14

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

Using PLAXIS 2D, it is possible to simulate dynamic soil-structure interaction. Here the influence of a vibrating source on its surrounding soil is studied. Oscillations caused by the generator are transmitted through the footing into the subsoil.

The physical damping due to the viscous effects is taken into consideration via the Rayleigh damping. Also, due to axisymmetry 'geometric damping' can be significant in attenuating the vibration.

The modelling of the boundaries is one of the key points. 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

- Defining a **Dynamic** calculation
- Defining dynamic loads
- Defining dynamic boundary conditions (viscous)
- 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 as shown in .
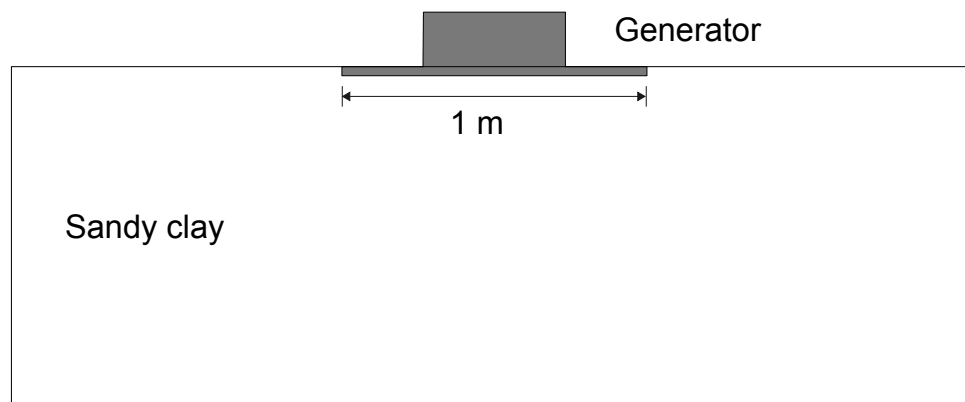

Figure 190: Generator founded on elastic subsoil

14.1 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 select** dialog box.
2. In the **Project** tabsheet of the **Project properties** window, enter an appropriate title.
3. Due to the three dimensional nature of the problem, an axisymmetric model is used. In the **Model** tabsheet select the **Axisymmetric** option for **Model** and keep the default option for **Elements** (**15-Noded**).
4. Keep the default values for units and constants and set the model contour to x_{min} = 0 m, x_{max} = 20 m, y_{min} = -10 m and y_{max} = 0 m.

> **Note:** The model boundaries should be sufficiently far from the region of interest, to avoid disturbances due to possible reflections. Although special measures are adopted in order to avoid spurious reflections (viscous boundaries), 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.

14.2 Define the soil stratigraphy

The subsoil consists of one layer with a depth of 10 m. The ground level is defined at y = 0. Note that water conditions are not considered in this example. To define the soil stratigraphy:

1. Click the **Create borehole** button and create a borehole at x = 0.
2. Create a soil layer extending from ground surface (y = 0) to a depth of 10 m (y = -10).
3. Keep the **Head** in the borehole at 0 m. This means that the sub-soil is fully saturated.

14.3 Create and assign material data sets

The soil layer consists of sandy clay, which is assumed to be elastic. Create a data set under the **Soil and interfaces** set type according to the information given in Table 30 (on page 215). The specified Young's modulus seems relatively high. This is because the dynamic stiffness of the ground is generally considerably larger than the static stiffness, since dynamic loadings are usually fast and cause very small strains.

Table 30: Material properties of the material

Parameter	Name	Value	Unit
General			
Soil model	Model	Linear elastic	-
Drainage type	Type	Drained	-
Unsaturated unit weight	γ_{unsat}	20	kN/m^3
Saturated unit weight	γ_{sat}	20	kN/m^3
Mechanical			
Stiffness	E'_{ref}	$50 \cdot 10^3$	kN/m^2
Poisson's ratio	v'	0.3	-
Initial			
K_0 determination	-	Automatic	-
Lateral earth pressure coefficient	$K_{0,x}$	0.50	-

Note: When using Mohr-Coulomb or linear elastic models the wave velocities V_p and V_s are calculated from the elastic parameters and the soil weight. V_p and V_s can also be entered as input; the elastic parameters are then calculated automatically. See also Elastic parameters and the Wave Velocity relationships in the *Mechanical Tabsheet* of the Reference Manual.

14.4 Define the structural elements

The generator is defined in the **Structures mode**.

