

Bearing Capacity of Shallow Foundation

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

This tutorial demonstrates how to model and analyse the construction of a shallow foundation and pier with a load applied at the top of the pier. The foundation is placed on weathered rock and has a width of 10 m. The geological stratification is composed of the following layers: fill material, silty sand, weathered rock and moderately weathered rock. Figure 1 shows the thickness of various layers we have.

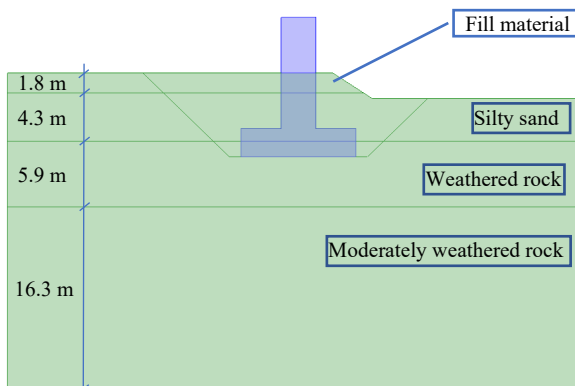


Figure 1: Foundation with geological conditions

Keywords

Bearing Capacity, Foundation, Displacement, Construction Stages, Excavation.

Associated Files

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



- bearing_capacity_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 required output from the analysis consists of:

- Analysis the stability of the foundation under actual loading
- Calculate the bearing capacity of the foundation

Preparing the Model Features

We create a new model, set the Analysis category as 2D, and specify the model units as kN,m,t,s,C.

Feature Geometry

- In LUSAS we have the ability to develop the model either by directly creating surface features or starting with lines which are then converted into surfaces. In both cases, it is crucial to ensure proper connection between surfaces and avoid any unintentional overlapping. Figure 2 shows the surfaces used to define this problem. Many facilities are available in LUSAS to help the user create these surfaces with ease.
- The model is divided, taking into consideration the excavation phases, and ensuring proper connectivity of different parts of the model while meshing (figure 2).

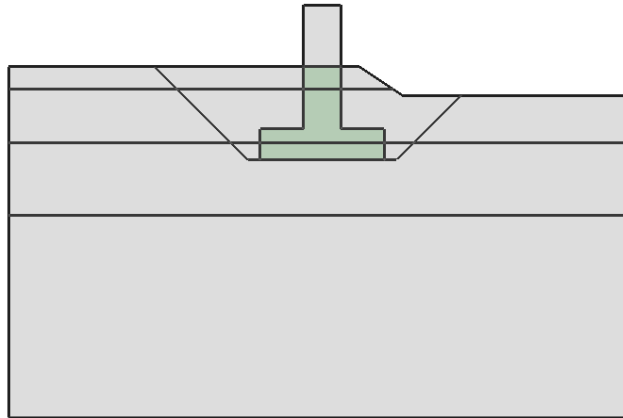


Figure 2: Decomposition of the model

Preparing the Model Attributes

Model attributes (mesh, material, geometric properties, etc.) are defined and assigned to the model. (Figure 3).

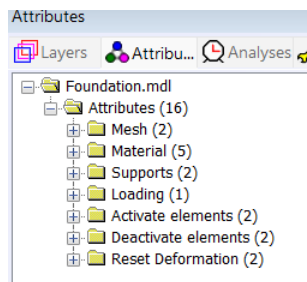


Figure 3: Model Attributes

Defining the Mesh

- The surface features are to be meshed using plane strain, quadrilateral, quadratic elements (QPN8) as illustrated in figure 4. Figure 5 shows the mesh assigned to the model.

