V7 User's Guide

2019

TABLE OF CONTENTS

QUICK OVERVIEW

MAIN FEATURES

The Visual Slope is a multifunction geotechnical engineering computer program developed for:

- ✧ Slope Stability Analysis
- ✧ Soil Nailing Design
- ✧ Retaining Pile Design
- ✧ MSE Wall Design
- ✧ Reinforced Slope Design
- ✧ Gravity Retaining Design
- ✧ Seepage Analysis
- ✧ Shoring Design
- ✧ Tunnel Lining Design
- ✧ Freezing Construction Design
- ✧ Two-dimensional Finite Element Method (FEM) Analysis
- ✧ Pile Supported Reinforced Embankment Design

All functions in the Visual Slope 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, the AutoCAD import method, or the image tracing method to help users establish input files, which allows a detailed and accurate modeling of a real situation to be achieved and which greatly reduces chances of input errors. Several quick generators can also be used to speed up the drawing process.

STEPS OF EXECUTION

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

1. Starting Project
2. Setting Up Material Properties
3. Modeling
4. Performing Analysis

This user guide will provide our users with tutoring on how to use Visual Slope. The following sections will describe each of the above four steps and the features in modeling and analysis.

INPUT FILE

Visual Slope allows users to save their work any time during using Visual Slope. No matter which module is in use, the saved file is a text file with the same format and same extension (.slp). The file size usually does not exceed 30 KB. Therefore, it is very easy to be emailed.

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 <<*Browse to Other Files*>>*.*

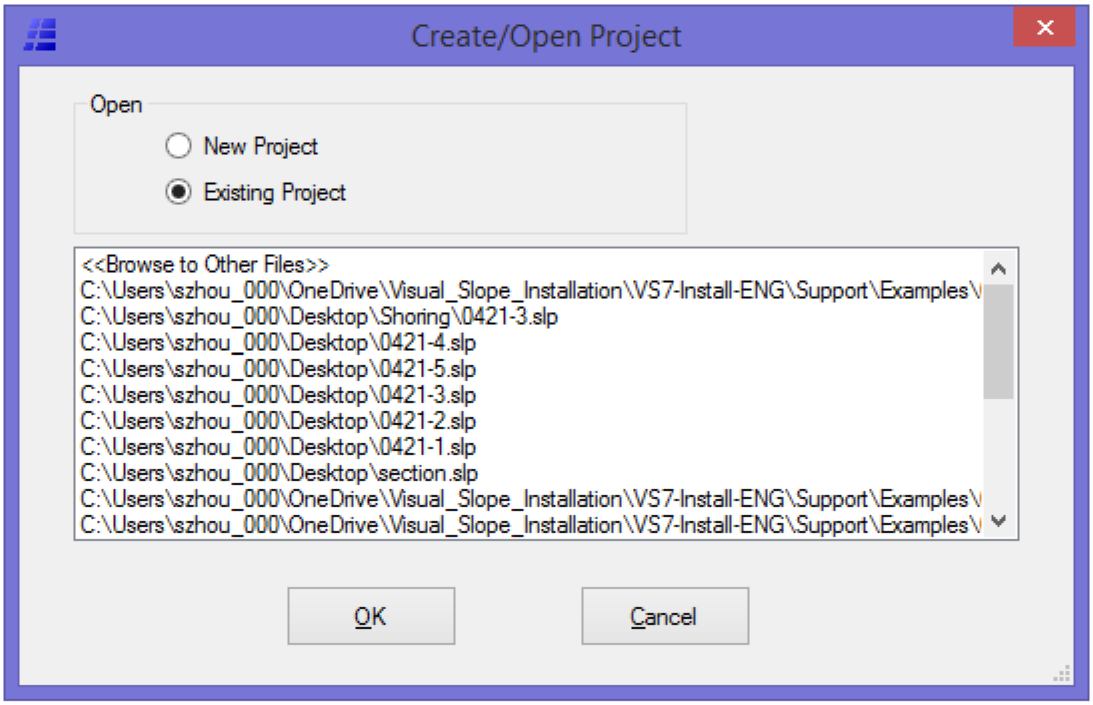


Figure 1: Create/Open Project

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.

On that page, the user can input the project information, select the units, 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 described in the next section.

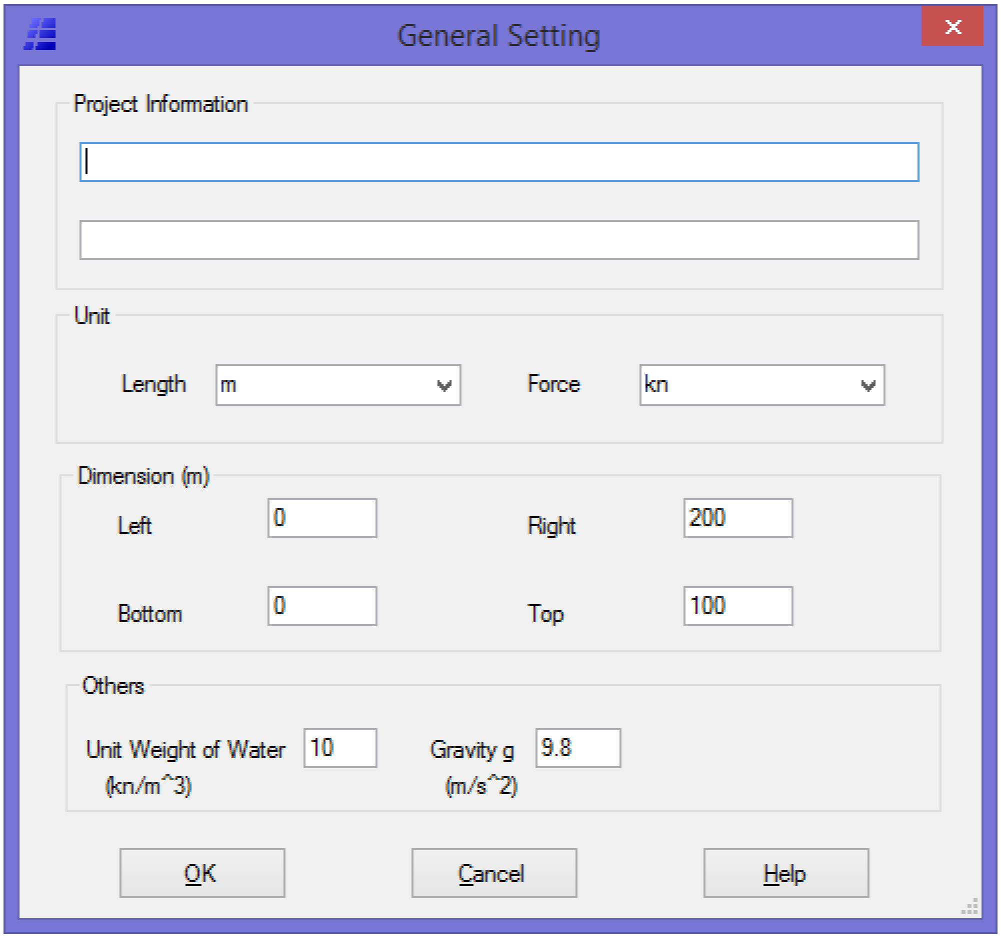


Figure 2: General Setting

FUNCTIONS OF BUTTONS

Visual Slope contains six layers of the top menu, a bottom status bar, and an *Analysis* panel on the right. To help users become familiar with the functions the menu, status bar, and *Analysis* panel, in this section, the buttons in the menu, status bar, and *Analysis* panel will be described in detail.

FILE AND SETTING

Under the File and Setting menu, there are fourteen buttons that are used to manage files. The following describes the function of each button.


The <*Exit*> button is for exiting Visual Slope.


The <*New*> button is for starting a new Visual Slope project.

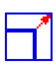

The <*Open*> button is for opening an existing Visual Slope project.


The <*Save*> button is for saving the current project (file).


The <*Save As*> button is for saving the current file as a file with a different file name.


The <*Print*> button is for printing the diagram showing on the Visual Slope screen.


The <*Setting*> button is for editing the *General Setting* page discussed in the Starting Project section.


The <*Scale*> button is for rescaling the current model. Once this button is clicked, the *ReScale* page (Figure 3) will appear. The user can scale and/or move the model horizontally and vertically.

R is a scaling factor to scale the model larger or smaller. *DX* is used to move the model farther right if set to a positive number, or farther left if set to a negative number. *DY* is used to move the model up if set to a positive number, or to move the model down if set to a negative number.

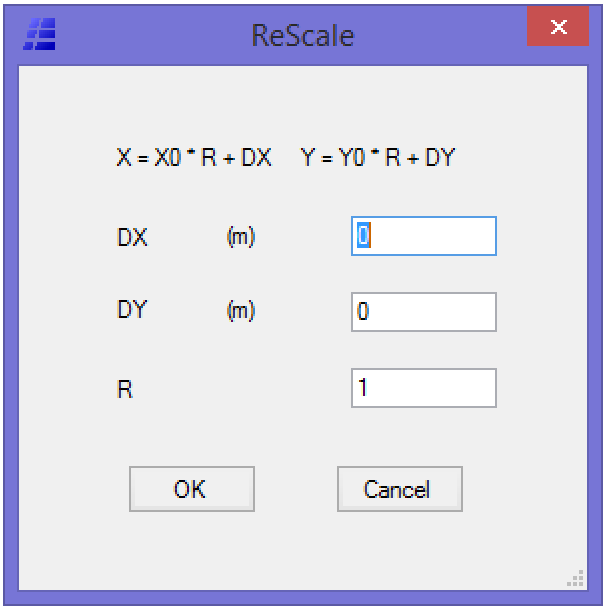
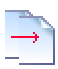
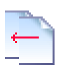


Figure 3: ReScale Page

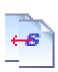


The <*DXF In*> button is for importing a DXF file. The DXF file must be formatted to the DXF 2000 version or earlier. The diagram in the DXF file must be formed with lines or polylines; other lines, such as hatches, will be ignored. For details, see the <u>AutoCAD File Import Method</u> section.



The <*DXF Out*> button is for converting the current Visual Slope file into a DXF file.

The <*STABL In*> button is for importing a STABL file and converting it to a Visual Slope file.

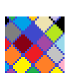


The <*STABL Out*> button is for converting the current Visual Slope file to a STABL file.

The <*Note-Line*> button is for drawing remark lines that are not for any analysis purposes.

The <*Set Remark Location*> button is for positioning a note/remark on the model after it has been typed in the *Note Box*.

MATERIAL



The <*Material*> button is for setting up or editing material properties to be used in the analysis. Details will be described in the <u>Material Properties</u> section.

MODEL

The <*Select*> button is for selecting a line or point to edit. Click this button, and move the cursor to the line or point to edit, and then click again. The *Line Edit* or *Point Edit* page will appear.

The <*Group Select*> button is for selecting multiple lines. With this button, the user can drag a box over lines to select them and can continue to select until all desired lines are selected. Then click the right mouse button to choose the next action, such as edit, delete, etc.

The <*Zoom In*> button is for zooming in a portion of the model for details. After clicking this button, move the cursor to the position to zoom in, and click again to define the start point. Once the start point is defined, drag the rectangular frame to define the zoom-in range, and click again. The model will be zoomed in. The zoom-in process can only be used consecutively for five times before zooming out.

The <*Zoom Out*> button is for zooming out. Each click will take the amplified model one step back from the previous zoom-in process.

The <*Move*> button is for moving the model in horizontal and vertical directions.

The <*Undo*> button is for undoing the last step.

The <*Flip*> button is for flipping the model from the right to the left or vice versa. Visual Slope requires a slope to face left for a slope stability analysis. If a slope faces right, it must be flipped before a slope stability analysis.

The buttons below are for drawing different types of lines in an analytical model.

The <*Geo Line*> button is for drawing soil/rock lines.

The <*Water*> button is for drawing water table lines.

The <*Geogrid/Strap*> button is for drawing geosynthetic or metal strap lines

The <*Unbonded Nail*> button is for drawing unbonded sections of soil nails or tiebacks.

The <*Bonded Nail*> button is for drawing bonded sections of soil nails or tiebacks.

The <*MSE*> button is for drawing MSE-wall lines but is not commonly used.

 The <*Pile*> button is for drawing pile lines or members subject to bending moments.

The <*Limit*> button is for drawing limit lines. Limit lines can be used to restrict failure surface search area, or as a part of a composite failure surface in a slope stability analysis, or as impervious lines in a seepage analysis, or a freezing pipe in a freezing construction design.

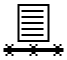
The <*Boundary*> button is for drawing boundary lines. Boundary lines are used to define the boards of a model and cannot be used inside the model.

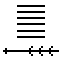
 The <*Quick Boundary*> button is for quickly setting the boundary of a model.

 The <*Rain*> button is for drawing rain lines. Rain lines can be used to analyze slope stability after rain.

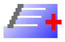
Once one of these buttons is clicked, this type of line can be drawn. The details for how to draw will be described in the modeling section.

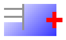
GENERATORS

 The <Geogrid/Strap Generator> button is for generating geosynthetic or metal strap lines if there are too many lines to draw. Before using this button, geogrid/metal strap properties must be set up first.

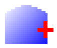
 The <*Soil Nail Generator*> button is for generating unbonded and bonded nail lines if there are too many nail lines to draw. Before using this button, nail properties must be set up first.

 The <Pile Generator> button is for generating vertical piles used in pile-supported embankment design.

 The <*MSE Wall Generator*> button is for generating an MSE wall section. See MSE wall section for detail.

 The <*Shoring System Generator*> button is for generating a shoring system model and staged excavation. See shoring section for detail.

 The <*Freezing Point Generator*> button is for generating a freezing system. See freezing construction design section for detail.



The <*Tunnel Lining Section Generator*> button is for generating a tunnel lining model. See tunnel lining design section for detail.

The <*Tracing*> button is for establishing a model by tracing an image. See modeling section for detail.

The <*Curve Line Generator*> is for generating curve lines in the model.

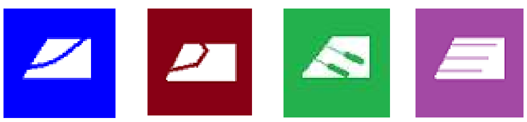
STATUS BAR

The <*Orthogonal*> button is for assistance in drawing vertical and horizontal lines.

The <*Auto-Connection*> button, when it is on, will help Visual Slope connect soil lines at their ends if the ends are very close, or at the lines' intersection.

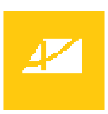
ANALYSIS PANEL

The buttons on this panel are for different analyses.



Those buttons are all for slope stability analyses. Currently, they serve the same function.

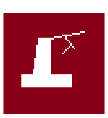
This button is for slope stability analysis under rainfall or seepage conditions.



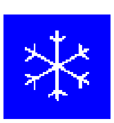
This button is for retaining pile design.

This button is for unreinforced segment retaining wall design.

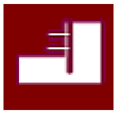
This button is for MSE wall design.



This button is for gravity wall design.



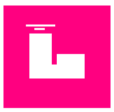
This button is for freezing construction design.



This button is for shoring design.

This button is for tunnel lining design.

This button is for phreatic line analysis.



This button is for seepage analysis under confined conditions.

MATERIAL PROPERTIES

Material properties in Visual Slope include:

1. Soil Properties
2. Geogrid/Metal Strap Properties
3. Soil Nail/Tieback Properties
4. MSE Wall Unit Properties
5. Wall Unit and Geogrid Connection Properties
6. Beam/Pile Properties

The material properties must be set up and assigned to the model before an analysis can be performed. The type of material needed depends on the type of analysis to be performed. In an MSE wall analysis, for example, soil, geogrid/metal strap, 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

To set up material properties, the user must first click the *Material* button (button section) under the *Material* menu. The *Material* page (Figure 4) will appear.

In the *Material* page, the user must first select the material type, such as *Soil Material, Geogrid/Metal Strap*, and so on, from the pull-down list (Figure 4) 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 5).

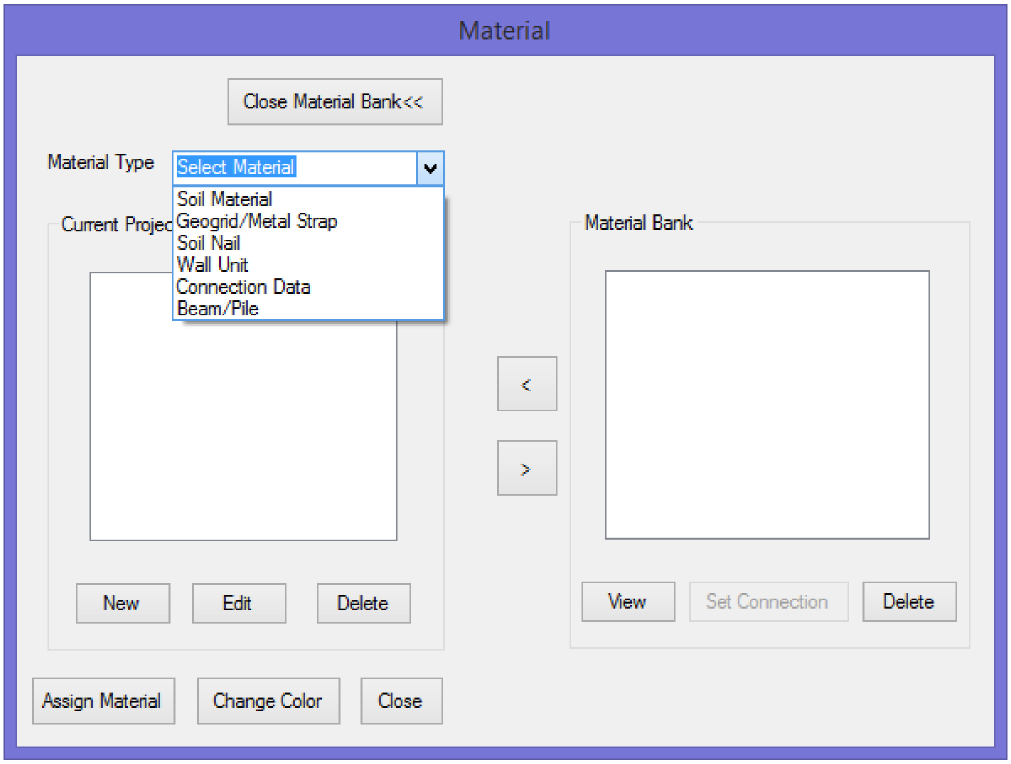


Figure 4: Material Page

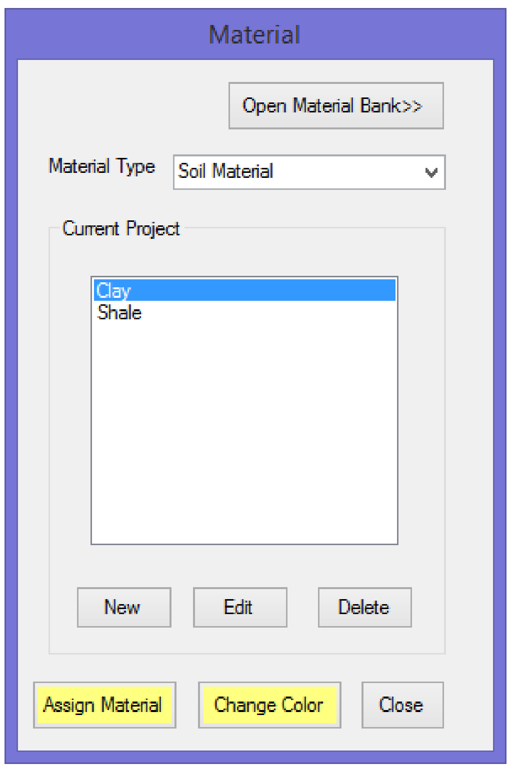


Figure 5: Select Material Type

To edit the properties of an existing material, select the material name listed in the list box, and then click the *Edit* button. The *Material Edit* page will appear (Figure 6). In the *Material Edit* page, the material parameters can be viewed or edited.

To add a new material, select the material type, and click *New* button on the *Material* page. The *Material Input* page will appear (Figure 6). The material parameters can be input in this page. After finishing, click the *Close* button. The material name will show in the list box. A similar process can be used to establish other material properties.

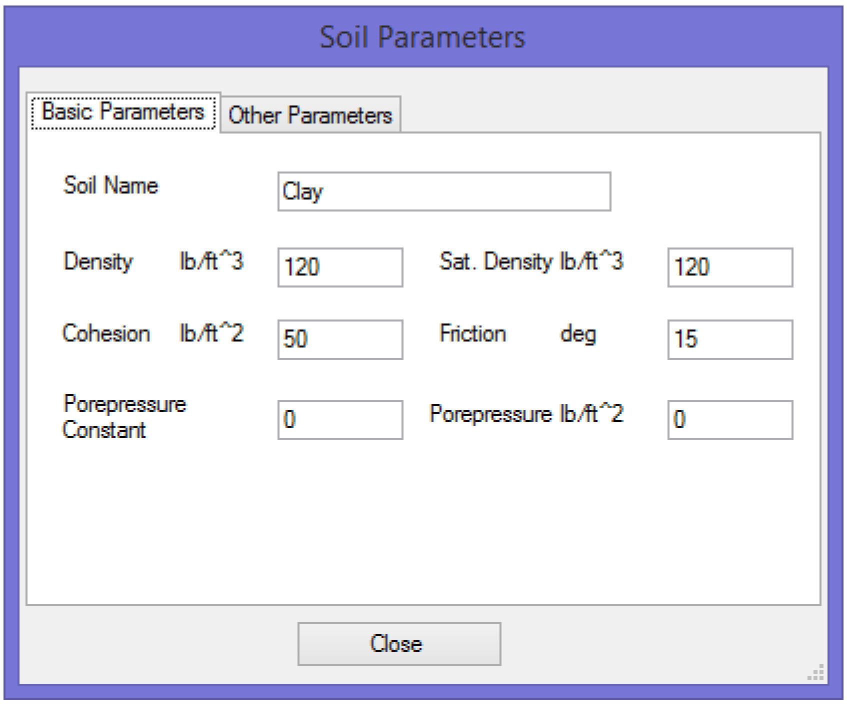


Figure 6: Material Input Page

SOIL PROPERTIES

Seven soil parameters and eight parameters are included in the *Basic Parameters* page and *Other Parameters* page, respectively.

- *Soil Name* is to identify a specific soil.
- *Density* is to specify the soil's natural (moist) density.
- *Saturated Density* is to specify the soil density when it is saturated. With this parameter the program can calculate the soil buoyance density by subtracting the water density from the saturated density.

- *Cohesion* is to specify the soil cohesion. If an effective analysis is to be performed, the effective cohesion parameter must be used; otherwise, the total cohesion should be used.
- *Friction* is to specify the soil friction angle in degree. If an effective analysis is to be performed, the effective friction angle must be used; otherwise, the total friction angle should be used.
- *Porepressure Parameter* defines excessive porepressure within the soil if applied.
- *Porepressure Constant* is a ratio of excessive porepressure increase with the depth if applied.

For all analyses in Visual Slope, except for a seepage analysis or a freezing construction design, the above parameters are required.

The parameters in the *Other Parameters* page are for MSE wall settlement analyses, seepage analyses, shoring design, and tunnel lining design.

- *Pc*, *Cc/(1+e)*, and *Cr/(1+e)* can be obtained from one-dimensional consolidation tests and are described in detail in regular soil mechanics books. Therefore, they will not be further discussed here.
- *Kx* and *Ky* are permeabilities in the horizontal direction and vertical direction, respectively.
- *Subgrade Modulus* is used to generate the spring constant in a shoring design analysis. If this parameter is not provided, Visual Slope will use the elastic modulus to calculate the spring constant.
- *Elastic Modulus* and *Poisson's Ratio* must be provided for a tunnel lining design and FEM analysis.

GEOSYNTHETICS/METAL STRAP PROPERTIES

Geosynthetics or metal straps are used for a reinforced slope design or MES wall design (Figure 7).

- *Reinforcement Name* is to identify a specific reinforcement.
- *Material Types* (plastic or metal) must be defined before inputting other parameters.
- *Long Term Design Strength* is used for a geosynthetic reinforcement and is obtained by dividing all factors of safety to the ultimate strength.
- For straps, the strength of a single strap should be used, which is obtained by multiplying the material strength to the cross area of the strap. The unit should be in force, such as lb or kn.

- When a layer of geosynthetics or metal strap is placed in the soil, the friction and cohesion on the interface between the soil and the reinforcement may not be the same as in the original soil; it usually will be less. Therefore, reduction factors (of less than 1) should be considered. However, if the friction and/or cohesion would be increased by some special treatment, a factor of greater than 1 can also be considered.
- *Front End Reduction Factor* defines the fixity condition of the reinforcement. If there is no fixity at the front end of reinforcement, the reduction factor should be 0; if the front end is completely fixed, the factor should be 1; if the condition is in between free and fixed, this factor should be between 0 and 1.
- If metal straps are used, the strap width and horizontal spacing must be provided.
- Finally, the fixity of the back end of the reinforcement must also be defined. If the back ends of the reinforcement are anchored into the back slope, the back end should be defined as fixed; otherwise, it should be considered as free.

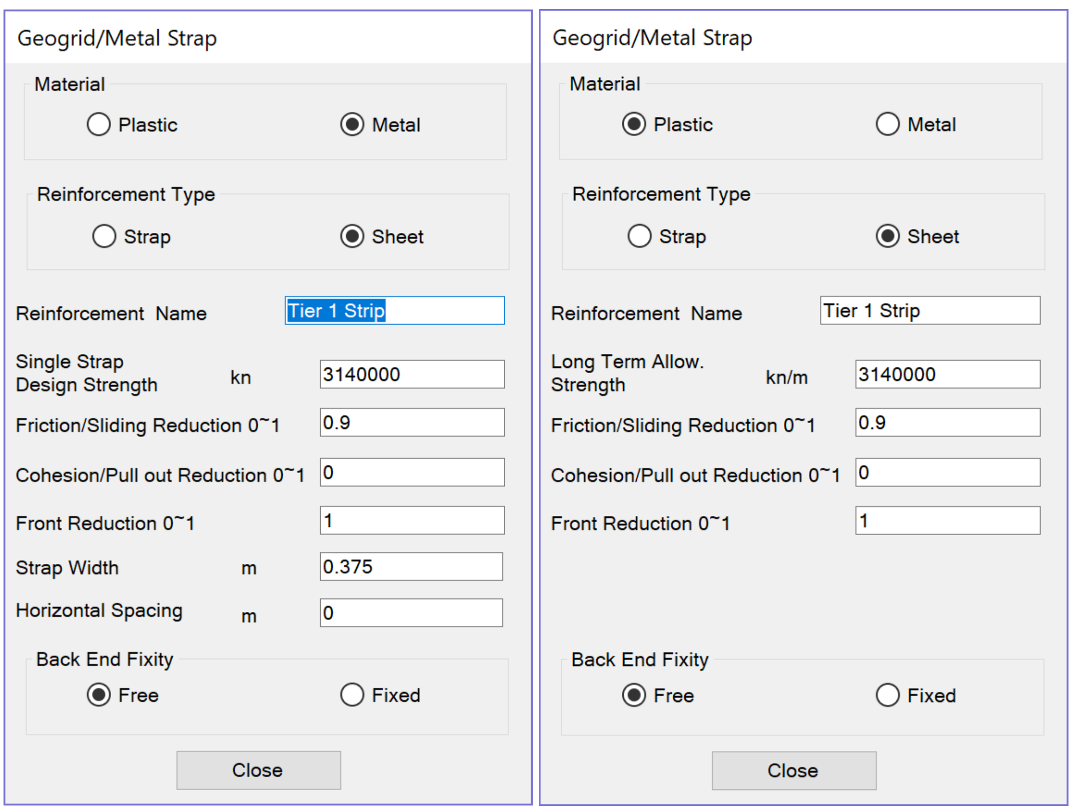


Figure 7: Geosynthetics and Metal Strap Properties

NAIL PROPERTIES

Nails can be used for soil nails, tiebacks of a shoring system, rock bolts for a tunnel, and shear keys. For convenience, only “nail” is used here. A nail usually consists of an unbounded portion and bounded portion or a bounded section only. The parameters (Figure 8) discussed are for the bounded portion. The unbounded portion only transfers the reinforcement from the bounded portion to the front end of the nail.

- *Nail Name* is to identify a type of nail.
- *Hole Diameter* is to provide the hole size. The bore hole will be grouted to fix the nail to the ground. The larger the hole, the larger the bounded area.
- *Bound Strength* is the strength between the grout and surrounding soil/rock.
- *Horizontal Spacing* defines the horizontal spacing of the nails.
- *Tendon Capacity* and *Head Capacity* define how much force the nail tendon and nail head can take.
- *EA* is compression or tension modulus that is only used for an FEM analysis.
- A nail can be used either for tension reinforcement or shear reinforcement. The shear option should be chosen if the nail plays a shear key role.
- In the *Function* option, if the nail option is chosen, the nail will provide reinforcement in a slope stability analysis; if the tieback option is selected, the nail only provides bracing to the retaining wall, but not reinforcement to the soil; if the *Nail + Tieback* option is chosen, the nail will provide both functions.

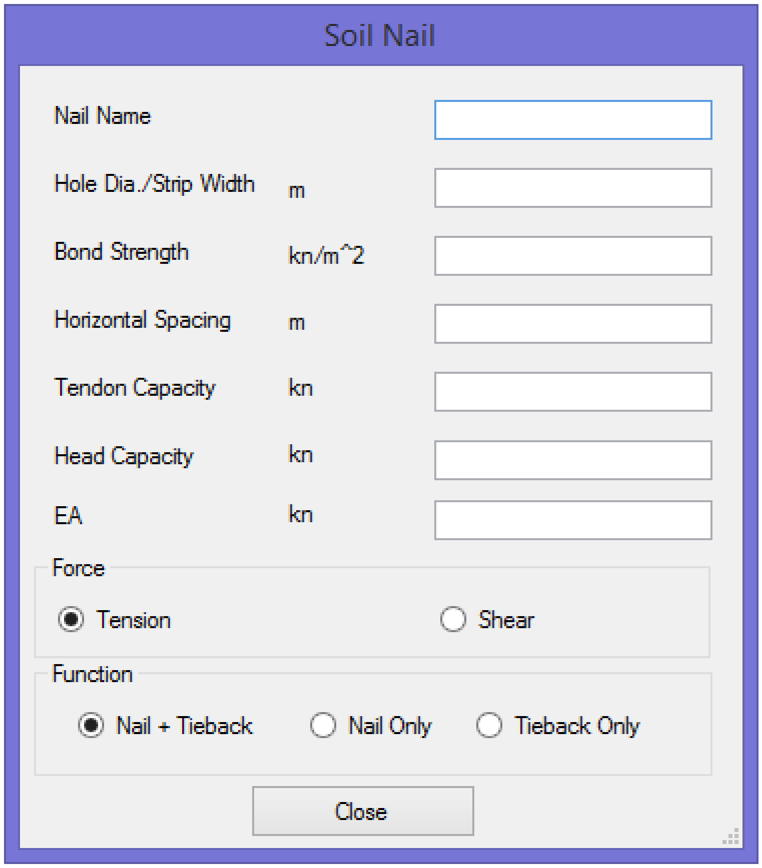


Figure 8: Nail/Anchor Properties

MSE WALL UNIT PARAMETERS

MSE Wall Unit parameters define an MSE wall facing (Figure 9).

