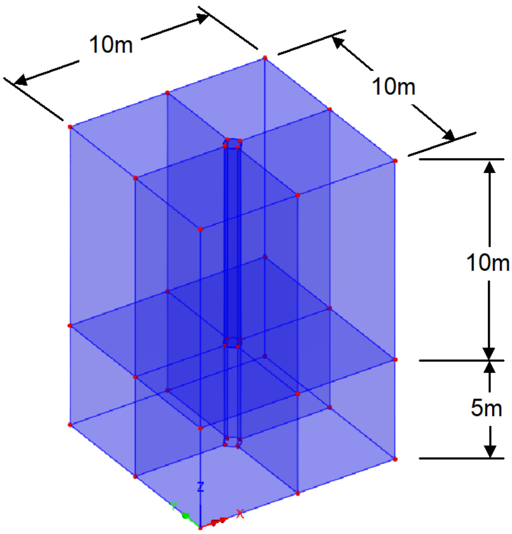
Pile Load Test

For LUSAS version:	24.0
For software product(s):	LUSAS Bridge plus or LUSAS Civil&Structural plus
With product option(s):	Geotechnical, Nonlinear

Problem Description

This example creates a model that simulates the behaviour of a pile during a load test. The 10m x 10m x 15m deep soil mass contains a centrally positioned pile of length 10m with a diameter of 0.8m. The pile is required to support a maximum load of 1720kN.



Model geometry

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.

A 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.

Note. This example, which requires the geometry defining the soil to intersect with the line representing the pile to be defined explicitly, was created prior to the availability of using embedded beams, joints and interfaces (which do not require such geometry considerations). As such, the use of embedded elements now greatly simplifies the modelling of piles in 2D and 3D continuum models. A separate worked example titled ‘Axially loaded pile’ shows the suggested way of modelling this type of problem.

Units of N, mm, t, s, C are used throughout.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_load_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.

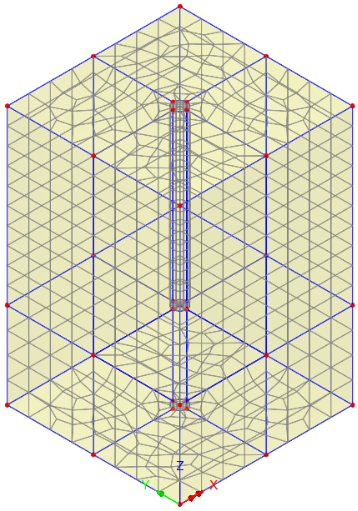
Objectives

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

Modelling overview

The model is created using an analysis category of **3D**, with model units of **kN,m,t,s,C**.

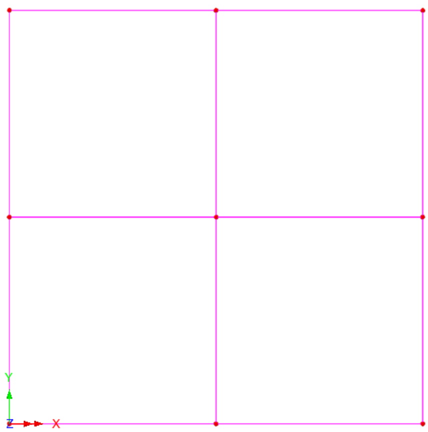
- After running the supplied script, turning off the supports and after choosing an isometric view the following view of the model will be seen.



The model created from running the supplied script

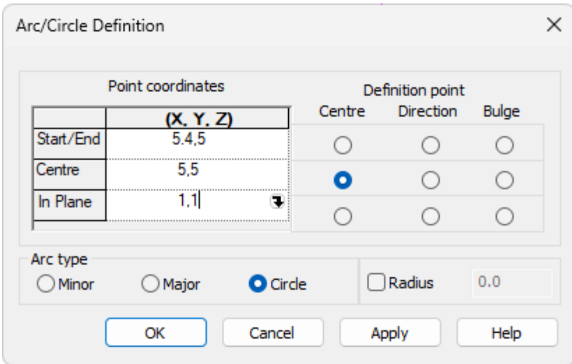
Feature Geometry

The model can be built in many ways. In this example, lines representing the bottom of the model can be created either by using the menu item **Geometry > Line By Points** or more easily by using the menu item **Geometry > Line > By Grid**.

