SOILVISION 10 Help Manual - 12/16/2019

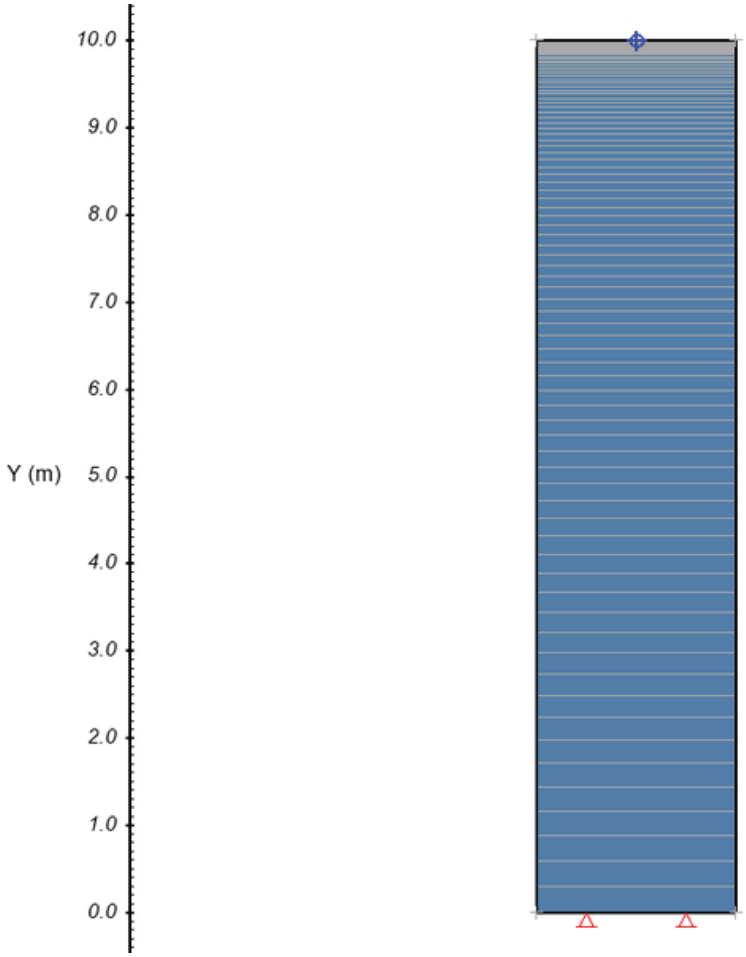
1D Consolidation - Instant Filling

This example introduces the coupling between SVSOLID and SVFLUX GT to solve large strain consolidation. The problem is 10 m instantaneously filled and the model is run for 30 years (10950 days). This example demonstrates the ability to simulate the large strain consolidation process of oil sands tailings and other soft soil whose settlements are large.

Project Name:	Consolidation
Model Name:	1D-ConsolidationInstantFilling

Minimum authorization required to complete this tutorial: 2D/3D SVSOLID Advanced, 2D SVFLUX

Model Description and Geometry



SOILVISION 10 Help Manual - 12/16/2019

Model Setup

In order to set up the model described in the preceding section, the following steps will be required. The steps fall under the general categories of:

a. Create model
b. Enter geometry
c. Stage Settings
d. Specify SVFLUX GT initial conditions
e. Specify SVFLUX GT boundary conditions
f. Apply SVFLUX GT material properties
g. Inputs for SVSOLID

h. Specify model output
i. Mesh settings
j. Analyze model
k. Results

a. Create Model

The following steps are required to create the model:

1. Open the SOILVISION Manager dialog ,
2. In EXPERT MODE, it is assumed that you are already familiar with creating Projects and Models in SVOFFICE. Select "MyProject" project or create a new project for this model. Press the *New* button under the Models heading,
3. Select the following:

Module:	**SVFLUX/SVSOLID - Consolidation**
System:	**1D Vertical**
Units:	**Metric**
Time Units:	**day**
Model Name:	**1DLSCINSTANT**

