

# Modelling of 3D Excavation

For LUSAS version:	22.0
For software product(s):	LUSAS Bridge plus or LUSAS Civil&Structural plus
With product option(s):	Geotechnical, Nonlinear

## Problem Description

In this example we are modelling a 3D excavation of a pit, taking into consideration the constructions stages of a retaining system. The retaining system consists of walers, struts and a diaphragm wall.

The model depicts a 10m by 20m excavation area with a temporary retaining system consisting of a diaphragm wall, walers, and struts. The maximum depth of the excavation is 7m. The excavation is divided into three stages with excavations to 3, 5, and 7 metres and with walers and struts installed at 2 and 4 metre depths. The height of the diaphragm wall is 12 m. The geological strata are divided into three layers: filling soil, colluvial soil, and weathered rock, with depths of 3, 10, and 22 metres, respectively.

## Keywords

**3D Modelling, Displacement, Diaphragm Wall, Struts, Walers.**

## Associated Files

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



- modelling\_3d\_excavation.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 purpose of this example is to review the stability of the retaining system and calculate displacements, stresses, and the interaction between retaining wall and structure members. This type of modelling also permits the study of the impact of an excavation on the surrounding ground and adjacent structures.

### Preparing the Model Features

The model sets the Analysis category as 3D and the model units as kN,m,t,s,C.

### Feature Geometry

- The model is built up using various features in LUSAS. The soil layers are represented by volume features, struts and walers by line features, and the diaphragm is a surface feature.
- The model uses points and lines to define the skeleton geometry (Figure 1).
- The definition of the points and lines take into consideration the excavation phases and the supporting elements installation. In doing so, we ensure proper connectivity of different parts of the model while meshing (Figures 2 and 3)

