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 0.2m thick concrete footing of 1m in diameter. Oscillations caused by the generator are transmitted through the footing into the subsoil (See Figure 125 (on page 151)). These oscillations are simulated as a uniform harmonic loading, with a frequency of 10Hz and amplitude of $10kN/m^2$. In addition to the weight of the footing, the weight of the generator is modelled as a uniformly distributed load of $8kN/m^2$.
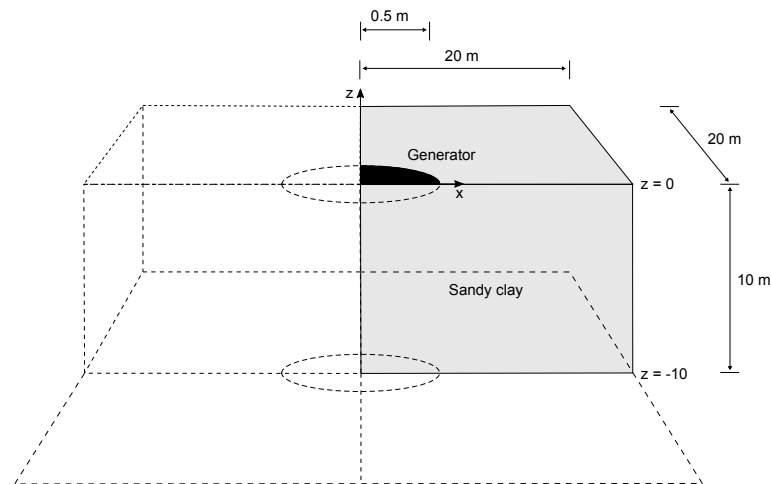
Figure 125: Generator founded on elastic subsoil

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 $x_{max} = 20$.
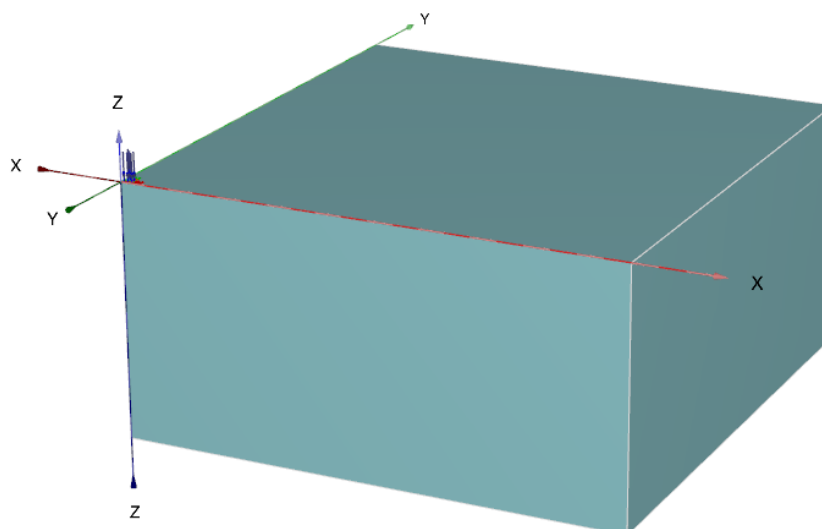   b. $y_{min} = 0$ and $y_{max} = 20$.

Figure 126: The geometry of the model

8.2 Define the soil stratigraphy

1. The subsoil consists of one layer with a depth of 10m. 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 Table 20 (on page 152) and assign it to the soil layer.

Table 20: Material properties

Property	Name	Sandy clay	Unit
General			
Soil model	Model	Linear elastic	-
Drainage type	Type	Drained	-
Unsaturated unit weight	γ_{unsat}	20.0	kN/m^3
Saturated unit weight	γ_{sat}	20.0	kN/m^3

Mechanical			
Young's modulus	E'_{ref}	$5 \cdot 10^4$	kN/m^2
Poisson's ratio	$v(nu)$	0.3	-
Interfaces			
Strength determination	-	Rigid	-
Initial			
K_0 determination	-	Automatic	-
Lateral earth pressure coefficient	$K_{0,x}$, $K_{0,y}$	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 as insertion point.
2. In the **General** tabsheet 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 Table 21 (on page 153).

