
PLAXIS

CONNECT Edition V21.01

PLAXIS 2D - Tutorial Manual - Pile driving [ULT]



Last Updated: March 16, 2021

1

Pile driving [ULT]

Pile driving is a dynamic process that causes vibrations in the surrounding soil. Moreover, excess pore pressures are generated due to the quick stress increase around the pile.

In this example focus is put on the irreversible deformations below the pile. In order to simulate this process most realistically, the behaviour of the sand layer is modelled by means of the Hardening Soil model with small-strain stiffness.

Geometry

This example involves driving a concrete pile through an 11 m thick clay layer into a sand layer. The pile has a diameter of 0.4 m.

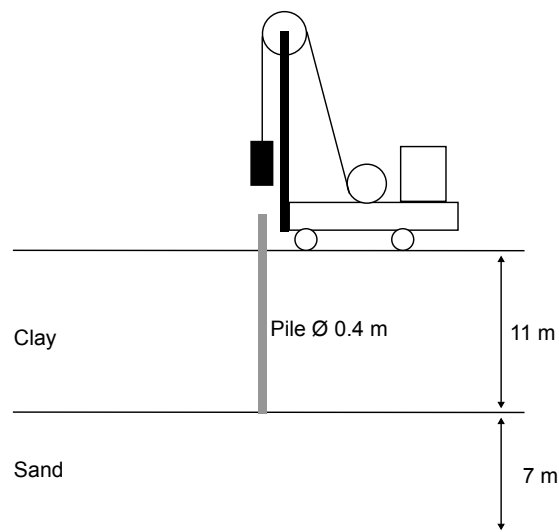


Figure 1: Pile driving situation

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 select the **Axisymmetry** option for **Model** and keep the default option for **Elements (15-Noded)**.


Pile driving [ULT]

Define the soil stratigraphy

4. Keep the default values for units and constants and set the model **Contour** to $x_{\min} = 0$ m, $x_{\max} = 30$ m, $y_{\min} = 0$ m and $y_{\max} = 18$ m.

Define the soil stratigraphy

The subsoil is divided into an 11 m thick clay layer and a 7 m thick sand layer. The phreatic level is assumed to be at the ground surface. Hydrostatic pore pressures are generated in the whole geometry according to this phreatic line. To define the soil stratigraphy:

1. Click the **Create borehole** button  and create a borehole at $x = 0$.
2. Create two soil layers extending from $y = 18$ to $y = 7$ and from $y = 7$ to $y = 0$.
3. Set the **Head** in the borehole at 18 m.

Create and assign material data sets

The clay layer is modelled with the Mohr-Coulomb model. The behaviour is considered to be **Undrained (B)**. An interface strength reduction factor is used to simulate the reduced friction along the pile shaft.

In order to model the non-linear deformations below the tip of the pile in a right way, the sand layer is modelled by means of the Hardening Soil model with small-strain stiffness. Because of the fast loading process, the sand layer is also considered to behave undrained. The short interface in the sand layer does not represent soil-structure interaction. As a result, the interface strength reduction factor should be taken equal to unity (rigid).

The layers have the following properties:

Table 1: Material properties of the clay layer and the pile

Parameter	Name	Clay	Pile	Unit
General				
Material model	Model	Mohr-Coulomb	Linear elastic	-
Type of material behaviour	Type	Undrained (B)	Non-porous	-
Soil unit weight above phreatic level	γ_{unsat}	16	24	kN/m ³
Soil unit weight below phreatic level	γ_{sat}	18	-	kN/m ³
Parameters				
Young's modulus (constant)	E'	$5.0 \cdot 10^3$	$30 \cdot 10^6$	kN/m ²
Poisson's ratio	ν'	0.3	0.1	-
Cohesion	c'_{ref}	-	-	kN/m ²
Undrained shear strength	$S_{u,ref}$	5.0	-	kN/m ²

Pile driving [ULT]

