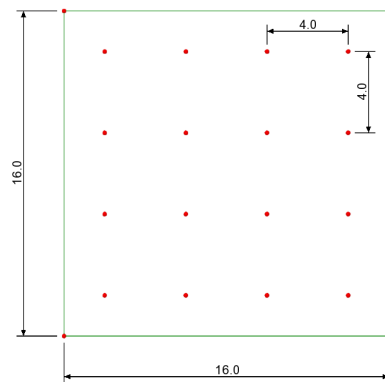


# Piled Raft Foundation

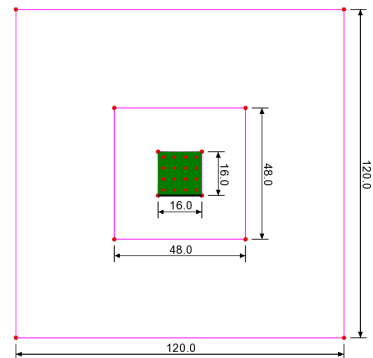
For LUSAS version:	24.0
For software product(s):	LUSAS Bridge plus or LUSAS Civil&Structural plus
With product option(s):	Geotechnical, Nonlinear

## Problem Description

A uniformly distributed load of  $30\text{kN/m}^2$  is applied to a piled raft constructed on a single layer of soft clay. Beneath this clay, a firm rock stratum lies at a depth of  $40\text{m}$  from the ground surface. The groundwater table is positioned  $3\text{m}$  below ground level. The foundation system consists of square precast concrete piles, each  $275\text{mm}$  wide,  $20\text{m}$  in length, supporting a  $16\text{m} \times 16\text{m} \times 0.5\text{m}$  thick raft. The piles are arranged in a regular grid with a spacing of  $4\text{m}$ .



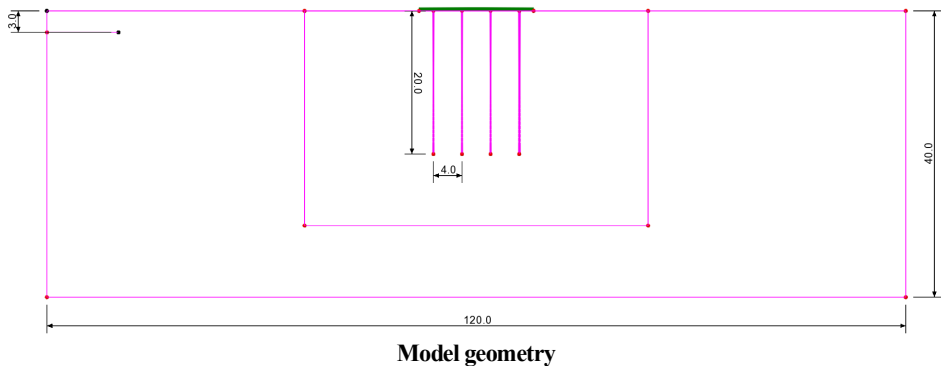
**Raft and pile layout**



**Plan geometry**

## Piled Raft Foundation

---



### Keywords

Embedded Piles, Shell Element, Interface, Joint, Water Table, Settlement, Axial Forces

### Associated Files

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



- piled\_raft\_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. This may take a few minutes to create the model.

### Objectives

The aim is to evaluate the behaviour and effects of pile installation with respect to settlement and the axial forces developed within the piles.