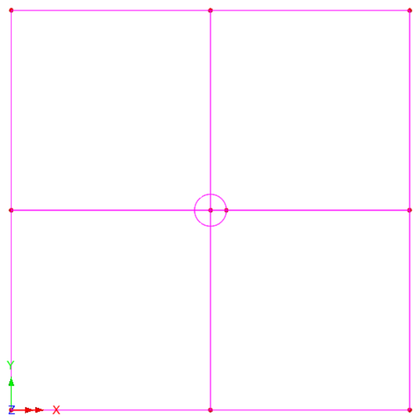


Lines that will be used to create surfaces

The circle representing the cross-section of the pile is generated by **Geometry > Line > Arc/Circle**.

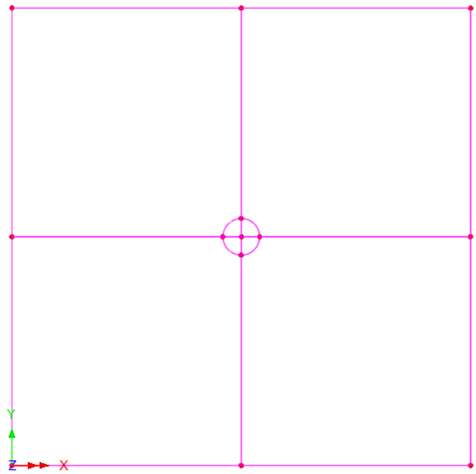


Inputs to create a circle of diameter 0.8m in the centre of model



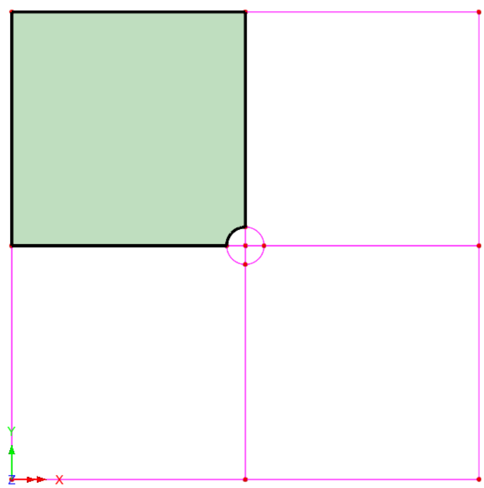
Circle added

The single line defining the circle that will be used to represent a pile is then split into 4 line divisions using **Geometry > Lines > By Splitting > In Equal Divisions**



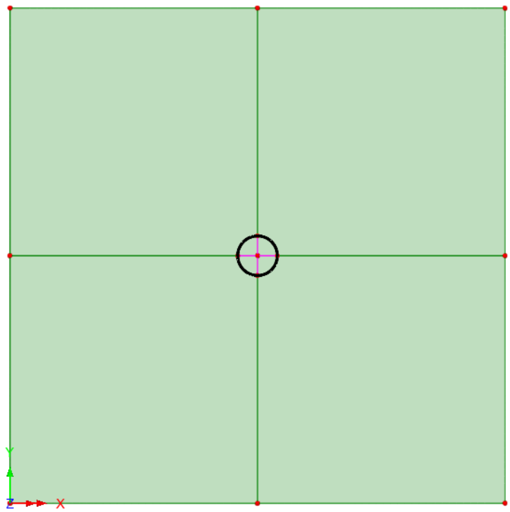
Circle split into 4 line divisions

Then, each point on the circle that intersects with a line can be selected along with the line and the menu item **Geometry > Line > By Splitting > At a Point** can be used to create separate lines. One arc and the appropriate four lines can then be selected and repeated to create each of the four main surfaces as shown below.

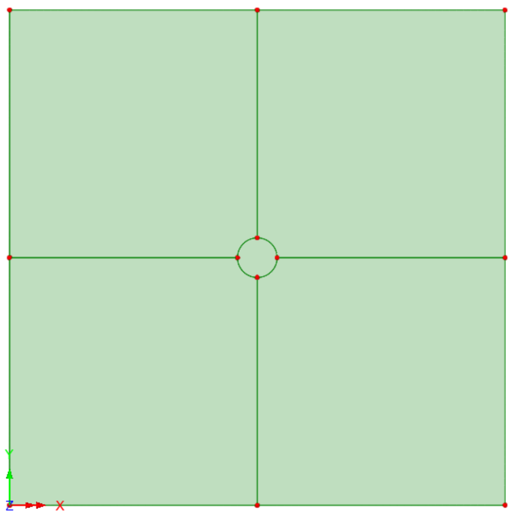


Surface creation from 5 selected lines

The 4 arcs defining the circle can also be selected and used to create a surface.

