



14 FREE VIBRATION AND EARTHQUAKE ANALYSIS OF A BUILDING

This example demonstrates the natural frequency of a 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 at the bottom of the model.

The building consists of 5 floors and a basement. It is 10 m wide and 17 m high including basement. The total height from the ground level is $5 \times 3 \text{ m} = 15 \text{ m}$ 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 underlain 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, compliant base and viscous)
- Defining earthquakes by means of dynamic multipliers
- Modelling free vibration of structures
- Modelling hysteretic behaviour by means of HS small model
- Evaluating the natural frequency from Fourier spectrum


14.1 INPUT

General settings

- Start the Input program and select the *Start a new project* from the *Quick select* dialog box.
- In the *Project* tabsheet of the *Project properties* window enter an appropriate title.
- In the *Model* tabsheet keep the default options for *Model (Plane strain)* and *Elements (15-Noded)*.
- Keep the default units and constants and set the model dimensions to $x_{min} = -80.0$, $x_{max} = 80.0$, $y_{min} = -40.0$ and $y_{max} = 15.0$.

14.1.1 DEFINITION OF SOIL STRATIGRAPHY

The subsoil is divided into a 15 m thick clay layer and a 25 m thick sand layer. The phreatic level is assumed to be at $y = -15.0 \text{ m}$. Hydrostatic pore pressures are generated in the whole geometry according to this phreatic line.

-  Create a borehole at $x = 0$. The *Modify soil layers* window pops up.
 - Add two soil layers extending from $y = 0.0$ to $y = -15.0$ and from $y = -15.0$ to $y = -40.0$.

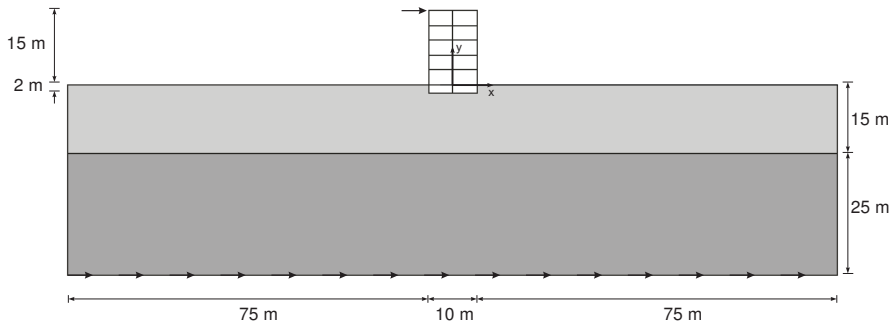


Figure 14.1 Geometry of the project

- Set the *Head* in the borehole at -15.0 m.

The upper layer consists of mostly clayey soil and the lower one is sandy. Both have HS small model properties. The presence of the groundwater is neglected. The soil layers with HS small model properties have inherent hysteretic damping.



Open the *Material sets* window.

- Create data sets under *Soil and interfaces* set type according to the information given in Table 14.1.
- Assign the material datasets to the corresponding soil layers in the borehole.

Table 14.1 Material properties of the subsoil layers

Parameter	Name	Upper clay layer	Lower sand layer	Unit
General				
Material model	Model	HS small	HS small	-
Type of material behaviour	Type	Drained	Drained	-
Soil unit weight above phreatic level	γ_{unsat}	16	20	kN/m ³
Soil unit weight below 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	28	°
Dilatancy angle	ψ	0	0	°
Shear strain at which $G_s = 0.722G_0$	$\gamma_{0.7}$	$1.2 \cdot 10^{-4}$	$1.5 \cdot 10^{-4}$	-
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 14.2 and 14.3 display the Modulus reduction curves, i.e. the decay of the shear modulus with strain. The upper curve shows the secant shear modulus and the lower curve shows the tangent shear modulus.

