1 **FLAC/SLOPE**

1.1 Introduction

1.1.1 Overview

**FLAC/Slope** is a mini-version of **FLAC** that is designed specifically to perform factor-of-safety calculations for slope stability analysis. This version is operated entirely from **FLAC**’s graphical interface (the GIIC) which provides for rapid creation of models for soil and/or rock slopes, and solution of their stability condition.

**FLAC/Slope** provides an alternative to traditional “limit equilibrium” programs for determining factor of safety. Limit equilibrium codes use an approximate scheme (typically based on the method of slices) in which a number of assumptions are made (e.g., the location and angle of interslice forces). Several assumed failure surfaces are tested, and the one giving the lowest factor of safety is chosen. Equilibrium is only satisfied on an idealized set of surfaces.

In contrast, **FLAC/Slope** provides a full solution of the coupled stress/displacement, equilibrium and constitutive equations. Given a set of properties, the system is determined to be stable or unstable. By automatically performing a series of simulations while changing the strength properties (“shear strength reduction technique” – see Section 1.5), the factor of safety can be found to correspond to the point of stability, and the critical failure (slip) surface can be located.

**FLAC/Slope** does take longer to determine a factor of safety than a limit equilibrium program. However, with the advancement of computer processing speeds (e.g., 1 GHz and faster chips), solutions can now be obtained in a reasonable amount of time. This makes **FLAC/Slope** a practical alternative to a limit equilibrium program, and provides advantages over a limit equilibrium solution (e.g., see Dawson and Roth 1999, and Cala and Flisiak 2001):

1. Any failure mode develops naturally; there is no need to specify a range of trial surfaces in advance.
2. No artificial parameters (e.g., functions for interslice force angles) need to be given as input.
3. Multiple failure surfaces (or complex internal yielding) evolve naturally, if the conditions give rise to them.
4. Structural interaction (e.g., rockbolt, soil nail or geo-grid) is modeled realistically as fully coupled deforming elements, not simply as equivalent forces.
5. The solution consists of mechanisms that are kinematically feasible. (Note that the limit equilibrium method only considers forces, not kinematics.)
1.1.2 Guide to the FLAC/Slope Manual

This volume is a user’s guide to FLAC/Slope. The following sections in the introduction, Sections 1.1.3 through 1.1.5, discuss the various features available in FLAC/Slope, outline the analysis procedure, and provide information on how to receive user support if you have any questions about the operation of FLAC/Slope. Also, in Section 1.1.6, we describe the concept of a “mini-version” of FLAC.

Section 1.2 describes the step-by-step procedure to install and start up FLAC/Slope, and provides a tutorial (in Section 1.2.2) to help you become familiar with its operation. We recommend that you run this tutorial first to obtain an overall understanding of the operation of FLAC/Slope.

The components of FLAC/Slope are described separately in Section 1.3. This section should be consulted for detailed descriptions of the procedures of operating FLAC/Slope.

Several slope stability examples are provided in Section 1.4. These include comparisons to limit analysis and limit equilibrium solutions.

FLAC/Slope uses the procedure known as the “strength reduction technique” to calculate a factor of safety. The basis of this procedure and its implementation in FLAC/Slope are described in Section 1.5.

1.1.3 Summary of Features

FLAC/Slope can be applied to a wide variety of conditions to evaluate the stability of slopes and embankments. Each condition is defined in a separate graphical tool.

1. The creation of the slope boundary geometry allows for rapid generation of linear, non-linear and benched slopes and embankments. The Bound tool provides separate generation modes for both simple slope shapes and more complicated non-linear slope surfaces. A bitmap or DXF image can also be imported as a background image to assist boundary creation.

2. Multiple layers of materials can be defined in the model at arbitrary orientations and non-uniform thicknesses. Layers are defined simply by clicking and dragging the mouse to locate layer boundaries in the Layers tool.

3. Materials and properties can be specified manually or from a database in the Material tool. At present, all materials obey either the Mohr-Coulomb or ubiquitous-joint yield model, and heterogeneous properties can be assigned. Material properties are entered via material dialog boxes that can be edited and cloned to create multiple materials rapidly. Material models are described in Section 2 in Theory and Background of the FLAC manual.

4. With the Interface tool, a planar or non-planar interface, representing a joint, fault or weak plane, can be positioned at an arbitrary location and orientation in the model. The
interface strength properties are entered in a properties dialog; the properties can be specified to vary during the factor-of-safety calculation, or remain constant.

Please be aware that FLAC/Slope is limited to slope configurations with no more than one interface. For analyses which involve multiple (and intersecting) interfaces or weak planes, full FLAC should be used. Interface logic in FLAC is described in Section 5 in Theory and Background of the FLAC manual.

5. An Apply tool is used to apply surface loading to the model in the form of either an areal pressure (surface load) or a point load.

6. A water table can be located at an arbitrary location by using the Water tool; the water table defines the phreatic surface and pore pressure distribution for incorporation of effective stresses and the assignment of wet and dry densities in the factor-of-safety calculation. Section 1.8.3 in Fluid-Mechanical Interaction of the FLAC manual describes the effective stress calculation procedure implemented in FLAC.

7. Structural reinforcement, such as soil nails, rockbolts or geo-textiles, can be installed at any location within the model using the Reinforce tool. Structural properties can be assigned individually for different elements, or groups of elements, through a Properties dialog. The reinforcement logic is based upon the cable element formulation in FLAC (see Section 1.4 in Structural Elements of the FLAC manual).

8. Selected regions of a FLAC/Slope model can be excluded from the factor-of-safety calculation. This is useful, for example, when studying complex slope geometries in which the user wishes to disregard selected regions, such as localized sloughing of the slope along the slope face.

1.1.4 Analysis Procedure

FLAC/Slope is specifically designed to perform multiple analyses and parametric studies for slope stability projects. The structure of the program allows different models in a project to be easily created, stored and accessed for direct comparison of model results.

A FLAC/Slope analysis project is divided into four stages. The modeling-stage toolbars for each stage are shown and described below.

Models Stage

Each model in a project is named and listed in a tabbed bar in the Models stage. This allows easy access to any model and results in a project. New models can be added to
the tabbed bar or deleted from it at any time in the project study. Models can also be restored (loaded) from previous projects and added to the current project. Note that the slope boundary is also defined for each model at this stage.

Build Stage

For a specific model, the slope conditions are defined in the Build stage. These include: changes to the slope geometry, addition of layers, specification of materials and weak plane (interface), application of surface loading, positioning of a water table and installation of reinforcement. Also, spatial regions of the model can be excluded from the factor-of-safety calculation. The build-stage conditions can be added, deleted and modified at any time during this stage.

Solve Stage

In the Solve stage, the factor of safety is calculated. The resolution of the numerical mesh is selected first (coarse, medium, fine or user-specified), and then the factor-of-safety calculation is performed. Different strength parameters can be selected for inclusion in the strength reduction approach to calculate the safety factor. By default, the material cohesion and friction angle are used.

Plot Stage

After the solution is complete, several output selections are available in the Plot stage for displaying the failure surface and recording the results. Model results are available for subsequent access and comparison to other models in the project.

All models created within a project, along with their solutions, can be saved, the project files can be easily restored, and the results viewed at a later time.
1.1.5 User Support

We believe that the support Itasca provides to code users is a major reason for the popularity of our software. We encourage you to contact us when you have a modeling question. We provide a timely response via telephone, electronic mail or fax. General assistance in the installation of FLAC/Slope on your computer, plus answers to questions concerning capabilities of the various features of the code, are provided free of charge. Technical assistance for specific user-defined problems can be purchased on an as-needed basis.

We can provide support in a more timely manner if you include an example FLAC/Slope model that illustrates your question. This can easily be done by including the project save file (i.e., the file with the extension “*.PSL”) as an email attachment with your question. See Section 1.3.2 for a description of the “*.PSL” file.

If you have a question, or desire technical support, please contact us at

Itasca Consulting Group Inc.
Mill Place
111 Third Avenue South, Suite 450
Minneapolis, Minnesota 55401 USA

Phone: (+1) 612-371-4711
Fax: (+1) 612-371-4717
Email: software@itascacg.com
Web: www.itascacg.com

We also have a worldwide network of code agents who provide local technical support. Details may be obtained from Itasca.

1.1.6 FLAC Mini-Version

The basis for FLAC/Slope is FLAC, Itasca’s numerical modeling code for advanced geotechnical analysis of soil, rock and structural support in two dimensions. FLAC/Slope actually runs FLAC, and the GIIC limits access to only those specific features in FLAC used for the slope stability calculations. That is why we call FLAC/Slope a mini-version of FLAC.

When you install FLAC/Slope, the full version of FLAC is also installed. If you wish, you may start up FLAC and evaluate its operation and features. See the installation and start-up instructions given below in Section 1.2.1.*

* You may also obtain a demo version of FLAC that can run small models (limited to approximately 600 Mohr-Coulomb zones) without a hardware lock. Contact Itasca if you would like a demo version. If you decide to upgrade to the full FLAC, it is only necessary to upgrade your hardware lock in order to operate FLAC as well as FLAC/Slope. Then, the full power of FLAC will also be available to you.
1.2 Getting Started

1.2.1 Installation and Start-Up Procedures

System Requirements – To install and operate FLAC/Slope, be sure that your computer meets the following minimum requirements:

1. At least 35 MB of hard disk space must be available to install FLAC/Slope. We recommend that a minimum of 100 MB disk space be available to save model project files.

2. For efficient operation of FLAC/Slope, your computer should have at least 128 MB RAM.

3. The speed of calculation is directly related to the clock speed of your computer. We recommend a computer with at least a 2 GHz CPU for practical applications of FLAC/Slope.

4. FLAC/Slope is a 32-bit software product. Any Intel-based computer capable of running Windows XP or later is suitable for operation of the code.

Installation Procedure – FLAC/Slope is installed in Windows from the Itasca CD-ROM using standard Windows procedures. Insert the Itasca CD in the appropriate drive. The installation procedure will begin automatically if the “autorun” feature on the drive is enabled. If not, enter “[cd drive]:\start.exe” on the command line to begin the installation process. The installation program will guide you through the installation. Make your selections in the dialogs that follow. Please note that the installation program can install all of Itasca’s software products. You can install FLAC/Slope either by clicking on the FLAC/Slope item or by clicking on the FLAC box in the Select Components dialog.*

By default, the electronic FLAC/Slope manual will be copied to your computer during the installation of FLAC/Slope. (After FLAC has been selected in the Select Components dialog, the option not to install the manual can be set by using the Change button.) To use the electronic manual, click on the FLAC Slope Manual icon in the “Itasca” group on the “Start” menu. All electronic volumes of the FLAC manual (including the FLAC/Slope manual) are PDF files that require the Adobe Acrobat Reader in order to be viewed. This software is freely available from Adobe Systems Incorporated.

The FLAC/Slope package can be uninstalled via the Add/Remove Programs icon in the Windows Control Panel.

When the installation is finished, a file named “INSTNOTE.PDF” will be found in the program sub-folder (“FLAC600”) that resides in the main installation folder. (This is the folder that is

* The full version of FLAC is also installed when FLAC/Slope is installed.
specified during the installation process as the location to which files will be copied; by default, this is “\ITASCA.”) The “INSTNOTE.PDF” file provides a listing of the directory structure that is created on installation, and a description of the actions that have been performed as part of the installation. This information may be used, in the unlikely event that it is necessary or desirable, to either manually install or manually uninstall FLAC/Slope. The specific directories related to FLAC/Slope are described below.

- The “\FLAC600” directory contains the files related to the operation of FLAC/Slope. There are three sub-directories: “FLAC600\EXE” contains the executable code that is loaded to run FLAC/Slope; “FLAC600\FLAC\SLOPE” contains the example files described in this manual; and “FLAC600\GUI” contains files used in the operation of the GIIC.

- The “\SHARED\JRE” directory contains the JAVA Runtime Environment (standard edition 1.5.0) that is used for operating the GIIC.

- The “\MANUALS\FLAC600” directory contains the complete FLAC manual, which includes the FLAC/Slope manual.

The first time you load FLAC/Slope you will be asked to specify a customer title. This title will appear on all hardcopy output plots generated by FLAC/Slope. The title information is written to the system registry. If you wish to rename the customer title at a later time, click on the **File / Customer Information** menu item.

Finally, be sure to connect the FLAC/Slope hardware key to your USB port before beginning operation of the code.
Start-Up – The default installation procedure creates an “Itasca” group with icons for FLAC/Slope and FLAC. To load FLAC/Slope, simply click on the FLAC/Slope icon. The code will start up and you will see the main window as shown in Figure 1.1.

