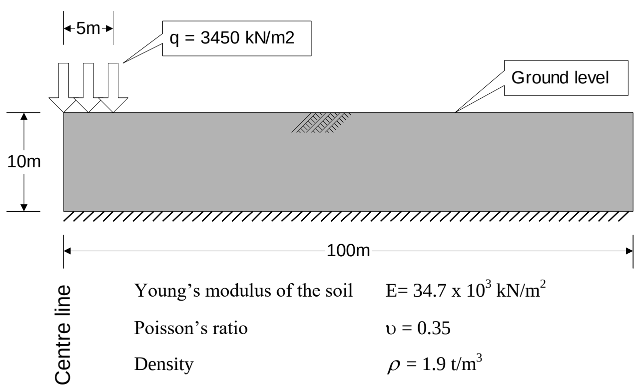
2D Consolidation under a Strip Footing

For LUSAS version:	20.0
For software product(s):	Any Plus version.
With product option(s):	Nonlinear, Dynamic.

Description

The pore water pressure dissipation and settlement in a soil following the application of a distributed load is to be investigated.

Horizontal displacement at the left edge is restrained to model the symmetrical boundary condition. Vertical displacement at the base of the soil is restrained as the soil rests on solid rock. The pore pressure at ground level is prescribed to zero.



Young's modulus of the soil	E= 34.7 x 10^3 kN/m^2
Poisson's ratio	υ = 0.35
Density	ρ = 1.9 t/m^3
Bulk modulus of water	K_w = 2.2 x 10^6 kN/m^2
Void ratio	e = 1.174
Unit weight of water	γ_w = 9.812 kN/m^3
Soil permeability	k_x=k_y=kz=1.0 x 10^{-8} m/s

Units of kN, m, t, s, C are used throughout.

Objectives

The required output from the analysis consists of:

- ❑ Settlement at the centre of the footing with time.
- ❑ Pore pressure distribution immediately after application of the load (undrained response)
- ❑ Pore water pressure dissipation with time at the centre of the footing

Keywords

2D, Plane Strain, Consolidation, Pore Water Pressure, Time Stepping, Nonlinear, Transient, Settlement, Two Phase Materials, Default Assignments, Graphs

Associated Files

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

- ❑ **pwp_modelling.lvb** carries out the modelling of the example.

Modelling

Running LUSAS Modeller

For details of how to run LUSAS Modeller, see the heading ***Running LUSAS Modeller*** in the *Introduction to LUSAS Worked Examples* document.

Note. This example is written assuming a new LUSAS Modeller session has been started. If continuing from an existing Modeller session select the menu command **File>New** to start a new model file. Modeller will prompt for any unsaved data and display the New Model dialog.

Creating a new model

- Enter a File name of **pwp**
- Use the default User-defined working folder.
- Ensure an Analysis type of **Structural** is set.
- Select an Analysis Category of **2D Inplane**
- Set Model units of **kN,m,t,s,C**
- Leave the Timescale units as **Seconds**
- Select a Startup template of **2D Plane Stress**.

- Ensure the Layout grid is set as **None**
- Enter a Title of **Consolidation under a Strip Footing**
- Click the **OK** button.

Note. The Undo button may be used to correct a mistake. The undo button allows any number of actions since the last save to be undone.

Mesh Definition

Since this analysis requires the modelling of pore water pressure, plane-strain two phase elements will be used.

Attributes
Mesh >
Surface...

- Select **Plane strain two phase, Quadrilateral, Quadratic** elements. Ensure the **Regular mesh** option is selected with **Automatic** chosen so that Modeller will use the mesh divisions assigned to each line.
- Give the attribute the name **Plane Strain Two Phase** and click the **OK** button to add the mesh attribute to the Treeview.
- In the Treeview click on the mesh attribute **Plane Strain Two Phase** with the right-hand mouse button and select the **Set Default** option**.** This will ensure all newly created surfaces will be assigned the elements defined in this mesh attribute.

