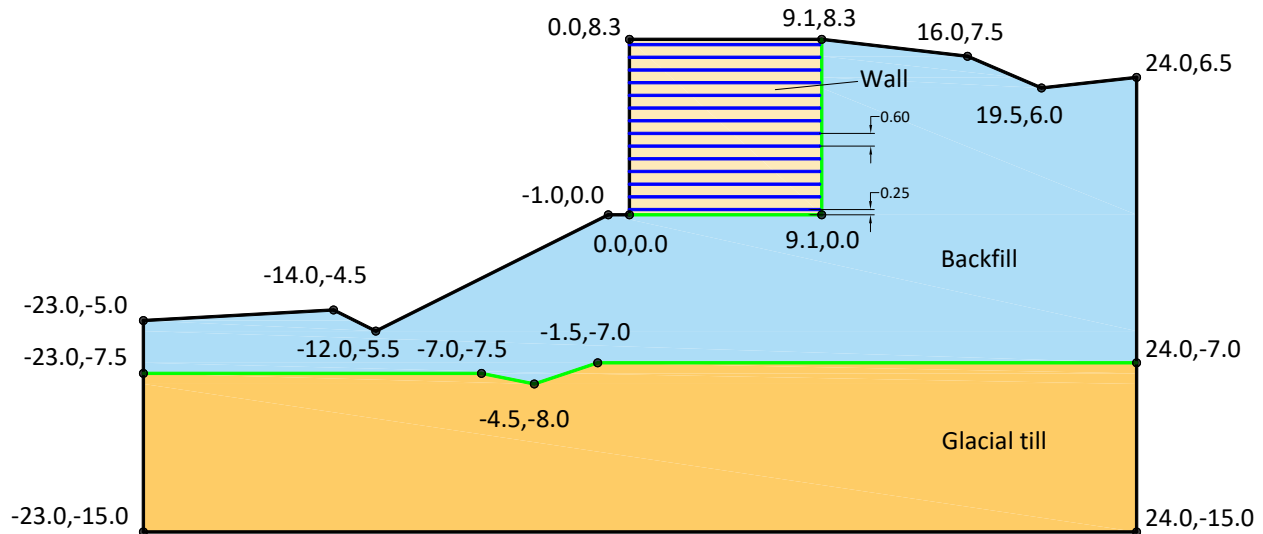


Stability of Geosynthetic Reinforced Retaining Wall

Created By: [Roozbeh Geraili Mikola, PhD, PE](#)

Email: hyrcan4geo@outlook.com

Web Page: www.geowizard.org



This tutorial will demonstrate the modeling of a geosynthetic reinforced retaining wall in **HYRCAN**. Five different types of slope reinforcement can be modeled in **HYRCAN**, including soil nails, geosynthetics, tiebacks, end anchored rock bolts and piles.

Project Settings

Various important modeling and analysis options are set in the Project Settings dialog, including Failure Direction, Units of Measurement, Analysis Methods and Groundwater property. For this analysis make sure the failure direction is set to "Right to Left" then press Apply. From the **Methods** tab you also will be able to select the limit equilibrium analysis method(s) to be used for the analysis.

Select: *Analysis* →



Project Settings

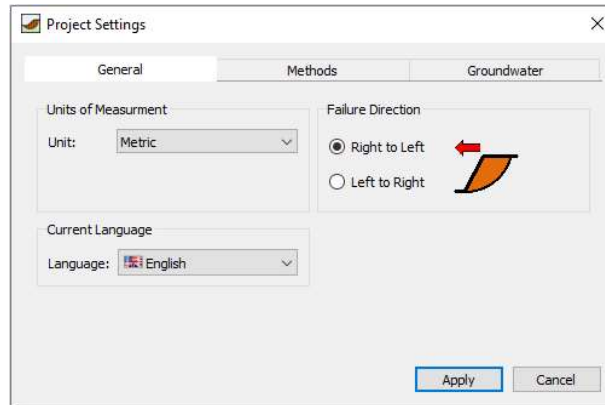
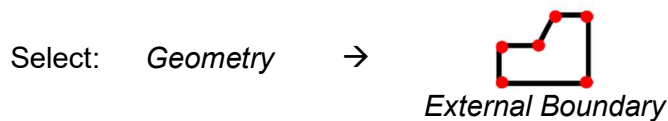


Figure 1- Project Settings dialog.

