8

Dynamics analysis of a generator on an elastic foundation [ULT]

In this tutorial the influence of a vibrating source on its surrounding soil is studied. To reduce the calculation time, only one-quarter of the overall geometry is modelled, using symmetry boundary conditions along the lines of symmetry. The physical damping due to the viscous effects is taken into consideration via Rayleigh damping. Also, due to radial wave propagation, 'geometric damping' can be significant in attenuating the vibration.

The modelling of the boundaries is one of the key points in the dynamics calculation. In order to avoid spurious wave reflections at the model boundaries (which do not exist in reality), special conditions have to be applied in order to absorb waves reaching the boundaries.

Objectives

- Performing a **Dynamic** calculation
- Defining dynamic boundary conditions (viscous)
- Defining dynamic loads by means of load multipliers
- Defining material damping by means of Rayleigh damping

Geometry

The vibrating source is a generator founded on a 0.2 m thick concrete footing of 1 m in diameter. Oscillations caused by the generator are transmitted through the footing into the subsoil. These oscillations are simulated as a uniform harmonic loading, with a frequency of 10 Hz and amplitude of 10 kN/m^2. In addition to the weight of the footing, the weight of the generator is modelled as a uniformly distributed load of 8 kN/m^2.
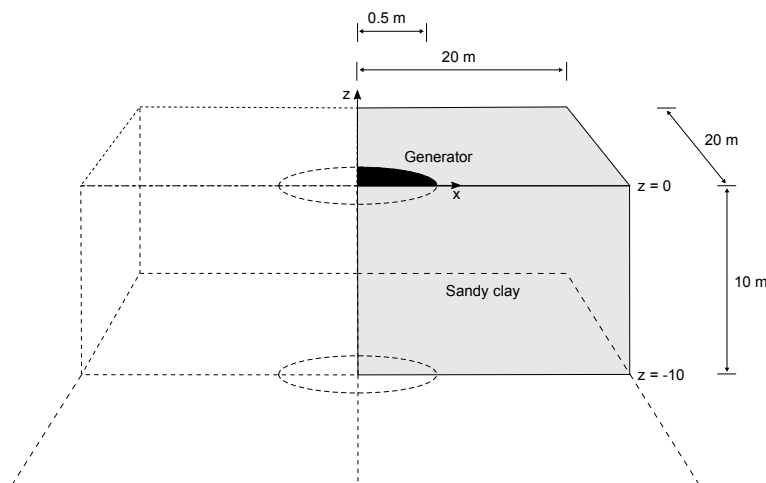

Figure 108: Generator founded on elastic subsoil

The model boundaries should be sufficiently far from the region of interest, to avoid disturbances due to possible reflections. Although special measures (absorbent boundaries) are adopted in order to avoid spurious reflections, there is always a small influence and it is still a good habit to put boundaries far away. In a dynamics analysis, model boundaries are generally taken further away than in a static analysis.

8.1 Create a new project

To create the geometry model, 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.** $x_{min} = 0$ and $x_{max} = 20$
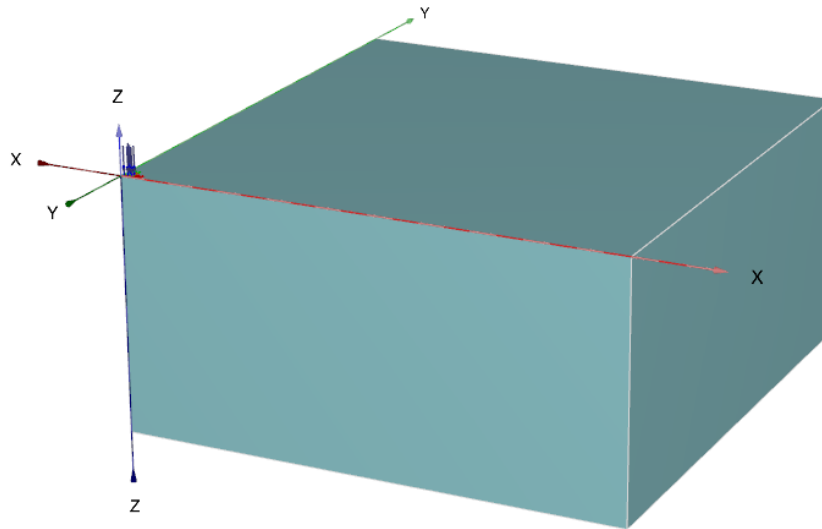   - **b.** $y_{min} = 0$ and $y_{max} = 20$

