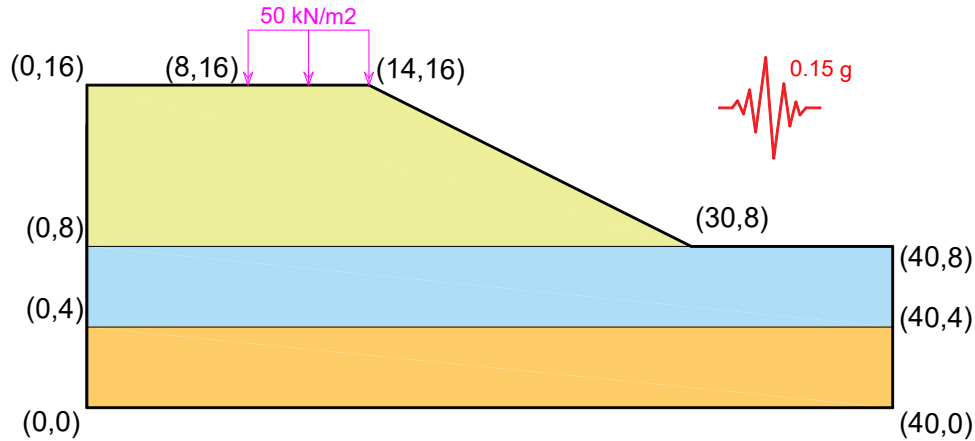


# Slope Stability Under Seismic Loading

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 how to use **HYRCAN** to calculate factor of safety for a layered slope, with a horizontal seismically induced acceleration of 0.15g included in the analysis. A uniformly distributed load of 50 kN/m<sup>2</sup> is applied over a width of 6 m as shown in the figure 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. For this analysis change the failure direction to “Left to Right” then press Apply.

Select: *Analysis* →



*Project Settings*

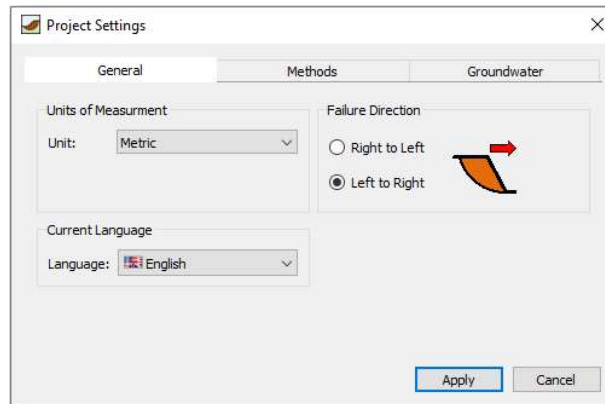


Figure 1- Project Settings dialog.

## 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* →



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

```
Enter vertex [esc=cancel]: 0 0
Enter vertex [esc=cancel]: 0 16
Enter vertex [esc=cancel]: 14 16
Enter vertex [c=close,esc=cancel]: 30 8
Enter vertex [c=close,esc=cancel]: 40 8
Enter vertex [c=close,esc=cancel]: 40 0
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:

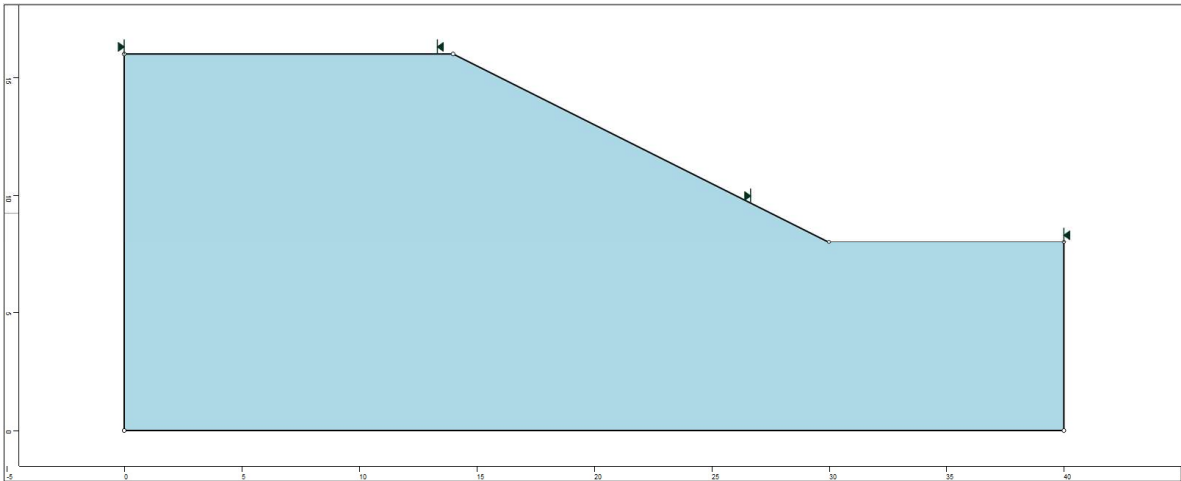


Figure 2- External boundary is created.

