Visual Slope User’s Guide
Version 6

2016
## Table of Contents

<table>
<thead>
<tr>
<th>Section</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Quick Overview</td>
<td>1</td>
</tr>
<tr>
<td>Starting Project</td>
<td>1</td>
</tr>
<tr>
<td>Establishing Profile</td>
<td>4</td>
</tr>
<tr>
<td>Start Line Drawing</td>
<td>4</td>
</tr>
<tr>
<td>Stop Line Drawing</td>
<td>4</td>
</tr>
<tr>
<td>Data Entering Method</td>
<td>5</td>
</tr>
<tr>
<td>Rules of Line Drawing</td>
<td>6</td>
</tr>
<tr>
<td>Snap On Feature</td>
<td>7</td>
</tr>
<tr>
<td>BRAKE Feature</td>
<td>7</td>
</tr>
<tr>
<td>Edit or Delete Line</td>
<td>7</td>
</tr>
<tr>
<td>Surcharge Load</td>
<td>7</td>
</tr>
<tr>
<td>Edit or Delete POINT</td>
<td>9</td>
</tr>
<tr>
<td>generators</td>
<td>9</td>
</tr>
<tr>
<td>Other Features</td>
<td>9</td>
</tr>
<tr>
<td>Zoom In and Zoom Out</td>
<td>9</td>
</tr>
<tr>
<td>Undo</td>
<td>10</td>
</tr>
<tr>
<td>Flip</td>
<td>10</td>
</tr>
<tr>
<td>Pan</td>
<td>10</td>
</tr>
<tr>
<td>Material Properties</td>
<td>10</td>
</tr>
<tr>
<td>Set up Material Properties</td>
<td>11</td>
</tr>
<tr>
<td>Save to Material Banks</td>
<td>15</td>
</tr>
<tr>
<td>Use Saved Materials</td>
<td>15</td>
</tr>
<tr>
<td>Set up Connection Data</td>
<td>15</td>
</tr>
<tr>
<td>Material Property Assignment</td>
<td>17</td>
</tr>
</tbody>
</table>
Soil Property Assignment

Material Properties Assignment to Other Objects

Second Way to Assign Material Properties

Analyses

Slope Stability Analyses

Soil/Rock Properties

Circular Failure Analysis

Irregular Failure Analysis

User Specified Single Failure Surface

Translational Failure Surface Analysis

Other Features

Report

Soil/Rock Nailing Design

Nail/Anchor Properties

Adding Nails to Slope

Analysis

Reinforcement

Reinforced Slope Design

Geosynthetic Properties

Adding Geogrid to Slope

Reinforcement

Resisting Pile Design

MSE Wall Design

Wall Block and Connection Properties

Wall Geometry

Perform Analysis

Shoring Design
QUICK OVERVIEW

The Visual Slope Series is a multi-function engineering computer program developed for:

1. Slope Stability Analysis
2. Soil Nailing Design
3. Resisting Pile Design
4. Reinforced Slope Design
5. MSE Wall Design
6. Shoring Design
7. Seepage Analysis
8. Tunnel Lining Design
9. Ground Freezing Construction Design

All functions in the Visual Slope Series have been designed to share a common analogy. Therefore, they are very easy to learn and use. Visual Slope uses convenient drawing procedures similar to AutoCAD to help users establish input files, which allows a detailed and accurate modeling for complicated projects and greatly reduces chances of input errors. Several quick generators can also be used to speed up the modeling process.

To perform the above analyses with the Visual Slope Series, the following five simple steps are required:

1. Starting Project
2. Setting Up and Assigning Material Properties
3. Establishing Profile
4. Performing Analysis
5. Generating Report

This User’s Guide will provide our users with tutoring on how to use Visual Slope. The following sections will describe each of the above five steps.

STARTING PROJECT

After Visual Slope starts, Visual Slope will prompt the following dialogue box (Figure 1) to let the user choose either an existing project (file) or a new project (file).
If the *Existing Project* option is chosen, the user can select the file from the project list or browse the file by selecting `<<Other Files>>`.

If the *New Project* option is chosen, Visual Slope will prompt the user to follow the *General Setting* page (Figure 2) to start a new project.

![Create/Open Project](image)

*Figure 1: Create/Open Project*
On that page, the user can input the project information, select the unit, and define the dimensions that should cover the range of the profile. The general settings can always be modified later from the File menu, as shown in Figure 3.
ESTABLISHING PROFILE

A Visual Slope profile (cross section) consists of zones and lines – soil lines, geogrid lines, soil nail/tieback lines, water table lines, etc., similar to those shown in Figure 4. A zone consists of closed soil lines or boundary lines. A profile can be drawn or generated in some cases. This section will focus only on the drawing method. The generating methods will be addressed in relevant sections. To draw different lines, the user must first click the corresponding line buttons on the Toolbar. Figure 5 shows the line buttons on the Toolbar.

After choosing the correct line type, the user can begin to draw a profile. There are two ways to draw a line: the direct drawing method or the coordinate input method. The following sections describe how to draw lines, edit lines, and delete lines.

START LINE DRAWING

After a line button is clicked, the program is in the drawing mode and the cursor becomes a cross hair. The user can move the cursor to the position at which the line will begin by referencing either the coordinates shown on the horizontal scale located immediately below the Toolbar and the vertical scale located on the left side of the screen, or the coordinates displayed at the lower left corner of the screen, as shown in Figure 6. Left click the mouse button to start a line. Move the cursor to the end point of the line, following the same process mentioned above. Once the start point of a line is selected, the user can also draw the line according to the angle (to the horizontal) and length of the line shown right to the coordinates display.

Repeat the above procedure for the following lines. Visual Slope will automatically start the next line at the end point of the previous line.

STOP LINE DRAWING

To stop drawing or to start a line from a different position, right click the mouse button or press the Esc key on the keyboard.
DATA ENTERING METHOD

Besides referencing the coordinates from the scales or from the coordinates display, the user can also type the horizontal and vertical coordinates into the Coordinate Entry box at the bottom.
of the screen. Both the horizontal and vertical coordinates should be separated by a space. For example, if the horizontal coordinate is 50.1 and the vertical coordinate is 635.4, the user can type 50.1 635.4 into the Coordinate box (Figure 7) and press the Enter key on the keyboard. The line will start at that point. The same method can be used for the end point.