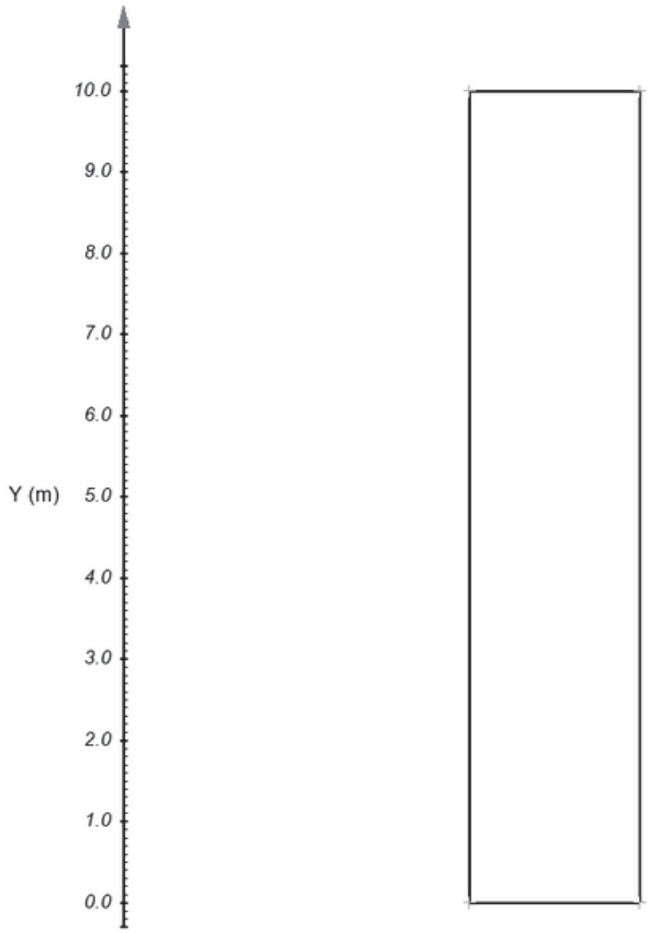
4. Click *OK* to close the dialog

b. Enter Geometry (Geometry)

The model being used has 1 single region with the following steps

1. Select *Geometry > 1D Thicknesses...*,
2. Enter 10 in the reference level box,
3. Enter 10 in thickness box,
4. Click *OK* to close the dialog.

Now your screen should look like the image below



c. Apply SVFLUX GT Material Properties (Materials)

The next step in defining the model is to enter the material properties for the SVFLUX GT materials used in the model.

Open the *Materials* dialog by selecting *Materials > Manager ...* from the menu,

1. Click the *New...* button to open the *New Materials* dialog,

2. Enter "Oil Sands" for the material name and *Saturated Consolidation* for *Category*,
3. Click ***OK***,
4. Click on the *HC Properties...* button to open the *Hydraulic Conductivity* dialog,
5. Select **Single Power Function Fit** in the *ksat Options* and click the *Data...* button to open *Ksat Vs. Void Ratio* dialog,
6. Copy the data from the table below and click Paste Points,
7. Click ***OK*** to close dialog
8. Press ***OK*** and ***OK*** to close *Hydraulic Conductivity and Materials Manager* dialogs respectively.

Void Ratio	Hydraulic Conductivity (m/day)
0.89	3.15e-06
1.04	1.32e-05
1.41	5.41e-05
2.53	7.30e-04
2.59	1.09e-03

Once the material has been entered and we must assign the material to the region using Stage Settings below.

I. Stage Settings (Geometry > Stage Settings)

1. Select *Geometry* > *Stage Settings* ,
2. Click the Add Stage button to add a new stage,
3. Enter the *Duration and Time Increments,*

   The final result should be the same as the below table.
4. Select the Region Stage Settings tab,
5. Select **Oil Sands** material for the region R1 for both stage 1 and 2,
6. Select Stage 1 and set the Region Name R1 as "Constructed" in the Action drop down
7. Click *Ok* to close Stage Settings dialog. (Click Yes to Time Update questions)

Stage Name	Duration	End Time	Initial Time Increment	Min. Time Increment	Max. Time Increment	Maximum Iteration	Body Load Coefficient	Include Displacement	Steady
Stage 1	1	1	1e-5	1e-5	0.001	6	0	checked	unchecked
Stage 2	10949	10950	10.95	1e-5	1095	6	0	checked	unchecked

