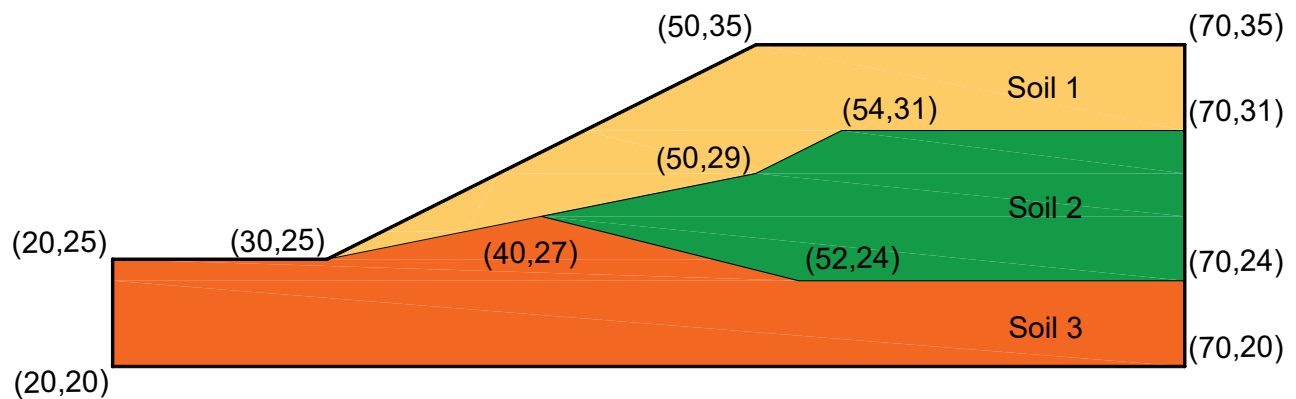


# Non-Homogeneous Slope

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

Email: [hyrcan4geo@outlook.com](mailto:hyrcan4geo@outlook.com)


Web Page: [www.geowizard.org](http://www.geowizard.org)



This tutorial will demonstrate some of the basic features of **HYRCAN** to calculate the factor of safety for a non-homogeneous three-layer slope as shown above.

## 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. In this analysis the default parameters have been selected.

Select: *Analysis* →  *Project Settings*

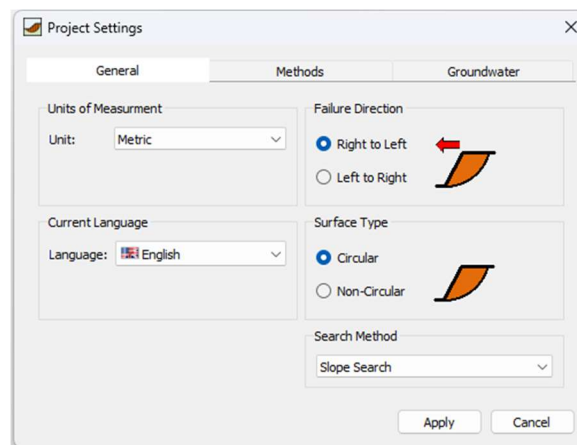



Figure 1- Project Settings dialog.

## 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.

Select: *Geometry* →  *External Boundary*

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

```
Enter vertex [esc=cancel]: 20 25
Enter vertex [esc=cancel]: 30 25
Enter vertex [esc=cancel]: 50 35
Enter vertex [c=close,esc=cancel]: 70 35
Enter vertex [c=close,esc=cancel]: 70 20
Enter vertex [c=close,esc=cancel]: 20 20
Enter vertex [c=close,esc=cancel]: c
```

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:

