Tension Crack Tutorial

1. Introduction

In slope stability analyses with cohesive soils, tension forces may be observed in the upper part of the slope. In general, soils cannot support tension, so the results of these analyses are not strictly correct. To obtain more accurate results, a tension crack boundary may be introduced. A tension crack essentially terminates the slip surface, thereby removing the tensile stresses from the calculations. In this tutorial, tension cracks are introduced into a **Slide2** model and different methods of determining the depth of the tension crack are explained.

The completed tutorial file is *Tutorial 16 Tension Crack.slmd*, which can be found in **File > Recent Folders > Tutorials Folder**.

2. Model with No Tension Cracks

Start the **Slide2** Model program. For this tutorial, we will start with the model from Tutorial 2. To open this file, go to **File > Recent Folders > Tutorials Folder** and choose *Tutorial 02 Materials and Loading.slmd*.

2.1 PROJECT SETTINGS

1. Open the **Project Settings** dialog from the **Analysis** menu and select **Methods** on the left.
2. Check the **Bishop simplified, GLE/Morgenstern-Price, Janbu simplified,** and **Spencer** methods.
   We need to use the **GLE/Morgenstern-Price** and **Spencer** analysis methods in order to obtain the thrust line, discussed later in the tutorial.
3. Click **OK** to close the **Project Settings** dialog.

2.2 COMPUTE

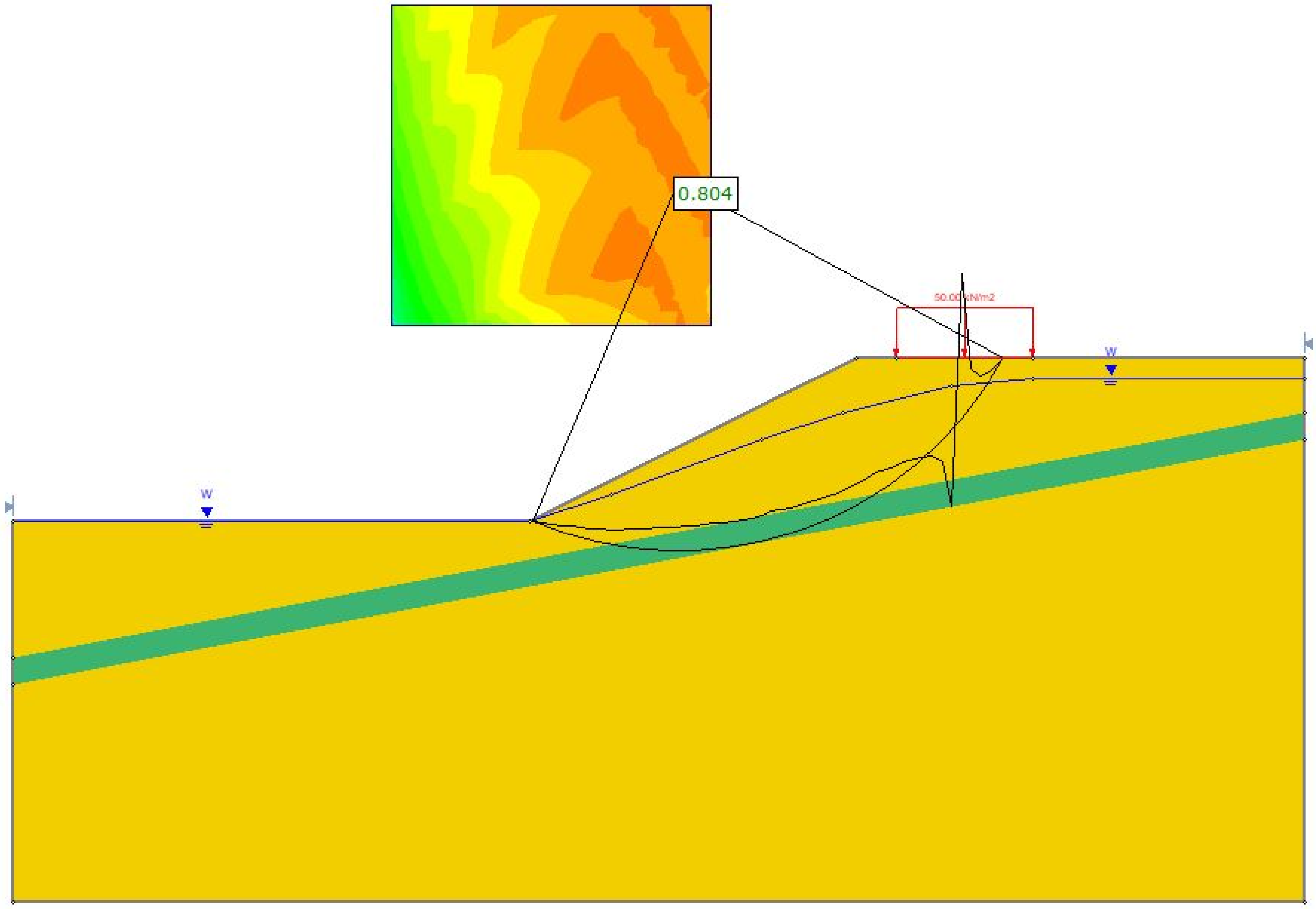
1. Save the model using the **Save As** option in the **File** menu.
   Be sure to name it something different so you don't overwrite the file for Tutorial 2.
2. Choose **Compute** from the **Analysis** menu to perform the analysis.
3. Choose **Interpret** from the **Analysis** menu to view the results.

2.3 INTERPRET

The Interpret program shows the results of the **Bishop Simplified** analysis by default.

1. Change the analysis to **Spencer** using the pull-down menu in the toolbar.
   You can see a factor of safety equal to about 0.8.
   For the Spencer (and GLE) methods we can plot a thrust line for the analysis.
2. Go to the **Query** menu and choose **Show Line of Thrust**.
3. The thrust line option is not available for Bishop or Janbu methods.