- **Material Boundaries**

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

Select: *Geometry* →



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

Enter vertex [esc=cancel]: 0 8  
 Enter vertex [d=done,esc=cancel]: 30 8  
 Enter vertex [d=done,esc=cancel]: d

And repeat it for the following coordinates:

Enter vertex [esc=cancel]: 0 4  
 Enter vertex [d=done,esc=cancel]: 40 4  
 Enter vertex [d=done,esc=cancel]: d

Your screen should now look as follows:

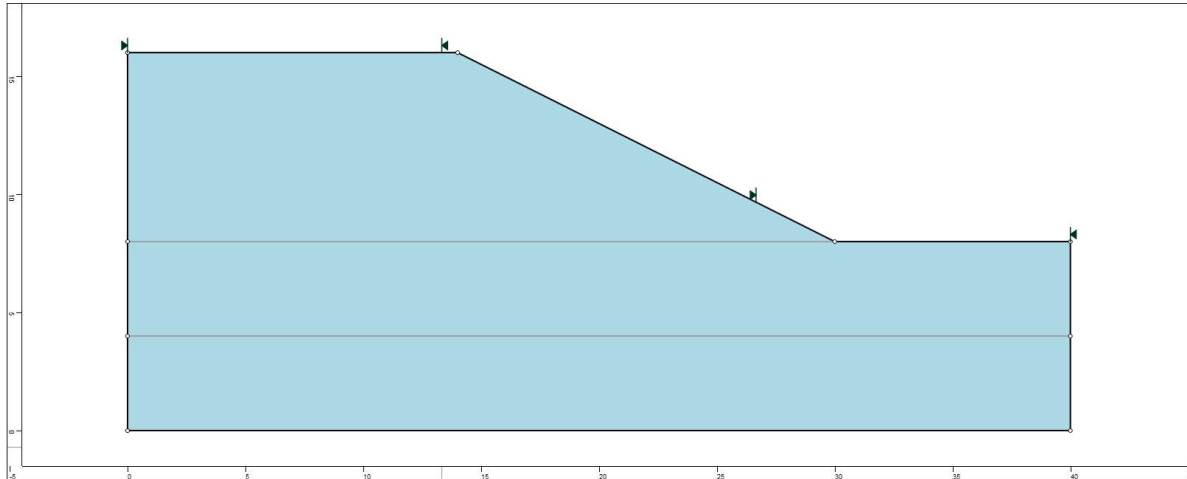



Figure 3- External and material boundaries added.

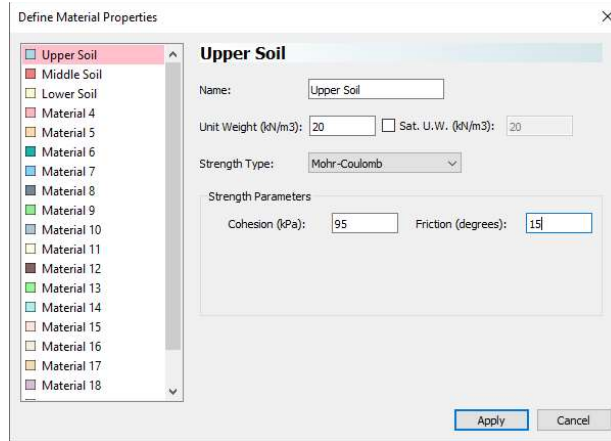
## Properties

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> )
Upper Layer	95.0	15.0	20
Middle Layer	15.0	0.0	20
Lower Layer	30.0	0.0	20

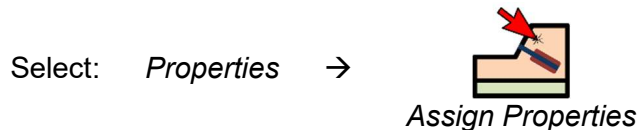
With the first (default) tab selected in the Define Materials dialog, enter the following properties:



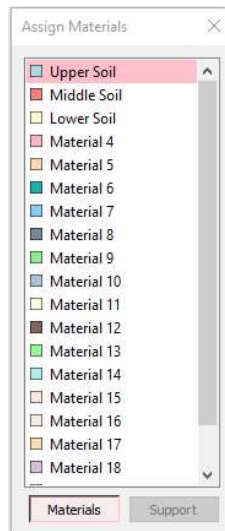
Enter the parameters shown above. When all parameters are entered for the first material, select the second and third tabs, and enter the properties for the middle and lower layers and press Apply when finished.