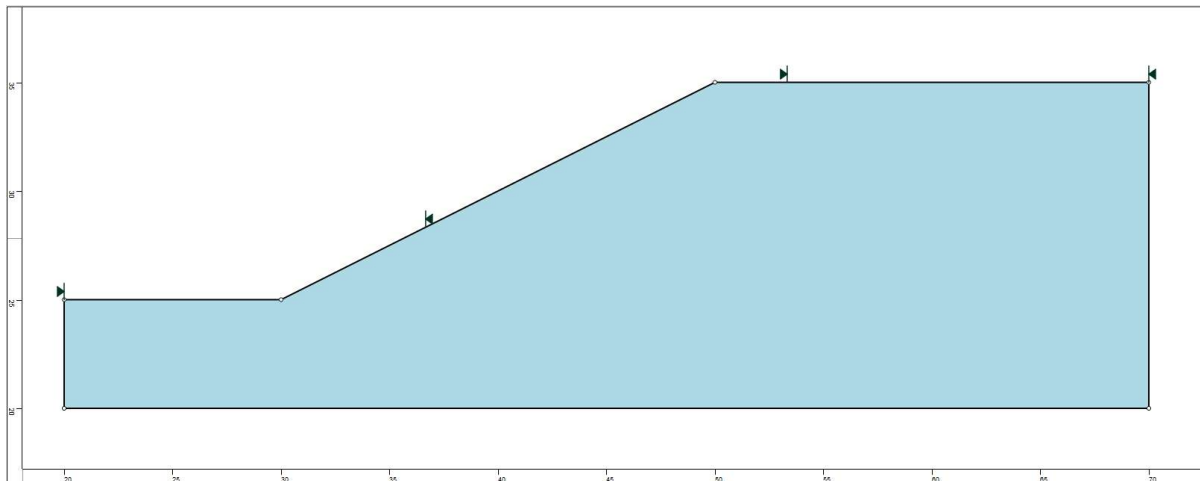



Figure 2- External boundary is created.

- **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]: 30 25  
 Enter vertex [d=done,esc=cancel]: 50 29  
 Enter vertex [d=done,esc=cancel]: 54 31  
 Enter vertex [d=done,esc=cancel]: 70 31  
 Enter vertex [d=done,esc=cancel]: d

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

Enter vertex [esc=cancel]: 40 27  
 Enter vertex [d=done,esc=cancel]: 52 24  
 Enter vertex [d=done,esc=cancel]: 70 24  
 Enter vertex [d=done,esc=cancel]: d

Your screen should now look as follows:

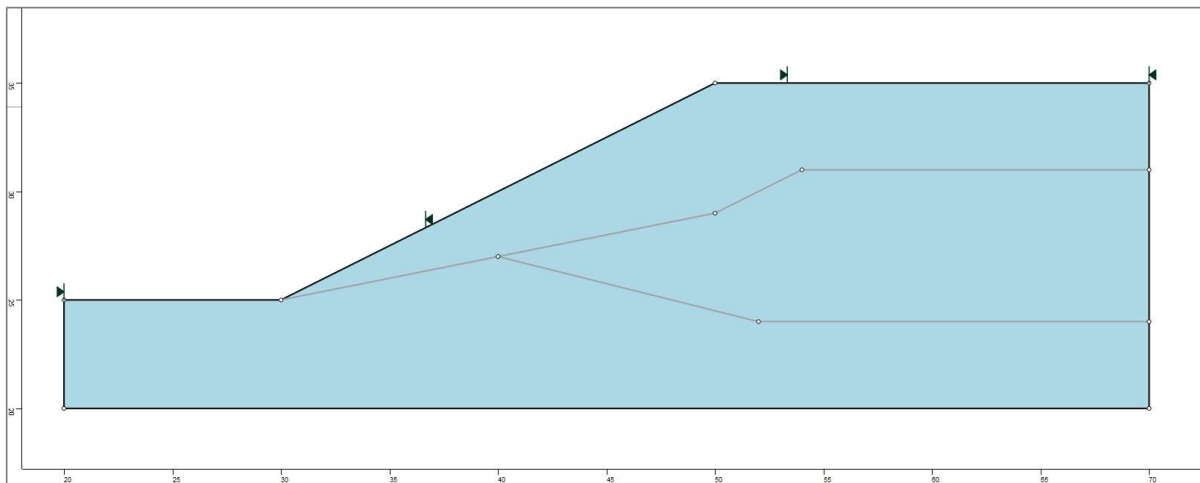



Figure 3- External and material boundaries added.

