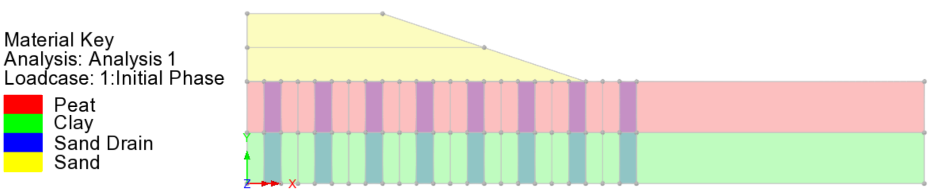
Stability of Embankment Constructed on Clayey Soil Treated with Sand Columns

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

Problem Description

The construction of a 4m high embankment built on clay soil is analysed. The embankment is constructed in stages and rests on two layers of clay and peat, each 3m thick. Sand drains are employed to speed up the soil consolidation process. The water table is at ground level. Because of symmetry, it is sufficient to only model one half of the embankment.



Half-model of embankment

The embankment is built in two stages over a period of fourteen days. The first layer is constructed over two days. The soil is then allowed to consolidate for a further ten days before a second layer is added, again over two days. Finally, the soil is allowed to consolidate until the maximum excess pore water pressure falls below 0.5kPa.

Keywords

Consolidation, Sand Columns, Pore Water Pressure, Settlement.

Associated Files

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

- ❑ **stability_of_embankment.lvb** carries out automated modelling of the example.

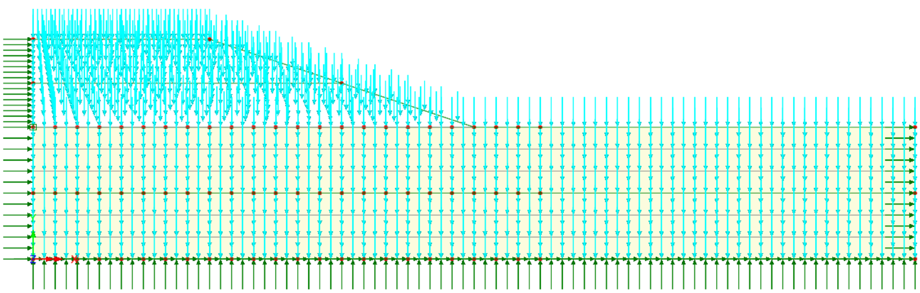
- Use **File > New** to create a new model of a suitable name in a chosen location.
- Use **File > Script > Run Script** to open the lvb file named above that was downloaded and placed in a folder of your choosing.

Objectives

Calculating the change of excess pore water pressure and settlement over time.

Modelling overview

The model is created using an analysis category of **2D Inplane**, with model units of **kN,m,t,s,C**. The **Time Scale** is set to **Days**.



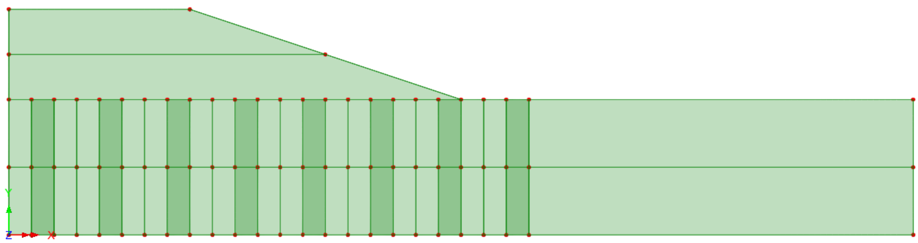
The model created from running the supplied script

Feature Geometry

The model is created through point and line features which are subsequently converted into surfaces. It is good practice to use the commands **Copy** and **Sweep** to reduce considerably the time needed to develop the model.

The following image shows the surfaces used to define this problem. At each sand drain location, two surfaces are defined, where one is assigned general soil material properties, and one is assigned sand drain properties. The location of the sand drains is seen by the additive colouration of the surfaces that represent them.

The water table lies at the ground surface.



Embankment model

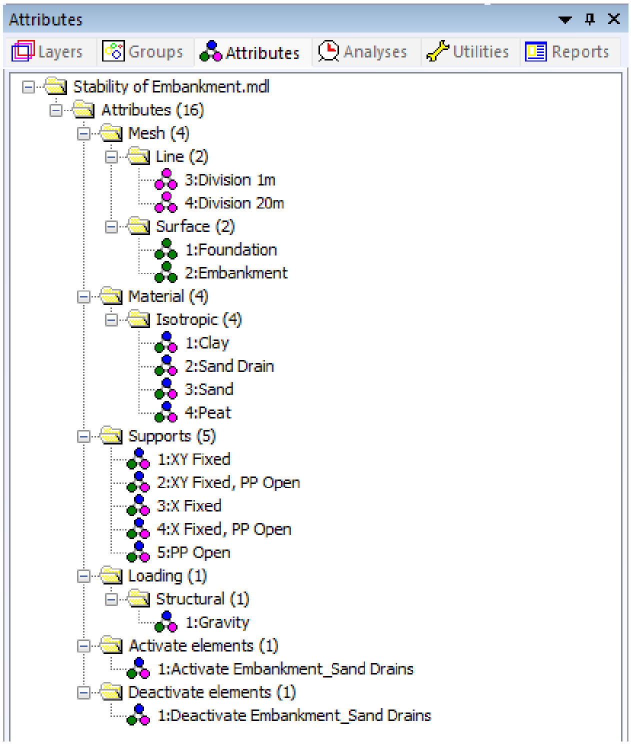
Replacing material within surfaces

Note. In situations where we need to deactivate soil and replace it with a different material (sand 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.

Attributes

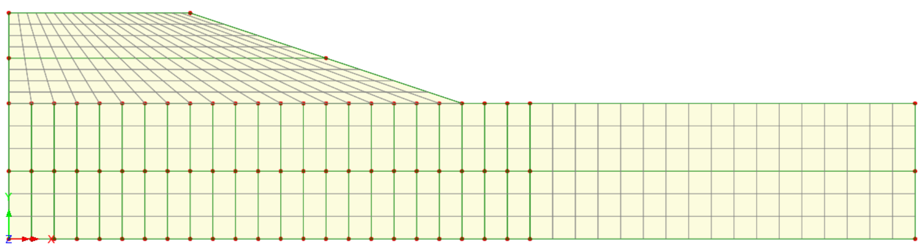
Model attributes (mesh, material, geometric properties, etc.) are defined and assigned to the model.