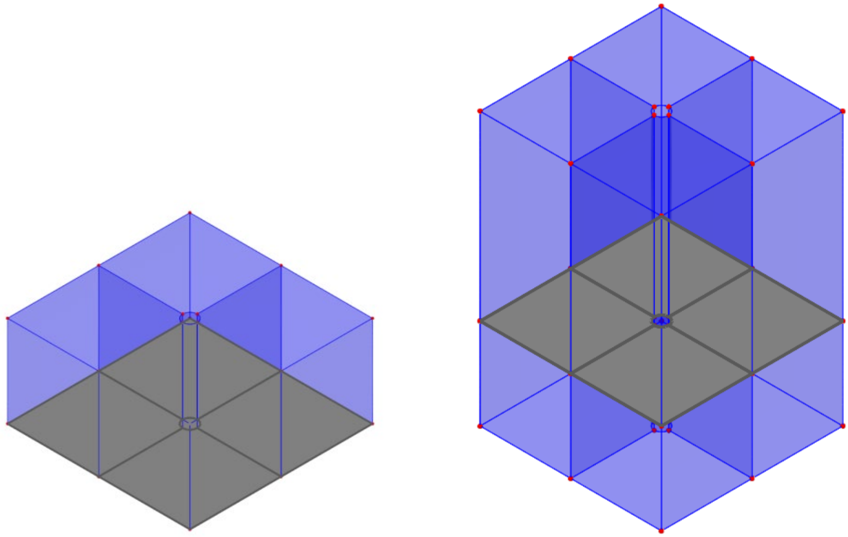


And the unwanted remaining lines and point in the middle of the circle then removed.



Top layout of soil model

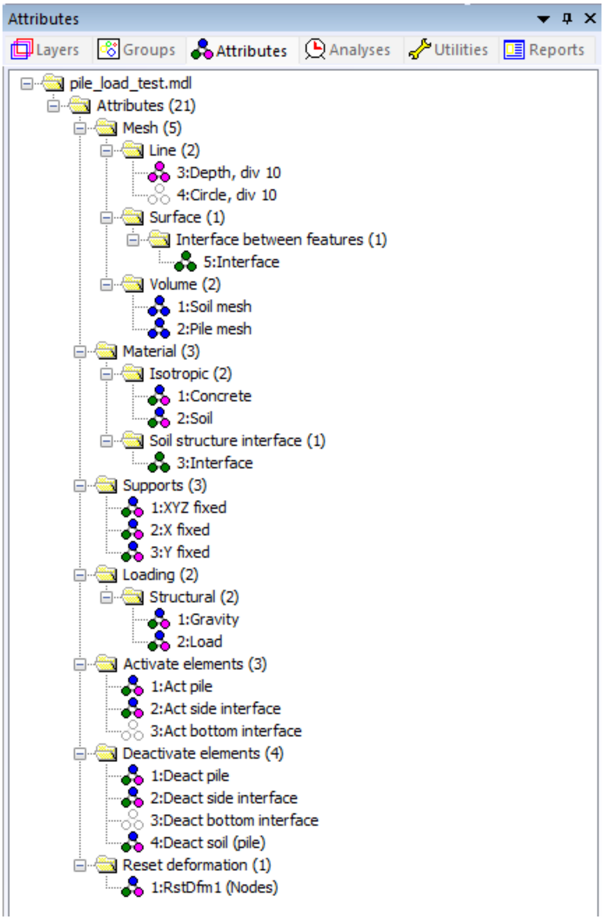
Volumes defining the soil mass are formed by selecting all surfaces and then using the menu item **Geometry > Volume > By Sweeping** to sweep those surfaces upwards through a distance of 5 in the Z direction initially. Then re-selecting the upper surfaces and sweeping those through a distance of 10 in the Z direction.



Sweeping of selected surfaces upwards to form volumes

Model Attributes

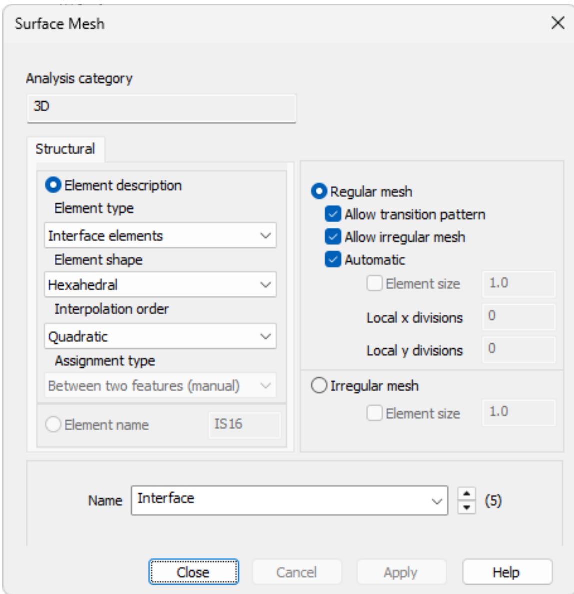
Model attributes (mesh, material, geometric properties, etc.) are defined and assigned to the model as shown in the Attributes treeview. Clicking on each will show more detail.