Create Geometry

- **External Boundaries**

The first boundary that must be defined for every model, is the external boundary. To add the external boundary, select **External Boundary** from the toolbar or the Boundaries menu.



Enter the following coordinates in the prompt line at the bottom right of the main window.

Enter vertex [esc=cancel]: -23.0,-5.0	Enter vertex [c=close,esc=cancel]: 16.0,7.5
Enter vertex [esc=cancel]: -14.0,-4.5	Enter vertex [c=close,esc=cancel]: 19.5,6.0
Enter vertex [esc=cancel]: -12.0,-5.5	Enter vertex [c=close,esc=cancel]: 24.0,6.5
Enter vertex [c=close,esc=cancel]: -1.0,0.0	Enter vertex [c=close,esc=cancel]: 24.0,-15.0
Enter vertex [c=close,esc=cancel]: 0.0,0.0	Enter vertex [c=close,esc=cancel]: -23.0,-15.0
Enter vertex [c=close,esc=cancel]: 0.0,8.3	Enter vertex [c=close,esc=cancel]: c
Enter vertex [c=close,esc=cancel]: 9.1,8.3	

Note that entering **c** after the last vertex has been entered, automatically connects the first and last vertices (closes the boundary) and exits the External Boundary option. Your screen should now look as follows:

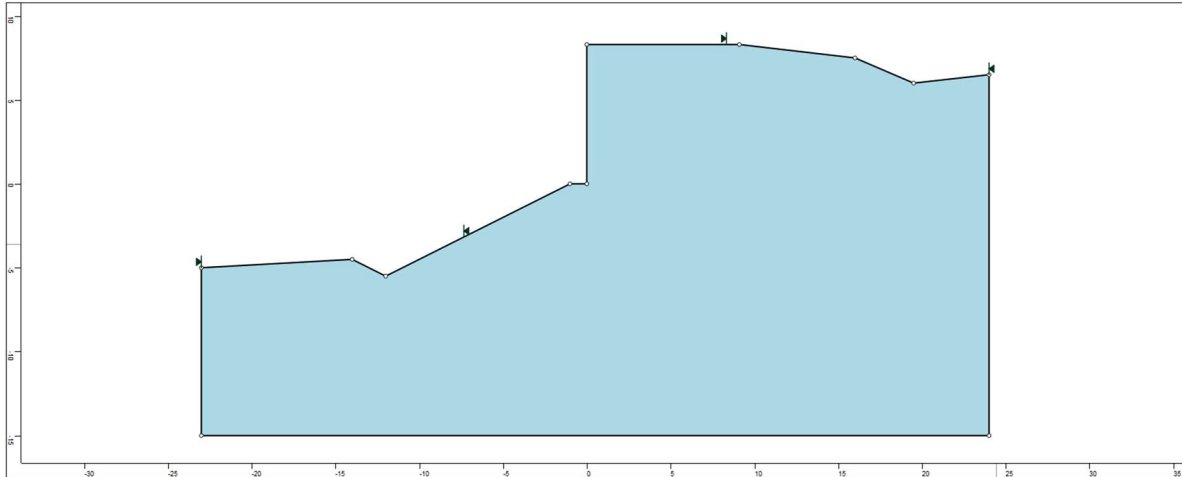



Figure 2- External boundary is created.

- **Material Boundaries**

Material boundaries are used in **HYRCAN** to define the boundaries between different material regions within the External Boundary. Let's add two material boundaries.

