PLAXIS

PLAXIS 2D 2023.2

Tutorial Manual 2D - Stabilisation of a rock slope

Last Updated: August 25, 2023

1 Stabilisation of a rock slope

This tutorial illustrates the use of PLAXIS 2D for simulating the excavation of a road in a rock mass of siltstone. Some new features, such as *Discontinuity* and *Cable* elements, are described in order to model the road cut in siltstone.

Objectives:

- Use the **Gravity loading** method to generate the initial stresses.
- Model the fault using the **Discontinuity** feature.
- Model the reinforcement with the **Cables** feature .
- Determine the influence of reinforcements on the factor of safety.

Geometry:

In this tutorial, the geometry of the natural siltstone sloping ground is considered, wherein a road cut slope is excavated. An unsupported excavation of the road cut will cause the instability of the rock slope. Therefore, the excavation is conducted in two stages. After the first 9 m of excavation, the excavated slope is reinforced with three rows of cables with 3 m spacing. And a 3 m retaining wall with backfill is constructed as soon as the final excavation is completed. A fault at a dip angle of 38.7° is located within the rock mass, and the fault daylights near the toe of the slope due to the excavation. The distance between the fault and the slope face at the top boundary is 12 m. The geometry of the rock slope, along with the fault and reinforcements, is shown in Figure 1 (on page 2).

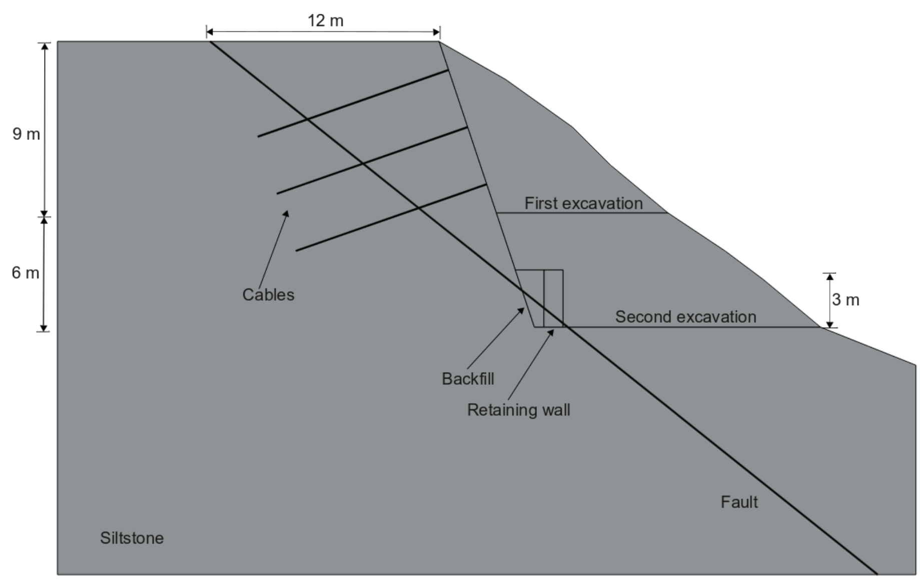


Figure 1: Geometry of the rock slope with reinforcement

1.1 Create a new project

To create a new project, follow the following steps:

1. Start the **Input program** and select **Start a new project** from the **Quick Start** dialog box.
2. In the **Project** tabsheet of the **Project properties** window, enter an appropriate title.
3. In the **Model** tabsheet keep the default options for **Model (Plane strain)**, and **Elements (15-Noded)**.
4. Set the model **Contour** to: x_{min} = 0 m, x_{max} = 45 m, y_{min} = -13 m and y_{max} = 15 m.
5. Keep the default values for units, constants, and general parameters and click **OK** to close the **Project properties** window.

1.2 Create and assign material data sets

In this project, the rock mass is a siltstone modelled using the Hoek Brown constitutive model. Table 1 (on page 3) shows the material properties of the rock mass.

Table 1: Material properties of the siltstone

Parameter	Name	Value	Unit
General			
Soil model	-	Hoek Brown	-
Drainage type	-	Drained	-
Unsaturated unit weight	γ_{unsat}	24	kN/m^3
Saturated unit weight	γ_{sat}	24	kN/m^3
Mechanical			
Young's modulus	E_{rm}	$1 \cdot 10^6$	kN/m^2
Poisson's ratio	ν	0.25	-
Uni-axial compressive strength intact rock	$\vert \sigma_{ci} \vert$	$25{\cdot}10^3$	kN/m^2
Intact rock parameter	m_i	10	°
Geological strength index	GSI	39	-

Mechanical			
Disturbance factor	D	0	-
Tension cut-off	-	True	-
Tensile strength	-	30	kN/m^2
Dilatancy angle	ψ_{max}	10	°
Dilatancy parameter	σ_ψ	50	kN/m^2

To create a material set for the rock mass, follow these steps:

1. Select the **Show materials** button and the **Material sets** window will pop up.
2. Click the **New** button in the **Material sets** window to create the data set as shown in the Table 1 (on page 3).

1.2.1 Define the rock mass stratigraphy

The geometry of the natural slope must be defined. To do so, follow these steps:

1. Click the **Structures** tab to proceed with the input of structural elements in the **Structures mode.**
2. Click the **Snapping options** button in the bottom toolbar. In the appearing window, set the **Spacing** to 0.5 and the **Number of snap intervals** to 1 as shown in Figure 2 (on page 4).

