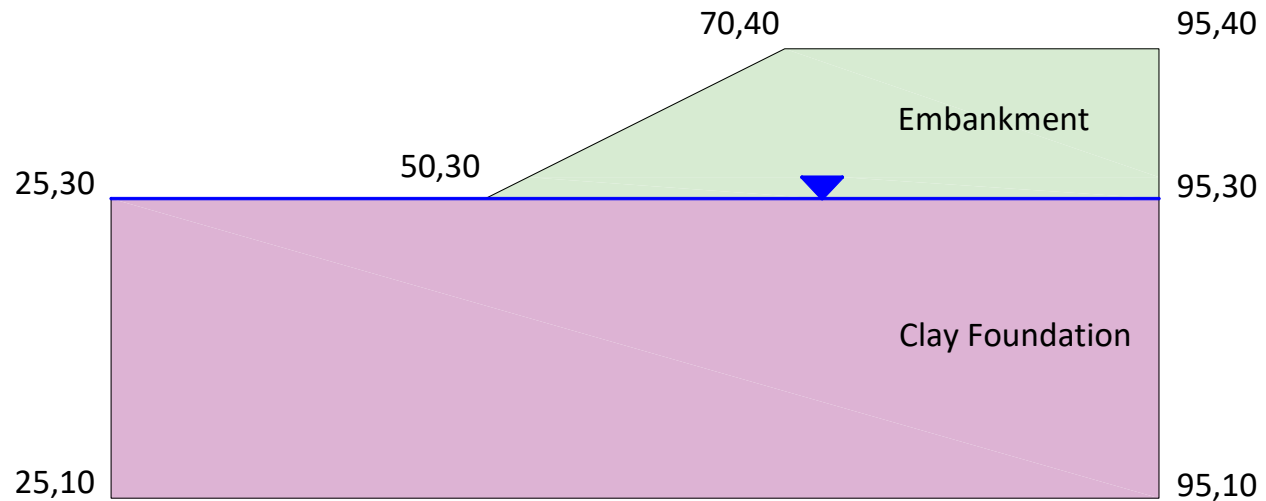# HYRCAN
## For Geo-Engineers

# Stability of Embankment on Clay Foundation using SHANSEP Method

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



This tutorial discusses the stability analysis of an embankment on a clay foundation in ***HYRCAN***. The clay foundation is modeled with the SHANSEP strength model. The layer of the embankment is modeled as Mohr-Coulomb materials.

## 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.

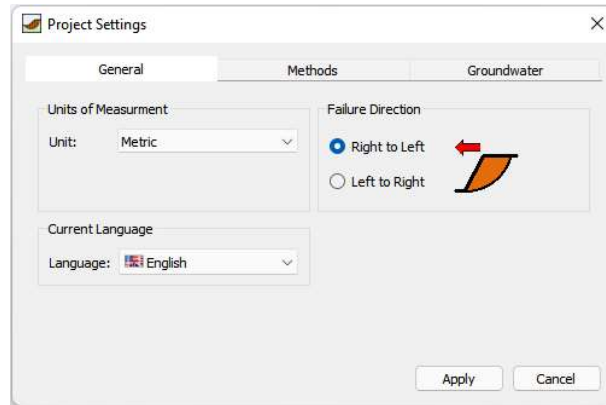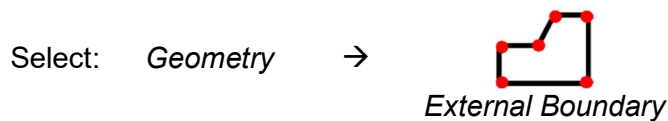Select:     *Analysis*   →

*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.