Besides using vertical and horizontal coordinates, the user can also use length and angle for the end point input. For example, if the line length is 10.3 and the slope is 20 degrees above the horizontal line, the user can type 10.3 <20 into the coordinate box (Figure 8) and press the Enter key on the keyboard. The length and the angle must be separated by a space.

![Figure 6: Coordinate Indicators](image)

**RULES OF LINE DRAWING**

1. A line must be drawn from left to right. Otherwise Visual Slope will ask the user to re-draw.

2. For soil lines, if they are connected, they must connect at their end points. If a new line starts or ends at the middle of an existing line, the later will be automatically broken into two lines, so that lines can connected at their ends.

3. Visual Slope does not require the lines to be drawn in a specific sequence. Lines can be drawn in any order. Lines can also be added or deleted anywhere as desired.
SNAP ON FEATURE
If a new line point is very close to an existing line point, the new line point will automatically snap on to the existing line point.

BRAKE FEATURE
If a new line starts or ends at the middle of an existing line, the later will be automatically broken into two lines, so that lines can connected at their ends.

EDIT OR DELETE LINE
To edit or delete a line, click the Select button on the Toolbar, and then move the cursor to the line to be edited or deleted and click the line. The Line Edit Dialog Box will appear (Figure 9). Select the line in the dialog box and click the Delete button. The selected line will be deleted. The coordinates of the line can also be edited in the Line Edit Dialog Box. However, it should be aware that once the coordinates of a line are altered, that line may no longer connect to the line(s) it previously connected to.

To delete multiple lines, click the Group Delete button on Toolbar, and then select the lines to be deleted by the dragging method. After being confirmed, the selected lines will be deleted.

SURCHARGE LOAD
Surcharge loads can be added by using a drawing method similar to drawing other lines. Click the Load button on the Toolbar. Then, draw the surcharge load from left to right. The initial
load intensity is 1. To specify the load intensity, the user can use the Line Edit method, referring to the last section. The user should also specify the load as "live" or "dead." This process is shown in Figure 10.

Figure 9: Edit or Delete a Line

Figure 10: Specify Load Intensity
EDIT OR DELETE POINT
To edit or delete a point, click the Select button on the Toolbar, and then move the cursor to the point to be edited or deleted and click the line. The Point Edit Dialog Box will appear (Figure 11). The coordinates of the line can also be edited in the Point Edit Dialog Box. If the Delete button is clicked, the selected point and all lines associated with that point will be deleted.

![Point Edit Dialog Box](image)

**Figure 11: Edit or Delete a Point**

GENERATORS
To speed up profile establishment, generators can be used. Generators shown in Figure 12 include generators for geogrid, nail/anchor, MSE wall, shoring system, freezing system, and tunnel lining.

![Generators](image)

**Figure 12: Generators**

OTHER FEATURES
To help users establish profiles, Visual Slope provides many other features, such as Zoom In, Zoom Out, Undo, Flip and Pan. as shown in Figure 13.

![Other Features](image)

**Figure 13: Other Features**

**Zoom In and Zoom Out**
The Zoom In and Zoom Out features help users:
1. Draw details in a small area

2. Perform an analysis focused on a specific area

To zoom in an area, click the Zoom In button, hold the left mouse button down, drag diagonally across the area to be zoomed in, and then release the button.

To zoom out, click the Zoom Out button.

Undo
To undo, click the Undo Button. The user can undo up to 5 steps back.

Flip
A slope, MSE wall, or shoring system to be analyzed must face left. However, in some cases the original slope may face right or have slopes on both sides that need to be analyzed, such as a dam. The Flip feature can be used to flip the slope from right to left for the analysis.

Pan
To move a profile towards a certain direction, click the Pan button, and then move the mouse in that direction with holding the left button of the mouse down.

MATERIAL PROPERTIES

Material properties in Visual Slope include:

1. Soil Properties
2. Geogrid Properties
3. Soil Nail/Tieback/Metal Strip Properties
4. MSE Wall Unit Properties
5. Wall Unit and Geogrid Connection Properties
6. Shoring Wall/Tunnel Lining Properties

The user must set up the material properties and assign them to the profile before running an analysis. The type of materials needed depends on the type of analysis to be performed. In an MSE wall analysis, for example, soil, geogrid, and wall unit, as well as connection properties,
are required. Visual Slope provides the most commonly used material properties in its material banks for users to employ. Users can also save their own material properties into the bank for future use.

The following sections describe how to:

1. Set up material properties
2. Use the material properties saved in the material banks
3. Save material properties into the material banks for future use
4. Assign the material properties to the profile

SET UP MATERIAL PROPERTIES

To set up material properties, the user must first click the Material button (Figure 14: Material Button) on the Toolbar. The Material page (Error! Reference source not found.) will appear.

![Figure 14: Material Button](image-url)
On the Material Property page, the user must first select the material type, such as Soil Material, Geogrid and so on, from the pull-down list, to input the corresponding material properties. Once a specific material type is chosen, if there are existing materials, they will appear in the list box (Figure 16: Select Material Type).
To edit the properties of an existing material, click the material name listed in the list box, and then click the *Edit Button*. The *Material Edit* page will appear (Figure 17). On that page, the user can revise or review the material parameters.

To add a new material, click *New* on the *Material* page under the selected material type. The *Material Input* page will appear (Figure 18: *Material Input Page*).

The user can input material parameters on that page. After finishing, click the *Close* button. The material name will show in the list box. Similar way can be used to establish other material properties.
Figure 17: Material Edit Page

Figure 18: Material Input Page
SAVE TO MATERIAL BANKS
To save a material to the corresponding bank, the user must first click the Open Material Bank button to open the material bank half. Then the user can select the row to be saved and click the > button. This material is saved to the database and can be used in the future for different projects.

