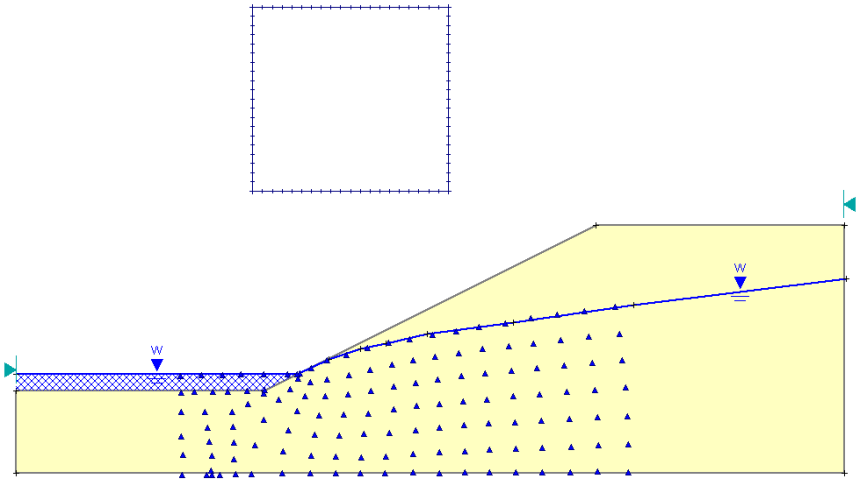
Water Pressure Grid Tutorial



This tutorial will demonstrate how to use a water pressure grid to model pore water pressure in *Slide*.

MODEL FEATURES:

- pore water pressure grid (total head)
- ponded water above slope, defined by water table
- circular surface search (Grid Search)

The finished product of this tutorial can be found in the **Tutorial 05 Water Pressure Grid.slim** data file. All tutorial files installed with *Slide* 6.0 can be accessed by selecting File > Recent Folders > Tutorials Folder from the *Slide* main menu.

This model is also presented in the Groundwater Tutorial (file: **Tutorial 07 Groundwater Seepage.slim)**. A seepage analysis is carried out, and results are compared with the Water Pressure Grid tutorial.

Model

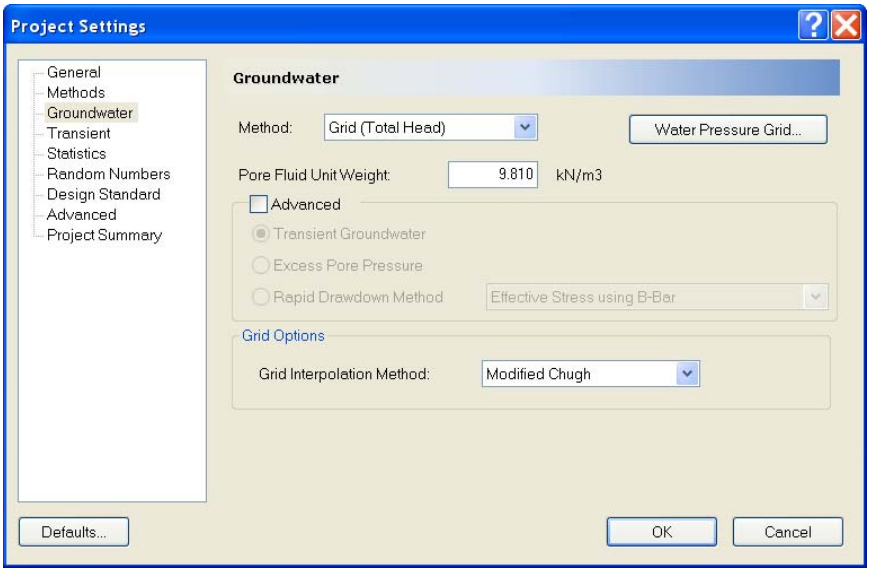
If you have not already done so, run the *Slide* Model program by double-clicking on the *Slide* icon in your installation folder. Or from the Start menu, select Programs → Rocscience → Slide 6.0 → Slide.

If the *Slide* application window is not already maximized, maximize it now, so that the full screen is available for viewing the model.

Project Settings

In order to use a water pressure grid for pore pressure calculations, we must first set the Groundwater Method to one of the three available pore pressure grid options (Total Head, Pressure Head or Pore Pressure) in the Project Settings dialog. In this case we will be using a grid of discrete Total Head values.

Select: Analysis → Project Settings



Select the **Groundwater** page, and set the Groundwater Method = Grid (Total Head).

Note that *Slide* can use one of several different methods for interpolating pressures at any point in the soil, from the grid values. We are using the default method (Modified Chugh). See the *Slide* Help system for a description of the interpolation methods available.