Table 21: Segment properties

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

4. Once the segments are created close the polycurve designer.
5. Right-click the polycurve and select the **Create** > **Create surface** option from the appearing menu.
6. Right-click the created surface and select the **Create** > **Create surface load** option in the appearing menu.
7. In the **Selection explorer**, for the **surface load** the **Uniform** distribution is valid. 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. ![add icon] 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 in Figure 127 (on page 154).
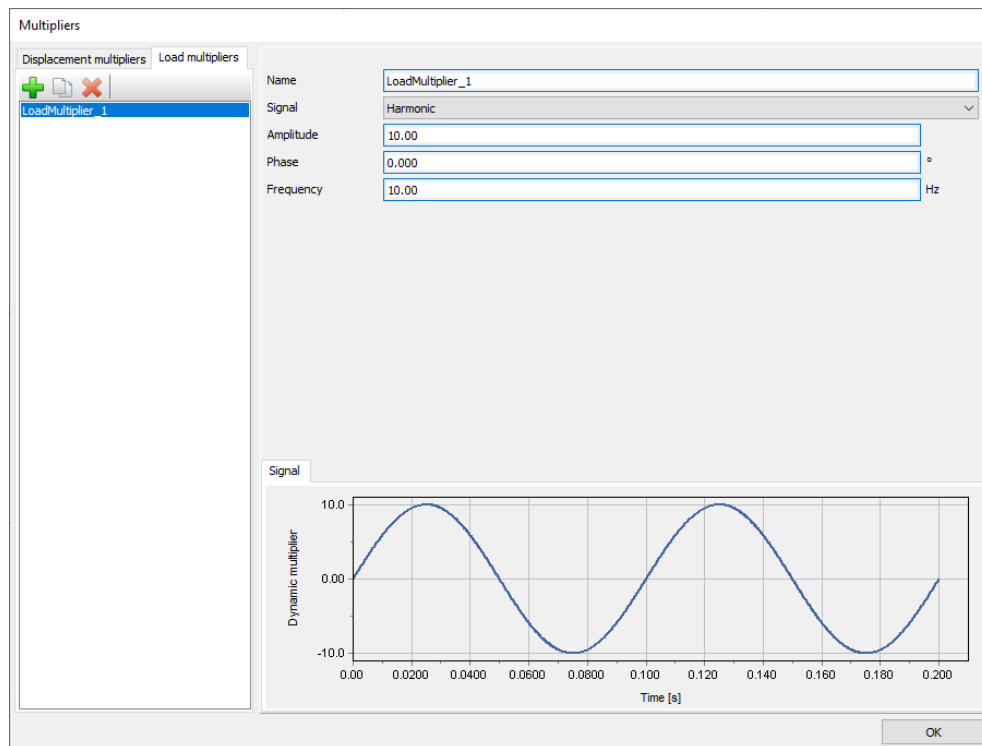
Figure 127: Definition of a Harmonic multiplier

6. ![icon] 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 **Structures mode** as well as in the **Stage construction mode**.

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. For **Element distribution** the **Medium** option will be used.
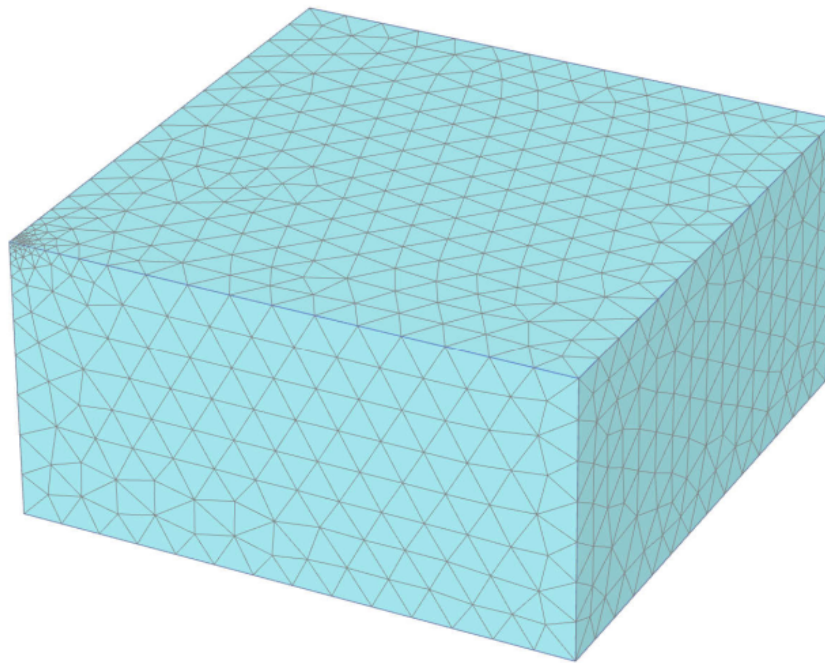4. View the generated mesh (See Figure 128).