The plot should look like this:



Line of thrust in Spencer Method results

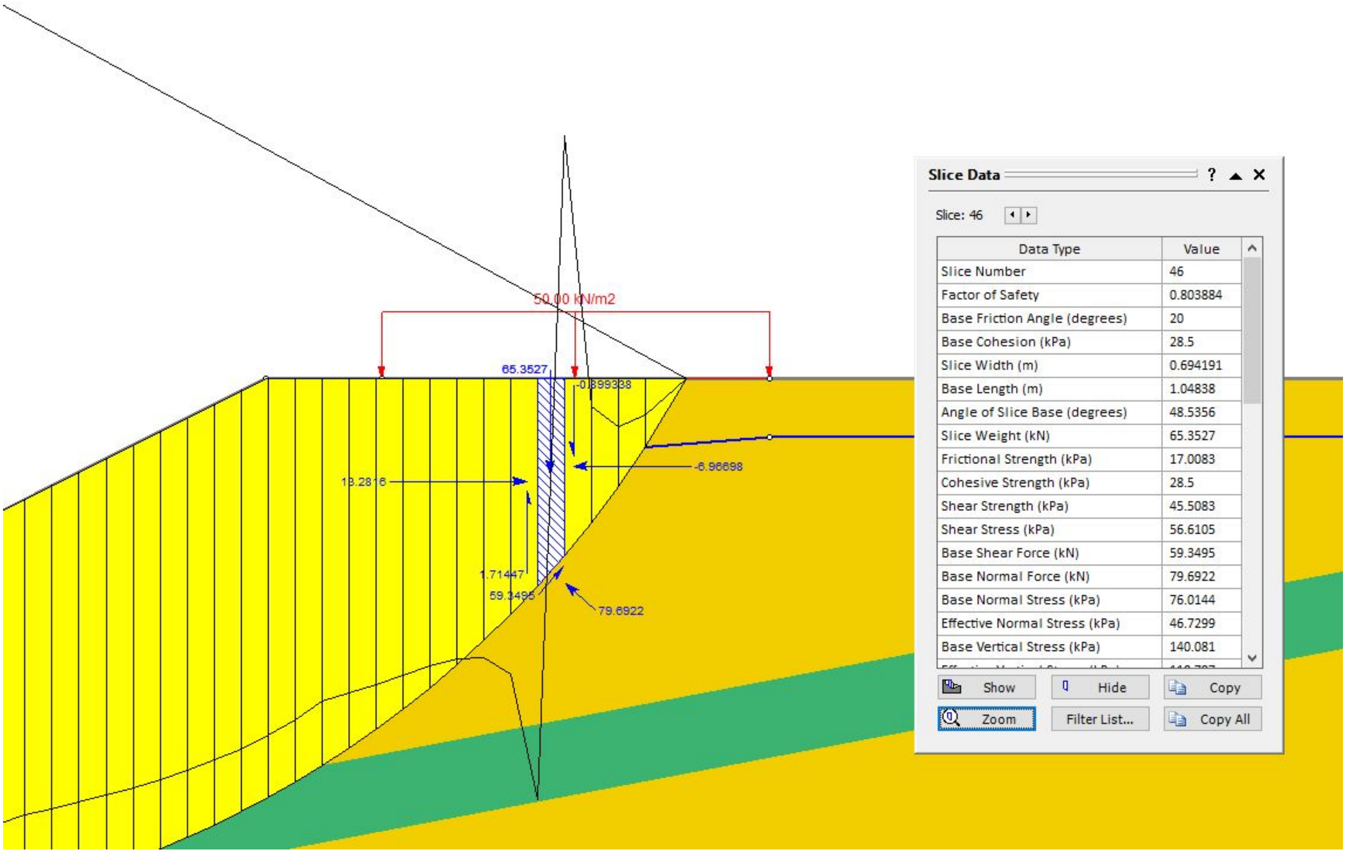
The thrust line gives the location of the resultant interslice forces (see Show Line of Thrust for more information). The important thing to observe here is that the thrust line extends outside of the sliding mass near the top of the slope. This generally indicates that tension is present.

To examine this further, we can view the force balance on each slice.

- From the **Query** menu, choose **Query Slice Data**.

When you click on individual slices you can see the forces. For slices near the top of the slope you can see that the interslice forces are negative (note the minus sign in front of the force magnitude).

The fifth slice from the top (Slice Number 46) shows negative forces on one side and positive forces on the other as shown. This slice represents the transition from tensile to compressive interslice forces in the sliding mass. The depth to the bottom of this slice can give a rough estimate as to the depth of the tension crack that will be required to eliminate the tension in the model. In this case, the depth is about 4.5 m.

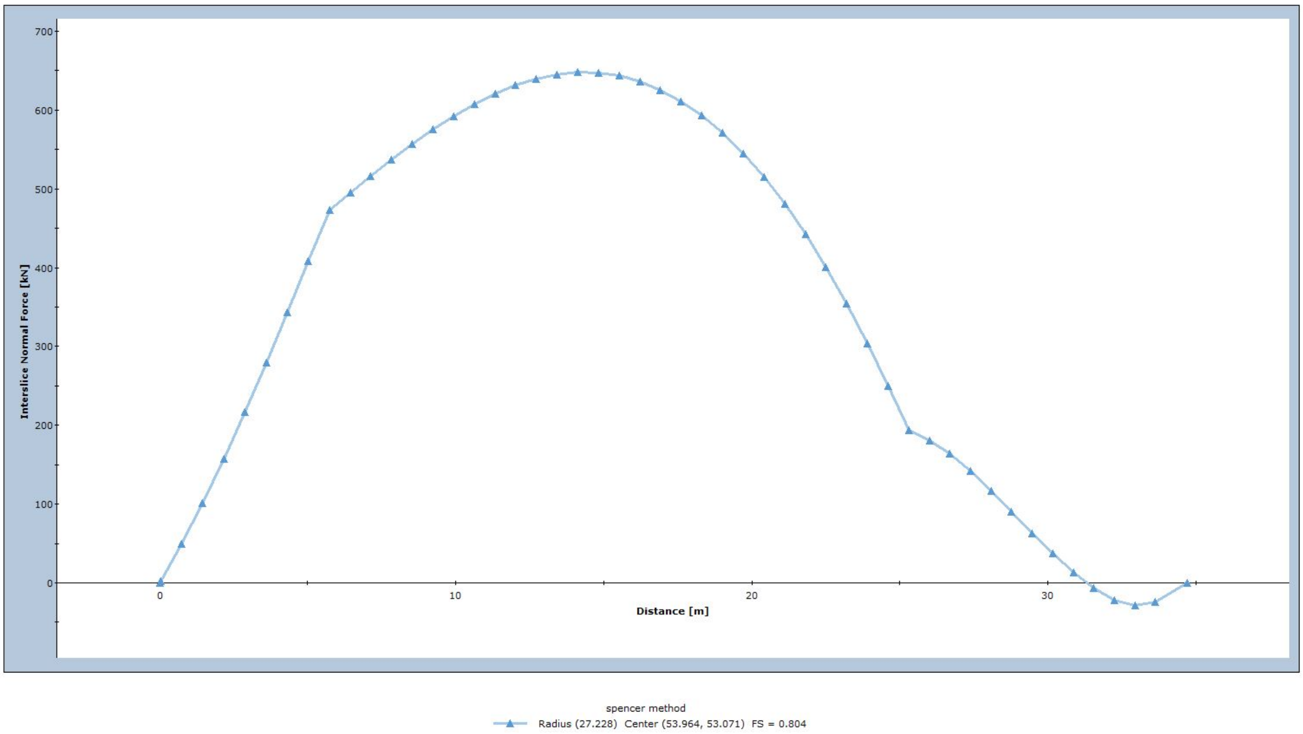


Interslice forces in slice number 46

We can also graph the interslice forces to observe the tension.