J. Specify SVFLUX GT Initial Conditions (Initial Conditions)

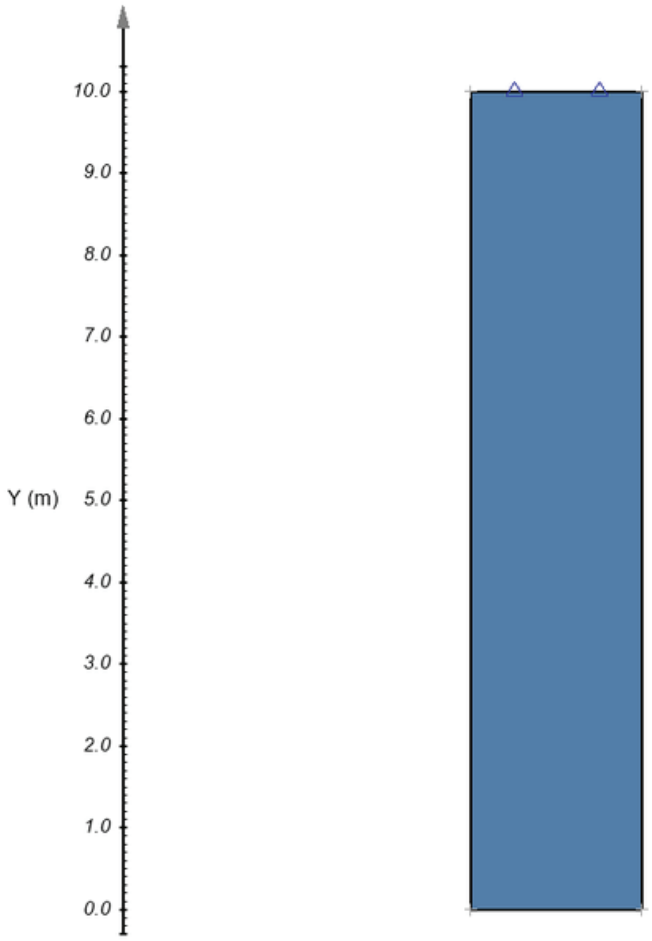
Initial conditions must be specified prior to solving a transient consolidation model. In this case we will specify a water table as an initial condition.

1. Select *Initial Conditions* > *Initial Head* ... from the menu,
2. Select the **h0 - Initial Head** in the *Variable* option and choose **Constant** in the *Type* option,
3. Enter **10m** in the head.

K. Specify SVFLUX GT Boundary Conditions (Boundaries)

Now that all of the regions and the model geometry have been successfully defined, the next step is to specify the boundary conditions

1. Select *Boundaries* > *Boundary Conditions* ... from the menu,
2. Select point Y (m) = 10 in the Boundary Conditions list,
3. Choose **Excess Pore Pressure > *Constant*** in the *Boundary Conditions* drop down,
4. Enter **0** in the *Constant* text box,
5. Change the *Boundary Name* to Surface,
6. Similarly, select point Y (m) = 0 m,
7. Choose **Zero Flux** as Boundary Condition,
8. Change *Boundary Name* to *Base,*
9. Select *OK* to close the dialog.



g. Inputs for SVSOLID

Similarly, initial conditions, boundary conditions and material properties are needed for SVSOLID

Click the *Model* > *SVSOLID* from the menu

Initial Conditions

1. Select *Initial Conditions* > *Initial Void Ratio* ... from the menu,
2. Chose **Initial Void Ratio** as *Variable,*
3. Enter **3.29** for *Constant Void Ratio,*
4. Click *Ok* to close dialog.

