Model created in COMSOL Multiphysics 6.4

Strength Reduction Method for Slope Stability

Introduction

The strength reduction method in combination with finite element analysis is a tool to find the factor of safety (FOS) in geomechanics, particularly for the stability of slopes and embankments. In the strength reduction method, the characteristic material properties are gradually reduced until failure occurs. Although the definition of an FOS depends on the context, in geotechnical problems it is defined with respect to the strength parameters of the soil, as discussed in Ref. 1.

The strength reduction method is applicable to linear failure criteria, like the Mohr–Coulomb criterion. When the Mohr–Coulomb criterion is used with the strength reduction method, the cohesion, the angle of internal friction, and the dilatation angle are simultaneously reduced until mechanical equilibrium is lost. Decreasing the material parameters results in a reduction of the shear strength of the soil, which eventually becomes unstable. This phenomena produces a collapse of the slope for a certain combination of loads, material parameters, and boundary conditions. The ratio between the initial cohesion and the cohesion at failure gives the FOS. More details of the method are given in Ref. 1 and Ref. 2.

The geometry, boundary conditions, loading conditions, and material parameters in this example are the same as discussed in Ref. 1. A similar example model can be found in *Slope Stability in an Embankment Dam*.

Model Definition

Figure 1 shows a cross section of the soil embankment. The lengths L_1 and L_2 are 85 m, and 20 m, respectively, and the heights of the embankment H_1 and H_2 are 20 m and 10 m, respectively. The slope angle α varies from 15° to 45°.

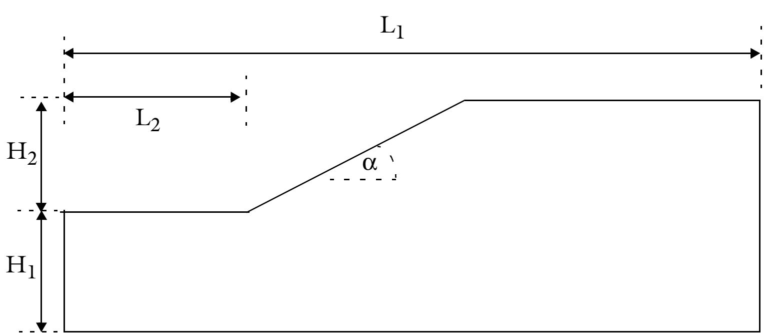

Figure 1: Geometry of the cross-section of an embankment.

The material properties for both associative and nonassociative plasticity are summarized in Table 1, and taken from Ref. 1.

TABLE 1: MATERIAL PROPERTIES.

Property	**Material case 1**	**Material case 2**
E	20 MPa	20 MPa
ν	0.3	0.3
c	20 kPa	20 kPa
ϕ	25°	25°
ψ	0°	25°
ρ	1940 kg/m^3	1940 kg/m^3

A plane strain approximation is used to model the soil embankment in 2D. The effect of gravity is 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 actual factor of safety is the value of the FOS parameter at which the model no longer converges, which is an indication of the collapse of the slope.

The Mohr–Coulomb yield function F and plastic potential Q are

$$F = m\sqrt{J_2} + \frac{\sin\Phi}{3}I_1 - C\cos\Phi \tag{1}$$

$$Q = m_q\sqrt{J_2} + \frac{\sin\Psi}{3}I_1 - C\cos\Psi \tag{2}$$

where I_1 is the first stress invariant and J_2 is the second deviatoric stress invariant. The parameterized cohesion C, parameterized angle of internal friction Φ, and parameterized dilatation angle Φ are given in terms of the FOS,

$$C = \frac{c}{\mathrm{FOS}},\ \Phi = \mathrm{atan}\left(\frac{\tan\phi}{\mathrm{FOS}}\right),\ \Psi = \mathrm{atan}\left(\frac{\tan\psi}{\mathrm{FOS}}\right) \tag{3}$$

where c is the cohesion, ϕ is the angle of internal friction, and ψ is the dilatation angle. Note that c, ϕ, and ψ are initial, unreduced material parameters. For the associative flow rule, $\phi = \psi$.