Model Attributes

Mesh

The foundation, comprising sand and peat layers, is meshed using plane strain two phase, quadrilateral, quadratic elements (QPN8P), whereas the embankment is meshed with plane strain, quadrilateral, quadratic elements (QPN8).



Model Meshing

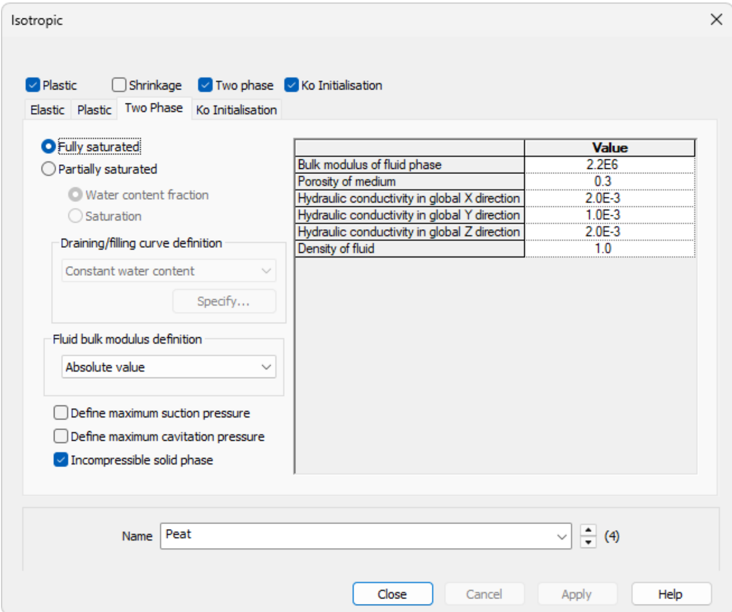
Materials

An isotropic nonlinear material utilising 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. Material properties for all materials are listed in Table 1.

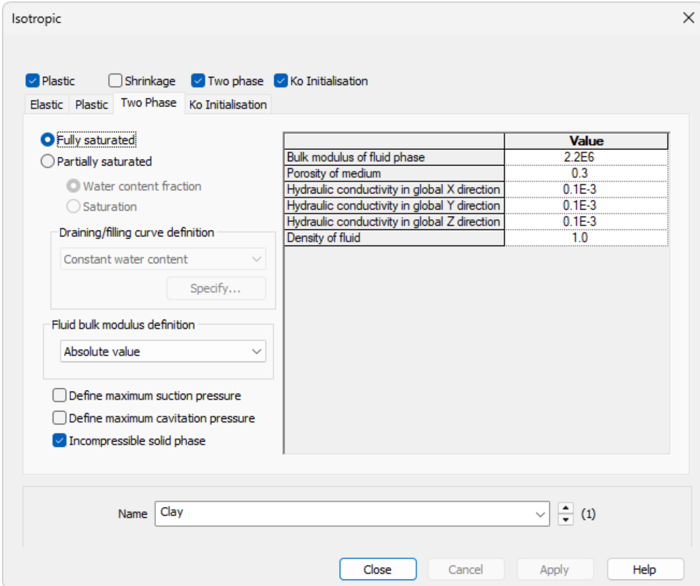
Table 1: material properties

Layer	Soil grain density	Young's modulus, E	Poisson's ratio, ν	Angle of friction, φ	Cohesion, c	K_0
Peat	1.143 t/m^3	350 kPa	0.35	20°	5 kPa	0.658
Clay	2.143 t/m^3	1.0E3 kPa	0.33	24°	2 kPa	0.593
Sand	1.6 t/m^3	3.0E3 kPa	0.3	30°	1 kPa	0.5
Sand drain	2.67 t/m^3	80.0E3 kPa	0.3	35°	10 kPa	-

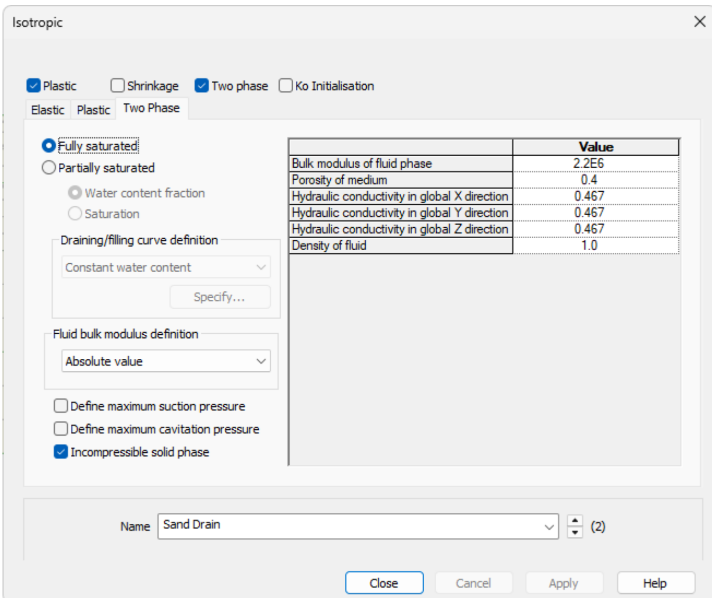
Two-phase properties for the relevant materials are given in the following images. Dilation is zero and the Rankine cut off prevents tensile stresses developing in the soil.