The code name and current version number are printed in the title bar at the top of the window, and a main menu bar is positioned just below the title bar. The main menu contains File, Show, Tools, View and Help menus. Beneath the main menu bar is the Modeling Stage toolbar, containing modeling-stage tabs for each of the stages: Models, Build, Solve and Plot. When you click on a modeling-stage tab, a set of tools becomes available: these tools are used to create and run the slope stability model. Separate sets of tools are provided for the models stage, the build stage, the solve stage and the plot stage (as discussed previously in Section 1.1.4).

Figure 1.1 The FLAC/Slope main window

Beneath the Modeling Stage toolbar is the model-view pane.* The model-view pane shows a graphical view of the model.

* If you are a user of full FLAC, you will also have access to a Console pane and Record pane. The Console pane shows text output and echoes the FLAC commands that are created when operating FLAC/Slope. This pane also allows command-line input (at the bottom of the pane). The Record pane contains a list of all the FLAC commands, which can be exported to a data file for input into full FLAC. The Console and Record panes are activated from the Show/ Resources menu item.
Directly above the *model-view* pane is a *View* toolbar. You can use the *View* tools to manipulate the *model-view* pane (e.g., translate or rotate the view, increase or decrease the size of the view, turn on and off the model axes). The *View* tools are also available in the *View* menu.

Whenever you start a new project, a *Model Options* dialog (shown in Figure 1.2) will appear in front of the *FLAC/Slope* main window. You have the option to include different features in the model, such as an interface (weak plane), a water table or reinforcement, exclude regions from the safety factor calculation, specify nonstandard gravity, and select the system of units for your project with this dialog. You can also choose to run in plane-strain mode (default) or in axisymmetric mode. Finally, you can open an existing *FLAC/Slope* model project from this dialog.

![Model Options dialog](image)

**Figure 1.2** The *FLAC/Slope* Model Options dialog

The menus and tools are described in detail in Section 1.3. An overview of the *FLAC/Slope* operation is provided in the *Help* menu. This menu also contains a list of Frequently Asked Questions about *FLAC/Slope*, and an index to all *GHC Help* files.
1.2.2 A Simple Tutorial

This section presents a simple tutorial to help you begin using FLAC/Slope right away. By working through this example, you will learn the recommended procedure to (1) define a project that includes different models; (2) build the slope conditions into each model; (3) calculate the factor of safety for each model; and (4) view the results.

The example is a simple slope in a layered soil. Figure 1.3 illustrates the conditions of the slope. The purpose of the project is to evaluate the effect of the water table on the stability of the slope. The project consists of two models: one model with a water table, and one without. In the following sections we discuss the four stages in the solution procedure for this problem.

If you have not done so already, start up FLAC/Slope following the instructions in Section 1.2.1. You will see the main FLAC/Slope window as shown in Figure 1.1 and the Model Options dialog as shown in Figure 1.2. You can now begin the tutorial.

### Materials

<table>
<thead>
<tr>
<th></th>
<th>Upper Soil</th>
<th>Lower Soil</th>
</tr>
</thead>
<tbody>
<tr>
<td>unsaturated density</td>
<td>1500 kg/m³</td>
<td>1800 kg/m³</td>
</tr>
<tr>
<td>saturated density</td>
<td>1800 kg/m³</td>
<td>2100 kg/m³</td>
</tr>
<tr>
<td>cohesion</td>
<td>5,000 Pa</td>
<td>10,000 Pa</td>
</tr>
<tr>
<td>friction angle</td>
<td>20°</td>
<td>25°</td>
</tr>
<tr>
<td>tensile strength</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

**Figure 1.3 Conditions of the simple slope**
Defining the Project – We begin the project by checking the box in the Model Options dialog. The water table tool will be made available for our analysis. We also select the SI: meter-kilogram-second system of units. Press OK to include these options in the project analysis.

We now click on File/Save Project As ... to specify a project title, a working directory for the project and a project save file. The Project File dialog opens, as shown in Figure 1.4, and we enter the project title and project save file names. The working directory location for the project is selected in this dialog. In order to change to a specific directory, we press in this dialog. An Open dialog appears to allow us to change to the working directory of our choice. We specify a project save file name of “SLOPE” and note that the extension “.PSL” is assigned automatically (i.e., the file “SLOPE.PSL” is created in our working directory). We click OK to accept these selections.

Figure 1.4 Project File dialog

We next click on the Models tool and enter the Models stage to specify a name for the first model in our project. We click on New and use the default model name Model 1 that appears in the New Model dialog. There will be two models in our project: Model 1, which does not contain a water table; and Model 2, which does. We will create Model 2 after we have completed the factor-of-safety calculation for Model 1. (Note that, alternatively, we can create both models first before performing the calculation.)

There are several types of model boundaries available to assist us in our model generation. For this tutorial, we select the Simple boundary button.

When we press in the New Model dialog, an Edit slope parameters dialog opens and we enter the dimensions for our model boundary, as shown in Figure 1.5. Note that we click on Mirror Layout to reverse the model layout to match that shown in Figure 1.3. We click OK to view the slope boundary that we have created. We can either further edit the boundary, or accept it. We press OK to accept the boundary for Model 1. The layout for the Model 1 slope is shown in Figure 1.6*. A tab is also created with the model name (Model 1) at the bottom of the view. Also, note that an icon is shown in the upper-left corner of the model view, indicating the direction and magnitude of the gravity vector. The project save file name, title and model name are listed in the legend to the model view. Additional information will be added as we build the model.

* We have increased the font size of the text in the model view. We click on the File/Preference Settings ... menu item and change the font size to 16 in the Preference settings dialog.
Figure 1.5  Edit slope parameters dialog

Figure 1.6  Model 1 layout
Building the Model – We click on the **Build** tool tab to enter the Build stage and begin adding the slope conditions and materials to Model 1. We first define the two soil layers in the model. By clicking on the **Layers** button, we open the Layers tool. (See Figure 1.7.) A green horizontal line with square handles at each end is shown when we click on the mouse inside the slope boundary; this line defines the boundary between two layers. We locate this line at the level $y = 9$ m by right-clicking on one of the end handles and entering 9.0 in the Enter vertical level dialog. We press **OK** in the dialog and then **OK** in the Layers tool to create this boundary between the two layers. The result is shown in Figure 1.8.

![Figure 1.7 Layers tool](image)

*Figure 1.7 Layers tool*
There are two materials in the slope. These materials are created and assigned to the layers using the Material tool. After entering this tool, we first click on the Create button, which opens the Define Material dialog. We create the two materials, upper soil and lower soil, and assign the densities and strength properties using this dialog. (Note that after one material is created, it can be cloned using the Clone button, and then the properties can be modified to create the second material.) The properties assigned for the upper soil material are shown in Figure 1.9. (A Class, or classification name, is not specified; this is useful if materials are stored in a database – see Section 1.3.5.)

Please be aware that we enter the (mass) density of the material, and not the unit weight. The relation between density, \( \rho \), unit weight, \( \gamma \), and gravitational magnitude, \( g \), is

\[
\rho = \frac{\gamma}{g}
\]

(1.1)

Note that Mass-Density and the system of units are shown in the dialog to emphasize that the input should be density and not unit weight.

In the dialog shown in Figure 1.9, the in-situ density of the material above the water table (unsaturated density in Figure 1.3) is assigned under “Mass-Density,” and the in-situ density below the water table is input under “Wet Density.” The relation between unsaturated and saturated in-situ densities is discussed in Section 1.3.5.
Figure 1.9  Properties input in the Define Material dialog for upper soil

After the materials are created, they are assigned to the two layers. We highlight the material in the List pane, and then click on the model view inside the layer we wish to assign the material. The material will be assigned to this layer, and the name of the material will be shown at the position that we click on the mouse inside this layer. The result after both materials are assigned is shown in Figure 1.10. We press OK to accept these materials in Model 1.

Figure 1.10  Materials assigned to the two layers in the MATERIAL tool
Calculating a Factor of Safety – We are now ready to calculate the factor of safety. We click on the [Solve] tool tab to enter the factor-of-safety calculation stage. When we enter this stage, we must first select a numerical mesh for our analysis. We choose a “fine-grid” model by pressing the [Fine] button, and the grid used for the FLAC solution appears in the model view. See Figure 1.11.

![Figure 1.11 Fine grid for Model 1](image)

We now press the [Solve FoS] button to begin the calculation. A Factor-of-Safety parameters dialog opens (Figure 1.12), we accept the default solution parameters, and press [OK]. FLAC/Slope begins the calculation mode, and a Model cycling dialog provides a status of the solution process. When the calculation is complete, the calculated factor of safety is printed; in this case, the value is 1.56.

![Figure 1.12 Factor-of-Safety parameters dialog](image)
Viewing the Results – We now click on the [Plot] tool tab to view the results. An [fc] button is shown, corresponding to the solution conditions (fine-zoned grid, friction angle and cohesion included in the calculation). When we click on this button, we view the factor-of-safety plot for this model, as shown in Figure 1.13:

![Factor-of-safety plot for medium-grid Model 1](image)

This plot indicates the type of failure that would develop when the cohesion and friction angle are reduced to the state that is the onset of failure. Failure is indicated by two overlaid plots: shear-strain rate contours and velocity vectors. The shear-strain rate contours delineate the location of the failure surface, and the velocity vectors indicate the failure mode (e.g., rotational failure).

The value for factor of safety is also printed in the plot legend. This is the ratio of the in-situ strength properties to the reduced properties at the onset of failure (see Eqs. (1.7) and (1.8) in Section 1.5).

Performing Multiple Analyses – We wish to compare this result to the case with a water table. We click on the [Models] tool tab to create the second model. We will start with Model 1 conditions by clicking on the [C] button. An Input dialog will appear again, but this time the default model name is copy of Model 1. We enter “Model 2” and accept this name by pressing [OK]. A Model 2 tab is now shown at the bottom of the view. All of the model conditions from Model 1 have been copied into Model 2. The only condition left to add is the water table. We go to the [Build] stage and click on the [Water] button. A horizontal line with square handles is shown in the Water tool. We position this line to match the location of the water table as shown in Figure 1.3. The line can either be repositioned by left-clicking the mouse on the line and dragging the line to the water table location, or by right-clicking the mouse on the line, which opens a dialog to specify coordinates.
of the water table. We define the water table by five points at coordinates: (0,10), (10,10), (20,8), (35,3) and (45,3). The result is shown in Figure 1.14:

![Figure 1.14 Positioning water table in the Water tool](image)

We are now ready to solve Model 2, so we go to the Solve stage, select the fine-grid model and press the Solve FOS button. To determine the factor of safety, we follow the same procedure as before. A factor of 1.46 is shown when the calculation stops. We now go to the Plot stage to produce the factor-of-safety plot for this model. The result is shown in Figure 1.15. Note that the water table is added to this plot by opening a factor-of-safety Plot items dialog via the ITEMS button. The results for Model 2 can easily be compared to those for Model 1 by clicking on the model-name tabs at the bottom of the model view.
**Making Hardcopy Plots** – Several different printer formats are available to create plots from FLAC/Slope. We click on the **Setup** button in the **Plot** toolbar to open a *Print setup* dialog, as shown in **Figure 1.16**:

![Print setup dialog](image)

**Figure 1.16  Print setup dialog**

For example, we have two choices if we wish to create a plot in an enhanced metafile format for insertion into a Microsoft Word document:
(1) We can click on the Enhanced Metafile radio button. We select the name of the file and the directory in which to save the file by using the File radio button. As shown in Figure 1.16, we save the factor-of-safety plot to a file named “MODEL2.EMF.” We press OK to save these printer settings. Then, we press Print in the Plot tool to send the plot to this file.

(2) Alternatively, we can copy the plot to the clipboard by clicking the Clipboard button. We press OK to save this setting. Then, press Print in the Plot tool to send the plot to the clipboard and, finally, paste the plot directly into the Word document.

The plot is shown in Figure 1.17. Note that hardcopy plots are formatted slightly differently than the screen plots.