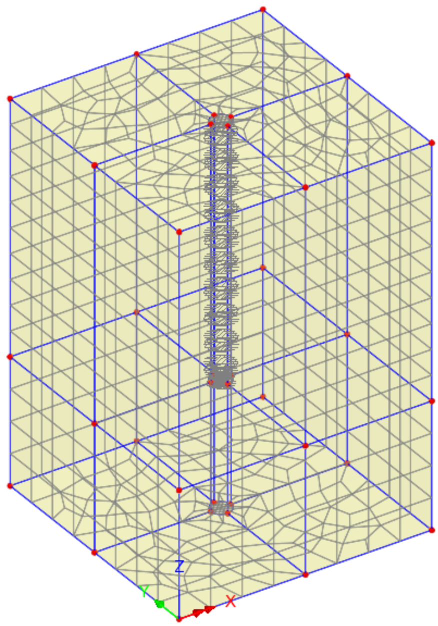
Model attributes

Mesh

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



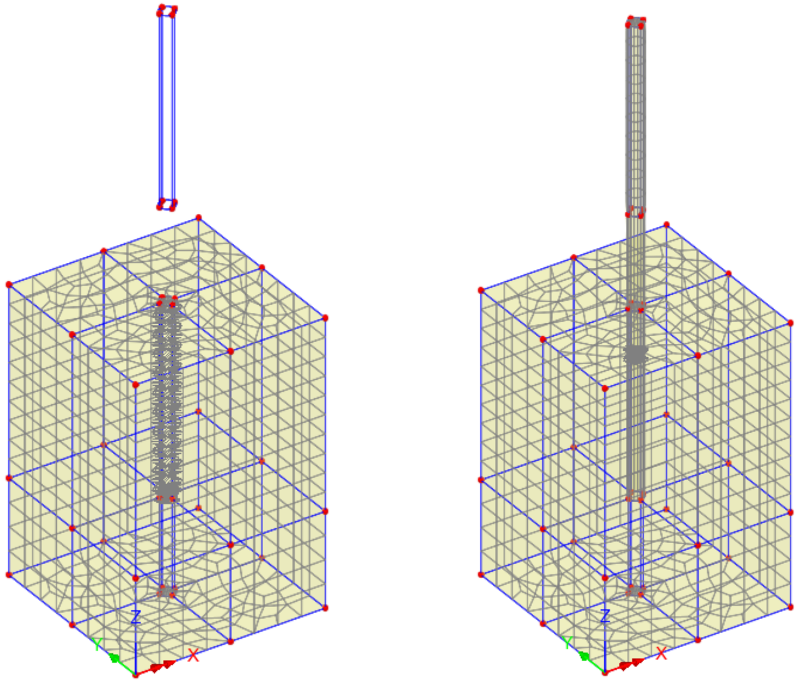
Interface mesh parameters



Meshed model

To model the pile, a separate volume is required so that pile material can be assigned and used when the pile is activated in the model. The following is done:

1. The volume representing the soil (Volume 2 in the supplied model) is copied to an upper location to create the volume (Volume 11) that represents the pile.
2. The interface mesh is assigned between the surrounding soil, side and bottom and the pile volume (Volume 11)
3. Before the pile volume is moved back, the volume, as well as its lines and points are made "**unmergeable**" so that it remains separate from the geometry defining the soil mass in that location.



Copying the volume representing the soil (left) and assigning interface mesh (right)

Materials

The soil material is 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

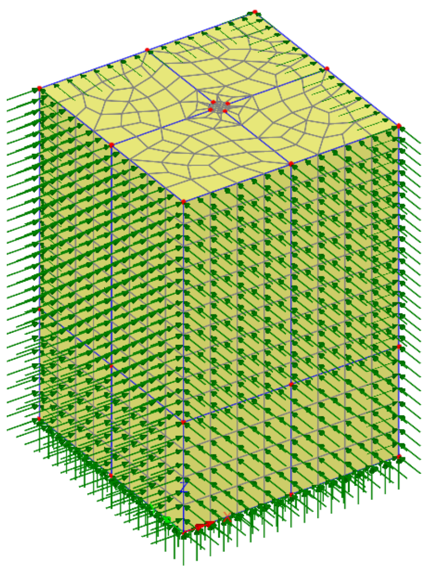
Material	Mass Density	Young's modulus, E	Poisson's ratio, ν	Angle of friction, φ	Cohesion, c	K_0
Soil	1.8 t/m^3	50E3 kPa	0.3	38°	10 kPa	0.384
Concrete	7 t/m^3	28.5E6 kPa	0.4	-	-	-