Two-phase properties for peat layer



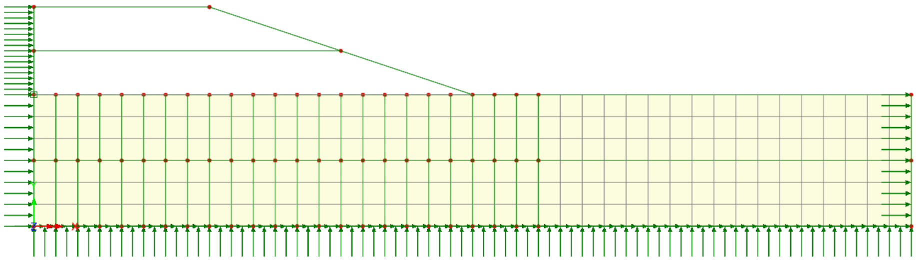
Two-phase properties for clay layer



Two-phase properties for sand drain

Supports

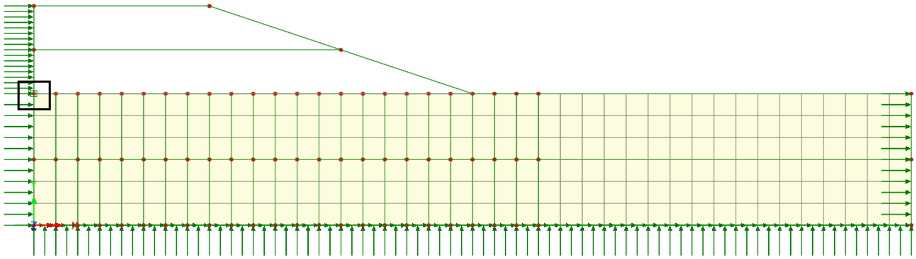
The model is restrained in X and Y directions at its base and in the X direction for the lateral sides as shown. These supports are activated at the initial analysis phase.



Boundary conditions at initial stage

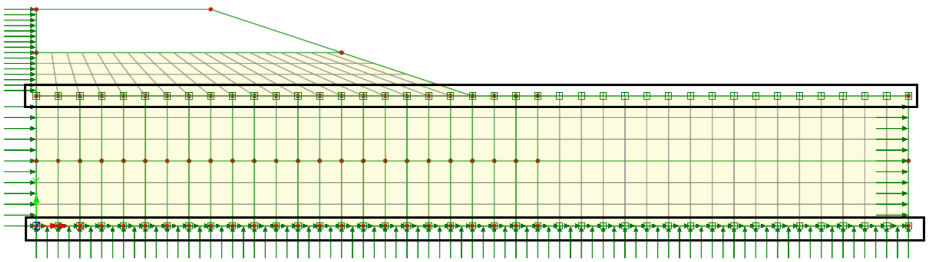
Water table

To establish the position of the water table, the pore pressure support is set to **Open** and the X-direction restrained and then assigned to the point at the upper -left of the existing ground (as shown below). This support condition is activated during the first stage as well.



Pore water boundary condition at initial stage

During construction of the embankment, the pore pressures from the hydrostatic pressure distribution established in the initial phase are fixed at the top and bottom of the foundation by setting the pore pressure to **Open** and assigning the pore pressure support to the lines at those locations.



Pore water boundary conditions at embankment construction stage

Loads

Gravity load is defined for assignment to each loadcase using **Attributes > Loading > Body Force**.

Other Attributes

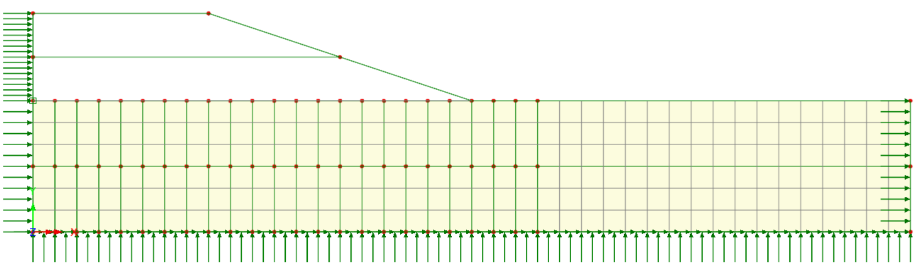
Deactivate and Activate attributes are employed to simulate the construction of the embankment and sand drains as explained in more detail in the following section.

Construction stages considered

Analysis 1

Initial Phase

This stage establishes the initial stress and water pressure distribution with gravity acting as a load and the model being restricted from movement at the base and sides. During this stage, the surfaces representing the embankment and sand drains are deactivated n the model by being assigned the **Deactivate** attribute.



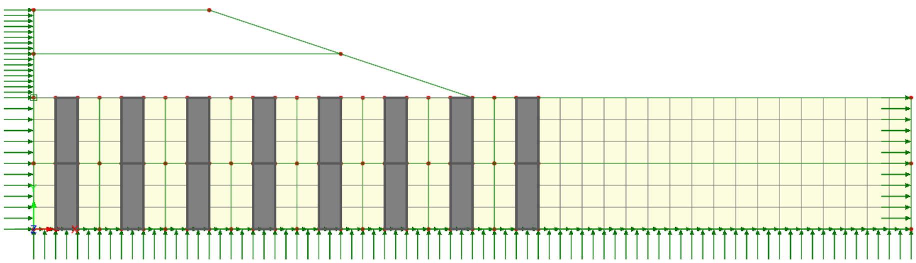
Analysis 1: Initial phase

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

Analysis 1

Installation of Sand Drains

As explained earlier, the surfaces assigned with the sand drain material are activated to simulate the installation process, while the clay and peat materials occupying the same location as the sand drains are deactivated. Surfaces assigned with sand drain material are shown selected below.



Analysis 1: Sand drain installation

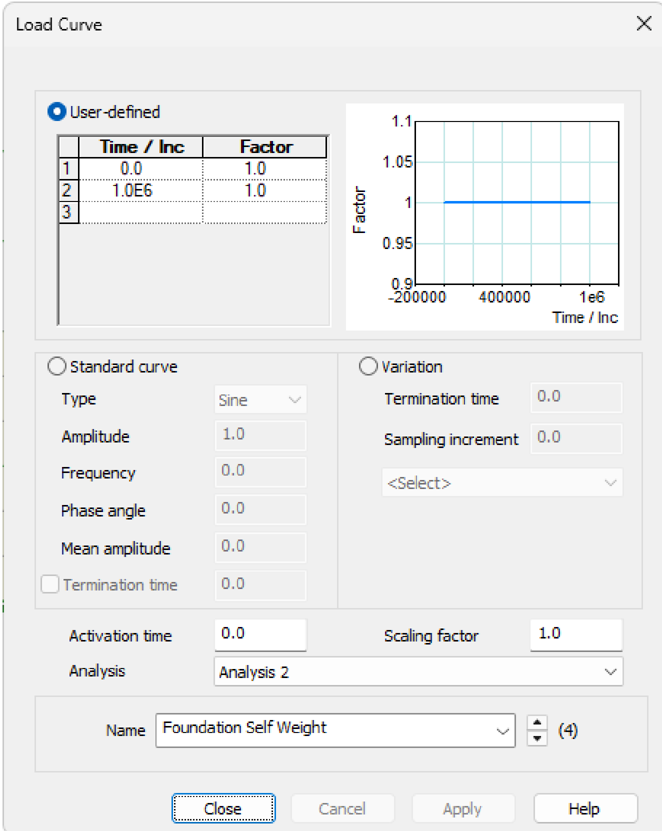
Nonlinear analysis control properties are defined for this phase, with all parameters are left at their default values.

