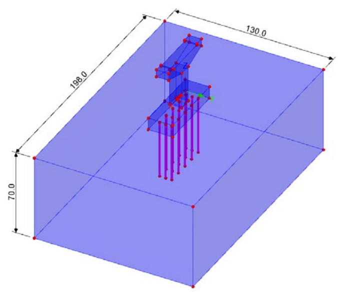
Pile-Supported Bridge Pier Foundation

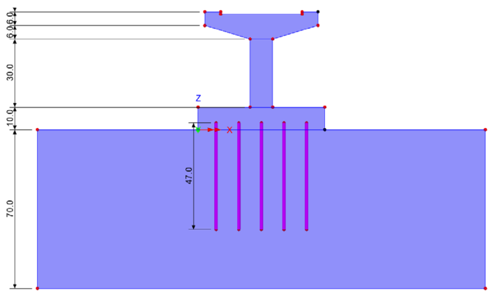
For LUSAS version:	24.0
For software product(s):	LUSAS Bridge plus or LUSAS Civil&Structural plus
With product option(s):	Geotechnical, Nonlinear

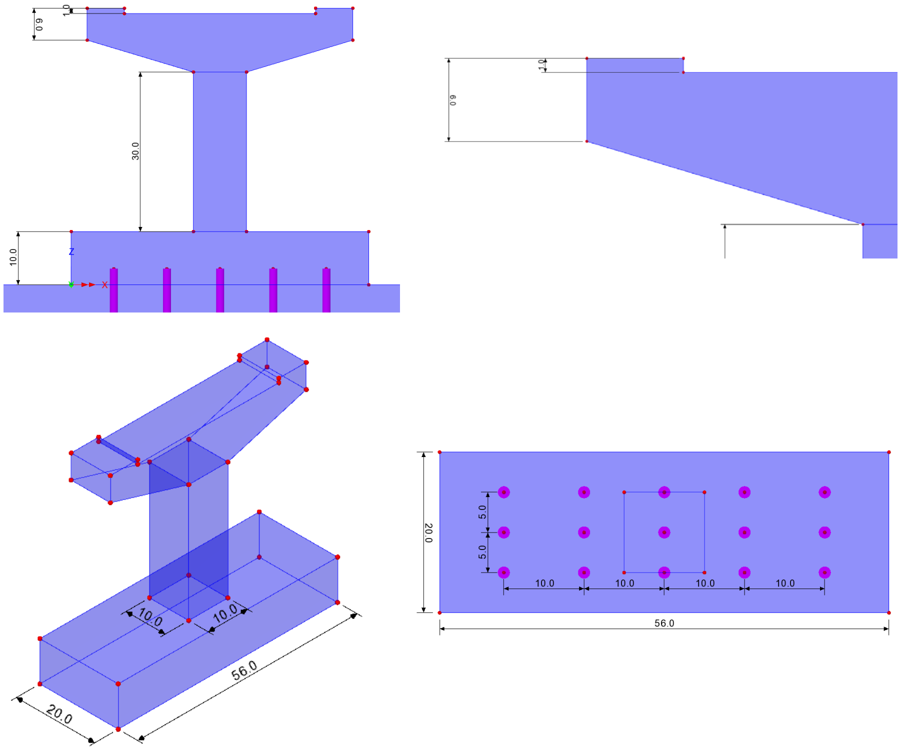
Problem Description

This example presents a three-dimensional model of a bridge pier foundation system supported by a pile group, with each pile having a diameter of 1.5m. The bridge pier is subjected to two non-uniform applied loads of 300kN/m² and 500kN/m².

The geometric configuration and dimensions of the model are shown in the following images.







Model geometry

Keywords

Embedded Piles, Volume Element, Interface, Joint, Settlement, Axial Forces

Associated Files

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

- **bridge_pier_foundation.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.