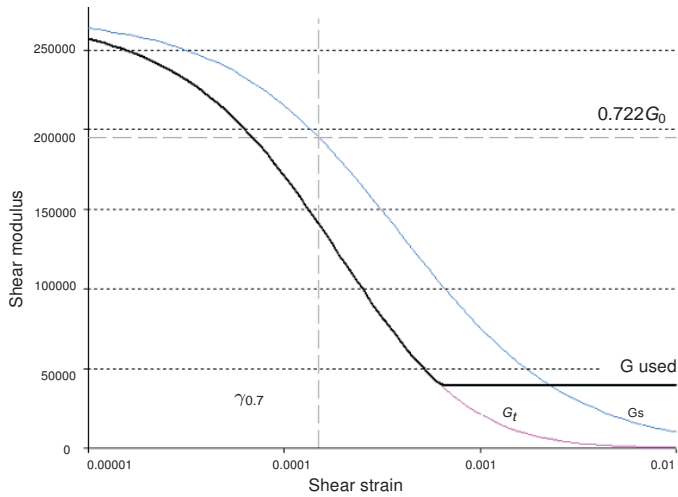


Figure 14.2 Modulus reduction curves for the upper clayey layer

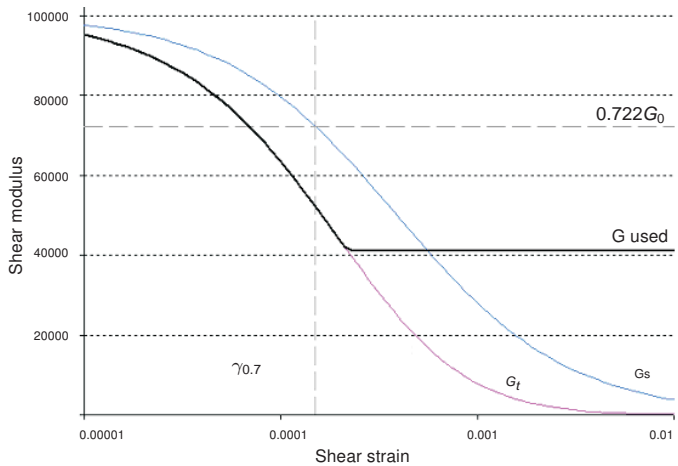


Figure 14.3 Modulus reduction curve for the lower sandy layer

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

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

The values of G_{ur}^{ref} for the *Upper clay layer* and *Lower sand layer* and the ratio to G_0^{ref} are shown in Table 14.2. This ratio determines the maximum damping ratio that can be obtained.

Table 14.2 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}	-	6.75	2.5

Figures 14.4 and 14.5 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*.

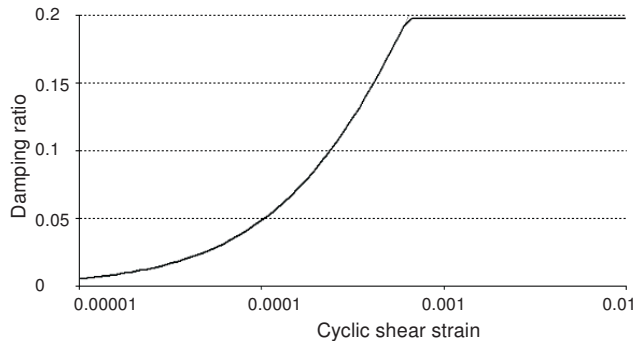


Figure 14.4 Damping curve for the upper clay layer

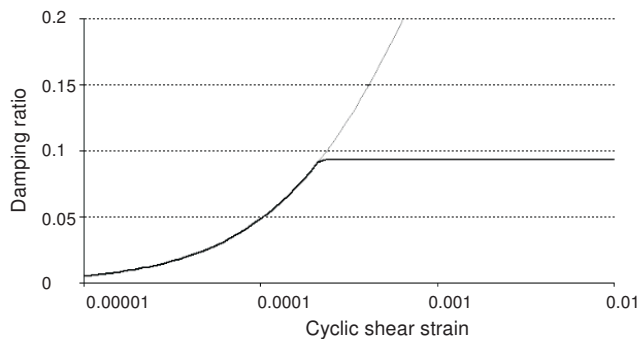


Figure 14.5 Damping curve for the lower sand layer

14.1.2 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. A value of 5 kN/m² is taken as the weight of the floors and the walls. The total height from the ground level is 5 x 3 m = 15 m and the basement is 2 m deep. To define the structure:

