

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 to determine 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., rock bolt, soil nail or geogrid) 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, nonlinear 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 the Mohr-Coulomb 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.
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.

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.
7. Structural reinforcement, such as soil nails, rock bolts or geotextiles, 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.
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 tool bars 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. This includes: 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 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 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](#). The solve facility is turned off in this evaluation version. If you decide to upgrade to the full *FLAC*, it is only necessary to upgrade your hardware lock 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 1 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 95 or later is suitable for operation of the code.

By default, plots from *FLAC/Slope* are sent directly to the Windows native printer. Plots can also be directed to the Windows clipboard, or files encoded in PostScript, Enhanced Metafile format, and several bitmap formats (PCX, BMP or JPEG). Instructions on creating plots are provided in [Section 1.3.12](#).

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 *must* click on the *FLAC* box in the *Select Components* dialog in order to install *FLAC/Slope* on your computer (note that selecting the *FLAC* box is the correct choice for *both* *FLAC* and *FLAC/Slope* installations).*

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(R) in order to be viewed. This software is freely available from Adobe Systems Incorporated.

* The full version of *FLAC* will also be installed when *FLAC/Slope* is installed. You may start up full *FLAC* and operate the code in *GIIC* mode to evaluate the features in the full version. Please note that the solve facility is turned off in the evaluation version. If you decide to upgrade to the full *FLAC*, it is only necessary to upgrade your hardware lock to operate *FLAC* as well as *FLAC/Slope*.

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 (“FLAC500”) that resides in the main installation folder. (This is the folder that is 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 “\FLAC500” directory contains the files related to the operation of *FLAC/Slope*. There are three sub-directories: “FLAC500\EXE” contains the executable code that is loaded to run *FLAC/Slope**; “FLAC500\FLAC_SLOPE” contains the example files described in this manual; and “FLAC500\GUI” contains files used in the operation of the *GIIC*.
- The “\SHARED\JRE” directory contains the JAVA(TM) Runtime Environment (standard edition 1.5.0) that is used for operating the *GIIC*.
- The “\MANUALS\FLAC500” 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 LPT1 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* tool bar, containing modeling-stage tabs for each of the stages: , , and . When you click on a modeling-stage tab, a set of tools becomes available: these tools are used to create and run the

* The executable code used for *FLAC/Slope* is the single-precision version (“FLACV_SPE.EXE”). This version is better-suited to factor-of-safety calculations than the double-precision version because it runs approximately 1.5 to 2.0 times faster, and the single-precision calculation is sufficient for this type of analysis. (See [Sections 2.1.3](#) and [2.9](#) in the **User’s Guide**.)

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](#)).

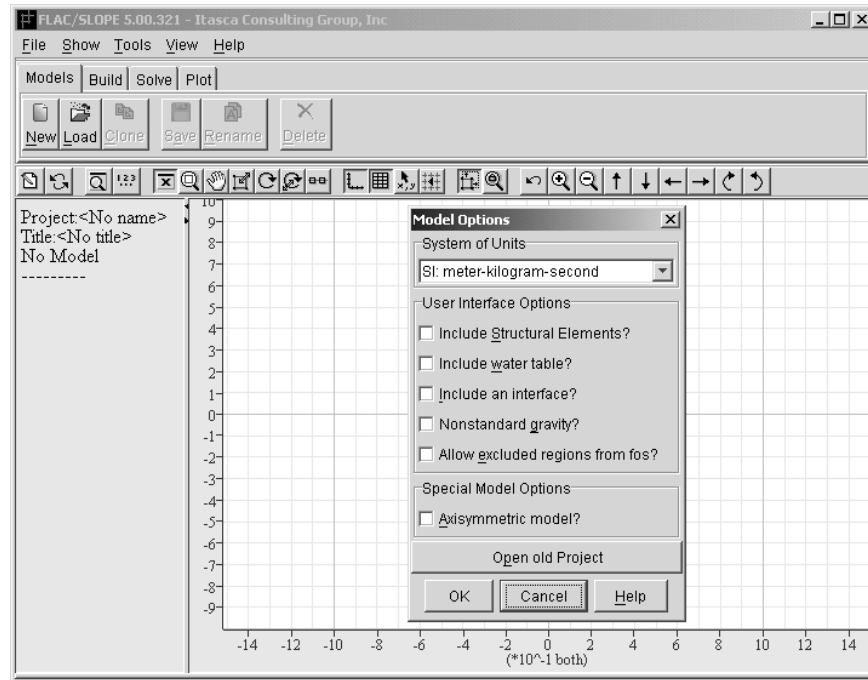


Figure 1.1 The *FLAC/Slope* main window

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

Directly above the *model-view* pane is a *View* tool bar. 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 will appear, as shown in [Figure 1.1](#). You have the option to include different features, such as an interface (weak plane), a water table or reinforcement, in the model and specify the system of units for your project with this 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 *GIIC Help* files.

* 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.

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 1.2](#) 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](#). You can now begin the tutorial.

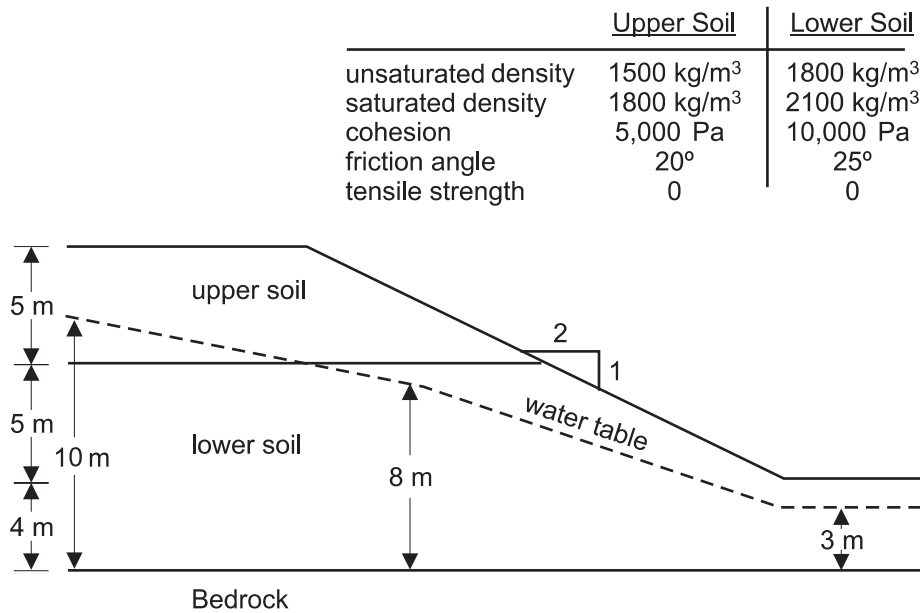


Figure 1.2 Conditions of the simple slope

Defining the Project — We begin the project by checking the INCLUDE WATER TABLE? 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 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 Save* dialog opens, as shown in [Figure 1.3](#), 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 to accept these selections.

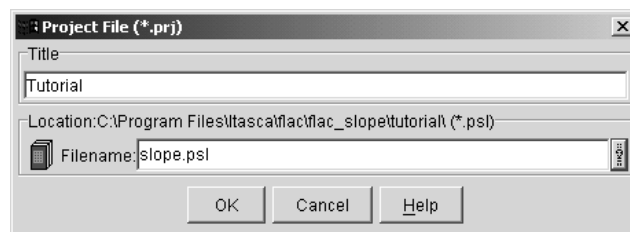


Figure 1.3 *Project Save dialog*

We next click on the tool and enter the *Models* stage to specify a name for the first model in our project. We click on 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 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 1.4](#). Note that we click on to reverse the model layout to match that shown in [Figure 1.2](#). We click to view the slope boundary that we have created. We can either edit the boundary further or accept it. We press to accept the boundary for *Model 1*. The layout for the *Model 1* slope is shown in [Figure 1.5](#)*. 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.

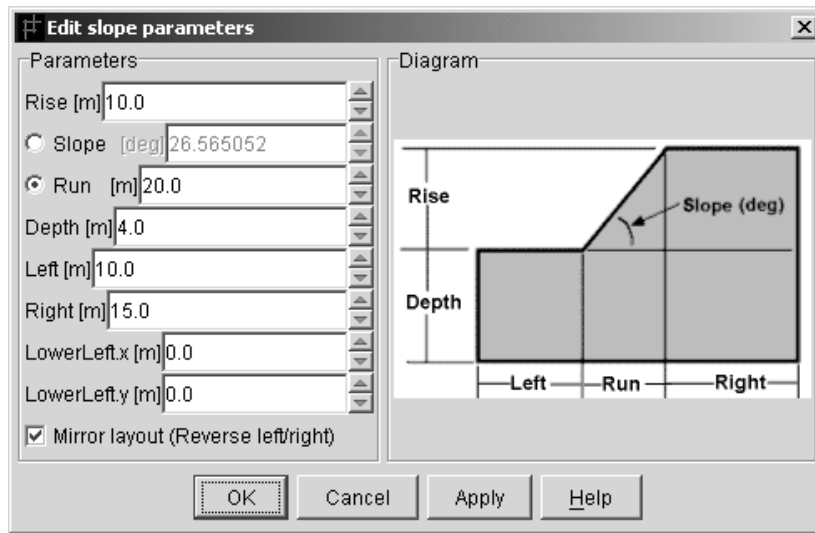


Figure 1.4 Edit Slope Parameters dialog

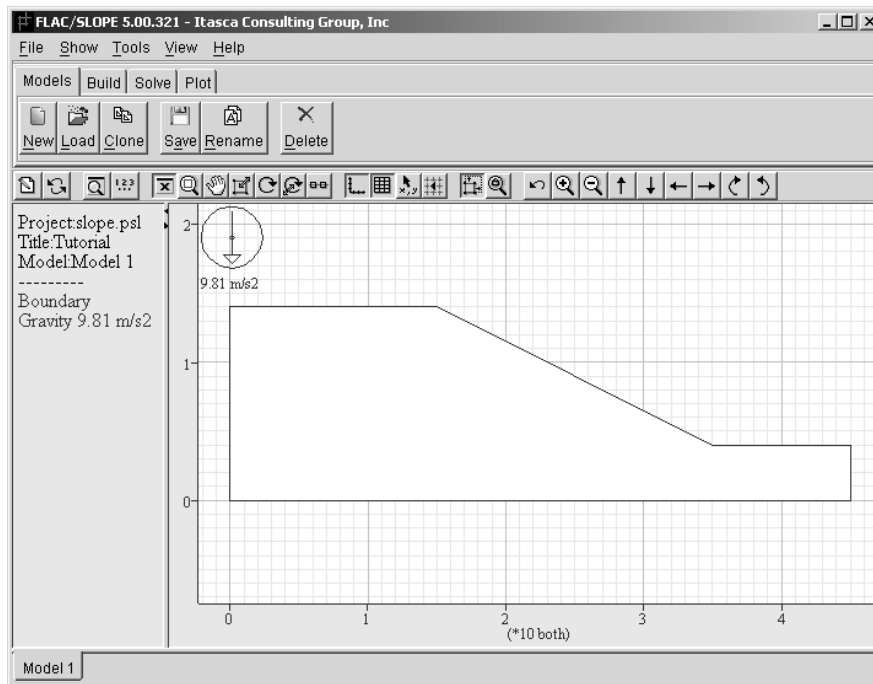


Figure 1.5 Model 1 layout

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 1.6.) 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 1.7.

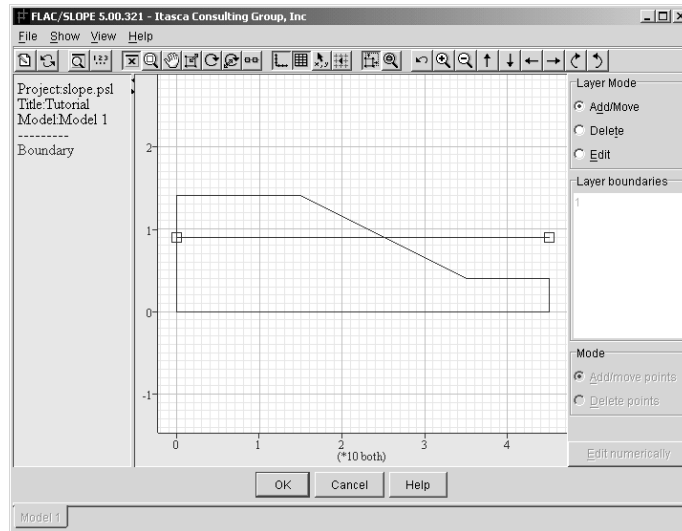


Figure 1.6 Layers tool

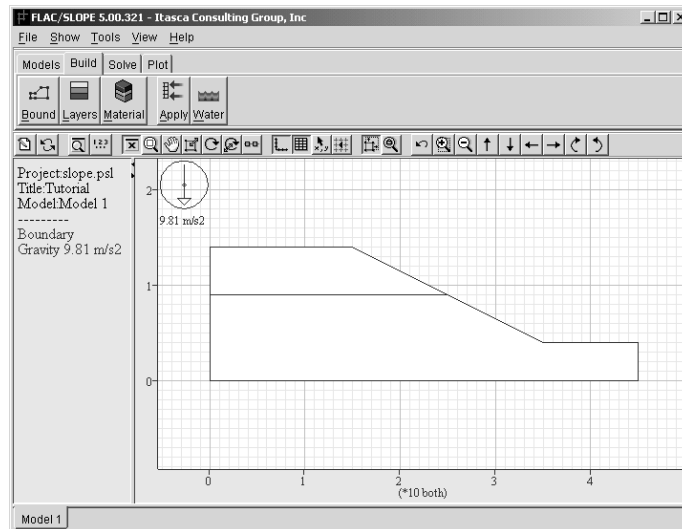


Figure 1.7 Two layers created by the Layers tool

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 1.8](#). (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, ρ , unit weight, γ , and gravitational magnitude, g , is

$$\rho = \frac{\gamma}{g} \quad (1.1)$$

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 1.8](#), the in-situ density of the material *above* the water table (*unsaturated density* in [Figure 1.2](#)) 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](#).

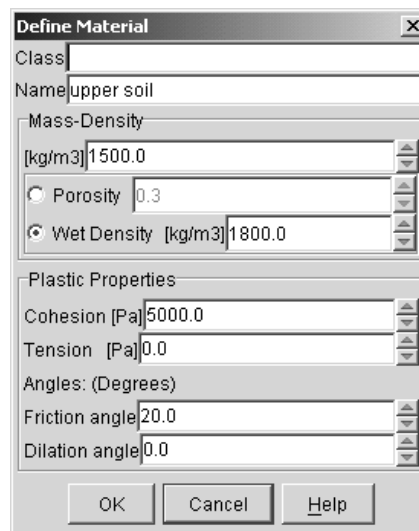


Figure 1.8 Properties input in the *Define Material* dialog for upper soil

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 1.9](#). We press **OK** to accept these materials in *Model 1*.

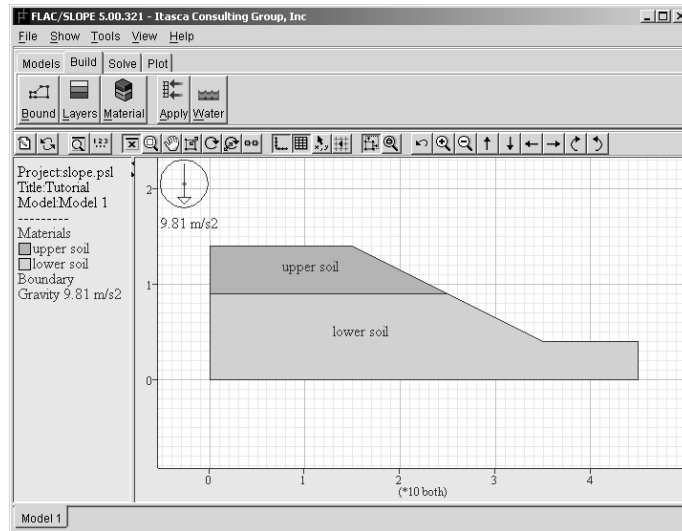


Figure 1.9 Materials assigned to the two layers in the **MATERIAL** tool

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 “medium-grid” model by pressing the **MEDIUM** button, and the grid used for the *FLAC* solution appears in the model view. See Figure 1.10.

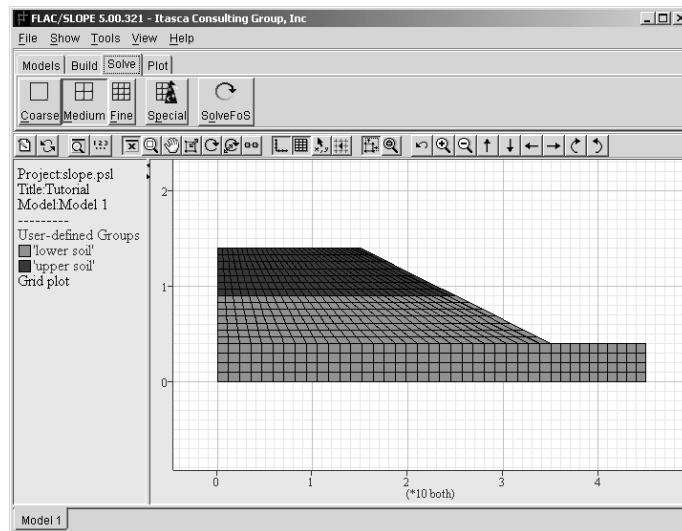


Figure 1.10 Medium-grid for Model 1

We now press the **SOLVE FOS** button to begin the calculation. A *Factor-of-Safety parameters* dialog opens (Figure 1.11), 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 1.60.

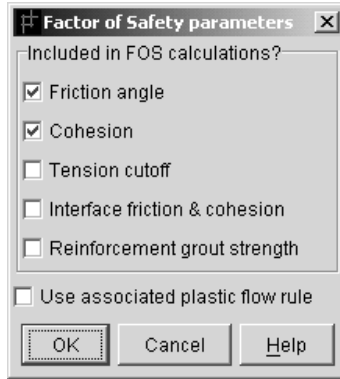


Figure 1.11 Factor-of-Safety parameters dialog

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 (medium-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 1.12.

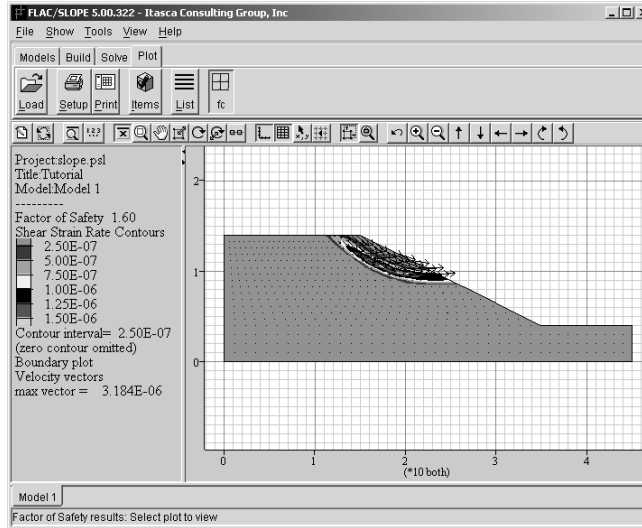


Figure 1.12 Factor-of-safety plot for medium-grid Model 1

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 **CLONE** 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 1.2. The line can either be re-positioned 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 1.13.

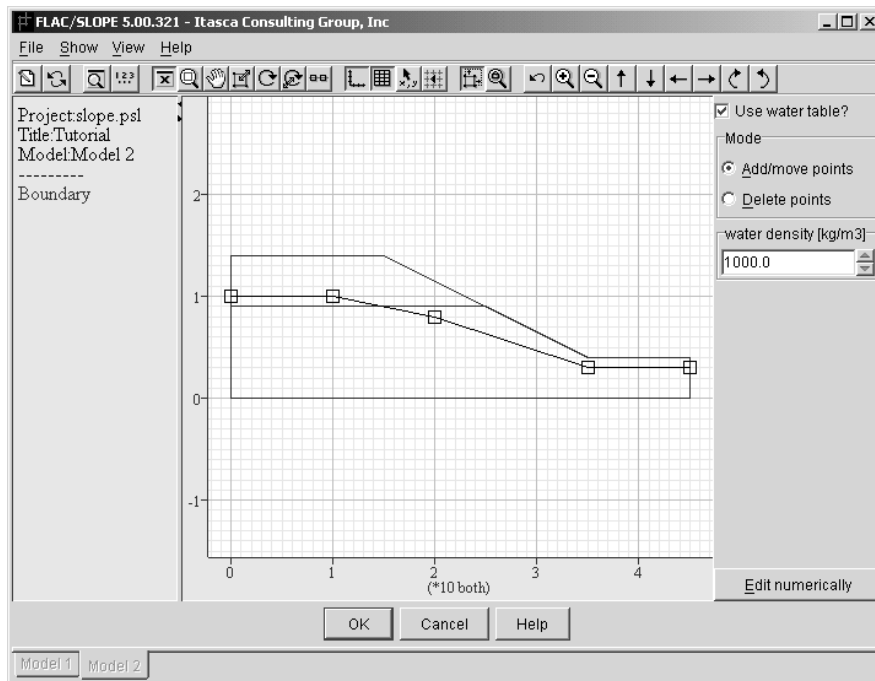


Figure 1.13 Positioning water table in the *Water* tool

We are now ready to solve *Model 2*, so we go to the *Solve* stage, select the medium-grid model and press the **SOLVE FOS** button. We follow the same procedure as before to determine the factor of safety. A factor of 1.47 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 1.14. 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.