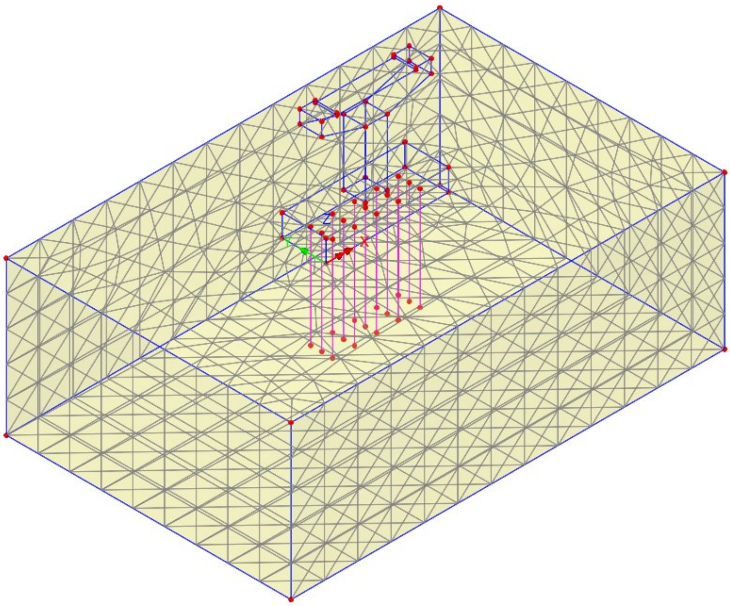
Objectives

The objective is to evaluate and compare the settlement behaviour in two scenarios: with piles and without piles.

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 and then pressing the **Isometric** button and turning off the **Supports** the following view of the model is obtained.



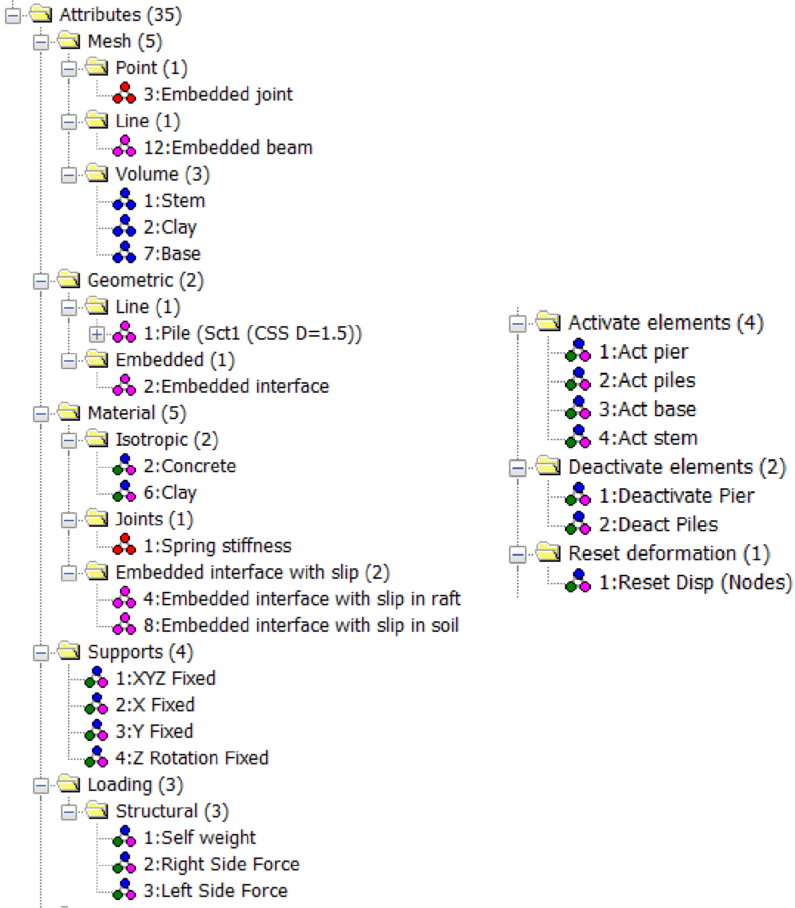
Model created from running the supplied script

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 examples provided in the help manual.

Model Attributes

Model attributes (mesh, material, geometric properties, etc.) have been defined and assigned to the model as shown in the following images.



