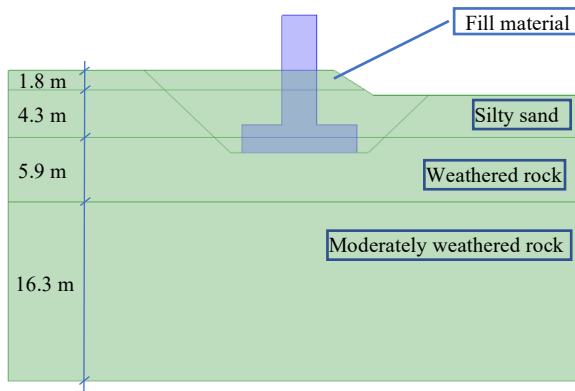


Bearing Capacity of Shallow Foundation

For LUSAS version:	23.0
For software product(s):	LUSAS Bridge plus or LUSAS Civil&Structural plus
With product option(s):	Geotechnical, Nonlinear

Problem Description

This overview example 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 10m. The geological stratification is composed of the following layers: fill material, silty sand, weathered rock and moderately weathered rock. The following image shows the thickness of each layer.



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_of_shallow_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.

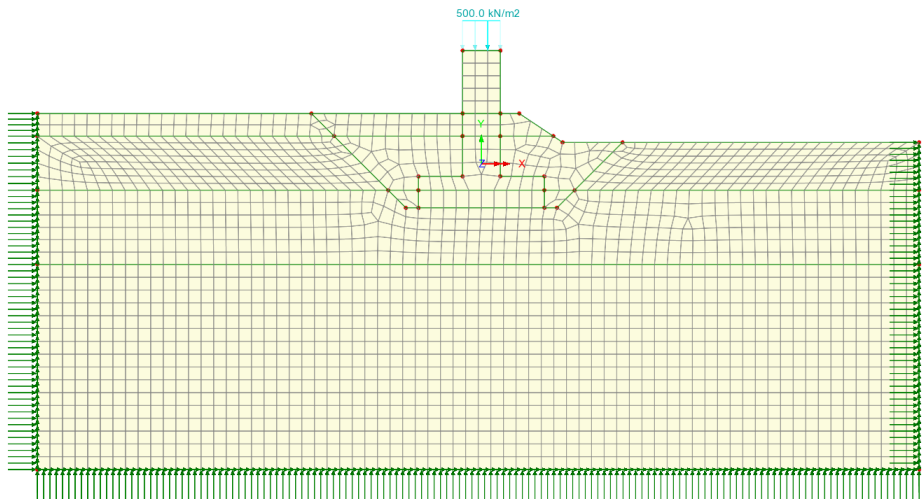
Objectives

The required output from the analysis consists of:

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

Model overview

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

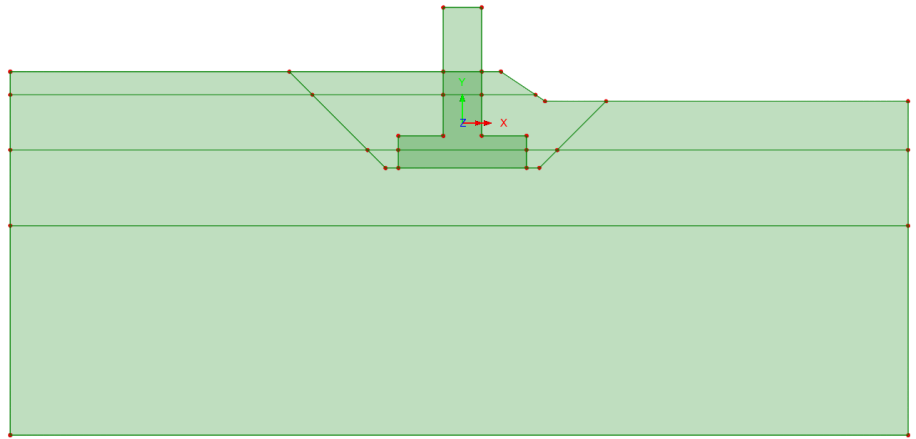


The model created from running the supplied script

Feature Geometry

In LUSAS a model can be developed either by directly creating surface features or by 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.