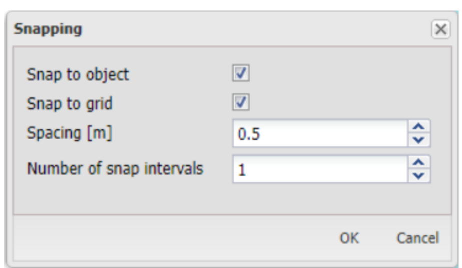


Figure 2: Window showing Snapping options

3. Click the **Create soil polygon** button in the side toolbar and select the **Create soil polygon** option.
4. Create the rock mass either in the command line or by clicking on points at coordinates:
```
(0 -13) (0 15) (20 15) (23.5 13) (27 10.5) (29 8.5) (32 6) (35 4) (37 2.5) (40 0)
(42.5 -1) (45 -2) (45 -13)
```
5. Select the created polygon, right click on it, and assign the Material to the soil polygon as shown in Table 1 (on page 3).

1.3 Define the structural elements

The creation of the fault, the cable elements, and the retaining wall are described below.

1.3.1 Define the fault

The fault is modelled using a discontinuity in this model, and its material properties are defined in Table 2 (on page 5) below:

Table 2: Material properties of the fault (discontinuity)

Parameter	Name	Value	Unit
General			
Material model	-	Mohr-Coulomb	-
Drainage type	-	Drained	-
Parameter	**Name**	**Value**	**Unit**
Mechanical			
Normal stiffness	k_n	$1 \cdot 10^6$	kN/m^3
Shear stiffness	k_s	$100 \cdot 10^3$	kN/m^3
Strength method	-	Peak	
Cohesion	c'	5	kN/m^2
Friction angle	φ'	24	o
Dilatancy angle	ψ	0	o
Consider gap closure	-	True	-

1. In the **Structures** mode, click the **Create line** button and select the **Create discontinuity** option .
2. Draw the discontinuity through the points (8 15) and (43 -13).
3. Select the discontinuity, and in the **Selection explorer**, click on **Material**. This will display a drop-down menu and a plus button next to it, as shown in Figure 3 (on page 6).
4. Click the **Add** button , to create and assign a new material set for the discontinuity.
5. Define the material data set for the discontinuity based on the properties listed in Table 2 (on page 5).

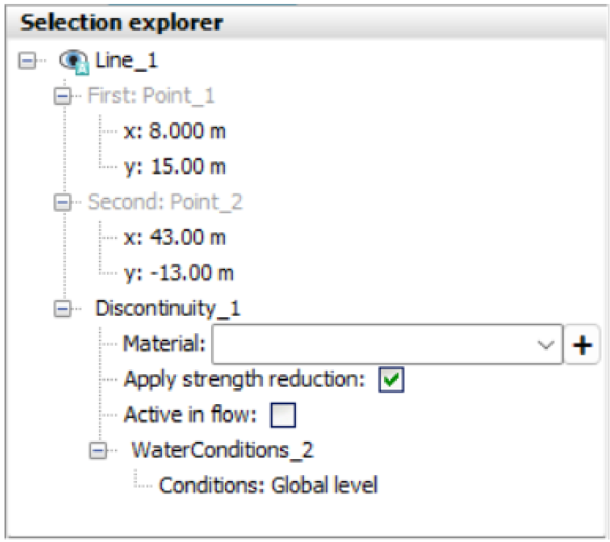


Figure 3: Material assignment of discontinuity in the Selection explorer

1.3.2 Define the excavation stages

The road cut is excavated in two stages with a total height of 15 m. The first excavation stage corresponds to the upper 9 m and the remaining 6 m is the second excavation stage. To define the excavation stages follow these steps:

1. Define the slope face of the road cut by drawing a line passing through points (20 15) and (25 0).
2. Define the bottom of excavation (roadway) by drawing a line through points (25 0) and (40 0).
3. Define the upper 9 m of excavation by drawing a horizontal line from (23 6) to the point where it intersects with the initial slope surface at (32 6).

1.3.3 Define the cables

The reinforcement of the upper slope after the first excavation stage is achieved by creating three rows of 25.4 mm diameter grouted cable elements installed at a spacing of 3 m.

The coordinates of the cables and the material properties of the cables are listed in Table 3 (on page 6) and Table 4 (on page 7) respectively.

Table 3: Cable element coordinates

Cable location	First point	Second point
Top	(20.5 13.5)	(10.5 10)

Cable location	First point	Second point
Middle	(21.5 10.5)	(11.5 7)
Bottom	(22.5 7.5)	(12.5 4)

Table 4: Properties of the cables

Parameter	Name	Value	Unit
General			
Material type	-	Elastoplastic	-
Mechanical			
Cable bolt spacing	$L_{spacing}$	3	m
Cross section type	-	Predefined	-
Predefined cross section type	-	Solid circular beam	-
Diameter	D	0.0254	m
Stiffness	E	$98.6 \cdot 10^6$	kN/m^2
Compressive yield strength	$N_{p,comp}$	0	kN
Tensile yield strength	$N_{p,tens}$	548	kN
Shear stiffness	k_s	$15 \cdot 10^6$	kN/m^2
Strength distribution	-	Uniform	-
Cohesive strength	-	800	kN/m
Bond frictional angle	φ_{bond}	20	o
Failure surface perimeter	-	Predefined	-

1. Click on the **Create line** button and then select the **Create cable** option .
2. Now draw a line from (20.5 13.5) to (10.5 10) to create the first row of cables.
3. Repeat step 2 to define the middle and bottom rows of the cables using the coordinates given in Table 3 (on page 6).
4. Create the Cable material data set based on the parameters listed in Table 4 (on page 7).

5. Multi-select all the cables in the drawing area, then in the **Selection explorer** assign the cable material by choosing the corresponding option in the **Material** drop-down menu .

1.3.4 Define the retaining wall and backfill

The retaining wall is modelled with a thickness of 1 m, constructed at the toe of the rock slope. The backfill is provided between the slope face and the retaining wall, with its thickness varying along the height. The material properties of the retaining wall and the backfill are shown in Table 5 (on page 8) and Table 6 (on page 8) respectively.