Figure 109: The geometry of the model

8.2 Define the soil stratigraphy

1. The subsoil consists of one layer with a depth of 10 m. The ground level is defined at $z = 0$.
2. Note that water conditions are not considered in this example and the hydraulic head is set at $z = -10$.

8.3 Create and assign material data sets

Create the material data set according to the table below and assign it to the soil layer.

Table 20: Material properties

Parameter	Name	Sandy clay	Unit
General			
Material model	Model	Linear elastic	-
Drainage type	Type	Drained	-
Unit weight above phreatic level	γ_{unsat}	20.0	kN/m^3
Unit weight below phreatic level	γ_{sat}	20.0	kN/m^3
Parameters			
Young's modulus	E'	$5 \cdot 10^4$	kN/m^2
Poisson's ratio	ν'	0.3	-
Interfaces			
Interface strength	-	Rigid	-
Initial			
K$_0$ determination	-	Automatic	-
Lateral earth pressure coefficient	K_0	0.5	-

8.4 Definition of structural elements

The generator is defined in the **Structures** mode. The **Polycurve** feature is used to define the geometry.

1. Click the **Create polycurve** button in the side toolbar and click on (0 0 0) in the drawing area.
2. In the **General** tabsheet the default option for shape (**Free**) and the default orientation axes (x-axis, y-axis) are valid for this polycurve.
3. In the **Segments** tabsheet three segments are defined as given in the table below. The insertion point is located at (0 0 0).

Table 21: Segment properties

Segment	Segment 1	Segment 2	Segment 3
Segment type	Line	Arc	Line
Segment properties	Relative start angle = 0° Length = 0.5 m	Relative start angle = 90° Radius = 0.5 m Segment angle = 90°	Relative start angle = 90° Length = 0.5 m

4. Right-click the polycurve and select the **Create surface** option from the appearing menu.
5. Right-click the created surface and select the **Create surface load** option in the appearing menu.
6. In the **Selection explorer**, the **Uniform** distribution is valid for the surface load. Assign (0 0 -8) to the pressure components.

8.4.1 Definition of dynamic multipliers

Dynamic loads are defined on the basis of input values of loads or prescribed displacements and corresponding time-dependent multipliers.

To create the multipliers of the dynamic load:

1. In the **Model explorer** expand the **Attributes library** subtree.
2. Right-click the **Dynamic multipliers** subtree and select the **Edit** option from the appearing menu.

   The **Multipliers** window pops up.
3. Click the **Load multipliers** tab.
4. Click **Add** button to introduce a multiplier for the loads.
5. Define a **Harmonic** signal with an **Amplitude** of 10, a **Frequency** of 10 Hz and a **Phase** of 0° as shown below:
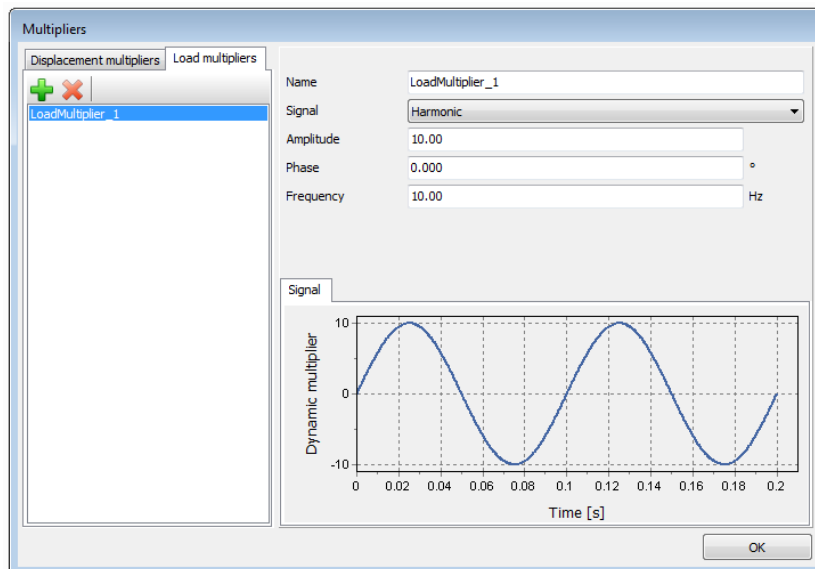