Boundary Conditions

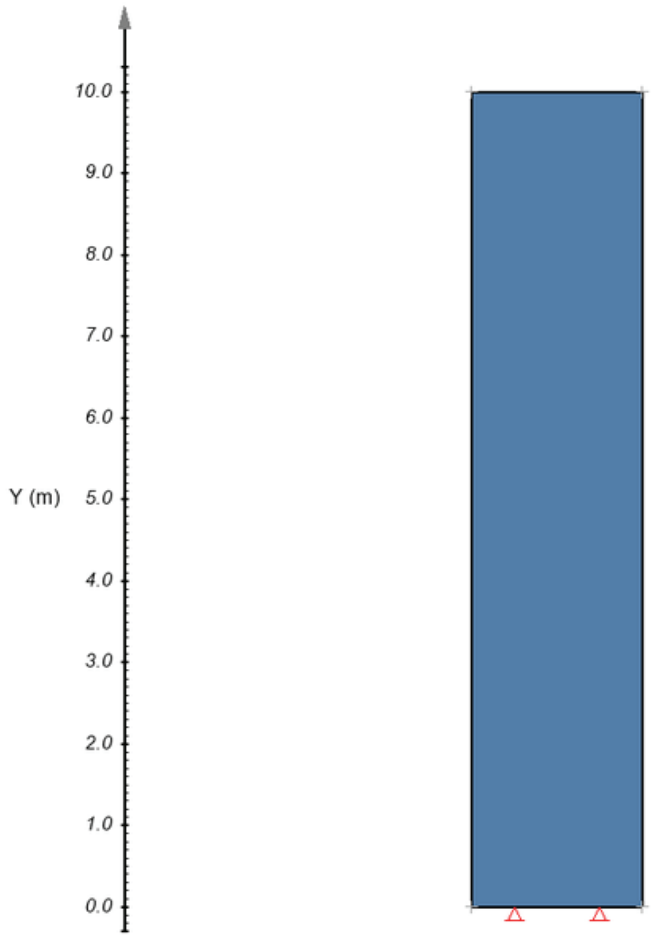
1. Select *Boundaries* > *Displacements*,
2. Select point Y = 10 and choose **Y Free** as Boundary Condition,
3. Select Y = 0 and choose **Y Fixed** as Boundary Condition,
4. Click *Ok* to close dialog.

Material Properties

1. Select *Materials* > *Material Manger* from the menu,
2. Select **Oil Sands** in the *Material Name* and click *Properties* button,
3. Enter **0.3** for Poisson's Ratio,
4. Click *Data* button to open Compression data dialog,
5. Copy the data in the table below and click **Paste** on the opened dialog,
6. Click *OK* to close dialog,
7. Click ***Apply Fit*** button to calculate A and B values,
8. Enter Specific Gravity value of **2.28** and Minimum Stress Limit value of **0.1** kPa,
9. Select *Loading* tab and enter **Ko = 0.6**,
10. Click *OK* to close the *Power Function* dialog,
11. Click *OK* to close the *Materials Manager* dialog.

Stress (kPa)	Void Ratio
1.1	2.53
2.2	2.27

8.7	1.85
68.8	1.25
274.3	0.89



h. Specify Model Output (Results)

A number of relevant output plots will be generated by default for both SVFLUX GT and SVSOLID. For instructions on customizing the output plots see the User Manual or other Tutorial examples.

SVFLUX GT

1. Select *Model* > *SVFLUX* ...,
2. Select *Results* > *Graph Manager* ...,
3. On the Range tab, click ***Add Defaults***,
4. Select all the elevation entries and click the ***Multiple Update*** button,
5. In the Update Method tab, change time ***Increment*** to **365** days and click ***Ok*** to close the dialog,
6. Click on Flux Sections tab and select both flux sections present,
7. Click on ***Multiple Update*** button,
8. On the update method tab, change time ***Increment*** to **365** days and click ***Ok*** to close the dialog,
9. Click *Ok* to close Graph manager.

SVSOLID

1. Select *Model* > *SVSOLID* ...,
2. Select *Results* > *Graph Manager* ...,
3. On the Range tab, click ***Add Defaults***,
4. Select all the range data and click on Multiple Update,
5. In the Update Method tab, change time ***Increment*** to **365** days and click *Ok* to close the dialog,
6. Click the Ground Surface tab,
7. Click *Add New Graph button* at the bottom left corner of the dialog,
8. Select **Ym** (y deformed coordinate*)* under the variable drop list,
9. Select update Method tab and change time ***increment*** to **365** days.
10. Click *Ok* and Click *Ok* to close Graph Manager dialog.

i. Mesh Settings (Mesh > Settings)

The next step is to change the mesh settings.