Table 5: Material properties of the retaining wall

Parameter	Name	Value	Unit
General			
Soil model	-	Mohr-Coulomb	-
Drainage type	-	Non-porous	-
Unsaturated unit weight	γ_{unsat}	24	kN/m^3
Parameter	**Name**	**Value**	**Unit**
Mechanical			
Young's modulus	E_{ref}	$27 \cdot 10^6$	kN/m^2
Poisson's ratio	ν	0.15	-
Cohesion	c_{ref}	500	kN/m^2
Friction angle	φ	35	o
Dilatancy angle	ψ	5	o
Tension cut-off	-	True	
Tensile strength	-	750	kN/m^2

Table 6: Material properties of the backfill

Parameter	Name	Value	Unit
General			
Soil model	-	Mohr-Coulomb	-

Parameter	Name	Value	Unit
General			
Drainage type	-	Drained	-
Unsaturated unit weight	γ_{unsat}	20	kN/m^3
Saturated unit weight	γ_{sat}	20	kN/m^3
Mechanical			
Young's modulus	E'_{ref}	$100 \cdot 10^3$	kN/m^2
Poisson's ratio	ν	0.3	-
Cohesion	c'_{ref}	5	kN/m^2
Friction angle	φ'	45	o
Dilatancy angle	ψ	15	o

To define the geometry of the retaining wall and backfill, follow these steps:

1. Click on the **Create line** button and then select the **Create line** option .
2. Draw the retaining wall by defining lines through the points (25.5 3), (26.5 3), then (26.5 0), (26.5 3) and (25.5 0), (25.5 3).
3. Define the backfill by drawing a line from the point (25.5 3) of the retaining wall to the sloping face of the rock cut at (24 3).
4. Create the separate material data sets for the retaining wall and the backfill from the **Soil and interfaces** type, using the parameters listed in Table 5 (on page 8) and Table 6 (on page 8) which will be assigned later in the **Staged construction** mode during calculation.

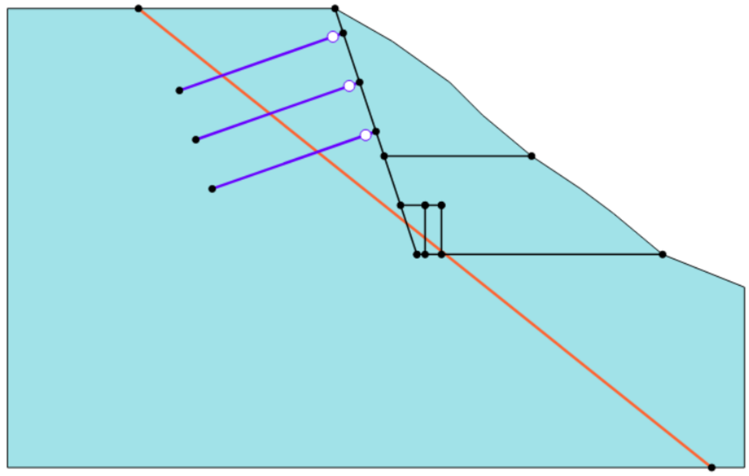


Figure 4: Final geometry of the model

1.4 Generate the mesh

1. Proceed to the **Mesh mode**.
2. Click the **Generate mesh** button to generate the mesh. The **Mesh options** window will appear.
3. Select the **Medium** option in the **Element distribution** list, and generate the mesh.
4. Click the **View mesh** button to view the mesh. The generated mesh is shown in Figure 5 (on page 10).
5. Select the **Close** button on the top left of the Output program to close the mesh view.

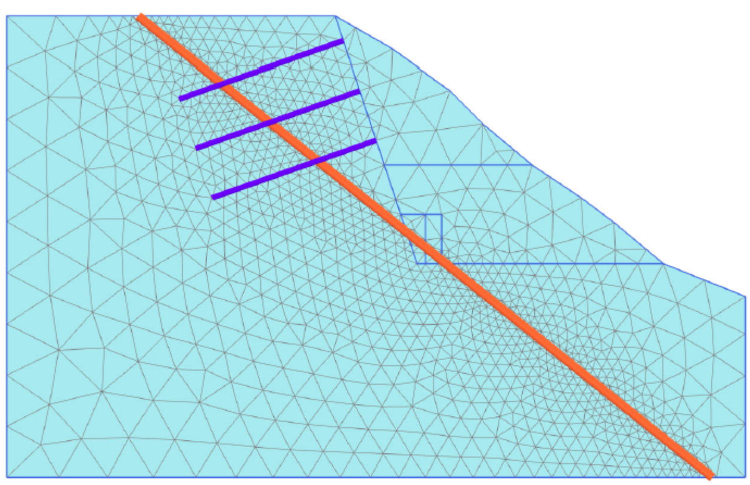


Figure 5: The generated mesh

1.5 Define and perform the calculation

The calculation of this project consists of six phases.

- In the initial phase, the initial stresses are generated using the **Gravity loading** method.
- Phase 1 involves the excavation of the top 9 m.
- Phase 2 includes the installation and connection of the cables to the face of the rock cut.
- Phase 3 entails a further 6 m excavation to reach the designed level of the roadway.
- Phase 4 involves the construction of the retaining wall near the toe of the road cut.
- Phase 5 involves placing the backfill between the slope face and the retaining wall.

Additionally, a safety analysis is performed for both Phase 3 and Phase 5.

1.5.1 Initial phase: Initial conditions

The initial stress field is generated by means of the Gravity loading method.

1. Proceed to the **Staged construction** mode.
2. Activate all the polygons of the original sloping rock mass.
3. Activate all the lines of the discontinuity. All other structural components (cables) should remain deactivated.
4. In the **Phases explorer**, double-click the initial phase and for **Calculation type** select the **Gravity loading** option. The other default parameters for the initial phase remain the same, and water will not be considered in this example.
5. Click **OK** to close the **Phases** window.