- *Block Name* identifies the wall facing.
- *Block Height* and *Block Width* define the wall facing dimensions.
- *Block Density* can be used with the wall facing dimensions to calculate the weight of the wall block.
- *Shear Strength* and *Friction* define the wall facing strength.
- *Block Density*, *Shear Strength*, and *Friction* are not used in the FHWA method.
- *Wall Angle* in degree is to define the sit back of the MSE wall. For a vertical wall, this parameter should be set to a very small value, like 0.1.

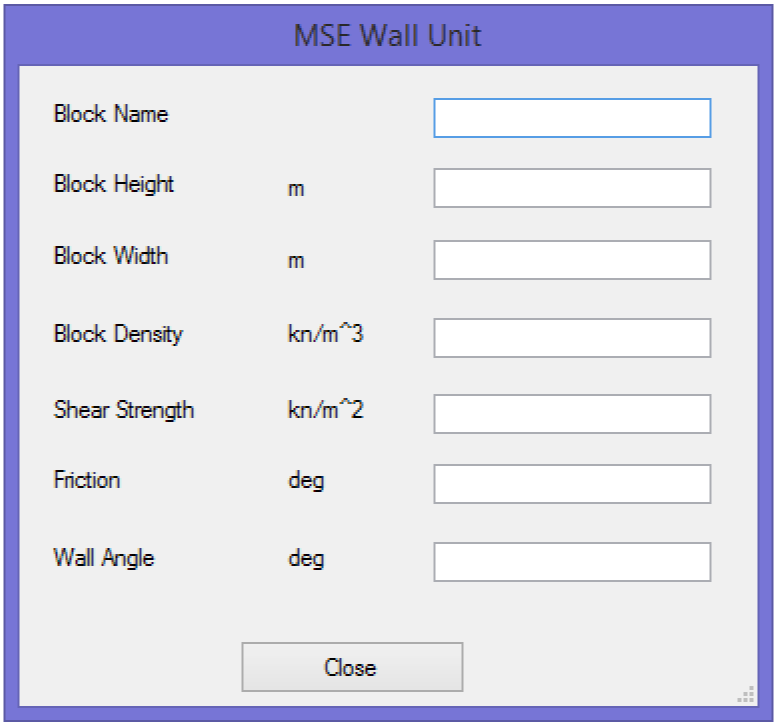


Figure 9: MSE Wall Unit Properties

CONNECTION PARAMETER

The connection data (Figure 10) are only needed for MSE wall design with the NCMA method. Once the data are set up, they 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. The *Connection* page will appear, as shown in Figure 10. 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 in the remaining connection data. The connection data should be from the wall unit manufacturer. If the connection data are not provided, Visual Slope will generate the connection based on the front-end condition of the reinforcement.

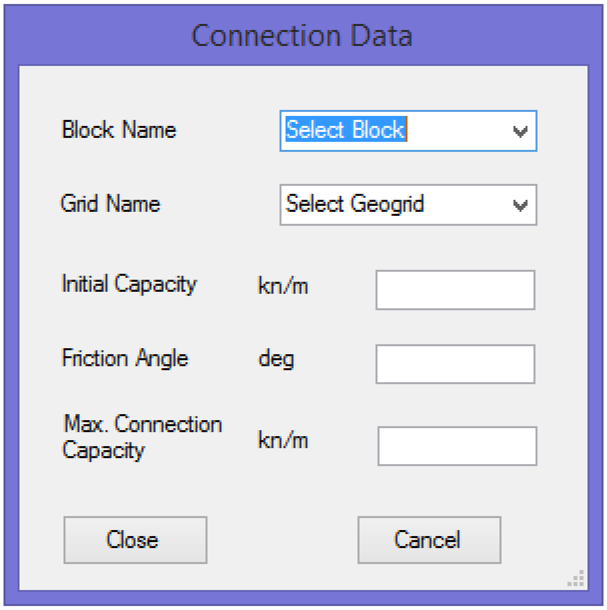


Figure 10: Connection Properties

PILE/BEAM PARAMETERS

Piles only provide vertical support and can be used in pile supported embankment design. Only the axial force is considered. Beams are used to provide lateral support to excavations and tunnels (Figure 11).

The *Pile* or *Beam* option identifies whether the properties are for piles or beams.

Pile/Beam Name gives a name to the properties.

For the beam option, the following parameters must be provided:

- *EI:* Moment inertial.
- *EA:* Compression/tension modulus.
- The other two parameters are *Poisson's Ratio* and *Self-Weight*.

For piles the following parameters must be provided:

- *Cross Area*: The pile cross area.
- *Cap Area*: The pile cap area.
- Other parameters are *Elastic Modulus*, *Pile Capacity*, and the pile *Longitudinal Spacing*.

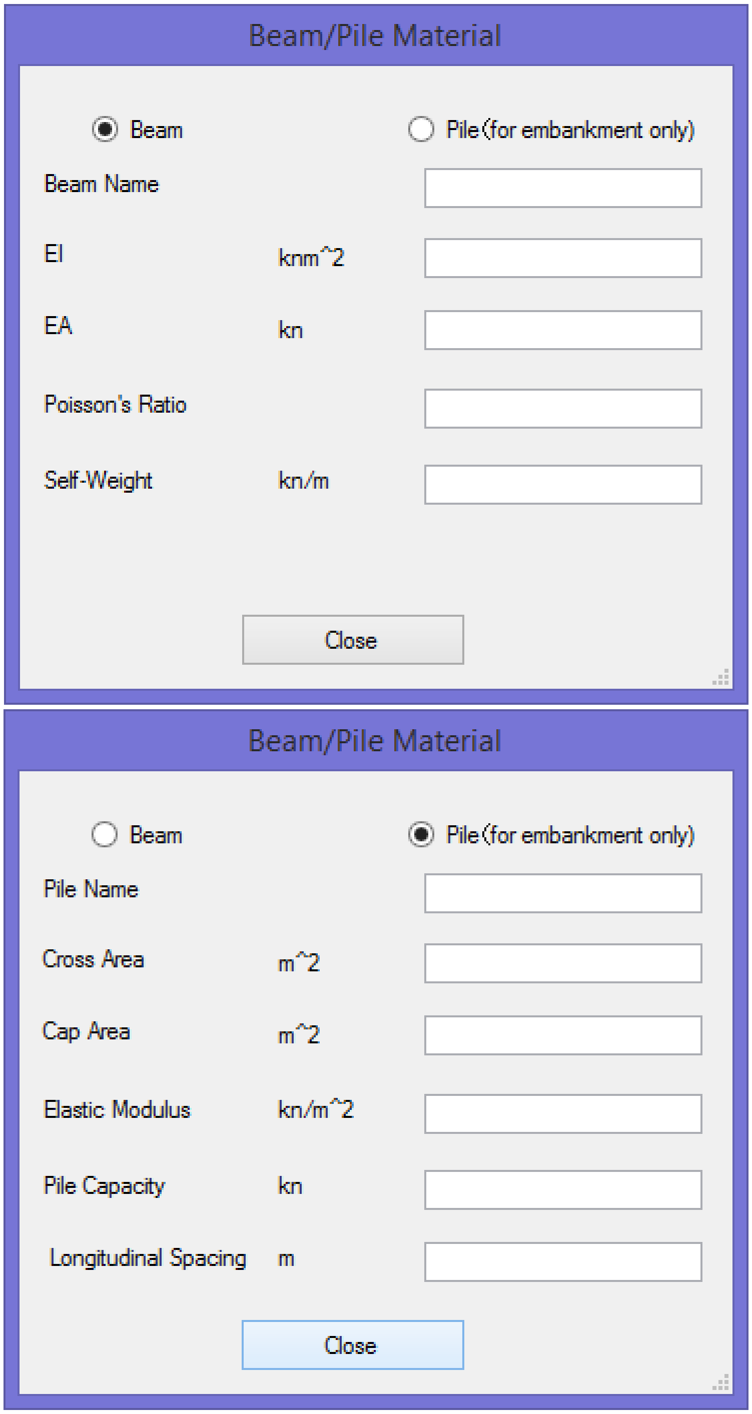


Figure 11: Beam/Pile Properties

SAVE TO MATERIAL BANKS

To save a material to the corresponding bank, click the *Open Material Bank* button first to open the material bank. Select the material name to be saved, and click the > button in Figure 4. This material is saved to the database and can be used for different projects.

USE SAVED MATERIALS

Opposite to saving a material to the material bank, materials saved in the material bank can be withdrawn and used for the current project. Choose the material name in the *Material Bank* first, and then click the < button in Figure 4. The material will be imported to the current project.

MATERIAL PROPERTY ASSIGNMENT

There are two ways to assign material properties to the model. Once the *Material* page is closed, material buttons will appear on the *Material* tray, as shown in Figure 19. The materials can be assigned to the model.

Soil Property Assignment

To assign soil properties to soil layers, click the *Soil* button first, and then click the zone to which the soil properties should be assigned. The color of the soil layer will change to the same color as that of the button. To view the material name of a soil layer, move the cursor to that layer, and right-click the mouse button. The soil name will show.

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. 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. The second way of assigning material properties is very similar to the method discussed in the previous section. The difference is that instead of using the buttons in the material bar, the button in the *Material* page is used, as shown in Figure 22. In the *Material* page, choose the material to be assigned first; click the *Assign Material* button. Then move the cursor to the material to be assigned, and click again.

MODELING

A Visual Slope model can be called a cross section or a profile. For consistency, it is called a model in this manual. To assist in modeling, Visual Slope provides many ways, such as the drawing method, AutoCAD DXF file import method, tracing method, and generating method, depending on the availability of data, user's preference, and the type of analysis. The following sections describe the first three methods in detail; the fourth method will be addressed in the analysis section because it is directly related to the type of analysis. However, no matter which method is used to create a model, the material properties to be used in the analysis must be set up first.

DRAWING METHOD

A Visual Slope model is composed of lines, such as a soil line, a water line, etc. To draw a line, a particular line type must be chosen first by clicking the corresponding button. The coordinates of the ends of this line must also be known. Coordinates on the screen can be specified by referencing the coordinate rulers, or through direct data input or coordinate edits. The following sections provide the details.

Start Line Drawing

After a line button is clicked, the program is in the drawing mode, and the cursor becomes a crosshair. 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 12. Left-click the mouse button to begin a line. Move the cursor to the end point of the line, and left-click the mouse button again. The line is completed.

Repeat the above procedure for the rest of line 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.

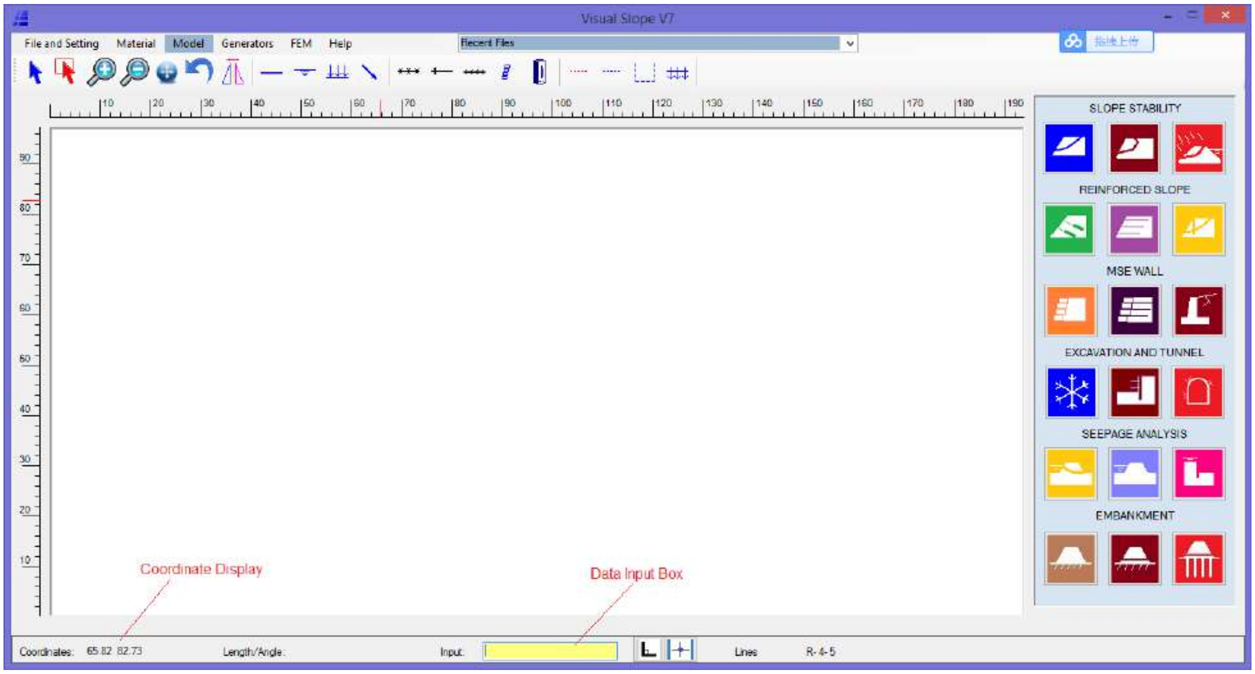


Figure 12: Visual Slope Main Page

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 in yellow color. 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 12) 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 12) and press the *Enter* key on the keyboard. The length and the angle must be separated by a space.

Coordinate Editing Method

The most efficient way to draw a model is to first sketch the model without considering the actual coordinates too much. After the sketch is completed, the user then edits each point of the line to correct its coordinates. To edit a point, first click the *Select* button, and then move the cursor to the point to edit and click that point. The coordinate edit page will appear, from where the coordinates of the point can be changed to the correct ones.

Rules of Line Drawing

1. A line must be drawn from left to right; otherwise Visual Slope will ask the user to redraw. No vertical soil lines are allowed. A vertical soil line should be converted to a near vertical line.

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 latter will be automatically broken into two lines, so that lines can be 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.

4. Each soil scatter must be in a closed polygon formed with soil lines. The left, right, and bottom boards of the model must be surrounded by boundary lines to form closed polygons.

Tracking and Snap-On Feature

During the drawing process, a red box will appear around an existing point as the cursor is approaching that point to help the user snap onto that point.

Break Feature

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

Edit Line or Point

To edit or delete a line or point, click the *Select* button under the *Model* menu, and then move the cursor to the line or point to edit or delete, and click the line or point. The *Line Edit Dialog* page (Figure 13) or *Point Edit* page (Figure 14) will appear. Select the line in the dialog box, and click the *Delete* button. The selected line will be deleted. The coordinates of the line can be edited in the *Line Edit Dialog Box*. However, one 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 edit a point, please refer the Coordinate Editing Method section.

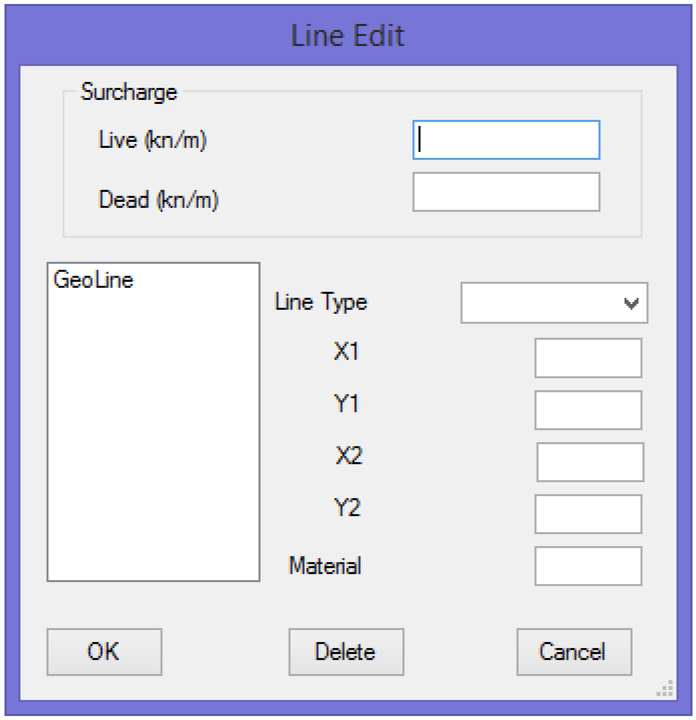


Figure 13: Line Edit Page

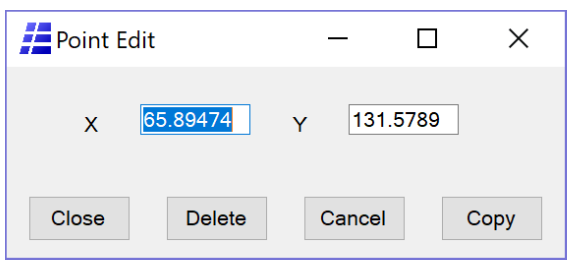


Figure 14: Point Edit Page

Group Selection

To select more than one line, the group selection feature can be used. The user first clicks the *Group Select* button and then drags a box over lines to select them and continues to select until all desired lines are selected. The user can then click the right mouse button to choose the next action, such as edit, delete, etc.

Surcharge Load

Surcharge loads can be added by using a drawing method similar to drawing other lines. Click the *Load* button under the *Model* menu. 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 load should be specified as “live” or “dead,” as shown in (Figure 13).

Rain Line

Rainfall can be added to the surface of a model, similar to a surcharge line. The rainfall property should be defined in the *Line Edit* page (Figure 13), including penetration depth and pore pressure.

OTHER FEATURES

To help users establish a model, Visual Slope provides many other features, such as *Zoom In*, *Zoom Out*, *Undo, Flip*, and *Move*.

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 on an area, click the *Zoom In* button, hold the left mouse button down, drag diagonally across the area to zoom 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 five 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.

Move

To move a cross section toward a certain direction, click the *Move* button, and then move the mouse in that direction while holding the left button of the mouse down.

Video Example

The following video example demonstrates how to set up material properties, create a model by using the drawing method, and use the functions of some key buttons.

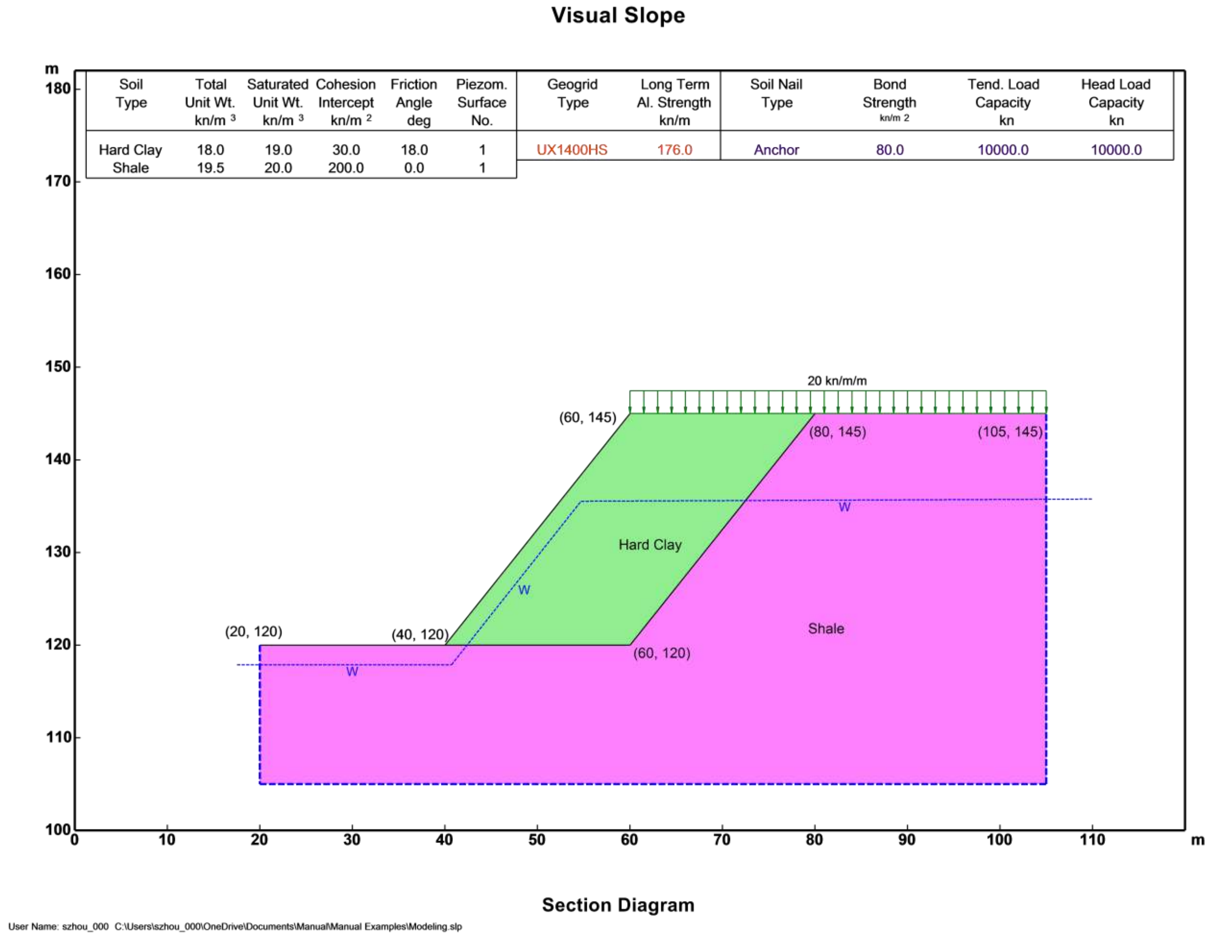


Figure 15: Drawing Method Example

AUTOCAD FILE IMPORT METHOD

Visual Slope is capable of converting an AutoCAD file into a Visual Slope model. The AutoCAD file to be converted must be saved as a DXF 2000 or earlier format file. The cross section in the AutoCAD file should consist of lines or polylines. Other types of lines will be ignored during AutoCAD-Visual Slope conversion. A conversion includes the following steps:

1. Import the DXF file.
2. All lines coming from the DXF file are considered as soil lines. If there are lines that are not supposed to be soil lines, they should be converted to the correct line type manually.

Prior to conversion, the Auto-Connection feature should be turned off. After all the lines have been converted, that feature should be turned back on.

3. If there are points that are supposed to be connected but are not, they should be connected manually using the group selection button.
4. If the model is not on the correct scale, it should be re-scaled to the correct one.

Video Example

This video example presents the details of above process.

IMAGE TRACING METHOD

If there are no detailed coordinates of a cross section available or if slight coordinate errors in a model will not affect the analytical results significantly, the image tracing method can be used to establish a Visual Slope model. Image tracing method includes the following steps:

1. Import a cross-section image as background.
2. Trace the image with corresponding lines.
3. Set up boundary lines.
4. Calibrate the model to convert the model to scale.
5. Assign material properties to the model.

Video Example

This video example presents the procedure of image tracing method.

ANALYSES

SLOPE STABILITY ANALYSES

The Visual Slope slope stability module is developed based on the widely accepted limit equilibrium theory. The finite element method (FEM) has been included in Visual Slope V7. The limit equilibrium theory and FEM for slope stability analysis were discussed in detail by Y. Huang (1983). This manual will not repeat this information. The slope stability module can be used to perform a slope stability analysis for soil, rock, or mixed slopes, no matter how complicated the ground conditions are. Visual Slope provides several different analyzing methods:

1. Modified Bishop method
2. Janbu method
3. Spencer method
4. Morgenstern and Price method
5. Finite element method

Almost any failure surface, no matter how complex it is, can be analyzed with Visual Slope. Common failure surface shapes are:

1. Circular failure surface
2. Planar failure surface
3. User specified failure surface
4. Translational failure surface
5. Composite failure surface
6. Random failure surface

The following sections describe how to perform slope stability analyses for those failure surfaces.

Circular and Irregular Failure Surfaces

A circular failure analysis or an irregular failure analysis (Figure 16) is specified by an entry area defined by two coordinates (X1, the coordinate of the leftmost initiation point, and X2, the coordinate of the left termination point) and an exit area defined by two coordinates (X3, the coordinate of the rightmost initiation point, and X4, the coordinate of the right termination point). The user should also provide the number of failure surfaces to be generated.

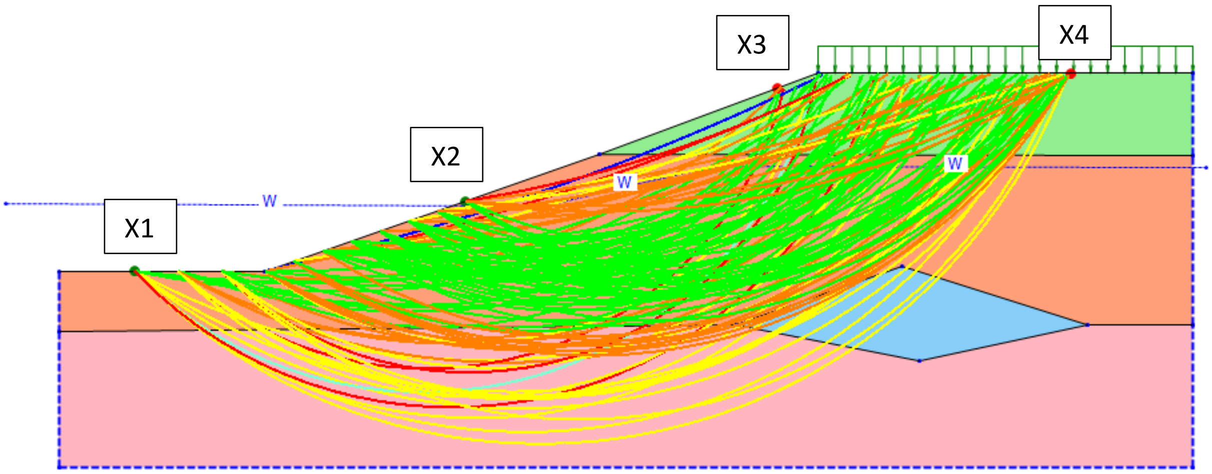


Figure 16: Circular Failure Analysis

To perform circular failure or irregular failure analysis, click the slope stability analysis button from the *Analysis* panel. The *Slope Stability* page will appear (Figure 17). 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. A left click moves the point vertically, while a right click moves the point horizontally. 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.

The number of failure surfaces should be an integer. It is recommended that this number be at least 500.

After the entry area and exit area are defined, click the *Analysis* button. After the analysis is completed, click the *Curve* buttons to see the failure surfaces. The video example may help users further understand the details of the procedure. The model used in the example is the same as that used in the [Modeling with Drawing Method](#).

Video Example

This [video example](#) presents the procedure of a circular failure analysis.

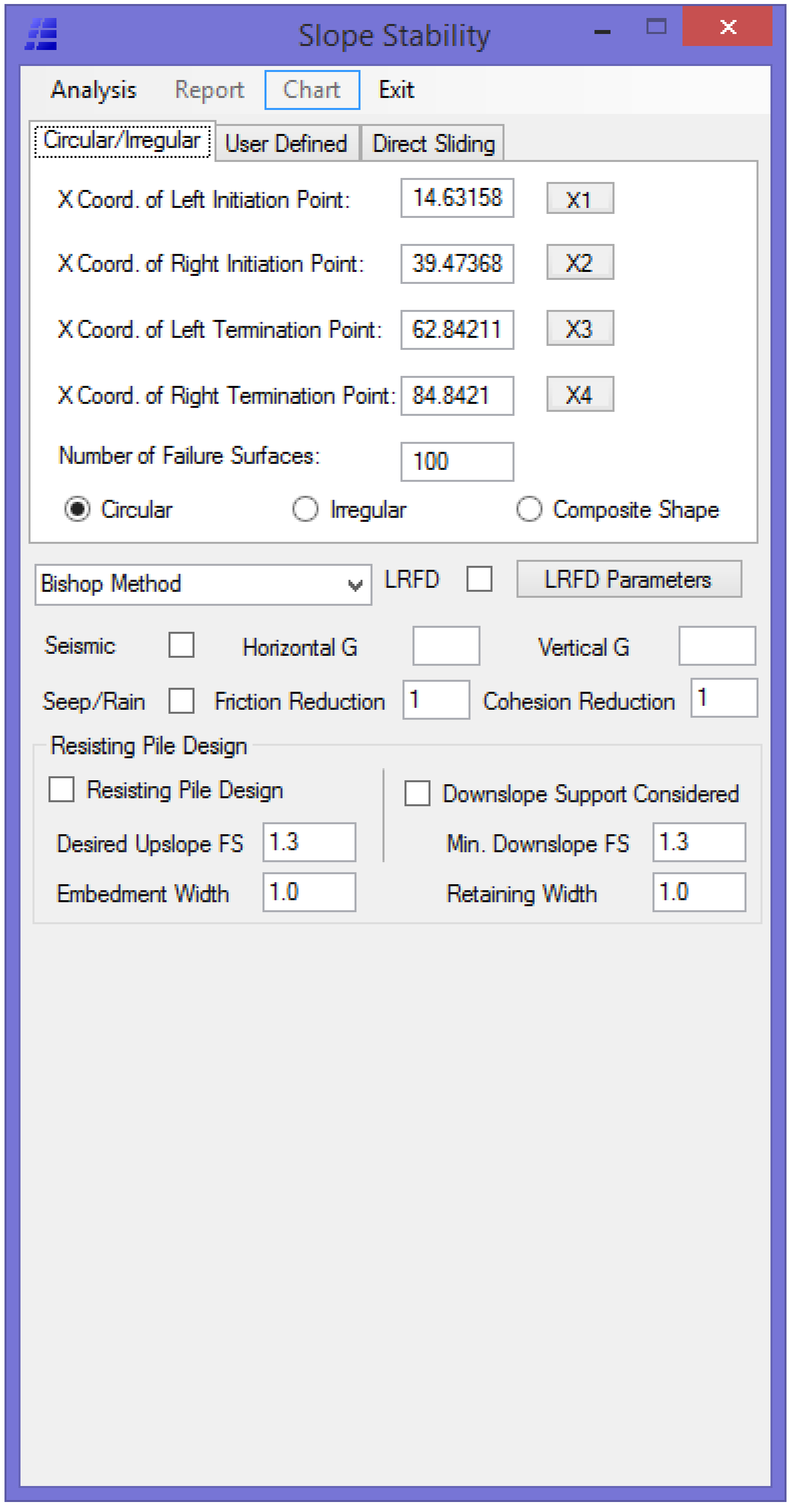


Figure 17: Set up Circular or Irregular Failure Analysis

User Defined Failure Surfaces

The user can specify his or her own 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 points with 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 red button will appear on the right, as shown in Figure 18. 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.

Repeating the above process, the user can draw a complete failure surface. 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. A user specified failure surface must consist of more than four lines.

After the potential failure surface is defined, specify the number of failure surfaces to 1, and then click the *Analysis* button. After the analysis is completed, click the *Curve* buttons to see the result of the user defined failure surfaces.

Video Example

This video example presents the procedure of a user defined failure analysis.

Translational Failure Surface Analysis

Visual Slope greatly simplifies the approach to perform a translational failure surface analysis by extending search area around a user defined failure surface. Therefore, the approach to perform a translational failure surface analysis is the same as that to perform a user defined failure surface analysis, except to set the number of failure surfaces to more than 10 and search width (how far from the original user defined failure surface) to a user desired value.

Video Example

This video example presents the procedure of a translational failure analysis.