For the nonassociative flow rule, ψ is kept constant as long as it is smaller than Φ, see Ref. 2 for details. However, in this example, ψ is zero when using the nonassociative flow rule, so no special treatment is needed, and Equation 3 is applicable to the associative as well as the nonassociative flow rule.

For the nonassociative flow rule, the strength reduction method might trigger numerical instabilities, which in turn can result in a nonunique failure surface and corresponding FOS. To avoid potential instabilities and convergence issues, the *Davis procedure B* approach, as suggested in Ref. 1 and Ref. 2, is used. In this approach, the associative flow rule is applied with reduced values of the cohesion and the angle of internal friction to capture the effects of the nonassociative flow rule. The reduced cohesion c' and the reduced angle of internal friction ϕ' are given by

$$c' = \beta c, \phi' = \operatorname{atan}(\beta \tan\phi) \tag{4}$$

where the *reduction factor* β is

$$\beta = \frac{\cos\left(\operatorname{atan}\left(\frac{\tan\phi}{\mathrm{FOS}}\right)\right)\cos\left(\operatorname{atan}\left(\frac{\tan\psi}{\mathrm{FOS}}\right)\right)}{1-\sin\left(\operatorname{atan}\left(\frac{\tan\phi}{\mathrm{FOS}}\right)\right)\sin\left(\operatorname{atan}\left(\frac{\tan\psi}{\mathrm{FOS}}\right)\right)}$$

Hence, Equation 3 is rewritten in terms of the reduced cohesion c' and the reduced angle of internal friction ϕ' for the associative and nonassociative flow rules, as the reduction factor β is unity for the associative flow rule.

Results and Discussion

The factor of safety (FOS) for different slope angles is shown in Figure 2. The FOS decreases as the slope angle increases, which is expected. The FOS for the same slope inclination with the nonassociative flow rule (material case 1) is always smaller than for the associative flow rule (material case 2), but the influence of the nonassociative flow rule is marginal for the material parameters chosen. The results presented in Figure 2 are in good agreement with the results presented in Figure 4 of Ref. 1. As a verification that we have reached failure, the maximum displacement of the soil embankment is shown in Figure 3 as a function of the FOS for all analyzed cases.

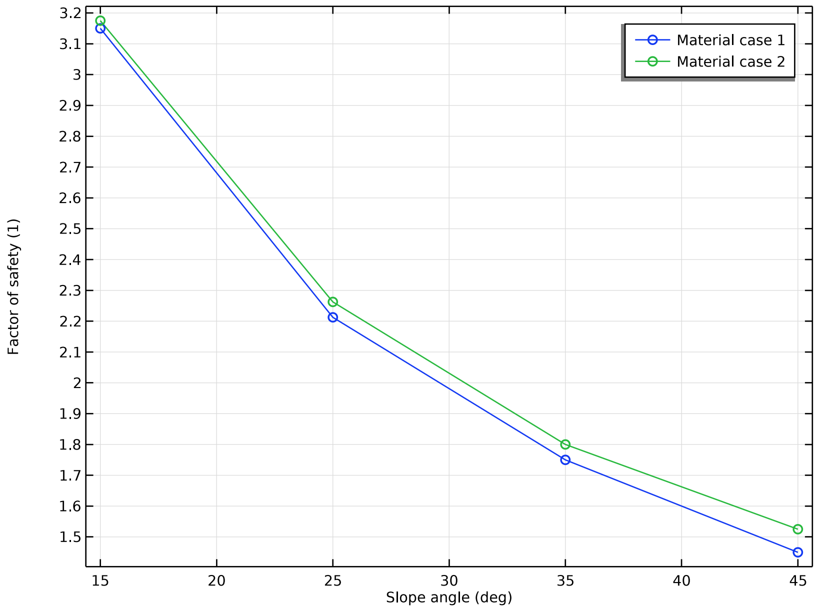

Figure 2: Factor of safety versus slope angle.