USE SAVED MATERIALS
Opposite to saving a material to the material bank, to use the materials saved in the material bank, choose the material in the Material Bank first, and then click the < button. The material will be imported to the current project.

The procedure of importing and banking material properties is shown graphically in Figure 19.

SET UP CONNECTION DATA
The connection data are only used for MSE wall design. Once the data is set up, it can be used by all projects. Visual Slope will search the database for the connection data during an MSE wall analysis. To set up connection data, choose Connection Date in the Material Type list first, and then click the Set Connection button. In the Block Name list of the Connection page, select the wall unit that has been saved in the Wall Bank; and select the geogrid that has been saved in the Geogrid Bank from the Grid Name list. After selection of the wall unit and geogrid, type, input the remaining connection data. The connection data should be from the wall unit manufacturer. The procedure of setting up connection data is shown in Figure 20.
Figure 19: Material Bank

Figure 20: Set up Connection Data
MATERIAL PROPERTY ASSIGNMENT

There are two ways to assign material properties. Once the Material page is closed, material buttons will appear on the Material Bar as shown in Figure 21. The user can assign the materials to the profile.

![Material Bar](image1.png)

Figure 21: Material Bar

Soil Property Assignment

Soil properties can only be assigned to zones that are formed with closed soil or boundary lines. To assign soil properties to soil zones, click the Soil button first and then click the zone to which the soil properties should be assigned. The color of the zone will change to the same color as that of the button. If the color of the zone does not change, the zone is not closed. This process is shown in Figure 22.

![Assign Soil Properties](image2.png)

Figure 22: Assign Soil Properties
Material Properties Assignment to Other Objects

To assign material properties to other objects, such as soil nail, geogrid, or MSE wall, click the *Material* button first and then click the corresponding object, as shown in Figure 23. The color of the object line becomes the same color as that of the button.

Second Way to Assign Material Properties

If the material bar is not large enough to hold all the materials used in a project, there is another way to assign material properties, which is very similar to the method discussed in the previous section. The difference is that, instead of using the buttons in the material bar, use the button on the Material page as shown in Figure 24. On the material page, choose the material to be assigned first; click the *Assign Material* button; then move the cursor to where the material is to be assigned and click again.
ANALYSES

SLOPE STABILITY ANALYSES

Visual Slope can perform three types of slope analyses:

1. Circular Failure Surface with Modified Bishop Method
2. Irregular Failure Surface with Janbu Method
3. Translational Failure Surface with Janbu Method or Transfer Coefficient Method
4. User Specified Failure Surface with either Modified Bishop Method or Janbu Method depending on the shape of the specified failure surface

Except for irregular failure surface, all other three types of failure surfaces can also be analyzed with the Spencer Method or Morgensten Method, if those are chosen. The following sections describe how to set up soil/rock properties for slope stability analysis and how to perform these four types of analyses with Visual Slope.
Soil/Rock Properties
The first step for slope stability analysis is to set up the soil/rock properties required for the analysis. The soil/rock properties include the moist unit weight, the saturated unit weight, the cohesion, the friction angle, a porepressure parameter, and a porepressure constant.

If there is a water table, moist unit weight will be used for the soils above the water table and buoyancy unit weight will be used for the soils below the water table.

Either an effective stress analysis ($\phi'$, $c'$) or total stress analysis ($c$, $\phi$=0) may be performed by using the appropriate values for the Mohr-Coulomb strength parameters.

Excess porepressure due to shear can be assumed to be related to the overburden by the single parameter. The porepressure constant of a soil type defines a constant porepressure for any point within the soil described. Either or both of these two options for specifying porepressure may be used, in combination with porepressure related to a specified piezometric surface, to describe the porepressure regime.

For how to set material properties, please see the section of SET UP MATERIAL PROPERTIES.

Circular Failure Analysis
A circular failure analysis (Figure 25) is specified by five numbers: 1) X1-coordinate of leftmost initiation point; 2) X2-coordinate of left termination point; 3) X3-coordinate of rightmost initiation point; 4) X4-coordinate of right termination point; and 5) Number of failure surfaces.

To perform circular failure analysis, click the Circular Failure button from the Analysis panel. The Slope Stability page will appear (Figure 26). Choose the Circular tab.

The user can type X-coordinates into the corresponding data boxes. More conveniently, the user can click the button next to the data box and then move the cursor to the position where the user wants the X-coordinate to be and click. The X-coordinate of this position will be input into the data box and a dot will appear on the top boundary with that X-coordinate. Those steps are shown in Figure 26.
The number of failure surfaces should be an integer. It is recommended that this number be at least 100.
If the user wants to use the Spencer Method, LRFD Method, or consider seismic effect for the analysis, the relevant check boxes should be checked and horizontal and vertical accelerations should be filled if seismic effect is included.

For the analysis, click the Analysis button. After the analysis is completed, click the Curve buttons to see the failure surfaces (Figure 27).

**Irregular Failure Analysis**

Performing an irregular failure analysis is exactly the same as performing a circular failure analysis, except choosing the *irregular* option. Figure 27 presents the result of an irregular failure analysis.

---

**Figure 27: Result of Analysis**
User Specified Single Failure Surface

Visual Slope allows the user to specify a potential failure surface for an analysis. The method for the analysis will depend on the shape of the failure surface. The failure surface is defined by a series of X and Y coordinates. The user can type the coordinates into the data cells manually, or use the drawing method. To use the drawing method, the user can single click the X-coordinate cell. A button will appear, as shown in Figure 29. The user can then click the button and move the cursor to the point on the failure surface and click at that point. The X and Y coordinates of that point will be input into the data cells. A failure surface line will appear on the screen after the second point is defined (Figure 30).

