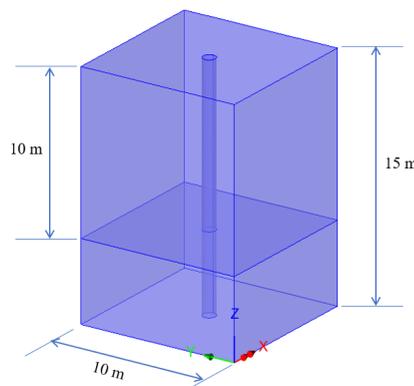


# Pile Load Test

For LUSAS version:	21.0
For software product(s):	LUSAS Bridge plus or LUSAS Civil&Structural plus
With product option(s):	Geotechnical, Nonlinear

## Problem Description

The Pile Load Test is a type of load testing that is conducted on deep foundation piles used in construction projects to determine the safe load-carrying capacity of the pile. The test involves applying a gradually increasing load to the top of the pile using a hydraulic jack or a load cell and monitoring the pile's deformation and the corresponding load. In a static load test, the load is applied slowly, and the pile's deflection is measured at different load levels. The test results are used to develop a load-settlement curve, which can be used to calculate the pile's ultimate bearing capacity and estimate its settlement under different loads.



**Figure 1: Pile model geometry**

In this example we create a numerical model that simulates the behaviour of a pile during a load test. The solution of this type of problem is very dependent on mesh density of the soil adjacent to the pile, the interface shear stiffness as well as the Gauss point integration scheme used. This example is provided for illustrative purposes only; however, it is essential to validate the model's results with actual load test data before using it for design or construction purposes.

## Pile Load Test

---

The pile length is 10m with a diameter of 0.8m, as shown in figure 1, and is required to support a maximum load of 1720 kN.

### Keywords

**Pile, Interface, 3D Modelling, Load Curve, Construction Stages, Loading, Unloading.**

### Associated Files

Associated files can be downloaded from the user area of the LUSAS website.



**pile\_test.lvb** carries out automated modelling of the example.

- Use **File > New** to create a new model of a suitable name in a chosen location.
- Use **File > Script > Run Script** to open the lvb file named above that was downloaded and placed in a folder of your choosing.

### Objectives

- Modelling of pile load test and plotting the load-settlement curve
- Determination of the maximum settlement.

## Preparing the Model Features

We create a new model, set the Analysis category as 3D, and specify the model units as kN,m,t,s,C.

### Feature Geometry

- To start building the model, a surface feature is created from “Lines” using the command **Geometry > Surface > By Lines**. The lines themselves are defined by points (Figure 2). The circle inside is generated by **Geometry > Line > Arc/Circle** and is then used to create a surface.

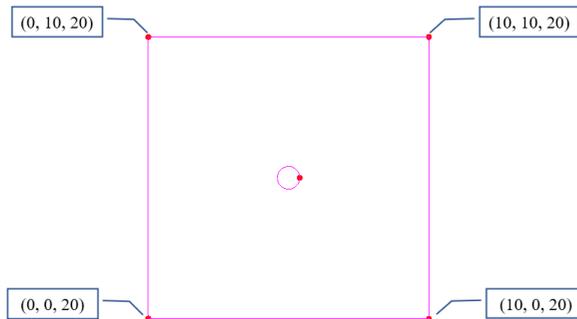


Figure 2: Top layout of soil model

- Next, a hole needs to be created which is done by selecting first the surrounding surface and then the inner circular surface, then **Geometry > Surface > Holes**. We uncheck “*delete geometry defining holes*” to keep the circular surface (Figure 3).

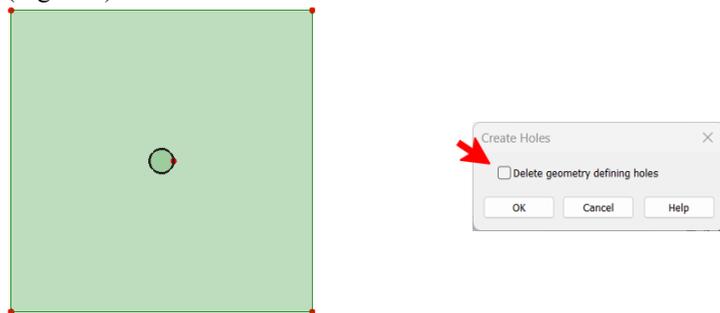


Figure 3: Creating hole in the top surface

- Volumes are then formed by sweeping the surfaces in Z direction by the command **Geometry > Volume > By Sweeping** (Figure 4)

