Slope Stability in an Embankment Dam

Introduction

Slope stability analysis is an essential technique for predicting the settlement, deformation, and slippage of soil due to various loading and environmental conditions. In an embankment dam, slope stability analysis is important to determine the safety of the dam.

The present model is inspired by an example included in Ref. 1. The pore pressure in the soil is modeled by Darcy's law, while the Mohr-Coulomb criterion is used for the elastoplastic analysis.

The technique used for studying the slope stability is called the *strength reduction* method, where the Mohr-Coulomb material parameters are functions of the Factor of Safety (FOS). Decreasing the material parameters results in reduction of the shear strength of the soil, which eventually becomes unstable. This phenomena can produce a collapse of the embankment for a certain combination of loads. More details of this technique are given in Ref. 1.

Model Definition

Figure 1 shows the cross section of the embankment dam. The lengths L_1, L_2, and L_3 are 24 m, 5 m, and 24 m, respectively, and the height of the embankment L_4 is 12 m. The water level is 10 m and the possible seepage height is 4 m. The total width of the embankment is $L_1 + L_2 + L_3$.

.

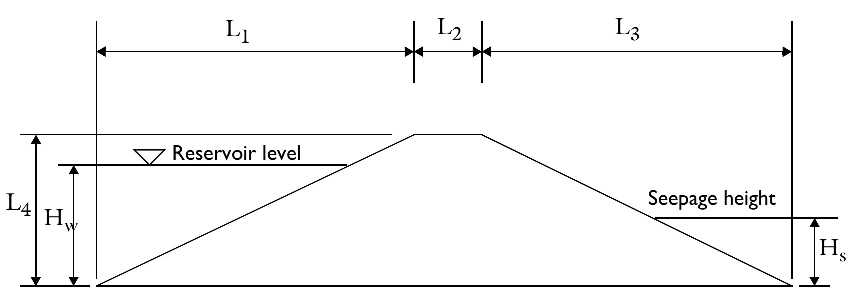

Figure 1: The illustration of the embankment dam.

In this example, a plane strain approximation is used to model the embankment dam in 2D. The effects of gravity and hydrostatic pressure are also included. The material properties for the Mohr–Coulomb model are parameterized with respect to a factor of safety parameter, FOS. A parametric study increases the FOS parameter, thereby reducing the strength of the soil with every parameter step. The model does not converge for parameter values above 1.915, which indicates a collapse of the slope.

The Mohr-Coulomb yield function and associated plastic potential is

$$F = Q = m\sqrt{J_2} + \alpha I_1 - k \tag{1}$$

with

$$\alpha = \frac{\sin\Phi}{3}, k = C\cos\Phi \tag{2}$$

The parameterized cohesion C and angle of internal friction Φ are given as

$$C = \frac{c}{\text{FOS}}, \Phi = \text{atan}\left(\frac{\tan\phi_u}{\text{FOS}}\right)(p < 0) + \text{atan}\left(\frac{\tan\phi_s}{\text{FOS}}\right)(p \geq 0) \tag{3}$$

where c is the material cohesion, ϕ_u and ϕ_s are angles of internal friction for unsaturated and saturated soils, and p is the pore pressure given by Darcy's law.

Results and Discussion

The pressure head in the embankment dam is shown in Figure 2; it varies from 0 m to 10 m on the submerged wall, while it is 0 m on the seepage face. Positive pressure head means positive pore pressure, which indicates a saturated soil, while unsaturated soil is represented with zero pressure head. The zero pressure head line in the figure is the location of a phreatic surface that divides saturated from the unsaturated soil.

The elastoplastic analysis does not converge for FOS values greater than 1.915; hence, the simulation is performed until its value becomes 1.915, which is the value at which the slope collapses due to increase in plastic strains and subsequent reduction of shear strength.