- Create the vertical walls of the building passing through (-5.0 0.0) to (-5.0 15.0) and through (5.0 0.0) to (5.0 15.0).
- Use the same feature to define the vertical walls of the basement passing through (-5.0 -2.0) to (-5.0 0.0) and through (5.0 -2.0) to (5.0 0.0).
- Define the floors and the basement of the building as plates passing through (-5.0 -2.0) to (5.0 -2.0), (-5.0 0.0) to (5.0 0.0), (-5.0 3.0) to (5.0 3.0), (-5.0 6.0) to (5.0 6.0), (-5.0 9.0) to (5.0 9.0), (-5.0 12.0) to (5.0 12.0) and (-5.0 15.0) to (5.0 15.0).

* 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

The plates, representing the walls and the floors in the building, are considered to be linear elastic. Note that two different material datasets are used, one for the basement and the other for the rest of the building. The physical damping in the building is simulated by means of Rayleigh damping. A description of Rayleigh damping parameters is given in Section 6.1.1 of the Reference Manual.



Define the material datasets for the structural elements in the building according to Table 14.3.

Table 14.3 Material properties of the building (plate properties)

Parameter	Name	Rest of building	Basement	Unit
Material type	Type	Elastic; Isotropic	Elastic; Isotropic	-
Normal stiffness	EA	$9.0 \cdot 10^6$	$1.2 \cdot 10^7$	kN/m
Flexural rigidity	EI	$6.75 \cdot 10^4$	$1.6 \cdot 10^5$	kNm ² /m
Weight	w	10	20	kN/m/m
Poisson's ratio	ν	0.0	0.0	-
Rayleigh damping	α	0.2320	0.2320	-
	β	$8.0 \cdot 10^{-3}$	$8.0 \cdot 10^{-3}$	-

- Assign the *Basement* material dataset to the vertical plates (2) and the lowest horizontal plate (all under the ground level) in the model.
- Assign the *Rest of the building* material dataset to the remaining plates in the model.
- Use the *Node-to-node anchor* feature to define the column at the centre of the building connecting consecutive floors, (0.0 -2.0) to (0.0 0.0), (0.0 0.0) to (0.0 3.0), (0.0 3.0) to (0.0 6.0), (0.0 6.0) to (0.0 9.0), (0.0 9.0) to (0.0 12.0) and (0.0 12.0) to (0.0 15.0).
- Define the properties of the anchor according to Table 14.4 and assign the material dataset to the anchors in the model.

Table 14.4 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
Spacing out of plane	$L_{spacing}$	3.0	m



Define an interface to model the interaction between soil and building around the basement floor.



Create a point load at the top left corner of the building.

- Specify the components of the load as (10.0 0.0).

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 point load at the top left corner of the building.

- Specify the components of the load as (10.0 0.0).

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




Define a prescribed displacement at the bottom of the model, through (-80.0 -40.0)

and (80.0 -40.0).


- Set the x-component of the prescribed displacement to *Prescribed* and assign a value of 1.0. The y-component of the prescribed displacement is *Fixed*. The default distribution (*Uniform*) is valid.

To define the dynamic multipliers for the prescribed displacement:

- Expand the *Dynamic displacement*
-  Click the *Multiplier_x* drop down menu and click on the appearing plus button. The *Multipliers* window pops up and a new displacement multiplier is automatically added.
- 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>). Copy all the data to a text editor file (e.g. Notepad) and save the file in your computer with the extension *.smc*.
- In the *Multipliers* window click the *Open* button and select the saved *.smc* 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. The defined multiplier is displayed (Figure 14.6).

