Tensile Strength Option

1. Introduction

This tutorial illustrates the Tensile Strength option in the Material Properties dialog. With this option, the user does not have to manually define a tension crack. After defining a tensile strength cutoff, the tension crack depth is automatically found.

The slope stability analysis is first carried out without using any tension crack. Then, the effect of a manually-added tension crack is examined, followed by the tensile strength option. The results are compared to an SSR analysis in RS2.

2. Model with No Tension Crack (Tensile Stresses Allowed)

From the main menu select **File > Recent Folders > Tutorials Folder**. Open the file *Tutorial 29 Tensile Strength (initial).slmd*.

MATERIAL PROPERTIES

Select **Define Materials** from the toolbar or the **Properties** menu. Notice that for all three materials, the **Tensile Strength** option is unchecked. This means that, in this analysis, for $\sigma_n' < c/\tan\phi,\ \tau = 0$.

Also, note that the **Upper Soil Layer** has a high cohesion (95 kPa) and relatively low friction angle (15 degrees).

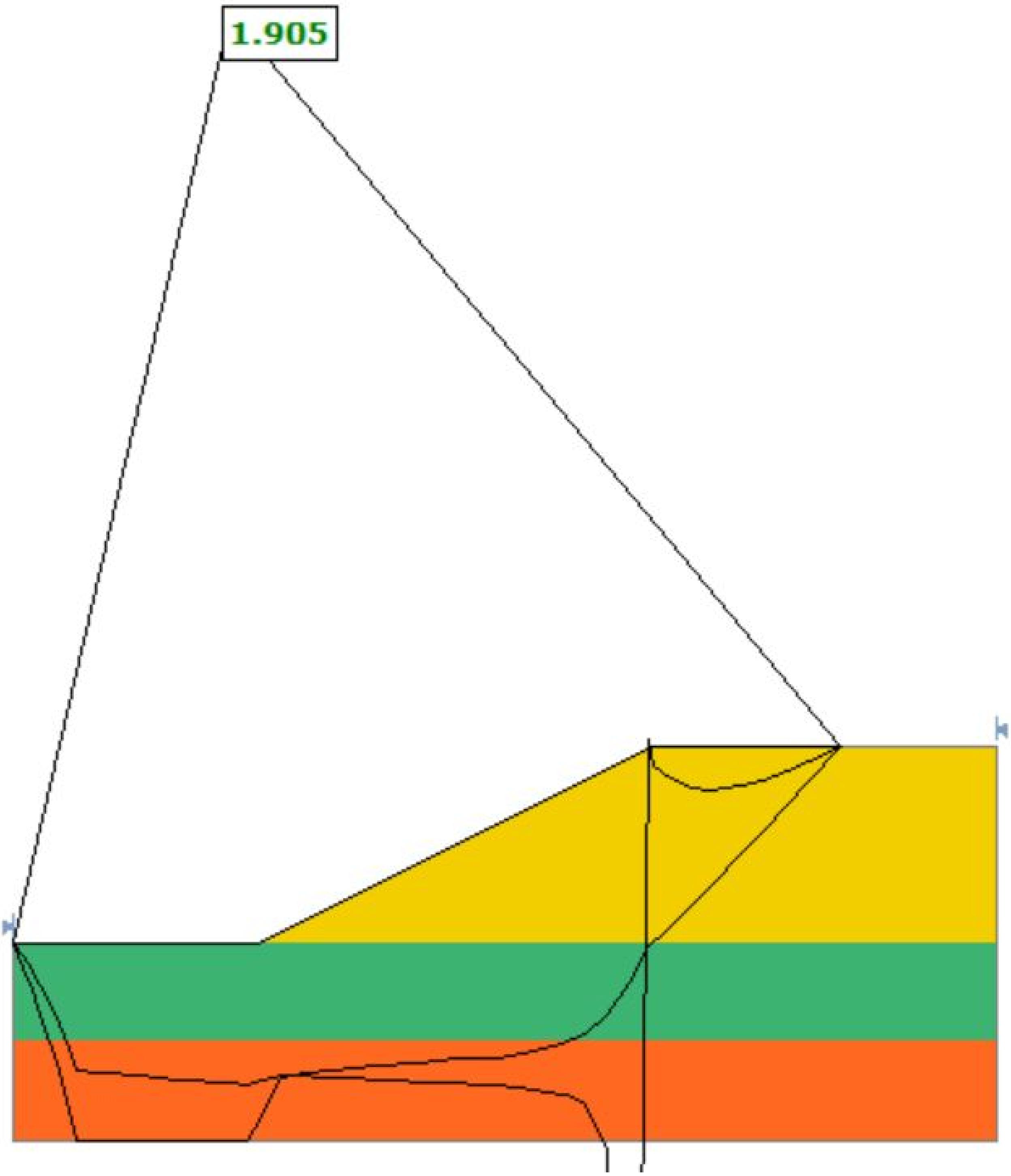
When there are cohesive soils in the upper slope layers, this often leads to tensile forces between slices or on the base of slices, in the limit equilibrium calculations. This usually leads to incorrect safety factors, for the following reasons (Ref. 1):

