rocscience

FOS for Geosynthetic Reinforced Retaining Wall

1.0 Introduction

For a reinforced retaining wall, it is crucial to evaluate the minimum safety factor of slip surfaces, where they can be impacted by boundary conditions.

In this tutorial, a geosynthetic reinforced retaining wall will be modelled in **Slide2** with the Cuckoo Search method for surface search. This tutorial will help you become familiar with accessing and using the **Manufacturer Library** available in the program (including **Geotextiles, Geogrids,** and **Strips**)

Topics Covered in this Tutorial:

- Adding Geosynthetic Supports to a retaining wall
- Slip Surface Boundaries
- Cuckoo Search (non-circular surface search)
- Support Forces
- Minimum Factor of Safety

Finished Product:

The finished product of this tutorial can be found in the *Tutorial 37 Geosynthetic Retaining Wall.slmd* data file. All tutorial files installed with **Slide2** can be accessed by selecting **File > Recent Folders > Tutorials Folder** from the **Slide2** main menu.

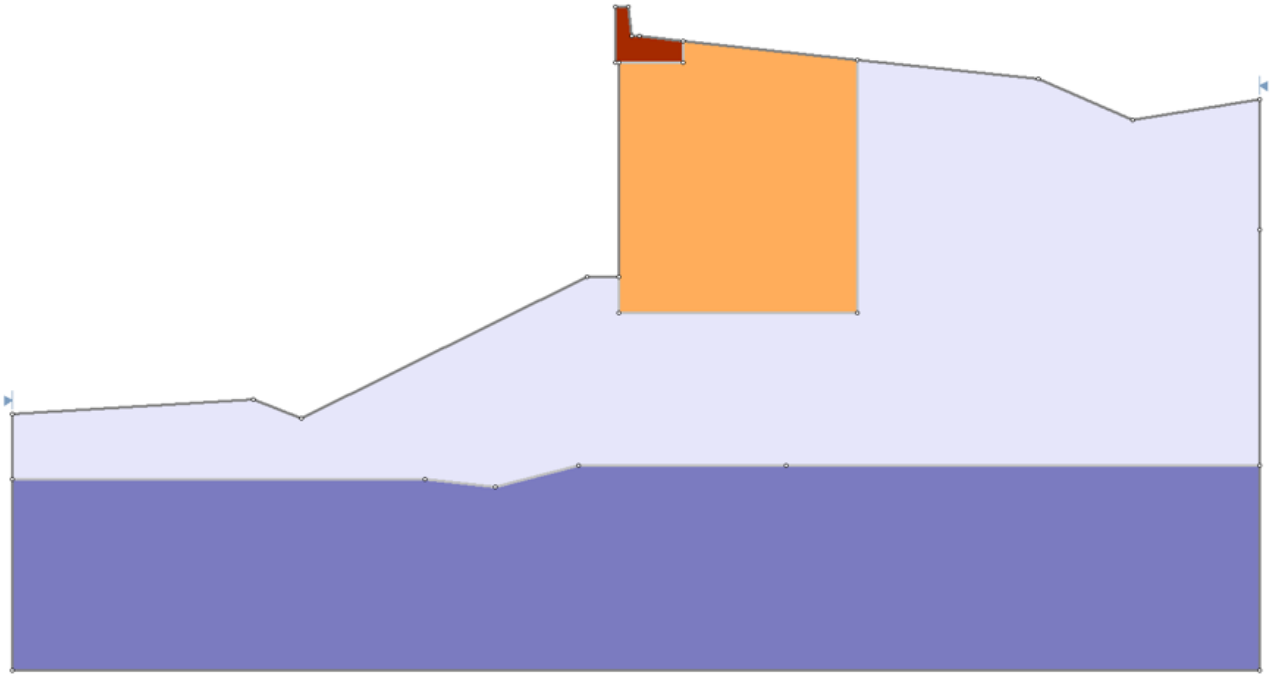
2.0 Model

An initial Slide2 model has been created for this tutorial with proper project settings and material properties. To learn more about basic modelling functions, please refer to Tutorial 01 – Quick Start.

To open the tutorial file:

1. Select **File > Recent > Tutorials** and open the file *Tutorial 37 Geosynthetic Retaining Wall_initial (no rein).slmd*

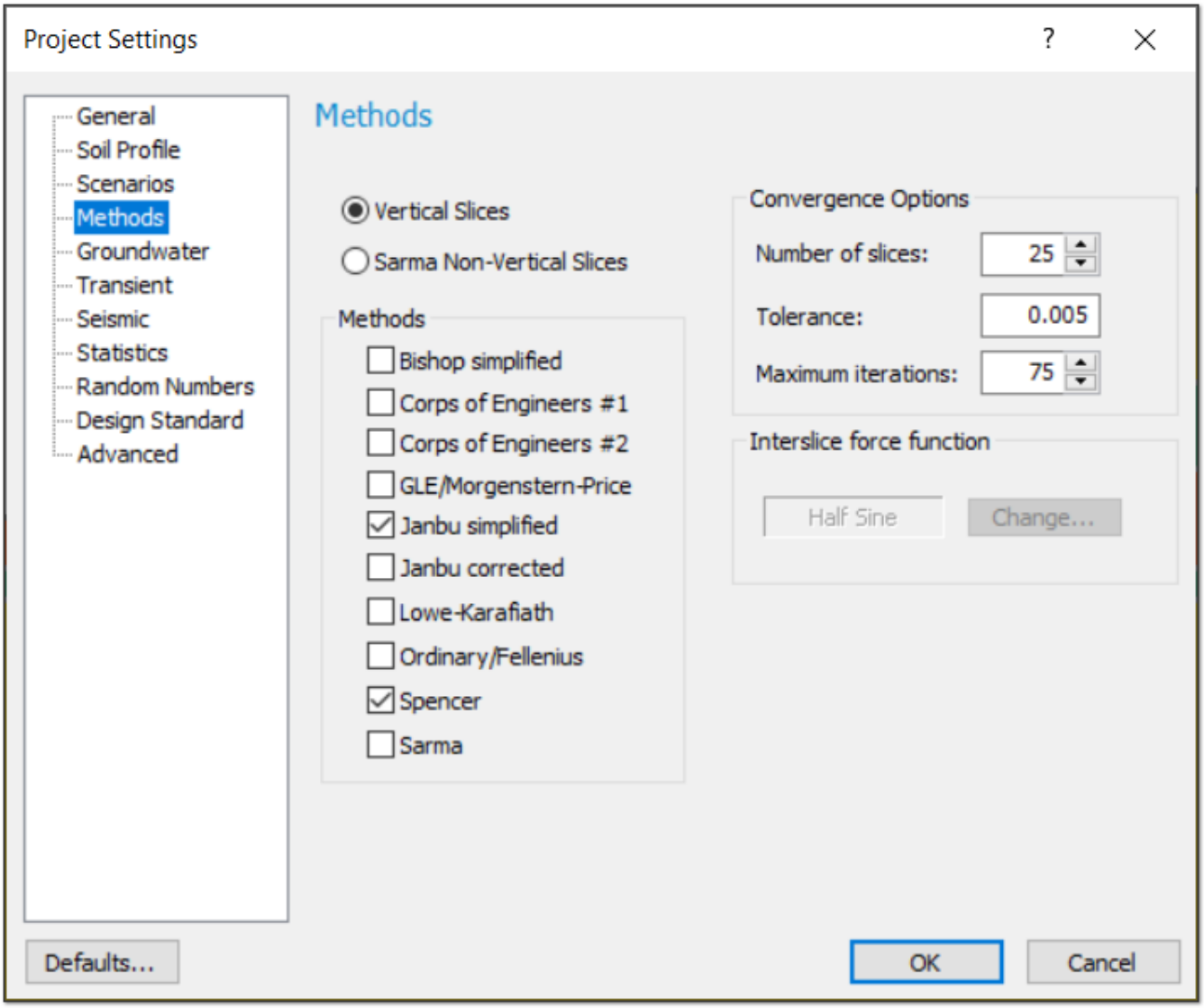
The model is shown below:



2.1 PROJECT SETTINGS

As noted above, the **Project Settings** are already selected in the starting project file. To view the Project Settings for this tutorial:

1. Select **Analysis > Project Settings** in the menu or click on the **Project Settings** icon in the toolbar.
2. Select **Methods** in the side menu.