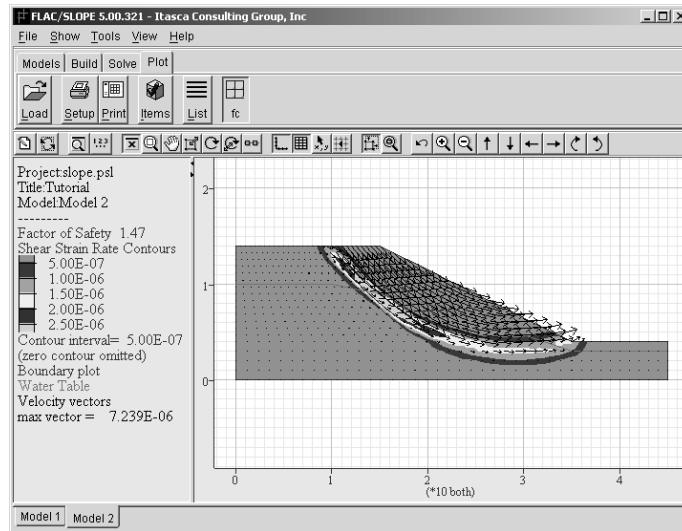


Figure 1.14 Factor-of-safety plot for medium-grid Model 2

Making Hardcopy Plots — Several different printer formats are available to create plots from *FLAC/Slope*. We click on the **SETUP** button in the **PLOT** tool bar to open a *Print setup* dialog, as shown in Figure 1.15.

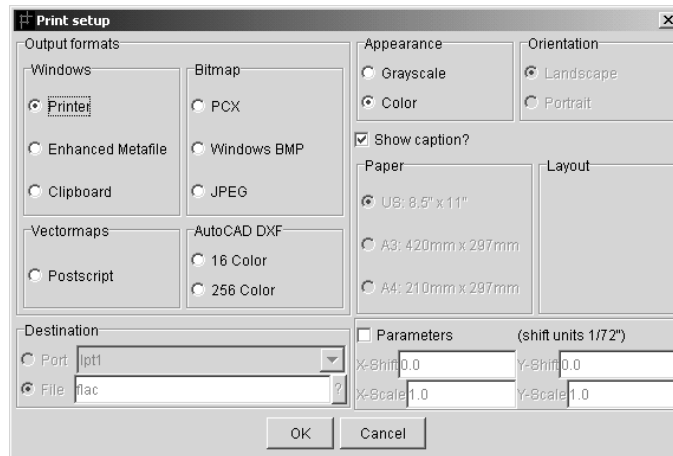


Figure 1.15 Print setup dialog

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 the figure, 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 button. We press to save this setting. Then, press in the tool to send the plot to the clipboard and, finally, paste the plot directly into the Word document.

The plot is shown in [Figure 1.16](#). Note that hardcopy plots are formatted slightly differently than the screen plots.

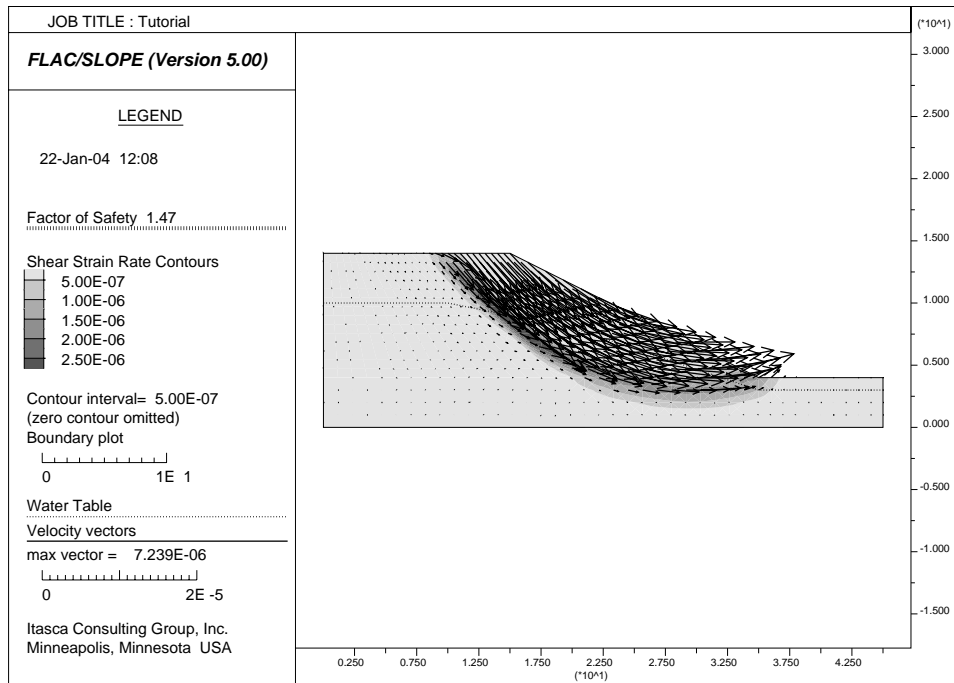


Figure 1.16 Hardcopy plot for Model 2 result

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 1.17](#). This will create an HTML-formatted file listing various information and plots for the study. For example, by selecting the and items in the dialog, tables listing the material properties and the calculated factors of safety will be produced, as shown in [Figure 1.18](#).

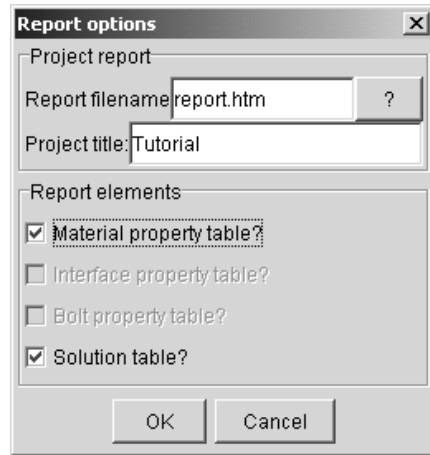


Figure 1.17 Report Options dialog

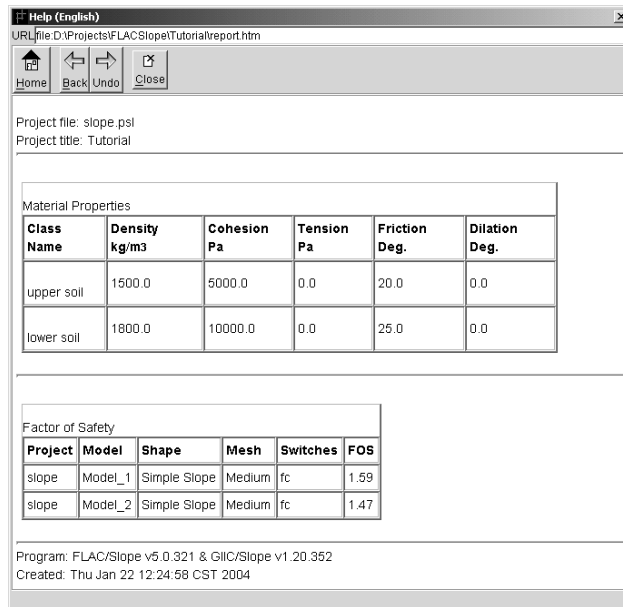


Figure 1.18 Results tables

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 **FINE** button. This will create a finer mesh than the medium-mesh model. After solving for the factor of safety, a new plot button will be added in the *Plot* tool bar for *Model 1*. You can then compare this result for a coarse mesh directly with the medium mesh result by clicking on the plot buttons. 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 1.19](#). 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.

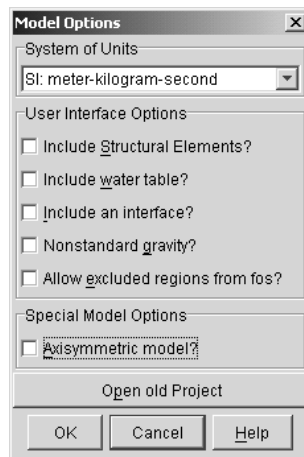


Figure 1.19 *Model Options dialog*

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?`, `INCLUDE WATER TABLE?` or `INCLUDE AN INTERFACE?` box is checked, the corresponding tool is added to the `BUILD` tool bar. 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 your choosing from a *Gravity* tool in the BUILD tool bar. Note that pseudo-static horizontal accelerations can be applied by using non-vertical gravity.

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 tool bar 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 tool bar and the view tool bar. Open the SHOW menu in the main menu to change the look-and-feel of the *FLAC/Slope* pane and tool bars. 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\FLAC500\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 Save* dialog as shown in [Figure 1.20](#). 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 ? in this dialog to select a directory in which to save the project and model-state save files.

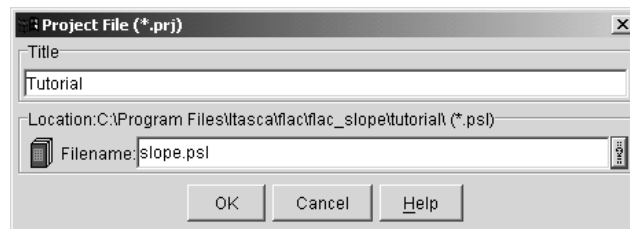


Figure 1.20 *Project Save dialog*

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 1.21](#). 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](#).

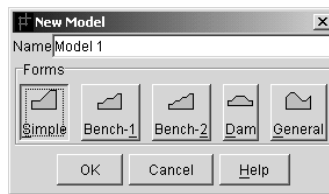


Figure 1.21 *New Model dialog*

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 1.22](#). 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.

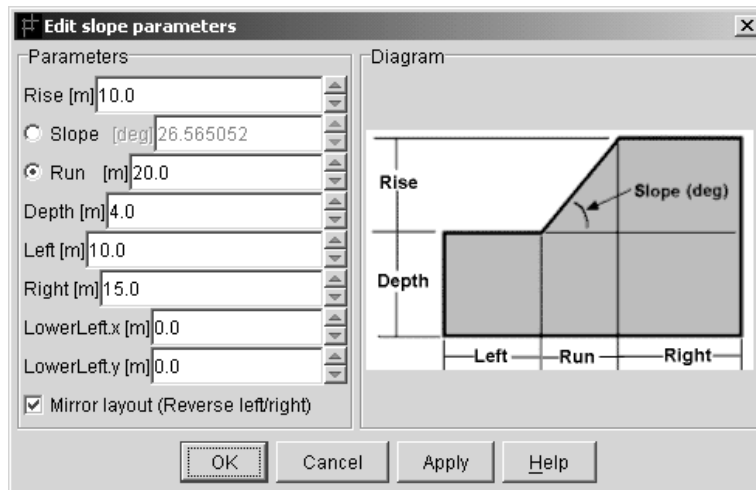


Figure 1.22 *Edit Slope Parameters dialog*

When selecting the dimensions for , , and , it is important that these dimensions are large enough such 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 and boxes in the *Edit Slope Parameters* dialog.

When you press , the dialog will close and the outline of the slope model will be drawn in a boundary view, as shown in [Figure 1.23](#). 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.

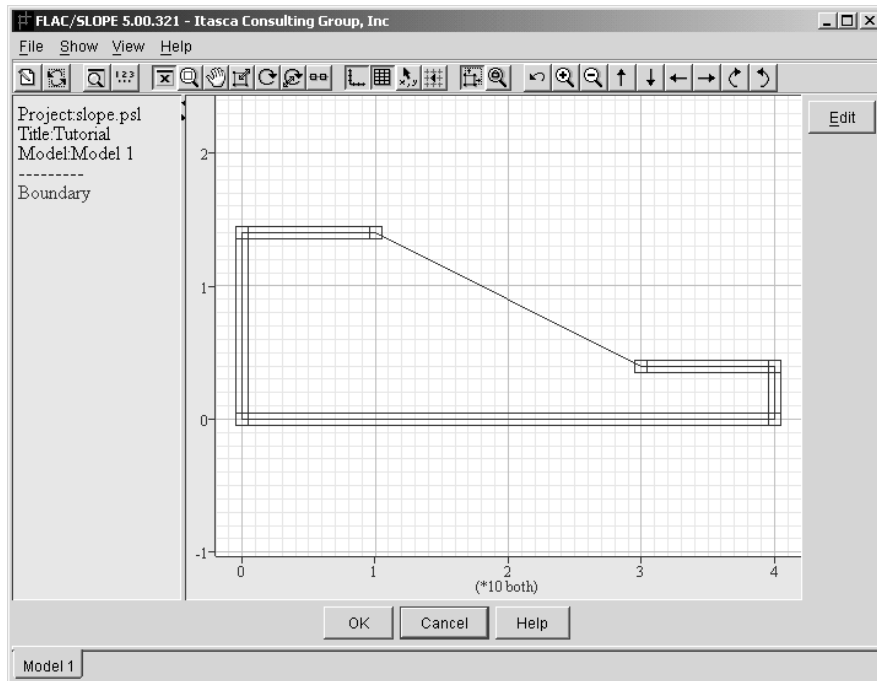


Figure 1.23 *Boundary view*

Once you are satisfied, press . The model boundary will now be drawn in the model view, as shown in [Figure 1.24](#). Note that a tab with the model name will appear at the bottom of the model view when a model is created.

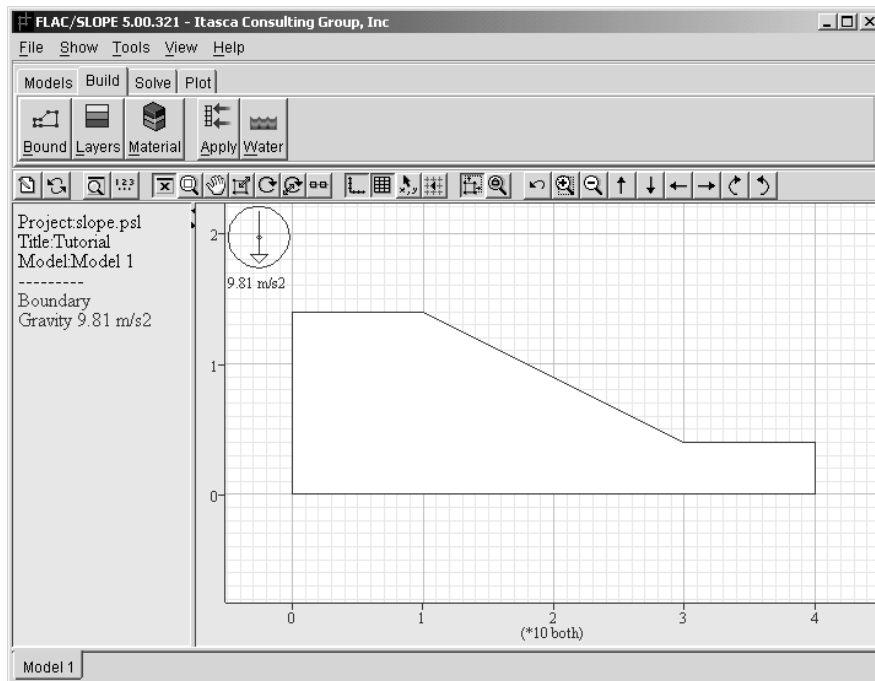


Figure 1.24 Model view

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** tool bar. 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** tool bar. 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 1.25](#) shows a model with two layer-boundary lines visible in the *Layers* tool.

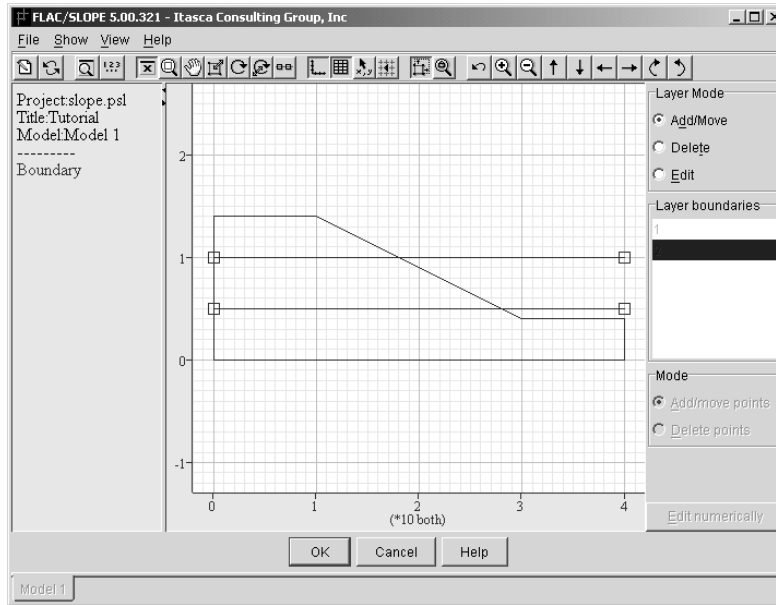


Figure 1.25 Slope model with two layer-boundary lines in the Layers tool

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** 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 1.26](#), 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.

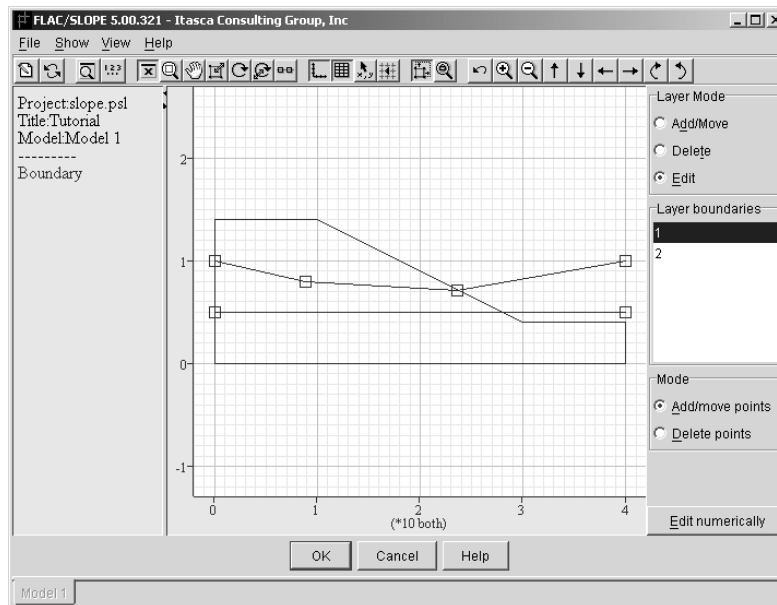


Figure 1.26 The upper layer-boundary line is edited to include two points

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 1.27](#), 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 1.27](#) there is a slight offset in the data points listed for the new line. By doing this, each line will be uniquely defined. [Figure 1.28](#) 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.

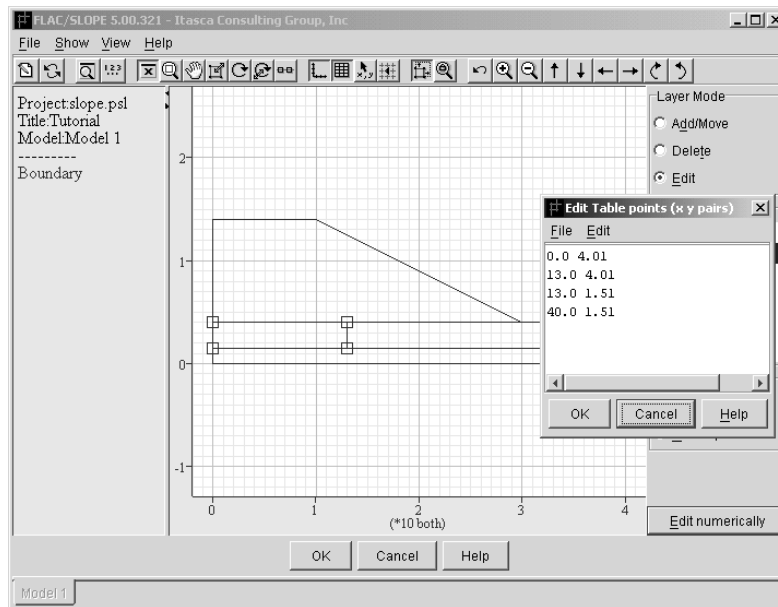


Figure 1.27 The new boundary line, 3, is offset slightly to avoid coinciding with the existing lines, 1 and 2

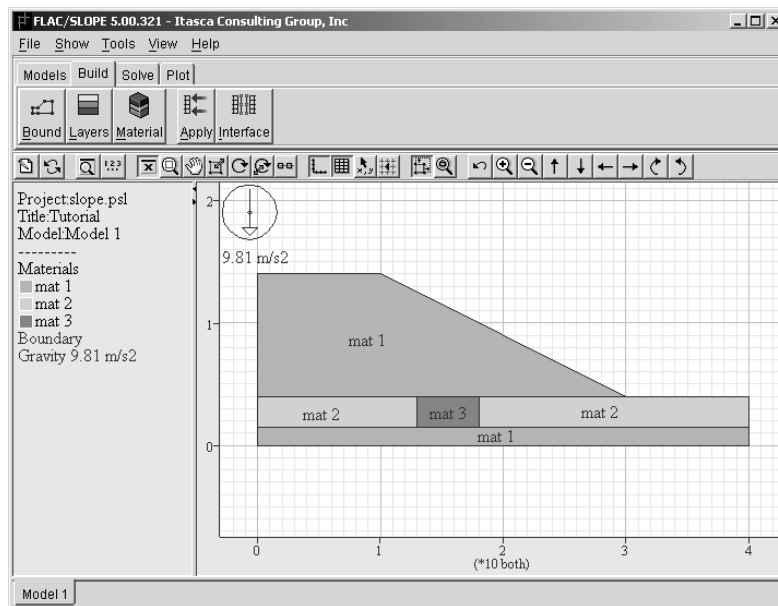





Figure 1.28 Model with vertical column within horizontal layer

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  button.

Materials are created by clicking on the  button to open the *Define Material* dialog. The dialog is shown in [Figure 1.29](#). 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  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

$$\rho^{\text{wet}} = \rho + n \rho_w \quad (1.2)$$

where ρ^{wet} is the wet in-situ density, ρ is the moist in-situ density, n is the porosity, and ρ_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 the Mohr Coulomb plasticity model in terms of the cohesion and internal angle of friction. A tensile strength and dilation angle may also be specified for the material. 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](#).)

The elastic properties 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.