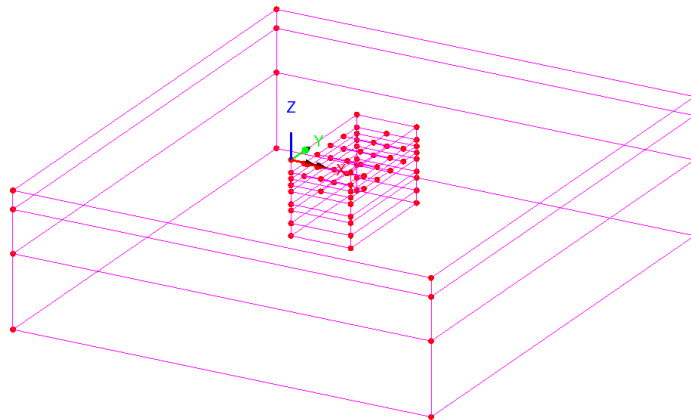


Figure 1: Model with lines feature

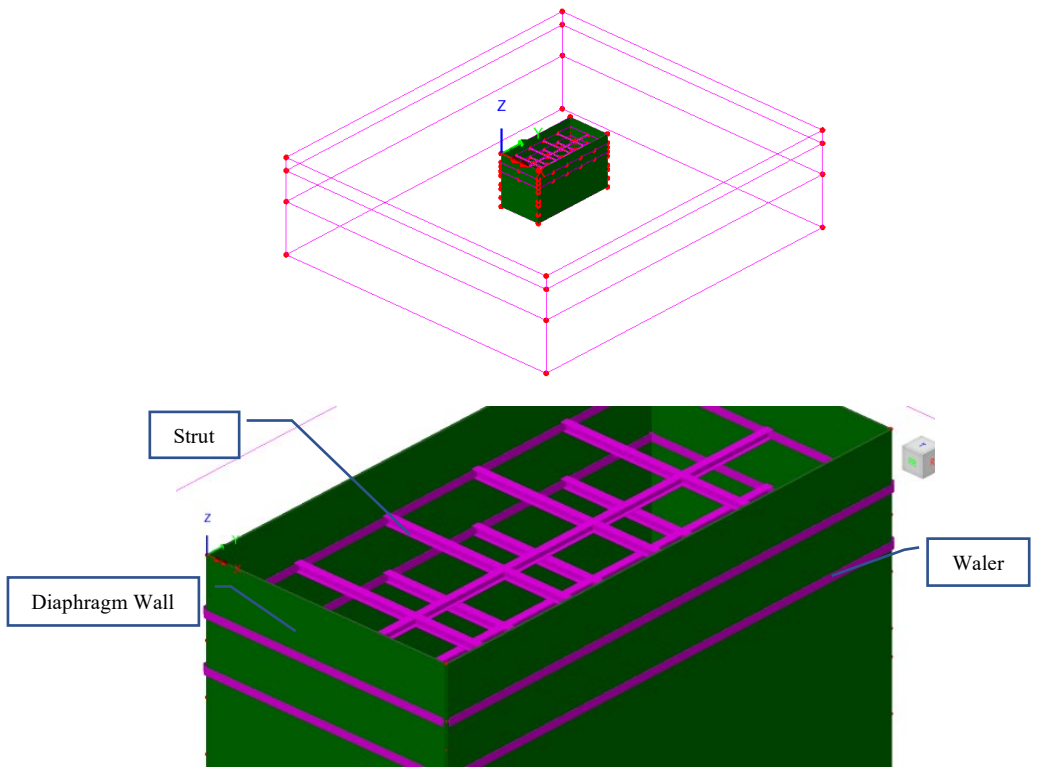


Figure 2: Structural elements

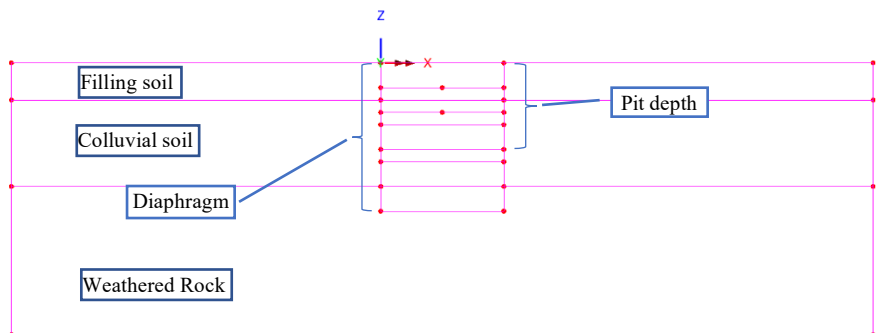


Figure 3: Model geometry

The struts and walers elements are meshed independently from the soil, with the struts lying on straight lines connecting opposite points of the wall. This way, we account for the deformation of the wall prior to their installation. It is not necessary for the line mesh to match up with the soil elements. Here are the steps to create the struts and walers:

- **Copy** the selected lines representing struts and walers with a Z translation of 10m (Figure 4)
- Make all the copied lines and points unmergeable except those on the perimeter which are to be connected to the diaphragm wall (Figure 5).
- Move the lines back to their original location using the command **Move** with a Z translation of -10m.

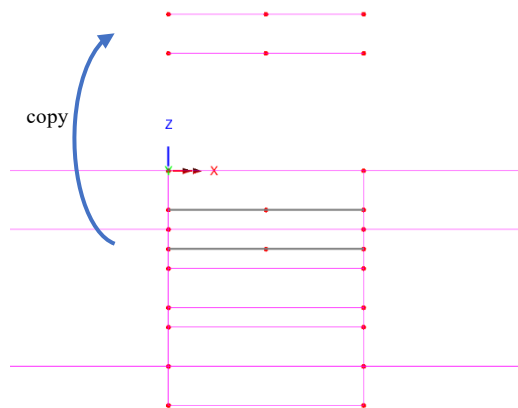


Figure 4: Creating Struts and Walers

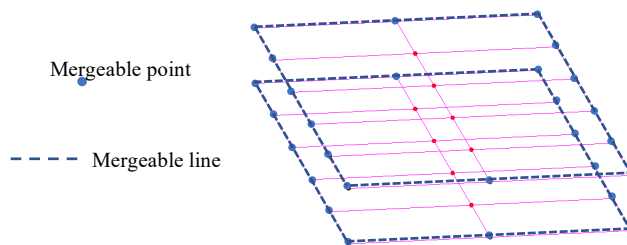


Figure 5: Mergeable Perimeter Points and Lines to connect with diaphragm wall.