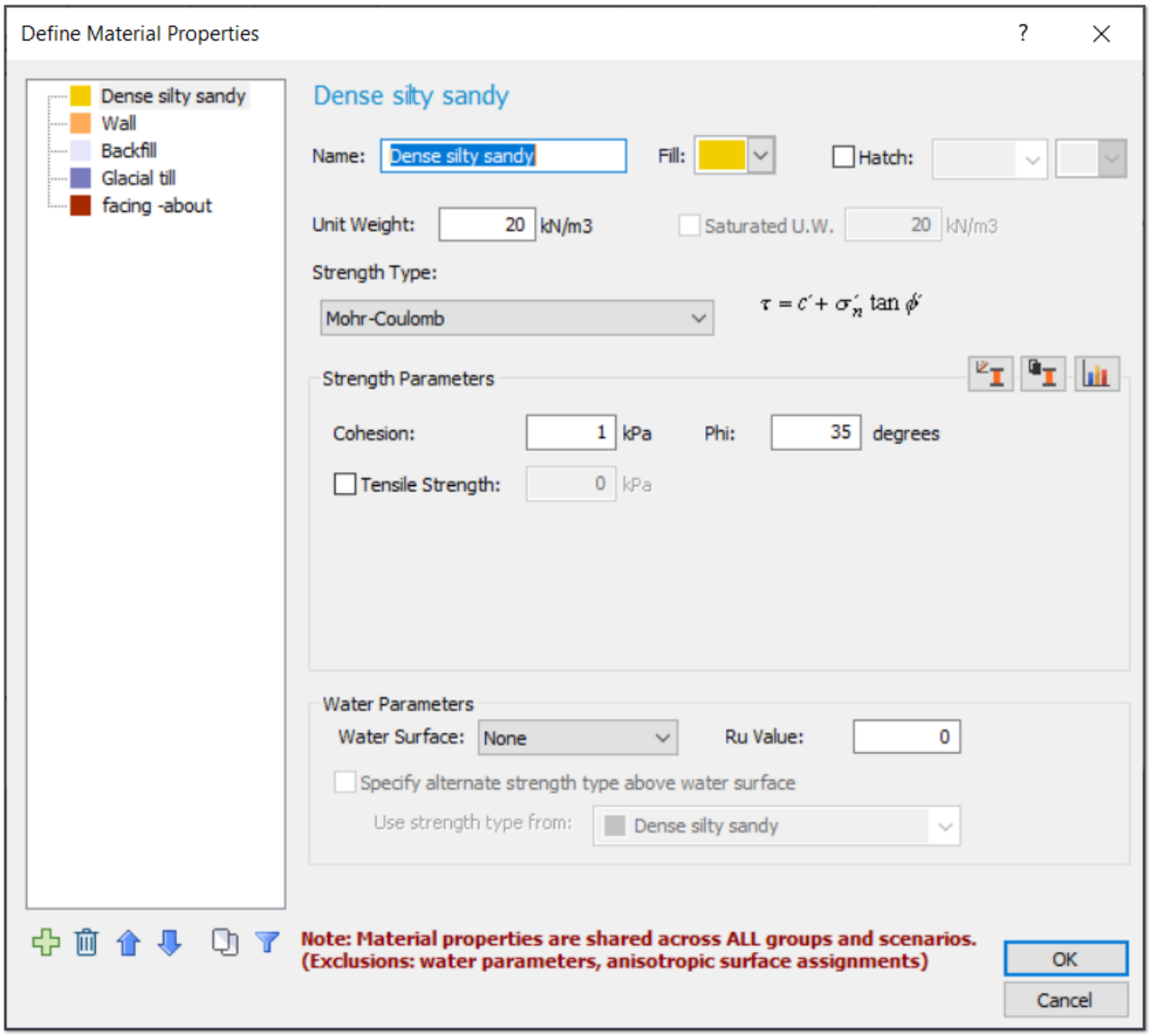


For this tutorial, the **Bishop simplified** checkbox has been cleared and the **Spencer** checkbox has been selected. All other program defaults remain in place.

2.2 MATERIAL PROPERTIES

As noted above, the **Material Properties** are already defined in the starting project file. To view the Material for this tutorial:

1. Select **Properties > Define Material Properties** in the menu or click on the **Define Material Properties** icon in the tool bar.



Five materials have been defined in the model: **Dense silty sandy, Wall, Backfill, Glacial till**, and **Facing-about**. Each material has a different unit weight and strength parameter that can be viewed in the **Define Material Properties** dialog.

2. Click **OK** or **Cancel** to close the dialog.

3.0 Define Support Properties

We will be defining 3 types of **Common Geosynthetic Supports** in this model. Each support has a different tensile strength.

Tensile strength and pullout strength, especially tensile strength, are the two main properties in determining a Geosynthetic support's strength.

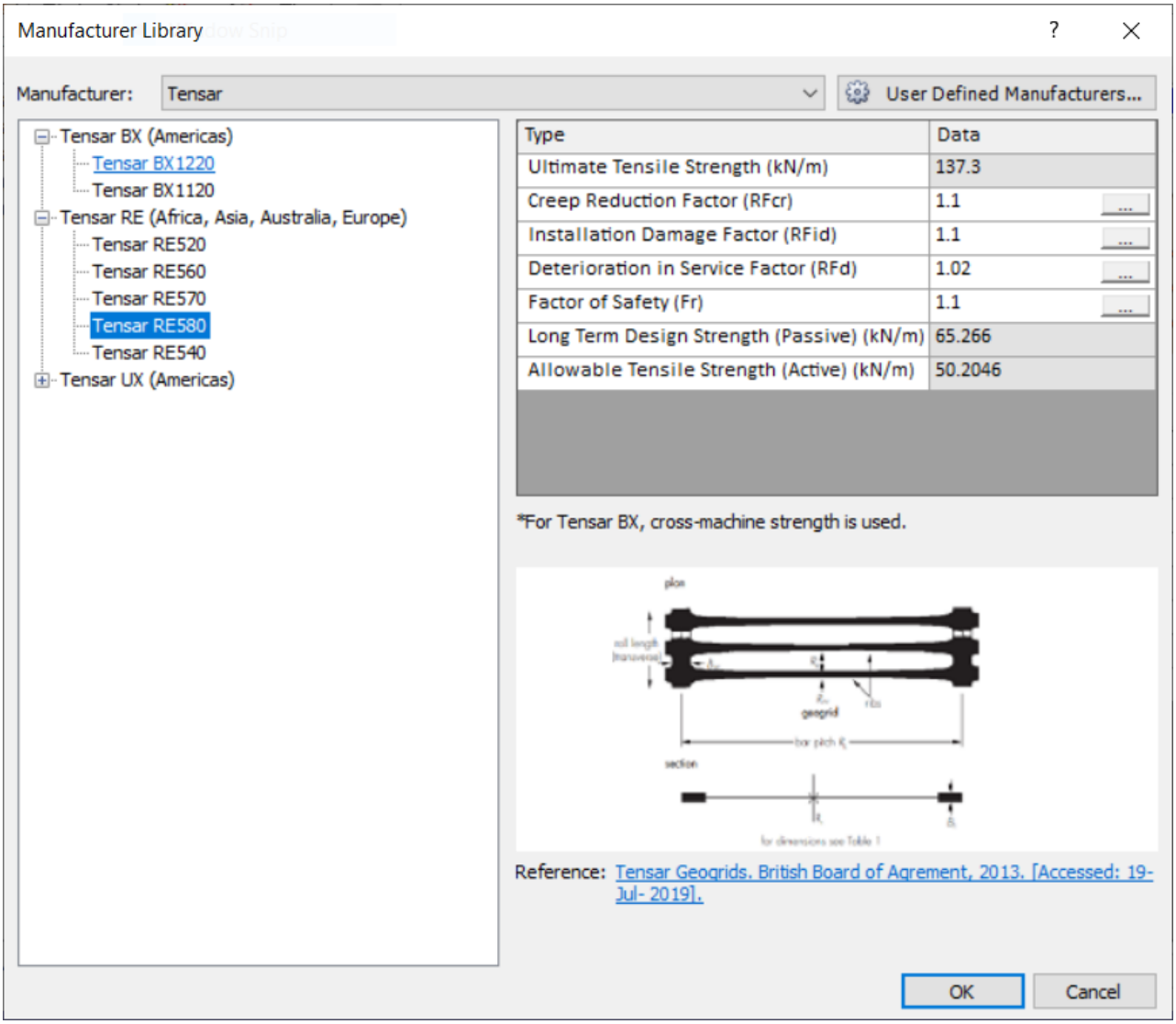
To begin defining the supports:

1. Select **Properties > Define Support** from the menu or click on **Define Support** icon in the toolbar.
2. Select **Support 1** on the left.
3. Change the name to **Wall C-type 1**.
4. Under the **Support Type** drop-down, select **Geosynthetic**.