Analysis 2

Construction of 1st layer of Embankment

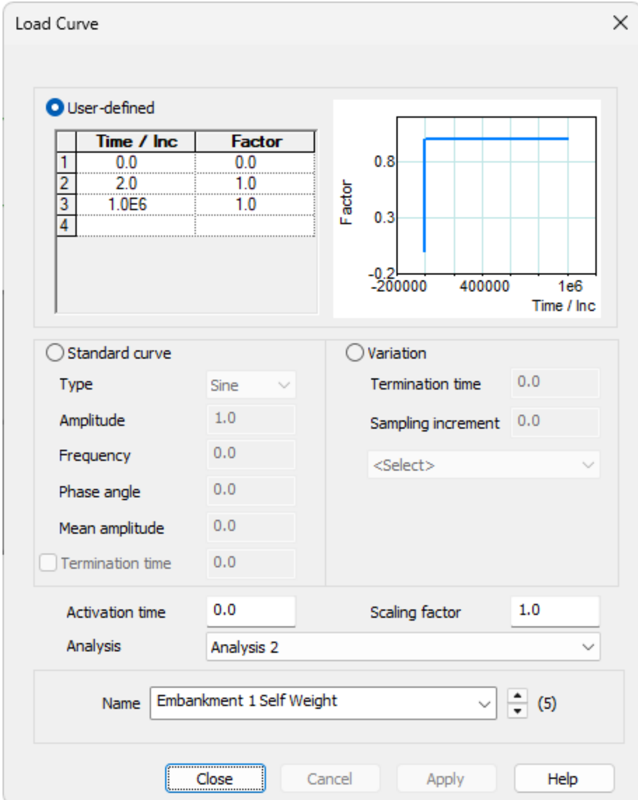
To model the construction of the embankment over time, the gravity loading needs to be assigned gradually over time. This can be achieved by using Load Curves. These can be added by right-clicking on the **Analysis 2** entry in the Analyses treeview and selecting **New > Load Curve.**

In this way three load curves are created as follows:



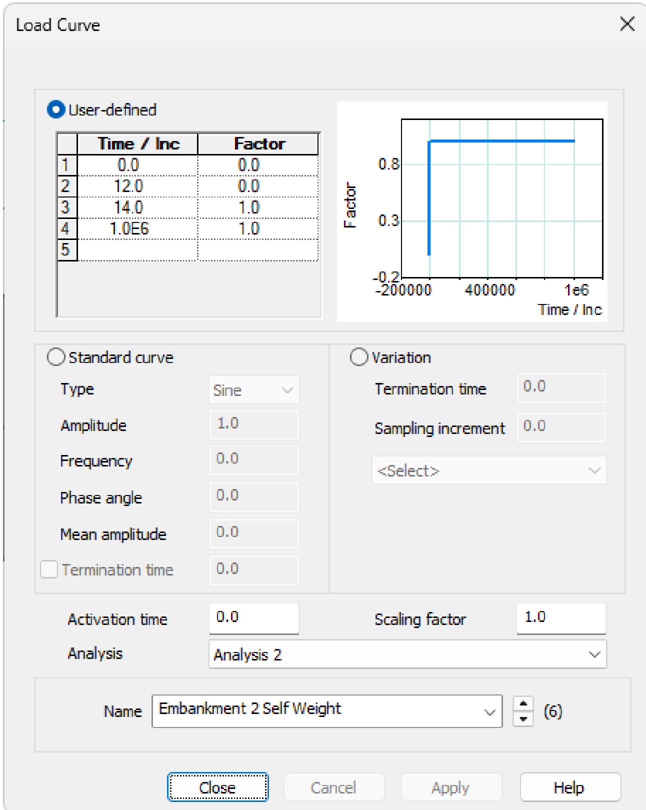
Load curve settings to model foundation self weight

On the dialog above, gravity is assigned to the foundation layers from Time day 0 to day 1E6.



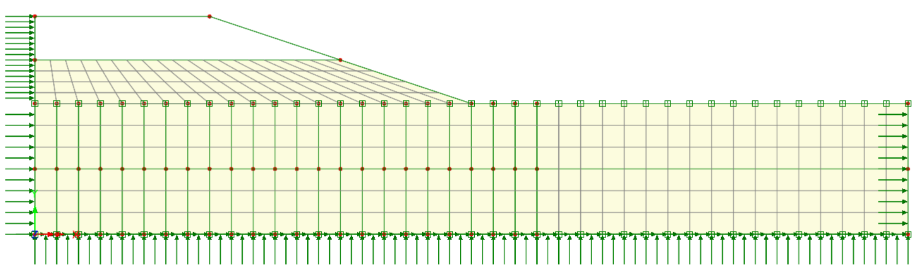
Load curve settings to model embankment 1 self weight

On the dialog above, gravity is increased from 0 on day 0 to 1 on day 2 to model the construction of the first layer of embankment over two days. It then remains constant until the end of the analysis.



