

SOILVISION 10 Help Manual - 12/16/2019

Model Setup



The following steps will be required in order to set up the model described in the preceding section. The steps fall under the general categories of:

- a. Create model
- b. Enable multi-plane analysis
- c. Specify multi-plane analysis settings
- d. Analyze Model
- e. Results

The details of these outlined steps are given in the following sections.

NOTE:

Any values on the dialogs that are not specifically mentioned in the steps below are assumed to be the default values currently present.

a. Create Model

In order to create the tutorial model, save a copy of the [Open Pit Analysis](#) model. This is accomplished through the following steps:

1. Open the SOILVISION [Manager](#) dialog,
2. Go to **Expert Mode**,
3. Select the **MyProject** project and open the **Open_Pit** model, you may also begin with the **Open_Pit_wFault** model under the **Slopes_3D** if the Open Pit tutorial was not created,
4. Select *Models > Save Selected Model As* ... from the menu,
5. Type the name **MPA3** and click OK.

b. Enable Multi-Plane Analysis (Model > Settings)

In SVSLOPE the *Settings* dialog is used to specify the method for determining the critical slip surface and the details of the applicable search techniques to be used in the analysis. For this model the settings will be entered as follows:

1. Select *Model > Settings* ... from the menu,
2. Move to the *Multi-Plane Analysis* tab,
3. Select **Enabled - Three dimensional** Analysis. The default other settings in this tab are appropriate in most cases,
4. Move to the *Convergence* tab,
5. Check the **Minimum Slide Surface Volume** box and enter **5000000** m³. This is done because the automatic search methods will sometimes find very small or shallow slip surfaces, which is often not desirable.
6. Press *OK* to close the dialogs.

c. Specify Multi-Plane Analysis Settings (Model > Multi-Plane Analysis)

At this point, the existing search methods are disabled, and the model is ready for defining the multi-plane analysis.

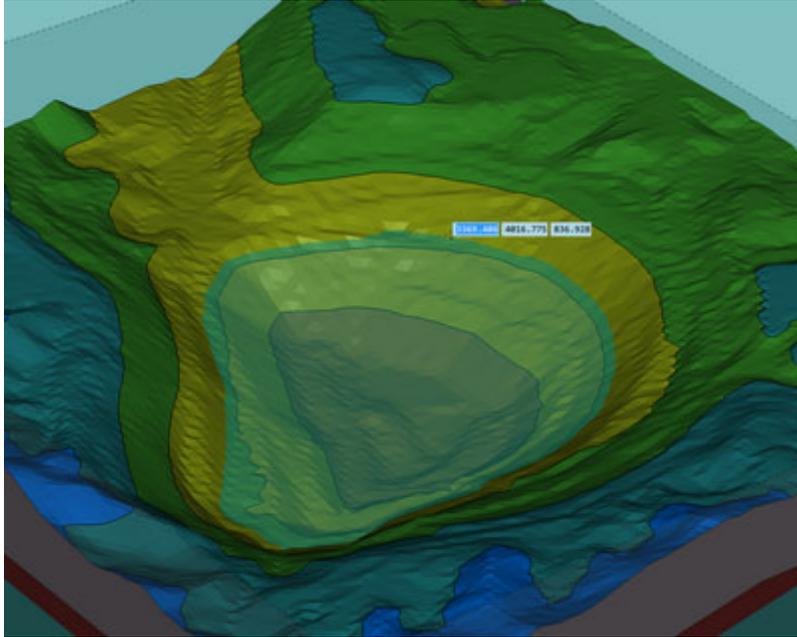
Create Planes

There are five ways to create planes. These are grouped into creating multiple planes automatically, or creating planes manually. Two of them (the elevation contour and polyline methods) are used to automatically create multiple planes. The other three (the draw planes, new and from points methods) are used to manually create individual planes. In this tutorial, the "From Elevation Contour" method will be used to create planes. This feature can be used to add many planes all around the pit in one action.

