9

Free vibration and earthquake analysis of a building [ULT]

9.1 Introduction

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.

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 Hardening Soil model with small-strain stiffness
- Calculating the natural frequency by means of Fourier spectrum

9.2 Geometry

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

9.3 Define the 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 3m will be considered in the model in order to decrease the model size. To create the geometry follow these steps:

1. Start the Input program and select **Start a new project** from the **Quick select** dialog box.
2. In the **Project properties** window, enter an appropriate title.
3. Keep the default units and set the model dimensions to:
   - **a.** x_{min} = -80 and x_{max} = 80
   - **b.** y_{min} = 0 and y_{max} = 3

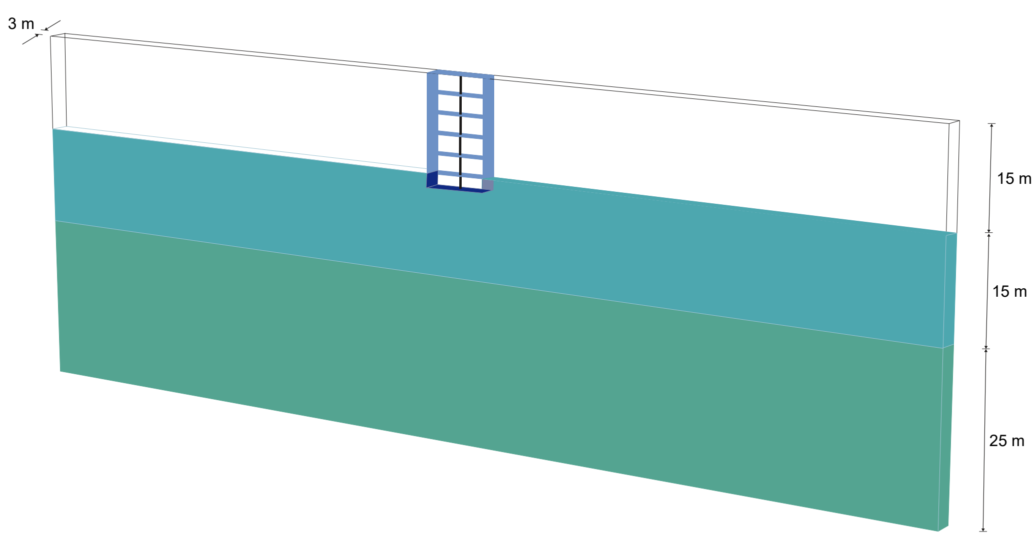

Figure 9–1: The geometry of the model

9.4 Define the soil stratigraphy

1. The subsoil consists of two layers. The **Upper clayey layer** lies between the ground level (z = 0) and z = -15.
2. The underlying **Lower sandy layer** lies to z = -40.

3 Define the phreatic level by assigning a value of -15 to the **Head** in the borehole.

9.5 Create and assign material data sets

Two material data sets are needed for this tutorial. The properties and some details of the material model are displayed in Table 9–1 (p. 162).

Table 9–1: Material properties

Property	Name	Upper clayey layer	Lower sandy layer	Unit
General				
Soil model	Model	HS small	HS small	-
Drainage type	Type	Drained	Drained	-
Unsaturated unit weight	γ_{unsat}	16	20	kN/m^3
Saturated unit weight	γ_{sat}	20	20	kN/m^3
Mechanical				
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
Poisson's ratio	ν_{ur}	0.2	0.2	-
Power for stress-level dependency of stiffness	m	0.5	0.5	-
Shear modulus at very small strains	G_0^{ref}	$2.7 \cdot 10^5$	$1.0 \cdot 10^5$	kN/m^2
Shear strain at which G_s = 0.722 G_0	$\gamma_{0.7}$	$1.2 \cdot 10^{-4}$	$1.5 \cdot 10^{-4}$	-
Cohesion	c'_{ref}	10	5	kN/m^2
Friction angle	φ'	18.0	28.0	°
Dilatancy angle	ψ	0.0	0.0	°

When subjected to cyclic shear loading, the Hardening Soil model with small-strain stiffness will show typical hysteretic behaviour. Starting from the small-strain shear stiffness, G_0^{ref}, the actual stiffness will decrease with increasing shear. Figure 9–2 (p. 163) and Figure 9–3 (p. 163) display the Modulus reduction curves, i.e. the decay of the shear modulus with strain.