Load curve settings to model embankment 2 self weight

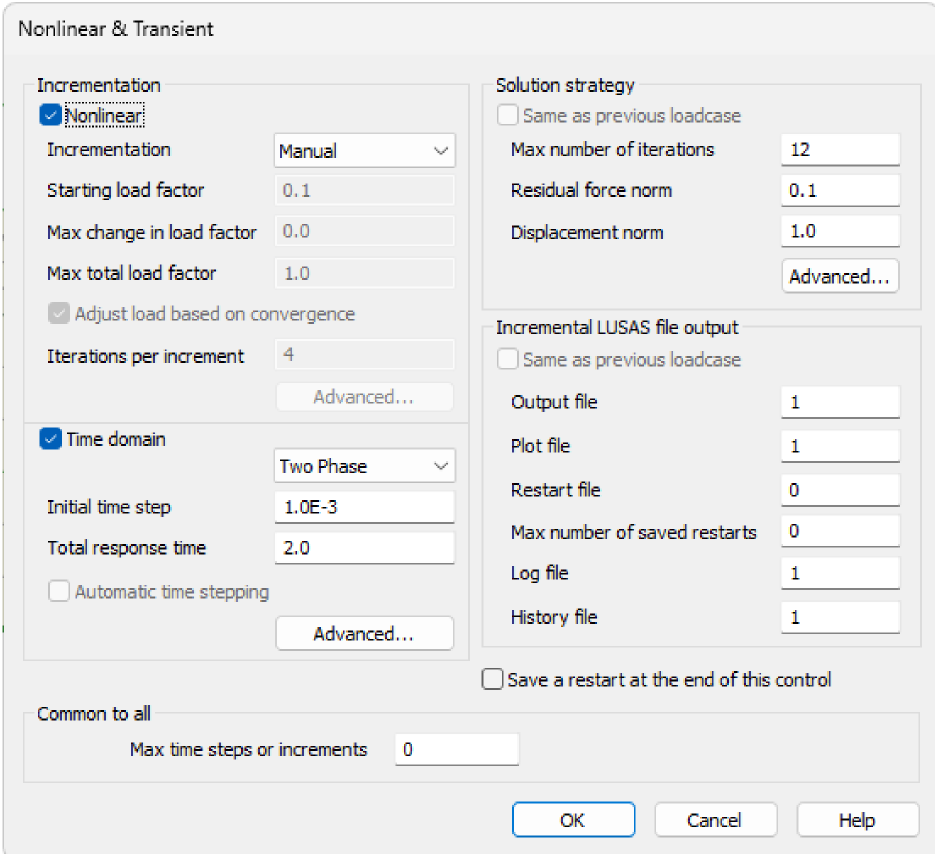
On the dialog above, gravity is increased from 0 on day 12 to 1 on day 14 to model the construction of the second layer of embankment over two days. It then remains constant until the end of the analysis



Analysis 2: First layer of embankment stage

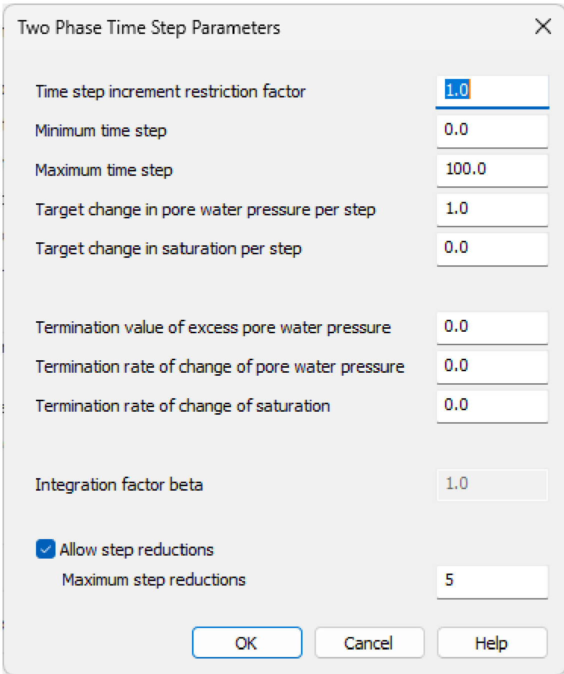
In this construction stage, the first layer of the embankment is activated in the model by using the Activate facility. The excess pore water freely dissipates through the upper and lower boundary of the foundation layers.

Nonlinear analysis control properties for this stage are set as shown below. The total response time is set to 2 days with a starting time step is 0.001 days. Automatic time stepping is used for this stage, and in fact for the other stages as well, with a target change in pore water pressure per step of 1kPa.



Nonlinear analysis control parameters

Advanced timestep parameters are shown in the next dialog.

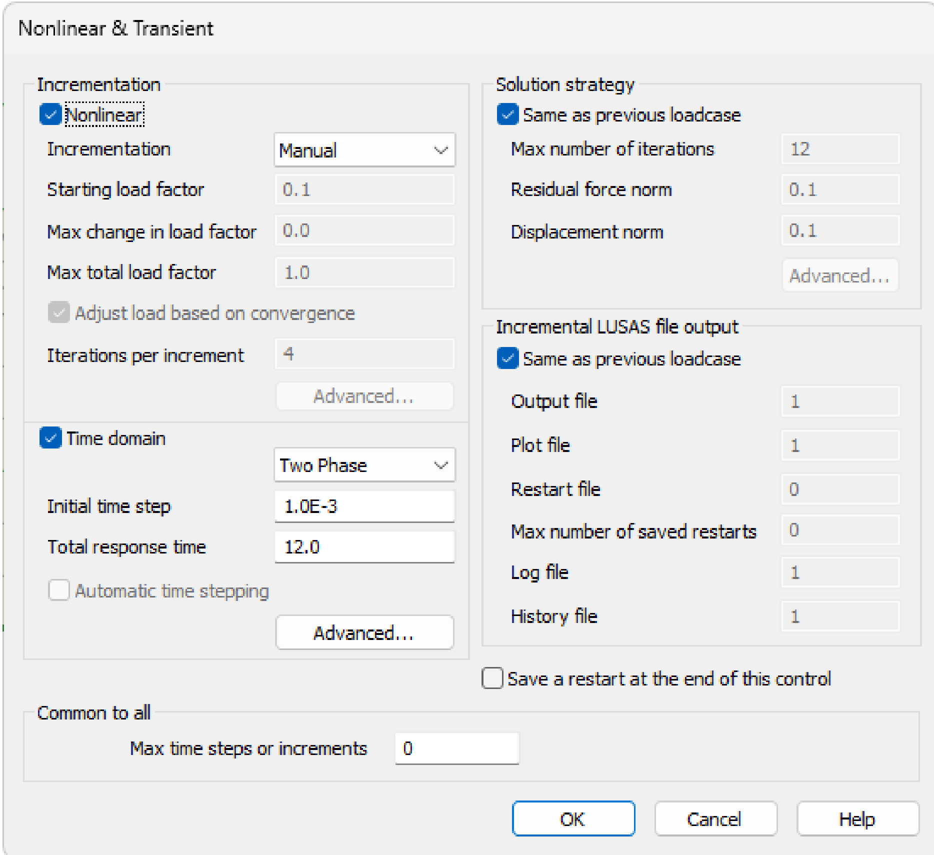


Nonlinear analysis control parameters – advanced time domain

Analysis 2

Consolidation of 1st layer of Embankment

This phase allows the embankment to consolidate over 10 days. Nonlinear analysis control properties are set as shown below. As the soil is allowed to consolidate for 10 days, the total time is now set to 12 days.