Select the **Project Summary** page and enter a project title – Water Pressure Grid Tutorial. Select OK.

Add External Boundary

The first boundary that must be defined for every *Slide* model is the External Boundary. To add the external boundary, select Add External Boundary from the toolbar or the Boundaries menu.

Select: Boundaries → Add External Boundary

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

```
Enter vertex [esc=cancel]: 15 20
Enter vertex [u=undo,esc=cancel]: 65 20
Enter vertex [u=undo,esc=cancel]: 65 35
Enter vertex [c=close,u=undo,esc=cancel]:50 35
Enter vertex [c=close,u=undo,esc=cancel]:30 25
Enter vertex [c=close,u=undo,esc=cancel]: 15 25
Enter vertex [c=close,u=undo,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 Add External Boundary option.

Adding a Water Pressure Grid

Now let's add the water pressure grid to the model. To add a water pressure grid, select the Water Pressure Grid option from the Boundaries menu.

Select: Boundaries → Water Pressure Grid

The points defining a water pressure grid can be entered in this dialog, by entering X and Y coordinates, and a value (in this case, Total Head), defining the pressure at each grid point.

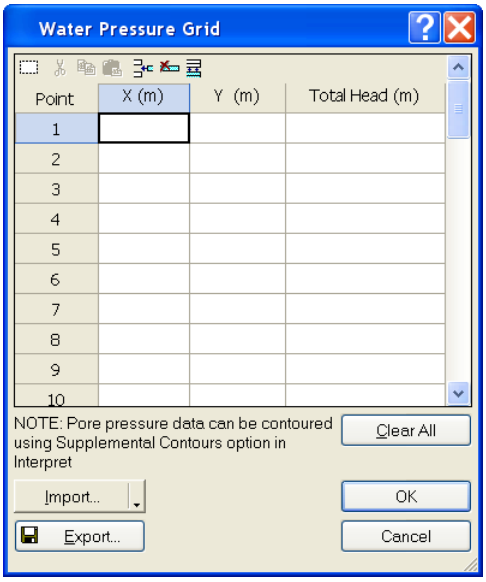
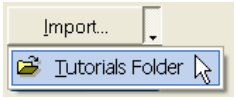


Figure 5-1: Water Pressure Grid dialog.

Rather than entering the data manually, a water pressure grid file has been provided, which you can simply read in using the **Import** option in the Water Pressure Grid dialog.

1. Select the drop-arrow at the right of the Import button. You will see a shortcut to the Tutorials folder appear.



2. Click on the Tutorials folder shortcut and you will see an Open file dialog. Water pressure grids can be imported from various file formats including .**pwp** files (these are simple text files where each line of the file contains X, Y and VALUE for one grid point) or .**dxf** format files (useful if a flownet has been digitized using AutoCAD, for example).

3. We will read in a .pwp file. Open the file called **Tutorial 05 Water Pressure Grid.pwp**. The grid data appears in the Water Pressure Grid dialog.

4. Now select OK in the Water Pressure Grid dialog and the grid will be added to the model. Each blue triangular symbol represents one grid point.

Your model should appear as follows:

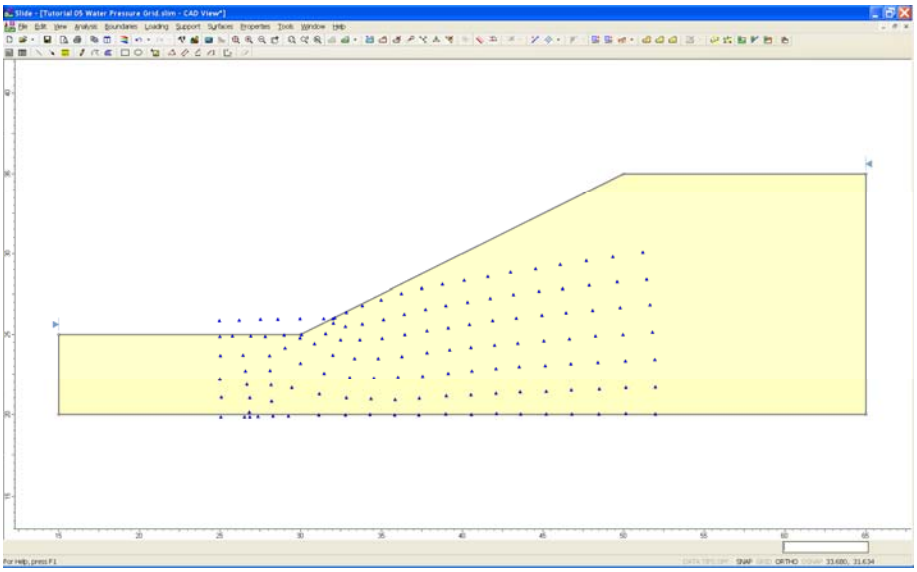


Figure 5-2: Water Pressure Grid added to model.

Water pressure grid data such as this could come from a flow net, field measurements, or a numerical analysis, such as the groundwater seepage analysis which is available within the *Slide* or *Phase2* programs.

In this case, the values at each grid point are Total Head values, which were originally obtained by digitizing the flownet in Figure 5-3, using a digitizing tablet and AutoCAD. (The grid was originally saved as a .DXF file, and then converted to a .PWP file).

