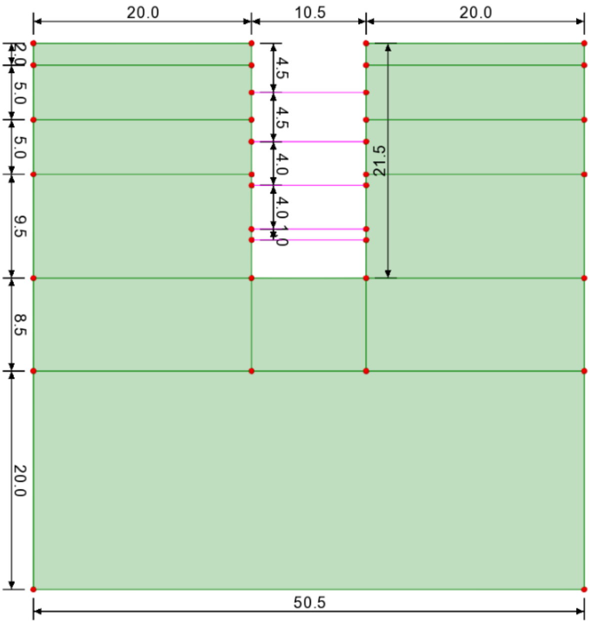
Modelling of a Braced Sheet Pile

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

The development of multi-level basements and tall buildings has become increasingly common. To ensure stability and prevent collapse during deep excavations, braced sheet pile systems are frequently employed. In this study, a numerical analysis of a braced sheet pile arrangement is performed using LUSAS. The sheet pile wall has a total depth of 30m, with struts installed at 4.5m, 9m, 13m, 17m, and 18m below ground level. The problem Geometry is shown in figure 1.



Model geometry

Keywords

Sheet Pile, Modified Mohr-Coulomb, Activation/Deactivation, Interface Elements.

Associated Files

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

- **braced_sheet_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

The aim is to model the behaviour of the sheet wall during construction and view resultant displacements and bending moments.

Modelling overview

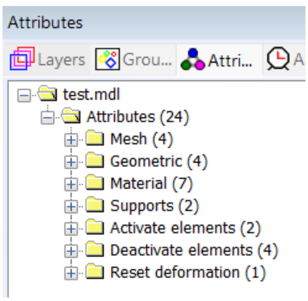
The model is created using an analysis category of 2D, with model units of kN,m,t,s,C

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 in the help manual.

Model attributes

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



Model Attributes

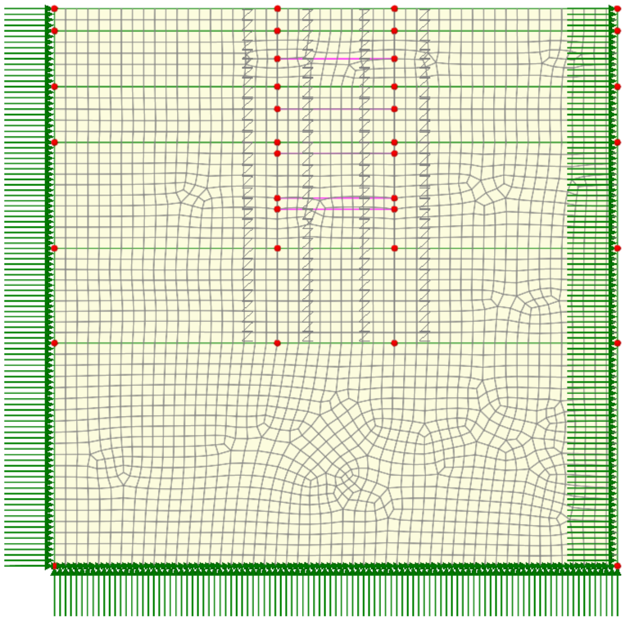
Mesh

The surface features are meshed using plane strain quadrilateral quadratic elements (QPN8) with an element size of 1.

The lines representing the sheet pile are meshed using plane strain beam quadratic elements (BMI3N) with an element length of 1.

The interface between the soil and the sheet pile is modelled using a single-feature interface element.

Strut supports are modelled using bar element.



Model mesh with boundary conditions

Geometric Properties

The sheet wall is 0.28m thick whilst the cross-sectional areas are listed in the table below.

Table 1: Sheet wall information

Strut	Elevation (m)	Strut spacing (m)	Section area (m^2)
1	-4.5	6	0.02696
2	-9	6	0.02696

Strut	Elevation (m)	Strut spacing (m)	Section area (m^2)
3	-13	6	0.053992
4	-17	6	0.08000
5	-18	6	0.08000

Materials

An isotropic elastic material is used for the sheet pile and struts whilst an isotropic nonlinear material utilizing the Modified Mohr-Coulomb (HS) failure surface is used for the soil. The initial stress state in the soil is defined by the coefficient of lateral earth pressure, K_0. Table 2 gives the material properties.