Interface properties

Material	Normal stiffness factor K_n	Tangential stiffness factor Ks	Angle of friction, φ	Cohesion, c
Soil	10	0.15	38°	10 kPa

Supports

The model is restrained on all faces in the X and Y direction. Fully fixed supports are assigned to the base.



Boundary conditions (mesh transparency turned off)

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**.

Other Attributes

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

Construction stages considered

The following analysis and construction stages are considered.

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

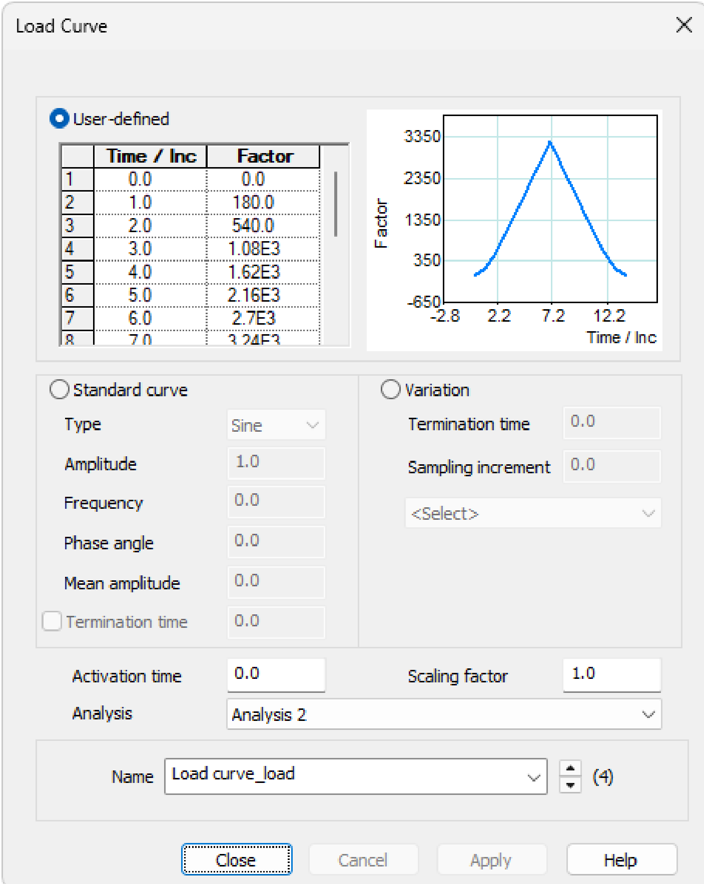
To install the pile in this phase, we need to make the pile volume and the associated interfaces active and the corresponding soil volume inactive.