## Preparing the Model Attributes

Attributes describe the characteristics of the model and can be defined using the Attributes dialog. These attributes are displayed in the Attributes Treeview.

## Defining the Mesh

- The soil layers are volumes, so we create a volume mesh for them. We create an irregular volume mesh using triangular elements (TH10) with sizes of 1m, 4m, and 5m for the pit, filling layer, and the rest of the layers, respectively (Figure 6).

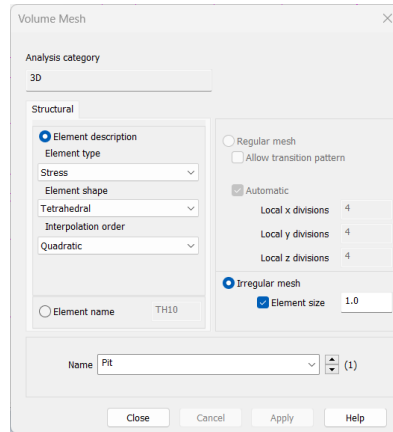


Figure 6: Volume mesh

- For the structural elements such as the struts and walers, a line mesh with a Thick Beam element type and element length of 1 metre is applied (Figure 7).

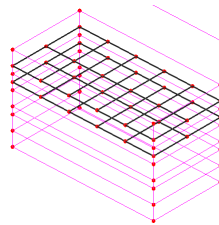
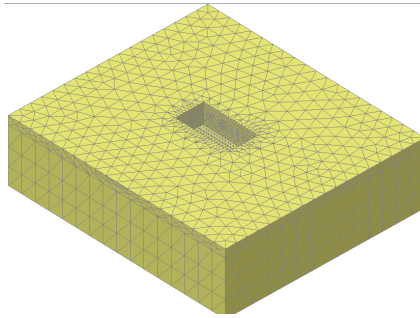


Figure 7: Structural elements lines

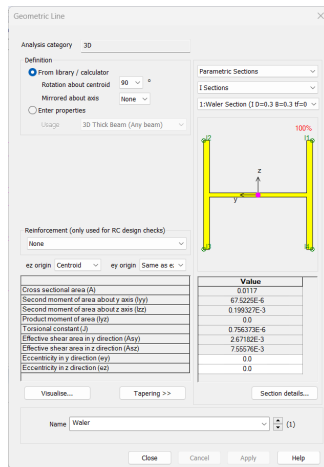
- Triangle thick shell elements with quadratic interpolation (TTS6) are used for meshing the diaphragm. The mesh of the excavated model is shown in figure 8.



**Figure 8: Model mesh of excavation**

## Defining the Geometric Properties

Two geometric attributes, representing the structural elements, struts, and walers, are created and added to the local library for use in the model as shown in figure 9. The diaphragm surface is given a thickness of 0.1m.



**Figure 9: Waler and strut section**

## Defining the Materials

An isotropic elastic material is used for the structural elements (Figure 10) while the various other soil materials are nonlinear and based on the Mohr-Coulomb failure criterion. The initial stress in the soil is determined by assigning a lateral earth pressure coefficient to the soil.

Table 1 gives the material properties for this example.

**Table 1: Soil material properties**

Layer	Mass Density	Young's modulus, E	Poisson's ratio, $\nu$	Angle of friction, $\varphi$	Cohesion, c	Earth pressure coefficient, $K_0$
Fill Material	1.6 t/m <sup>3</sup>	22.0E3 kPa	0.2	30°	1 kPa	0.5
colluvial soil	1.7 t/m <sup>3</sup>	43.0E3 kPa	0.2	30°	1 kPa	0.5
Weathered Rock	2.0 t/m <sup>3</sup>	60.0E3 kPa	0.2	38°	1 kPa	0.7

**Structural element properties**

Mass Density	Young's modulus, E	Poisson's ratio, $\nu$
7.698 t/m <sup>3</sup>	210.0E6 kPa	0.3