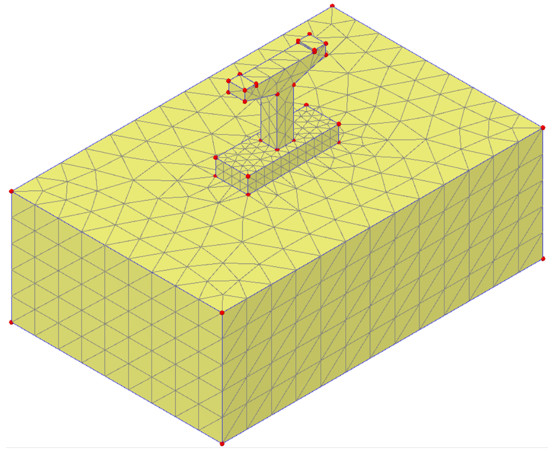
Model Attributes

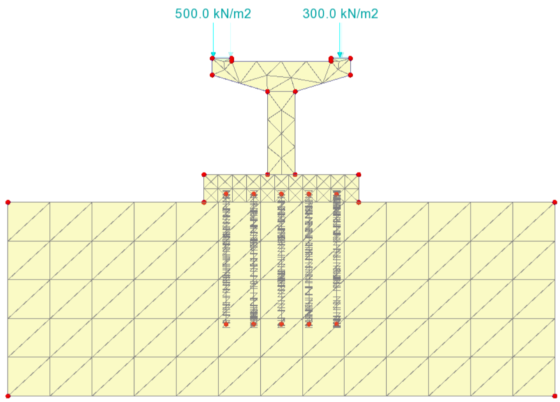
Mesh

Solid tetrahedral, quadratic stress elements (TH10) are used for meshing the volume parts.

A rigid connection is assumed between the piles and the raft and modelled using an Embedded Joint.

Pile capacity is derived from both shaft friction and end bearing. To achieve this, the lines representing the piles are meshed with Embedded Beam with Interface elements (BMI31) and an Embedded Joint is assigned at the toe of each pile.





Meshing

Geometric Properties

The diameter of the piles is 1.5m.

The elastic zone diameter is set to be 1.5m.

Materials

The clay material is represented using a Mohr-Coulomb model with the parameters provided in Table 1, while the piles are assumed to be constructed from concrete.

The parameters adopted for the interface and the joints along the pile shaft and at the pile tip are presented in Table 2.

Table 1: Soil material properties

	Young's modulus, E (kPa)	Poisson's ratio, ν	Mass Density (t/m^3)	Cohesion, c (kPa)	Angle of friction, φ	Earth pressure coefficient, K_0
Clay	50000	0.35	2	20	27	0.574
Concrete	2E7	0.3	2.5	-	-	-

Table 2: Interfaces material properties

	Cohesion, c (kPa)	Angle of friction, φ	Normal stiffness factor	Tangential stiffness factor
Embedded interface with slip in soil	15	25	10E6	0.15
Embedded interface with slip in raft	1E3	45	1E9	1
Embedded joint	-	-	Elastic spring stiffness: 1E12	

Supports

Fully fixed boundary conditions are applied at the model base, while the lateral boundaries are restrained in the horizontal directions. The piles are also constrained against rotation about the Z-axis.

Loads

There are three types of loads that are being taken into consideration: self-weight, distributed load acting at the left and right bearings of 500kN/m^2 and 300kN/m^2 respectively.

Other Attributes

To simulate the construction stages of the bridge pier, we need to use the Activate and Deactivate attributes.

Analyses Considered

For comparison purposes, four construction stages are adopted, with two branches originating from the initial stage. The following cases are considered:

- ❑ **Initial stage → Pier construction (no loads applied, no piles installed)**
  Used to determine the settlement after pier construction and prior to load application.
- ❑ **Initial stage → Pier construction (loads applied, no piles installed)**
  Used to determine the settlement after load application without piles.
- ❑ **Initial stage → Piles installation → Pier construction → Loading**
  Used to determine the settlement after pile installation, completion of pier construction, and load application.

A comparison can then be made between these cases and the other considered conditions.

Initial Phase

We stablish the initial stress state of the soil at this stage. The bridge pier and piles are deactivated.

Manual nonlinear analysis control is specified for this phase, with all parameters retained at their default values.