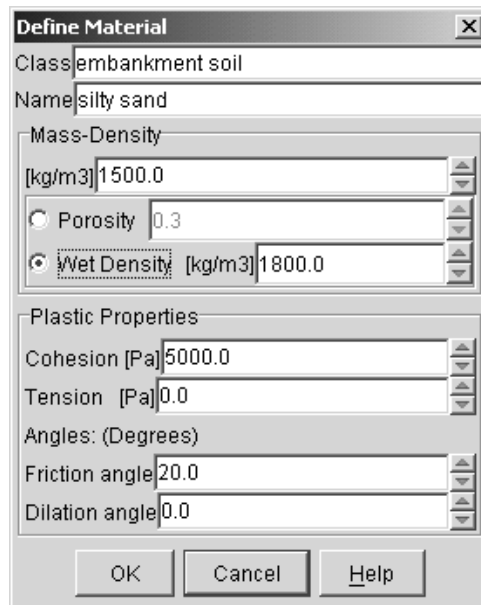


Figure 1.29 Define Material dialog

If the material button is pressed, a *Material list* dialog will open as shown in [Figure 1.30](#). 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 to send these materials to the *List* shown in the *Material* tool.

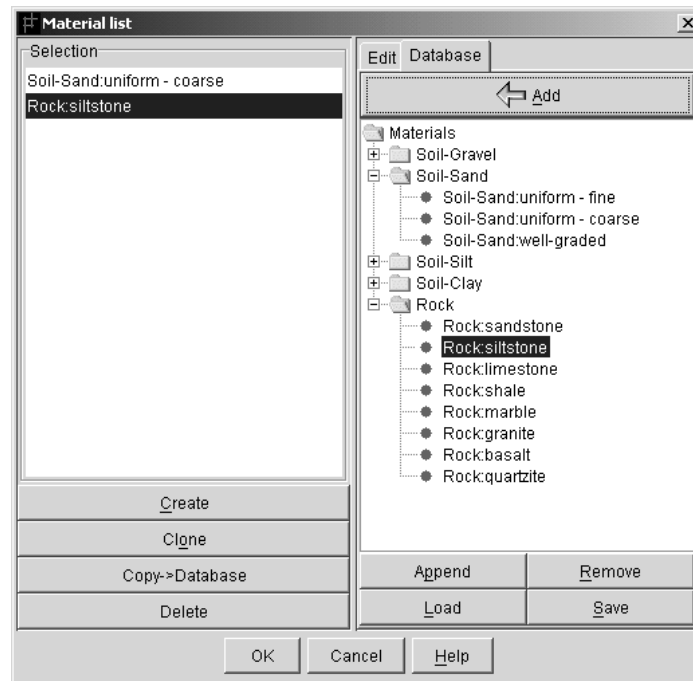


Figure 1.30 *Material List dialog*

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 1.31](#). 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 1.30](#)) 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.

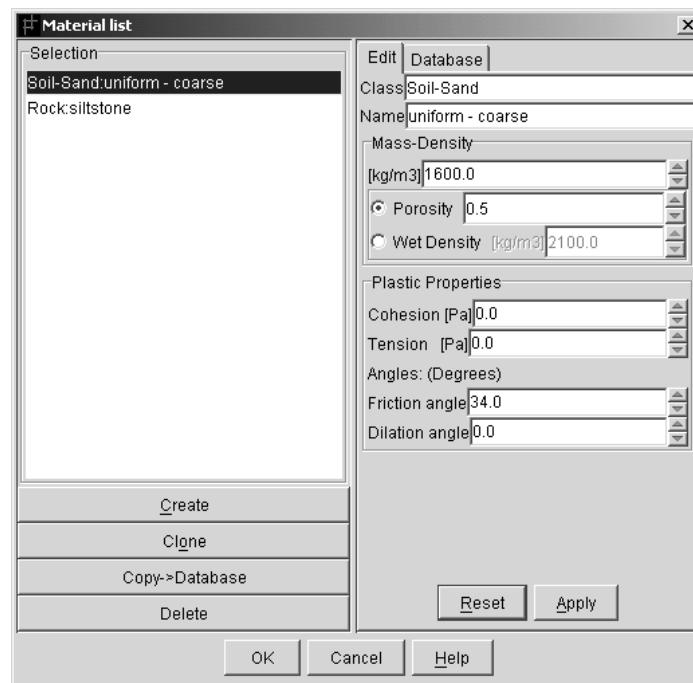


Figure 1.31 Material List dialog in Edit pane

1.3.6 Adding a Weak Plane (Interface)

A weak plane or interface can be added to the slope model by clicking on the **INTERFACE** button in the **BUILD** tool bar. This opens the *Interface* tool, as shown in Figure 1.32. 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 *Table* dialog to input x - and y -coordinates for the points. The interface-line 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 line are listed; points can be input and edited in this dialog. Figure 1.33 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 *FLAC/Slope*. *FLAC* should be used if it is necessary to model multiple or sub-vertical interfaces.

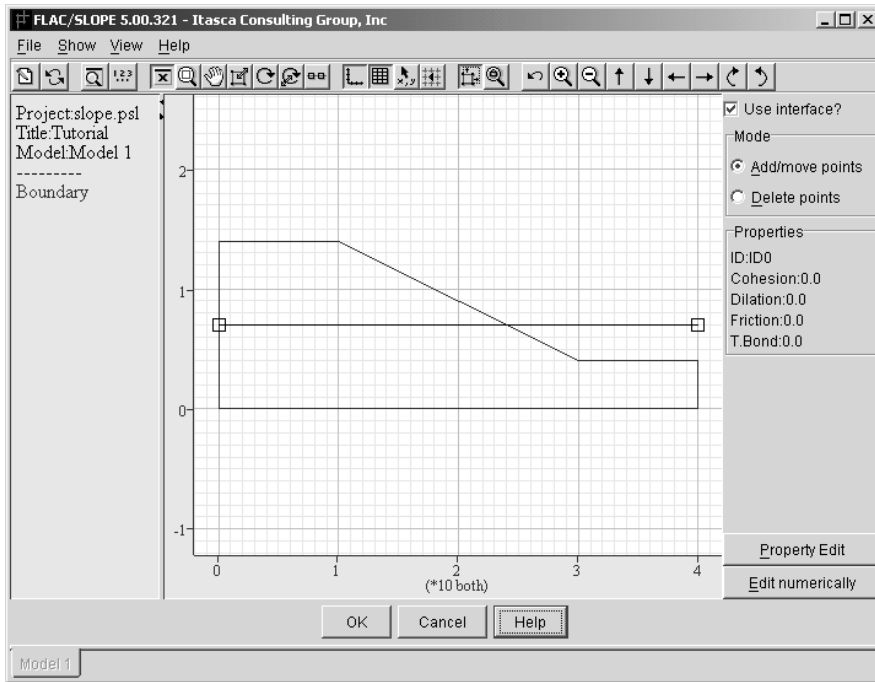


Figure 1.32 Interface tool

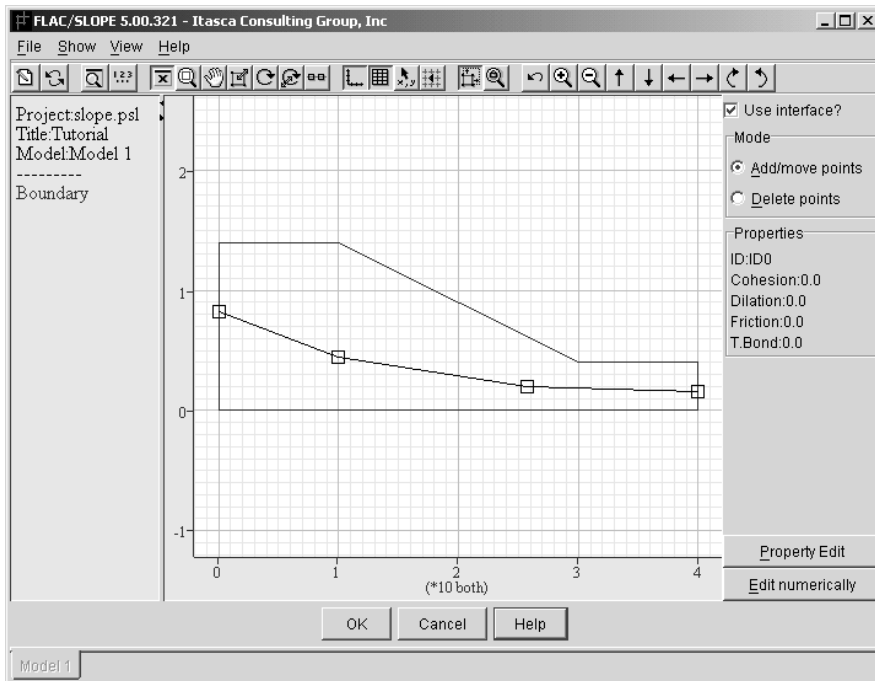


Figure 1.33 Interface line repositioned

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 1.34](#). The interface is defined by a classification and name — e.g., classification: *bedding plane* and name: *smooth*. The interface properties are then prescribed to this interface material and applied by pressing **APPLY**. 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.

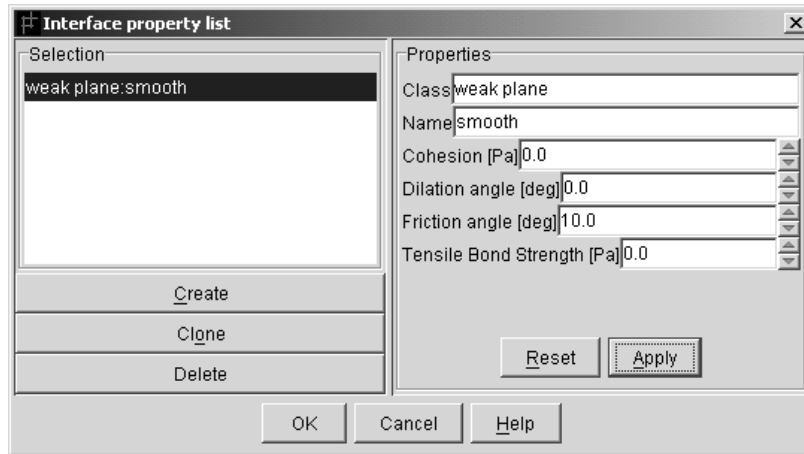


Figure 1.34 *Interface Property list dialog*

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

$$\tau_{\max} = c_i + \sigma_n \tan \phi_i \quad (1.3)$$

where c_i = cohesion (in stress units) along the interface, ϕ_i = friction angle of the interface surface, and σ_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, ψ_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 4.4.1](#) 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** tool bar. This opens the *Water* table tool, as shown in [Figure 1.35](#). 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 1.36](#) shows the water table line repositioned with two handle points added along the line.

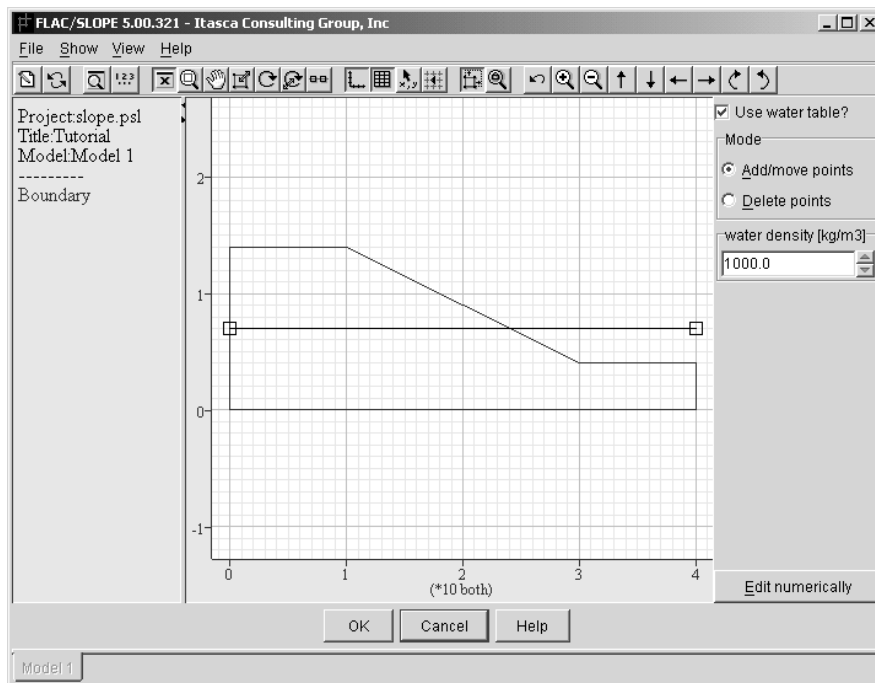


Figure 1.35 Water table tool

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 the water table.

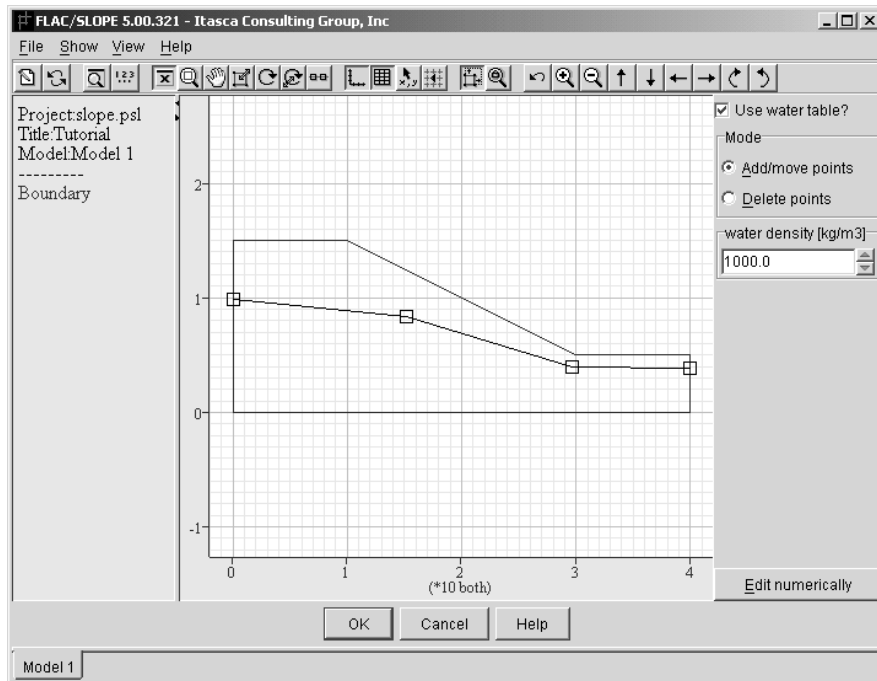


Figure 1.36 Water table repositioned with two handle points added

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 1.37](#). 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.

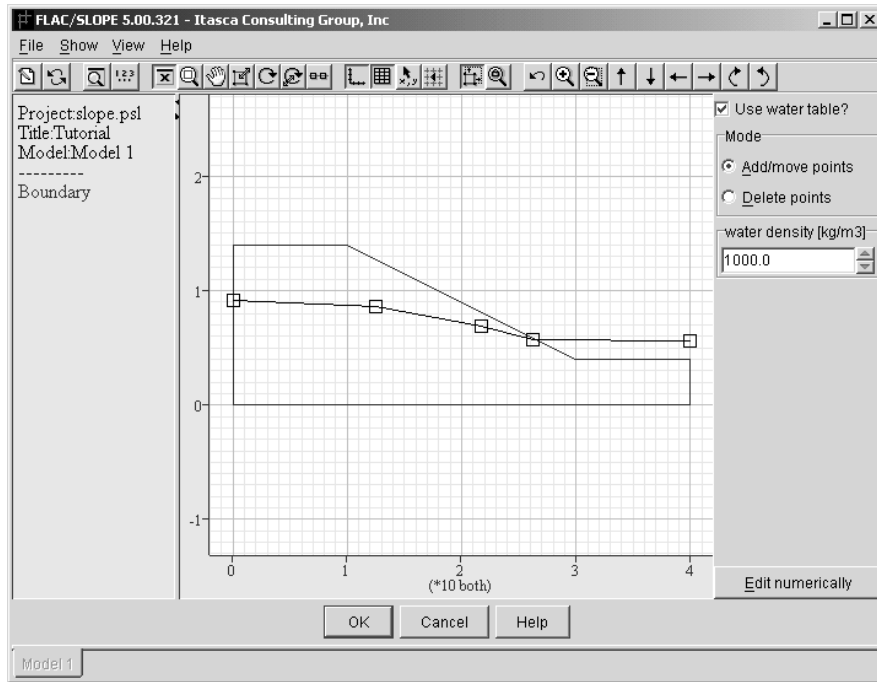


Figure 1.37 *Water table repositioned to intersect the slope and coincide with a reservoir level*

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 1.38](#). 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 1.39](#) shows arrows corresponding to the surface water pressure applied in [Figure 1.38](#).

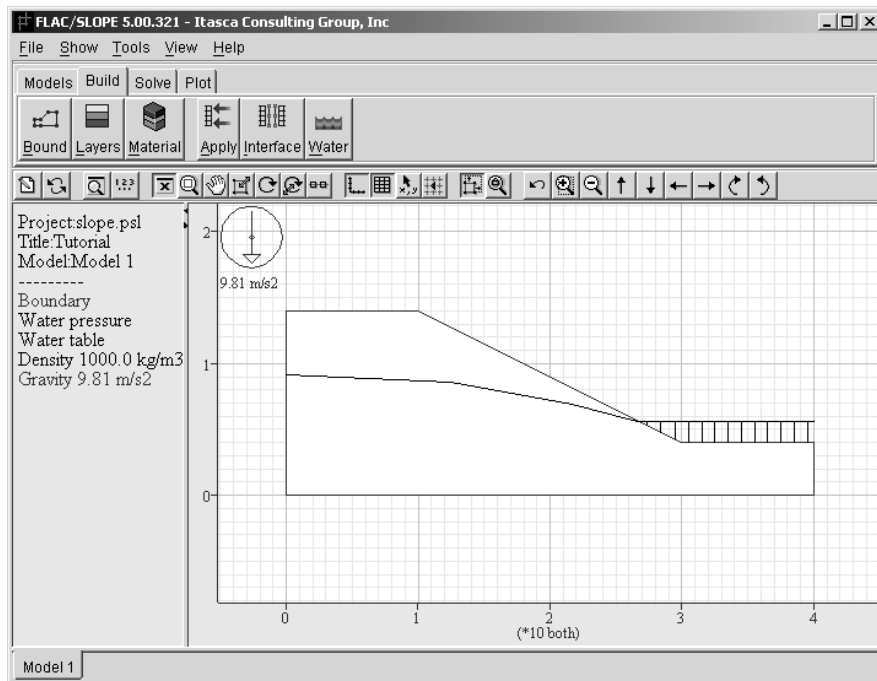


Figure 1.38 Water pressure acting along slope surface shown in model view

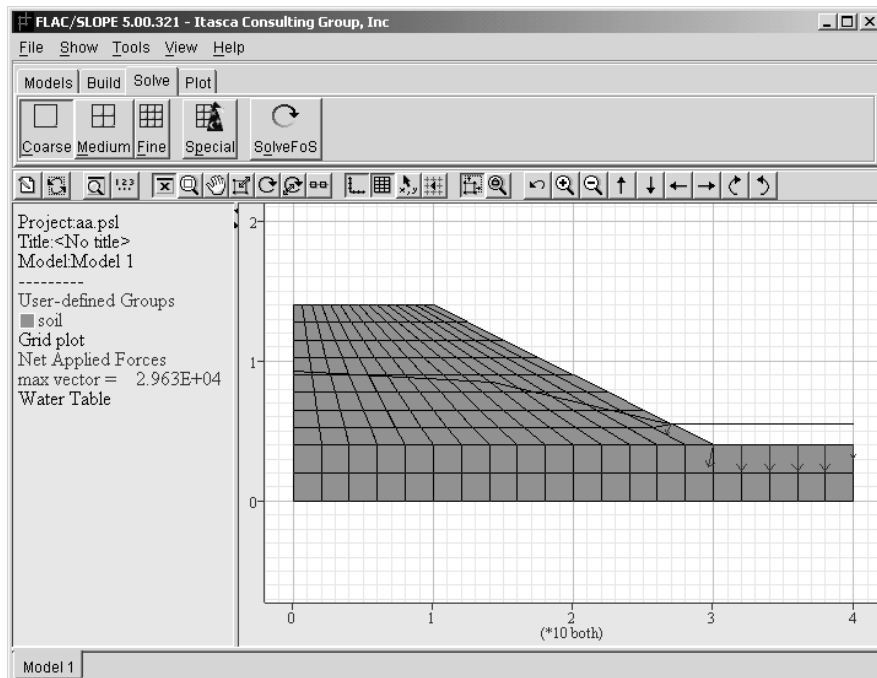


Figure 1.39 Applied forces corresponding to the surface water pressure applied in Figure 1.38

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** tool bar. This opens the *Apply* tool, as shown in Figure 1.40. 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 you wish to apply the load. For example, in Figure 1.40, a pressure is applied at the top of the slope along the region designated by the pressure bar.

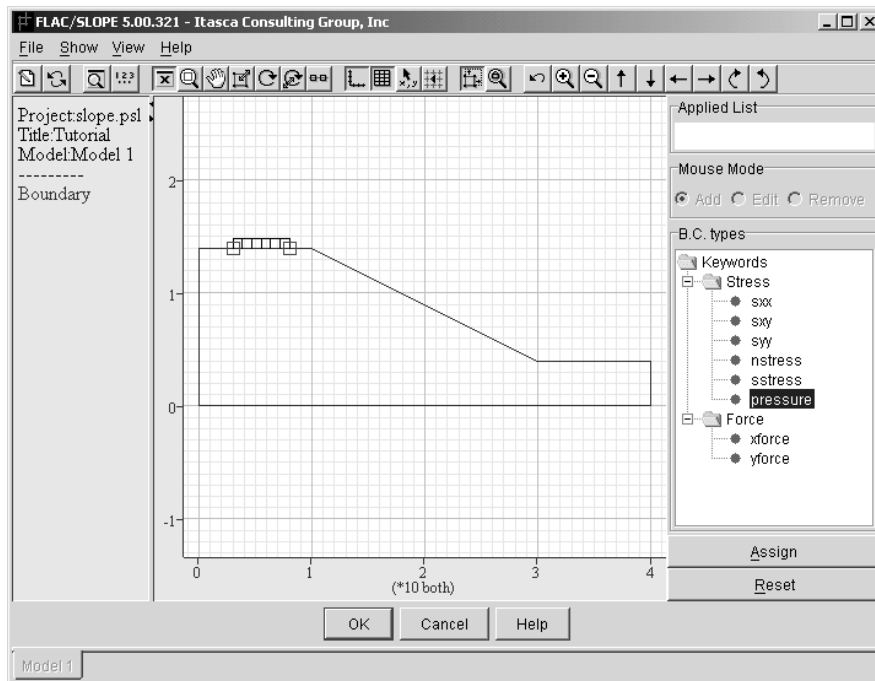


Figure 1.40 *Apply tool*

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 1.41, 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.