- Most soils do not have significant tensile strength, therefore calculated tensile stresses are unrealistic and inappropriate.
- When significant tension develops, this can cause numerical problems in the slope stability calculations.

RESULTS

Compute the file and open **Interpret**. You can see a factor of safety equal to about 1.9.

Go to the **Query** menu and choose **Show Line of Thrust.** The plot should look like this.



The thrust line gives the location of the resultant interslice forces, as mentioned in Tutorial 16: Tension Crack. The important thing to observe here is that the thrust line extends outside of the sliding mass. This generally indicates that tension is present.

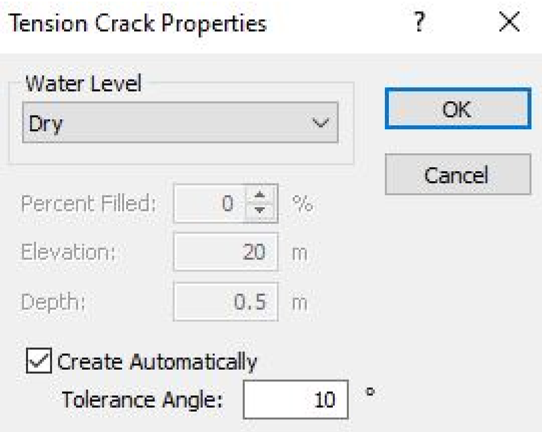
Tutorial 16: Tension Crack goes into detail about viewing the force balance on each slice and looks at the **Query Slice Data** option. We will not go into detail about this here. The key point is that because of this tension, and the fact that it is not accounted for in the model, the factor of safety is likely incorrect.

3. Model with Manual Tension Crack

Go back to the Slide2 modeller. To manually eliminate the tension, we'll add **a tension crack zon**e. We'll add the zone to the entire upper layer.

To add the tension crack:

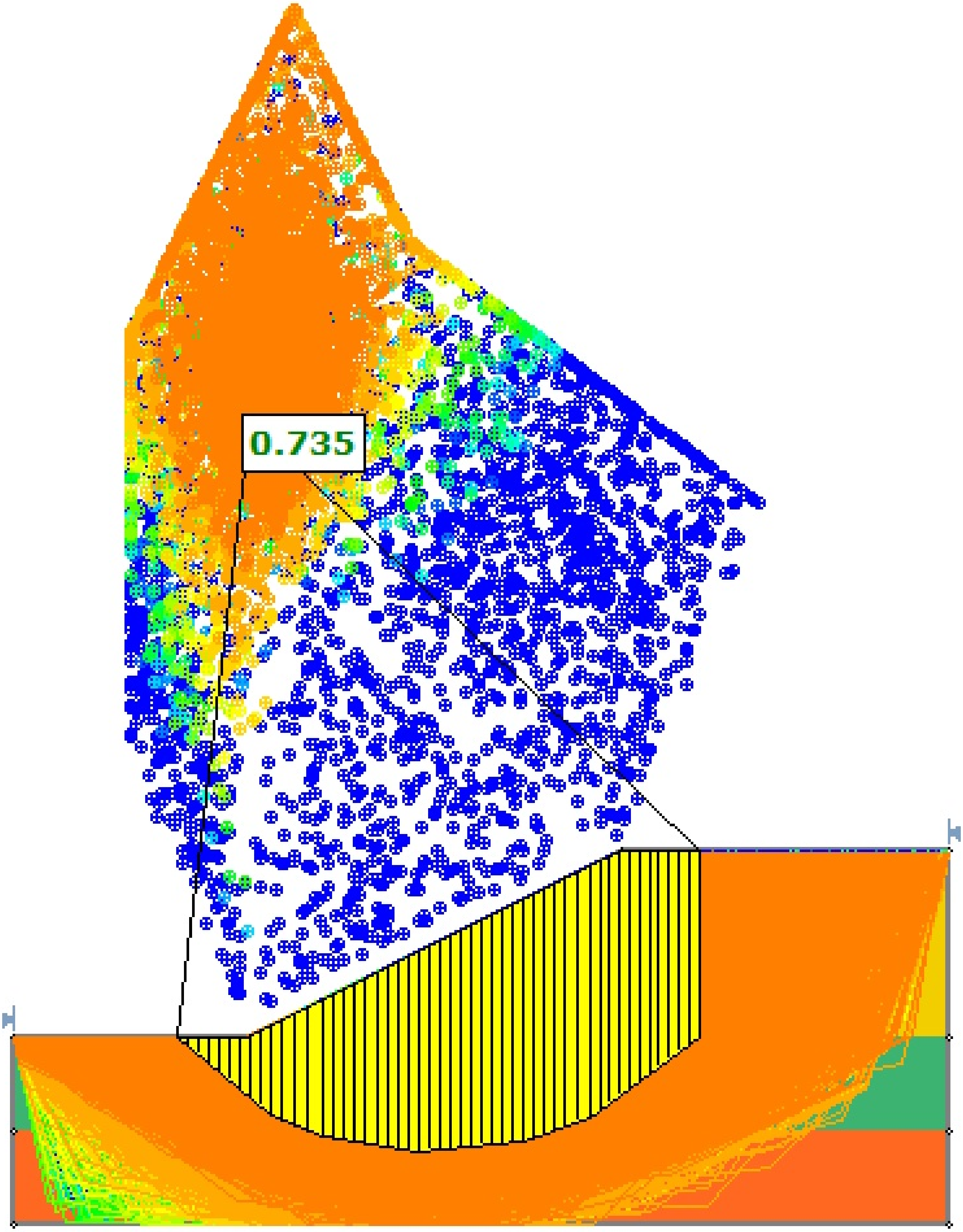
1. Go to the **Boundaries** menu and select **Add Tension Crack**.
2. Click on the two points **(10, 8) and (40, 8)** that define the upper soil layer in order to define the tension crack. By default, the **Water Level** of the tension crack is set to **Filled**.
3. Go to the **Properties** menu and select **Define Tension Crack**.
4. Set the **Water Level** to **Dry**.



RESULTS

Save the model, making sure to save it under a different name so that you don't overwrite the first file.

Compute the file and open Interpret. When you display the **Line of Thrust**, the results should look like this:



This plot shows several key differences from the example with no tension crack:

- Where the failure surface intersects the tension crack boundary, a vertical tension crack forms that extends to the ground surface.
- The line of thrust is completely inside the failure surface indicating that there is no tension in the soil mass.

- The factor of safety has decreased to about 0.73.