Material Properties

For consolidation analysis both the elastic and two-phase soil properties need to be defined. The overall 'equivalent' bulk modulus of the soil is related to the bulk modulus of the pore fluid and the bulk modulus of the solid soil particles by the formula:

$$\frac{1}{Ke}=\frac{n}{Kf}+\frac{(1-n)}{Ks}\approx\frac{n}{Kf}$$

Where

Ke is the equivalent bulk modulus of the soil

Kf is the bulk modulus of the pore fluid

Ks is the bulk modulus of the solid soil particles

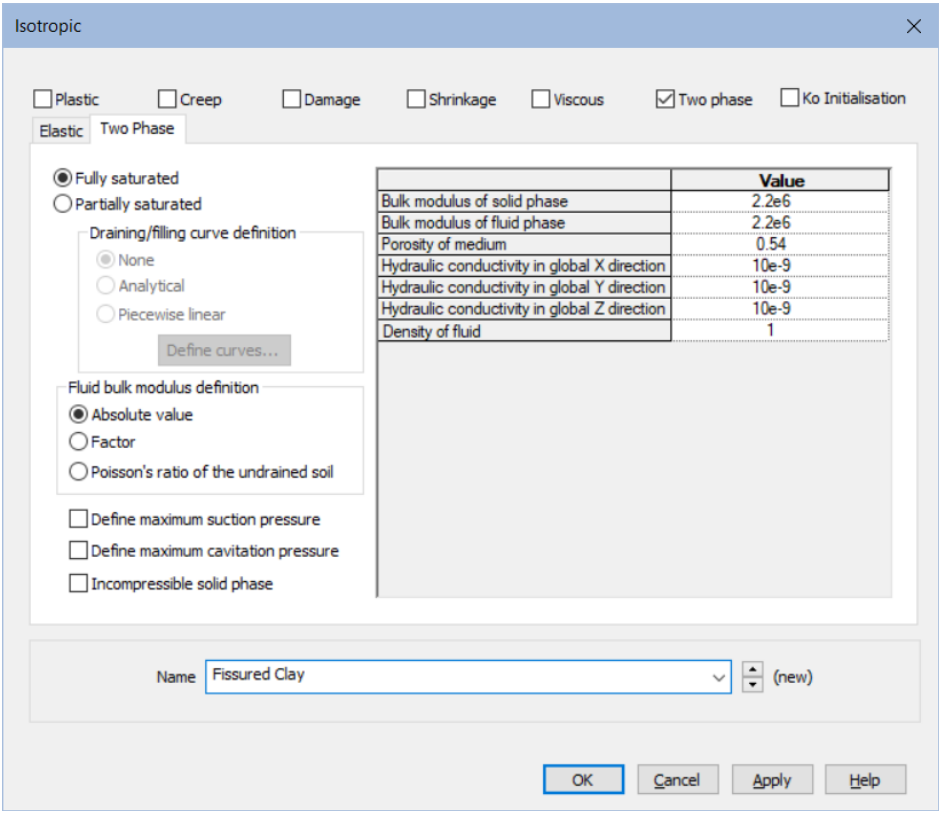
n is the porosity of the soil

The porosity of the soil is related to the void ratio by the formula:

$$\text{Porosity} \quad n = \frac{e}{1+e} = 0.54$$

Attributes
Material >
Isotropic...

- Enter the isotropic material properties for the soil as Young's Modulus **34.7E3**, Poisson's Ratio **0.35** and Mass density **1.9** (Note that the 'Mass density' field is renamed to be 'Fully saturated soil density' when the Two phase tab is selected)
- Click on the **Two phase** check box on the top of the dialog.



- For a **Fully saturated condition** enter the properties as shown in the dialog above.
- Enter the attribute name as **Fissured Clay** and click the **OK** button to add the attribute to the Treeview.
- In the Treeview click on the attribute **Fissured Clay** with the right-hand mouse button and select the **Set Default** option**.** This will ensure all newly created features will be assigned the properties defined in this material attribute.

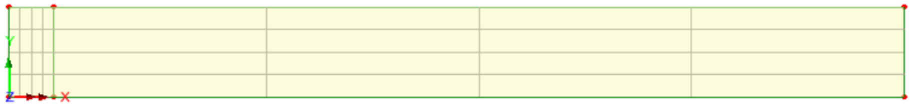
Feature Geometry

Geometry
Surface >
By Coords...

Enter coordinates of (**0, 0**), (**5, 0**), (**5, 10**) and (**0, 10**) to define the soil under the load. Use the Tab key to move to the next entry field on the dialog. When all coordinates have been entered click the **OK** button.

Geometry
Surface >
By Coords...