Figure 110: Definition of a Harmonic multiplier

6. In the **Selection explorer**, under DynSurfaceLoad_1 specify the components of the load as (0 0 -1).

7. Click Multiplier$_z$ in the dynamic load subtree and select the LoadMultiplier_1 option from the appearing menu.

Note:

The dynamic multipliers can be defined in the **Geometry** modes as well as in the **Calculation** modes.

8.5 Generate the mesh

1. Proceed to the **Mesh** mode.

2. Refine the surface corresponding to the generator by assigning a **Coarseness factor** of 0.125.

3. Click the **Generate mesh** button. The **Medium** option will be used for **Element distribution**.

4. View the generated mesh.
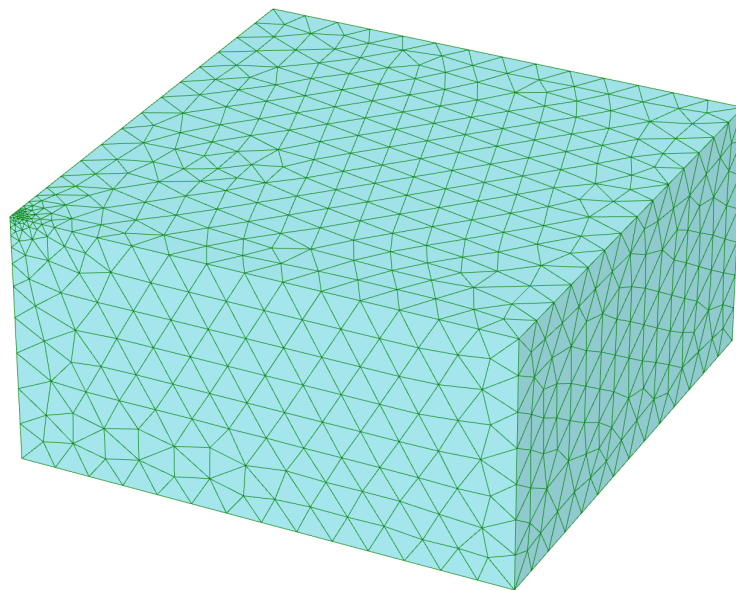
Figure 111: The generated mesh

Note: In all dynamics calculations, the user should pay special attention to the element size to decrease numerical dispersion of waves. It should be noted that large elements are not able to transmit high frequencies. The transmission of waves is governed by both wave speed and wave length. If dynamic input contains high frequencies, either high frequencies should be filtered out or a finer mesh should be used.

8.6 Define and perform the calculation

The calculation consists of 4 phases. The initial phase consists of the generation of the initial stresses using the **K0 procedure**. The second phase is a **Plastic** calculation where the static load is activated. The third phase is a **Dynamic** calculation where the effect of the functioning generator is considered. The fourth and final phase is a **Dynamic** calculation as well where the generator is turned off and the soil will vibrate freely.

8.6.1 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.

