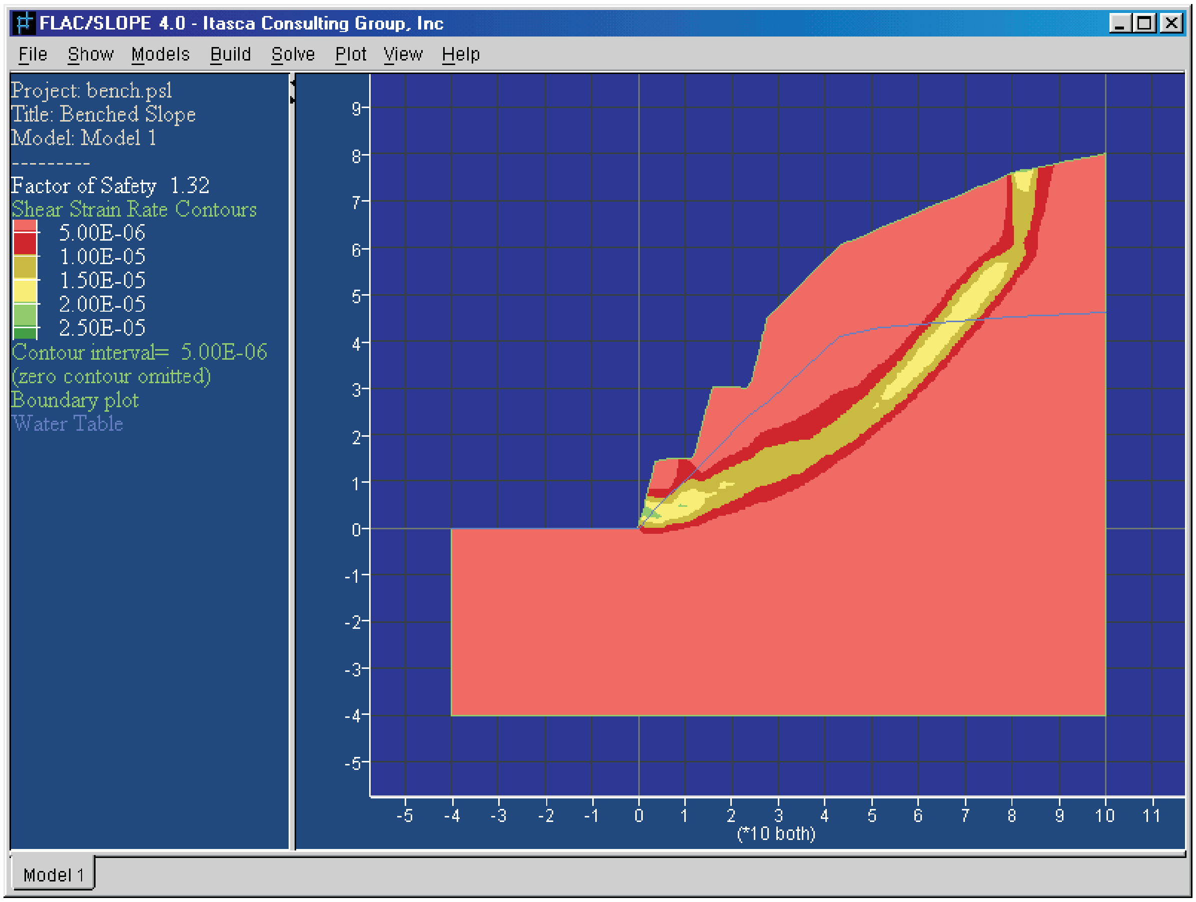
FLAC/Slope

User's Guide

*a mini-version of FLAC * to calculate factor of safety for slopes*



```
Project: bench.psl
Title: Benched Slope
Model: Model 1
---------
Factor of Safety  1.32
Shear Strain Rate Contours
      5.00E-06
      1.00E-05
      1.50E-05
      2.00E-05
      2.50E-05
Contour interval=  5.00E-06
(zero contour omitted)
Boundary plot
Water Table
```

Itasca Consulting Group, Inc.
Thresher Square East
708 South Third Street, Suite 310
Minneapolis, Minnesota 55415 USA

Phone: (1) 612-371-4711
Fax: (1) 612·371·4717
E-Mail: software@itascacg.com
Web: www.itascacg.com

---

* **F**ast **L**agrangian **A**nalysis of **C**ontinua — ***FLAC*** is a general-purpose Itasca program for numerical modeling of continuous materials.

First Edition February 2002

TABLE OF CONTENTS

1 *FLAC/Slope*

TABLES

FIGURES

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 corresponding 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 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 inter-slice 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 feasible kinematically. (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 mini-versions of *FLAC* and our plans for future mini-versions.

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 on 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.

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.

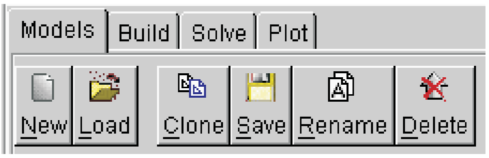
Please be aware that *FLAC/Slope* is limited to slope configurations with sub-horizontal layering and no more than one interface. For analyses which involve multiple (and intersecting) interfaces and sub-vertical layering or weak planes, full *FLAC* should be used.

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.

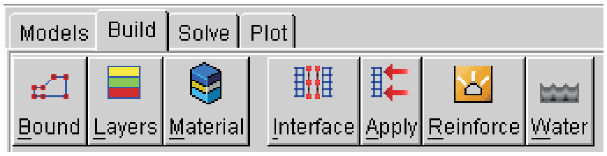
A *FLAC/Slope* analysis project is divided into four stages. The modeling-stage tool bars for each stage are shown and described below.

Models Stage



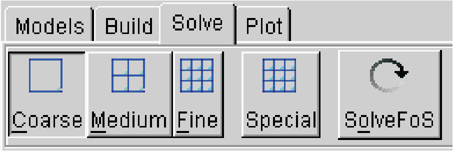
Each model in a project is named and listed in a tabbed bar in the *Models* stage. This allows easy access to any model and results in a project. New models can be added to the tabbed bar or deleted from it at any time in the project study. Models can also be restored (loaded) from previous projects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

Build Stage



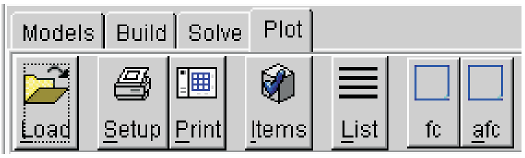
For a specific model, the slope conditions are defined in the *Build* stage. This includes: changes to the slope geometry, addition of layers, specification of materials and weak plane (interface), application of surface loading, positioning of a water table and installation of reinforcement. The conditions 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 that 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.
Thresher Square East
708 South Third Street, Suite 310
Minneapolis, Minnesota 55415 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-Versions

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 specific features in *FLAC* used for the slope stability calculations. That is why we call *FLAC/Slope* a mini-version of *FLAC*. We plan to develop several different mini-versions of *FLAC* for a variety of different geo-engineering applications.

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 clockspeed 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 1.3.11.

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 Codes” 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. Users who do not have the Reader may install it from the Itasca CD.

---

* 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.

A default directory structure will be created when using the install program. The root directory is "\ITASCA"; the sub-directories and their contents are summarized in Table 1.1 and described below.

Table 1.1 Contents of Itasca directories for FLAC/Slope

Directory	Sub-directory	Section Files
FLAC		
	EXE	executable codes
	FLAC_SLOPE	project files for examples in manual
	GUI	Graphical User Interface — JAVA class files
JRE		JAVA runtime environment
MANUALS	FLAC	FLAC electronic manual
SYSTEM		hardware key drivers, FLAC.CFG
UTILITY		README files, UPDATE.EXE

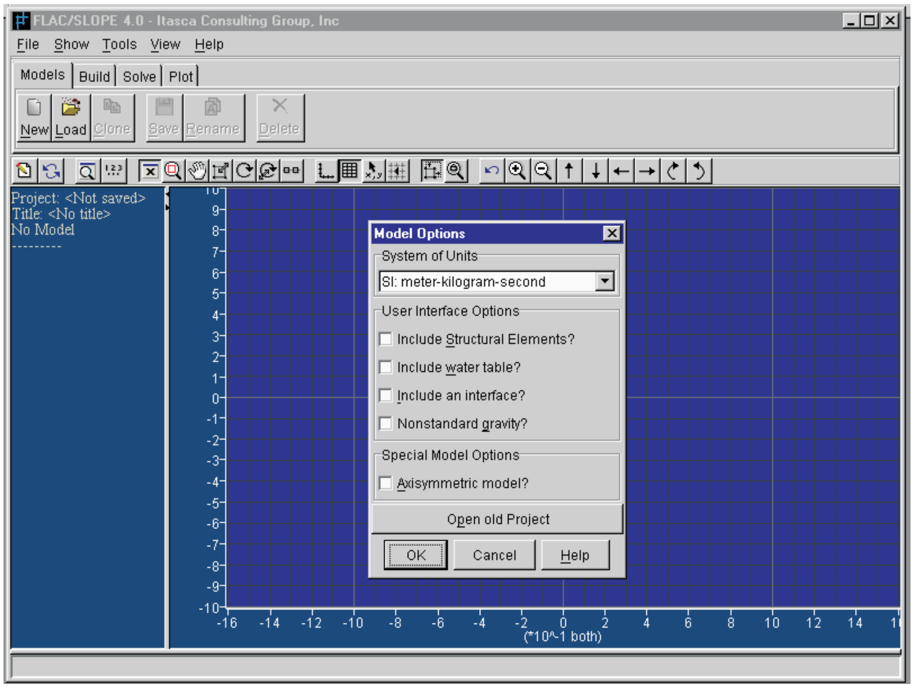
- The "\FLAC" directory contains the files related to the operation of *FLAC/Slope*. There are three sub-directories: "FLAC\EXE" contains the executable code that is loaded to run *FLAC/Slope*; "FLAC\FLAC_SLOPE" contains the example files described in this manual; and "FLAC\GUI" contains files used in the operation of the *GIIC*.
- The "\JRE" directory contains the JAVA(TM) Runtime Environment (standard edition 1.2.2) that is used for operating the *GIIC*.
- The "\MANUALS\FLAC" directory contains the complete *FLAC* manual, which includes the *FLAC/Slope* manual.
- The "\SYSTEM" directory contains the files related to the hardware lock.
- The "UPDATE.EXE" file located in the "\UTILITY" directory is used to upgrade the hardware key if the full version of *FLAC* is purchased.

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 a file named "FLAC.CFG," which is located in "ITASCA\SYSTEM." If you wish to rename the customer title at a later time, delete "FLAC.CFG" and restart *FLAC/Slope*.

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 Codes” 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: Models, Build, Solve and Plot. When you click on a modeling-stage tab, a set of tools becomes available: these tools are used to create and run the slope-stability model. Separate sets of tools are provided for the models stage, the build stage, the solve stage and the plot stage (as discussed previously in Section 1.1.4).



Project: <Not saved>
Title: <No title>
No Model

System of Units: SI: meter-kilogram-second

User Interface Options
☐ Include Structural Elements?
☐ Include water table?
☐ Include an interface?
☐ Nonstandard gravity?

Special Model Options
☐ Axisymmetric model?

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.

---

* 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 echos the *FLAC* commands that are created when operating *FLAC/Slope*. This pane also allows command-line input (at the bottom of the pane). The *Record* pane contains a list of all the *FLAC* commands, which can be exported to a data file for input into full *FLAC*. The *Console* and *Record* panes are activated from the Show/Resources menu item.

Directly above the model-view pane is a *View* 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.

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.

	Upper Soil	Lower Soil
dry density	1500 kg/m^3	1800 kg/m^3
wet density	1800 kg/m^3	2100 kg/m^3
cohesion	5,000 kPa	10,000 kPa
friction angle	20°	25°
tensile strength	0	0

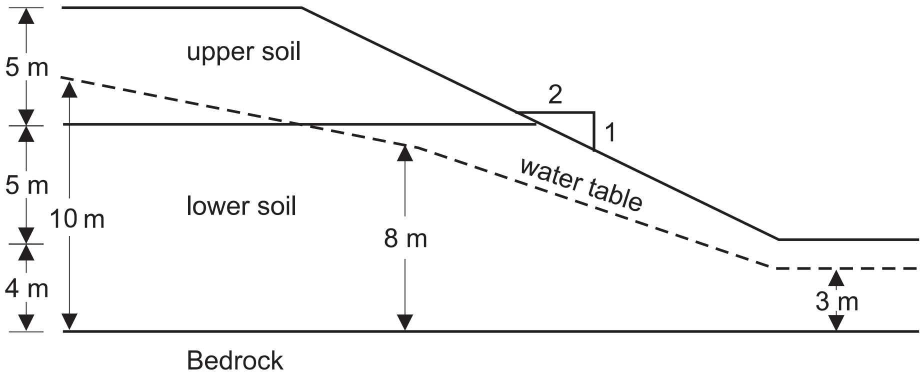
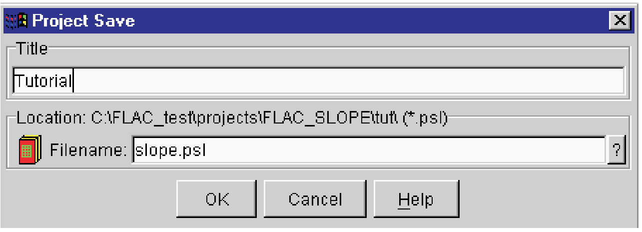


Figure 1.2 Conditions of the simple slope

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 OK to include these options in the project analysis.

We now click on File / Save Project As ... to specify a project title, a working directory for the project and a project save file. The *Project 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 OK to accept these selections.



Title: Tutorial

Location: C:\FLAC_test\projects\FLAC_SLOPE\tut\ (*.psl)

Filename: slope.psl

Figure 1.3 Project Save dialog

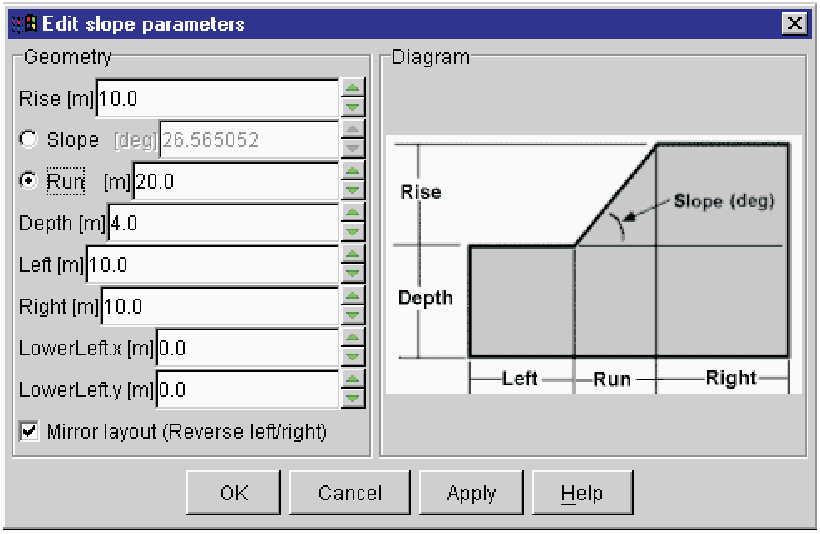
We next click on the Models tool and enter the *Models* stage to specify a name for the first model in our project. We click on New and use the default model name *Model 1* that appears in the *New Model* dialog. There will be two models in our project: *Model 1* which does not contain a water table,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

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

When we press OK 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 Mirror Layout to reverse the model layout to match that shown in Figure 1.2. We click OK to view the slope boundary that we have created. We can either edit the boundary further or accept it. We press OK to accept the boundary for *Model 1*. The layout for the *Model 1* slope is shown in Figure 1.5*. 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.



Edit slope parameters

Geometry

Rise [m] 10.0

○ Slope [deg] 26.565052

● Run [m] 20.0

Depth [m] 4.0

Left [m] 10.0

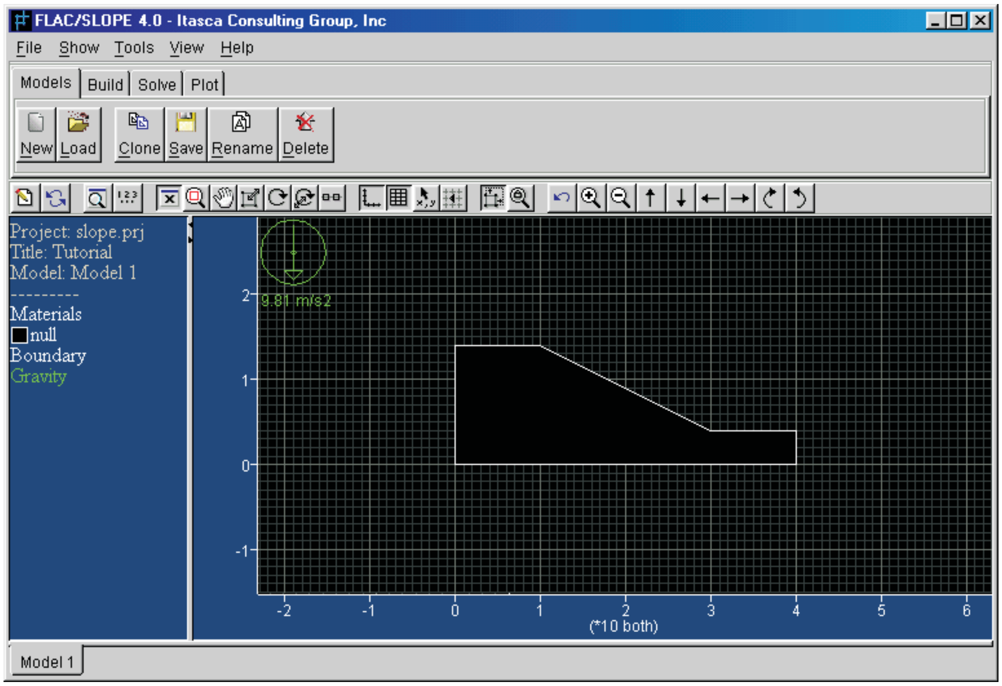
Right [m] 10.0

LowerLeft.x [m] 0.0

LowerLeft.y [m] 0.0

☑ Mirror layout (Reverse left/right)

Figure 1.4 Edit Slope Parameters dialog



FLAC/SLOPE 4.0 - Itasca Consulting Group, Inc

Project: slope.prj
Title: Tutorial
Model: Model 1

Materials
null
Boundary
Gravity

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.

