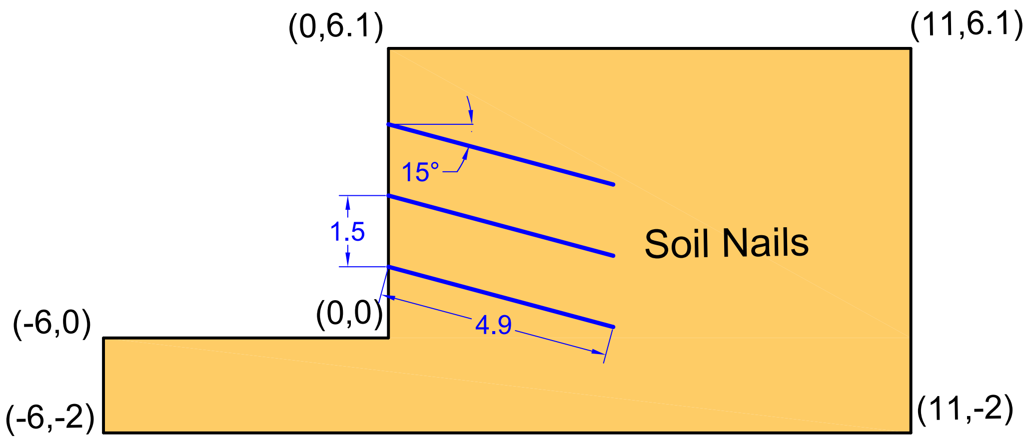
Stability of Slope Reinforced with Soil Nails

Created By: **Roozbeh Geraili Mikola, PhD, PE**
Email: **hyrcan4geo@outlook.com**
Web Page: **www.geowizard.org**



This tutorial will demonstrate the modeling of support in ***HYRCAN***. Three different types of slope reinforcement can be modeled in ***HYRCAN***, including soil nails, tiebacks and end anchored rock bolts.

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.

Select: *Analysis* →  *Project Settings*

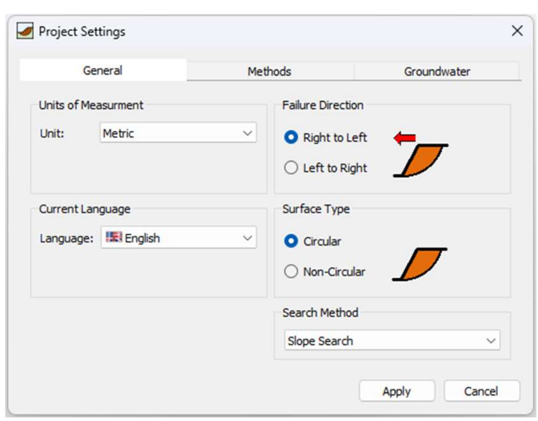
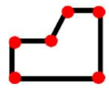


Figure 1- Project Settings dialog.

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]: 0 0
Enter vertex [esc=cancel]: 0 6.1
Enter vertex [esc=cancel]: 11 6.1
Enter vertex [c=close,esc=cancel]: 11 -2
Enter vertex [c=close,esc=cancel]: -6 -2
Enter vertex [c=close,esc=cancel]: -6 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:

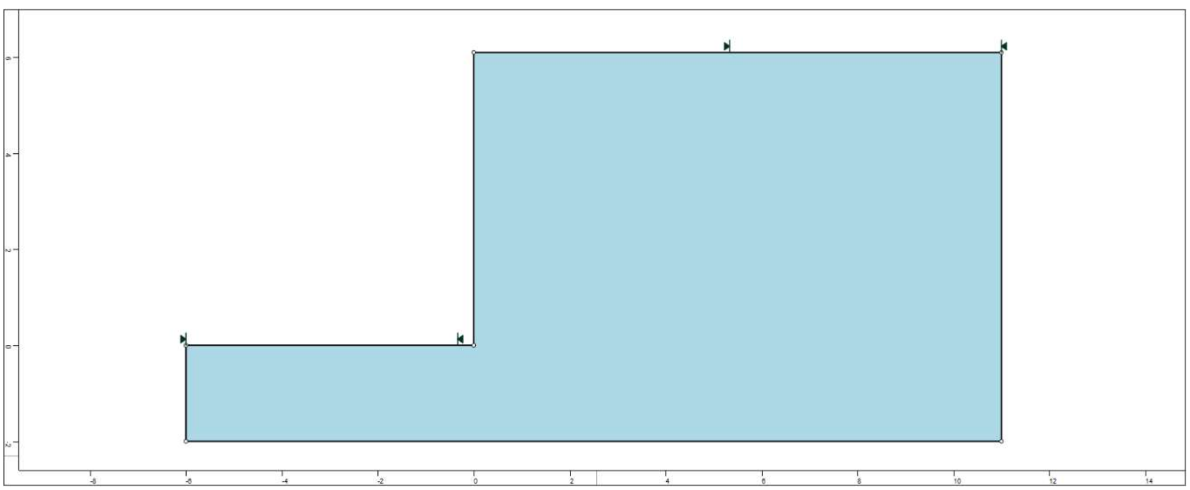
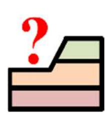


Figure 2- External boundary is created.