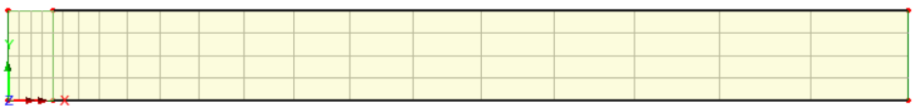
Enter the coordinates of **(5, 0), (100, 0), (100, 10), (5, 10)** to define the remainder of the soil.



Mesh Grading

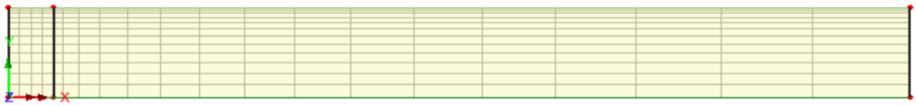
Attributes
Mesh >
Line...

- Define a Line mesh with element type **None** and **16** divisions.
- Select the **Spacing** button.
- Choose the **Uniform transition** option and set the Ratio of first to last element to **0.1**
- Click **OK** to accept the spacing properties.
- Give the attribute name as **Null 16 Graded** and click the **Apply** button to add the attribute to the Treeview and keep the dialog active.
- Define a Line mesh with **12** divisions.
- Select the **Spacing** button.
- Enter the **Uniform transition** option and set the Ratio of first to last element to **5**
- Click **OK** to accept the spacing properties.
- Give the attribute name as **Null 12 Graded** and click **OK** to add the attribute to the Treeview.
- Select the two horizontal lines of the larger surface as shown and assign the mesh attribute **Null 16 Graded**



Note. When the mesh is graded with the smaller elements at the wrong end of a line the affected line(s) can be reversed

- Select the incorrectly oriented line and use the **Geometry>Line>Reverse** menu.
- Select the three vertical lines as shown (Use the **Shift** key to add to the initial selection) and assign the mesh attribute **Null 12 Graded**



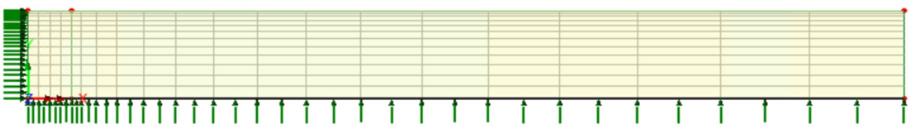
- Select any incorrectly oriented line(s) and use the **Geometry>Line>Reverse** menu.

The end result should be a mesh that looks like the above image

Supports

Assign the supports to the model.

- Select the vertical Line on the left-hand side of the model and drag and drop the support attribute **Fixed in X** from the Treeview onto the selection. With the **Assign to lines** option selected click **OK** to assign the support attribute to **All analysis loadcases**
- Select the 2 horizontal Lines representing the base of the soil and drag and drop the support attribute **Fixed in Y** from the Treeview onto the selection. With the **Assign to lines** option selected click **OK** to assign the support attribute to **All analysis loadcases**



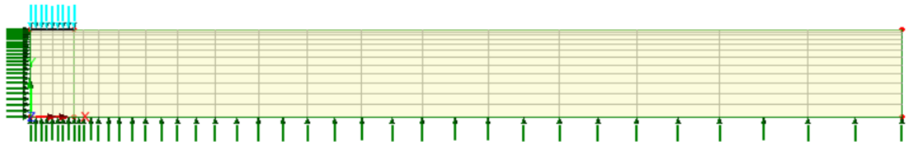
Loading

To define the distributed load:

Attributes
Loading...

- Select the **Face** option and click **Next**
- Enter loading of **3450** in the **y Direction**
- Enter the attribute name of **Distributed Load** and click the **Finish** button.

- Select the top line on the left-hand side of the model and drag and drop the **Distributed Load** attribute onto the selection. With the **Assign to lines** option selected click **OK** to assign the loadcase attribute to **All loadcases**.