Create interfaces on the boundary

Free-field and *Compliant base* boundaries require the 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:

-  Click the *Create interfaces on the boundary* button to automatically generate the interfaces at the boundary of the model.

The geometry of the model is shown in Figure 14.7.

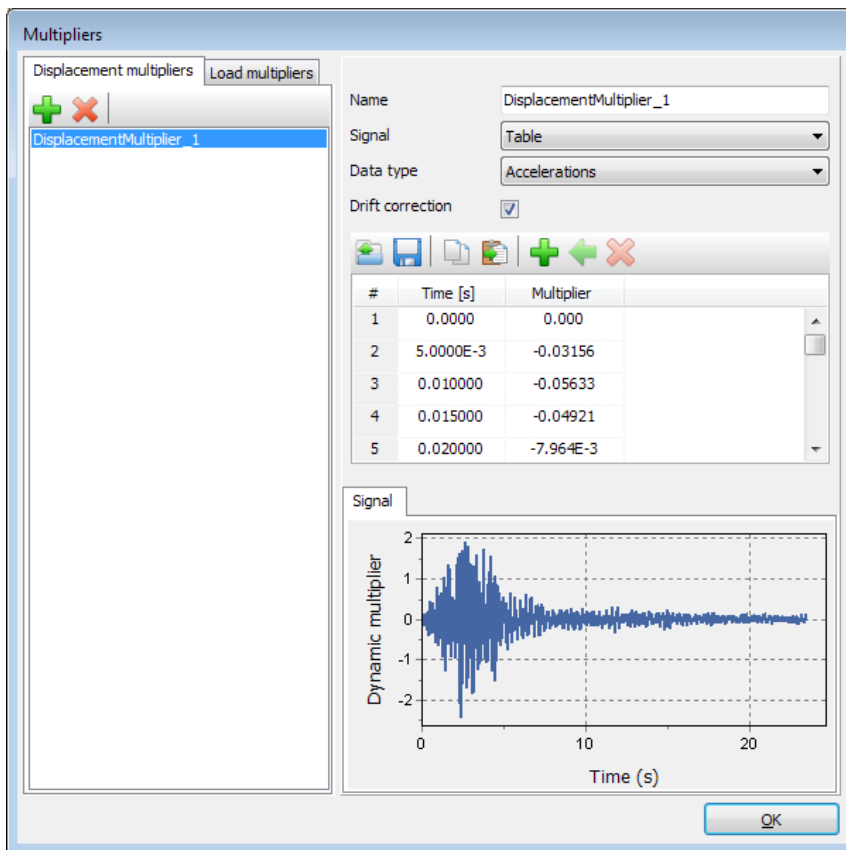
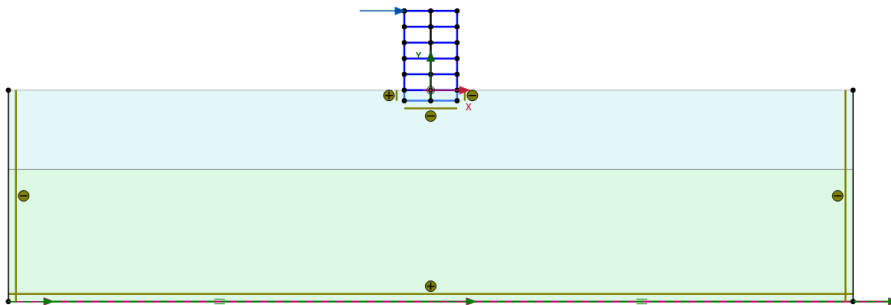


Figure 14.6 *Dynamic multipliers* window

Figure 14.7 Geometry of the model

14.2 MESH GENERATION

- Proceed to the *Mesh* mode.
-  Click the *Generate mesh* button. Set the element distribution to *Fine*.
-  View the generated mesh. The resulting mesh is shown in Figure 14.8.
- Click on the *Close* tab to close the Output program.