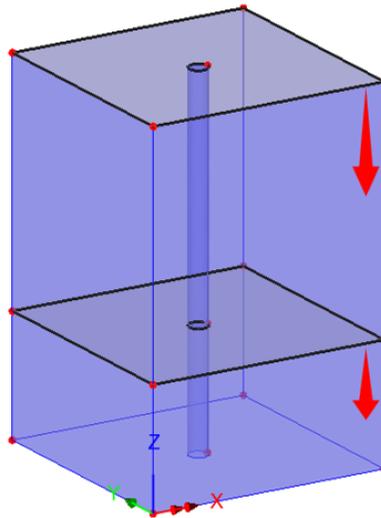


Figure 4: Sweeping surfaces to form volumes

### Preparing the Model Attributes

Model attributes (mesh, material, geometric properties, etc.) are defined and assigned to the model. Figure 5 shows the attributes of model. The fine element integration scheme is used to improve convergence.

### Defining the Mesh

Solid hexahedral, quadratic stress elements (HX20) are used for meshing the volume parts, the interface is meshed with hexahedral, quadratic elements (IS16) (Figures 6 and 7).

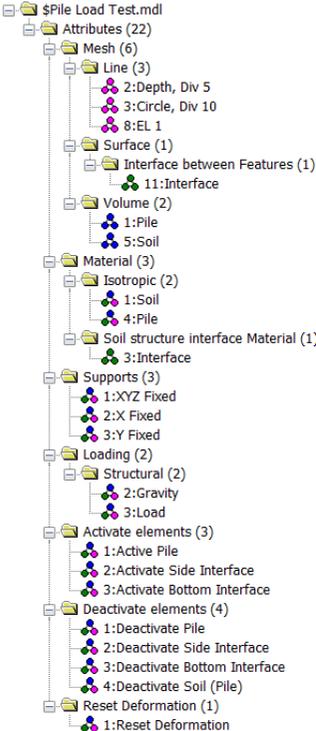
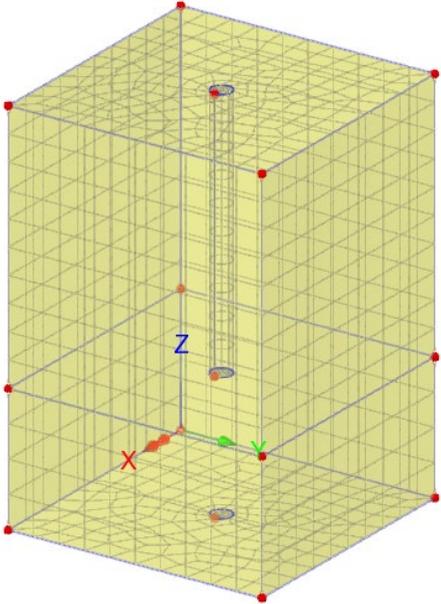
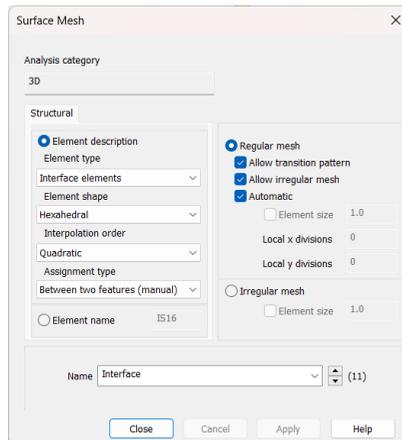


Figure 5: Model Attributes



**Figure 7: Interface mesh parameters**

In order to create the pile material, we implement these steps:

- We copy the sand volume (volume 2) to an upper location to create the pile volume (volume 5) (Figure 8).
- We assign the interface mesh between the surrounding soil, side and bottom (volume 1) and the pile volume (volume 5) (Figure 9).
- Before moving the pile volume back, we make the volume, as well as its lines and points **“unmergeable”**.