## Assigning Properties

Since we have defined three 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.



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.

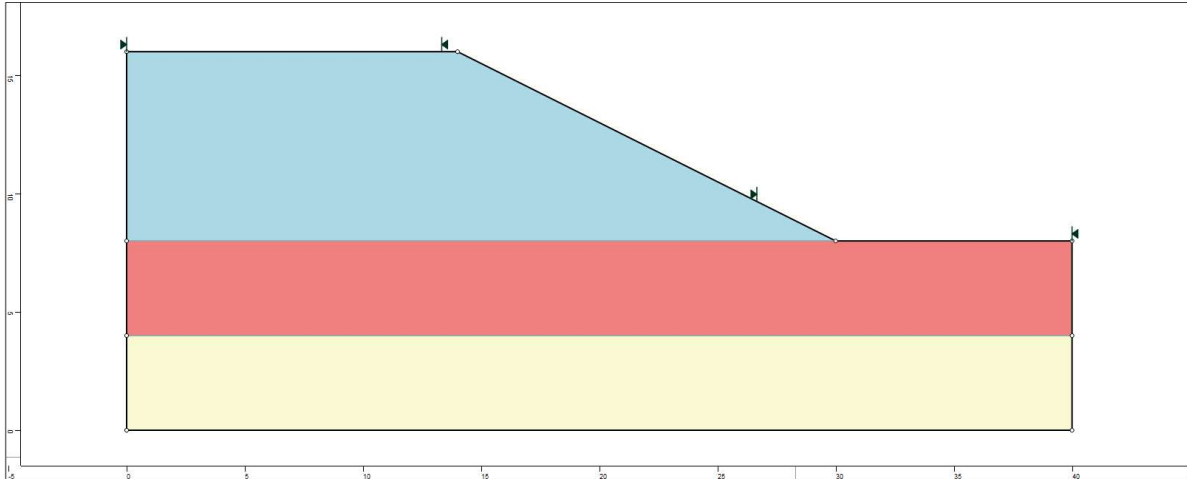
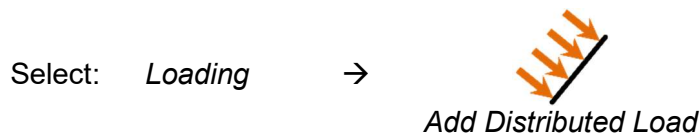


Figure 4- Geometry setup after assigning the properties.

## Add Distributed Load

To add distributed load select Add Distributed Load from toolbar or the Loading menu.



The Add Distributed Load dialog will show up.

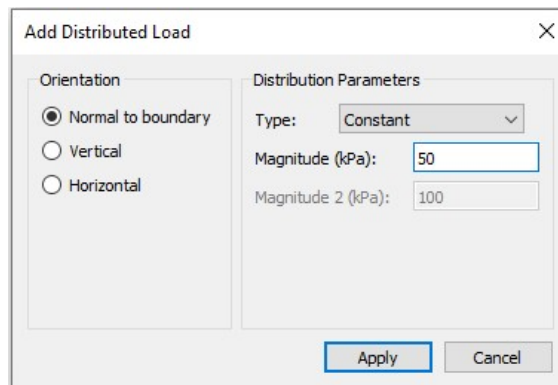


Figure 5- Add Distributed dialog.

Select constant distributed load type and enter 50 kPa as loading magnitude 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 the load graphically, by clicking the left mouse button when the red cross is at the desired starting and ending points of the distributed load. However, to enter exact coordinates, it is easier in this case to enter the coordinates in the prompt line.