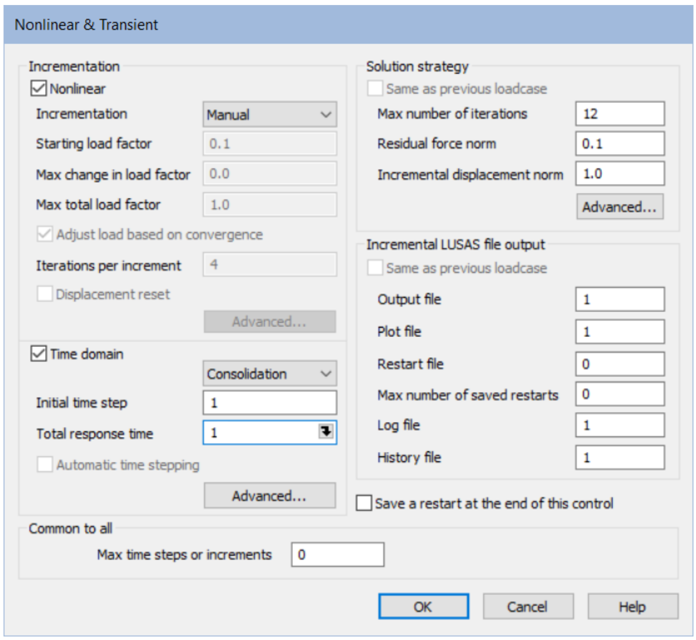


Analysis Control

Undrained loading stage

The undrained loading of the soil must be calculated to establish the initial conditions of the consolidation analysis.

- In the Treeview expand **Analysis 1** then right-click on **Loadcase 1** and select **Nonlinear & Transient** from the **Controls** menu.



In the Incrementation section:

- Select the **Nonlinear** option and choose **Manual** incrementation.
- Select the **Time domain** option and chose **Consolidation** from the drop down list.

- Enter an **Initial time step** of **1**
- Enter **Total Response Time** of **1**.
- Set **Max time steps or increments** to **0.**
- Click **OK**.

Consolidation stage

To allow dissipation of the excess pore-water pressures a drainage path must be defined which for this example is through the ground level of the soil. A new loadcase needs to be created for this.

Analyses
Loadcase…

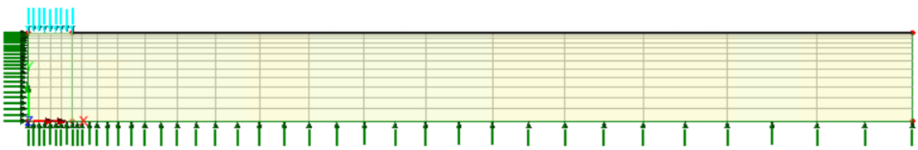
- Click **OK** to create **Loadcase 2**
- Click **OK** to acknowledge that assignments to loadcase 2 have been made from assignments made previously that related to a range of loadcases. See the Help button if more information is required.

Supports

Define a support to prescribe zero pore pressure.

Attributes
Support…

- Specify the **Pore pressure** to be **Fixed.**
- Enter the attribute name as **Drainage Path** and click the **OK** button.
- Select the Line at ground level on the right-hand side of the model (see following mage) and drag and drop the support attribute **Drainage Path** from the Treeview onto the selection. With the **Assign to lines** option selected, choose the **From loadcase (nonlinear and transient) option** to assign the supports to **Analysis 1** and **Loadcase 2**. Click **OK**.

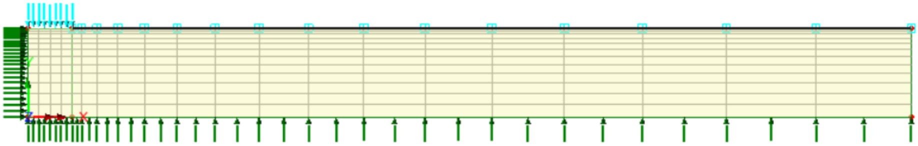


Loading

Attributes
Loading...

- Select the **Prescribed displacement** option and click **Next**
- With the **Total** option selected, set the Pore Pressure to be **Fixed** and enter a prescribed displacement of 0. Name the attribute **Surface Excess Pore Water Pressure** and click **Finish**.

- With the line at ground level on the right-hand side of the model still selected (see previous image), drag and drop the loading attribute **Surface Excess Pore Water Pressure** from the Treeview onto the selection for a **Single loadcase** named **Loadcase 2**. The applied loading will be visualised as shown below.