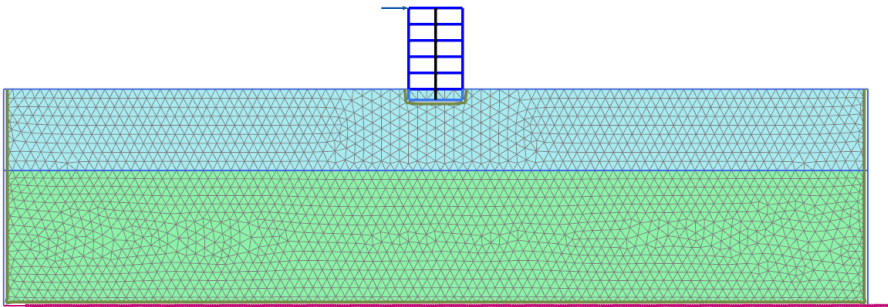


Figure 14.8 The generated mesh

14.3 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 (Figure 14.9).

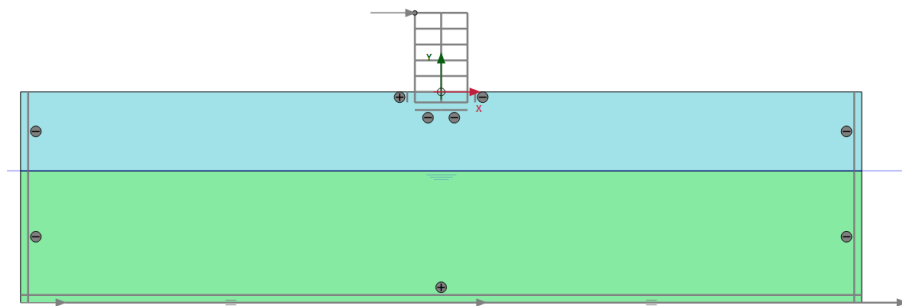


Figure 14.9 Initial phase

Phase 1 - Building

- 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 interfaces and the anchors) and deactivate the basement volume (Figure 14.10).

Phase 2 - Excitation

- Add a new phase (Phase_2).

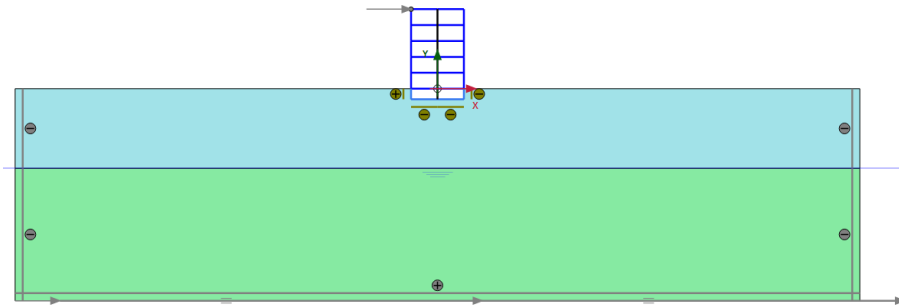



Figure 14.10 Construction of the building

- 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 load. The value of the load is already defined in the *Structures* mode.

Phase 3 - Free vibration


 Add a new phase (Phase_3).

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

- Set the *Dynamic time interval* parameter to 5 sec.
- In the *Staged construction* mode release (deactivate) the point load.
- In the *Model explorer* expand the *Model conditions* subtree.
- Expand the *Dynamics* subtree. Check the boundary conditions *BoundaryXMin*, *BoundaryXMax* and *BoundaryYMin* are viscous (Figure 14.11) which is the default option.

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 - Earthquake

 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

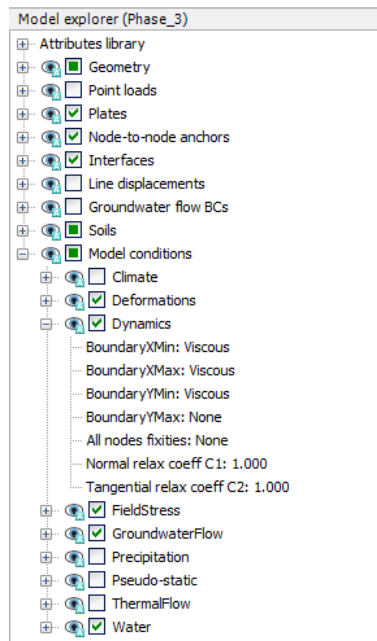


Figure 14.11 Boundary conditions for *Dynamic calculations*

calculation phase.