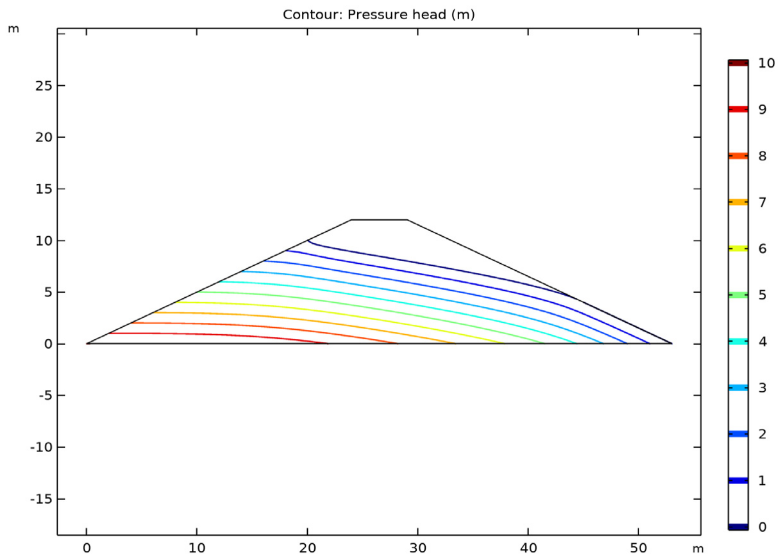

Figure 2: Pressure head in the embankment dam. The zero pressure head line shows the location of phreatic surface.

The equivalent plastic strain just before the collapse shows a different pattern, which gives an indication of the failure mechanism (see Figure 3).

The slip surface is shown in Figure 4. The arrows show the direction of displacement for the soil particles. This figure illustrates the phenomenon of soil slippage. The soil near the lower-right corner does not slip because of the fixed constraint on the lower boundary. The slip surface figure matches qualitatively well with the results given in Ref. 1.

A 3D visualization of displacement is shown in Figure 5 with the help of an extrusion dataset. The results indicate that a 2D analysis of an embankment dam is an efficient way to predict soil instability with plane strain approximation, and 3D visualization is possible with help of postprocessing tools. This approach avoids solving a bigger numerical problem in 3D.

The plot of the maximum displacement versus FOS is shown in Figure 6. The maximum displacement increases significantly at around FOS = 1.9, which indicates the onset of the collapse of the slope.

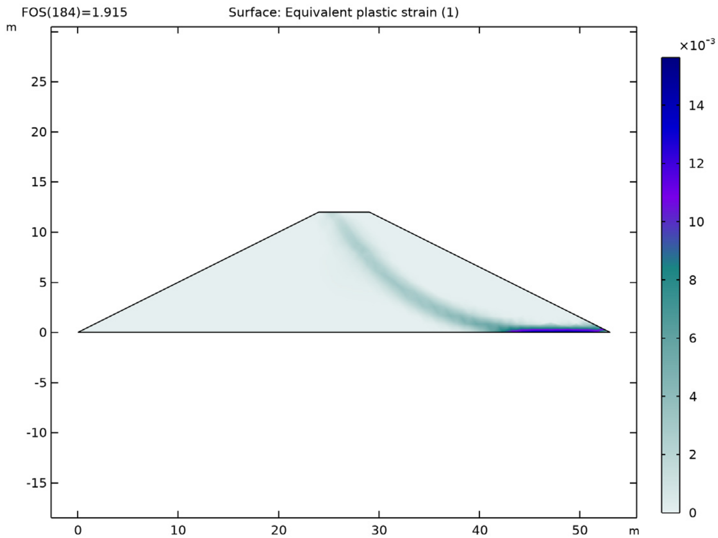

Figure 3: Equivalent plastic strain just before the collapse of the slope.