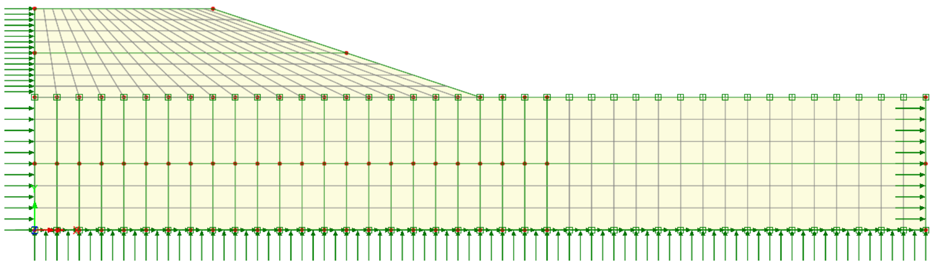


Nonlinear analysis control parameters – embankment 1 consolidation

Analysis 2

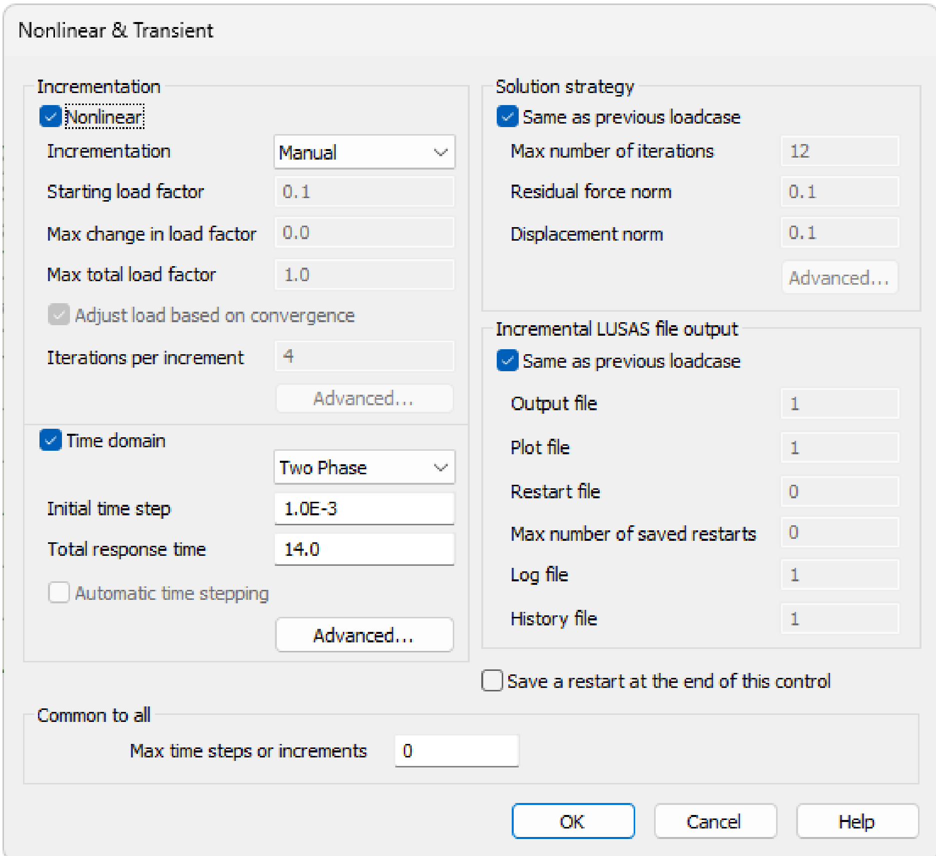
Construction of 2nd layer of Embankment

The second layer of the embankment is activated.



Analysis 2 : Second construction stage activated

Nonlinear analysis control properties are set as shown below. The total time is set to 14 days.