Select:    *Geometry*    →    

*External Boundary*

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

| |
|---|
| Enter vertex [esc=cancel]: 25.0,10.0 |
| Enter vertex [esc=cancel]: 95.0,10.0 |
| Enter vertex [esc=cancel]: 95.0,40.0 |
| Enter vertex [c=close,esc=cancel]: 70.0,40.0 |
| Enter vertex [c=close,esc=cancel]: 50.0,30.0 |
| Enter vertex [c=close,esc=cancel]: 25.0,30.0 |
| 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:
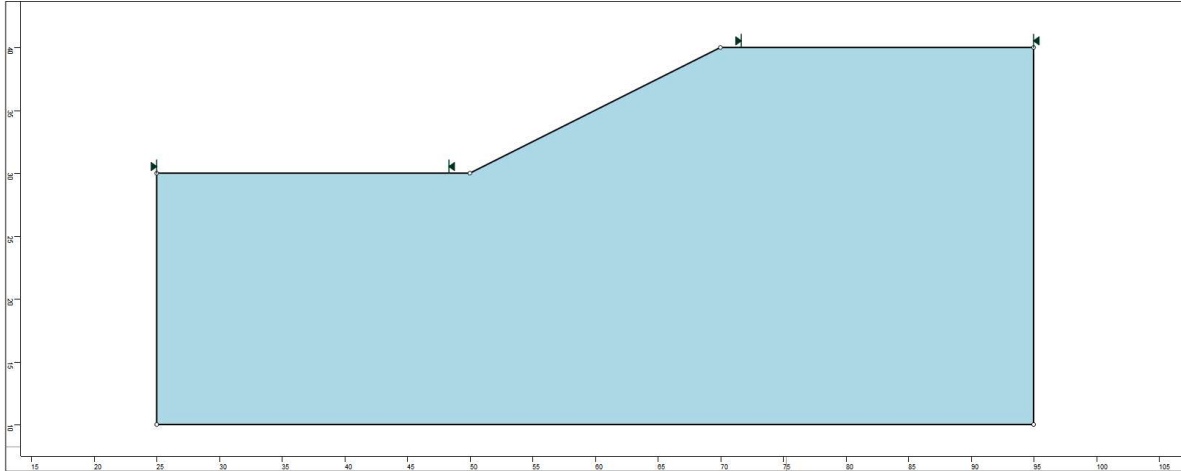
Figure 2- External boundary is created.

- ***Material Boundaries***

Material boundaries are used in **HYRCAN** to define the boundaries between different material 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]: 50.0,30.0
Enter vertex [d=done,esc=cancel]: 95.0,30.0
Enter vertex [d=done,esc=cancel]: d

Your screen should now look as follows:



Figure 3- External and material boundaries added.

- ***Water Table***

To add water table:

Select:  *Geometry*  →

*Add Water Table*

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

> Enter vertex [esc=cancel]: 25.0,30.0
> Enter vertex [d=done,esc=cancel]: 95.0,30.0
> Enter vertex [d=done,esc=cancel]: d

