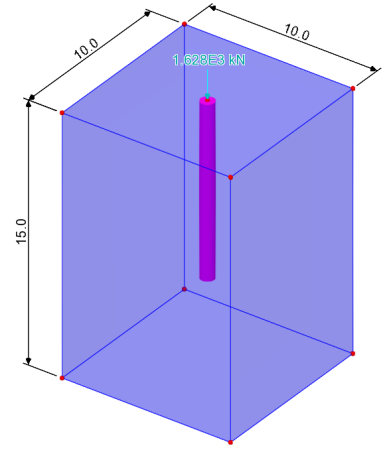
Modelling of an Axially Loaded Pile

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 simulates the response of a pile subjected to a single applied load. The soil domain, measuring 10m×10m in plan and extending to a depth of 15m, contains a centrally located pile with a length of 10m and a diameter of 0.8m. The pile is designed to carry a maximum load of 1627.78kN.



Model geometry

Keywords

Embedded Beam, Embedded Joint, Embedded Interface with Slip, 3D Modelling.

Associated Files

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

- **axially_loaded_pile.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 axially loaded pile and obtaining corresponding settlement.

Modelling overview

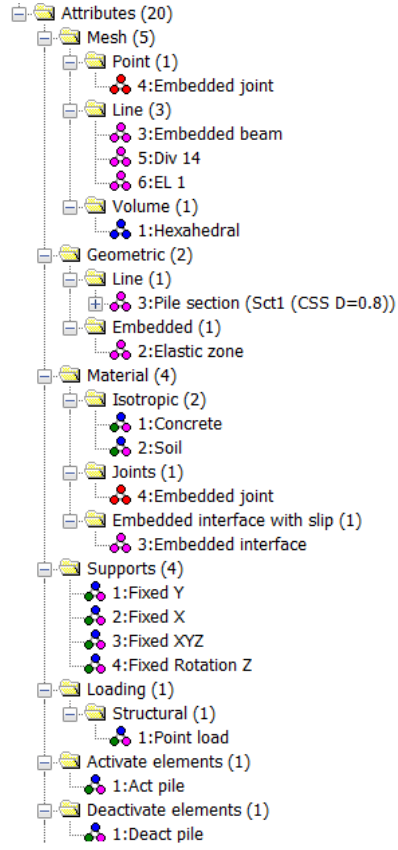
The model is created using an analysis category of **3D**, with model units of **kN,m,t,s,C**.

Feature Geometry

Regardless of the method used to create the model, it is essential to ensure proper connections between surfaces and to avoid any unintended overlaps. If additional guidance is required, users are strongly encouraged to review the step-by-step worked examples provided.

Model Attributes

Model attributes (mesh, material, geometric properties, etc.) are defined and assigned to the model.



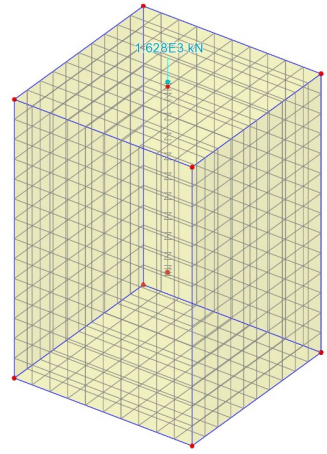
Model Attributes

Mesh

Solid hexahedral, quadratic stress elements (HX20) are used for meshing the volume representing the soil.

The line representing the pile is meshed using Embedded Beam with Interface elements (BMI31).

To account for the contribution of end-bearing resistance at the pile tip, an Embedded Joint (EJE_3D) is incorporated at the pile toe. This allows the model to explicitly represent interaction between the pile base and the underlying soil. It ensures that load transfer is not governed solely by shaft friction, but also includes resistance mobilised at the pile tip. This leads to a more realistic representation of pile behaviour, particularly where significant support is provided by a stiff stratum at depth.



Model mesh

Geometric Properties

The pile has a cross-sectional diameter of 0.8m.

The elastic zone diameter is set to be 0.9m.

Materials

The soil is modelled using Modified Mohr-Coulomb (HS) model, with parameters given in Table 1. The pile and soil-pile interface parameters are also summarised in Table 1.

Table 1: Soil material properties

	Mass Density (t/m^3)	Young's modulus, E (kPa)	Poisson's ratio, ν	Angle of friction, φ	Cohesion, c (kPa)	Earth pressure coefficient, K_0
Soil	1.8	1E5	0.3	38	10	0.3843
Concrete	2.4	28.5E6	0.2	-	-	-
Embedded interface with slip	-	-	-	38	10	-
Embedded point joint		1E12 kN/m				

Supports

Fully fixed supports are assigned to the base, while the lateral sides are fixed in the horizontal directions.