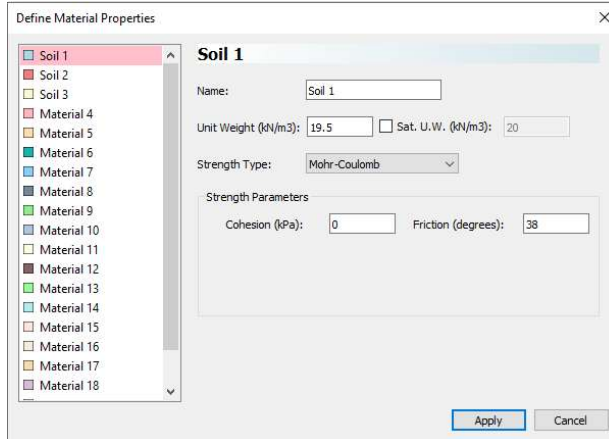
## Properties

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

Select: *Properties* →   
*Define Materials*

Material	$c$ (kN/m <sup>2</sup> )	$\phi$ (degrees)	$\gamma$ (kN/m <sup>3</sup> )
Soil 1	0.0	38.0	19.5
Soil 2	5.3	23.0	19.5
Soil 3	7.2	20.0	19.5

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



**Define Material Properties**

**Soil 1**

Name: Soil 1

Unit Weight (kN/m<sup>3</sup>): 19.5 ☐ Sat. U.W. (kN/m<sup>3</sup>): 20

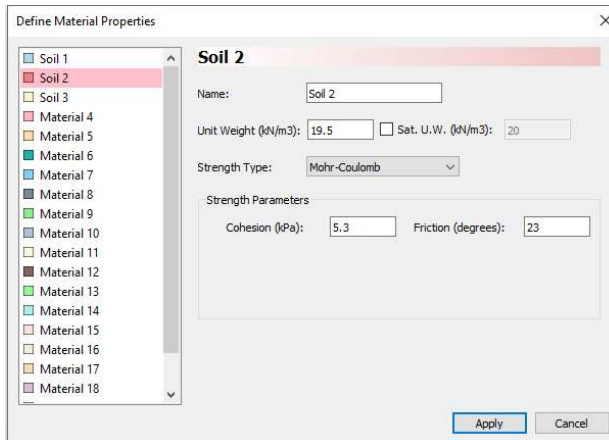
Strength Type: Mohr-Coulomb

Strength Parameters

Cohesion (kPa): 0 Friction (degrees): 38

Apply Cancel

Enter the parameters shown above. When all parameters are entered for the first material, select the second and third tabs, and enter the properties for each soil layer.



**Define Material Properties**

**Soil 2**

Name: Soil 2

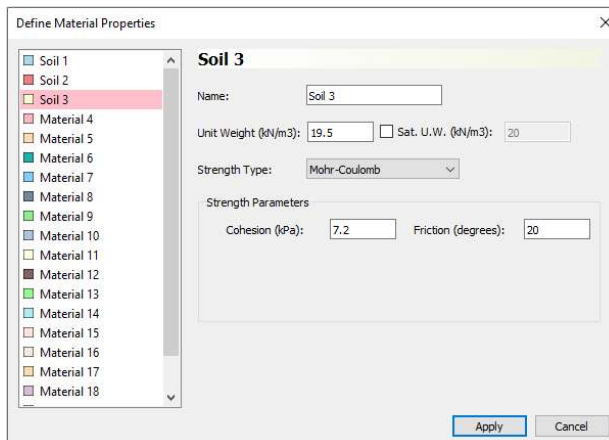
Unit Weight (kN/m<sup>3</sup>): 19.5 ☐ Sat. U.W. (kN/m<sup>3</sup>): 20