Create and assign material data sets

Parameter	Name	Clay	Pile	Unit
Young's modulus increment	E'_{inc}	$1.0 \cdot 10^3$	-	kN/m ²
Reference level	y_{ref}	18	-	m
Undrained shear strength increment	$S_{u,inc}$	3	-	kN/m ²
Reference level	y_{ref}	18	-	m
Interface				
Interface strength type	Type	Manual	Rigid	-
Interface strength	R_{inter}	0.5	1.0	-
Initial				
K_0 determination	-	Automatic	Automatic	-
Lateral earth pressure coefficient	$K_{0,x}$	0.5000	0.5000	-

Table 2: Material properties of the sand layer

Parameter	Name	Sand	Unit
General			
Material model	Model	HS small	-
Type of material behaviour	Type	Undrained (A)	-
Soil unit weight above phreatic level	γ_{unsat}	17	kN/m ³
Soil unit weight below phreatic level	γ_{sat}	20	kN/m ³
Parameters			
Secant stiffness in standard drained triaxial test	E_{50}^{ref}	$50 \cdot 10^3$	kN/m ²
Tangent stiffness for primary oedometer loading	E_{oed}^{ref}	$50 \cdot 10^3$	kN/m ²
Unloading / reloading stiffness	E_{ur}^{ref}	$150 \cdot 10^3$	kN/m ²
Power for stress-level dependency of stiffness	m	0.5	-
Cohesion	c'_{ref}	0	kN/m ²
Friction angle	φ'	31	°
Dilatancy angle	ψ	0	°

Pile driving [ULT]

Define the structural elements

Parameter	Name	Sand	Unit
Shear strain at which $G_s = 0.722 G_0$	$\gamma_{0.7}$	$0.1 \cdot 10^{-3}$	-
Shear modulus at very small strains	G_0^{ref}	$120 \cdot 10^3$	kN/m ²
Poisson's ratio	ν_{ur}	0.2	-
Interface			
Interface strength	-	Rigid	-
Initial			
K_0 determination	-	Automatic	-

1.  Create the material data sets according to [Table 33](#) (on page 3) and [Create and assign material data sets](#) (on page 3)

Define the structural elements

The pile is defined as a column of 0.2 m width. The **Interface** elements are placed along the pile to model the interaction between the pile and the soil. The interface should be extended to about half a meter into the sand layer. Note that the interface should be defined only at the side of the soil. A proper modelling of the pile-soil interaction is important to include the material damping caused by the sliding of the soil along the pile during penetration and to allow for sufficient flexibility around the pile tip.

Note: Use the **Zoom in** feature to create the pile and the interface.

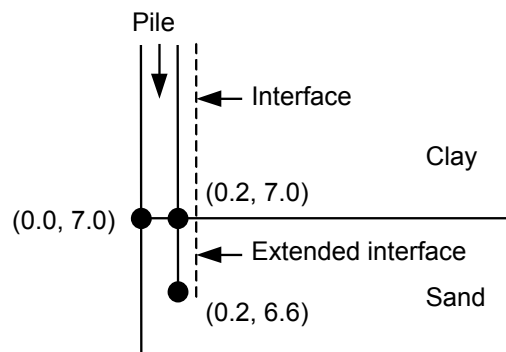




Figure 2: Extended interface

Define the pile

To define the concrete pile:

Pile driving [ULT]



Define the structural elements

1. Click the **Structures** tab to proceed with the input of structural elements in the **Structures** mode.
2. Select the **Create line** feature  in the side toolbar and draw a line from (0.2 6.6) to (0.2 18).
3.  Assign a negative interface to the line to model the interaction of the pile with the surrounding soil.

The pile is made of concrete, which is modelled by means of the linear elastic model considering non-porous behaviour. In the beginning, the pile is not present, so initially the clay properties are present in the area the represents the pile.

Define a load

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

1.  Define a distributed load by clicking on **Create load > Create line load** from the tool bar and then on (0 18.0) and (0.2 18.0).
2. The load components will be defined in the **Selection explorer**. Note that the static component of the load will not be used in this project. The program will neglect the static components of the load if it (static load) is not activated.
3. Expand the **Dynamic load** subtree and specify a unit load in the gravity direction.
4. Click the Multiplier_y drop down menu and click on the appearing plus button . The **Multipliers** window pops up and a new load multiplier is automatically added.
5. Define a **Harmonic** signal with an **Amplitude** of 5000, a **Phase** of 0° and a **Frequency** of 50 Hz and as shown in the figure below. During the pile driving phase, we will only consider half a cycle (0.01 s) of this signal.

