16

Free vibration and earthquake analysis of a building [ULT]

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.

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 Hardening Soil model with small-strain stiffness
- Evaluating the natural frequency from Fourier spectrum

Geometry

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 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 rather loose sand of 15 m depth underlain by a deep denser sand layer. In the model, 25 m of the deep sand layer will be considered as shown in .
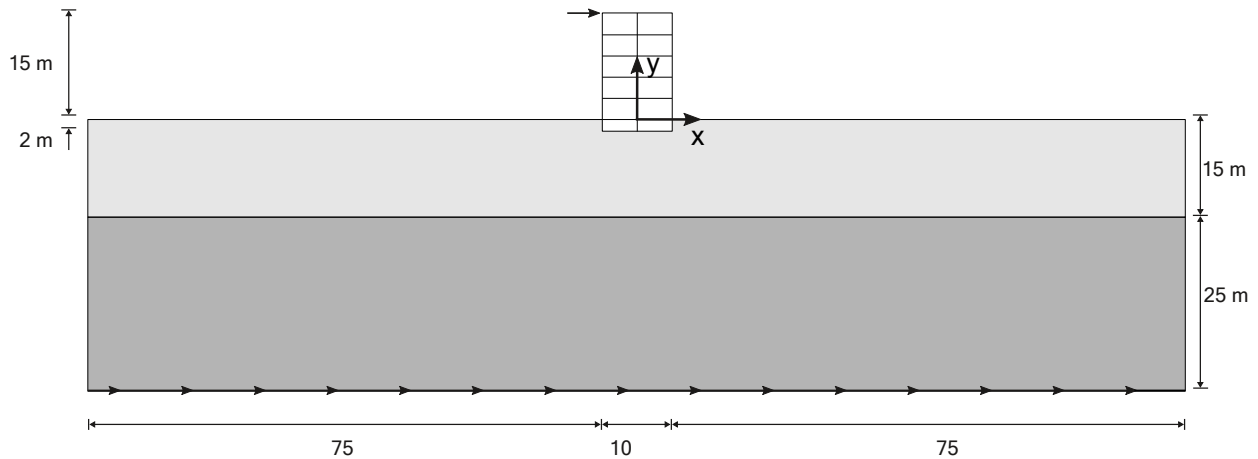
Figure 211: Geometry of the project

16.1 Create new project

To create the new project, follow these steps:

1. Start the Input program and select **Start a new project** from the **Quick start** dialog box.
2. In the **Project** tabsheet of the **Project properties** window, enter an appropriate title.
3. In the **Model** tabsheet keep the default options for **Model** (**Plane strain**) and **Elements** (**15-Noded**).
4. Keep the default units and constants and set the model **Contour** to x_{min} = -80 m, x_{max} = 80 m, y_{min} = -40 m and y_{max} = 15 m.

16.2 Define the soil stratigraphy

The subsoil is divided into a 15 m thick rather loose sand layer and a 25 m thick medium dense sand layer. The phreatic level is assumed to be at y = -15 m. Hydrostatic pore pressures are generated in the whole geometry according to this phreatic line.

1. Click the **Create borehole** button and create a borehole at x = 0.
   The **Modify soil layers** window pops up.
2. Add two soil layers extending from y = 0 to y = -15 and from y = -15 to y = -40.
3. Set the **Head** in the borehole at -15 m.

16.3 Create and assign material data sets

The upper layer consists of rather loose sand and the lower one is a medium dense sand. Both have Hardening Soil model with small-strain stiffness properties. The presence of the groundwater is neglected. The soil layers with Hardening Soil model with small-strain stiffness properties have inherent hysteretic damping.

- ![icon] Open the **Material sets** window.
- Create data sets under **Soil and interfaces** set type according to the information given in Table 34 (on page 241).
- Assign the material datasets to the corresponding soil layers in the borehole.

Table 34: Material properties

Parameter	Name	Upper sand layer	Lower sand layer	Unit
General				
Soil model	-	HS small	HS small	-
Drainage 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}	$20 \cdot 10^3$	$30 \cdot 10^3$	kN/m^2
Tangent stiffness for primary oedometer loading	E_{oed}^{ref}	$26 \cdot 10^3$	$36 \cdot 10^3$	kN/m^2
Unloading / reloading stiffness	E_{ur}^{ref}	$95 \cdot 10^3$	$110 \cdot 10^3$	kN/m^2
Poisson's ratio	v'_{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}	$270 \cdot 10^3$	$100 \cdot 10^3$	kN/m^2
Shear strain at which $G_s = 0.722\ G_0$	$\gamma_{0.7}$	$0.15 \cdot 10^{-3}$	$0.1 \cdot 10^{-3}$	-
Cohesion	c'_{ref}	10	5	kN/m^2