Strength Type: Mohr-Coulomb

Strength Parameters

Cohesion (kPa): 5.3 Friction (degrees): 23

Apply Cancel



**Define Material Properties**

**Soil 3**

Name: Soil 3

Unit Weight (kN/m<sup>3</sup>): 19.5 ☐ Sat. U.W. (kN/m<sup>3</sup>): 20

Strength Type: Mohr-Coulomb

Strength Parameters

Cohesion (kPa): 7.2 Friction (degrees): 20

Apply Cancel

Enter the properties, and press Apply when finished.

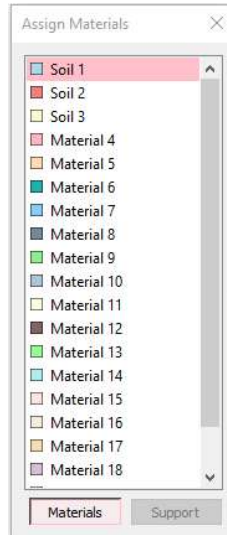
## Assigning Properties

Since we have defined more than one material, it will be necessary to assign properties to the correct regions of the model, using the Assign Material dialog. Select Assign Properties from the toolbar or the Properties menu.

Select: *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.

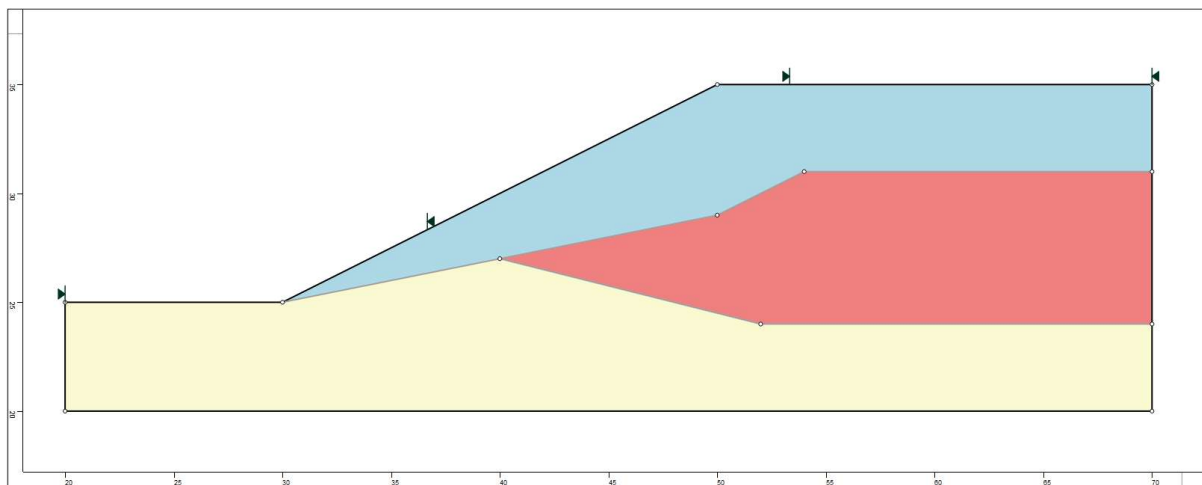
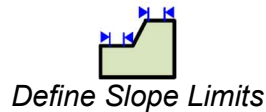


Figure 4- Geometry setup after assigning the properties.

## 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:

Right x coordinate:

☒ Second set of limits

Limits

Left x coordinate:

Right x coordinate:

In this tutorial in order to cover the crest of the slope the value of “Left x coordinate” of the Second Limit is modified to 45 as shown in the figure above. 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 is completed, you are ready to view the results in Result Tab. By default, when Result Tab opened, the Global Minimum slip surface, for the Bishop Simplified analysis method will be shown. This resulted in total of 7500 trial slip surfaces. The result of the factor of safety calculation is shown in Figure 5. Table 1 summarize the comparisons of calculated factor of safety for the same model using different commercial programs.