- In the *Model explorer* expand the *Model conditions* subtree.
- Expand the *Dynamics* subtree. Set the *BoundaryXMin* and *BoundaryXMax* to *Free-field*. Set the *BoundaryYMin* to *Compliant base* (Figure 14.12).
- Interface elements do not need to be active to enable the use of *Free-field* or *Compliant base* boundaries.
- In the *Model explorer* activate the *Line displacements* and its dynamic component. Make sure that the value of $u_{x,start,ref}$ is set 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 bedrock (within) motion.



Select a point at the top of the building for curves (0.0 15.0).



Calculate the project.



Save the project.

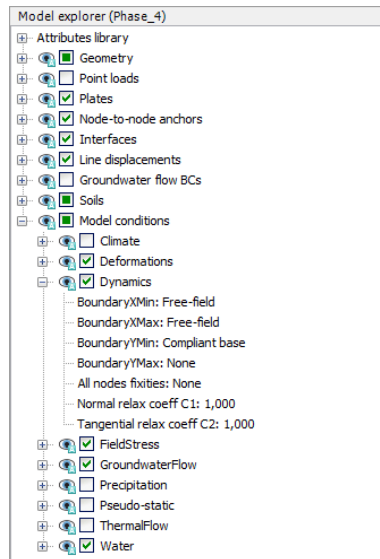


Figure 14.12 Boundary conditions for *Dynamic calculations* (Phase_4)

14.4 RESULTS

Figure 14.13 shows the deformed structure at the end of the Phase 2 (application of horizontal load).

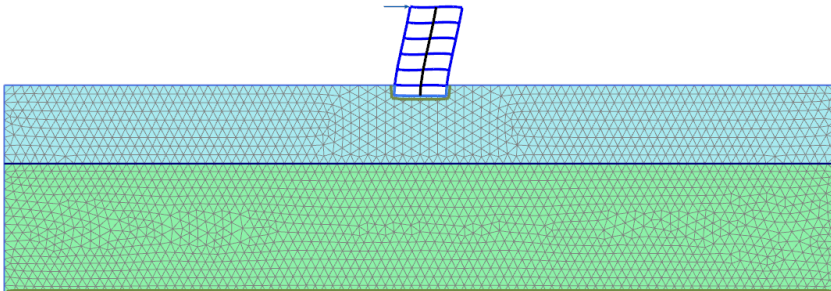


Figure 14.13 Deformed mesh of the system

Figure 14.14 shows the time history of displacements of the selected points A (0 15) 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.