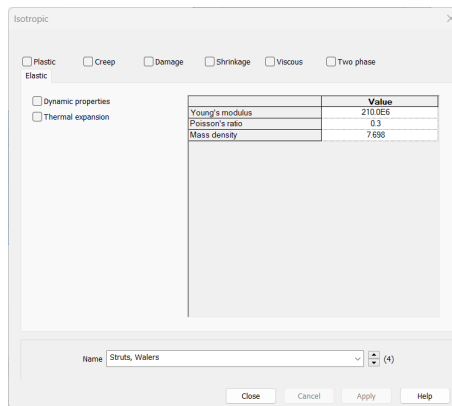


Figure 10: Struts, walers and diaphragm properties

**Defining the Supports**

Fully fixed supports are assigned to the base, while the lateral sides are only fixed in the horizontal directions as shown in the figure 11.

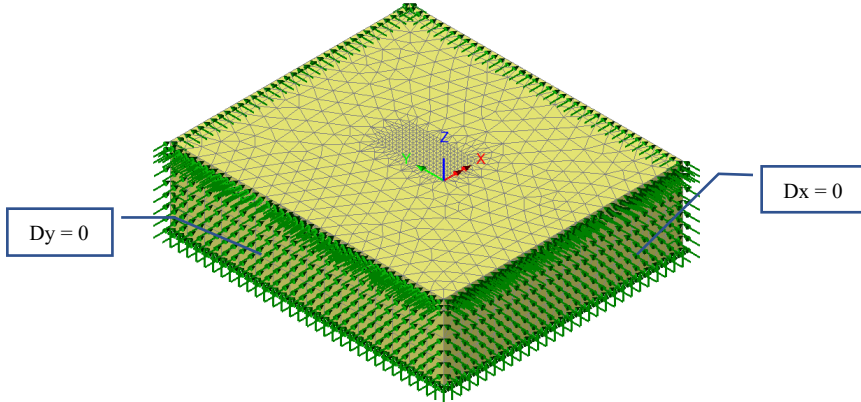


Figure 11: Horizontal and vertical supports

### Defining the Loads

The load considered in the analysis is the self-weight, which can be added to any load case by right-clicking on the load case in the Treeview and selecting the option for **Gravity** (Figure 12).

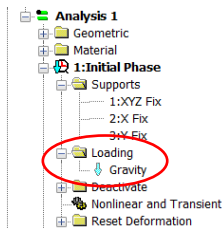


Figure 12: Adding gravity to the model

### Defining Other Attributes

To accurately simulate the excavation process, the deactivation and activation function must be utilised to remove soil and then install the supporting system. The excavation and installation of the supporting system will be completed in a total of seven load stages.

To take account of the fact that the supporting structural elements for the excavation are not installed initially, LUSAS allows the user to activate these elements in a specific order based on the construction stages. Deactivated elements in LUSAS are still included in the analysis, but their stiffness is greatly reduced so that they do not affect the overall stiffness of the system. In the example being discussed, a default value has been chosen, as shown in Figure 13.

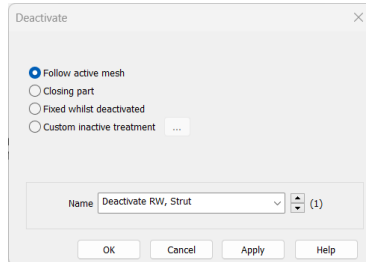


Figure 13: Deactivation options available in LUSAS

## Running the Analysis

Nonlinear analysis control properties are defined for all the load cases (Figure 14), all the parameters are left at their default values. The load stages defined for the simulation are the following (Figure 15):

- Ground initialisation phase
- Installing diaphragm wall
- Excavation stage 1
- Installing top-level support
- Excavation stage 2
- Installing bottom-level support
- Excavation stage 3