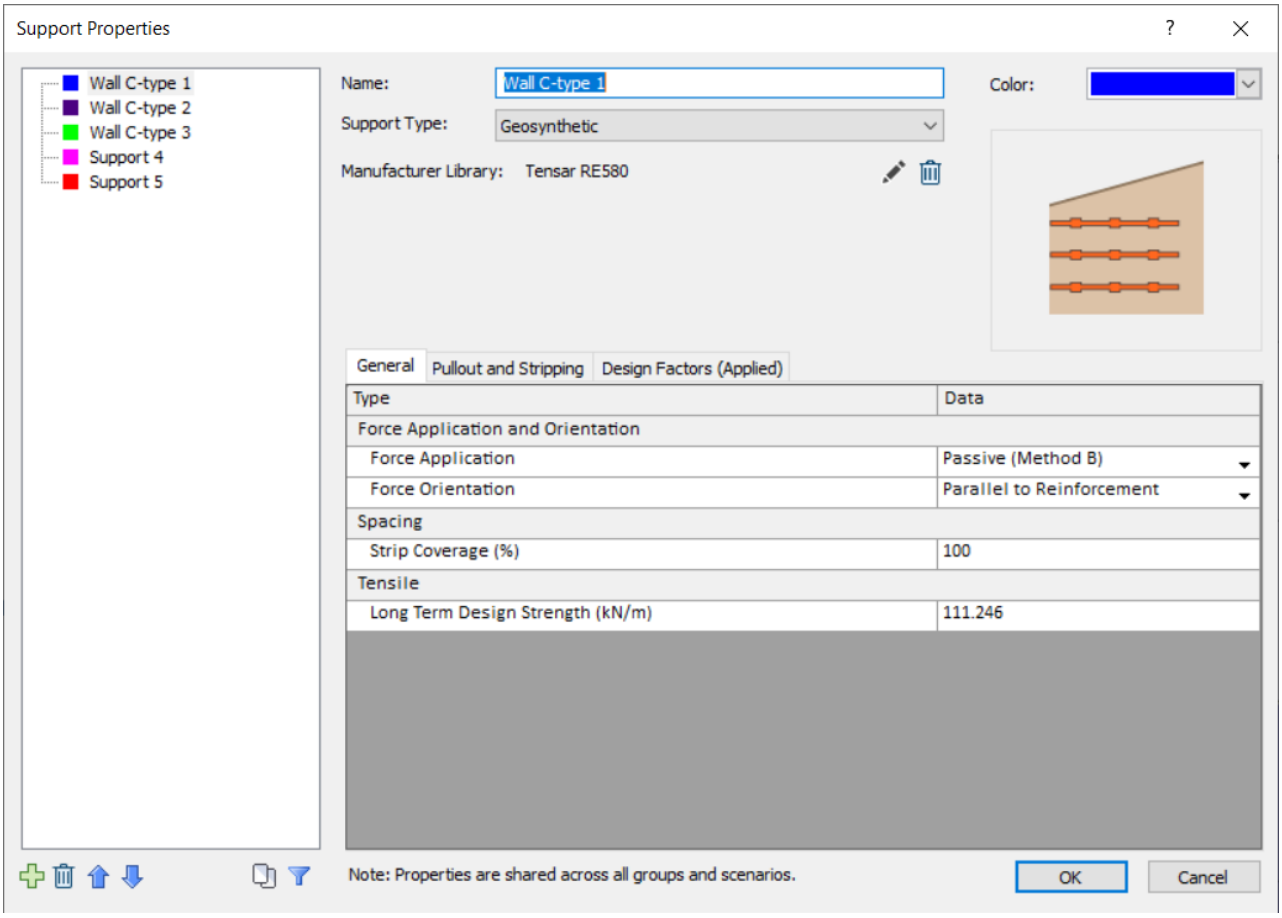
5. Click the **Manufacturer Library** button. A **Manufacturer Library** dialog will appear.
6. Select the following geosynthetic support:
    - Manufacturer = Tensar International
    - Production Set = Tensar RE
    - Type = Tensar RE580.

    On the right-hand side of the dialog, the default values for the selected support type are displayed. We will change the “Deterioration in Service Factor (RFd)” parameter to 1.02.

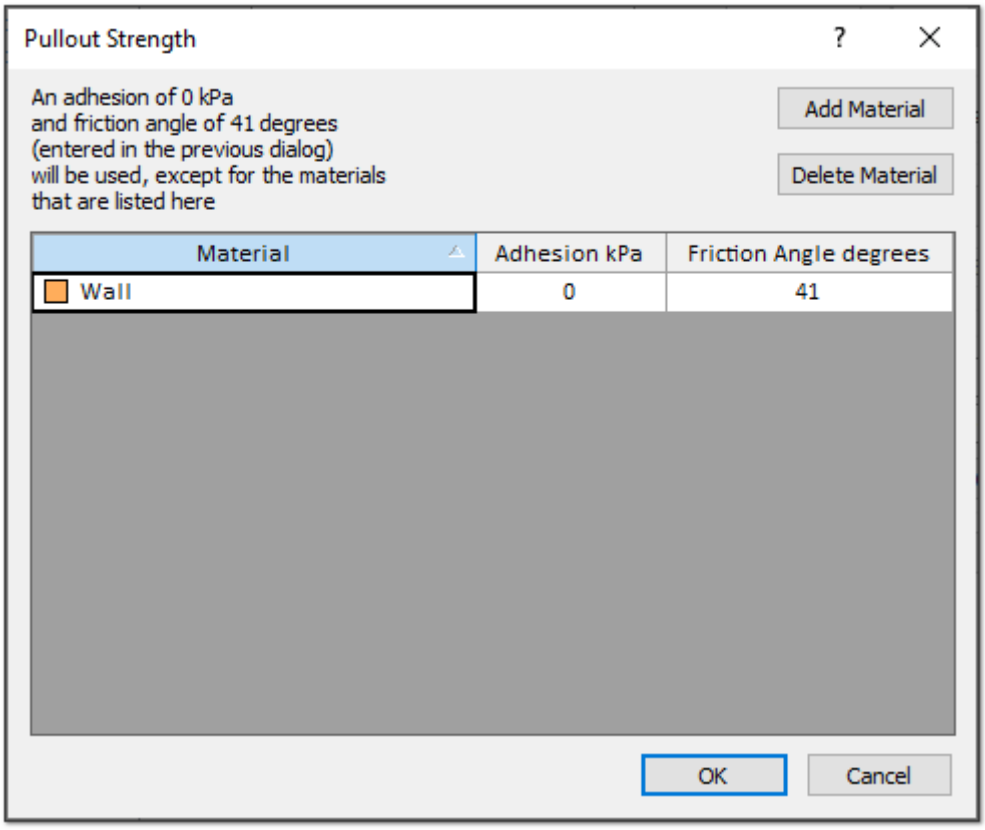
> **Note**
>
> This parameter needs to be changed for all three added supports.



7. Click **OK** to close the dialog.



8. Now in the **Support Properties** dialog, in the **General** tab, select **Force Application** = **Passive (Method B).**
9. Under the **Force Orientation** section select **Parallel to Reinforcement**.
10. Ensure **Strip coverage = 100%** and **Long Term Design Strength = 111.246 kN/m** (according to the selected support type).
11. Under the **Pullout and Stripping** tab, enter **Adhesion = 0 kPa** and **Friction Angle = 41 degrees**. Set the **Material Dependent** to **Yes** and then click the **Define** button below it. The following dialog should appear:



12. Select **Wall** as the **Material**, ensure the **Adhesion = 0 kPa** and **Friction Angle = 41 degrees**.
13. Click **OK** to close the Material Dependencies dialog.
14. Back in the **Define Support Properties** dialog, select:
    - **Shear Strength Model = Linear**
    - **Anchorage = Slope Face**
    - **Connection Strength = 111.246 kN/m** (assume same as tensile)

    Note: In the **Design Factors** tab, you can click on **Define Manufacturer Library Values** to see the Reduction Factor Calculation.