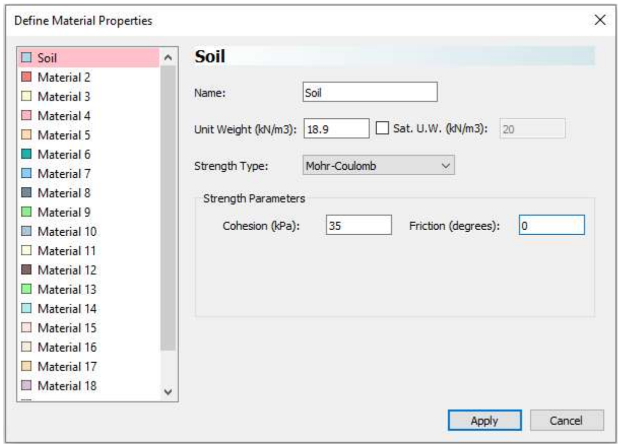
Properties

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/m²)	*φ* (degrees)	*γ* (kN/m³)
Soil	35.0	0.0	18.9

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 press Apply.

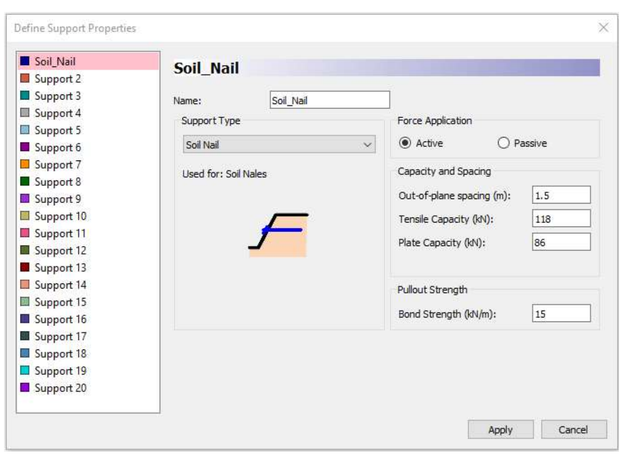
Add Support Pattern

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.

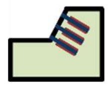
Select: *Properties* → 
Define Support

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

Support Type	Name	Out-of-plane Spacing (m)	Tensile Capacity (kN)	Plate Capacity (kN)	Bond Strength (kN/m)
Soil Nail	Soil_Nail	1.5	118	86	15



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. Set the Orientation = Angle from Horizontal, Angle = –15 degrees, Length = 4.9, and Distance between supports = 1.5. press Apply.

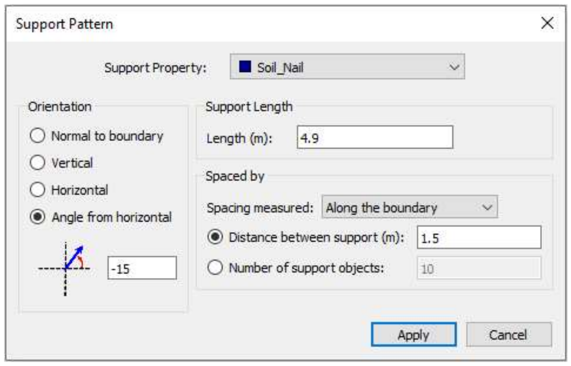


Figure 3- Add Distributed dialog.

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.

Enter first point on boundary [esc=cancel]: 0 1.5
Enter second point on boundary [esc=cancel]: 0 5