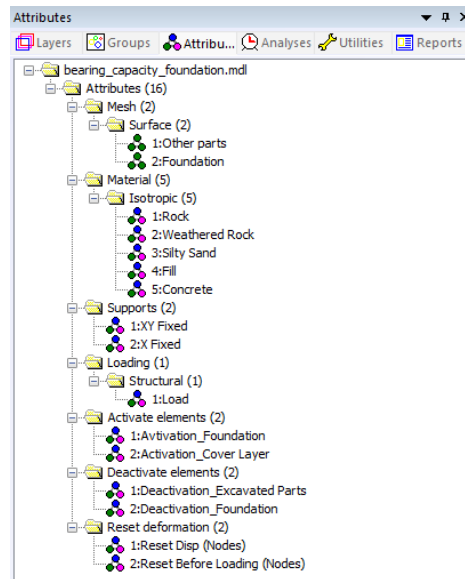
The image below shows the surfaces used to define this problem. The model is divided, taking into consideration the excavation phases, and ensuring proper connectivity of different parts of the model while meshing. Many facilities are available in LUSAS to help you create these surfaces with ease.



Surface features used to define the problem

Model Attributes

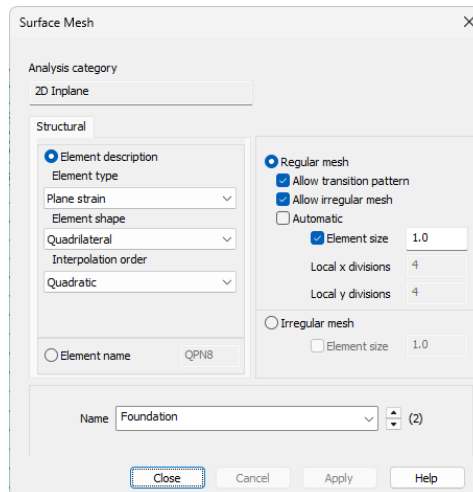
Model attributes (mesh, material, geometric properties, etc.) are defined and assigned to the model.



Model Attributes

Mesh

The surface features are meshed using plane strain, quadrilateral, quadratic elements (QPN8).



Surface mesh

Materials

An isotropic elastic material is used for the concrete foundation whilst an isotropic nonlinear material utilizing the Modified Mohr-Coulomb 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 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	25	1.00
Moderately Weathered Rock	2.5 t/m ³	2.0E6	0.25	35	200	1.00

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 on the top of the pier. The load applied to the pier is 500 kPa.

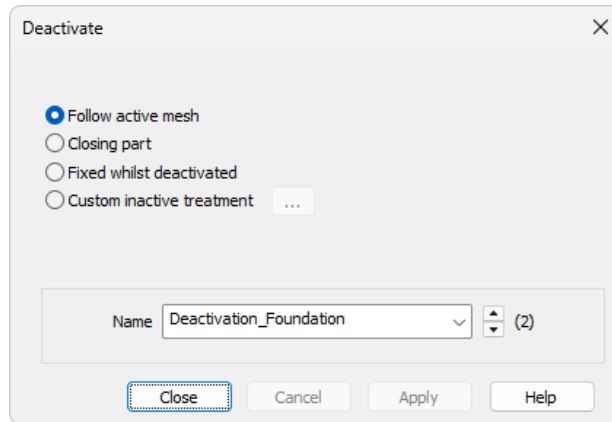
Self-weight can be added to any load case by right-clicking on the load case in the Analyses Treeview and selecting the option **Gravity**.

Other Attributes

In addition to the previous attributes, to simulate the construction stages of the foundation, the following attributes are used:

- ☐ 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. 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.



Deactivation option used in the actual case

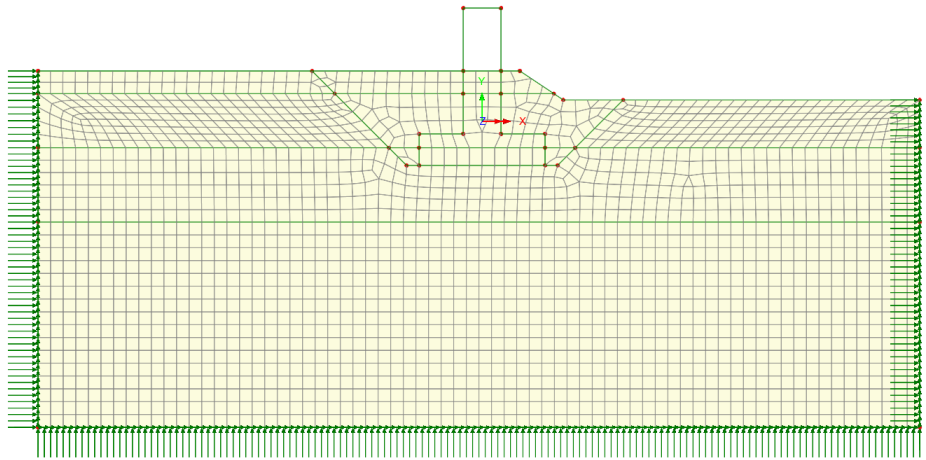
Construction stages considered

Four stages for the construction of the foundation are considered in the model.