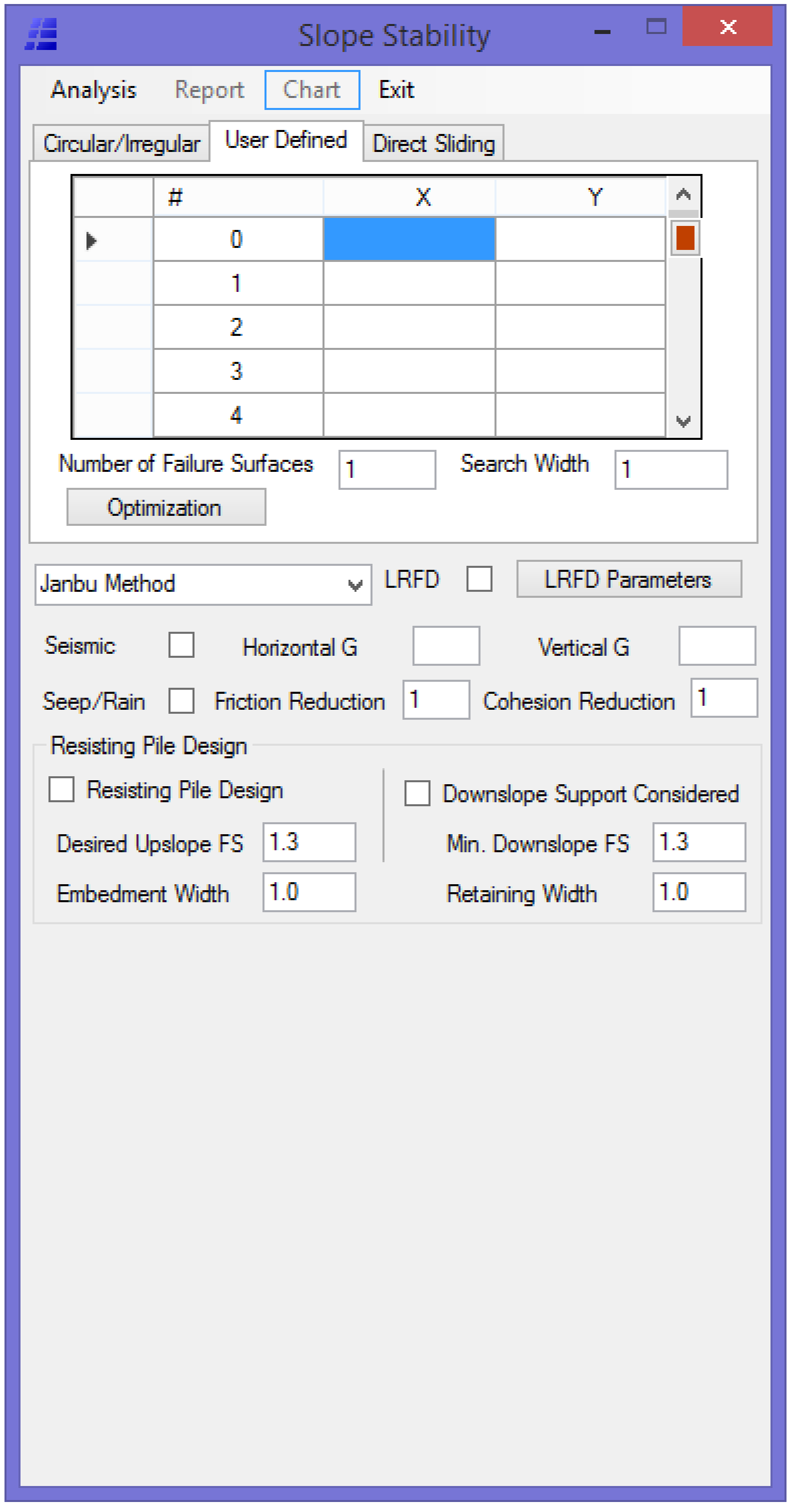


Figure 18: Draw Failure Surface

Composite Failure Surface Analysis

A composite failure surface is a combination of a user defined failure surface and a program generated circular failure surface, as shown in Figure 19, and is commonly used for a soil layer sliding over the bedrock interface, or a clay cap sliding over a geomembrane interface in a landfill project. The interface portion of a composite failure surface is defined by limit lines; the circular portion is similar to what has been described in the section Circular and Irregular Failure Surfaces. After a composite failure surface analysis is specified, choose the *Composite* shape option in Figure 17, and then click the *Analysis* button. After the analysis is completed, click the *Curve* buttons to see the result of the user defined failure surfaces.

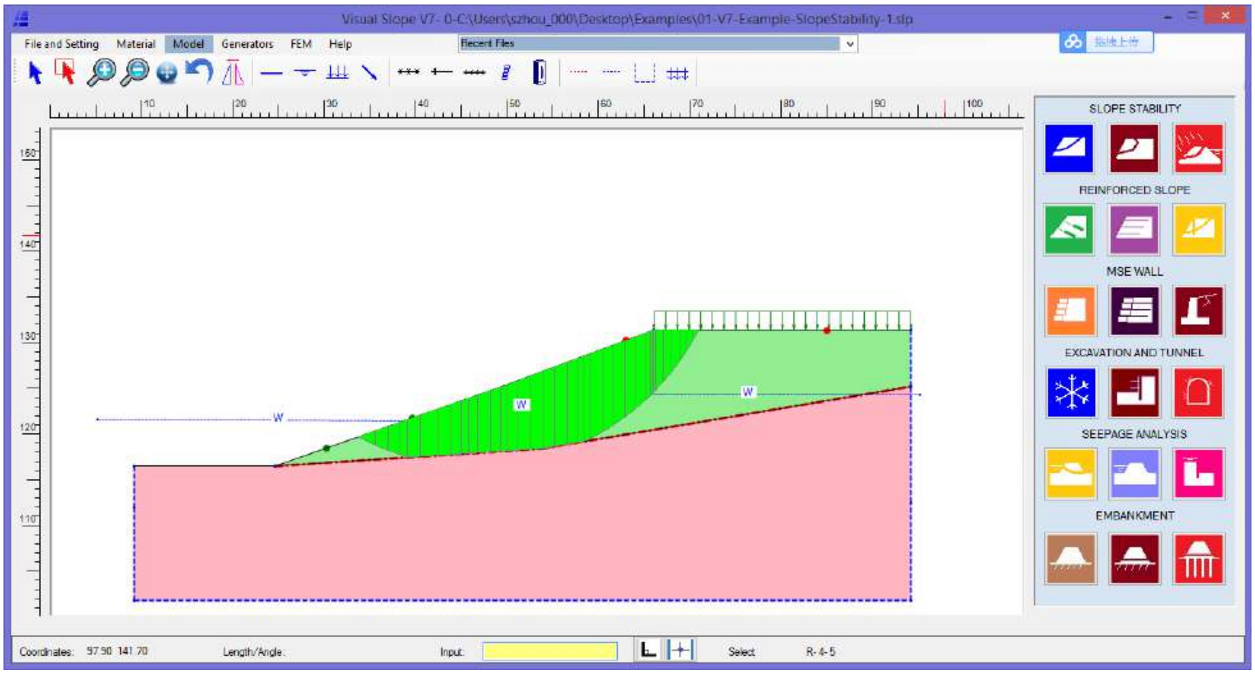


Figure 19: Composite Failure Surface

When the user draws limit lines, they should be drawn inside the weaker material; if it is a soil and rock interface, for example, the limit lines should be drawn slightly in the soil layer. If the strength of the interface is weaker than that of the original material, a strength reduction for the interface is allowed. First choose the limit line(s), and in the line *Edit* page (Figure 20), enter a reduction factor less than 1. Please watch the tutorial video for details.

Video Example

This video example presents the procedure of a composite failure analysis.

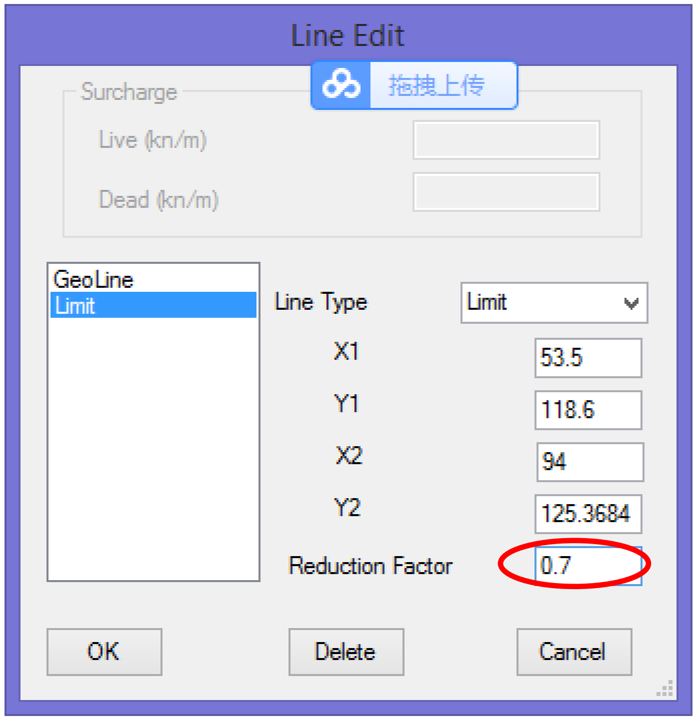


Figure 20: Setup Reduction Factor

Other Features

Seismic Analysis

To include seismic fact in a slope stability analysis, choose the *Seismic* option in Figure 17, and provide horizontal and vertical peak accelerations. The unit of accelerations is g; "0.25," for example, means "0.25g."

Strength Reduction

Commonly, the strength of a failure surface is lower than that of the original soils and rocks. Visual Slope allows the user to simulate this condition by reducing the strength of failure surfaces. To reduce the strength of failure surfaces, choose the strength reduction option, and provide the reduction factors (<1) to cohesion and friction, referring to Figure 17.

LRFD Option

The LRFD method has been revised in Visual Slope V7, according to the latest FHWA recommendation. The FHWA recommends that instead of factoring in soil parameters, a reduction factor of 0.75 should be applied to the resisting force. Results from this new LRFD method will be more consistent with the ASD method. To perform an LRFD analysis, choose the LRFD option; the rest will be the same as an ASD analysis.

Report

After an analysis, both text and graphic reports can be generated. To produce a text report, click the *Report* butting of the *Slope Stability* page. A text report will include all the input information and the detailed analysis results. To generate a graphic report, choose the chart type from the graphic list. On the report preview page, the report can be printed or saved to a PDF, Word, or Excel file.

REINFORCED SLOPE DESIGN

If a soil fill slope is too steep, it may not meet the slope stability requirement. To increase slope stability, geosynthetic reinforcement 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 use Visual Slope to design a reinforced slope. Most modeling processes have been discussed in the previous sections. Therefore, they will not be repeated here.

Adding Geosynthetic Reinforcement to Slope

Drawing Method

To add geosynthetic reinforcement 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 properties to the geogrid.

Grid Array

The grid array method is easier than the drawing method to add geosynthetic reinforcement layers to a slope. The grid array method can add many geosynthetic reinforcement layers that have the same type and same length into a slope at one time. To use the grid array method, the user must set up the geosynthetic reinforcement properties first, using the *Material Setup* dialog box. After materials have been set up, the user can click the *Grid Array* button under the *Generators* menu. The *Grid Array* dialog box will appear (Figure 21). The user must select the geosynthetic reinforcement type from the pulldown list, and then provide the start elevation, end elevation, vertical spacing, and geogrid length in the corresponding data cells. Once the user clicks the *Close* button, geogrid layers will appear in the model.

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.

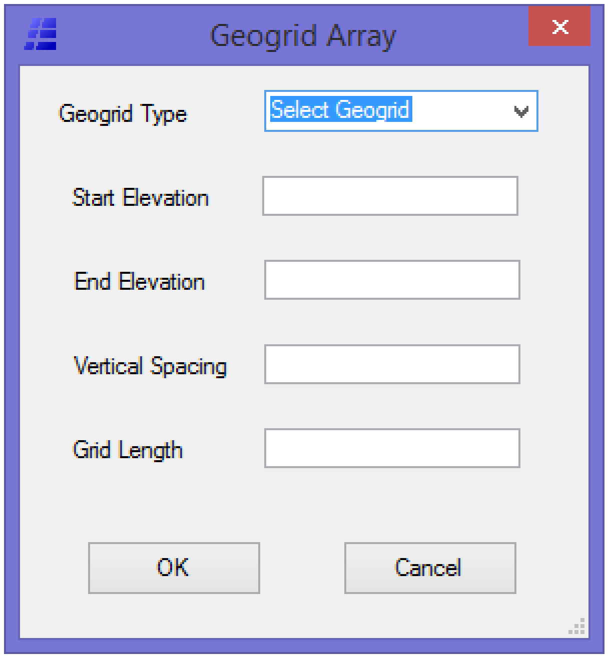


Figure 21: Geosynthetic Layer Generator

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 shown in Figure 22.

Direct Sliding Analysis

To perform an analysis for direct sliding over the geogrid in a reinforced slope design, the user should select the *Direct Sliding* tab in the *Slope Stability* page. After the analysis is completed, the user can see six of the most critical failure surfaces by clicking the *Failure Surface* buttons and can also see the most critical failure surface of each geogrid layer, as shown in Figure 23.

Reinforcement

After an analysis is completed, color spectra will display along geogrid layers (Figure 23). 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 the geogrid is providing, but how much reinforcement that geogrid will provide according to the color at a point, if a failure surface passes that specific point on the geogrid.

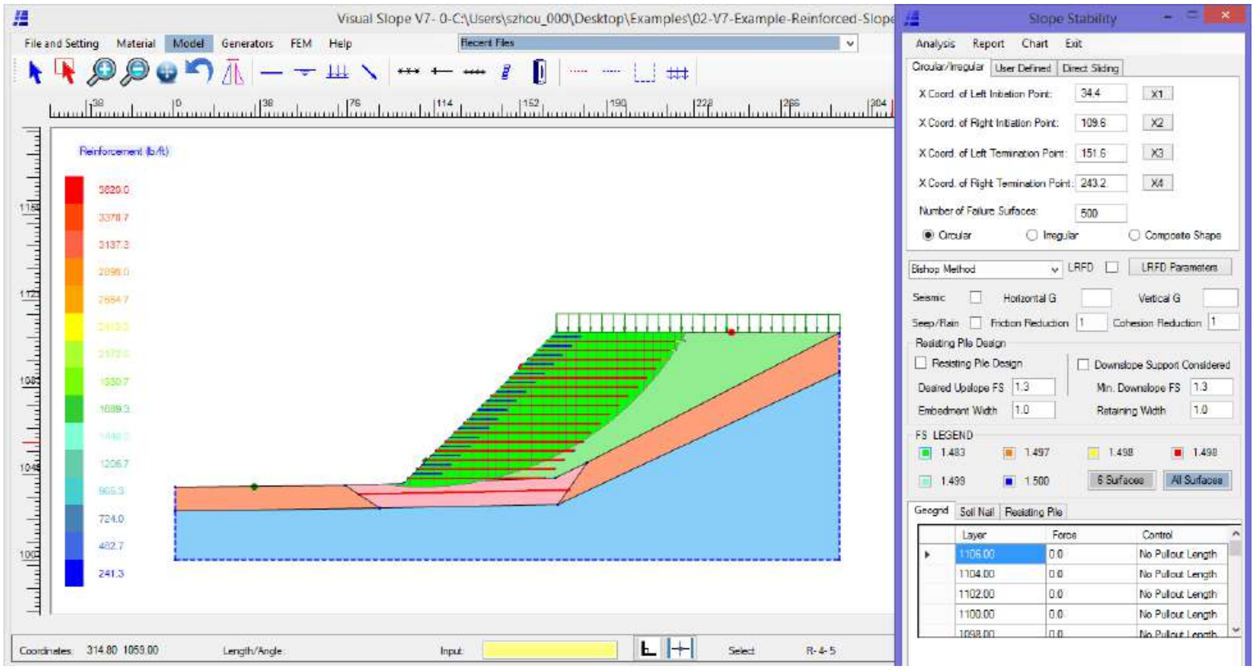


Figure 22: Circular Failure

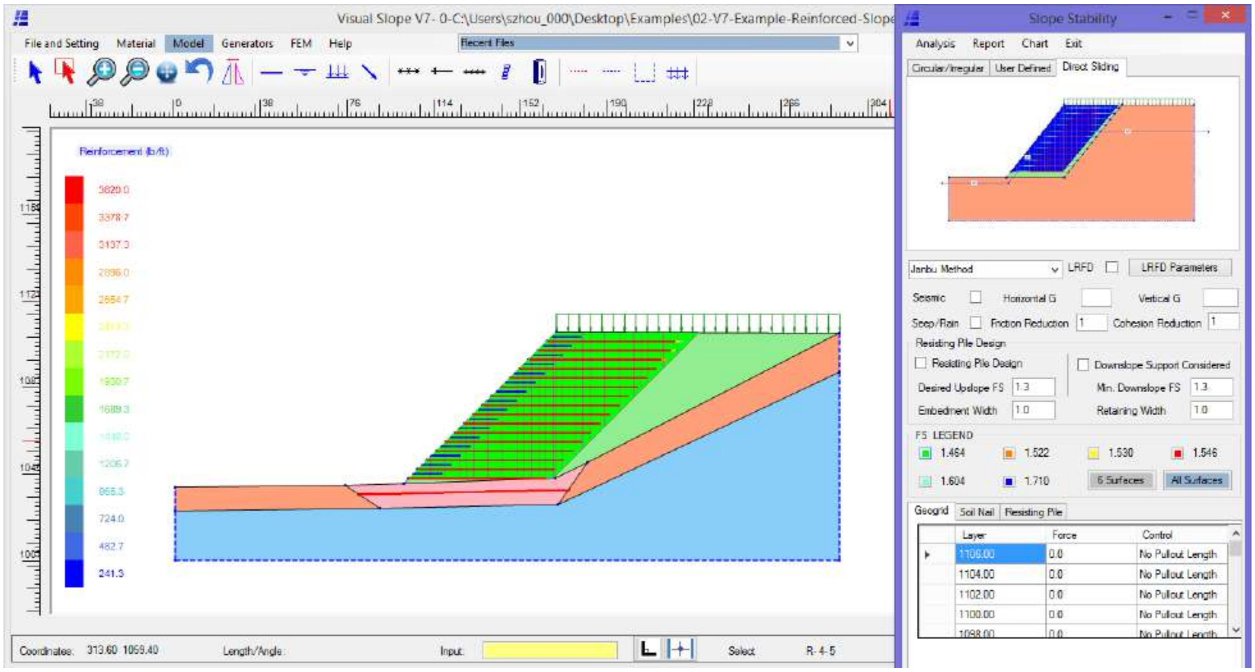


Figure 23: Direct Sliding

Settlement and Contact Pressure Evaluation

To estimate the foundation settlement and contact pressure between a reinforced slope and its foundation, the 2-D FEM module of Visual Slope can be used. Please reference the Finite Element Method section for details.

Video Example

This video example presents the procedure of reinforced slope design.

MSE WALL DESIGN

Both the NCMA method and the AASHTO method can be used in Visual Slope to perform a mechanically stabilized earth (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 2009 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 or concrete panels 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 straps. 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.

Modeling

Although an MSE wall model can be established by using the modeling methods described previously, to simplify the process, the *MSE Wall Model Generator* can be used to develop an MSE model. Under the *Generators menu,* click the *MSE Generator* button, and fill out the *MSE Generator* page (Figure 24).

Wall Dimensions

The wall dimensions include:

1. MSE wall (embedded) bottom coordinates
2. Wall height, including embedment
3. Front slope in degree
4. Back slope in degree
5. Back slope height (if back slope=0, this can be any number)
6. Reinforced zone depth (commonly equals the reinforcement length)

Soil and Wall Unit

Reinforced Soil (select from the list)

Retained Soil (select from the list)

Foundation Soil (select from the list)

Wall Unit (select from the list). If *Mixed Units* is chosen, wall units must be defined course by course from the bottom course up in the provided wall course table.

Reinforcement

If *Add Reinforcement* option is checked, the following information must be provided.

1. Start from which block (bottom)
2. Interval (blocks)
3. Reinforcement length
4. Reinforcement type

If the *Add Reinforcement* option is not checked, reinforcement can also be added to model by drawing method or array method described previously.

After filling out the *MSE Generator* page, click the *Close* button. The wall model will appear in the *Main* page (Figure 25).

If it is a tiered MSE wall, as in Figure 26, the *Wall Generator* page can be used repeatedly for each tier. After all tiers have been generated, use the drawing tools to add necessary lines and delete unwanted lines to finalize the model.

Analyzing

Internal and External Stability

After the wall profile is completed, to perform an external stability analysis for the MSE wall analysis, 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 39). On this page, the user should first choose the wall to be analyzed. If it is a tiered wall, all tiers of the wall will be listed. The user should analyze them one tier 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.

After the analysis is completed, click the *Report* button for a detailed report.

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. In contrast, the compound stability analysis should focus on the reinforced zone. Figure 28 shows a compound stability result.

2-D FEM Assistance

The 2-D FEM module can be used to assist in an MSE wall design for estimating the wall base settlement and the wall lateral deformation. For a tiered MSE wall, the 2-D FEM module can also be used to verify the vertical pressure from the upper walls to the lower walls.

Video Example

This video example presents the procedure of MSE wall design.