Repeating the above process, the user can draw a complete failure surface (Figure 30). The failure surface should be specified consecutively from left to right. The start point and end point of the failure surface should be slightly beyond the ground surface of the slope. The total points of a failure surface should not exceed 30.
Figure 29: Draw Failure Surface

1. Click Here
2. Click Here

Figure 30: Continue to Draw

1. Click here
2. Click here
Translational Failure Surface Analysis

Translational Failure Surfaces are defined by 1) an original potential failure surface; 2) the number of failure surface to be searched; and 3) the search width. The original potential failure surface is defined the same way that a user specified failure surface is defined (see the section above). Visual Slope will search the number of failure surfaces around the original potential failure within the range of the search width. Figure 31 shows the result of a translational failure surface analysis.
Other Features

Seismic Analysis – To perform a seismic analysis, seismic option should be chosen and horizontal and vertical accelerations should be provided. The unit of the acceleration is g. If 0.1 is input, an acceleration of 0.1g will be used in the calculation.

Strength Reduction – Along a failure surface, the soil/rock is usually weaker than the original soil/rock. To simulate this condition, reduction factors of can be specified for cohesion and friction.

LRFD – Load Resistance Factored Design can be performed if desired.

Report

After an analysis, both text and graphical reports can be generated. To produce a text report, click the Report button on the Slope Stability menu. A text report will include all the input information and the detailed analysis results. To generate a graphical report, click the Chat pull-down menu of the Slope Stability page and choose the chat type to be generated. On the report preview page, the report can be printed or saved to a PDF, Word or Excel file.

SOIL/ROCK NAILING DESIGN

If a soil or rock slope is not stable, nails can be used to stabilize the slope. Visual Slope is capable of soil or rock nailing design. A nail-reinforced excavation must meet the minimum
factors of safety for different failure conditions. The following sections describe how to set up nail properties and use Visual Slope for soil/rock nailing design. A soil/rock nailing design is similar to a slope stability analysis. Please read the section of SLOPE STABILITY ANALYSES first.

Nail/Anchor Properties

Nail/Anchor properties include the nail name, bore hole diameter (d), bond strength (s), horizontal spacing, tendon capacity, head capacity, force type, and function.

The tension reinforcement increment:

\[ dT = \pi d^2 s dL \] (dL is the increment of bonded nail length)

The tendon capacity is the allowable load the tendon can sustain. The head capacity is the allowable load the nail/anchor head connection can sustain.

The force type can be set to tension or shear according to the function of the nail/anchor. If the force type is set to shear, the shear capacity equals the tendon capacity.

The nail/anchor function can be set to Nail+Tieback, Nail Only, or Tieback Only. If a nail/anchor is not used to support a retaining system should be set to nail only. Figure 32 shows the nail/anchor property input box.
Adding Nails to Slope

Drawing Method
To add nails to a profile, the user can simply draw nails one by one, similar to drawing soil lines (Figure 33). After drawing, the user can assign the material to the nails. Using length and angle input or display, such as 10 < -20, will be more convenient for nail drawing.

Nail Array
The nail array method is easier than the drawing method to generate nails with the same type and same length at one time. To use the nail array method, the user must set up the nail properties first using the Material Setup dialog box. After material set up, the user can click the Nail Array button on the Toolbar. The Grid Array dialog box will appear (Figure 34). The user must select the nail type from the pull down list and then provide the start elevation, end elevation, vertical spacing, angle of inclination, unbonded length, and bonded length for the corresponding data boxes. Once the user clicks the Close button, nails will appear on the profile (Figure 35).
Figure 33: Adding Nails with Drawing Method

Figure 34: Nail Array
Analysis

The nailing design must satisfy the factor of safety for all potential failure conditions. The circular failure model is always a good one with which to begin.

Reinforcement

Like geogrid reinforced slope analysis, after a soil nail analysis is completed, color spectrums will display along soil nails (Figure 36). The legend of the color spectrum on the left of the screen gives the magnitude of reinforcement. However, please note that the magnitudes of reinforcement along a layer of soil nail do not represent the actual reinforcement that soil nail is providing, but the potential magnitudes of reinforcement that soil nail can provide, if a failure surface passes that specific point on the soil nail.
If a soil fill slope is too steep, it may not meet the slope stability requirement. To increase slope stability, geogrid or geotextile can be used as reinforcement for the slope. A reinforced slope must meet the minimum factors of safety for circular failure and direct sliding over geosynthetics. The following sections describe how to set up geosynthetic properties and use Visual Slope to design a reinforced slope.

**Geosynthetic Properties**

Geosynthetic properties include the name, long-term allowable strength, friction reduction, cohesion reduction, front reduction, creep factor of safety, and material type.

The long-term allowable strength of the geosynthetic should be obtained from the manufacturer. Friction and cohesion between geosynthetic layers and soil layers may be reduced from the original soil friction and cohesion. In the front of a slope, geosynthetic may not be well wrapped, which will cause reinforcement reduction. The front reduction factor is used to account that fact. The creep factor of safety is only for a seismic analysis, in which the design strength of the geosynthetic will equal to multiplication of creep factor of safety and the original long term
design strength. If the reinforcement is from a metal material, it should be specified. Figure 37 shows the geosynthetic property input box.

![Geosynthetic Property Input Box]

**Figure 37: Geosynthetic Property Input Box**

**Adding Geogrid to Slope**

**Drawing Method**
To add geogrid layers to a slope, the user can draw in the geogrid layer by layer, similar to drawing soil lines. After drawing, the user can assign the material to the geogrid (Figure 38).

**Grid Array**
The grid array method is easier than the drawing method to generate geogrid layers with the same type and same length at one time. To use the grid array method, the user must set up the geogrid properties first, using the *Material Set Up* dialog box. After materials have been set up, the user can click the *Grid Array* button on the *Toolbar*. The *Grid Array* dialog box will appear (Figure 39). The user must select the geogrid type from the pull down list, and then provide the start elevation, end elevation, vertical spacing, and geogrid length for the corresponding data cells. Once the user clicks the *Close* button, geogrid layers will appear on the profile.
Figure 38: Draw Geogrid

Figure 39: Geogrid Array

The user can use the grid array method repeatedly to generate different types or different lengths of geogrid layers.
The maximum layers of geogrid should not exceed 300.