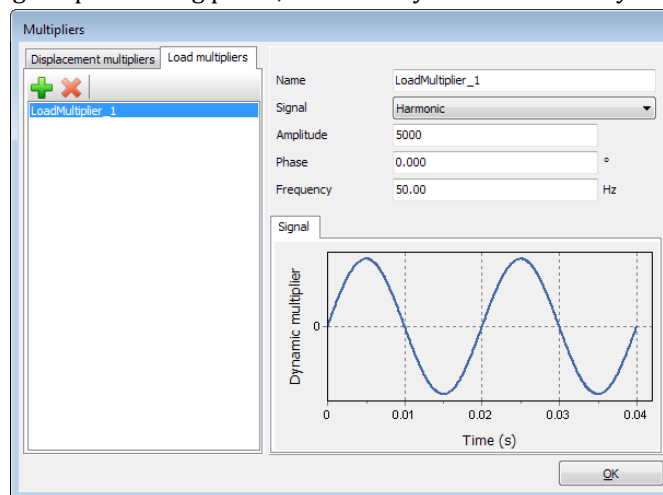


Figure 3: Definition of an Harmonic multiplier

Note:

Note that dynamic multipliers can be defined by right-clicking the **Dynamic multipliers** subtree under **Attributes library** in the **Model explorer**.

Pile driving [ULT]

Generate the mesh



Note that dynamic multipliers are attributes and as such it is possible to define them in all the program's modes.

The final geometry model is shown in the following figure:



Figure 4: The geometry model

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. Note that the mesh is automatically refined under the footing.

Pile driving [ULT]

Define and perform the calculation

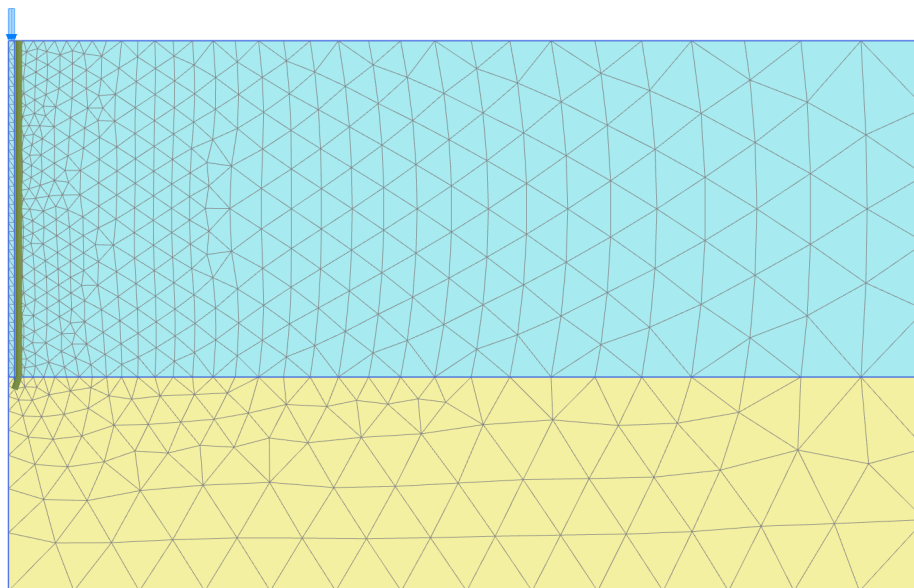


Figure 5: The generated mesh

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


Define and perform the calculation

The calculation consists of 3 phases. In the Initial phase, the initial stress conditions are generated. In the Phase 1 the pile is created. In the Phase 2 the pile is subjected to a single stroke, which is simulated by activating half a harmonic cycle of load. In the Phase 3 the load is kept zero and the dynamic response of the pile and soil is analysed in time. The last two phases involve dynamics calculations.