Table 1- Comparison of Minimum Factor of Safety for Non-Homogeneous model

Method	Slide2	SVSLOPE	HYRCAN
Bishop Simplified	1.405	1.405	1.404

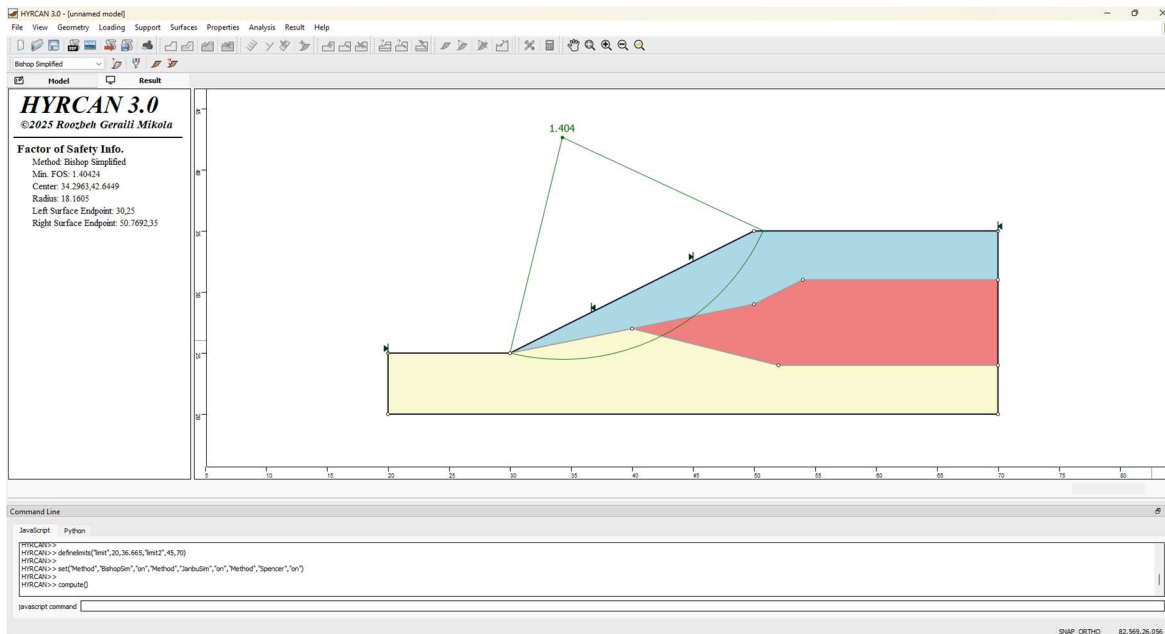



Figure 5- Result of automatic slope search.