1. Select *Mesh* > *Settings...* from the menu,
2. In the Global tab enter the mesh data as shown in table below,
3. Press *OK* to close the *Meshing Settings* dialog, and accept the Mesh Reset message.

Global Meshing Settings Option	Total Nodes
Total Nodes	100
Mesh Layout	Denser at the top

j. Analyze model (Solve > Analyze)

The current model is run by selecting the *Solve* > *Analyze* menu option.

k. Results (Solve > Results)

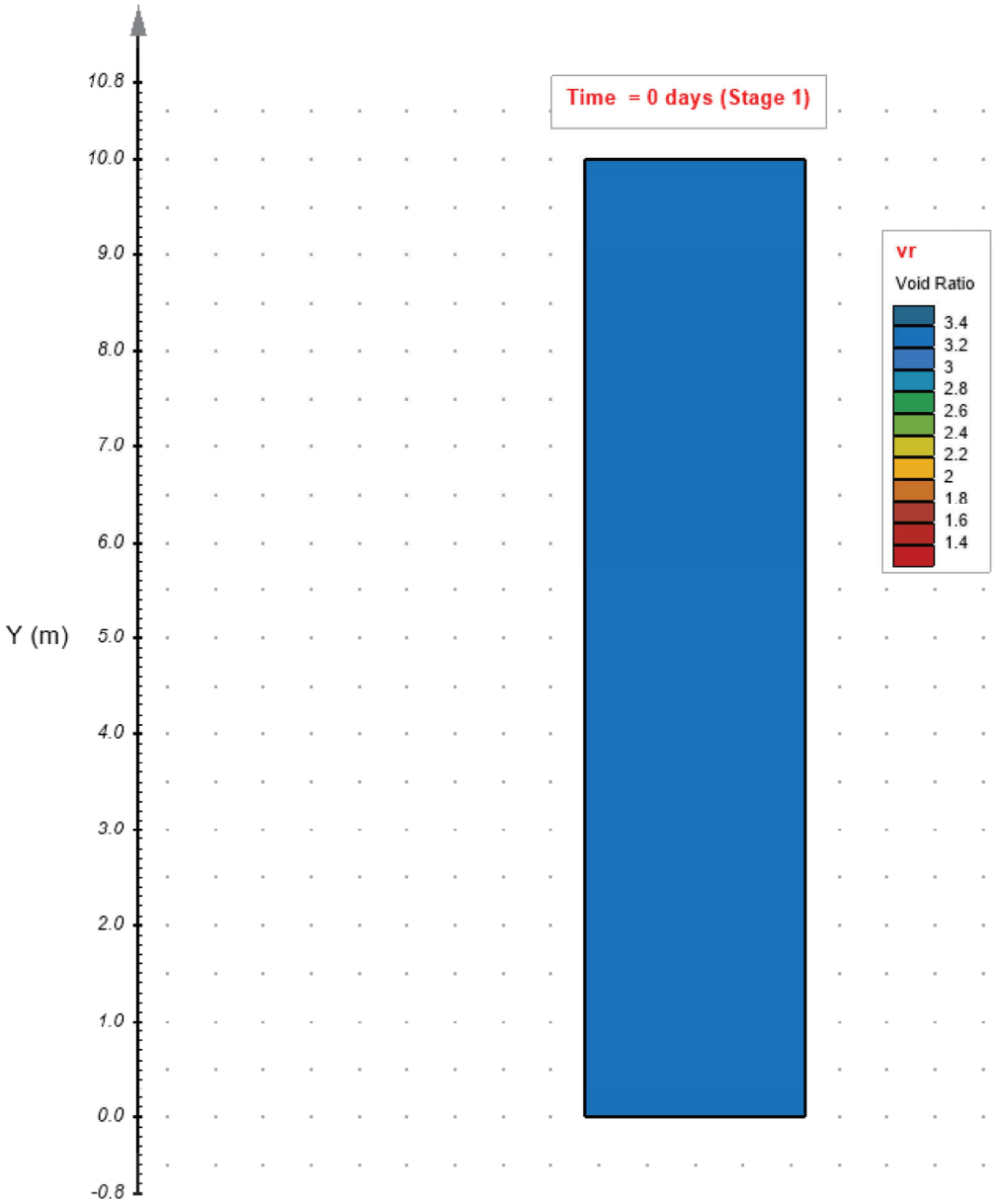
Upon completion of the solver, the visual results for the current model may be examined by selecting the *Solve* > *Results* menu option.