Figure 128: 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 fisrt phase is a **Plastic** calculation where the static load is activated. The second phase is a

Dynamic calculation where the effect of the functioning generator is considered. The third and final phase are **Dynamic** calculations 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: Footing

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**, as displayed on , activate the static component of the surface load. Do not activate the dynamic load.
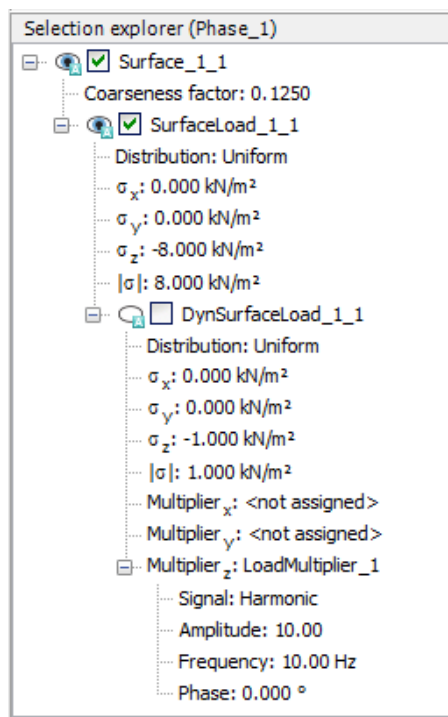

Figure 129: Applied load in the Phase_1

8.6.3 Phase 2: Start generator

In this phase, a vertical harmonic load, with a frequency of 10 Hz and amplitude of 10 kN/m2, is applied to simulate the vibrations transmitted by the generator. Five cycles with a total time interval of 0.5 sec are considered.

1. ⊞Add a new calculation phase (Phase_2).
2. In the **General** subtree in the **Phases** window, as calculation type select the **Dynamic** option.
3. Set the **Time interval** parameter to 0.5s.
4. In the **Deformation control parameters** subtree 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 (see Figure 130 (on page 157)).
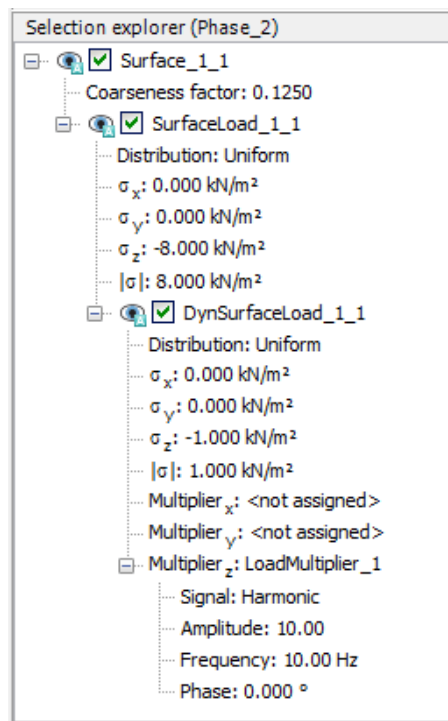