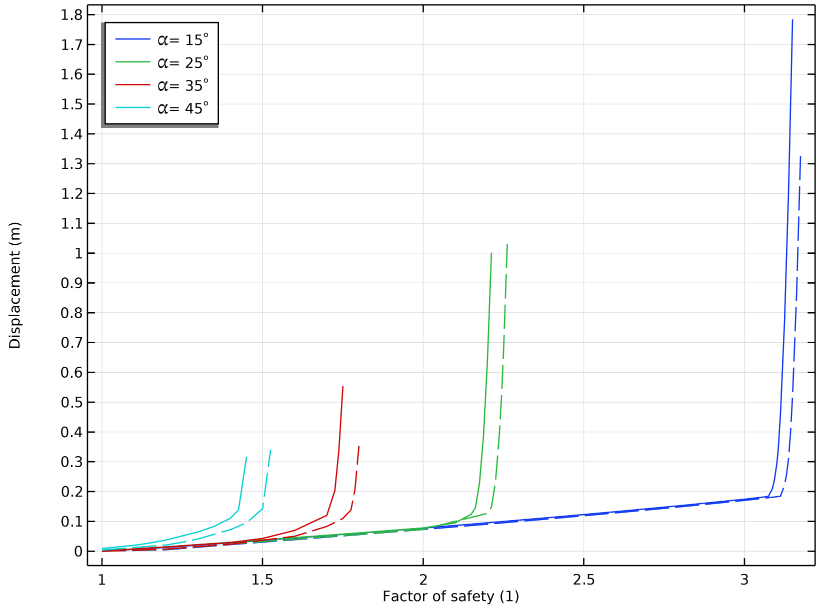

Figure 3: Maximum displacement in the soil embankment versus the factor of safety for material case 1 (solid) and material case 2(dashed).

The equivalent plastic strain for different slope angles just before collapse is shown in Figure 4 and Figure 5 for the two material sets. The localization of plastic strains in the figures gives an indication of the failure surface for different slope angles. It is evident that for lower slope angles, multiple failure surfaces develop in the embankment.

A 3D visualization of the displacement for a slope angle equal to 45° is shown in Figure 6. The results are qualitatively in good agreement with the results presented in Figure 2 of Ref. 1. The figures also clearly show how parts of the embankment outside the slip surface start sliding once the material becomes unstable.

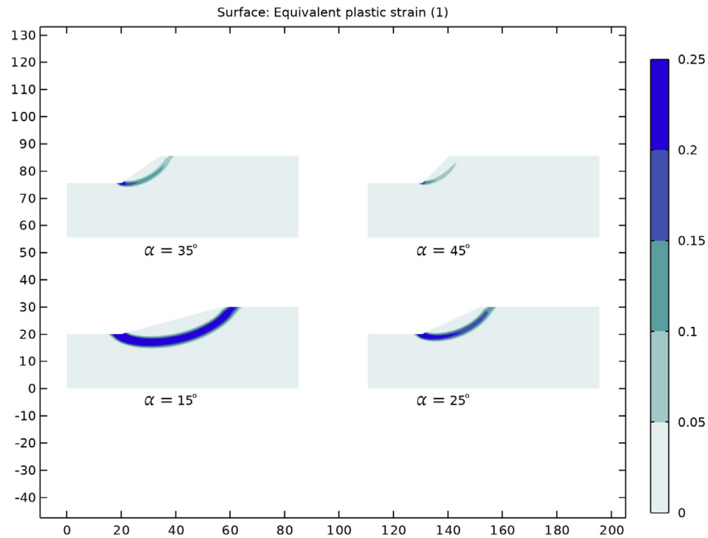

Figure 4: Equivalent plastic strain just before collapse for material case 1 (nonassociative flow).