15. Make sure "**Use external loads in strength computation**" is set to **Yes**.
16. Similar to Support 1, enter the following properties for Support 2 and Support 3. Leave the other options with default settings.

	Support 1	Support 2	Support 3
Name	Wall C-type 1	Wall C-type 2	Wall C-type 3
Support Type	Geosynthetic		

Manufacturer Library	Manufacturer	Tensar International		
	Production Set	Tensar RE		
	Type	Tensar RE580	Tensar RE560	Tensar RE520
Force Application		Passive (Method B)		
Force Orientation		Parallel to Reinforcement		
Pullout Strength	Adhesion (kPa)	0		
	Friction Angle (degrees)	41		
Material Dependent		Wall		
Anchorage		Slope Face		
Connection strength (kN/m)		111.246	71.868	42.78
Shear Strength Model		Linear		
Strip Properties	Strip Coverage (%)	100		
	Long Term Design Strength (kN/m)	111.246	71.868	42.78

17. Click **OK** to close the Support Properties dialog.

4.0 Add Support Patterns

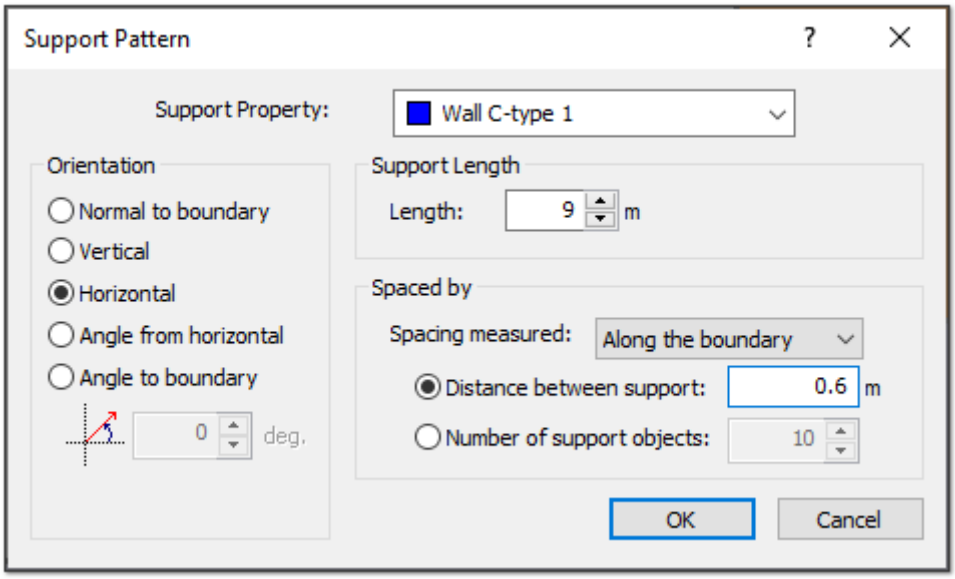
We will now add the support patterns.

1. Select the **Support** workflow tab.

2. Select **Support > Add Support Pattern** in the menu or click on the **Add Support Pattern**  icon in the toolbar. A **Support Pattern** dialog will open.

3. Enter the following:

    - Support Property = Wall C-type 1
    - Orientation = Horizontal
    - Support Length = 9 m
    - Distance between support = 0.6 m



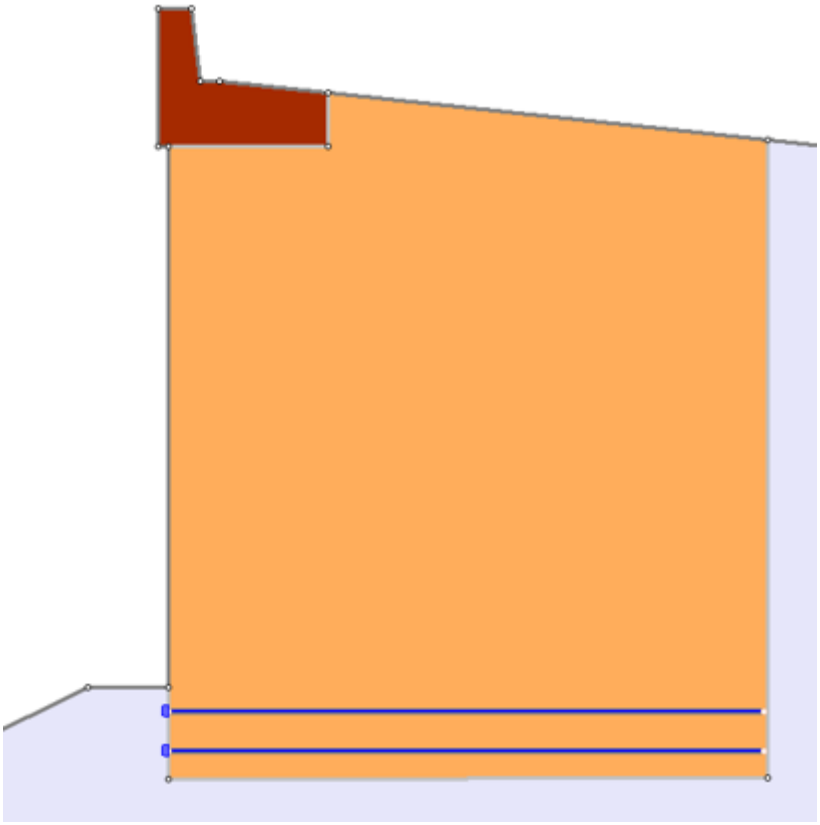
4. Select **OK** to close the dialog.

    You now should see a cursor on the screen. You can either select the position with your mouse or enter coordinates in the bottom right of the screen to place the support.

5. Enter coordinates = **{-6.896, 101.2}**. Hit **Enter** to place the support.

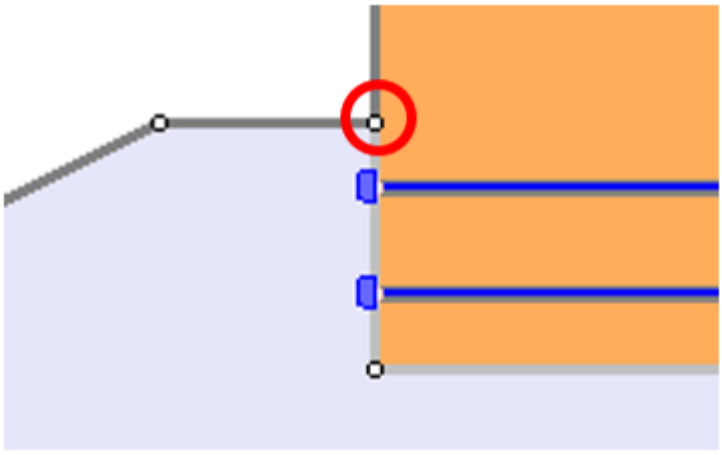
> **Note**
>
> The x coordinate = -6.896 ensures all supports are anchored along the left edge of the Wall (orange).

A geosynthetic support will be positioned at the entered coordinates. It is anchored at the left edge of the Wall (orange). As you move the mouse above the first support, second support will appear on top of it as shown in the following figure:



6. Click the left mouse button to complete the selection.

   The spacing between the two supports is 0.6 m. Supports further above the two supports cannot be extended because a boundary vertex shows within 0.6 m above the second support as circled in red below:



   To build the rest of the supports, we need to add a second support pattern.

