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. Enter geometry
c. Specify SSR settings
d. Apply material properties
e. Specify boundary conditions
f. Mesh settings
g. Analyze model
h. Results

a. Create Model

The following steps are required to create the model:

1. Open the SOILVISION Manager dialog,
2. In LEARNING MODE, select the SVSOLID module icon and click New Model. The model is automatically stored in "MyProject" project.
3. Select the following:
   Module: **SVSOLID**
   System: **3D**
   Units: **Metric**
   Model Name: **3DSSR**
4. Click *OK* to close the dialog.

b. Enter Geometry (Geometry)

Model geometry is defined as a set of Regions and a series of Surfaces. Geometry can be either drawn by the user or defined as a set of coordinates. Model Geometry can also be imported from sources such as .DXF files or from existing models. In this example the geometry will be created by copying and pasting the geometry into the model.

1. Open the *Regions* dialog by selecting *Geometry* > *Regions*... from the menu,
2. Select R1 and click the ***Properties...*** button,
3. Click the ***New Polygon...*** button to open the *New Region Polygon* dialog,
4. Copy the points for Region *R1* from the table provided below and paste them into the *New Region Polygon* dialog by clicking the ***Paste*** button,
5. Click *OK* to close the dialog,

R1

X	Y
0	0
21	0
21	22
0	22

This model consists of two surfaces with differing dimensions and grid densities. By default, every model initially has two surfaces.

- **Define Surface 1**

This surface will be defined by providing a constant elevation.

1. Select *Geometry* > *Surfaces...* from the menu to open the *Surfaces* dialog,
2. Select the row containing **Surface 1** in the surface list,
3. Click the ***Properties...*** button,
4. For the *Definition Option*, select **Constant** from the drop-down,
5. Click on the *Constant* tab,
6. Enter an Elevation of **0**,
7. Click *OK* to close the dialog.

- **Define Surface 2**

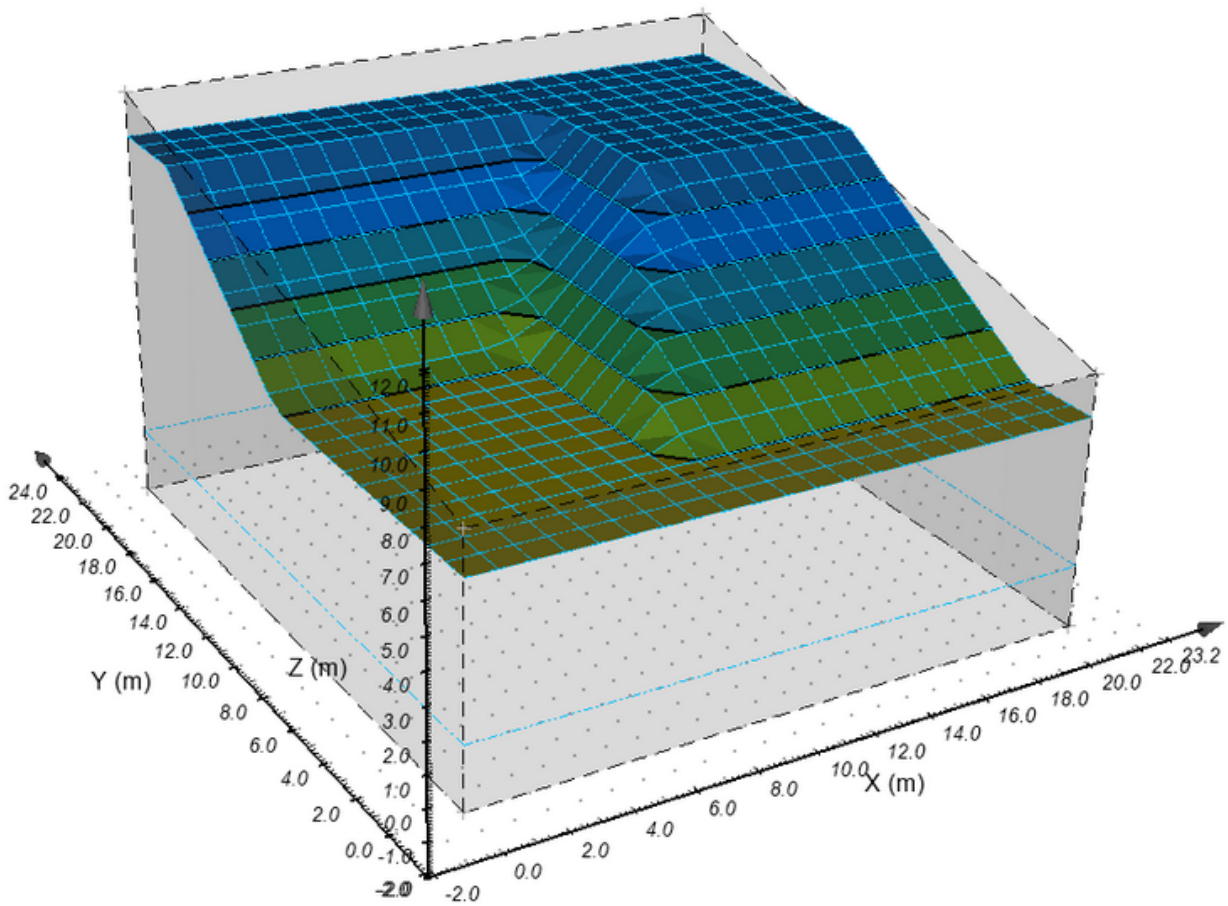
This surface will be defined by providing a regular grid of X and Y grid lines and corresponding elevations.