1. Close the **Slice Data** dialog.
2. Right-click on the slip surface and select **Graph Query** from the popup menu.
3. In the **Graph Slice Data** dialog, select **Interslice Normal Force** as the **Primary Data** to plot
4. Click the **Create Plot** button.

You should see the following graph. Notice the negative forces for the slices at the top of the slope. Also note that the absolute magnitude of the tension is relatively small, compared to the compressive forces further down the slope.



Interslice normal force (kN) vs. Distance (m)

We will now rerun the model with a tension crack boundary to try to eliminate the tension.

3. Model with Tension Crack Boundary

Go back to the **Slide2** model program. We now wish to eliminate the tension in the model. Duncan and Wright (2005, chapter 14) provide an informative discussion on tension in the active zone and describe how adding a tension crack can eliminate the effects of tension in the slope stability calculations. In **Slide2**, we can insert a tension crack boundary to delineate the lower extent of any possible tension cracks. The trick is to determine the depth at which to put this boundary.

As mentioned above, you could use the depth of the first slice in which tension is observed (~ 4.5 m). Alternatively, analytical equations have been derived to determine the depth for a tension crack, for example, Abramson et al. (2002) give this relationship:

$$z_c = \frac{2c}{\gamma}\tan\left(45 + \tfrac{1}{2}\phi\right)$$

Where zc is depth of tension crack, c is the material cohesion, φ is the angle of friction and γ is the unit weight of the soil material. This equation is derived in terms of effective stress parameters for a single homogenous material (although undrained strength parameters could also be used when appropriate). Using this equation a depth of 4.3 m is calculated, which is very close to our estimated value of 4.5.

A tension crack depth of 4.3 meters translates into a y-coordinate of 35.7 m, so we will use this to define the tension crack boundary.

We'll add a new scenario to look at the effect of the tension crack. Under **Master Scenario** add a new scenario and name it "Tension Crack". We will now edit this scenario.

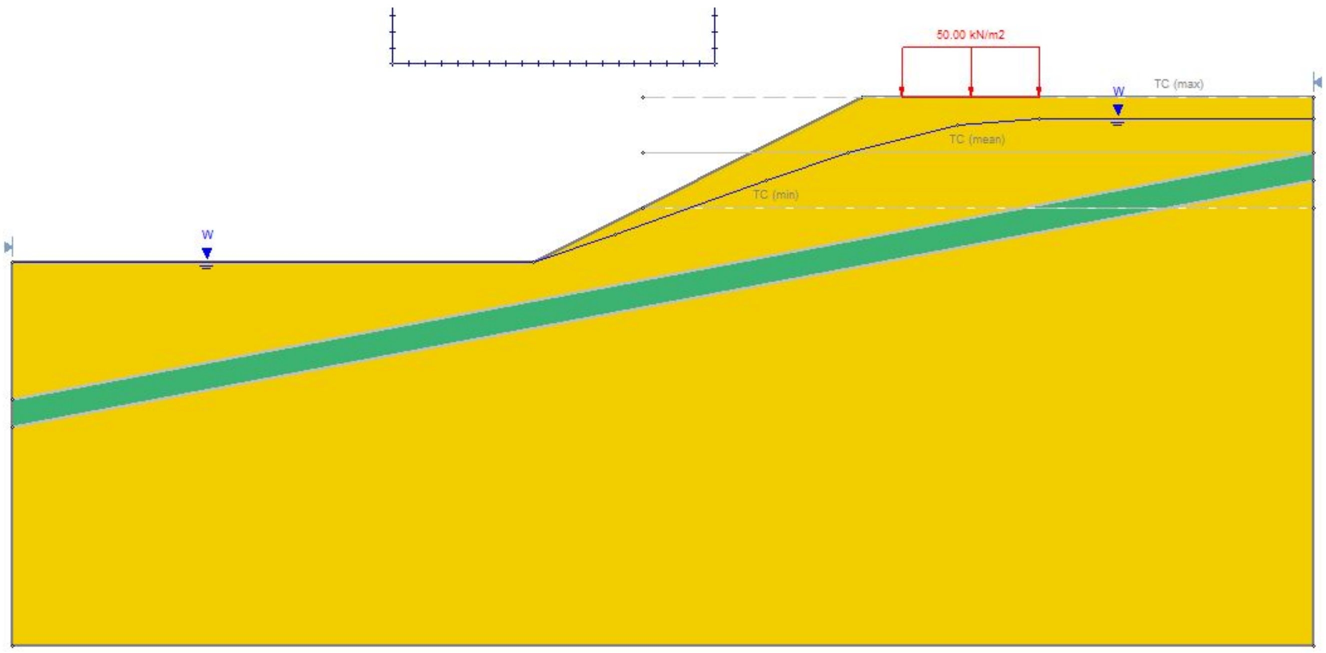
1. To add the tension crack boundary, go to the **Boundaries** menu and select **Add Tension Crack**.
2. Enter the right vertex (**100, 35.7)**.
3. We want the boundary to extend horizontally to intersect the slope, so right-click and ensure that **Snap**, **Ortho** and **OSnap** are all checked.
4. Close the context menu and draw a horizontal line that intersects the slope face.
5. Hit Enter to finish entering points.

You can see how the boundary delineates a zone of tension cracking. When a potential failure surface hits this line, it will ascend vertically to the ground surface to create a tension crack.