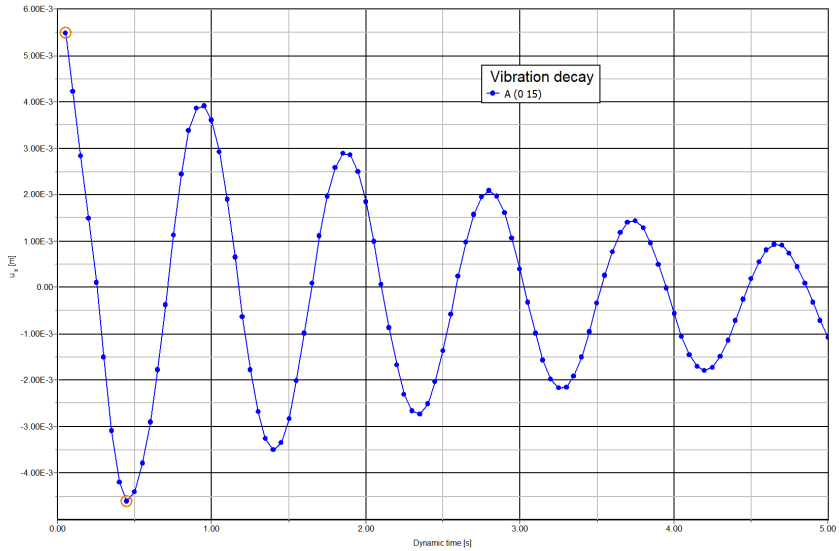


Figure 14.14 Time history of displacements at selected points

In the *Fourier* tabsheet of the *Curve generation* window select the *Power (spectrum)* and click *OK* to create the plot. The plot is shown in Figure 14.15. From this figure it can be evaluated that the dominant building frequency is around 1 Hz.

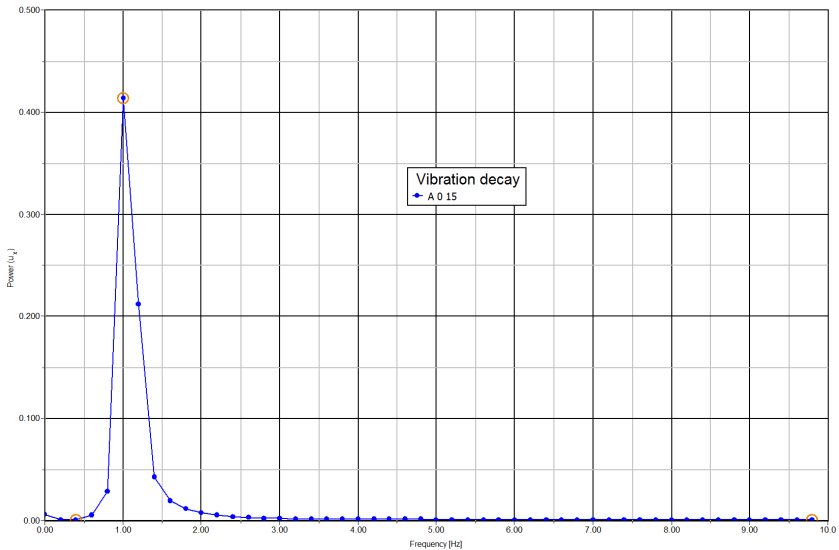


Figure 14.15 Frequency representation (spectrum)

Figure 14.16 shows the time history of the lateral acceleration of the selected points A (0 15) for the earthquake phase (dynamic analysis). For a better visualisation of the results animations of the free vibration and earthquake can be created.

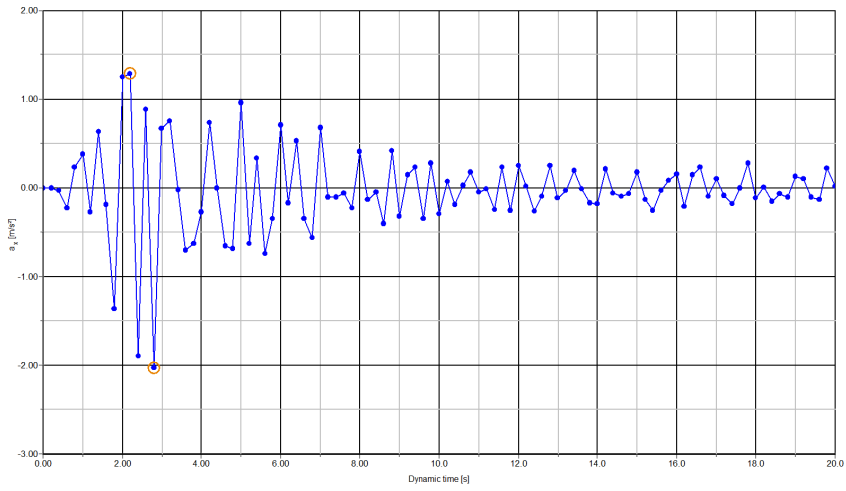


Figure 14.16 Variation of acceleration in dynamic time