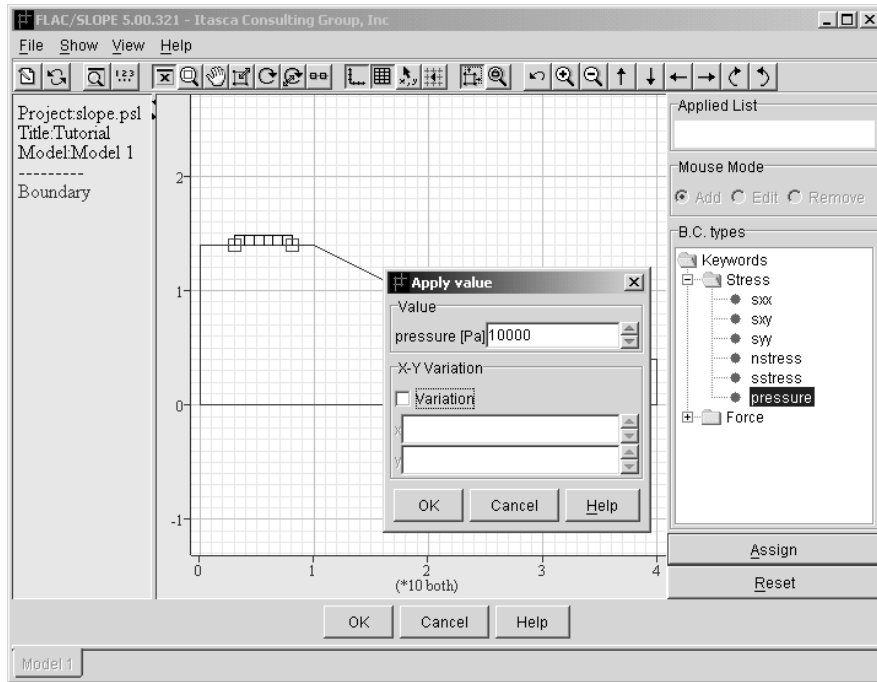


Figure 1.41 Apply value dialog in the Apply tool

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 1.42 shows the dialog. The sign conventions and formula for applying a spatial variation in load are described below.

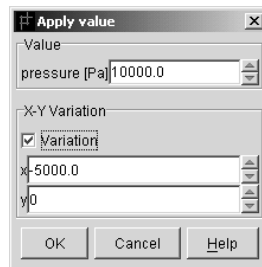


Figure 1.42 Editing the applied value in the Apply value dialog

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 *sstress*-type name. The sign convention for the stress types, *sxx*, *syy* and *nstress*, is that positive values indicate tension. The sign convention for shear stress types,

s_{xy} and s_{stress} , is illustrated by [Figure 2.43](#) in the **User's Guide** of the full *FLAC* manual. The sign convention for *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). *pressure* and *nstress* apply the same type of loading, but with an opposite sign convention.

When stresses are applied in *FLAC/Slope*, they are converted into forces applied at boundary grid-points 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, $xforce$ and $yforce$ (shown in [Figure 1.40](#)), can also be applied to represent a point (i.e., line) load on the boundary. A positive x - or y -force acts in the positive x - or y -direction.

The sense of the applied stress or force can be checked by entering the *Solve* stage after pressing to leave the *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 1.43](#) illustrates the applied forces that correspond to the applied pressure variation prescribed in [Figure 1.42](#).

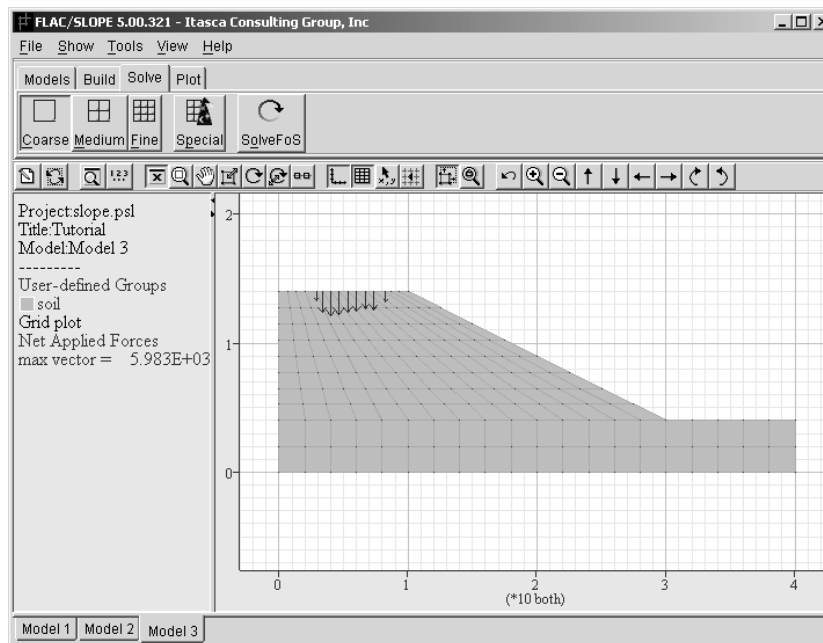


Figure 1.43 Applied forces corresponding to the applied pressure in [Figure 1.42](#)

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

$$v = v^{(s)} + \frac{x - x^{(s)}}{x^{(e)} - x^{(s)}} v_x + \frac{y - y^{(s)}}{y^{(e)} - y^{(s)}} v_y \quad (1.4)$$

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 1.42](#), the coordinate range is $(x^{(s)}, y^{(s)}) = (3, 14)$ and $(x^{(e)}, y^{(e)}) = (8, 14)$. The variation in pressure is only in the x -direction ($v_y = 0$). Thus, by using [Eq. \(1.4\)](#), we get

$$v = 10,000 - 1,000(x - 3.0) \quad (1.5)$$

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 1.43](#).

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 button in the tool bar. This opens the *Reinforcement* tool, as shown in [Figure 1.44](#). Cable elements are installed in a slope by first checking the 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 1.44](#). 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 radio button. Then, press and drag the end node with the mouse. Cables can be deleted from the slope by checking the radio button. You can then click the mouse over the cable(s) you wish to delete.

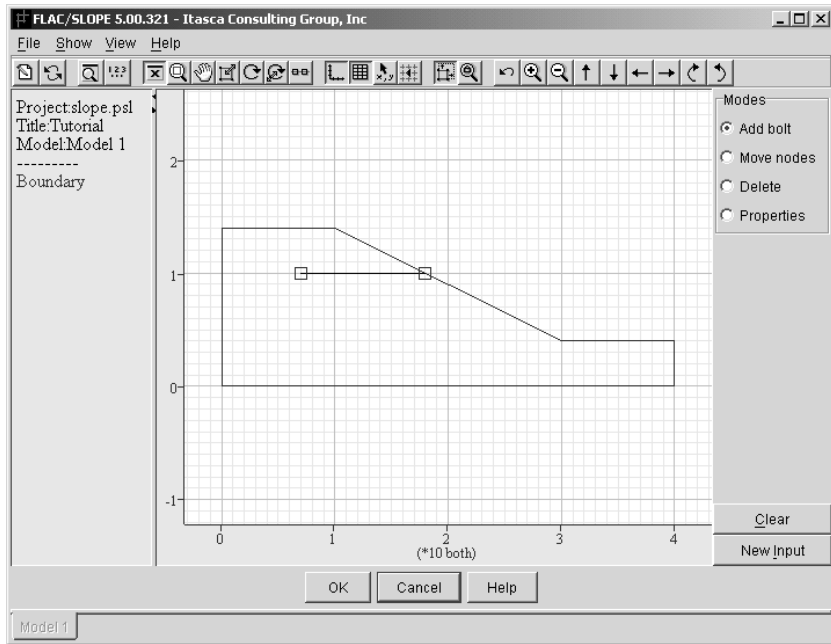


Figure 1.44 Reinforcement tool

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 1.45.

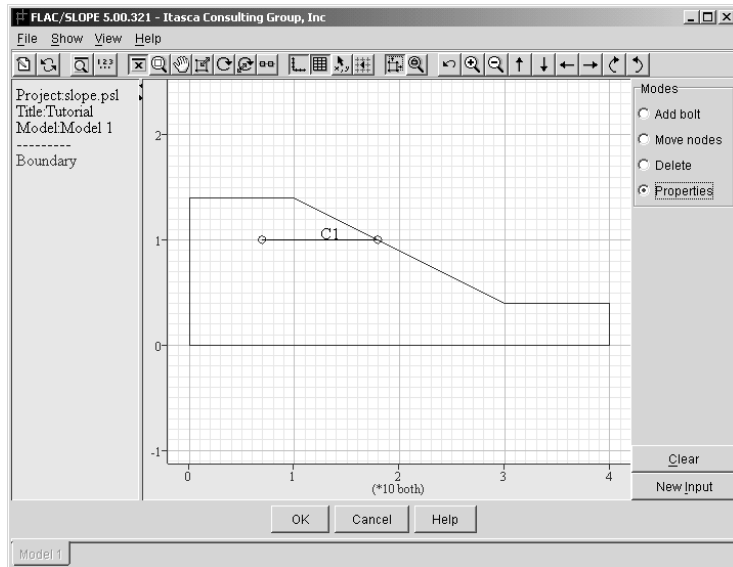


Figure 1.45 Property identification number for reinforcement

By clicking the mouse over the property number, a *Cable Element Properties* dialog will open, as shown in [Figure 1.46](#). Properties are then assigned to a specific property number.

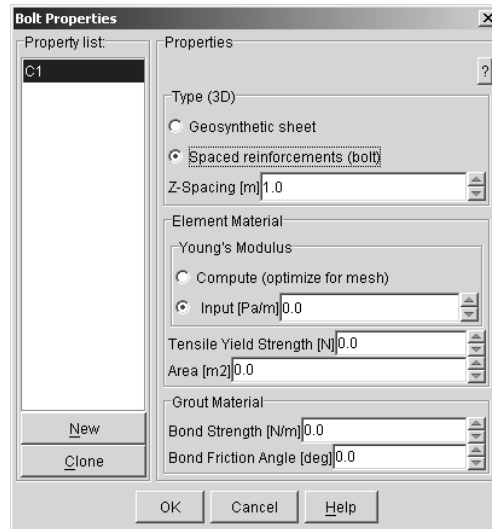


Figure 1.46 *Cable Element Properties dialog*

Two types of reinforcement can be simulated: a continuous **GEOSYNTHETIC SHEET** or a **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 **Structural Elements** of the full *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 *Cable Element 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 *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^{max} , that can develop along the interface per length, L , of the cable:

$$\frac{F_s^{max}}{L} = c_b + \sigma'_c \times \tan(\phi_b) \times p \quad (1.6)$$

where: c_b = bond strength or cohesion [force/cable length];
 σ'_c = mean effective confining stress normal to the element;
 ϕ_b = bond friction angle [degrees]; and
 p = perimeter of the element (based on input area).

See [Section 1.4.1.2](#) in **Structural Elements** of the full *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 4.4.1](#) in **Theory and Background** of the full *FLAC* manual for more information on the rationale for selection of stiffness values.)

The reinforcement properties are assigned to a property number — in [Figure 1.46](#), this is *C1*. Additional property numbers can be created by pressing the button in the *Cable Element Properties* dialog. A new property number, *C2*, will be added to the *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 is pressed.

Different segments along a cable can also be assigned different property numbers — e.g., to simulate bonded and unbonded portions of a grouted bolt. [Figure 1.47](#) shows a bolt composed of two segments. This is created in the 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 , we can then assign properties for the unbonded segment to *C1*, and the bonded segment to *C2*. We change the left portion of the bolt in [Figure 1.47](#) to *C2* by highlighting *C2* in the *Cable Element Properties* dialog.

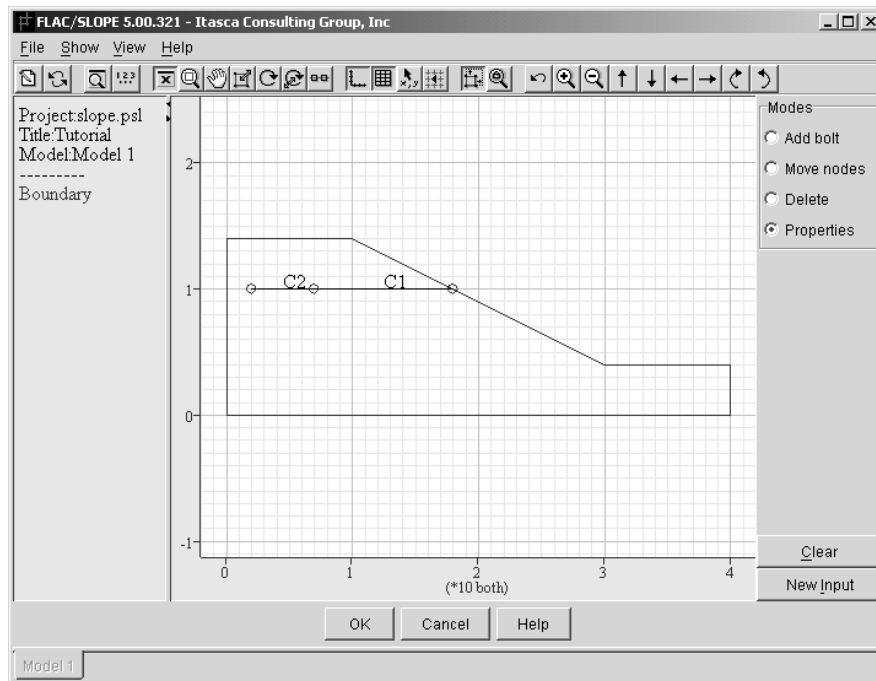


Figure 1.47 *Creating a grouted and ungrouted bolt*

Once we are satisfied with all the reinforcement conditions and properties we have specified, we click 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 **BUILD** tool bar, the tool opens as shown in [Figure 1.48](#). 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 1.48](#), 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.

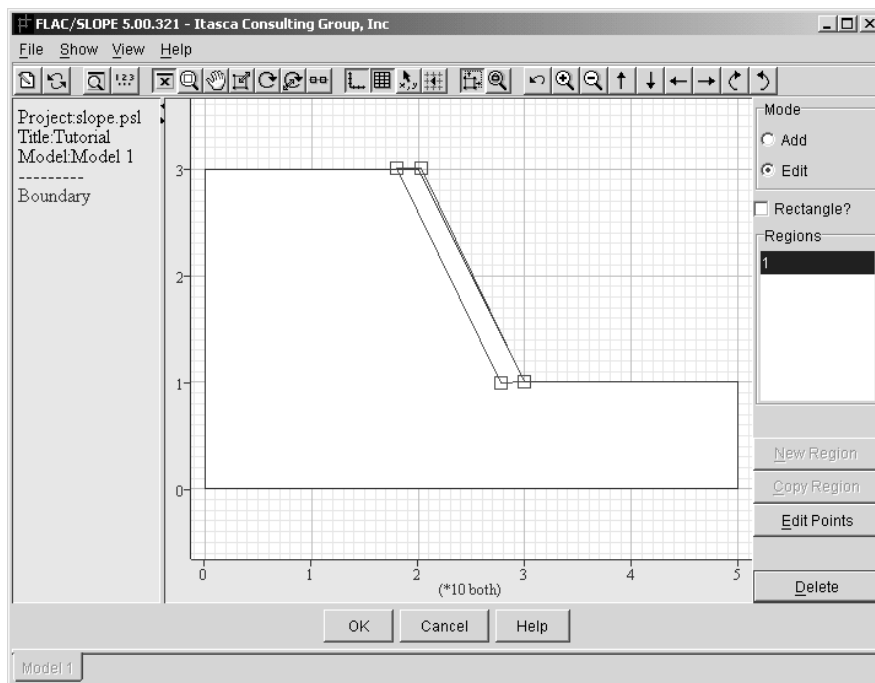


Figure 1.48 Excluded region tool

After the excluded region is accepted, by pressing **OK**, 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 1.49](#).

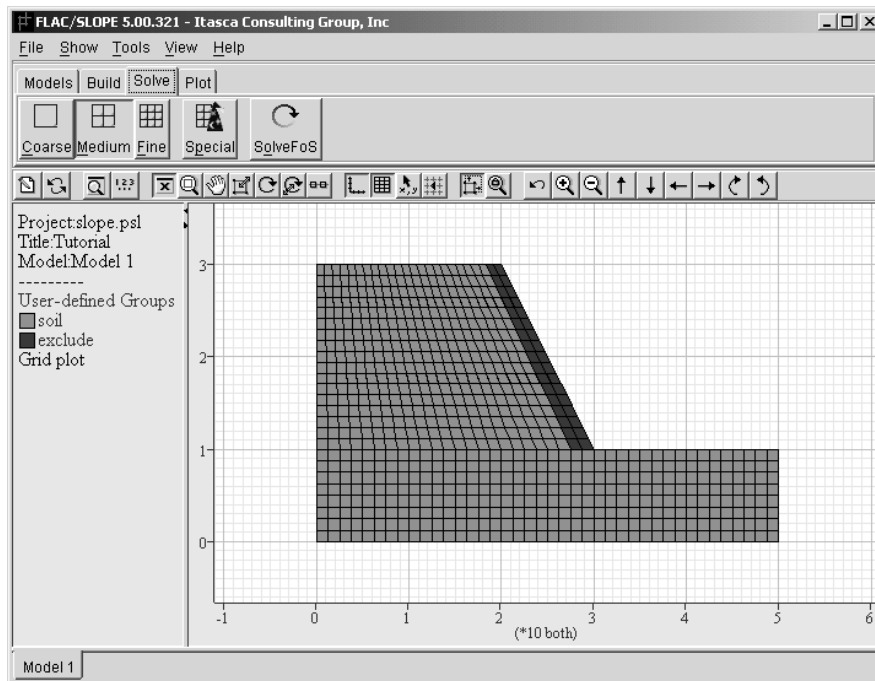


Figure 1.49 Medium-zoned mesh showing zones excluded from the factor-of-safety calculation

The effect of using the *Exclude* tool is shown by comparing factor-of-safety results for the slope shown in Figure 1.48, with and without the excluding region. Figure 1.50 presents the factor-of-safety plot for the case without an excluded region, and Figure 1.51 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.

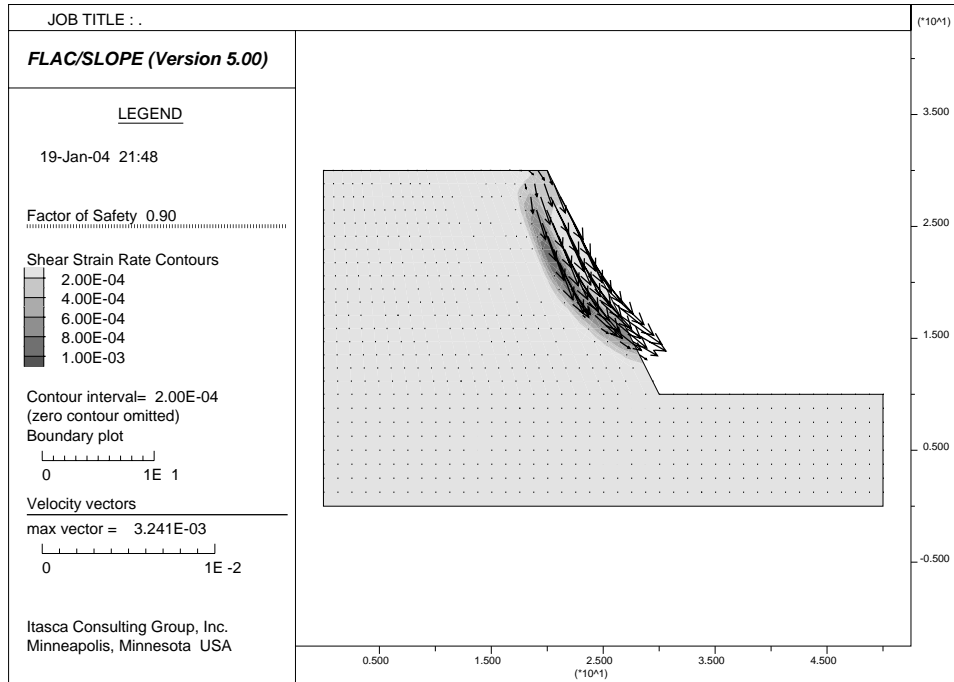


Figure 1.50 Factor-of-safety calculation with no excluded region

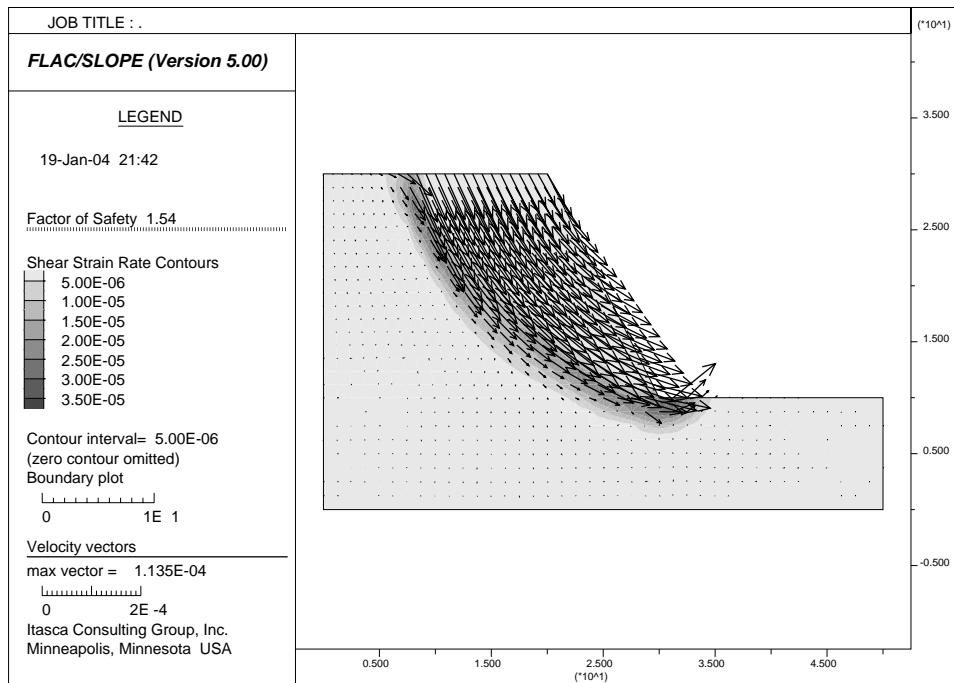


Figure 1.51 Factor-of-safety calculation with excluded region

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 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 button, a “coarse-grid” model is created, as shown in [Figure 1.52](#). If the button is pressed, a “medium-grid” model appears, as shown in [Figure 1.53](#), and if the button is pressed, a “fine-grid” model appears, as shown in [Figure 1.54](#). 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 1 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 comprehensive studies. The results for this type of zoning are found to be in good 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 1 GHz computer the solution typically requires a few minutes to complete.