Mechanical				
Friction angle	φ'	31	28	°
Dilatancy angle	ψ	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. The figures below display the Modulus reduction curves, i.e. the decay of the shear modulus with strain. Figure 212 (on page 242) shows the secant shear modulus and Figure 213 (on page 243) shows the tangent shear modulus.
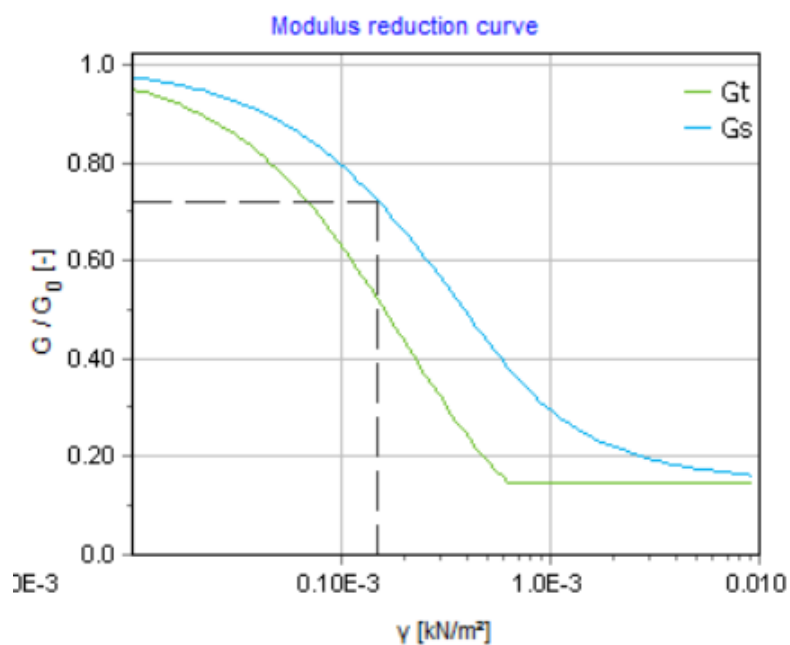
Figure 212: Modulus reduction curves for the upper sand layer
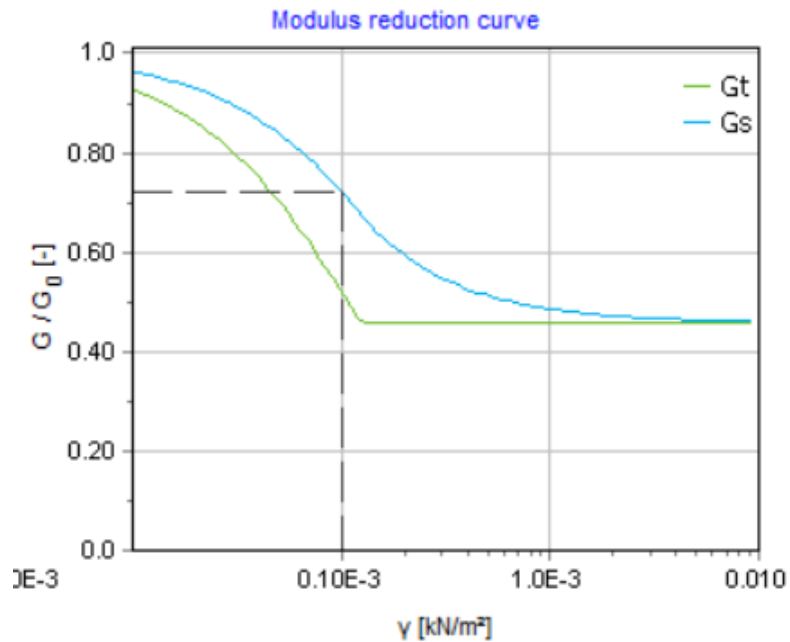
Figure 213: Modulus reduction curve for the lower sand layer

In the Hardening Soil model with small-strain stiffness, the tangent shear modulus is bound by a lower limit, G_{ur}.

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

The values of G_{ur}^{ref} for the **Upper sand layer** and **Lower sand layer** and the ratio to G_0^{ref} are shown in Table 35 (on page 243) . This ratio determines the maximum damping ratio that can be obtained.

Table 35: G_{ur} values and ratio to G_0^{ref}

Parameter	Upper sand layer	Lower sand layer	Unit
G_{ur}	39.10^3	45.10^3	kN/m^2
G_0^{ref}/G_{ur}	6.82	2.18	-

Figure 214 (on page 244) and Figure 215 (on page 244) show the damping ratio as a function of the shear strain for the material used in the model. A more detailed description about the modulus reduction curve to the damping curve can be found in the literature.[6]