Circular Failure Analysis

Please refer to the Circular Failure Analysis of Slope Stability Analyses for performance of a circular failure analysis for a reinforced slope. An example of circular failure is show in Figure 40.

Direct Sliding Analysis

To perform an analysis for direct sliding over the geogrid in reinforced slope design, the user can select Direct Sliding Over Geogrid from the Analysis menu. On the resulting page, the user can see seven of the most critical failure surfaces by clicking the Failure Surface buttons. The user can also see the most critical failure surface of each geogrid layer, as shown in Figure 41.

Figure 40: Circular Failure
After an analysis is completed, color spectrums will display along geogrid layers (Figure 40). The legend of the color spectrum on the left of the screen gives the magnitude of reinforcement. However, please note that the magnitudes of reinforcement along a layer of geogrid do not represent the actual reinforcement that geogrid is providing, but the potential magnitudes of reinforcement that geogrid can provide, if a failure surface passes that specific point on the geogrid.

**RESISTING PILE DESIGN**

One of the common approaches to remediate an instable slope is installing a row of resisting piles near the toe of the slope to buttress the upper slope. When designing a resisting pile, the engineer must know how much resisting force from the pile is required to support the upper slope to increase the factor of safety to a desired value. With this resisting force, the embedment and section of the pile can then be determined. However, the resisting force varies with the location of the pile. Most slope-stability-analyzing program will only provide factor of safety for an entire sliding body by averaging the factor of safety of each slice; however, if a slope is divided into upslope and downslope at the location where a resisting pile is to be installed, the factor of safety of the upslope usually is much lower than that of the downslope.
many cases, even though the factor of safety of the entire sliding body is lower than the required factor of safety, the factor of safety of the downslope could still be above the required factor of safety and provide supporting force to the upslope. The design process is complicated if all of the factors mentioned above are considered.

Visual Slope greatly simplifies the resisting pile design process. The user only needs to:

1. Perform a slope stability analysis for the slope without a resisting pile. If the calculated factor of safety is below the desired value, a resisting pile can be considered.
2. Identify 1) where the pile is located (by drawing the pile into the slope model), 2) the factor of safety required for the upper slope, 3) the factor of safety desired for the downslope.
3. Check the resisting pile design option. Check the downslope supporting option, if a support from the downslope is considered.
4. Perform a slope stability analysis again.
5. After the analysis is completed, click the analysis button at the bottom as shown in Figure 42. Click the results button. The result page will appear.

![Figure 42: Resisting Pile Design Process](image)

Visual Slope will automatically adjust the pile embedment and provide the internal forces of the pile for the section design.
If a tiebacks is needed, after slope stability analysis (after Step 4), tieback can be drawn in, and then click the analysis Button (Step 5) as shown in Figure 43.

![Figure 43: Resisting Pile with Tieback](image)

**MSE WALL DESIGN**

Both the NCMA method and the AASHTO method can be used in Visual Slope to perform an MSE wall design. The NCMA method used in Visual Slope is based on the third edition of the *NCMA Segmental Retaining Wall Design Manual*, while the AASHTO method in Visual Slope is, in general, in accordance with the AASHTO 2012 MSE Wall Design Guideline. Visual Slope is also capable of performing LRFD analyses incorporated with the AASHTO method. Visual Slope is also a perfect tool for tiered wall design, which will be discussed in the following sections.

An MSE wall is commonly constructed from dry-stacked units that are usually connected through concrete shear keys or mechanical connectors. An MSE wall can be constructed as an unreinforced gravity retaining wall or as a retaining wall with reinforcement, such as a geogrid, geotextile, metal grid, or metal strips. The soils in a reinforced MSE wall analysis can be divided into three zones. The soil within the reinforcement zone is called “reinforced soil.” The reinforced soil and reinforcement (such as geogrid), as well as dry-stacked units, work together
and act as a compound gravity wall. The soil behind the reinforced zone is called “retained soil.” The soil that the MSE wall, which includes the dry-stacked column and the reinforced zone, bears on is called “foundation soil.”

An MSE wall analysis includes:

1. Internal Stability
2. External Stability
3. Global Stability
4. Compound Stability
5. Settlement

Visual Slope is capable of providing all those analyses with one simple input file. The following sections describe how to perform an MSE wall analysis with Visual Slope.

**Wall Block and Connection Properties**

Prior to an MSE wall design, reinforcement properties, wall block properties, and connection properties must be specified. The reinforcement properties have been discussed in the section of REINFORCED SLOPE DESIGN. The wall block properties and connection properties should be obtained from the manufacturers.

The wall block properties include block name, block height, block width, block density, shear strength (between blocks), friction (between blocks), and wall inclination. Figure 44 shows the wall block/unit input box.

The connection properties are related to the connection between wall blocks and reinforcement. The connection data setup process has been discussed in Material Property Setup section.
Wall Geometry

Similar to a slope stability analysis, the user should use the Line Tool to draw the retaining wall cross section to scale as a design drawing, which includes:

1. The reinforced zone, retained zone, and foundation zone
2. The back slope, a broken back slope, or a front slope, if they exist

Soil strata in the foundation zone are for global stability and settlement analyses. All lines must be drawn from left to right! Do not draw wall units in detail!

To simplify the process, the user can use the MSE Wall Profile Generator to develop an MSE profile. From the Toolbar, click the Wall Profile button (Figure 45). Fill out the Wall Profile dialog box (Figure 46). The wall profile will appear (Figure 47).
Figure 46: Wall Profile Generator
Perform Analysis

Internal and External Stability
After the wall profile is completed, to perform an external stability analyses for the MSE wall, click the Reinforced MSE Wall analysis button or Unreinforced MSE Wall button in the Analysis Panel, depending on the wall type. The MSE wall analysis page appears (Figure 48). On this page, the user should first choose the wall to be analyzed. If it is a tiered wall, all levels of the wall will be listed. The user should analyze them one level a time.