Remember that *Slide* also has the capability of using pressure head or pore pressure grids, as selected in the Project Settings dialog.

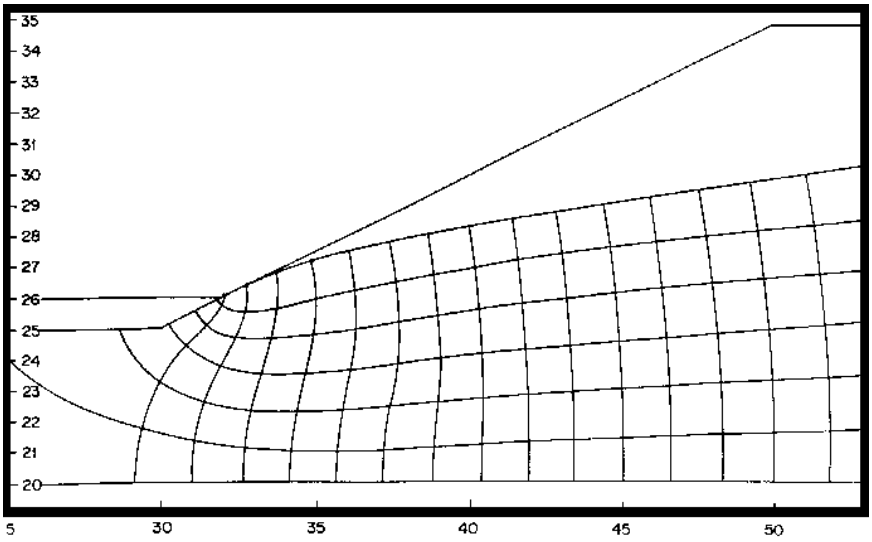


Figure 5-3: Flownet used to obtain total head measurements.

The actual values at each grid point can be displayed on the model with the Display Options dialog. Let's take a quick look. Right-click the mouse and select Display Options from the popup menu.

In the Display Options dialog, select the Water Pressure Grid Values checkbox, and select Done. The values will be displayed on the model.

If the values overlap, use one of the Zoom options (e.g. Zoom Window, Zoom Mouse or simply rotate the mouse wheel forward), to zoom in to approximately the center of the grid, so that the values are readable. The total head values were obtained at the intersections of each flowline and equipotential line of the flownet in Figure 5-3.

Now select Zoom All to bring the whole model back into view. Tip: you can use the F2 function key as a shortcut to Zoom All.

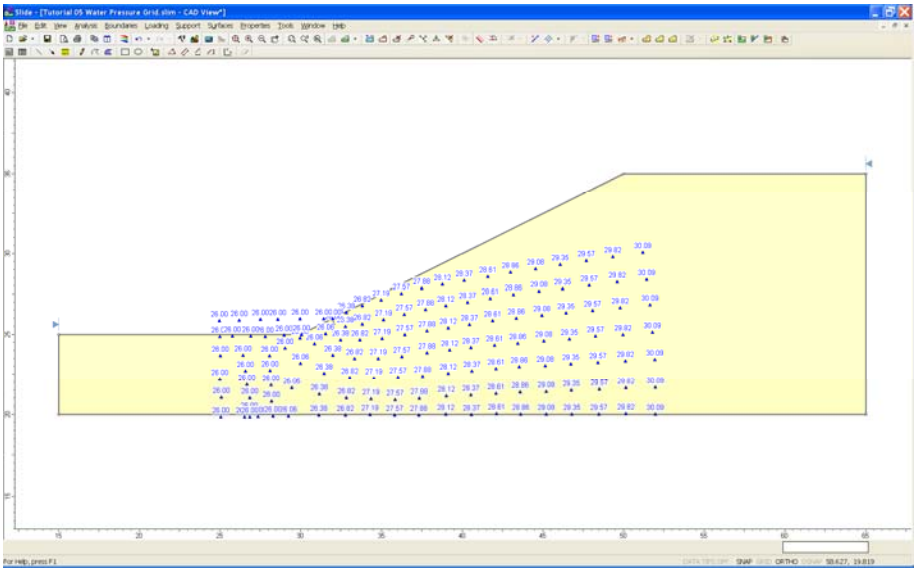


Figure 5-4: Water pressure grid values displayed on model.

Now hide the grid values again. Right-click the mouse and select Display Options. Clear the Water Pressure Grid Values checkbox, and select Done.

Defining Ponded Water

You will notice that some of the water pressure grid points, at the left of the model, are above the ground surface. That is because this model will include ponded water at the foot of the slope, which we have not yet defined.

Ponded water in *Slide* can be created as follows:

- If a Water Table is drawn above the External Boundary, *Slide* will automatically create a region of ponded water below the Water Table and above the External Boundary. This is the simplest method of defining ponded water.

NOTE: a Water Pressure Grid CANNOT define ponded water. A Water Pressure Grid is only used to obtain values of pore pressure within the soil. The Grid DOES NOT simulate the weight and hydrostatic forces which act on the slope due to the ponded water.

Add Water Table

As demonstrated in previous tutorials, a Water Table can be used in *Slide* to define pore pressure conditions for a slope model. ***In this tutorial, the Water Table will NOT be used for pore pressure calculations, since the water pressure grid will be used for this purpose***.

Regardless of the method of pore pressure definition (with the exception of the Finite Element Analysis method), a Water Table can always be used to define ponded water above a slope. Let's add a Water Table to see how this works.

Select: Boundaries → Add Water Table

Enter the following coordinates in the prompt line.