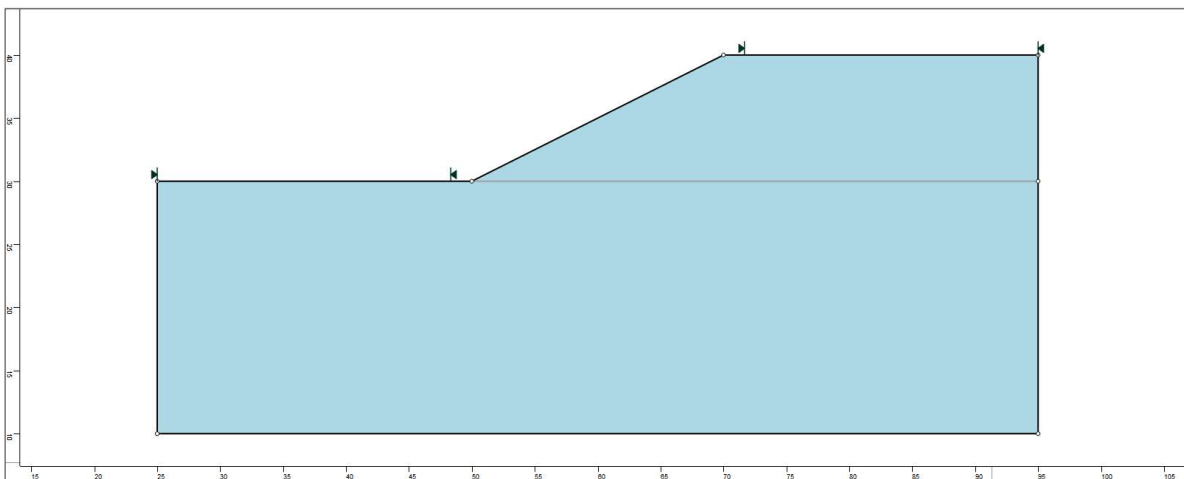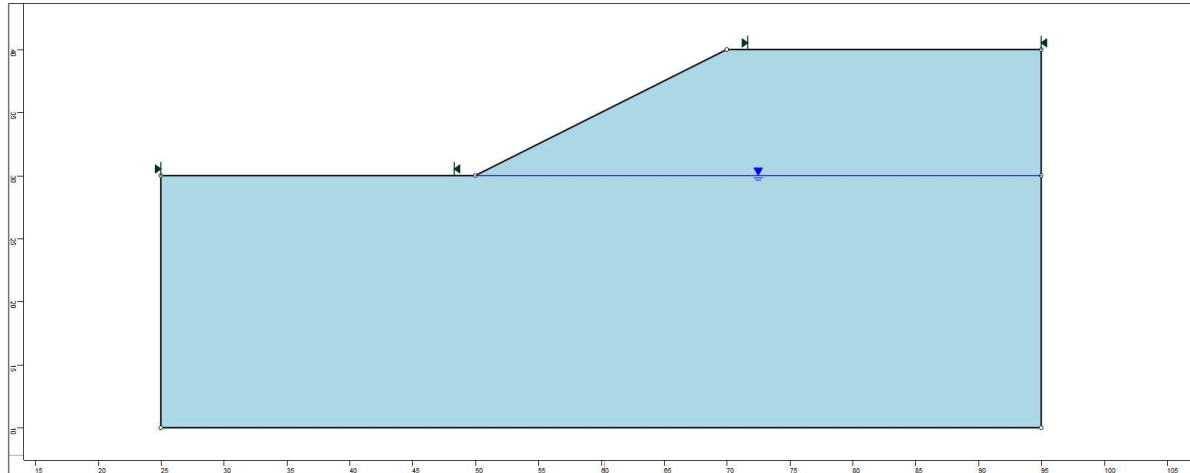Your screen should now look as follows:



Figure 4- Water table added.

# Properties

The embankment is considered to be a free draining material (i.e., sand). The undrained strength of the clay foundation is to be represented by the SHANSEP material model. Research has demonstrated that the undrained strength of normally consolidated clay soils can be represented by a constant strength ratio defined as $S_u/\sigma'_v = constant$. The undrained strength is normalized with respect to the vertical effective overburden stress. For a soil subjected to a given stress path, the following equation describes the undrained shear strength.

$\tau = A + \sigma'_v S (OCR)^m$

where:

$\tau$ = undrained shear strength
$A$ = minimum undrained shear strength
$\sigma'_v$ = in situ effective vertical stress
$S$ = normally consolidated ratio of $\left(\tau/\sigma'_v\right)_{nc}$