![Figure 1.17 Hardcopy plot for Model 2 result](image)

It is also possible to create tables that summarize the results of the study. Click on the File/Create Report... menu item to open the dialog as shown in Figure 1.18. This will create an HTML-formatted file listing various information and plots for the study. For example, by selecting the Material property table and Solution table items in the dialog, tables listing the material properties and the calculated factors-of-safety will be produced, as shown in Figure 1.19.
This completes the simple tutorial. We recommend that you try additional variations on this project to help increase your understanding. For example, if you wish to evaluate the effect of zoning on the calculated safety factor, return to the Solve stage for Model 1 and click on the Medium button. This will create a medium-mesh model. After solving for the factor of safety, a new plot button will be added in the Plot toolbar for Model 1. You can then compare the result for a fine mesh directly with the medium mesh result by clicking on the plot buttons. The factor of safety for the medium-mesh model will be very close to that for the fine-mesh model. However, the failure surface will not be as well defined by the shear-strain rate contour plot. See Section 1.3 for more information on the components of FLAC/Slope and recommended procedures to perform slope stability calculations.
1.3 Details on Using FLAC/Slope

FLAC/Slope is designed to perform a series of analyses for a slope stability project. A parametric study involving several model simulations can easily be set up, executed, and the results viewed. Each model simulation involves four modeling stages: Models, Build, Solve and Plot. Several tools are associated with each stage to assist with the model analysis. Each of the tools is described in the following sections.

1.3.1 Selecting Model Options

When you first begin a FLAC/Slope analysis you will see a Model Options dialog box, as shown in Figure 1.20. The Model Options dialog will appear every time you start FLAC/Slope or begin a new project. The dialog allows different conditions and optional facilities to be set for the project.

You can select the system of units for your analysis from this dialog. Parameters in the model will then be labeled with the corresponding units, and predefined values, such as gravitational magnitude and material properties in the material database, will be converted to the selected system. A selection for system of units must be done at the beginning of the analysis.

When the INCLUDE STRUCTURAL ELEMENTS or INCLUDE INTERFACE box is checked, the corresponding tool is added to the BUIL toolbar. See Section 1.3.6 for a description of the interface tool, Section 1.3.7 for a description of the water table tool, and Section 1.3.9 for a description of the structural elements tool.

By default, the standard value for gravitational acceleration is used in the analysis. A gravity icon will appear in the model view (when the model is created), with a gravitational vector pointing downward and magnitude corresponding to the selected system of units. If you check the box NONSTANDARD GRAVITY, you will be able to assign a gravitational acceleration magnitude and direction of

Figure 1.20  Model Options dialog

FLAC Version 6.0
your choosing from a Gravity tool in the Build toolbar. Note that pseudostatic accelerations can be applied by using non-vertical gravity (see Section 1.4.7).

It is possible to exclude selected spatial regions of the model from the factor-of-safety calculation by selecting the Allow excluded regions from fos? box. A tool will then be added to the Build toolbar to allow excluded regions to be delineated in the model.

By default, a two-dimensional plane-strain analysis is performed. Alternatively, by clicking the Axisymmetric model? box, you can perform an axisymmetric analysis. In this mode, cylindrical coordinates are used; \( x = 0 \) is the axis of symmetry, the positive \( x \)-direction corresponds to the radial coordinate, the \( y \)-direction to the axial coordinate, and the out-of-plane direction (the \( z \)-direction) to the circumferential coordinate. This geometry mode may be applied, for example, to cylindrical-shaped mounds or circular open pits.

After you have selected which model options you wish to apply during your analysis, you can save these preferences so that these selections are active each time you start FLAC/Slope. Also, you can save your preferences for the look-and-feel of FLAC/Slope on start-up. You can select the size of the Model-view pane, and the layout for the modeling stage toolbar and the view toolbar. Open the Show menu in the main menu to change the look-and-feel of the FLAC/Slope pane and toolbars. Once you are satisfied, click File/Save Preferences in the main menu. The FLAC/Slope start-up preferences are stored in the file “STARTUP2.GPF,” located in the “ITASCA\FLAC600\GUI” directory.

1.3.2 Setting Up the Slope Project

When beginning a project, first select the File/Save Project As ... menu item in order to set up a project save file. This opens a Project File dialog as shown in Figure 1.21. The title and project save-file name selected for the project will be printed in the plot legend for each plot created in the project. The project save file will have the extension “*.PSL.” This file contains the project record and also allows access to all the model save states (saved as “*.SAV” files) and factor-of-safety calculation save states (saved as “*.FSV” files) for each model analysis in the project. Note that you can click on \( \textit{\textsuperscript{?}} \) in this dialog to select a directory in which to save the project and model-state save files.

![Figure 1.21 Project File dialog](image)

You can stop working on a project at any stage, save it (by pressing the File/Save Project menu item) and reopen it at a later time simply by opening the project file (from the File/Open Project
menu item); the entire project and associated model save and calculation save files will be accessible as before.

1.3.3 Creating a Slope Model

After you have set up the project save file, you can enter the Models stage of the analysis. In this stage, click on the [New] button to begin a new model analysis and assign a name to the model (the default name is Model 1). Model naming is done in the New Model dialog as shown in Figure 1.22. Note that you can also select the type of slope boundary to create for this model: a simple, linear boundary; or more complex boundaries, such as bench slope, dam embankment or nonlinear slope shapes. Advanced slope building is discussed in Section 1.3.13.

![Figure 1.22 New Model dialog](image)

If you select the [Simple] boundary and then press [OK], an Edit slope parameters dialog will open for you to input the dimensions for the simple slope model. This dialog is shown in Figure 1.23. A diagram is included in this dialog to guide you in the selection of geometry parameters. If you press [Apply] after inputting the parameters, the dialog will remain open and the slope boundary will be plotted. You can then make alterations to the boundary and view the results directly.

![Figure 1.23 Edit slope parameters dialog](image)
When selecting the dimensions for Depth, Left and Right, it is important that these dimensions are large enough that artificial boundaries (i.e., left, right and bottom boundaries) do not influence the development of the failure surface. If the final calculated slip surface is found to intersect any of these boundaries, then the model should be rerun with a larger dimension so that the surface does not intersect the boundary.

Please note that the coordinate axes for FLAC/Slope models are such that the axes origin is located at the bottom-left corner of the model, the y-axis is positive pointing upward in the vertical direction, and the x-axis is positive pointing to the right in the horizontal direction. The axes origin can be relocated by using the LowerLeftx and LowerLefty boxes in the Edit slope parameters dialog.

When you press OK, the dialog will close and the outline of the slope model will be drawn in a boundary view, as shown in Figure 1.24. The boundary can be edited further in this view, either by dragging the mouse to move the boundaries or by pressing the button to open the Edit slope parameters dialog again.

Figure 1.24  Boundary view

Once you are satisfied, press OK. The model boundary will now be drawn in the model view, as shown in Figure 1.25. Note that a tab with the model name will appear at the bottom of the model view when a model is created.
Several options are available once the model boundary is created. The model name can be changed with the Rename button. The model can also be removed from the project with the Delete button.

Individual models can be saved at this stage by pressing the Save button. A Save As dialog will open, and you can select a directory in which to save the model. The model file will automatically have the extension “.SLP.” You can then load this model into another project, if desired, by pressing the Load button; the loaded model will be automatically added to the model list for that project.

You can also make a copy of a model by using the Clone button. This will copy all information on the model into a new model; the Input dialog will open to assign a model name.

You can alter a model boundary using the Bound button in the Build toolbar. This will open the Edit slope parameters dialog, and allow changes to the boundary. However, this should be used with caution. For example, boundaries in a model should not be changed after layers, interfaces and/or a water table have been defined. These items will become invalid if the edge positions of the boundary are changed.
1.3.4 Adding Layers

If the slope stability analysis involves layered materials, layer boundaries should be defined first in the model. This is accomplished by clicking on the **Layers** button in the **Build** toolbar. The **Layers** tool will then open. To add layer boundaries in a model, click the mouse on a position within the model close to the location of the boundary between two layers. A green horizontal line with square handles at each end will appear. Figure 1.26 shows a model with two layer-boundary lines visible in the **Layers** tool.

![Figure 1.26 Slope model with two layer-boundary lines in the Layers tool](image)

Each line corresponds to a table of points that defines the location of the layer boundary. When the **Add/Move** radio button is pressed, lines can be added or moved within the model. To move a line, click and hold the left mouse button over one of the square handles, and drag the mouse in the vertical direction. The line will move up or down.

The shape of the boundary line can be modified by adding more handle points along the line, and then dragging these points to different positions. Click on the **Edit numerically** radio button to add points along the line. To select a line to edit, click on the line number in the **Layer boundaries** list and the selected line will turn white. For example, in Figure 1.27, the upper-layer boundary (boundary 1) has been edited by adding two points which are then dragged to new positions.

Handle points can be located at specific $x$- and $y$-coordinate positions by right-clicking the mouse over the handle. A **Table** dialog will open to enter the coordinates. The line tables can also be edited by clicking on the **Edit numerically** button. This opens an **Edit Table points** dialog in which the
$x$- and $y$-coordinates for all of the table points for the line are listed. Points can be input and edited in this dialog.

![Image of FLAC/Slope User's Guide](image)

**Figure 1.27** The upper layer-boundary line is edited to include two points

The *Layers* tool works best for sub-horizontal layers. However, it is possible to create models containing sub-vertical layers, provided certain rules are followed. The boundary lines must run continuously from the left model boundary to the right. In order to create a sub-vertical boundary, handle points are added along the line to create a vertical segment. For example, in **Figure 1.28**, a vertical column is created within a horizontal layer by adjusting the handle points along a boundary line to create a vertical segment. When creating this line, the handle points should be offset slightly from the existing horizontal lines so that the handle points of the new line do not coincide with those of the existing lines. (Note that in **Figure 1.28** there is a slight offset in the data points listed for the new line.) By doing this, each line will be uniquely defined. **Figure 1.29** displays the model with a vertical column of material (*mat 3*) located within the horizontal layer (*mat 2*).

Layer-boundary lines can extend beyond the boundary; upon tool execution, the lines will terminate at the boundary. Be careful to not make the layers too thin, because a bad zoning geometry may result when the model zoning is performed in the *Solve* stage. *FLAC* should be used to model more complex layering, involving, for example, pinched-out layers.
Figure 1.28  The new boundary line is offset slightly to avoid coinciding with the existing horizontal lines

Figure 1.29  Model with vertical column within horizontal layer
1.3.5 Assigning Materials and Properties

After all layer boundaries have been defined in the model, materials can be assigned to each layer. This assignment is a two-step process. First, the material is created and its associated properties are prescribed. Then, the material is assigned to a specific layer. Material creation and assignment are both done within the Material tool, which is opened by pressing the Material button.

Materials are created by clicking on the Create button to open the Define Material dialog. The dialog is shown in Figure 1.30. A material is defined by its classification and name (for example, classification: embankment soil; and name: silty sand). The classification is useful if you choose to create a database to store common materials to use on different projects. The database is accessed via the Database button located at the bottom-right corner of the Material tool. The database is described later in this section.

The (mass) density and material strength properties are assigned for each material. Note that the corresponding units for each property are shown in the dialog, depending on the system of units selected in the Model Options dialog. Density is specified in [mass/volume] units. This value times the gravitational magnitude is equal to the unit weight of the material ([weight/volume] units) (see Eq. (1.1)).

If the water table tool is not active, only the “unsaturated” (or “moist”) in-situ density is assigned. This is the density of the material above the water table in situ. If the water table tool is active, then either a porosity or a “saturated” (or “wet”) density must also be assigned. The relation between “saturated” and “moist” densities is defined in FLAC/Slope by the formula

$$\rho_{\text{wet}} = \rho + n \rho_w$$

where $\rho_{\text{wet}}$ is the wet in-situ density, $\rho$ is the moist in-situ density, $n$ is the porosity, and $\rho_w$ is the density of water. When the water table is assigned to the model, all zones with centroids located below the water table are assumed to be fully saturated, and will automatically be assigned the value for wet density for the factor-of-safety calculation.

Material failure is defined by either the Mohr Coulomb or ubiquitous-joint plasticity model. Both models require the intact material strength properties, cohesion and internal angle of friction. A tensile strength and dilation angle may also be specified for both material models. If associated plastic flow is specified for the analysis, the dilation angle will be automatically adjusted to match the friction angle. (See Section 1.3.11.2.) For the ubiquitous-joint model, joint cohesion, friction angle, tensile strength and dilation angle are also input. In addition, the orientation of the ubiquitous-joint set is specified. (The angle is measured counterclockwise from the x-axis.)