Note: Make sure to activate the small triangular soil polygon between the retaining wall, discontinuity and bottom of the second excavation stage.

1.5.2 Phase 1: First excavation stage

1. Click the **Add phase** button to create a new phase.
2. In the **Staged construction** mode, deactivate the upper 9 m cluster of the excavation.

The model for the Phase 1 is shown in Figure 6 (on page 12) below:

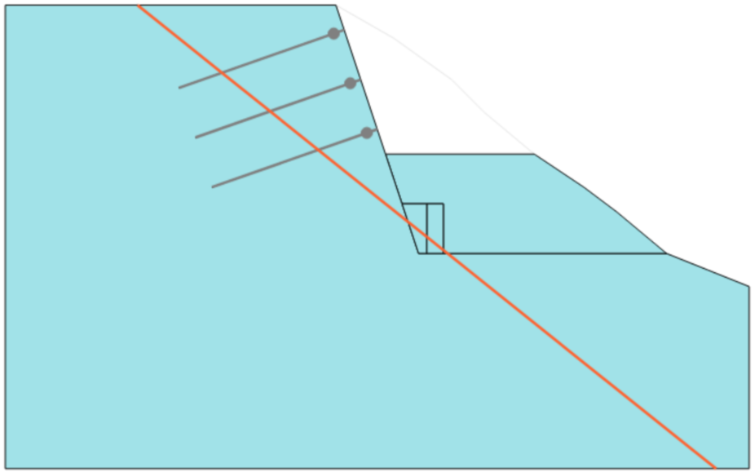


Figure 6: Configuration of Phase 1

1.5.3 Phase 2: Installation of cables

1. In the **Phases explorer**, click on the **Add phase** button .
2. Select all the cable elements and then activate them by clicking on the checkbox in front of the **Cables** in the **Model explorer**.
3. In the **Selection explorer**, click the checkbox for **Adjust prestress** and assign a pre-stress force of **200 kN.**

The model for the Phase 2 is shown in Figure 7 (on page 12) below:

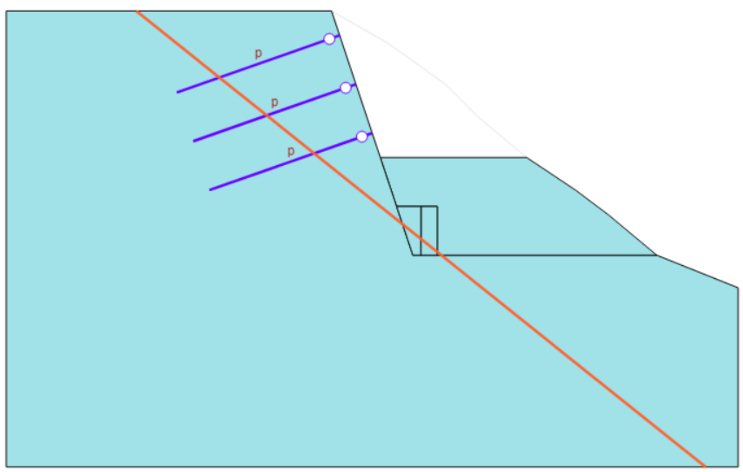


Figure 7: Configuration of Phase 2

1.5.4 Phase 3: Second excavation stage

1. Click the **Add phase** button to add a new phase.
2. Deactivate the soil polygon of the second excavation.
3. Deactivate the lines of the discontinuity that are inside the second excavation.

The model configuration for Phase 3 is shown in Figure 8 (on page 13) below:

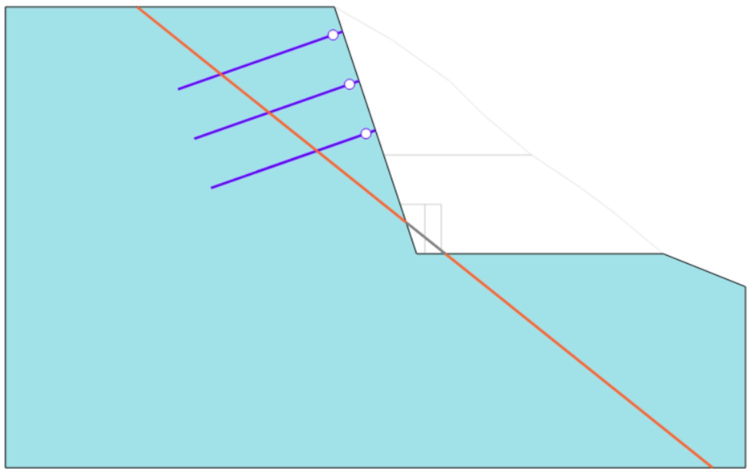


Figure 8: Configuration of Phase 3

Note: Make sure to deactivate the small triangular soil polygon and the discontinuity element between the retaining wall, discontinuity and bottom of the second excavation stage.

1.5.5 Phase 4: Construction of the retaining wall

1. Click the **Add phase** button to add a new phase.
2. Activate the retaining wall and assign the material to the retaining wall according to Table 5 (on page 8).