8.6.2 Phase 1

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 activate the static component of the surface load. Do not activate the dynamic load.
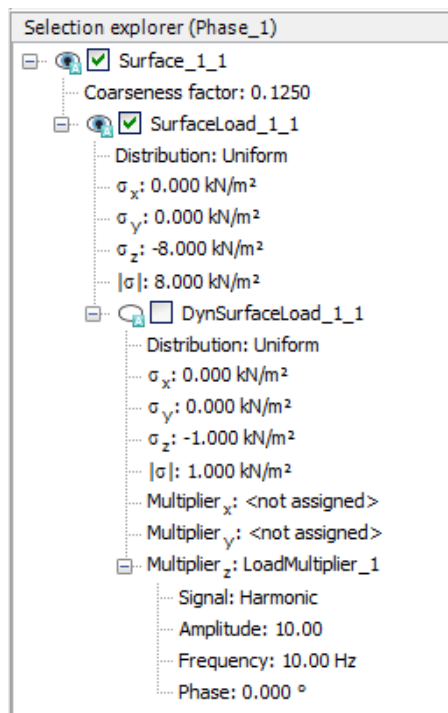
Figure 112: Applied load in the Phase_1

8.6.3 Phase 2

1. Add a new calculation phase (Phase_2).

2. In the **General** subtree in the **Phases** window, select the **Dynamic** option as calculation type.

3. Set the **Time interval** parameter to 0.5 s.

4. In the **Deformation control parameters** subtree in the **Phases** window select the **Reset displacement to zero** parameter. The default values of the remaining parameters will be used for this calculation phase.

5. 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**.

6. Set the **Max steps** to 250.

7. In the **Staged construction** mode activate the dynamic component of the surface load. Note that the static component of the load is still active.
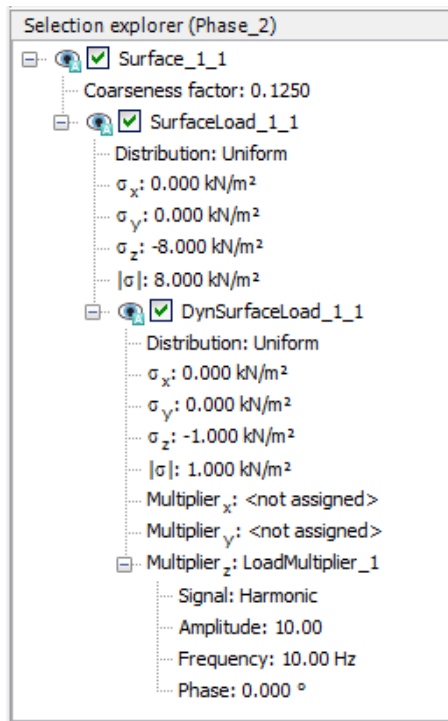
Figure 113: Applied load in the Phase_2

Special boundary conditions have to be defined to account for the fact that in reality the soil is a semi-infinite medium. Without these special boundary conditions the waves would be reflected on the model boundaries, causing perturbations. To avoid these spurious reflections, viscous boundaries are specified at Xmax, Ymax and Zmin. The dynamic boundaries can be specified in the **Dynamics** subtree located under the **Model conditions** in the **Model explorer**.

Figure 114: Boundary conditions for dynamics calculations

8.6.4 Phase 3

1. Add a new calculation phase (Phase_3).
2. In the **General** subtree in the **Phases** window, select the **Dynamic** option as calculation type.
3. Set the **Dynamic time interval** parameter to 0.5 s.
4. 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**.
5. Set the **Max steps** to 250.
6. In the **Staged construction** mode deactivate the dynamic component of the surface load. Note that the static load is still active. The dynamic boundary conditions of this phase should be the same as in the previous phase.

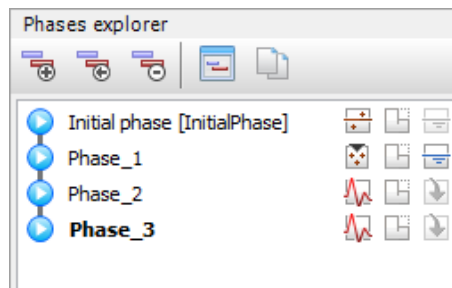The following figure shows the **Phases** explorer of this tutorial.