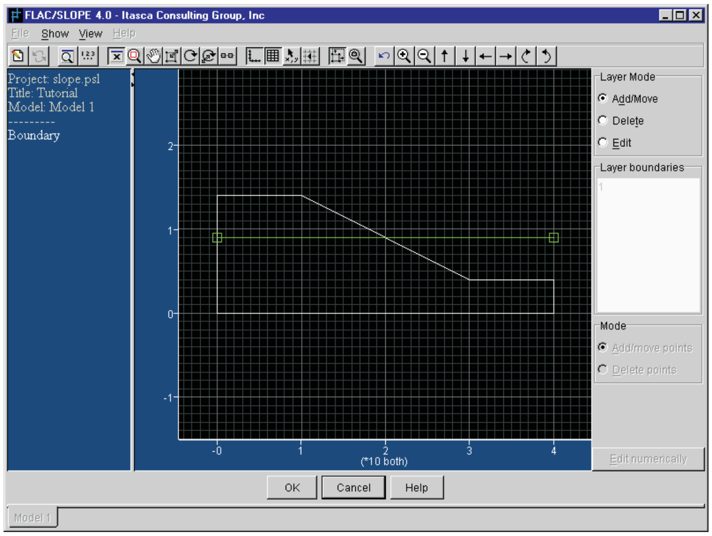


Figure 1.6 Layers tool

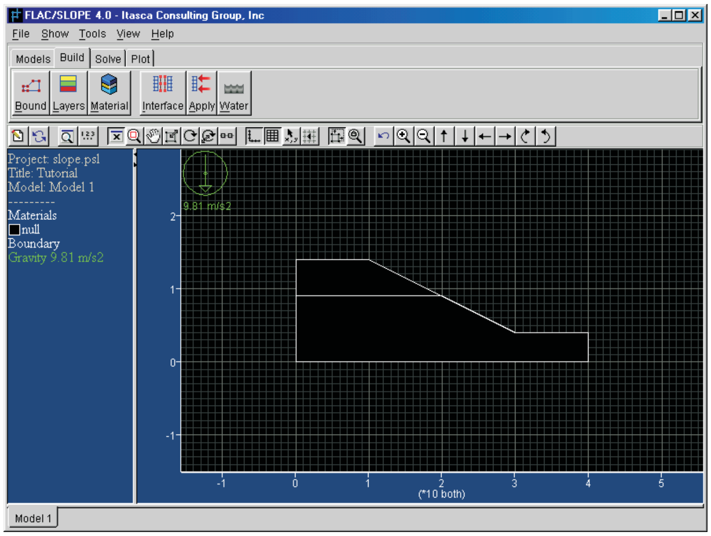
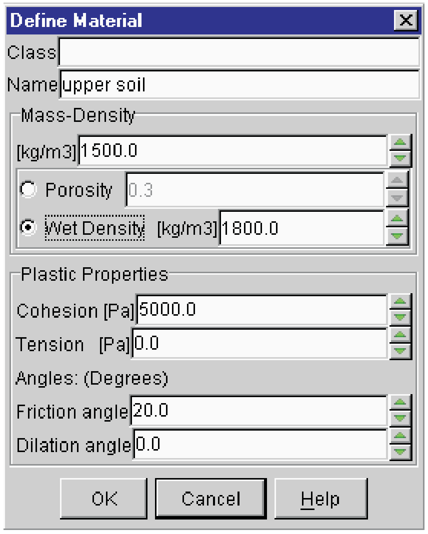


Figure 1.7 Two layers created by the Layers tool

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.)



Define Material

Class

Name upper soil

Mass-Density

[kg/m3] 1500.0

○ Porosity 0.3

● Wet Density [kg/m3] 1800.0

Plastic Properties

Cohesion [Pa] 5000.0

Tension [Pa] 0.0

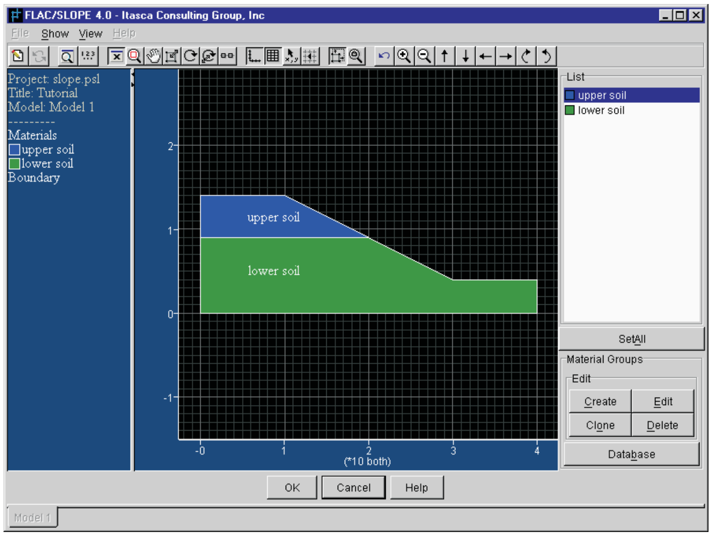
Angles: (Degrees)

Friction angle 20.0

Dilation angle 0.0

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

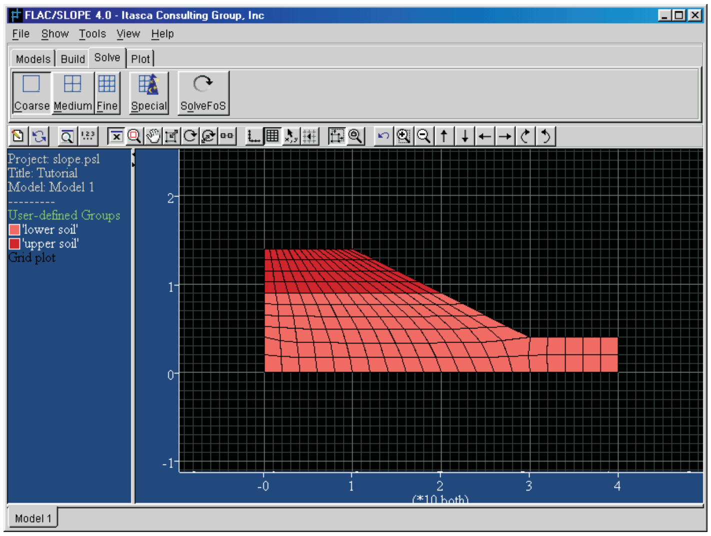
After the materials are created, they are assigned to the two layers. We highlight the material in the *List* pane and then click on the model view inside the layer we wish to assign the material. The material will be assigned to this layer, and the name of the material will be shown at the position 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*.



Project: slope.psl
Title: Tutorial
Model: Model 1

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

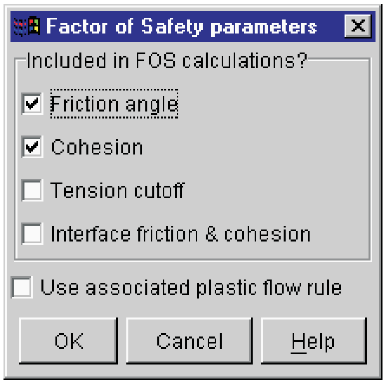
Calculating a Factor of Safety — We are now ready to calculate the factor of safety. We click on the SOLVE tool tab to enter the factor-of-safety calculation stage. When we enter this stage, we must first select a numerical mesh for our analysis. We choose a "coarse-grid" model by pressing the COARSE button, and the grid used for the *FLAC* solution appears in the model view. See Figure 1.10.



Project: slope.psl
Title: Tutorial
Model: Model 1

Figure 1.10 Coarse-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 1.68.



Factor of Safety parameters

Included in FOS calculations?

☑ Friction angle

☑ Cohesion

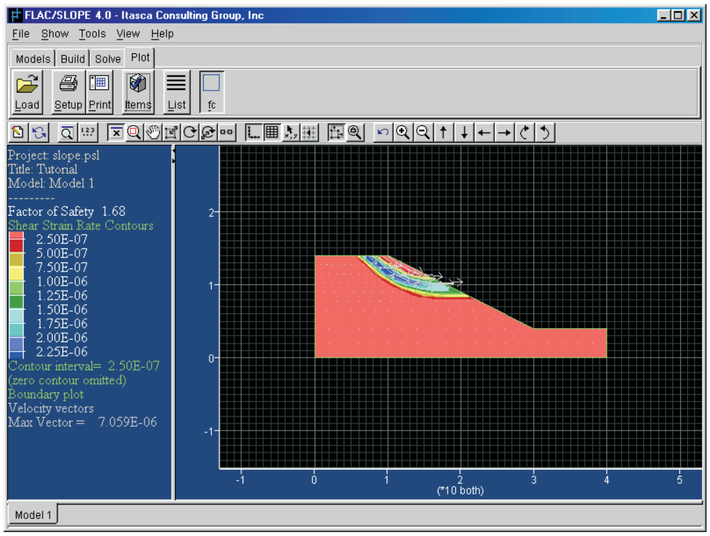
☐ Tension cutoff

☐ Interface friction & cohesion

☐ Use associated plastic flow rule

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 (coarse grid, friction angle and cohesion included in the calculation). When we click on this button, we view the failure plot for this model, as shown in Figure 1.12.



FLAC/SLOPE 4.0 - Itasca Consulting Group, Inc

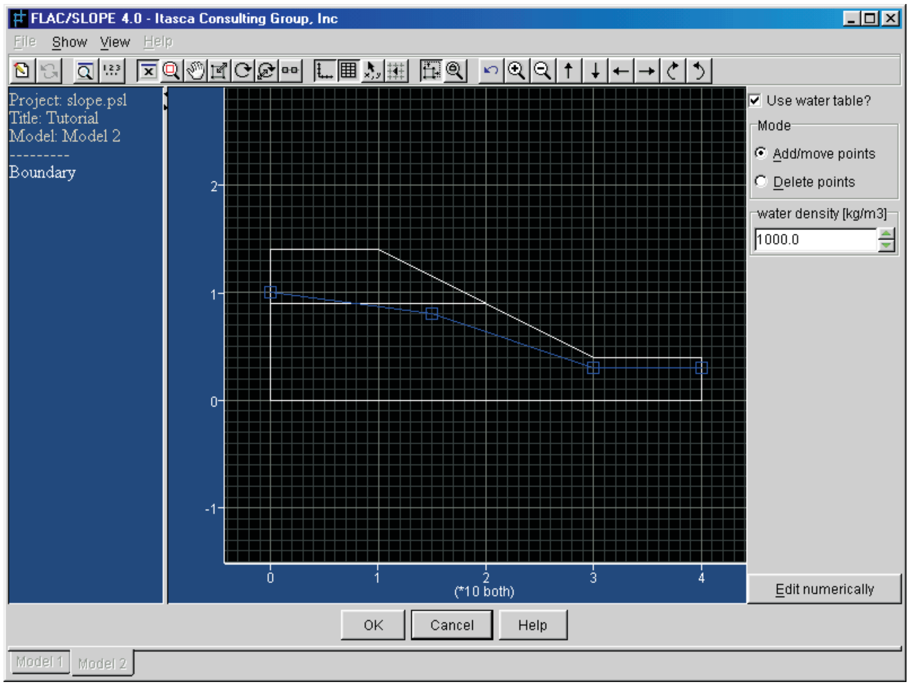
Project: slope.psl
Title: Tutorial
Model: Model 1

Factor of Safety 1.68
Shear Strain Rate Contours
2.50E-07
5.00E-07
7.50E-07
1.00E-06
1.25E-06
1.50E-06
1.75E-06
2.00E-06
2.25E-06
Contour interval= 2.50E-07
(zero contour omitted)
Boundary plot
Velocity vectors
Max Vector = 7.059E-06

Figure 1.12 Failure plot for coarse-grid Model 1

This plot shows the failure surface that develops for these model conditions (delineated by the shear strain-rate contours and velocity vectors). The value for factor of safety is also printed in the plot legend.

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 *Model 2*. We accept this name by pressing OK. A *Model 2* tab is now shown at the bottom of the view. All the model conditions from *Model 1* have been copied into *Model 2*. The only remaining condition to add is the water table. We go to the BUILD stage and click on the WATER button. A blue 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 four points at coordinates: (0,10), (15,8), (30,3) and (40,3). The result is shown in Figure 1.13.



Project: slope.psl
Title: Tutorial
Model: Model 2

Boundary

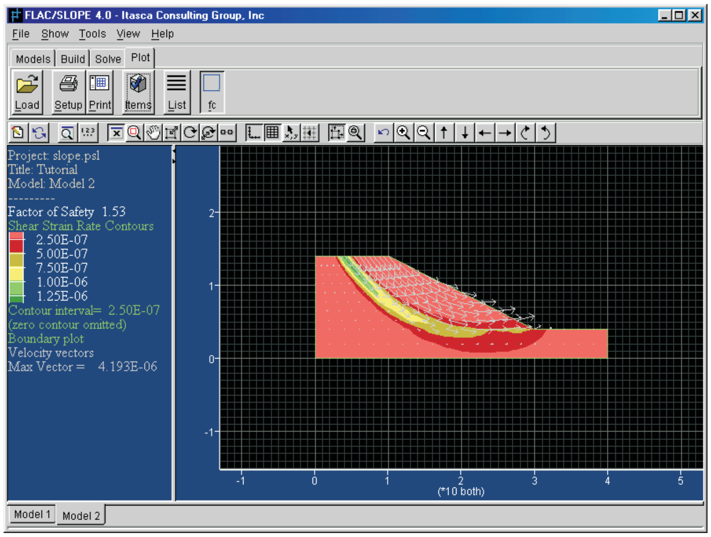
☑ Use water table?

● Add/move points
○ Delete points

water density [kg/m3] 1000.0

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 coarse-grid model and press the SOLVE FOS button. We follow the same procedure as before to determine the factor of safety. A factor of 1.53 is shown when the calculation stops. We now go to the *Plot* stage to produce the failure 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 *Failure 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.

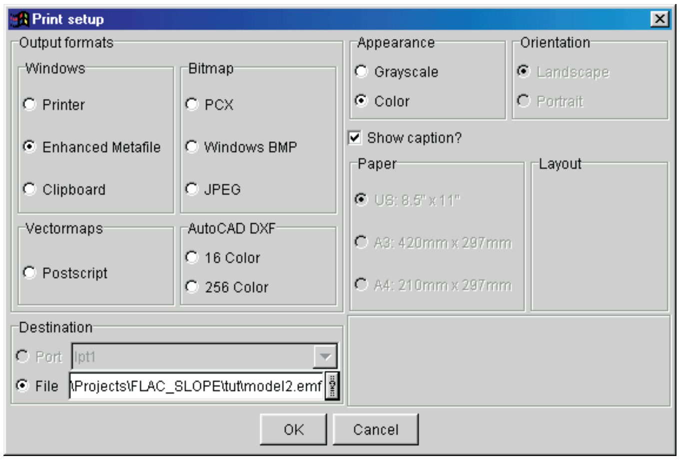


Project: slope.psl
Title: Tutorial
Model: Model 2

Factor of Safety 1.53
Shear Strain Rate Contours
2.50E-07
5.00E-07
7.50E-07
1.00E-06
1.25E-06
Contour interval= 2.50E-07
(zero contour omitted)
Boundary plot
Velocity vectors
Max Vector = 4.193E-06

Figure 1.14 Failure plot for coarse-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.



○ Printer
● Enhanced Metafile
○ Clipboard
○ PCX
○ Windows BMP
○ JPEG
○ Postscript
○ 16 Color
○ 256 Color
○ Grayscale
● Color
● Landscape
○ Portrait
☑ Show caption?
● US: 8.5" x 11"
○ A3: 420mm x 297mm
○ A4: 210mm x 297mm
○ Port lpt1
● File \Projects\FLAC_SLOPE\tut\model2.emf

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 failure plot to a file named “MODEL2.EMF.” We press OK to save these printer settings. Then, we press PRINT in the PLOT tool to send the plot to this file.

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

The plot is shown in Figure 1.16. Note that hardcopy plots are formatted slightly differently from the screen plots.

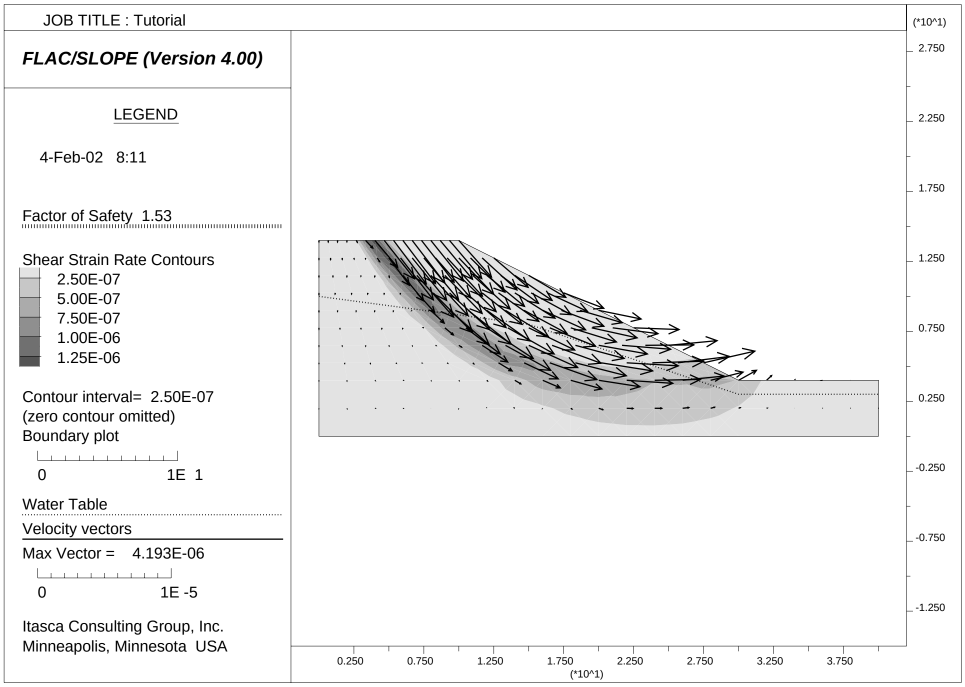


Figure 1.16 Hardcopy plot for Model 2 result

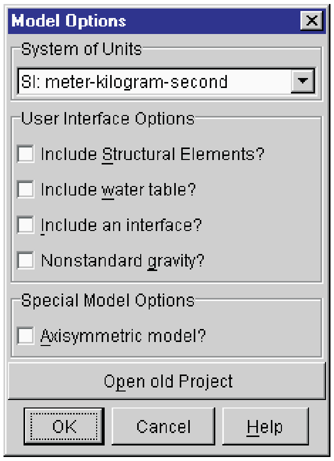
This completes the simple tutorial. We recommend that you try additional variations on this project to help increase your understanding. For example, if you wish to evaluate the effect of zoning on the calculated safety factor, return to the *Solve* stage for *Model 1* and click on the Medium button. This will create a finer mesh than the coarse 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 medium mesh directly with the coarse 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 1.17. 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.



Model Options

System of Units

SI: meter-kilogram-second

User Interface Options

☐ Include Structural Elements?

☐ Include water table?

☐ Include an interface?

☐ Nonstandard gravity?

Special Model Options

☐ Axisymmetric model?

Figure 1.17 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.

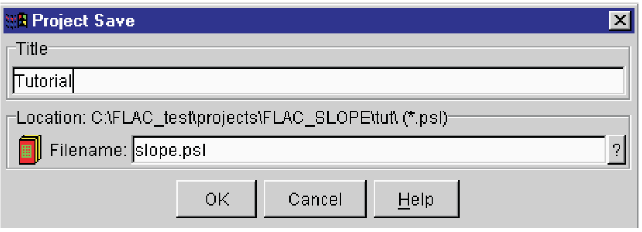
By default, the standard value for gravitational acceleration is used in the analysis. A gravity icon will appear in the model view (when the model is created) with a gravitational vector pointing downward and magnitude corresponding to the selected system of units. If you check the box Nonstandard gravity?,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

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\FLAC\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 1.18. 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.



Title: Tutorial

Location: C:\FLAC_test\projects\FLAC_SLOPE\tut\ (*.psl)

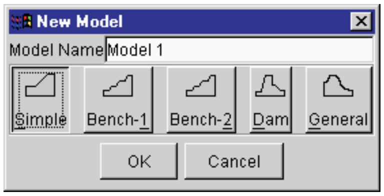
Filename: slope.psl

Figure 1.18 Project Save dialog

You can stop working on a project at any stage, save it (by pressing the File / Save Project menu item) and re-open 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 1.19. 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 1.3.12.

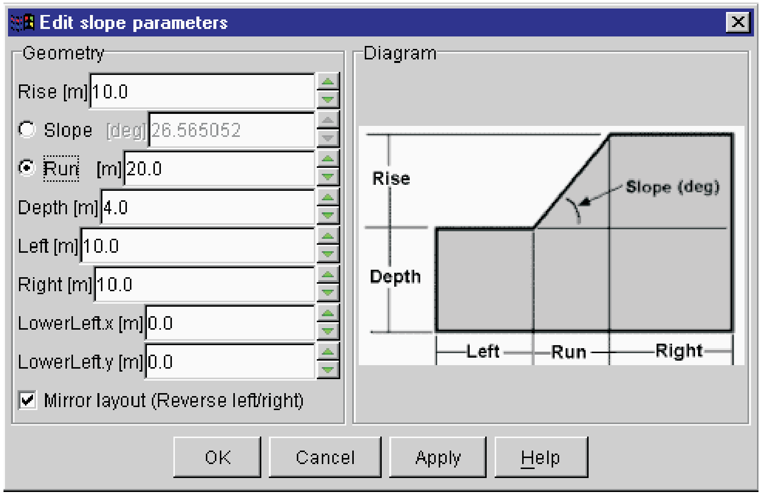


Model Name: Model 1

Figure 1.19 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 1.20. 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.