By default, the tension crack zone is assumed to be filled. A filled tension crack represents the worst-case scenario (it will give the lowest factor of safety). However, since we want to compare this model to the model with no tension crack, we will use the actual water table to define the crack saturation. To do this:

1. Right-click on the tension crack zone and select **Tension Crack Properties**.
2. For **Water Level**, choose **Use Water Table**.
3. Click **OK** to close the dialog.

The model will now look like this:



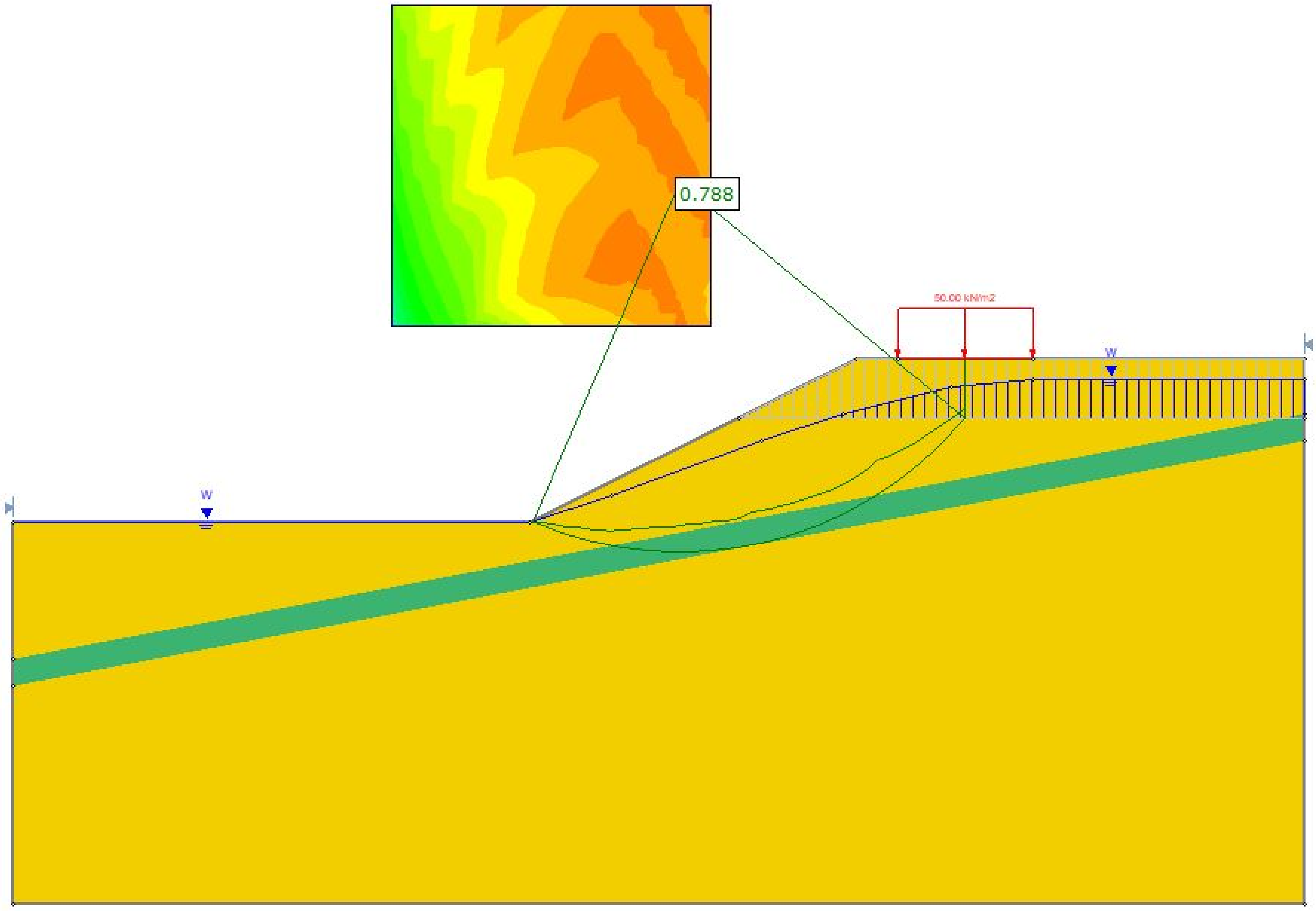
You can see how the potential tension cracks are now shown as saturated only up to the water table.

4. Compute

Save and Compute the model, then view the results in Interpret.

5. Interpret

As before, the Interpret program shows the results of the Bishop Simplified analysis. Change the plot to Spencer. Add the thrust line (Query → Show Line of Thrust) and the results should look like this:

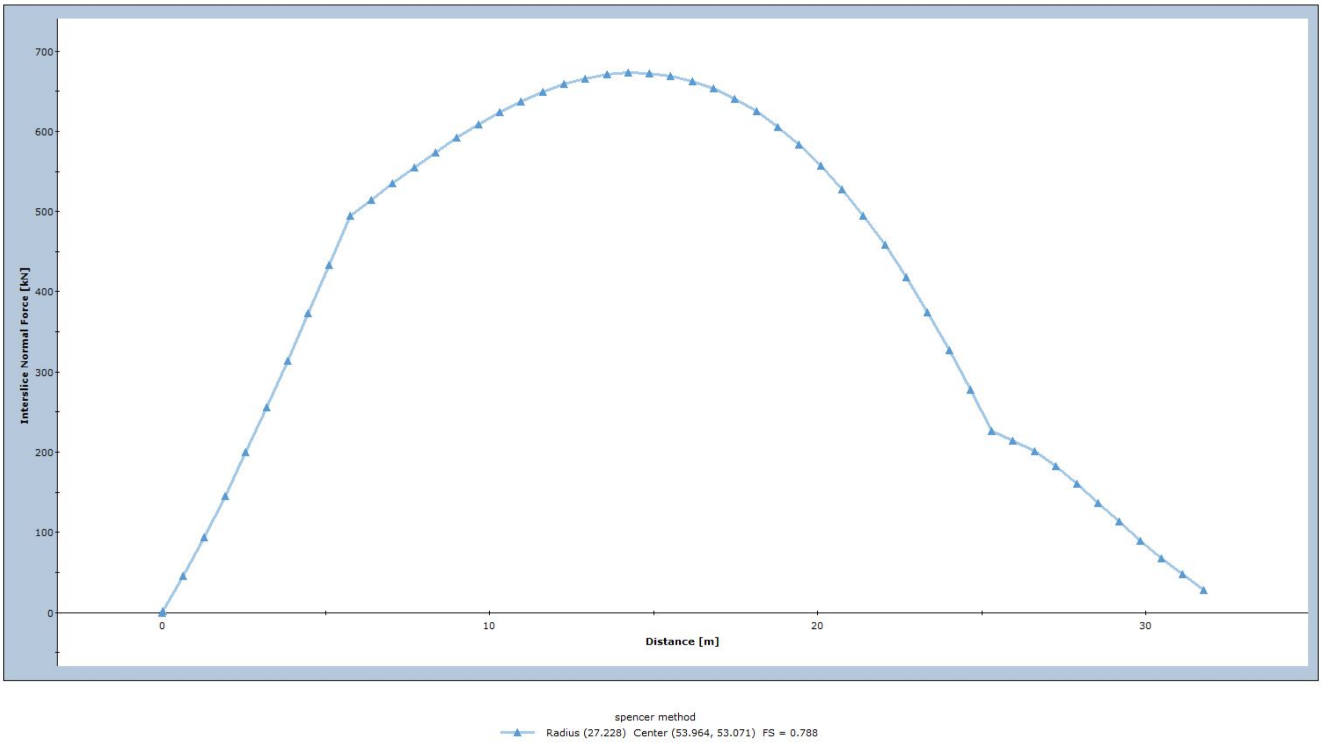


This plot shows several key differences from the example with no tension cracking:

- Where the failure surface intersects the tension crack boundary, a vertical tension crack forms that extend 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 slightly to about 0.79.

Although the difference in factor of safety is small, in general, it is good practice to introduce a tension crack zone for models which exhibit tensile interslice forces to obtain a more accurate failure surface and to eliminate possible numerical stability problems. For models with more extensive tensile zones or larger tensile forces, it may be essential to introduce a tension crack zone in order to obtain realistic results.

If we graph the interslice normal force as we did previously, the graph indicates compressive interslice forces for all slices with no tension present, as shown in the next figure. Note that the normal force on the last slice is not zero – this is due to the hydrostatic water force in the tension crack. If the tension crack zone were dry, the normal force on the side of the last slice would be zero.



Interslice normal force (kN) vs. Distance (m) after a tension crack is defined

5. Tension Crack Sensitivity Analysis

The tension crack depth calculated from equation 1 (4.3 m) eliminates the tension in the model and therefore produces more reliable results. There are many valid possibilities for tension crack depth that will eliminate the tension in the model. We now wish to determine the tension crack depth that minimizes the factor of safety. We can calculate this depth using a sensitivity analysis.

1. Go back to the **Slide2** model program.
2. Duplicate the **Master Scenario**, then delete the scenario under it.
3. Rename the new **Group 2 Master Scenario** to **Sensitivity**.
4. Open the **Project Settings** from the **Analysis** menu.
5. Click on **Statistics** on the left side and check the box for **Sensitivity Analysis**.
6. Then go to the **Methods** tab and reduce the tolerance to **0.001**.
7. Click **OK** to close the dialog.

We will now define the upper and lower limits of the tension crack boundaries, and **Slide2** will test 50 possible tension crack boundary locations in between these limits.

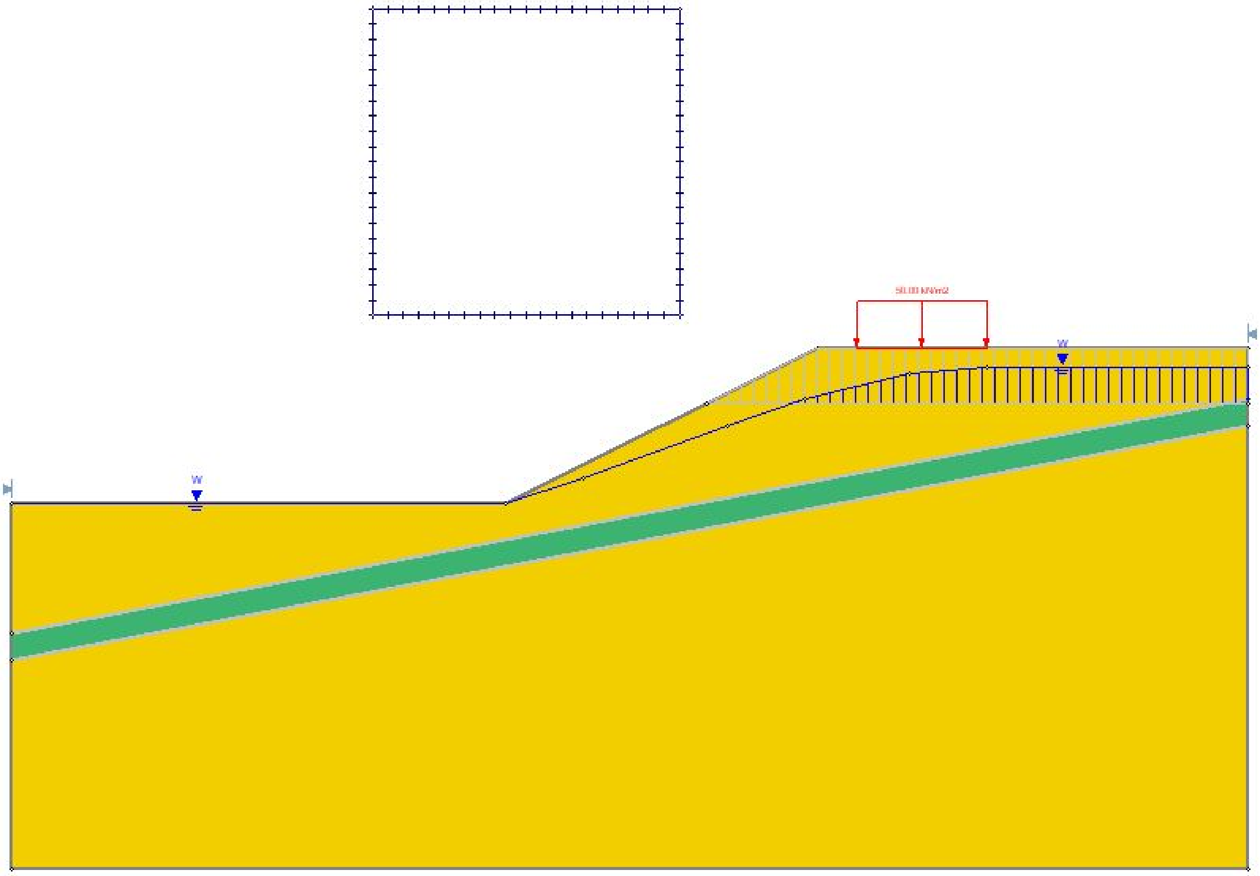
1. Go to the **Statistics** menu and select **Tension Crack > Draw Min Tension Crack**

1. Go to the **Statistics** menu and select **Tension Crack > Draw Min Tension Crack**.
2. Enter **100, 32** for the right coordinate.
   As before, ensure that all snapping is turned on by right-clicking and turning on all snap options.
3. Draw a horizontal line that intersects the slope surface.
4. Click the left mouse button at the intersection point.
5. Hit Enter to finish entering points.