Enter first point on boundary [esc=cancel]: 8 16  
 Enter second point on boundary [esc=cancel]: 14 16

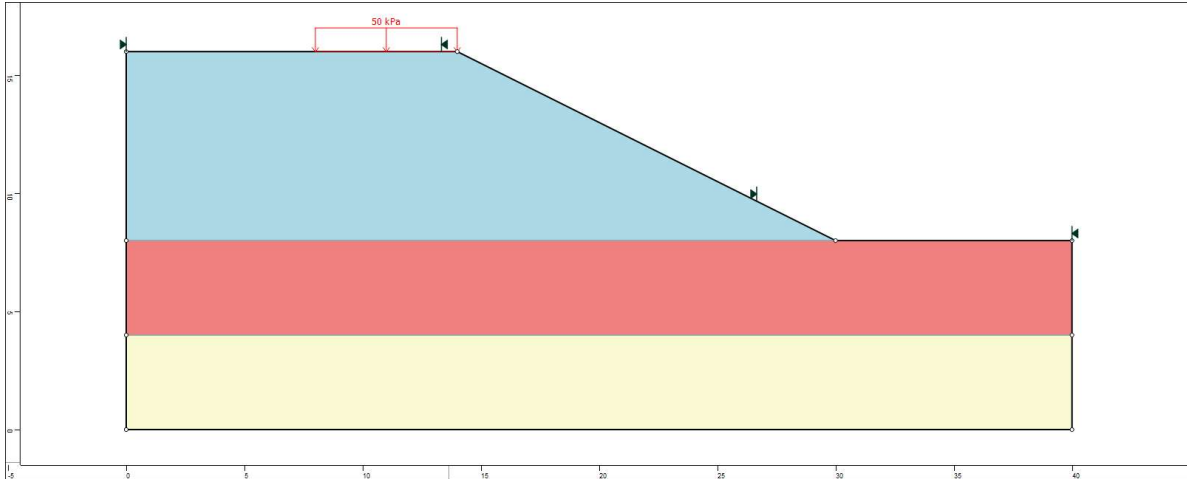
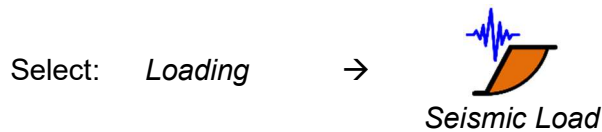


Figure 6- Geometry setup with applied distributed load.

## Apply Seismic Load

To apply seismic load, select Seismic Load from toolbar or the Loading menu.



The Seismic Load dialog will show up.

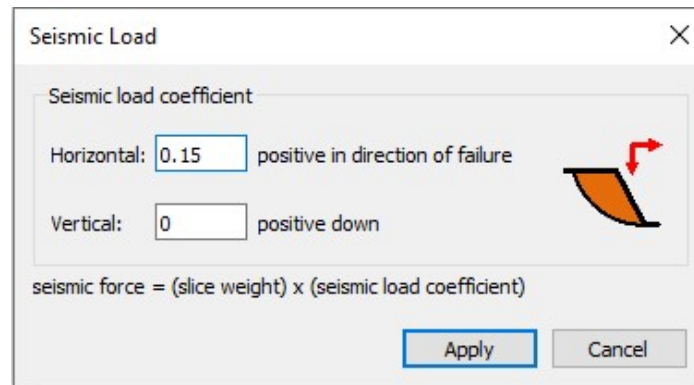


Figure 7- Seismic Load dialog.

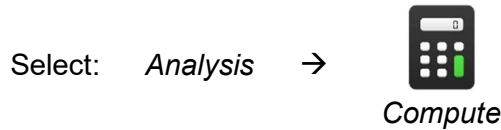
Enter 0.15 as magnitude of horizontal seismic coefficient then press Apply.

### Notes:

- The horizontal seismic coefficient is always positive, and represents a horizontal seismic force directed out of the slope.
- A positive vertical seismic coefficient represents a vertical seismic force directed downwards and a negative vertical seismic coefficient represents a vertical seismic force directed upwards.

## Compute

The model is now ready to run.