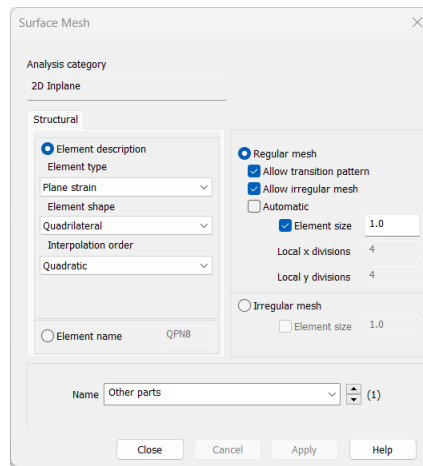


Figure 4: Surface mesh

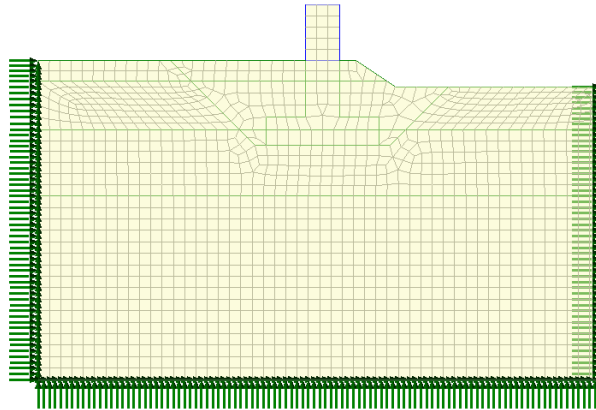


Figure 5: Model mesh with boundary conditions

Defining the Materials

An isotropic elastic material will be used for the concrete foundation whilst an isotropic nonlinear material utilizing the Modified Mohr-Coulomb failure surface will be used for the soil. The initial stress state in the soil is defined by the coefficient of lateral earth pressure, K_0 .

Table 1 gives the material properties for this example.

Table 1: Soil material properties

Layer	Mass Density	Young's modulus, E	Poisson's ratio, ν	Angle of friction, φ	Cohesion, c	Earth pressure coefficient, K_0
Concrete	2.5 t/m ³	21.0E6 kPa	0.3	-	-	-
Fill Material	1.7 t/m ³	17.0E3 kPa	0.3	28	26 kPa	1.00
Silty Sand	1.8 t/m ³	9.0E3 kPa	0.3	30	10	1.00
Weathered Rock	2.1 t/m ³	500.0E3 kPa	0.3	33	20	1.00
Moderately Weathered Rock	2.5 t/m ³	2.0E6	0.25	35	200	1.00

Defining the Supports

Fully fixed supports are assigned to the base, while the lateral sides are fixed in the horizontal directions as shown in the figure 5.

Defining the Loads

There are two types of loads that are being taken into consideration: the self-weight and a distributed load acting on the top of the pier. The load applied to the pier is 500 kPa.

Self-weight can be added to any load case by right-clicking on the load case in the Treeview and selecting the option **Gravity**.

Defining 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

During the simulation of the construction stages of the foundation, we need to deactivate the soil material and activate the concrete material (Figure 6). 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.

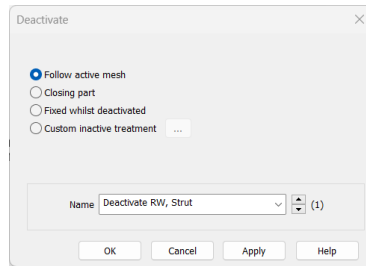


Figure 6: Deactivation option used in the actual case

Running the Analysis

Four stages for 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. In this phase, initial stresses are calculated before the foundation is installed, so the surface features related to the foundation are deactivated. The deformation caused by the weight of the soil is reset for this load case.

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

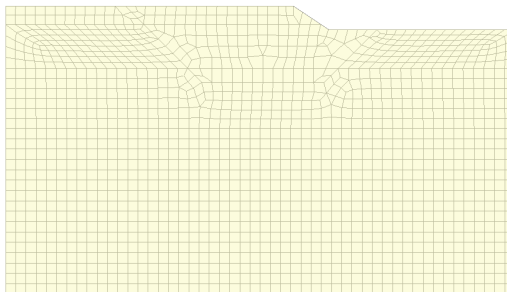


Figure 7: Initial phase

Installing Foundation

To install the foundation, the surface features related to the concrete materials are enabled and the soil features in the excavated areas are disabled, as shown in Figure 8. Nonlinear analysis control properties are defined for this phase, all the parameters are left at their default values.