Table 31: Material properties of the footing

Parameter	Name	Value	Unit
Material type	-	Elastic	-
Weight	w	5	kN/m/m
Isotropic	-	Yes	-
Axial stiffness	EA_1	$7.6 \cdot 10^6$	kN/m

Parameter	Name	Value	Unit
Flexural rigidity	EI	$24 \cdot 10^3$	kNm²/m
Poisson's ration	ν	0	-

1. 🔧 Create a plate extending from (0 0) to (0.5 0) to represent the footing.
2. 📋 Define a material data set for the footing according to the information given in Table 31 (on page 215). The footing is assumed to be elastic and has a weight of 5 kN/m².
3. Apply a distributed load on the footing to model the weight of the generator as well as the vibrations that it produces. The actual value of the load will be defined later.
   The model is shown in Figure 191 (on page 216):
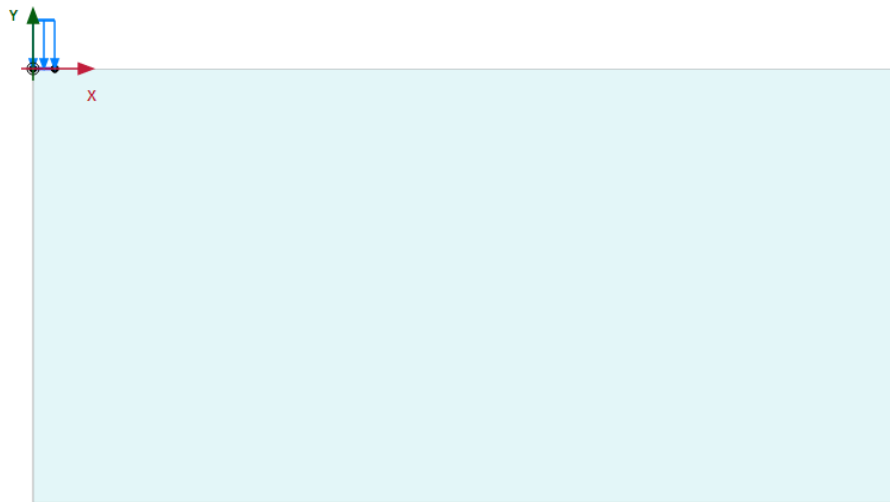
Figure 191: Model layout

14.5 Generate the mesh

1. Proceed to the **Mesh mode** .
2. Click the **Generate mesh** button 🔘 to generate the mesh. Use the default option for the **Element distribution** parameter (**Medium**).
3. Click the **View mesh** button 🔍 to view the mesh.

   The resulting mesh is shown in Figure 192 (on page 217) . Note that the mesh is automatically refined under the footing.
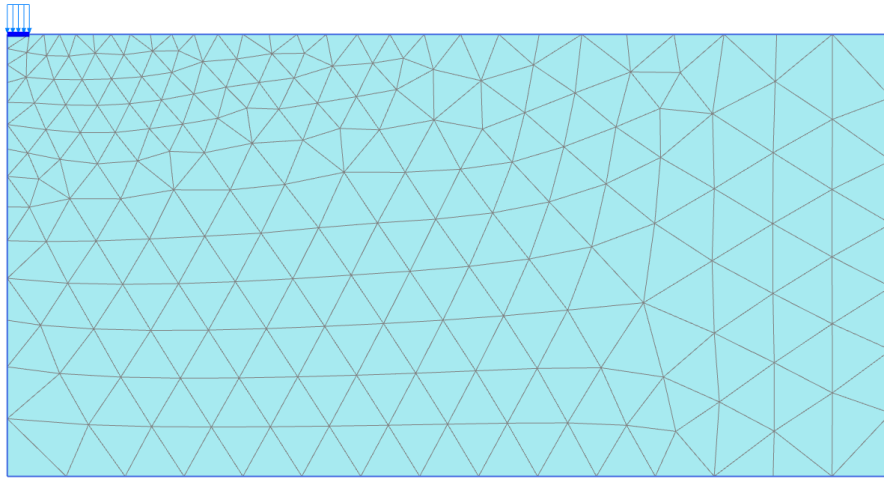
Figure 192: The generated mesh

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

14.6 Define and perform the calculation

The calculation consists of 4 phases and it will be defined in the **Staged construction mode** .