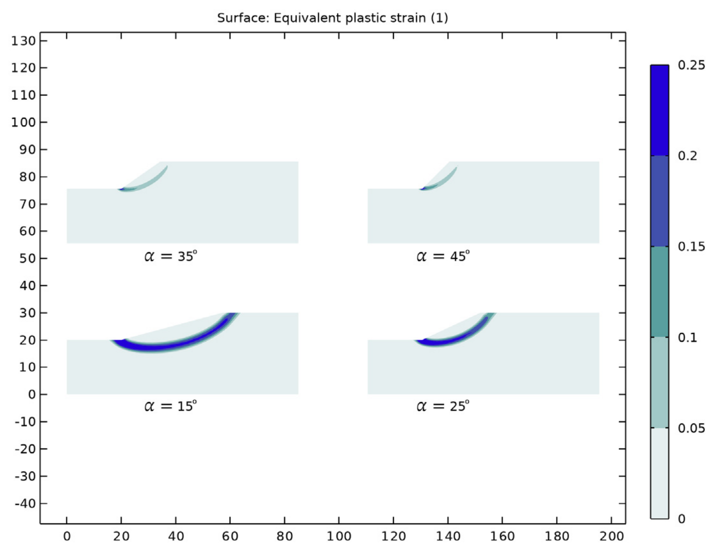

Figure 5: Equivalent plastic strain just before collapse for material case 2 (associative flow).

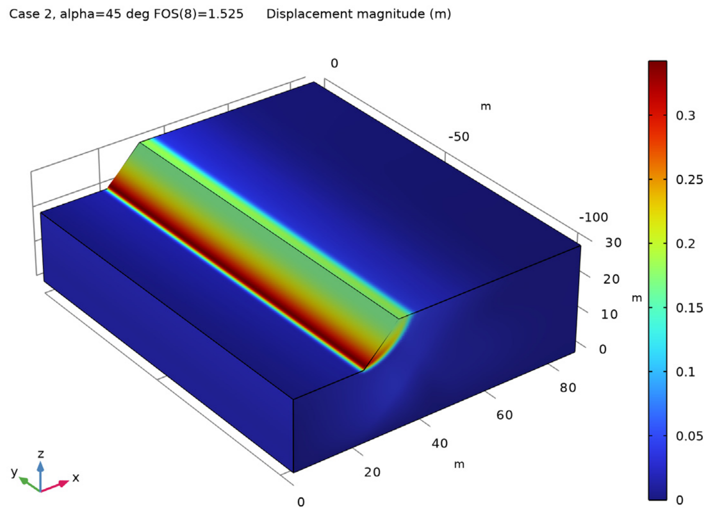

Figure 6: Displacement magnitude in the soil embankment.

Notes About the COMSOL Implementation

Two stationary study steps are added. The first study step computes the in-situ stresses due to gravity. The Mohr–Coulomb criterion is added in the second study to compute the elastoplastic failure due to the combined effect of gravity and the reduction in strength, where the initial stresses generated in the first step are incorporated in the analysis with the help of an **Initial Stress and Strain** node.

Two outer parametric sweeps are added to change the slope angle and the set of material parameters. A **Stop Condition** is added to the continuation parameter sweep such that the solution stops when the maximum displacement in the embankment exceeds a tenth of the length of the slope.

References

1. H.F. Schweiger "Strength reduction technique with finite element method for slopes without stabilization measures," *Benchmark, International Magazine for Engineers, Designers and Analysts from NAFEMS*, pp. 51–58, 2020.

2. S. Oberhollenzer, F. Tschuchnigg, and H.F. Schweiger "Finite element analysis of slope stability problems using non-associated plasticity," *Journal of Rock Mechanics and Geotechnical Engineering*, vol. 10, pp. 1091–1101, 2018.

Application Library path: `Geomechanics_Module/Verification_Examples/strength_reduction_method`

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 **Structural Mechanics** > **Solid Mechanics (solid)**.

3 Right-click and choose **Add Physics**.

4 Click **Study**.

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

6 Click **Done**.

Parameters for the model geometry, material, and solver are available in the appended text files.

GLOBAL DEFINITIONS

Geometry and Solver Parameters

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

2 In the **Settings** window for **Parameters**, type `Geometry and Solver Parameters` in the **Label** text field.

3 Locate the **Parameters** section. Click **Load from File**.

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

Material Parameters

1 In the **Home** toolbar, click **Parameters** and choose **Add** > **Parameters**.

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

3 Locate the **Parameters** section. Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `strength_reduction_method_material_parameters1.txt`.

5 In the **Home** toolbar, click **Parameter Case**.

6 In the **Home** toolbar, click **Parameter Case**.

7 In the **Settings** window for **Case**, locate the **Parameters** section.

8 Click **Load from File**.

9 Browse to the model's Application Libraries folder and double-click the file `strength_reduction_method_material_parameters2.txt`.