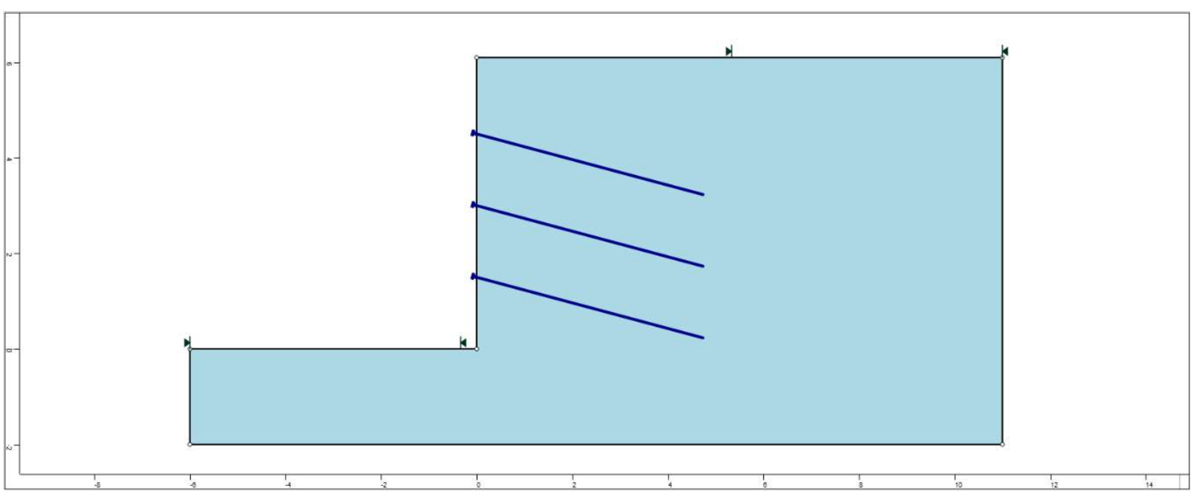
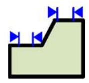
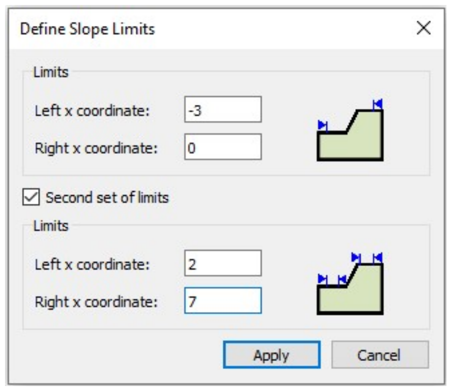


Figure 4- Geometry setup with applied distributed load.

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



In this tutorial the left and right coordinates are set to -3 and 0 and the left and right coordinates of the second set of limits are set to 2 and 7. 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* → 
Compute

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 7500 trial slip surfaces. The results of the factor of safety calculation is shown in Figure 6. 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	1.347	1.290

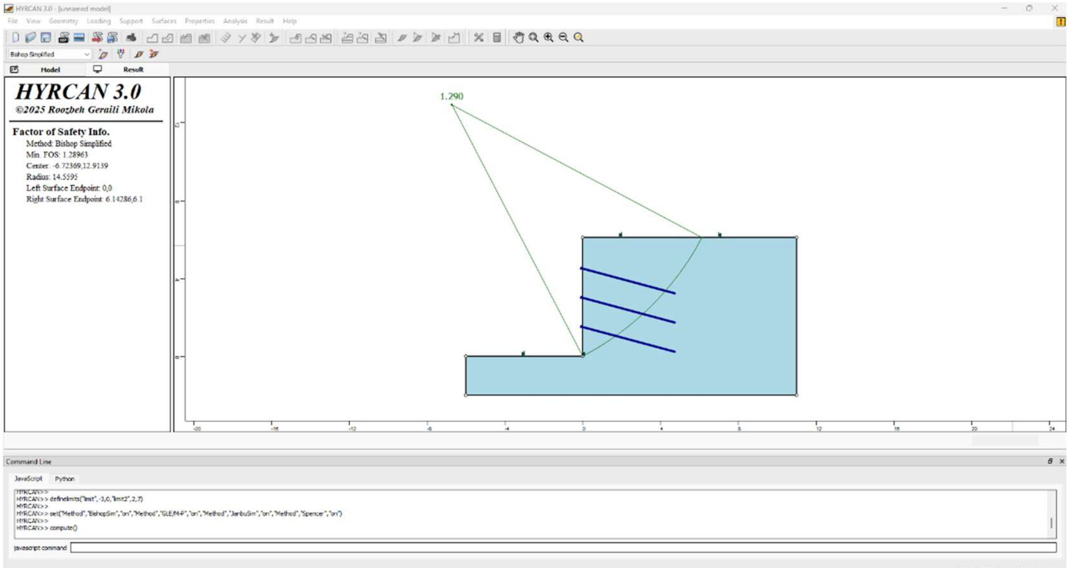


Figure 5- Result of automatic slope search.