```
Enter vertex [esc=cancel]: 15 26
Enter vertex [u=undo,esc=cancel]: 32 26
Enter vertex [enter=done,esc=cancel]: 33.9 26.9
Enter vertex [enter=done,esc=cancel]:35.8 27.5
Enter vertex [enter=done,esc=cancel]:37.3 27.9
Enter vertex [enter=done,esc=cancel]: 39.8 28.3
Enter vertex [enter=done,esc=cancel]: 45 29.1
Enter vertex [enter=done,esc=cancel]: 52.3 30.2
Enter vertex [enter=done,esc=cancel]: 65.1 31.8
Enter vertex [enter=done,esc=cancel]: press Enter
```

Note that pressing Enter with nothing in the prompt line, after the last vertex has been entered, adds the Water Table to the model, and exits the Add Water Table option. Your model should now appear as follows:

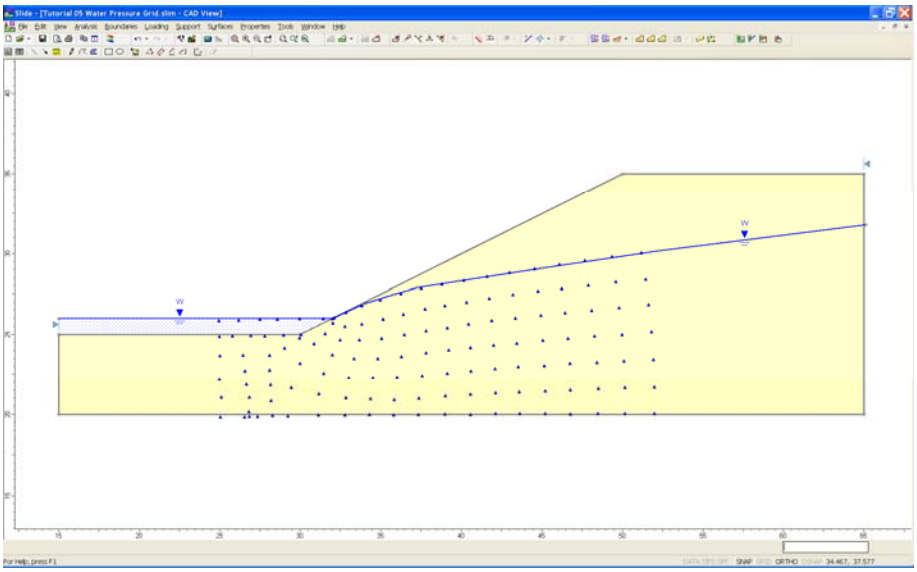


Figure 5-5: Water table added to define ponded water.

As you can see at the left of the model, the region above the ground surface and below the water table is filled with a blue hatched pattern. This region is automatically determined by *Slide* when the water table is drawn above the slope, and indicates the existence of ponded water.

As we have already emphasized, pore pressures for this model will be calculated using the water pressure grid and NOT the Water Table, since we have configured the method of pore pressure calculation in the Project Settings dialog. However, we will point out one extra feature of using a Water Table in conjunction with a pore pressure grid:

- All points ABOVE the Water Table will automatically be assigned a ZERO pore pressure, even if the water pressure grid interpolation procedure, determines a non-zero pore pressure for a point above the water table. This may be useful in some situations, for example, if a water pressure grid is defined by an insufficient number of points.

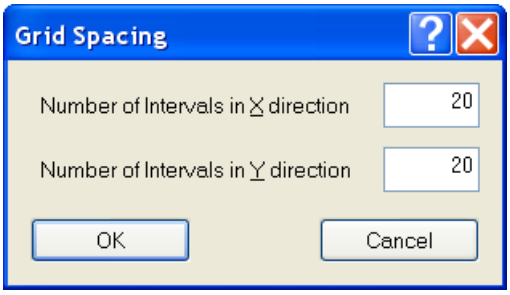
Slip Surfaces

For this tutorial, we will be performing a Grid Search, to attempt to locate the critical circular slip surface (i.e. the slip surface with the lowest safety factor).

A Grid Search requires a grid of slip centers to be defined. We will use the Auto Grid option, which automatically locates a grid for the user.