The elastic properties typically have an insignificant effect on the factor-of-safety calculation, and therefore these properties are not required as input. By default, the bulk modulus and shear modulus of all materials in the model (assuming SI units) are set to 100 MPa and 30 MPa, respectively.
If the material database button is pressed, a Material list dialog will open as shown in Figure 1.31. The database is divided into groups, designated by classification names, and shown in a collapsible tree structure. The database can be used to store sets of common materials and their properties for use on different projects. By default, a database of soil and rock materials is provided, as shown in the Database listed in the figure. Materials are selected from this list by double-clicking on a material name; the material will then be added to the Selection list. After choosing the materials for a project, press OK to send these materials to the List shown in the Material tool.
You can edit the properties in the database by pressing the "Edit" tab, which will switch from the Database pane to an Edit pane, as shown in Figure 1.32. Press "Apply" to apply the edited properties to the material. You can also create new materials with the "Create" button, and clone and delete materials in the list with the other buttons in the Material list dialog. You can store the altered or new materials back in the database by pressing the "Copy->Database" button.

The buttons beneath the Database list (shown in Figure 1.31) allow you to store this altered database as a new database file. By pressing "Save", a Save As dialog opens, and you can save your database with the extension "*.GMT." You can then load this database in a different project by pressing the "Load" button when working in this project.

Figure 1.31  Material list dialog
1.3.6 Adding a Weak Plane (Interface)

A weak plane or interface can be added to the slope model by clicking on the \textit{Interface} button in the \textit{Build} toolbar. This opens the \textit{Interface} tool, as shown in Figure 1.33. The tool contains a blue horizontal line with square handles at each end. The line corresponds to a table of points that define the location of the interface. The line can be positioned in the model in the following manner. By clicking on and dragging the square handles, the ends of the line can be moved up and down in the model. By clicking on points along the line, new handles can be added, and these handles can be moved to distort the line as needed to fit the interface location. Handle points can also be right-clicked with the mouse to open a \textit{Table} dialog to input $x$- and $y$-coordinates for the points. The interface-line table can also be edited by clicking on the \textit{Edit numerically} button, which opens an \textit{Edit Table points} dialog. The $x$- and $y$-coordinates for all of the table points for the line are listed; points can be input and edited in this dialog. Figure 1.34 shows the interface line repositioned with two handle points added along the line.

WARNING: Please note that only one interface can be included in the model. Also, the interface must be oriented such that it intersects the left and right boundaries of the model. Sub-vertical interfaces cannot be modeled in \textit{FLAC/Slope}. \textit{FLAC} should be used if it is necessary to model multiple or sub-vertical interfaces.
Figure 1.33  Interface tool

Figure 1.34  Interface line repositioned
After the interface is located in the model, interface properties should be prescribed. This is done by pressing the [Property Edit] button to open the Interface property list dialog. The dialog is shown in Figure 1.35. The interface is defined by a classification and name (e.g., classification: bedding plane; name: smooth). The interface properties are then prescribed to this interface material, and applied by pressing the [Apply] button. Several interface materials can be created at one time in this dialog. The highlighted material will be applied to the interface when [OK] is pressed. The interface material and properties are listed in the Properties list in the Interface tool.

![Interface property list dialog](image)

The behavior of the interface is defined by the Coulomb slip criterion, which limits the shear stress, $\tau_{\text{max}}$, along the interface by the relation

$$\tau_{\text{max}} = c_i + \sigma_n \tan \phi_i$$

(1.3)

where $c_i$ = cohesion (in stress units) along the interface, $\phi_i$ = friction angle of the interface surface, and $\sigma_n$ is the normal stress acting on the interface.

In addition, the interface may dilate at the onset of slip. Dilation is governed in the Coulomb model by a specified dilation angle, $\psi_i$.

If a tensile bond strength is specified for the interface, the interface acts as if it is glued, while the tensile normal stress acting on the interface is below the bond strength. If the tensile normal stress exceeds the bond strength, the bond breaks, and separation and slip can occur.

The elastic shear and normal stiffnesses associated with the interface behavior do not affect the solution for the factor of safety. Therefore, default values are assigned automatically to optimize the solution convergence. (See Section 5.4.1 in Theory and Background of the full FLAC manual for more information on the rationale for selection of stiffness values.)
1.3.7 Locating a Water Table

A water table can be added to the slope model by clicking on the Water button in the Build toolbar. This opens the Water table tool, as shown in Figure 1.36. The tool contains a blue horizontal line with square handles at each end. The line corresponds to a table of points that define the location of the water table (piezometric surface). The line can be positioned in the model in the following manner. By clicking on and dragging the square handles, the ends of the line can be moved up and down in the model. By clicking on points along the line, new handles can be added, and these handles can be moved to distort the line as needed to fit the water table location. Handle points can also be right-clicked with the mouse to open a Table dialog to input $x$- and $y$-coordinates for the points. The table can also be edited by clicking on the Edit numerically button, which opens an Edit Table points dialog. The $x$- and $y$-coordinates for all of the table points for the water table line are listed; points can be input and edited in this dialog. Figure 1.37 shows the water table line repositioned with two handle points added along the line.

![Figure 1.36 Water table tool](image)

The water table can be turned on and off in the model by clicking on the Use water table? box. The water density is assigned automatically with a value corresponding to the selected system of units. The density value can be set manually in the water density box.

When the water table is active, failure in the factor-of-safety calculation is controlled by the effective-stress state of the model. The value for water density is used in the calculation of the pore-pressure distribution, which is then applied to determine the effective stresses in all zones below the water table.
The location of the water table is also used to determine if unsaturated or saturated density is used to compute material weight. Unsaturated density is assigned to all zones in the model with zone centroids located above the water table, and saturated density is assigned to all zones with centroids below.

*Figure 1.37 Water table repositioned with two handle points added*

The water table can be located so that it intersects the slope surface and coincides with a reservoir level, such as the case shown in Figure 1.38. In this case, the weight of the water corresponding to the reservoir elevation is automatically included as a mechanical pressure acting on free surfaces below the reservoir line.
Figure 1.38 Water table repositioned to intersect the slope and coincide with a reservoir level

When we press OK to accept this location in the Water table tool, the surface water pressure is depicted in the model view by a pressure bar acting along the slope boundary. See Figure 1.39. When we click on the Solve tool tab to enter the Solve stage and create the zoned mesh for this model, the surface water pressure is shown in the model view by arrows located at gridpoints along the slope surface. The arrow lengths correspond to applied mechanical forces that are derived from the value for the water pressure times the boundary length associated with each gridpoint. Figure 1.40 shows arrows corresponding to the surface water pressure applied in Figure 1.39.
Figure 1.39  Water pressure acting along slope surface shown in model view

Figure 1.40  Applied forces corresponding to the surface water pressure applied in Figure 1.39
1.3.8 Applying Surface Loads

Point loads and areal stresses can be applied along a slope surface by clicking on the **Apply** button in the **Build** toolbar. This opens the **Apply** tool, as shown in **Figure 1.41**. Various forms of loads can then be applied to the slope surface; the types of loads are listed in a collapsible tree structure in the **B.C. types** pane in this tool. To apply a specific load, click on the name in the tree, and then click-and-drag the mouse over the portion of the boundary to which you wish to apply the load. For example, in **Figure 1.41**, a pressure is applied at the top of the slope along the region designated by the pressure bar.

![Figure 1.41 Apply tool](image)

When you release the mouse button, an **Assign** button becomes active. By clicking on this button, an **Apply value** dialog opens. A constant value or a linearly varying value can be applied for the boundary load. In **Figure 1.42**, a constant pressure of 10,000 is applied in the dialog. By pressing **OK**, the value is added to the **Applied List**. Several loads can be added in this manner.
If it is necessary to make a change to the applied value, highlight the apply type to be edited in the Applied List, and click on the Edit button. For example, if we wish to vary the pressure in the x-direction, we highlight the pressure, click on Edit, and make the change in the Apply value dialog. Figure 1.43 shows the dialog. The sign conventions and formula for applying a spatial variation in load are described below.

The applied loading types are divided into two categories, Stress and Force, in the Apply tool list. The stress types sxx, syy and sxy refer to stresses applied in the x-direction, y-direction or as a xy-shear stress along a specified boundary, respectively. Alternatively, a stress can be applied in the normal direction to the boundary with the nstress-type name or pressure-type name, and in
the shear direction with the \textit{sstress}-type name. The sign convention for the stress types, \textit{sxx}, \textit{syy} and \textit{nstress}, is that positive values indicate tension. The sign convention for shear stress types, \textit{sxy} and \textit{sstress}, is illustrated by Figure 2.42 in the \textit{User's Guide} of the full \textit{FLAC} manual. The sign convention for \textit{pressure} is that a positive pressure acts normal to, and in a direction toward, the surface of a body (i.e., push towards the free surface). \textit{pressure} and \textit{nstress} apply the same type of loading, but with an opposite sign convention.

When stresses are applied in \textit{FLAC/Slope}, they are converted into forces applied at boundary gridpoints after the zoned mesh is created. The applied forces are derived from the value for stress (or pressure) times the boundary segment length associated with each gridpoint.

Directional forces, \textit{xforce} and \textit{yforce} (shown in Figure 1.41), can also be applied to represent a point (i.e., line) load on the boundary. A positive \textit{x}- or \textit{y}-force acts in the positive \textit{x}- or \textit{y}-direction.

The sense of the applied stress or force can be checked by entering the \textit{Solve} stage after pressing \textit{OK} to leave the \textit{Apply} tool. When the zoned mesh is created in this stage, the applied loading condition will be depicted on the model view by arrows with lengths corresponding to applied forces, acting at gridpoints along the model boundary. For example, Figure 1.44 illustrates the applied forces that correspond to the applied pressure variation prescribed in Figure 1.43.

\begin{figure}[h]
\centering
\includegraphics[width=\textwidth]{figure144.png}
\caption{\textit{Figure 1.44} Applied forces corresponding to the applied pressure in \textit{Figure 1.43}}
\end{figure}
The spatial variation in applied stress or force values is defined by the following formula. For a stress or force variation applied along a boundary within the range \( x = x^{(s)} \) to \( x^{(e)} \) and \( y = y^{(s)} \) to \( y^{(e)} \), then the applied stress or force magnitude value, \( v \) is

\[
v = v^{(s)} + \frac{x - x^{(s)}}{x^{(e)} - x^{(s)}} v_x + \frac{y - y^{(s)}}{y^{(e)} - y^{(s)}} v_y \tag{1.4}
\]

where \( (x^{(s)}, y^{(s)}) \) is the coordinate of the starting point, \( (x^{(e)}, y^{(e)}) \) is the coordinate of the ending point, \( v^{(s)} \) is the starting value of the stress or force entered under Value in the Apply value dialog, and \( v_x \) and \( v_y \) are the variation values entered under X-Y Variation in the dialog.

For the example defined by the Apply value dialog in Figure 1.43, the coordinate range is \( (x^{(s)}, y^{(s)}) = (5,14) \) and \( (x^{(e)}, y^{(e)}) = (10,14) \). The variation in pressure is only in the \( x \)-direction \( (v_y = 0) \). Thus, by using Eq. (1.4), we get

\[
v = 10,000 - 1,000(x - 5.0) \tag{1.5}
\]

When the zoned model is created, this pressure is converted into forces applied at gridpoints with a variation as depicted by the vector lengths shown in Figure 1.44.

**1.3.9 Installing Structural Reinforcement**

Structural element logic is provided in FLAC/Slope to simulate the effect of reinforcement in a slope or embankment. The FLAC cable element is used to represent this reinforcement in FLAC/Slope. See Section 1.4 in Structural Elements of the full FLAC manual for a detailed description of the cable element logic.

Reinforcement is installed in a slope by clicking on the Reinforce button in the Build toolbar. This opens the Reinforcement tool, as shown in Figure 1.45. Cable elements are installed in a slope by first checking the Add bolt radio button, and then pressing the mouse button at one endpoint of the cable, dragging the mouse to the other endpoint, and then releasing the button. A yellow line with square white handles will be drawn, as shown in Figure 1.45. Any number of cables can be installed within the slope in this manner.

The end nodes of the cable can be positioned more precisely by right-clicking on the handles. This opens a Coordinate dialog to enter \( x \)- and \( y \)-coordinates of the end node. End nodes can also be relocated by checking the Move nodes radio button. Then, press and drag the end node with the mouse. Cables can be deleted from the slope by checking the Delete radio button. You can then click the mouse over the cable(s) you wish to delete.
After the reinforcement is installed in the slope, the next step is to assign material properties to the reinforcement. This is done by checking the Properties radio button. Properties are assigned to cable elements in FLAC/Slope via a property identification number. This number will appear over each cable when the Properties button is pressed. By default, all cables are given the property number C1. See Figure 1.46.