Loads

There are two types of loads that are being taken into consideration: the self-weight and a point load applied at pile top (1627.78kN).

Other Attributes

- ❑ Activate and Deactivate.
- ❑ Reset Deformation.

Analyses Considered

Three stages are considered.

Initial Phase

Initial stresses are determined for a soil body at this phase. Gravity is included in this phase by right-clicking on the phase name and selecting **gravity**.

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

Construction (Installation of Pile)

To install the pile, the thick beam elements are activated. The deformation caused by the weight of the soil is reset for this load case.

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

Loading

Axial load is applied at the pile top. Automatic nonlinear analysis control properties are defined.

Running the Analysis


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

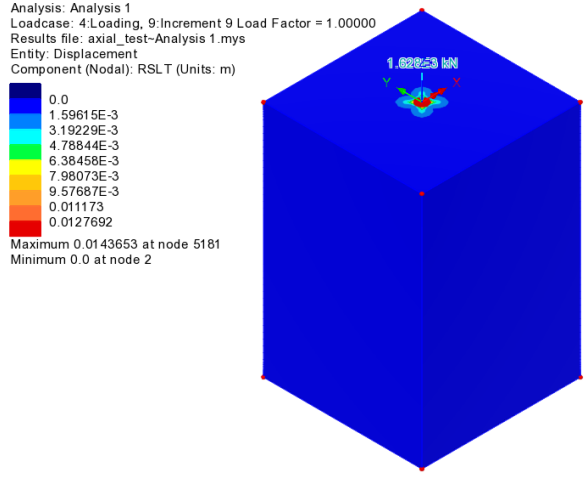
Viewing the Results

Analysis loadcase results are present in the Treeview.

- Press the **Isometric** button.
- Turn off display of the supports.

Displacements

- Turn off the display of the **Mesh** and **Deformed mesh** layers.
- Set active the **Loading** loadcase.
- Add the **Contours** layer to plot contours of entity **Displacement** and component **RSLT**

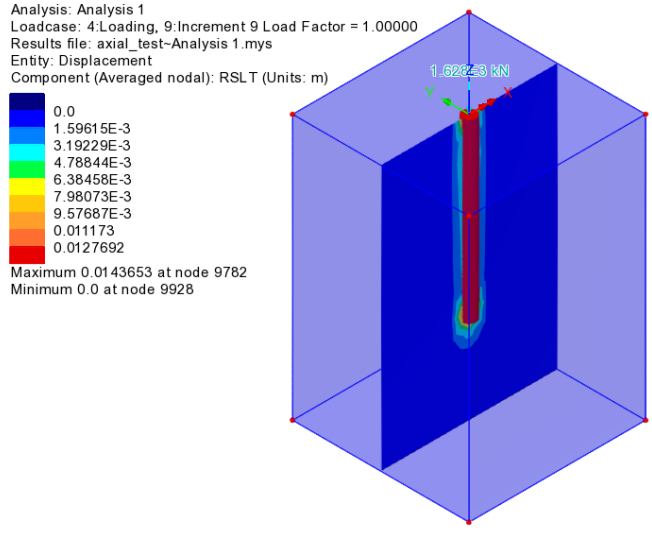


Displacements (m)

To see contours of displacement within the continuum:

- View the model directly from above.

- Select **Utilities > Section through 3D** and slice the model horizontally on the screen along Y=0.
- Choose an isometric view or otherwise rotate the model to see a view like the following one



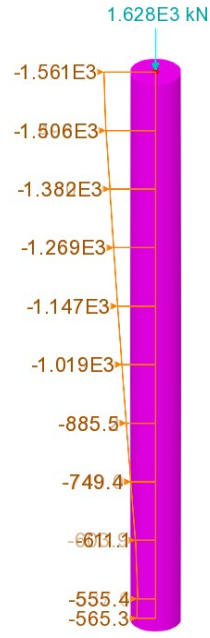
Displacements shown on slice section through model

Axial Forces

Moments and forces can also be determined for structural elements.

- Select the line representing the pile and set as only visible.
- Turn on the display of the **Diagrams** layer and choose to plot entity **Force/Moment – Thick 3D Beam** and component **Fx**.
- Turn off the display of the **Contours** layers.
- Click in the view window to deselect the line.

The next image shows the axial force in the pile.



Axial force (kN)