SOILVISION 10 Help Manual - 12/16/2019

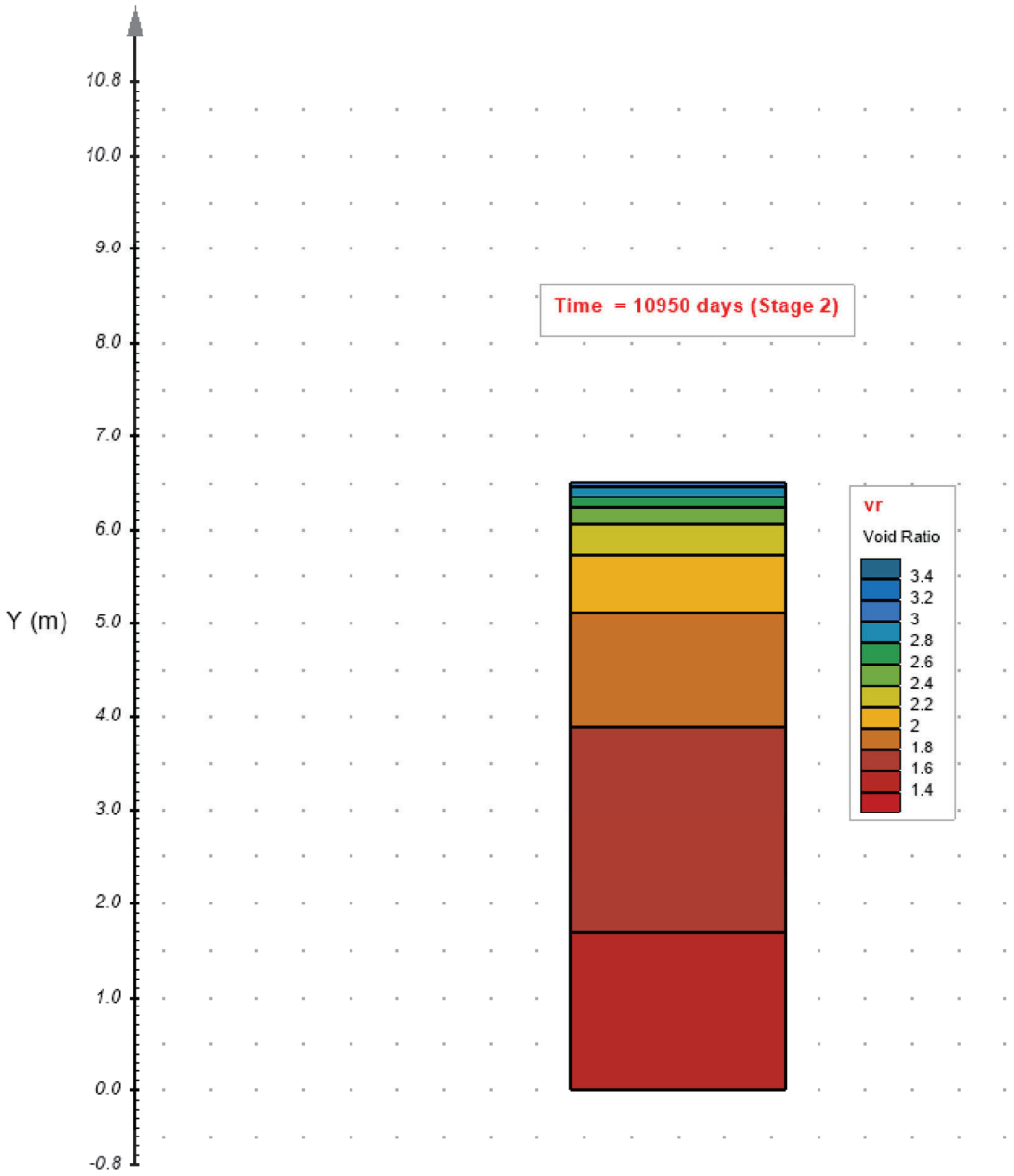
Results and Discussion

After the computation for the model has been completed, the below figures show the results at 3650 days (10 years), 7300 (20 years) and 10950 days (30 years)