1. Select the row containing **Surface 2** in the surface list,
2. Click the ***Properties...*** button,
3. Select **Grid** from the Definition Options,
4. Click the ***Paste Data Grid...*** button to set up the grid for the selected surface,
5. Click on the ***Import From File...*** button,

6. Open the folder **"C:\Program Files\Bentley\Geotechnical\SoilVision V10\Tutorials"** in the Select A XYZ/Grid Data File dialog and select the file **SVSOLID Tutorial 3D SSR Surface 2.csv**,

   NB: This path is dependent on the folder chosen by the user when they installed SOILVISION, but it will always be in the sub-folder "Tutorials" of whatever path they chose to use.
7. Click *OK* to close the dialog, Click *No* on the pop-up asking if you want to keep existing grid points,
8. Click *OK* to close the *Surface Properties* dialog,
9. Click *OK* to close the *Surface* dialog.

Now your screen should look like the image below.



c. Specify SSR settings (Model > Settings)

The next step is to define the shear strength reduction settings.

1. Open the *Model Settings* dialog by selecting *Model* > *Settings...* from the menu,
2. Check the Calculate Shear Strength Reduction (SSR) factor,
3. Move to the *Shear Strength Reduction* tab,
4. Enter the parameters are shown in the table below,
5. Press *OK* to close the *Model Settings* dialog.

Settings	
Initial SSR Factor	1.0
Tolerance of SSR Factor	0.001
Step Size	0.1
SSR Searching Area	
X Minimum = 6	X Maximum = 16
Y Minimum = 2	Y Maximum = 12
Z Minimum = 0	Z Maximum = 10

d. Apply Material Properties (Materials)

The next step in defining the model is to enter the material properties for the material comprising the model. The model consists of one material. This section will provide instructions on inputting data for the material.

1. Open the *Materials Manager* dialog by selecting *Materials* > *Manager* ... from the menu,
2. Click the ***New...*** button to create a material,

NOTE:
When a "new" material is created, you can specify the display color of the material using the Fill Color box on the Material Properties menu. The color for the soil will be displayed for any Region that has a material assigned.