DEFINITIONS

Define the parameterized cohesion and angle of internal friction for the Mohr–Coulomb criterion.

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
beta_f	cos(atan(tan(phi)/FOS))*cos(atan(tan(psi)/FOS))/(1-sin(atan(tan(phi)/FOS))*sin(atan(tan(psi)/FOS)))		Reduction factor
c_r	beta_f*c	Pa	Reduced cohesion
phi_r	atan(beta_f*tan(phi))	rad	Reduced friction angle
c_p	c_r/FOS	Pa	Parameterized cohesion
phi_p	atan(tan(phi_r)/FOS)	rad	Parameterized friction angle

GEOMETRY 1

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+Lslope,L2,0`.

5 In the **y** text field, type `0, 0, H1+H2,H1+H2,H1,H1`.

6 Click **Build Selected**.

SOLID MECHANICS (SOLID)

Linear Elastic Material 1

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

2 In the **Settings** window for **Linear Elastic Material**, locate the **Quadrature Settings** section.

3 Select the **Reduced integration** checkbox.

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 Model** section.

3 From the F_f list, choose **Mohr–Coulomb**.

4 Click to expand the **Nonlocal Plasticity Model** section. From the list, choose **Implicit gradient**.

5 In the l_{int} text field, type `0.1`.

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 two study steps in order to account for the in situ stresses due to gravity. Add the in situ stresses computed in the first study step as initial stresses for the second study step. You can access these stresses using the `withsol` operator as follows:

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

3 Specify the S_0 matrix as

`withsol('sol2', solid.sxx)`	`withsol('sol2', solid.sxy)`	`withsol('sol2', solid.sxz)`
withsol('sol2',solid.sxy)	`withsol('sol2', solid.syy)`	`withsol('sol2', solid.syz)`
withsol('sol2',solid.sxz)	withsol('sol2',solid.syz)	`withsol('sol2', solid.szz)`

Gravity 1

In the **Physics** toolbar, click **Global** and choose **Gravity**.

Roller 1

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

2 Select Boundaries 1 and 6 only.

Fixed Constraint 1

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

2 Select Boundary 2 only.

MATERIALS

Soil Material

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 Material` in the **Label** text field.

3 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	`E_soil`	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	`nu_soil`	1	Young's modulus and Poisson's ratio
Density	rho	`rho_soil`	kg/m³	Basic
Initial cohesion	cohesion0	`c_p`	Pa	Soil material
Friction angle	phis	`phi_p`	rad	Soil material

MESH 1

Mapped 1

In the **Mesh** toolbar, click **Mapped**.

Size 1

1 Right-click **Mapped 1** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section.

5 Select the **Maximum element size** checkbox. In the associated text field, type `0.75`.

6 In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.

The material strengths are parameterized with the help of the FOS parameter. Add an auxiliary sweep for FOS in the second study step.

Add two **Parametric Sweep** nodes to change the slope angle and the material parameters.

STUDY 1

Step 1: Stationary

1 In the **Model Builder** window, under **Study 1** 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** checkbox.

4 In the 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**.

5 Right-click and choose **Disable**.

Step 2: Stationary 2

1 In the **Study** toolbar, click **Stationary**.

2 In the **Settings** window for **Stationary**, click to expand the **Values of Dependent Variables** section.

3 Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.

4 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** checkbox.

5 Click **Add**.

6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
FOS (Factor of safety)	1 4	

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
alpha (Slope angle)	`range(15,10,45)`	`deg`

Parametric Sweep 2

1 In the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 From the **Sweep type** list, choose **Parameter switch**.

4 Click **Add**.

5 In the table, enter the following settings:

Switch	Cases	Case numbers
Material Parameters	All	range(1,1,2)

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 2** node, then click **Parametric 1**.