A fine-grid model is recommended as a check on analyses made with the medium-grid model. The factor-of-safety calculation with the fine-grid model should agree very closely with that from the medium-grid model. However, because this type grid takes longer to calculate a safety factor, using fine-grid models for comprehensive studies is usually not warranted.

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 model, 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 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, the zoning tool containing a vertical column within a horizontal layer is used for the model, as described previously in [Figure 1.28](#), 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 1.55](#). Also, the box is not checked in order to create a uniform zoning.

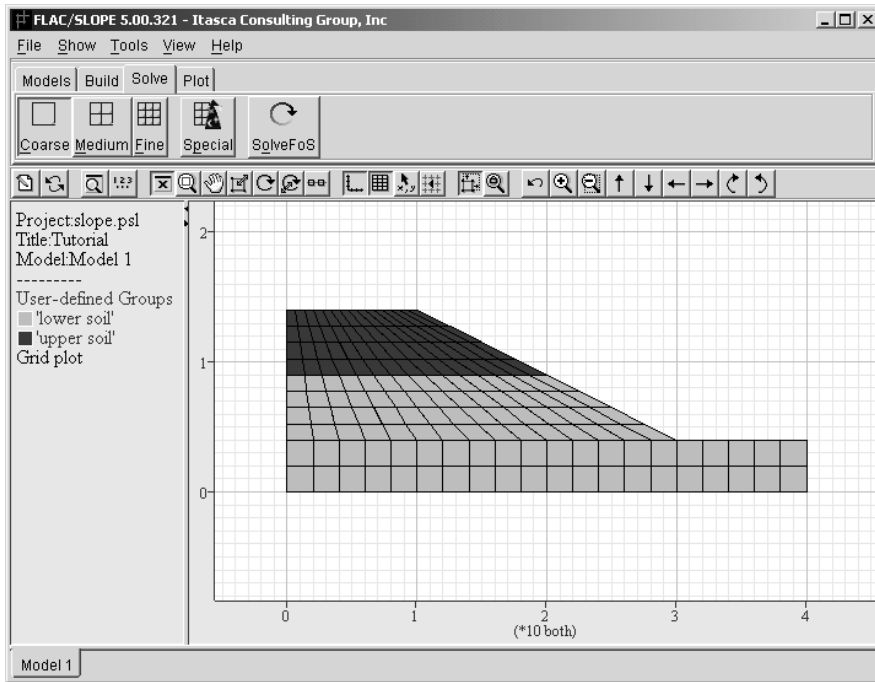


Figure 1.52 Coarse-grid model

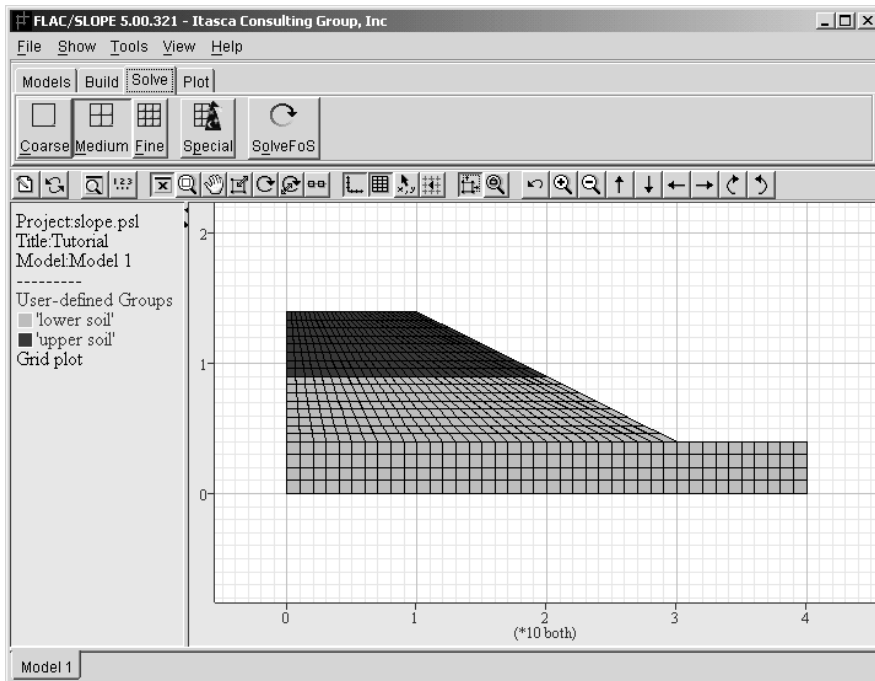


Figure 1.53 Medium-grid model

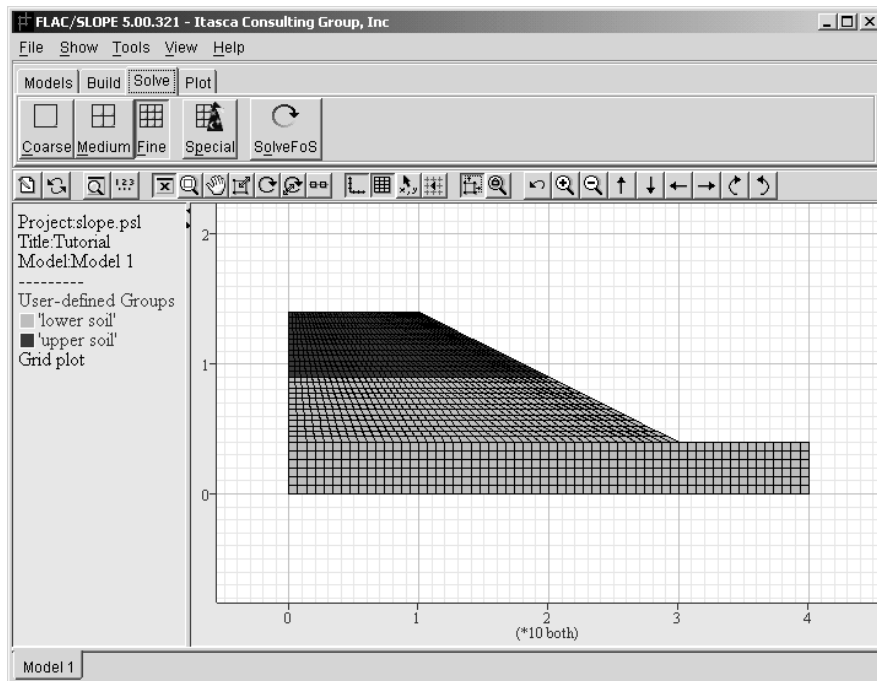


Figure 1.54 Fine-grid model

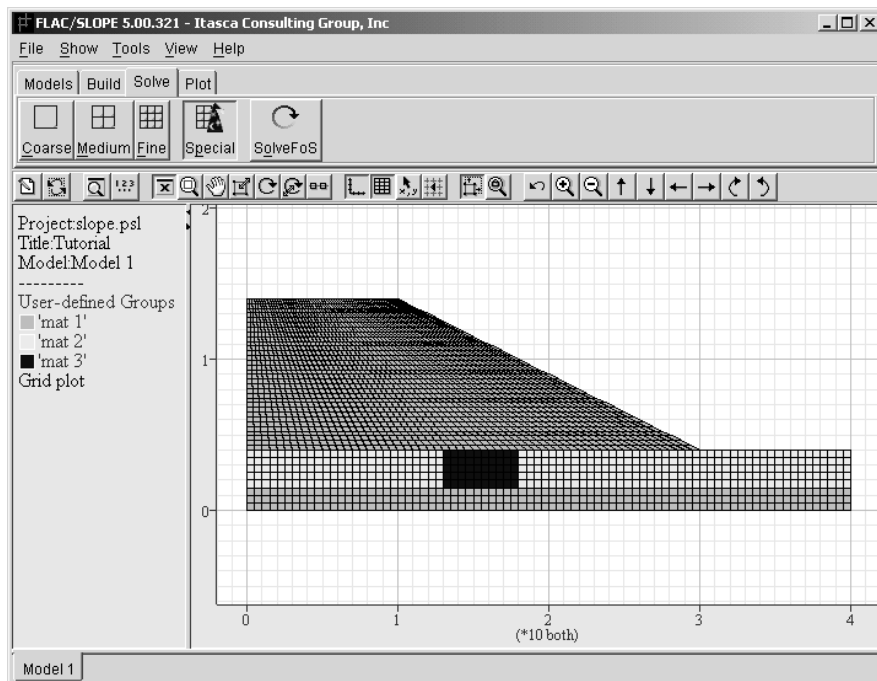


Figure 1.55 Special-zoning model with 80-zone mesh density

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 medium-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_Medium.sav.” Note that this save file is deleted after the factor-of-safety calculation is completed. (See [Section 1.3.11.3](#).)

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 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 is pressed, a *Factor of Safety parameters* dialog opens, with the and boxes checked, as shown in [Figure 1.56](#). By pressing , the calculation will commence.

It is also possible to include other strength parameters in the safety-factor calculation. By checking the 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 box should be checked to include these interface strength properties in the strength reduction solution. If structural reinforcement is included in the model, the box should be checked to include grout bond strength and bond friction angle properties 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.

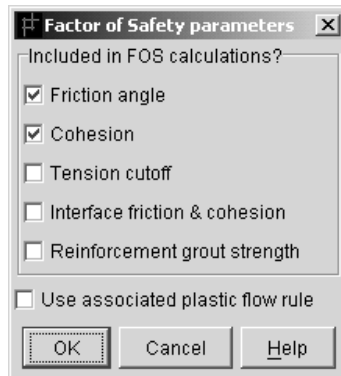


Figure 1.56 *Factor-of-safety parameters dialog*

Associated or non-associated plastic flow can also be specified for the factor-of-safety calculation via the 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 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 dilation angle will be set equal to the friction angle during the safety-factor calculation; otherwise, it will be held constant at its assigned value.

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 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 1.57](#). 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.

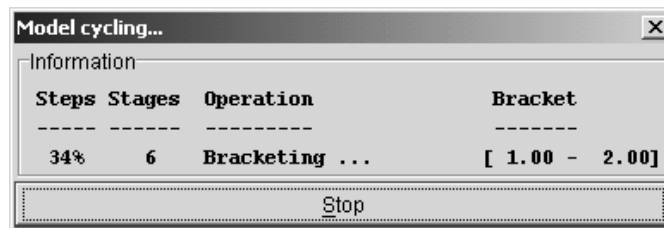


Figure 1.57 *Model cycling dialog*



After the first bracketing values have been found in the series, the run can be interrupted, by pressing the 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 complete.

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_Medium_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
- 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  button. When a calculation is complete, a “factor-of-safety”-plot button is added to the  tool bar with a name corresponding to the type of zoning and factor-of-safety parameters selected for the calculation. For example, in [Figure 1.58](#), the button contains a four-square symbol, indicating a medium-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.

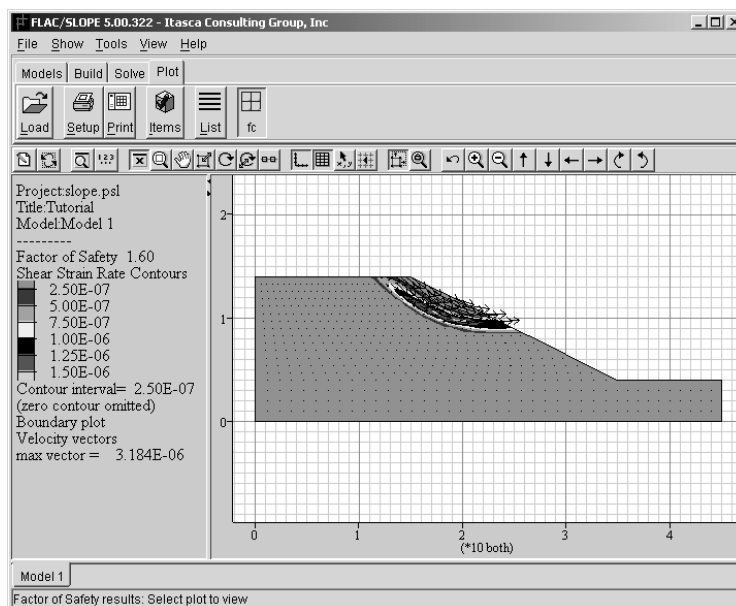


Figure 1.58 Factor-of-safety plot for medium grid model with friction angle and cohesion included in the factor-of-safety calculation

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 Section 1.2.2, these plot items show a well-defined failure surface and indicate a rotational failure mode, as illustrated in Figure 1.58.

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 1.59. 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 CENTROID (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 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.

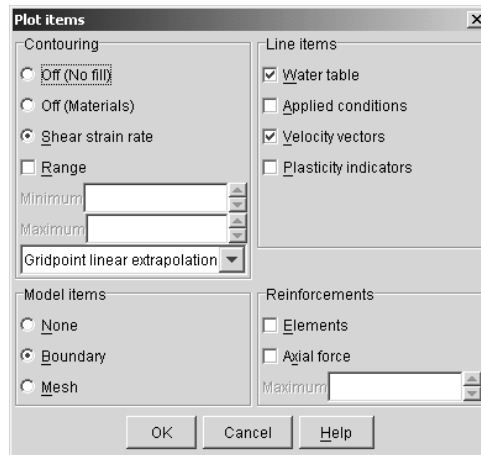


Figure 1.59 Factor-of-safety Plot items dialog

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 also 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 1.90](#) and [1.91](#).

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](#) tool bar for the loaded model. The list of factor-of-safety plots can be edited, and factor-of-safety plots removed from the tool bar, 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 *Print setup* dialog, which is opened by pressing the [SETUP](#) button. The dialog is shown in [Figure 1.60](#). 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.

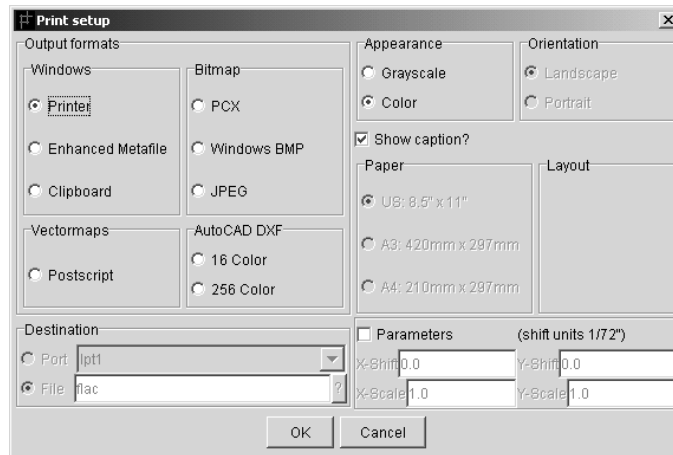


Figure 1.60 *Print setup dialog*

A report mode is also provided to summarize the results of a study. Click on the FILE/CREATE REPORT ... menu item to open the dialog as shown in [Figure 1.61](#). This will create an HTML-formatted file listing various tables of information for the study. This file can then be pasted into a report document, such as a Microsoft Word file.

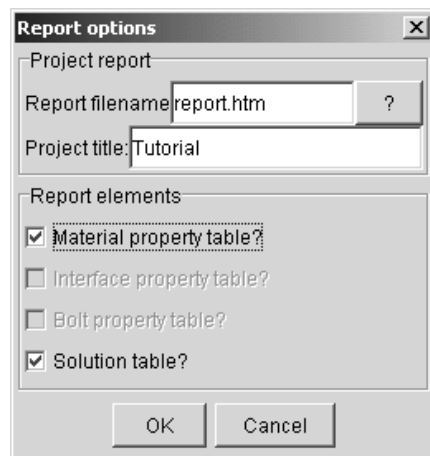


Figure 1.61 *Report options dialog*

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 1.62](#). 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.

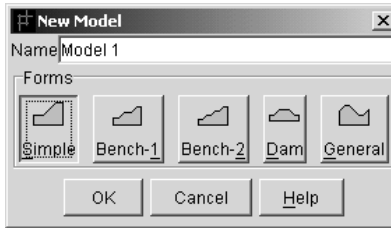


Figure 1.62 *New Model dialog*

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 1.63](#). The dimensions for the bench are defined in the diagram included in this dialog. For example, using the dimensions shown in [Figure 1.63](#), a bench boundary is produced, as illustrated in [Figure 1.64](#). A two-bench slope is produced in a similar fashion when the **BENCH-2** button is pressed.

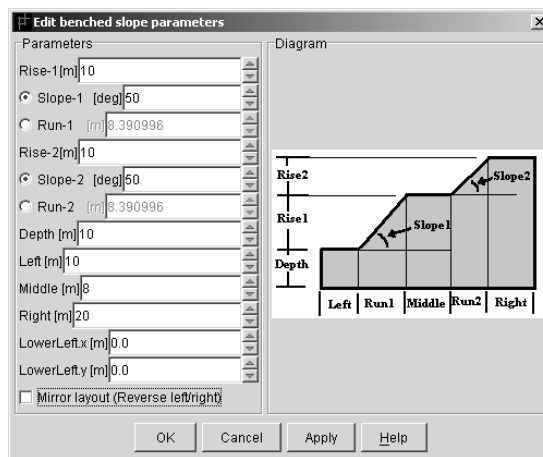


Figure 1.63 *Edit benched slope parameters dialog*

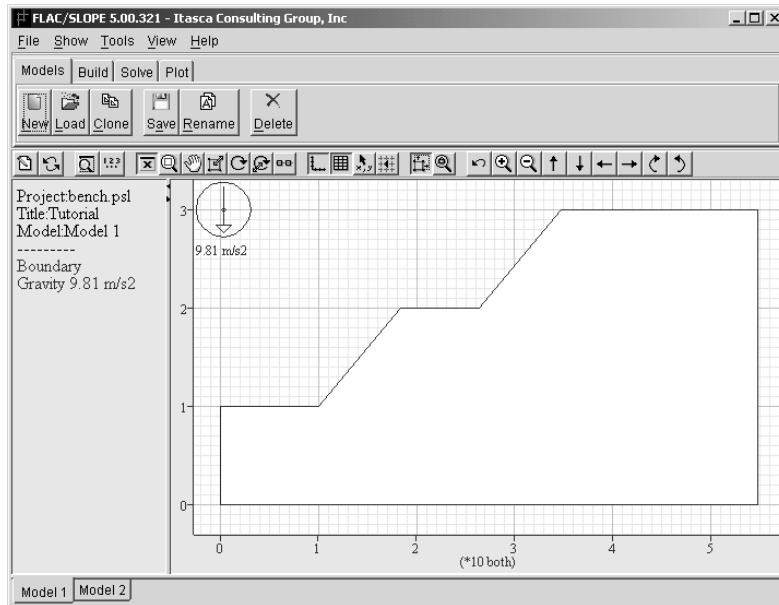


Figure 1.64 Single bench slope boundary

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 1.65. The dimensions for the dam are defined in the diagram included in this dialog. For example, using the dimensions shown in Figure 1.65, a dam boundary is produced as illustrated in Figure 1.66.

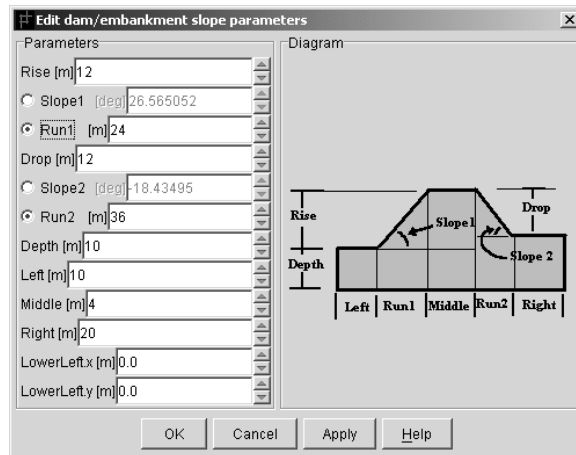


Figure 1.65 Edit dam/embankment slope parameters dialog

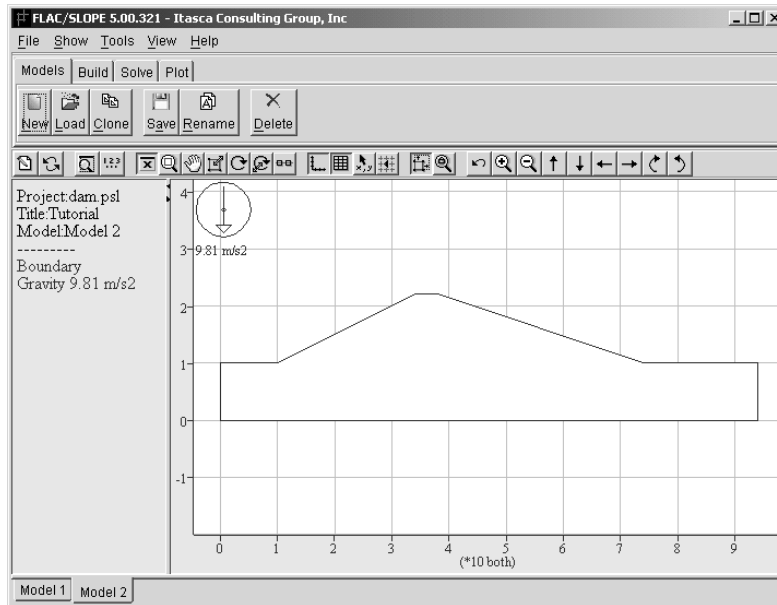


Figure 1.66 Dam boundary

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 1.67. 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.