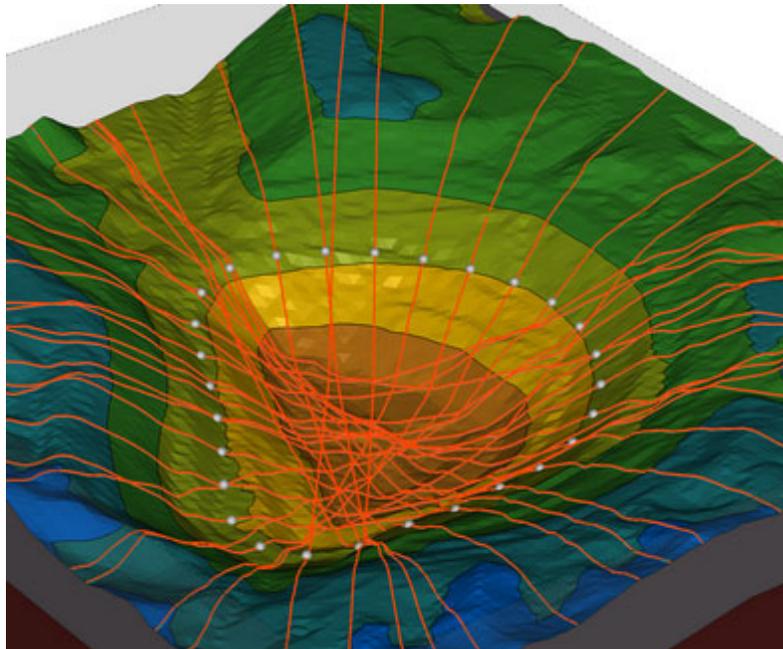
- From Elevation Contour method
 1. Select *Model > Multi-Plane Analysis...* from the menu,

2. In the *Create/Delete Planes* tab, set the **Distance between new planes** to **250 m**. This setting controls the distance between plane slope points for any planes that are created thereafter.
3. Then click the **Pick Elevation** button...
4. Click on any point between elevations of **600 and 850 m** in the open pit. Avoid picking a location high enough to spill outside of the main circular pit area.

Your screen should look like the image below when performing step 4.



Your model will look like the image below if the planes have been set correctly. The lines projected on top of the model represent the new planes. You should see a number of lines, each indicating a location and direction for a plane that represents a sub-model that will be created for analysis. Each of these slices will become a full 3D SVSLOPE model, and will be analyzed as any other such 3D model. You do not have to manage these individual sub-models yourself, since the system does that automatically. However, you are free to examine and modify the sub-models if desired.



Configure Planes

The next step is to configure the newly created slices. The configuration controls are in the Slice Data and Search Method tabs. Every setting in those two tabs acts only upon the multi-plane analysis planes that are currently selected. Each plane that was created is shown both graphically on the model, and in the tree view on the left side of the multi-plane analysis dialog. Since you have just created a set of planes, the newly created planes are already selected. You can confirm this by making sure they are orange-red colored on the graphical representation, and have a blue background behind their text in the tree view in the dialog.

1. Move to the *Slice Data* tab,
2. Change the **Search Limit Crest** to **900** m. The default settings span the entire extents of the model. While this is acceptable, more efficient results are produced by restricting the slope limits,
3. Change the **Search Limit Toe** to **900** m,
Keep the default settings of the slip direction and slope direction controls. The multi-orientation settings allow for testing multiple similar rotation angles for each plane,
4. Move to the *Search Method* tab,
5. Select **Slope Search** method from the Search Method droplist,
6. Change the **Number of Surfaces** to **300**,
7. Click **OK** to close the *Multi-Plane Analysis* dialog.

d. Analyze model (Solve > Analyze)

The next step is to analyze the model.

1. Select *Solve > Analyze*  from the menu. The **SVSLOPE Solver** dialog will pop-up and automatically solve.
2. Select the **Visualize** button to view results.

e. Results (Solve > Results)

The visual results for the current model may be examined by selecting the *Solve > Results* menu option or click on Results icon .