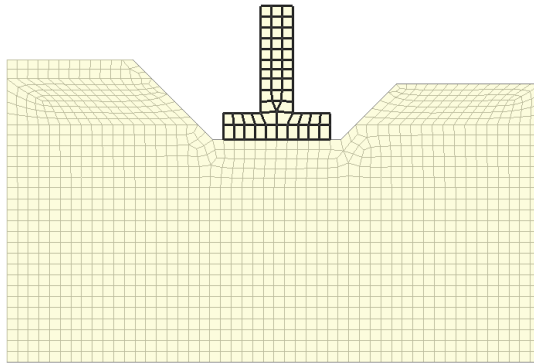


Figure 8: Installing foundation

Backfilling

After the foundation is installed, the previously set aside soil is used to backfill the foundation. To do this, the surface related to the soil backfill is activated, as shown in Figure 9. Nonlinear analysis control properties are defined for this phase, all the parameters are left at their default values. It is necessary to reset the deformation in this phase in order to accurately plot the displacement caused by loading in the next phase and determine the "bearing capacity" of the soil.

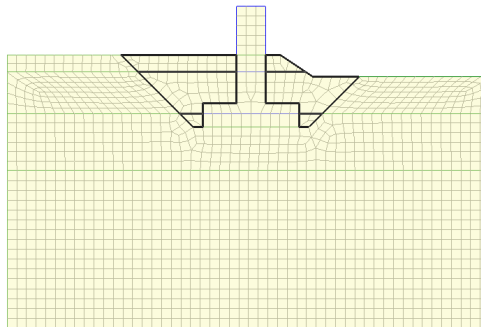


Figure 9: Backfilling phase

Note. In situations where we need to deactivate soil and replace it with a different material such as concrete, the following procedure is followed:

- Create a second, superimposed surface for the foundation and ensure it is made unmergeable.
- Ensure the two surfaces in question have exactly the same mesh density.

Bearing Capacity of Shallow Foundation

- ❑ Equivalence these surfaces using an equivalence attribute. This will connect the two meshes.
- ❑ Assign concrete material to one surface and soil to the other surface.
- ❑ Deactivate the surface with the concrete material in the initial phase.
Deactivate soil at the excavation stage and activate the concrete.

Loading

In Addition to gravity, we assign the actual load applied to the pier of 500 kPa. Nonlinear analysis control properties are defined for this phase as well, all the parameters have default values.

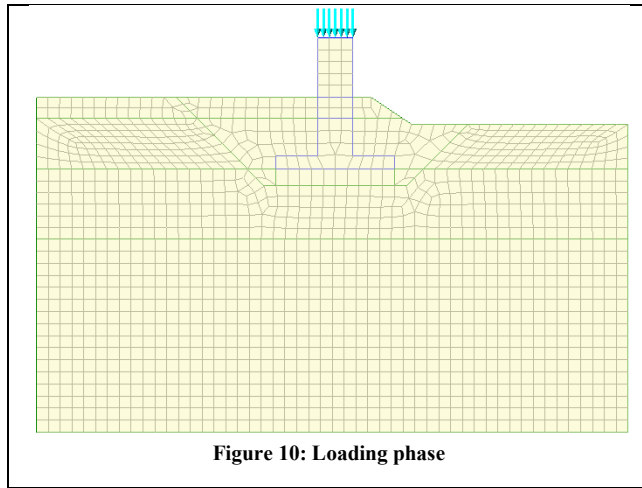


Figure 10: Loading phase

Loading Branch > Bearing Capacity

It is of interest to calculate the bearing capacity for the geotechnical engineer. Bearing capacity is simply the capacity of soil to support the loads that are applied to the ground above. LUSAS can calculate the bearing capacity using the arclength calculation technique whereby the maximum load that the foundation can support can be determined by gradually increasing the load up to failure. However, the allowed bearing capacity is not only controlled by shear failure, but large settlements may occur under loaded foundations without actual shear failure occurring; in such cases, the allowable bearing capacity is based on the maximum allowable settlement. Here, also, LUSAS has the facility needed to limit the maximum calculation of displacement of a specific point to a certain value.

Nonlinear analysis control properties are defined for this phase, the parameters to be used in this case can be found in figure 11. We notice in figure 12 the parameters to be set to limit the settlement of a point located on the pier to a value of 40 mm.