$OCR$ = over consolidation ratio
$m$ = exponent typically between 0.75 and 1

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

**HYRCAN**
**For Geo-Engineers**

Select: *Properties* →



*Define Materials*

In the Define Materials dialog, enter the following properties:

| Material | Strength Type | γ (kN/m³) | c (kN/m²) | φ (degrees) | A | S | m | OCR |
|----------|---------------|-----------|-----------|-------------|---|---|---|-----|
| Embankment | Mohr-Coulomb | 20 | 5.0 | 30.0 | - | - | - | - |
| Clay Foundation | SHANSEP | 19 | - | - | 0 | 0.3 | 0.8 | 2 |
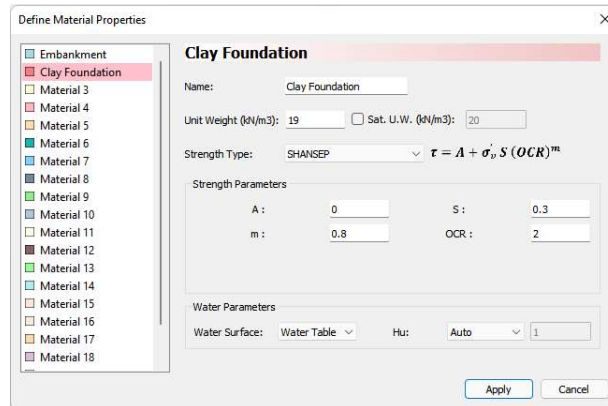


When all parameters are entered press Apply.

# 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.
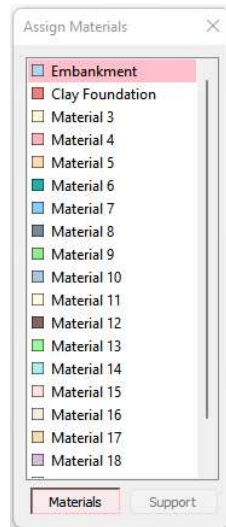
To assign properties to the soil layers:

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.
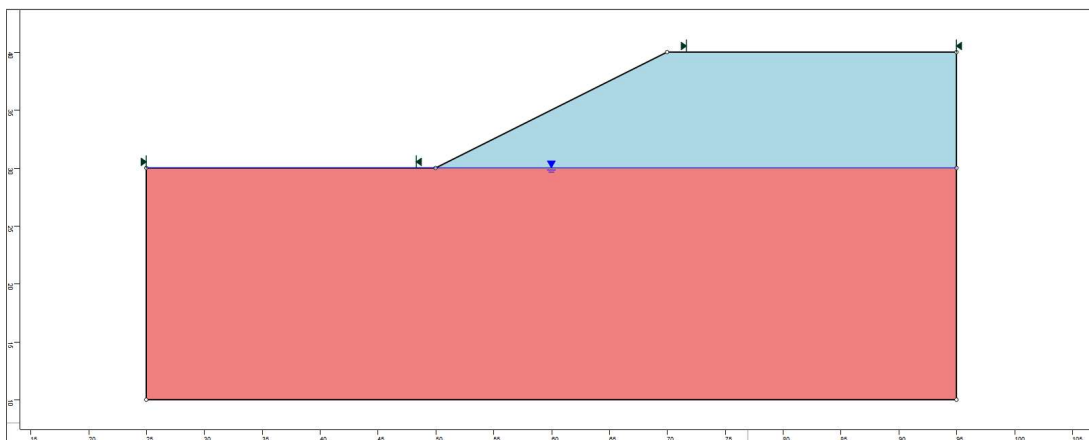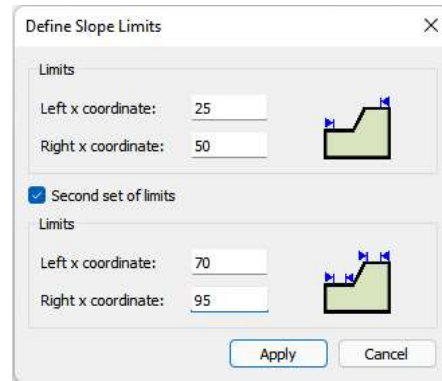


Figure 5- Geometry setup after assigning the properties.