Select: **Geometry** → 
Material Boundary

Enter the following coordinates in the prompt line at the bottom right of the main window.

```
Enter vertex [esc=cancel]: -23.0,-7.5
Enter vertex [d=done,esc=cancel]: -7.0,-7.5
Enter vertex [d=done,esc=cancel]: -4.5,-8.0
Enter vertex [d=done,esc=cancel]: -1.5,-7.0
Enter vertex [d=done,esc=cancel]: 24.0,-7.0
Enter vertex [d=done,esc=cancel]: d
```

Repeat the sequence and enter the following coordinates in the prompt line.

```
Enter vertex [esc=cancel]: 0.0,0.0
Enter vertex [d=done,esc=cancel]: 9.1,0.0
Enter vertex [d=done,esc=cancel]: 9.1,8.3
Enter vertex [d=done,esc=cancel]: d
```

Your screen should now look as follows:

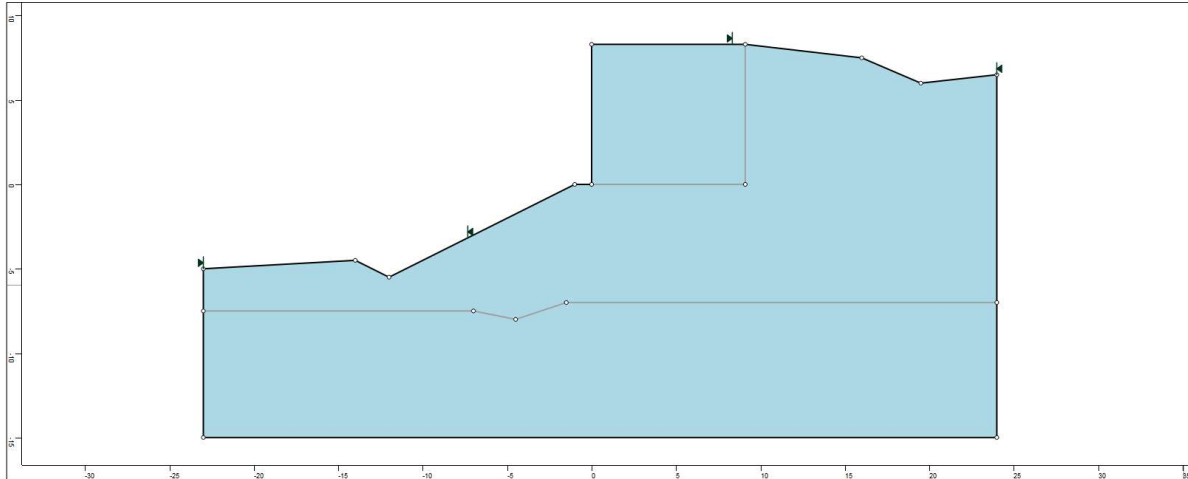



Figure 3- External and material boundaries added.

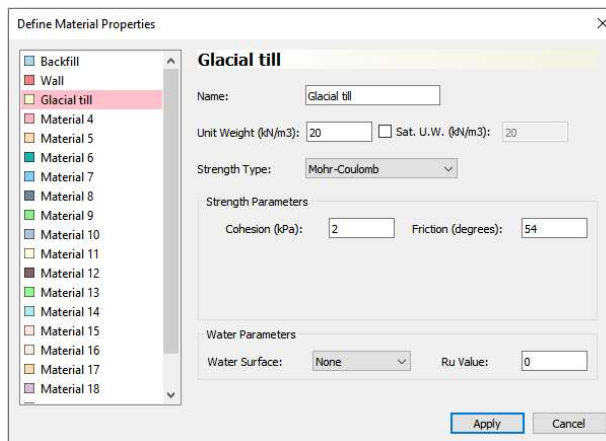
Properties

It's time to define our material properties. Select **Define Materials** from the toolbar or the Properties menu.