Select: Surfaces → Auto Grid

You will see the Grid Spacing dialog.



Enter a 20 x 20 spacing. Select OK. The Grid will be added to the model, and your screen should appear as follows:

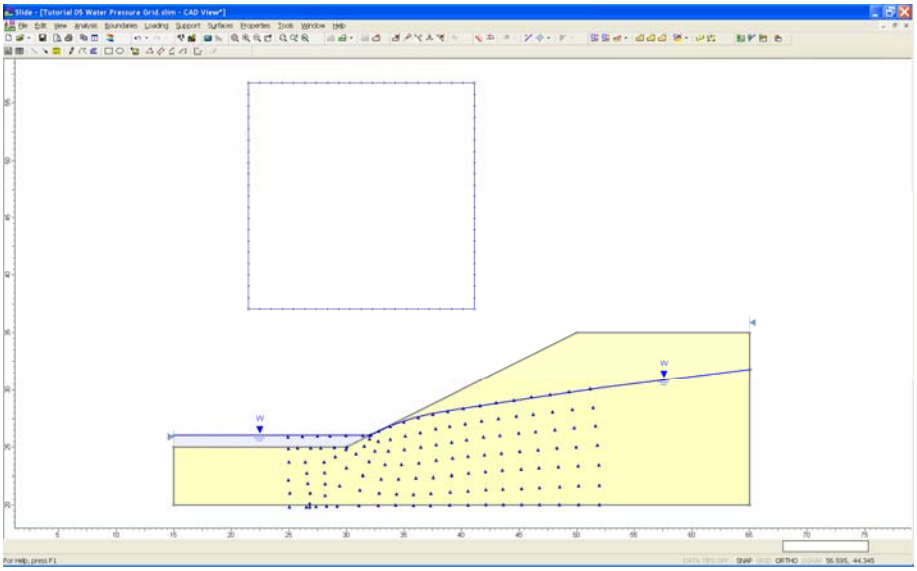


Figure 5-6: Slip center grid added to model.

NOTE: the Grid Search is discussed in detail in the Quick Start Tutorial. Please refer to that tutorial, or the *Slide* Help system, for more information.

Properties

To complete our modeling, we still have to define our material properties, and we will then be ready to run the analysis.

Select: Properties → Define Materials

In the Define Material Properties dialog, enter the following parameters, with the first (default) material selected.

✓ *Enter:*
✓ Name = soil 1
Unit Weight = 20
Strength Type = Mohr-Coul
✓ Cohesion =11
✓ Phi = 28
Grid (Total Head) = On

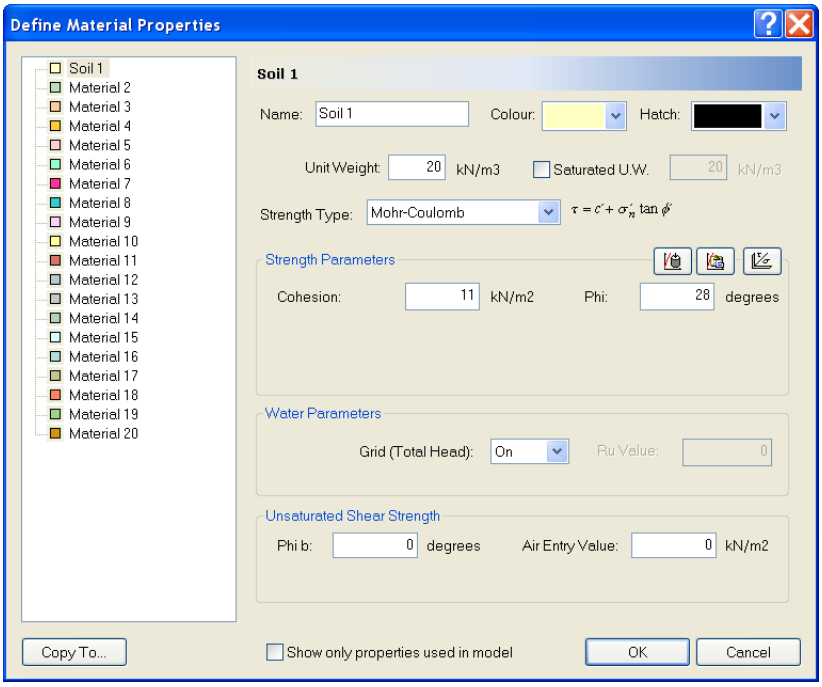


Figure 5-7: Define Material Properties dialog.

Enter Name = Soil 1, Cohesion = 11 and Phi = 28. Select OK.

In the Define Material Properties dialog, you will notice the Grid (Total Head) On / Off toggle, under Water Parameters. This allows you to toggle the effect of a Water Pressure Grid ON or OFF for any given soil. If the water pressure grid is turned OFF, then pore pressure will be ZERO for that soil. In this example, we are of course leaving the grid ON, since we want to see the results of using the water pressure grid.

ALSO NOTE: Since we are dealing with a single material model, and since you entered properties with the first (default) material selected, you do not have to Assign these properties to the model. *Slide* automatically assigns the default properties (i.e. the properties of the first material in the Define Material Properties dialog) for you. (For multiple material models, it is necessary for the user to assign properties with the Assign Properties option. This is discussed in Tutorial 2.)

We are now finished with the modeling, and can proceed to run the analysis and interpret the results.

Compute

Before you analyze your model, save it as a file called **WPG.slim**. (*Slide* model files have a .**slim** filename extension).