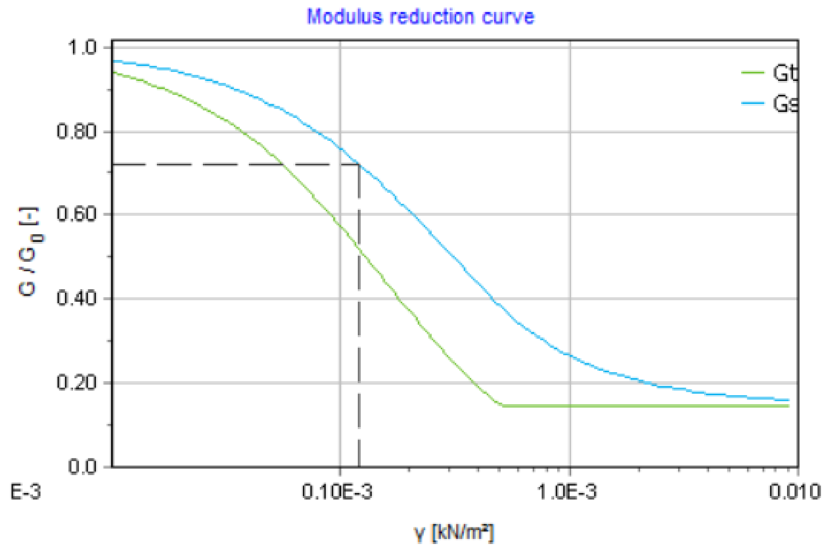

Figure 9–2: Modulus reduction curves for the upper clayey layer

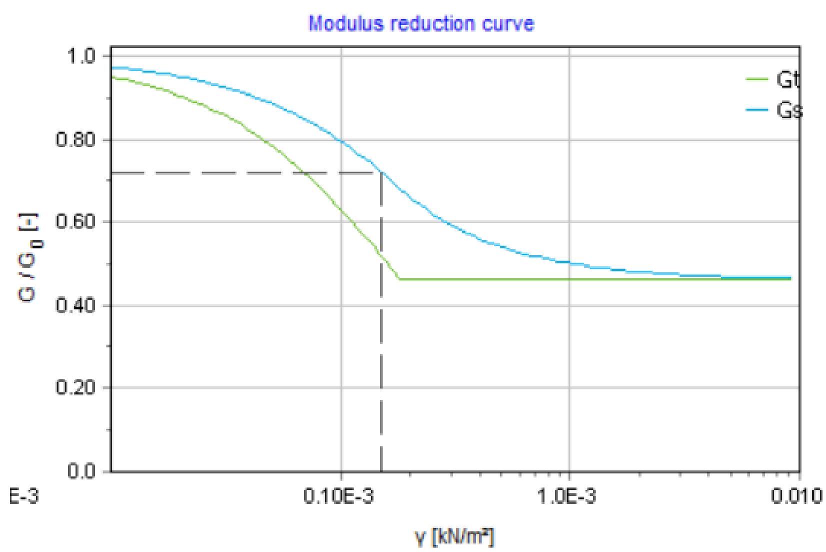

Figure 9–3: Modulus reduction curve for the lower sandy layer

In the Hardening Soil model with small-strain stiffness, 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 (p. 163). 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

Parameter	Unit	Upper clayey layer	Lower sandy layer
G_0^{ref}/G_{ur}	-	6.83	2.43

Figure 9–4 (p. 164) and Figure 9–5 (p. 164) 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. See 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.

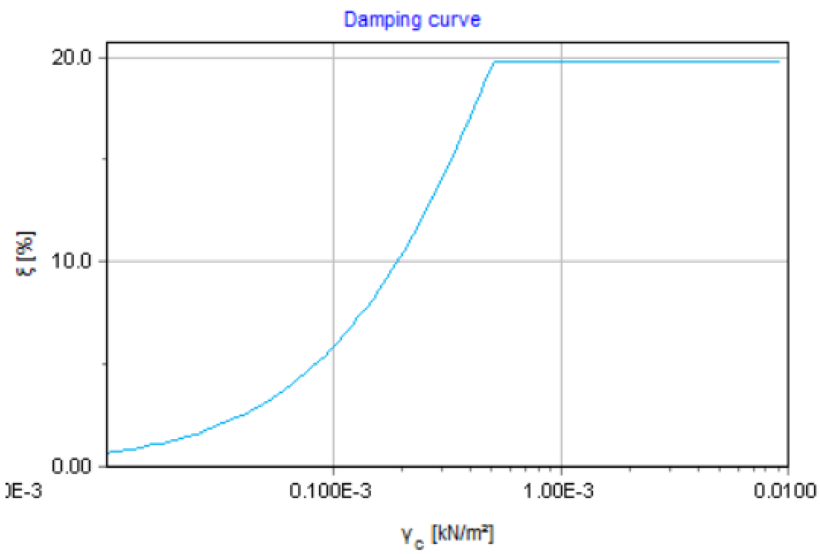

Figure 9–4: Damping curve for the upper clayey layer

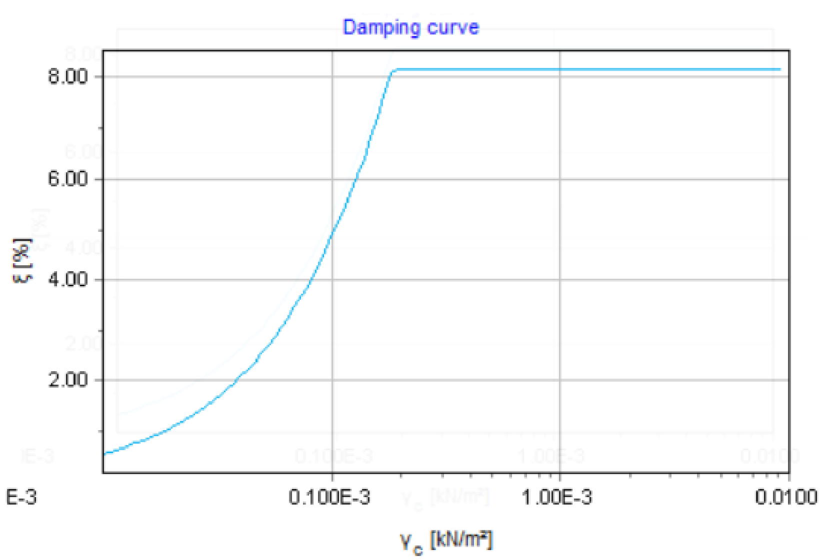

Figure 9–5: Damping curve for the lower sandy layer

1. Create the material data set according to Table 9–1 (p. 162) and
2. Assign it to the corresponding soil layers. The upper layer consists of mostly clayey soil and the lower one consists of sandy soil.