The model configuration for Phase 4 is shown in Figure 9 (on page 14) below:

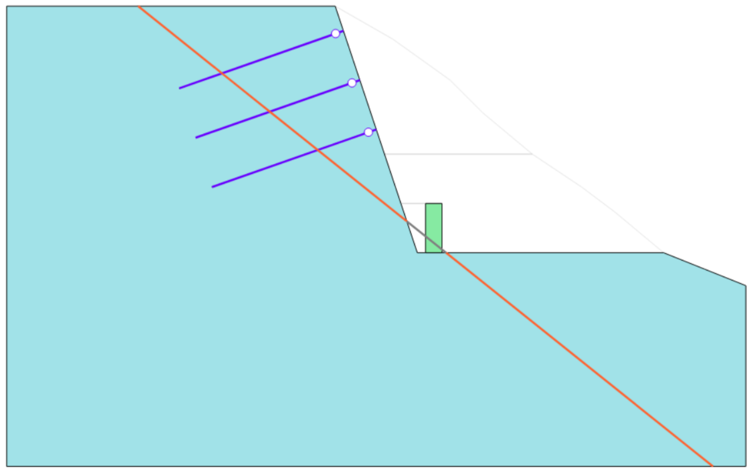


Figure 9: Configuration of Phase 4

1.5.6 Phase 5: Backfilling

1. Click the **Add phase** button to add a new phase.
2. Activate the backfill and assign the material to the backfill according to Table 6 (on page 8).

The model configuration for Phase 5 is shown in Figure 10 (on page 14) below:

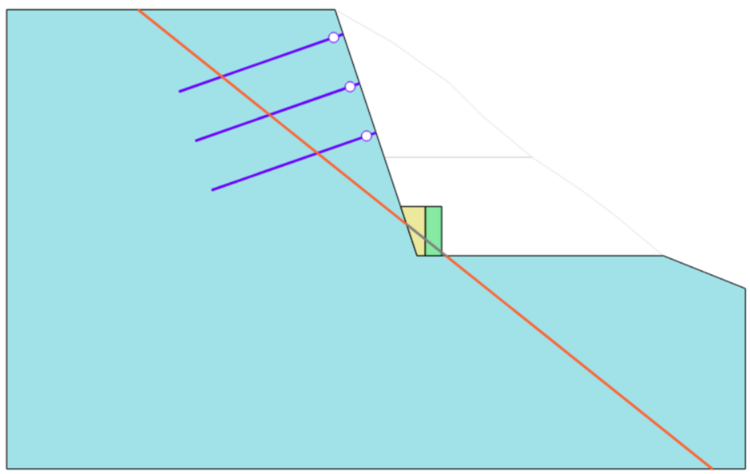


Figure 10: Configuration of Phase 5

1.5.7 Phase 6: Safety analysis

To design the road cut slope, it is important to consider the stability during the excavation and also after the completion of construction. Therefore, it is necessary to evaluate the global safety factor of Phase 3 (second excavation stage), and Phase 5 (construction of the backfill). To calculate the global safety factor of the second excavation, follow these steps:

1. Select Phase 3 in the **Phases** explorer.
2. Add a new calculation phase.
3. Double-click on the new phase to open the **Phases** window.
4. In the **Phases** window, Phase 3 is automatically selected from the **Start from phase** drop-down menu.
5. In the **Calculation type** box, select **Safety** .
6. In the **Loading type** box, the **Incremental multipliers** option is already selected. The first increment of the multiplier that controls the strength reduction process, **Msf**, is set to 0.1.
7. To exclude the existing deformations from the resulting failure mechanism, in the **Deformation control parameters** subtree, select the **Reset displacements to zero** option.
8. In the **Numerical control parameters** subtree, deselect the **Use default iter parameters** option, and in the **Max steps** box input **120** for the safety calculation.
9. The first safety calculation is now defined for Phase 3.
10. Follow the same steps as mentioned above to create a new calculation phase that analyses the stability of the final phase after the construction of the backfill (Phase 5).

The phases after defining the safety calculations are displayed in Figure 11 (on page 15).

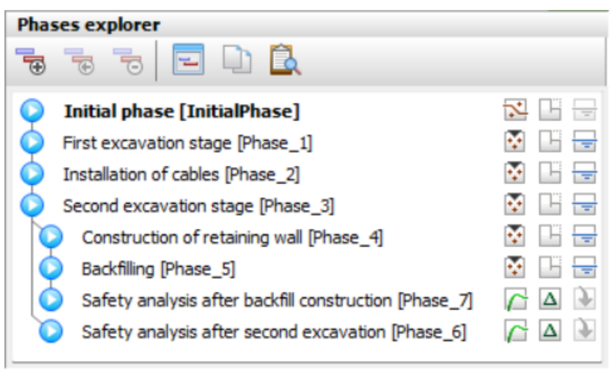


Figure 11: Safety calculation phases

1.6 Calculate

Before initiating the calculation, it is recommended to select some nodes in order to generate the plots to determine the factor of safety at a later stage. To do this, follow these steps.

1. Click on the **Select points for curves** button in the side toolbar.
2. Select the characteristic point (23 6) for curves and click on **Update**.
3. Click on the **Calculate** button to compute the project.
4. Once the calculation is completed, save the project by clicking on the **Save** button .

1.7 Results

For this project, the results for the excavation phases and the safety analysis are evaluated.

1.7.1 Evaluation of results

1. After completing the calculations, select Phase 3 (second excavation stage) and click on the **View calculation results** button . The Output program will start, showing the deformed mesh after the second excavation stage.
2. Click on the **Scale factor** from the top menu and select the **Manual scale**. Input **300** to adjust the scale of the deformed mesh. Then, zoom in the toe area to inspect the deformed mesh.
3. From the button in front of the drop-down menu in the toolbar, choose to view the results for Phase 5. Now the deformed mesh after the construction of the retaining wall and backfill is displayed.

   Figure 12 (on page 17) shows the deformed mesh for the above two phases.