*Figure 1.45  Reinforcement tool*
Figure 1.46 Property identification number for reinforcement

By clicking the mouse over the property number, a *Bolt Properties* dialog will open, as shown in Figure 1.47. Properties are then assigned to a specific property number.

Figure 1.47 Bolt Properties dialog
Two types of reinforcement can be simulated: a continuous \textit{geosynthetic sheet} or a \textit{spaced reinforcement}. If spaced reinforcement is selected (e.g., to simulate soil nails), the spacing in the out-of-plane direction is also specified. The spacing parameter is used to automatically scale properties and parameters to account for the effect of the distribution of the cables over a regularly spaced pattern. (See Section 1.9.4 in \textbf{Structural Elements} of the full \textit{FLAC} manual for more information on the simulation of spaced reinforcement.) Please note that the actual properties of the cables, not scaled properties, are entered in the \textit{Bolt Properties} dialog.

You can input a Young’s modulus for the reinforcement, or you can choose to allow the modulus to be computed automatically to optimize the calculation process. It is recommended that, if the modulus of the reinforcement is two orders of magnitude or more greater than the elastic stiffness of the slope material, the computed value for modulus be selected. If the reinforcement modulus is more than two orders of magnitude greater than the slope material stiffness, the calculated factor of safety will be essentially the same for the input modulus as for the computed modulus, but the solution convergence will be very slow.

In addition to the Young’s modulus, the tensile yield strength and cross-sectional area of the reinforcement must be input. For a geosynthetic sheet, the area is equal to the thickness of the sheet times a unit depth in the out-of-plane direction.

The properties describing the shear interaction at the reinforcement/slope material interface are input under the \textit{Grout Material} heading in the dialog. These properties are prescribed in terms of a cohesive or bond strength and a bond friction angle.

The following relation is used to determine the maximum bond shear force, $F_{s}^{\text{max}}$, that can develop along the interface per length, $L$, of the cable:

$$
\frac{F_{s}^{\text{max}}}{L} = c_{b} + \sigma'_{c} \times \tan(\phi_{b}) \times p
$$

where:
- $c_{b}$ = bond strength or cohesion [force/cable length];
- $\sigma'_{c}$ = mean effective confining stress normal to the element;
- $\phi_{b}$ = bond friction angle [degrees]; and
- $p$ = perimeter of the element (based on input area).

See Section 1.4.1.2 in \textbf{Structural Elements} of the full \textit{FLAC} manual for more information on the shear behavior.

The elastic shear stiffness at the interface does not affect the calculation of the factor of safety. Therefore, it is computed automatically to optimize the solution convergence. (See Section 5.4.1 in \textbf{Theory and Background} of the full \textit{FLAC} manual for more information on the rationale for selection of stiffness values.)

The reinforcement properties are assigned to a property number (in Figure 1.47, this is C1). Additional property numbers can be created by pressing the [NEW] button in the \textit{Bolt Properties} dialog. A new property number, C2, will be added to the \textit{Property List}, and a different set of properties can be prescribed for that number. Several property sets can be created in this manner. The property
number that is highlighted in the Property List will be assigned to the active cable when OK is pressed.

Different segments along a cable can also be assigned different property numbers (e.g., to simulate bonded and un-bonded portions of a grouted bolt). Figure 1.48 shows a bolt composed of two segments. This is created in the Add Bolt mode by creating one segment and then clicking the mouse over one existing end node to start the second segment. The second segment will automatically be connected to the first. After checking Properties, we can then assign properties for the un-bonded segment to C1, and the bonded segment to C2. We change the left portion of the bolt in Figure 1.48 to C2 by highlighting C2 in the Bolt Properties dialog.

![Figure 1.48 Creating a grouted and un-grouted bolt](image)

Once we are satisfied with all the reinforcement conditions and properties we have specified, we click OK to accept this reinforcement in the model. The reinforcement will then be drawn in the model view.

Axial forces are calculated in the cables during the factor-of-safety calculation. These values can be added to the output plots (see Section 1.3.12). Please note that the sign convention for axial forces in cables is that forces are negative in tension. Also, note that cables in FLAC/Slope cannot sustain a load in compression.
1.3.10 Excluding Regions from the Factor-of-Safety Calculation

Regions of the model can be excluded from the factor-of-safety calculation by using the Exclude tool. When the Exclude button is pressed in the toolbar, the tool opens as shown in Figure 1.49. By clicking the mouse within the model boundary, a polygonal box with four corner handles opens. The handle points can be dragged to position the box to cover the excluded region. In Figure 1.49, the box is repositioned to cover a thin region along the slope face.

Multiple regions can be selected for exclusion, and corner handles can be edited to reposition regions. If Rectangle? is selected, the region is restricted to a rectangular shape. Regions can also be deleted.

After the excluded region is accepted, by pressing the OK button, the region boundary is shown in the model view. When we click on the Solve tool tab to enter the Solve stage and create a zoned mesh, the excluded zones will be identified, as shown in Figure 1.50.
The effect of using the *Exclude* tool is shown by comparing factor-of-safety results for the slope shown in Figure 1.49, with and without the excluding region. Figure 1.51 presents the factor-of-safety plot for the case without an excluded region, and Figure 1.52 shows the result with the excluded region. In the first case, the failure surface intersects the slope; in the second case, the failure surface extends below the slope and into the base.

Note that the *Exclude* tool only applies to zones in the *FLAC/Slope* model. Interfaces or structural reinforcement that lie within an excluded region will still be affected by the factor-of-safety calculation, if interface strength or grout strength parameters are selected as factor-of-safety parameters in the calculation.
**Figure 1.51**  Factor-of-safety calculation with no excluded region

**Figure 1.52**  Factor-of-safety calculation with excluded region
1.3.11 Solving for a Factor of Safety

The calculation for the factor of safety is done in the Solve stage, which is accessed from the Solve button. There are three steps in the Solve stage: grid generation, factor-of-safety parameter selection and factor-of-safety solution.

1.3.11.1 Grid Generation

When the Solve stage is entered, a numerical mesh must first be created. Four zoning choices are available: coarse, medium, fine and user-selected (special). These can be selected by pressing the different buttons shown in the Solve tool. For example, by pressing the Coarse button, a “coarse-grid” model is created, as shown in Figure 1.53. If the Medium button is pressed, a “medium-grid” model appears, as shown in Figure 1.54, and if the Fine button is pressed, a “fine-grid” model appears, as shown in Figure 1.55. The fineness of zoning affects the accuracy of the factor-of-safety calculation: the finer the zoning, the better the accuracy of the solution.

The coarse-grid model is recommended for preliminary analyses. The solution for this model is quite rapid: on a 2 GHz computer, a solution time is typically only a few seconds. A project with several models can easily be run to provide a quick estimate for the effect of different conditions on the factor of safety.

A medium-grid model is recommended for more detailed studies. The results for this type of zoning are found to be in reasonable agreement with limit analyses and limit equilibrium model results (e.g., see Sections 1.4.1 and 1.4.2). A medium-grid model takes longer to calculate the factor of safety: on a 2 GHz computer the solution typically requires a few minutes to complete.

A fine-grid model is recommended for comprehensive analyses and as a check on calculations made with the medium-grid model. The factor-of-safety calculation from the fine-grid model should agree very closely with that from the medium-grid model. The failure surface delineated by the concentration of shear strain contours will also be better defined with a fine-grid model than with a medium-grid model. However, the fine-grid model will take longer to calculate a safety factor than the medium-grid model. Comparisons between medium-grid and fine-grid models are recommended before conducting detailed parameter studies.
Figure 1.53  Coarse-grid model

Figure 1.54  Medium-grid model
Figure 1.55 Fine-grid model

For cases in which there are fairly irregular surfaces in the model (e.g., irregular slope surface, material boundary layers or interface), it may be necessary to use a “special” grid model. If a “bad geometry” message appears during the grid generation using the coarse-, medium- and fine-grid models, it will not be possible to perform a safety-factor calculation. In this case, a special-grid model should be applied using the user-defined Special zoning tool. This tool provides more control over the zoning parameters. (If there is still a problem with grid generation, then it will be necessary to return to the Build stage and adjust the irregular surface.)

For example, parameters are set in the Special zoning tool shown in Figure 1.56 for the model containing a vertical column within a horizontal layer, as described previously in Figure 1.29. These parameters are specified in order to create appropriate zoning for the irregular boundary layers. An 80-zone mesh density is selected to create the mesh shown in Figure 1.57. Also, the aspect ratio of the zones is changed to 1.5:1, and the box Conform to material boundaries is not checked, in order to create a uniform zoning in the vertical column.

Each time one of the zoning buttons is pressed, a set of FLAC commands, corresponding to the model created in the Build tool, is executed to create the model for the factor-of-safety calculation. The state of the model is also saved at this stage, with a file extension of “*.SAV.” The name of the save file is defined by the project and model names and type of zoning. For example, when the fine-grid model is created for the tutorial example in Section 1.2.2, a model save file is created with the name “slope_Model_1_Fine.sav.” Note that this save file is deleted after the factor-of-safety calculation is completed. (See Section 1.3.11.3.)
Figure 1.56  **Special** zoning tool

Figure 1.57  **Special**-zoning model with 80-zone mesh density and zone aspect ratio = 1.5:1
1.3.11.2 Factor-of-Safety Parameters

After the grid generation is complete, the safety factor can be calculated. The calculation is performed by pressing the `SolveFoS` button. The factor-of-safety calculation is based on the strength reduction technique, as described in Section 1.5. By default, the material strength parameters, cohesion and friction angle are reduced in accordance with Eqs. (1.7) and (1.8), given in that section. When `SolveFoS` is pressed, a Factor of Safety parameters dialog opens, with the `Friction angle` and `Cohesion` boxes checked, as shown in Figure 1.58. By pressing `OK`, the calculation will begin.

It is also possible to include other strength parameters in the safety-factor calculation. By checking the `Tension cutoff` box, the material tensile strength can be reduced in a fashion similar to that used with the material cohesion and friction angle. If a weak plane is included in the model, the `Interface friction & cohesion` box should be checked to include these interface strength properties in the strength reduction solution. If structural reinforcement is included in the model, the `Reinforcement grout strength` box should be checked to include grout bond strength and bond friction angle properties in the strength reduction solution. If the `Friction (Ubiquitous-Joint)`, `Cohesion (Ubiquitous-Joint)` or `Tension (Ubiquitous-Joint)` boxes are checked, then the ubiquitous-joint material properties (joint friction, joint cohesion or joint tensile strength, respectively), are included in the strength reduction solution. (The equations used for reduction of these additional strength parameters are described in Section 1.5.) If these boxes are not checked, the corresponding assigned properties will not be changed during the safety-factor calculation.

Associated or non-associated plastic flow can also be specified for the factor-of-safety calculation via the `Use associated plastic flow rule` check box. The material plastic flow rule quantifies the effect of shear dilatancy that occurs in a material at the onset of failure. This is generally expressed by the relation between the friction angle of the Mohr-Coulomb or ubiquitous-joint intact material model and the dilation angle; the dilation angle is related to the ratio of plastic volume strain to plastic shear strain. For associated plastic flow, the dilation angle is equal to the friction angle. If `Use associated plastic flow rule` and `Friction angle` are checked, then the dilation angle will be set equal
Note that for soils, rocks and concrete, the dilation angle is generally significantly smaller than the friction angle of the material. Associated plastic flow is not observed in triaxial testing or shear testing of these materials. See Section 3.7.4.1 in the User's Guide of the full FLAC manual for additional information. Care should be taken when selecting this check box. If associated flow is checked for a physically unrealistic dilation angle (e.g., if the friction angle is greater than 30°), the factor-of-safety calculation may fail to converge.

1.3.11.3 Factor-of-Safety Solution

When \( \text{OK} \) is pressed in the Factor of Safety parameters dialog, the factor-of-safety calculation begins. A series of simulations will be made as described in Section 1.5, and the status of the calculation will be reported in a Model cycling dialog, as shown in Figure 1.59. This dialog displays the percentage of steps completed for an individual solution stage (based on a “characteristic response time,” as defined in Section 1.5), the total number of solution stages that have been performed thus far in the series, the operation currently being performed, and the bracketing values of the factor of safety; the bracket range will continuously decrease until the final value is determined. The run stops when the difference between the upper and lower bracket values becomes smaller than 0.005. When the calculation is complete, the final value is reported.

![Model cycling dialog](image)

After the first bracketing values have been found in the series, the run can be interrupted by pressing the \( \text{Stop} \) button. An estimate for factor of safety will be reported based on the current bracketing limits, but this will be less accurate than if the operation had been allowed to finish.