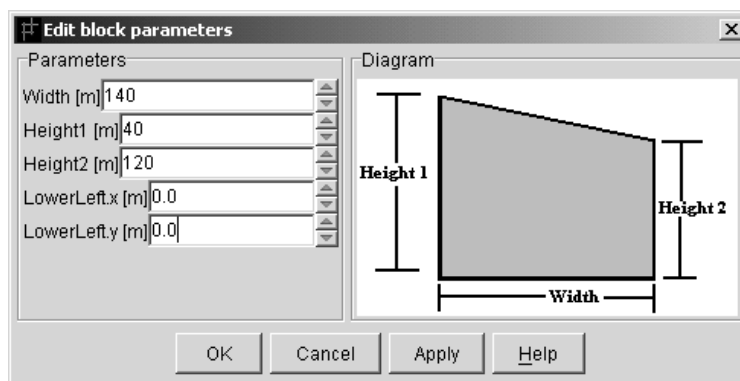


Figure 1.67 Edit block parameters dialog

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 1.68 shows the *General boundary* tool with a nonlinear slope defined by seven handle points. Figure 1.69 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 1.70, 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 1.71. The final slope boundary is shown in Figure 1.72.

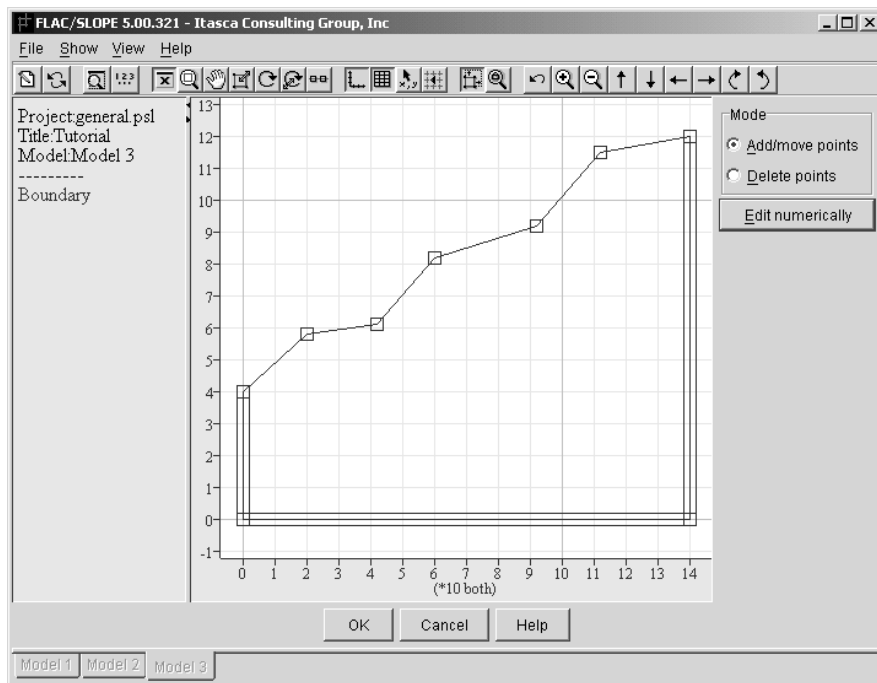


Figure 1.68 *General boundary tool*

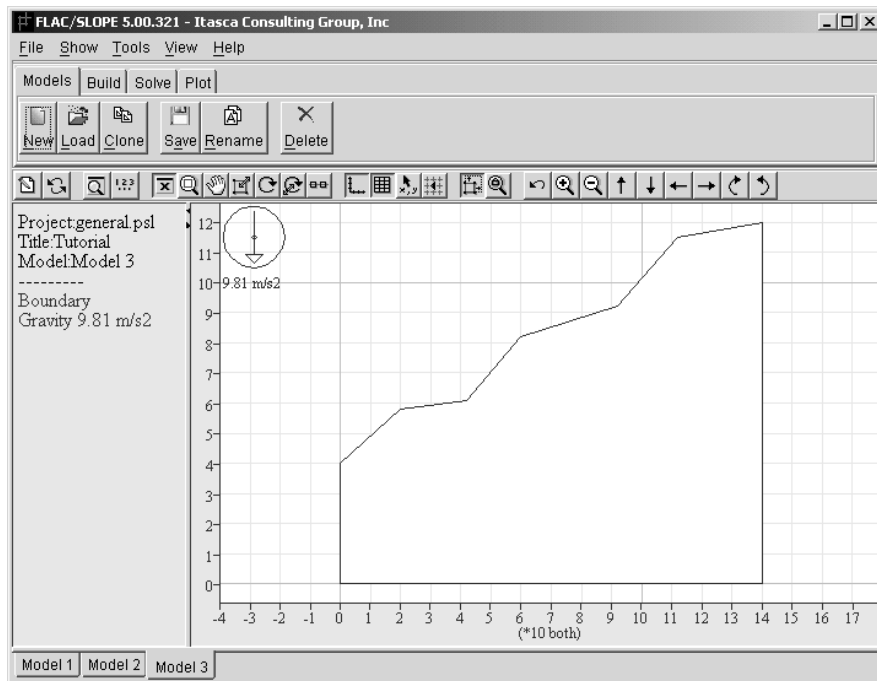


Figure 1.69 Nonlinear slope boundary

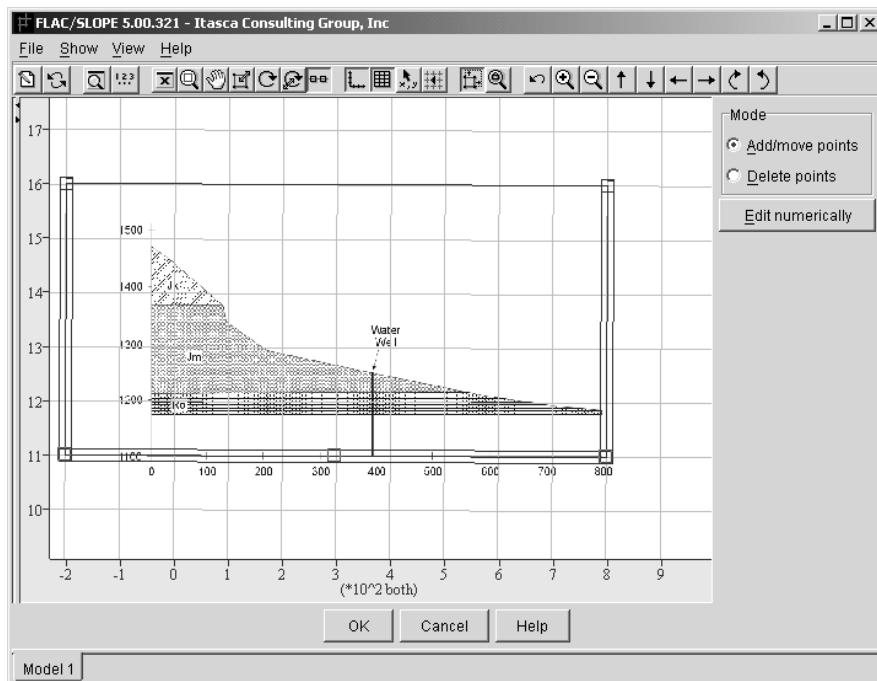


Figure 1.70 Bitmap image imported onto the model view

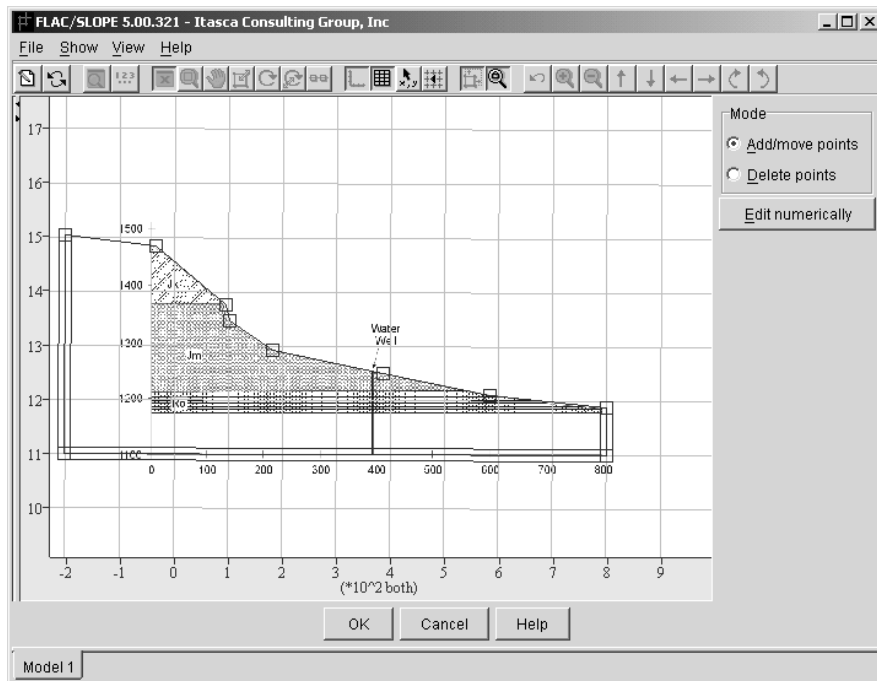


Figure 1.71 Model slope adjusted to fit slope of bitmap image

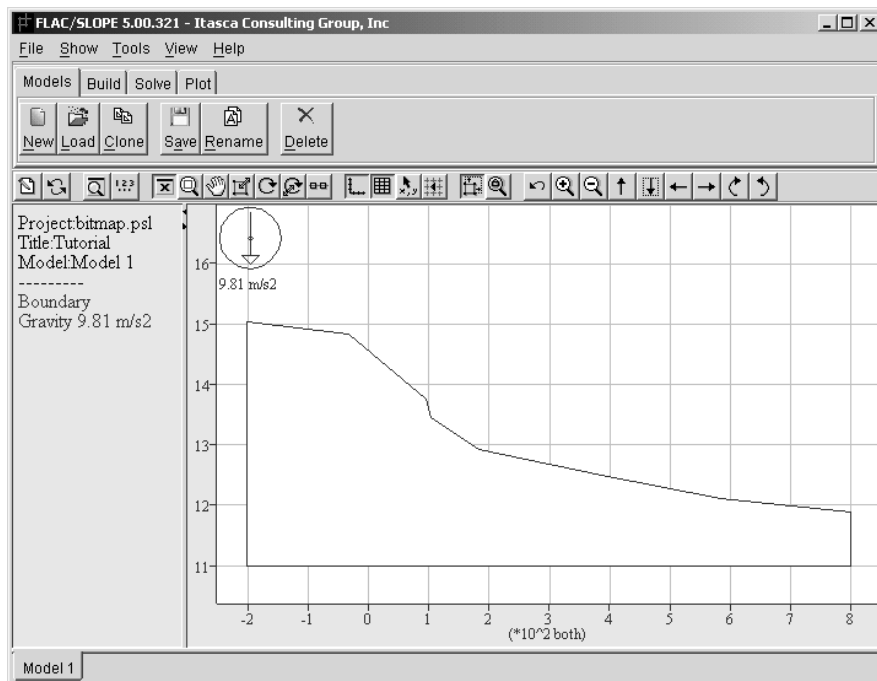


Figure 1.72 Slope boundary created from bitmap image

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 “FLAC500\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 (1975). The problem setting is a homogeneous embankment of height $H = 10$ m, slope angle $\beta = 45^\circ$, unit weight $\gamma = 20$ kN/m³, cohesion $c = 12.38$ kPa and friction angle $\phi = 20^\circ$. A gravitational acceleration of 10.0 m/sec² 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 1.73 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.”

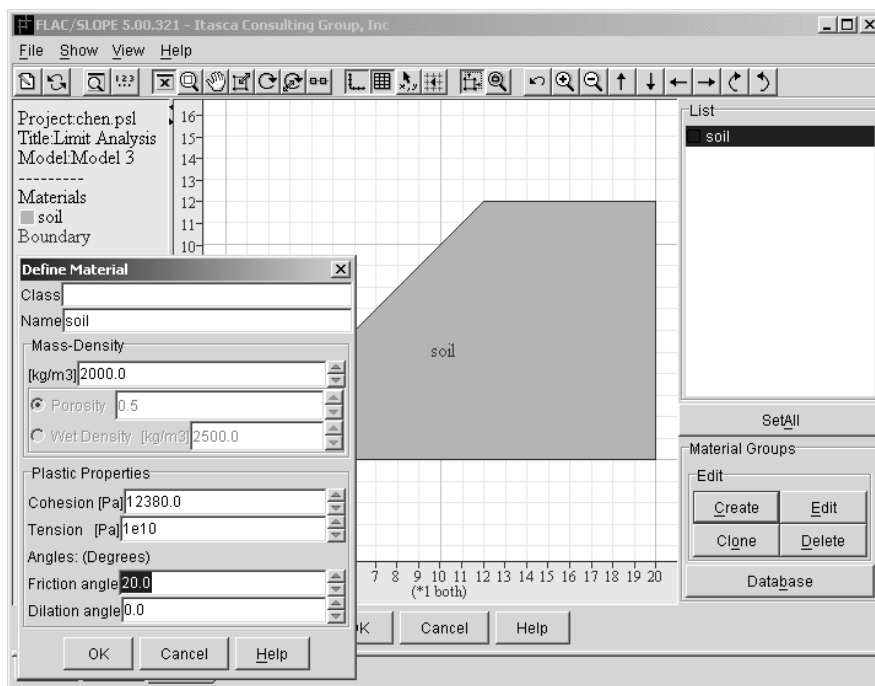


Figure 1.73 Material properties for homogeneous embankment example

We use the *Medium* grid mode in the *Solve* stage; the resulting grid is shown in [Figure 1.74](#). We perform the factor-of-safety calculation and calculate a factor of 1.01. The failure surface is indicated in [Figure 1.75](#). 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.

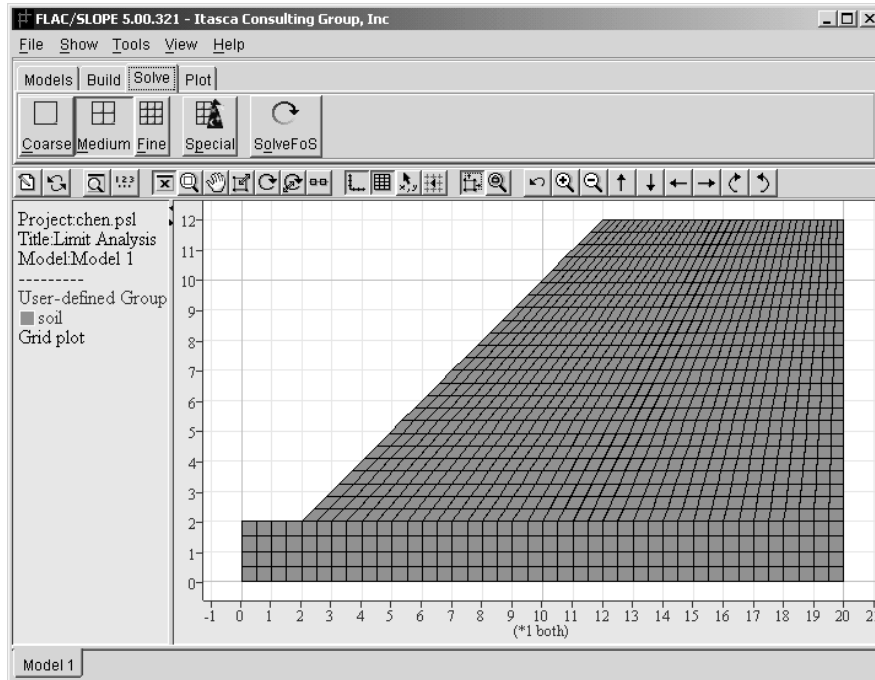


Figure 1.74 *Medium-grid zoning for homogeneous embankment example*

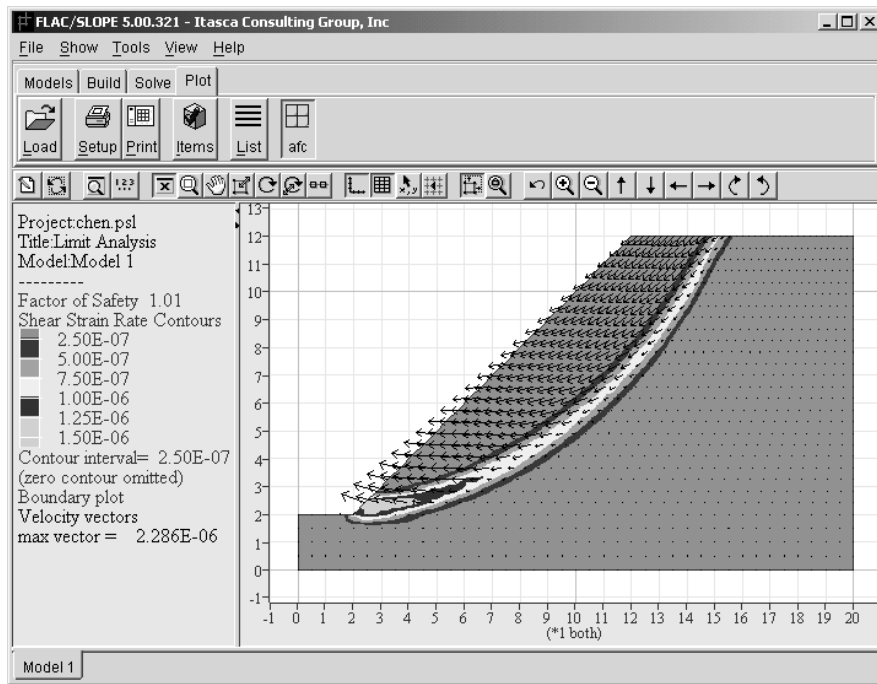


Figure 1.75 Failure surface calculated for homogeneous embankment

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 1.76. Four of the cases analyzed by Fredlund and Krahn (1977) are re-analyzed 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 1.77 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 1.78.

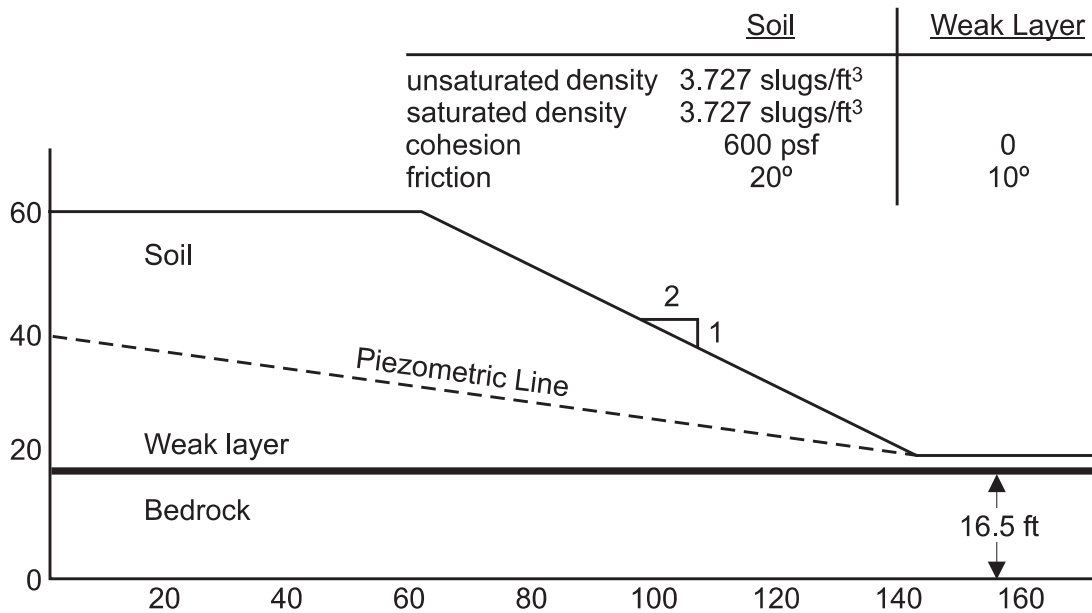


Figure 1.76 Slope stability example (from Fredlund and Krahn, 1977)

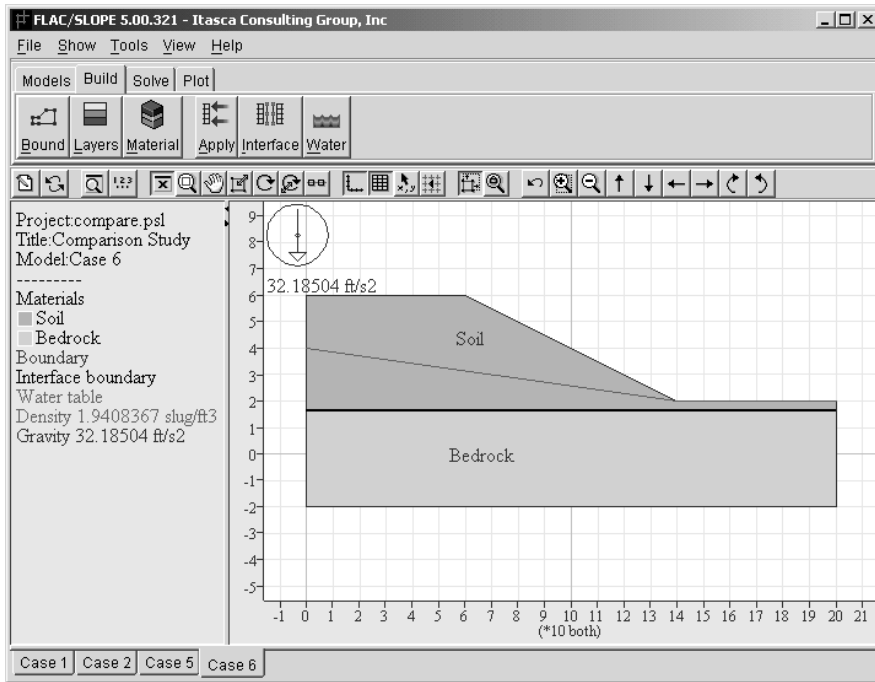


Figure 1.77 FLAC/Slope geometry for Case 6

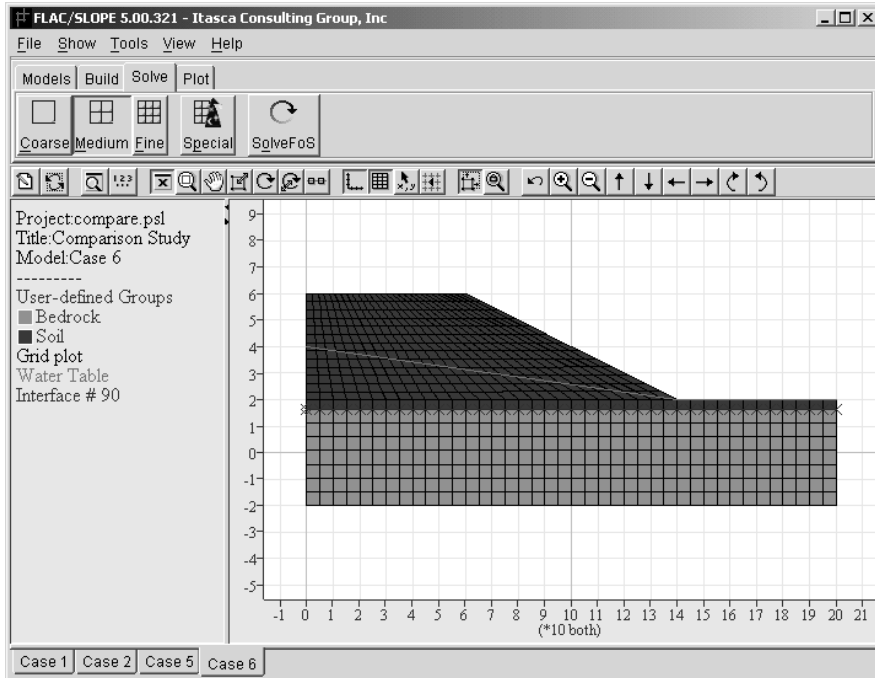


Figure 1.78 FLAC/Slope grid for Case 6

The result for the factor-of-safety calculation for Case 6 is illustrated in Figure 1.79. 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.