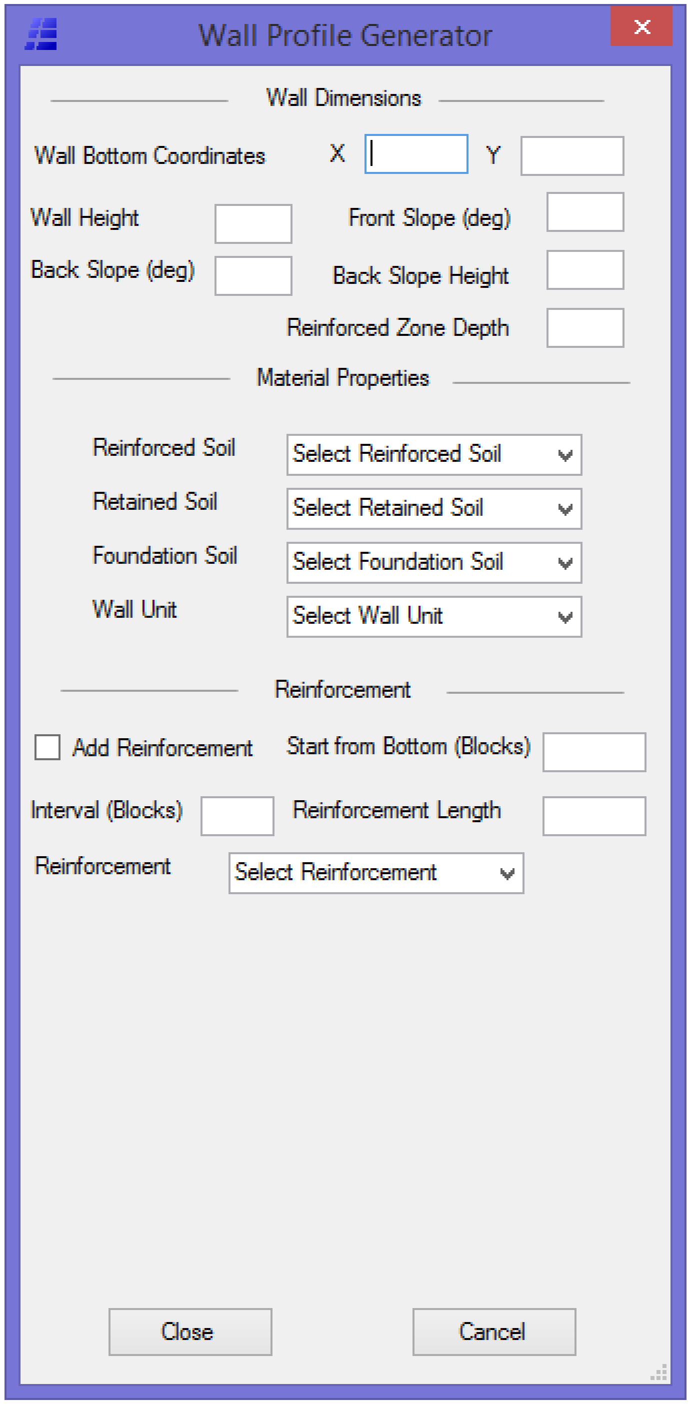


Figure 24: MSE Wall Generator

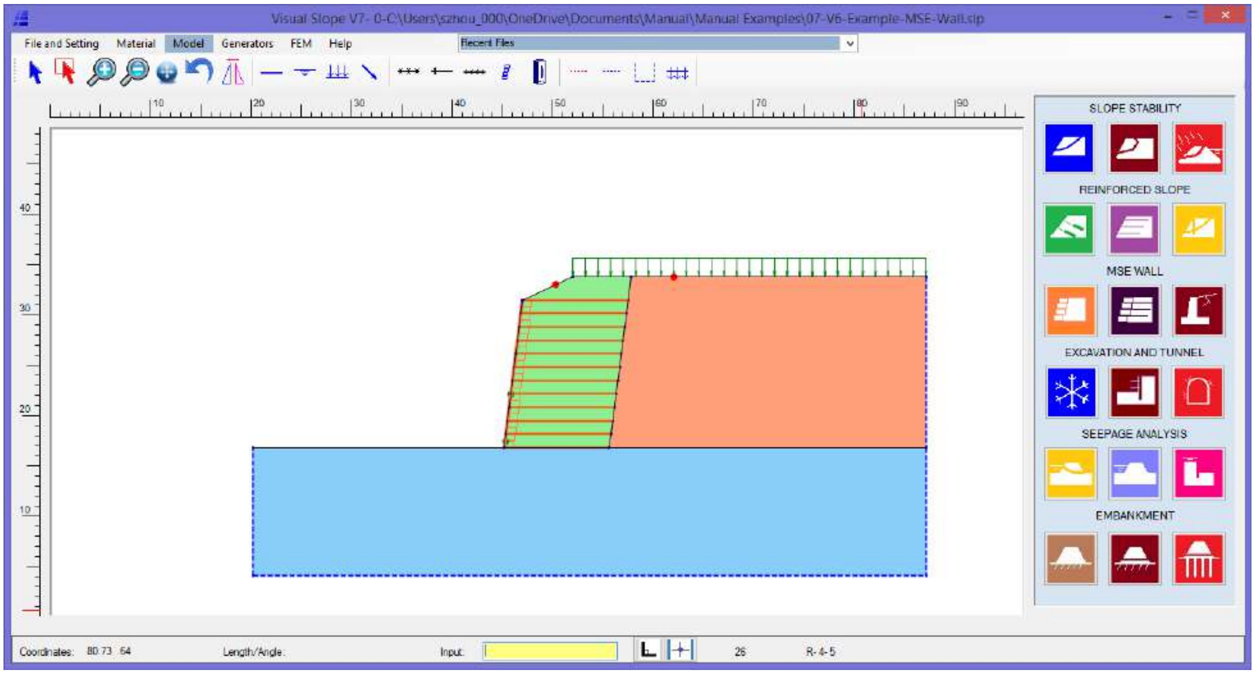


Figure 25: MSE Wall Model

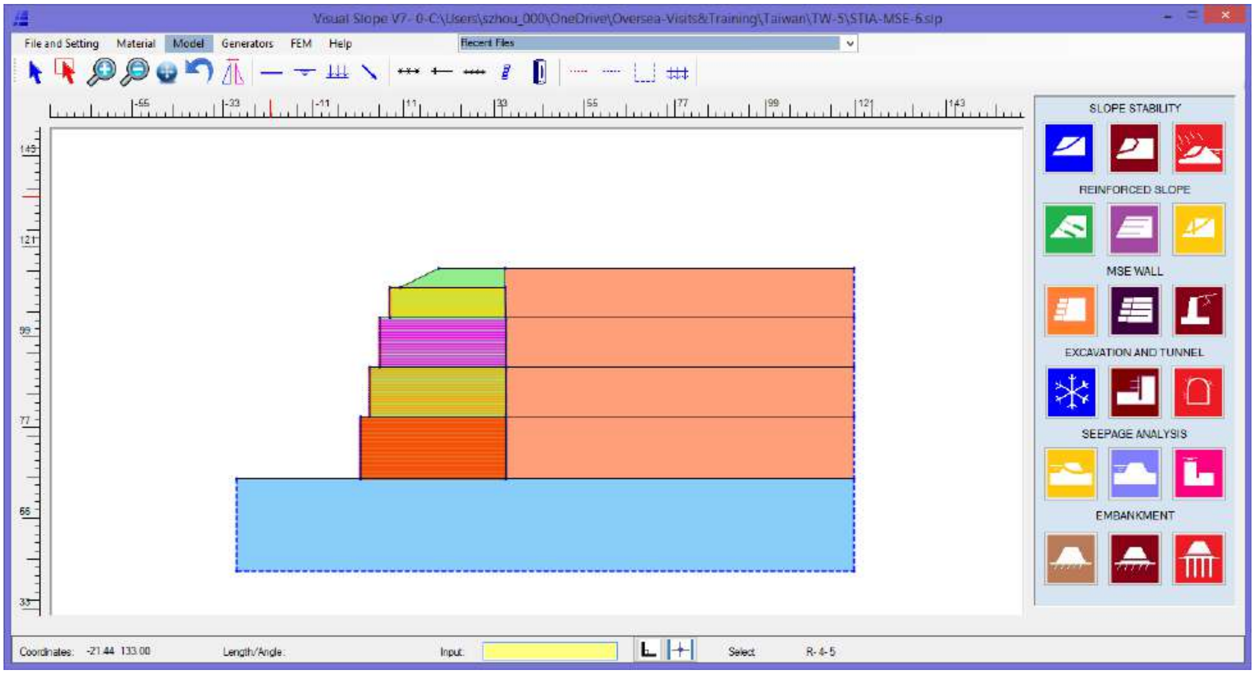


Figure 26: Tiered MSE Wall Model

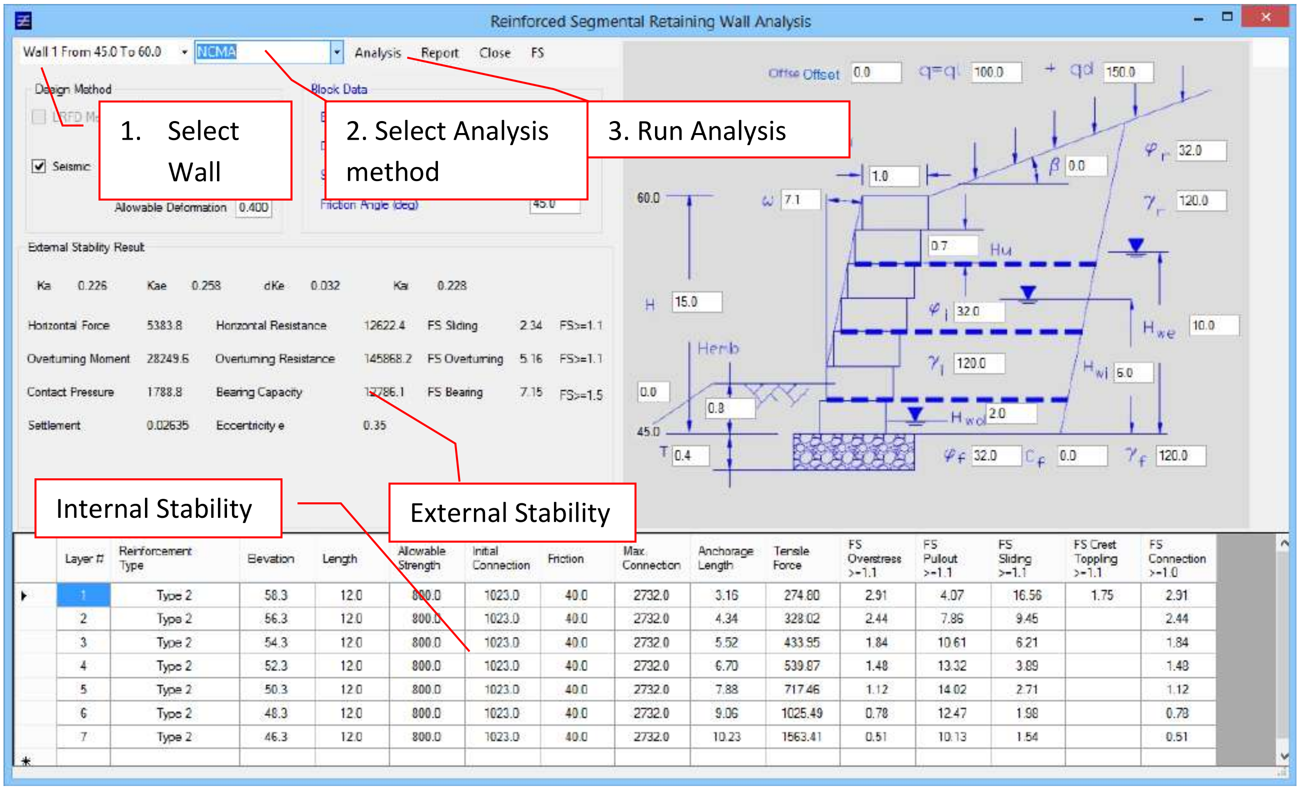


Figure 27: Internal and External Stabilities

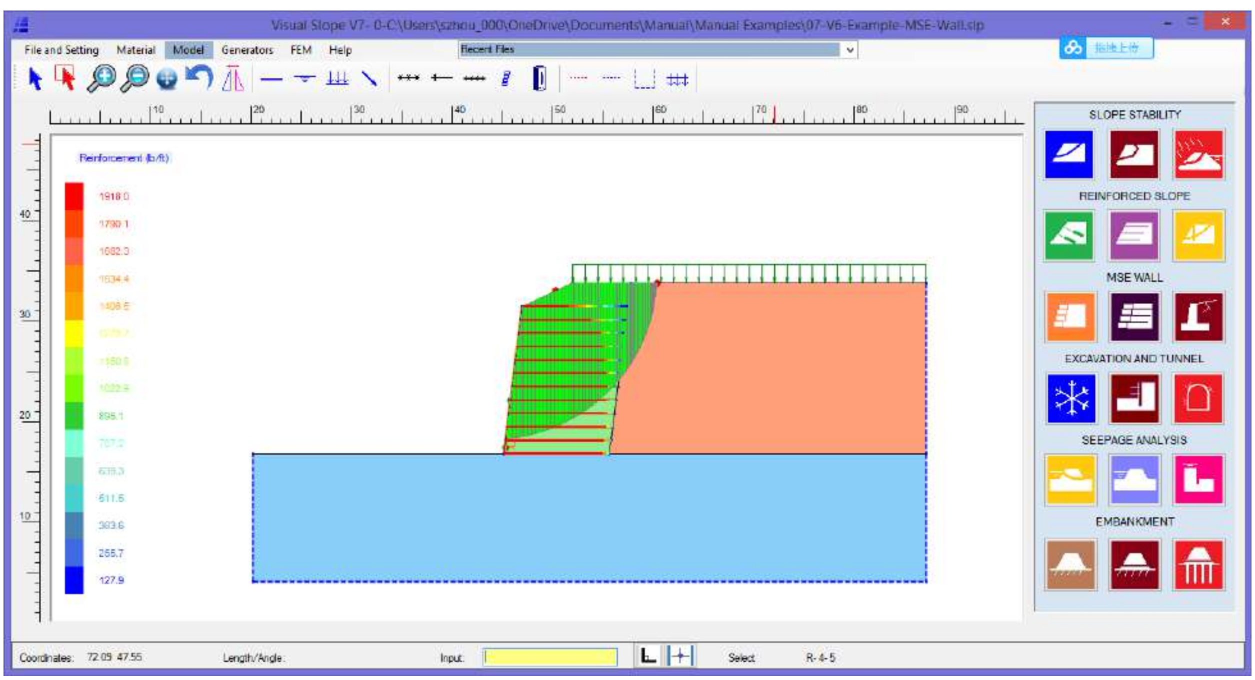


Figure 28: Compound Stability Analysis

GRAVITY RETAINING WALL DESIGN

Being widely used in geotechnical engineering practice, a gravity retaining wall is a type of retaining wall that uses its mass to buttress instable soil. To successfully retain instable soil, a gravity retaining wall must 1) resist the horizontal movement against the horizontal pressure from the retained soil or rock by using the friction between the retaining wall and foundation soil, 2) balance the overturning moment from the retained soil with the mass of the retaining wall, and 3) meet the foundation soil bearing capacity requirement.

The Visual Slope gravity retaining wall design module provides a straightforward method to design a gravity retaining wall. The following sections describe the procedure of design.

Material Property Setup

Prior to modeling, the properties of the materials to be used in the design must be set up first. The materials should include retained soil, foundation soil, and retaining wall material. A retaining wall material can be considered as a special type of soil. The material properties used for gravity retaining wall design are the same as those used for slope stability analysis. For information on how to set up material properties, see the Material Properties section.

Modeling and Analyzing

To set up a model for gravity retaining wall design, click the *Gravity Retaining* button in the *Analysis* panel, and the *Gravity Retaining Wall* page appears, as shown in Figure 29.

Select Material

First select materials for retaining soil, foundation soil, and the retaining wall itself.

Dimension Setup

The dimensions listed in Figure 29 can be varied to simulate retaining walls of different shapes.

Analysis

After materials have been selected and dimensions have been defined, an analysis can be performed. The result will appear in Figure 29.

Generate Profile

Once the analysis is completed, click the *Generate Profile* button, and the gravity retaining wall will appear in the Visual Slope *Main* page as shown in Figure 30. The model in the *Main* page can be used for slope stability analyses.

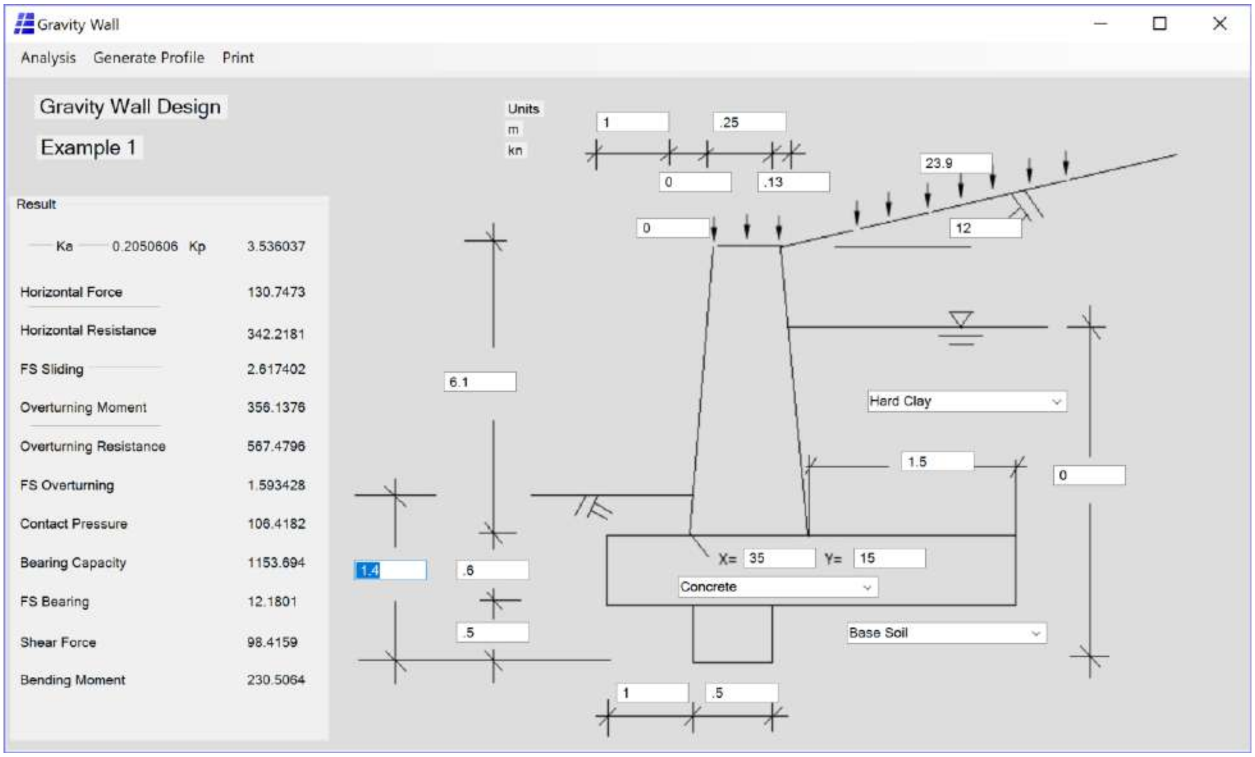


Figure 29: Gravity Retaining Wall Page

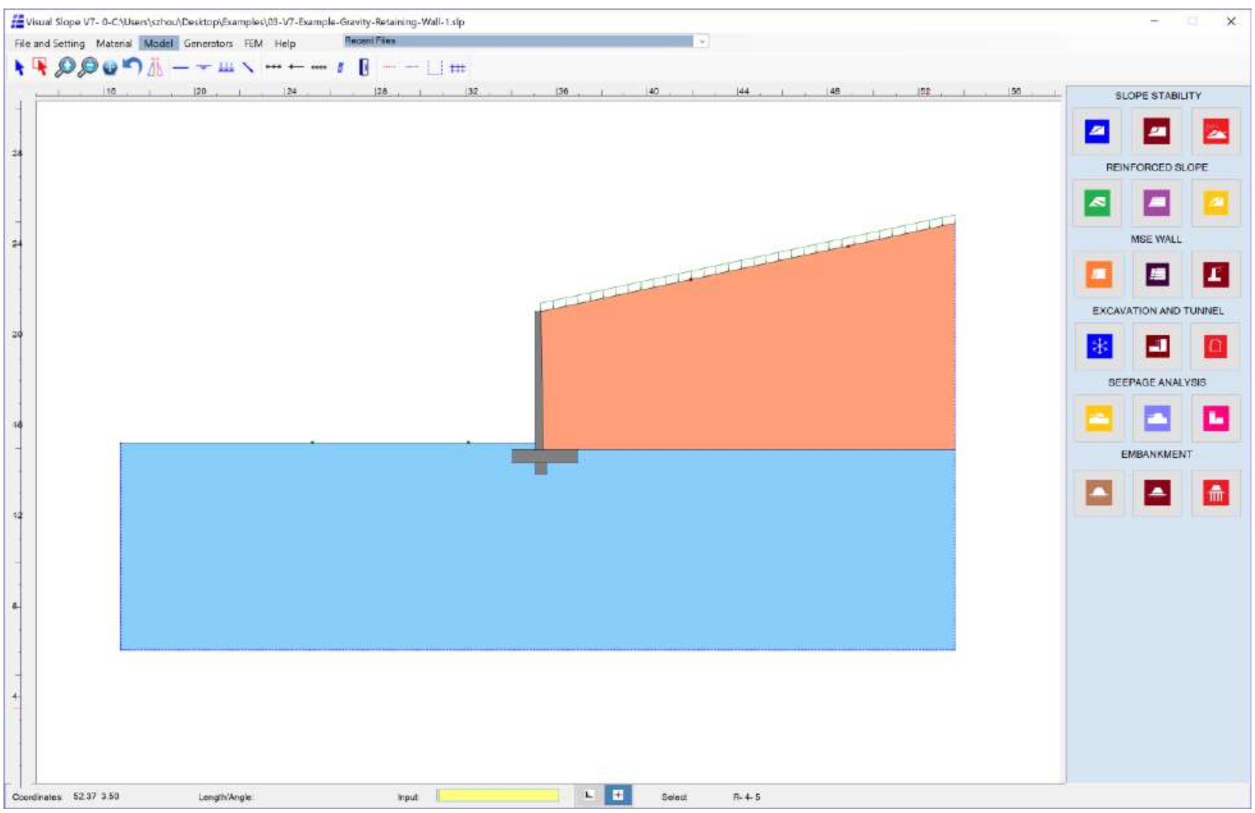


Figure 30: Model in Main Page

Video Example

This video example presents the procedure of gravity retaining wall design.

SOIL/ROCK NAILING DESIGN

If a soil or rock cut is not stable, nails can be used to stabilize the cut. 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. Visual Slope is, in general, in accordance with *AASHTO LRFD Bridge Design Specifications, 7th Edition, Soil Nail Walls Reference Manual*. The following sections describe how to use Visual Slope for soil/rock nailing design.

Adding Nails to Slope

Drawing Method

To add nails to a model, the user can simply draw nails one by one, similar to drawing soil lines (Figure 31). 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 drawing nails.

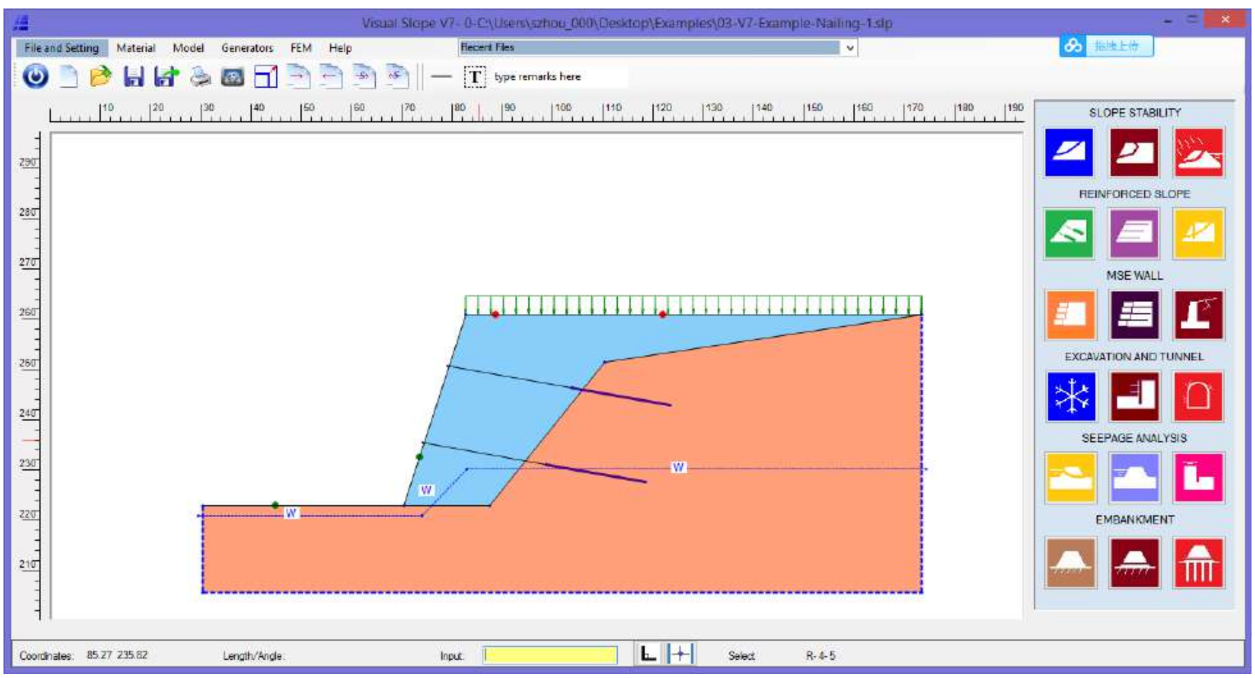


Figure 31: Adding Nails

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 Set Up* dialog box. After material is set up, the user can click the *Nail Array* button on the *Nail Array* button under the *Generators*. The *Grid Array* dialog box will appear (Figure 32). The user must select the nail type from the pulldown list and then provide the start elevation, end elevation, vertical spacing, angle of inclination, unbounded length, and bonded length for the corresponding data boxes. Once the user clicks the *Close* button, nails will appear on the model.

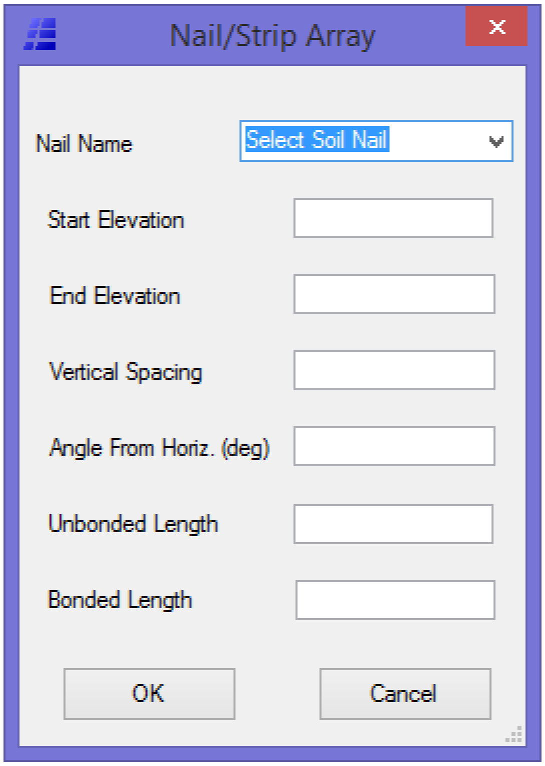


Figure 32: Nail Array

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 spectra will display along soil nails (Figure 33). 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 soil nail do not represent the actual reinforcement that the soil nail is providing, but how much reinforcement that soil nail will provide according to the color at a point, if a failure surface passes that specific point on the soil nail.

Video Example

This video example presents the procedure of soil/rock nail design.

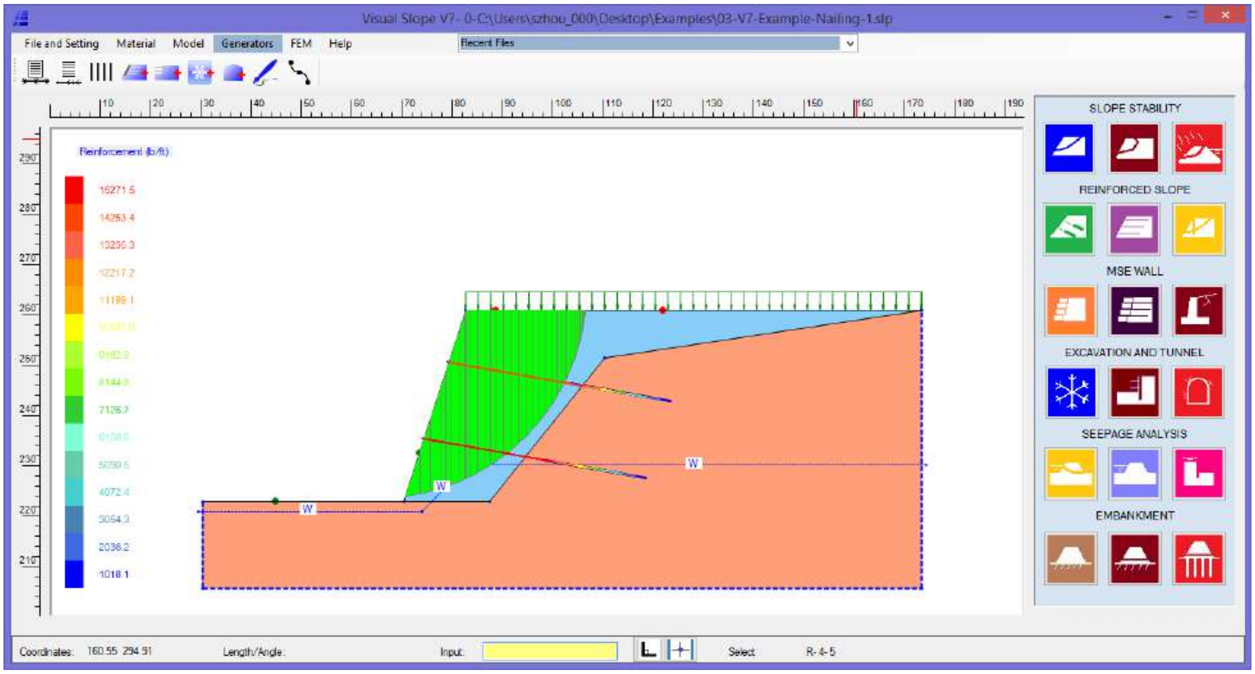


Figure 33: Soil/Rock Nail Design

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 multilevel braced, depending on the height of the wall. Associated with staged excavation, a shoring system commonly transfers from cantilevered to braced with the depth of excavation. Visual Slope is capable of simulating those conditions. In addition, the Visual Slope shoring design model and 2-D FEM module can work together, which allows users to analyze not only the internal forces and deflection of retaining walls, but also the ground deformation. Similar to that of designing an MSE wall, the process of designing a shoring system includes the following three steps:

1. Setting up material properties

2. Establishing model and staged excavation

3. Performing analysis

Setting up material properties has been described in the *Set Up Material Properties* section. The following sections describe the last two steps.

Model and Staged Excavation Setup

In a shoring model setup, the user must use 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 under the *Generators* menu. The *Staged Excavation Generator* page will appear (Figure 34). This page contains several sections that will be described in detail below.

Wall and Back Slope Coordinates

First the user must provide the wall coordinates, including wall top X and Y, wall tip Y, and back slope data, as shown in the diagram in the *Generator* page. If there is no back slope, the slope angle should be set to zero, and the back-slope value can be any number.

Wall Sections

Visual Slope allows the wall section to vary with the depth. Referring the wall section input table at the top-right corner of Figure 34, the user must set the wall sections from the top to bottom with the elevations at each depth, where the wall section changes in column 1, and the corresponding the section properties in column 2. The first elevation must be the top of the wall elevation. A section property will be used between two elevations. For example, in the wall section input table of Figure 34, the wall top elevation is 129 (in column 1, row 1); the wall section property is shoring-1 (in column 2, row 1) that will be used between elevations 129 and 120, where the wall section changes to shoring-2. The wall section shoring-2 will be used till the next specified elevation if provided. If no elevation is specified below 120, shoring-2 will be used from element 120 to the tip of the wall.

Ground Properties

Similar to the wall sections, the user must provide the ground conditions layer by layer from the top to bottom in the ground condition input table in Figure 34. The first column is for elevations, and the second column is for ground properties. The ground property will be used between the elevation next to it and the elevation below it. For example, sand 2 will be used between elevations 129 and 126. The last ground property will be used between the elevation next to it and the bottom of the model.

Bracing Properties

The user must provide bracing properties layer by layer from the top to bottom. The bracing properties consist of the elevation, stiffness, magnitude of preload, and horizontal spacing. If bracings are tiebacks, the tieback option must be checked, and the tieback angle should also be provided. However, if tieback lengths or angles vary, tiebacks can be modified with the drawing method prior to analysis.

Excavation Sequence

The user must provide the excavation sequence in a top-down order. The excavation sequence consists of elevations, water tables in front of the wall, water tables behind the wall, and surcharges. An excavation level should be repeated; if there are more than one, events will occur at that level. For instance, once the ground is excavated to elevation 122, a bracing at elevation 127 will first be preloaded and then locked. In the excavation table, the user must provide two excavation levels at elevation 122, one for preloading and the other for locking.

Stage Setup Button

After all required data of a shoring system have been provided, the user must click the *Stage Setup* button to confirm and store the data prior to setting up staged excavation sequence.

Staged Excavation

In the staged excavation setting area of Figure 34, there are two bracing lists. Once an excavation level is selected from the pulldown *Select Stage* list, the bracing list on the left shows bracings available for installation, and the bracing list on the right shows the bracings that have been installed. To install new bracings, select the bracing listed on the left, and click the >> button. That bracing will move to the right. If a bracing is in a preload condition, double-click that bracing in the right list. If a bracing needs to be dissembled, select that bracing in the right list, and then click the << button.

View This Stage

To view the model of a particular stage, select that stage from the *Select Stage* list, and click the *View This Stage* button. The model will appear in the *Main* page.

Code Setting

Before performing a shoring analysis, the user must set up the design code. To set up the design code, click the *Shoring Analysis* button on the *Analysis* panel. Once the *Shoring Design* page appears (Figure 35), click the *Code Setting* Button. The *Shoring Design Setting* page will appear

(Figure 36). The design code includes Code Name, Wall-Soil Friction, Water Pressure type, Calculation Method, Load Combinations, Factors of Safety, and Seismic Angles. The code should be set up according to the local code or the current project specifications. Once the code is set up and saved, it can be used for other projects if the code applies. The user can preset six different codes for future use. The last one saved will be the current code. The user can continue to use the current code till it is changed.

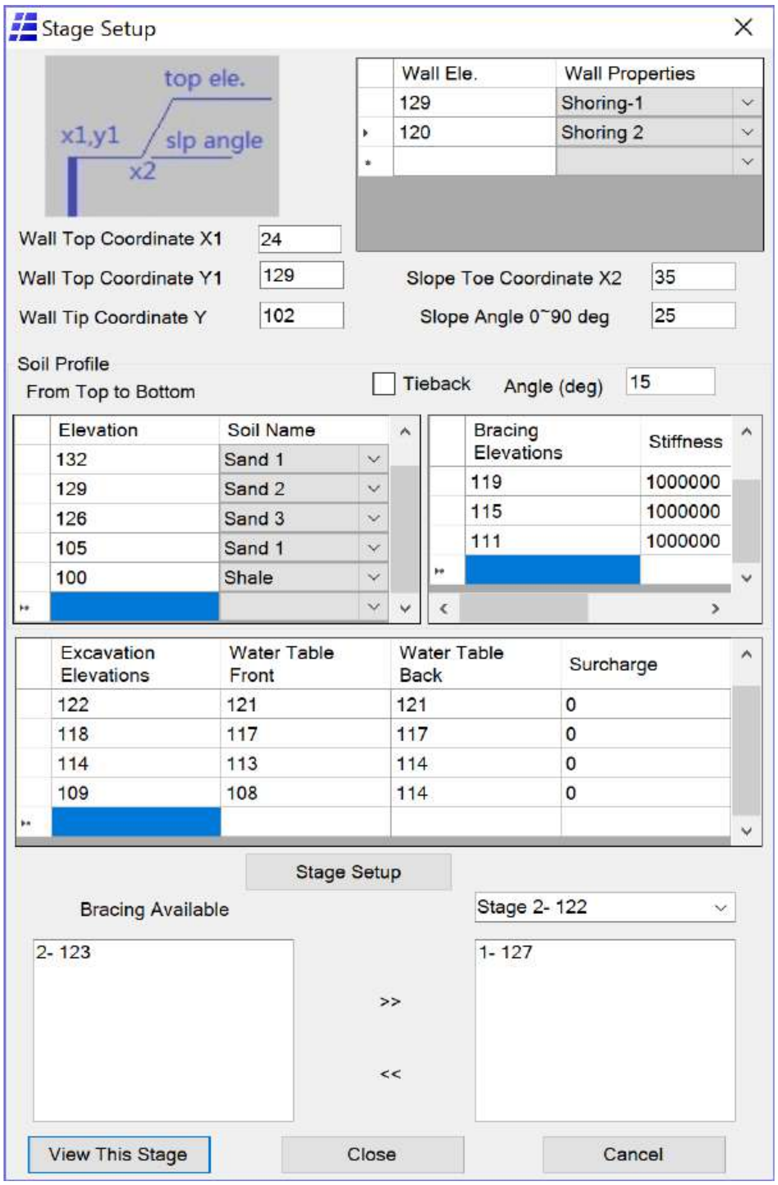


Figure 34: Staged Excavation Setup Page

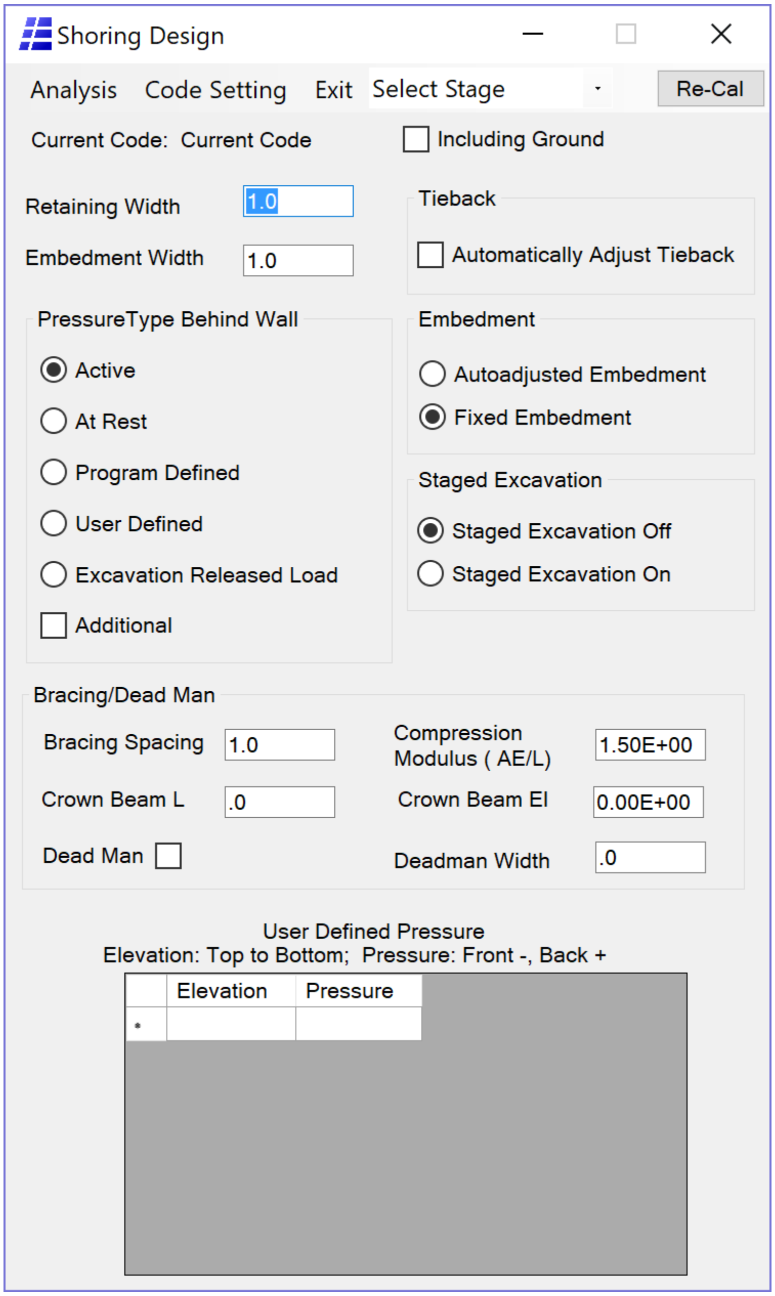


Figure 35: Shoring Design Page

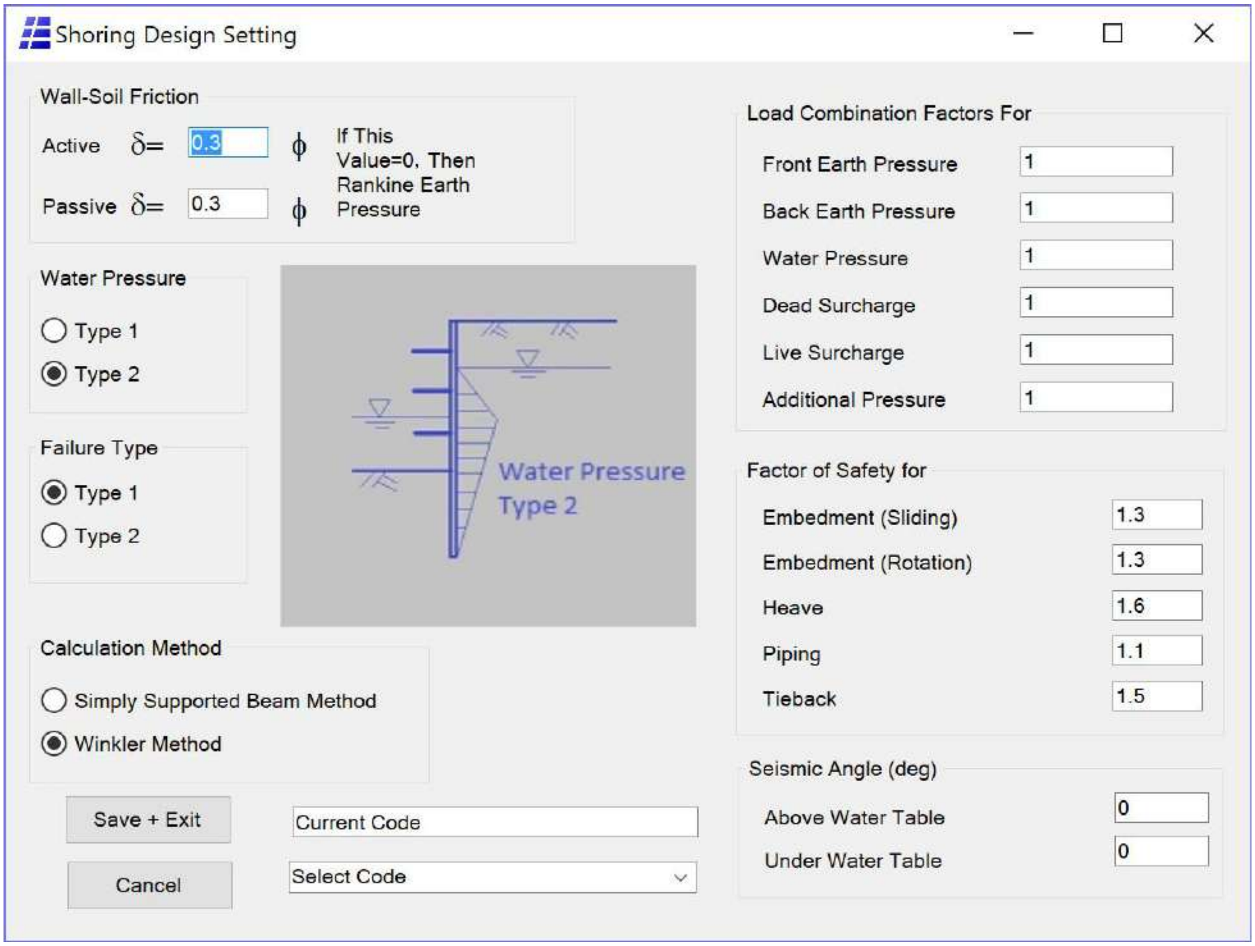


Figure 36: Code Setting Page

Analyzing

After completing the model and setting design code, the user can perform an analysis. To perform a shoring analysis, click the *Shoring Analysis* button on the *Analysis* panel. Visual Slope is able to detect what type of shoring system the user is working on based on the provided model. The following sections describe the contents in the shoring design page (Figure 35) and how to perform analyses considering or not considering the staged excavation fact, including or not including the ground movement evaluation.