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

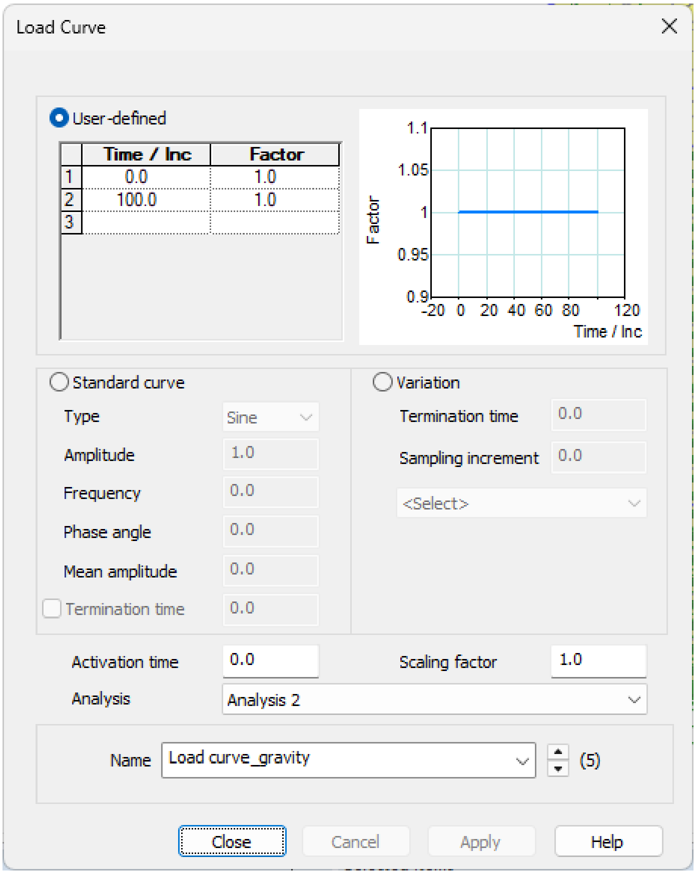
Analysis 2

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 right-clicking on the **Analysis 2** entry in the Analyses treeview and selecting **New > Load Curve**. Using load curves, we assign the gravity load and applied load over 14 time units (loading and unloading). The maximum applied load is 3240 kN/m^2.

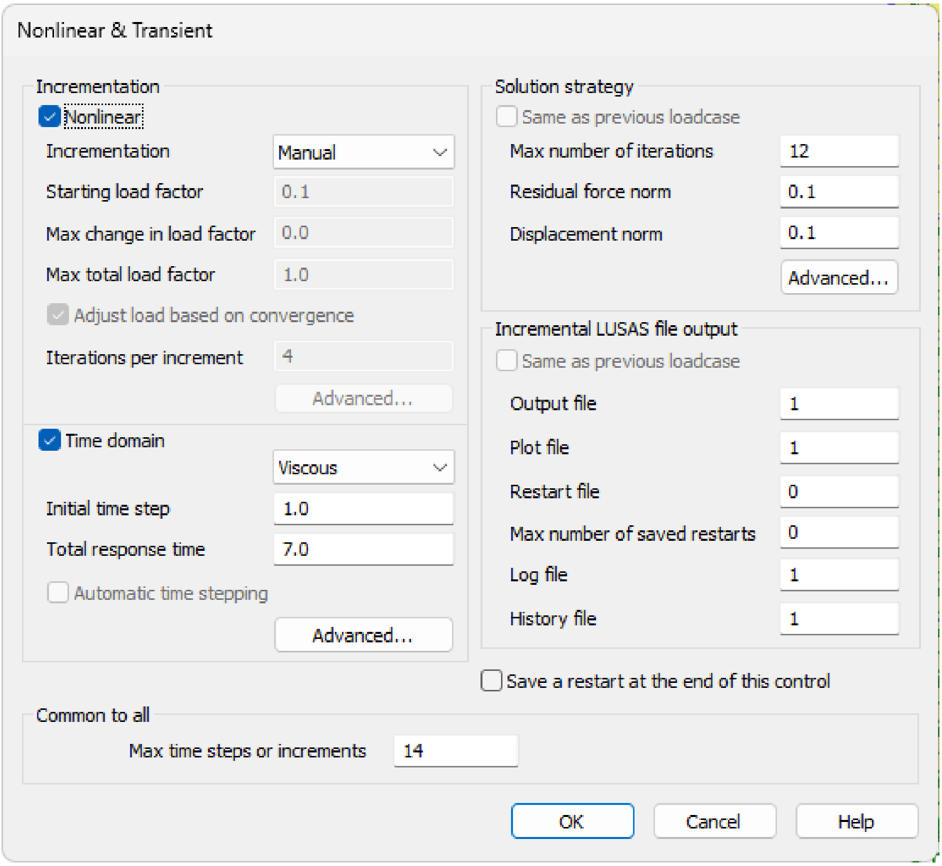


Load curve load - used in analysis 2



Load curve gravity - used in analysis 2

Viscous control properties are set as shown in the following image. The total response time is set to 7 time units with a time step of 1, while the maximum number of increments is set to14 to allow the solution to step halve if necessary.