Figure 130: Applied load in the Phase_2

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 **Model Explorer > Model conditions > Dynamics** as displayed in Figure 131 (on page 158).
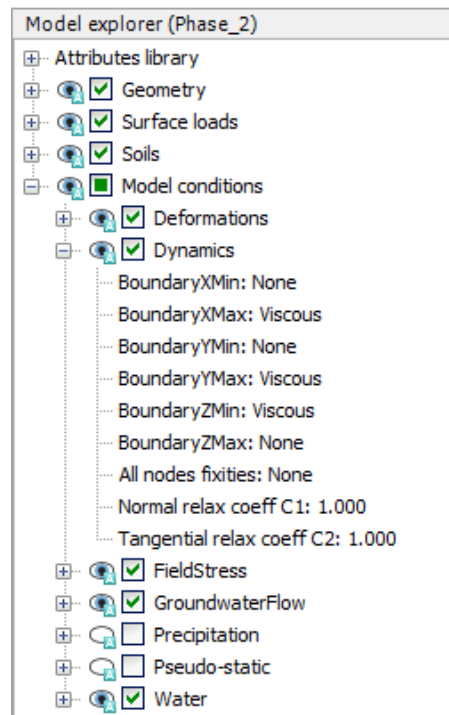

Figure 131: Boundary conditions for dynamics calculations

8.6.4 Phase 3: Stop generator

1. Add a new calculation phase (Phase_3).
2. In the **General** subtree in the **Phases** window, as **Calculation type** select the **Dynamic** option.
3. Set the **Dynamic time interval** parameter to 0.5s.
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.

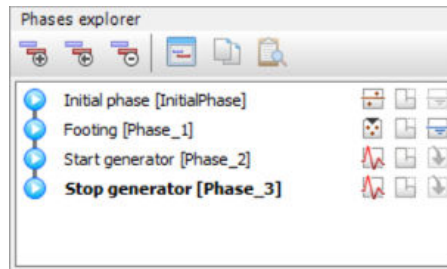Figure 132 (on page 159) shows the **Phases** explorer of this tutorial.

Figure 132: 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 under **Rayleigh damping** select as Input method the **SDOF equivalent** option (See Figure 133 (on page 160)).
4. Set a value of 5% for both ξ_1 and ξ_2. .
5. Set respectively values of 9 and 11 for the frequency targets, f_1 and f_2.
6. Notice that the values of α and β are automatically calculated by the program.
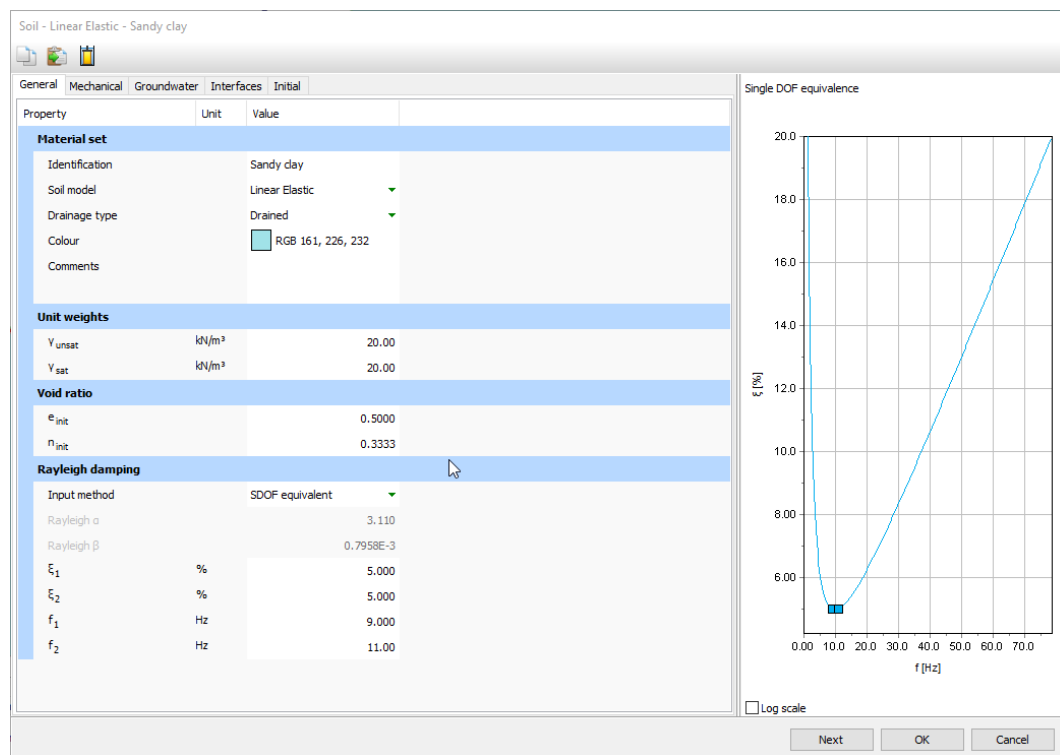
Figure 133: Input of Rayleigh damping