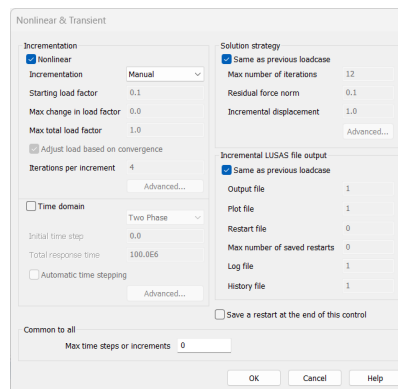
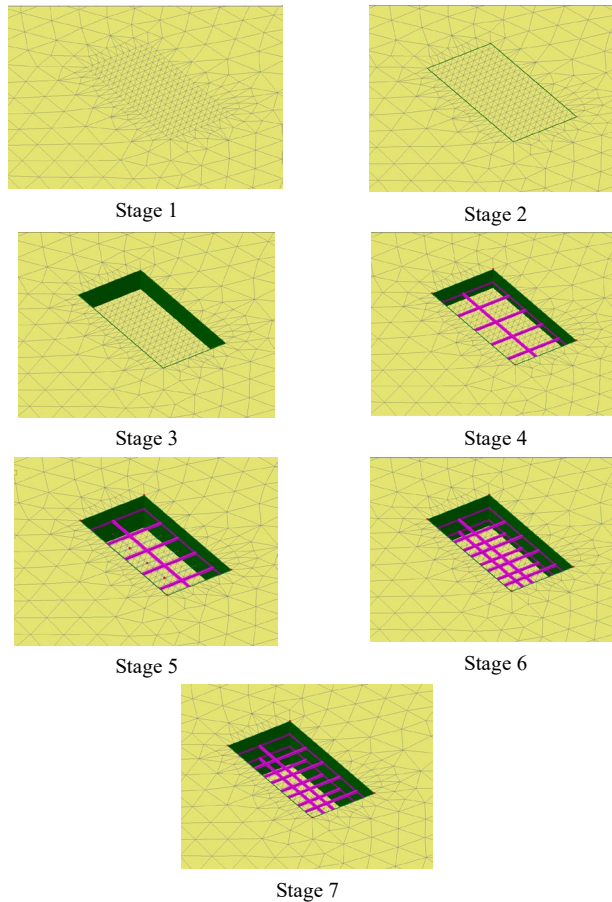


Figure 14: Control parameters



**Figure 15: Construction Stages**

## Viewing the Analysis

Analysis loadcase results are present in the Treeview.

### Deformed Mesh

It is important to review the deformed shape of the model after running the file to ensure there are no obvious errors, such as excessive displacements in unexpected areas, which may be caused by incorrect properties, incorrect load placement, or incorrect support conditions (Figures 16 and 17).

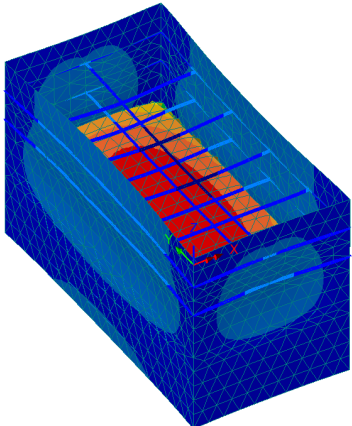


Figure 16: Deformed mesh

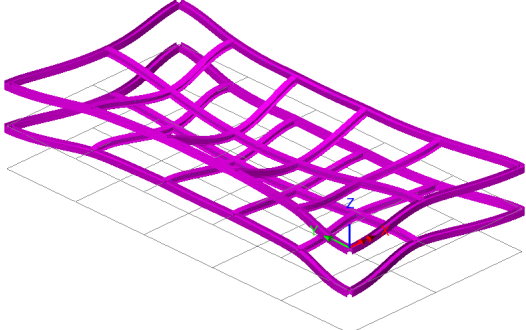


Figure 17: Deformed structural elements

A graph of the deformation of the wall top over a range of excavation loadcases is created using the graph wizard and results are exported into Excel. The maximum resultant displacement is 4.5 mm (Figure 18)