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

Select: *Result* →  *All Surfaces*

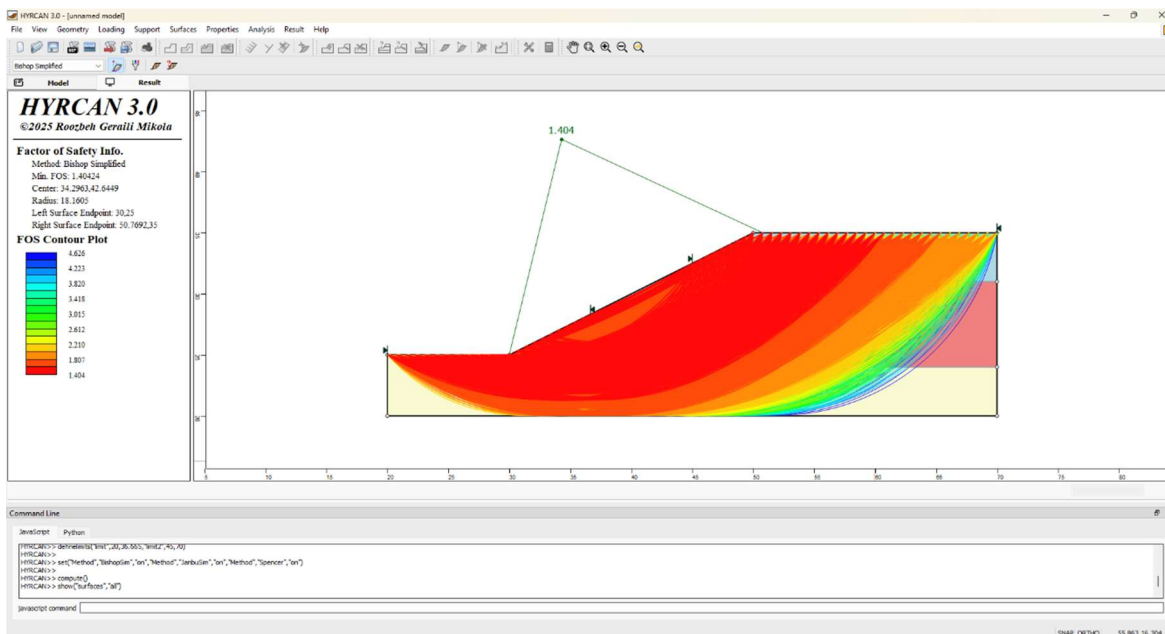


Figure 6- Circular surface search – All surfaces shown.

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

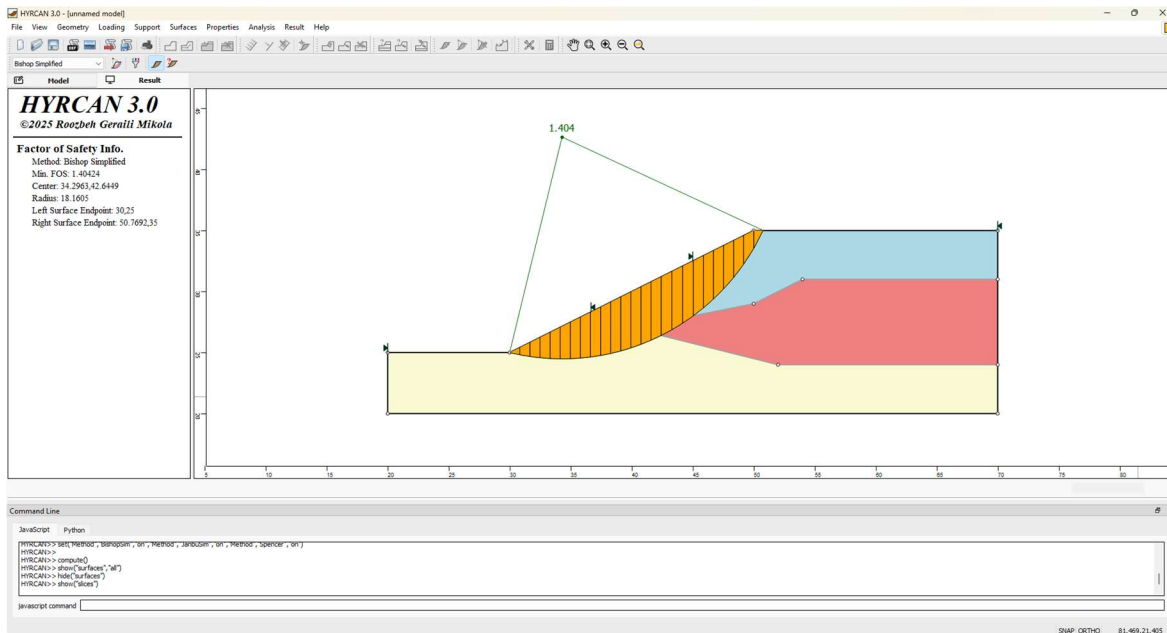
Select: *Result* →


Figure 7- Slice displayed.