Initial Phase > Pier construction (no loads, no piles)

The bridge pier is activated.

Manual nonlinear analysis control is specified for this phase, with all parameters retained at their default values.

Initial Phase > Pier construction (with loads, no piles)

The bridge pier is activated and loads at bearings are considered.

Manual nonlinear analysis control is specified for this phase, with all parameters retained at their default values.

Pile installation

The piles are activated together with the base.

Manual nonlinear analysis control is specified for this phase, with all parameters retained at their default values.

Pier Construction

The pier is activated.

Automatic nonlinear analysis control is specified for this phase, with all parameters retained at their default values.

Lock-in phase

This manual loasdcase is to ensure that the weight of the pier will be inherited in the ‘Loading’ loadcase that follows.

Loading

Loads at bearings are included in this stage.

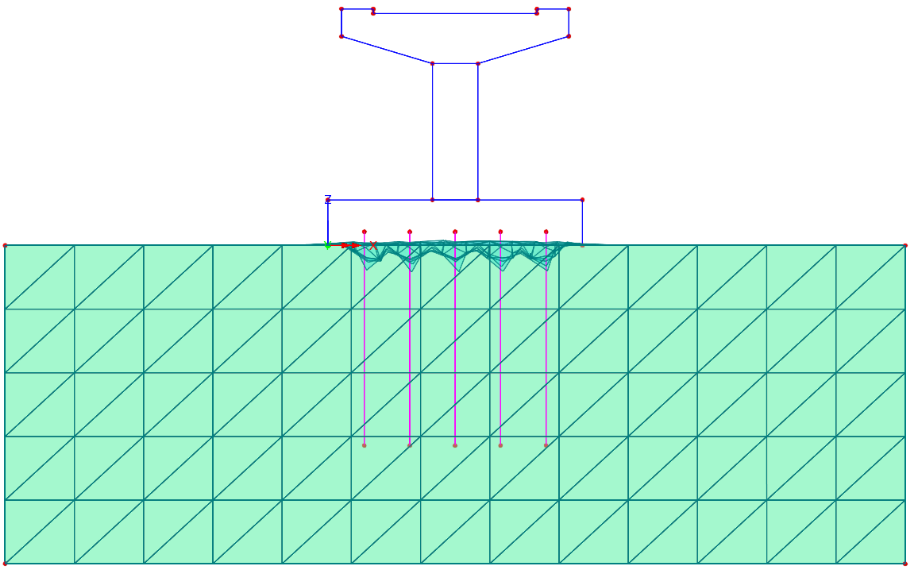
Automatic nonlinear analysis control is specified for this phase, with all parameters retained at their default values.

Running the Analysis

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

Viewing the Results

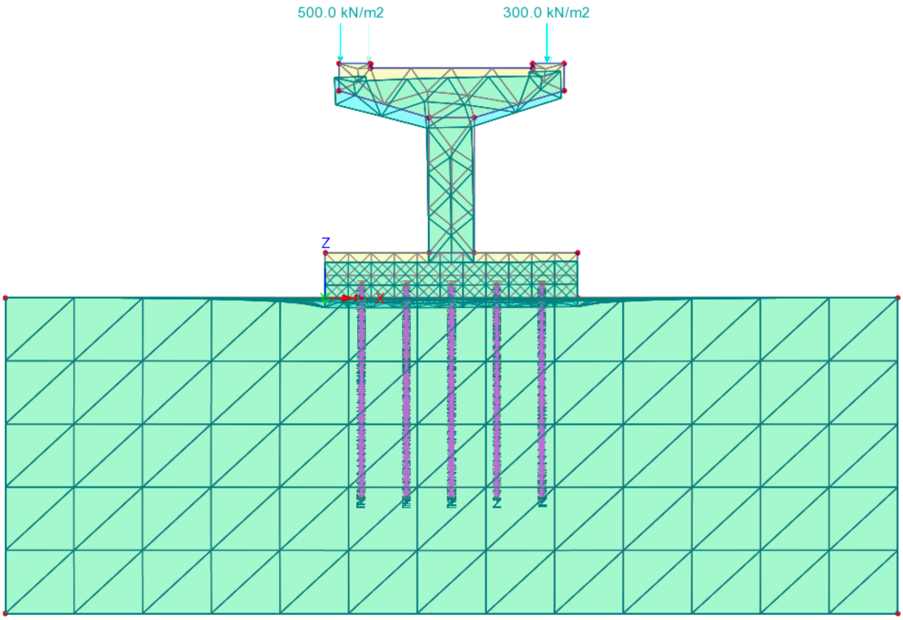
Analysis loadcase results are present in the Analyses treeview. When viewed from along the Y-axis the following view will be seen:



View along Y-axis

Deformations and Displacements

- Set active the **Loading** loadcase (the final loadcase).

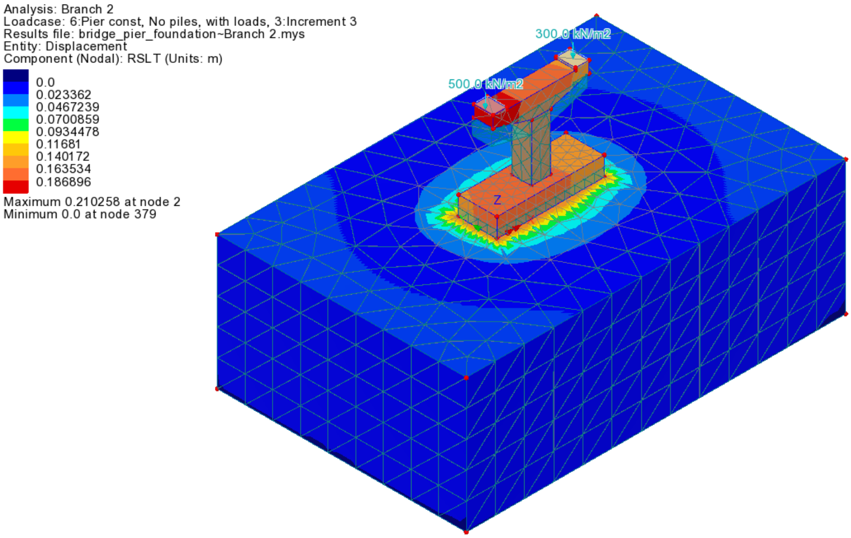


Deformed mesh at final stage.

Displacements

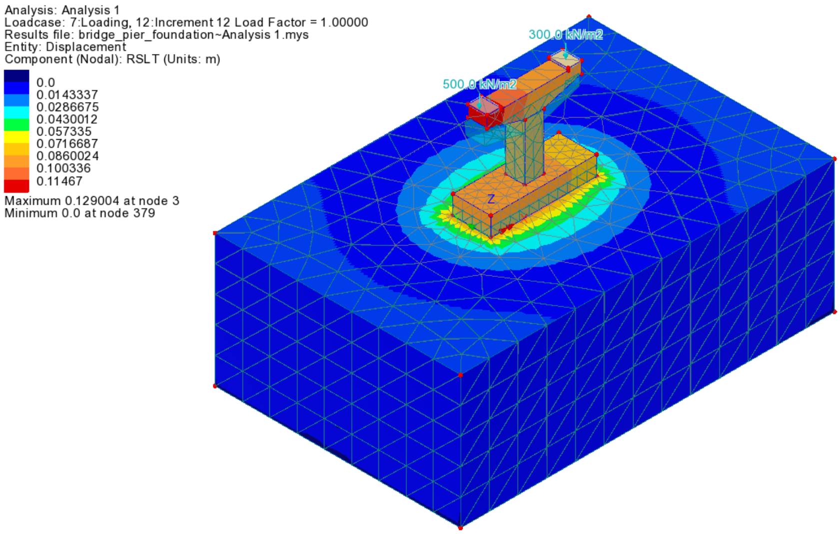
The displacement contours for the cases with and without piles are to be compared.