Select: File → Save

Use the Save As dialog to save the file. You are now ready to run the analysis.

Select: Analysis → Compute

The *Slide* Compute engine will proceed in running the analysis. This should only take a few seconds. When completed, you are ready to view the results in Interpret.

Interpret

To view the results of the analysis:

Select: Analysis → Interpret

This will start the *Slide* Interpret program. You should see the following figure:

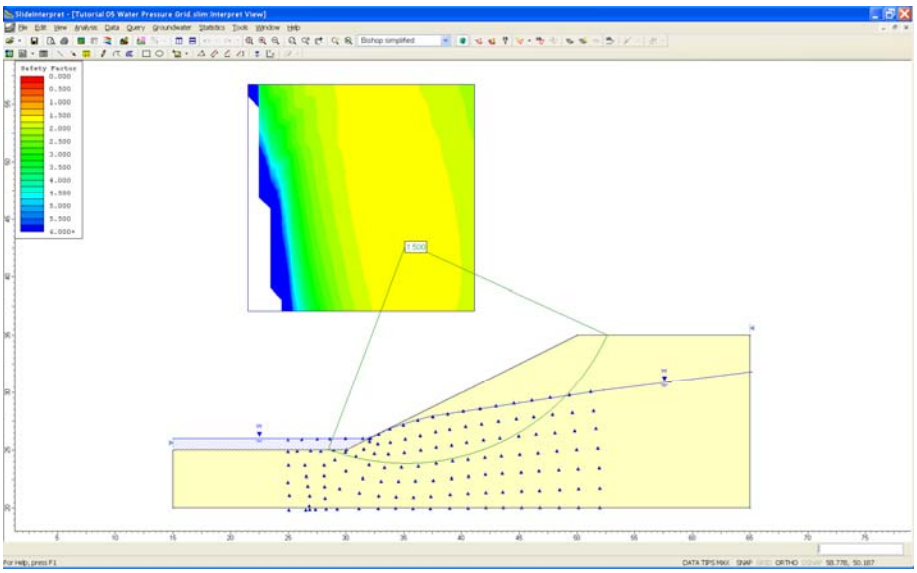


Figure 5-8: Results of circular surface Grid Search.

By default, the Global Minimum surface for a Bishop analysis, is initially displayed. The minimum safety factor = 1.500.

Notice the slip center grid, in this case, has a blank (white) area which is not contoured, at the left of the grid. This occurs when no valid slip circles are generated at one or more slip center grid points. For this grid, most circles generated at these points have intersected the horizontal segment of the external boundary at the left of the model. This generally leads to zero driving force and an invalid slip surface (safety factor cannot be calculated).

TIP: when a slip center grid displays blank areas such as this, you may want to go back to the modeler, and modify the grid size or location. This is left as an optional exercise after completing this tutorial. You can edit grids using right-click shortcuts, or use the Edit sub-menu in the Surfaces menu.

The appearance of the contours in the slip center grid can be customized by the user with the Contour Options dialog. Let's try this now. Contour Options is available in the View menu, however, a convenient shortcut is to right-click the mouse and select Contour Options from the popup menu.

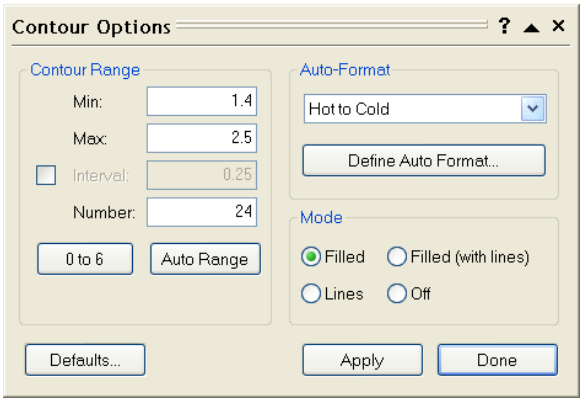


Figure 5-9: Contour Options dialog.

In the Contour Options dialog, enter a Contour Range Min = 1.4 and Max = 2.5. Select Apply. Now use the "roll-up" arrow ▲ to minimize the dialog without closing it. You can also minimize / maximize the dialog, by double-clicking on the title bar of the dialog.