Analysis Control

Consolidation stage

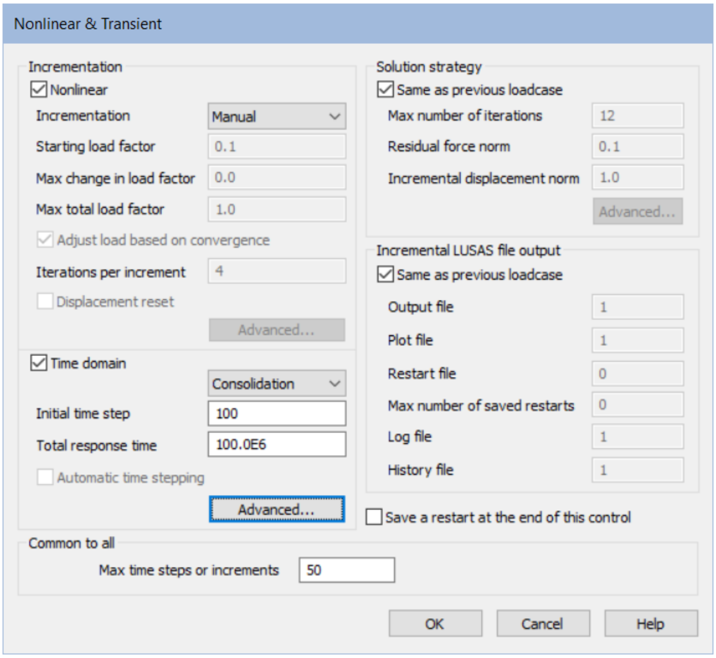
With this consolidation problem an automatic time stepping procedure is adopted. This is because consolidation is a typical diffusion process in which the field changes rapidly at the start of the process before settling down to a steady state condition a considerable time after the initial load is applied. The automatic time stepping procedure enables the time step to be modified so that the small time steps required at the start of the problem can be increased as the analysis progresses. In some cases the overall response time can be orders of magnitude larger than the initial time step.

The initial time step is important since the early variations in pore pressure must be accurately accounted for. Vermeer and Verruijt suggest the following criteria for determining the initial time step where Δh is the minimum distance between nodes.

$$\Delta t \ \geq \ \frac{\gamma_{\omega}}{6Ek}(\Delta h)^2 = \frac{9.812}{6 \text{ x } 34.7E3 \text{ x } 1E-8} \text{ x } 0.14^2 \approx 100 \text{ secs}$$

Note. The distance between two nodes can be determined by selecting the two nodes and then picking the **Tools> Mesh> Distance between Nodes** menu item.

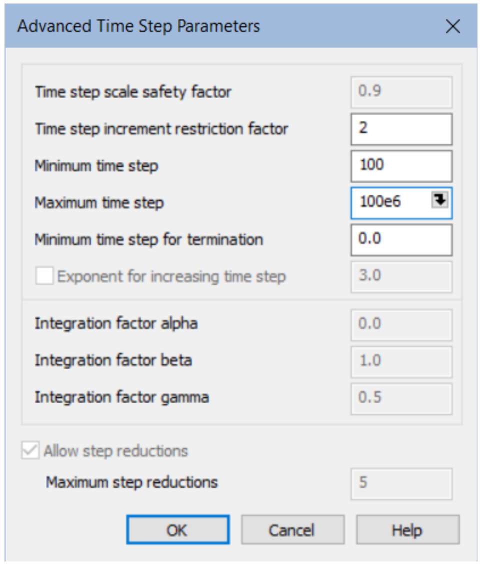
- In the Treeview expand **Analysis 1** then right-click on **Loadcase 2** and select **Nonlinear & Transient** from the **Controls** menu.



In the Incrementation section:

- Select the **Nonlinear** option and choose **Manual** incrementation.
- Select the **Time domain** option and chose **Consolidation** from the drop down list.
- Enter an **Initial time step** of **100**
- Enter **Total Response Time** of **100E6**.
- Set **Max time steps or increments** to 5**0.**
- Select the **Advanced** button in the Time domain section of the dialog.

- On the Advanced time step parameters dialog set the 'Time step increment restriction factor' to **2**
- Set the 'Minimum time step' to **100**
- Ensure the 'Maximum time step' is set to **100E6**
- Ensure the 'Minimum time step for termination' is set to **0**
- Click **OK** to return to the Nonlinear and Transient control dialog.
- Click **OK** to set the loadcase control.



Saving the model

File
Save

Save the model file.