Select: *Properties* →  *Define Materials*

Material	γ (kN/m ³)	c (kN/m ²)	ϕ (degrees)
Backfill	21.7	1.0	33.0
Wall	20.4	0.0	41.0
Glacial till	20	2.0	54.0

With the first (default) tab selected in the Define Materials dialog, enter the following properties:



Define Material Properties

Backfill
 Wall
Glacial till
 Material 4
 Material 5
 Material 6
 Material 7
 Material 8
 Material 9
 Material 10
 Material 11
 Material 12
 Material 13
 Material 14
 Material 15
 Material 16
 Material 17
 Material 18

Glacial till

Name: Glacial till

Unit Weight (kN/m³): 20 ☐ Sat. U.W. (kN/m³): 20

Strength Type: Mohr-Coulomb

Strength Parameters

Cohesion (kPa): 2 Friction (degrees): 54

Water Parameters

Water Surface: None Ru Value: 0

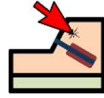
Apply Cancel

Enter the parameters shown above. When all parameters are entered press Apply.

Assigning Properties

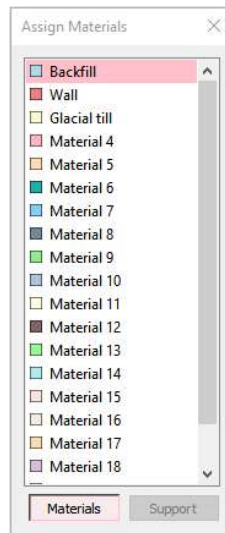
Since we have defined multiple materials, it will be necessary to assign properties to the correct regions of the model, using the Assign Properties option. Select **Assign Properties** from the toolbar or the Properties menu.

Select: *Properties* →



Assign Properties

You will see the **Assign Materials** dialog, shown below.



To assign properties to the soil layers:

1. Use the mouse to select the soil material, in the Assign Materials dialog (notice that the material names are the names you entered in the **Define Material Properties** dialog).
2. Now place the cursor anywhere in the soil region and click the left mouse button. Repeat the same sequence for other soil materials until all the materials are assigned.

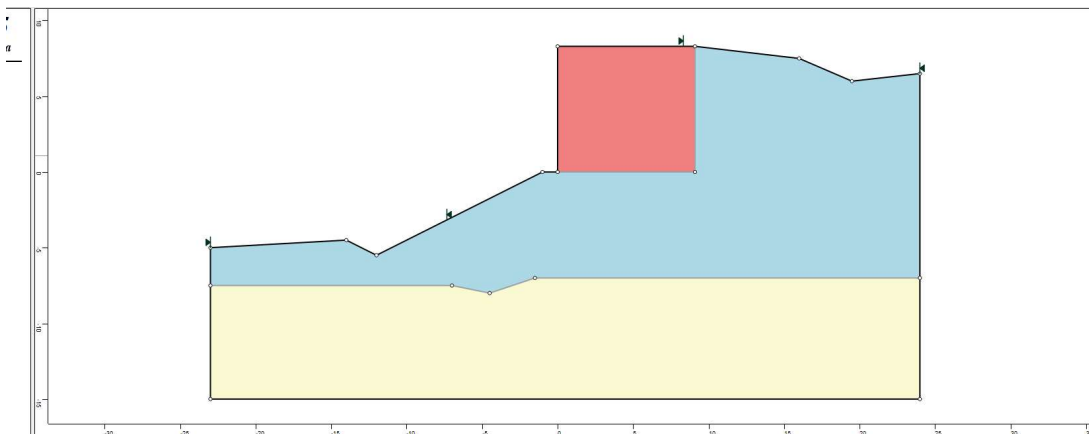


Figure 4- Geometry setup after assigning the properties.

Add Support Pattern

Before adding the support let's first define the support type and corresponding properties. To do so, select **Define Support** from the toolbar or the Properties menu.