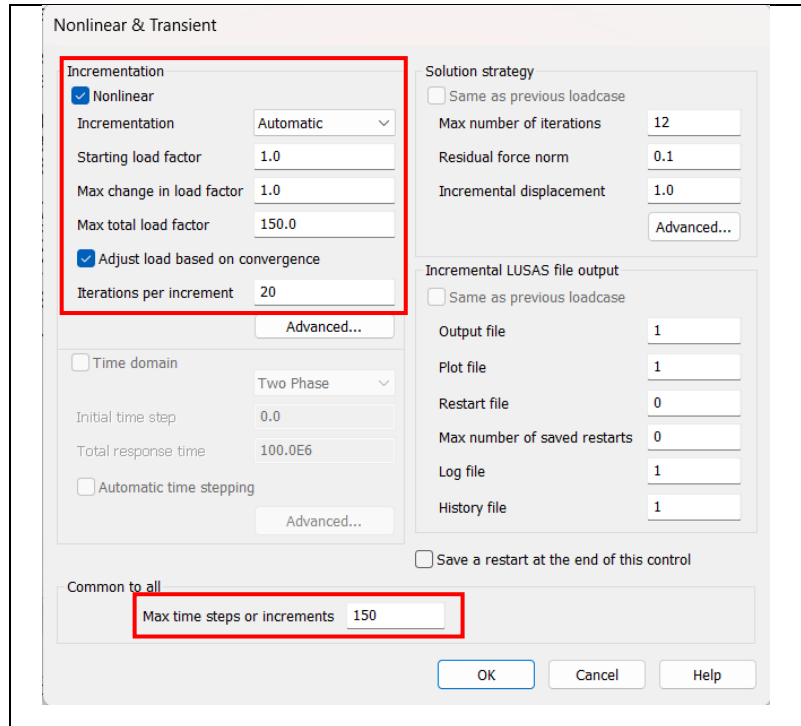


Figure 11: Nonlinear analysis control for branch analysis

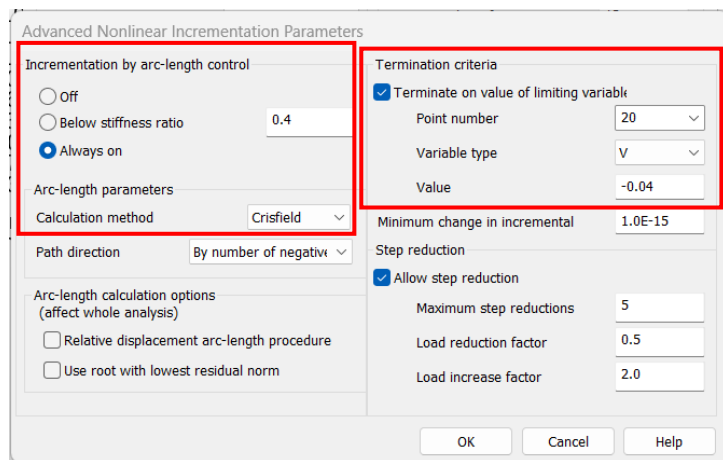


Figure 12: Advanced parameters used in branch analysis

Note. As this type of calculation runs until solution failure, it is best done in a branched analysis. Analysis branches may be added to the Analysis Treeview by selecting the **New > Branch** context menu item for any loadcase in the Analysis Treeview that has (or inherits) a nonlinear or transient control. They allow the creation and solution of one or more sub-analyses to investigate the response of the model at a particular loadcase or "stage". More information can be found in LUSAS help.

Viewing the Analysis

Analysis loadcase results are present in the Treeview.

Loading Stage Results

Figure 18 illustrates the deformed shape of the model after loading the foundation with the actual load of 500 kPa, which gives resultant displacement of 3.4 mm. (Figures 18 and 19).