<u>Retaining Width and Embedment Width</u>

To maintain a shoring system stable, the pressures behind the wall and in front of the wall must be in an equilibrium condition. A portion of the pressures in front the wall is from earth and water pressures against the embedment of the wall. However, the embedment width of a retaining wall may not necessarily equal the width of the retaining width. For the soldier pile wall, as an example, with a pile diameter of 0.6 m and pile spacing of 3 m, the embedment width should be 0.6 m and retaining width 3 m. 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 both retaining width and embedment width.

Pressures Type behind Wall

Visual Slope can detect what type of bracing system there is. If there is a multibraced 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 textbooks. Therefore, only defined program and user defined pressures will be discussed in this manual.

Program Defined Pressure

The program defined earth pressure is based on:

$$P = 0.65\gamma H K_a$$

The shapes of the earth pressure are:

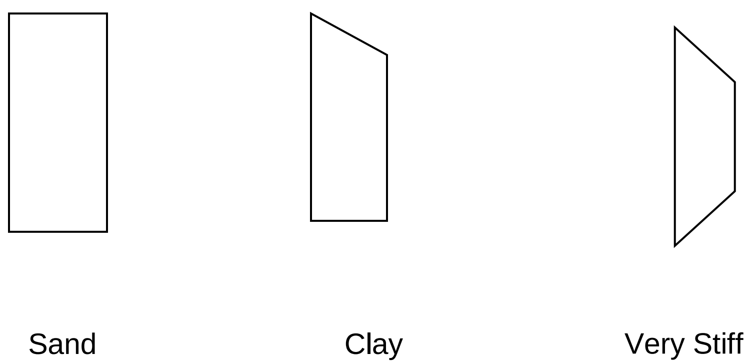


Figure 37: Program Defined Pressure

User Defined Pressure

The user can also define his or her own earth pressure pattern for a special case. The user can type elevations and earth pressure values into the input box (Figure 35). The pressure value is positive if the pressure is acting on the back of the wall, and it is negative if the pressure is acting on the front of the wall. The user defined pressure must be input in a top-down order. If the input data do 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 a calculation.

Additional Pressure

In many cases, besides common pressures, such as earth pressure, water pressure, and surcharge pressure, other pressures, such as wave pressure, should be considered in design in addition to the common pressures. 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 together with the user defined pressure option, as mentioned in the section above.

Staged Construction

Staged Construction is another option. If *Staged Construction* is set to *off*, the bracing will be considered to be added to the retaining wall at one time. However, if *Staged Construction* is set to *on*, the bracing will be considered to be added to the retaining wall layer by layer with excavation. Therefore, the staged construction calculation will result in a larger retaining wall deflection. Stage Construction will **not** affect design if a support calculation method is simply used in code setting. If the *Include Ground* option is chosen, *Staged Construction* will automatically be set to *on*.

Once *Staged Construction* is set to *on*, analyses must be performed stage by stage. The deflection and internal forces of the wall in the current stage are accumulations of the results of current analysis and the results of analyses for the previous stages.

Include Ground

If the *Include Ground* option is chosen, *Staged Construction* will automatically be set to *on*, and the 2-D FEM mode will be automatically invoked. For each stage, the analysis will consist of two portions. In the first portion, the retaining wall alone will be analyzed for the deflection and internal forces according to specified earth and water pressures, and bracing conditions. The first portion of the analysis is the same as the traditional method of retaining wall analysis. The second portion of the analysis only includes the ground but not the wall and bracings. However, the wall deflection calculated in the first portion will be used as the boundary condition along the wall in the 2-D FEM analysis for ground movement.

To exclude the ground movement and include ground stresses due to the initial conditions, such as the weight of the ground and water conditions, the initial condition that is prior to any excavation will always be calculated as stage 1.

Other Options

Automatically-Adjust-Tieback

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.

Automatically-Adjust-Embedment

If the *Automatically-Adjust-Embedment* option is selected, Visual Slope will adjust the embedment so that the retaining wall will not only satisfy the equilibrium condition, but will also have an adequate factor of safety against the bottom heave and piping.

Analyzing

After all options in Figure 35 have been correctly set, an analysis can be performed. To execute an analysis, choose the stage to be calculated from the *Stage List*, and click the *Analysis* button. Once the analysis is completed, the *Shoring Analysis Result* page will appear, as shown in Figure 38.

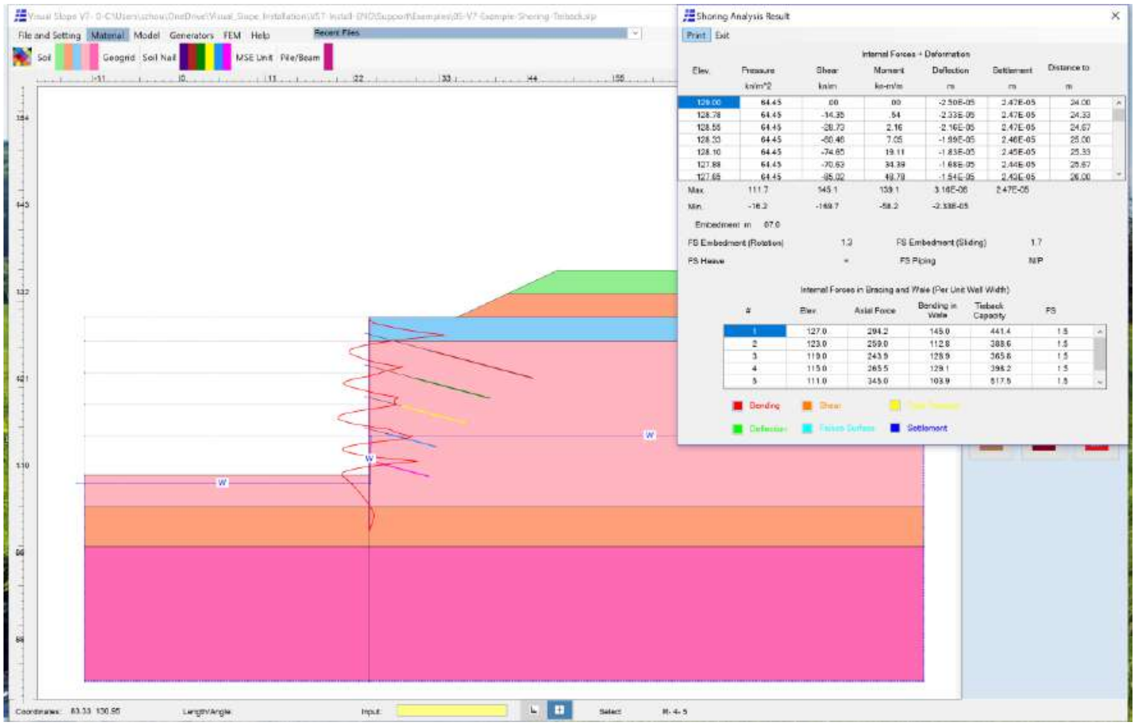


Figure 38: Bending Moment in Retaining Wall

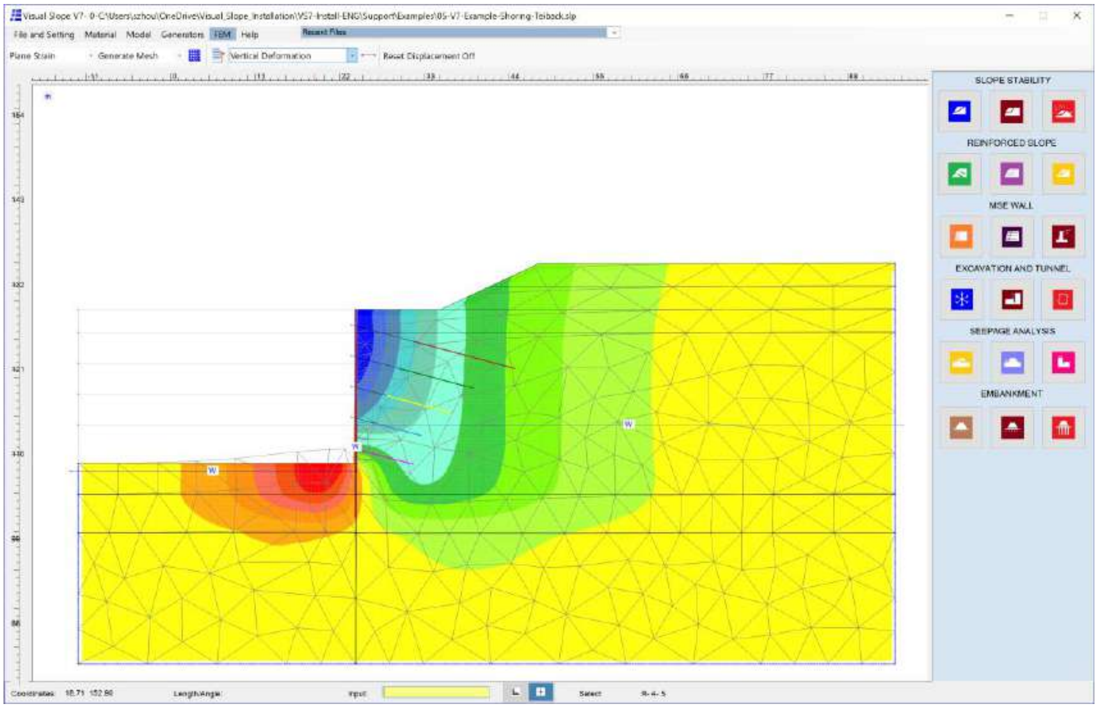


Figure 39: Ground Vertical Movement

RETAINING PILE DESIGN

One of the common approaches to remediate an instable slope is installing a row of retaining piles near the toe of the slope to buttress the upper slope. When designing a retaining pile, the engineer must know how much resisting force is required to support the upper slope. With this resisting force, the embedment and section of the pile can then be determined. Therefore, a retaining pile design involves two steps. The first step is to determine how much resisting force is required to stabilize the slope. The magnitude of the resisting force depends on: 1) how stable the slope is (the more stable the slope is, the less additional resisting force is required), and 2) the location of the pile. The second step is to determine the embedment depth, internal forces of the pile, and whether tiebacks are required. Hence, the design process is a combination of a slope stability analysis and a shoring system design. Visual Slope greatly simplifies the resisting pile design process. The user only needs to perform a slope stability analysis on the slope in question. If the slope does not meet the safety requirement, retaining piles are required, and the user locates the pile by drawing the pile into the slope model, as shown in Figure 40.

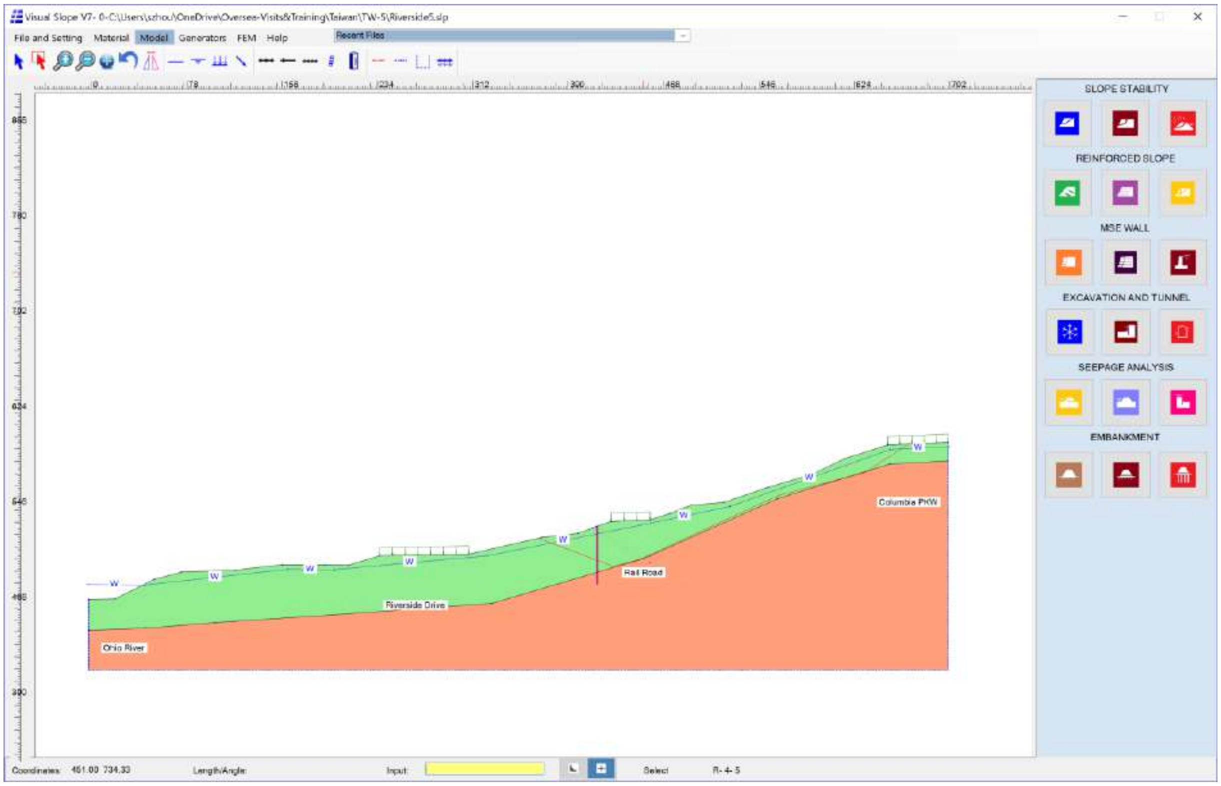


Figure 40: Retaining Pile Design Model

Once the pile has been drawn in, click the *Retaining Pile Design* button on the *Analysis Panel* of the *Main* page. The retaining pile design page (Figure 41), the same as slope stability analysis page (Figure 17), will appear. In that page, the *Resisting Pile Design* option must be checked; the factor of safety required for the upper slope must be specified regarding whether the support from the lower slope will be considered, and the factor of safety desired for the lower slope if the support from the lower slope is considered. The user must specify the embedment width and retaining width. Once those data are provided, run slope stability analysis again to accomplish the Step 1 analysis, in which the magnitude of the resisting force is determined. Then the Step 2 analysis can be performed by clicking the *Analysis* button in Figure 41. You can also add tiebacks in if needed.

Visual Slope will ask you if you want Visual Slope to automatically adjust the pile embedment and optimize the calculation. Once the analysis is completed, click the *Result* button to see the results, as shown in Figure 41.

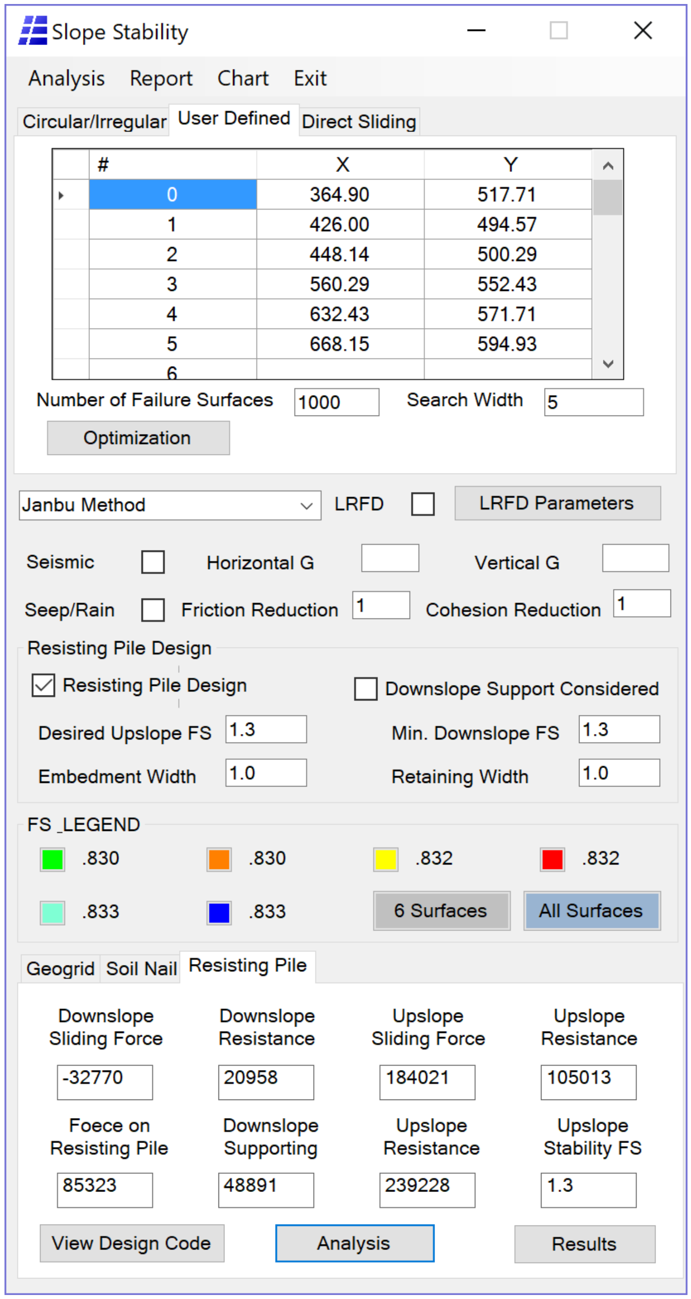


Figure 41: Retaining Pile Design

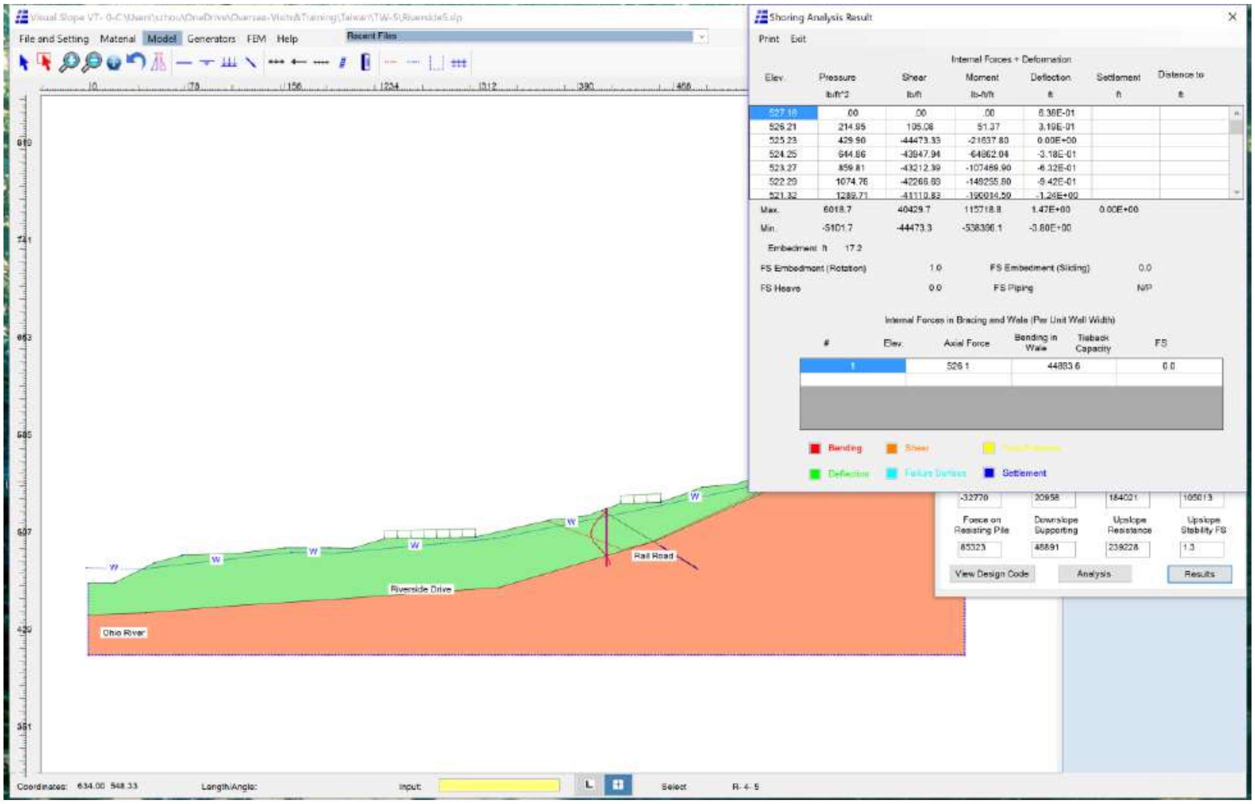


Figure 42: Retaining Pile Results

Video Example

This [video example] presents the procedure of retaining pile design.

SEEPAGE ANALYSIS

The Visual Slope seepage analysis module is capable performing static seepage analysis for a profile with isotropic, anisotropic, homogeneous, or nonhomogeneous soils. Drain and artesian aquifer can also be simulated. 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 by using the cutting cross-section feature. The following sections describe how to establish a model, input soil properties, set up boundary conditions, and simulate drain and artesian conditions.

Modeling

The method of establishing a model for a seepage analysis is the same as that used to establish a model for slope stability analysis. Therefore, please refer the Establishing Model section.

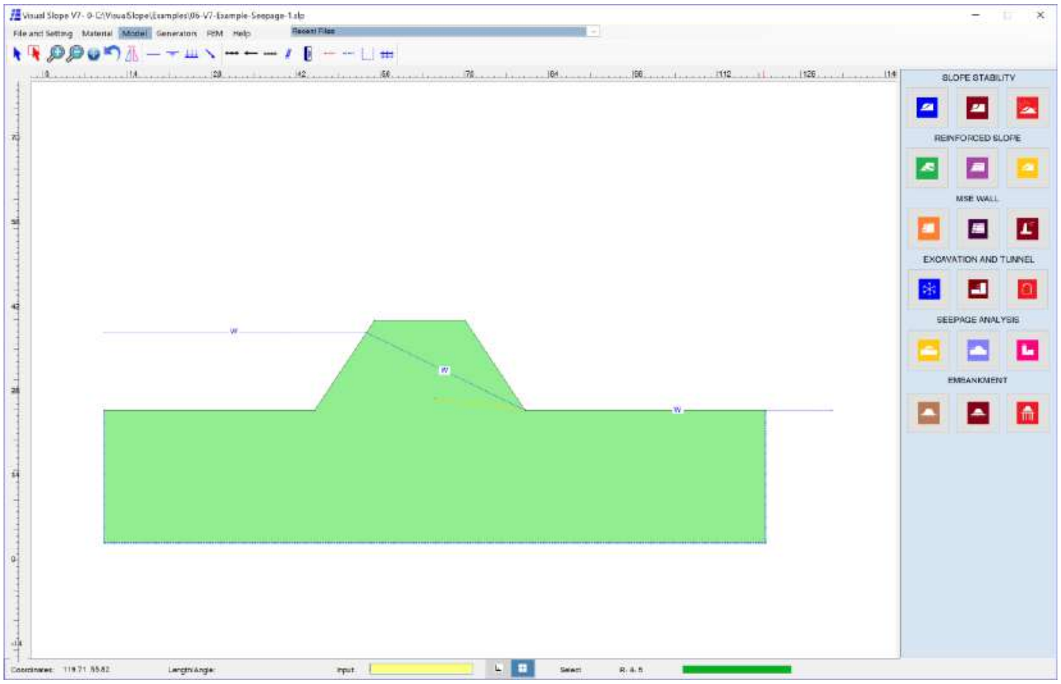


Figure 43: Typical Seepage Analysis Model

Input Soil Data

The soil data required for a seepage analysis are permeability (conductivity) in horizontal and vertical directions (*Kx* and *Ky*). Please 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 two types of boundary conditions: equipotential boundary and no-flow boundary.

An equipotential boundary is a surface on which the total head is fixed, like the surface in Figure 43. Some part of the surface is submerged under water, where the equipotential equals the water head, and some part of the surface is exposed, where the equipotential equals zero. To set up an equipotential boundary, water lines can be used, as shown in Figure 43.

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 43 should also be considered as no-flow boundaries, since they are far away, where the hydraulic gradient equals zero across the boundary. To establish a no-flow boundary, boundary lines can be used, as shown in Figure 43.

Drains

If a model contains drains, geosynthetic lines without material properties can be used to simulate the drains. Along a drain, water pressure is always zero.

Cutoff Walls

If a seepage analysis includes cutoff walls, like sheet piles, through which water cannot flow, limit lines can be used to simulate the cutoff walls.

Artesian

If there is an artesian soil layer, where water head is higher than that in other layers and remains constant, in a seepage analysis model, the water head can be set in the soil material input page, as shown in Figure 6. If a soil layer has a specified water head value, Visual Slope will consider it as an artesian layer.

Analyzing

After the model and boundary conditions are completed, seepage analysis can be performed. However, the user must choose between a confined seepage analysis and unconfined analysis.

<u>Confined Seepage Analysis</u>

If the boundaries of a model are completely defined either by boundary lines across which there is no water flow, or by equipotential lines like water tables, then a confined seepage analysis should be conducted. In a confined seepage analysis, the soils within the model are saturated.

<u>Unconfined Seepage Analysis</u>

If some boundaries of a seepage model are not defined by either boundary lines or equipotential lines, those boundaries are free boundaries that can interact with air. In that case, an unconfined seepage analysis should be performed, in which saturated zones and unsaturated zones will be defined through the analysis. The boundaries between saturated and unsaturated zones are called phreatic lines.

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 44.

To calculate flow rates and y-exit gradient, click the *Section* button, and then draw a section on the model. The flow rate and y-exit gradient of that section will appear, as shown in Figure 44.

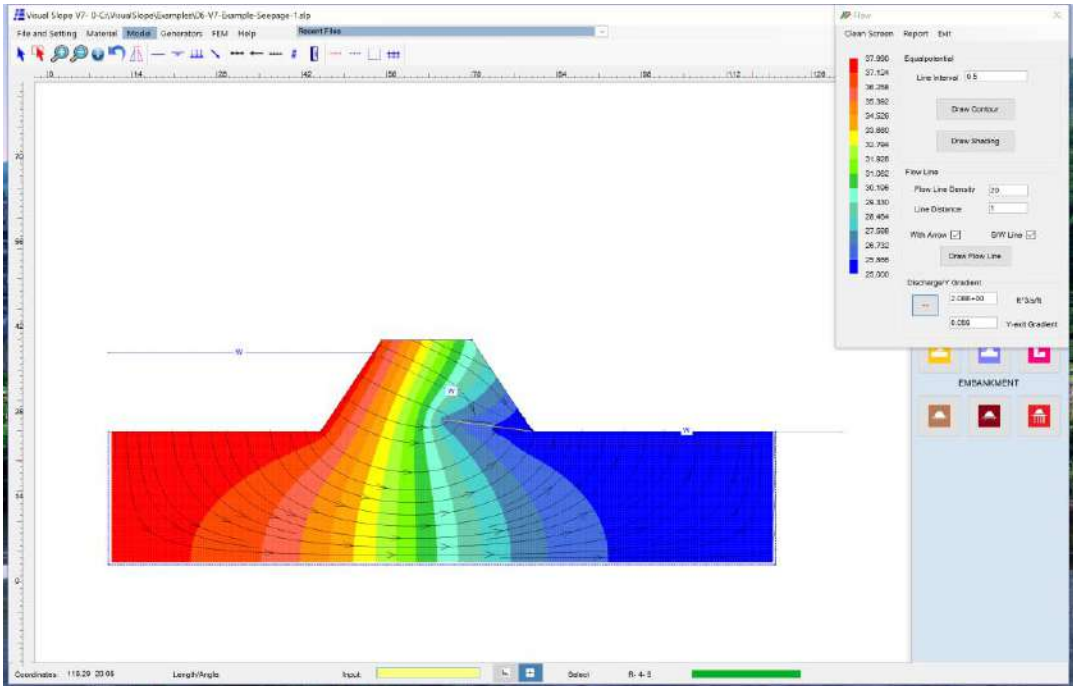


Figure 44: Seepage Analysis Result

Video Example

This [video example](#) presents the procedure of seepage analysis.