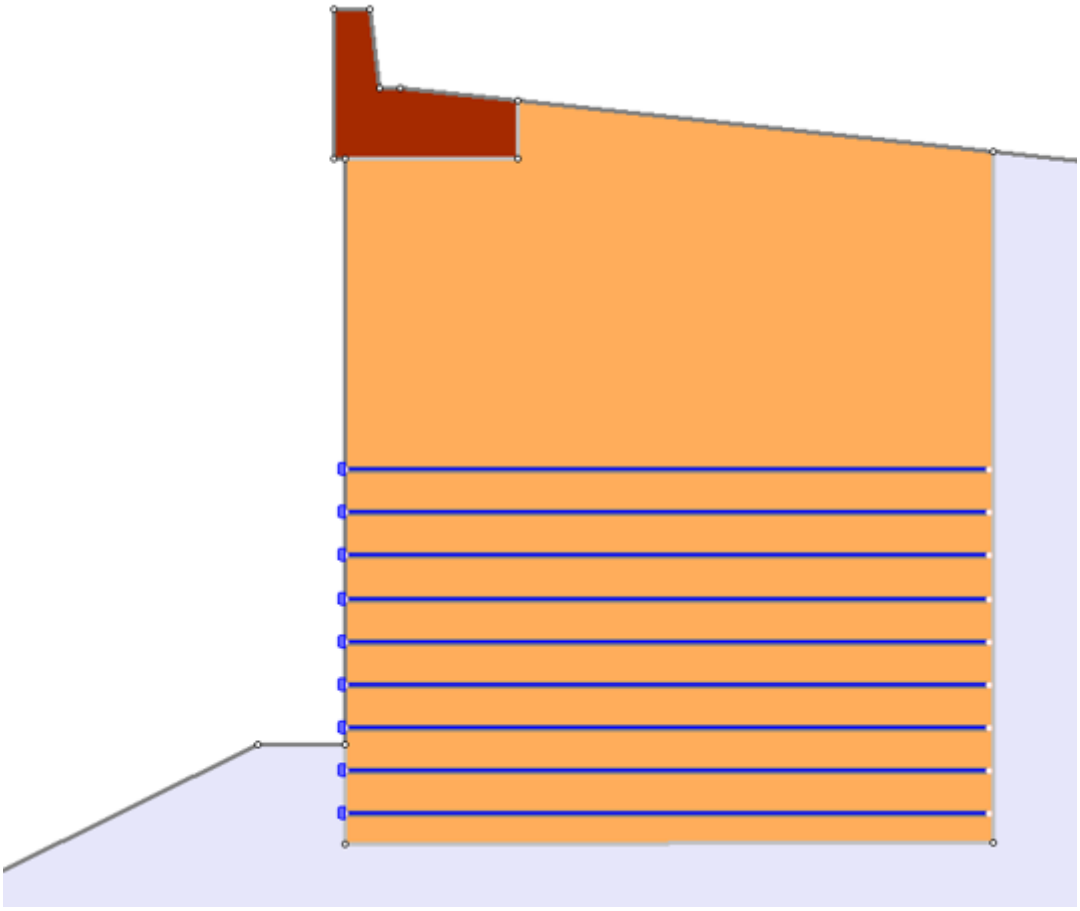
7. Select **Support > Add Support Pattern** in the menu or click on the **Add Support Pattern** icon in the toolbar. The Support Pattern dialog will open.

8. All factors should stay the same as the first time. Select **OK** to close the dialog.

   You now should see a cursor on the screen.

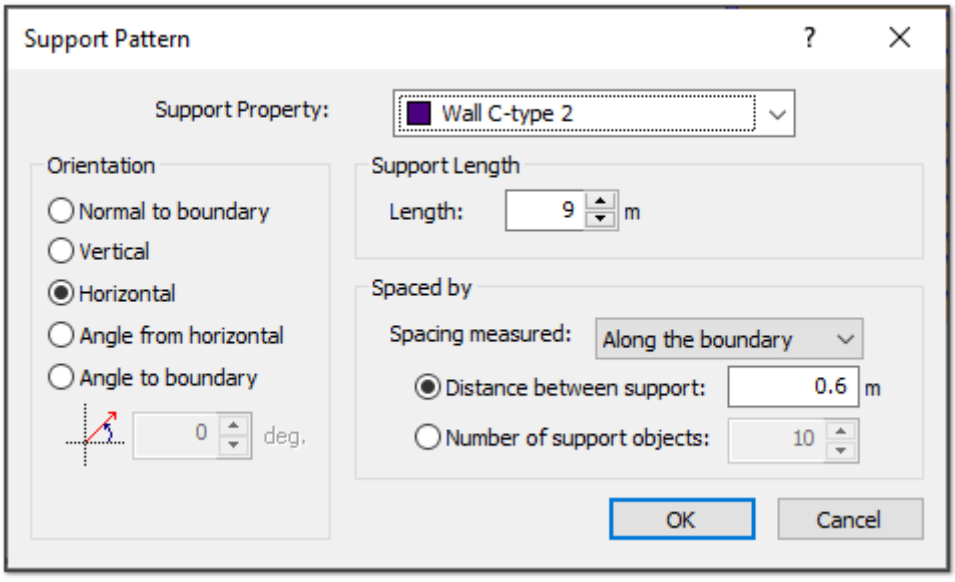
9. Enter coordinates = {-6.896, 102.4} this time and hit **Enter**.

10. Move the mouse above the first support. A series of supports will show along the boundary.
11. Click the left mouse button when a total of **seven supports** are positioned as the following:



Now we will add other types of support.

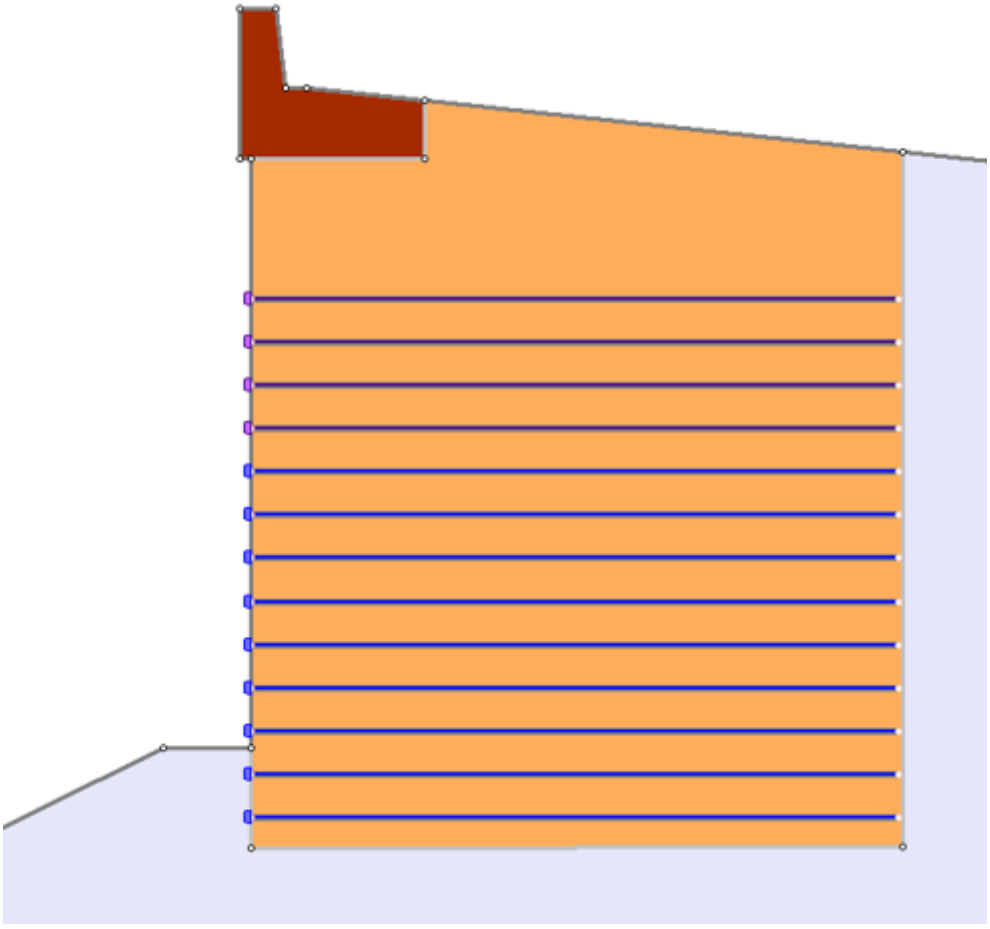
12. Select **Support > Add Support Pattern**.