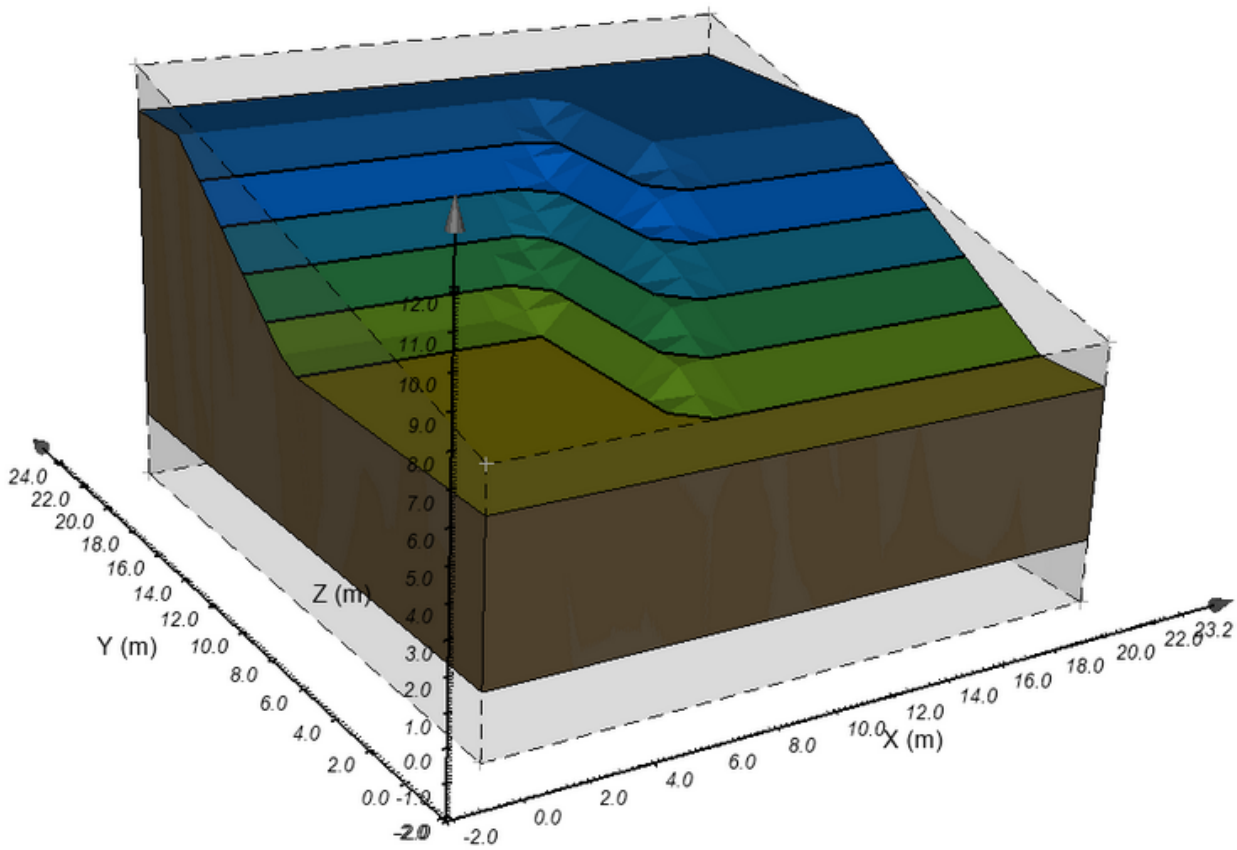
3. Enter **Soil** for the material name and set *Mohr Coulomb* as Method Type, then press *OK*,
4. The *Material Properties* dialog will automatically open,
5. Move to the *Parameters* tab,
6. Enter a *Poisson's Ratio* value of **0.4**,
7. Enter a *Young's Modulus* value of **50,000 kPa**,
8. Move to the *Loading* tab,
9. Enter the Coefficient of earth pressure at rest, Ko value of **1**,
10. Enter the *Unit Weight* as **25 kN/m^3**,
11. Move to *Shear Strength* tab,
12. Enter the Cohesion of **5 kPa**,
13. Enter the Friction Angle, phi of **12 deg**,
14. Enter the Dilation Angle of **0 deg**,
15. Press *OK* to close the dialog,
16. Press *OK* to close the *Materials Manager* dialog.