SLOPE STABILITY ANALYSIS UNDER SEEPAGE PRESSURE

If water lines are across the entire model from the left to the right, as shown in Figure 45, seepage pressures will be automatically considered in regular slope stability analyses. However, in some cases, the water table under the ground surface is not clear, as shown in Figure 46. To correctly perform a slope stability analysis, three steps (shown below) must be taken.

1. An unconfined seepage analysis must be conducted first to obtain the phreatic line (Figure 47), which is the water table under the ground surface.
2. Once the phreatic line is obtained, you can draw water lines by tracing the phreatic line and obtain Figure 45.
3. Then a slope stability analysis under seepage pressure can be performed.

Alternatively, you can click the *Slope Stability Analysis under Seepage Analysis* button in the *Analysis* panel, or check the *Seep/Rain* option in Figure 17. Visual Slope will perform those three steps together. The result is shown in Figure 48.

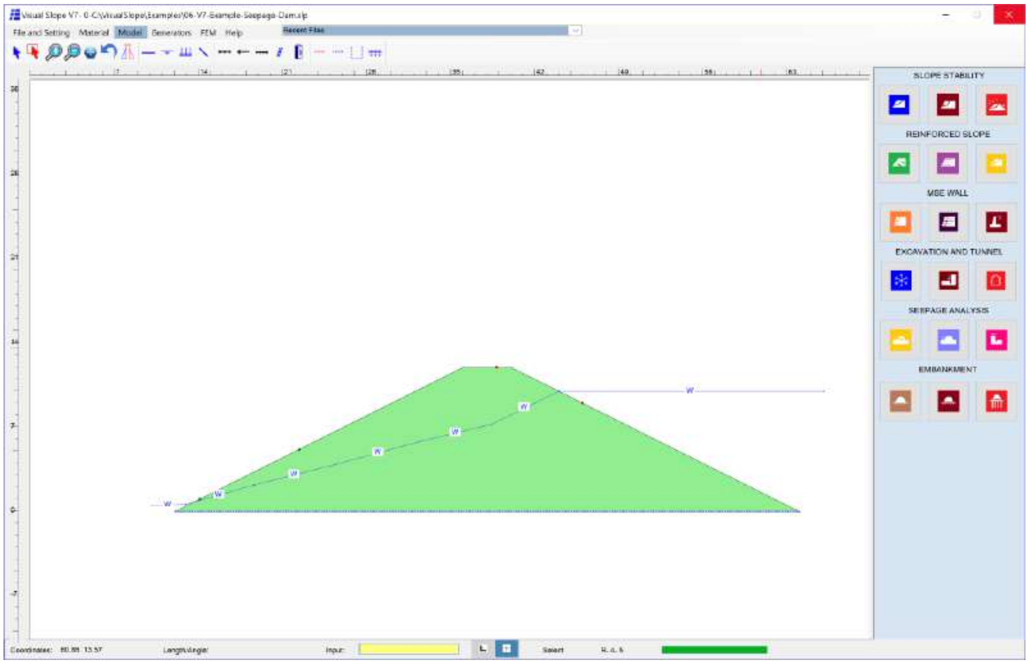


Figure 45: Slope Stability Analysis under Seepage Pressure

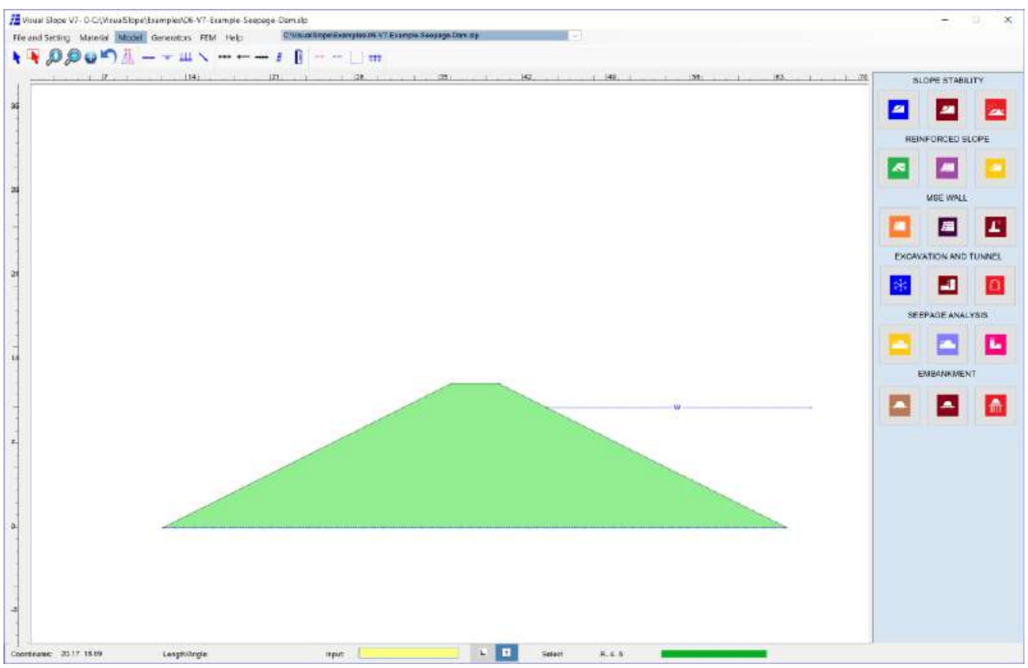


Figure 46: Slope Stability Analysis under Incompletely Defined Water Table

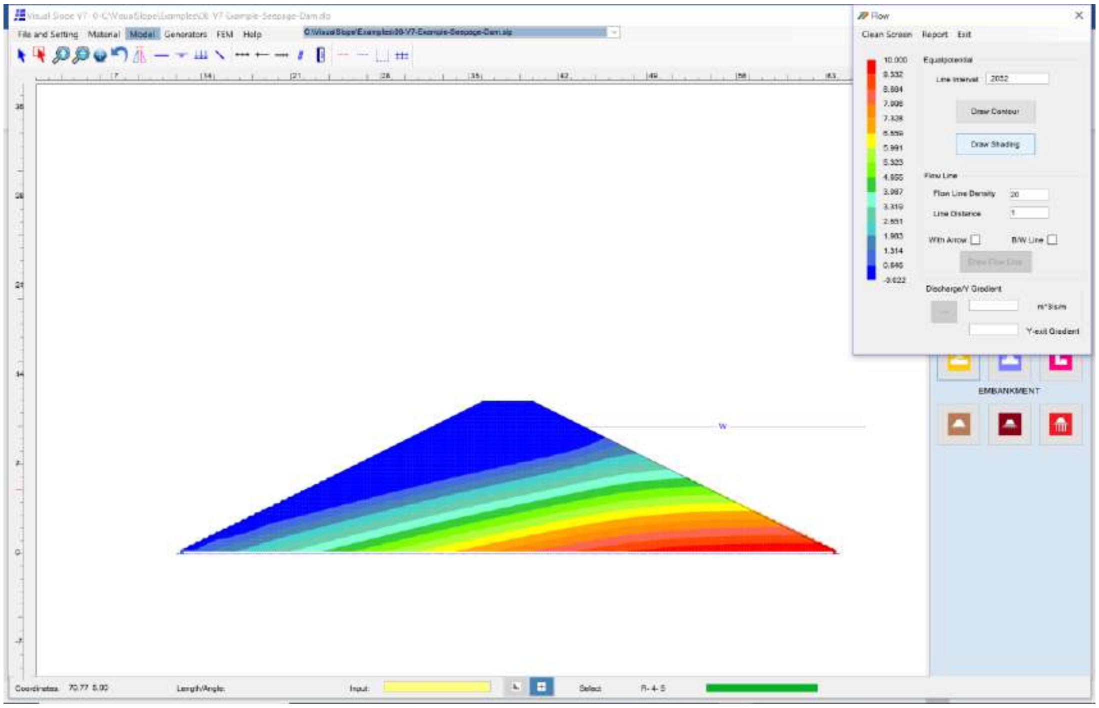


Figure 47: Phreatic Analysis

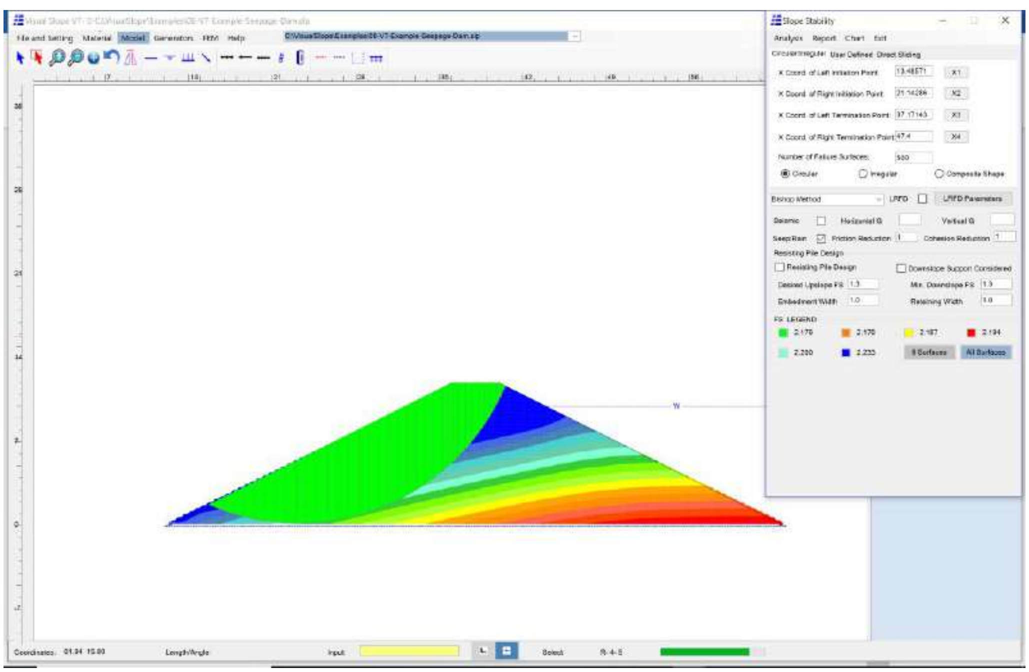


Figure 48: Slope Stability Analysis under Seepage Pressure

Video Example

This [video example] presents the procedure of slope stability analysis under seepage pressure.

TUNNEL LINING DESIGN

In tunnel-lining design, two types of design models—ground-structure model and load-structure model—are commonly used.

The ground-structure model is usually used for complicated projects to enable the engineers to examine not only the internal forces and deflections in the tunnel lining but also the stresses and deformation in the surrounding ground, based on the interaction between the tunnel lining and the ground. This design model requires 2-D or 3-D numerical analysis. However, the results from this model usually are not directly used for the actual tunnel-lining design but more for verification purposes.

The load-structure model, on other hand, is commonly adopted in actual tunnel design. In this model, the interaction between the tunnel lining and the surrounding ground is simulated with normal and tangential springs; load combinations can also be included. However, this model is unable to provide stresses and deformation in the surrounding ground.

Visual Slope V7 seamlessly combines those two models to enhance their advantages and eliminate their shortcomings. All load combinations can be calculated with just one click.

Modeling and Load Combination

To establish a tunnel lining model, the user must use the *Tunnel Lining Generator*. All the material properties to be used in the design must be set up prior to modeling. Click the *Tunnel Lining Generator* button on the *Generators* menu; the *Tunnel Lining Generator* page will appear (Figure 49). Currently, Visual Slope only allows a tunnel section and horizontal loads in symmetrical conditions. This page contains several sections that will be described in detail below.

<u>Common Input</u>

No matter what shape the tunnel lining is, the data circled in Figure 49 must be defined.

Cross-Section Thickness

The *Cross-Section Thickness* is the longitudinal thickness to be analyzed.

Spring Constants

Spring constants are for normal and tangential springs that are used to simulate the interaction between the tunnel lining and the surrounding rock. If the spring constants are not given, Visual Slope will calculate them in accordance with the Army Corps of Engineers' Recommendation (EM 1110-2-290).

Spring constant for normal springs:

$$K_r = \frac{wlE_r}{R(1 + \mu_r)}$$

Spring constant for tangential springs:

$$K_t = \frac{0.5K_r}{(1 + \mu_r)}$$

Where:

Kr = Radial Spring Constant

Kt = Tangential Spring Constant

Er = Elastic Modulus of Surrounding Material

μ_r=Poisson's Ratio of Surrounding Material

w = Longitudinal Calculation Depth

l = Element Length

The user can use his or her own spring constants by checking the *User Defined Spring Constants* option and spring constants for normal and tangential springs.

Tangential Spring and Compression Only Options

If the *Tangential Spring* option is off, tangential springs will not be generated in the model. It is recommended that tangential springs be used for circular tunnels to avoid rotational movement in calculation.

The *Compression Only* option is recommended for simulating the interaction between the tunnel lining and surrounding soil/rock.

Lining Property

The lining property must be set up prior to model generation. For details on how to set up the lining property, see Pile/Beam Parameters section. Select the lining property from the list for the tunnel lining. If the tunnel lining section varies, after the model is generated, each section properties can be manually assigned.

Hinges

If the tunnel lining includes hinges, after the model has been generated, click the *Add Hinge* button, and then move the cursor to the node in the lining where a hinge is needed, and click again. A hinge will be added to that node. The bending moment will be zero at nodes that have hinges.

Circular Shape Lining

To create a circular shape tunnel lining model, click the *Circular* button on the top menu first. A circular tunnel section will appear in Figure 49. For a circular tunnel, the data below are required.

1. Circular center coordinates, X and Y.
2. The center radius of the tunnel section.

TBM Tunnel Lining

If a TBM lining section is to be generated, check the *TBM Lining* option. The TBM lining input table will appear. For each segment connection, a row of data is required, which includes the angle in degree, hinge option, and hinge stiffness factor. Angles must input consecutively in a clockwise pattern, referring to 12:00 as zero degree. If a hinge factor needs to be considered at the connection, check the hinge option. The hinge stiffness factor varies from 0 to 1. If 0 is chosen, the hinge is a perfect hinge, at which the bending moment equals zero. If 1 is chosen, the hinge stiffness equals the stiffness of the segment, or in other words, no hinge in considered. If a value between 0 and 1 is chosen, a fraction of the segment stiffness will be used as the hinge stiffness.

Circular Crown and Straight Wall Closed Tunnel Section

If this option is chosen, besides the data for a circular tunnel, additional data for the wall height are required.

Unclosed Horseshoe-Shaped Tunnel Section

Tunnel shape options 3 and 4 are for an unclosed horseshoe-shaped tunnel section that is composed of three smoothly connected walls, a crown, and two side walls. The difference between option 3 and option 4 is that option 3 is for a hinge connected bottom, while option 4 is for a fixed bottom. For those two options, the data below are required.

1. Circular center coordinates, X and Y
2. The center radius of the tunnel section
3. Wall height for the side wall height
4. Bottom width
5. Top width

The top width cannot be wider than the diameter of the crown. If the top width equals the diameter of the crown and the bottom width, the side walls will be vertical straight walls; otherwise, they will be curved walls.

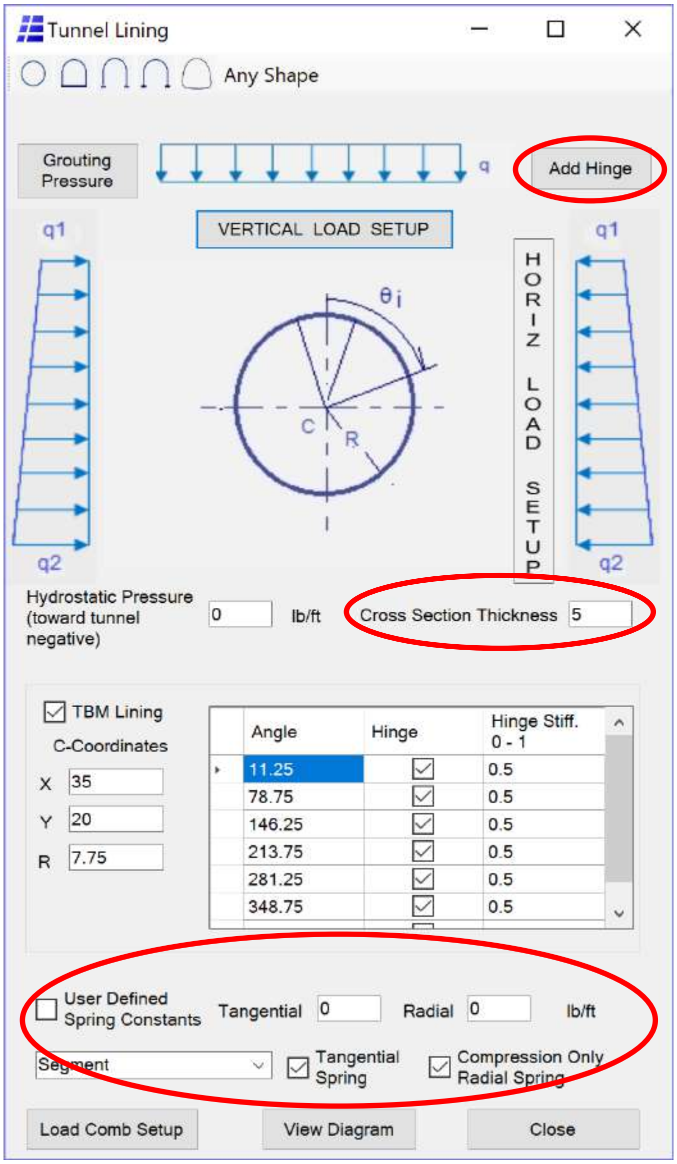


Figure 49: Tunnel Lining Generator

<u>Closed Horseshoe-Shaped Tunnel Section</u>

Tunnel shape option 5 is for a closed horseshoe-shaped tunnel section that is composed of six smoothly connected walls, as shown in Figure 50. For this option, the data below are required.

1. Circular center coordinates, X and Y
2. The center radius of the tunnel section
3. Wall height for the side wall height
4. Bottom width
5. Top width
6. R1, the inverted radius
7. R2, the bottom corner radius

The top width cannot be wider than the diameter of the crown. If R1 is set to 0, the bottom will be a horizontal straight line. If R2 is set to 0, there will be no corner curves.

Any Shape

Visual Slope allows users to create tunnel sections of any shapes, a combination of curves, and straight lines. Since symmetrical conditions are considered, only the right side of the structure needs to be created; the left side will be the mirror image of the right side. Date must be input consecutively in a clockwise order, referring to the first point at the 12:00 position (Figure 51).

Arc Only Section

If the *Arc Only* option is chosen, the tunnel section will consist of a series of arcs that are connected tangentially (smoothly) one by one. After the coordinates of the first point are set, input the center coordinates of each arc in a clockwise order. No radii are required. Visual Slope will calculate the proper radii based on the input data and tangential connection conditions.

Line and Curve Section

If the *Arc Only* option is unchecked, the tunnel section will consist of a series of curves and straight lines that are connected one by one, not necessarily in a smooth manner. After the coordinates of the first point are set, the coordinates of the end point of each line must be provided in a clockwise order. If the line is a curve, the coordinates of the center of that curve must also be provided. The tunnel section can be a closed or unclosed section.

View Model

To view the tunnel section model, click the *View Diagram* button in Figure 51. The tunnel model will appear in the *Main* page of Visual Slope (Figure 52).

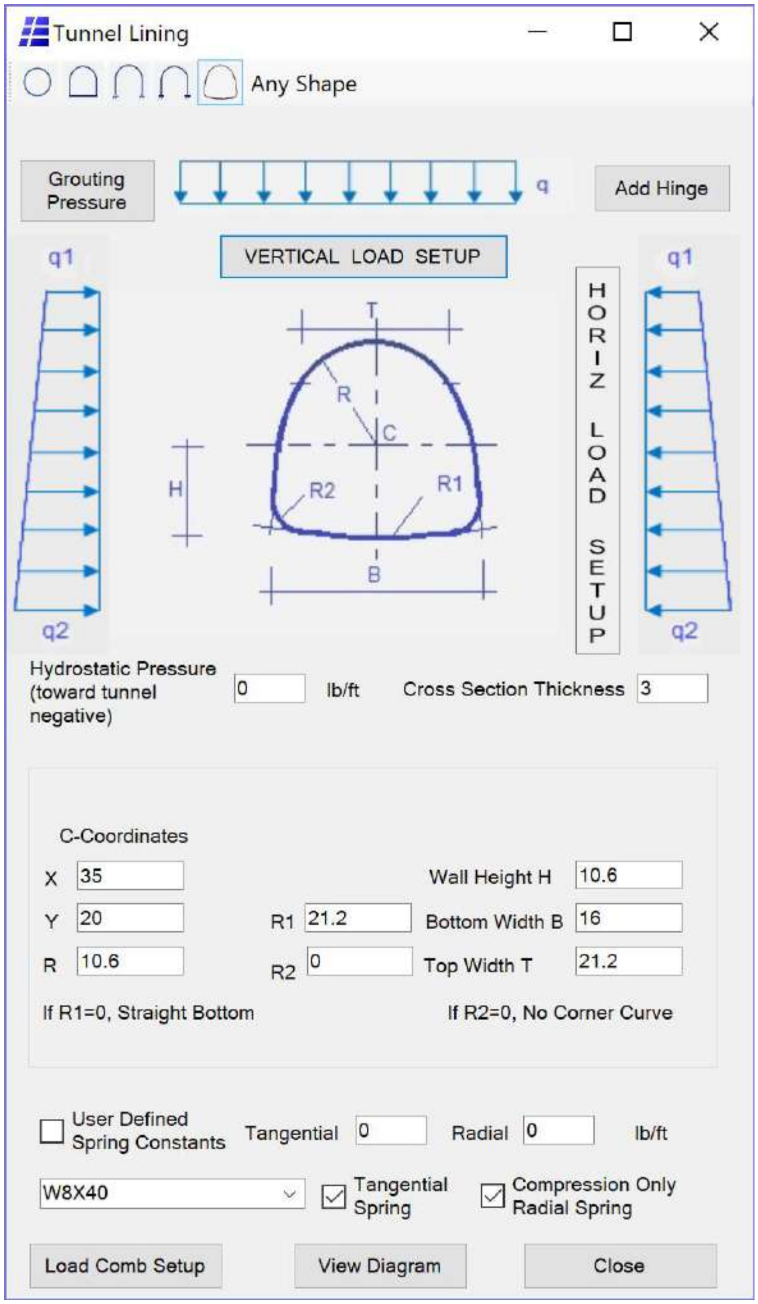


Figure 50: Closed Horseshoe Section

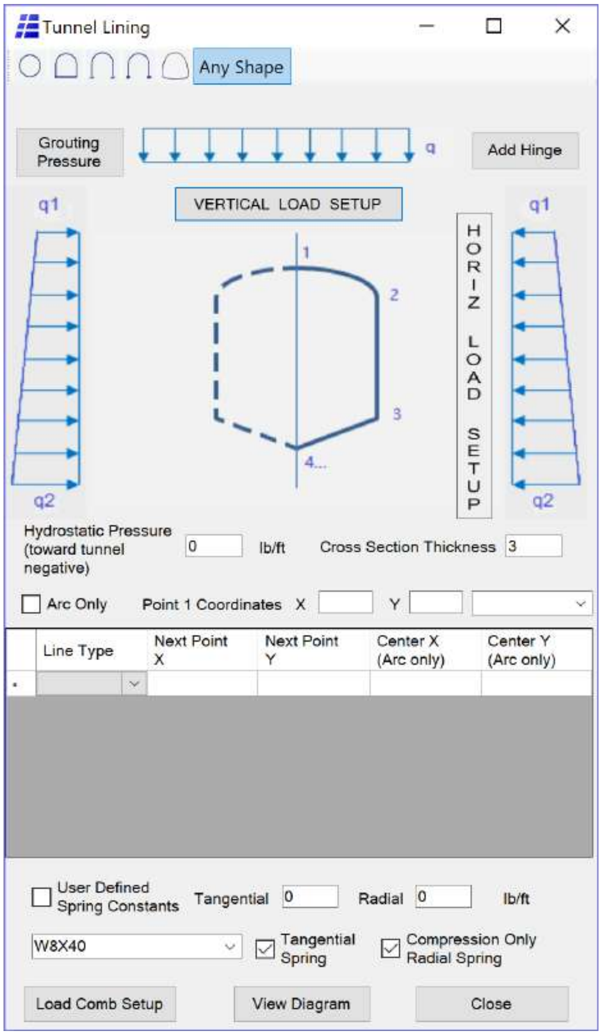


Figure 51: Tunnel of Any Shape

<u>Load Conditions</u>

The convention for a load is that if the load is toward the tunnel, it is considered negative, and vice versa. For example, water pressure inside the tunnel is positive; water pressure outside the tunnel is negative.

Hydraulic Pressure

A hydraulic pressure is an all-around pressure. If the pressure is toward the tunnel, it is negative. Otherwise, it is positive.

Grouting Pressure

Grouting pressure should be set point by point in clockwise order, referring to 12:00 as zero degree. For each point, the start angle, end angle, and pressure are required (Figure 53).

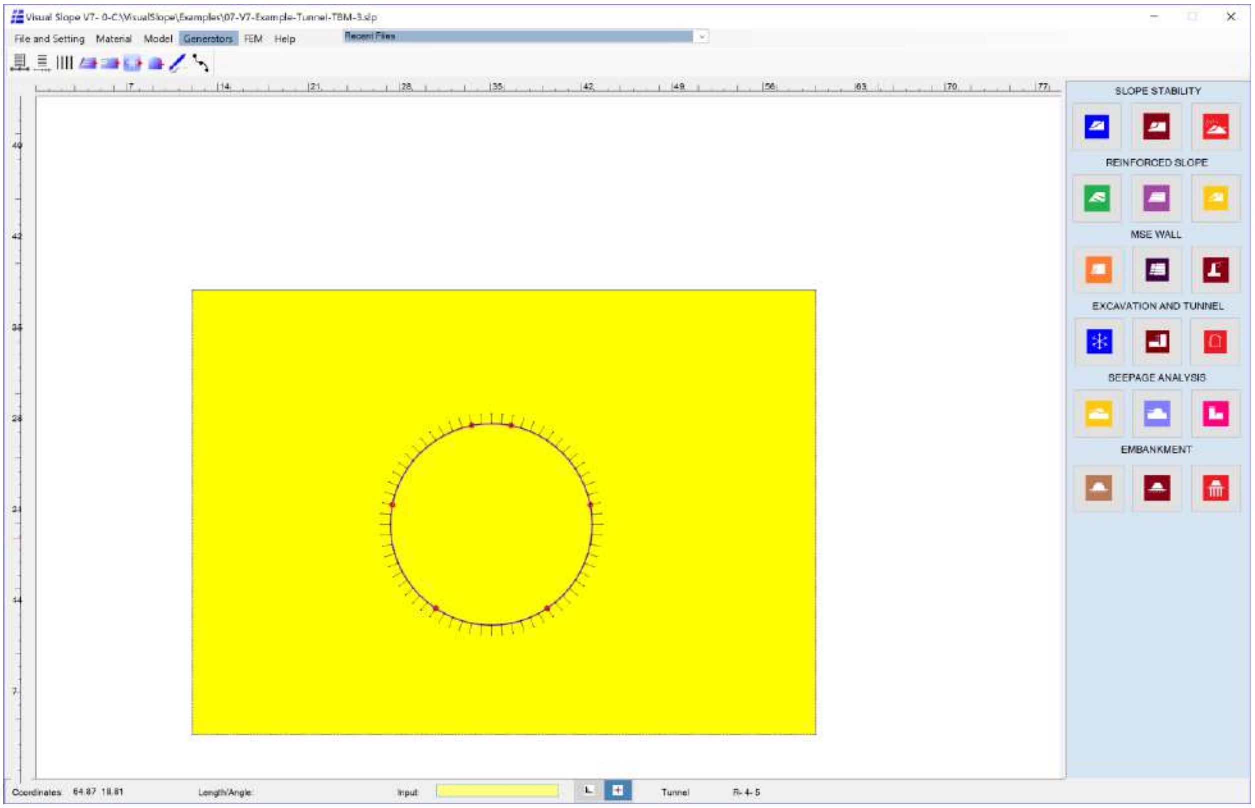


Figure 52: Tunnel Model

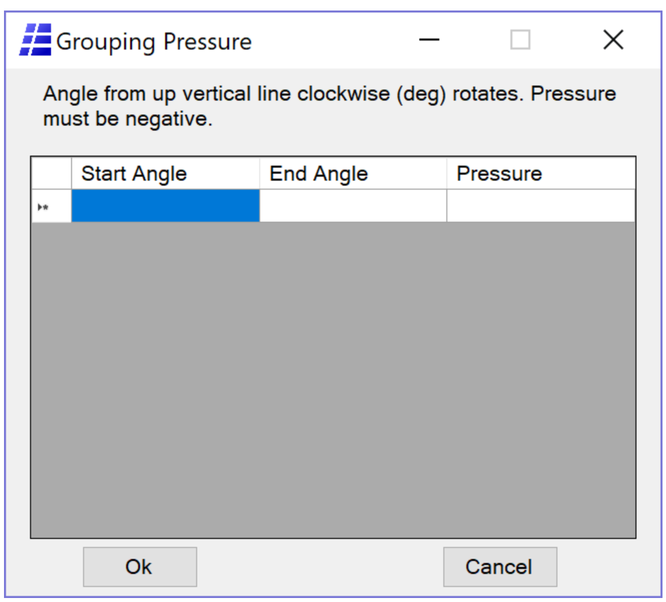


Figure 53: Grouting Pressure Input Page

Vertical and Horizontal Pressure

To input vertical or horizontal pressure, click the *Vertical Pressure Setup* button or *Horizontal Pressure Setup* button (Figure 51), respectively. The input page (Figure 54) will appear. Visual Slope allows the user to input up to four vertical pressures and up to four horizontal pressures. For each pressure, the user needs to input the pressure name, the intensities of point 1 and point 2, and the distances from edge 1 and edge 2.

<u>Load Combinations</u>

Once the load conditions have been set up, the user can set up the load combinations in accordance with code or project requirements. To set up load combinations, click the *Load Comb. Setup* button in Figure 51, and the *Load Combination* page will appear, as shown in Figure 55. The *Load Combination* page contains two columns. The first column lists all existing load names, and the second column is for load factors. A load factor can be any number. If it is 0, the load associated with it will not be considered in the calculation of this combination; if it is greater than 1, the load associated with this factor will be amplified. And if it is negative, the load will be changed to its opposite direction. In Figure 55, for example, in load combination 1, only *Grout-Press.* is considered. Once the load combination 1 has been established, the user can continue to establish load combinations 2, 3, and 4 if needed. Visual Slope allows the user to establish up to four different load combinations for each analysis.