1. 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.

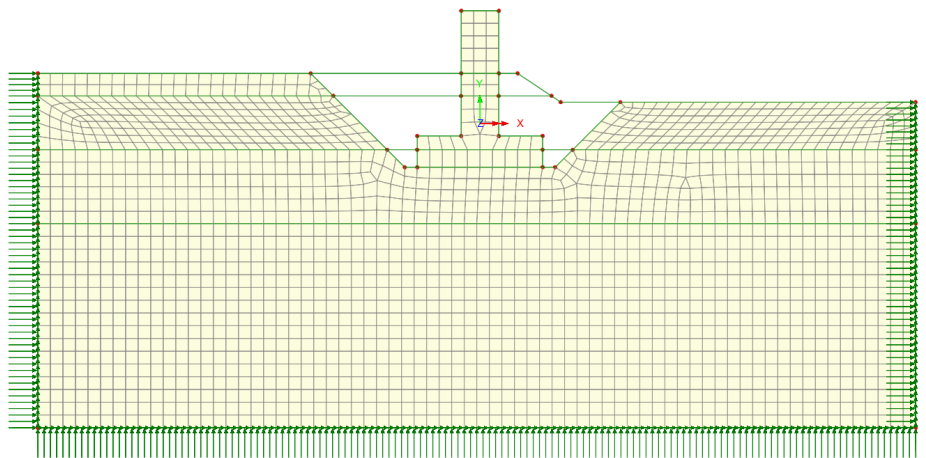
Nonlinear analysis control properties are defined for phase. All the parameters are left at their default values.



Initial phase

2. Installing Foundation

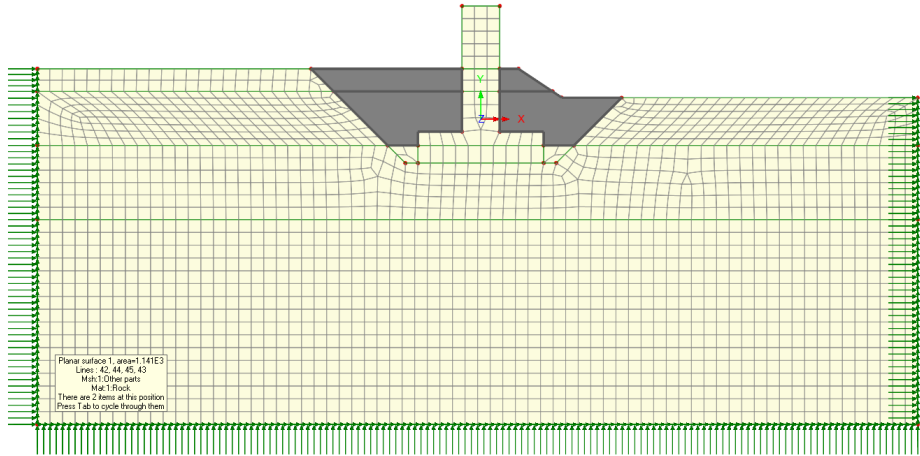
To install the foundation, the surface features related to the soil features in the excavated areas are disabled and the features representing the concrete materials are enabled. Nonlinear analysis control properties are defined for this phase, with all the parameters left at their default values.



Installing the foundation

3. Backfilling

After the foundation is installed, the previously set aside soil is used to backfill the foundation. To do this, the surfaces related to the soil backfill are activated, as shown in the following image.



Backfilling phase

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

It is necessary to reset the deformation in this phase to accurately plot the displacement caused by loading in the next phase and determine the "bearing capacity" of the soil.