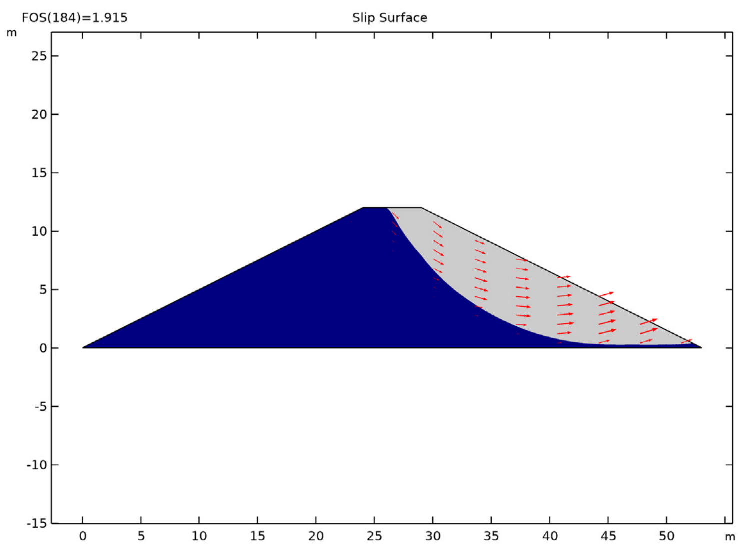

Figure 4: Slip circle just before the collapse of the slope.

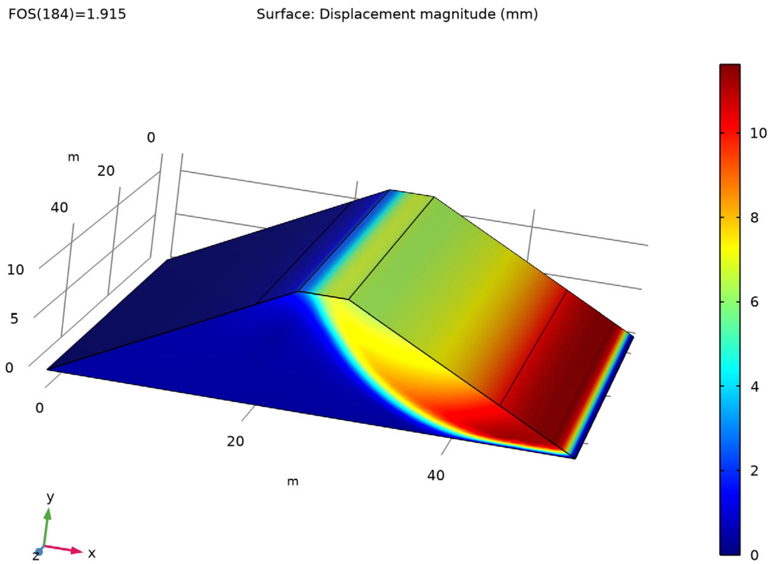

Figure 5: Displacement magnitude of the embankment dam just before the collapse of the slope.

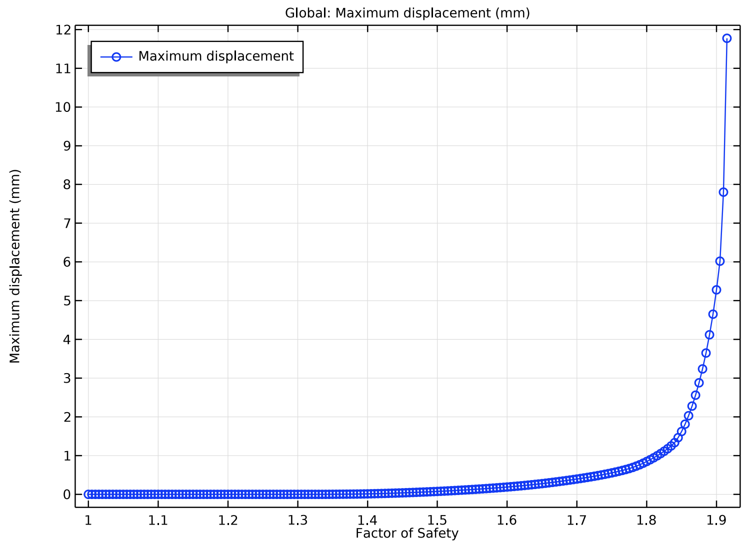

Figure 6: Maximum displacement versus FOS.