4 In the **Settings** window for **Parametric**, locate the **General** section.

5 From the **On error** list, choose **Skip parameter step**.

6 Click to expand the **Continuation** section. Select the **Tuning of step size** checkbox.

7 In the **Initial step size** text field, type `0.2`.

8 In the **Maximum step size** text field, type `0.2`.

9 From the **Predictor** list, choose **Constant**.

10 Click to expand the **Output** section. From the **Parameters to store** list, choose **Steps taken by solver**.

Add a **Stop Condition** to stop the solution when the maximum displacement in the embankment exceeds a tenth of the length of the slope.

11 Right-click **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 2** > **Parametric 1** and choose **Stop Condition**.

DEFINITIONS

Maximum 1 (maxop1)

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

2 Select Domain 1 only.

STUDY 1

Solution 1 (sol1)

1 In the **Model Builder** window, under **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 2** > **Parametric 1** click **Stop Condition 1**.

2 In the **Settings** window for **Stop Condition**, locate the **Stop Expressions** section.

3 Click **Add**.

4 In the table, enter the following settings:

Stop expression	Stop if	Active	Description
`comp1.maxop1(comp1.solid.disp)>Lslope/20`	True (>=1)	√	Stop expression 1

5 In the **Model Builder** window, under **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 2** click **Fully Coupled 1**.

6 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.

7 From the **Nonlinear method** list, choose **Constant (Newton)**.

8 In the **Maximum number of iterations** text field, type 8.

9 In the **Study** toolbar, click **Compute**.

Create a plot showing the FOS as a function of the slope angle for the two sets of material parameters.

RESULTS

Evaluation Group 1

1 In the **Results** toolbar, click **Evaluation Group**.

2 In the **Settings** window for **Evaluation Group**, click to expand the **Format** section.

3 From the **Include parameters** list, choose **Off**.

Global Evaluation 1

1 Right-click **Evaluation Group 1** and choose **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (sol3)**.

4 From the **Material Parameters** list, choose **From list**.

5 In the **Material Parameters** list box, select **Case 1**.

6 From the **Parameter selection (FOS)** list, choose **Last**.

7 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`alpha`	`deg`	`Slope angle`

Global Evaluation 2

1 Right-click **Global Evaluation 1** and choose **Duplicate**.

2 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`FOS`	`1`	`Factor of safety`

Global Evaluation 3

1 Right-click **Global Evaluation 2** and choose **Duplicate**.

2 In the **Settings** window for **Global Evaluation**, locate the **Data** section.

3 In the **Material Parameters** list box, select **Case 2**.

4 In the **Evaluation Group 1** toolbar, click **Evaluate**.

FOS vs. Slope Angle

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

2 In the **Settings** window for **1D Plot Group**, type `FOS vs. Slope Angle` in the **Label** text field.

3 Locate the **Plot Settings** section.

4 Select the **y-axis label** checkbox. In the associated text field, type `Factor of safety (1)`.

Table Graph 1

1 Right-click **FOS vs. Slope Angle** and choose **Table Graph**.

2 In the **Settings** window for **Table Graph**, locate the **Data** section.

3 From the **Source** list, choose **Evaluation group**.

4 Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.

5 Click to expand the **Legends** section. Select the **Show legends** checkbox.

6 From the **Legends** list, choose **Manual**.

7 In the table, enter the following settings:

Legends
Material case 1
Material case 2

8 In the **FOS vs. Slope Angle** toolbar, click **Plot**.

Create a plot showing the maximum displacement in the embankment as a function of the FOS for all analysis cases.

Displacement vs. FOS

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

2 In the **Settings** window for **1D Plot Group**, type `Displacement vs. FOS` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **None**.

4 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

5 Locate the **Plot Settings** section.

6 Select the **x-axis label** checkbox. In the associated text field, type `Factor of safety (1)`.

Global 1

1 Right-click **Displacement vs. FOS** and choose **Global**.

2 In the **Settings** window for **Global**, click to expand the **Legends** section.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (sol3)**.