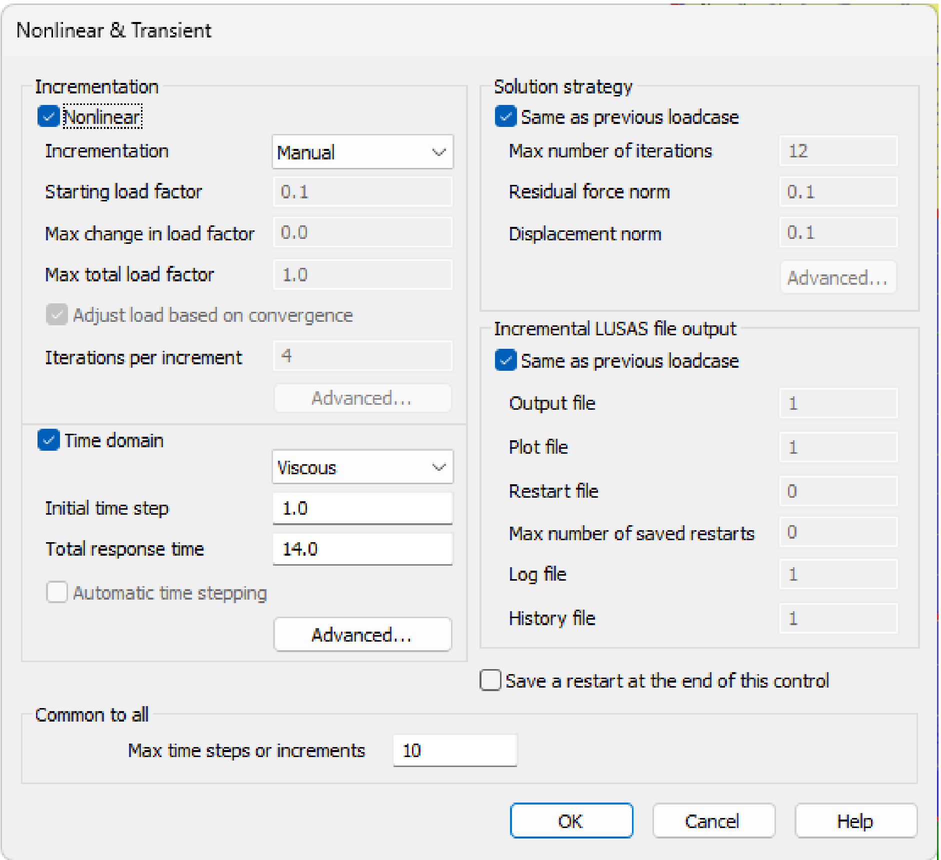


Nonlinear analysis control parameters - loading

Analysis 2

Unloading

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



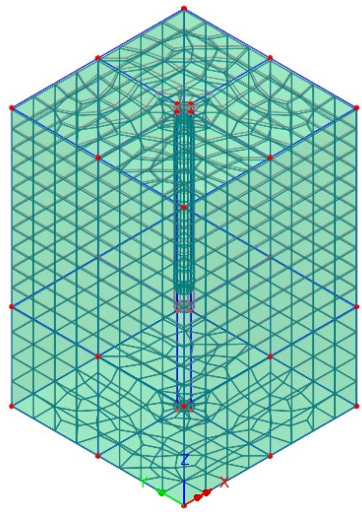
Nonlinear analysis control parameters - unloading

Running the Analysis

 Ensure all analyses are selected and press **OK** to solve.

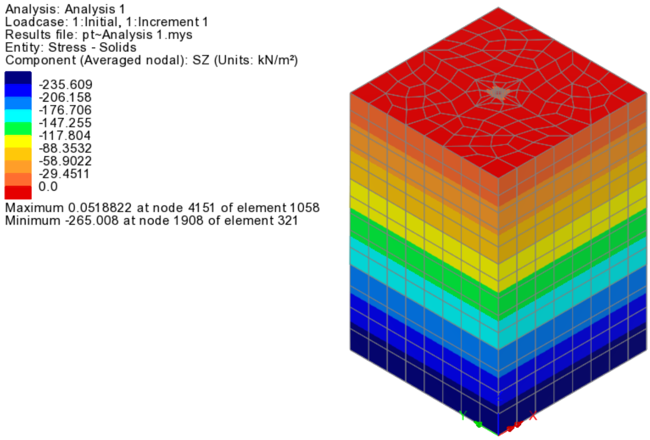
Viewing the results

Analysis loadcase results are present in the Treeview. The following view of the model will be seen.



Stress

- Turn off the display of the geometry and deformed mesh layers
- Add the **Contours** layer to plot contours of entity **Stress – Solids** and component **SZ**



Vertical stress at the initial stage (kN/m^2

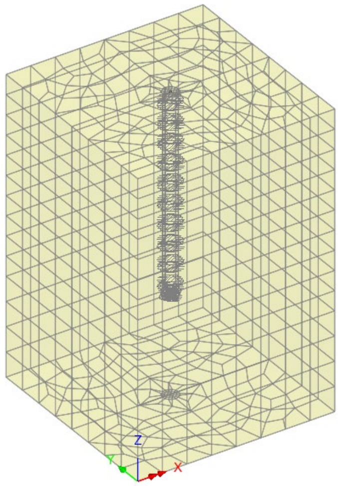
Graphing displacement of pile

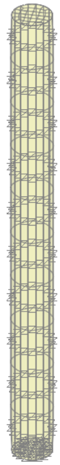
To plot a graph of the vertical displacement of the pile under loading and unloading:

- First, turn off the display of the contours layer.
- Set active the **Install pile** loadcase because this is the loadcase in which the pile is activated.