## Modelling overview

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

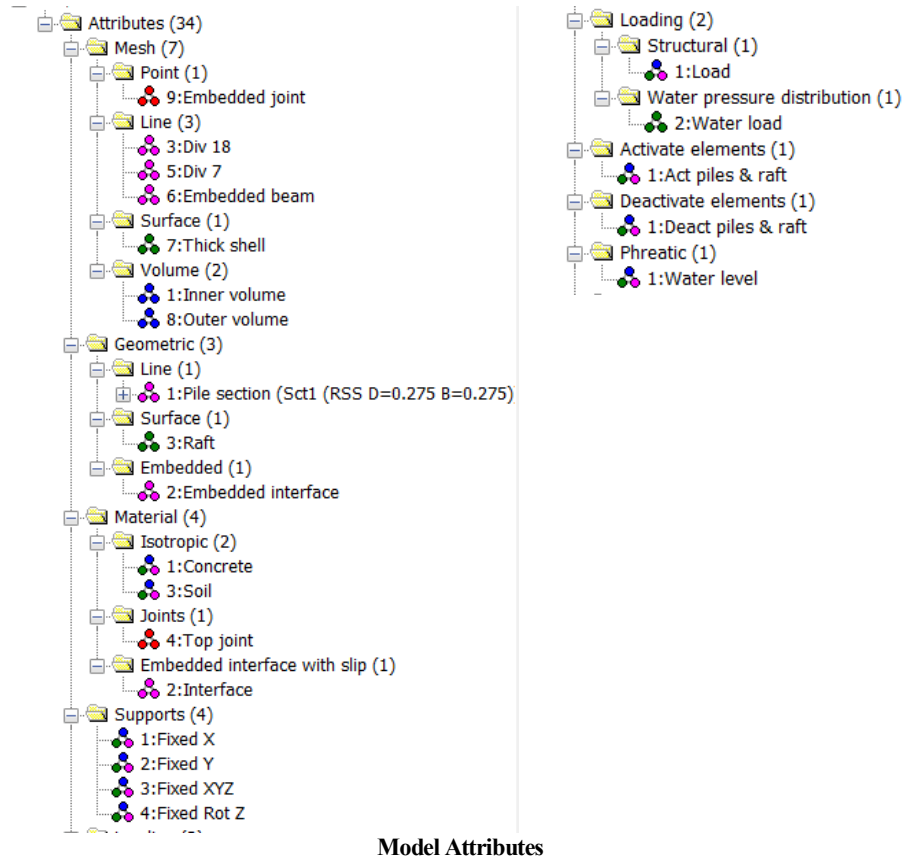
### Feature Geometry

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 examples provided.

## Model attributes

Model attributes (mesh, material, geometric properties, etc.) are defined and assigned to the model. (Figure 2).



## Mesh

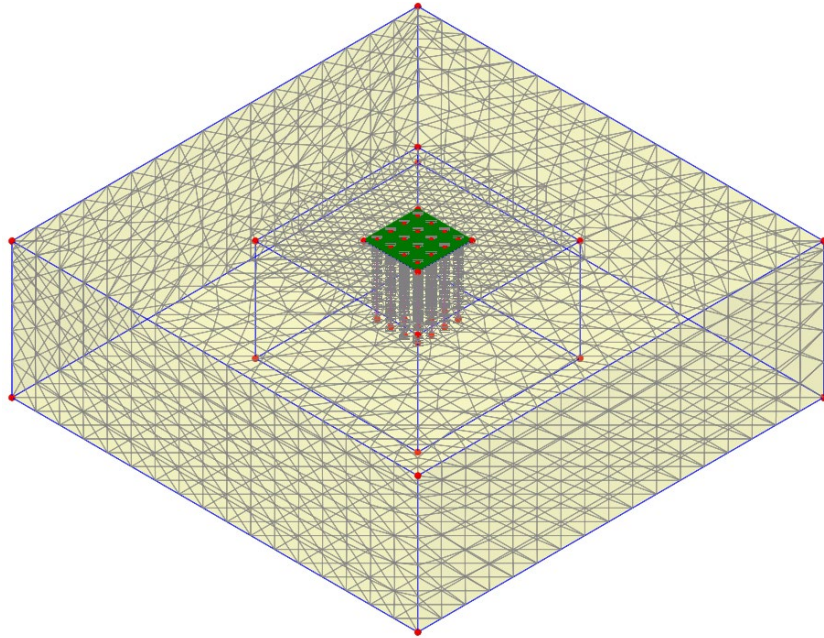
Solid tetrahedral, quadratic stress elements (TH10) are used for meshing the volume parts, the raft is meshed with Thick Shell, Triangle, quadratic elements (TTS6).

The lines representing the piles are meshed using the Embedded Beam with Interface element (BMI31).