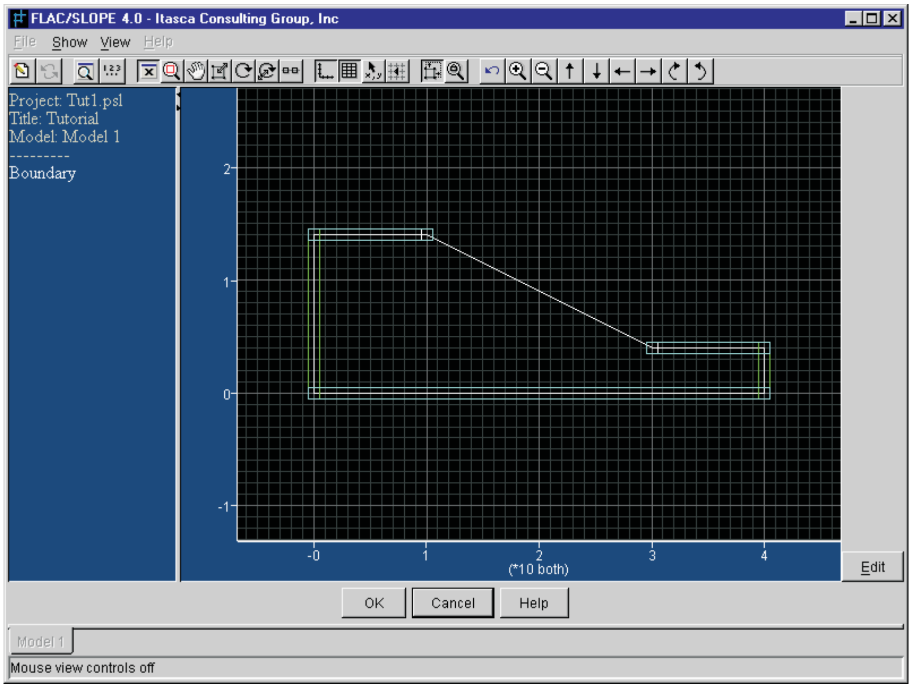
When selecting the dimensions for Depth, Left and Right, it is important that these dimensions are large enough such that artificial boundaries (i.e., left, right and bottom boundaries) do not influence the development of the failure surface. If the final calculated slip surface is found to intersect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



Rise [m] 10.0
○ Slope [deg] 26.565052
● Run [m] 20.0
Depth [m] 4.0
Left [m] 10.0
Right [m] 10.0
LowerLeft.x [m] 0.0
LowerLeft.y [m] 0.0
☑ Mirror layout (Reverse left/right)

Figure 1.20 Edit Slope Parameters dialog

When you press OK, the dialog will close and the outline of the slope model will be drawn in a boundary view as shown in Figure 1.21. The boundary can be edited further in this view, either by dragging the mouse to move the boundaries or by pressing the EDIT button to open the *Edit slope parameters* dialog again.



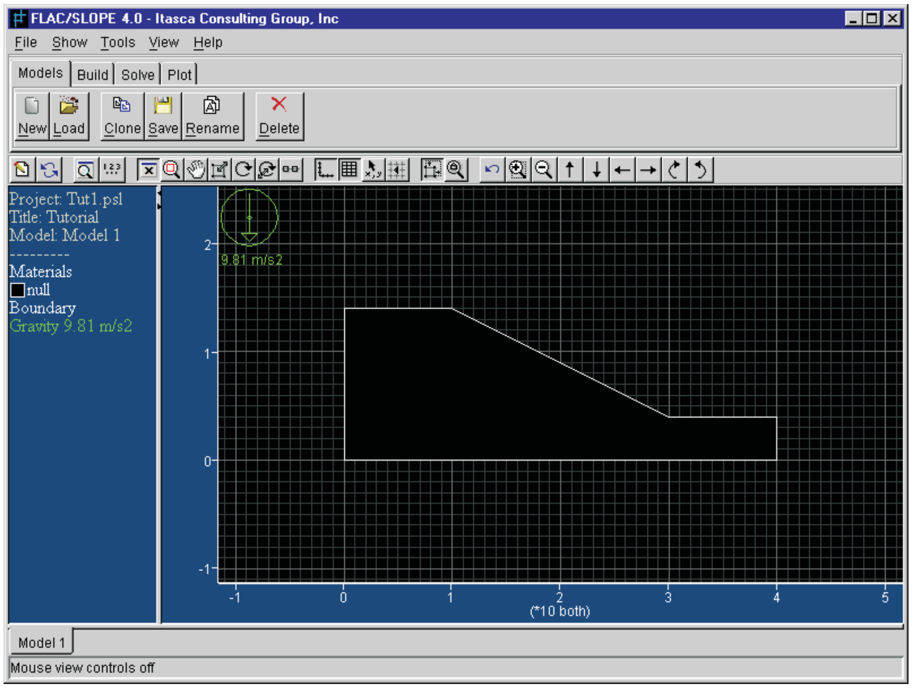
Project: Tut1.psl
Title: Tutorial
Model: Model 1

Boundary

Mouse view controls off

Figure 1.21 Boundary view

Once you are satisfied, press OK. The model boundary will now be drawn in the model view, as shown in Figure 1.22. Note that a tab with the model name will appear at the bottom of the model view when a model is created.



Project: Tut1.psl
Title: Tutorial
Model: Model 1

Materials
■null
Boundary
Gravity 9.81 m/s2

Mouse view controls off

Figure 1.22 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

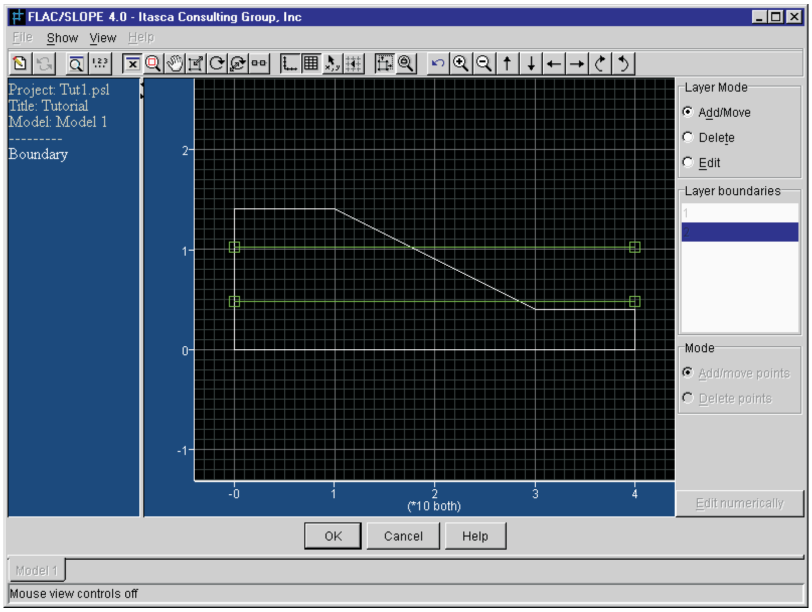
Individual models can be saved at this stage by pressing the Save button. A *Save As* dialog will open and you can select a directory in which to save the model. The model file will automatically have the extension ".SLP." You can then load this model into another project, if desired, by pressing the 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 1.23 shows a model with two layer-boundary lines visible in the *Layers* tool.



Project: Tut1.psl
Title: Tutorial
Model: Model 1

Layer Mode: ● Add/Move ○ Delete ○ Edit

Mode: ● Add/move points ○ Delete points

Mouse view controls off

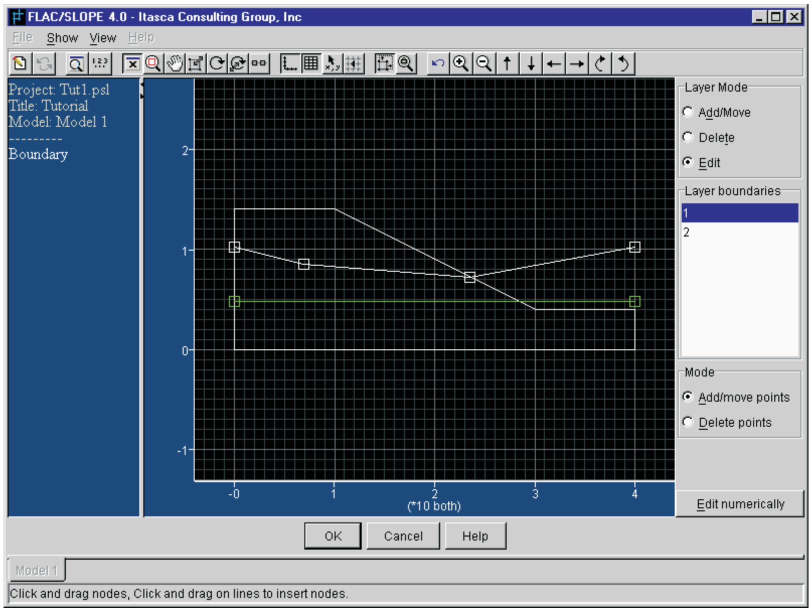
Figure 1.23 Slope model with two layer-boundary lines in the Layers tool

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 1.24 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.

WARNING: The layers must be sub-horizontal; sub-vertical layers cannot be modeled. The boundary lines must run continuously from the left model boundary to the right. The 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, sub-vertical and/or pinched-out layers.



Project: Tut1.psl
Title: Tutorial
Model: Model 1

Boundary

Click and drag nodes, Click and drag on lines to insert nodes.

Figure 1.24 The upper layer-boundary line is edited to include two points

1.3.5 Assigning Materials and Properties

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

Materials are created by clicking on the CREATE button to open the *Define Material* dialog. The dialog is shown in Figure 1.25. A material is defined by its classification and name — for example, classification: *embankment soil*, and name: *silty sand*. The classification is useful if you choose to create a database to store common materials to use on different projects. The database is accessed via the DATABASE button located at the bottom-right corner of the *Material* tool. The database is described later in this section.

The mass density and material strength properties are assigned for each material. Note that the corresponding units for each property are shown in the dialog, depending on the system of units selected in the *Model Options* dialog. Mass 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).

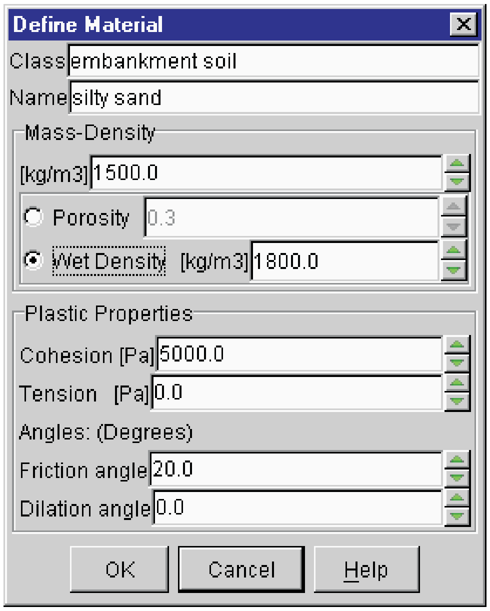
If the water table tool is not active, only the “dry” (or “moist”) mass density is assigned. This is the density of the material above the water table in situ. If the water table tool is active, then either a porosity or a “wet” (or “saturated”) mass density must also be assigned. The relation between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

$$\rho^{\text{WET}} = \rho^{\text{DRY}} + n\,\rho_w \tag{1.1}$$

where ρ^{WET} is the wet density, ρ^{DRY} is the dry 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 1.3.10.2.)

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.



Class embankment soil
Name silty sand
Mass-Density
[kg/m3] 1500.0
○ Porosity 0.3
● Wet Density [kg/m3] 1800.0
Plastic Properties
Cohesion [Pa] 5000.0
Tension [Pa] 0.0
Angles: (Degrees)
Friction angle 20.0
Dilation angle 0.0

Figure 1.25 Define Material dialog

If the material DATABASE button is pressed, a *Material list* dialog will open as shown in Figure 1.26. The database is divided into groups, designated by classification names, and shown in a collapsible tree structure. The database can be used to store sets of common materials and their properties for use on different projects. By default, a database of soil and rock materials is provided, as shown in the *Database* listed in the figure. Materials are selected from this list by double-clicking on a material name; the material will then be added to the *Selection* list. After choosing the materials for a project, press OK to send these materials to the *List* shown in the *Material* tool.

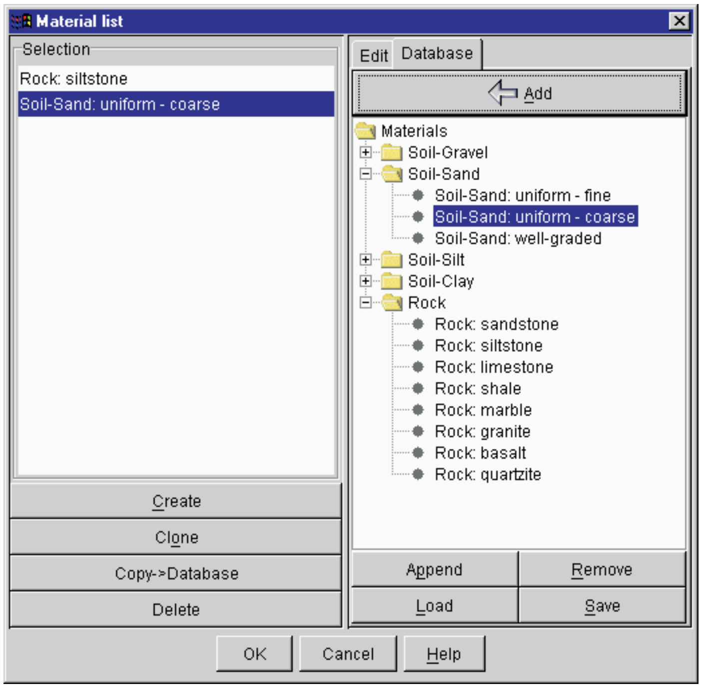
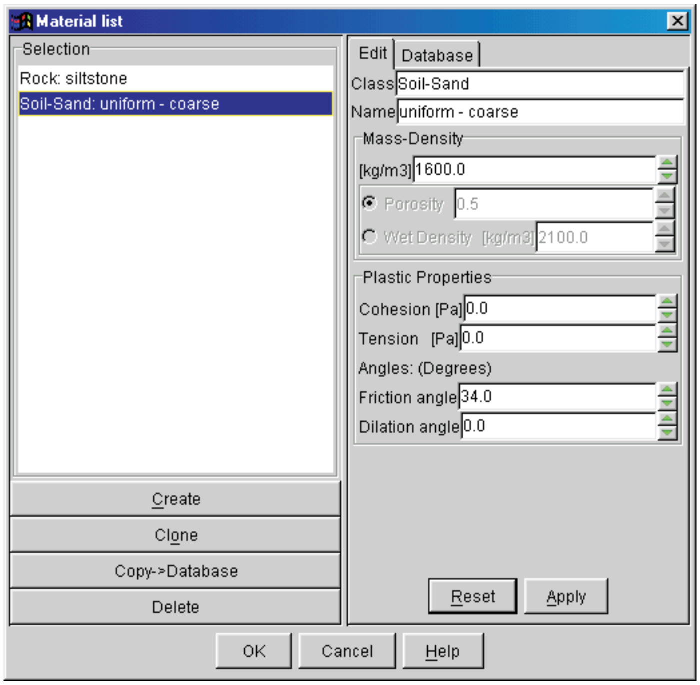


Figure 1.26 Material List dialog

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 1.27. 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 1.26) 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.



Rock: siltstone
Soil-Sand: uniform - coarse

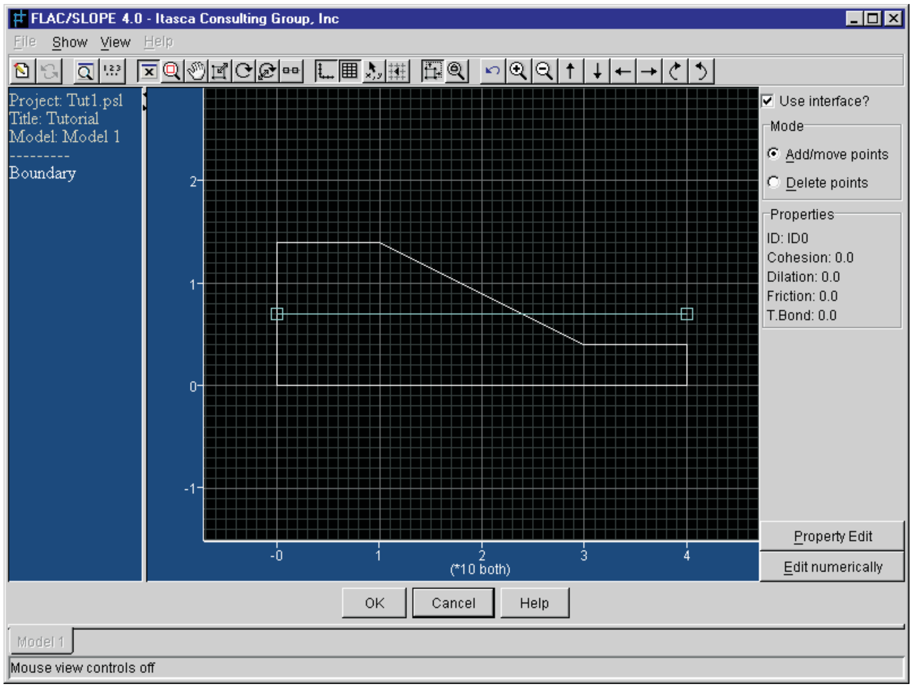
Class Soil-Sand
Name uniform - coarse
[kg/m3] 1600.0
● Porosity 0.5
○ Wet Density [kg/m3] 2100.0
Cohesion [Pa] 0.0
Tension [Pa] 0.0
Friction angle 34.0
Dilation angle 0.0

Figure 1.27 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 1.28. 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 1.29 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.



Project: Tut1.psl
Title: Tutorial
Model: Model 1

Boundary

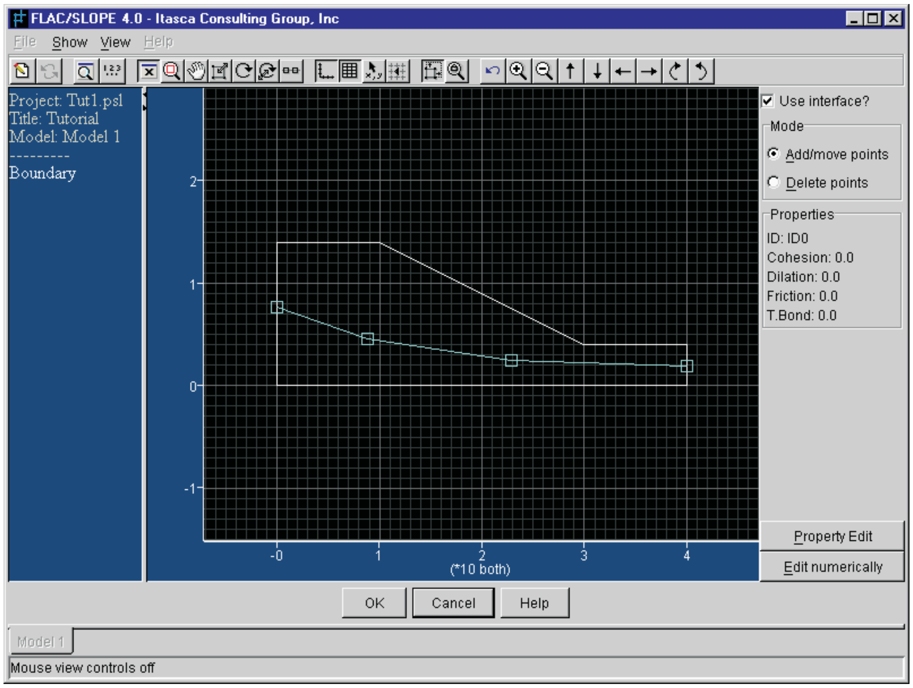
☑ Use interface?