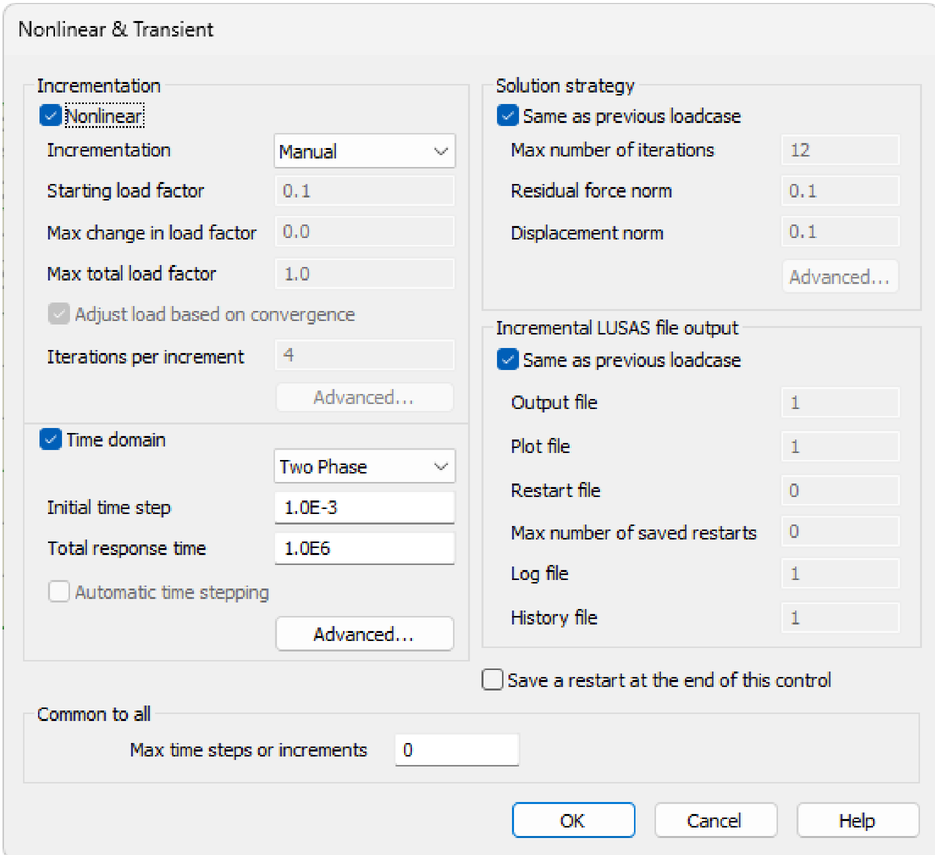


Nonlinear analysis control parameters – embankment 2 construction

Analysis 2

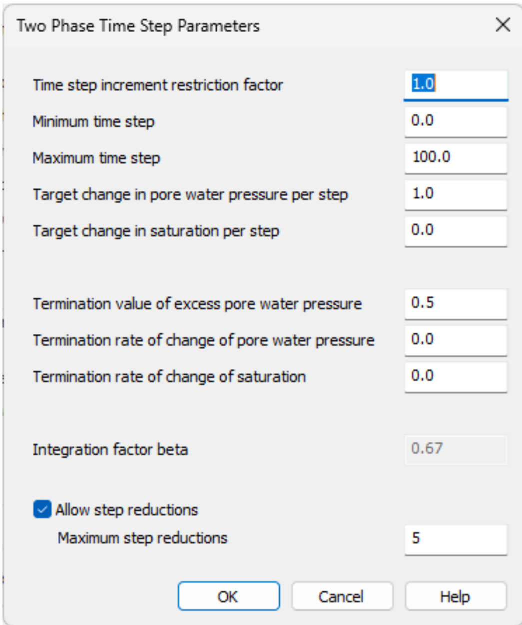
Consolidation of 2nd layer of Embankment

At the final stage, the model is allowed to stabilise until the maximum excess pore water pressure falls below 0.5 kPa to obtain the final overall settlement. Nonlinear analysis control properties are set as shown below.



Nonlinear analysis control parameters – embankment 2 consolidation

Advanced timestep parameters are shown in the next dialog.



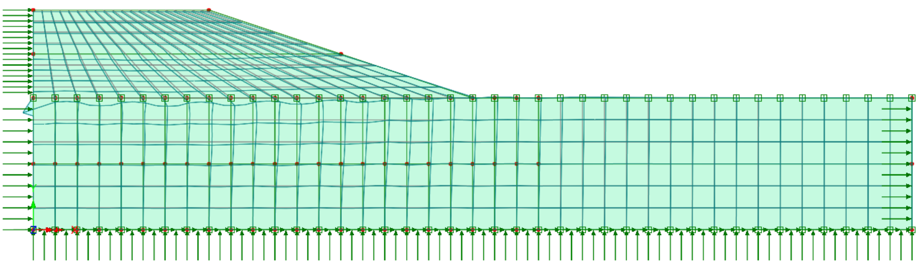
Nonlinear analysis control parameters – advanced time domain

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 Treeview. The initial view will show this.

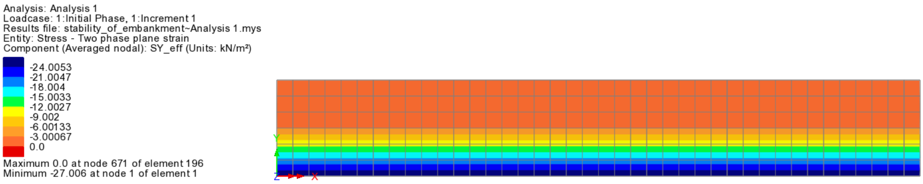


- Set active the Analysis **Initial phase**.

- Turn off the display of the geometry and deformed mesh layer and the supports and loading.

Stress

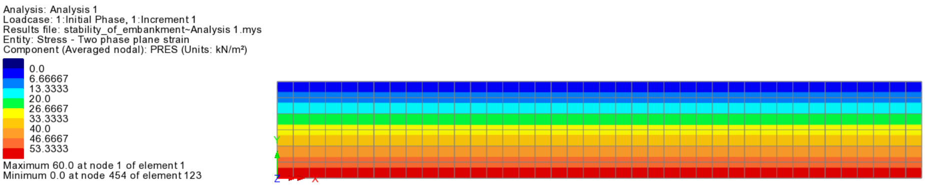
- Add the **Contours** layer to plot contours of entity **Stress – Two phase plane strain** and component **Sy_eff**



Analysis 1: Effective stress at the initial stage (kN/m^2)

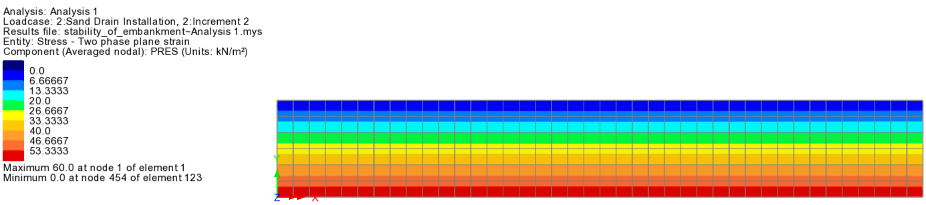
Pore Pressure

- Change the contours layer to show contours of entity **Stress – Two phase plane strain** and component **PRES**

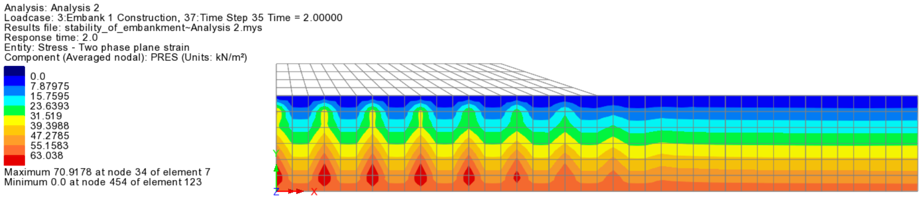


Analysis 1: Pore pressure at the initial stage (kN/m^2)

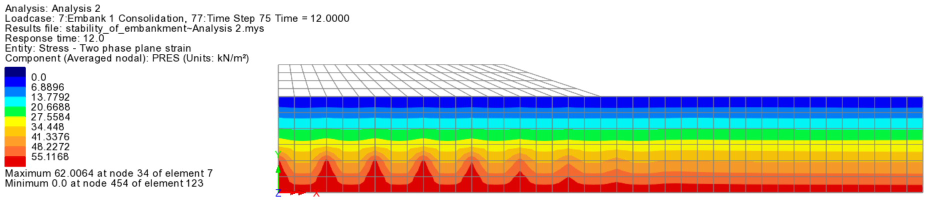
- By setting each following loadcase active in turn, the distribution of pore pressure in different stages can be seen. These clearly illustrate the generation of the excess pore pressures and their dissipation with time/stages.