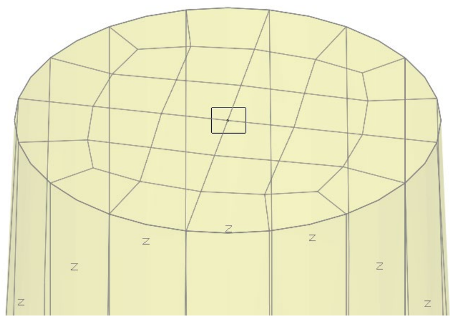
Caution. Failure to set the ‘Install pile’ loadcase active will result in node numbers for the soil being present, and give different results.

- Set the **Pile mesh** element attribute as ‘only visible’ to display just the pile.





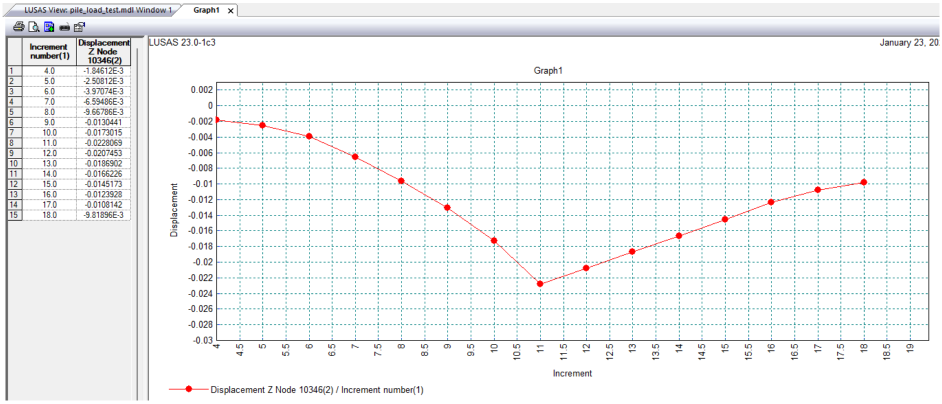
- Enlarge the view of the top of the pile and select the node in the centre. This will be node 10346.



To plot a graph of nodal displacement against time:

- Select the menu item **Utilities > Graph wizard**
- On the dialog select **Time history** and press **Next**
- Select **Named** then set Whole Analysis to **Analysis 2** and press **Next**
- Select **Increment number** and press **Next**
- Select **Nodal** and press **Next**,
- Select entity **Displacement** of component **DZ** and press **Next**.

- Finally enter an X and Y axis name of **Increment number** and **Displacement** respectively and press **OK**.



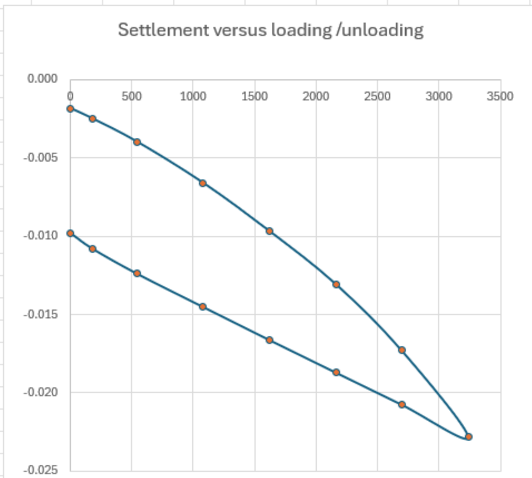
Loading-Unloading displacement (DZ)

The maximum displacement for the selected node is shown by the graph data to be 23mm

Alternative graphing

The following graph shows a loading / unloading curve that can be obtained by manually plotting the corresponding increment number for the load curve load factor against displacement obtained for the same increment.

Load curve load factor	Displacement
0	-1.85E-03
180	-2.51E-03
540	-3.97E-03
1.08E+03	-6.59E-03
1.62E+03	-9.67E-03
2.16E+03	-1.30E-02
2.70E+03	-1.73E-02
3.24E+03	-2.28E-02
2.70E+03	-2.07E-02
2.16E+03	-1.87E-02
1.62E+03	-1.66E-02
1.08E+03	-1.45E-02
540	-1.24E-02
180	-1.08E-02
0	-9.82E-03



Loading-Unloading displacement (DZ)