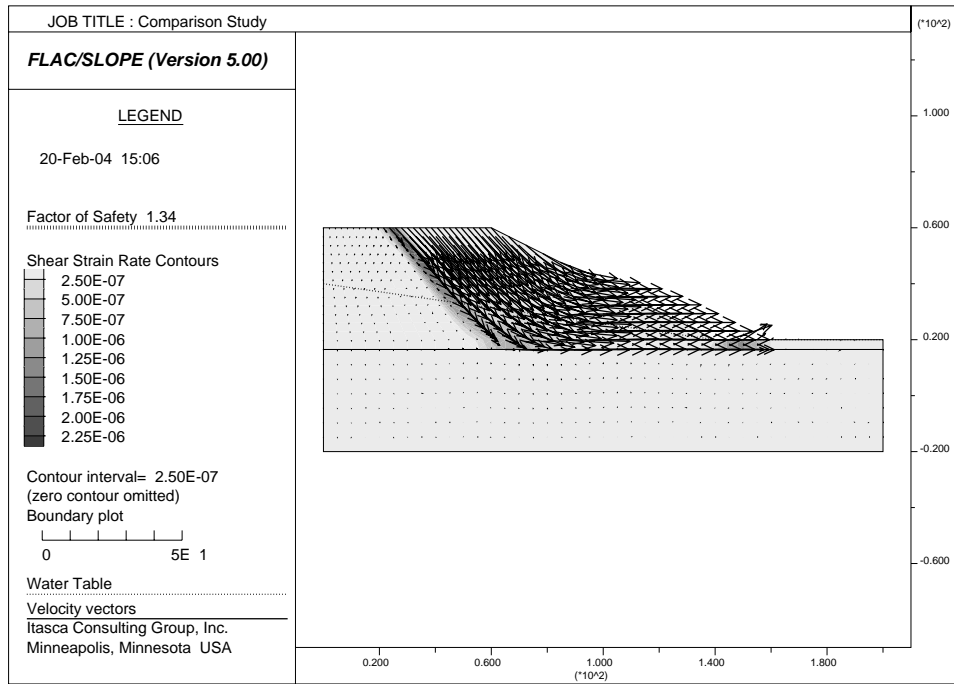


Figure 1.79 Factor-of-safety results for Case 6

Table 1.1 Results from Fredlund and Krahn (1977) study compared to *FLAC/Slope*

Case	Simplified Bishop Method	Spencer's Method	Janbu's Rigorous Method	Morgenstern-Price Method	<i>FLAC/Slope</i>
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 1.80. 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.

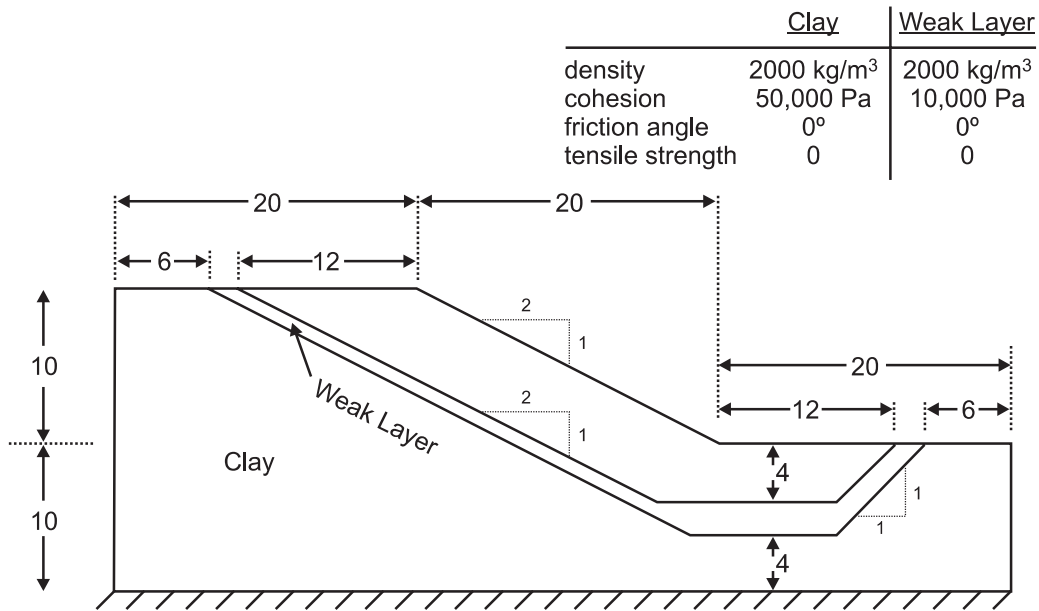


Figure 1.80 Clay slope containing a thin weak layer (from Griffiths and Lane, 1999)

The thin layer is created in the *FLAC/Slope* model by adjusting two layer boundaries to match the locations denoted in Figure 1.80. 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 1.81. The resulting model is shown in Figure 1.82. A fine-grid model is necessary to represent the thin layer — see Figure 1.83. Three cases are analyzed: $c_l/c = 0.2, 0.6$ and 1.0 . The project save file for this example is “THIN.PSL.”

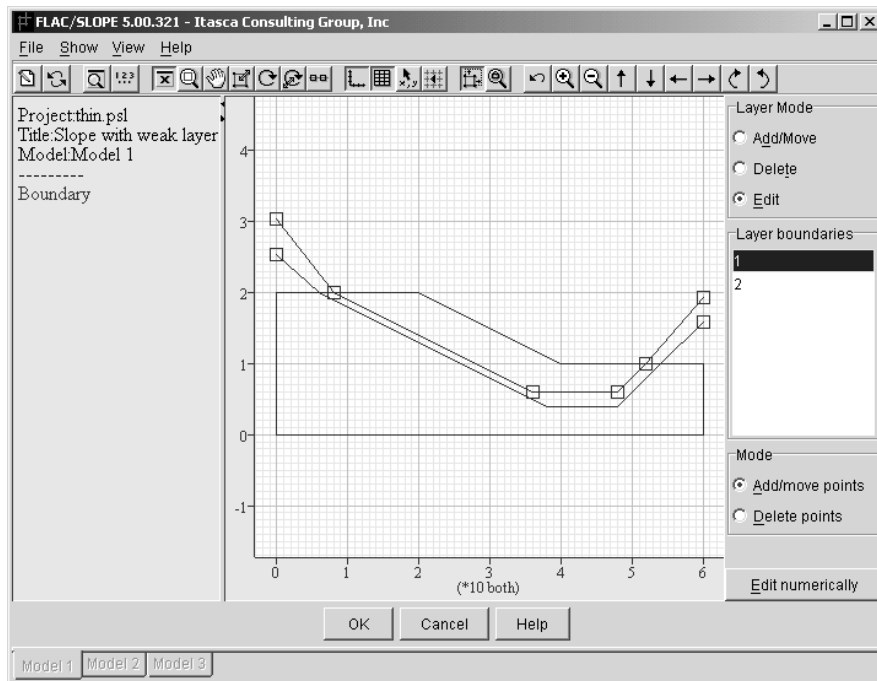


Figure 1.81 Weak layer boundaries created in the Layers tool

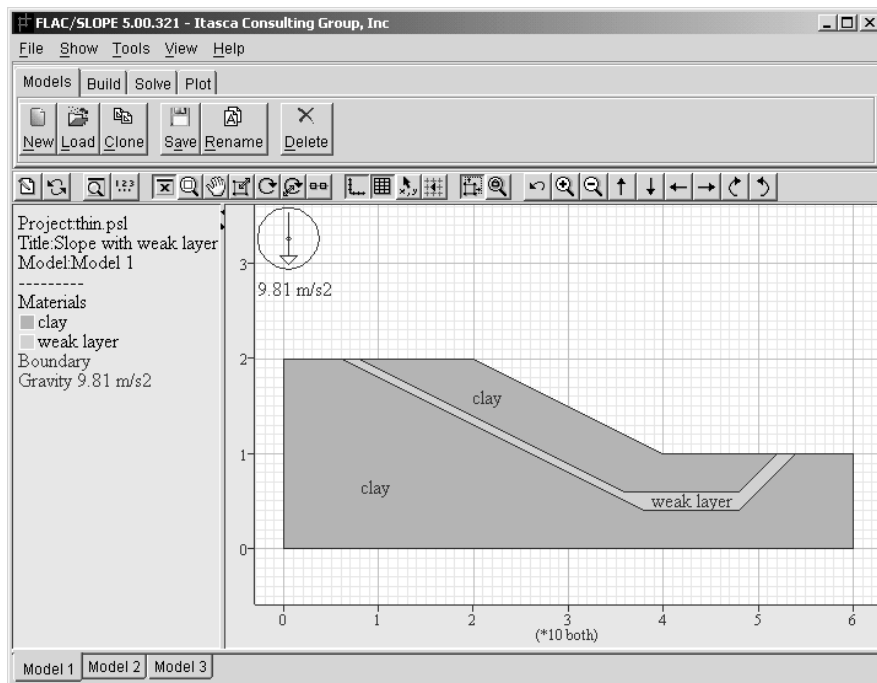


Figure 1.82 FLAC/Slope model of slope with a thin weak layer

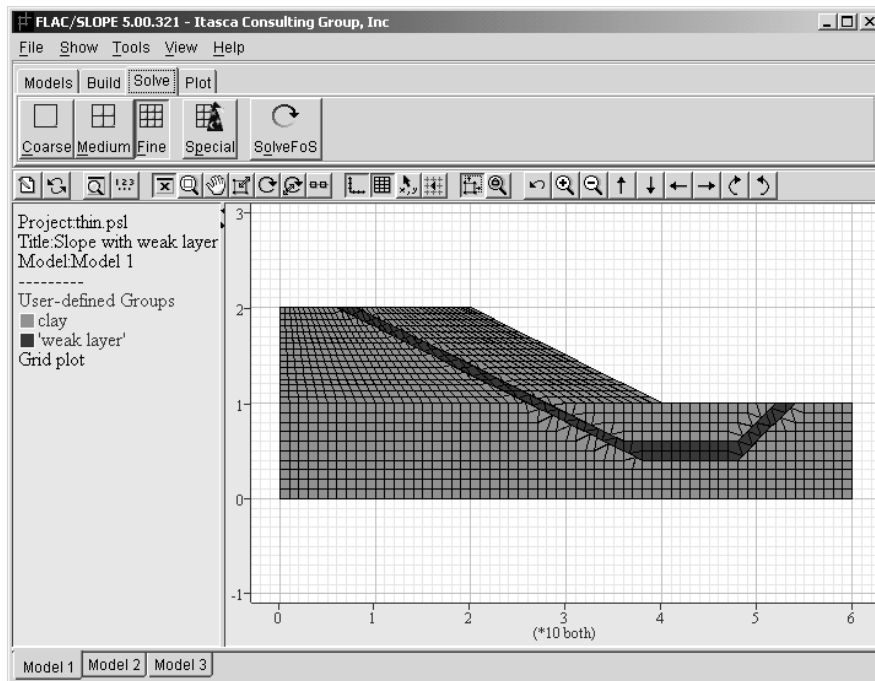


Figure 1.83 Fine-grid model for slope with thin weak layer

The factor-of-safety plots for the three cases are shown in [Figures 1.84](#) through [1.86](#). 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 1.84](#), the failure surface indicates localized slip along the weak plane, while in [Figure 1.86](#), a circular failure surface develops in the homogeneous material. [Figure 1.85](#) 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 $c_l/c = 0.6$ 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.

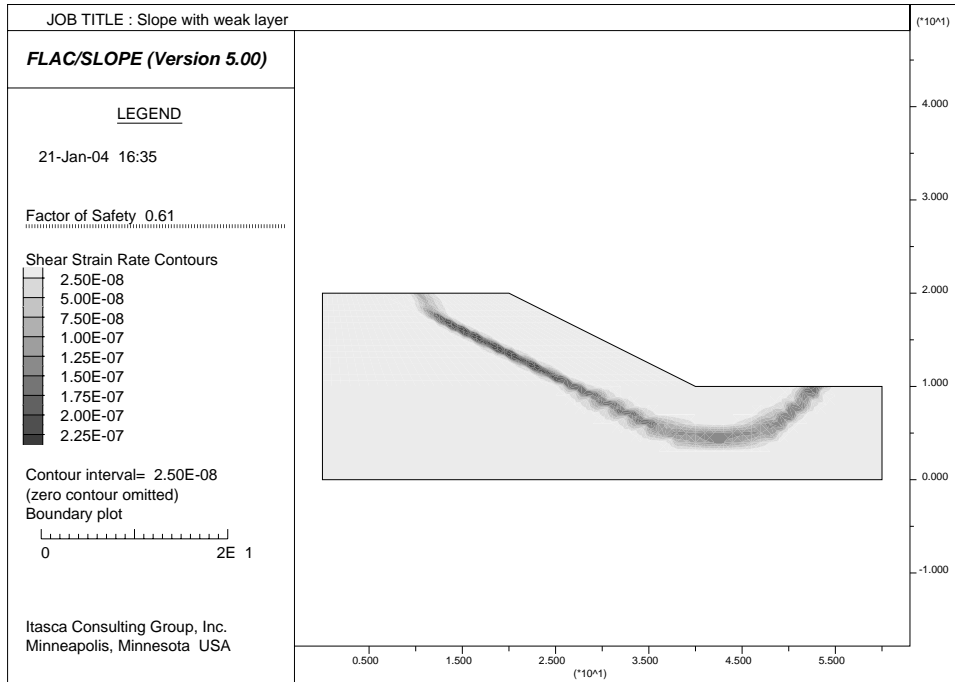


Figure 1.84 Factor-of-safety plot for $c_1 / c = 0.2$

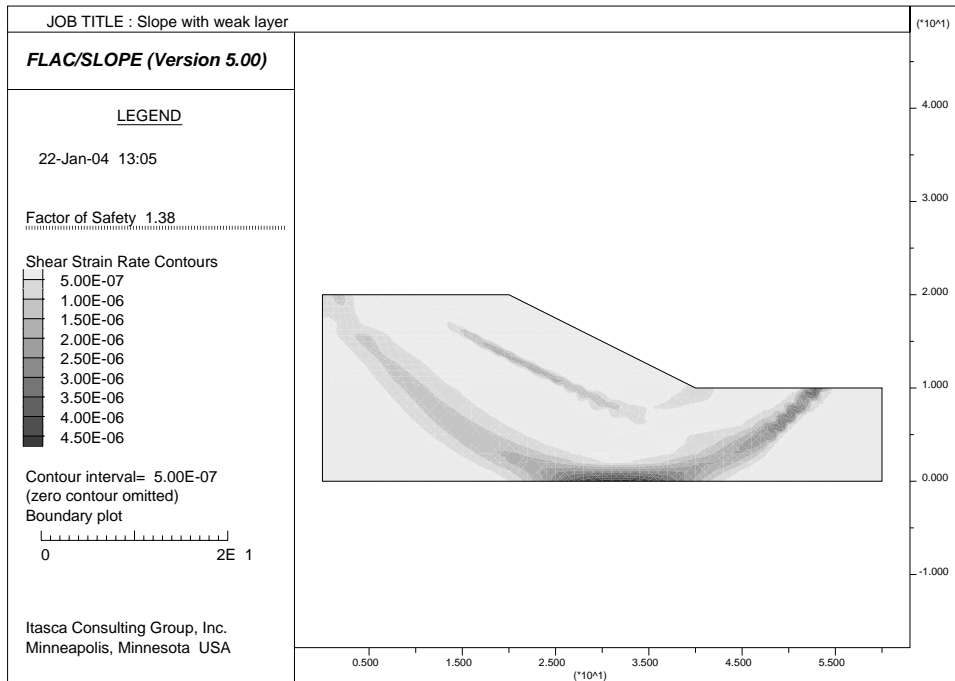


Figure 1.85 Factor-of-safety plot for $c_1 / c = 0.6$

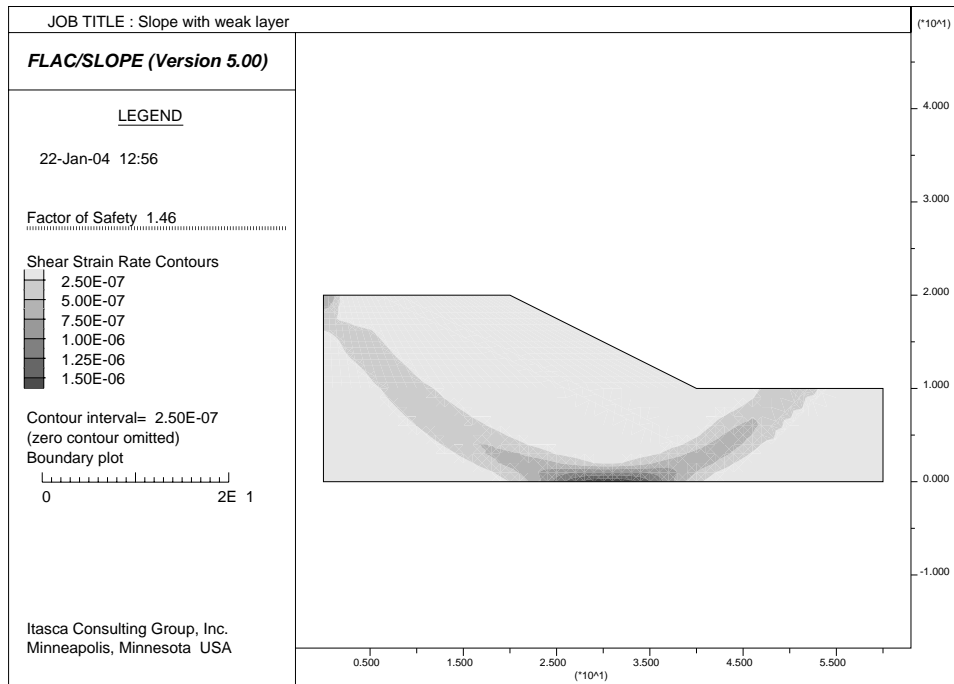


Figure 1.86 Factor-of-safety plot for $c_l / c = 1.0$

1.4.4 Slope with Geogrid Reinforcement

In this example, two layers of geogrid are used to stabilize a slope. The slope conditions and material properties for this model are shown in [Figure 1.87](#). The project save file is “GEOGRID.PSL.” A *Medium* mesh is used for this example.

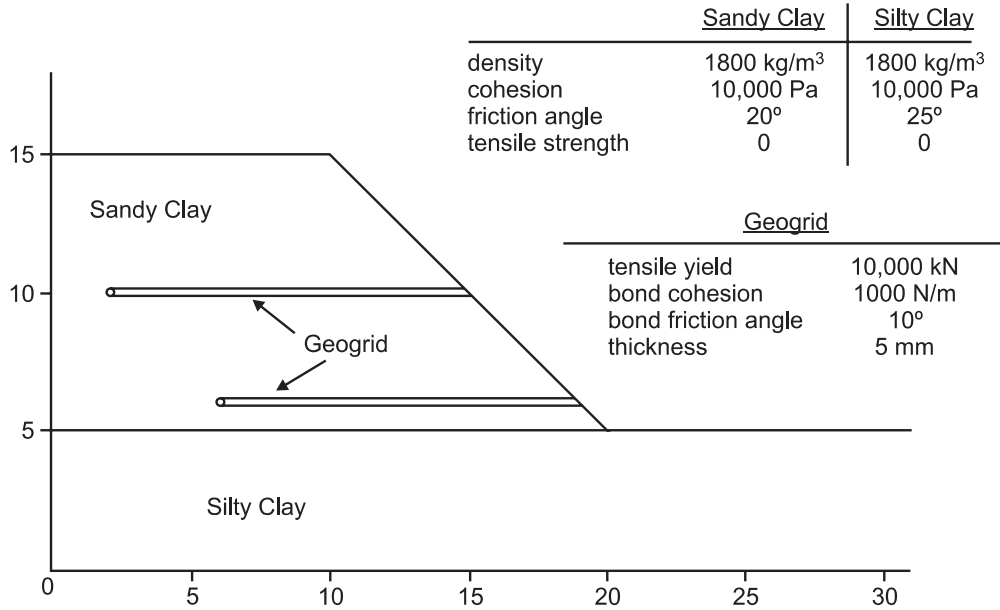


Figure 1.87 Slope with geogrid reinforcement

The slope is unstable without the geogrid reinforcement. The results for the unsupported case are shown in [Figure 1.88](#). The factor of safety is calculated to be 0.93.