Replacing material within surfaces

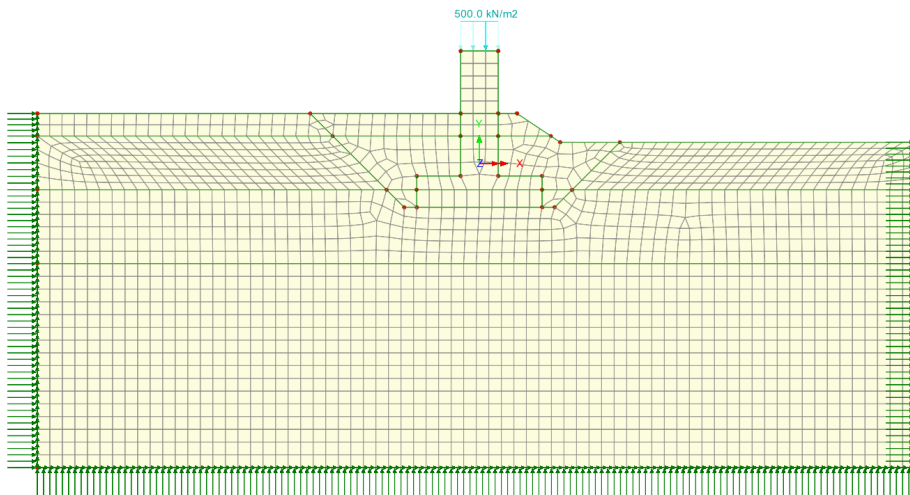


Note. In situations where we need to deactivate soil and replace it with a different material (concrete in this case), the following procedure is followed:

1. Create a second, superimposed surface for the foundation and ensure it is made unmergeable.
2. Ensure the two surfaces in question have the same mesh density.
3. Equivalence these surfaces using an equivalence attribute. This will connect the two meshes.
4. Assign the replacement material to one surface and soil to the other surface.
5. Deactivate the surface with the replacement material in the initial phase.
6. Deactivate soil at the excavation stage and activate the replacement material.

4. Loading

In addition to gravity, we assign the actual load applied to the pier (500 kPa). Nonlinear analysis control properties are also defined for this phase, with all parameters using default values.



Loading phase

Use of a loading branch to evaluate bearing capacity

It is of interest to calculate the bearing capacity of the soil to ensure it can support the loads that are applied to it.

LUSAS can calculate the bearing capacity using the arc-length 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.

As this type of calculation runs until solution failure, it is best done in a branched analysis. Analysis branches may be added to the Analyses Treeview by selecting the **New > Branch** context menu item for any loadcase in the Analyses 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.

Bearing Capacity of Shallow Foundation

The nonlinear analysis control properties are defined for this phase, and the parameters used in this case can be found in the following image.

Nonlinear & Transient

Incrementation

- ☒ Nonlinear
- Incrementation: Automatic
- Starting load factor: 1.0
- Max change in load factor: 1.0
- Max total load factor: 150.0
- ☒ Adjust load based on convergence
- Iterations per increment: 20
- Advanced...

☐ Time domain: Two Phase

Initial time step:

Total response time:

☐ Automatic time stepping

Advanced...

Common to all

Max time steps or increments: 150

Solution strategy

- ☐ Same as previous loadcase
- Max number of iterations: 12
- Residual force norm: 0.1
- Displacement norm: 1.0
- Advanced...

Incremental LUSAS file output

- ☐ Same as previous loadcase
- Output file: 1
- Plot file: 1
- Restart file: 0
- Max number of saved restarts: 0
- Log file: 1
- History file: 1

☐ Save a restart at the end of this control

OK Cancel Help

Nonlinear analysis control for branch analysis

Advanced incrementation parameters (see next image) are also set to limit the settlement of a point located on the pier to a value of 40 mm.

Advanced Nonlinear Incrementation Parameters

Incrementation by arc-length control ☐ Off ☐ Below stiffness ratio 0.4 ☒ Always on		Termination criteria ☒ Terminate on value of limiting variable Point number 20 Variable type V Value -0.04 Minimum change in incremental load 1.0E-15	
Arc-length parameters Calculation method Crisfield Path direction By number of negative		Step reduction ☒ Allow step reduction Maximum step reductions 5 Load reduction factor 0.5 Load increase factor 2.0	
Arc-length calculation options (affect whole analysis) ☐ Relative displacement arc-length procedure ☐ Use root with lowest residual norm			