At the completion of the calculation, a factor-of-safety save file is automatically created with the extension “*.FSV.” This file corresponds to the last non-equilibrium state of the model, at which the calculation stopped. The results of this file can then be used to plot variables, such as shear strain contours and velocity vectors, that identify the critical failure surface in the model (see Section 1.3.12). This save file is identified by the project name, model name, type of zoning and factor-of-safety parameters that were selected for the simulation. For example, the factor-of-safety save file for Model 1 in the tutorial example in Section 1.6 is named “slope_Model_1_Fine_fc.fsv.”
The “fc” descriptor identifies that friction angle and cohesion are included in the calculation. The following code names are used as descriptors for the factor-of-safety parameters:

- \( f \) = friction angle
- \( c \) = cohesion
- \( t \) = tensile strength
- \( uf \) = ubiquitous-joint friction angle
- \( uc \) = ubiquitous-joint cohesion
- \( ut \) = ubiquitous-joint tensile strength
- \( i \) = interface friction and cohesion
- \( s \) = structural element grout strength
- \( a \) = associated plastic flow rule

1.3.12 Producing Output

The results of the factor-of-safety calculation are viewed in the Plot tool, which is accessed by pressing the Plot button. When a calculation is complete, a “factor-of-safety” plot button is added to the toolbar with a name corresponding to the type of zoning and factor-of-safety parameters selected for the calculation. For example, in Figure 1.60 the button contains a nine-square symbol, indicating a fine-grid model, and the descriptors \( fc \), indicating that friction angle and cohesion were included in the calculation. Note that the name can be changed by right-clicking the mouse over the button. Be careful to keep the name short, however, because the entire text is included on the button.
Figure 1.60  Factor-of-safety plot for fine-grid model with friction angle and cohesion included in the factor-of-safety calculation

The factor-of-safety plot displayed in this tool contains, by default, a filled contour plot of shear strain contours and velocity vectors.* The shear strain contours indicate the location of the failure surface, and the velocity vectors indicate the failure mode, at the initiation of failure. This plot is created at the solution stage for which the strengths are reduced to the values at the onset of failure. The factor-of-safety value (i.e., the ratio of the actual strength to the strength at which failure occurs, as defined in Eqs. (1.7) and (1.8)) is also displayed in the plot legend. For the tutorial Model 1 in

* The shear strain contours are identified as “shear strain rate contours” in the plot legend. Shear strain rate is a basic variable calculated in the FLAC solution method for every zone in a FLAC mesh. The “rate” refers to the zone strain calculated during one computational step. For details on this solution scheme, see Section 1.1.2 in Theory and Background of the full FLAC manual; and for the definition of the shear strain rate, see Section 1.3.3.1 in Theory and Background. Shear strain rate contours identify regions in the FLAC model where shear strain localizes. Bands of shear localization, or “shear bands” that develop in the model during a calculation correspond to failure surfaces.

Velocity vectors are also basic variables in the FLAC calculation. Velocities are calculated at all gridpoints in a FLAC mesh. If a coherent velocity field is identified in a velocity vector plot, this indicates that continuous failure (i.e., plastic flow of material) is occurring. For the factor-of-safety calculation, velocity vectors are not related to a real-time movement. They only provide a sense of the pattern of motion at any selected point in the calculation.
Section 1.2.2, these plot items show a well-defined failure surface and indicate a rotational failure mode, as illustrated in Figure 1.60.

Different parameters can be displayed in the factor-of-safety plot. By pressing the **Items** button, a *Plot items* dialog opens as shown in Figure 1.61. For example, the range of the contouring can be controlled; this is useful to define a common contour level if several model results are compared. Also, note that the shear strain-rate contours are derived from strain-rate values calculated in FLAC at zone centroids. The contours for shear strain-rate terminate at zone centroids; they do not extend to model boundaries. An extrapolation function is available to extend the contours to the boundaries. The function uses a simple linear averaging extrapolation. (The extrapolation procedure is described in “EXTRAP.FIS” in Section 3 in the *FISH volume* of the full FLAC manual.) The two contouring approaches can be accessed by clicking on **Zone centroids (exact)** or **Gridpoint linear extrapolation** in the pull-down menu of the *Plot items* dialog. In most instances, **Gridpoint linear extrapolation** provides the clearest representation of the failure surface.

Other optional plots that can be included on the plot are the mesh elements, the water table line and the applied conditions. Plasticity indicators can be included; these identify the type of failure (e.g., shear or tensile failure). The orientation angle of ubiquitous joints can also be displayed.

![Figure 1.61 Factor-of-safety Plot items dialog](image)

If structural elements are included in the model, the location of the reinforcement, and the axial force that develops at the last non-equilibrium state, can be included in the factor-of-safety plot. The maximum value of the axial force can be set so that values from different models are scaled to a specific value. For example, in the example given in Section 1.4.4, the maximum axial force is set to 90,000 N so that the results from two different cases can easily be compared, as shown in Figures 1.92 and 1.93.

Results from other projects can be included in the *Plot* tool by loading the selected “*.FSV” file with the **Load** tool. A failure-plot button will be added to the **Plot** toolbar for the loaded model. The list of factor-of-safety plots can be edited, and factor-of-safety plots removed from the toolbar, by pressing the **List** button to open a *FoS Plots* dialog.
A hardcopy printout of the factor-of-safety plot can be created in the \textit{Print setup} dialog, which is opened by pressing the [Set up] button. The dialog is shown in Figure 1.62. This dialog controls the type and format of graphics hardcopy output. The output types include: Windows printer, Windows clipboard, Windows enhanced metafile, Windows bitmap, PCX, JPEG, Postscript and AutoCad data exchange format (DXF). The default setting is a Windows color printer. The appearance, orientation and settings of the plot, and the destination and name of the plot file, can also be controlled in this dialog. Press [OK] when you have completed your selections. To create the plot, press [Print] in the [Plot] tool and the plot will be sent to the selected hardcopy type.

\textbf{Figure 1.62} \textit{Print setup dialog}

A report mode is also provided to summarize the results of a study. Click on the \textit{File/Create Report ...} menu item to open the dialog as shown in Figure 1.63. This will create an HTML-formatted file listing various tables of information for the study. For an example of an HTML-formatted file, see Figure 1.19. This file can then be pasted into a report document, such as a Microsoft Word file.

\textbf{Figure 1.63} \textit{Report options dialog}
1.3.13 Building More Complex Slopes

Several tools are available to facilitate the creation of different types of slope models. The tools are accessed when a new model is defined in the New Model dialog, as shown in Figure 1.64. These tools define common slope shapes which can be used as a starting point for creation of similarly shaped models. Three general boundary shapes are given: bench slope; dam or embankment; and general, nonlinear slope. The procedures for creating slopes for these three types are described below.

![New Model dialog](image)

**Figure 1.64 New Model dialog**

1.3.13.1 Building a Benched Slope

Two boundary tools are provided to generate bench slopes; these create slopes with one or two benches. If more than two benches are required, then the General boundary tool should be used. When the Bench-1 button is pressed in the New Model dialog, an Edit benched slope parameters dialog opens for a single bench slope, as shown in Figure 1.65. The dimensions for the bench are defined in the diagram included in this dialog. For example, using the dimensions shown in Figure 1.65, a bench boundary is produced, as illustrated in Figure 1.66. A two-bench slope is produced in a similar fashion when the Bench-2 button is pressed.

![Edit benched slope parameters dialog](image)

**Figure 1.65 Edit benched slope parameters dialog**
1.3.13.2 Building a Dam Embankment

An earth dam or an embankment boundary is created using the **Dam** button in the **New Model** dialog. This opens the **Edit dam/embankment slope parameters** dialog, as shown in Figure 1.67. The dimensions for the dam are defined in the diagram included in this dialog. For example, using the dimensions shown in Figure 1.67, a dam boundary is produced as illustrated in Figure 1.68.
1.3.13.3 Building a Nonlinear-Shaped Model

A nonlinear slope boundary can be created using the General button in the New Model dialog. This opens the Edit block parameters dialog, as shown in Figure 1.69. The left, right and bottom dimensions of the boundary are entered in this dialog. When OK is pressed, a General boundary tool opens, showing the left, right and bottom boundaries, and the slope boundary. The shape of the slope boundary line can be modified by adding handle points along the line, and then dragging the points to different locations. Alternatively, handle points can be located at specific x- and y-coordinate positions by right-clicking the mouse over the handle. A Table dialog will open to enter the coordinates.
The slope line corresponds to a table of points that define the slope surface. The line table can be edited by clicking on the **Edit Numerically** button in the *General boundary* tool; this opens an *Edit Table points* dialog in which the $x$- and $y$-coordinates for all the slope points are listed. Points can be input and edited in this dialog.

**Figure 1.70** shows the *General boundary* tool with a nonlinear slope defined by seven handle points. **Figure 1.71** illustrates the final slope boundary.

A digital bitmap or DXF background image can be imported onto the model view from the pop-up *Plot* menu. This menu is opened by right-clicking the mouse over the model view. Click on the **Images/Bitmap** or **Images/DXF** menu item to import a bitmap or DXF file. The general slope boundary can then be adjusted to fit this image.

For example, in **Figure 1.72**, a bitmap image of a rock slope is imported onto the model view in the *General boundary* tool. The dimensions of the model are adjusted to correspond to the scale of the bitmap drawing. Then, the slope line is manually altered to coincide with the slope line on the bitmap, as shown in **Figure 1.73**. The final slope boundary is shown in **Figure 1.74**.

**Figure 1.70  General boundary tool**

---

**FLAC Version 6.0**
Figure 1.71  Nonlinear slope boundary

Figure 1.72  Bitmap image imported onto the model view
**Figure 1.73** Model slope adjusted to fit slope of bitmap image

**Figure 1.74** Slope boundary created from bitmap image
1.4 Stability Analysis Examples

Several examples are presented to validate and demonstrate the application of FLAC/Slope for slope stability analysis. The project file for each example (identified by the extension “.PSL”) is provided in the “FLAC600\FLAC_SLOPE” directory. Use the File/Open Project... menu item to re-create the example and perform the slope stability analysis.

1.4.1 Homogeneous Embankment at Failure

This example compares FLAC/Slope to a limit analysis solution given by Chen (2007). The problem setting is a homogeneous embankment of height $H = 10$ m, slope angle $\beta = 45^\circ$, unit weight $\gamma = 20$ kN/m$^3$, cohesion $c = 12.38$ kPa and friction angle $\phi = 20^\circ$. A gravitational acceleration of 10.0 m/sec$^2$ is also specified. For these parameters, Chen calculates a factor of safety of exactly 1.0. This example problem is also presented in the publication by Dawson et al. (1999), which compares and validates the FLAC solution for several variations of the homogeneous embankment conditions.

We enter the embankment conditions in the FLAC/Slope model in the Build stage. Figure 1.75 shows a plot of the slope geometry and the properties listed in the Define Material dialog of the Material tool. Note that the limit analysis solution by Chen assumes that the material behavior corresponds to the Mohr-Coulomb yield criterion with an associated flow rule (dilation angle $\psi = \phi$). Also, the tensile strength of the material is set to a high value to prevent use of the tension cutoff, for comparison to the Chen solution. The project save file for this example is “CHEN.PSL.”

![Figure 1.75 Material properties for homogeneous embankment example](image-url)
We use the Medium grid mode in the Solve stage; the resulting grid is shown in Figure 1.76. We perform the factor-of-safety calculation, and calculate a factor of 1.01. The failure surface is indicated in Figure 1.77. Note that for a Coarse mesh, the calculated factor-of-safety is 1.03.

We also investigate the effect of assuming an associated flow behavior. If non-associated flow is selected (with $\psi = 0$) in the SolveFoS dialog, the calculated factor-of-safety is 1.00 for the coarse grid and 0.98 for the medium grid.

