Modelling of 3D Excavation

For LUSAS version:	21.0
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.

π

excavation_3d.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 3: Model geometry

The struts and walers elements are meshed independently from the soil, with the structs 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 expect 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 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).

30		
Structural		
Element description	O Regular mesh	
Element type	Allow transition patter	m
Element shape	Automatic	
Tetrahedral V	Local x divisions	4
Interpolation order	Local v divisions	4
Quadratic ~	Local y divisions	4
	LOCAI Z DIVISIONS	-
	 Irregular mesh 	
C Element name TH10	🔽 Element size	1.0
Name	~ 📫	(1)

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.

V Profit Ibraty / calculator	Parametric Sections
Rotation about centroid 90 V °	
Mirrored about axis None	I Sections
C Enter properties	1:Waler Section (1D=0.3 8=0.3 tf=0
Usage 30 Thick Beam (key beam)	2 1009
None	
None ez origin Centroid v ey origin Same as e: v	
None ez origin Centroid v ey origin Same as e: v	Value 0.0117
Nome ex origin Centroid ex origin Centroid cross sectional area (A) Cross sectional area (A)	Value 0.0117 675225E 6
Nome determine (unity dised ion Ac design cliences) None ez origin Centroid v ey origin Same as ei v Cross sactional area (A) Saccond moment of area about y axis (lyy) Second moment of area about y axis (lz)	Value 0.0117 675228-6 0.19927E-3
Reinice many basic lonic design decks // fane ext origin [Centroid] ~ every origin [Same as ext origin Cross sectional area (A) Second moment of area about y axis (ky) Second moment of area about y axis (kz) Product moment of area (by)	Value 00117 67.5258-6 0.199327E-3 0.0
Reinitic contraint, unity daes into inc. design directs) ténere ex origin. Centrud v groppin Same as ex- Cross sectional area (A). Second moment of area about z assi (bz). Product moment (A). Product moment of area (byz).	Value 00117 675226-6 0.1993276-3 0.0 0.755736-6
Ideole ez origin [Centrol	Value 0.017 67.5226-6 0.199327-3 0.0 0.755378-6 267126-3
Ince exercisin Central or program Same as a exercisin Central or program Same as a Coss sectional area (A) Second moment of area about y aski (by) Second moment of area about y aski (by) Product moment of area (by) Product moment of area (by) Fichal or a second second (c) Effective shear area in y direction (Are)	Value Value 0017 675225-6 0199275-3 0 0 0.55736-6 2671025-3 7555016-3
termin termin (onry used in no. versign centrol) e origin Centrol version (no. versign centrol) Cross sectional area (A) Second memore of area about y set (by) Second memory of a bout y	Value 00117 675226-6 0199276-3 0 0755776-6 2.671226-5 7555765-3 0.0
Another the second seco	Value 0017 673226 0 0.7583726 2.671626 2.671626 0.0 0.0
Annual Construction (Service) (Service) Annual Construction (Serv	Value 00117 675226-6 019827-3 00 0758378-6 267102-3 000 000 000 000 000 000

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, v	Angle of friction, φ	Cohesion, c	Earth pressure coefficient, K ₀
Fill Material	1.6 t/m ³	22.0E3 kPa	0.2	30°	1 kPa	0.5
colluvial soil	1.7 t/m ³	43.0E3 kPa	0.2	30°	1 kPa	0.5
Weathered Rock	2.0 t/m ³	60.0E3 kPa	0.2	38°	1 kPa	0.7

Structural element properties

Mass Density	Young's modulus, E	Poisson's ratio, v
7.698 t/m ³	210.0E6 kPa	0.3

Dynamie	c properties		Value	
Therma	I expansion	Young's modulus	210.0E6	
		Poisson's ratio	0.3	
		mass density	7.000	

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.



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.

O Fo	Ilow active mesh
Oc	osing part
⊖ Fi	ked whilst deactivated
0 c i	istom inactive treatment
	Name Deactivate RW, Strut

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



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 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

Figure 20: Vertical displacement at stage 7 (section at the middle in YZ plane)

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)

Properties			×
Diagram Plot	Diagram Display	Scale	
Entity	Force/Moment	~	
Component	My	~	
Location	Internal points	~	
d	ose Cano	el Apply	Help
		D 1	• .

Figure 22: Diagram plot



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