---

[6] 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.
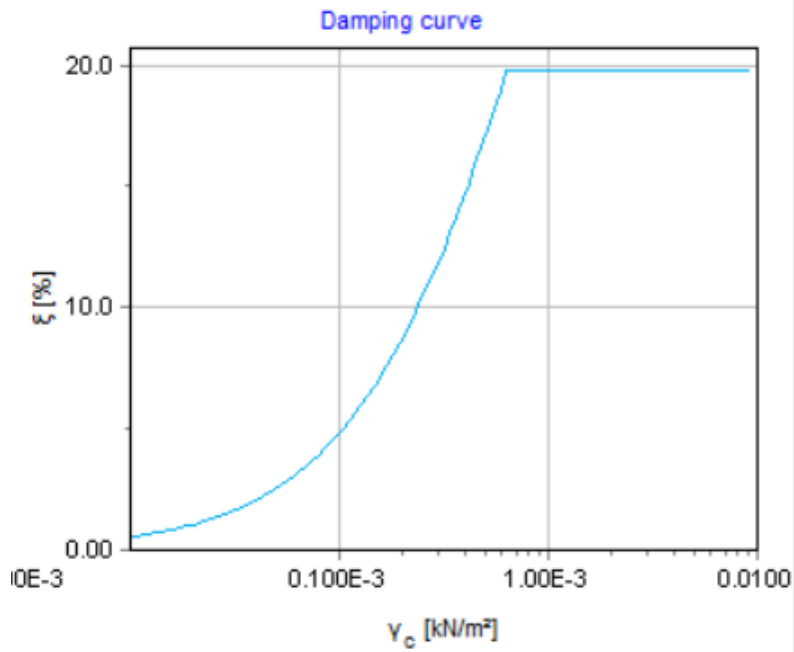
Figure 214: Damping curve for the upper sand layer
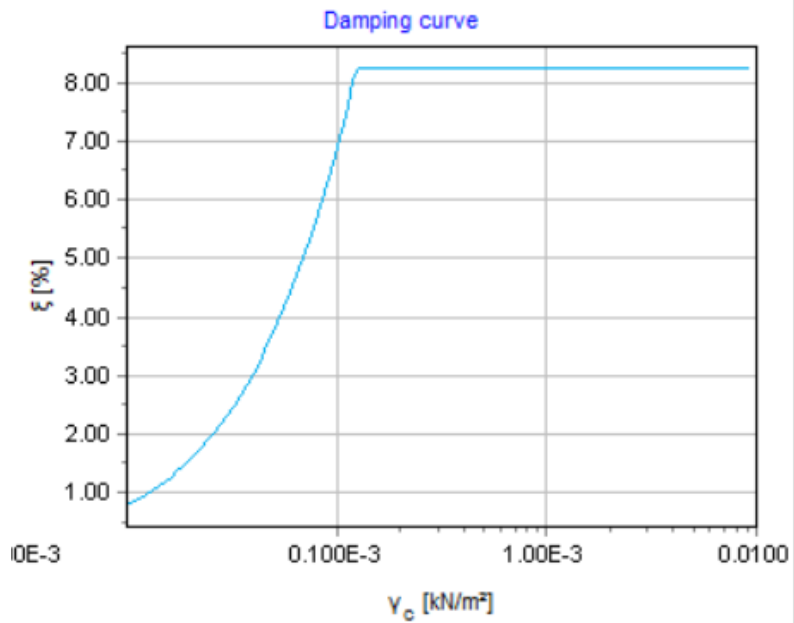
Figure 215: Damping curve for the lower sand layer

16.4 Define the structural elements

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

16.4.1 Define the 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/m^2 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.

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 the Reference Manual .

Table 36: Material properties of the building (plate properties)

Parameter	Name	Building	Basement	Unit
General				
Material type	-	Elastic	Elastic	-
Weight	w	10	20	kN/m/m
Input method	-	Direct	Direct	-
Rayleigh α	-	0.2320	0.2320	-
Rayleigh β	-	$8.0{\cdot}10^{-3}$	$8.0{\cdot}10^{-3}$	-
Prevent punching	-	No	No	-
Mechanical				
Isotropic	-	Yes	Yes	-
Axial stiffness	EA_1	$9{\cdot}10^6$	$12{\cdot}10^6$	kN/m
Bending stiffness	EI	$67.5{\cdot}10^3$	$160{\cdot}10^3$	kNm^2/m
Poisson's ratio	ν	0	0	-

Table 37: Material properties of the node-to-node anchor

Parameter	Name	Column	Unit
Material type	Type	Elastic	-
Out-of-plane spacing	$L_{spacing}$	3	m
Axial stiffness	EA	$2.5 \cdot 10^6$	kN

1. ⬍ Use plates to create the vertical walls of the building from (-5 0) to (-5 15) and from (5 0) to (5 15).
2. Again with plates, now define the vertical walls of the basement from (-5 -2) to (-5 0) and from (5 -2) to (5 0).
3. Define the basement and ground level of the building as plates from (-5 -2) to (5 -2) and from (-5 0) to (5 0).
4. Define the floors by copying the basement floor 5 times. To do so, select the basement floor and choose the