Material	Soil
Method Type	Mohr Coulomb
Young's Modulus, E	50,000 kPa
Poisson's Ratio, n	0.4
Coefficient of earth pressure at rest, Ko	1
Unit Weight, g	25 kN/m3
Cohesion	5 kPa
Friction Angle, phi	12 deg
Dilation Angle	0 deg

Once the material property has been entered, we must apply the material to the appropriate Region. Each Region will cut through all the Layers in a model, creating a separate "block" on each Region in each Layer. Each block can be assigned a material or be left as void. In this model there is only one Region and one Layer. The material is assigned to this block as follows.

1. Select *Geometry* > *Stage Settings...* from the menu to open the *Material Regions* dialog,
2. Select the Region Stage Settings tab,
3. Select the **Soil** Material for **Region 1** from the drop down,
4. Press the *OK* to close the dialog.

Now your screen should look like the image below.



e. Specify Boundary Conditions (Boundaries)

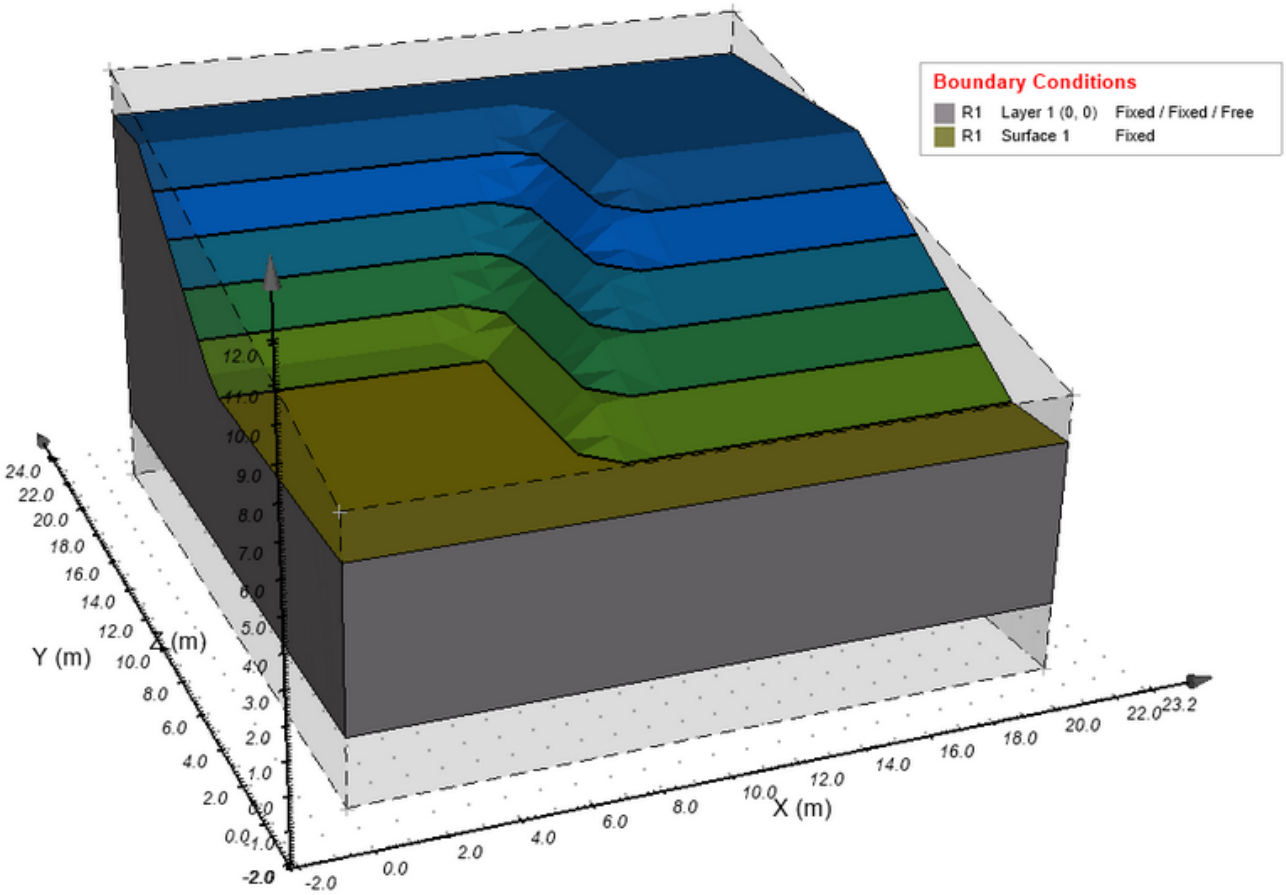
More information on boundary conditions can be found in *Boundaries* in the User's Manual.