# 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*

In this tutorial the left and right coordinates are set to 25 and 50 and the left and right coordinates of the second set of limits are set to 70 and 95. Later, 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 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 6. 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

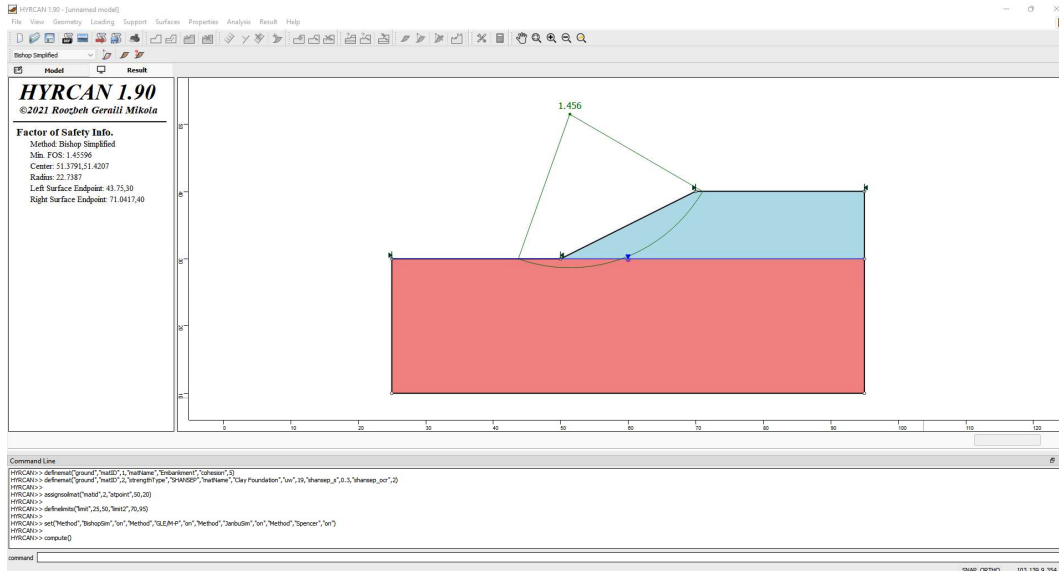| Method | Slide2 | HYRCAN |
|---|---|---|
| Bishop Simplified | 1.459 | 1.456 |
| GLE/Morgenstern-Price | 1.382 | 1.379 |
| Janbu Simplified | 1.330 | 1.330 |
| Spencer | 1.393 | 1.390 |

Figure 6- 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*     →     
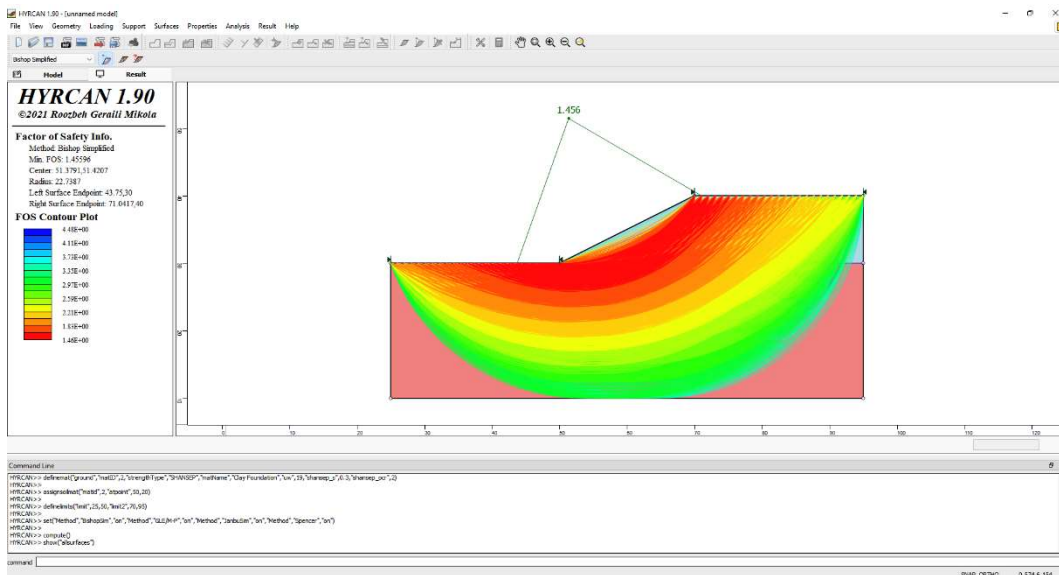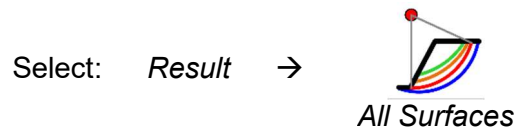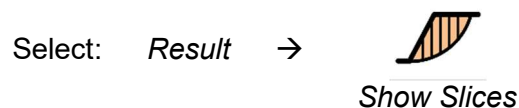
*All Surfaces*