14.6.1 Initial phase

1. Click 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.

14.6.2 Phase 1: Footing

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. Activate the footing.
3. Activate the static component of the distributed load. In the **Selection explorer** set $q_{y,start,ref}$ value to -8 kN/m/m. Do not activate the dynamic component of the load as shown in Figure 193 (on page 218).
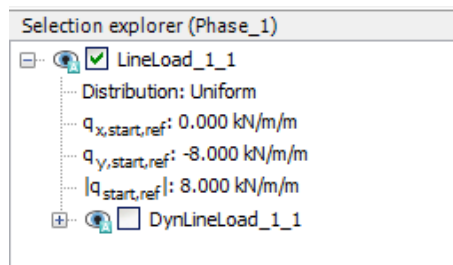
*Figure 193: Specification of the static load component in the **Selection explorer***

14.6.3 Phase 2: Start generator

In this phase, a vertical harmonic load, with a frequency of 10 Hz and amplitude of 10 kN/m^2, is applied to simulate the vibrations transmitted by the generator. Five cycles with a total time interval of 0.5 sec are considered.

1. Click the **Add phase** button to create a new phase.
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 **Deformation control parameters** subtree in the **Phases** window select the **Reset displacements to zero** parameter. The default values of the remaining parameters will be used for this calculation phase.
5. In the **Model explorer** expand the **Attributes library** subtree.
6. Right-click the **Dynamic multipliers** subtree and select the **Edit** option from the appearing menu. The **Multipliers** window pops up.
7. Click the **Load multipliers** tab.
8. Click the **Add** button to introduce a multiplier for the loads.
9. Define a **Harmonic** signal with an **Amplitude** of 10, a **Phase** of 0° and a **Frequency** of 10 Hz and as shown in
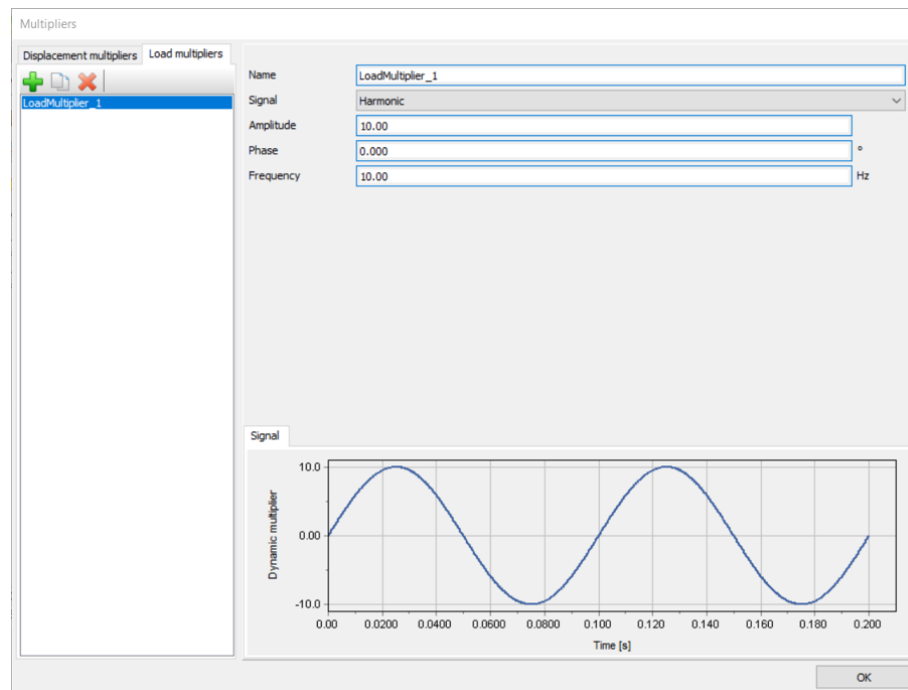
*Figure 194: Definition of a **Harmonic** multiplier*

10. In the **Selection explorer**, activate the dynamic component of the load (DynLineLoad_1).

11. Specify the components of the load as ($q_{x, start, ref}$, $q_{y, start, ref}$) = (0.0, -1.0). Click Multiplier_y in the dynamic load subtree and select the LoadMultiplier_1 option from the drop-down menu as shown in Figure 195 (on page 220).
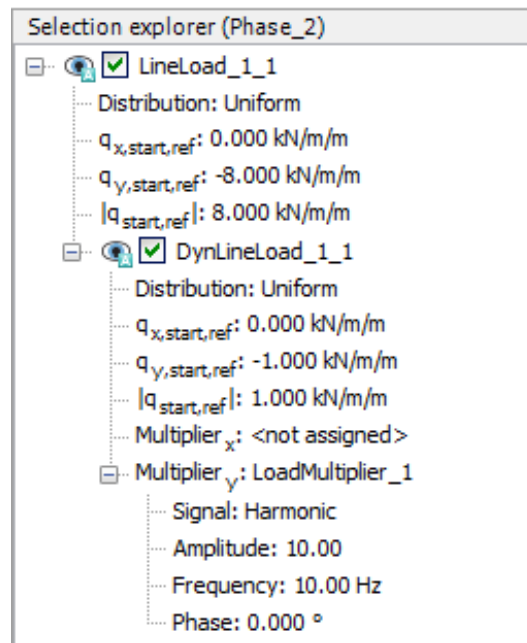
*Figure 195: Specification of the dynamic load component in the **Selection explorer***