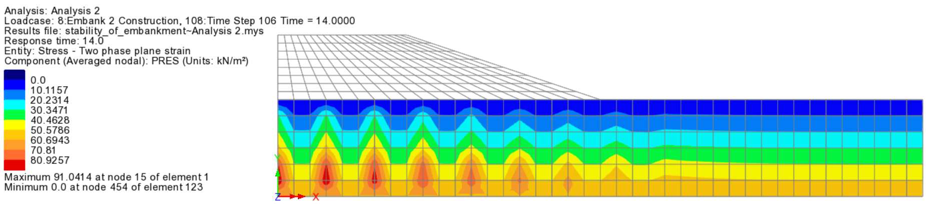
Analysis 1: Installation of sand drains



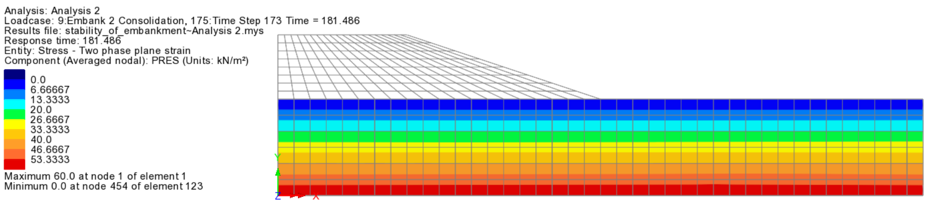
Analysis 2:Embankment 1 construction



Analysis 2: Embankment 1 consolidation



Analysis 2:Embankment 2 construction

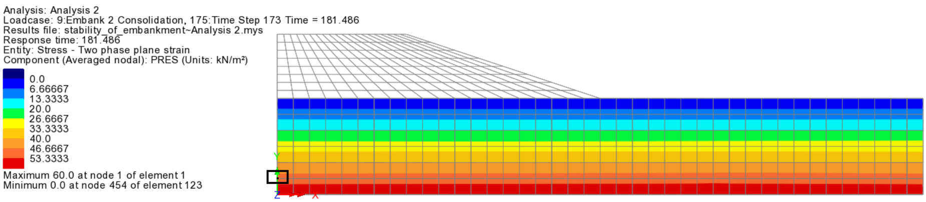


Analysis 2: Embankment 2 consolidation

Settlement

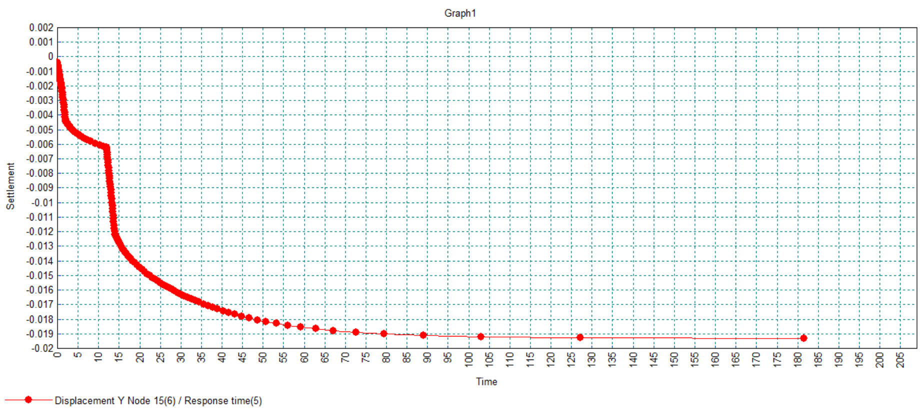
To plot a graph of settlement against time for a selected node.

- Select the node shown on the lower left of the model. (node 15 has been used here)



To plot a graph of settlement against time:

- Select the menu item **Utilities > Graph wizard**
- On the dialog select **Time history** and press **Next**
- Select **Named** and press **Next**
- Select **Response time** and press **Next**
- Select **Nodal** and press **Next**,
- Select entity **Stress – Two phase plane strain** of component **Displacement** and press **Next**.
- Finally enter an X and Y axis name of **Time** and **Displacement** respectively and press **OK**.



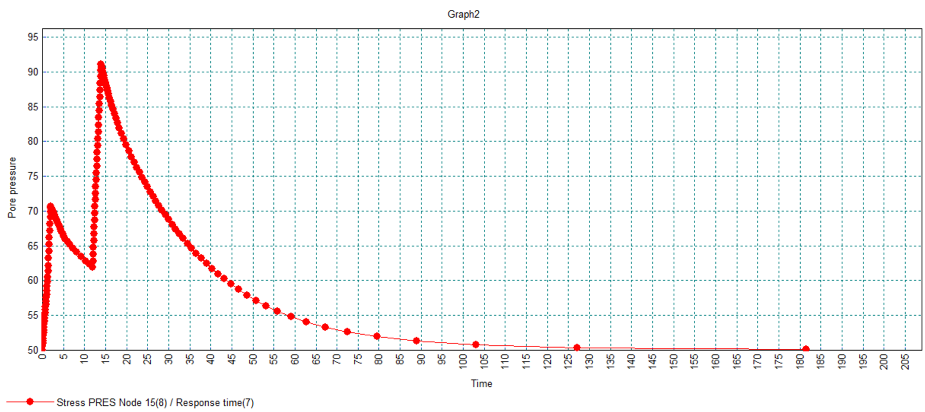
Settlement with time

This graph and its associated table data show the settlement for node 15 reaching 19 mm.

Pore pressure

To plot a graph of pore pressure against time (with the chosen node still selected).

- Select the menu item **Utilities > Graph wizard**
- On the dialog select **Time history** and press **Next**
- Select **Named** and press **Next**
- Select **Response time** and press **Next**
- Select **Nodal** and press **Next**
- Select entity **Stress – Two phase plane strain** of component **PRES** and press **Next**.
- Finally enter an X and Y axis name of **Time** and **Pore pressure** respectively and press **OK**.



Variation of pore pressure with time