Select: *Properties* →

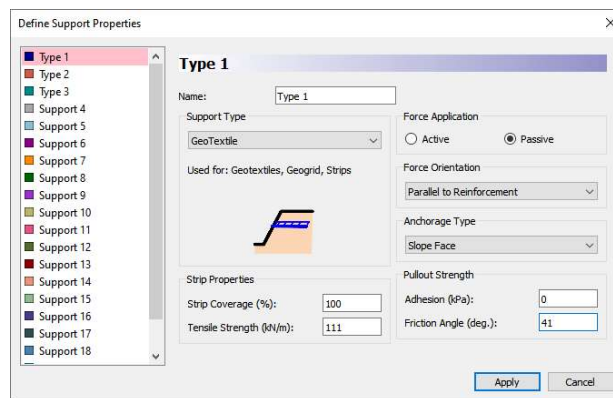


Define Support

With the first (default) tab selected in the **Define Materials** dialog, enter the following properties:

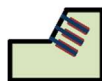
Support Type	Name	Strip Coverage (%)	Tensile Strength (kN/m)	Adhesion (kPa)	Friction Angle (deg.)
Geotextile	Type 1	100	111	0	41
	Type 2		71		
	Type 3		42		

Please Note that, the “**Force Application**” and “**Force Orientation**” for all three support types are “**Passive**” and “**Parallel to Reinforcement**” respectively.



Now let's add Support elements. Support elements can be added to a model individually, with the Add Single Support option in the Support menu. If multiple support elements in a regular pattern are to be added, you can use the Add Support Pattern option in the Support menu. In this tutorial We will use the Add Support Pattern option, to add a uniformly spaced support pattern to the slope.

Select: *Support* →



Add Support Pattern

You will first see the Support Pattern dialog. Select support property “**Type 1**” and set the Orientation = Horizontal, Length = 9.0, and Distance between supports = 0.6 as shown in Figure 5 then press Apply. Now as you move the cursor, you will see a small black cross which follows the cursor and snaps to the nearest point on the nearest boundary. You can enter the location of start and end points of the pattern graphically, on the external boundary. However, to enter exact coordinates, it is easier and more accurate in this case to enter the coordinates in the prompt line. Let's enter the first and second points on the boundary to install the first seven geotextiles:

Enter first point on boundary [esc=cancel]: 0,0.25
Enter second point on boundary [esc=cancel]: 0,3.9

Repeat the sequence to install the next four geotextiles (i.e. **Type 2**) and enter the points as follow:

Enter first point on boundary [esc=cancel]: 0,4.45
Enter second point on boundary [esc=cancel]: 0,6.5

And for last three geotextiles (i.e. **Type 3**):

Enter first point on boundary [esc=cancel]: 0,6.85
Enter second point on boundary [esc=cancel]: 0,8.3

Note that **Add Single Support** could be used instead of **Add Support Pattern** to install the geotextiles individually.

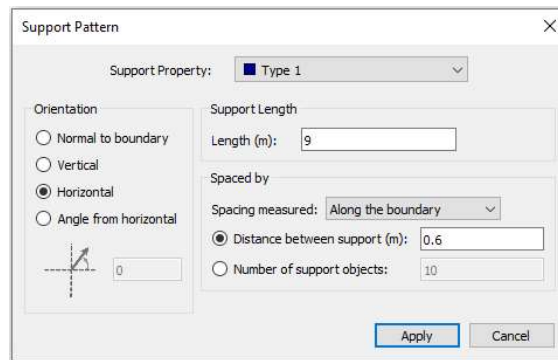


Figure 5- Add Support Pattern dialog.