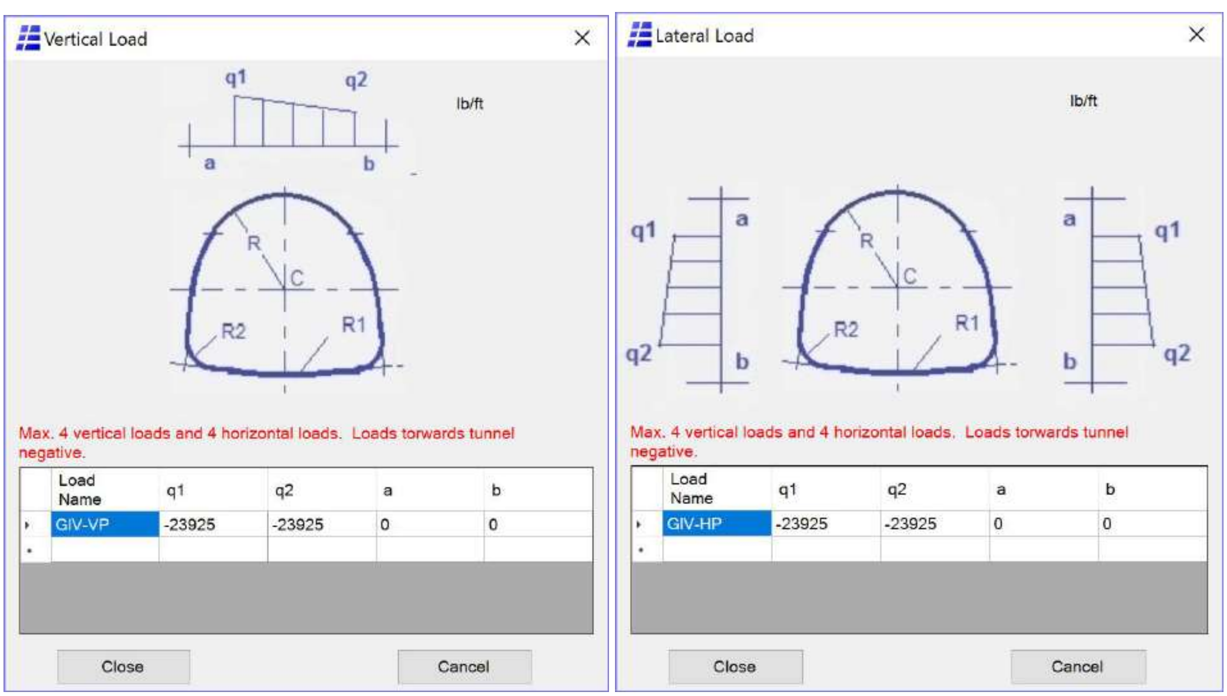


Figure 54: Vertical and Horizontal Pressure Input Page

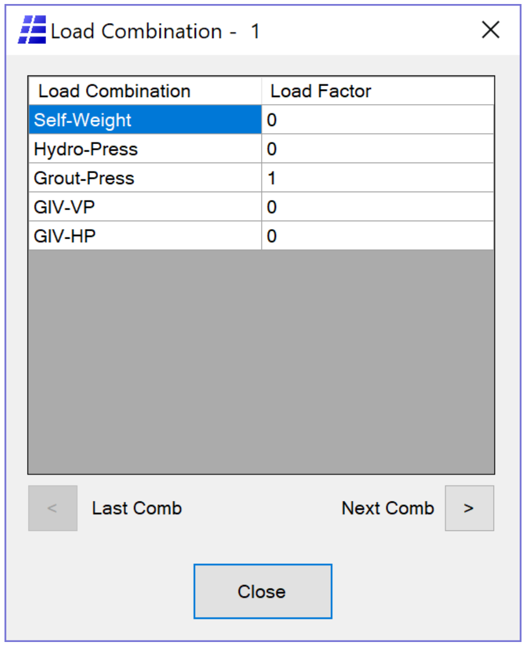


Figure 55: Load Combination Page

Performing Analysis

After the model and loads have been set up, analyses can be performed. The following sections describe the background of analyses and steps of execution.

<u>Background of Analyses</u>

To explain the background of analyses clearly, Figure 52 is referenced. If a user only needs to know the deformation and internal forces of the tunnel lining, then the load-structure model, which is recommended by the Army Corps of Engineers' Recommendation (EM 1110-2-290) and as shown in Figure 56, will be used for the analysis. Calculations will be based on the 1-D FEM and conducted for each load combination. After the analysis is completed, the user will be able to view the results of deformations and internal forces of the tunnel lining under each load combination. Figure 57 presents the tunnel lining deformation under load combination 2.

If a user needs to know not only the deformation and internal forces of the tunnel lining, but also the deformation and stress changes in the surrounding ground due to tunneling, then both the 1-D and 2-D FEM will be used. First, Visual Slope will use the 2-D FEM to calculate the initial stresses and deformation prior to tunnel excavation, as shown in Figure 58. Second, Visual Slope will use the 1-D FEM to perform a load-structure model calculation, as discussed in the last paragraph and shown in Figure 57. Third, once the result of a particular load combination is chosen, the deformation of the tunnel lining under that load combination plus the specified contraction value will be used as the boundary conditions along the tunnel lining in the 2-D FEM model to calculate the stress changes and deformation of the ground due to tunneling. The total stresses in the ground equal the combination of the initial stresses calculated from the first step and the stress changes calculated in the third step. Figure 59 presents the ground deformation due to tunneling under load combination 2. Along the tunnel lining, the ground deformation will match exactly with the tunnel lining deformation calculated in the second step.

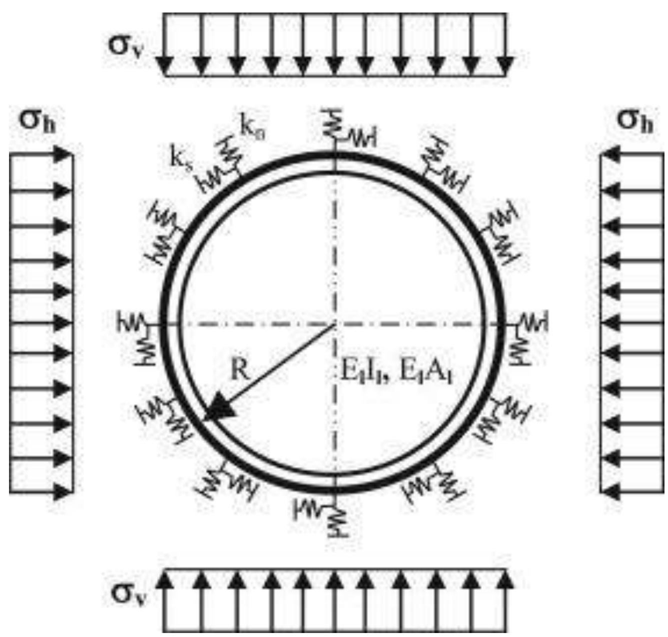


Figure 56: Load-Structure Model

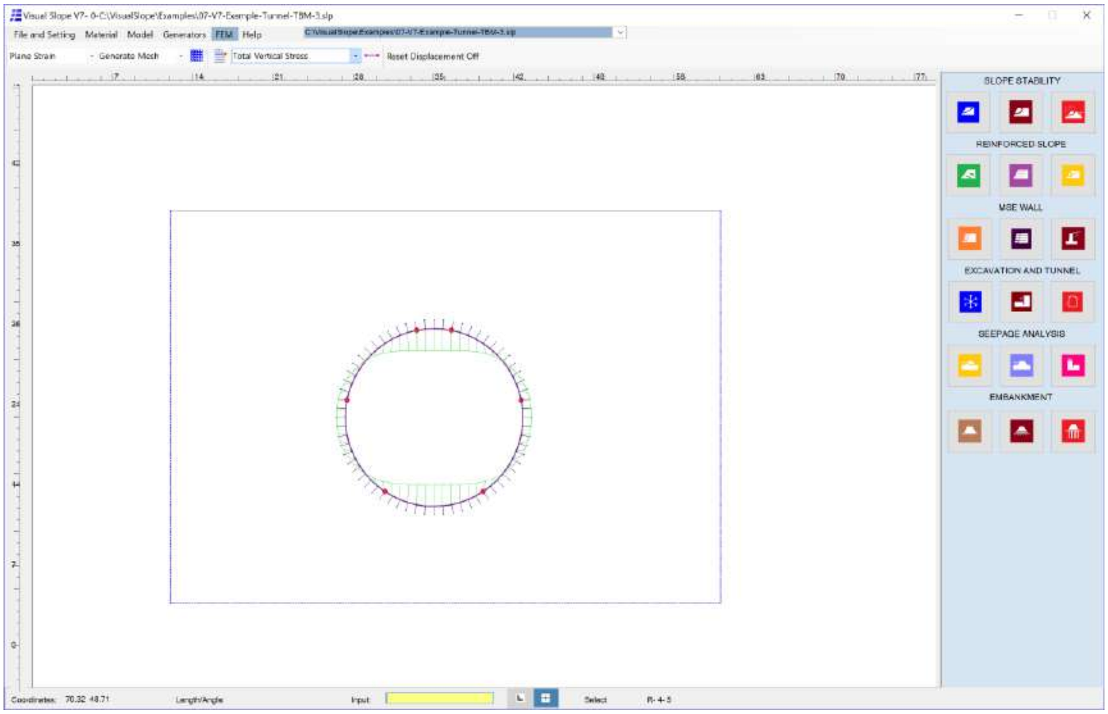


Figure 57: Tunnel Lining Deformation under Load Combination 2

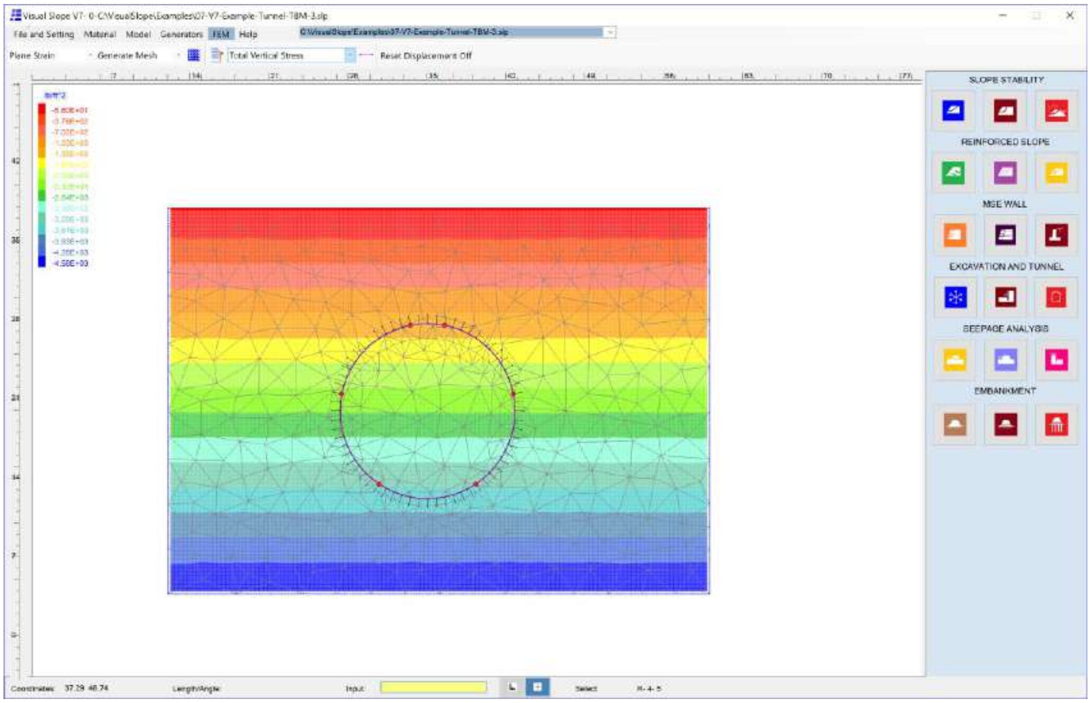


Figure 58: Ground Initial Vertical Stress and Settlement

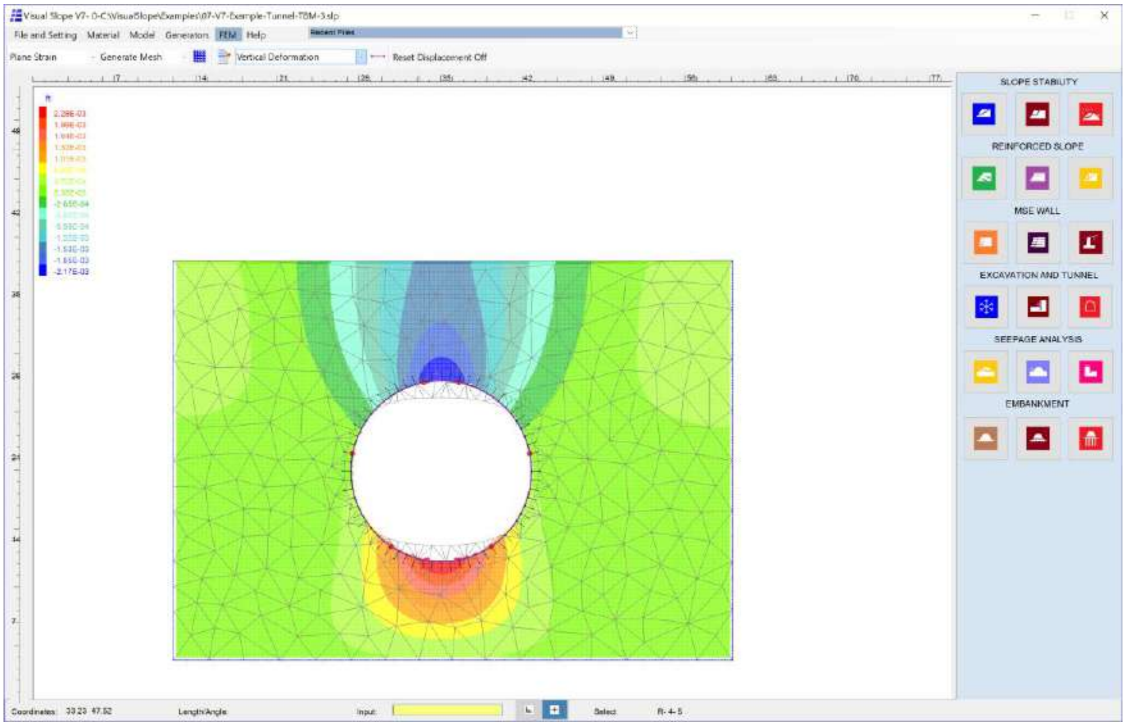


Figure 59: Ground Deformation (Mesh) and Settlement (Contour) under Load Combination 2

<u>Execution of Calculation</u>

To execute a tunnel lining analysis, click the *Tunnel Lining Design* button. The *Tunnel Lining Design* page will appear, as shown in Figure 60.

Including Ground

If both the tunnel lining and the surrounding ground need to be analyzed, this option needs to be selected. For details, see Background of Analyses section. Once this option is chosen, the ground contraction value can be specified if it exists. The ground contraction is the ground deformation along the tunnel excavation before the lining is installed.

Analysis Types

Visual Slope provides several different methods for tunnel lining analyses (not for ground).

1. FEM, which is the 1-D FEM (refer to Figure 56).
2. Ranken Method and Schwartz-Einstein Method, which are closed-form elasticity calculations only for circular tunnels without hinge effects.

3. Use of excavation load, in which all load combinations will be ignored and the load acting on the tunnel lining will be calculated from 2-D FEM and then used as the load in the 1-D FEM analysis.

Once the *Including Ground* option has been specified, and the analysis type has been chosen, an analysis will be executed. After the analysis is completed, the result will appear in the result tables (Figure 60). Different diagrams, such as deformation and bending moment, can be displayed in the *Main* page. Diagrams and a detailed report can also be printed. Once the Tunnel Lining Design page is closed, the ground deformation and stresses will be calculated by the 2-D FEM if the *Including Ground* option is checked. For the details of 2-D FEM, see the 2-D FEM section.

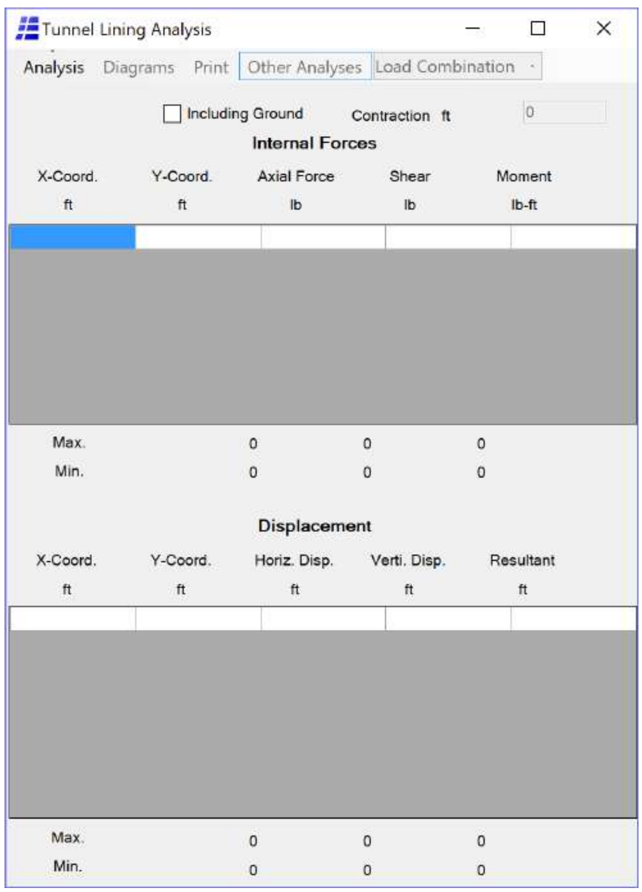


Figure 60: Tunnel Lining Design Page

Video Example

This video example presents the procedure of tunnel lining design.

GROUND FREEZING CONSTRUCTION DESIGN

Ground freezing is a construction technique used when the soil needs to be stabilized to prevent it from collapsing next to excavations and to prevent contaminates in soil from being leached away. 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.

The Visual Slope Ground Freezing module allows users to design the number of freeze pipes and layouts much more accurately and efficiently through 2-D analyses. In Visual Slope, for a freezing construction design, the units must be in meters for dimensions and in Celsius for temperature. Although Visual Slope Freezing Construction Design is a 2-D program, separately, it can analyze not only the temperature distribution in the radial direction of freezing pipes, but also the temperature along freezing pipes. The sections below describe how to use Visual Slope for ground freezing construction design.

Radial Direction Analysis

<u>Overview Modeling</u>

Overview modeling is used to establish freezing pipe layout over the ground surface or excavation surface to analyze the temperature distribution in the radial direction of freezing pipes. In an overview analysis, freezing pipes go into and are perpendicular to the surface. The *Freezing Construction Setup* generator under the *Generators* menu must be used to lay out freezing pipes. The *Freezing Construction Setup* page (Figure 61) will appear after the *Freezing Construction Setup* button is clicked. The setup page can be divided into four portions.

General Setup

The general setup contains 1) the ground initial temperature, 2) the ground thermal conductivity (in J/s/m/derC), 3) freezing pipe temperature, and 4) days of freezing, which must be set up first for the entire project.

Pipe Line Generator

The *Pipe Line Generator* can be used to generate a row of pipes. The coordinates of the start point and the end point of the line, and the total number of pipes in the line, must be given. After the *Generate* button is clicked, a line of pipes will appear in the *Main* page.

Pipe Circle Generator

If freezing pipes distribute in a circular pattern, the Pipe Circle Generator can be used to generate either an upper circle, a lower circle, or both. If only a portion of the circle is required, those unneeded points can be deleted by the *Select-Delete* method.

Individual Point

To draw freezing pipe points individually, go to the pulldown list, and click an empty row. A red button will appear on the right of that row. Click the red button, and then move the cursor to where the pipe should be located in the *Main* page, and click again. The coordinates of the pipe will be recorded in the list. The other way to set up a pipe is simply to enter the coordinates directly into the list.

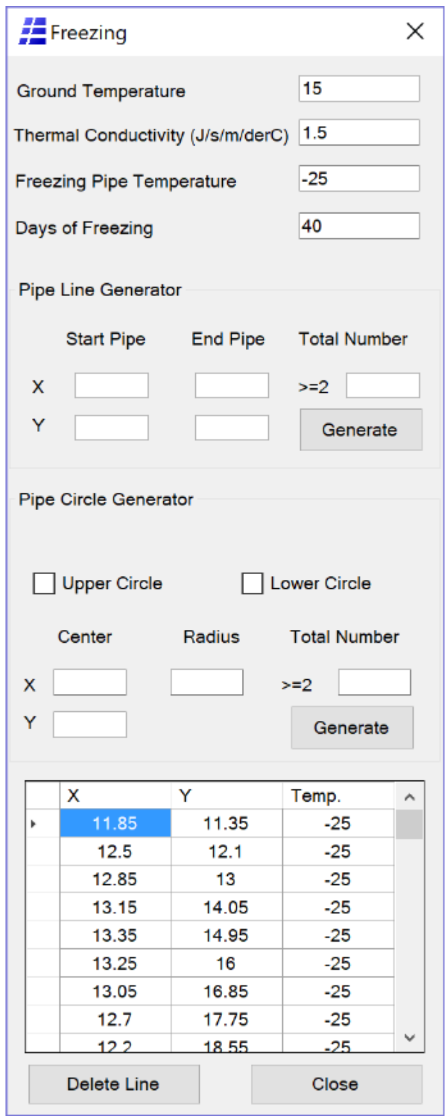


Figure 61: Freezing Construction Setup Page

Analyzing

Once the model is established, as shown in Figure 62, to perform the analysis, click the *Freezing Construction* button in the *Analysis* panel. The result of the analysis is shown in Figure 63.

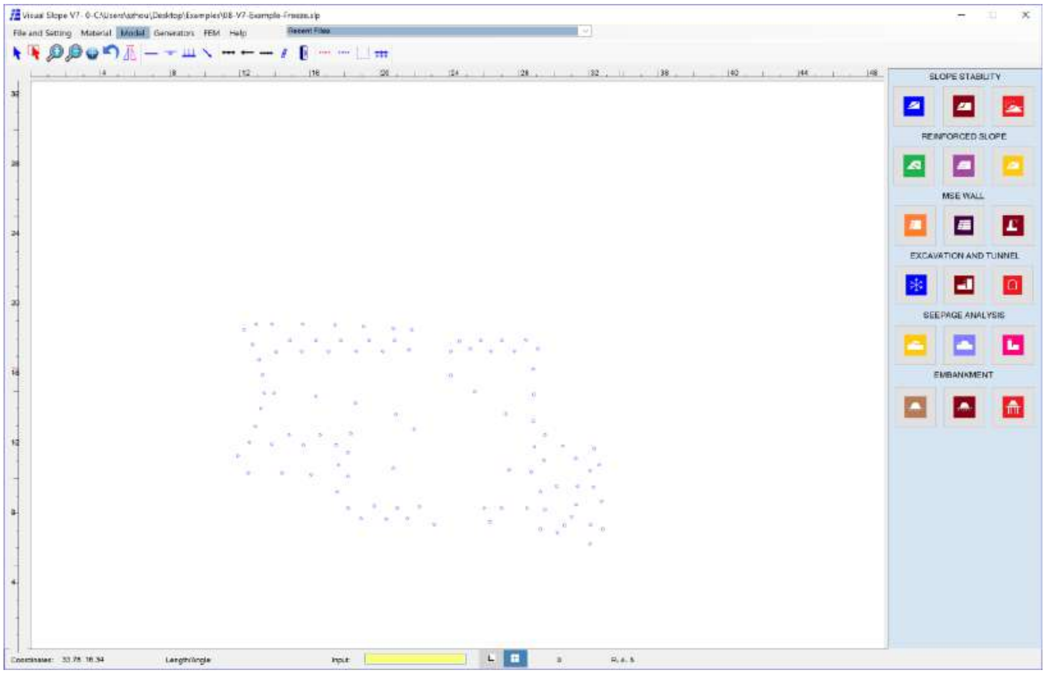


Figure 62: Freezing Pipe Layout

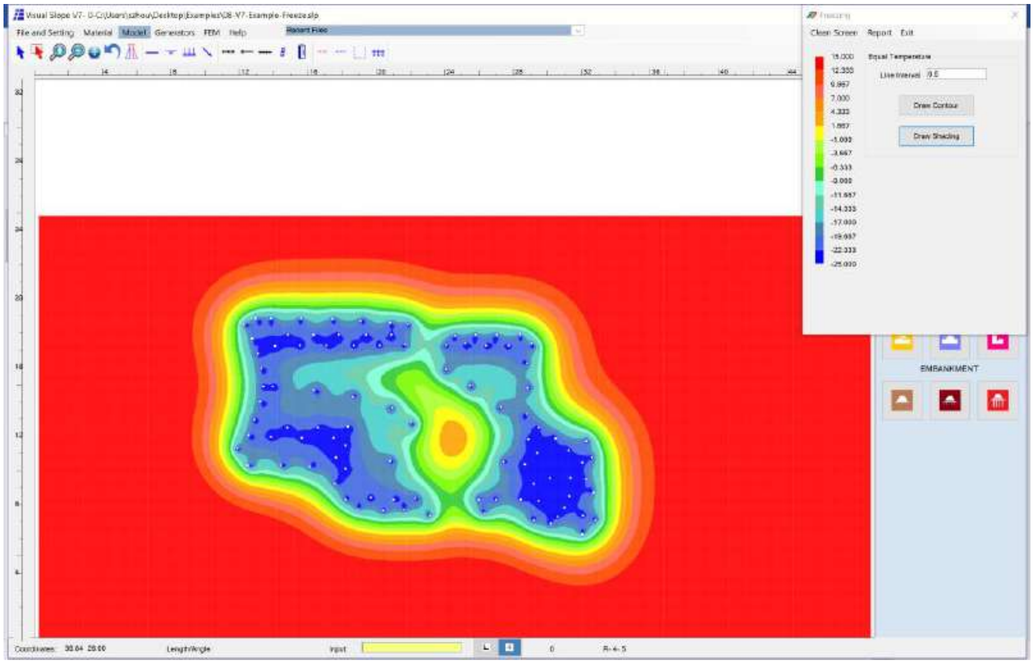


Figure 63: Temperature Distribution

Along Pipe Analysis

Sideview Modeling

A sideview modeling is used to analyze the temperature distribution along freezing pipes. For this type of analysis, freezing pipes are parallel to the surface and simulated with *Limit* lines, as shown in Figure 64, of which the temperatures can be set up individually through *Line Editor* (Figure 65). The *Reduction Factor*, in this case, is the temperature of the freezing pipe. The ground conditions must be set in accordance with the previous General Setup section.

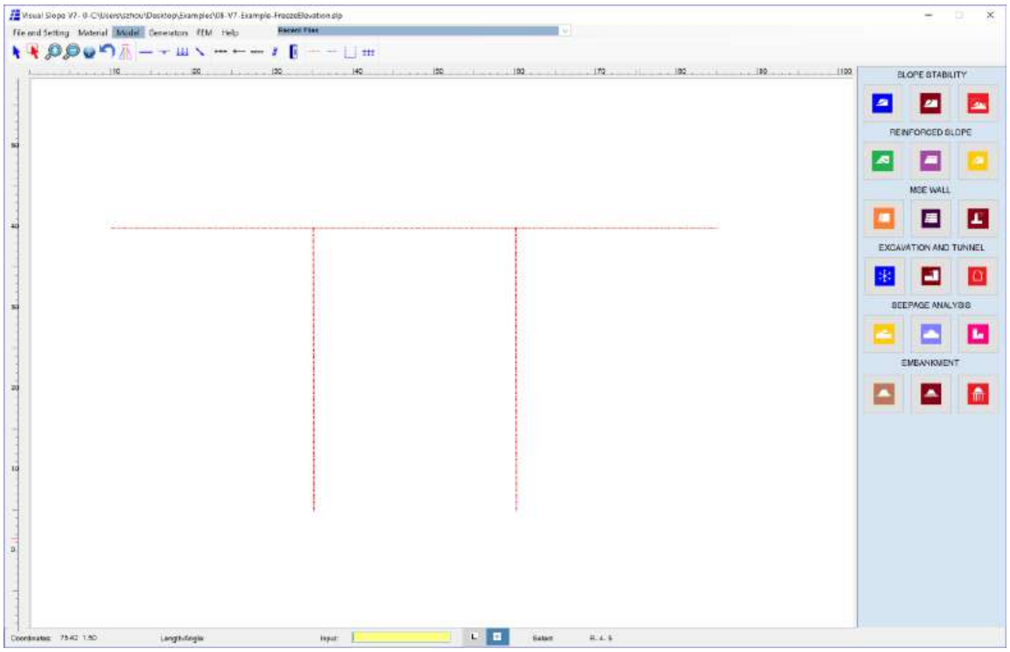


Figure 64: Sideview Modeling

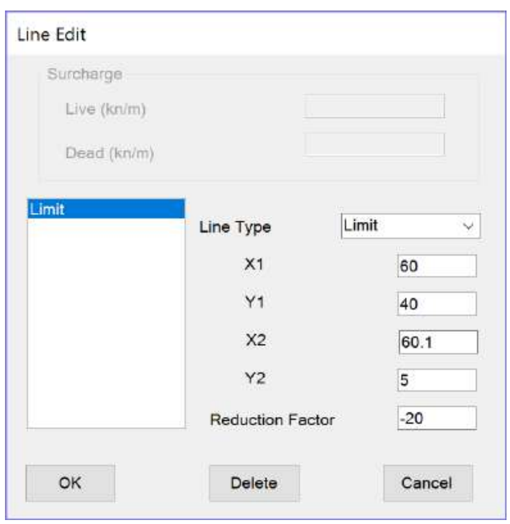


Figure 65: Line Editor for Limit Lines

<u>Analyzing</u>

Once the model is established, as shown in Figure 64, to perform the analysis, click the *Freezing Construction* button in the *Analysis* panel. The result of the analysis is shown in Figure 66.