4 From the **Material Parameters** list, choose **From list**.

5 In the **Material Parameters** list box, select **Case 1**.

6 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
maxop1(solid.disp)	m	Displacement

7 Locate the **Legends** section. From the **Legends** list, choose **Evaluated**.

8 In the **Legend** text field, type \alpha = eval(alpha,deg)<sup>\circ</sub>.

Global 2

1 Right-click **Global 1** and choose **Duplicate**.

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

3 In the **Material Parameters** list box, select **Case 2**.

4 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

5 From the **Color** list, choose **Cycle (reset)**.

6 Locate the **Legends** section. Clear the **Show legends** checkbox.

7 In the **Displacement vs. FOS** toolbar, click **Plot**.

Add plots to visualize the failure of the embankment for all cases.

RESULT TEMPLATES

1 In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

3 In the tree, select **Study 1/Parametric Solutions 1 (sol3)** > **Solid Mechanics** > **Equivalent Plastic Strain (solid)**.

4 Click the **Add Result Template** button in the window toolbar.

5 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

RESULTS

Equivalent Plastic Strain (Case 1)

1 In the **Settings** window for **2D Plot Group**, type Equivalent Plastic Strain (Case 1) in the **Label** text field.

2 Locate the **Data** section. From the **Dataset** list, choose **None**.

3 Click to expand the **Title** section. From the **Title type** list, choose **Custom**.

4 Find the **Solution** subsection. Clear the **Solution** checkbox.

5 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

6 Click to expand the **Plot Array** section. From the **Array type** list, choose **Square**.

Surface 1

1 In the **Model Builder** window, expand the **Equivalent Plastic Strain (Case 1)** node, then click **Surface 1**.

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

3 From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (sol3)**.

4 Click to expand the **Range** section. Select the **Manual color range** checkbox.

5 In the **Maximum** text field, type `0.25`.

6 Locate the **Coloring and Style** section. In the **Number of bands** text field, type `5`.

Solution Array 1

1 Right-click **Surface 1** and choose **Solution Array**.

2 In the **Settings** window for **Solution Array**, locate the **Data** section.

3 From the **Material Parameters** list, choose **First**.

4 From the **Parameter selection (FOS)** list, choose **Last**.

5 Locate the **Plot Array** section. From the **Array shape** list, choose **Square**.

Equivalent Plastic Strain (Case 1)

In the **Model Builder** window, under **Results** click **Equivalent Plastic Strain (Case 1)**.

Table Annotation 1

1 In the **Equivalent Plastic Strain (Case 1)** toolbar, click **More Plots** and choose **Table Annotation**.

2 In the **Settings** window for **Table Annotation**, locate the **Data** section.

3 From the **Source** list, choose **Local table**.

4 In the table, enter the following settings:

x-coordinate	y-coordinate	Annotation
25	0	`\[\alpha \;=\;15^\circ\]`
125	0	`\[\alpha \;=\;25^\circ\]`

x-coordinate	y-coordinate	Annotation
25	55	`\[\alpha \;=\;35^\circ\]`
125	55	`\[\alpha \;=\;45^\circ\]`

5 Select the **LaTeX markup** checkbox.

6 Locate the **Coloring and Style** section. Clear the **Show point** checkbox.

7 In the **Equivalent Plastic Strain (Case 1)** toolbar, click **Plot**.

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

Set up a duplicate plot group for the second material parameter case.

Equivalent Plastic Strain (Case 2)

1 Right-click **Equivalent Plastic Strain (Case 1)** and choose **Duplicate**.

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

3 In the **Model Builder** window, expand the **Equivalent Plastic Strain (Case 2)** node.

Solution Array 1

1 In the **Model Builder** window, expand the **Results** > **Equivalent Plastic Strain (Case 2)** > **Surface 1** node, then click **Solution Array 1**.

2 In the **Settings** window for **Solution Array**, locate the **Data** section.

3 From the **Material Parameters** list, choose **Last**.

4 In the **Equivalent Plastic Strain (Case 2)** toolbar, click **Plot**.

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

Create a 3D visualization of the displacements in the embankment at failure.

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 1/Parametric Solutions 1 (sol3)**.

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

5 Find the **Embedding** subsection. From the **Map plane to** list, choose **xz-plane**.

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

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

Deformation 1

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

2 In the **Settings** window for **Deformation**, locate the **Scale** section.

3 Select the **Scale factor** checkbox. In the associated text field, type 1.

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

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