The model results will be displayed. To view the results in more detail proceed to [Results and Discussion](#).

NOTE:

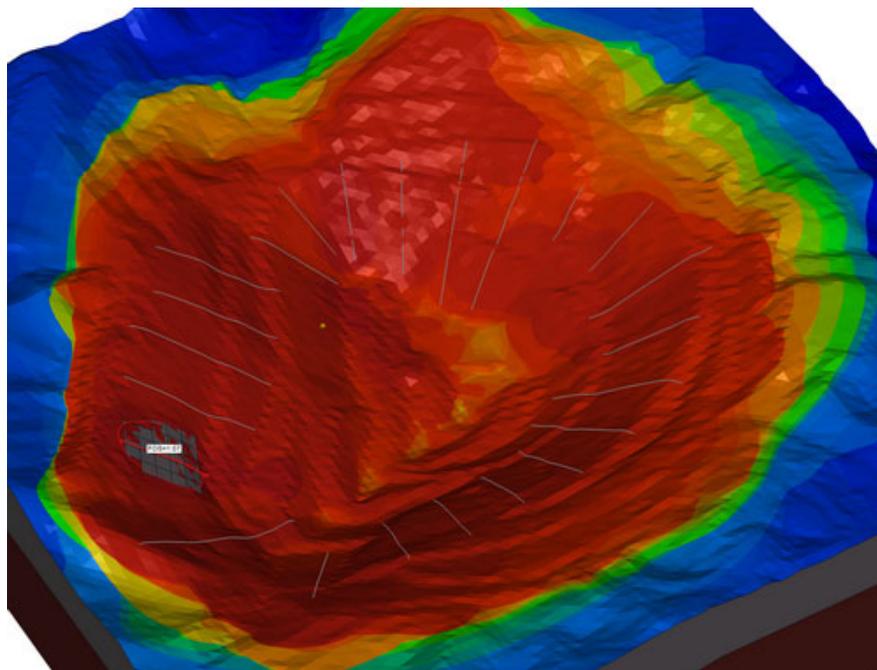
To transfer from viewing results to the SVSLOPE design module click on the SVSLOPE icon  found on the left vertical tool bar.