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 manager** 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. (on page 161) 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.
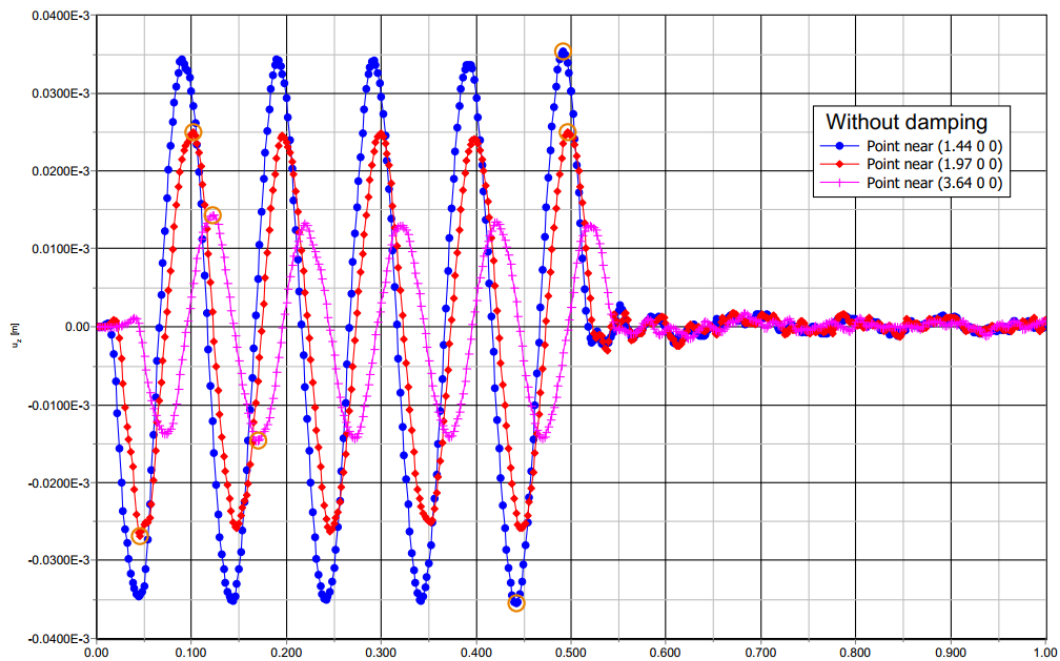
Figure 134: Vertical displ.- time on the surface at different distances to the vibrating source (without damping)

Figure 135 (on page 161) 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.
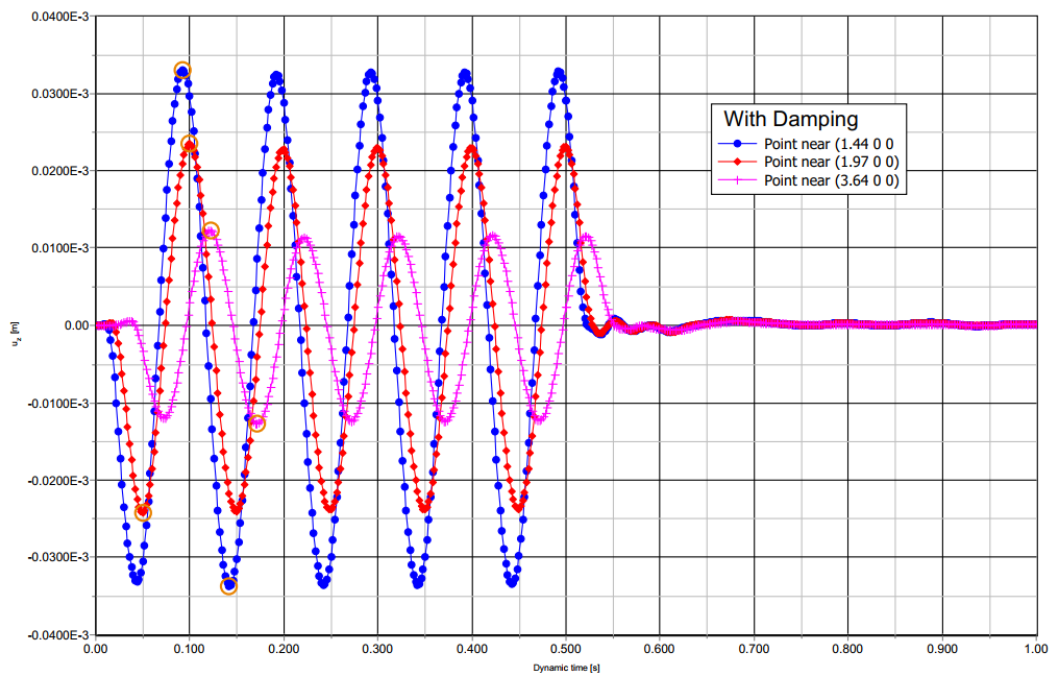