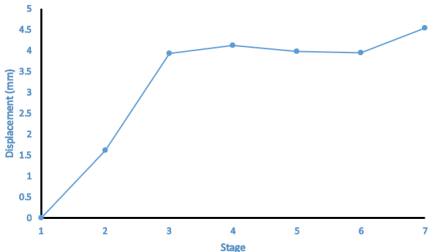
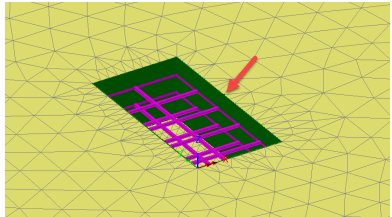
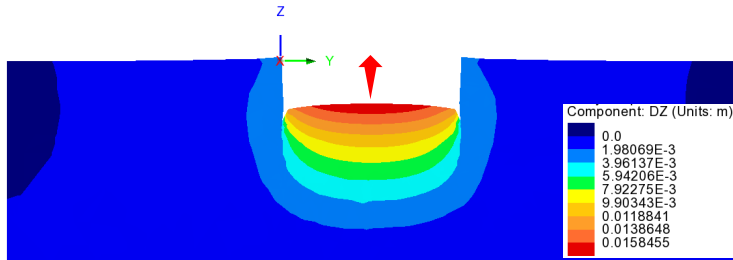
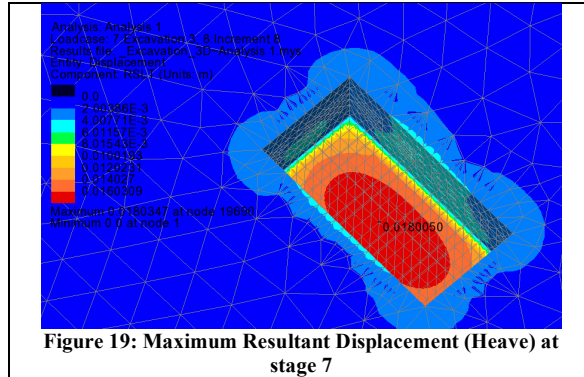


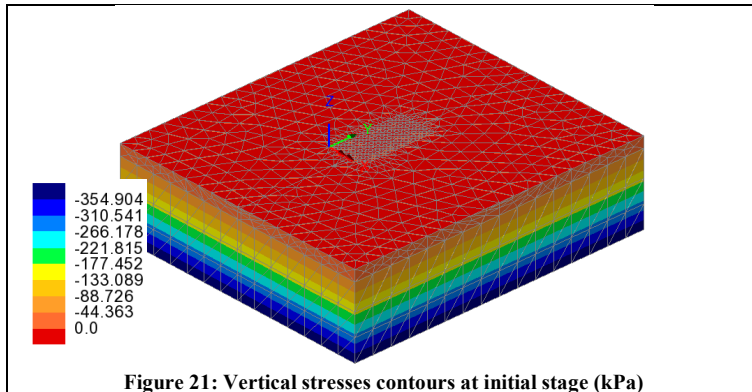
Figure 18: Resultant Displacement of node

Figures 19 and 20 show the maximum heave at stage 7



## Stress Contours

Figure 21 illustrates the initial stress state in the soil, showing that the vertical stresses increase with depth.



### Structural Elements Diagrams

Maximum moments of walers and struts are checked by selecting **Layers > Diagrams** (Figure 22 and 23)

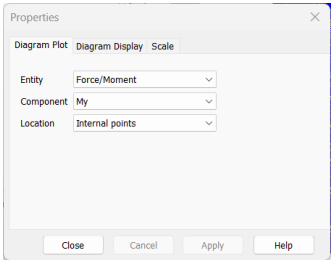


Figure 22: Diagram plot

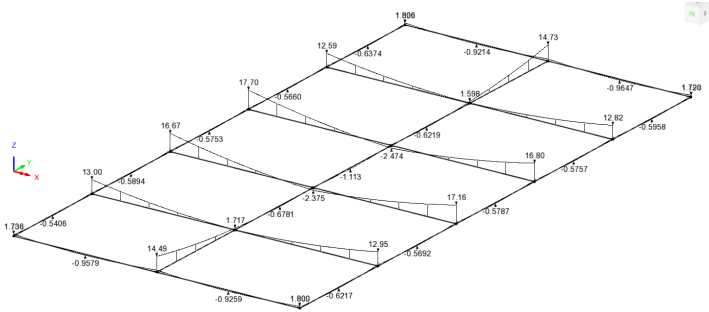


Figure 23: My moments at the top support (stage 7) (kN.m)