   Array button ⬚. Now specify that we want to copy it in y-direction, 6 copies (be aware: number of copies includes the original) and an intermediate distance of 3 m.
5. 🖩 Define the material datasets for the structural elements in the building according to Table 36 (on page 245).
6. Assign the Basement material dataset to the vertical plates (2) and the lowest horizontal plate (all below ground level) in the model.
7. Assign the Building material dataset to the remaining plates in the model.
8. ⌇ Use the Node-to-node anchor feature to define the column at the centre of the building connecting consecutive floors, hence (0 -2) to (0 0), (0 0) to (0 3), (0 3) to (0 6), (0 6) to (0 9), (0 9) to (0 12) and (0 12) to (0 15). Of course this can also be done by drawing one column and use the Array function to copy the others.
9. Define the properties of the anchor according to Table 37 (on page 246) and assign the material dataset to the anchors in the model.
10. ⊞ Define an interface to model the interaction between soil and the building.

16.4.2 Define the loads

1. In order to model the driving force, a point load is created on top of the pile. To create a dynamic load:

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

   b. Set $F_{x,ref}$ = 10 kN/m and $F_{y,ref}$ = 0 kN/m.
2. The earthquake is modelled by imposing a prescribed displacement at the bottom boundary. To define the prescribed displacement:

   a. ⬍ Define a line displacement at the bottom of the model, from (-80 -40) to (80 -40).

   b. Set the x-component of the line displacement to **Prescribed** and assign a value of 1.0. The y-component of the line displacement is **Fixed**. The default distribution (Uniform) should be kept.
3. To define the dynamic multipliers for the line displacement:

   a. Expand the Dynamic line displacement.

b. 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.

c. From the **Signal** drop-down menu select the **Table** option.

d. The file containing the earthquake data is available in the [Bentley Communities](#): search for the tutorial Free vibration and earthquake analysis of a building. Download the earthquake signal file `225a.smc`.

e. In the **Multipliers** window click the **Open** button. In the appearing window change in the drop-down menu **Plain text files *.txt** for **Strong motion CD-ROM files** option and select the appearing saved .smc file.

f. Select the **Acceleration** option in the **Data type** drop-down menu.

g. Select the **Drift correction** options and click **OK** to finalize the definition of the multiplier.
The defined multiplier is displayed in [Figure 216](#) (on page 247) :
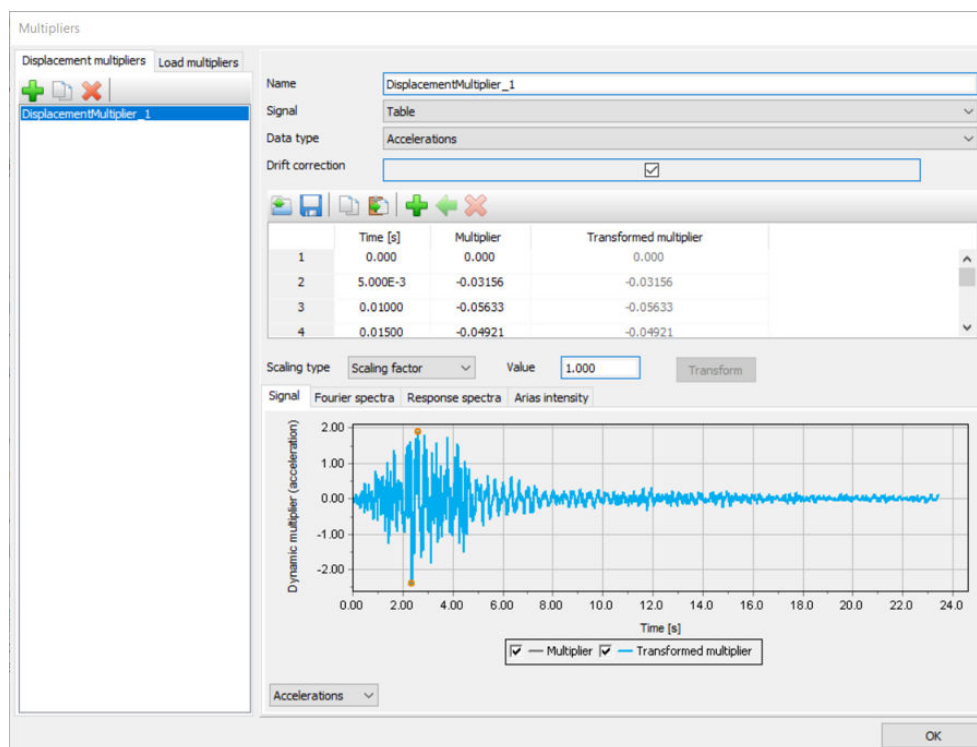
Figure 216: Dynamic multipliers window

16.4.3 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:

1. ▦ 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 217](#) (on page 248) :
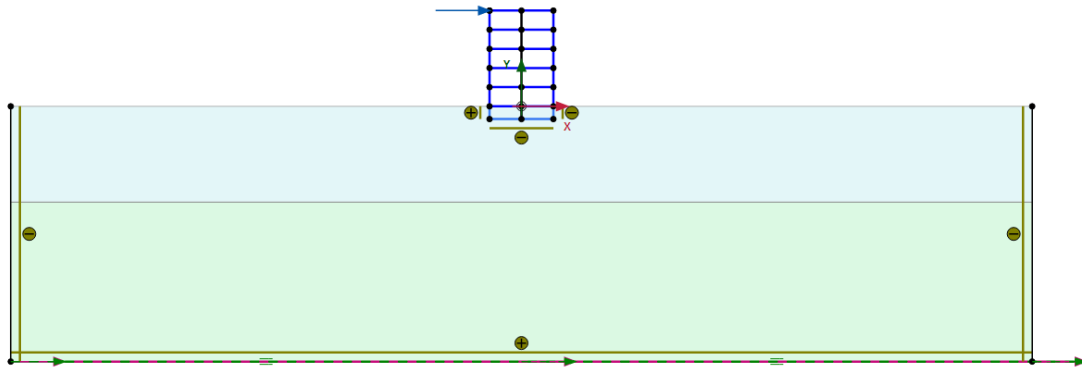
Figure 217: Geometry of the model

16.5 Generate the mesh

1. Proceed to the **Mesh mode** .
2. Reset the mesh refinement on the boundaries by changing the **Coarseness factor** on the boundaries to 1.
3. Select both soil layers and set their **Coarseness factor** to 0.3.
4. Click the **Generate mesh** button to generate the mesh. Set the **Element distribution** to **Medium**.
5. Click the **View mesh** button to view the mesh.
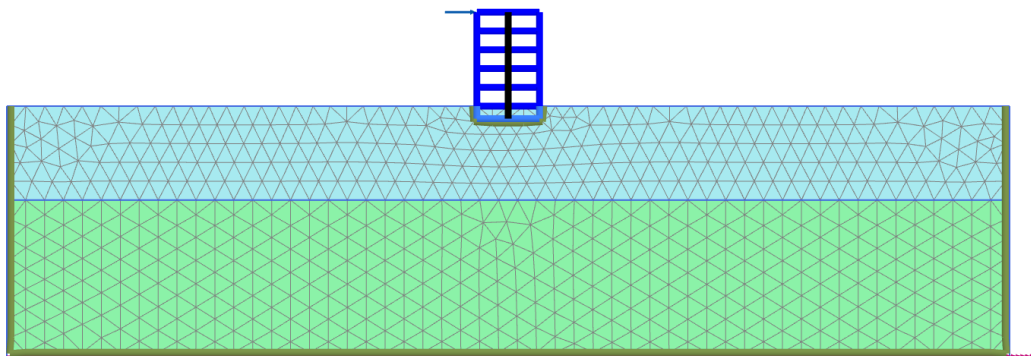   The resulting mesh is shown in Figure 218 (on page 248).:

Figure 218: The generated mesh

6. Click the **Close** tab to close the Output program.

16.6 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.

16.6.1 Initial phase

1. Click on the **Staged construction** tab to proceed with 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.
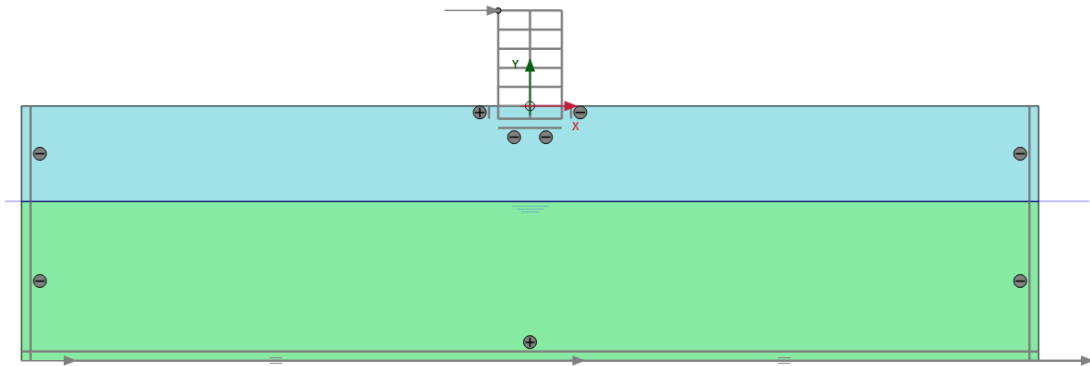4. The model for the initial phase is shown in Figure 219 (on page 249).