Now that all of the Regions, surfaces, and materials have been successfully defined, the next step is to specify the boundary conditions on the Region shapes. The vertical boundaries of the slope will be fixed as will the base. The steps for specifying the boundary conditions are as follows:

1. From the menu select *Boundaries* > *Displacements...*,
2. Select **Surface 1**,
3. Select the *Sidewalls* tab at the top of the dialog,
4. Select point **(0,0)**,
5. From the *X* Boundary Condition drop-down for the first point, select a **"X Fixed"** Boundary Condition,
6. From the *Y* Boundary Condition drop-down for the first point, select a **"Y Fixed"** Boundary Condition,
7. From the Z Boundary Condition drop-down for the first point, select a **"Z Free"** Boundary Condition,
8. Ensure that the boundary condition for the other points are set to **Continue**,
9. Select the *Surface* tab at the top of the dialog,
10. From the *X* Boundary Condition drop-down select a **"X Fixed"** Boundary Condition,
11. From the *Y* Boundary Condition drop-down select a **"Y Fixed"** Boundary Condition,
12. From the *Z* Boundary Condition drop-down select a **"Z Fixed"** Boundary Condition,
13. Click *OK* to close the *Displacements* dialog.

NOTE:
The *Fixed* boundary condition for the point (0,0) becomes the boundary condition for the following sidewall segments that have a *Continue* boundary condition applied until a new boundary condition is specified.

Now your screen should look like the image below.