The properties selected for the geogrid reinforcement are assigned in the *Cable Element Properties* dialog, as shown in [Figure 1.89](#). 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 1.90](#). The safety factor is now increased to 1.13.

The effect of the bonding resistance provided at the geogrid/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 1.91](#).

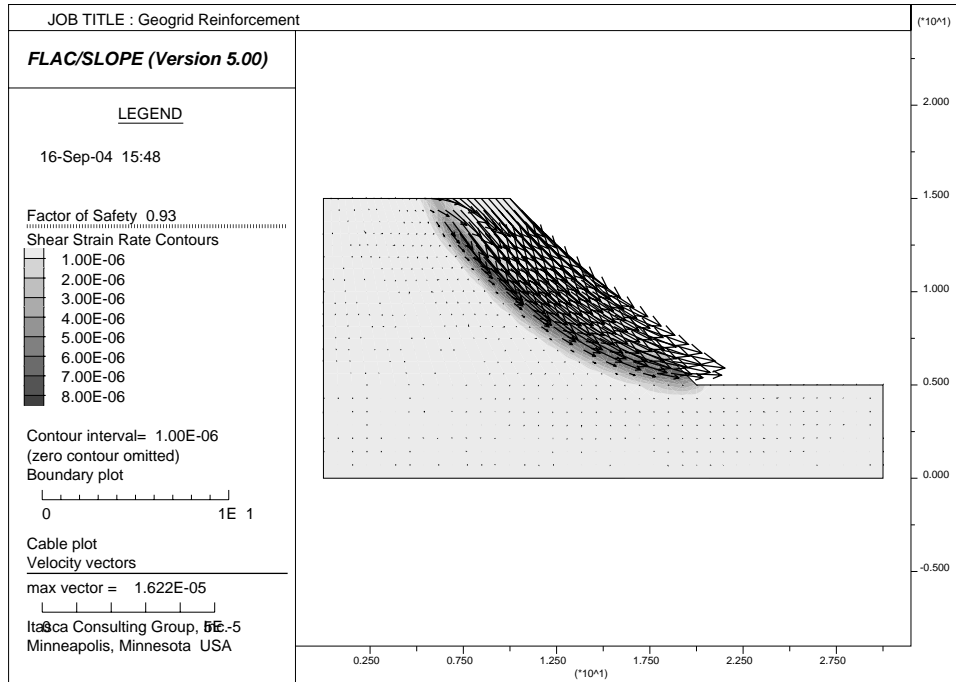


Figure 1.88 Factor-of-safety results for unsupported slope

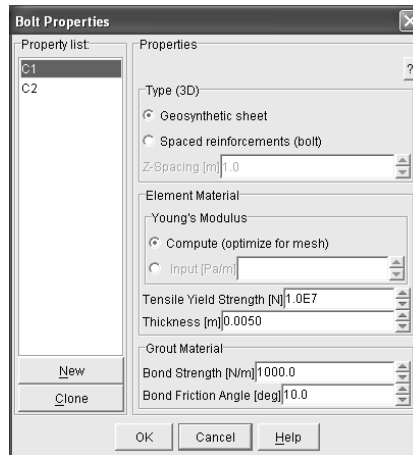


Figure 1.89 Geogrid properties specified in Cable Element Properties dialog

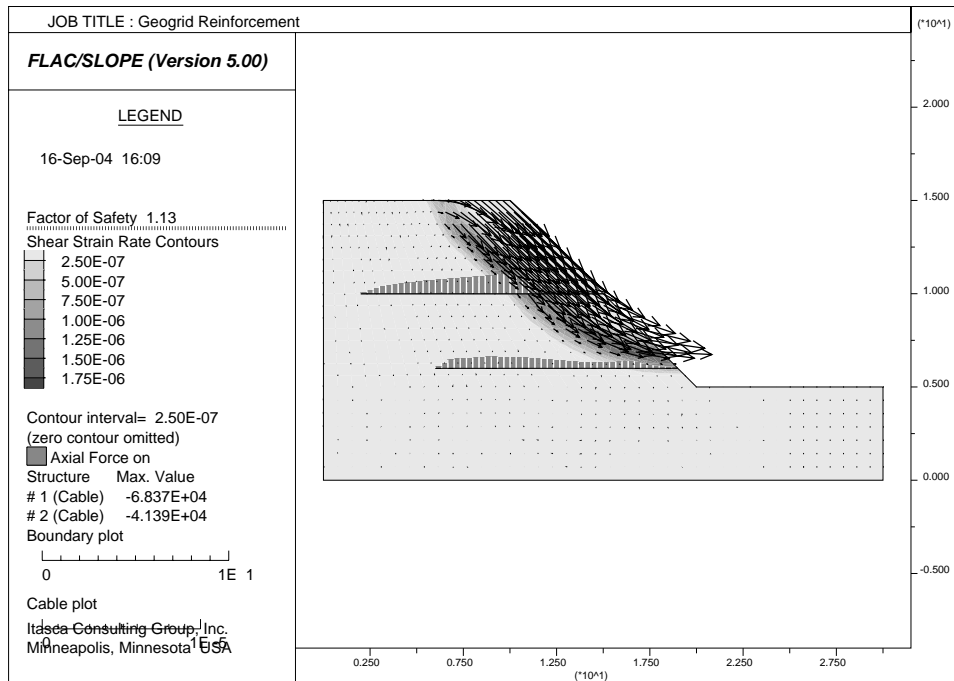


Figure 1.90 Factor-of-safety results for geogrid support with bond cohesion = 1000 N/m

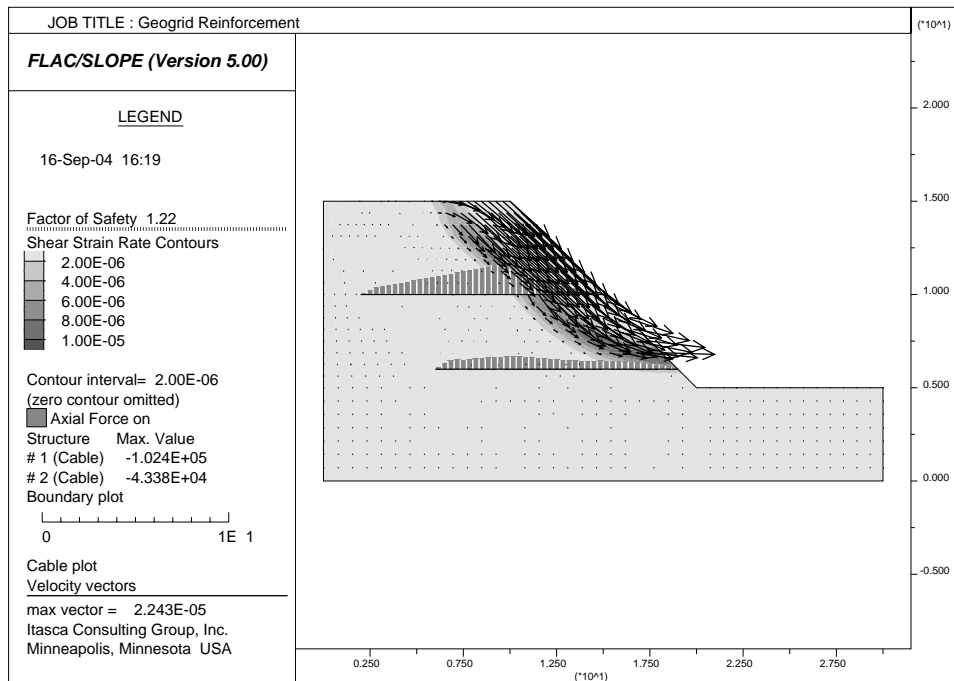


Figure 1.91 Factor-of-safety results for geogrid support with bond cohesion = 10,000 N/m

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° to the horizontal, and the top of the slope is cut at 45° from the top of the third bench to the ground surface. Figure 1.92 illustrates the geometry of the slope. This example is taken from Hoek and Bray (1981).

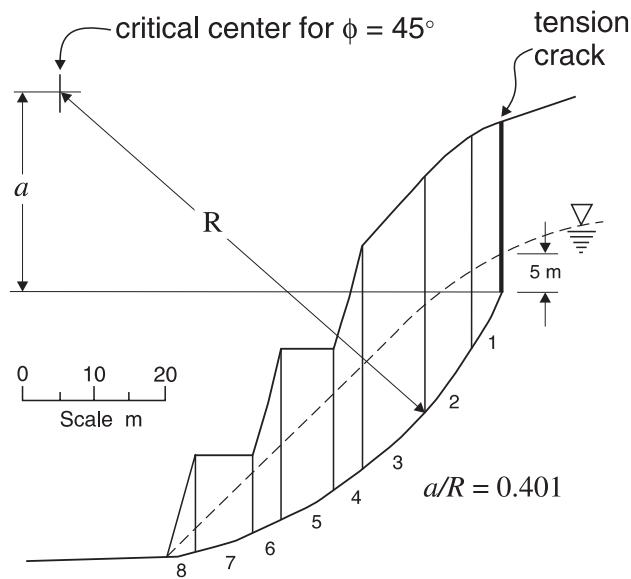


Figure 1.92 Failure surface solution from Bishop's method for a rock slope (Hoek and Bray 1981)

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

$$\begin{aligned}
 m &= 0.13 \\
 s &= 0.00001 \\
 \sigma_c &= 150 \text{ MPa} \\
 \sigma_c^m &= \sqrt{s} \sigma_c = 0.47 \text{ MPa}
 \end{aligned}$$

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):

$$\begin{aligned}
 \phi &= 45^\circ \\
 c &= 0.14 \text{ MPa}
 \end{aligned}$$

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 1.92, 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 1.92, and a factor of safety of 1.423.

The *FLAC/Slope* model is created using the **GENERAL** boundary tool in the *New Model* dialog to specify the coordinates of bench locations along the slope face. Figure 1.93 shows the tool. The **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 1.92. The **WATER** tool is used to input data points defining the water table, as shown in Figure 1.94.

The model is run using the *Fine* grid. The project save file for this example is “BENCH.PSL.”

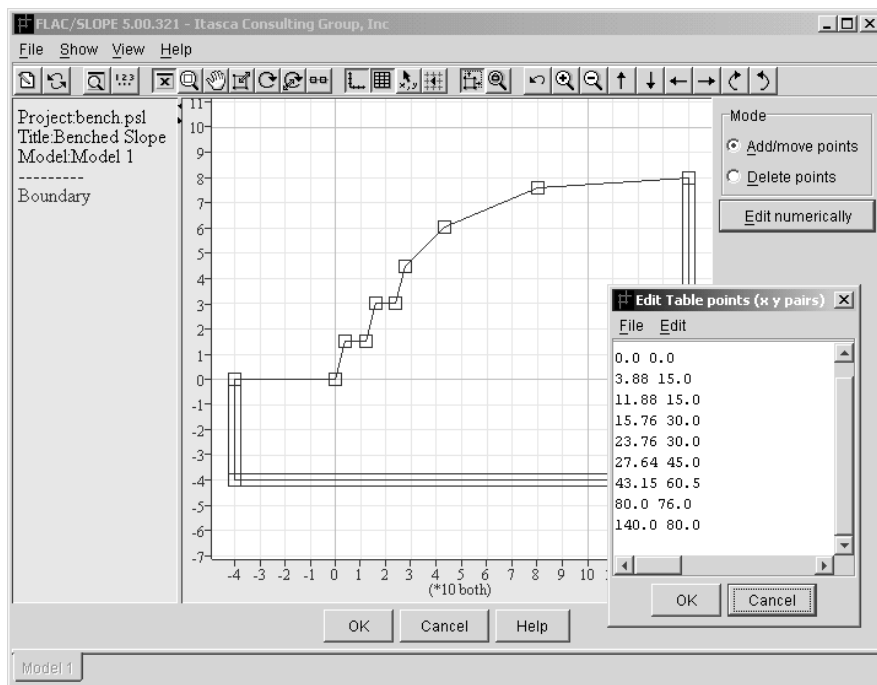


Figure 1.93 **GENERAL** boundary tool

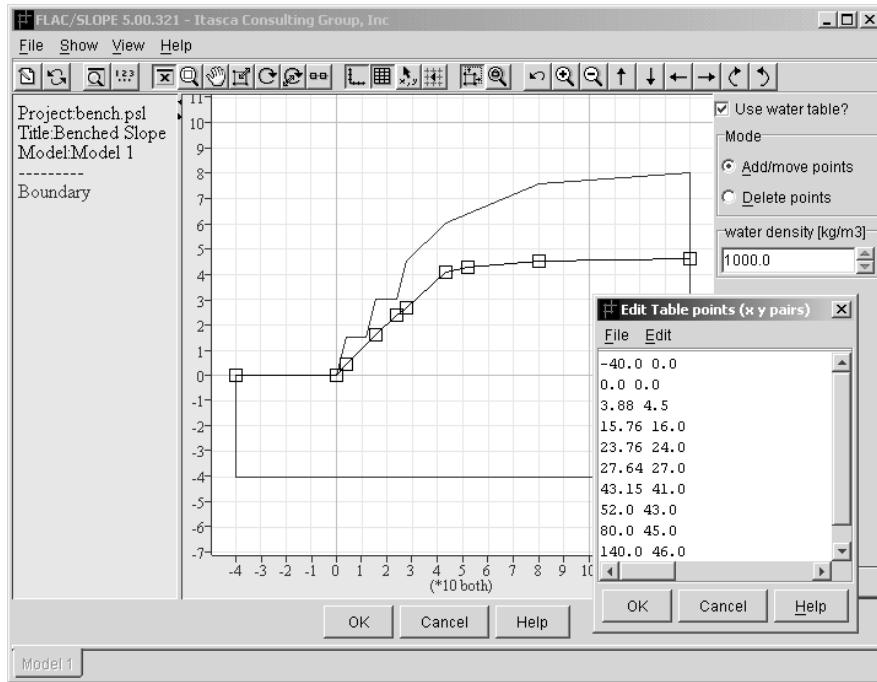


Figure 1.94 WATER tool

Figure 1.95 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 1.96). This progressive failure cannot be identified in a limit-equilibrium solution.

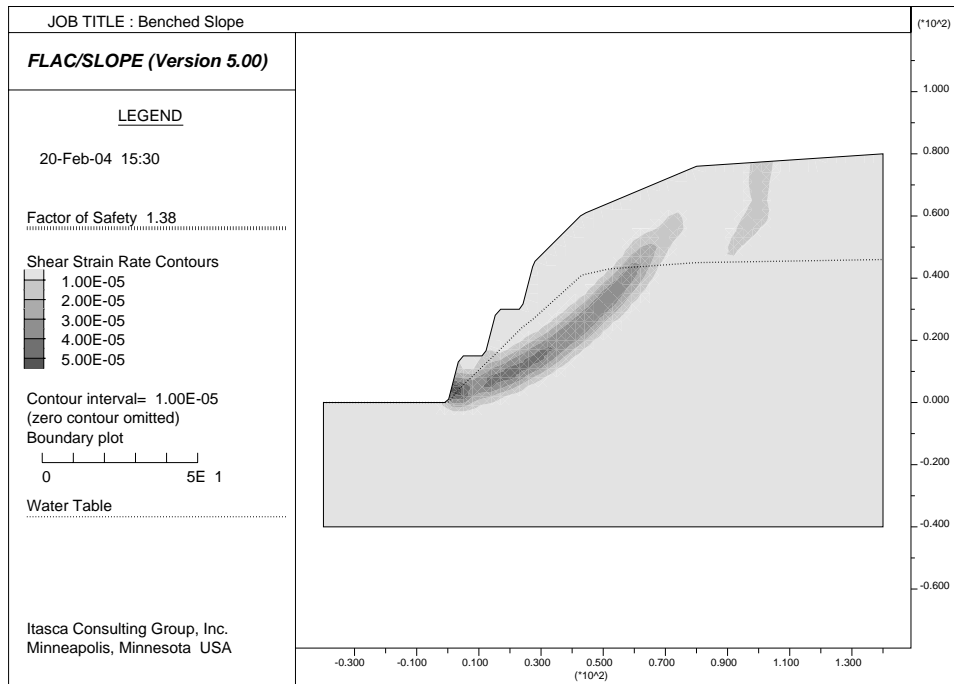


Figure 1.95 Factor-of-safety plot for rock slope with benches

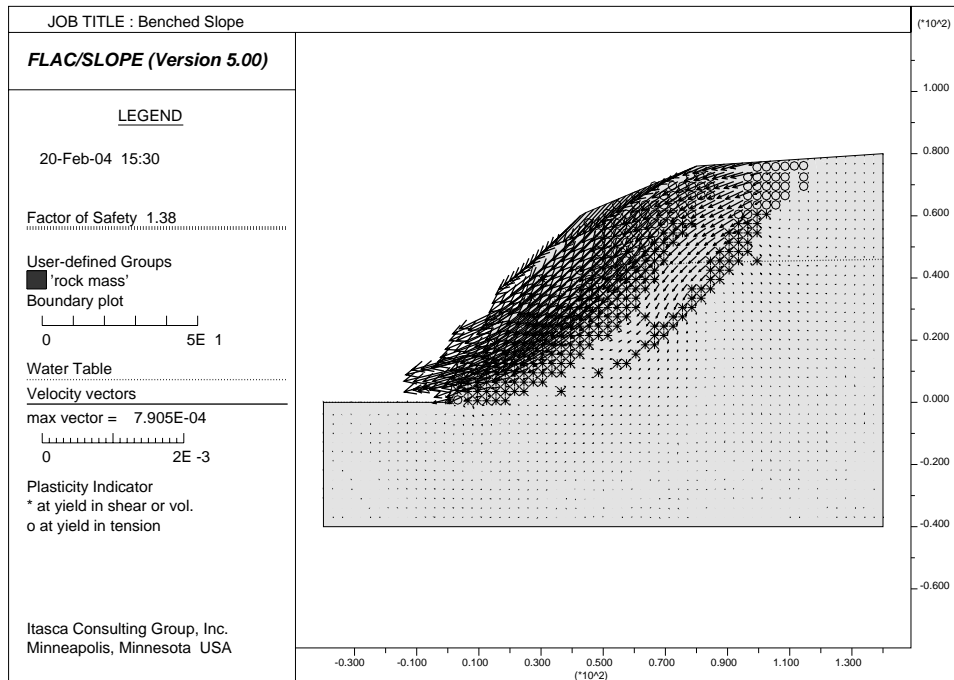


Figure 1.96 Factor-of-safety plot for rock slope with benches — velocity vectors and plasticity indicators

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:

$$c^{\text{trial}} = \frac{1}{F^{\text{trial}}} c \quad (1.7)$$

$$\phi^{\text{trial}} = \arctan\left(\frac{1}{F^{\text{trial}}} \tan \phi\right) \quad (1.8)$$

A series of simulations are made using trial values of the factor F^{trial} to reduce the cohesion, c , and friction angle, ϕ , until slope failure occurs. (Note that if the slope is initially unstable, c and ϕ 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 procedure in *FLAC/Slope* 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 either: (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.

* 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 1.46](#)).

† 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.

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.

If tensile strength, interface friction and cohesion, 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 σ^t , the reduction equation is

$$\sigma^{t(trial)} = \frac{1}{F^{trial}} \sigma^t \quad (1.9)$$

and, for the interface strength values c_i and ϕ_i , the equations are:

$$c_i^{trial} = \frac{1}{F^{trial}} c_i \quad (1.10)$$

$$\phi_i^{trial} = \arctan\left(\frac{1}{F^{trial}} \tan \phi_i\right) \quad (1.11)$$

For the reinforcement grout strength values c_b and ϕ_b , the strength-reduction equations are:

$$c_b^{trial} = \frac{1}{F^{trial}} c_b \quad (1.12)$$

$$\phi_b^{trial} = \arctan\left(\frac{1}{F^{trial}} \tan \phi_b\right) \quad (1.13)$$

These values are then used in the safety-factor calculation.

1.6 References

Bishop, A. W. "The Use of the Slip Circle in the Stability Analysis of Earth Slopes," *Géotechnique*, **5**, 7-17 (1955).

Cala, M., and J. Flisiak. "Slope Stability Analysis with FLAC and Limit Equilibrium Methods," in *FLAC and Numerical Modeling in Geomechanics — 2001 (Proceedings of the 2nd International FLAC Symposium on Numerical Modeling in Geomechanics, Ecully-Lyon, France, October 2001)*, pp. 113-114. D. Billiaux, X. Rachez, C. Detournay and R. Hart, Eds., Rotterdam: A. A. Balkema, 2001.

Dawson, E. M., and W. H. Roth. "Slope Stability Analysis with FLAC," in *FLAC and Numerical Modeling in Geomechanics (Proceedings of the International FLAC Symposium on Numerical Modeling in Geomechanics, Minneapolis, Minnesota, September 1999)*, pp. 3-9. C. Detournay and R. Hart, Eds. Rotterdam: A. A. Balkema, 1999.

Dawson, E. M., W. H. Roth and A. Drescher. "Slope Stability Analysis by Strength Reduction," *Géotechnique*, **49**(6), 835-840 (1999).

Fredlund, D. G., and J. Krahn. "Comparison of Slope Stability Methods of Analysis," *Can. Geotech. J.*, **14**, 429-439 (1977).

Griffiths, D. V., and P. A. Lane. "Slope Stability Analysis by Finite Elements," *Géotechnique*, **49**(3), 387-403 (1999).

Hoek, E., and J. Bray. *Rock Slope Engineering*. London: IMM, 1981.