Initial phase

Initial effective stresses are generated by the **K0 procedure**, using the default values. Note that in the initial situation the pile does not exist and that the clay properties should be assigned to the corresponding cluster. The phreatic level is assumed to be at the ground surface. Hydrostatic pore pressures are generated in the whole geometry according to this phreatic line.

Phase 1: Pile activation

1. Click the **Add phase** button  to create a new phase.
2. In the **General** subtree in the **Phases** window, the **Plastic** option is selected as **Calculation type**.
3. The **Staged construction** option is by default selected as **Loading type**.
4. In the **Staged construction** mode assign the pile properties to the pile cluster.
5. Activate the interface in the **Clay** layer only. The model for the Phase 1 in the **Staged construction** mode is displayed below.


Pile driving [ULT]

Define and perform the calculation



Figure 6: Configuration of Phase 1 in the *Staged construction* mode

Phase 2: Pile driving

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** to 0.01 s.
4. In the **Deformation control parameters** subtree select **Reset displacements to zero**.
5. In the **Staged construction** mode activate the dynamic component of the distributed load.
The activated dynamic component of the load in **Selection explorer** is shown in the figure below.

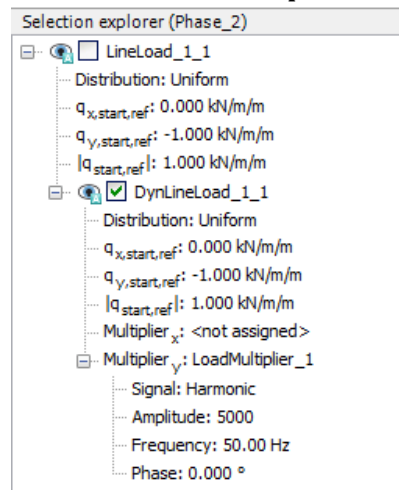


Figure 7: The dynamic load component in the *Selection explorer*

6. Expand the subtree **Model explorer > Model conditions > Dynamics**.

Pile driving [ULT]

Define and perform the calculation

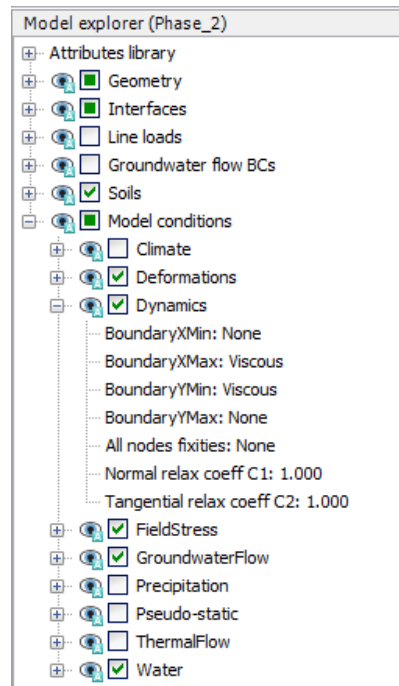



Figure 8: Boundary conditions for *dynamics calculations*




7. Specify viscous boundaries at x_{\max} and y_{\min} .

The result of this phase is half a harmonic cycle of the external load. At the end of this phase, the load is back to zero.

Phase 3: Fading

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** to 0.19 s.
4. In the **Staged construction** mode de-activate the dynamic load.

Execute the calculation

1. Click the **Select points for curves** button  in the side toolbar and select a node at the top of the pile for load displacement curves.
2. Click the **Calculate** button  to calculate the project.
3. After the calculation has finished, save the project by clicking the **Save** button .

Results

[Figure 187](#) (on page 11) shows the settlement of the pile (top point) versus time. From this figure the following observations can be made:

- The maximum vertical settlement of the pile top due to this single stroke is almost 14 mm. However, the final settlement is about 9.5 mm.
- Most of the settlement occurs in phase 3 after the stroke has ended. This is due to the fact that the compression wave is still propagating downwards in the pile, causing additional settlements.
- Despite the absence of Rayleigh damping, the vibration of the pile is damped due to soil plasticity and the fact that wave energy is absorbed at the model boundaries.

