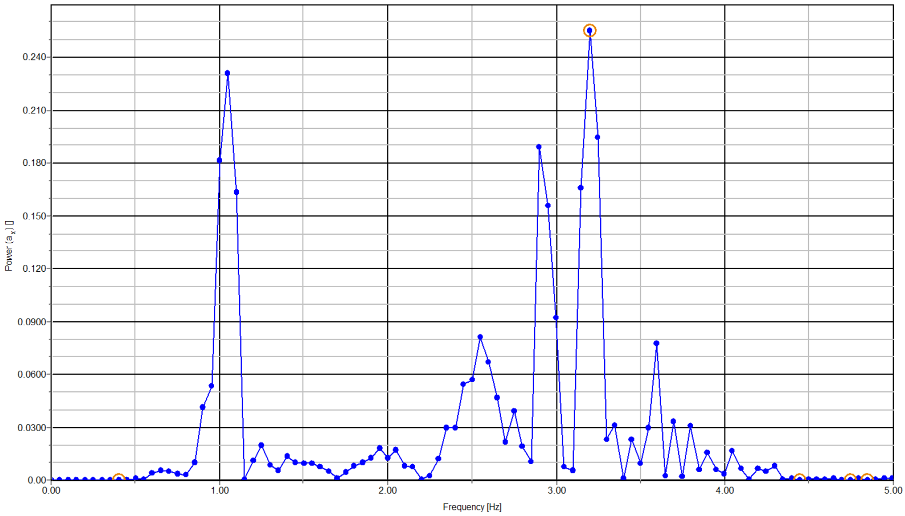

Figure 9.15 Acceleration power spectra at (0 1.5 15)