Mode
● Add/move points
○ Delete points

Properties
ID: ID0
Cohesion: 0.0
Dilation: 0.0
Friction: 0.0
T.Bond: 0.0

Mouse view controls off

Figure 1.28 Interface tool



Project: Tut1.psl
Title: Tutorial
Model: Model 1

Boundary

☑ Use interface?

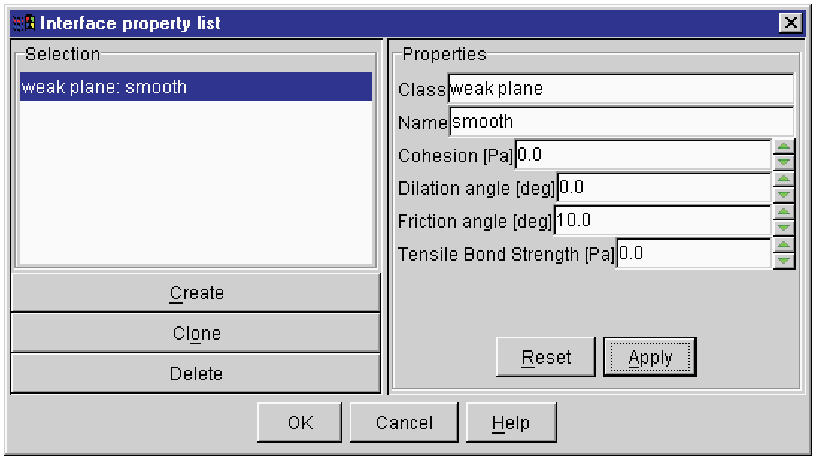
Mode
● Add/move points
○ Delete points

Properties
ID: ID0
Cohesion: 0.0
Dilation: 0.0
Friction: 0.0
T.Bond: 0.0

Mouse view controls off

Figure 1.29 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 1.30. 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.



weak plane: smooth

Class weak plane
Name smooth
Cohesion [Pa] 0.0
Dilation angle [deg] 0.0
Friction angle [deg] 10.0
Tensile Bond Strength [Pa] 0.0

Figure 1.30 Interface Property list dialog

The behavior of the interface is defined by the Coulomb slip criterion which limits the shear force, F_{smax}, along the interface by the relation

$$F_{\text{smax}} = cL + \tan\phi \; F_n \tag{1.2}$$

where c = cohesion (in stress units) along the interface, L = effective contact length, ϕ = friction angle of the interface surface, and F_n is the normal force acting on the interface.

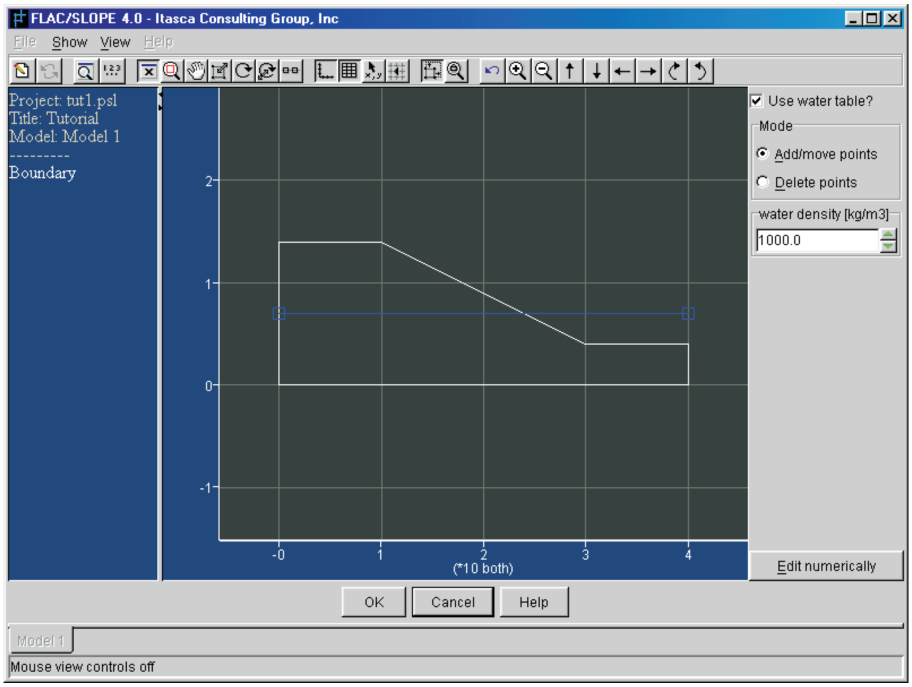
In addition, the interface may dilate at the onset of slip. Dilation is governed in the Coulomb model by a specified dilation angle, ψ.

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 the **Theory and Background** volume 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 1.31. 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 1.32 shows the water-table line repositioned with two handle points added along the line.



Project: tut1.psl
Title: Tutorial
Model: Model 1

Boundary

☑ Use water table?

● Add/move points
○ Delete points

water density [kg/m3]: 1000.0

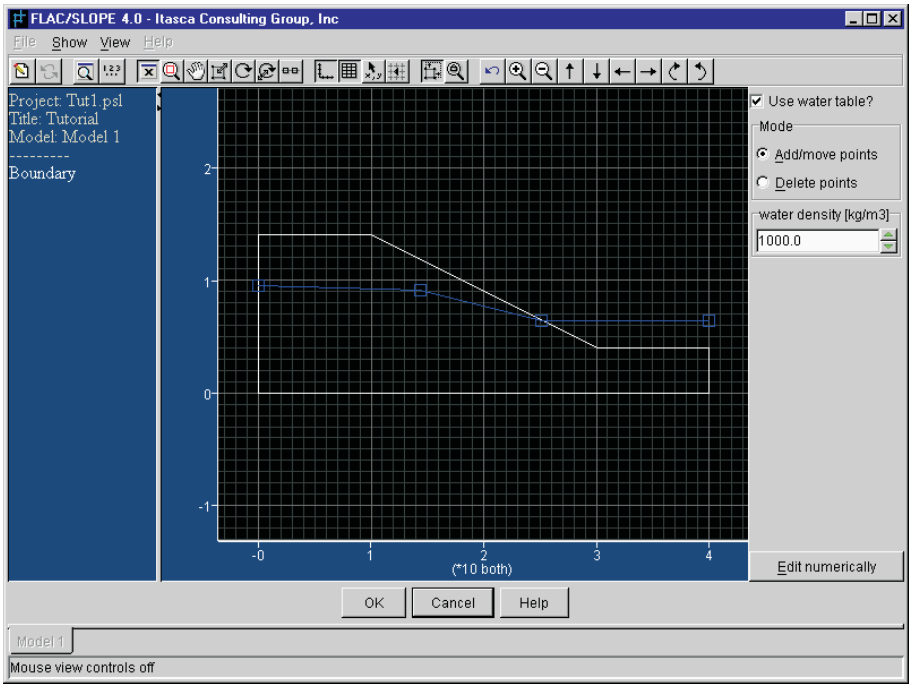
Mouse view controls off

Figure 1.31 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 dry or wet density is used to compute material weight. If the water table is located so that it intersects the slope surface, such as the case shown in Figure 1.32, then the weight of the water is automatically included as a mechanical pressure acting on free surfaces below the water table.



Project: Tut1.psl
Title: Tutorial
Model: Model 1

Boundary

☑ Use water table?

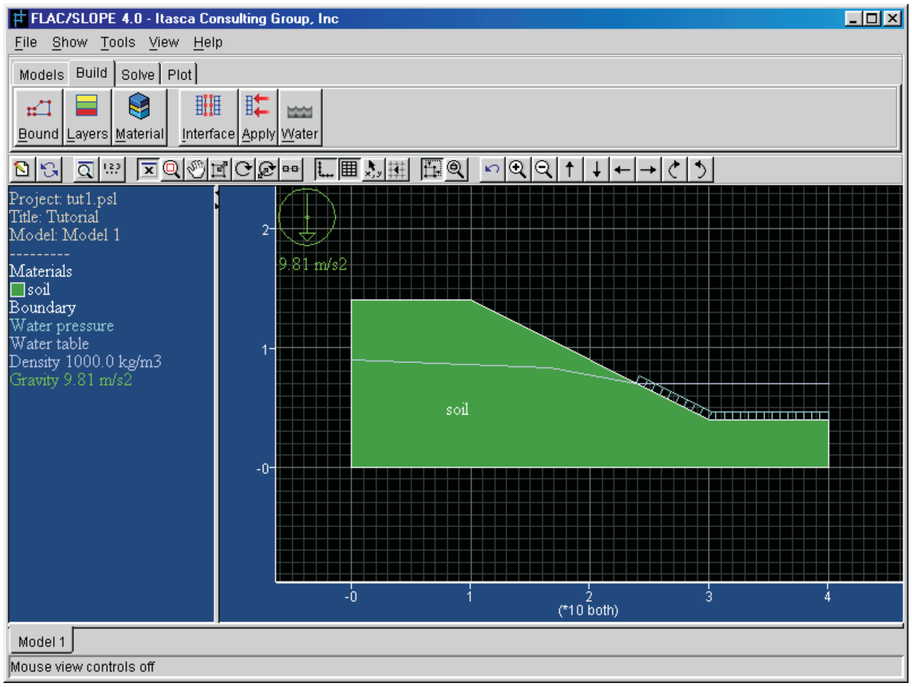
● Add/move points
○ Delete points

water density [kg/m3]: 1000.0

Mouse view controls off

Figure 1.32 Water Table repositioned with two handle points added

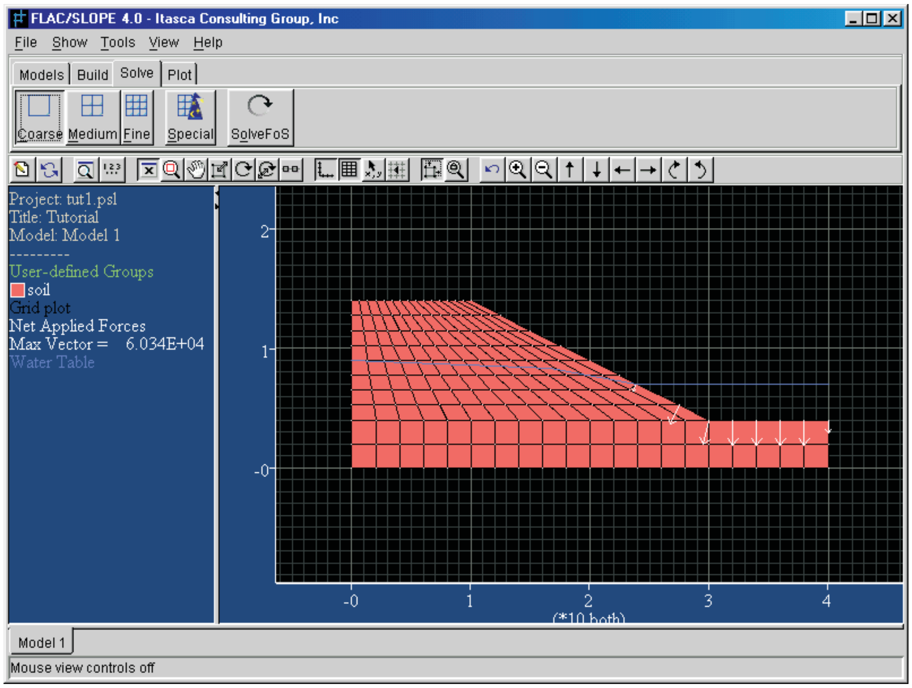
When we press OK to accept this location in the *Water* table tool, the surface water pressure is depicted in the model view by a pressure bar acting along the slope boundary. See Figure 1.33. When we click on the SOLVE tool tab to enter the *Solve* stage, the surface water pressure is shown in the model 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 1.34 shows arrows corresponding to the surface water pressure applied in Figure 1.33.



Project: tut1.psl
Title: Tutorial
Model: Model 1

Materials
🟩 soil
Boundary
Water pressure
Water table
Density 1000.0 kg/m3
Gravity 9.81 m/s2

Figure 1.33 Water pressure acting along slope surface shown in model view



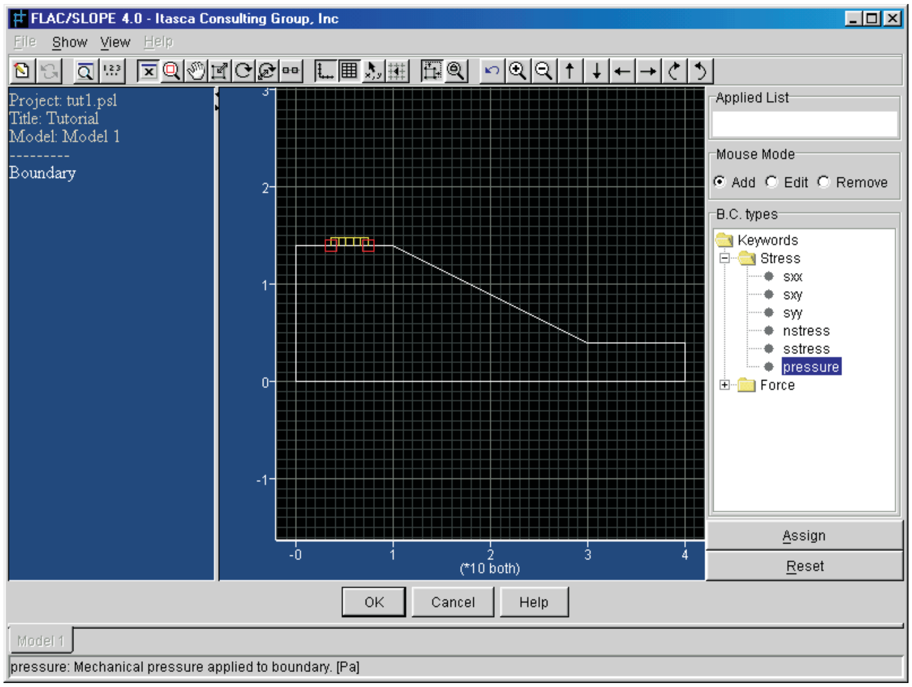
Project: tut1.psl
Title: Tutorial
Model: Model 1

User-defined Groups
🟥 soil
Grid plot
Net Applied Forces
Max Vector = 6.034E+04
Water Table

Figure 1.34 Applied forces corresponding to the surface water pressure applied in Figure 1.33

1.3.8 Applying Surface Loads

Point loads and areal pressures 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 1.35. 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 1.35 a pressure is applied at the top of the slope along the region designated by the pressure bar.



Project: tut1.psl
Title: Tutorial
Model: Model 1

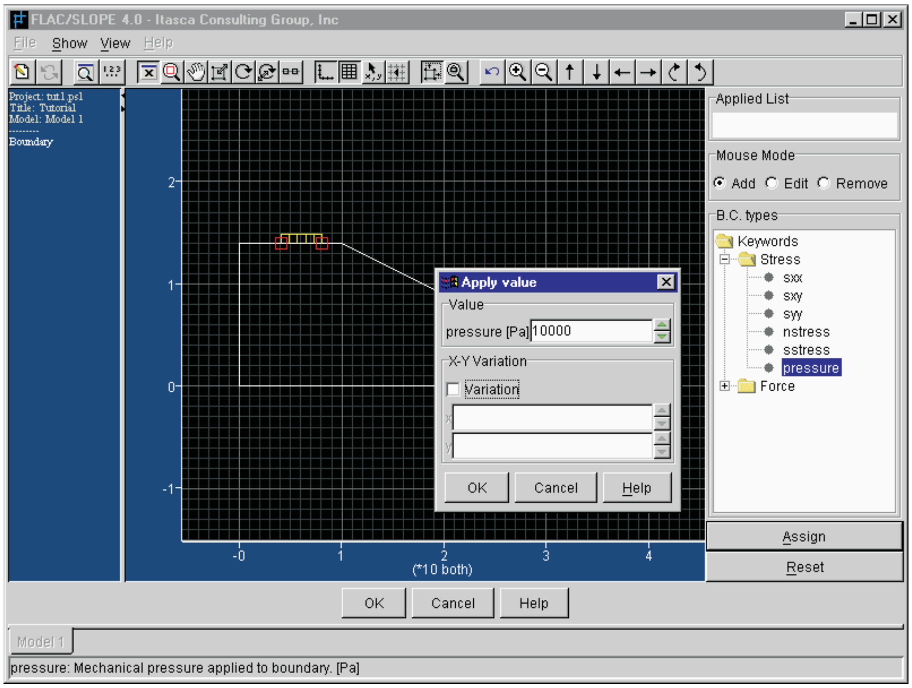
Boundary

Mouse Mode: ● Add ○ Edit ○ Remove

pressure: Mechanical pressure applied to boundary. [Pa]

Figure 1.35 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 1.36 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.



Project: tut1.psl
Title: Tutorial
Model: Model 1

Boundary

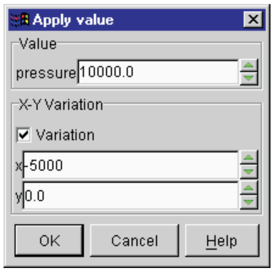
Mouse Mode: ● Add ○ Edit ○ Remove

Apply value
Value
pressure [Pa] 10000
X-Y Variation
☐ Variation

pressure: Mechanical pressure applied to boundary. [Pa]

Figure 1.36 Apply value dialog in the Apply tool

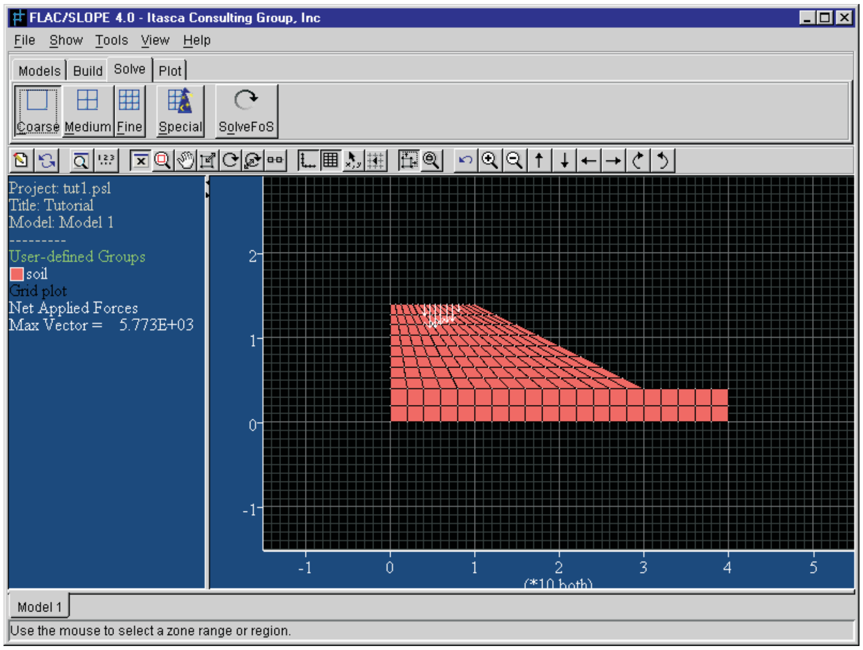
If it is necessary to make a change to the applied value, highlight the apply type to be edited in the *Applied List* and click on the EDIT button. For example, if we wish to vary the pressure in the x-direction, we highlight the pressure, click on EDIT and make the change in the *Apply value* dialog. Figure 1.37 shows the dialog. The sign conventions and formula for applying a spatial variation in load are described below.