4. Model with Tensile Strength set to Zero

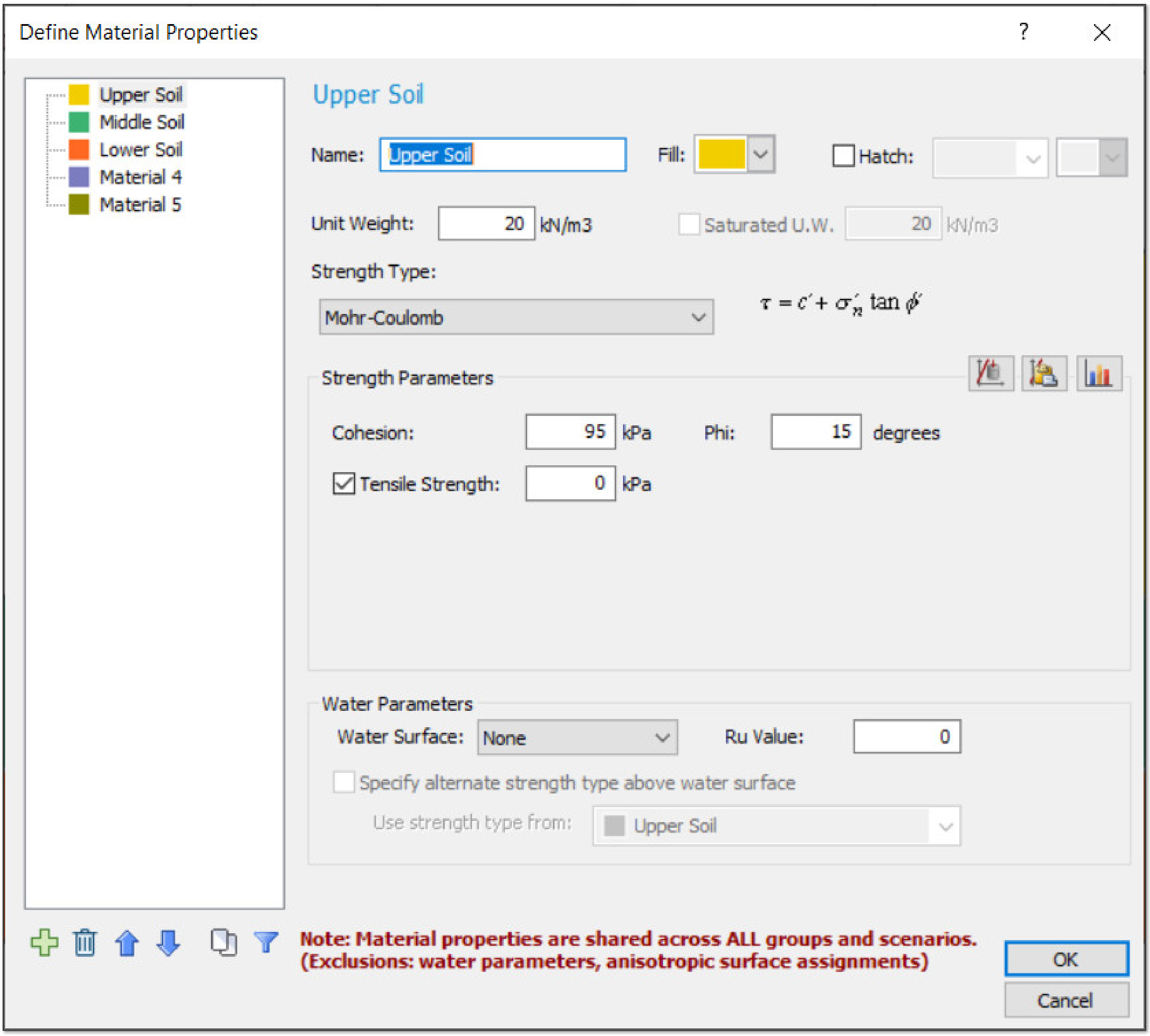
Instead of introducing a tension crack, the local factor of safety on a slice can be adjusted so that the effective normal stress is zero on the base of the slice. In this way, the tension crack is automatically defined.

Go back to the Slide2 modeller. Delete the tension crack (right-click on the tension crack and select **Delete Boundary**). In this analysis, we'll look at the use of the **Tensile Strength** option to automatically eliminate tensile forces.

MATERIAL PROPERTIES

Open the Material Properties dialog by selecting **Define Materials** from the **Properties** menu.