Running the Analysis

Open the **Solve Now** dialog. Ensure **Analysis 1** is selected and press **OK** to run the analysis.

A LUSAS Datafile will be created from the model information. The LUSAS Solver uses this datafile to perform the analysis.

If the analysis is successful...

Analysis loadcase results are added to the Treeview.

In addition, 2 files will be created in the Associated Model Data directory where the model file resides:

- **pwp.out** this output file contains details of model data, assigned attributes and selected statistics of the analysis.
- **pwp.mys** this is the LUSAS results file which is loaded automatically into the Treeview to allow results processing to take place.

If the analysis fails...

If the analysis fails, the output file will provide information relating to the nature of the error encountered. Any errors listed in the output file should be fixed in LUSAS Modeller before saving the model and re-running the analysis.

Rebuilding a Model

If it proves impossible for you to correct the errors reported a file is provided to enable you to re-create the model from scratch and run an analysis successfully. You may download this associated file from the user area of the LUSAS website.

- **pwp_modelling.lvb** this file carries out the modelling of the example.

File
New…

Start a new model file. If an existing model is open Modeller will prompt for unsaved data to be saved before opening the new file.

- Enter the file name as **pwp**
- Use the default **User-defined** working folder.
- Ensure an Analysis type of **2D Inplane** is set.
- Click the **OK** button.

Note. There is no need to enter any other new model details when a script is run to build a model, since the contents of the script will overwrite any other settings made.

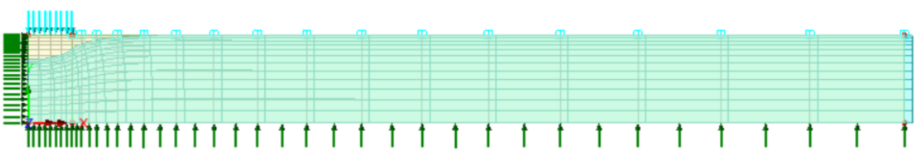
File
Script >
Run Script...

To recreate the model, select the file **pwp_modelling.lvb** that was downloaded and placed in a folder of your choosing.

Solve the analysis to generate the results.

Viewing the Results

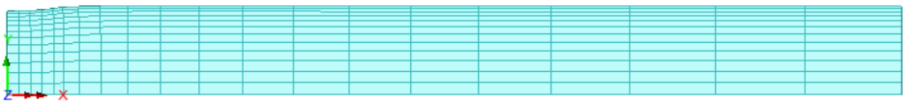
Loadcase results can be seen in the Treeview. For a nonlinear analysis the last results time step is set active by default.



- If present, turn off the **Geometry**, **Mesh** and **Attributes** layers in the Treeview.

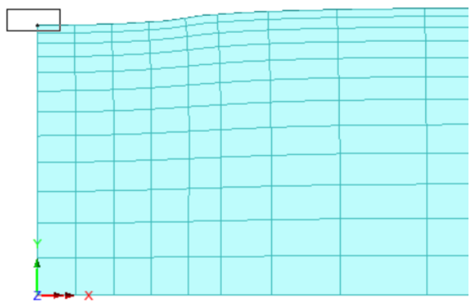
Settlement

- In the Treeview, right-click on the first load increment **Time Step 0** and select the **Set Active** option to view results immediately after the loading is applied.
- Click the **Deformations** button in the Treeview and select the **Specify factor** option. Specify a factor of **1** and click **OK** to visualise the deformed mesh for the first time step.



A graph of the deformation over time will be created using the graph wizard.

- Zoom into the left-hand side of the model and select only the node on the centre line representing the bottom of the footing.



Graphing settlement

Utilities
Graph Wizard...

- With the **Time history** option selected click **Next**

Firstly define the data to be used for the X axis.