Value
pressure 10000.0
X-Y Variation
☑ Variation
x -5000
y 0.0

Figure 1.37 Editing the applied value in the Apply value dialog

The sense of the applied stress or force can be checked by entering the *Solve* stage after pressing OK to leave the *Apply* tool. The applied loading condition will be depicted on the model view in this stage by arrows with lengths corresponding to applied forces acting at gridpoints along the model boundary. The applied forces are derived from the value for stress (or pressure) times the boundary segment length associated with each gridpoint. For example, Figure 1.38 illustrates the applied forces that correspond to the applied pressure variation prescribed in Figure 1.37.



Project: tut1.psl
Title: Tutorial
Model: Model 1

User-defined Groups
🟥 soil
Grid plot
Net Applied Forces
Max Vector = 5.773E+03

Use the mouse to select a zone range or region.

Figure 1.38 Applied forces corresponding to the applied pressure in Figure 1.37

The applied loading types are divided into two categories, stress and force. Directional stresses sxx (xx-component of stress), syy (yy-component of stress) and sxy (xy-component of stress) can be specified as boundary loads. Also, the stress component applied in the normal direction to the boundary and the stress component applied in the shear direction can be specified with the $nstress$- and $sstress$-type names. The sign convention for the direct stress components, sxx, syy and $nstress$, is that positive values indicate tension. The sign convention for shear stress components, sxy and $sstress$, is illustrated by Figure 2.34 in the **User's Guide** volume 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).

The spatial variation in an applied pressure or stress value is defined by the following formula. For a pressure or stress applied along a boundary within the range $x = x^{(s)}$ to $x^{(e)}$ and $y = y^{(s)}$ to $y^{(e)}$, then the applied pressure V is

$$V = V^{(s)} + \frac{x - x^{(s)}}{x^{(e)} - x^{(s)}} v_x + \frac{y - y^{(s)}}{y^{(e)} - y^{(s)}} v_y \qquad (1.3)$$

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 pressure or stress 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.

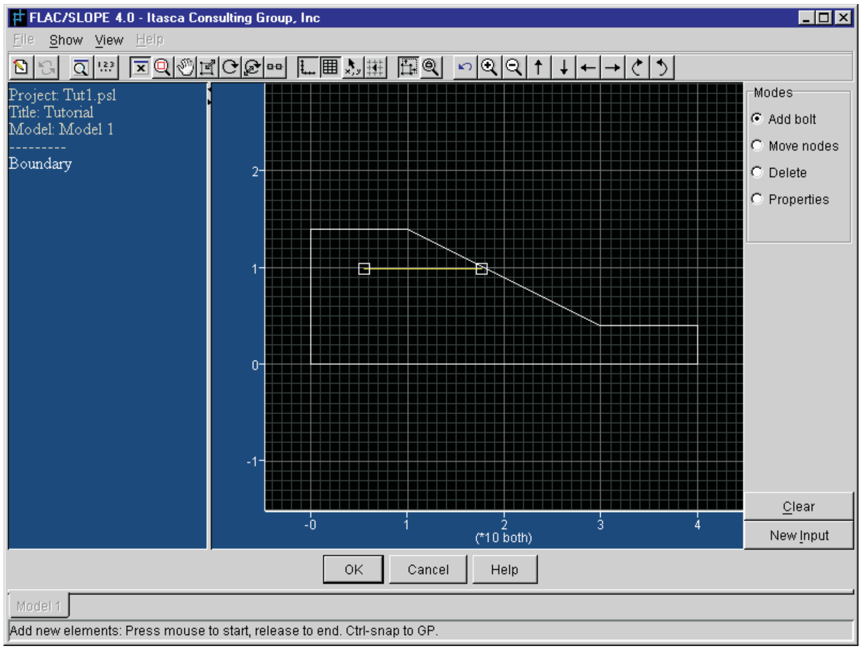
Directional forces, $xforce$ and $yforce$ can also be applied to represent a point (i.e., line) load on the boundary. A positive x- or y-force acts in the positive x- or y-direction.

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 1.3 in the **Structural Elements** volume of the full *FLAC* manual for a detailed description of the cable element logic.

Reinforcement is installed in a slope by clicking on the REINFORCE button in the BUILD tool bar. This opens the *Reinforcement* tool, as shown in Figure 1.39. Cable elements are installed in a slope by first checking the ADD BOLT radio button, and then pressing the mouse button at one endpoint of the cable, dragging the mouse to the other endpoint, and then releasing the button. A yellow line with square white handles will be drawn, as shown in Figure 1.39. Any number of cables can be installed within the slope in this manner.

The end nodes of the cable can be positioned more precisely by right-clicking on the handles. This opens a *Coordinate* dialog to enter x- and y-coordinates of the end node. End nodes can also be relocated by checking the MOVE NODES radio button. Then, press and drag the end node with the mouse. Cables can be deleted from the slope by checking the DELETE radio button. You can then click the mouse over the cable(s) you wish to delete.



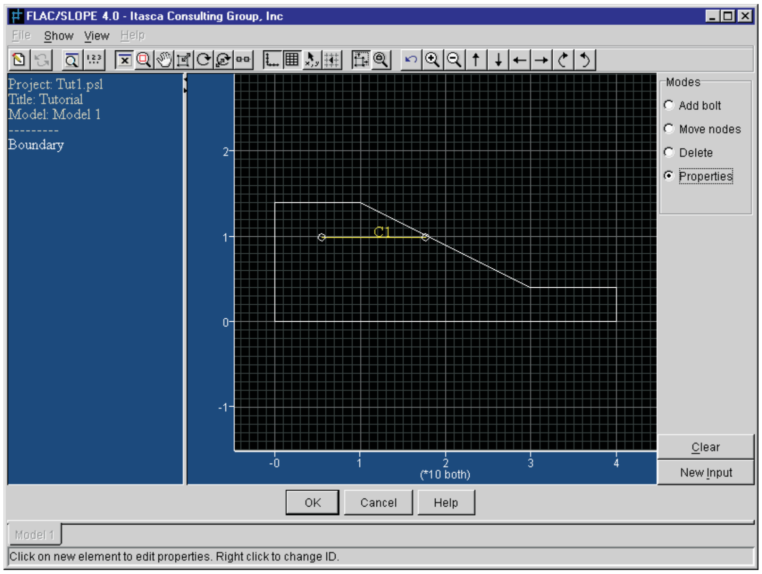
Project: Tut1.psl
Title: Tutorial
Model: Model 1

Boundary

Add new elements: Press mouse to start, release to end. Ctrl-snap to GP.

Figure 1.39 ***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 1.40.



Project: Tut1.psl
Title: Tutorial
Model: Model 1

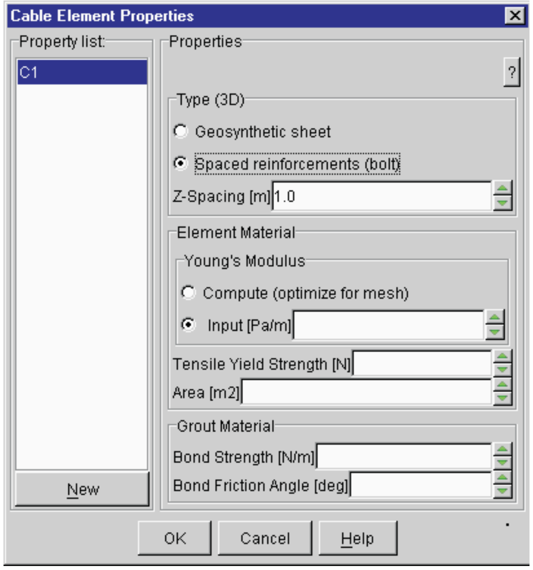
Boundary

Modes:
○ Add bolt
○ Move nodes
○ Delete
● Properties

Click on new element to edit properties. Right click to change ID.

Figure 1.40 Property identification number for reinforcement

By clicking the mouse over the property number, a *Cable Element Properties* dialog will open, as shown in Figure 1.41. Properties are then assigned to a specific property number.



C1

Type (3D)
○ Geosynthetic sheet
● Spaced reinforcements (bolt)
Z-Spacing [m] 1.0

○ Compute (optimize for mesh)
● Input [Pa/m]
Tensile Yield Strength [N]
Area [m2]

Bond Strength [N/m]
Bond Friction Angle [deg]

Figure 1.41 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 1.6.4 in the **Structural Elements** volume of the full *FLAC* manual for more information on the simulation of spaced reinforcement.) Please note that the actual properties of the cables are entered in the *Cable Element Properties* dialog, not scaled properties.

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

$$\frac{F_s^{max}}{L} = S_{bond} + \sigma_c' \times \tan(S_{friction}) \times perimeter \tag{1.4}$$

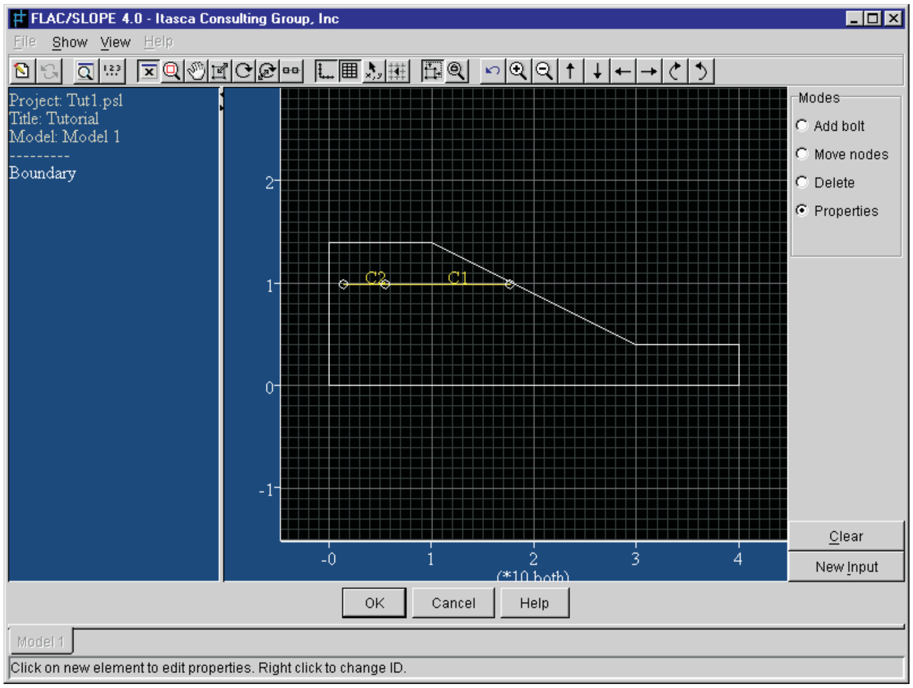
where S_{bond} = bond strength or cohesion;
σ_c' = mean effective confining stress normal to the element;
$S_{friction}$ = bond friction angle; and
$perimeter$ = perimeter of the element (based on input area).

See Section 1.3.1.2 in the **Structural Elements** volume 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 the **Theory and Background** volume 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 1.41 this is $C1$. Additional property numbers can be created by pressing the NEW 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 OK 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 1.42 shows a bolt composed of two segments. This is created in the ADD BOLT mode by creating one segment and then clicking the mouse over one existing end node to start the second segment. The second segment will automatically be connected to the first. After checking PROPERTIES, we can then assign properties for the unbonded segment to $C1$, and the bonded segment to $C2$. We change the left portion of the bolt in Figure 1.42 to $C2$ by highlighting $C2$ in the *Cable Element Properties* dialog.



Project: Tut1.psl
Title: Tutorial
Model: Model 1

Boundary

● Properties

Click on new element to edit properties. Right click to change ID.

Figure 1.42 Creating a grouted and ungrouted bolt

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

Axial forces are calculated in the cables during the factor-of-safety calculation. These values can be added to the output plots; see Section 1.3.11. 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 Solving for a Factor of Safety

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

1.3.10.1 Grid Generation

When the *Solve* stage is entered, a numerical mesh must first be created. Four zoning choices are available: coarse, medium, fine and user-selected (special). These can be selected by pressing the different buttons shown in the *Solve* tool. For example, by pressing the COARSE button, a "coarse-grid" model is created, as shown in Figure 1.43. If the MEDIUM button is pressed, a "medium-grid" model appears as shown in Figure 1.44. The fineness of zoning affects the accuracy of the factor-of-safety calculation; the finer the zoning, the better the accuracy of the solution.

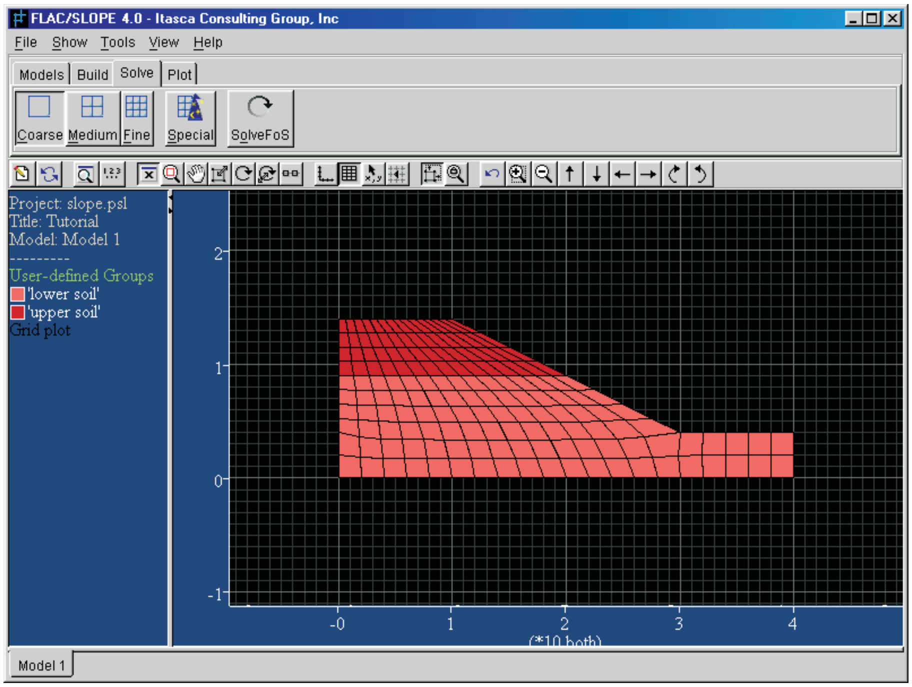
The coarse-grid model is recommended for preliminary analyses. The solution for this model is quite rapid; on a 1 GHz computer, a solution time is typically only a few seconds. A project with several models can easily be run to provide a quick estimate for the effect of different conditions on the factor 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, it usually is not warranted to use fine-grid models for comprehensive studies.

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 finer grid model. If a "bad geometry" message appears during the grid generation, it will not be possible to perform a safety-factor calculation with the current model. In this case, a finer-grid model should be applied. If there is still a "bad-geometry" problem when using a fine-grid model, then the user-defined SPECIAL zoning tool should be tried. 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.

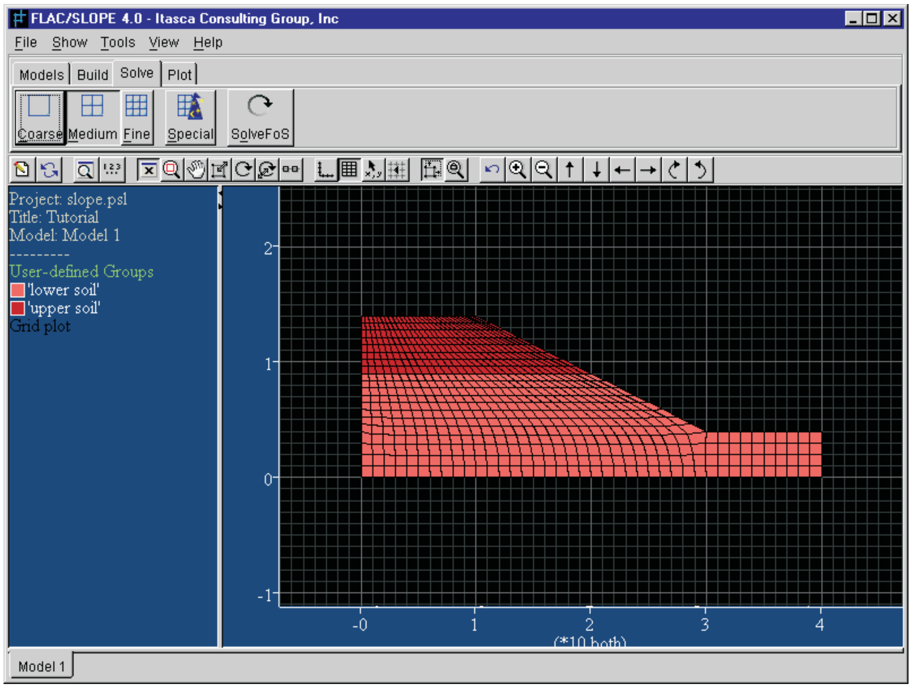
Each time one of the zoning buttons is pressed, a set of *FLAC* commands, corresponding to the model created in the *Build* tool, is executed to create the model for the factor-of-safety calculation. The state of the model is also saved at this stage, with a file extension of "*.SAV." The name of the save file is defined by the project and model names and type of zoning. For example, when the coarse model is created for the tutorial example in Section 1.2.2, a model save file is created with the name "slope_Model_1_Coarse.sav."



Project: slope.psl
Title: Tutorial
Model: Model 1

User-defined Groups
🟥 'lower soil'
🟥 'upper soil'
Grid plot

Figure 1.43 Coarse-grid model



Project: slope.psl
Title: Tutorial
Model: Model 1

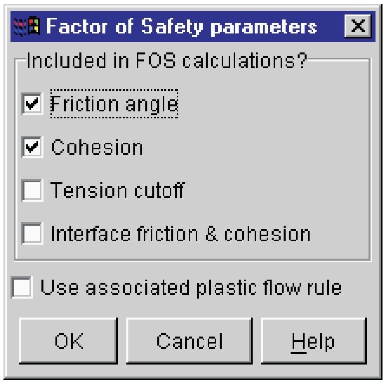
User-defined Groups
🟥 'lower soil'
🟥 'upper soil'
Grid plot

Figure 1.44 Medium-grid model

1.3.10.2 Factor-of-Safety Parameters

After the grid generation is complete, the safety factor can be calculated. The calculation is performed by pressing the SolveFoS button. The factor-of-safety calculation is based on the strength reduction technique, as described in Section 1.5. By default, the material strength parameters, cohesion and friction angle are reduced in accordance with Eqs. (1.1) and (1.2). When SolveFoS is pressed, a *Factor of Safety parameters* dialog opens, with Friction angle and Cohesion boxes checked, as shown in Figure 1.45. By pressing OK, the calculation will commence.

It is also possible to include other strength parameters in the safety factor calculation. By checking the Tension cutoff button, the material tensile strength can be reduced in a similar fashion to the material cohesion and friction angle. If a weak plane is included in the model, the Interface friction & cohesion button should be checked to include these interface strength properties in the strength reduction solution. If these buttons are not checked, the corresponding assigned properties will not be changed during the safety factor calculation.



Included in FOS calculations?

☑ Friction angle
☑ Cohesion
☐ Tension cutoff
☐ Interface friction & cohesion

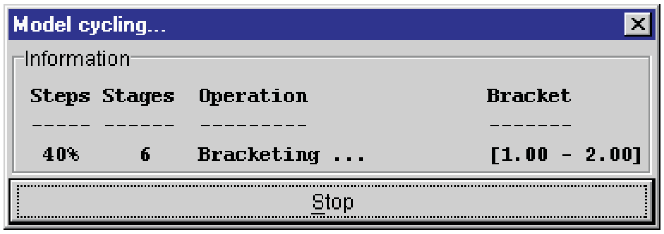
☐ Use associated plastic flow rule

Figure 1.45 Factor of Safety parameters dialog

Associated or non-associated plastic flow can also be specified for the factor-of-safety calculation via the Use associated plastic flow rule check box. The material plastic flow rule quantifies the effect of shear dilatancy that occurs in a material at the onset of failure. This is generally expressed by the relation between the friction angle of the Mohr-Coulomb 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.