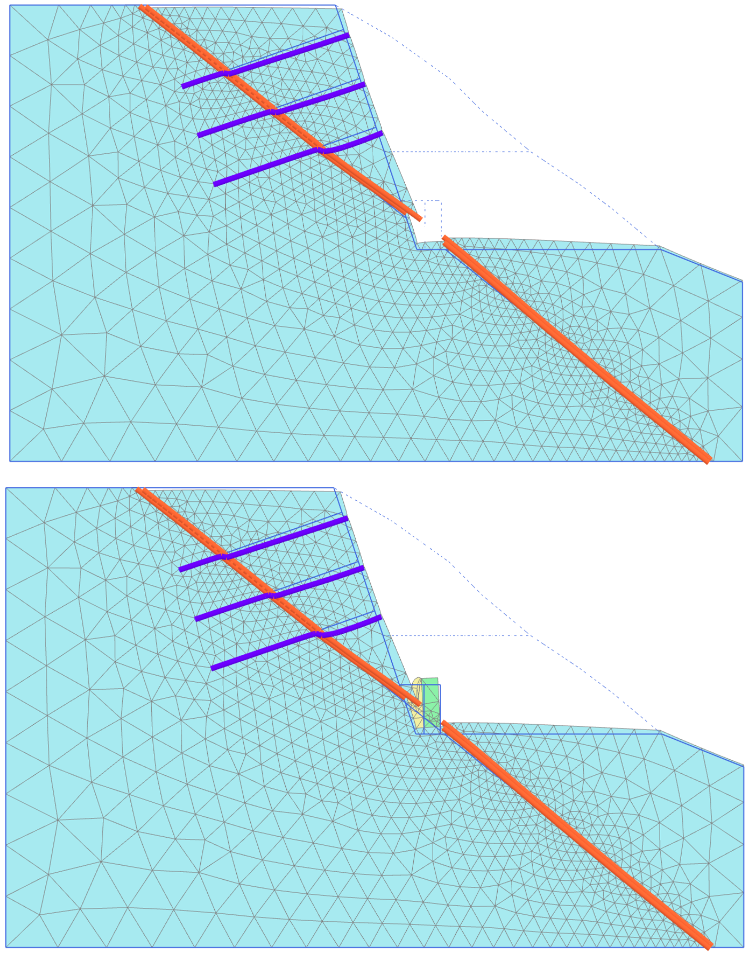
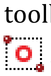


Figure 12: The deformed mesh after the second excavation stage (top) and after the construction of the retaining wall (bottom)

4. To inspect the resulting axial force in the cables, from the button in front of the drop-down menu in the toolbar, go to Phase 3 (Second excavation stage). Click on the **Drag a window to select structures** button  in the side toolbar and drag the mouse to define a rectangle encompassing all the cables. Select the **Cable** option in the appearing window, as shown in Figure 13 (on page 18).

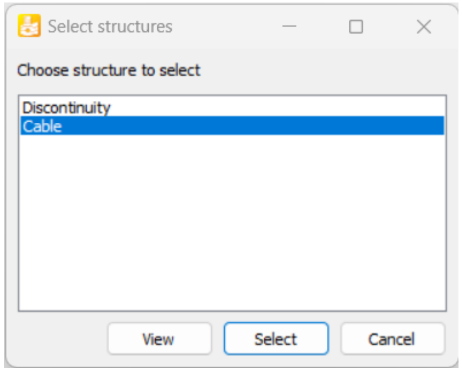


Figure 13: Select structures window

5. Click **View**. Note that the cables are displayed in the Structures view.
6. From the **Forces** menu, select the $T_{s,bond}$ option. Click on the **Scale factor** from the top menu and select **Manual scale** and input **0.01** to adjust the scale and view the shear force developed in the grouted segments as shown in Figure 14 (on page 18).
7. Again from the **Forces** menu, select the **Axial forces N** option and ensure that the **Scale factor** is still set to **0.01**, now the cable axial force after the second excavation stage is shown in Figure 15 (on page 19).

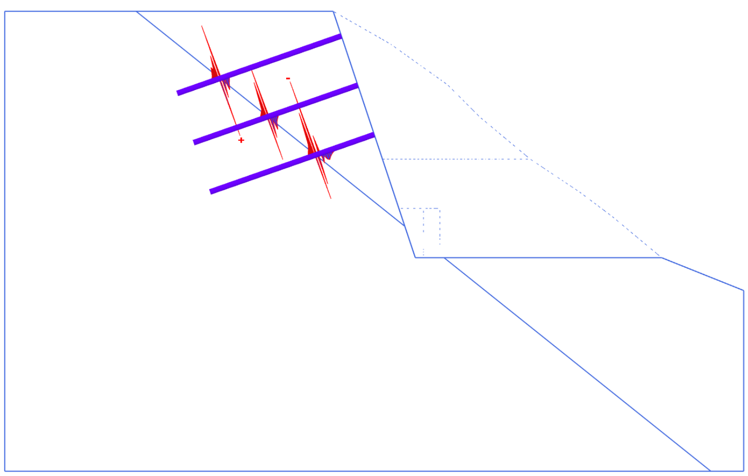


Figure 14: Resulting shear force in the bond $T_{s,bond}$ after the second excavation stage

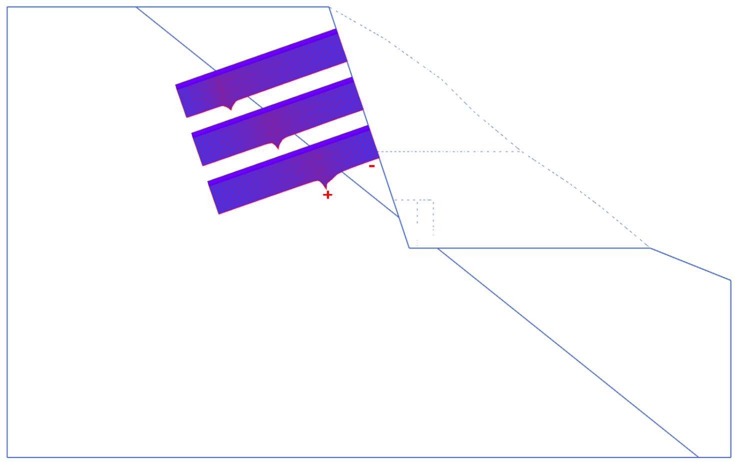


Figure 15: Axial forces in the cables after the second excavation stage

Figure 14 (on page 18) and Figure 15 (on page 19) illustrates that the axial forces are considerably intensified at the intersection of cables and the fault, where the shear force in the bond also develops.