The new contour range makes the low safety factor area of the slip center grid more apparent, as shown in Figure 5-10. Many different contour options are available to the user, and customized contour formats can be saved for future use with the Define Auto-Format option. The user is encouraged to experiment with these options after completing this tutorial.

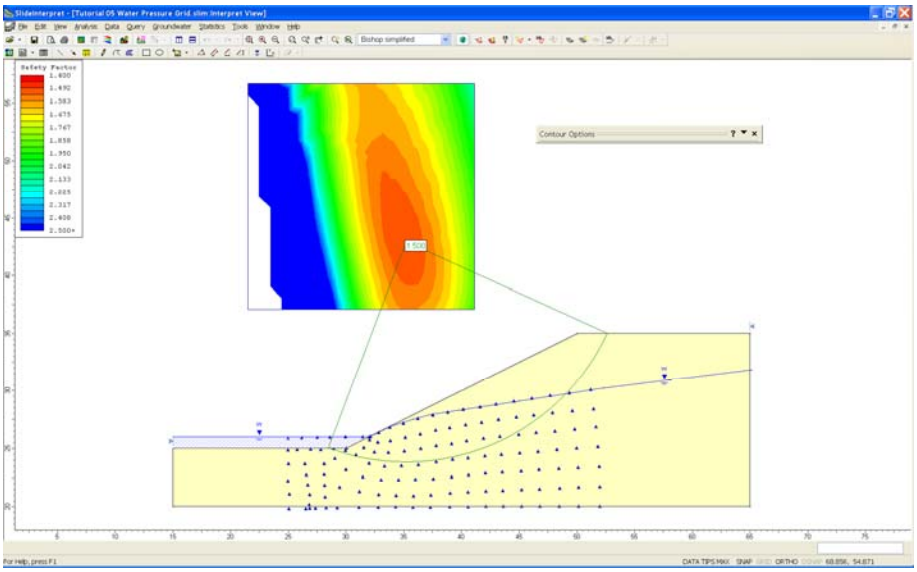


Figure 5-10: Custom contour range in slip center grid.

Select Done or X, to close the Contour Options dialog.

Now display the minimum circles at the slip center grid points.

Select: Data → Minimum Surfaces

The minimum surfaces are displayed. Note that the circle colours correspond to the safety factor contour colours in the slip center grid, and the legend at the upper left of the view.

Let's view the results for another analysis method. Select Janbu Simplified from the drop-list in the toolbar.

For this model, the Janbu Global Minimum slip surface is different from the Bishop surface. Global Minimum surface information, for each analysis method, is always available in the Info Viewer.

Select: Analysis → Info Viewer

Scroll down the Info Viewer, to view the Global Minimum surface information. Note that each surface has different center coordinates, and radius. Close the Info Viewer view, by selecting the X in the upper right corner of the view (make sure you select the view X and not the application X, so you don't close the INTERPRET program!)

NOTE: for the Janbu Simplified analysis method, you will have noticed a blank (white) area in the low safety factor area of the slip center grid contours. This is because we customized the contour range for the Bishop results, but results for the Janbu Simplified method were outside of our custom range. Let's restore the default contour range.

Right-click the mouse and select Contour Options. Select the “0 to 6” button in the Contour Options dialog, to restore the default 0 to 6 safety factor contour range. Select Done.

Now select different analysis methods again from the toolbar, and observe the default contours for each method.