13. Enter the following:
    - Support Property = Wall C-type 2

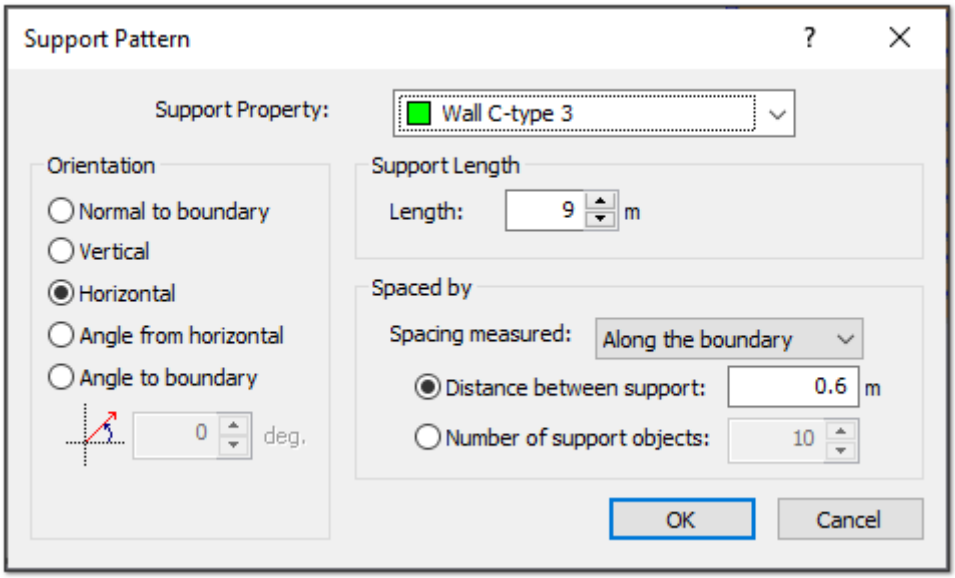
- Orientation = Horizontal
- Length = 9 m
- Distance between support = 0.6 m

14. Click **OK** to close the dialog.
15. Enter coordinates = **{-6.896, 106.6}**.
16. Move the mouse above the support. When a total of **four supports** are positioned (as shown in the figure below), click the left mouse button to place them.

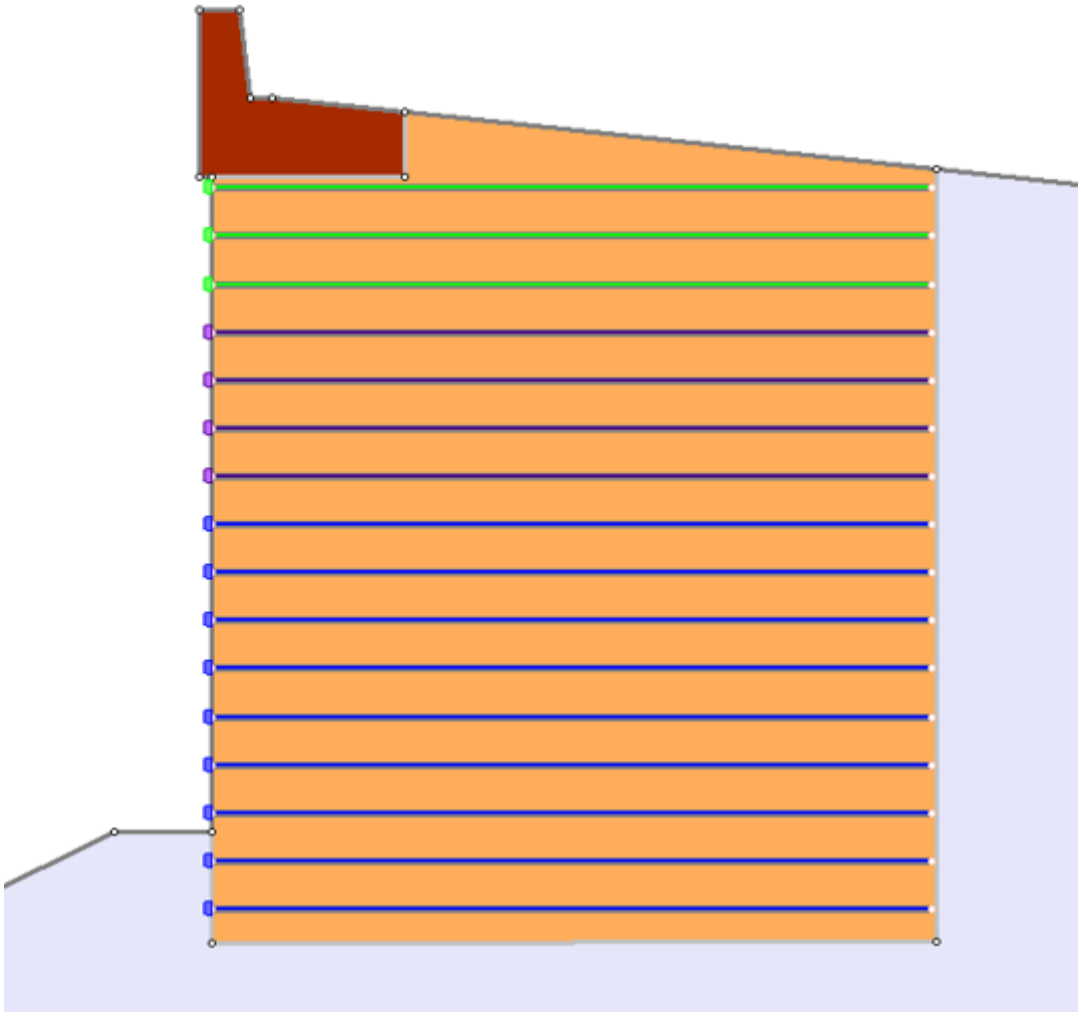


We will now add the third support property.

17. Select **Support > Add Support Pattern.**



18. Enter the following values:
    - Support Property = Wall C-type 3
    - Orientation = Horizontal
    - Length = 9 m
    - Distance between support = 0.6 m
19. Click **OK** to close the dialog.
20. Enter coordinates = **{-6.896, 109.0}**. Move the mouse above the support. When **two supports** are positioned above the support (as shown below), click the left mouse button to place.



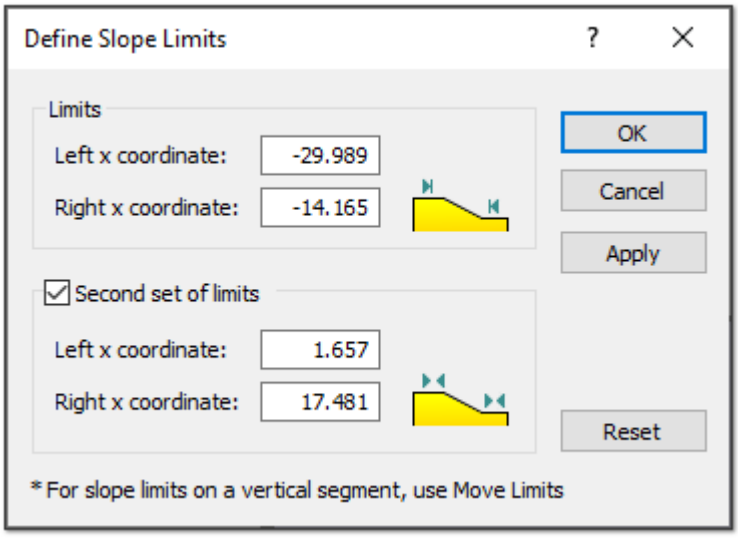
5.0 Slip Surface Boundaries

By default, the Slide2 model has generated a slope limit based on input geometry. This defines the **slip surface boundaries**.

In order to avoid a shallow slip surface failure, a **secondary set of limits** is needed. It further defines the boundaries of a slip surface.

To define the second set of limits:

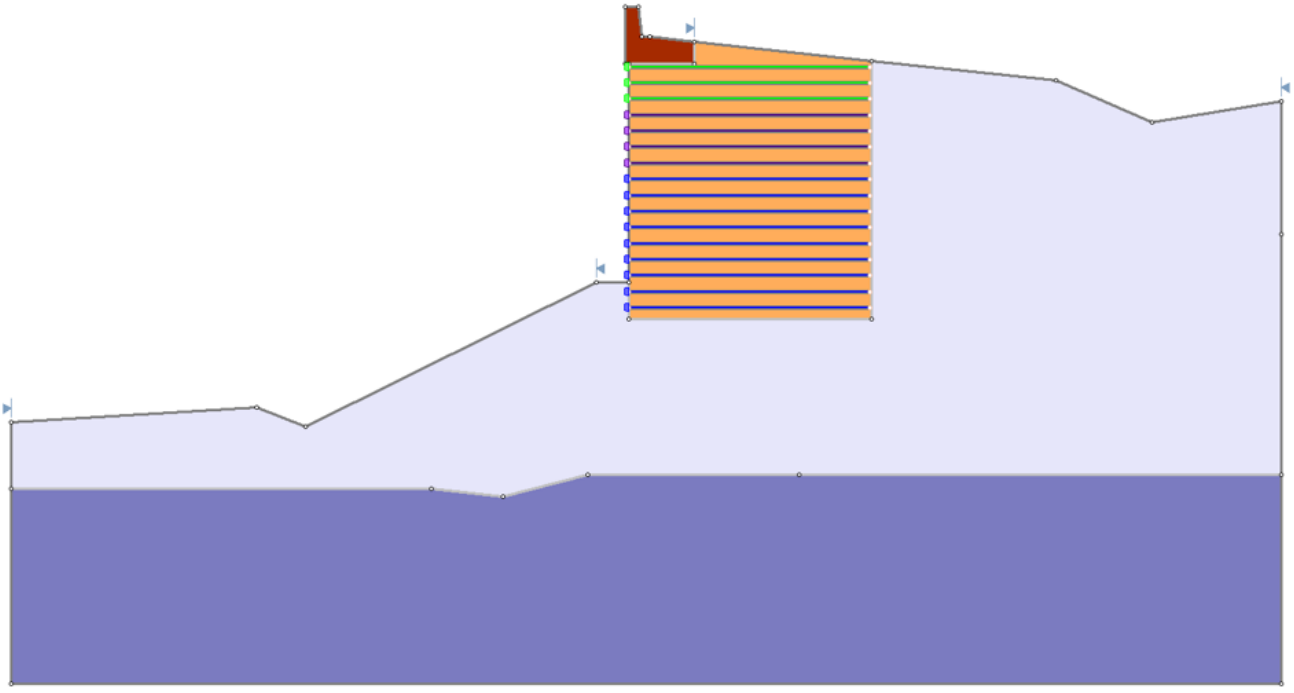
1. Right-click on the primary slope limit select Define Limits (or select **Surfaces > Slope Limits > Define Limits**).



2. In the**Define Slope Limits** dialog, select the checkbox for **Second set of limits**. Leave coordinates as the defaults for now.
3. Click **OK** to close the dialog.

Now a secondary set of limits should display.

1. Right-click on one secondary slope limit. Select **Move To**.
2. Use the mouse to drag the slope limit to the edge of the wall. Hit **Enter** to complete.
3. Repeat the same for the other secondary slope limit. When completed, your model should look like the following:

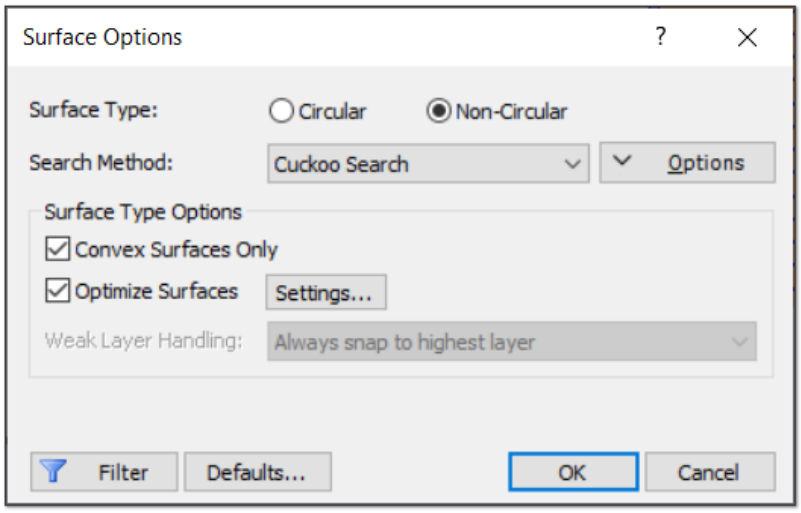


6.0 Surface Options

In this tutorial, we will be performing the **Cuckoo Search Method (Non-Circular Surface)** to generate slip surfaces.

1. Select the **Surfaces** workflow tab.

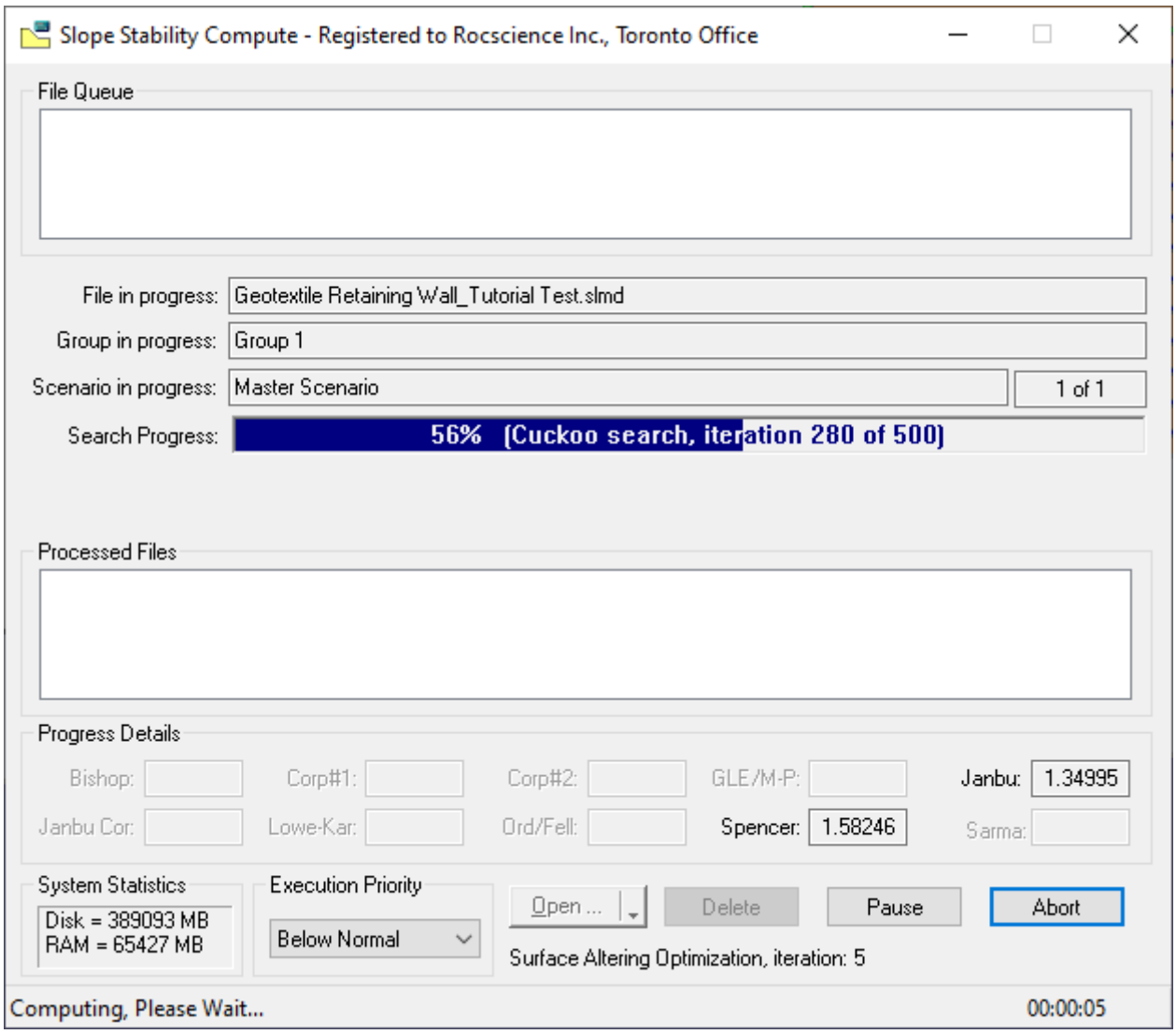
2. Select **Surfaces > Surface Options** in the menu or click on the **Surface Options**  icon in the toolbar.



3. Under**Surface Type Section**, select **Non-Circular**.
4. Keep **Search Method = Cuckoo Search**.
5. Leave other options as default settings.
6. Click **OK** to close the dialog.

7.0 Compute

1. Select **File > Save As**, save the file as a separate file.
2. Select **Analysis > Compute** in the menu or click on the **Compute**  icon in the toolbar.