Notes About the COMSOL Implementation

Three stationary studies are created in order to account for the effects of pore pressure and gravity loads in the stability of the slope. In the first study, only *Darcy's Law* is computed to get the pore pressure profile. In a second study, the embankment is simulated with this hydrostatic pressure and gravity. In the third study step the pore pressure generated in the first study, and the initial stresses generated by gravity loads in the second study are taken into account by adding a **Initial Stress and Strain** node. The Mohr-Coulomb criterion is added in the third study to study the elastoplastic failure of the soil due to the combined effect of gravity and variable pore pressure.

The angle of internal friction is different for saturated and unsaturated soils, hence a combination based on the pore pressure is used. No external stresses are applied in regions of unsaturated soil, since pores are considered interconnected and at constant atmospheric pressure.

An additional extrusion dataset is created to generate a 3D plot from the 2D dataset, and the 3D view is adjusted in order to properly visualize it. As stated in Ref. 1, the non-convergence of the simulation is considered as an indicator of slope failure.

References

1. D.V.Griffiths and P.A.Lane, "Slope Stability Analysis by Finite Elements," *Geotechnique*, vol. 49, no. 3, pp. 387–403, 1999.

Application Library path: `Geomechanics_Module/Tutorials/slope_stability`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D**.

2 In the **Select Physics** tree, select **Fluid Flow>Porous Media and Subsurface Flow>Darcy's Law (dl)**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.

5 Click **Add**.

6 Click **Study**.

7 In the **Select Study** tree, select **General Studies>Stationary**.

8 Click **Done**.

GEOMETRY 1

Model parameters and interpolation function data are available in the appended text files.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `slope_stability_parameters.txt`.

Interpolation 1 (int1)

1 In the **Home** toolbar, click f(X) **Functions** and choose **Global>Interpolation**.

2 In the **Settings** window for **Interpolation**, locate the **Definition** section.

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `slope_stability_interpolation.txt`.

5 In the **Function name** text field, type `cond`.

6 Locate the **Units** section. In the **Arguments** text field, type `m`.

7 In the **Function** text field, type `m/s`.

GEOMETRY 1

Construct the 2D geometry using a polygon.

Polygon 1 (pol1)

1 In the **Geometry** toolbar, click **Polygon**.

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 From the **Data source** list, choose **Vectors**.

4 In the **x** text field, type `0, L1, L1+L2, L1+L2+L3`.

5 In the **y** text field, type `0, L4, L4, 0`.

Add points at the reservoir level and the possible seepage level to partition the sides of the dam.

Point 1 (pt1)

1 In the **Geometry** toolbar, click **Point**.

2 In the **Settings** window for **Point**, locate the **Point** section.

3 In the **x** text field, type `Hw*L1/L4`.

4 In the **y** text field, type `Hw`.

Point 2 (pt2)

1 Right-click **Point 1 (pt1)** and choose **Duplicate**.

2 In the **Settings** window for **Point**, locate the **Point** section.

3 In the **x** text field, type `L1+L2+L3-Hs*L1/L4`.

4 In the **y** text field, type `Hs`.

5 Click **Build Selected**.

DEFINITIONS

Variables 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions** node.

2 Right-click **Definitions** and choose **Variables**.

3 In the **Settings** window for **Variables**, locate the **Variables** section.

4 In the table, enter the following settings:

Name	Expression	Unit	Description
Saturated	dl.Hp>=0		Boolean variable for saturated region
Unsaturated	dl.Hp<0		Boolean variable for unsaturated region
K	cond(dl.Hp)	m/s	Hydraulic conductivity
C	c/FOS	Pa	Parameterized cohesion
PHI	atan(tan(phi_un)/FOS)*Unsaturated+atan(tan(phi_sat)/FOS)*Saturated	rad	Parameterized friction angle

Maximum 1 (maxop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Maximum**.

2 Select Domain 1 only.

Add two blank materials, one for the soil and one for the water, then rename them accordingly. For the water material, keep the domain selection empty.

MATERIALS

Soil

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Soil` in the **Label** text field.

Water

1 Right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Water` in the **Label** text field.

DARCY'S LAW (DL)

Fluid and Matrix Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Darcy's Law (dl)** click **Fluid and Matrix Properties 1**.