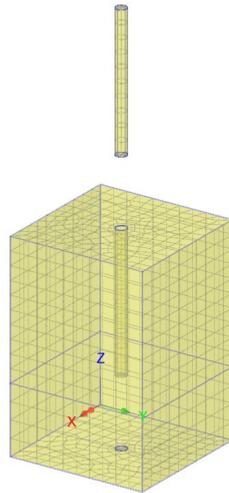


Figure 8: Model meshing

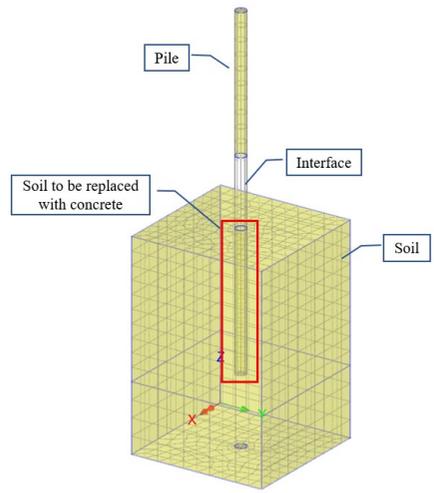


Figure 9: Model meshing

## Defining the Materials

The soil material will be modelled using an isotropic nonlinear material with a Modified Mohr-Coulomb failure surface, while the pile will be assumed to have elastic behaviour. The initial stress state in the soil is established by assigning a coefficient for lateral earth pressure,  $K_0$ , to the soil.

All material properties are listed in table 1.

Table 1: Material properties

Layer	Mass Density	Young's modulus, E	Poisson's ratio, $\nu$	Angle of friction, $\varphi$	Cohesion, c	$K_0$
Soil	1.8 t/m <sup>3</sup>	50E3 kPa	0.3	38°	10 kPa	0.384
Pile	7 t/m <sup>3</sup>	28.5E6 kPa	0.4	-	-	-

Interface properties					
Layer	Normal stiffness factor $K_n$	Tangential stiffness factor $K_s$	Angle of friction, $\varphi$	Cohesion, c	
Soil	10	0.15	38°	10 kPa	

## Defining the Supports

The model is restrained in X and Y directions and fully fixed supports are assigned to the base (Figure 7).

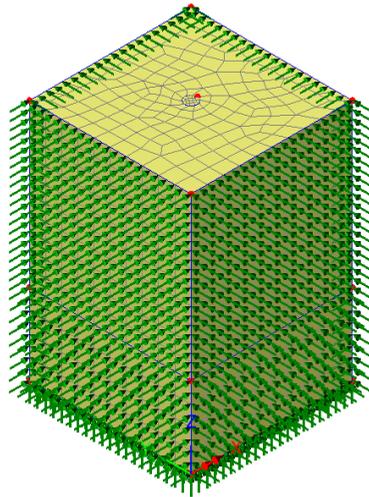


Figure 7: Boundary conditions

### Defining the Loads

In addition to the weight of the structure, a gradual load is applied to the top of the pile until it reaches the value specified by the designer. In pile load testing, loading and unloading cycles are considered. The gradual load is applied using a **Load Curve**.

### Defining Other Attributes

Deactivate and Activate attributes are used to simulate the installation phase of the pile. The following paragraphs provide more details.

## Running the Analysis

We consider the following analysis and construction stages.

### Analysis 1 > Initial Phase

This stage establishes the initial stress distribution under gravity load. The pile volume and associated interfaces are deactivated in this phase.

Nonlinear analysis control properties are defined for this phase, all the parameters are left at their default values.

## Analysis 1 > Install Pile

To install the pile in this phase, we need to make the pile volume (volume 5) and the associated interfaces active and the corresponding soil volume (volume 2) inactive.

Nonlinear analysis control properties are defined for this phase, all the parameters are left at their default values.

## Analysis 2 > Loading

In analysis 2, we simulate the loading applied to the top of the pile. For this purpose, we first create a **Load Curve** by the command **Analysis2 (right click) > New > Load Curve**. Using load curves, we assign the gravity load and applied load over 14 time units (loading and unloading) (Figure 8). The maximum applied load is 3240 kN/m<sup>2</sup>.