8.0 Interpret

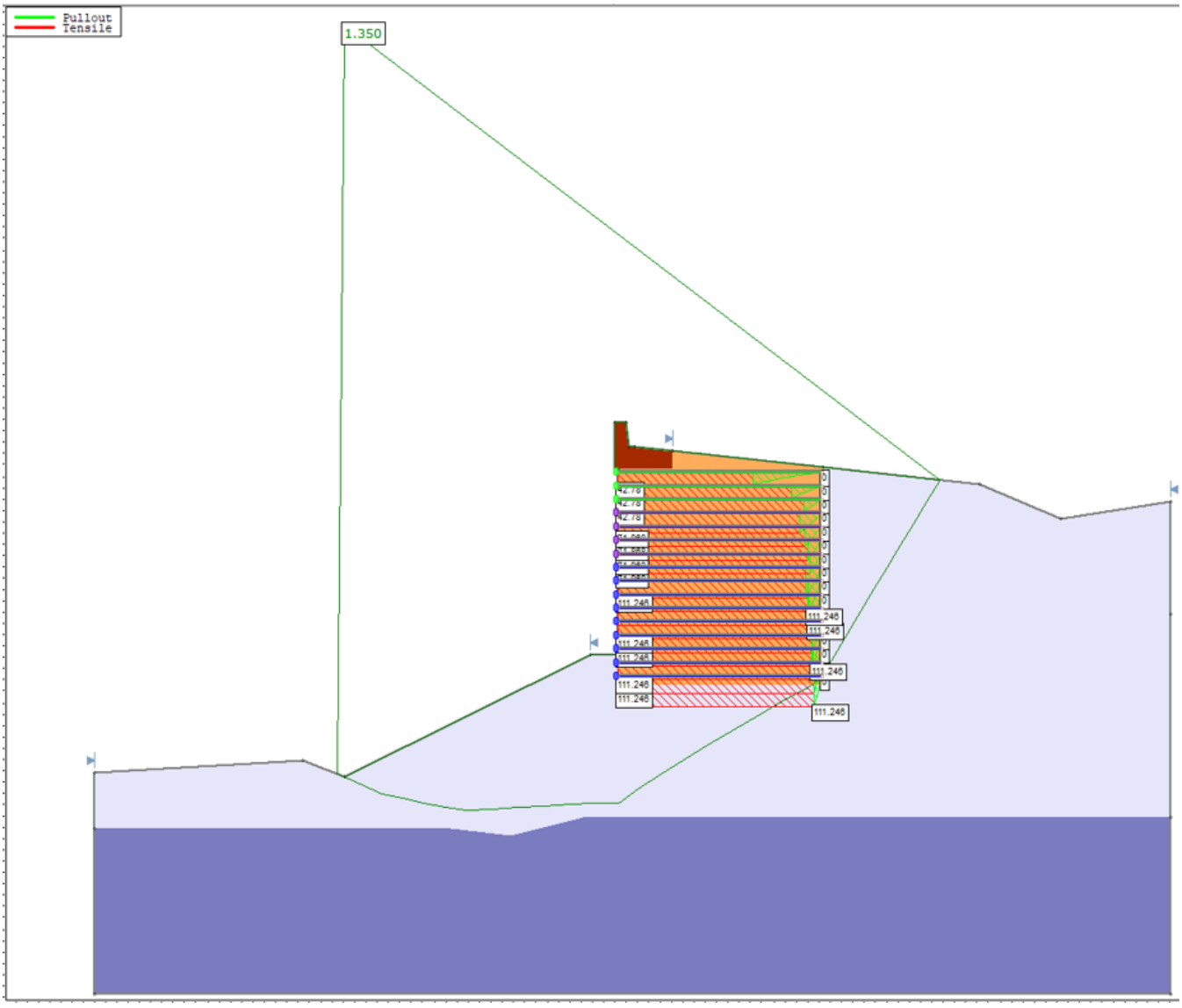
1. Select **Analysis > Interpret** in the menu or click on the **Interpret** icon in the toolbar.

   The result will be displayed in the Slide2 Interpret program.

   In the toolbar, the analysis method can be selected. In this case, it can be either **Janbu simplified** or **Spencer**.

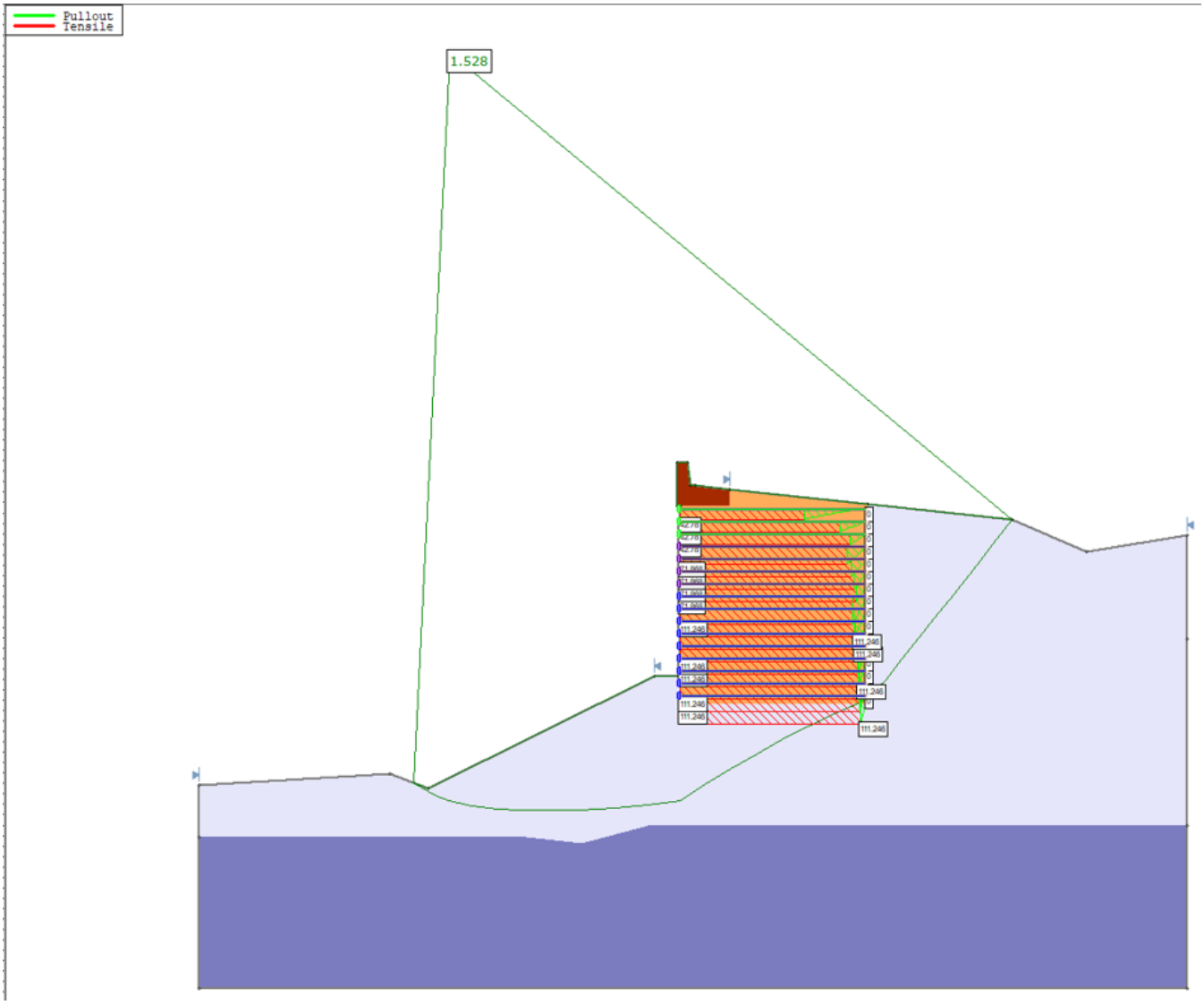
2. In the top toolbar, click on the **Show Support Force Diagram** icon to display the force for each support.

With **Janbu Method**, the following results are shown:



The Global Minimum slip surface is displayed. The minimum factor of safety is **1.350**.

With **Spencer Method**, the following results are shown:



The Global Minimum slip surface is displayed. The minimum factor of safety is **1.528.**

This concludes the tutorial on using Common Geosynthetic Supports in your model. You may now close the Slide2 program.

References

Yu, Y., Bathurst, R.J., & Allen, T.M. (2016) *Numerical Modeling of the SR-18 Geogrid Reinforced Modular Block Retaining Walls*. Journal of Geotechnical and Geoenvironmental Engineering. 142:5