Figure 115: Phases explorer

8.6.5 Execute the calculation

1. Select nodes located at the ground surface (ex: (1.4 0 0), (1.9 0 0), (3.6 0 0)) to consider in curves.
2. Execute the calculation.
3. Save the project.

8.6.6 Additional calculation with damping

In a second calculation, material damping is introduced by means of Rayleigh damping. Rayleigh damping can be entered in the material data set. The following steps are necessary:

1. Save the project under another name.
2. Open the material data set of the soil.
3. In the **General** tabsheet click the box next to the **Rayleigh α** parameter.
   Note that the display of the **General** tabsheet has changed displaying the **Single DOF equivalence** box.
4. Set the value of the **ξ** parameter to 5% for both targets.
5. Set the frequency values to 9 and 11 for the **Target 1** and **Target 2** respectively.
6. Click on one of the definition cells of the Rayleigh parameters. The values of **α** and **β** are automatically calculated by the program.

Dynamics analysis of a generator on an elastic foundation [ULT]
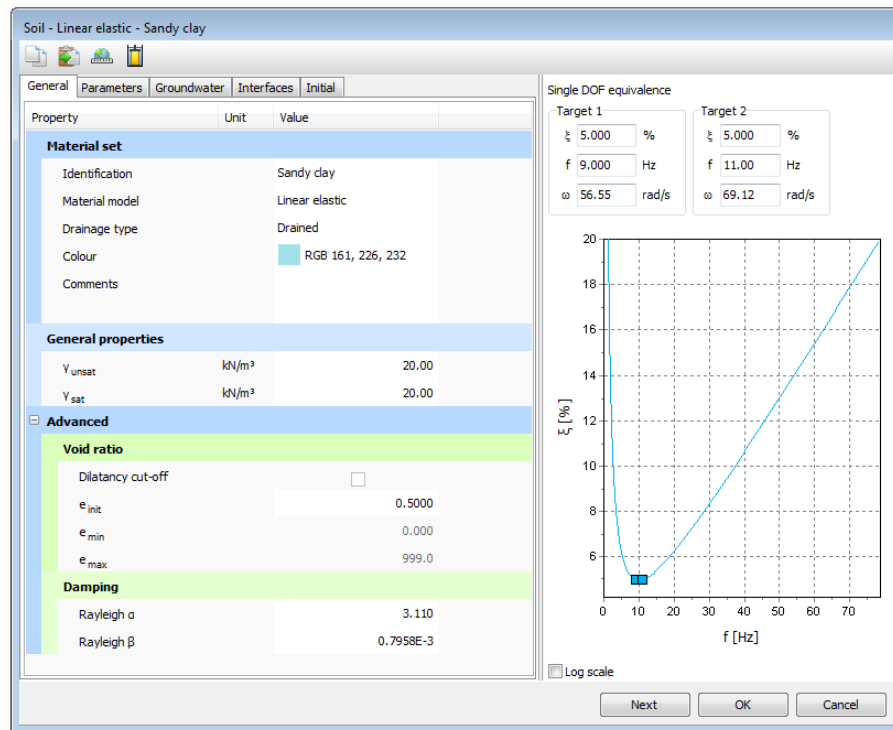Define and perform the calculation

Figure 116: Input of Rayleigh damping

7. Click **OK** to close the data base.
8. Check whether the phases are properly defined (according to the information given before) and start the calculation.