After the wall is chosen, the MSE Wall page should include all the information necessary for the analysis from the provided profile. The user should then select the design method and determine if LRFD and/or seismic effect should be considered. Click the Analysis button on that page for analysis.
Global and Compound Stability

The global and compound stabilities are similar to the regular slope stability analysis with circular failure surfaces. For the global stability analysis, the area to be analyzed should generally be out of the reinforced zone as shown in Figure 49. In contrast, the compound stability analysis should focus on the reinforced zone. Figure 50 shows a compound stability result.
Figure 49: Global Stability Analysis

Figure 50: Compound Stability Analysis
SHORING DESIGN

Shoring systems commonly consist of sheet piles, diaphragm walls, soldier pile walls with lagging, etc. A shoring system can be cantilevered, single braced, or multi-level braced, depending the height of the wall. Visual Slope is capable of designing all those types of retaining walls. Similar to that of designing an MSE wall, the process of designing a shoring system includes the following three steps:

1. Setup material properties
2. Establishing soil profile (cross section), similar to that for a Slope Stability Analysis
3. Adding wall and braces (if needed)
4. Performing analysis

Setup Material Properties
Material properties for shoring design include soil properties, tieback properties, and pile properties. For setting up soil, tieback properties, please see SET UP MATERIAL PROPERTIES and Nail/Anchor Properties, respectively. Please note that subgrade modulus must be set up for the soils. Pile properties include bending stiffness EI (for mending members), compression or tension stiffness EA, self-weight, and Poison’s Ratio. For shoring design, only bending stiffness EI is needed.

Establishing Soil Profile
To design a shoring system, the user must create the soil profile (cross section) first. If you are not familiar with creating a profile, please read the Establishing Profile section. The dredge side must be on the left. The wall face should still be a slope facing the left (x1<x2), even though it is near vertical. The profile can include different soil strata, surcharges and water table. Similar to performing a slope stability analysis, the soil properties must be assigned to each soil stratum.

Drawing Wall and Brace
To draw a wall, click the Shoring Pile button on the Toolbar first. The wall line must be drawn from the dredge line to the top. The user does not have to draw the embedment. The program will adjust the embedment length automatically after calculation. Figure 51 shows the retaining wall.
Once the retaining wall is in place, braces can be drawn in to their locations. Drawing braces is similar to drawing nails. If you are not familiar with drawing nails, please read the Soil/rock nailing Design sections. If regular braces are used, click the Unbond Nail Button first. The braces can then be drawn in. The length of bracing is not important but the bracing must intercept the shoring wall.

Establishing Excavation Profile with Slope Generator

More conveniently, the user can establish an excavation with retaining system by using the Staged Excavation Generator. To use the Staged Excavation Generator, the user must first set up all the material properties to be used, and then click the Staged Excavation Generator button on Generator menu. The Staged Excavation Generator dialog box will appear (Figure 52).

The generator contains five parts. The first part is related to top of the excavation including the position of the retaining wall and the back slope. If there is no back slope, the back slope angle should be set to zero.

The second part is for soil profile from top to bottom. The top elevation equals the top elevation of the back slope. If there is no back slope, the top elevation should be equal to the top of the retaining wall.
The third part is to set up bracing elevation from top to bottom. If tiebacks are used, the tieback option should be checked and the tieback angle should be provided.

The forth part is to define excavation stages from top to bottom. The excavation elevations, water table elevations (if exist), and surcharges (if exist) must be provided.

The last part is to set up bracings with excavation. First choose the excavation level, then add or remove bracings.

After the dialog box is filled out, choose a specific stage and click View This Stage Button, the excavation profile will appear just as shown in Figure 51.

Details can be added to the model after being generated using the drawing method.
Before perform a shoring analysis, the user must setup the design code. To set up the design code, click the Shoring Analysis button on the Analysis panel. Once the Shoring Design page appears, click the Code Setting Button. The Shoring Design Setting page will appear (Figure 53). The design code includes Code Name, Wall-Soil Friction, Water Pressure type, Calculation Method, Load Combinations, Factor of Safety, and Seismic Angles. The code should be setup according to the local code or the project’s specifications. Once the code is set up and saved, it
can be used for other projects, if the code applies. Six different codes can be pre-set for future use. The last one saved will be the current code, which can be continually used until it is changed.

Figure 53: Shoring Design Setting

**Perform Shoring Analysis**

After completing the profile and setting design code, the user can perform the analysis. To perform a shoring analysis, click the *Shoring Analysis* button on the Analysis panel. Visual Slope is able to detect what type of retaining system the user is working on based on the provided profile. The following sections describe how to perform an analysis for a cantilevered, braced, or tieback shoring system.

**Cantilevered Wall**

If the profile is a cantilevered wall, after the user clicks the *Shoring Analysis* button on the Analysis panel, the shoring analysis page (Figure 54) will appear:
Retaining Width
For a continuous wall, such as a sheet pile wall or a diaphragm wall, a unit width of 1 ft or 1 m can be used for analysis. For a soldier pile wall, the retaining width should be the spacing of the soldier piles.

Embedment Width
For a continuous wall, the embedment width should be the same as the retaining width. For a soldier pile wall, the embedment width should be the width of the soldier pile or follow the code requirement.

After filling out the page, click the Analysis button for an analysis. For a cantilever shoring system, active earth pressure is always used in the analysis. Once the analysis is completed, the Moment Diagram will appear (Figure 55). The user can use the diagram buttons to display
the moment, shear, earth pressure, settlement, and failure surface diagrams. The detailed results are tabulated in the results table. A detailed text report and charts can also be generated.

Figure 55: Results of Analysis

**Braced Retaining Wall**

The example in Figure 51 is used to explain how to perform a braced retaining wall analysis. After completion of the profile, the user can choose *Shoring Analysis* from the *Analysis* panel. The *Shoring Analysis* page (Figure 46) will appear.
Pressures Type behind Wall
Based on the provided profile, Visual Slope can detect the type of bracing system. If it is a multi-braced retaining wall, Visual Slope will allow the user to select the different types of earth pressure — active, at rest, program-defined, and user defined. Active and at rest earth pressures are well defined in many text books. Therefore, only program defined and user defined pressures will be discussed in this manual.