The Query Slice Data option allows you to view detailed analysis results for individual slices.

 Select: *Result* →


After selecting the Query Slice Data, the Slice Data dialog will show up, which prompts you to "Click on a slice to view slice data". Click on any slice, and the data for the slice will be displayed in the dialog, as shown below:



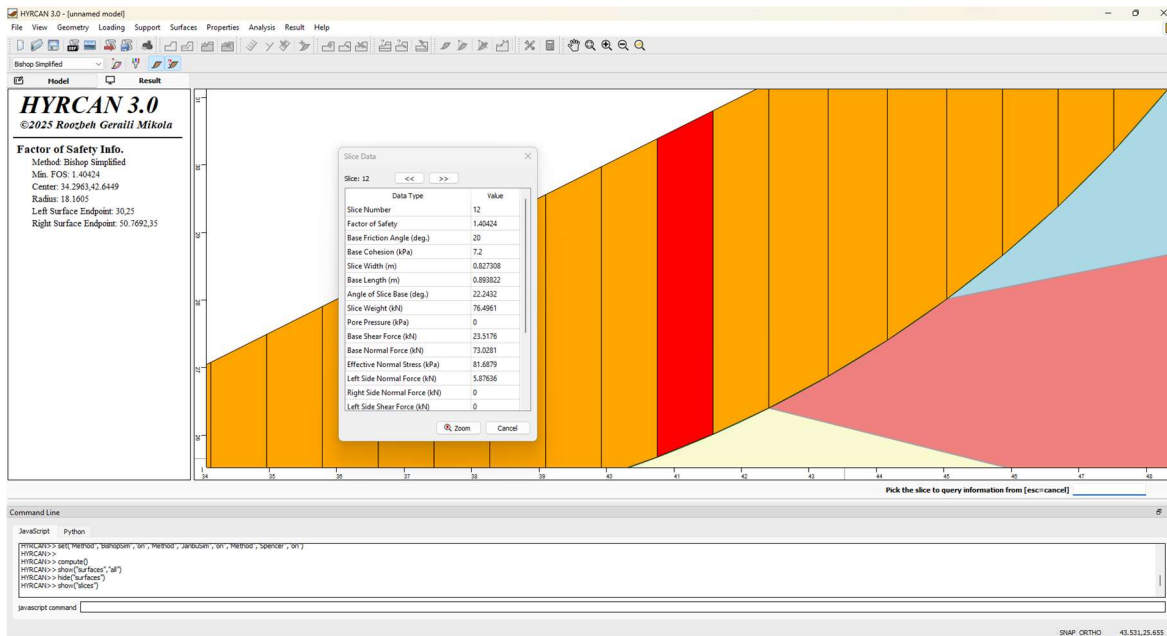


Figure 8- Slice Data dialog.

## 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(20,25,30,25,50,35,70,35,70,20,20,20,20,25)

matboundary(30,25,50,29,54,31,70,31)
matboundary(40,27,52,24,70,24)

definemat("ground", "matID",1,"matName", "Soil 1", "uw", 19.5, "cohesion", 0, "friction", 38)
definemat("ground", "matID",2,"matName", "Soil 2", "uw", 19.5, "cohesion", 5.3, "friction", 23)
definemat("ground", "matID",3,"matName", "Soil 3", "uw", 19.5, "cohesion", 7.2, "friction", 20)

assignsoilmat("matid",1,"atpoint",53.5982,32.8461)
assignsoilmat("matid",2,"atpoint",56.9309,28.517)
assignsoilmat("matid",3,"atpoint",34.2547,22.0234)

definelimits("limit",20,36.665,"limit2",45,70)

compute()
```