- Selected the **Named** option and click **Next**
- Select **Response Time** from the drop down list and click **Next**

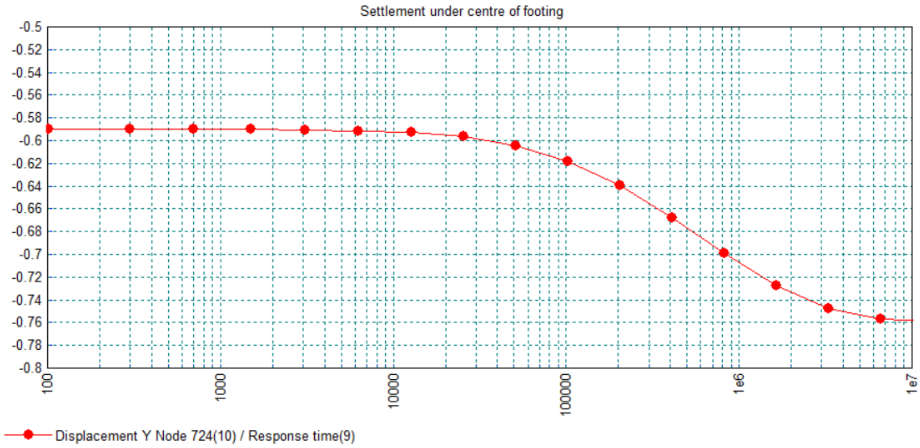
Secondly define the data for the Y axis.

- With the **Nodal** option selected and click **Next**
- Select **Displacement** from the Entity drop down list and **DY** from the Component drop down list.
- The selected node number will appear in the **Specified single node** drop down list. Click **Next**

For the X Scale enter **Manual** values of minimum **100** and maximum **10e6** respectively.

- Select the **Use logarithmic scale** option.

- Title the graph as **Settlement under centre of footing**
- Click **Finish** to display the graph of settlement over time under the centre of the strip footing.

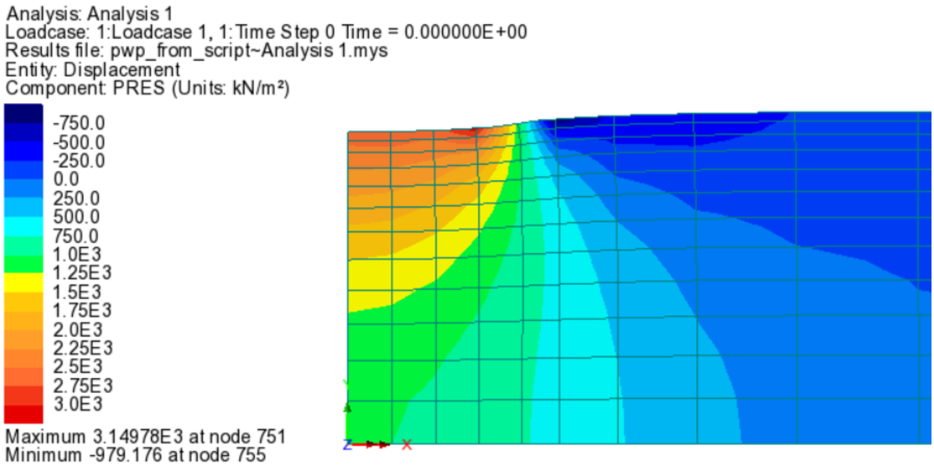


Close the graph window

Pore water pressure

The distribution of pore pressure is to be shown using contours.

- With **Time Step 0** set active in the Treeview, and with no features selected, click the right-hand mouse button in a blank part of the view window and select **Contours** to add the contours layer to the Treeview.
- Select **Displacement** from the Entity drop down list and **PRES** from the component drop down list.
- Select the **Appearance** tab and for the **Classic** contour type press the **Set** button and set the **Interval** to **250**
- Click the **OK** button to exit the dialog, and again to display contours of the undrained pore pressure distribution (Time step 0) immediately after the loading is applied.



- To observe the distribution of pore water pressure at a particular time after the application of loading, activate the appropriate time step from the Treeview by selecting the time step with the right-hand mouse button and choosing the **Set Active** option.

Note. The dissipation of pore water pressure over time may also be observed by creating an animation of the contour display.

Graphing pore water pressure

The dissipation of pore water pressure under the footing is to be presented on a graph.

- In the Treeview turn off the display of the Contours layer.
- Select the node on the centreline under the footing as before.

Utilities
Graph Wizard...