8.6.7 Results

The **Curve generator** feature is particularly useful for dynamics analysis. You can easily display the actual loading versus time (input) and also displacements, velocities and accelerations of the pre-selected points versus time. The evolution of the defined multipliers with time can be plotted by assigning **Dynamic time** to x-axis and u_z to the y-axis. Figure 117 (on page 145) shows the response of the pre-selected points at the surface of the structure. Even with no damping, the waves are dissipated thanks to the geometric damping.
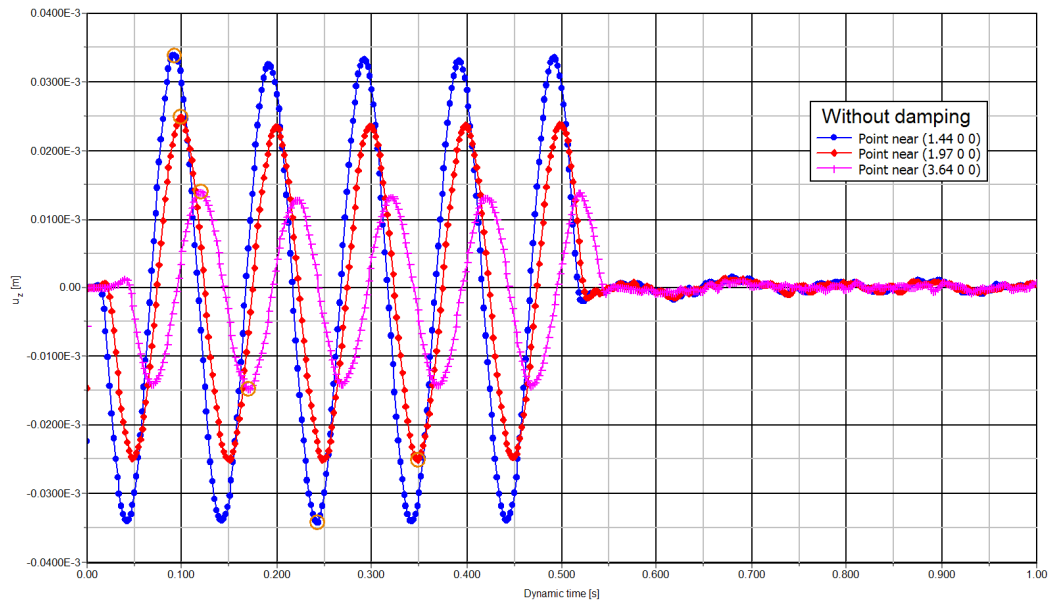
Figure 117: Vertical displ.- time on the surface at different distances to the vibrating source (without damping)

Figure 118 (on page 145) shows the response of the pre-selected points at the surface of the structure with material damping. The vibration is seized when some time is elapsed after the removal of the dynamic component of the surface load (at t = 0.5 s). Comparing the results without damping with the results with damping, also the displacement amplitudes are lower.
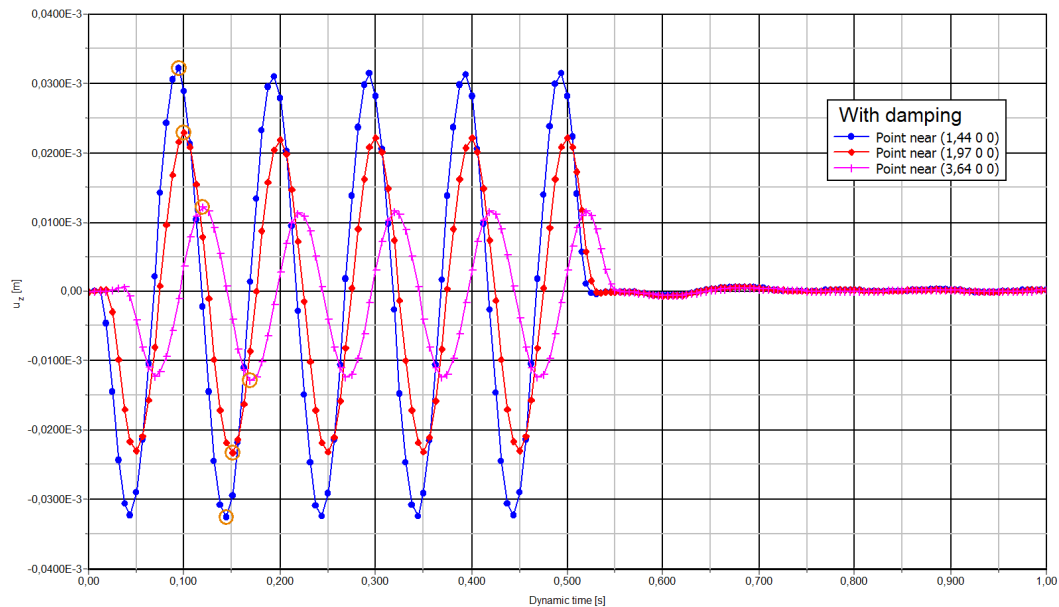