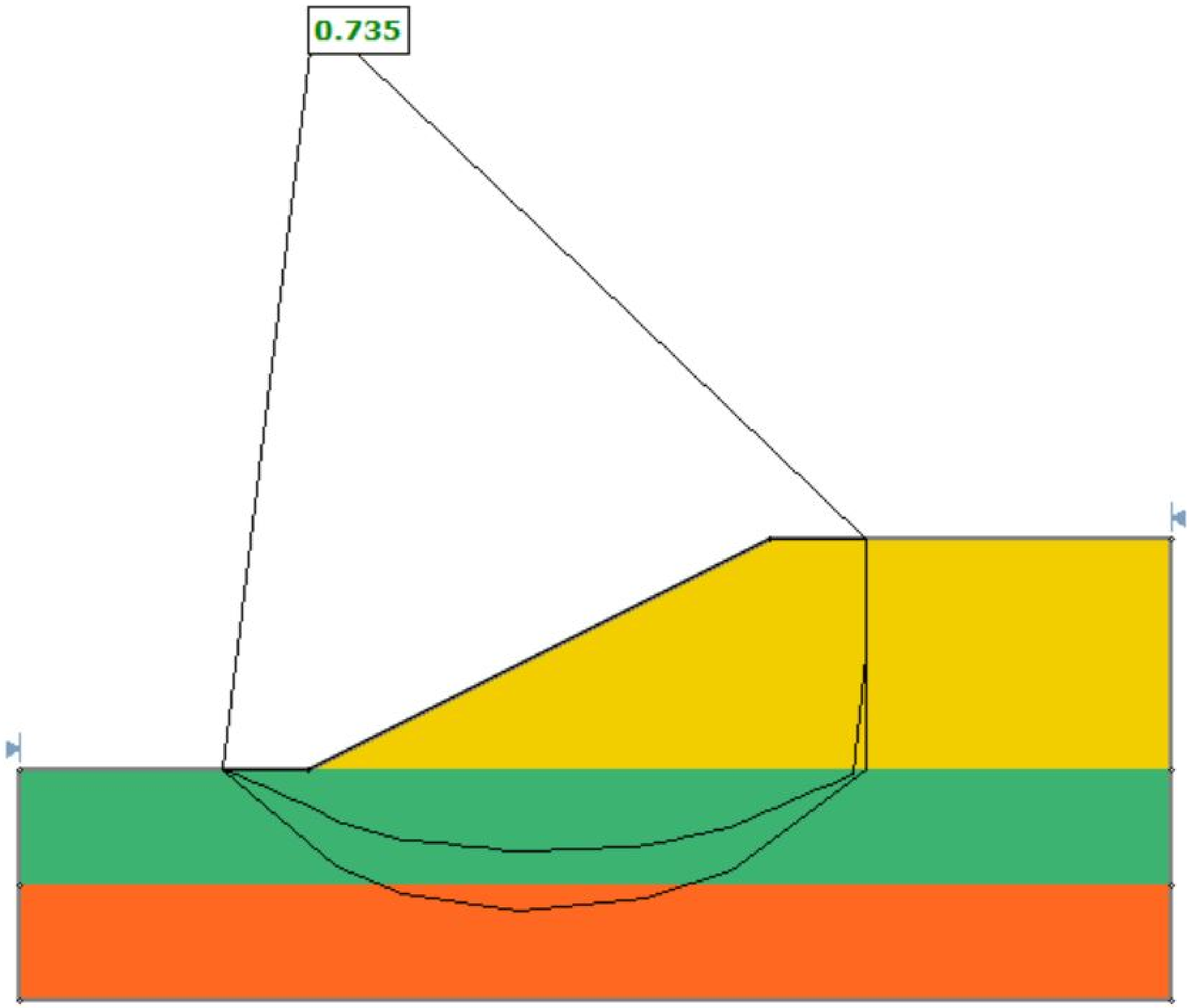
For the **Upper Soil,** select the **Tensile Strength** checkbox, as shown below. Leave the **Tensile Strength** at its default value of **0 kPa**.



RESULTS

Save the model, making sure to save it under a different name so that you don't overwrite the previous file.

Compute the file and open Interpret. The results should look like this when you display the line of thrust:



Note that the factor of safety is 0.73, the same as when the tension crack boundary was used.

5. Comparison with RS2 Analysis

For the comparison with RS2, we'll import the Slide2 file. Note that you'll have to export it to a single scenario file first.

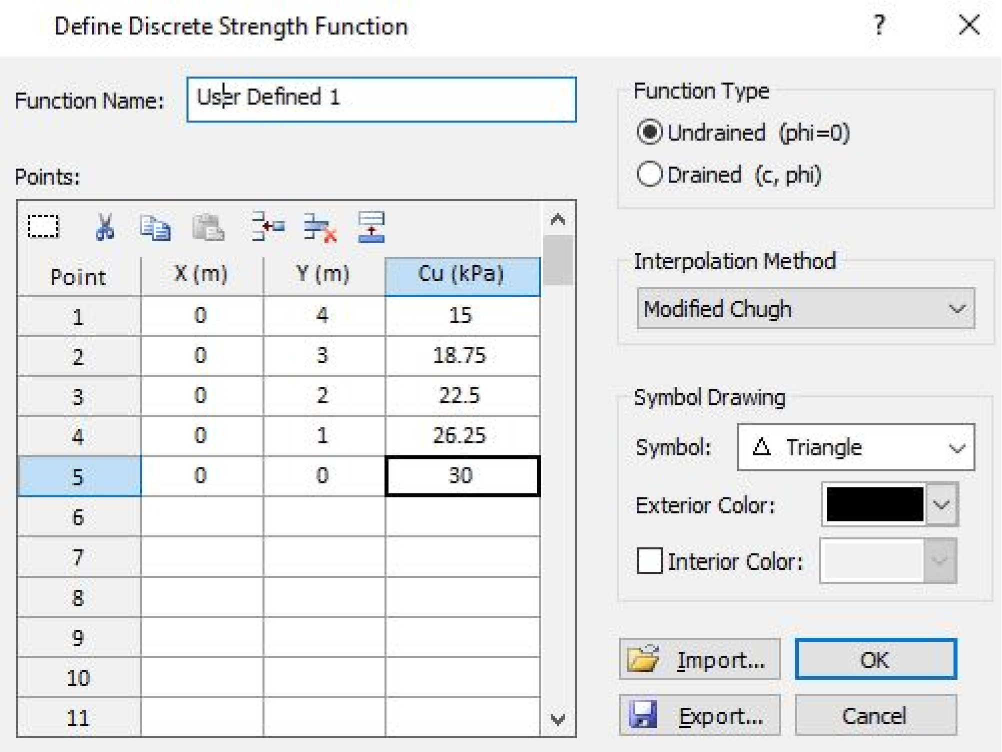
When you import the file into RS2, you'll get a warning message that the *Undrained F(depth) material model used in Slide2 is not available in RS2*. To fix this, we'll change the material properties for the **Lower Soil layer.**