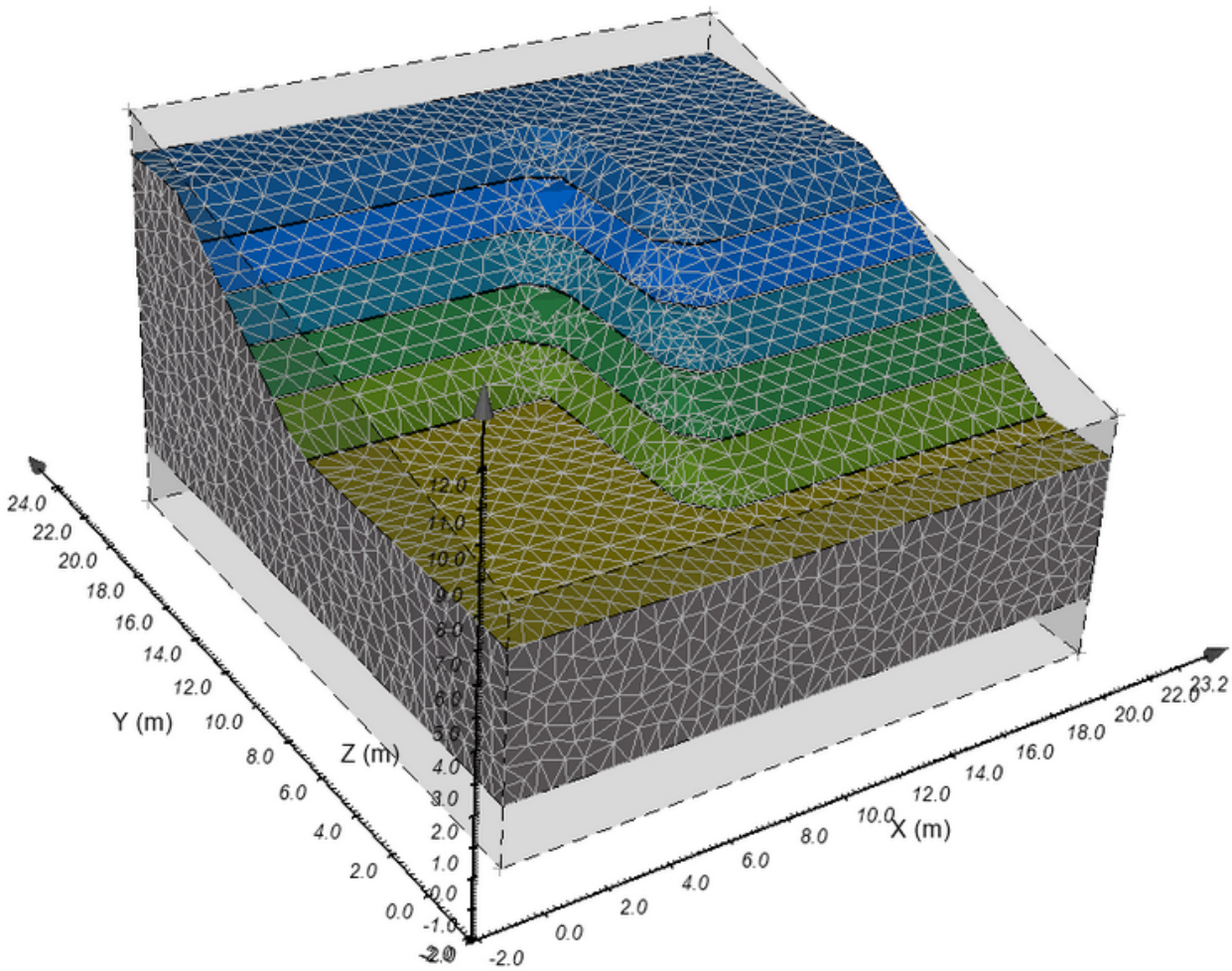


f. Mesh settings (Mesh > Settings)

The next step is to change the mesh settings.

1. Select *Mesh* > *Settings...* from the menu,
2. In the Global tab set the Maximum Tetrahedron Volume to **0.075** m^3,
3. Press *OK* to close the *Meshing Settings* dialog.

Now your screen should look like the image below.



g. Analyze model (Solve > Analyze)

The model is now ready for the analysis to be performed. Select *Solve > Analyze* from the menu. This action will write a descriptor file and open the SVSOLID solver. The solver will automatically begin solving the model.

h. Results (Solve > Results)

After the model has been run, you can click the *Open Results* icon on the left side toolbar. The SVSOLID screen will then change to visualize the results as shown in the Results and Discussion section. To switch back and forth between your original geometry and the results click on the SVSOLID or Results icon which appears below the toolbars on the top left hand side of the screen.