Table 2: Soil material properties

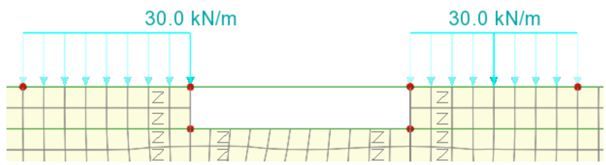
Layer	Mass Density (t/m^3)	Young's modulus, E (kPa)	Poisson's ratio, ν	Angle of friction, φ	Cohesion, c	Earth pressure coefficient, K_0
Sheet pile	6.28	31.622E6	0.15	-	-	-
Strut	7.8	200E6	0.2	-	-	-
Silty sand 1	1.8	10.0E3	0.3	30	1	0.5
Silty sand 2	1.8	12.3E3	0.3	30	1	0.5
Silty sand 3	1.9	34.5E3	0.3	34	1	0.4408
Silty sand 4	1.9	38.3E3	0.3	36	1	0.4122

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 distributed load acting at ground surface (30 kPa).



Applied loads

Other Attributes

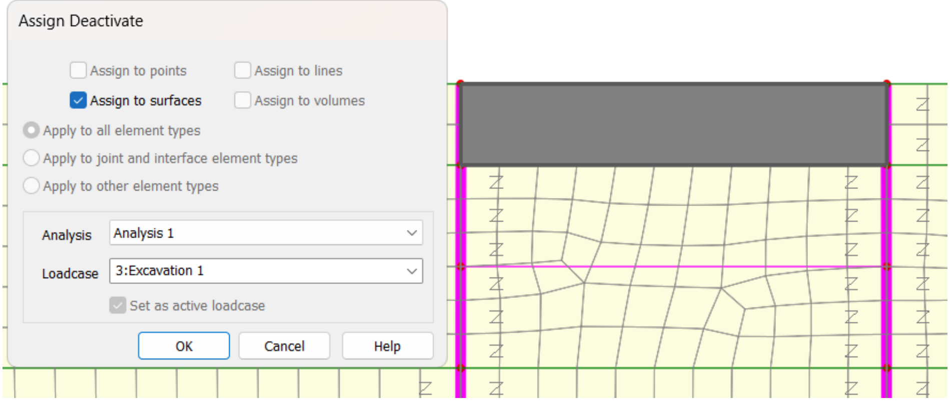
In addition to the previous attributes, to simulate the construction stages of the foundation, we need to use the following attributes:

- ❑ Activate and Deactivate
- ❑ Reset Deformation

Reset deformation is required in the initial phase as the displacements resulting from the application of the gravity force are not of interest in the actual example.

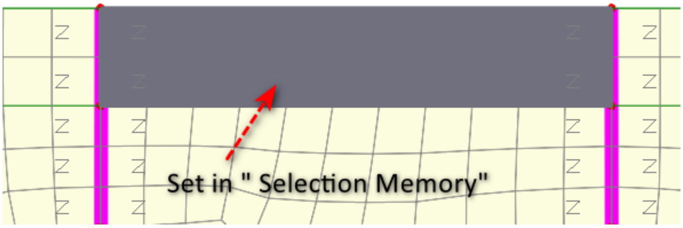
During the simulation of the construction stages, the soil material must be deactivated. The following steps illustrate how to deactivate a layer, using the first layer as an example.

1. Select the surface feature representing the soil layer and assign the Deactivate attribute.



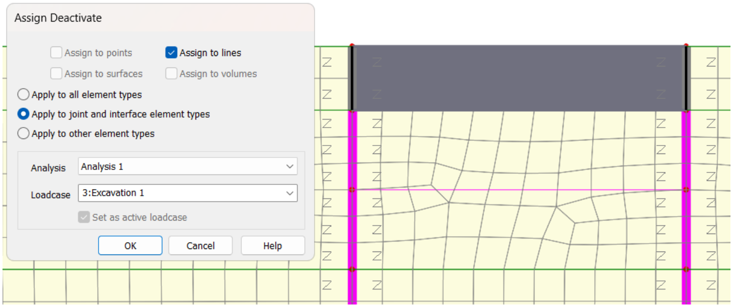
Deactivating soil layer

2. To deactivate the interfaces at the excavation sides, select the surface feature representing the soil layer and set it to Selection Memory.

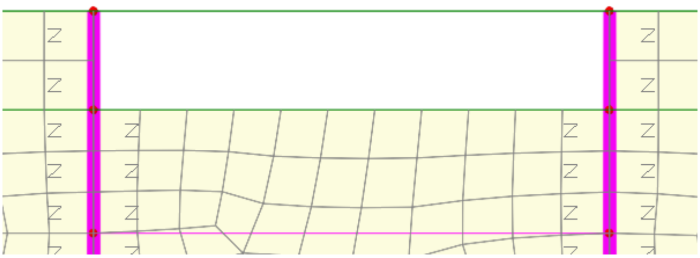


Soil put into Selection memory

3. Then assign the Deactivate attribute to the selected lines located at the excavation sides.



Lines to side of excavation deactivated



Deactivating interfaces

Analyses considered

Six stages relating to the construction of the foundation are considered in the model as follows:

Initial Phase

In geotechnical engineering, initial stresses must be determined for a soil body. These stresses are affected by the weight and formation of the soil. The deformation caused by the weight of the soil is reset for this load case.