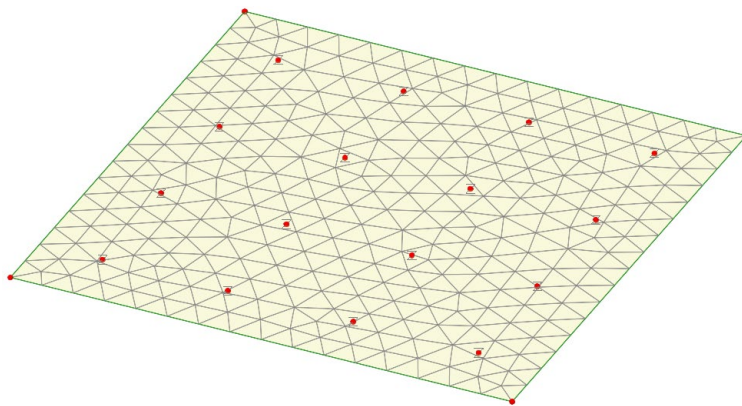
## Piled Raft Foundation

---

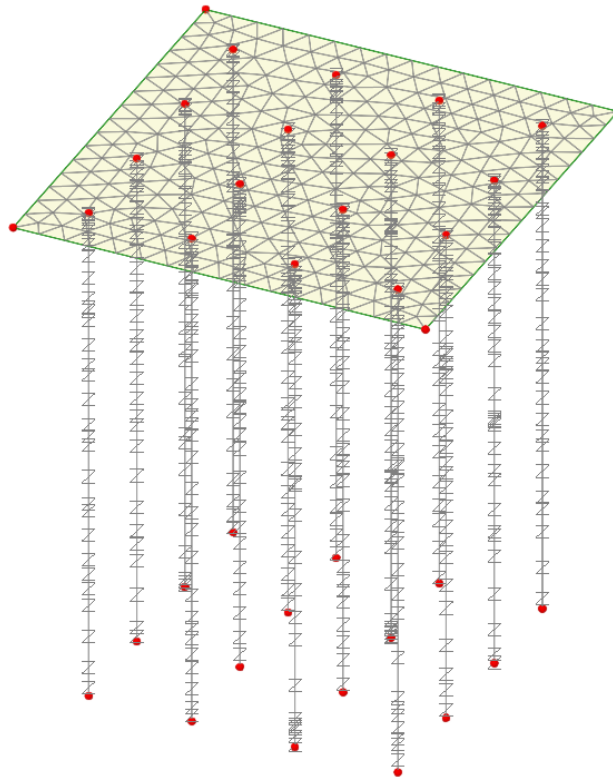
A rigid connection is assumed between the piles and the raft. This is meshed using an Embedded Joint.



**Model mesh**



**Top rigid connection with the raft (Embedded Joint)**



Piles meshed with embedded beam with interface

## Geometric Properties

The piles have a cross-sectional dimension of  $0.275\text{m} \times 0.275\text{m}$ .

The raft thickness is 0.5m.

Elastic zone diameter is set to be 0.3m.

## Materials

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

## Piled Raft Foundation

---

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**

	Mass Density (t/m <sup>3</sup> )	Young's modulus, E (kPa)	Poisson's ratio, $\nu$	Angle of friction, $\phi$	Cohesion, c (kPa)	Earth pressure coefficient, $K_0$
Clay	1.8	5000	0.35	30	4	0.5
Concrete	2.5	35.0E6	0.2	-	-	-

**Table 2: Interfaces material properties**

	Cohesion, c (kPa)	Angle of friction, $\phi$	Normal stiffness	Tangential stiffness
Embedded interface with slip	3.2	24.79	50.0E3	6.0E3
Embedded joint	-	-	Elastic spring stiffness: 1E12	

## Supports

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

## Loads

Three types of loads are taken into consideration: self-weight, a distributed load acting at raft (30 kPa) and the effect of water.

The water table, as earlier stated, is located 3m below ground surface.

## Other Attributes

In addition to the previous attributes, to simulate the construction stages of the foundation, Activate and Deactivate attributes are used.