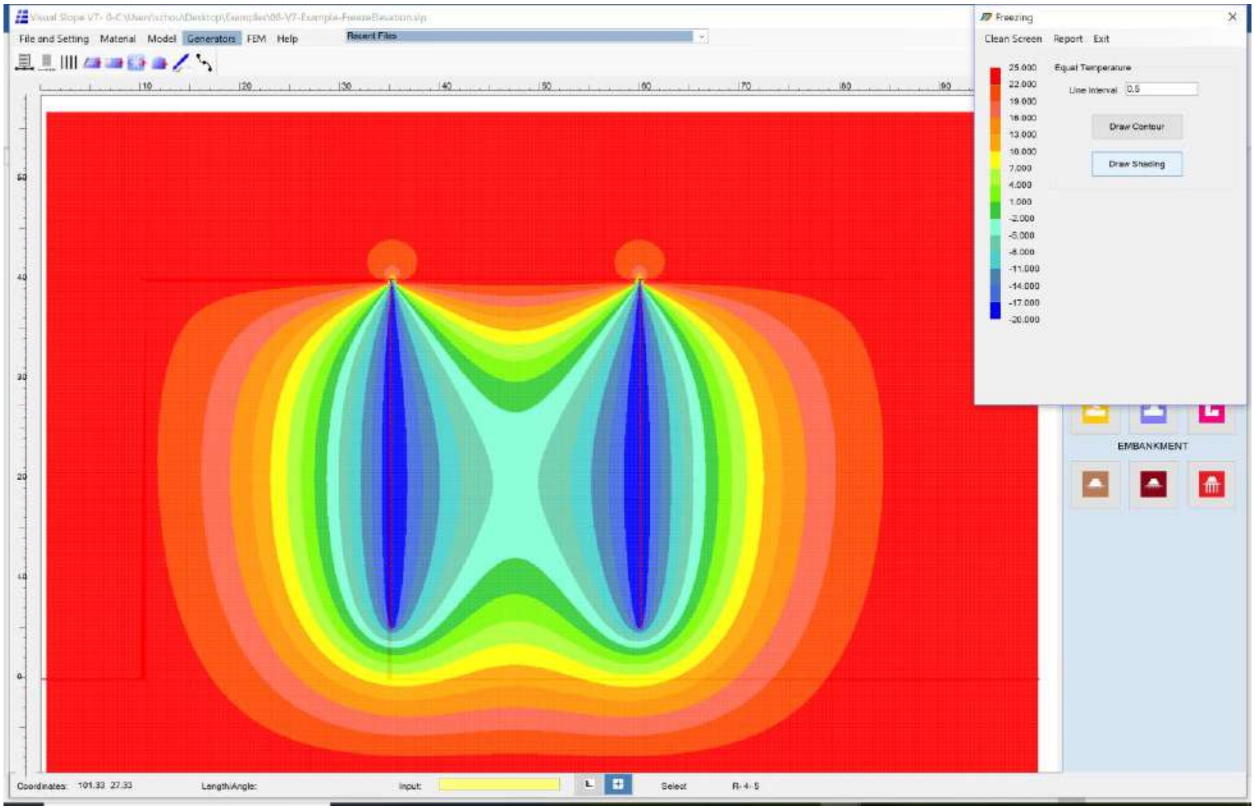


Figure 66: Temperature Distribution

Video Example

This video example presents the procedure of freezing construction design.

PILE SUPPORTED REINFORCED EMBANKMENT DESIGN

With modern pile and soil reinforcement technology, pile supported embankments will frequently provide the best combination of economic efficiency, construction speed, settlement performance, reliability, installation/QC simplicity, and environmental compliance in comparison to any other ground improvement method or bridge construction technique. However, there are not many design tools available for engineers to use to design this type of application. Visual Slope developed a pile supported embankment design module based on German EBGEO recommendations. This design module is easy to use and reliable.

Modeling

Establishing a pile supported reinforced embankment model is similar to establishing a model for slope stability analysis. Piles can be added in with a drawing method or an array method. The pile property must be set as *Pile*. See the material property for details. Figure 67 is a typical pipe supported embankment model, which consists of layered ground, an embankment, two layers of geosynthetic reinforcement, and piles.

Analyzing

To perform an analysis, click the *Pile Supported Reinforced Embankment* button in the *Analysis* panel. After analysis, the results will show as in Figure 68.

Video Example

This video example presents the procedure of pile supported reinforced embankment analysis.

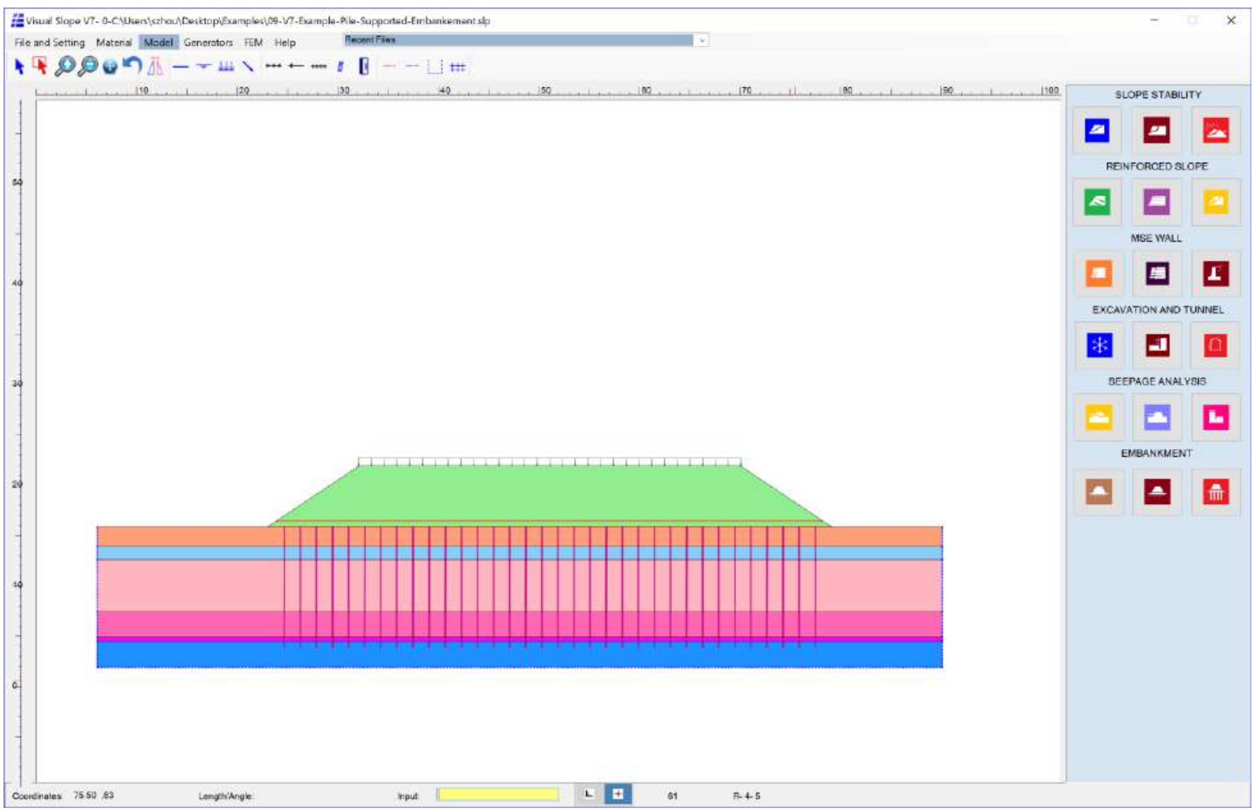


Figure 67: Pile Supported Reinforced Embankment

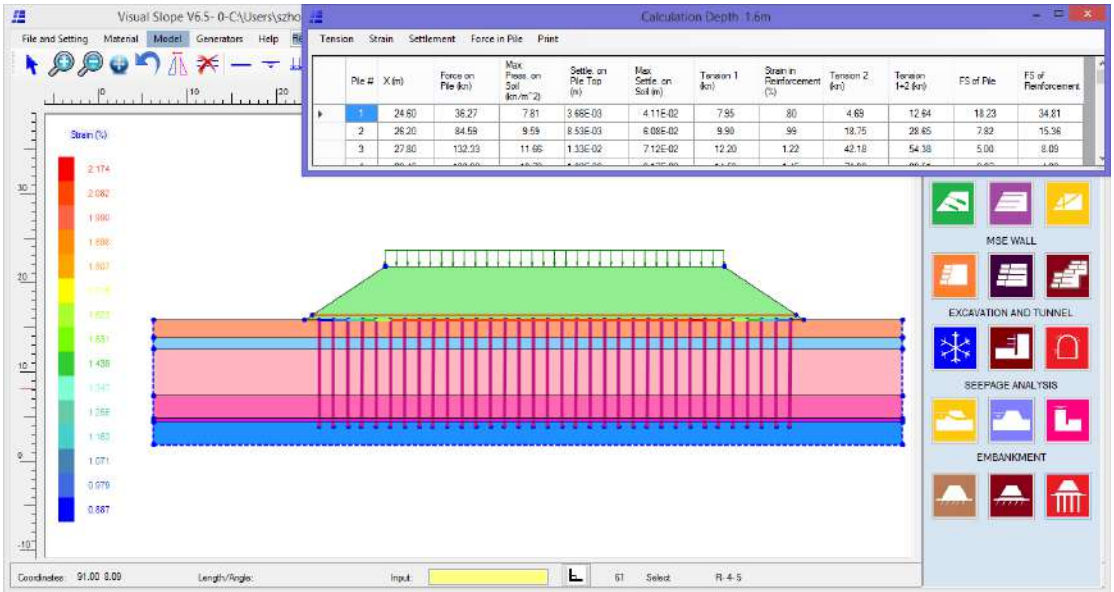


Figure 68: Results of Pile Supported Reinforced Embankment

2-D FEM ANALYSIS

The 2-D FEM module developed in Visual Slope V7 is based on the elasticity theory and mainly to assist other modules in design. With the 2-D FEM module, users can not only obtain the internal forces and deformation of the structures being designed, but also the stresses and deformation of surrounding ground, which may affect adjacent existing structures, such as in a shoring system or a tunnel lining design case. 2-D FEM can also provide an additional means to verify an existing calculation, such as in a slope stability analysis.

Besides the functions mentioned above, the 2-D FEM module can also be used as stand-alone module for general FEM analyses in a limited capacity. This section will describe the 2-D FEM module for that purpose, including modeling, mesh generation, staged construction, and view of results.

Element Types and Modeling

Currently, the Visual Slope 2-D FEM module allows two element types to be used: triangle solid elements for soil or rock and truss elements for anchors. Triangle elements used in Visual Slope are six-node and six-gaussian-point isoparametric elements. Truss elements used Visual Slope are two-node elements. Beam elements cannot be used directly through the 2-D FEM module but can be used through the shoring design module and tunnel lining design module. For the reinforcement, like geosynthetics or metal straps, used in a reinforced slope or an MSE wall, since the vertical spacing between reinforcement layers commonly are very narrow, if reinforcement layers are generated as FEM elements, it would cause elements to be overcrowded. Therefore, in order to count the fact of reinforcement, Visual Slope combines horizontal stiffness with the soil stiffness to increase the total horizontal stiffness.

The modeling procedure for an FEM analysis is the same as that for a slope stability analysis. Review the modeling section for details.

In boundary conditions, currently, the Visual Slope 2-D FEM module allows perfect vertical boundary lines that restrict horizontal movement and perfect horizontal boundary lines that restrict vertical movement. At a corner of vertical and horizontal boundary lines, both horizontal and vertical movements are restricted.

Analysis Types

The Visual Slope 2-D FEM module is capable of performing three types of analyses: plane strain, plane stress, and axisymmetric analyses. Users should choose a proper analysis according to the project conditions.

Plane Strain Analysis

If a project is very long compared with the dimensions of the cross-section, and the load conditions do not vary significantly in the longitudinal direction, the deformation in the longitudinal direction can be ignored. Then a cross-section can be cut out for a plane strain analysis. For example, plane strain analyses can be used for long tunnel lining design, or a slope stability analysis for a long slope.

Plane Stress Analysis

If an object with cross-section dimensions much larger than its thickness, like a plate, and loads are all parallel to the cross-section, then a plane stress analysis is suitable to use.

Axisymmetric Analysis

A 3-D object that is formed through a plane rotating about an axis z is call an axisymmetric body, such as a cylinder and a donut, and is most easily described in cylindrical coordinates, where z is called the axis of symmetry, which in Visual Slope is always located at coordinate 0 of the radial axis r. If the geometry, support conditions, loads, and material properties are all axially symmetric (all are independent of θ), then the problem can be idealized as a 2-D one. Problems such as soil masses subjected to circular footing loads can often be analyzed using axisymmetric analysis.

Mesh Generation

Once an analytical model is established, choose mesh generation options in the *Generate Mesh* list. Options include *Basic Mesh*, *Coarse Mesh*, *Fine Mesh*, and *Very Fine Mesh*. *Basic Mesh* is strongly recommended.

Example of Staged Construction

To explain the procedure of a 2-D FEM analysis, a simple staged construction example (Figure 69) is used, in which an embankment will be placed on the existing ground, and the goal is to calculate the ground settlement and final stresses due the fill placement. A plane strain analysis is used. The sections below describe the procedure of the analysis.

Model Setup

Figure 69 presents the model of this example, which consists of the existing ground in brown and the embankment in green.

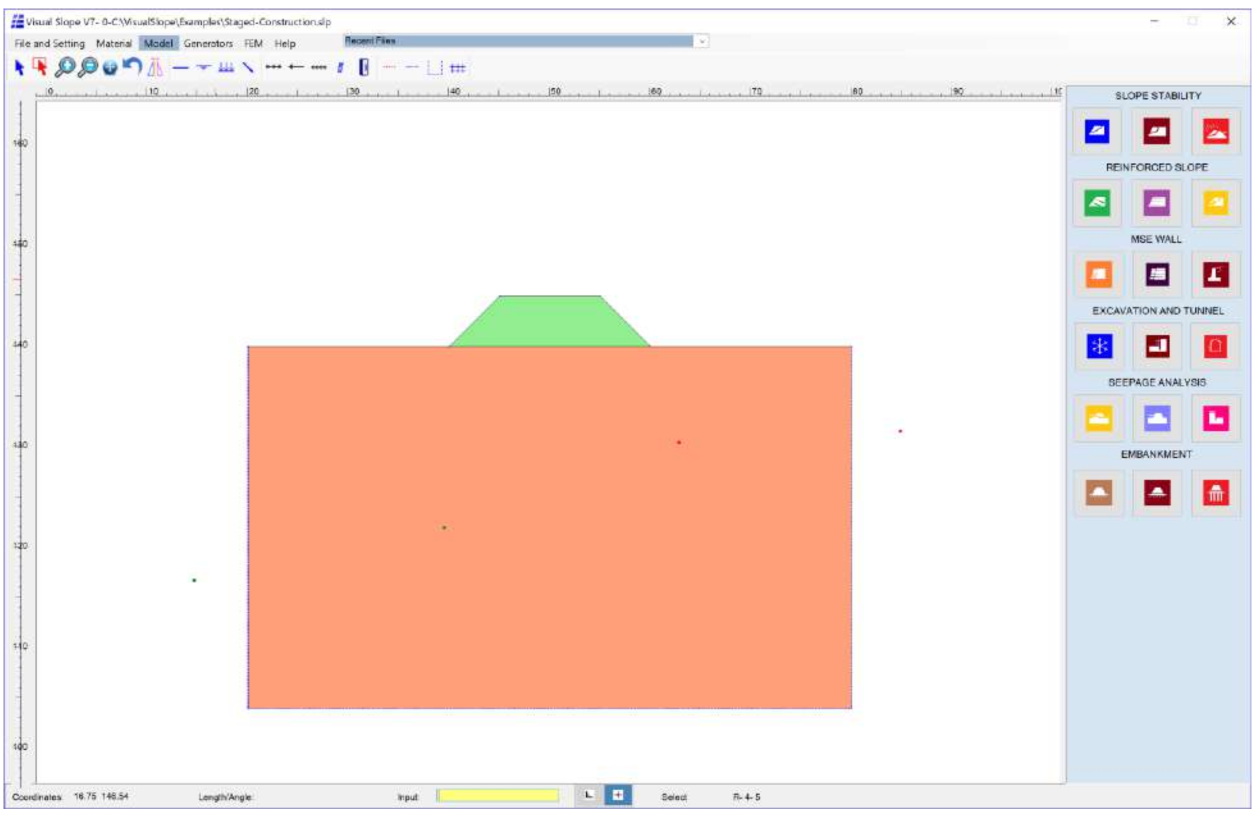


Figure 69: Staged Construction Example

Mesh Generation

From the *Generate Mesh* list, choose the *Basic Mesh* option to generate mesh, as shown in Figure 70.

Initial Stress and Movement Calculation

Prior to the fill placement, under the weight of ground soil, the ground has experienced stresses and movement, of which the vertical portion is settlement. Those portions of stress and movement are called initial stress and initial movement, respectively. To calculate the initial stress and movement, the embankment must be excluded. First, use the group select feature to select the embankment; then right-click the mouse button and choose the *Deactivate* option to deactivate the embankment, as shown in Figure 71. Then click the *FEM Calculation* button to calculate the initial conditions. The result of the initial conditions is shown in Figure 72.

Final Stress and Movement Calculation

After the initial conditions have been calculated, reactivate the embankment (see *Group Selection* for details) for the final calculation. The final calculation must include the initial stresses but exclude the initial movement, because only the settlement due to the fill placement is interested. To achieve that, the *Reset Displacement* option under the *FEM* menu must be set to *On*; then click the *FEM Calculation* button. The initial stress and the stress due to the fill placement together is the final stress, as shown in Figure 73. The final settlement only includes the settlement due to the fill placement, as shown in Figure 74.

Data Collection

To collect particular data, such as settlement along the base of the embankment, use the *Date Line* tool under FEM menu. Click the *Date Line* tool, and then draw a line along where the data need to be collected. The data will appear in the list. as shown in Figure 74. Use the *Copy Data* button in the data list page; the data can then be copied and pasted to a spreadsheet.

Video Example

This video example presents the procedure of FEM analyses for staged construction.

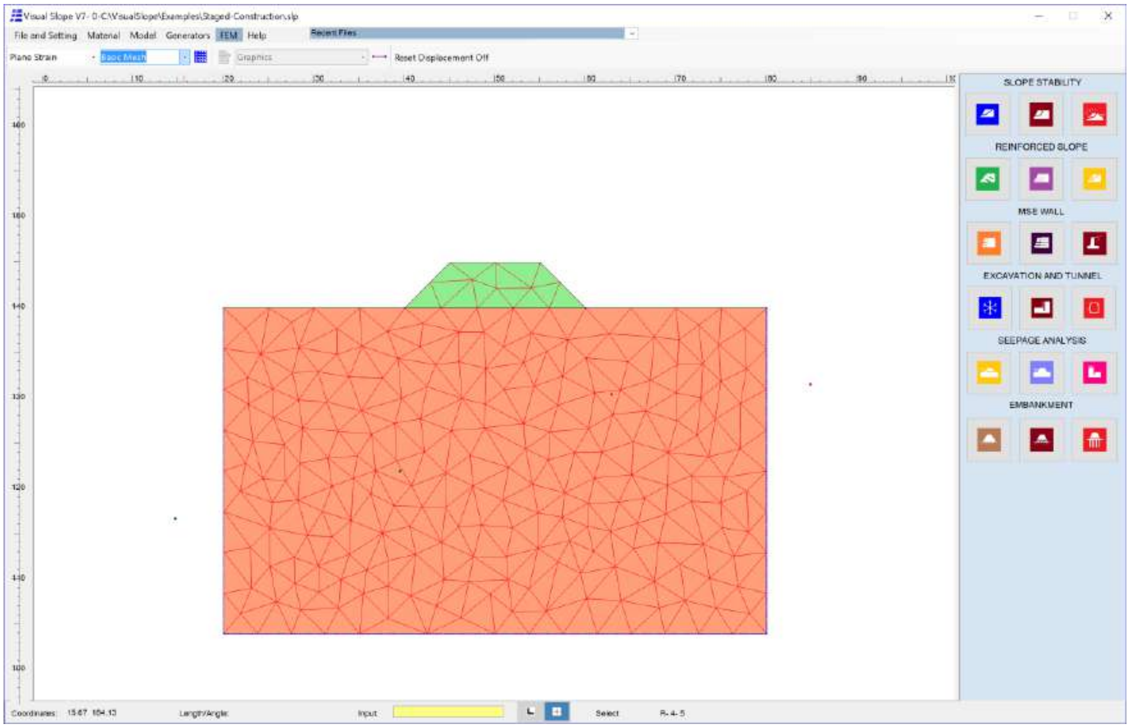


Figure 70: Mesh Generation

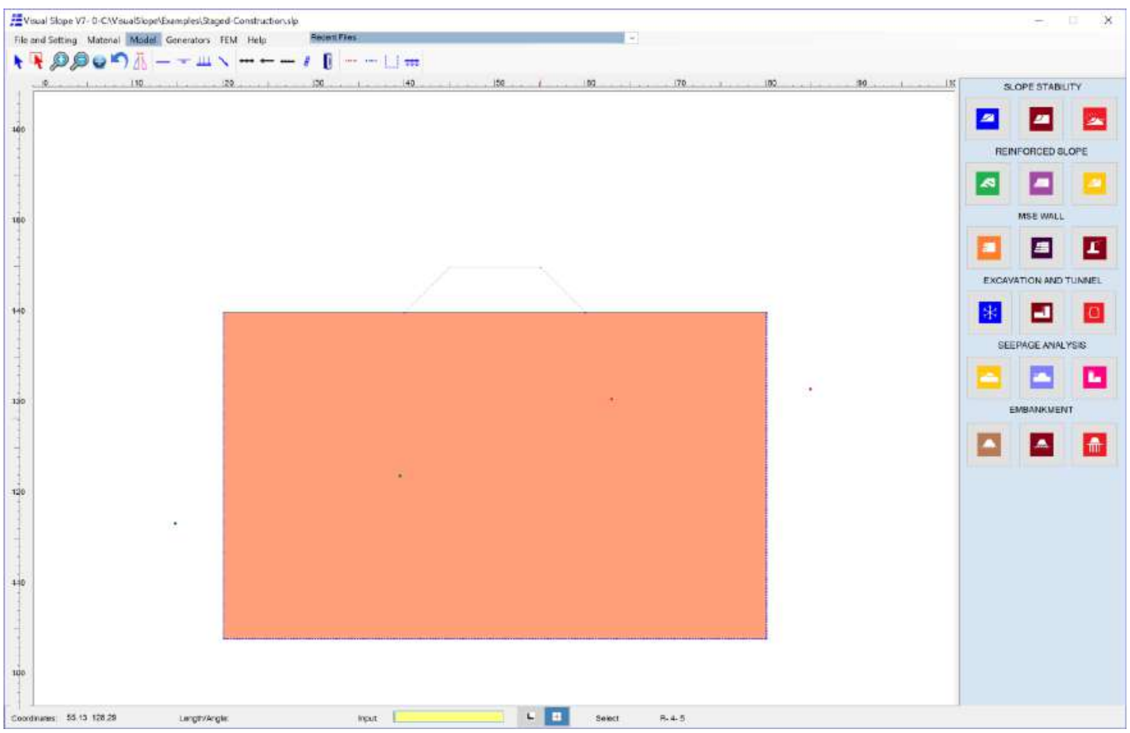


Figure 71: Deactivate Embankment

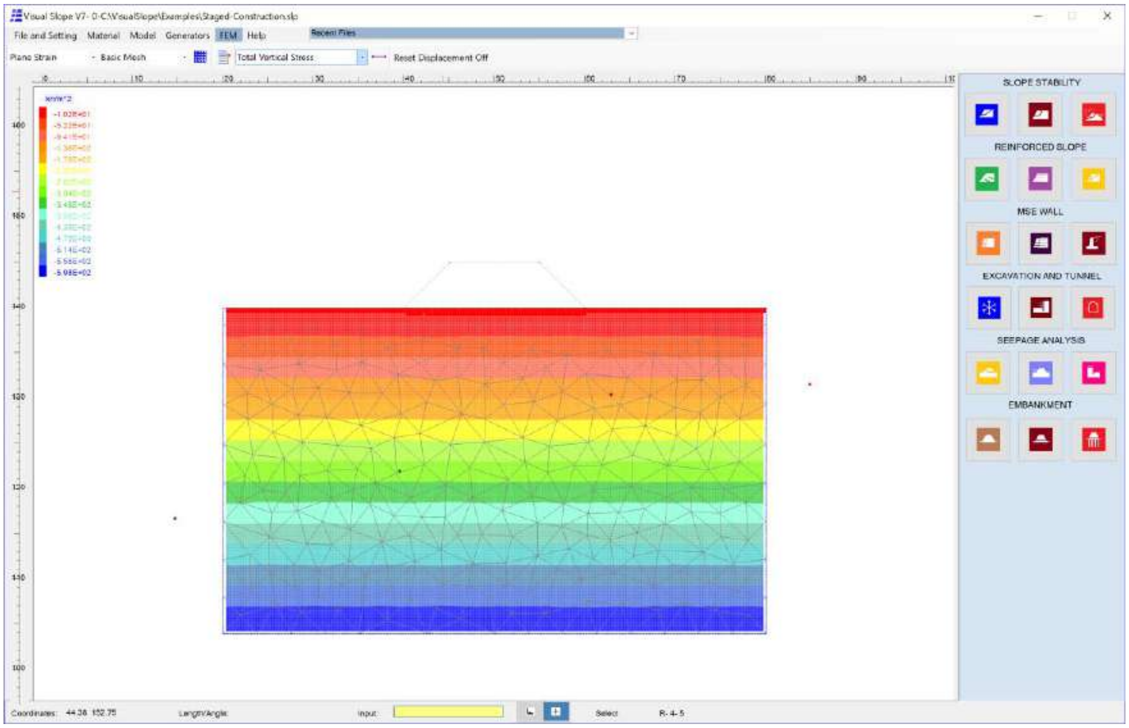


Figure 72: Initial Conditions

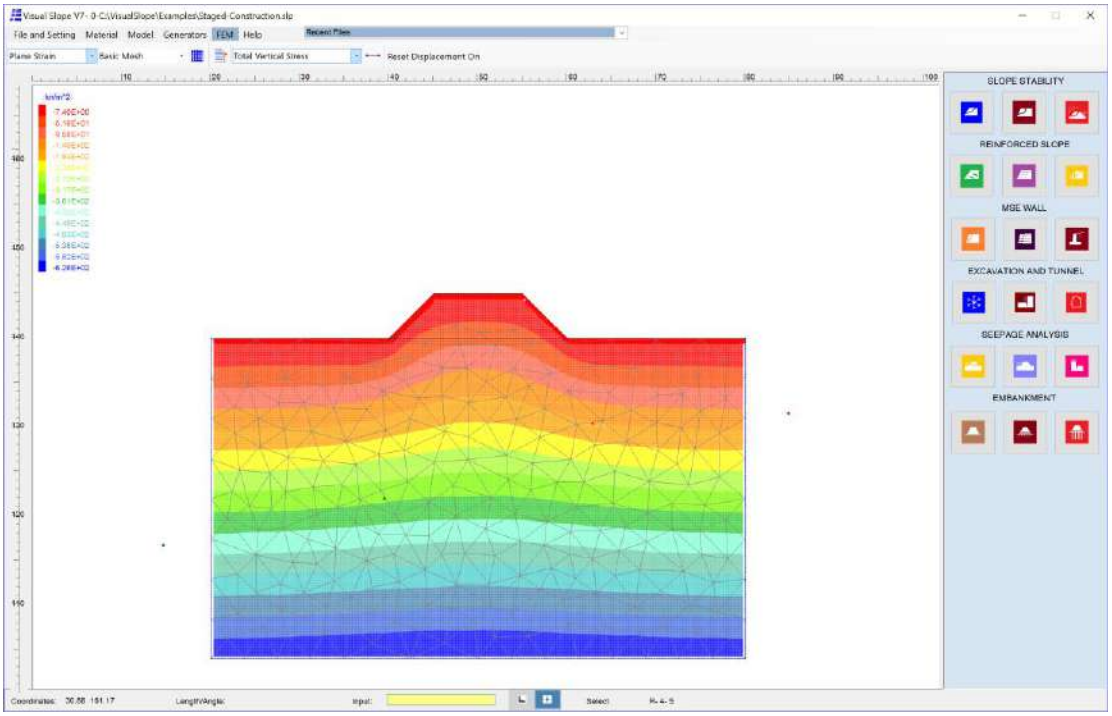


Figure 73: Final Vertical Stress

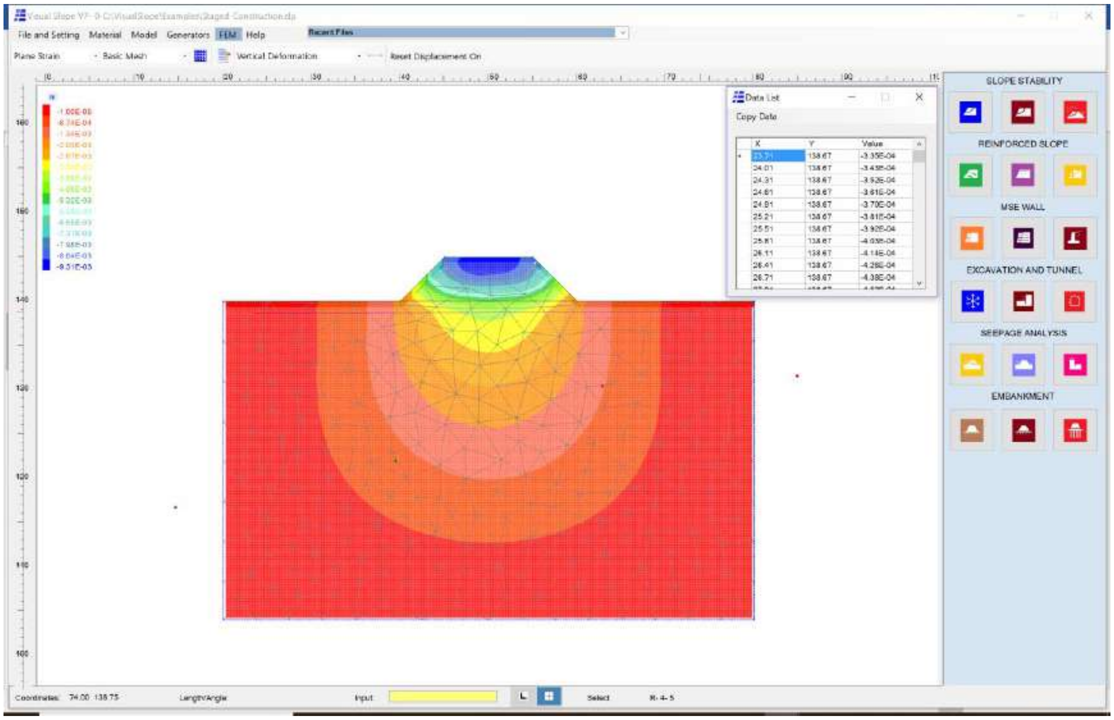


Figure 74: Final Settlement Due to Fill Placement

REFERENCES

Das, B. M. *Principles of Geotechnical Engineering.* 50th ed. Brooks/Cole Thomson Learning, 2002.

Department of the Army: Army Corps of Engineers. *Tunnels and Shafts in Rock*. EM 1110-2-2901, 1997.

Hinton, E. and D. R. J. Owen. *Finite Element Programming*. Academic Press, 1977.

Huang, Y. H. *Stability Analysis of Earth Slopes*. Van Nostrand Reinhold Company, 1972.

Kempfert, H. G. "German Recommendations for Reinforced Embankments on Pile-Similar Element." *Bauingenieur Band* 77, 2002.

National Concrete Masonry Association. *Design Manual for Segmental Retaining Walls.* 2nd ed. 1997.

U.S. Department of Transportation Federal Highway Administration. *Load and Resistance Factor Design (LRFD) for Highway Bridge Substructures*. Publication No. FHWA HI-98-032, 2001.

U.S. Department of Transportation Federal Highway Administration. *Soil Nail Walls Reference Manual*. Publication No. FHWA-NHI-14-007, 2015.