Program Defined Pressure
The program defined earth pressure is based on:

\[ P = 0.65\gamma HK_u \]

The shapes of the earth pressure are:
User Defined Pressure
The user can also define a specific earth pressure according to the project conditions. The user can type elevations and earth pressure values into the input box (Figure 58). The pressure value is positive if the pressure is acting on the back of the wall, and is negative if the pressure is acting on the front of the wall. The user defined pressure must be input from top to bottom. If the input data does not go deep enough to cover entire shoring depth, Visual Slope will extrapolate the data to the depth needed using the last two sets of input data. Please note that if a user defined pressure is used, all other pressures, such as water pressure, surcharge pressure, etc. will be ignored in calculations.

Additional Pressure
In many cases, in addition to the common pressures, such as earth pressure, water pressure and surcharge pressure, other pressures, such as wave pressure, should be considered when designing. To include an additional pressure, the user can check the Additional check box; the additional pressure input table will appear. Please refer to the User Defined Pressure section for how to input additional pressure. The additional pressure option cannot be used with the user defined pressure option as mentioned in the section above.
**Other Options**

For a multi-braced retaining wall, the user can select *With or Without Embedment*. If the *Automatically-Adjust-Tieback* option is selected, Visual Slope will adjust the tieback free length so that the bonded zone will be beyond the failure zone and it will adjust the bonded length to meet the factor of safety requirement.
Staged Construction is another option. If Staged Construction is off, the bracing will be added to the retaining wall at the same time. However, if Staged Construction is on, the bracing will be added to the retaining wall layer by layer with excavation. Therefore, staged construction calculation will result in a larger retaining wall deflection. Staged Construction will be affected, but only if the Winkler calculation method is used in code setting.

Other Features
Visual Slope also adjusts the embedment so that the retaining wall will not only satisfy the equilibrium condition, but also have an adequate factor of safety against the bottom heave and piping.

Visual Slope will also estimate ground surface settlement according to ground loss due to retaining wall deflection. The estimated settlement is for reference only.

Figure 59 is the analysis result.

Figure 59: Result of Tieback Retaining Wall

SEEPAGE ANALYSIS
The Visual Slope seepage analysis module is capable of performing static seepage analysis for a profile with isotropic or anisotropic, homogeneous or non-homogeneous, and saturated or
unsaturated soils. Drains and artesian aquifers can also be simulated. Visual Slope seepage analysis module can be used for confined boundary seepage analysis, free surface boundary seepage analysis, and phreatic line analysis. The results of a calculation including equipotential lines, equipotential shading and flow lines (a vector field) can be presented. The user can also obtain a flow rate and exit gradient by using the cutting cross-section feature.

Profile
The method of establishing a profile for a seepage analysis is the same as that used to establish a profile for slope stability analysis. Therefore, please refer the Establishing Profile section. Figure 60 is a typical profile for a seepage analysis.

![Figure 60: Typical Seepage Analysis Profile](image)

Input Soil Data
The soil data required for a seepage analysis is permeability (conductivity) in horizontal and vertical directions (Kx and Ky). To input the soil data, click the Material button in the toolbar. The material input page will appear. Select Seepage Analysis from the pull-down list on input page for Kx and Ky input (see Figure 61).

If a soil layer is an artesian layer, its water head should be input in the Porepressure column as shown in Figure 17.

Please also see the Material Properties section of this manual for more details.
Boundary Conditions

Boundary conditions must be set up, before a seepage analysis can be conducted. There are four types of boundary conditions, equipotential boundary, no-flow boundary, in-flow boundary, free-flow boundary.

An equipotential boundary is submerged under water, where the equipotential equals the water head. To set up an equipotential boundary, water lines can be used as shown in Figure 62.

A no-flow boundary is a surface across which there is no water flowing out or in (for example, an impermeable soil layer or an impermeable wall). The boundaries on each side in Figure 62 should also be considered as a no-flow boundary, since they are far away where the hydraulic gradient equals zero across the boundary. To establish a no-flow boundary, boundary lines, limit lines or pile lines can be used, as shown in Figure 62. In a confined seepage analysis, if a boundary is not defined, it is considered as a no-flow boundary.

An in-flow boundary specifies the rate of water flow through the boundary. An in-flow boundary can be used to simulate rainfall on surface.
A free-flow boundary is a boundary that water can flow in or out freely. In a phreatic line analysis and an unconfined analysis, if a boundary is not defined, it is considered as a free-flow boundary.

**Figure 62: Boundary Conditions**

**Run Analysis**
After the profile and boundary conditions are completed, a confined boundary seepage analysis, free surface boundary seepage analysis, or phreatic line analysis can be performed by clicking the corresponding button on the *Analysis* panel.

**View Results**
After the analysis is completed, the seepage frame will appear. To draw contours, shading, or flow lines, click the corresponding buttons as shown in Figure 63.

To calculate flow rates and Y-exit gradient, click the *Section* button and then draw a section on the profile. The flow rate and Y-exit gradient of that section will appear, as shown in Figure 63.
Drain and Artesian
Geogrid lines can be used to simulate drains, as shown in Figure 62. To simulate an artesian layer, a soil layer with specified water head can be used.

TUNNEL LINING DESIGN

Visual Slope-Tunnel Design module uses so called load-structure model for tunnel lining design. An initial steel set or final lining is modeled with FEM beam elements. The interaction between the tunnel structure and the surrounding soil/rock is simulated with radial and tangential springs. Vertical and horizontal pressures and/or all-round pressure with different load combinations can be applied to the tunnel structure.

The user can use the tunnel generator to set up calculation models with different shapes in just a few minutes. All load combinations can be calculated with just one click.

Similar to that of designing a shoring system, the process of designing a tunnel lining includes the following three steps:

1. Setup material properties
2. Establishing soil profile (cross section), similar to that for a Slope Stability Analysis

3. Generating tunnel lining

4. Specifying load conditions

5. Performing analysis

**Setup Material Properties**

Material properties for tunnel lining design include soil properties, lining properties, and spring properties. For setting up soil/rock please see SET UP MATERIAL PROPERTIES. Please note that elastic modulus and Poison’s ratio must be set up for the soils/rocks. Lining properties include bending stiffness EI (for mending members), compression or tension stiffness EA, self-weight, and Poison’s Ratio. The radial and tangential spring properties will be automatically generated from the elastic modulus and Poison’s ratio of surrounding soil/rock. If the user wants to specify the spring properties, the user can do that too.