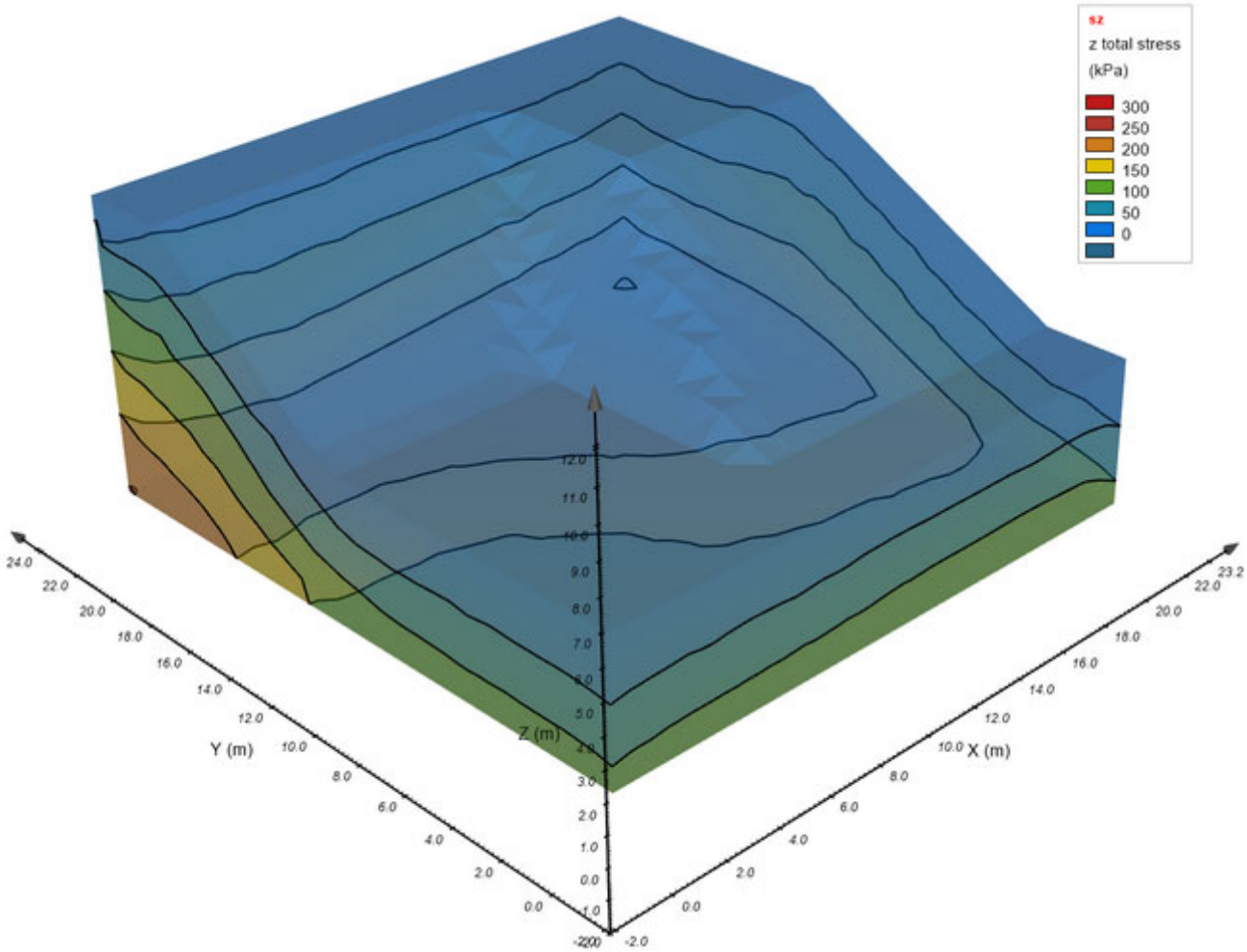
SOILVISION 10 Help Manual - 12/16/2019

Results and Discussion

When the computations associated with the analysis are complete, the user can visualize output plots using the Results module. In order to view plots, select *Solve* > *Results* from the menu.