You have now defined the minimum (bottom) tension crack boundary.

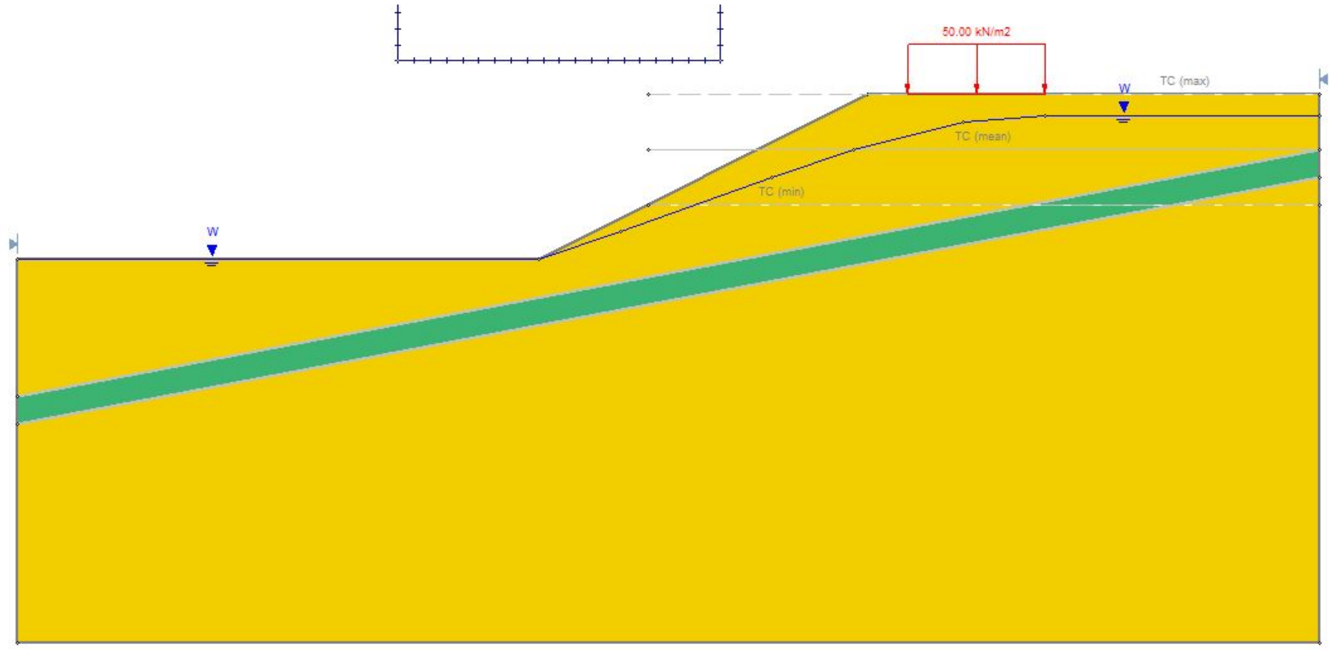
To draw the maximum (top boundary):

1. Go to the **Statistics** menu again and select **Tension Crack > Draw Max Tension Crack**.
2. Click on the top right corner of the model (**100, 40**).
   Now, it is important that the top boundary extends the same horizontal distance as the bottom boundary.
3. To get the second point, move the cursor to the left point of the bottom boundary and hold it there for a second but don't click!
   You will now see a dashed red line extending vertically.
4. Move up the vertical line so that you are drawing a horizontal line as shown.



5. Click the left mouse button to establish a point directly above the left point on the lower boundary.
6. Hit Enter to finish entering points.

You will now see the minimum, maximum and mean tension crack boundaries as shown



As before, we only want the tension crack saturated below the water table.

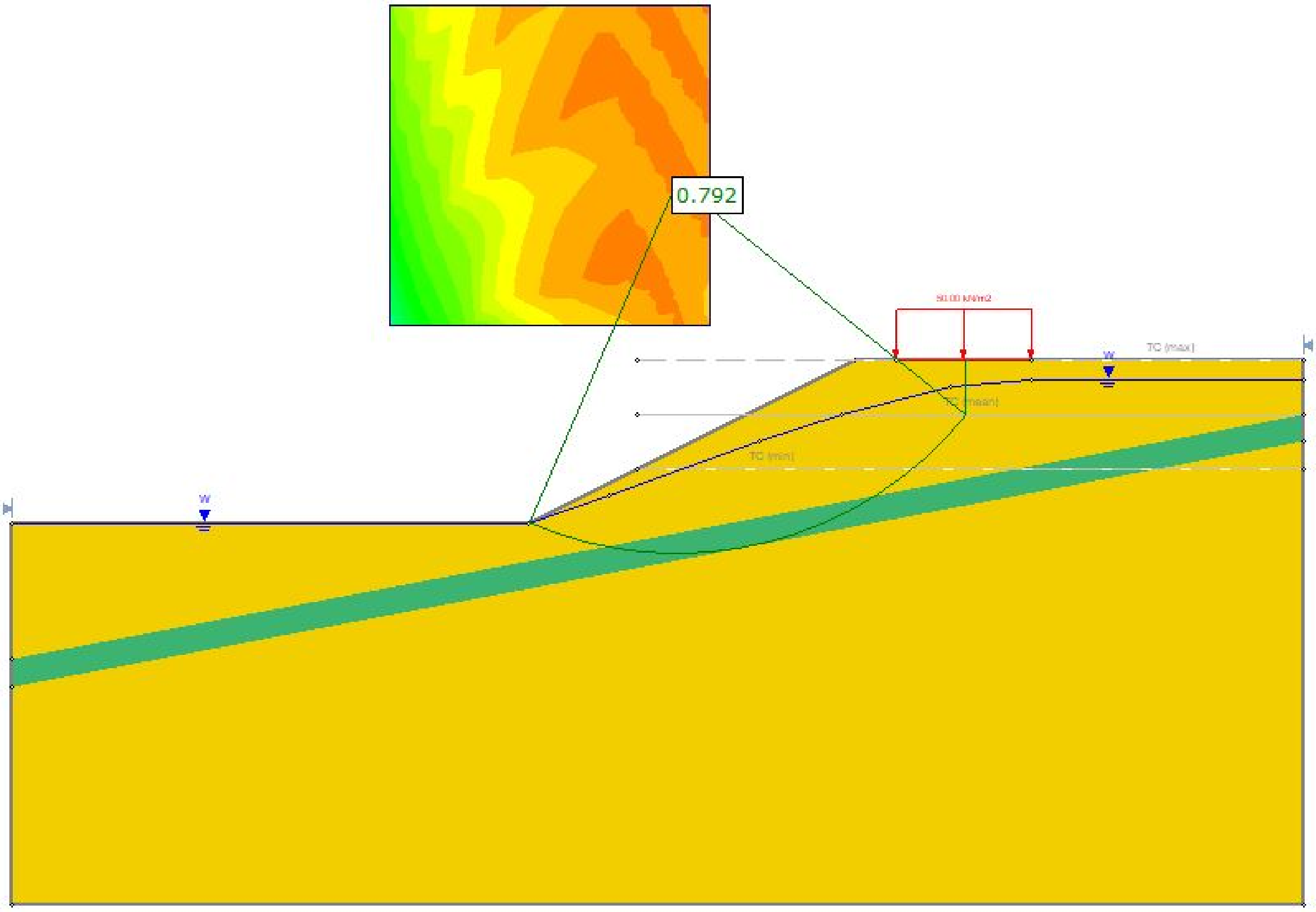
1. From the **Properties** menu, choose **Define Tension Crack**.
2. Set the **Water Level** to **Use Water Table**.
3. Click **OK** to close the dialog.