Figure 118: Vertical displ.- time (with damping)

It is possible in the Output program to display displacements, velocities and accelerations at a particular time, by choosing the appropriate option in the **Deformations** menu. Figure 119 (on page 146) shows the total accelerations in the soil at the end of phase 2 (t = 0.5 s).
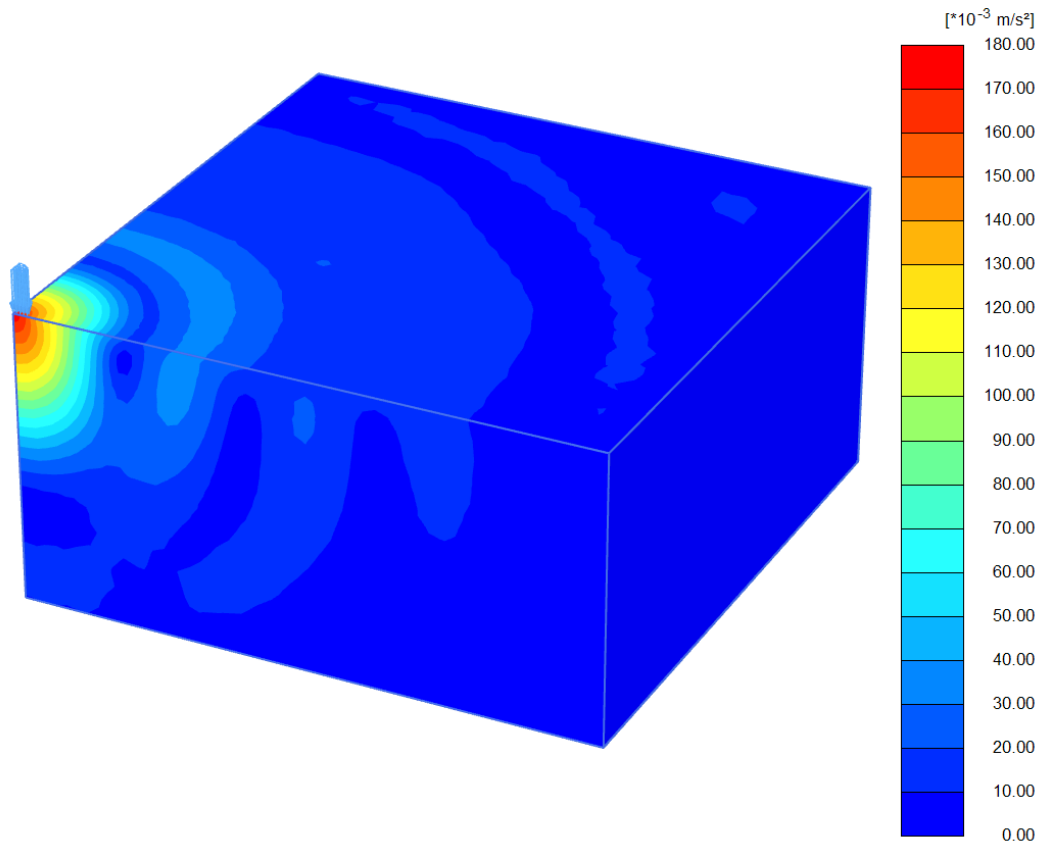
Figure 119: Total accelerations in the soil at the end of Phase 2 (with damping)