2 In the **Settings** window for **Fluid and Matrix Properties**, locate the **Matrix Properties** section.

3 From the **Permeability model** list, choose **Hydraulic conductivity**.

4 In the K text field, type K.

5 Locate the **Fluid Properties** section. From the **Fluid material** list, choose **Water (mat2)**.

6 Locate the **Matrix Properties** section. From the **Porous material** list, choose **Soil (mat1)**.

MATERIALS

Soil (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Soil (mat1)**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Porosity	epsilon	psi	1	Basic
Young's modulus	E	E_soil	Pa	Basic
Poisson's ratio	nu	nu_soil	1	Basic
Density	rho	rho_soil	kg/m³	Basic

Water (mat2)

1 In the **Model Builder** window, click **Water (mat2)**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	rho_wat	kg/m³	Basic

DARCY'S LAW (DL)

Add a pressure head to the submerged parts of the downstream and upstream sides of the dam.

Pressure Head 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Pressure Head**.

2 Select Boundary 1 only.

3 In the **Settings** window for **Pressure Head**, locate the **Pressure Head** section.

4 In the H_{p0} text field, type `Hw-y`.

Pressure Head 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Pressure Head**.

2 Select Boundary 6 only.

3 In the **Model Builder** window, click **Darcy's Law (dl)**.

4 In the **Settings** window for **Darcy's Law**, locate the **Gravity Effects** section.

5 Select the **Include gravity** check box.

Gravity 1

1 In the **Model Builder** window, click **Gravity 1**.

2 In the **Settings** window for **Gravity**, locate the **Gravity** section.

3 From the **Specify** list, choose **Elevation**.

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 From the **Element size** list, choose **Extra fine**.

4 Click **Build All**.

STUDY 1-DARCY'S LAW

Disable the default plots for this study.

1 In the **Model Builder** window, click **Study 1**.

2 In the **Settings** window for **Study**, type `Study 1-Darcy's Law` in the **Label** text field.

3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

Disable the **Solid Mechanics** physics from the study to solve only for the pressure variable.

Step 1: Stationary

1 In the **Model Builder** window, under **Study 1-Darcy's Law** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for **Solid Mechanics (solid)**.

4 In the **Home** toolbar, click **Compute**.

Add a contour plot of the pressure head.

RESULTS

Pressure Head

1 In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Pressure Head` in the **Label** text field.

Contour 1

1 Right-click **Pressure Head** and choose **Contour**.

2 In the **Settings** window for **Contour**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Darcy's Law> dl.Hp - Pressure head - m**.

3 Locate the **Levels** section. From the **Entry method** list, choose **Levels**.

4 In the **Levels** text field, type `range(0,1,10)`.

5 Locate the **Coloring and Style** section. From the **Contour type** list, choose **Tube**.

6 Select the **Radius scale factor** check box.

7 In the associated text field, type `0.07`.

8 In the **Pressure Head** toolbar, click **Plot**.

SOLID MECHANICS (SOLID)

Add the water pressure as a boundary load on the downstream side of the dam.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

Boundary Load 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.

2 Select Boundary 1 only.

3 In the **Settings** window for **Boundary Load**, locate the **Force** section.

4 From the **Load type** list, choose **Pressure**.

5 In the p text field, type `p`.

Fixed Constraint 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.

2 Select Boundary 2 only.

Gravity 1

1 In the **Physics** toolbar, click **Domains** and choose **Gravity**.

2 Select Domain 1 only.

Linear Elastic Material 1

In the **Model Builder** window, click **Linear Elastic Material 1**.

External Stress 1

1 In the **Physics** toolbar, click **Attributes** and choose **External Stress**.

2 In the **Settings** window for **External Stress**, locate the **External Stress** section.

3 From the **Stress input** list, choose **Pore pressure**.

4 In the p_A text field, type `p*Saturated`.

5 From the α_B list, choose **User defined**. In the associated text field, type `1`.

6 In the p_{ref} text field, type `0`.

Add another study to get the in situ stresses generated by gravity and pore pressure. Disable the default plots for this study.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Stationary**.