## Analyses considered

Three stages for the construction of piled raft foundation are considered in the model as follows:

### Initial Phase

In geotechnical engineering, the initial stress state of the soil must first be established. Gravity is included in this phase by right-clicking on the phase and selecting the gravity option. The water table is also considered from the start of this phase, while the raft and piles are deactivated.

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

### Installation of raft and piles

Raft foundation and piles are activated.

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

### Loading

Distributed Load at raft foundation is activated at this stage. Automatic nonlinear analysis control properties are defined.

## Running the Analysis



Ensure all analyses are selected and press **OK** to solve. This may take around 25 minutes to solve depending upon the machine specification.

## Viewing the Results

Analysis loadcase results are present in the Treeview.

### Displacements

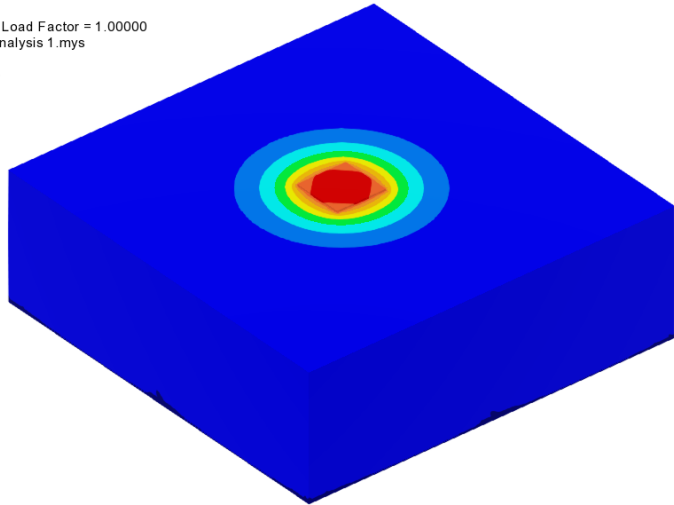
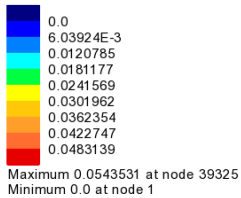
- Turn off the display of the **Deformed mesh** layer, the **Mesh** layer and the **Geometry** layer, and the **Supports** and **Loading**.
- Add the **Contours** layer to plot contours of entity **Displacement** and component **RSLT**

## Piled Raft Foundation

---

- Set active the **Loading** loadcase (Loadcase 3)
- View the model like shown below.

Analysis: Analysis 1  
Loadcase: 3>Loading, 6>Increment 6 Load Factor = 1.00000  
Results file: piled\_raft\_foundation-Analysis 1.mys  
Entity: Displacement  
Component (Nodal): RSLT (Units: m)

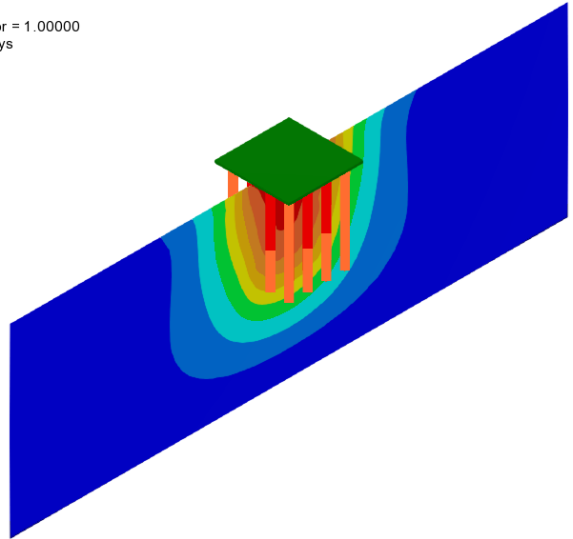
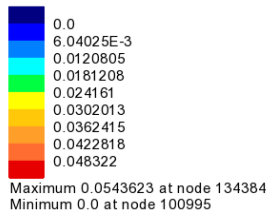