6. Compute

1. Save the model using the **Save As** option in the **File** menu.
2. Choose **Compute** from the **Analysis** menu to perform the analysis.
3. Choose **Interpret** from the **Analysis** menu to view the results.

7. Interpret

As before, change the plot to Spencer. Add the thrust line (**Query > Show Line of Thrust**) and the results should look like this:

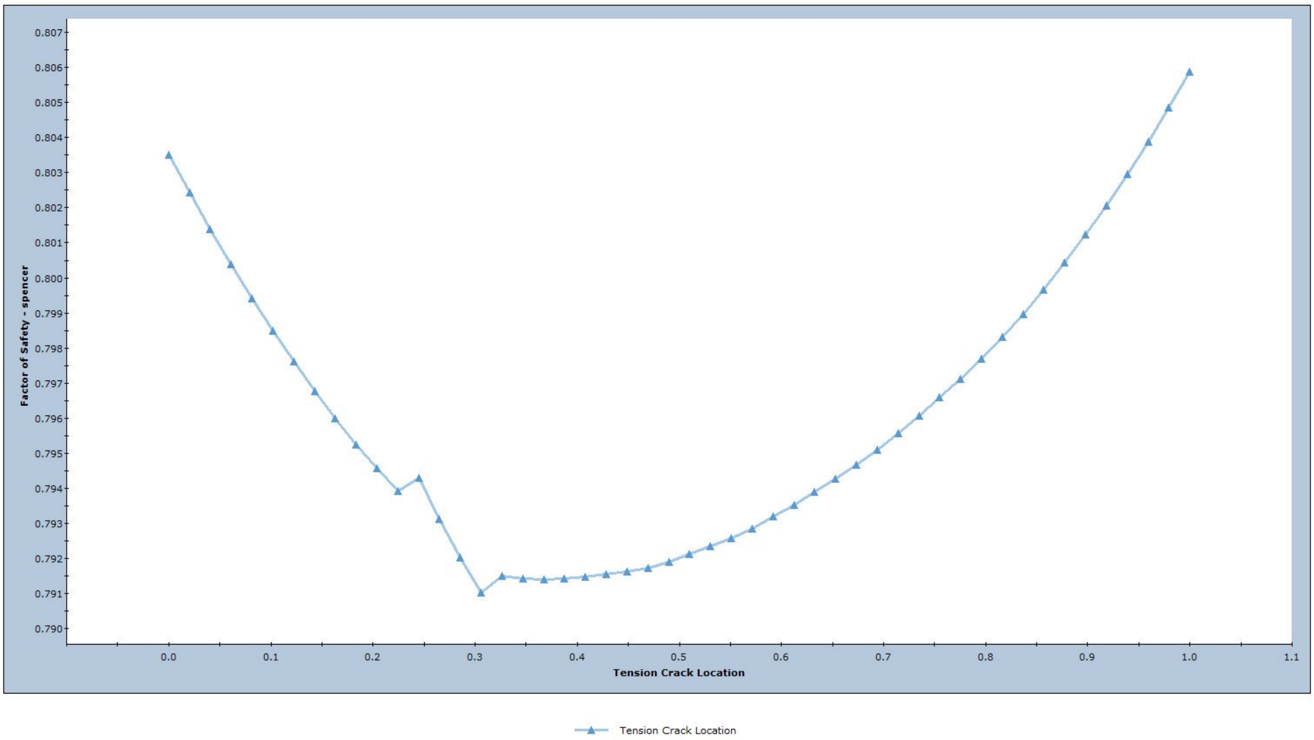


This plot is showing the results for the mean tension crack location. For the mean boundary (at a depth of 4 m), the thrust line is completely inside of the sliding mass indicating no tension, and the factor of safety is 0.792.