1.3.10.3 Factor-of-Safety Solution

When OK is pressed in the *Factor of Safety parameters* dialog, the factor-of-safety calculation begins. A series of simulations will be made as described in Section 1.5, and the status of the calculation will be reported in a *Model cycling* dialog, as shown in Figure 1.46. This dialog displays the percentage of steps completed for an individual simulation stage, the total number of 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. When the calculation is complete, the final value is reported.



```
Steps Stages  Operation                   Bracket
----- ------  ---------                   -------
 40%     6    Bracketing ...              [1.00 - 2.00]
```

Figure 1.46 Model cycling dialog

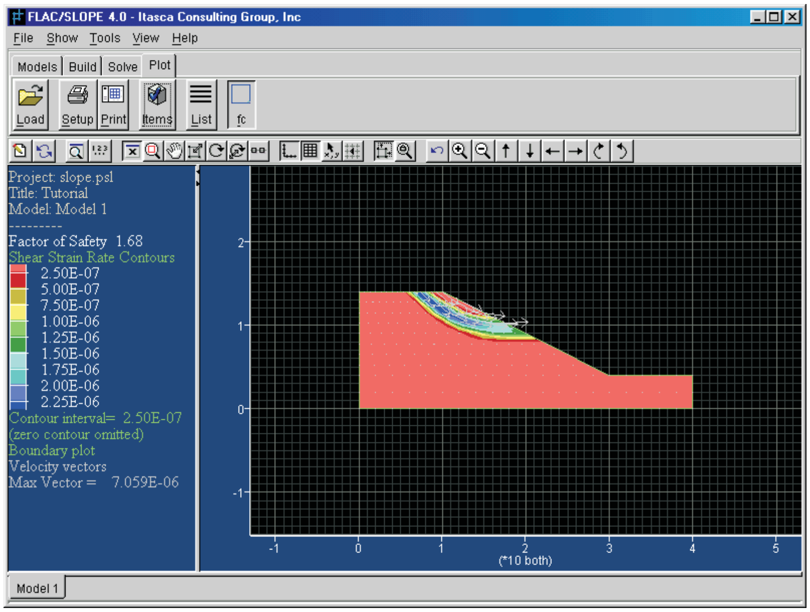
After the first lower-bound value has been found in the series, the run can be interrupted, by pressing the STOP button. An estimate for factor of safety will be reported based on the current bracketing limits, but this will be less accurate than if the operation had been allowed to complete.

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 1.3.11. 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_Coarse_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
- a = associated plastic flow rule

1.3.11 Producing Output

The results of the factor-of-safety calculation are viewed in the *Plot* tool which is accessed by pressing the PLOT button. When a calculation is complete, a failure-plot button is added to the PLOT tool bar with a name corresponding to the type of zoning and factor-of-safety parameters selected for the calculation. For example, in Figure 1.47, the button contains a single square, indicating a coarse grid model, and the descriptors *f c*, 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.



```
Project: slope.psl
Title: Tutorial
Model: Model 1
---------
Factor of Safety  1.68
Shear Strain Rate Contours
     2.50E-07
     5.00E-07
     7.50E-07
     1.00E-06
     1.25E-06
     1.50E-06
     1.75E-06
     2.00E-06
     2.25E-06
Contour interval=  2.50E-07
(zero contour omitted)
Boundary plot
Velocity vectors
Max Vector =    7.059E-06
```

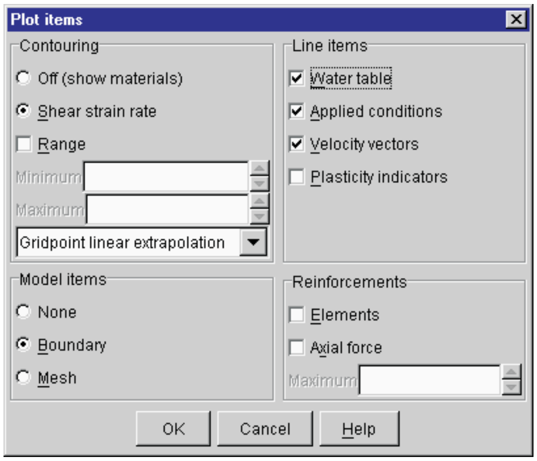
Figure 1.47 Failure plot for coarse grid model with friction angle and cohesion included in the factor-of-safety calculation

The failure plot displayed in this tool contains, by default, a filled contour plot of shear strain-rate contours and velocity vectors. These parameters indicate the extent of the failure region at the last non-equilibrium state, and show a well-defined failure surface, as illustrated in Figure 1.47. Different parameters can be displayed in the failure plot. By pressing the ITEMS button, a *Plot items* dialog opens as shown in Figure 1.48.

The shear strain-rate contours are derived from strain-rate values calculated in *FLAC* at zone centroids. (See Section 1.3 in the **Theory and Background** volume of the full *FLAC* manual for a description of the strain-rate calculation in a *FLAC* analysis.) 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 two different procedures: a simple linear averaging and a bilinear least-squares fit. (The extrapolation procedure is described in “EXTRAP.FIS” in Section 3 in the ***FISH* volume** of the full *FLAC* manual) The three contouring approaches can be accessed by clicking on ZONE CENTROID (EXACT), GRIDPOINT LINEAR EXTRAPOLATION or GRIDPOINT BILINEAR 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.

In addition, the range of the contouring can be controlled; this is useful to define a common contour level if several model results are compared.

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, and whether the stress state in the zone is currently at the yield surface (“at yield”) or has been at the yield surface but currently is below the yield surface (“at yield in past”).



Plot items

Contouring
○ Off (show materials)
● Shear strain rate
☐ Range
Minimum
Maximum
Gridpoint linear extrapolation

Line items
☑ Water table
☑ Applied conditions
☑ Velocity vectors
☐ Plasticity indicators

Model items
○ None
● Boundary
○ Mesh

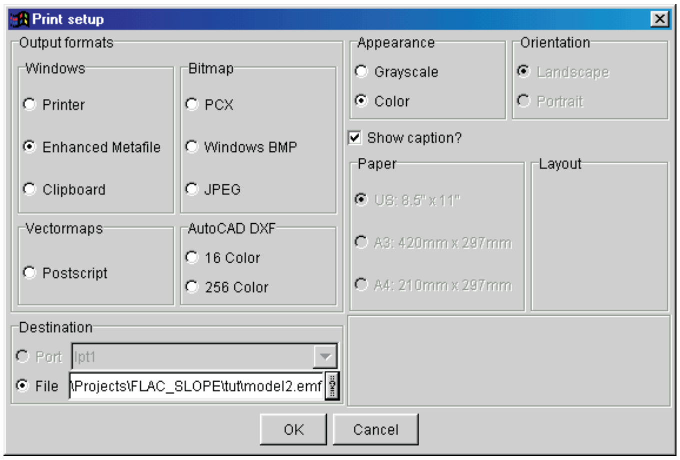
Reinforcements
☐ Elements
☐ Axial force
Maximum

Figure 1.48 Failure 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 failure 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 100,000 N so that the results from two different cases can easily be compared, as shown in Figures 1.75 and 1.76.

Results from other projects can be included in the *Plot* tool by loading the selected “*.FSV” file with the LOAD tool. A failure-plot button will be added to the PLOT toolbar for the loaded model. The list of failure plots can be edited and failure plots removed from the toolbar by pressing the LIST button to open a *FoS Plots* dialog.

A hardcopy printout of the failure plot can be created in the *Print setup* dialog, which is opened by pressing the SETUP button. The dialog is shown in Figure 1.49. 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.



Print setup

Output formats

Windows
○ Printer
● Enhanced Metafile
○ Clipboard

Bitmap
○ PCX
○ Windows BMP
○ JPEG

Vectormaps
○ Postscript

AutoCAD DXF
○ 16 Color
○ 256 Color

Appearance
○ Grayscale
● Color

Orientation
● Landscape
○ Portrait

☑ Show caption?

Paper
● US: 8.5" x 11"
○ A3: 420mm x 297mm
○ A4: 210mm x 297mm

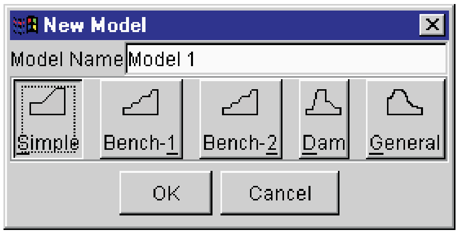
Layout

Destination
○ Port lpt1
● File \Projects\FLAC_SLOPE\tut\model2.emf

Figure 1.49 Print setup dialog

1.3.12 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 1.50. These tools define common slope shapes which can be used as a starting point for creation of similarly shaped models. Three general boundary shapes are given: bench slope, dam or embankment, and general, nonlinear slope. The procedures for creating slopes for these three types are described below.



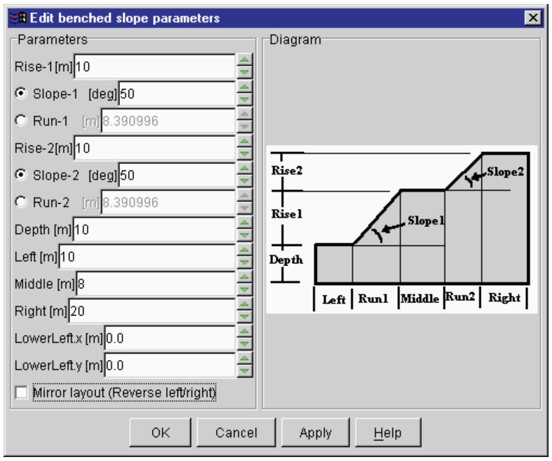
New Model

Model Name: Model 1

Figure 1.50 New Model dialog

1.3.12.1 Building a Benched Slope

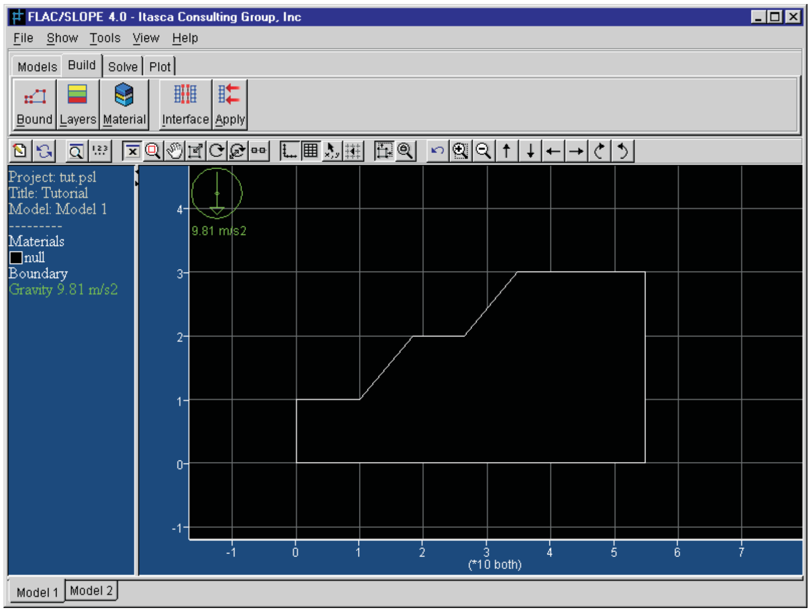
Two boundary tools are provided to generate bench slopes; these create slopes with one or two benches. If more than two benches are required, then the GENERAL boundary tool should be used. When the BENCH-1 button is pressed in the *New Model* dialog, an *Edit benched slope parameters* dialog opens for a single bench slope, as shown in Figure 1.51. The dimensions for the bench are defined in the diagram included in this dialog. For example, using the dimensions shown in Figure 1.51, a bench boundary is produced as illustrated in Figure 1.52. A two-bench slope is produced in a similar fashion when the BENCH-2 button is pressed.


Edit benched slope parameters

Parameters

Rise-1 [m] 10
● Slope-1 [deg] 50
○ Run-1 [m] 8.390996
Rise-2 [m] 10
● Slope-2 [deg] 50
○ Run-2 [m] 8.390996
Depth [m] 10
Left [m] 10
Middle [m] 8
Right [m] 20
LowerLeft.x [m] 0.0
LowerLeft.y [m] 0.0
☐ Mirror layout (Reverse left/right)

Figure 1.51 Edit benched slope parameters dialog

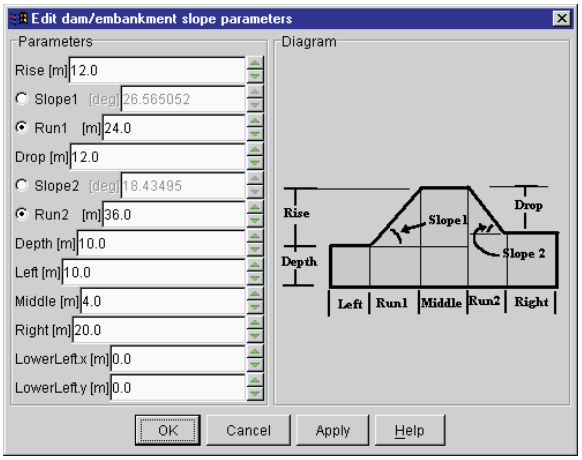

Project: tut.psl
Title: Tutorial
Model: Model 1

Materials
null
Boundary
Gravity 9.81 m/s2

Figure 1.52 Single bench slope boundary

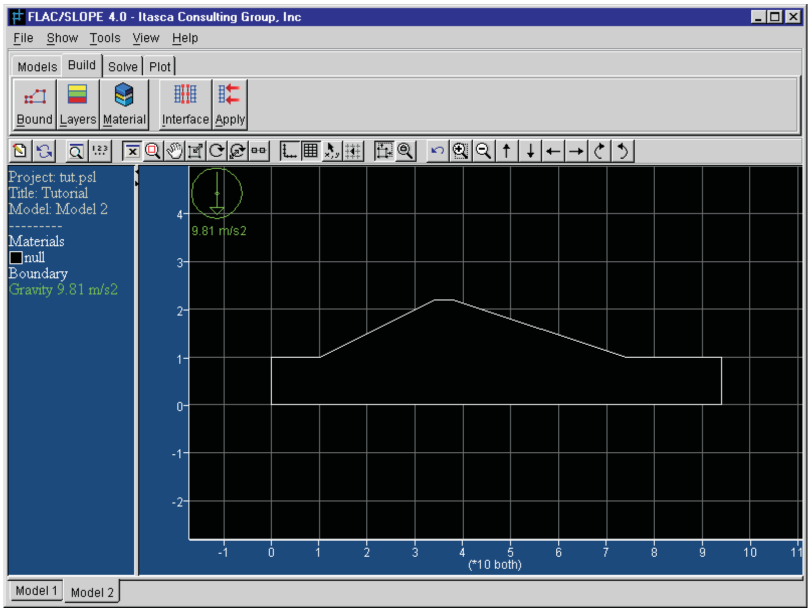
1.3.12.2 Building a Dam Embankment

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 1.53. The dimensions for the dam are defined in the diagram included in this dialog. For example, using the dimensions shown in Figure 1.53, a dam boundary is produced as illustrated in Figure 1.54.



Rise [m] 12.0
○ Slope1 [deg] 26.565052
● Run1 [m] 24.0
Drop [m] 12.0
○ Slope2 [deg] 18.43495
● Run2 [m] 36.0
Depth [m] 10.0
Left [m] 10.0
Middle [m] 4.0
Right [m] 20.0
LowerLeft.x [m] 0.0
LowerLeft.y [m] 0.0

Figure 1.53 Edit dam/embankment slope parameters dialog



Project: tut.psl
Title: Tutorial
Model: Model 2

Materials
■ null
Boundary
Gravity 9.81 m/s2

Figure 1.54 Dam boundary

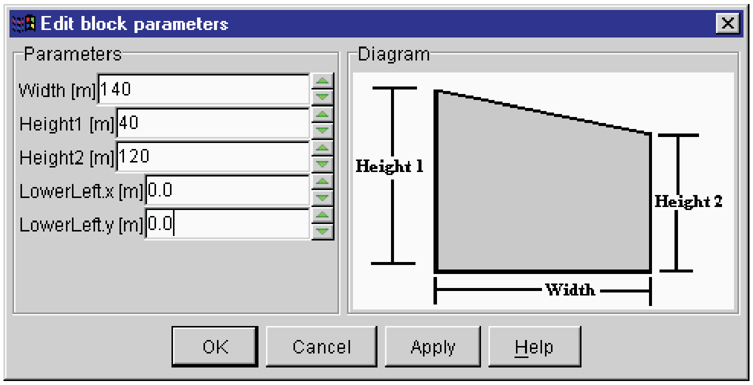
1.3.12.3 Building a Nonlinear-Shaped Model

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 1.55. 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 in green and the slope boundary in yellow. The shape of the yellow line can be modified by adding handle points along the line, and then dragging the points to different locations. Alternatively, handle points can be located at specific x- and y-coordinate positions by right-clicking the mouse over the handle. A *Table* dialog will open to enter the coordinates.

The slope line corresponds to a table of points that define the slope surface. The line table can be edited by clicking on the Edit Numerically button in the *General boundary* tool; this opens an *Edit Table points* dialog in which the x- and y-coordinates for all the slope points are listed. Points can be input and edited in this dialog.

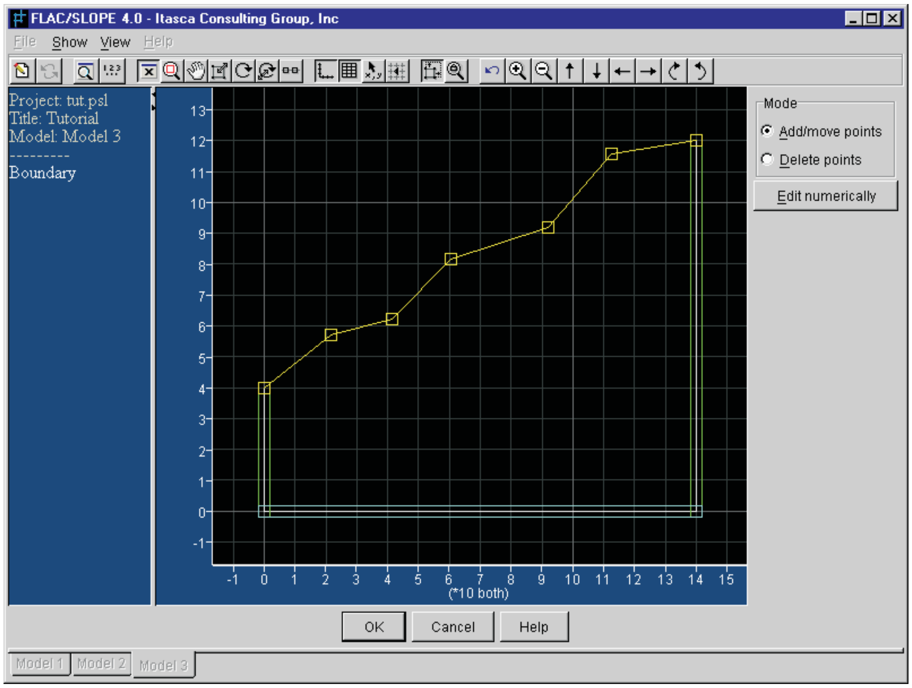
Figure 1.56 shows the *General boundary* tool with a nonlinear slope defined by seven handle points. Figure 1.57 illustrates the final slope boundary.

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.