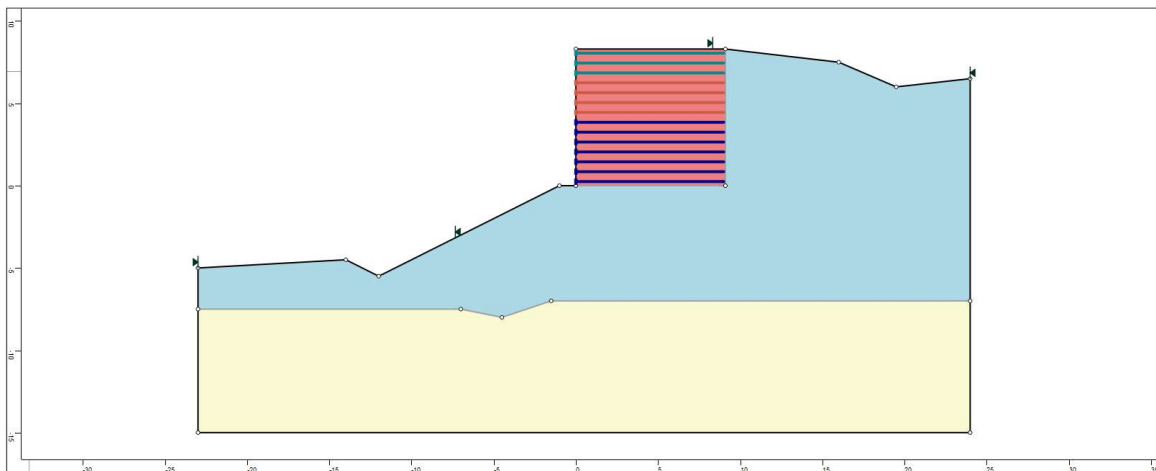
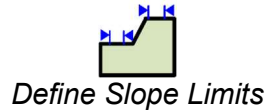


Figure 6- Geometry setup with installed geotextiles.

Modifying the Slope Limits

The Slope Limits are automatically calculated by **HYRCAN** as soon as the **External Boundary** is created. If you wish to narrow the Slope Search to more specific areas of the model, the Slope Limits can be customized with the Define Limits dialog.

Select: *Surfaces* →



Define Slope Limits

Limits

Left x coordinate: -14

Right x coordinate: -1

☒ Second set of limits

Limits

Left x coordinate: 2

Right x coordinate: 19.5

Apply

Cancel

In this tutorial the left and right coordinates are set to -14 and -1 and the left and right coordinates of the second set of limits are set to 2 and 19.5. Later on, by refining the slope limits you will be able to estimate more accurate global minimum slip surface. We are now finished creating the model, and can proceed to run the analysis and interpret the results.

Compute

The model is now ready to run.

Select: *Analysis* →



The engine will proceed in running the analysis. When completed, you are ready to view the results in Result Tab.

Results and Discussions

When calculation completed, you are ready to view the results in Result Tab. By default, when Result Tab opened, the Global Minimum slip surface, for the first activated limit equilibrium analysis method will be shown. This resulted in total of approximately 5000 trial slip surfaces. The results of the factor of safety calculation are shown in Figure 7. Table 1 summarize the comparisons of calculated factor of safety for the same model using **Slide2**, a well-known commercial program.

Table 1- Comparison of Minimum Factor of Safety

Method	Slide2	HYRCAN
Bishop Simplified	1.493	1.459
GLE/Morgenstern-Price	1.495	1.459
Janbu Simplified	1.328	1.323
Spencer	1.494	1.458