To examine the results of the sensitivity analysis:

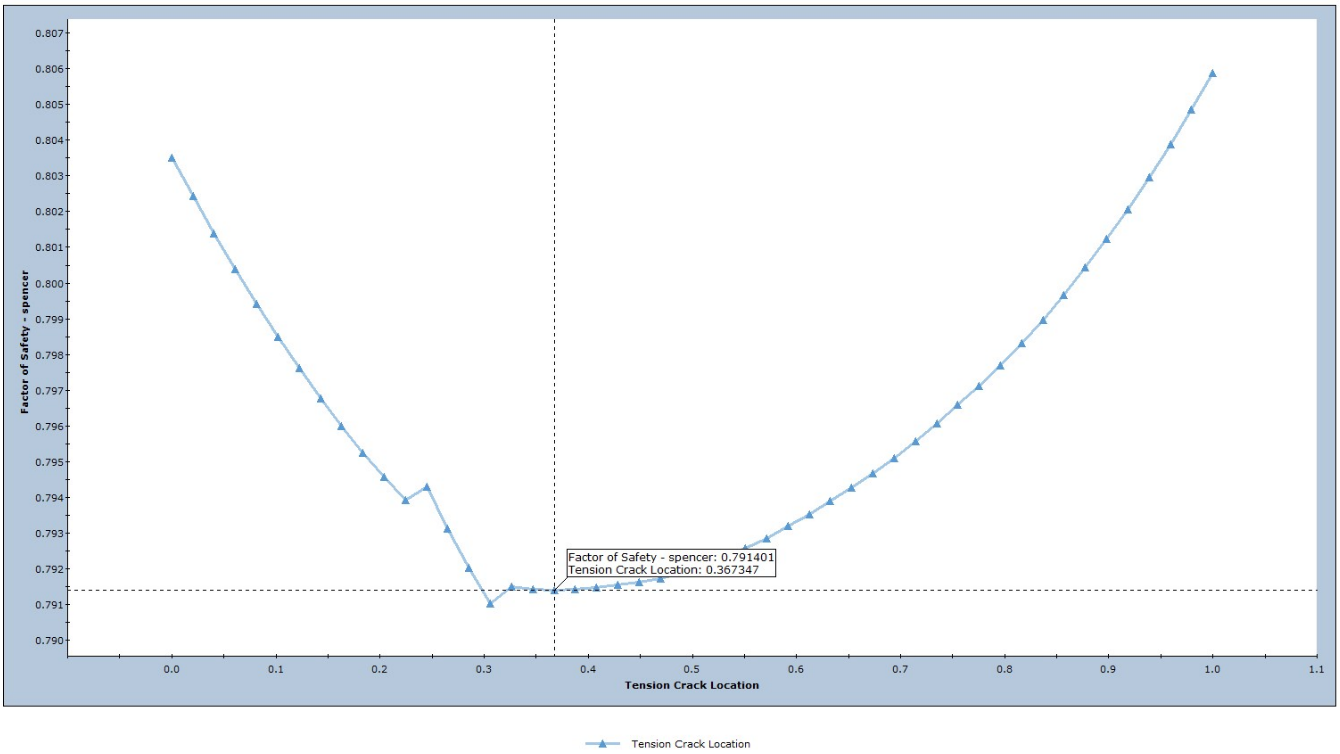
1. Go to the **Statistics** menu and choose **Sensitivity Plot**.
2. Choose the **Spencer** analysis method and select the check box for **Sensitivity – Tension Crack Location** under **Data to Plot**.
3. Click the **Plot** button.

You will see the factor of safety plotted versus the tension crack boundary location.



The tension crack boundary location is defined as a fraction of the distance from the minimum (32 m) to the maximum (40 m). The plot shows a clear minimum at a value of about 0.325. This, therefore, corresponds to a y-coordinate of about 34.6 m and therefore a depth of about 5.4 m.

To obtain more precise values from the graph, you can use the **Sampler** options. For example, right-click on the chart and choose the **Sampler > Show Sampler** option. Now hover the sampler on the lowest point of the graph. The minimum FOS tension crack location is indicated to be 0.367347, as shown in the following figure.



The minimum factor of safety is also obtained using the Sampler.

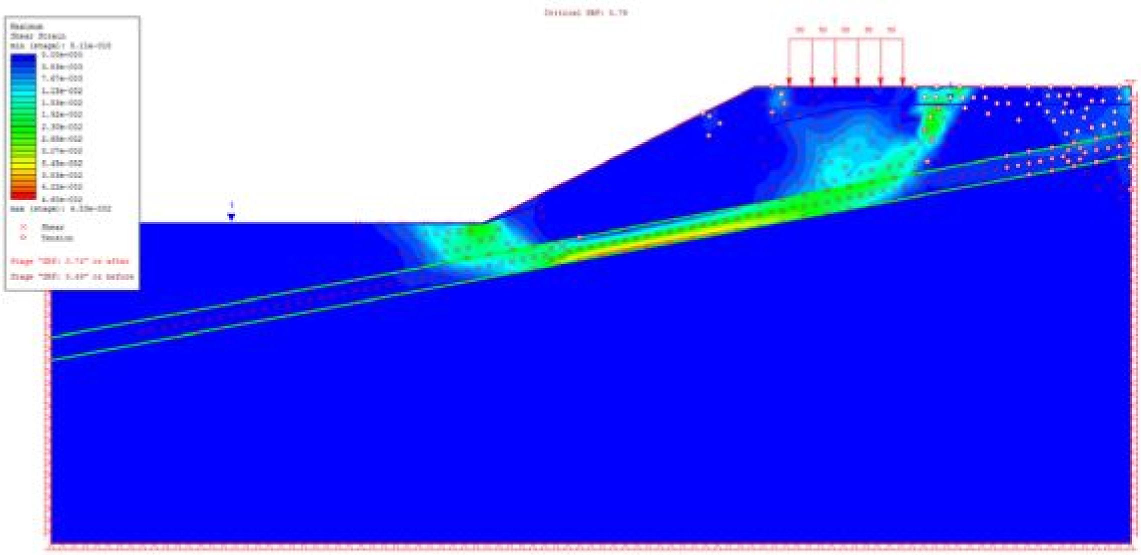
To summarize, the tension crack depth which minimizes the factor of safety for this model is about 5.4 m, as determined by the sensitivity analysis. This is about 1 meter more than the depth given by Equation 1 (4.3 meters) and the estimated depth of 4.5 meters, based on the tensile interslice forces of the original model.

This concludes the Tension Crack tutorial.

8. Additional Exercises

The same analysis can be performed in **RS2** (a finite element program from Rocscience) using the Shear Strength Reduction method. Start **RS2** and import the **Slide2** file from Tutorial 2. Set the number of elements to 1800 during the import process. Run the analysis.

The following plot shows the results at the critical Strength Reduction Factor of 0.79. The white circles indicate tensile failure. You can see that the zone of tensile failure varies from about 6 to 9 m deep.



9. References

Abramson, L.W., Lee, T.S., Sharma, S. and Boyce, G.M., 2002. *Slope stability and stabilization methods, second edition*, John Wiley & Sons Inc., New York.

Duncan, J.M. and Wright, S.G., 2005. *Soil strength and slope stability*, John Wiley & Sons Inc., New York.