Width [m] 140
Height1 [m] 40
Height2 [m] 120
LowerLeft.x [m] 0.0
LowerLeft.y [m] 0.0

Figure 1.55 Edit block parameters dialog

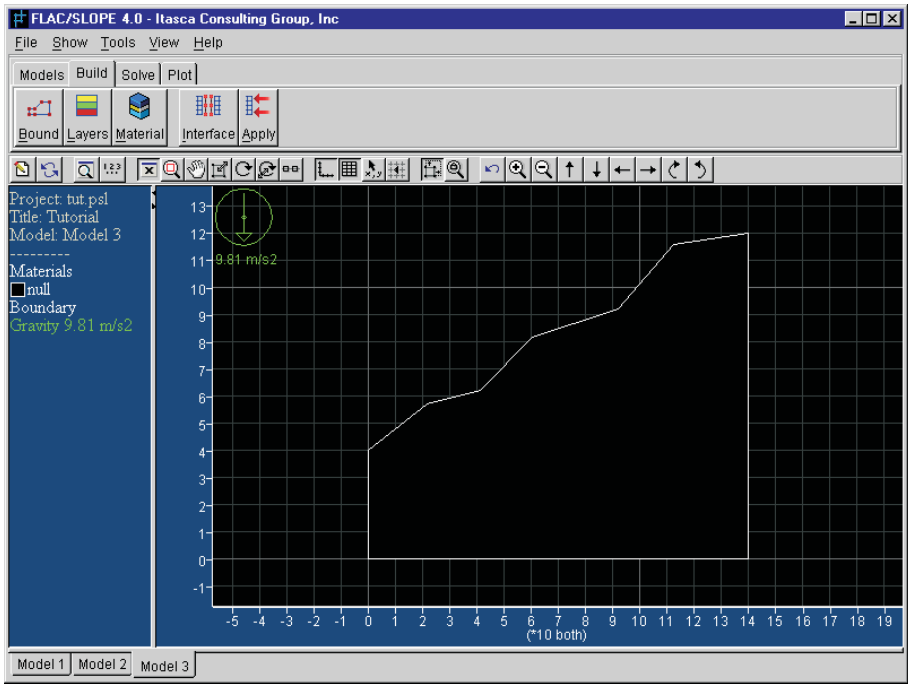


Project: tut.psl
Title: Tutorial
Model: Model 3

Boundary

Mode: ● Add/move points ○ Delete points

Figure 1.56 General boundary tool



Project: tut.psl
Title: Tutorial
Model: Model 3

Materials
■null
Boundary
Gravity 9.81 m/s2

Figure 1.57 Nonlinear slope boundary

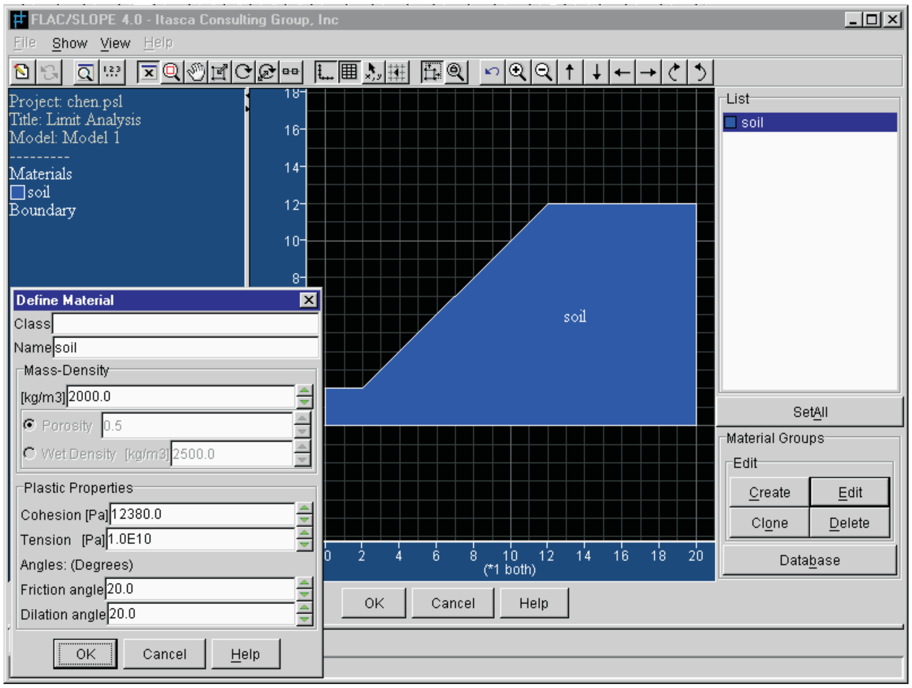
1.4 Stability Analysis Examples

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 "FLAC\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 β = 45°, unit weight γ = 20 kN/m^3, cohesion c = 12.38 kPa and friction angle ϕ = 20 °. A gravitational acceleration of 10.0 m/sec^2 is also specified. For these parameters, Chen calculates a factor of safety of exactly 1.0. This example problem is also presented in the publication by Dawson et al. (1999), which compares and validates the *FLAC* solution for several variations of the homogeneous embankment conditions.

We enter the embankment conditions in the *FLAC/Slope* model in the *Build* stage. Figure 1.58 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 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 cut-off, for comparison to the Chen solution. The project save file for this example is "CHEN.PSL."



Project: chen.psl
Title: Limit Analysis
Model: Model 1

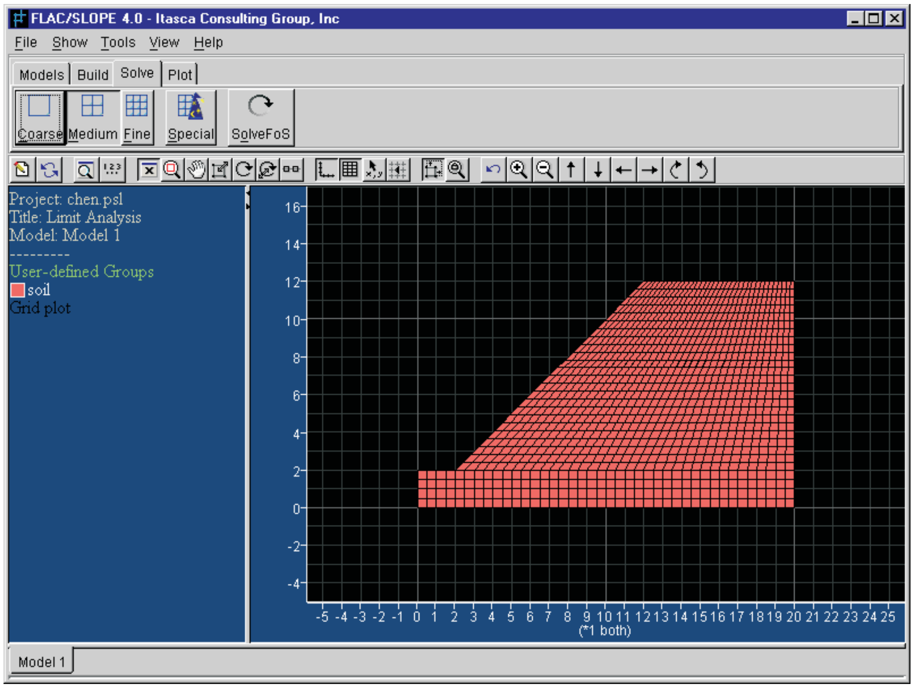
Materials
☐ soil
Boundary

Define Material
Class
Name soil
Mass-Density
[kg/m3] 2000.0
● Porosity 0.5
○ Wet Density [kg/m3] 2500.0
Plastic Properties
Cohesion [Pa] 12380.0
Tension [Pa] 1.0E10
Angles: (Degrees)
Friction angle 20.0
Dilation angle 20.0
OK Cancel Help

Figure 1.58 Material properties for homogeneous embankment example

We use the *Medium* grid mode in the *Solve* stage; the resulting grid is shown in Figure 1.59. We perform the factor-of-safety calculation and calculate a factor of 1.01. The failure surface is indicated in Figure 1.60. Note that for a *Coarse* mesh, the calculated factor of safety is 1.03.

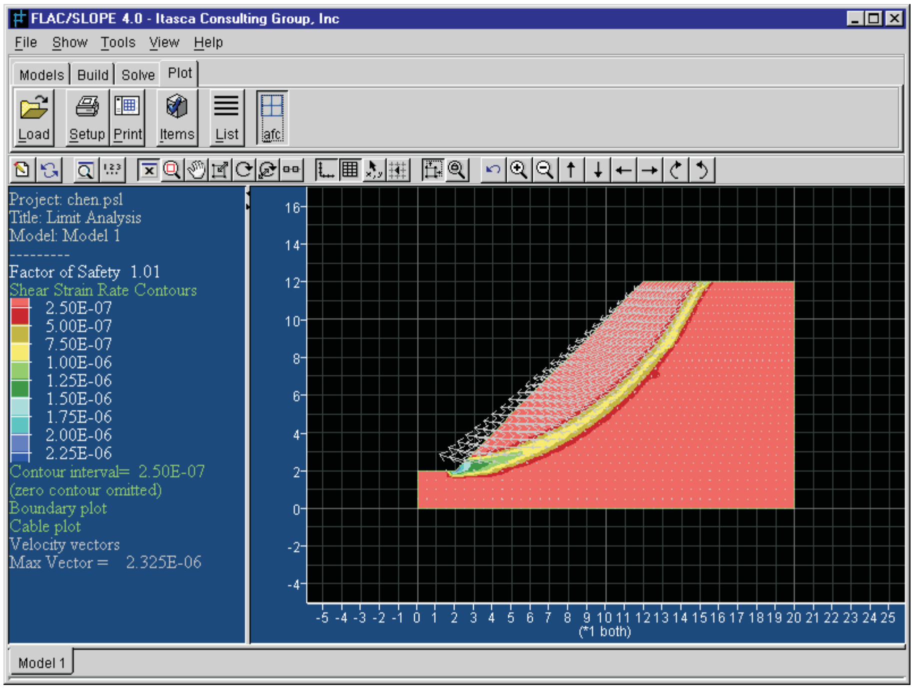
We also investigate the effect of assuming an associated flow behavior. If non-associated flow is selected (with $\psi = 0$) in the *SOLVE FOS* dialog, the calculated factor of safety is 1.00 for the coarse grid and 0.98 for the medium grid.



Project: chen.psl
Title: Limit Analysis
Model: Model 1

User-defined Groups
🟥 soil
Grid plot

Figure 1.59 Medium-grid zoning for homogeneous embankment example



Project: chen.psl
Title: Limit Analysis
Model: Model 1

Factor of Safety 1.01
Shear Strain Rate Contours
2.50E-07
5.00E-07
7.50E-07
1.00E-06
1.25E-06
1.50E-06
1.75E-06
2.00E-06
2.25E-06
Contour interval= 2.50E-07
(zero contour omitted)
Boundary plot
Cable plot
Velocity vectors
Max Vector = 2.325E-06

Figure 1.60 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 1.61. Four of the cases analyzed by Fredlund and Krahn (1977) are re-analyzed with *FLAC/Slope*. The descriptions of these cases are:

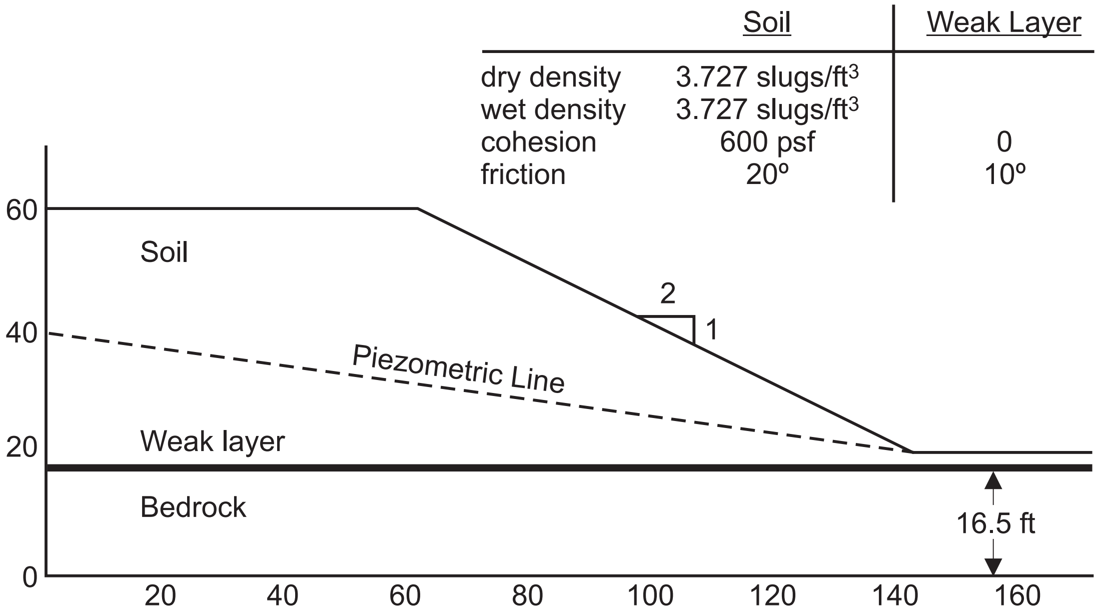
Case 1: Simple 2:1 slope, 40 ft high, $\phi' = 20°$, $c' = 600$ psf, no weak layer, no bedrock

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

Case 5: Same as Case 1 with piezometric line

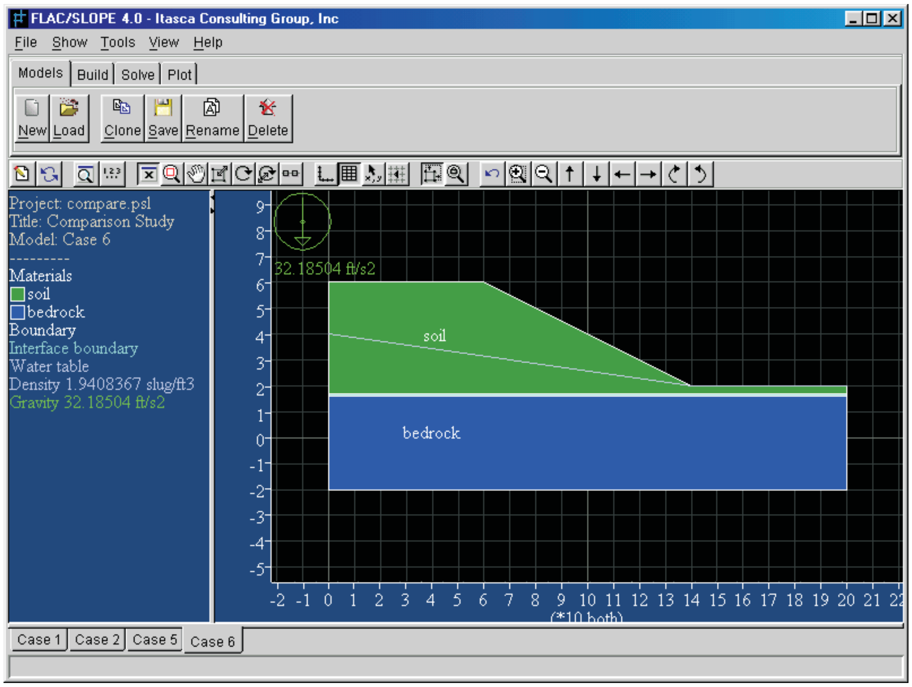
Case 6: Same as Case 2 with piezometric line

The four cases are created in *FLAC/Slope* as four separate models. The project save file for this example is “COMPARE.PSL.” Figure 1.62 shows the model for the Case 6 conditions. Note that the weak layer is represented by an interface in the model. The *Medium* grid for this model is shown in Figure 1.63.



	Soil	Weak Layer
dry density	3.727 slugs/ft^3	
wet density	3.727 slugs/ft^3	
cohesion	600 psf	0
friction	20°	10°

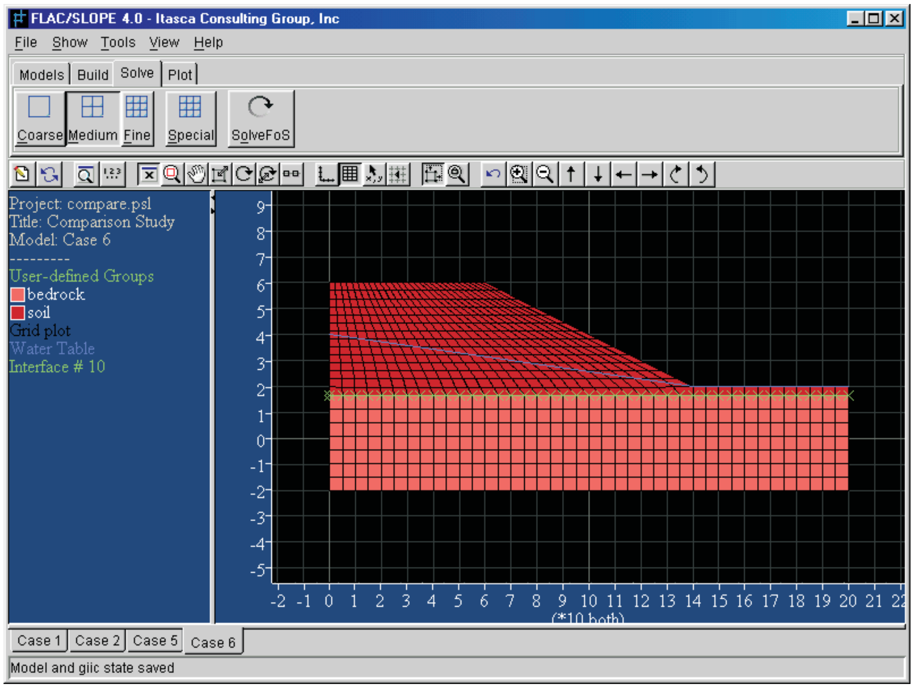
Figure 1.61 Slope stability example (from Fredlund and Krahn, 1977)



Project: compare.psl
Title: Comparison Study
Model: Case 6

Materials
🟩 soil
🟦 bedrock
Boundary
Interface boundary
Water table
Density 1.9408367 slug/ft3
Gravity 32.18504 ft/s2

Figure 1.62 FLAC/Slope geometry for Case 6



Project: compare.psl
Title: Comparison Study
Model: Case 6

User-defined Groups
🟥 bedrock
🟥 soil
Grid plot
Water Table
Interface # 10

Model and glic state saved

Figure 1.63 FLAC/Slope grid for Case 6

The result for the factor-of-safety calculation for Case 6 is illustrated in Figure 1.64. The *FLAC/Slope* results for all four cases are summarized in Table 1.2. The *FLAC/Slope* results are in good agreement with the results from the limit-equilibrium calculations.