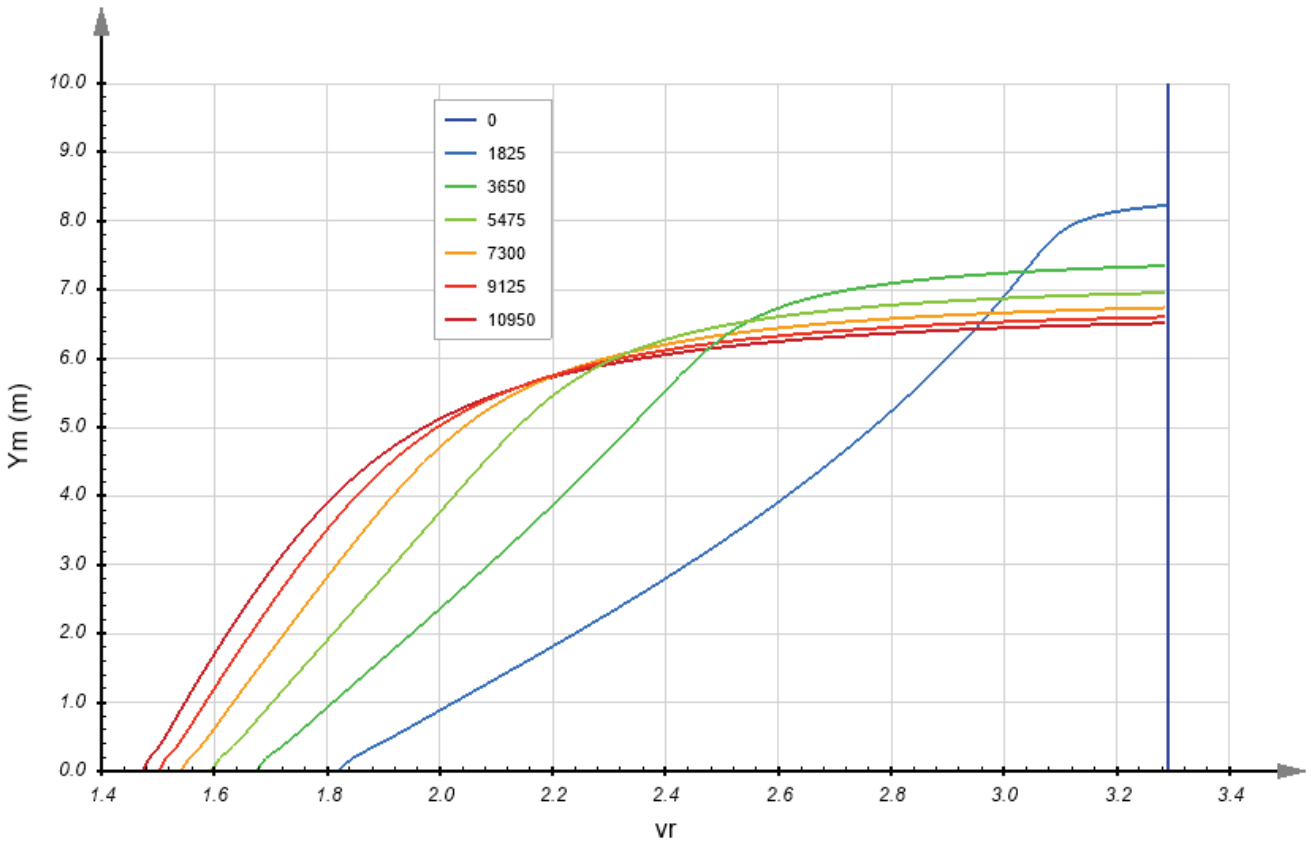
- **Initial thickness**



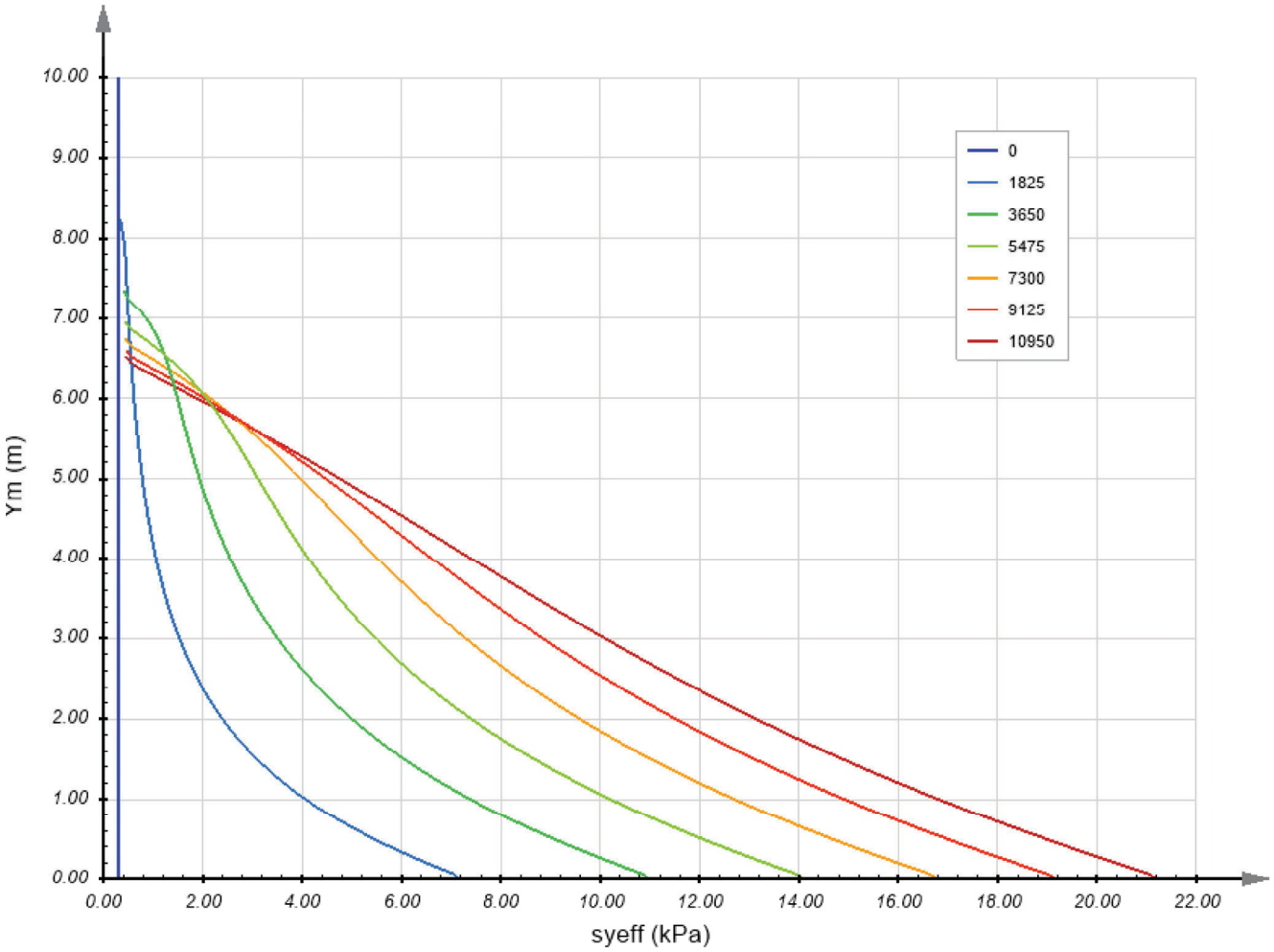
- **Final thickness after 30 years (10950 days)**


Time = 10950 days (Stage 2)
vr
Void Ratio
3.4
3.2
3
2.8
2.6
2.4
2.2
2
1.8
1.6
1.4
Y (m)
10.8
10.0
9.0
8.0
7.0
6.0
5.0
4.0
3.0
2.0
1.0
0.0
-0.8

- **Void Ratio**



- **Effective stress**



- **Output of Settlement**