9.6 Definition of structural elements

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

9.6.1 Create a building

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

For the building two material data sets are needed and their corresponding material properties are described in Table 9–3 (p. 165):

Table 9–3: Material properties of the building (plate properties)

Parameter	Name	Rest of building	Basement	Unit
General				
Material type	Type	Elastic	Elastic	-
Unit weight	γ	33.33	50	kN/m^3
Rayleigh damping (Direct)	α	0.2320	0.2320	-
	β	$8\cdot10^{-3}$	$8\cdot10^{-3}$	-
Mechanical				
Isotropic	-	Yes	Yes	-
Young's modulus	E_1	$3\cdot10^7$	$3\cdot10^7$	kN/m^2
Poisson's ratio	ν_{12}	0	0	-
Thickness	d	0.3	0.3	m

9.6.1.1 Structure definition

To create the floors and walls of the structure:

1. Define a surface passing through the points (-5 0 -2), (5 0 -2), (5 3 -2) and (-5 3 -2).
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 2m.
3. 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.
4. Define a surface passing through the points (5 0 -2), (5 3 -2), (5 3 15) and (5 0 15).
5. 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 -10m.
6. Multiselect the vertical surfaces and the horizontal surface located at z = 0.

7. 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.
8. Select all the created surfaces representing the building (basement, floors and walls), right-click and select **Create** > **Create plate** option from the appearing menu.
9. Define the material data set for the plates representing the structure according to Table 9–3 (p. 165). Note that two different material data sets are used for the basement and the rest of the building respectively.
10. Assign the **Basement** material data set to the horizontal plate located at z = -2 and the vertical plates located under the ground level.
11. Assign the corresponding material data set to the rest of the plates in the model.
12. 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.

9.6.1.2 | Central column

The central column of the structure is modelled using the **Node-to-node anchor** feature. The modelling procedure is described as follows:

1. Create a **Line** through points (0 1.5 -2) and (0 1.5 0) corresponding to the column in the basement floor.
2. Create a **Line** through points (0 1.5 0) and (0 1.5 3) corresponding to the column in the first floor.
3. 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 3m.
4. Select the created lines, right-click and select the **Create** > **Create node-to-node anchor** option from the appearing menu.
5. Create the material data set according to Table 9–4 (p. 166) 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

9.6.2 | Create the loads

9.6.2.1 | Creation of the static load

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

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

9.6.2.2 Earthquake definition

The earthquake is modelled by imposing a prescribed displacement at the bottom boundary and assigning dynamic multipliers to the prescribed displacements:

- **To define the prescribed displacement:**
  1. Create a surface prescribed displacement passing through (-80 0 -40), (80 0 -40), (80 3 -40) and (-80 3 -40).
  2. 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:**
  1. 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.
  2. To add a multiplier click the corresponding button in the **Multipliers** window.
  3. From the **Signal** drop-down menu select the **Table** option.
  4. The file containing the earthquake data is available on Bentley Communities.
  5. 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.
  6. 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.
  7. Select the **Acceleration** option in the **Data type** drop-down menu.
  8. Select the **Drift correction** options and click **OK** to finalize the definition of the multiplier.
  9. In the **Dynamic multipliers** window the table and the plot of the data is displayed (See Figure 9–6 (p. 168)).