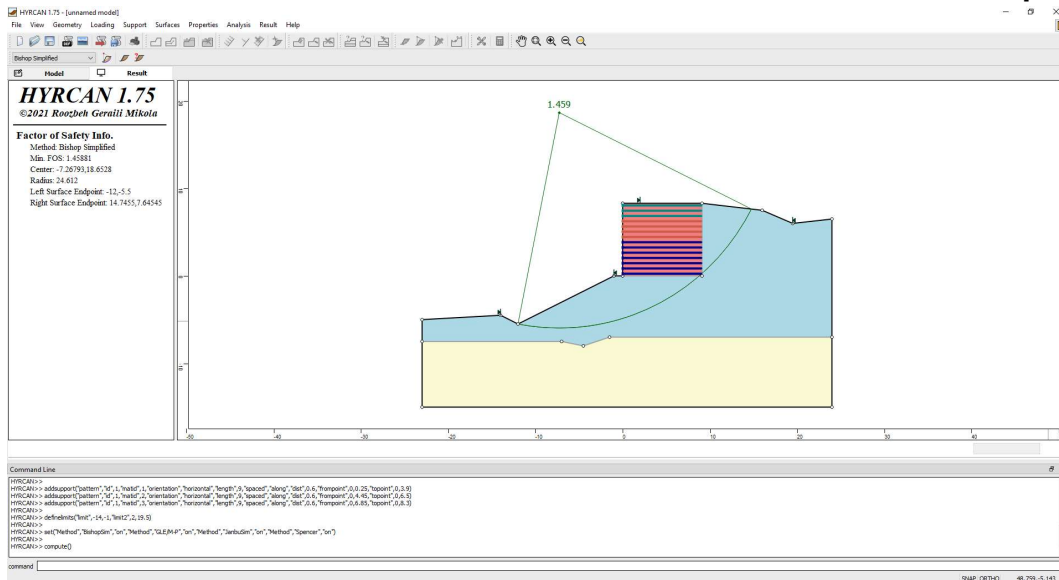


Figure 7- Result of automatic slope search.

To view ALL valid slip surfaces generated by the analysis, select the **All Surfaces** option from the toolbar or the Result menu.

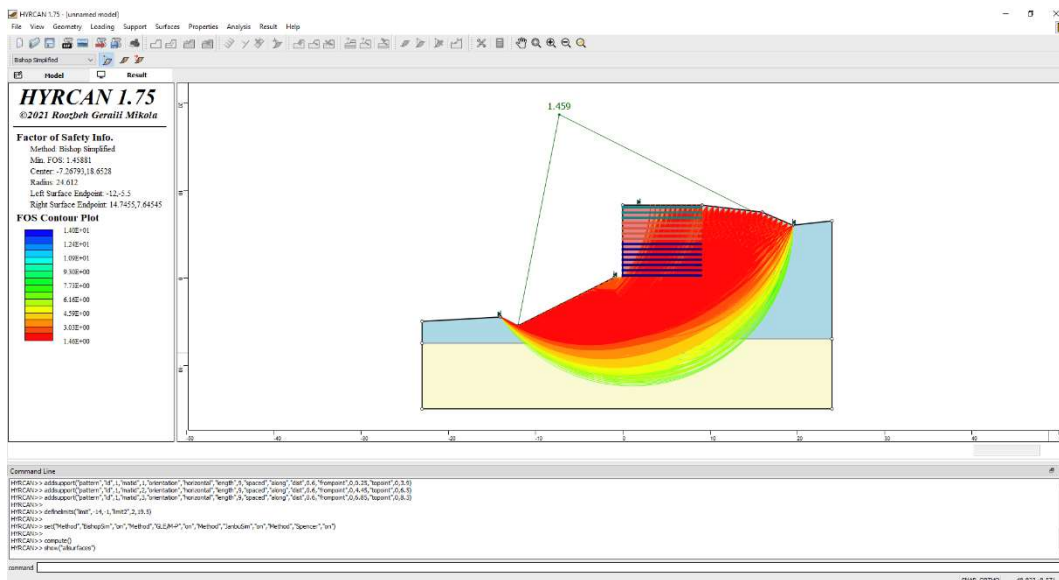
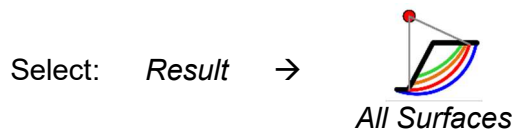
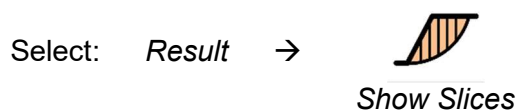


Figure 8- Circular surface search – All surfaces shown.

The Show Slices option can be used to display the actual slices used in the analysis.