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

12. 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 and Ymin. The dynamic boundaries can be specified in the **Model explorer** > **Model conditions** > **Dynamics** subtree as shown in Figure 196 (on page 221).
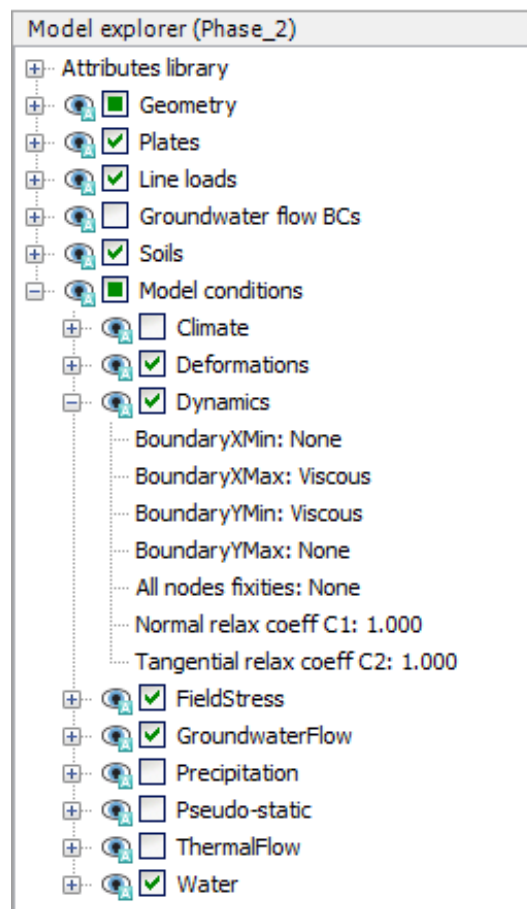
Figure 196: Boundary conditions for ***dynamics calculations***

14.6.4 Phase 3: Stop generator

1. Click the **Add phase** button 🔳 to create a new phase.

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 **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.

14.6.5 Execute the calculation

1. Click the **Select points for curves** button ▼ in the side toolbar and select nodes located at the ground surface, for instance at (1.4 0), (1.9 0) and (3.6 0), to consider in curves.