**Establishing Soil Profile**

To design tunnel lining, the user must create the soil profile (cross section) first. If you are not familiar with creating a profile, please read the Establishing Profile section. The section does not need to go all the way up to the ground surface. Only surrounding soil/rock should be included in the section. The profile can include different soil/rock strata. Water table and surcharge are not needed. Similar to performing a slope stability analysis, the soil/rock properties must be assigned to each soil stratum.

**Generating Tunnel Lining**

**Regular Tunnel Shape**

The tunnel lining to be designed must be generated through the tunnel generator (Figure 64) by clicking 📄. Different regular shapes of tunnel lining can be easily generated by specifying the required parameters. After parameters are specified, click the View Diagram button; the tunnel lining section appears on the left of the screen. Tunnel lining structure must be symmetrical.

**Any Tunnel Shape**

If a tunnel shape is not included in those regular shapes, any shape generator can be used. However, the tunnel shape must still be symmetrical. Only the parameters for the right side of
the tunnel are needed. The tunnel lining can be comprised of arcs and/or straight lines. The coordinates of the first point, which is the middle point of the tunnel crown, must be provided first, followed by those of each transition point from top to bottom. If between two points is a straight line, “Line” must be specified in the first column, otherwise “Arc” must be specified. If a section is an arc, the coordinates of the center must also be provided. A tunnel section can be open or closed. If a tunnel section is closed, the horizontal coordinate of the last point must equal to that of the “First Point”.

Specifying Load Conditions

Water Pressure

Water pressure is an all-round pressure. If a water pressure is from outside toward the tunnel, a negative value should be specified. Otherwise, if a water pressure is from inside of the tunnel, a positive value should be provided.

Vertical and Horizontal Pressures

Vertical and horizontal pressures can be specified by clicking the Vertical Load and Horizontal Load buttons, respectively. A total of four vertical pressures and four horizontal pressures can be defined, as shown in Figure 65.

Load Combinations

Once individual pressures have been input, load combinations can be specified. A total of four load combinations can be defined as shown in Figure 66.

Performing Analysis

After the tunnel lining section and load conditions are defined, tunnel lining design can be performed. First click the Tunnel Design button on the Analysis panel to open the tunnel design page as shown in Figure 67.

After the analysis is completed, structure deformation, bending moment, axial force, and shear force for different load combinations can be shown as in Figure 68.
Figure 64: Tunnel Section Generator
Max. 4 vertical loads and 4 horizontal loads. Loads towards tunnel negative.

**Figure 65: Load Setup**
### Figure 66: Load Combinations

<table>
<thead>
<tr>
<th>Load Combination</th>
<th>Load Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>Self-Weight</td>
<td>0</td>
</tr>
<tr>
<td>Hydro-Press</td>
<td>1</td>
</tr>
<tr>
<td>V-L1</td>
<td>0</td>
</tr>
<tr>
<td>H-LD1</td>
<td>2</td>
</tr>
</tbody>
</table>
**Figure 67: Tunnel Lining Design Page**

### Internal Forces

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>ft</td>
<td>ft</td>
<td>lb</td>
<td>lb</td>
<td>lb-ft</td>
</tr>
<tr>
<td>Max.</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>Min.</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

### Displacement

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>ft</td>
<td>ft</td>
<td>ft</td>
<td>ft</td>
<td>ft</td>
</tr>
<tr>
<td>Max.</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>Min.</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>
GROUND FREEZING CONSTRUCTION DESIGN

Ground freezing is a construction technique used under circumstances in which the soil needs to be stabilized to prevent it from collapsing next to excavations and to prevent contaminants in soil from being leached away. This technique can also be used to prevent ground water intruding a tunnel or an excavation. Ground freezing has been used for at least one hundred years. However, most ground freeze pipe layouts are estimated with rulers and compasses, which can result in inaccurate, inefficient and unsafe designs.

Visual Slope-Ground Freezing module allows users design the number of freeze pipes and layouts much more accurately and efficiently. To perform ground freezing design, following two steps should follow:

1. Defining freezing pipe layout
2. Performing analysis
Defining Freezing Pipe Layout
A freezing pipe layout must be defined through freezing pipe generator. After the Freezing Pipe Generator button is clicked, the freezing pipe generator page appears as shown in Figure 69. In this page, the user should input: Ground Temperature, Thermal Conductivity, Freezing Pipe Temperature, and Days of Freezing. Different patterns of pipe layout can be generated through the line generator or circular generator. If a pipe layout is irregular, click the X column, a red button will appear on the right. Click that button and move the cursor to where the pipe is to be located and click again. The coordinates of the pipe will appear in the table.

Performing Analysis
Once the layout is specified, analysis can be performed by clicking the Freezing Design button on the analysis panel. The results are shown as in Figure 70 and Figure 71.

OTHER FEATURES

Visual Slope includes many other features. This section briefly describes those features.

SEISMIC ANALYSIS
The seismic effect can be included in slope stability analysis, reinforced slope design, soil/rock nailing design, and MSE wall design. To include the seismic effect in the analysis, click the Seismic Menu and choose the Seismic On option.

LRFD/LIMIT STATE ANALYSIS
LRFD/Limit State analysis can be used in slope stability analysis, reinforced slope design, and MSE wall design. To use the LRFD method, click the LRDF/Limit State Menu and choose the LRDF/Limit State On option.

ONLINE TUTORIAL VIDEOS
The tutorial videos at www.visualslope.com can help users learn how to use Visual Slope. Users are encouraged to watch those videos.
Figure 69: Ground Freezing Design
Figure 70: Ground Freezing Design for Excavation (Plan View)

Figure 71: Ground Freezing Design for Excavation (Elevation View)