4 Click **Add Study** in the window toolbar.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2 - SOLID MECHANICS (IN SITU STRESS INITIALIZATION)

1 In the **Model Builder** window, click **Study 2**.

2 In the **Settings** window for **Study**, type `Study 2 - Solid Mechanics (In Situ Stress Initialization)` in the **Label** text field.

3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

4 In the **Home** toolbar, click **Compute**.

SOLID MECHANICS (SOLID)

Linear Elastic Material 1

In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Linear Elastic Material 1**.

Soil Plasticity 1

1 In the **Physics** toolbar, click **Attributes** and choose **Soil Plasticity**.

2 In the **Settings** window for **Soil Plasticity**, locate the **Soil Plasticity** section.

3 From the **Yield criterion** list, choose **Mohr-Coulomb**.

4 From the **Plastic potential** list, choose **Associated**.

Linear Elastic Material 1

In the **Model Builder** window, click **Linear Elastic Material 1**.

Initial Stress and Strain 1

1 In the **Physics** toolbar, click **Attributes** and choose **Initial Stress and Strain**.

Add the stresses computed in the second study step as initial stresses. You can access these stresses by using the `withsol` operator as follows:

2 In the **Settings** window for **Initial Stress and Strain**, locate the **Initial Stress and Strain** section.

3 In the S_0 table, enter the following settings:

withsol('sol2', solid.sx)	withsol('sol2', solid.sxy)	withsol('sol2', solid.sxz)
withsol('sol2',solid.sxy)	withsol('sol2', solid.sy)	withsol('sol2', solid.syz)
withsol('sol2',solid.sxz)	withsol('sol2',solid.syz)	withsol('sol2', solid.sz)

MATERIALS

Soil (mat1)

Add material properties to the **Mohr-Coulomb** model. These are functions of the factor of safety from the parameter list.

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Soil (mat1)**.

2 In the **Settings** window for **Material**, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Cohesion	cohesion	`C`	Pa	Mohr-Coulomb
Angle of internal friction	internalphi	`PHI`	rad	Mohr-Coulomb
Porosity	epsilon	`psi`	1	Basic
Young's modulus	E	`E_soil`	Pa	Basic
Poisson's ratio	nu	`nu_soil`	1	Basic
Density	rho	`rho_soil`	kg/m³	Basic

Disable the **Soil Plasticity** and the **Initial Stress and Strain** nodes from the second study step. Add a third study for the elastoplastic analysis.

STUDY 2 - SOLID MECHANICS (IN SITU STRESS INITIALIZATION)

Step 1: Stationary

1 In the **Model Builder** window, under **Study 2 - Solid Mechanics (In Situ Stress Initialization)** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** check box.

4 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Darcy's Law (dl)**.

5 Click **Disable in Solvers**.

6 In the **Physics and variables selection** tree, select **Component 1 (comp1)> Solid Mechanics (solid)>Linear Elastic Material 1>Soil Plasticity 1** and **Component 1 (comp1)>Solid Mechanics (solid)>Linear Elastic Material 1> Initial Stress and Strain 1**.

7 Click **Disable**.

8 Click to expand the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

9 From the **Method** list, choose **Solution**.

10 From the **Study** list, choose **Study 1-Darcy's Law, Stationary**.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Stationary**.

4 Click **Add Study** in the window toolbar.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 3 - SOLID MECHANICS (FACTOR OF SAFETY)

Disable the default plots for this study.

1 In the **Model Builder** window, click **Study 3**.

2 In the **Settings** window for **Study**, type `Study 3 - Solid Mechanics (Factor of Safety)` in the **Label** text field.

3 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

Step 1: Stationary

Disable the **Darcy's Law** physics in this study.

1 In the **Model Builder** window, click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 In the table, clear the **Solve for** check box for **Darcy's Law (dl)**.

The pressure variable is not solved for in this study step; instead, its value is taken from the first solution. Create an auxiliary sweep over the parameter `FOS`.

4 Locate the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

5 From the **Method** list, choose **Solution**.

6 From the **Study** list, choose **Study 1-Darcy's Law, Stationary**.

7 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** check box.