In the **Define Material Properties** dialog, change the **Failure Type** to **Discrete Function**. Define the following function:

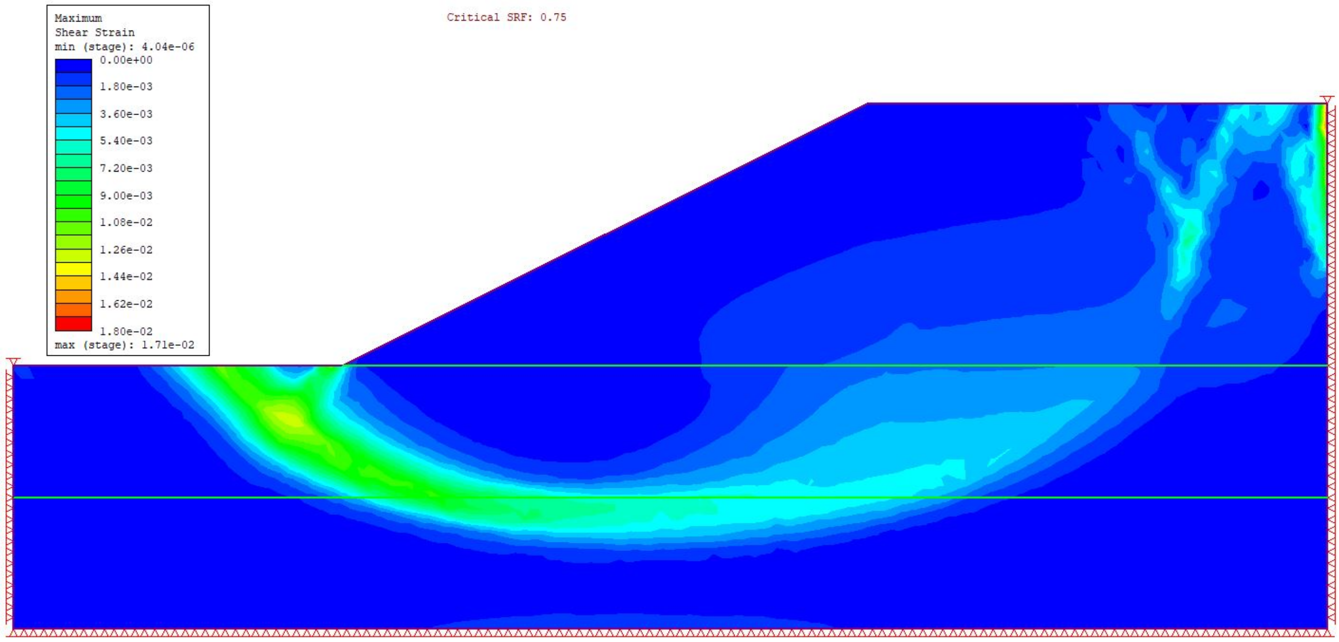
Function Name = User Defined 1

Function Type = Undrained

Point	X (m)	Y (m)	Cu (kPa)
1	0	4	15
2	0	3	18.75
3	0	2	22.5
4	0	1	26.5
5	0	0	30



The mesh can also be refined. Run the analysis and open Interpret. You should see the following results:



The Critical SRF of 0.76, is very similar to the factor of safety of 0.73 obtained from the Slide2 analyses.

6. References

Duncan, J.M., Wright, S.G. and Brandon, T.L. "*Soil Strength and Slope Stability*", 2nd edition, John Wiley & Sons, 2014.