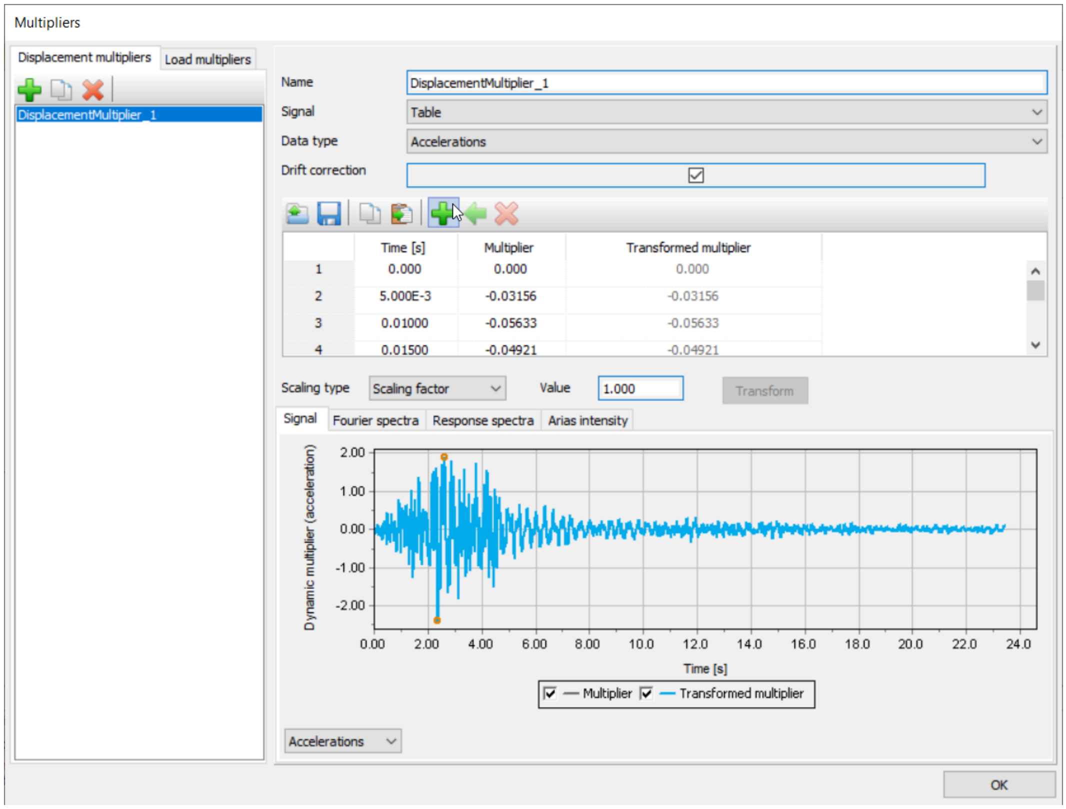

Figure 9–6: Dynamic multipliers window

10. In the **Model explorer** expand the **Surface displacements** subtree and in DynSurfaceDisplacement_1 assign the Multiplier $_x$ to the x- component by selecting the option in the drop-down menu.

9.6.3 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:

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

9.7 Generate the mesh

1. Proceed to the **Mesh Mode** .
2. Click the **Generate mesh** button. Set the element distribution to **Fine**.
3. View the generated mesh.

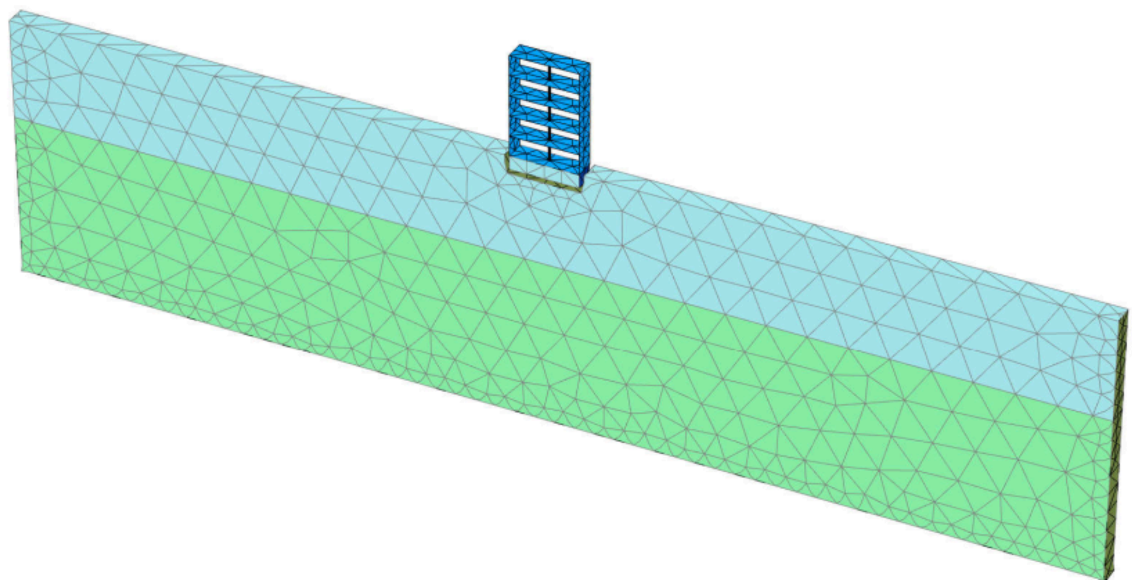

Figure 9–7: The generated mesh

9.8 Define and perform the calculation

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

9.8.1 Initial phase

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

9.8.2 Phase 1 - Building construction

1. Add a new calculation phase (Phase_1). The default settings of the added phase will be used for this calculation phase.
2. 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.

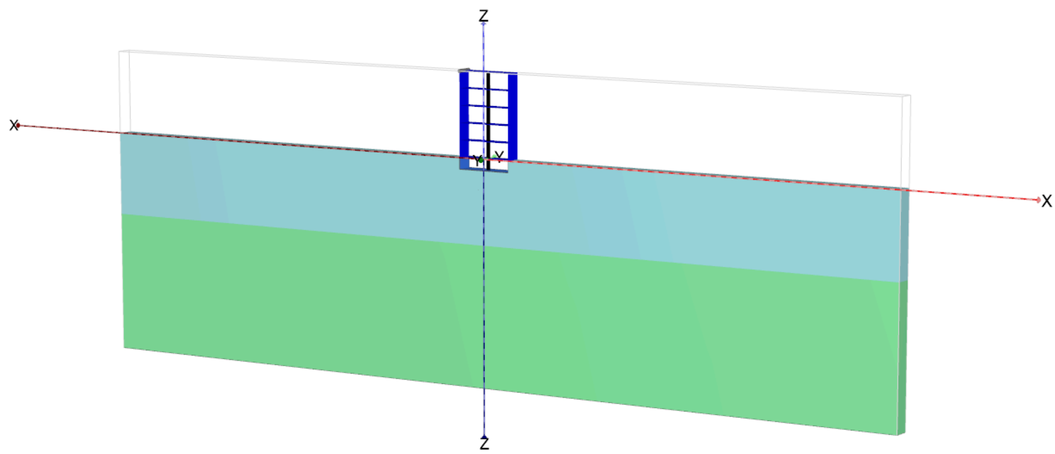

Figure 9–8: Construction of the building

9.8.3 | Phase 2 -Excitation

1. Add a new calculation phase (Phase_2).
2. In the **Phases** window in the **Deformation control parameters** subtree select the **Reset displacement to zero**. The default values of the remaining parameters will be used in this calculation phase.
3. In the **Staged construction mode** activate the line load. The value of the load is already defined in the **Structures mode** .