OK Cancel Help

Advanced parameters used in branch analysis

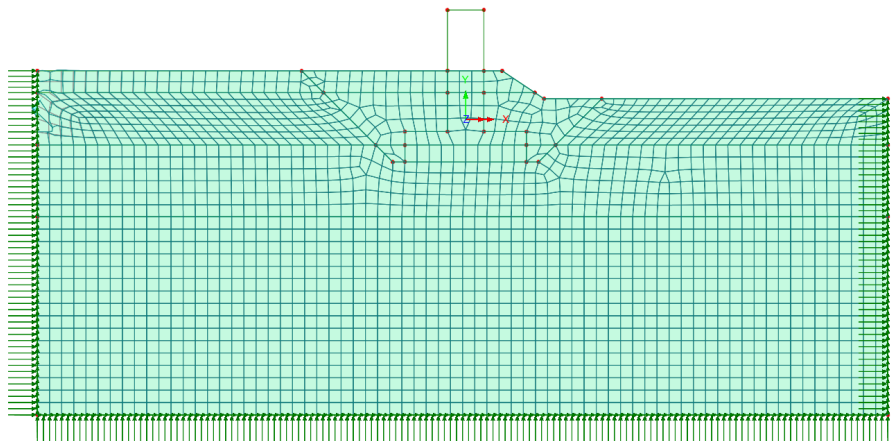
Running the Analysis



Ensure all analyses are selected and press **OK** to run the analysis.

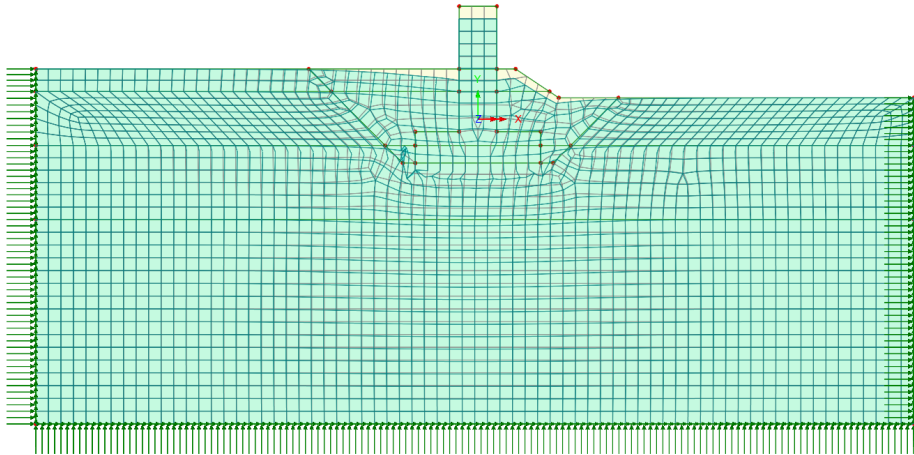
Viewing the Results

Analysis loadcase results are present in the Analyses Treeview. The initial view will be this.



Loading Stage Results

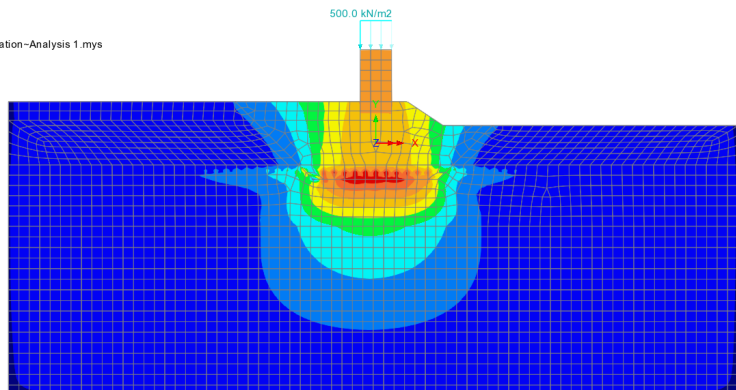
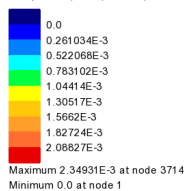
- Set active the loadcase **Loading** to see the deformed mesh from loading the foundation with the face load of 500 kPa.