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 Bishop Simplified analysis method will be shown. This resulted in total of approximately 5000 trial slip surfaces. The result of the factor of safety calculation is shown in Figure 8. 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

Method	Slide2	HYRCAN
Bishop Simplified	0.797	0.747

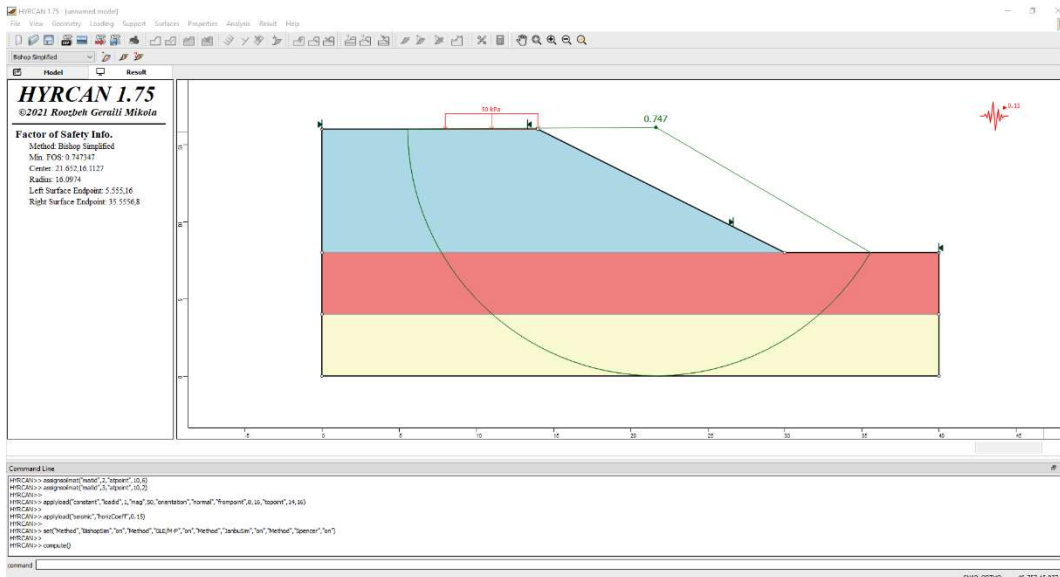
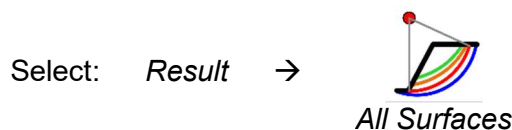


Figure 8- Result of automatic slope search.

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



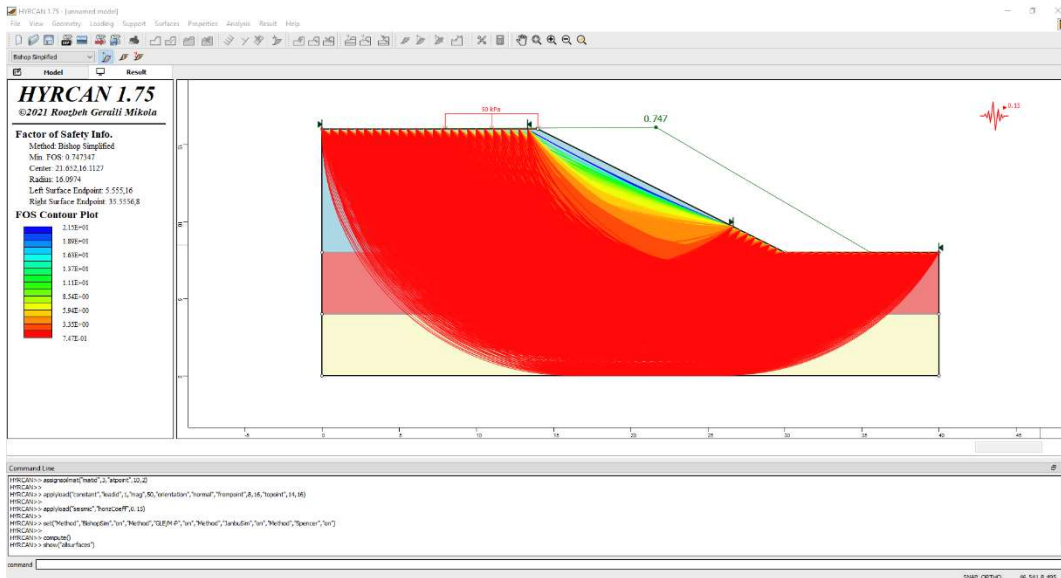



Figure 9- Circular surface search – All surfaces shown.