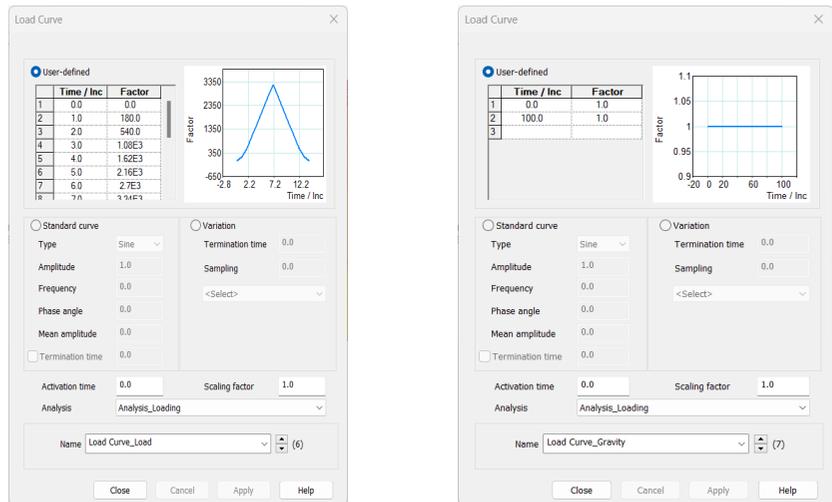


Figure 8: Load curves used in analysis 2

Viscous control properties are set as shown in figure 9. The total response time is set to 7 time units with a time step of 1, while the maximum number of increments is set to 14 to allow the solution to step halve if necessary.

The screenshot shows the 'Nonlinear & Transient' dialog box with the following settings:

- Incrementation:**  Nonlinear; Incrementation: Manual; Starting load factor: 0.1; Max change in load factor: 0.0; Max total load factor: 1.0;  Adjust load based on convergence; Iterations per increment: 4.
- Time domain:**  Time domain; Viscous; Initial time step: 1.0; Total response time: 7.0;  Automatic time stepping.
- Solution strategy:**  Same as previous loadcase; Max number of iterations: 20; Residual force norm: 0.1; Incremental displacement: 1.0.
- Incremental LUSAS file output:**  Same as previous loadcase; Output file: 1; Plot file: 1; Restart file: 0; Max number of saved restarts: 0; Log file: 1; History file: 0.
- Common to all:** Max time steps or increments: 14.

Figure 9: Nonlinear analysis control parameters

## Analysis 2 > Unloading

The viscous control properties are set as shown in figure 10 with the total response time of 14.

The screenshot shows the 'Nonlinear & Transient' dialog box with the following settings:

- Incrementation:**  Nonlinear; Incrementation: Manual; Starting load factor: 0.1; Max change in load factor: 0.0; Max total load factor: 1.0;  Adjust load based on convergence; Iterations per increment: 4.
- Time domain:**  Time domain; Viscous; Initial time step: 1.0; Total response time: 14.0;  Automatic time stepping.
- Solution strategy:**  Same as previous loadcase; Max number of iterations: 12; Residual force norm: 0.1; Incremental displacement: 1.0.
- Incremental LUSAS file output:**  Same as previous loadcase; Output file: 1; Plot file: 1; Restart file: 0; Max number of saved restarts: 0; Log file: 1; History file: 1.
- Common to all:** Max time steps or increments: 10.

Figure 10: Nonlinear analysis control parameters

## Viewing the Analysis

Analysis loadcase results are present in the Treeview.

## Stress

The following figure 11 shows the vertical stress at the initial stage.

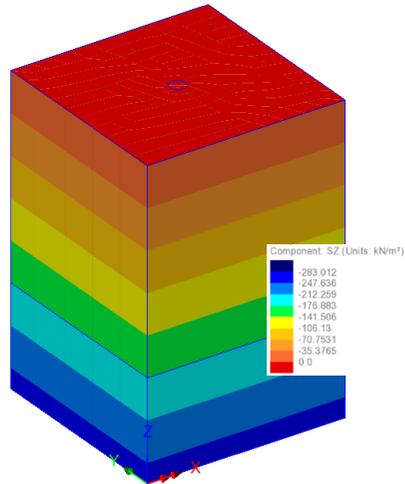


Figure 11 : Vertical stress at the initial stage (kN/m<sup>2</sup>)

## Displacement

The maximum resultant displacement is 19.8 mm. Figure 12 illustrates the loading-unloading plot.

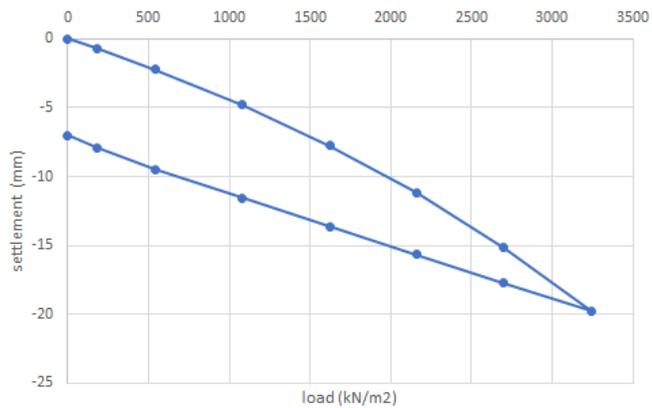


Figure 12: Loading-Unloading displacement (mm)