2. Click the **Calculate** button 🔢 to calculate the project.

3. After the calculation has finished, save the project by clicking the Save button 💾.

14.6.6 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 **Rayleigh damping** parameter is displayed .Set the **Input method** to **SDOF Equivalent.**
4. In order to introduce 5% of material damping, set the value of the ξ parameter to **5%** for both targets as ξ_1 and ξ_2 and set the frequency values to **1** and **10** for f_1 and f_2 respectively.
5. The values of α and β are automatically calculated by the program as shown in ..
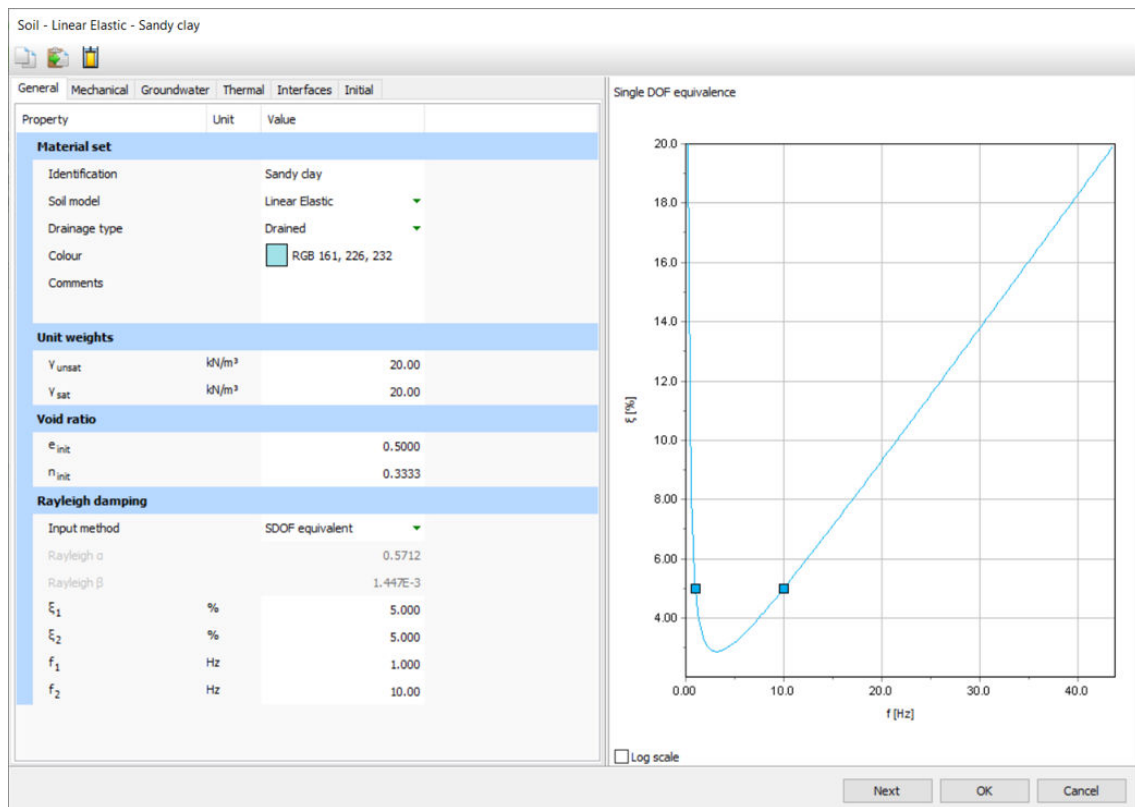
Figure 197: Input of Rayleigh damping

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

14.7 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 the x-axis and u_y to the y-axis.

Figure 198 (on page 224) shows the response of the pre-selected points at the surface of the structure. It can be seen that even with no damping, the waves are dissipated which can be attributed to the geometric damping.
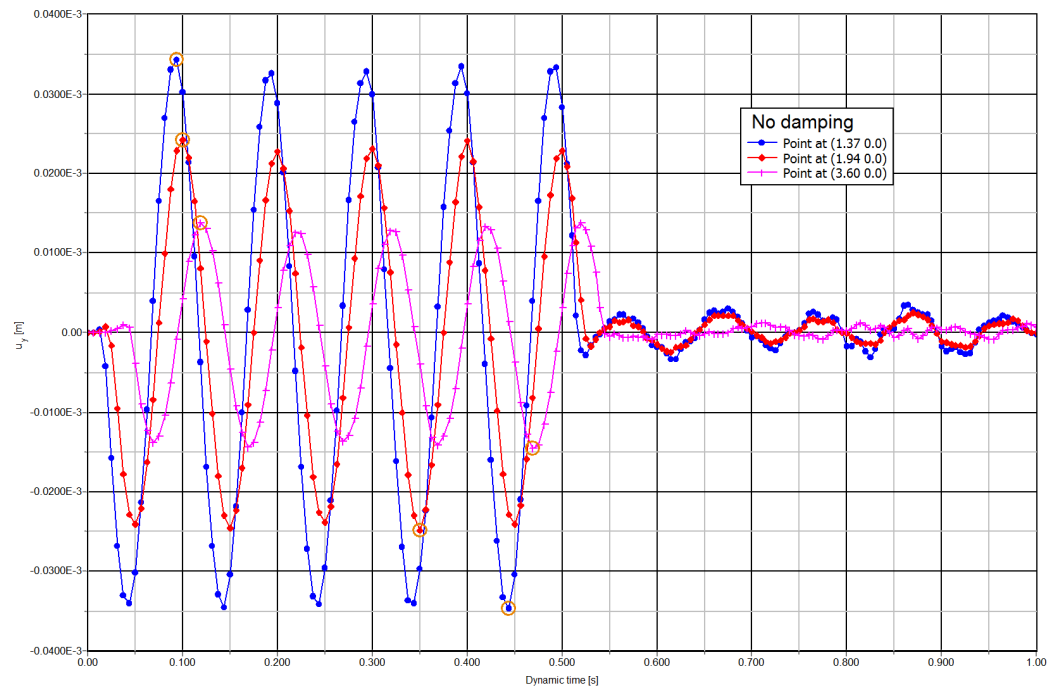
Figure 198: Vertical displacement vs time on the surface at different distances to the vibrating source (without damping)