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

Select: *Result* →   
 Show Slices

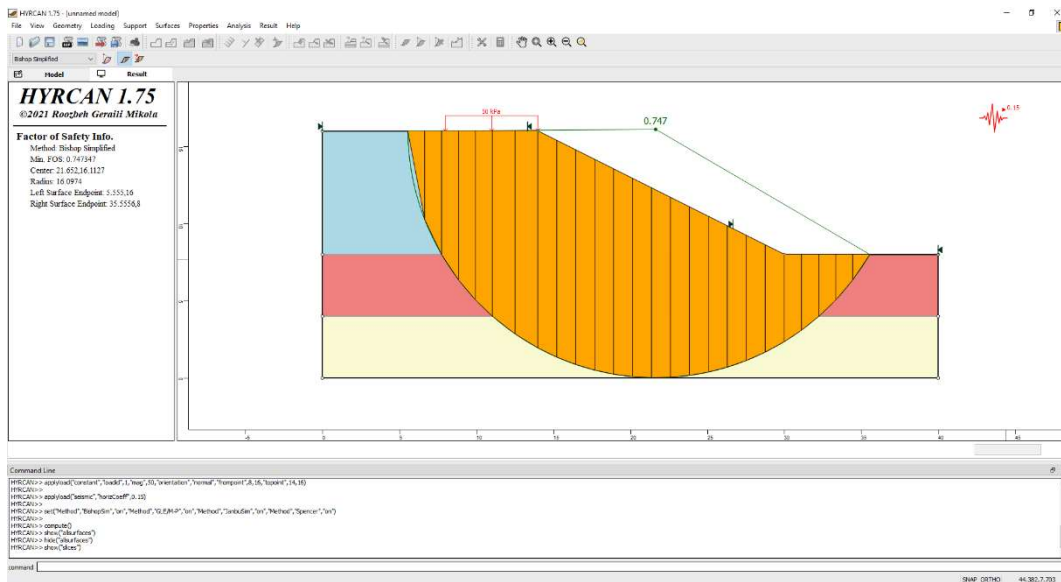



Figure 10- Slice displayed.

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

Select: *Result* →   
 Query Slice Data



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:

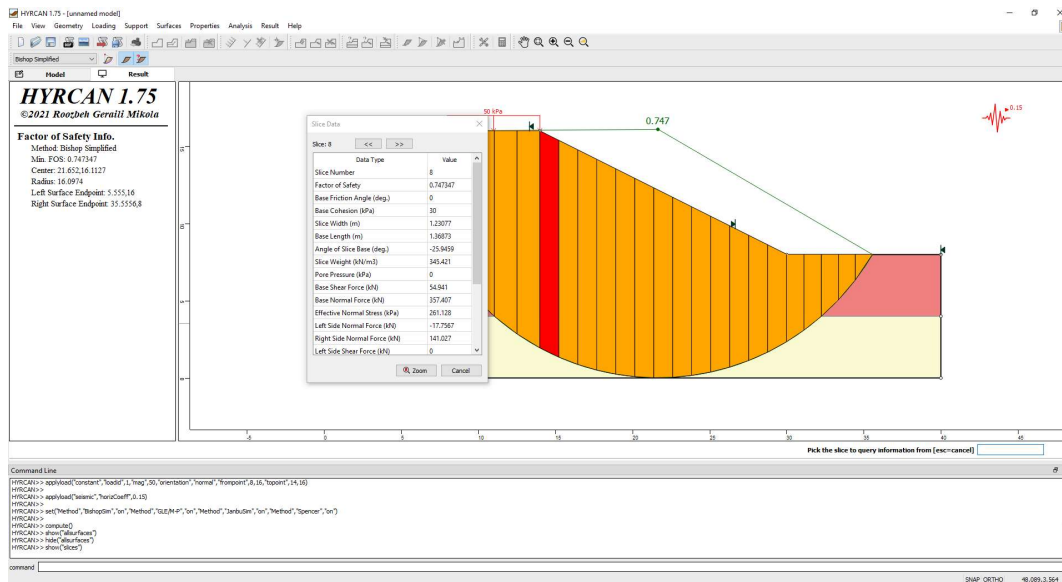


Figure 11- Slice Data dialog.

## 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", "l2r")

extboundary(0,0,0,16,14,16,30,8,40,8,40,0,0,0)

matboundary(0,8,30,8)
matboundary(0,4,40,4)

definemat("ground", "matID", 1, "matName", "Upper Soil", "uw", 20, "cohesion", 95, "friction", 15)
definemat("ground", "matID", 2, "matName", "Middle Soil", "uw", 20, "cohesion", 15, "friction", 0)
definemat("ground", "matID", 3, "matName", "Lower Soil", "uw", 20, "cohesion", 30, "friction", 0)

assignsoilmat("matid", 1, "atpoint", 10, 10)
assignsoilmat("matid", 2, "atpoint", 10, 6)
assignsoilmat("matid", 3, "atpoint", 10, 2)

applyload("constant", "loadid", 1, "mag", 50, "orientation", "normal", "frompoint", 8, 16, "topoint", 14, 16)

applyload("seismic", "horizCoeff", 0.15)

compute ()
```