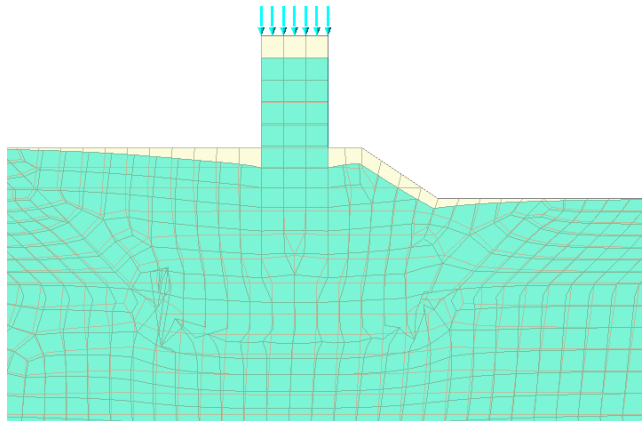


Figure 18: Deformed mesh at loading phase

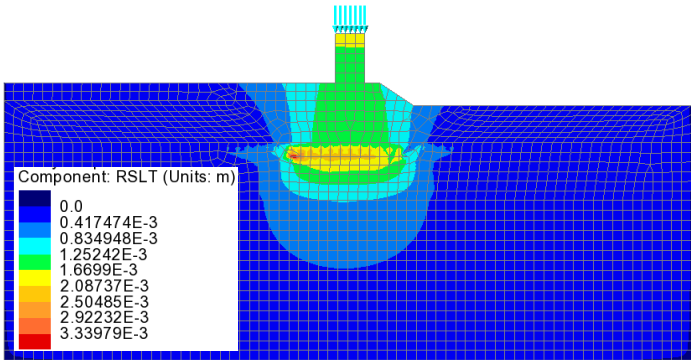


Figure 19: Resultant displacement (loading stage) (m)

Bearing Capacity branch Results

We show below in figures 20 and 21 some results for the bearing capacity branch, where we limiting the progressive failure to a specific settlement threshold of 40 mm.

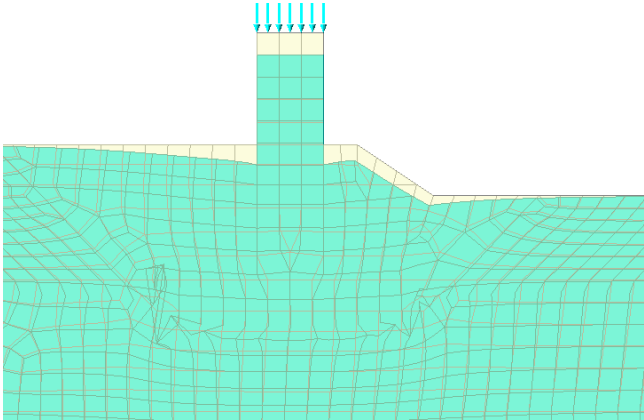


Figure 20: Deformed mesh at bearing capacity branch

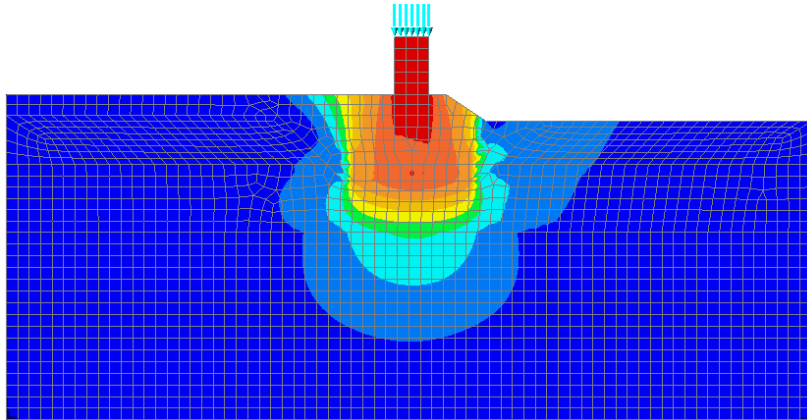


Figure 21: Resultant displacement (bearing capacity stage) (m)

A graph of the deformation (figure 22) of the pier top for the bearing capacity branch is created in LUSAS from which we can obtain the corresponding value of load factor leading to the 40 mm settlement. The bearing capacity in this case is $500+21.82*500 = 11410$ kPa. It is to be noted this value has to be factored by a safety factor related to design considerations.

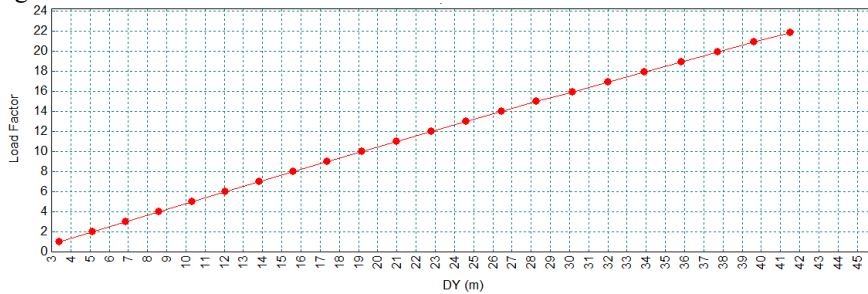


Figure 22: Resultant displacement (bearing capacity stage) (m)