- Add the **Contours** layer to plot contours of entity **Displacement** and component **RSLT**
- Choose an **Isometric** view.
- In the **Branch 2** analysis, set active loadcase **Pier const, No piles, with loads**



Displacements at final stage for the case without piles

- To see the displacement contours case with piles, set active the loadcase **Loading**

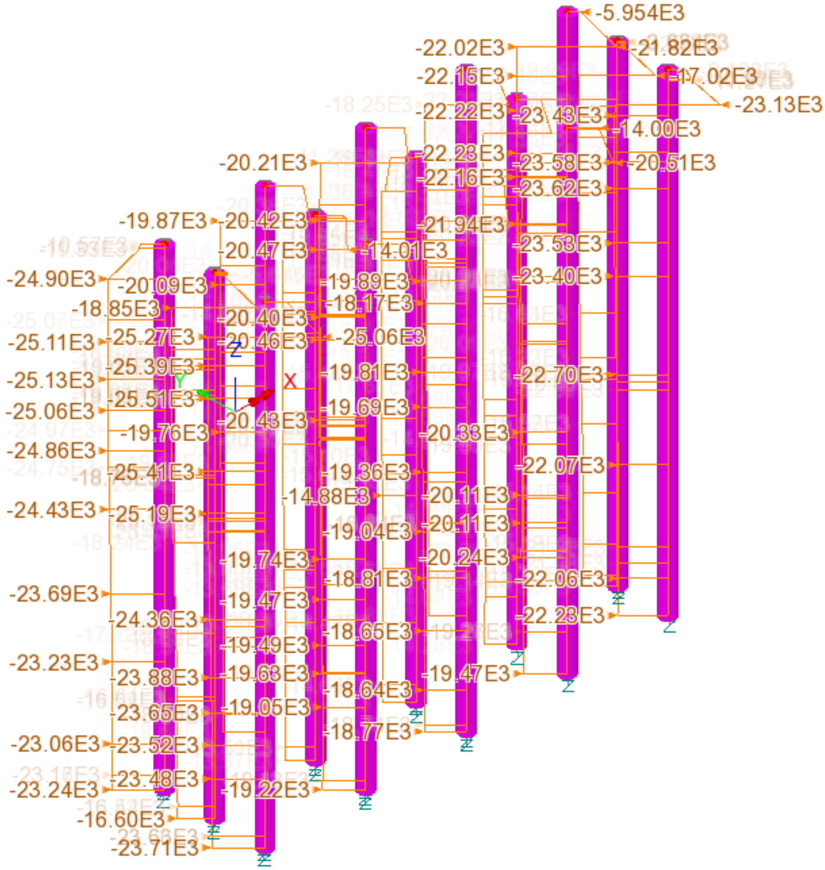


Displacements at final stage for the case with piles

The presence of piles reduces the settlement by around 55%

Axial Forces in Piles

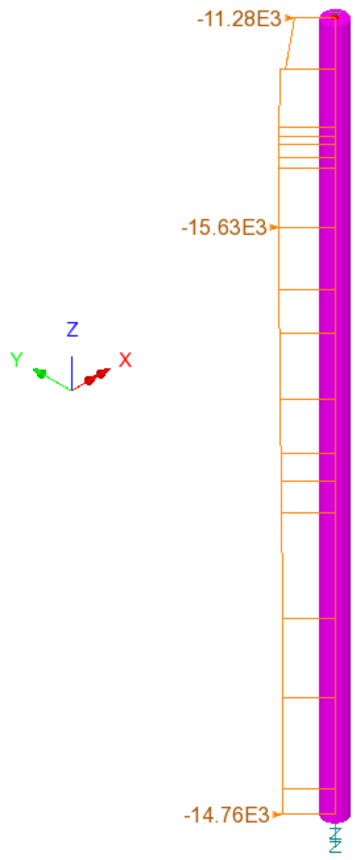
- Turn off the display of the **Contours** layer.
- In the Groups treeview, for the **Piles** group, choose **Set as Only Visible**.
- Add the **Diagrams** layer to plot component **Force/Moment - Thick 3D Beam** and entity **Fx**.



Axial forces in pile elements (kN)

To show results on the central pile:

- In the Groups treeview, for the **Central pile** group, choose **Set as Only Visible**.
- In the Layers treeview, for the **Diagram** layer, set the **Diagram display** to show **Peaks only.**



Axial forces (peaks only) in central pile (kN)

This completes the example.