9.8.4 | Phase 3 - Free Vibration

1. Add a new calculation phase (Phase_3).
2. In the **Phases** window as **Calculation type** select the **Dynamic** option.
3. Set the **Time interval** parameter to 5sec.
4. In the **Staged construction mode** deactivate the line load.
5. In the **Model explorer** expand the **Model conditions** subtree.
6. Expand the **Dynamics** subtree. By default the boundary conditions in the x directions are set to viscous. Select the **None** option for the boundaries in the y direction. Set the boundary Zmin to **Viscous** (See Figure 9–9 (p. 171)).

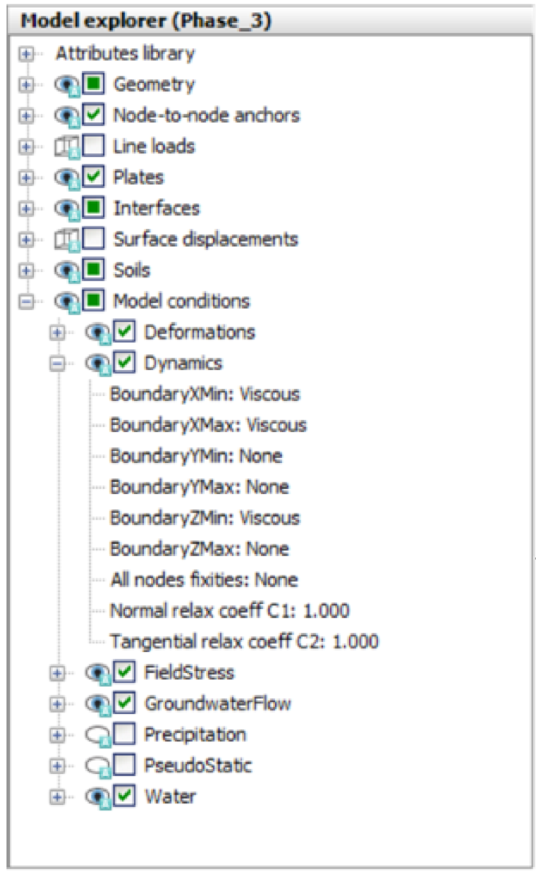

Figure 9–9: Boundary conditions for dynamics calculations (Phase_3)

Note: 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.

9.8.5 Phase 4 - Earthquake

1. Add a new phase (Phase_4).
2. In the **Phases** window set the **Start from phase** option to Phase 1 (construction of building).
3. As **Calculation type** select the **Dynamic** option.
4. Set the **Dynamic time interval** parameter to 20 sec.
5. In the **Deformation control parameters** subtree select the **Reset displacement to zero**. The default values of the remaining parameters will be used in this calculation phase.
6. In the **Numerical control parameters** subtree uncheck the **Use default iter parameters** checkbox.
7. Set the **Max steps** to 1000.

8 Set the **Time step determination** to **Manual** and the **Number of sub steps** to 4.

> **Note:**
>
> - The dynamic time interval is set to 20 seconds with a time interval of 0.005 seconds which gives 20/0.005=4000. So 4000 steps are required for the calculation. Therefore, the **Max steps** is set to 1000 and the **Number of sub steps** is set to 4. However, the automatic time stepping suggests smaller time steps due to one or two relatively small elements, which requires a sensitivity analysis for the time step size. For more information on time step for dynamic calculations please visit Bentley communities.
> - For more information on drift correction and input signal please visit Bentley communities.

9 In the **Model explorer** expand the **Model conditions** subtree.

10 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** (see Figure 9–10 (p. 173)).