Figure 219: Initial phase

16.6.2 Phase 1: Building

1. Click the **Add phase** button to create a new phase. 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.
3. The model for phase 1 is shown in Figure 220 (on page 249).
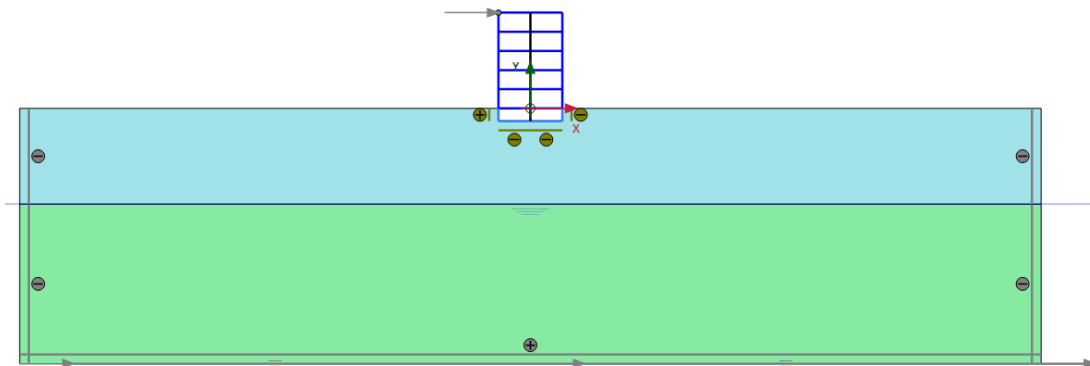
Figure 220: Construction of the building

16.6.3 Phase 2: Excitation

1. Click the **Add phase** button 🔲 to create a new phase (Phase_2).
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.
3. In the **Staged construction mode** activate the load. The value of the load is already defined in the **Structures mode** .

16.6.4 Phase 3: Free vibration

1. Click the **Add phase** button 🔲 to create a new phase (Phase_3).
2. In the **Phases** window select the **Dynamic** option 〜 as **Calculation type**.
3. Set the **Dynamic time interval** parameter to 5 sec.
4. In the **Staged construction** mode release (deactivate) the point load.
5. In the **Model explorer** expand the **Model conditions** subtree.
6. Expand the **Dynamics** subtree. Check the boundary conditions **BoundaryXMin**, **BoundaryXMax** and **BoundaryYMin** are viscous which is shown in
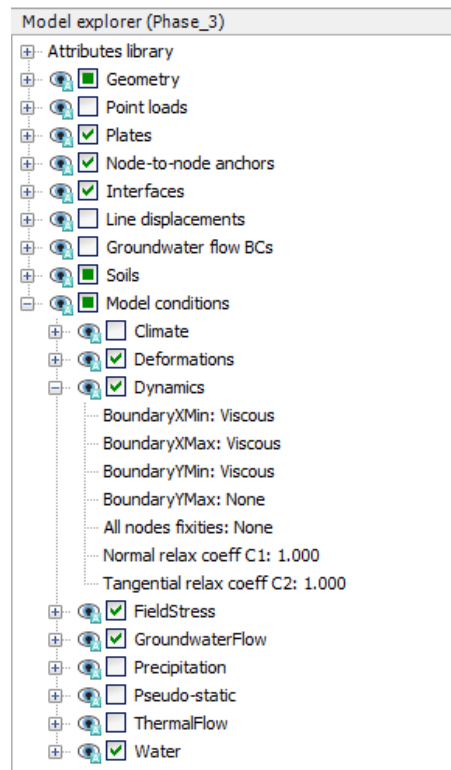
*Figure 221: Boundary conditions for **dynamics calculations***

> **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.

16.6.5 Phase 4: Earthquake

1. Click the **Add phase** button 🔲 to create a new phase (Phase_4).

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

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

4. Set the **Dynamic time interval** parameter to 20 sec.

5. Select the **Reset displacement to zero** in the **Deformation control parameters** subtree.

6. In the **Numerical control parameters** unselect the option **Use default iter parameters** and set **Max steps** to 1000 in order to get a more detailed time-acceleration curve.

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

8. Expand the **Dynamics** subtree. Set the **BoundaryXMin** and **BoundaryXMax** to **Free-field**. Set the **BoundaryYMin** to **Compliant base** as shown in Figure 222 (on page 252).
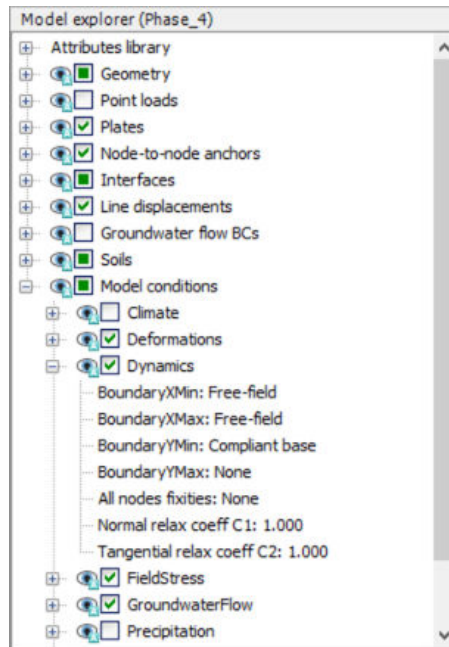
*Figure 222: Boundary conditions for **dynamics calculations** (Phase_4)*

9. Interface elements do not need to be active to enable the use of **Free-field** or **Compliant base** boundaries.
10. 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.