1.7.2 Evaluation of safety analysis results

To assess the stability of the road cut slope design, a safety analysis is performed for this project based on the strength reduction method. Due to the ongoing reduction of the soil strength, additional displacements are generated during a safety calculation. The incremental displacements in the final step of failure are evaluated to assess the possible failure mechanism. To view the incremental displacements of the slope resulting from the second excavation, follow these steps:

1. Select Phase 6 in the Input program, and click on the **View calculation results** button , which is the Safety phase starting from Phase 3.
2. In the Output program, select the menu **Deformations > Incremental displacements > |Δu|**, to display the contour plots of total incremental displacement of the safety calculation after the second excavation stage.
3. From the button in front of the drop-down menu in the toolbar, select Phase 7 to view the results of the safety calculation for the rock slope after the construction of the backfill and retaining wall.

   The results are shown in Figure 16 (on page 20).

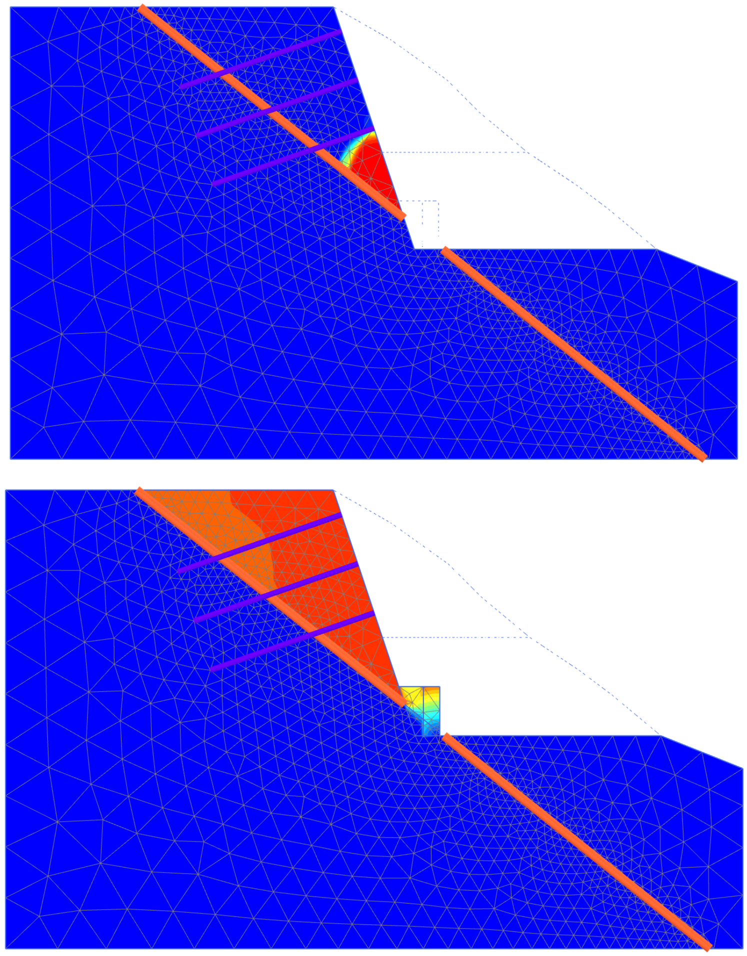


Figure 16: Shadings of the displacement increments indicating the most applicable failure mechanism for the second excavation stage (top) and after the construction of the retaining wall (bottom)

4. From the top toolbar, click the **Arrows** button, and the incremental displacement movement vectors will be displayed. The length of the arrow indicates the magnitude of the specific incremental displacement, while the arrow direction indicates the displacement increment direction, as shown in Figure 17 (on page 21) .

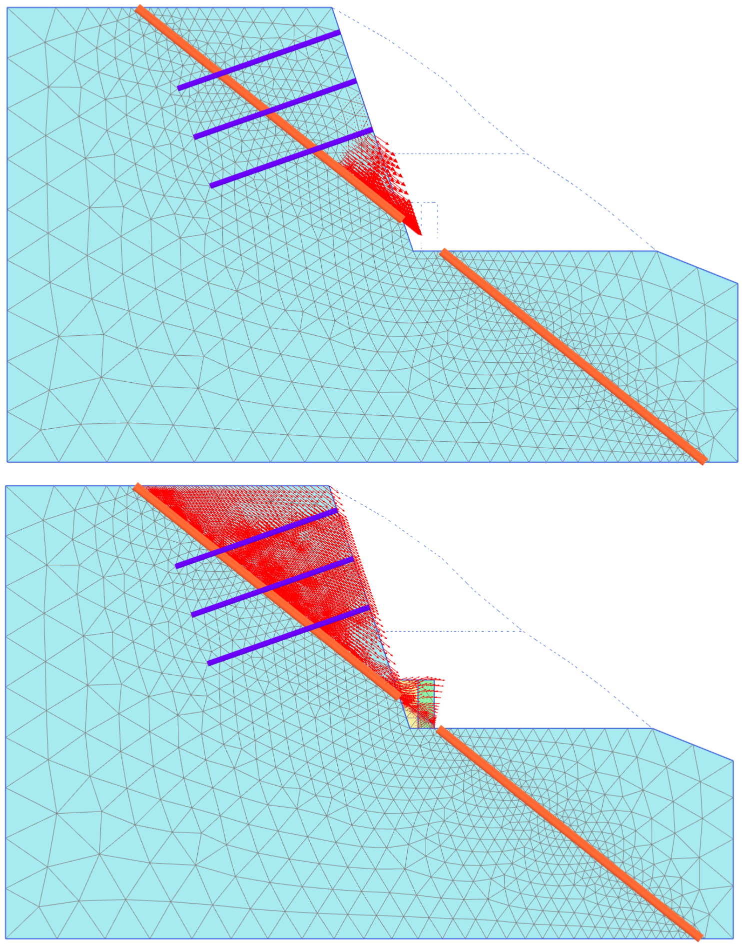


Figure 17: Displacement increments after the second excavation stage (top) and after the construction of the retaining wall (bottom)