Figure 135: Vertical displ.- time (with damping)

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 136 (on page 162) shows the total accelerations in the soil at the end of phase 2 (t = 0.5s).
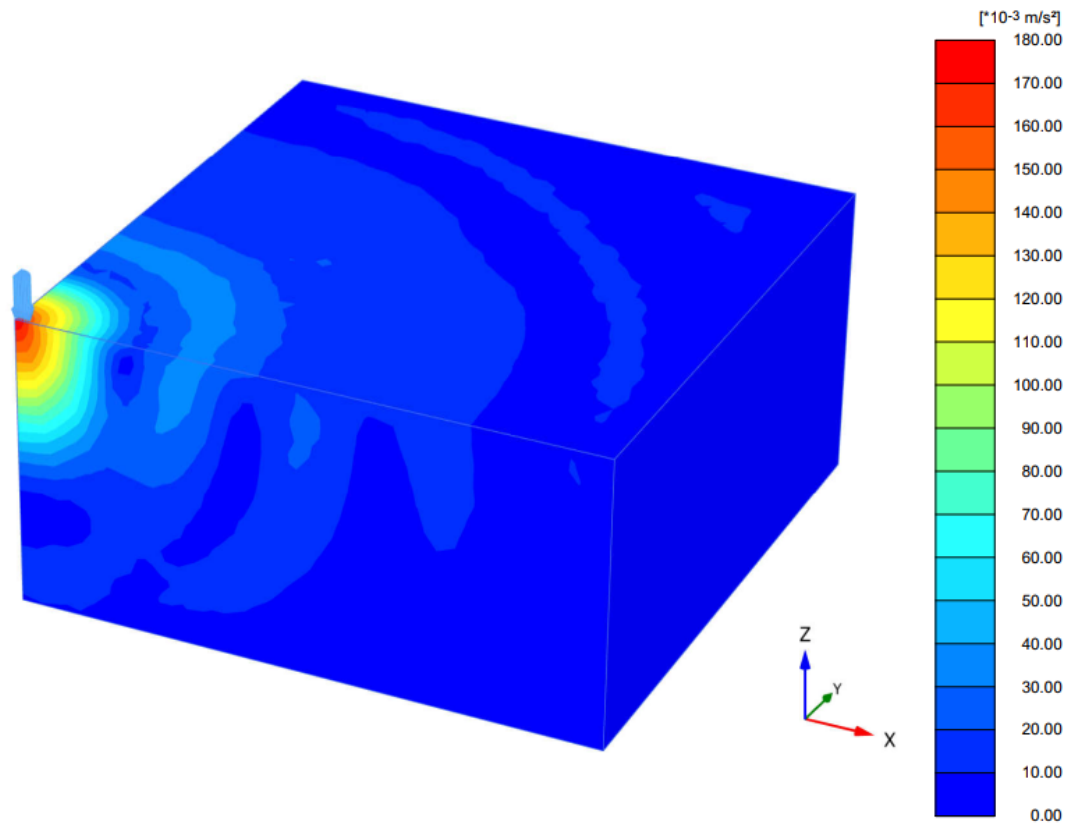

Figure 136: Total accelerations in the soil at the end of Phase 2 (with damping)