16.6.6 Execute the calculation

1. Click the **Select points for curves** button in the side toolbar and select a point at the top of the building, close to or at (0 15).
2. Click the **Calculate** button to calculate the project.
3. After the calculation has finished, save the project by clicking the **Save** button .

16.7 Results

Figure 223 (on page 253) shows the deformed structure at the end of the Phase 2 (application of horizontal load).
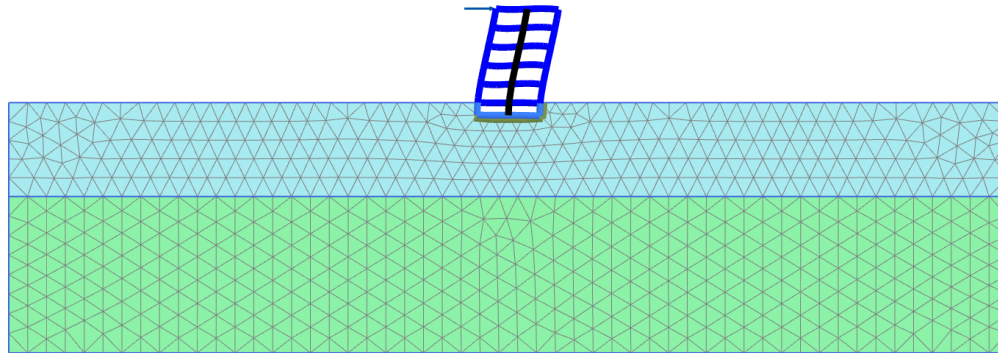
Figure 223: Deformed structure in phase 2

Figure 224 (on page 253) 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.
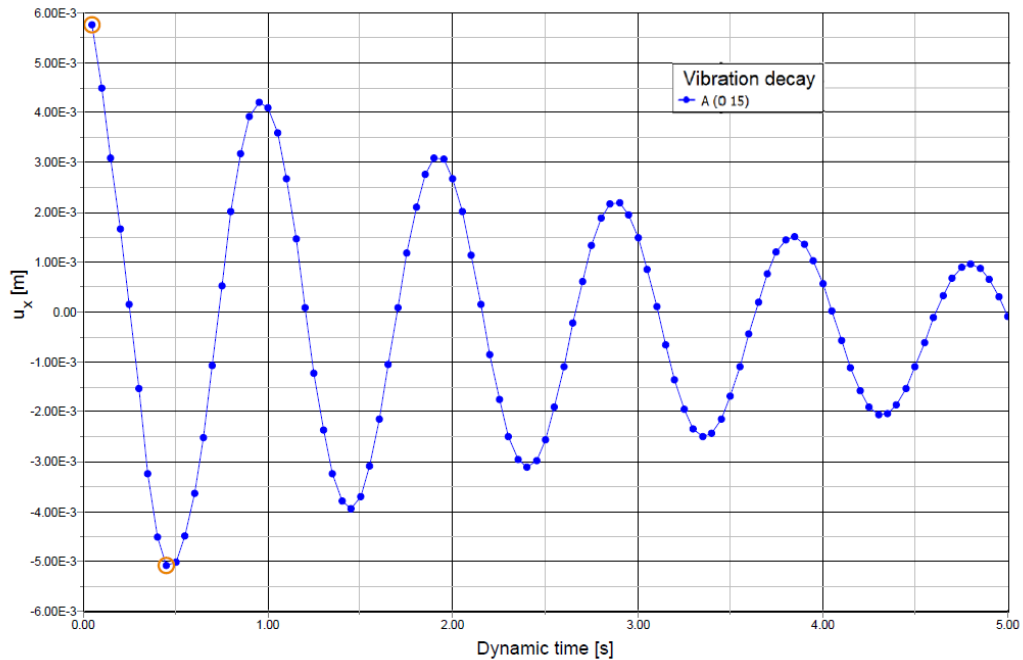