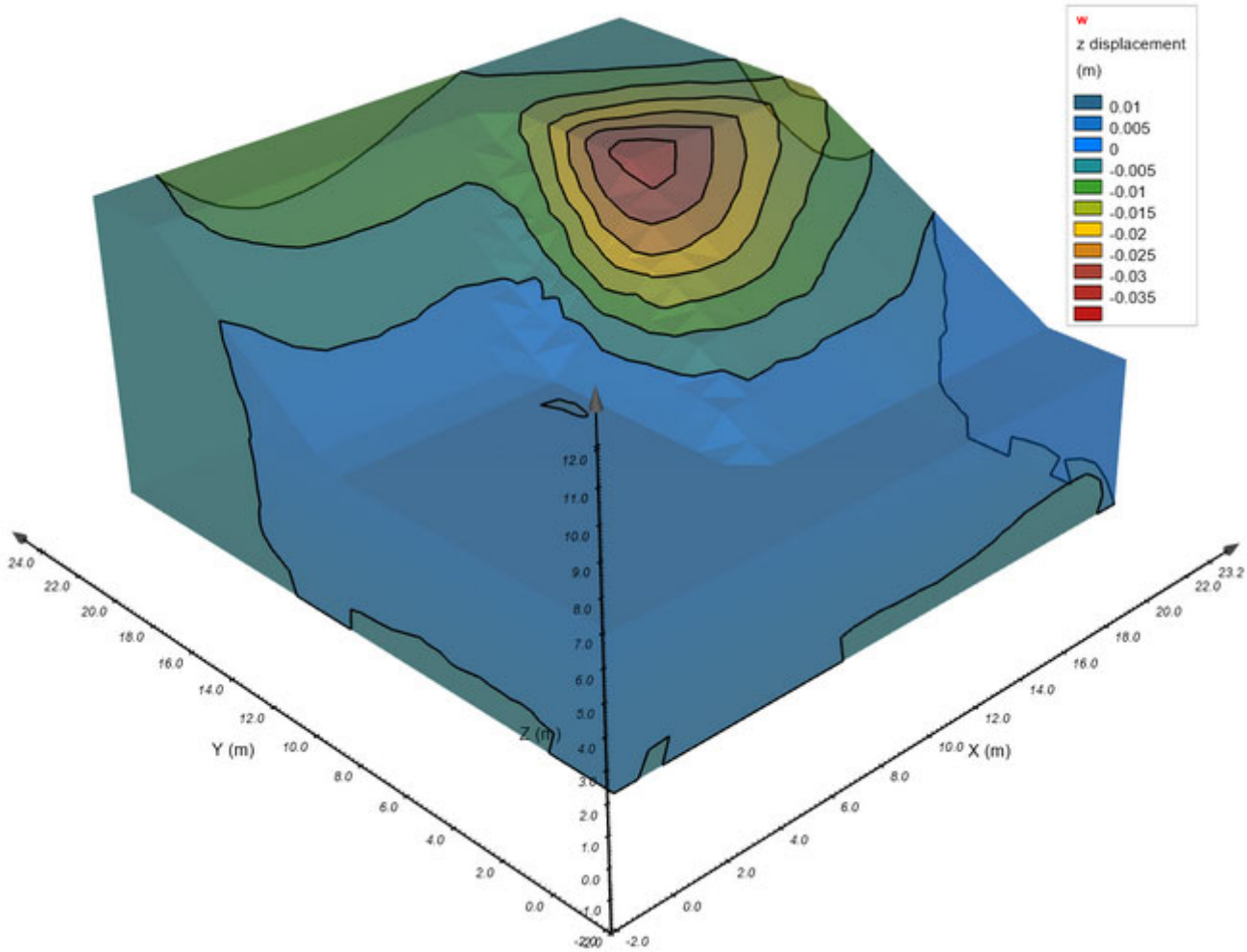
The calculated FOS for this model is 1.12.

- **Stress Contours**



The displacement contours are shown below. The areas with the most displacement indicating the weakest location are near the top corner of the slope.

- **Displacement Contours**


w
z displacement
(m)
0.01
0.005
0
-0.005
-0.01
-0.015
-0.02
-0.025
-0.03
-0.035
Y (m)
X (m)
Z (m)