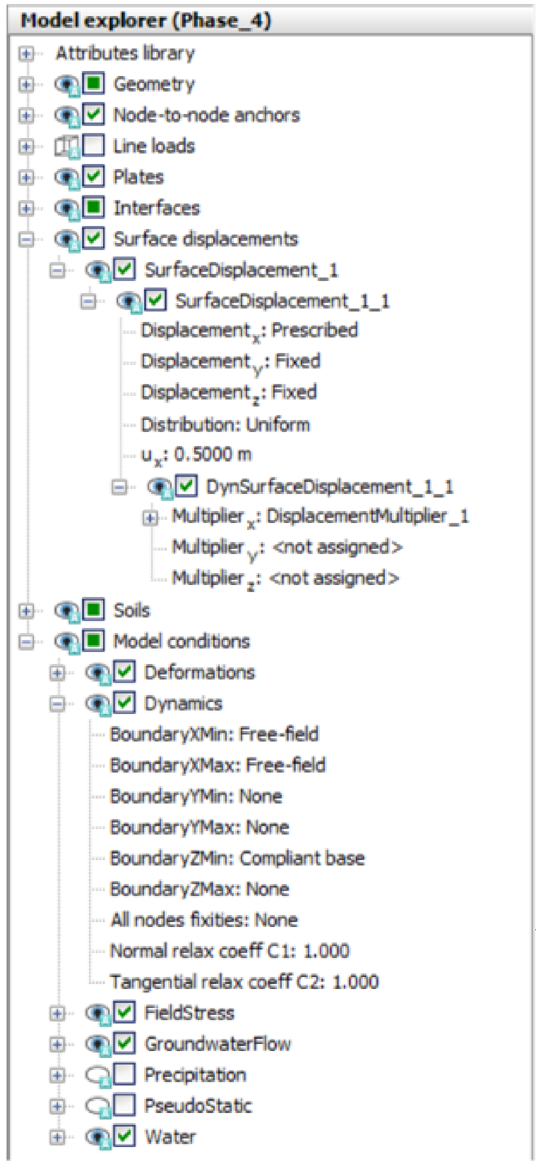

Figure 9–10: Boundary conditions for dynamics calculations (Phase_4)

11. Make sure that the interfaces on the boundary of the model are not activated in the **Model explorer**.

12. In the **Model explorer** activate the **Surface displacement** and its dynamic component. Set the value of u_x to 0.5m. 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.

9.8.6 Execute the calculation

1. Select points for load displacement curves at (0 1.5 15), (0 1.5 6), (0 1.5 3) and (0 1.5 -2).

2. Execute the calculation.

9.9 Results

Figure 9–11 (p. 174) shows the deformed structure at the end of the Phase 2 (application of horizontal load).

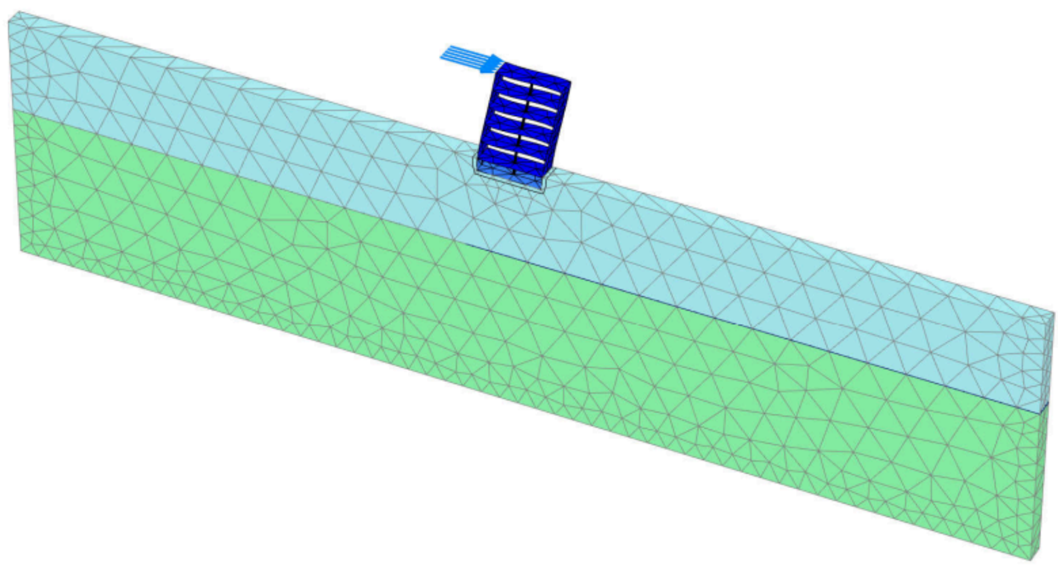

Figure 9–11: Deformed mesh of the system at the end of Phase_2

Figure 9–12 (p. 175) 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.