Figure 224: Time history of displacements at selected points

In the **Fourier** tabsheet of the **Curve generation** window select the **Power (spectrum) > Total displacements > Ux** and click **OK** to create the plot. The plot is shown in Figure 225 (on page 254). From this figure it can be evaluated that the dominant building frequency is around 1 Hz.
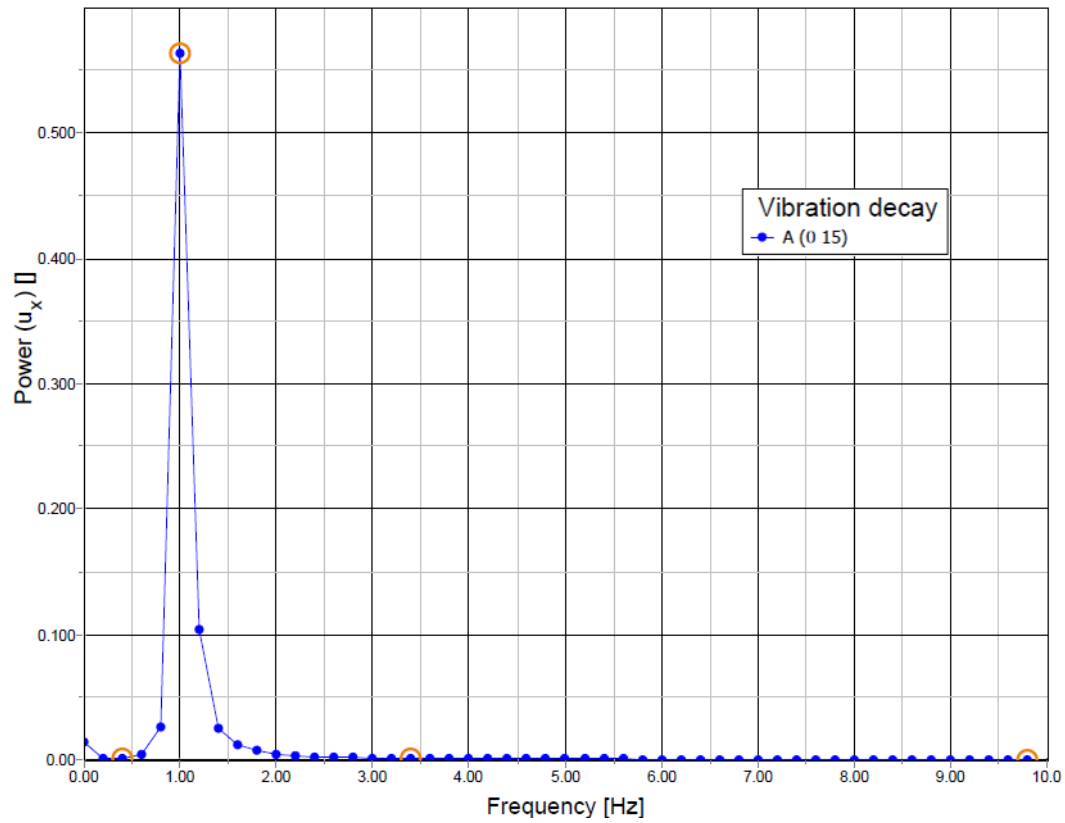
Figure 225: Frequency representation (spectrum)

Figure 226 (on page 255) shows the time history of the lateral acceleration of the selected point at (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.
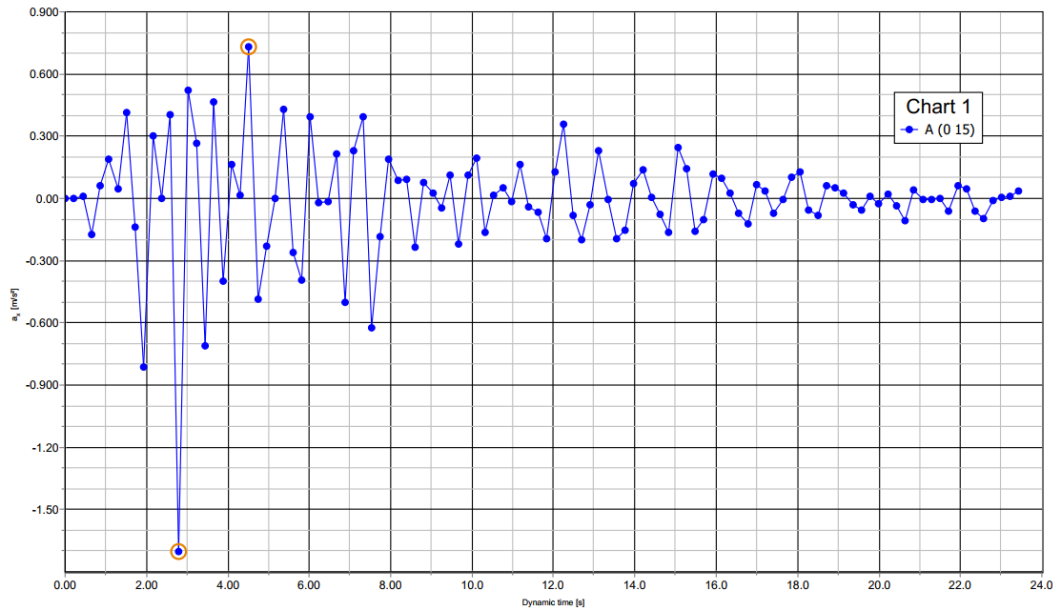
Figure 226: Variation of acceleration in dynamic time