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.

Installing Sheet piles

To install the sheet piles, the thick beam elements are activated.

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

Excavation 1

First soil layer with interfaces at the excavation sides are deactivated.

Distributed Loads at the ground surfaces are activated at this stage as well. Manual nonlinear analysis control properties are defined at each.

Excavation 2

Second soil layer with interfaces at the excavation sides are deactivated.

The first strut form top is activated.

Automatic nonlinear analysis control properties are defined at each.

Excavation 3

Third soil layer with interfaces at the excavation sides are deactivated.

The second strut form top is activated.

Automatic nonlinear analysis control properties are defined at each.

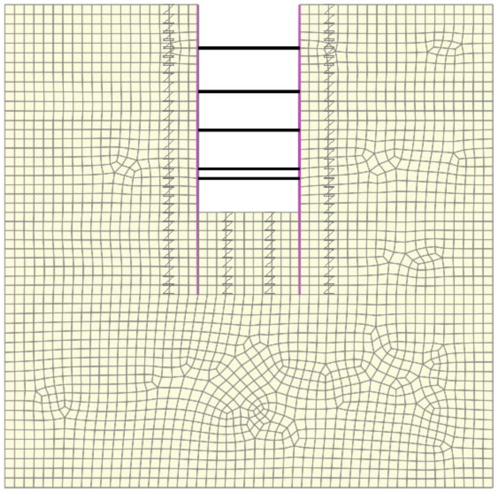
Excavation 4

Fourth soil layer with interfaces at the excavation sides are deactivated.

The last two struts are activated.

Automatic nonlinear analysis control properties are defined at each.

The following image shows the model at the fourth (and final) stage.



Final stage

Running the Analysis

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

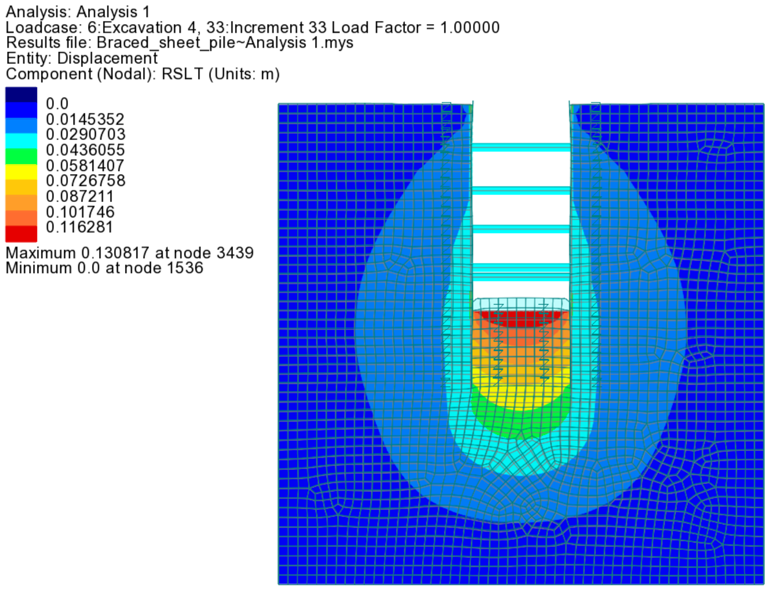
Viewing the Results

Analysis loadcase results are present in the Treeview.

Displacements

- Turn off the display of the **Deformed** mesh layer and **Supports**
- Add the **Contours** layer to plot contours of entity **Displacement** and component **RSLT**
- Set active the **Excavation 4** loadcase..

The next image shows the displacement contours.



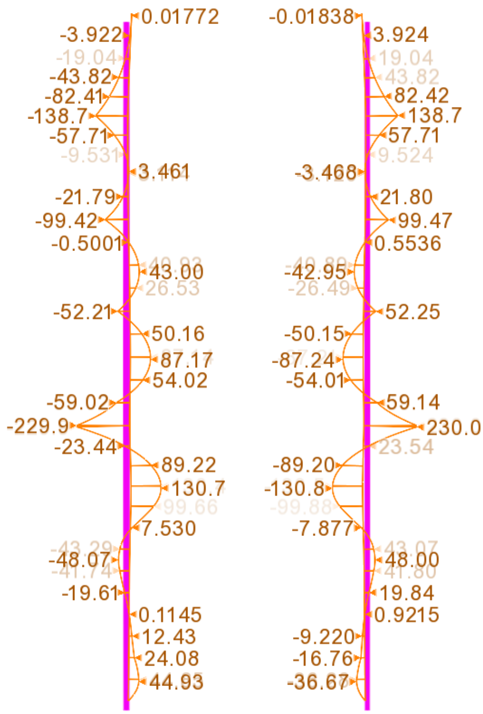
Displacements at final stage (m)

Moments

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 line attribute **Wall** and select **Set as only visible.**
- Right-click in the view window and add the **Diagrams** layer to plot entity **Force/Moment - Plane Strain Beam** and component **Mz**.

The next image shows the moments in the sheet piles.



Mz (kN.m)

This completes the example.