- With the **Time history** option selected click **Next**

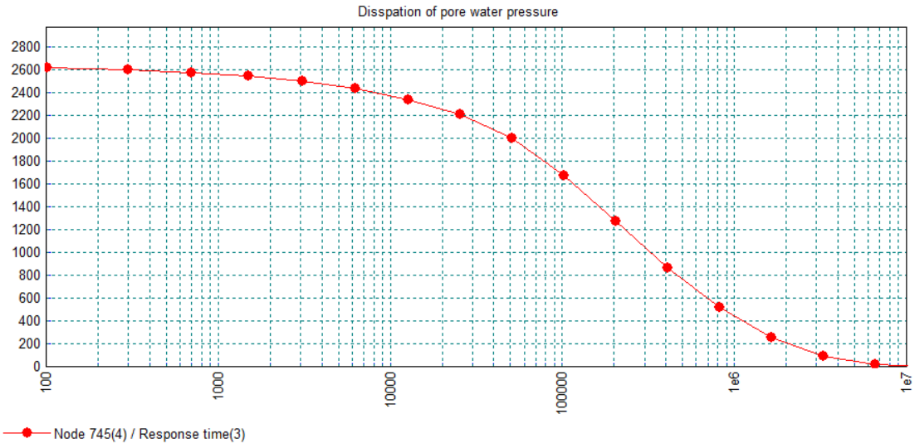
Firstly define the data to be used for the X axis:

- Selected the **Named** option and click **Next**
- Select **Response Time** from the drop down list and click **Next**

Secondly define the data for the Y axis:

- With the **Nodal** option selected and click **Next**
- Select **Displacement** from the Entity drop down list and **PRES** from the Component drop down list.

- The extent to be graphed will be set to **Specified single node** and the node selected will be seen in the Specify node field. Click **Next**
- For the X Scale ensure that **Manual** values of minimum **100** and maximum **10e6** respectively are used.
- Ensure the **Use Logarithmic scale** option is selected.
- For the Y Scale leave the values as the defaults.
- Title the graph as **Dissipation of pore water pressure**
- Click **Finish** to display the dissipation of pore water pressure over time under the centre of the strip footing.



Close the graph window.

Graphing effective stress in soil

As the pore water dissipates the load is carried by the soil. The increase in effective stress in soil can be observed on a graph of effective stress against response time.

- Ensure the node on the centreline under the footing still selected.

Utilities
Graph Wizard...

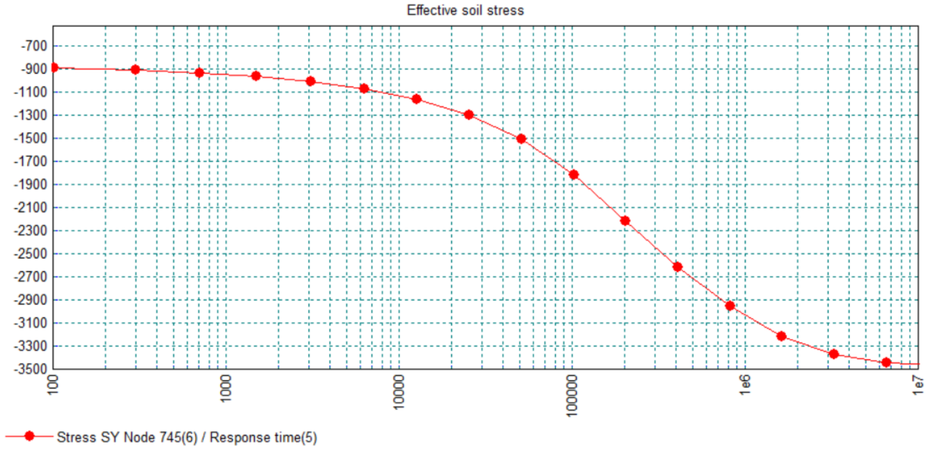
- With the **Time history** option selected click **Next**

Firstly define the data to be used for the X axis

- Select the **Named** option and click **Next**
- Select **Response Time** from the drop down list and click **Next**

Secondly define the data for the Y axis

- With the **Nodal** option selected and click **Next**
- Select **Stress** from the Entity drop down list and **SY** from the Component drop down list.
- The extent to be graphed will be set to **Specified single node** and the node selected will be seen in the Specify node field. Click **Next**
- For the X Scale ensure that **Manual** values of minimum **100** and maximum **10e6** respectively are used.
- Ensure the **Use Logarithmic scale** option is selected.
- For the Y Scale leave the values as the defaults.
- Title the graph as **Effective soil stress**
- Click **Finish** to display the graph of the effective soil stress over time under the centre of the strip footing.



This completes the example.