Add Query

Let’s now add a query on the Global Minimum for the Bishop analysis, and plot pore pressure along the slip surface.

First, select the Bishop analysis method from the toolbar, if it is not already selected.

Queries can be added with the Add Query option in the Query menu. However, a shortcut for adding a query corresponding to a Global Minimum slip surface, is to right-click anywhere on the slip surface, or on the radial lines joining the slip center to the slip surface endpoints, and select Add Query from the popup menu.

Do this now, for the Bishop analysis Global Minimum.

Note that the colour of the Global Minimum surface has changed to black, indicating that a query has been added. (Queries are displayed using black. The Global Minimum, before the query was added, was displayed in green).

Graph Pore Pressure

After a query has been added, data can be graphed using the Graph Query option.

Select: Query → Graph Query

Since only one Query exists (on the Global Minimum), it is automatically selected, and the Graph Slice Data dialog will appear.

TIP: if you select Graph Query BEFORE you have added any queries, *Slide* will automatically create a Query for the Global Minimum, and display the Graph Slice Data dialog. This saves the user the step of using the Add Query option.

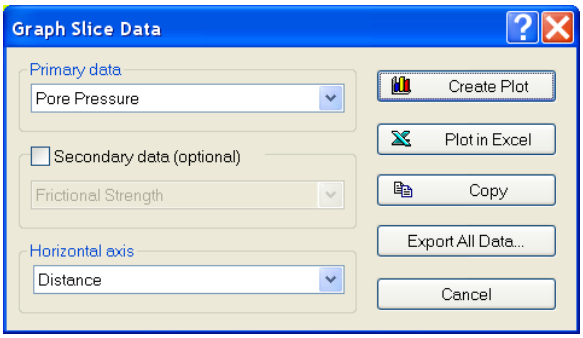


Figure 5-11: Graph Slice Data dialog.

Let's graph pore pressure along the slip surface.

Select Pore Pressure from the Primary Data drop-list. Select Create Plot.

You should see the plot shown below.

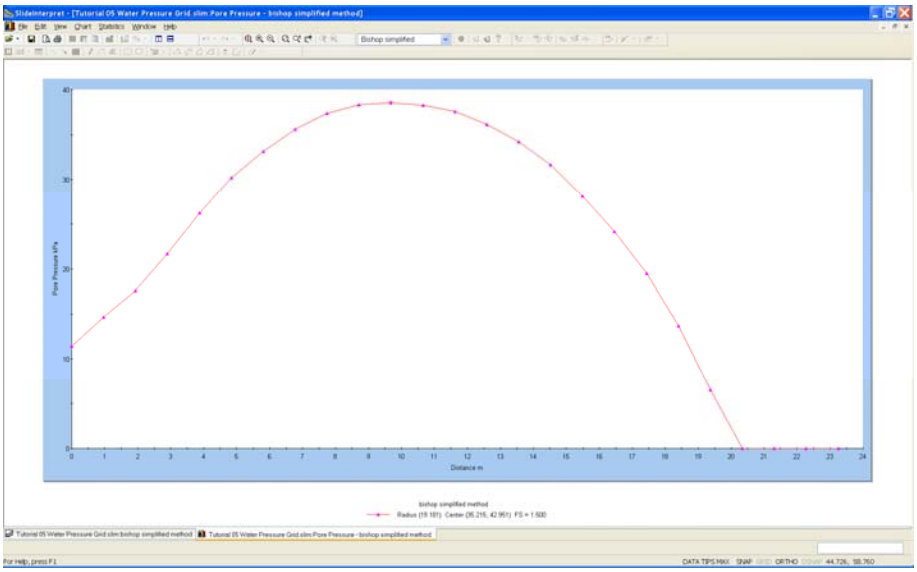


Figure 5-12: Pore pressure graphed along slip surface.

This graph shows the pore pressure calculated at the midpoint of the base of each slice, by interpolation from the water pressure grid values.

NOTE: you can customize the graph appearance, by right-clicking on the graph and selecting Chart Properties, or you can view different data for the same slip surface, by right-clicking and selecting Change Plot Data. This is left as an optional exercise.

That concludes this tutorial.