8 Click **Add**.

9 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
FOS (Factor of Safety)	`range(1,0.005,1.915)`	

10 In the **Home** toolbar, click **Compute**.

RESULTS

Slip Surface

1 In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Slip Surface` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 3 - Solid Mechanics (Factor of Safety)/Solution 3 (sol3)**.

4 Click to expand the **Title** section. From the **Title type** list, choose **Label**.

To show the slip surface, customize the settings for the Contour plot.

Contour 1

1 Right-click **Slip Surface** and choose **Contour**.

2 In the **Settings** window for **Contour**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Displacement>solid.disp - Displacement magnitude - m**.

3 Locate the **Levels** section. In the **Total levels** text field, type `1`.

4 Locate the **Coloring and Style** section. From the **Contour type** list, choose **Filled**.

5 Clear the **Color legend** check box.

6 From the **Color table** list, choose **Wave**.

Arrow Surface 1

1 In the **Model Builder** window, right-click **Slip Surface** and choose **Arrow Surface**.

2 In the **Settings** window for **Arrow Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Displacement>u,v - Displacement field**.

3 In the **Slip Surface** toolbar, click **Plot**.

Equivalent Plastic Strain

1 In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Equivalent Plastic Strain` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 3 - Solid Mechanics (Factor of Safety)/Solution 3 (sol3)**.

Surface 1

1 Right-click **Equivalent Plastic Strain** and choose **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Strain>solid.epe - Equivalent plastic strain**.

3 Locate the **Coloring and Style** section. From the **Color table** list, choose **AuroraAustralisDark**.

4 In the **Equivalent Plastic Strain** toolbar, click **Plot**.

Set up a 1D plot in order to visualize the maximum displacement in the domain.

Factor of Safety

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Factor of Safety` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 3 - Solid Mechanics (Factor of Safety)/Solution 3 (sol3)**.

Global 1

1 Right-click **Factor of Safety** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
maxop1(solid.disp)	mm	Maximum displacement

4 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

5 Click **Replace Expression** in the upper-right corner of the **x-Axis Data** section. From the menu, choose **Global definitions>Parameters>FOS - Factor of Safety**.

6 Click to expand the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.

7 From the **Positioning** list, choose **In data points**.

Factor of Safety

1 In the **Model Builder** window, click **Factor of Safety**.

2 In the **Settings** window for **1D Plot Group**, locate the **Legend** section.

3 From the **Position** list, choose **Upper left**.

4 In the **Factor of Safety** toolbar, click **Plot**.

Create an Extrusion dataset to use for visualizing the displacement field in 3D.

Extrusion 2D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Extrusion 2D**.

2 In the **Settings** window for **Extrusion 2D**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 3 - Solid Mechanics (Factor of Safety)/ Solution 3 (sol3)**.

4 Locate the **Extrusion** section. In the **z maximum** text field, type `L1+L2+L3`.

5 In the **z variable** text field, type `Z`.

Displacement

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Displacement` in the **Label** text field.

Surface 1

1 Right-click **Displacement** and choose **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics> Displacement>solid.disp - Displacement magnitude - m**.

3 Locate the **Expression** section. From the **Unit** list, choose **mm**.

4 In the **Displacement** toolbar, click **Plot**.

5 In the **Model Builder** window, expand the **Results>Views** node.

Camera

In order to better visualize the 3D plot, change the 3D view so that the thickness direction points out-of-plane.

1 In the **Model Builder** window, expand the **Results>Views>View 3D 2** node, then click **Camera**.

2 In the **Settings** window for **Camera**, locate the **Position** section.

3 In the **x** text field, type `83`.

4 In the **y** text field, type `100`.

5 In the **z** text field, type `410`.

6 Click **Update**.

7 Locate the **Up Vector** section. In the **x** text field, type `-0.15`.

8 In the **y** text field, type `0.9`.

9 In the **z** text field, type `-0.20`.

10 Click **Update**.

11 Click the **Zoom Extents** button in the **Graphics** toolbar.