![Medium-grid zoning for homogeneous embankment example](image)

**Figure 1.76** Medium-grid zoning for homogeneous embankment example
Figure 1.77  Failure surface calculated for homogeneous embankment
1.4.2 Comparison to Fredlund and Krahn (1977) Study

Fredlund and Krahn (1977) report a comparison of several different limit equilibrium methods for the solution of a slope stability example involving different combinations of slope material and piezometric conditions. The conditions are shown in Figure 1.78. Four of the cases analyzed by Fredlund and Krahn (1977) are reanalyzed with FLAC/Slope. The descriptions of these cases are:

Case 1: Simple 2:1 slope, 40 ft high, $\phi' = 20^\circ$, $c' = 600$ psf, no weak layer, no bedrock

Case 2: Same as Case 1 with thin weak layer ($\phi' = 10^\circ$, $c' = 0$) and bedrock

Case 5: Same as Case 1 with piezometric line

Case 6: Same as Case 2 with piezometric line

The four cases are created in FLAC/Slope as four separate models. The project save file for this example is “COMPARE.PSL.” Figure 1.79 shows the model for the Case 6 conditions. Note that the weak layer is represented by an interface in the model. Also, the tensile strength of the soil is set to a high value to prevent tensile failure, for comparison to the limit equilibrium solution. The Medium grid for this model is shown in Figure 1.80.
Figure 1.79  FLAC/Slope geometry for Case 6

Figure 1.80  FLAC/Slope grid for Case 6
The result for the factor-of-safety calculation for Case 6 is illustrated in Figure 1.81. The FLAC/Slope results for all four cases are summarized in Table 1.1. The FLAC/Slope results are in good agreement with the results from the limit equilibrium calculations.

![Figure 1.81 Factor-of-safety results for Case 6](image)

| Table 1.1 Results from Fredlund and Krahn (1977) study compared to FLAC/Slope |
|---------------------------------|-----------------|----------------|-----------------|-----------------|------------------|
| Case | Simplified Spencer’s Janbu’s Morgenstern- |
|      | Bishop Method Method Method Method |
|      | Method Method Method Method |
|      | FLAC/Slope |
| 1   | 2.08 2.07 2.01 2.08 2.03 |
| 2   | 1.38 1.37 1.43 1.38 1.39 |
| 5   | 1.83 1.83 1.78 1.83 1.81 |
| 6   | 1.25 1.25 1.30 1.25 1.34 |
1.4.3 Slope with a Thin, Weak Layer

A clay slope contains a thin layer of weaker material, which is located within the slope, as shown in Figure 1.82. The cohesion of the weak plane ($c_l = 10,000$ Pa) is 20% of the cohesion of the clay ($c = 50,000$ Pa). The strength of the weak plane is varied, while the strength of the clay is kept constant, to evaluate the effect of the weak plane on the resulting failure surface and the calculated factor-of-safety. This example is taken from the slope stability study presented by Griffiths and Lane (1999).

The thin layer is created in the FLAC/Slope model by adjusting two layer boundaries to match the locations denoted in Figure 1.82. The layer boundaries are positioned in the Layers tool by locating the handle points along the boundaries at the specified $x$- and $y$-coordinate positions, as shown in Figure 1.83. The resulting model is shown in Figure 1.84. A fine-grid model is necessary to represent the thin layer (see Figure 1.85). Three cases are analyzed: $c_l/c = 0.2, 0.6$ and $1.0$. The project save file for this example is “THIN.PSL.”
Figure 1.83  Weak layer boundaries created in the Layers tool

Figure 1.84  FLAC/Slope model of slope with a thin weak layer
Figure 1.85 Fine-grid model for slope with thin weak layer

The factor-of-safety plots for the three cases are shown in Figures 1.86 through 1.88. The shear-strain contour plots in the three figures illustrate the different failure surfaces that develop as the strength of the weak plane is changed. In Figure 1.86, the failure surface indicates localized slip along the weak plane, while in Figure 1.88, a circular failure surface develops in the homogeneous material. Figure 1.87 shows a combination of both weak plane failure and circular-slip failure. All of these results compare directly to those reported in the study by Griffiths and Lane (1999).

The safety factors calculated by FLAC/Slope for these three cases also correspond to those presented by Griffiths and Lane (1999). The factor is found to drop significantly as the strength of the weak plane is reduced. The case of $c_1/c = 0.6$ is shown by Griffith and Lane to be the strength ratio at which there is a transition from the weak-plane failure mode to the circular failure mode.
**Figure 1.86** Factor-of-safety plot for $c_l / c = 0.2$

**Figure 1.87** Factor-of-safety plot for $c_l / c = 0.6$
**Figure 1.88**  Factor-of-safety plot for $c_l / c = 1.0$
1.4.4 Slope with Geogrid Reinforcement

In this example, two layers of geo-grid are used to stabilize a slope. The slope conditions and material properties for this model are shown in Figure 1.89. The project save file is “GEOGRID.PSL.” A Medium mesh is used for this example.

![Figure 1.89 Slope with geo-grid reinforcement](image)

The slope is unstable without the geo-grid reinforcement. The results for the unsupported case are shown in Figure 1.90. The factor of safety is calculated to be 0.93.

The properties selected for the geo-grid reinforcement are assigned in the Bolt Properties dialog, as shown in Figure 1.91. Note that, with the reinforcement added, we now include the grout bond strength and friction angle as strength-reduction parameters in the safety-factor calculation.

The factor-of-safety calculation is run for this support in Model 2. The results are shown in Figure 1.92. The safety factor is now increased to 1.13.

The effect of the bonding resistance provided at the geo-grid/soil interface can be seen when we increase the bond cohesion from 1000 N/m to 10,000 N/m. A different cable property ID, C2, is defined to specify the higher bond cohesion. For this case (Model 3), the calculated factor-of-safety is now 1.22, as shown in Figure 1.93.
Figure 1.90  Factor-of-safety results for unsupported slope

Figure 1.91  Geogrid properties specified in Bolt Properties dialog
Figure 1.92  Factor-of-safety results for geo-grid support with bond cohesion = 1000 N/m

Figure 1.93  Factor-of-safety results for geo-grid support with bond cohesion = 10,000 N/m
1.4.5 Rock Slope with Benches

This example is a slope excavated in highly weathered granitic rock. The slope contains three 15 m high benches with two 8 m wide berms. The bench faces are inclined at $75^\circ$ to the horizontal, and the top of the slope is cut at $45^\circ$ from the top of the third bench to the ground surface. Figure 1.94 illustrates the geometry of the slope. This example is taken from Hoek and Bray (1981).

![Figure 1.94 Failure surface solution from Bishop’s method for a rock slope (Hoek and Bray 1981)](image)

The rock mass is classified as a Hoek-Brown material with strength parameters of:

\[
\begin{align*}
    m &= 0.13 \\
    s &= 0.00001 \\
    \sigma_c &= 150 \text{ MPa} \\
    \sigma_c^m &= \sqrt{s} \sigma_c = 0.47 \text{ MPa}
\end{align*}
\]

The tensile strength is estimated to be 0.012 MPa. In order to derive the Mohr-Coulomb properties from the Hoek-Brown parameters, a tangent to the curved Hoek-Brown failure envelope is drawn at a normal stress level estimated from the slope geometry. Mohr-Coulomb properties for friction angle and cohesive strength are then estimated to be (see “HOEK.FIS” in Section 3 in the FISH volume of the full FLAC manual):

\[
\begin{align*}
    \phi &= 45^\circ \\
    c &= 0.14 \text{ MPa}
\end{align*}
\]
The mass density of the dry rock mass is 2500 kg/m$^3$, and the mass density of the saturated rock mass is 2800 kg/m$^3$. The phreatic surface is located as shown in Figure 1.94, and the mass density of water is 1000 kg/m$^3$.

Hoek and Bray (1981) present a limit equilibrium solution for this problem derived from Bishop’s simplified method of slices (Bishop 1955). Based upon the above parameters, Hoek and Bray report that the Bishop method produces a location for the circular failure surface and tension crack, as shown in Figure 1.94, and a factor of safety of 1.423.

The $FLAC$/Slope model is created using the $\text{GENERAL}$ boundary tool in the New Model dialog to specify the coordinates of bench locations along the slope face. Figure 1.95 shows the tool. The $\text{Edit Numerically}$ button is selected to enter the data points that define the slope boundary.

The model also contains a water table at the position shown in Figure 1.94. The $\text{WATER}$ tool is used to input data points defining the water table, as shown in Figure 1.96.

The model is run using the $\text{Fine}$ grid. The project save file for this example is “BENCH.PSL.”

\textbf{Figure 1.95} \textit{GENERAL boundary tool}
Figure 1.96 \textbf{Water tool}

Figure 1.97 displays the factor-of-safety plot for this model. The calculated factor-of-safety is 1.38. The shear-strain contour plot closely resembles the failure surface produced from the Bishop solution, although the failure surface extends farther up the slope in the FLAC/Slope results.

The FLAC/Slope results indicate that tensile failure continues up the slope (as identified from the plot of velocity vectors and plasticity indicators, as shown in Figure 1.98). This progressive failure cannot be identified in a limit equilibrium solution.
Figure 1.97  Factor-of-safety plot for rock slope with benches

Figure 1.98  Factor-of-safety plot for rock slope with benches – velocity vectors and plasticity indicators
1.4.6  Slope in a Closely Jointed Rock Mass

The effect of joint orientation on the stability of a rock slope is illustrated in this example. The rock mass contains a single closely spaced joint set. Three orientations of the jointing are investigated: joint angles of 10°, 45° and 135°, measured counterclockwise from the horizontal plane.

The behavior of the jointed material is simulated with FLAC’s ubiquitous-joint model (see Section 2.4.3 in Theory and Background of the FLAC manual). The rock mass has the following intact material properties:

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Density</td>
<td>2500 kg/m³</td>
</tr>
<tr>
<td>Cohesion ($c$)</td>
<td>0.1 MPa</td>
</tr>
<tr>
<td>Friction angle ($\phi$)</td>
<td>45°</td>
</tr>
<tr>
<td>Tensile strength ($\sigma^t$)</td>
<td>0.0</td>
</tr>
</tbody>
</table>

The joint properties are:

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cohesion ($c_j$)</td>
<td>0.01 MPa</td>
</tr>
<tr>
<td>Friction angle ($\phi_j$)</td>
<td>40°</td>
</tr>
<tr>
<td>Joint tensile strength ($\sigma^t_j$)</td>
<td>0.0</td>
</tr>
</tbody>
</table>

The stability condition for a 30 m high 1:2 (H:V) slope is evaluated for the three jointing orientations. The project save file for this example is “UBISLOPE.PSL.”

For comparison, a simulation is first made neglecting the presence of the jointing (i.e., by using the Mohr-Coulomb material model and the intact rock properties). The calculated factor-of-safety for this case is 2.13, and the failure surface results are shown in Figure 1.99.

When the jointing is included with a joint orientation of 10°, the effect of the jointing is negligible. The factor of safety for this case is 2.12, and the failure surface results are very similar to the Mohr-Coulomb material case (compare Figure 1.100 to Figure 1.99). Note that the ubiquitous joint orientation is also shown on Figure 1.100.

As the joint orientation angle is increased, the effect of the jointing becomes more evident. For the case of the 45° joint orientation, the factor of safety is 1.71, and the failure surface is more aligned with the angle of the slope, indicating a toppling mode of failure. See Figure 1.101. The failure mode now includes both shear and tensile failure along the joints.

At a joint orientation of 135°, slip along the jointing is the predominant mode of failure. This is evident from the failure plot in Figure 1.102. The factor of safety in this case is 1.04.
Figure 1.99  Factor-of-safety plot for rock slope neglecting the presence of jointing – velocity vectors and shear strain contours

Figure 1.100  Factor-of-safety plot for rock slope with jointing oriented at 10° – velocity vectors and shear strain contours
Figure 1.101 Factor-of-safety plot for rock slope with jointing oriented at 45° – velocity vectors and shear strain contours

Figure 1.102 Factor-of-safety plot for rock slope with jointing oriented at 135° – velocity vectors and shear strain contours
1.4.7 Pseudostatic Analysis

Pseudostatic analysis is a procedure commonly used to estimate the seismic stability of earth structures by representing the effects of an earthquake with constant horizontal and/or vertical accelerations. In limit equilibrium analysis, dimensionless horizontal and vertical pseudostatic acceleration coefficients, $k_h$ and $k_v$, are multiplied times the weight of the failure mass to calculate a safety factor. The coefficients are defined as $k_h = a_h/g$ and $k_v = a_v/g$, where $a_h$ and $a_v$ are the horizontal and vertical pseudostatic accelerations and $g$ is the static gravitational magnitude. The magnitude of the coefficients is related to the severity of the earthquake. Kramer (1996) provides further information on the pseudostatic approach and a discussion on the selection of pseudostatic coefficients.

A pseudostatic analysis can be conducted with FLAC/Slope by checking the [Nonstandard gravity?] option in the Model Options dialog.* The gravitational acceleration magnitude and direction can now be assigned to include the pseudostatic acceleration. For example, if $k_h = 0.1$ and $k_v = 0.0$, then the gravitational vector is adjusted to include the pseudostatic acceleration by adding a horizontal acceleration component of $a_h = 0.1g$.