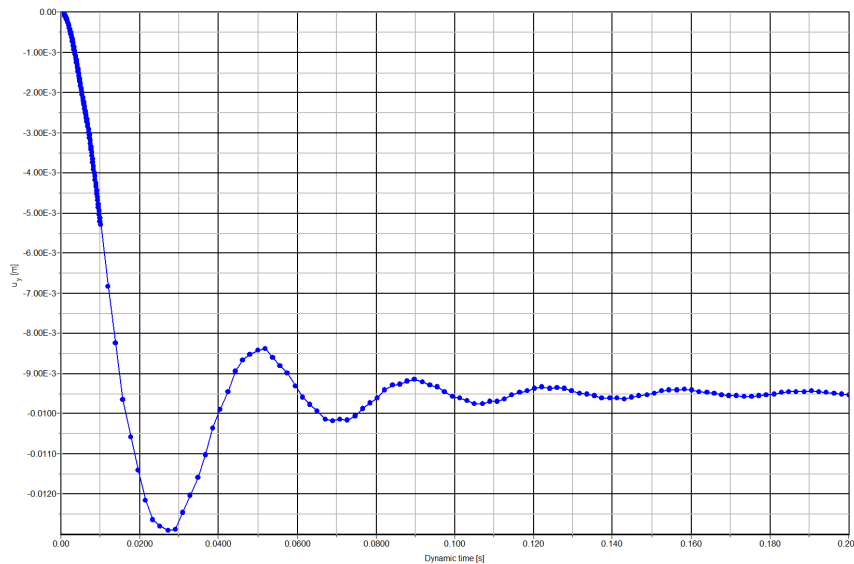


Figure 9: Pile settlement vs. time

When looking at the output of the second calculation phase ($t = 0.01$ s, i.e. just after the stroke), it can be seen that large excess pore pressures occur very locally around the pile tip. This reduces the shear strength of the soil and contributes to the penetration of the pile into the sand layer. The excess pore pressures remain also in the third phase since consolidation is not considered.

[Figure 188](#) (on page 12) shows the shear stresses in the interface elements at $t = 0.01$ s. The plot shows that the maximum shear stress is reached all along the pile, which indicates that the soil is sliding along the pile.

Pile driving [ULT]

Results

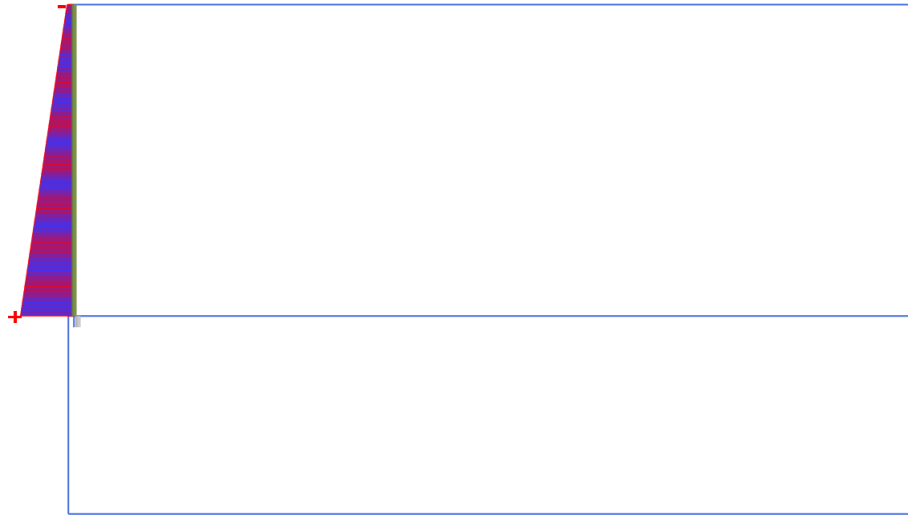


Figure 10: Maximum shear stresses in the interface at $t = 0.01$ s

When looking at the deformed mesh of the last calculation phase ($t = 0.2$ s), it can also be seen that the final settlement of the pile is about 9.5 mm. In order to see the whole dynamic process it is suggested to use the option **Create Animation** to view a 'movie' of the deformed mesh in time. You may notice that the first part of the animation is slower than the second part.