Deformed mesh at loading phase

- Turn off the display of the geometry and deformed mesh layers.
- Add the contours layer to plot contours of entity of **Displacement** and component **RSLT**

Analysis: Analysis 1
Loadcase: 4: Loading, 6: Increment 6
Results file: bearing_capacity_foundation-Analysis 1.mys
Entity: Displacement
Component (Nodal): RSLT (Units: m)



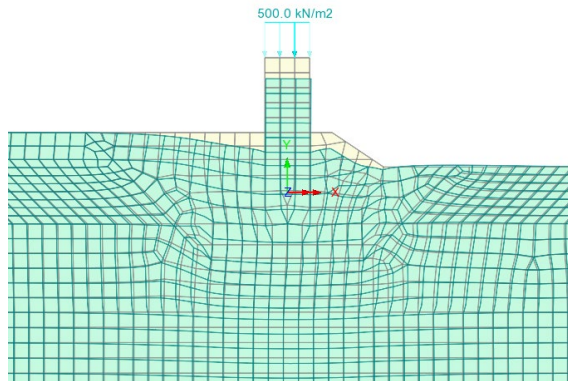
Resultant displacement at loading phase

A resultant displacement of 2.4mm is obtained for this phase.

Bearing Capacity Branch Results

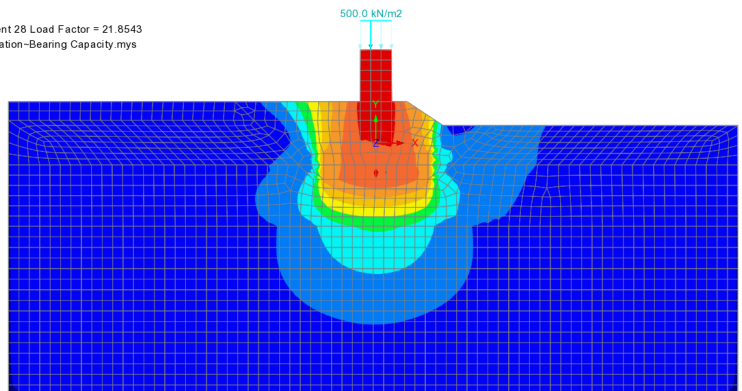
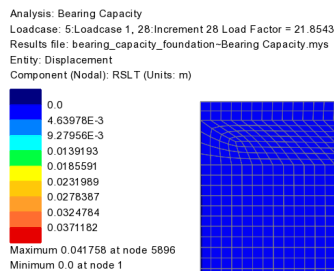
To view results for the bearing capacity branch, where the progressive failure is limited to a specific settlement threshold of 40 mm:

- Turn off the contours layer and turn-on the deformed mesh layer
- Set active the bearing capacity loadcase



Deformed mesh at bearing capacity branch

- Turn off the deformed mesh layer
- Turn on the contours layer.



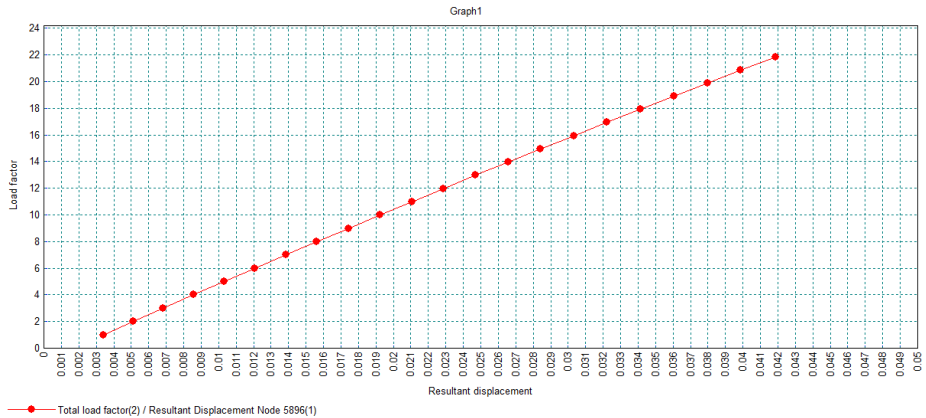
Resultant displacement (bearing capacity stage)

A resultant displacement of 42mm is obtained for this phase.

Bearing Capacity of Shallow Foundation

- Select the node at the top-right of the pier.
- Use **Utilities > Graph wizard** to create a graph of time history of nodal resultant displacement for the bearing capacity loadcase against total load factor.

A graph of the resultant displacement of the pier top for the bearing capacity branch is created in LUSAS. From this the value of load factor leading to the 40 mm settlement can be obtained. The bearing capacity in this case is $500 + 21.82 \times 500 = 11410$ kPa. It is to be noted this value must be factored by a safety factor related to design considerations.



Resultant displacement (bearing capacity stage) (m)