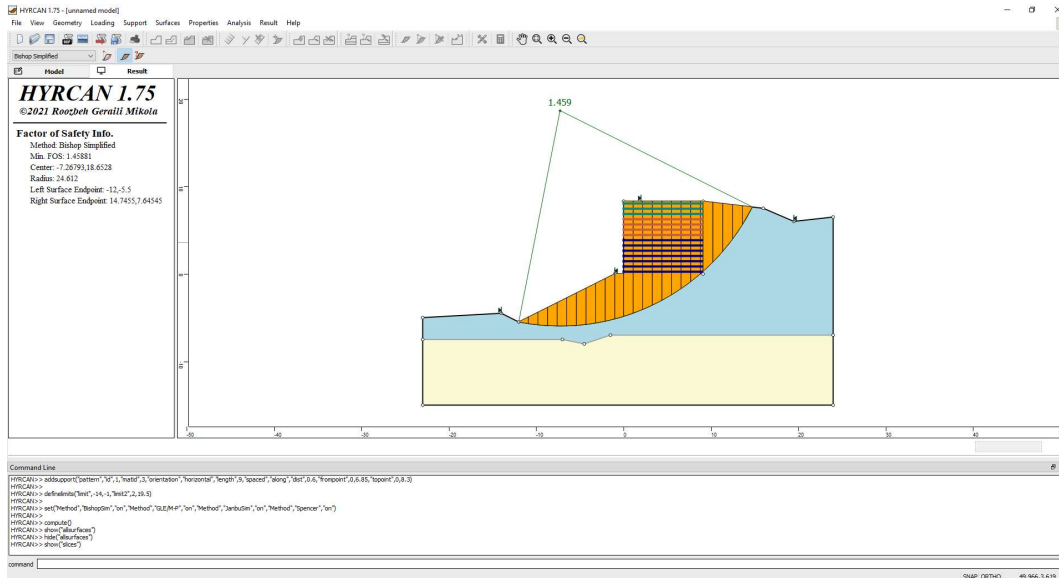


Figure 9- Slice displayed.

HYRCAN will also display the mobilized force in each support when Show Slices option is turned on.

Script

After finishing the model, you will be able to save the generated script by **HYRCAN** in the text file.

Select:



The commands for this tutorial are listed below.

```
newmodel()

set("failureDir","r21")

extboundary(-23,-5,-14,-4.5,-12,-5.5,-1,0,0,0,0,8.3,9.1,8.3,16,7.5,19.5,6,24,6.5,24,-15,-23,-15,-23,-5)

matboundary(-23,-7.5,-7,-7.5,-4.5,-8,-1.5,-7,24,-7)
matboundary(0,0,9.1,0,9.1,8.3)

definemat("ground","matID",1,"matName","Backfill","uw",21.7,"cohesion",1,"friction",33)
definemat("ground","matID",2,"matName","Wall","uw",20.4,"cohesion",0,"friction",41)
definemat("ground","matID",3,"matName","Glacial till","uw",20,"cohesion",2,"friction",54)

assignsoilmat("matid",2,"atpoint",7,5)
assignsoilmat("matid",3,"atpoint",7,-11)

definemat("support","matID",1,"supportType","GeoTextile","matName","Type 1","forceApp","passive","stripTensile",111,"stripAdhesion",0,"stripFricAngle",41)
definemat("support","matID",2,"supportType","GeoTextile","matName","Type 2","forceApp","passive","stripTensile",71,"stripAdhesion",0,"stripFricAngle",41)
definemat("support","matID",3,"supportType","GeoTextile","matName","Type 3","forceApp","passive","stripTensile",42,"stripAdhesion",0,"stripFricAngle",41)

addsupport("pattern","id",1,"matid",1,"orientation","horizontal","length",9,"spaced","along","dist",0.6,"frompoint",0,0.25,"topoint",0,3.9)
addsupport("pattern","id",1,"matid",2,"orientation","horizontal","length",9,"spaced","along","dist",0.6,"frompoint",0,4.45,"topoint",0,6.5)
```

```
addsupport("pattern","id",1,"matid",3,"orientation","horizontal","length",9,"spaced","along","dis  
t",0.6,"frompoint",0,6.85,"topoint",0,8.3)  
  
definelimits("limit",-14,-1,"limit2",2,19.5)  
  
set("Method","BishopSim","on","Method","GLE/M-  
P","on","Method","JanbuSim","on","Method","Spencer","on")  
  
compute()
```