Figure 7- Circular surface search – All surfaces shown.

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

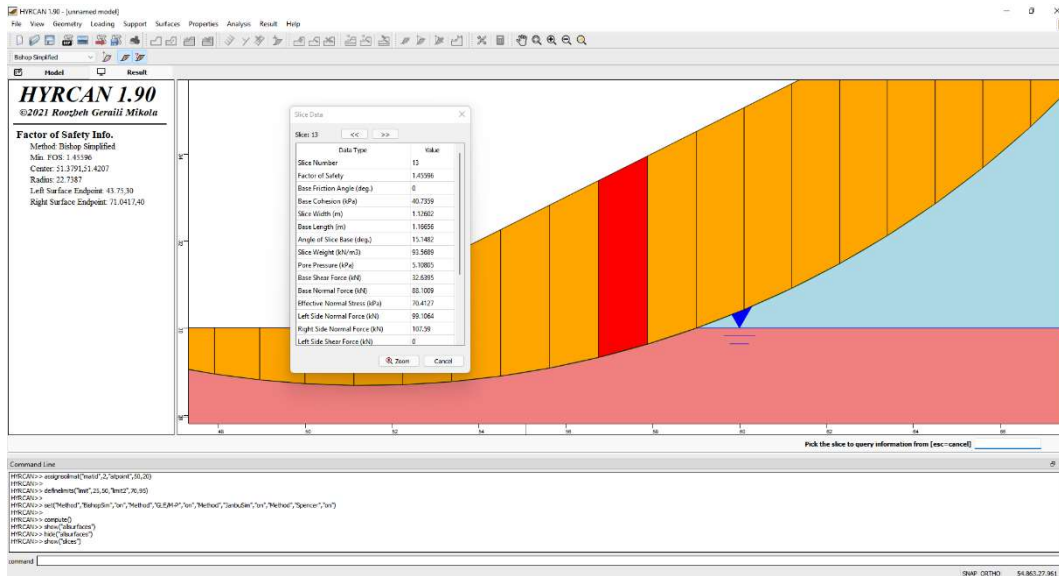Select:     *Result*     →     

*Show Slices*

Figure 8- 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")

set("unit","metric","waterUW",9.81)

extboundary(25,10,95,10,95,40,70,40,50,30,25,30,25,10)

matboundary(50,30,95,30)

addwatertable(25,30,95,30)

definemat("ground","matID",1,"matName","Embankment","cohesion",5)
definemat("ground","matID",2,"strengthType","SHANSEP","matName","Clay
Foundation","uw",19,"shansep_s",0.3,"shansep_ocr",2)

assignsoilmat("matid",2,"atpoint",50,20)

definelimits("limit",25,50,"limit2",70,95)

set("Method","BishopSim","on","Method","GLE/M-
P","on","Method","JanbuSim","on","Method","Spencer","on")

compute()
```