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*

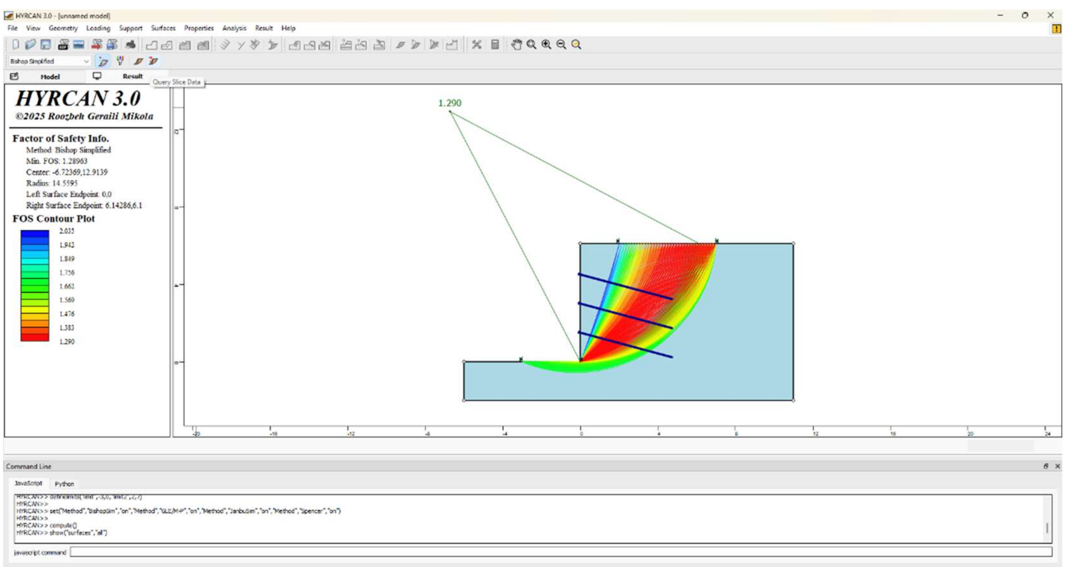


Figure 6- Circular surface search – All surfaces shown.

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

Select: *Result* →  *Show Slices*

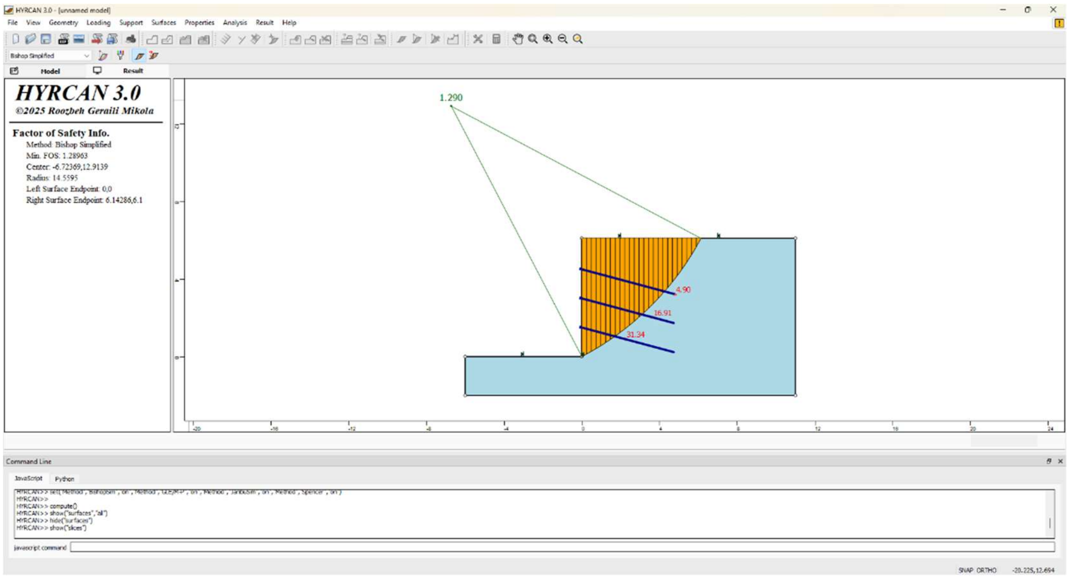


Figure 7- Slice displayed.

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","r2l")

extboundary(-6,0,0,0,0,6.1,11,6.1,11,-2,-6,-2,-6,0)

definemat("ground","matID",1,"matName","Soil","uw",18.9,"cohesion",35,"friction",0)

definemat("support","matID",1,"supportType","SoilNail","matName","Soil_Nail","spacing",1.5,"tensC
apacity",118,"plateCapacity",86,"bondStrength",15)

addsupport("pattern","id",1,"matid",1,"orientation","anglefromhoriz","angle",-
15,"length",4.9,"spaced","along","dist",1.5,"frompoint",0,1.5,"topoint",0,5)

definelimits("limit",-3,0,"limit2",2,7)

compute()
```