This procedure is demonstrated by comparing FLAC/Slope to a pseudostatic limit equilibrium analysis presented by Kramer (1996). A simple 30 ft high 2:1 (H:V) slope is subjected to an earthquake that is represented by $k_h = 0.1$ and $k_v = 0.0$. The slope consists of two soil layers overlying a rigid bedrock. Soil 1 is a 30 ft thick layer immediately above the bedrock, and Soil 2 is a 20 ft thick layer above Soil 1.

The properties of Soil 1 are:

- unit weight: 125 lb/ft$^3$
- cohesion ($c$): 1000 lb/ft$^2$
- friction angle ($\phi$): 0.0

The properties of Soil 2 are:

- unit weight: 110 lb/ft$^3$
- cohesion ($c$): 600 lb/ft$^2$
- friction angle ($\phi$): 0.0

The factor of safety reported by Kramer for the pseudostatic analysis of this slope with $k_h = 0.1$ and $k_v = 0.0$ is 1.28, assuming a circular failure surface. Kramer also reports that the factor of safety for a static limit equilibrium analysis of the slope is 1.79.

The pseudostatic accelerations are added in the Gravity Settings dialog that is available from the Gravity button in the Build tool after the [Nonstandard gravity?] option is checked. An additional gravity

* Note that a fully dynamic time-domain analysis can also be performed with the full version of FLAC. See Section 1 in Dynamic Analysis of the FLAC manual for further information.
component in the negative $x$-direction of $-0.1g$ ($-3.22$ ft/sec$^2$) is added to the static gravitational vector, as shown in Figure 1.103:

![Gravity Settings](image)

**Figure 1.103 Pseudostatic analysis of a simple slope**

The FLAC/Slope model for the given slope conditions is shown in Figure 1.104. Note that the orientation and magnitude of the gravity icon, shown in the model view, indicate the effect of including the pseudostatic acceleration. Also, for this FLAC/Slope analysis the tensile strengths of the soils are set to a high value to prevent tensile failure, for comparison to the limit equilibrium solution.
The factor of safety calculated from the pseudostatic FLAC/Slope run is 1.27. A circular failure mode is indicated in Figure 1.105. The results compare well with the pseudostatic limit equilibrium solution. The static factor-of-safety is also calculated to be 1.71, and the failure surface result is shown in Figure 1.106.
Figure 1.105  Factor-of-safety results for pseudostatic analysis of a simple slope

Figure 1.106  Factor-of-safety results for static analysis of a simple slope
1.5 Strength Reduction Technique

The “strength reduction technique” is typically applied in factor-of-safety calculations by progressively reducing the shear strength of the material to bring the slope to a state of limiting equilibrium. The safety factor $F$ is defined according to the equations

$$
c_{\text{trial}} = \frac{1}{F_{\text{trial}}} c
$$

$$
\phi_{\text{trial}} = \arctan \left( \frac{1}{F_{\text{trial}}} \tan \phi \right)
$$

A series of simulations are made using trial values of the factor $F_{\text{trial}}$ to reduce the cohesion, $c$, and friction angle, $\phi$, until slope failure occurs. (Note that if the slope is initially unstable, $c$ and $\phi$ will be increased until the limiting condition is found.) In FLAC/Slope, a bracketing approach similar to that proposed by Dawson, Roth and Drescher (1999) is used.

The strength reduction method implemented in FLAC/Slope (and FLAC) will always produce a valid solution: in the case of an unstable physical system, FLAC/Slope simply shows continuing motion in the model. An iteration solution, which is often used in the finite element method, is not used here. The FLAC/Slope solution is a dynamic, time-marching simulation in which continuing motion is as valid as equilibrium. There is also no iteration in the use of elastic-plastic constitutive laws: the stress tensor is placed exactly on the yield surface (satisfying equations, such as the flow rule and elastic/plastic strain decomposition) if plastic yield is detected. The stress state in FLAC/Slope at a safety factor = 1 is the actual stress state that corresponds to the yielding mechanism, not an arbitrary pre-yield stress state or an elastic stress state.

The detection of the boundary between physical stability and instability is based on an objective criterion in FLAC/Slope that decides whether the system is in equilibrium, or in a state of continuing motion. Finer incremental changes that may affect the solution in an iterative solution scheme are not needed in a time-marching scheme and do not affect the solution. In order to determine the boundary between physical stability and instability, a set of completely separate runs is made with different strength-reduction factors. Each run is then checked to determine whether equilibrium or continuing plastic flow is reached. The point of failure can be found to any required accuracy (typically 1%) by successive bracketing of the strength-reduction factors. This process should not be confused with taking finer solution steps; the solution scheme is identical for each run of the set (whether it results in equilibrium or continuing motion).
The procedure for implementing the strength reduction technique in FLAC/Slope (and FLAC) is as follows.

First, the code finds a “characteristic response time,” which is a representative number of steps (denoted by $N_r$) that characterizes the response time of the system. $N_r$ is found by setting the cohesion and tensile strength to large values, making a large change to the internal stresses, and finding how many steps are necessary for the system to return to equilibrium.*

Then, for a given factor of safety, $F$, $N_r$ steps are executed. If the unbalanced force ratio† is less than $10^{-3}$, then the system is in equilibrium. If the unbalanced force ratio is greater than $10^{-3}$, then another $N_r$ steps are executed, exiting the loop if the force ratio is less than $10^{-3}$. The mean value of force ratio, averaged over the current span of $N_r$ steps, is compared with the mean force ratio over the previous $N_r$ steps. If the difference is less than 10%, the system is deemed to be in non-equilibrium, and the loop is exited with the new non-equilibrium, $F$. If the above-mentioned difference is greater than 10%, blocks of $N_r$ steps are continued until: (1) the difference is less than 10%; (2) 6 such blocks have been executed; or (3) the force ratio is less than $10^{-3}$. The justification for case (1) is that the mean force ratio is converging to a steady value that is greater than that corresponding to equilibrium; the system must therefore be in continuous motion.

The following information is displayed during the solution process.

1. Number of calculation steps completed to determine a given value of $F$, as a percentage of $N_r$.
2. Number of completed solution cycles (i.e., tests for equilibrium or non-equilibrium).
3. Operation currently being performed.
4. Current bracketing values of $F$.

The factor-of-safety solution stops when the difference between the upper and lower bracket values becomes smaller than 0.005.

* A maximum limit of 50,000 is set for $N_r$. If the model does not reach equilibrium within 50,000 steps, the run will stop, and the factor-of-safety solution cannot be completed. If this happens, the user should review the parameters selected for the model. For example, if the user has selected cable support with a high value for Young’s modulus, this may affect the solution convergence time. In this event, the [Compute (Optimize for mesh)] button should be selected when setting the Young’s modulus for cables (see Figure 1.47).

† The unbalanced force is the net force acting on a FLAC gridpoint. The ratio of this force to the mean absolute value of force exerted by each surrounding zone is the unbalanced force ratio. Consult note 4 of Section 3.8 in the User’s Guide of the full FLAC manual for more information.
If tensile strength, interface friction and cohesion, ubiquitous-joint model friction, cohesion and tensile strength, and/or reinforcement grout strength are selected to be included in the safety-factor calculation, trial properties are calculated in a manner similar to that used with material friction and cohesion. For the tensile strength $\sigma^t$, the reduction equation is

$$\sigma^{t\,\text{trial}} = \frac{1}{F_{\text{trial}}} \sigma^t$$ \hspace{1cm} (1.9)

For the interface strength values $c_i$ and $\phi_i$, the equations are:

$$c_{i\,\text{trial}} = \frac{1}{F_{\text{trial}}} c_i$$ \hspace{1cm} (1.10)

$$\phi_{i\,\text{trial}} = \arctan\left(\frac{1}{F_{\text{trial}}} \tan \phi_i\right)$$ \hspace{1cm} (1.11)

For the ubiquitous-joint model strength values $c_j$, $\phi_j$ and $\sigma^t_j$, the equations are:

$$c_{j\,\text{trial}} = \frac{1}{F_{\text{trial}}} c_j$$ \hspace{1cm} (1.12)

$$\phi_{j\,\text{trial}} = \arctan\left(\frac{1}{F_{\text{trial}}} \tan \phi_j\right)$$ \hspace{1cm} (1.13)

$$\sigma^{t\,\text{trial}}_j = \frac{1}{F_{\text{trial}}} \sigma^t_j$$ \hspace{1cm} (1.14)

For the reinforcement grout strength values $c_b$ and $\phi_b$, the strength-reduction equations are:

$$c_{b\,\text{trial}} = \frac{1}{F_{\text{trial}}} c_b$$ \hspace{1cm} (1.15)

$$\phi_{b\,\text{trial}} = \arctan\left(\frac{1}{F_{\text{trial}}} \tan \phi_b\right)$$ \hspace{1cm} (1.16)

These values are then used in the safety-factor calculation.
1.5.1 Relation of FLAC/Slope Solution to Limit Equilibrium and Bound Theorem Limits

The bound theorems (presented in most textbooks on plasticity) provide rigorous limits on the collapse conditions of a system consisting of a perfectly plastic material obeying normality (associated flow rule). Of particular interest is the lower-bound theorem, which states (Davis and Selvadurai 2002)

*Collapse will not occur if any state of stress can be found that satisfies the equations of equilibrium and the traction boundary conditions and is everywhere “below yield.”*

In the above theorem, the words “equations of equilibrium” pertain to local equilibrium. Any stress field that satisfies the criteria of the lower-bound theorem is referred to as *statically admissible stress field*. Also, in a factor-of-safety calculation, a statically admissible stress field provides a lower-bound (conservative) estimate for the FoS.

It is also useful to recall the upper-bound theorem, which states that (Davis and Selvadurai 2002)

*Collapse must occur if, for any compatible plastic deformation, the rate of working of the external forces on the body equals or exceeds the rate of internal energy dissipation.*

In this statement, “compatible plastic deformation” means any deformation that satisfies all displacement boundary conditions and is possible kinematically according to the associated flow rule, which governs admissible dilation. Any deformation field that satisfies the criteria of the upper-bound theorem is referred to as *kinematically admissible deformation*.

It is interesting to point out that a limit equilibrium (LE) solution is never a lower bound for the load because, although global equilibrium is satisfied by the LE solution, local equilibrium is not guaranteed (none of the LE solutions are statically admissible).

Also, a strong statement made in the literature (e.g., Davis and Selvadurai 2002) is that the results from LE will always be the same as those from the upper-bound theorem for any translational collapse mechanism (i.e., a plane system of rigid soil blocks separated by thin shear surfaces). Thus, there exist cases for which a LE solution gives an upper bound for the load (Drescher and Detournay 1993).

One then may ask why an LE solution “works,” because not only is it not guaranteed to provide a lower bound for the factor of safety, but in some cases it is even proven to give an upper bound for the factor of safety. An answer, provided by Chen (2007), rests on the observation that most factor-of-safety analyses are concerned with slopes, and apparently, for most slopes, the LE solution provides a factor-of-safety value which is close to the exact solution.

On the other hand, consider the last stable state calculated by FLAC/Slope (the last lower bracket, which is typically 0.005 less than the final safety factor) for an associated problem. FLAC will provide an *approximate exact solution* to the problem at that state, in the sense that, local equilibrium may not be satisfied everywhere at the boundary between zones, but if the zone size is reduced to zero, local equilibrium will be satisfied to the limit. In particular, the limit stress field satisfies the lower-bound theorem. Also, the deformation field at the “failure state” calculated by FLAC/Slope (the last upper bracket) is a kinematically admissible deformation (i.e., it fulfills all the criteria of
the upper-bound theorem). Thus, one may say that, if the calculated factor of safety tends to a limit as the grid size is reduced, this limit may be considered to be very close (within 0.005) to the exact factor of safety for the problem.

In summary, in most cases FLAC/Slope (on a fine grid) and a LE solution will give factors of safety that are very similar. In some cases, FLAC/Slope will give a safety factor on a fine grid that is lower than that provided by a LE solution. This implies that the LE solution provides an upper bound for the safety factor. In other cases, FLAC/Slope will give a safety factor on a fine grid that is higher than that provided by a LE solution. This does not mean that FLAC is non-conservative, but instead that we have encountered a case where the LE solution cannot be relied upon (because it can never correspond to a lower bound for the load).

Note that the bound theorems only apply to an associated flow rule. This rule may not be very realistic in some cases, as it provides far too much dilation (Davis and Selvadurai 2002). FLAC/Slope adds generality as it has the capability to handle more complex flow rules, including non-associated flow.
1.6 References