### Displacements at final stage

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  $Y=60$ .
- Choose an isometric view, or otherwise rotate the model to see a view like this one:

Analysis: Analysis 1  
 Loadcase: 3:Loading, 6:Increment 6 Load Factor = 1.00000  
 Results file: piled\_raft\_foundation~Analysis 1.mys  
 Entity: Displacement  
 Component (Averaged nodal): RSLT (Units: m)



Slice section showing displacements at final stage

## Axial Forces

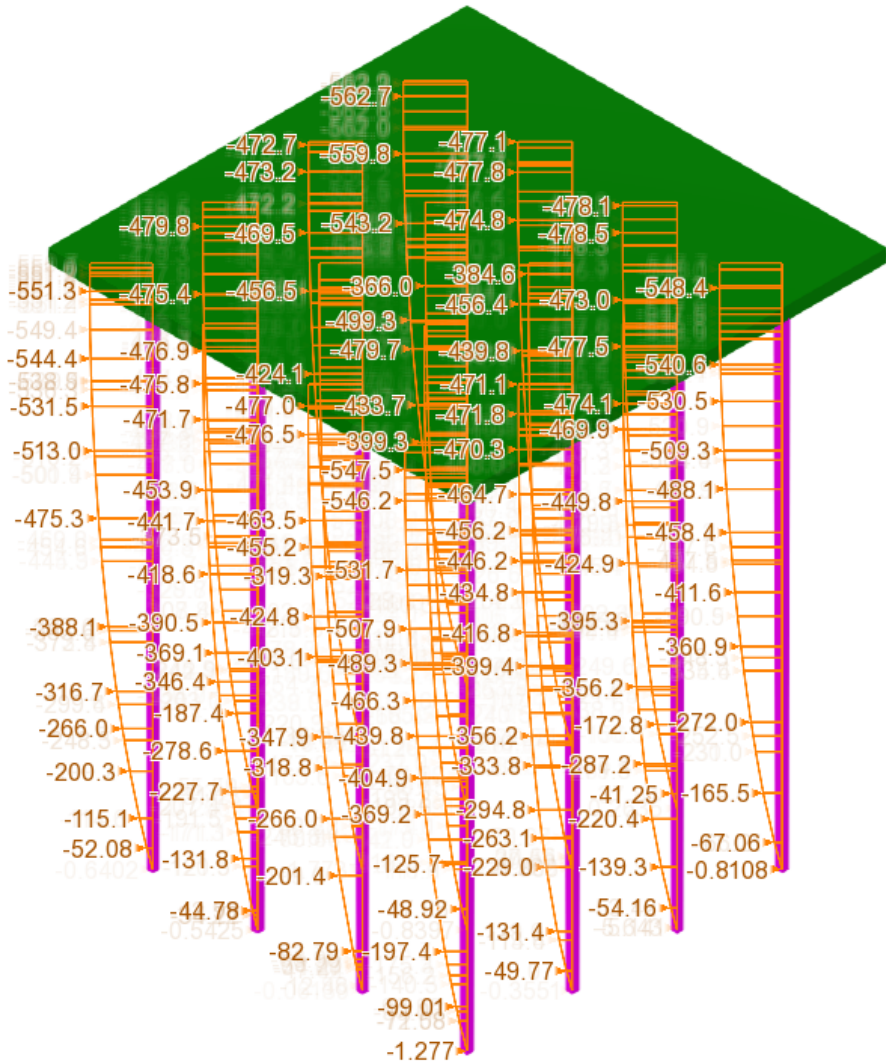
It is also possible to determine the moments and forces for structural elements.

- Turn off the display of the **Contours** layer.
- In the Attributes treeview right-click on the geometric surface attribute **Thick shell** and select **Set as Only Visible**.
- In the Attributes treeview right-click on the geometric line attribute **Embedded beam** and select **Visible**.
- Right-click in the view window and add the **Diagrams** layer and plot entity **Force/Moment – Thick 3D Beam** and component **Fx**.

The next image shows the axial forces in the pile elements.

## Piled Raft Foundation

---



Axial forces in pile elements (kN)

This completes the example.