Figure 16 (on page 20) and Figure 17 (on page 21) shows the contour plots and arrows of the incremental displacement from the safety calculations for both Phases 3 and 5. The magnitude of the displacement increments is not relevant, but it indicates whether or not a failure mechanism has developed. In Figure 16 (on page 20), the lower slope tends to slide down along the discontinuity, indicating the need for stabilization by constructing a retaining wall. However, in both figures it is seen that the potential failure mechanism is a planar failure of the rock block sliding along the discontinuity.

The safety factor can be obtained from the **Calculation info** option of the **Project** menu. The value of ΣMsf is the strength reduction factor that represents the factor of safety provided that failure has been reached. This can be shown by the value of ΣMsf remaining relatively constant from a certain step onwards while the displacements keep increasing. For an accurate evaluation of the safety factor, it is necessary to plot the changes of parameter ΣMsf against the displacements of a certain node. To calculate the safety factors, follow these steps:

1. To evaluate the factor of safety for Phases 3 and 5, click on the **Curves manager** button.
2. In the **Charts** tabsheet, click on **New** option.
3. In the **Curve generation** window, select the pre-calculation **Node** from the dropdown list for the x-axis. Select **Deformations > Total displacements > |u|**.
4. For the y-axis, select **Project > Multipliers > ΣMsf**.

5. Press **OK** to close the window and generate the chart.
6. Right-click on the chart and select the **Settings** option in the appearing menu. The **Settings** window pops up.
7. In the tabsheet corresponding to the node curve, click the **Phases** button.
8. In the **Select phases** window make sure only the safety calculation Phases 6 and 7 are selected as shown in Figure 18 (on page 22).

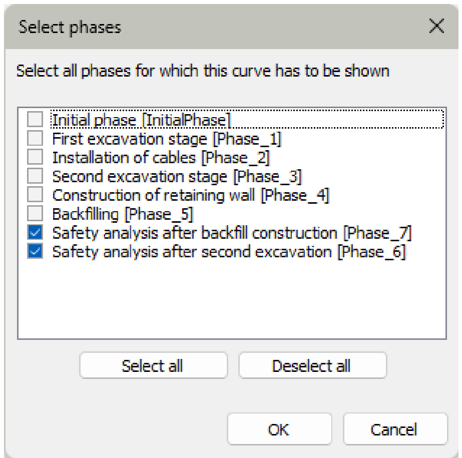


Figure 18: The Select phases window

9. Click **OK** to close the **Select phases** window.
10. In the **Settings** window change the titles and colour of the curves in the corresponding tabsheet.
11. Click **Apply** to update the chart according to the changes made and click **OK** to close the Settings window. The plot is shown in Figure 19 (on page 22).

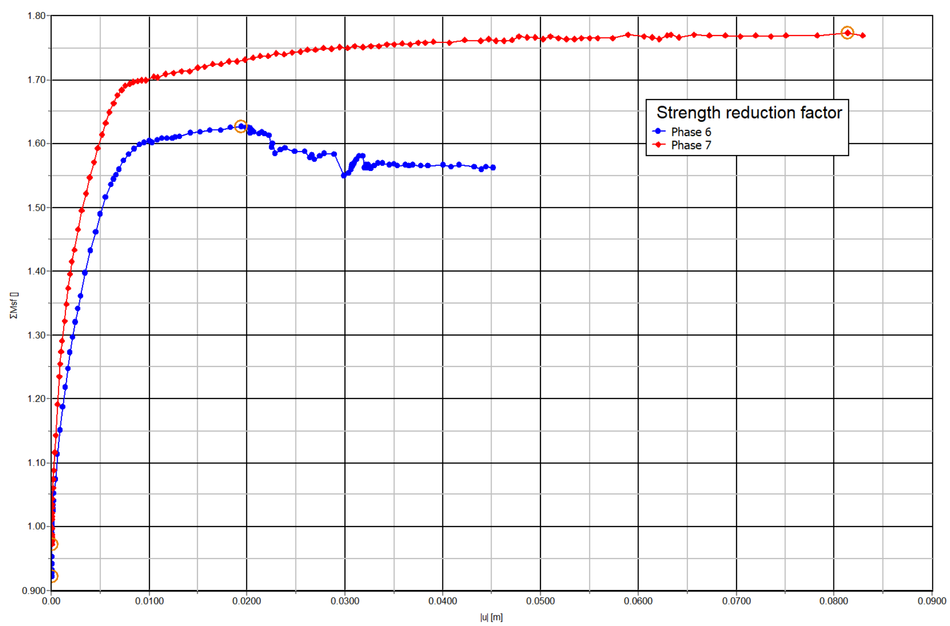


Figure 19: Evaluation of safety factor

By hovering the mouse cursor over a point on the curves, a box appears with the exact value of ΣMsf and the corresponding calculation phase. From this information, it can be determined that the upper curve, with a factor of safety of about 1.77, represents Phase 7, which corresponds to the final road cut slope with the construction of the backfill and retaining wall. Similarly, the lower curve, with a factor of safety of 1.56, represents Phase 6, the second stage of excavation with cable reinforcement. The maximum displacements calculated are irrelevant. It can be seen that for both the curves a more or less constant value of ΣMsf is obtained.