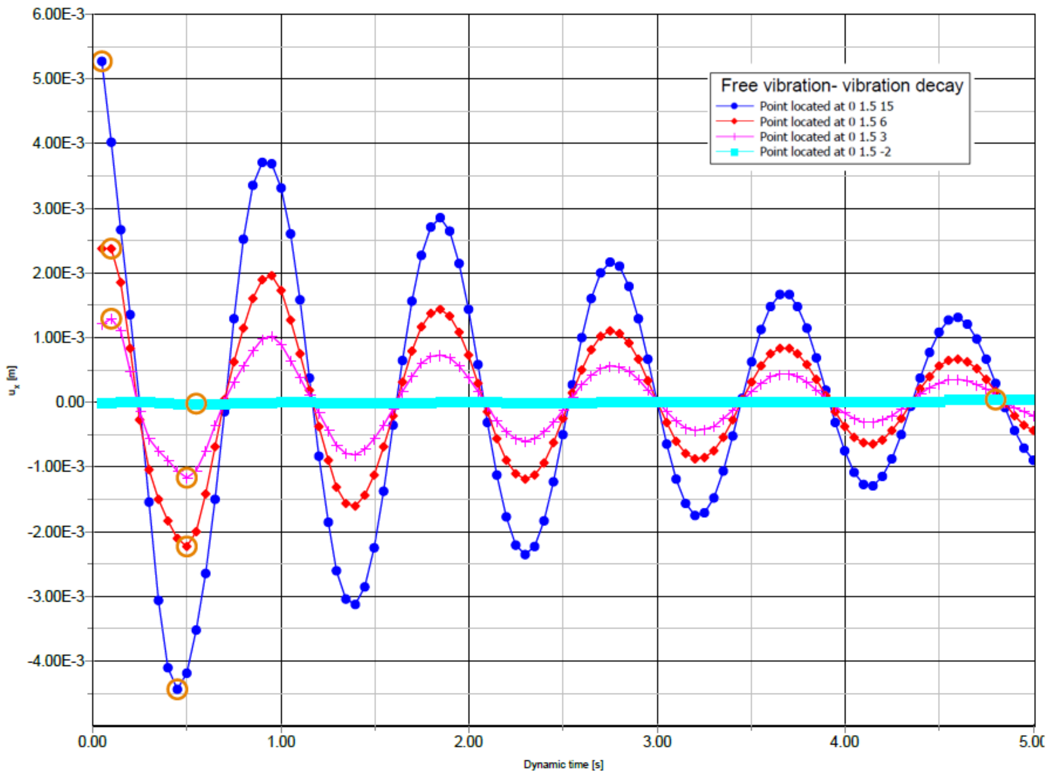

Figure 9–12: Time history of displacements (Free vibration)

In the **Curve generation** window under the **Fourier tabsheet** select **Power (spectrum)**, subsequently in the Total displacements subtree select u_x and click OK to create the plot. From Figure 9–13 (p. 176) 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.

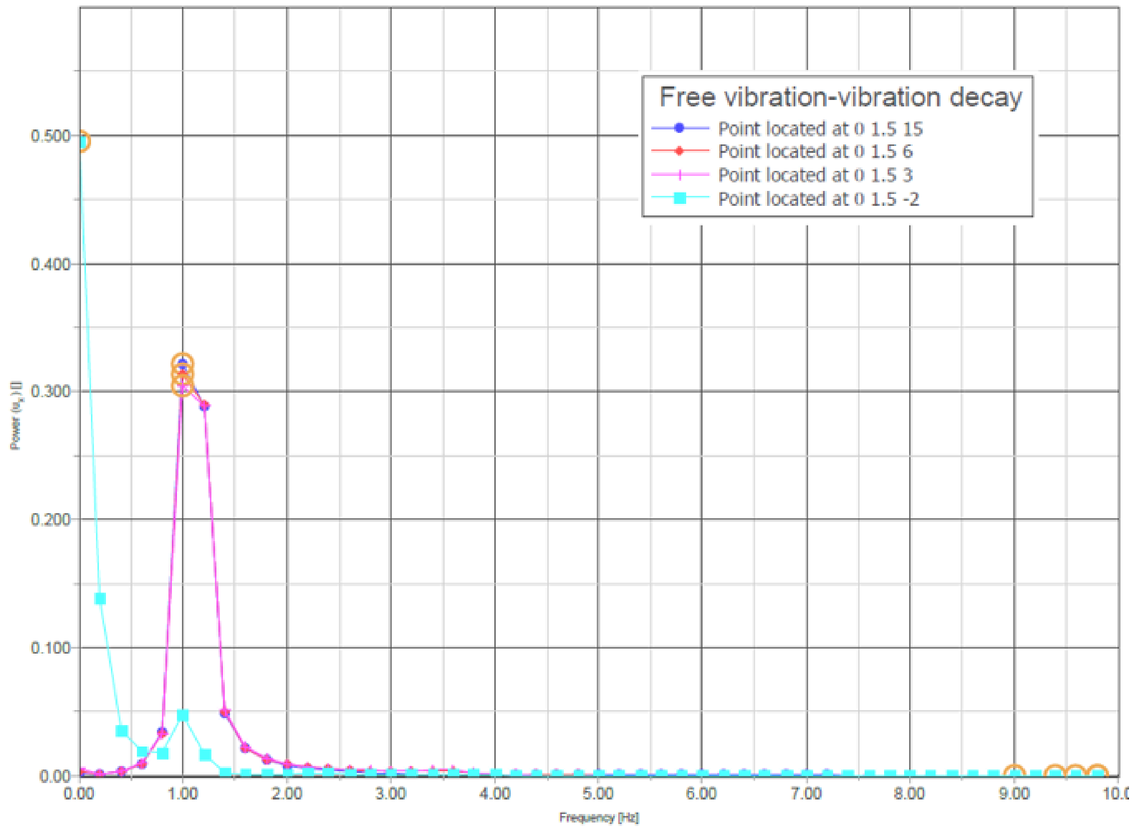

Figure 9–13: Frequency representation (spectrum - Free vibration)

Figure 9–14 (p. 176) 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.