SOILVISION 10 Help Manual - 12/16/2019

Results and Discussion



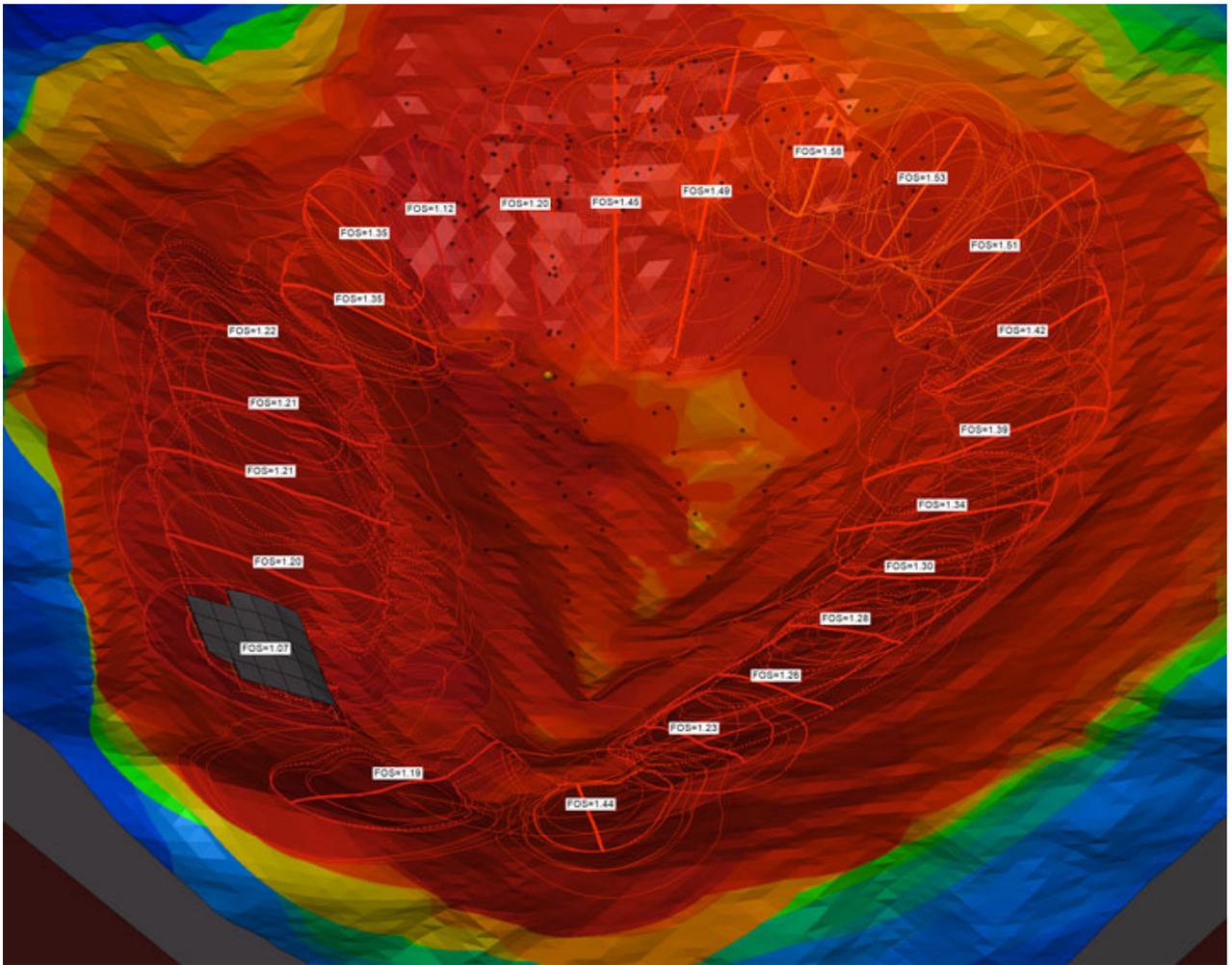
The model is displayed in the CAD with the lowest FOS for the whole model showing. The usual 3D SVSLOPE Results visualization features are shown as well: the critical sliding mass, the critical trial outline (where it intersects the model), and the factor of safety contouring.



1. Select *Slips > Multi-Plane Analysis Model Slices...* from the menu,
This dialog shows a breakdown of the results of each slice. Similar to the front-end controls, you may select a slice to see information about it such as the critical factor of safety.
2. Move to the *Visualization Options* tab and adjust the *Explosion Distance* slider. This will raise all the result visualizations upwards above the model, so that you can see the parts that would normally be hidden within the model.
3. It is usually also desirable to see the critical slip surfaces for the other planes. In the *Visualization Options* tab, from the "Show FOS for" drop-down, select one of the other two options that are not currently selected. Note the displayed FOS values and outlines for the critical surfaces throughout the pit. The bottom of the outlines will only be visible if the explosion distance slider is still raised, or if the model is made transparent in some way.

The user may also want to see some of the slip surface trials in each slice.

1. Select *Slips > Slip Surfaces...* from the menu,
2. Change the filter option to the 10 surfaces with the lowest factors of safety,
3. Then enable the checkbox to Show Trial Slip Surfaces,
4. Try changing the Results Filter Mode at the bottom right of the dialog. This setting allows you to filter each plane independently, or all planes together. For example, you can show the 10 most critical slip surface trials in the whole model, or the 10 most critical in each slice plane. The difference between "Per Slip Point" and "Per Slip Plane" is only apparent when multiple orientations are defined in the front-end dialog.



Trial slip surfaces at each plane location