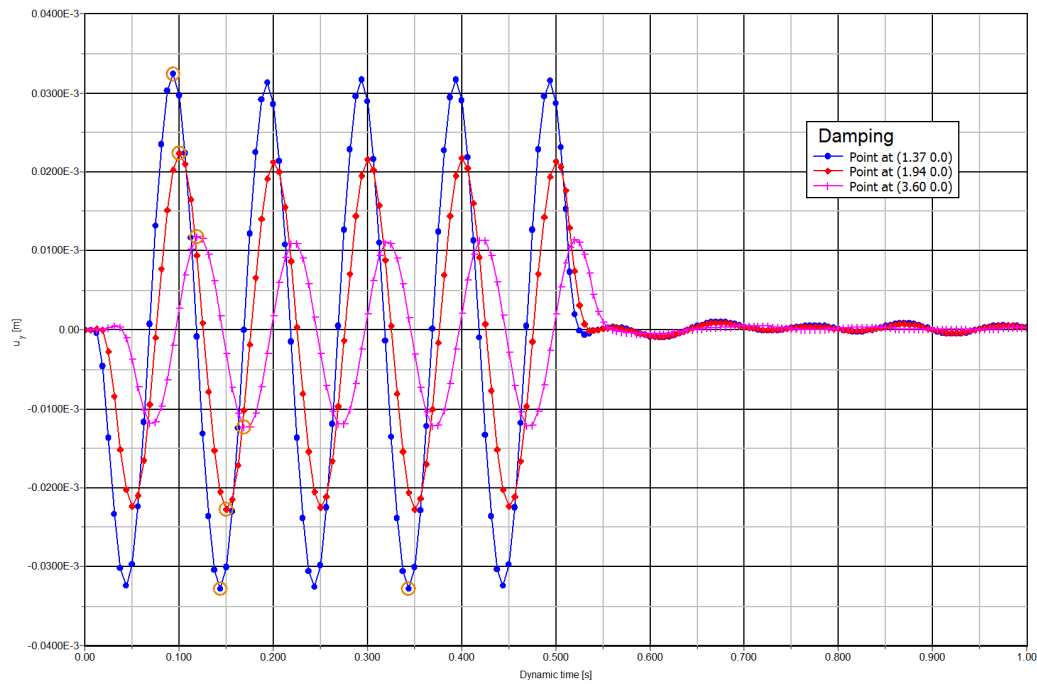The presence of damping is clear in Figure 199 (on page 225).

Figure 199: Vertical displacement vstime on the surface at different distances to the vibrating source (with damping)

It can be seen that the vibration is totally seized when some time is elapsed after the removal of the force (at t = 0.5 s). Also, the displacement amplitudes are lower. Compare the curves without and 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 200 (on page 225) shows the total accelerations in the soil at the end of phase 2 (t = 0.5 s).
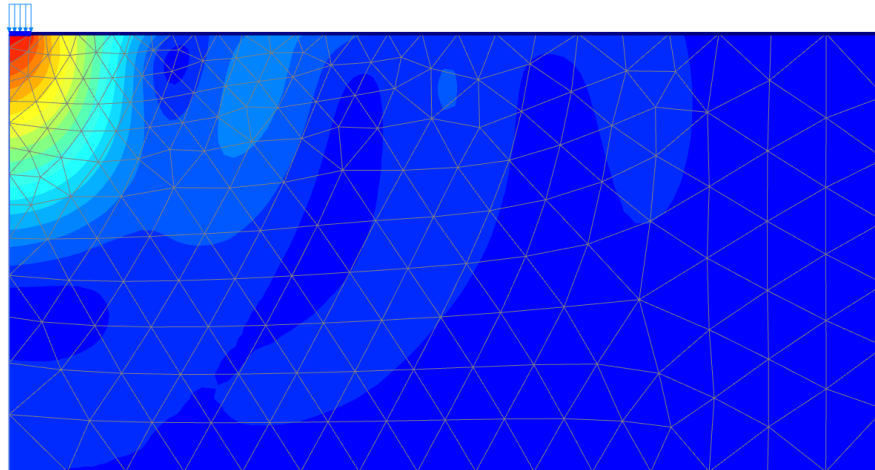

*Figure 200: Acceleration (|**a**|) in the soil at the end of phase 2 (with damping)*