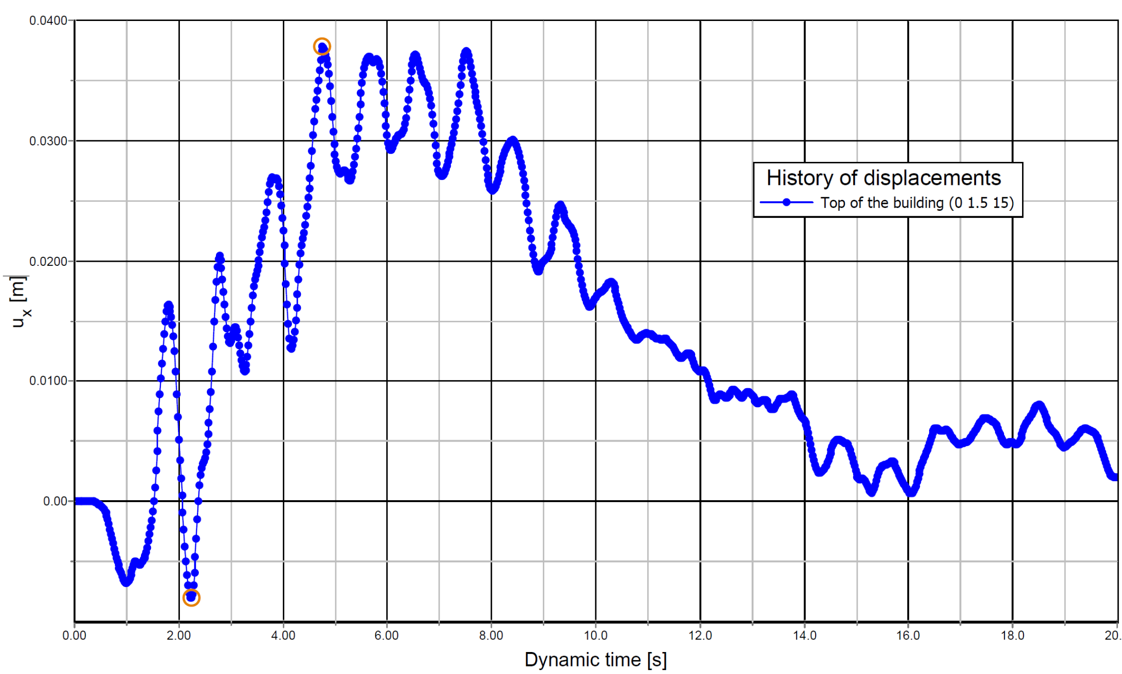

Figure 9–14: Time history of displacements of the top of the building (Earthquake)

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 (p. 177).

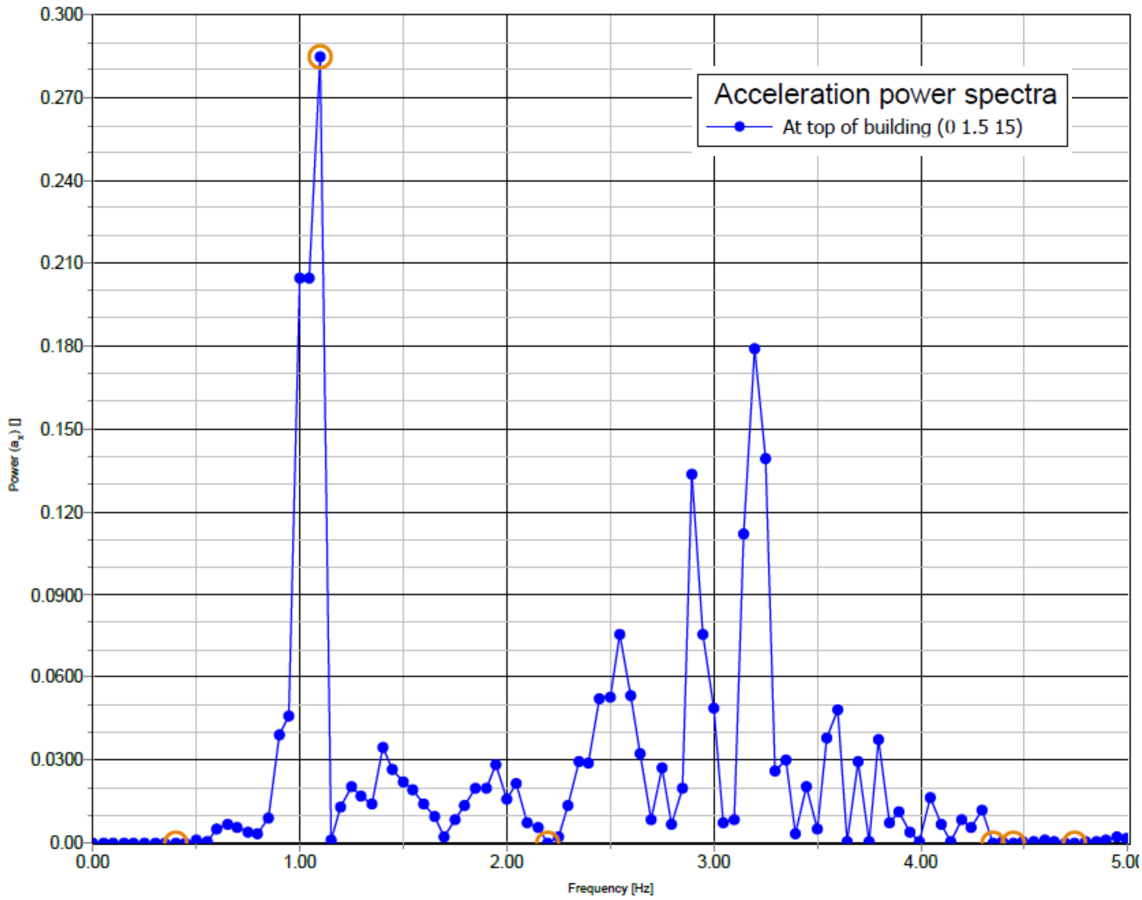

Figure 9–15: Acceleration power spectra at (0 1.5 15)