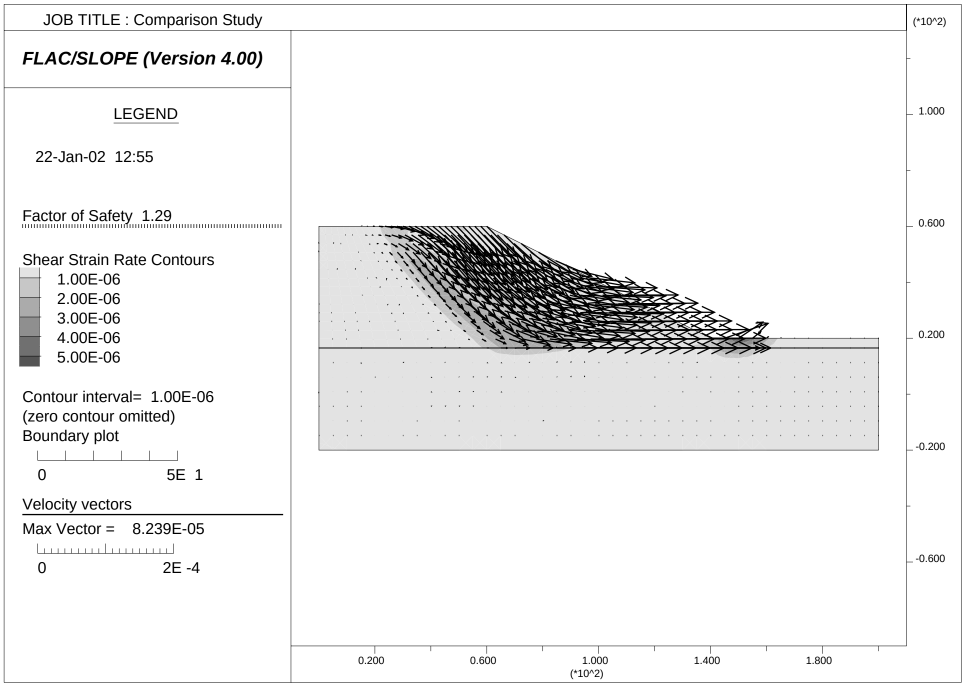


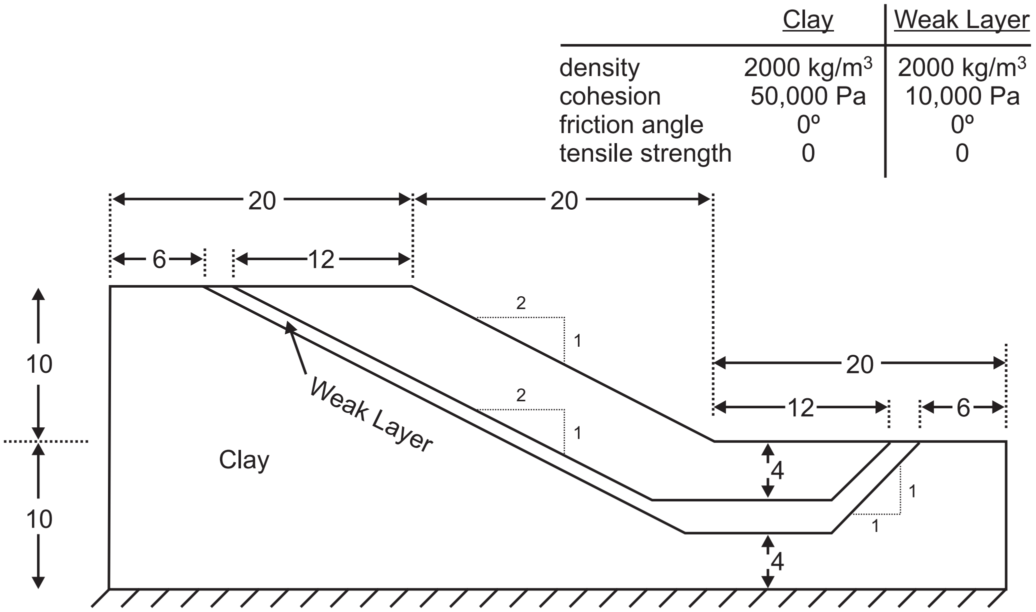
Figure 1.64 Factor-of-safety results for Case 6

Table 1.2 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	*FLAC/Slope*
1	2.08	2.07	2.01	2.08	2.00
2	1.38	1.37	1.43	1.38	1.35
5	1.83	1.83	1.78	1.83	1.79
6	1.25	1.25	1.30	1.25	1.29

1.4.3 Slope with a Thin, Weak Layer

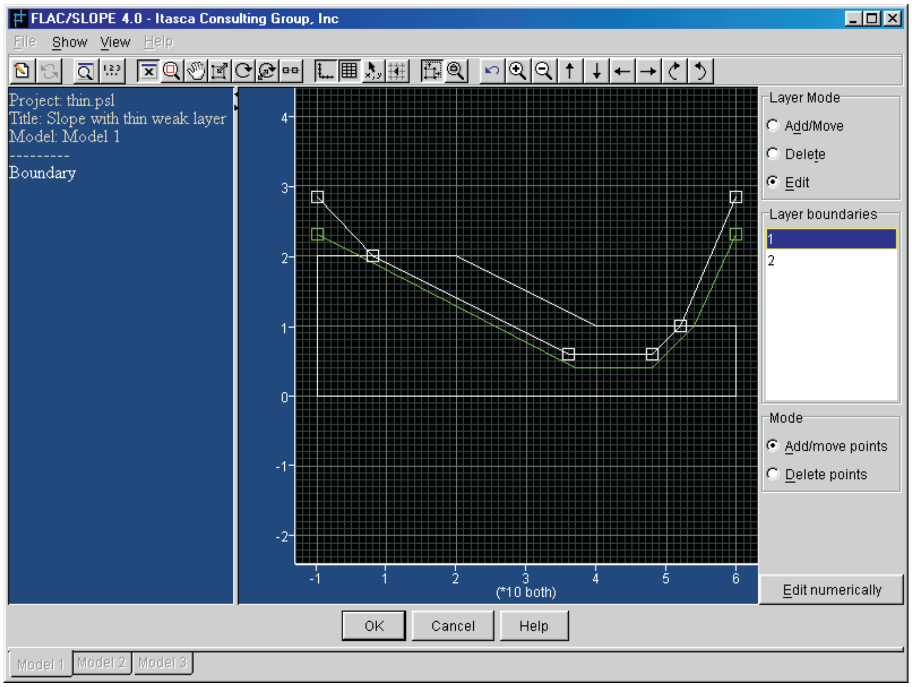
A clay slope contains a thin layer of weaker material, which is located within the slope as shown in Figure 1.65. The cohesion of the weak plane (c_l = 10,000 Pa) is 20% of the cohesion of the clay (c = 50,000 Pa). The strength of the weak plane is varied, while the strength of the clay is kept constant, 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.



	Clay	Weak Layer
density	2000 kg/m^3	2000 kg/m^3
cohesion	50,000 Pa	10,000 Pa
friction angle	0°	0°
tensile strength	0	0

Figure 1.65 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 1.65. 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 1.66. The resulting model is shown in Figure 1.67. A fine-grid model is necessary to represent the thin layer — see Figure 1.68. Three cases are analyzed: c_l/c = 0.2, 0.6 and 1.0. The project save file for this example is “THIN.PSL.”



Project: thin.psl
Title: Slope with thin weak layer
Model: Model 1

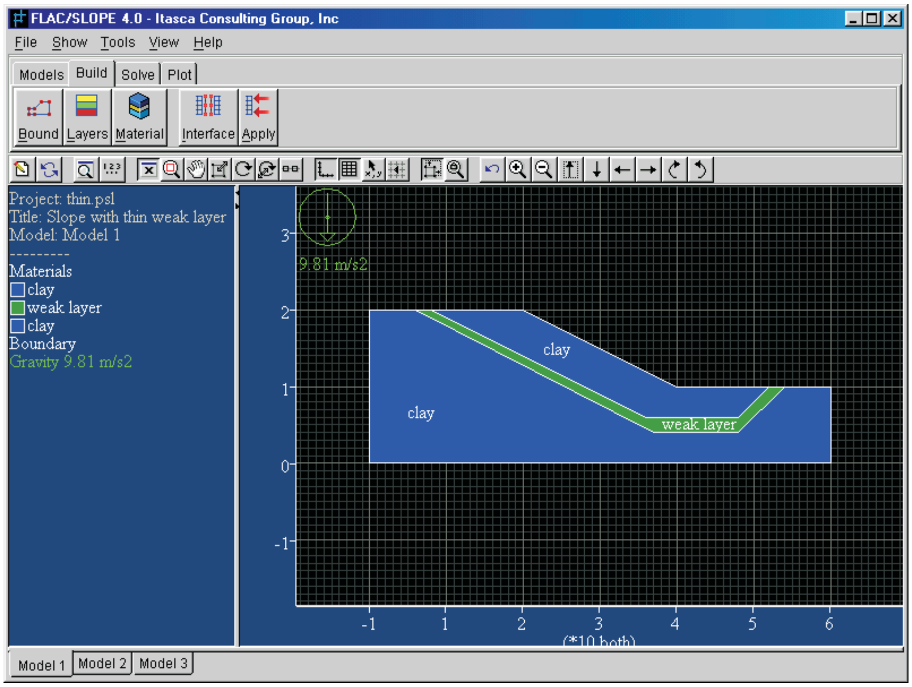
Boundary

Layer Mode
○ Add/Move
○ Delete
● Edit

Layer boundaries
1
2

Mode
● Add/move points
○ Delete points

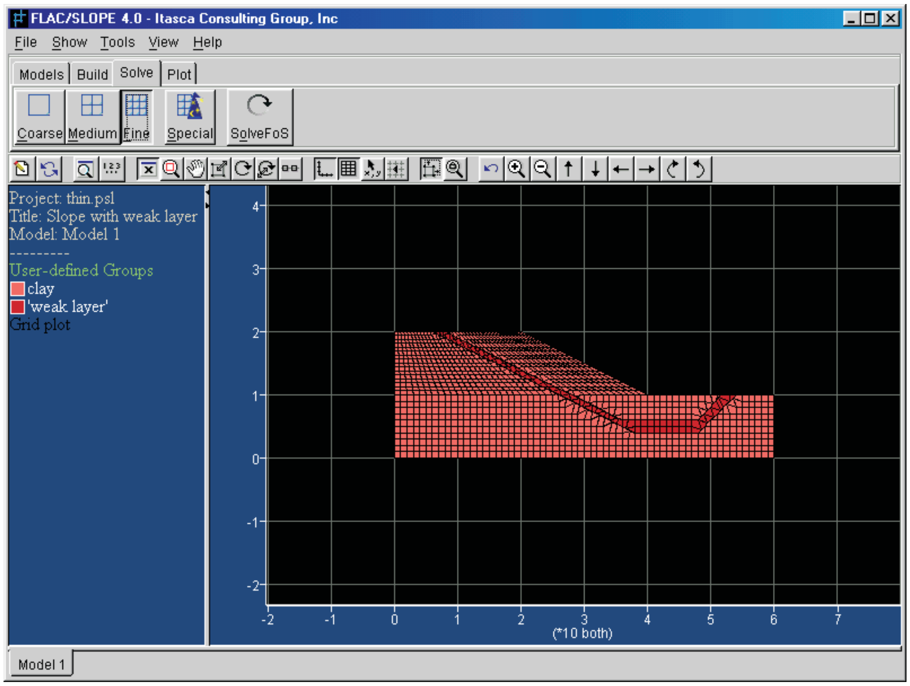
Figure 1.66 Weak layer boundaries created in the Layers tool



Project: thin.psl
Title: Slope with thin weak layer
Model: Model 1

Materials
🟦 clay
🟩 weak layer
🟦 clay
Boundary
Gravity 9.81 m/s2

Figure 1.67 FLAC/Slope model of slope with a thin weak layer



Project: thin.psl
Title: Slope with weak layer
Model: Model 1

User-defined Groups
🟥 clay
🟥 'weak layer'
Grid plot

Figure 1.68 Fine-grid model for slope with thin weak layer

The failure plots for the three cases are shown in Figures 1.69 through 1.71. 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 1.69, the failure surface indicates localized slip along the weak plane, while in Figure 1.71, a circular failure surface develops in the homogeneous material. Figure 1.70 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.

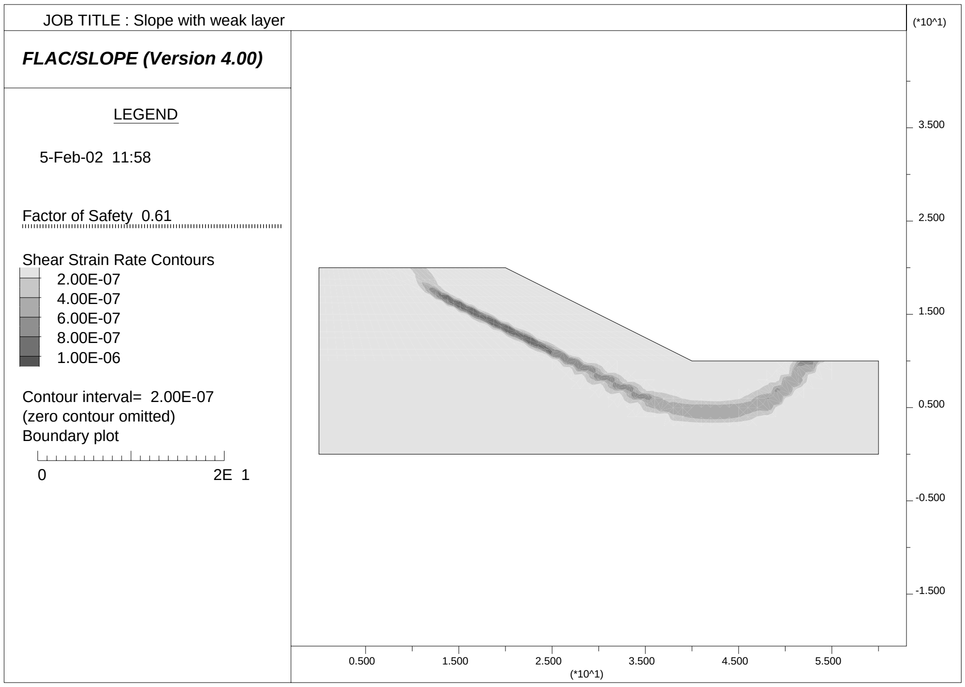


Figure 1.69 Failure plot for c_l / c = 0.2

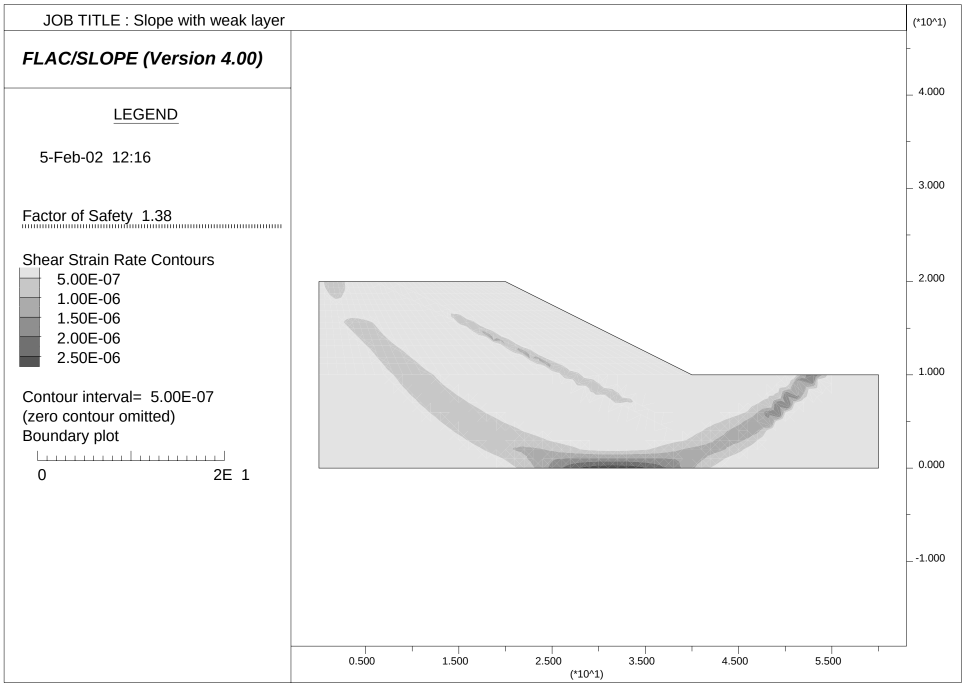


Figure 1.70 Failure plot for c_l / c = 0.6

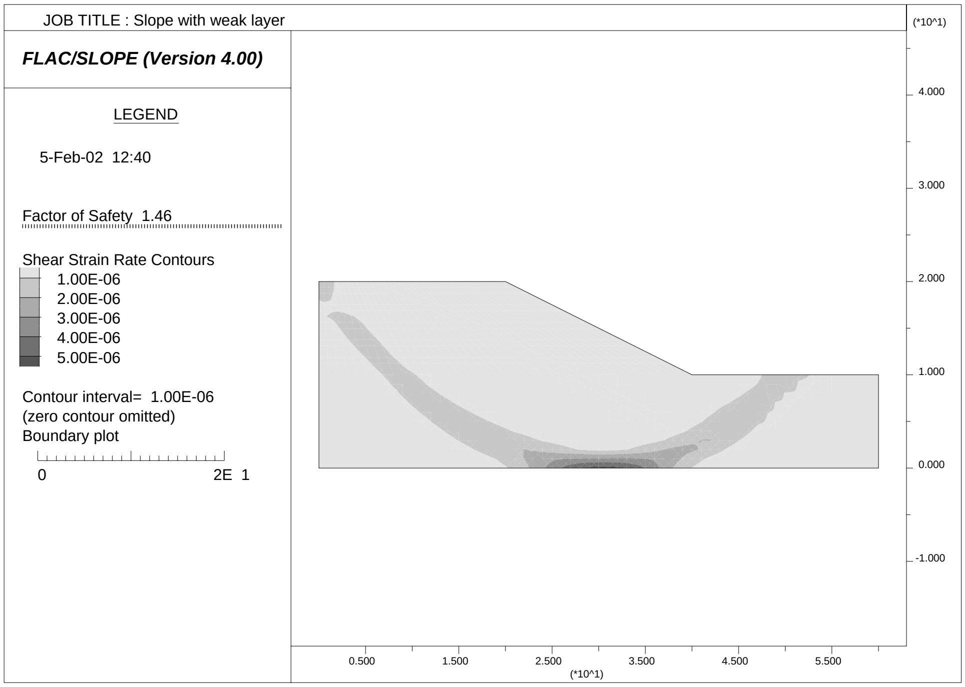
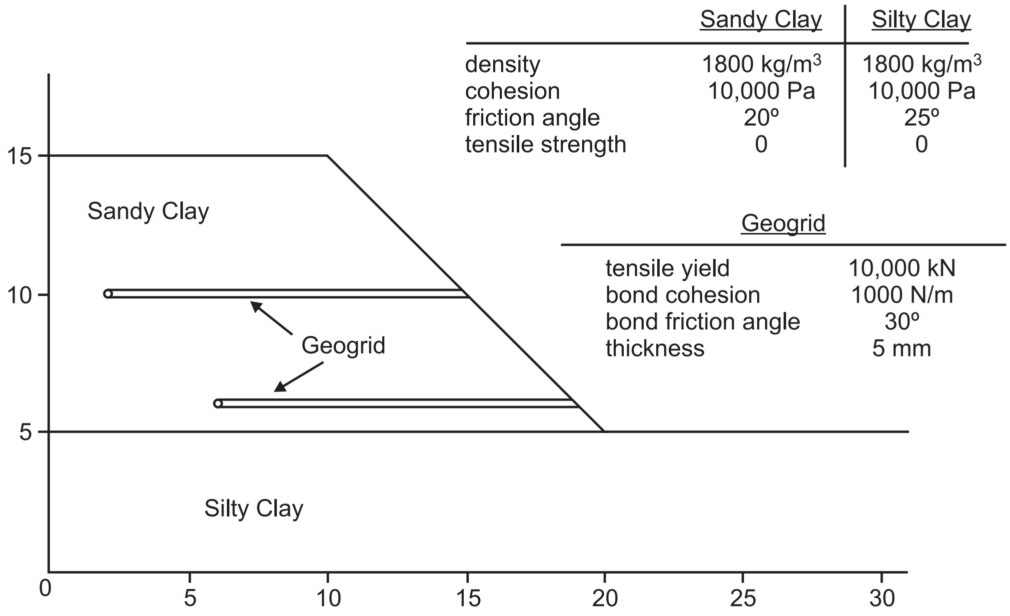


Figure 1.71 Failure plot for c_l / c = 1.0

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 1.72. The project save file is “GEOGRID.PSL.”



	Sandy Clay	Silty Clay
density	1800 kg/m^3	1800 kg/m^3
cohesion	10,000 Pa	10,000 Pa
friction angle	20º	25º
tensile strength	0	0

Geogrid	
tensile yield	10,000 kN
bond cohesion	1000 N/m
bond friction angle	30º
thickness	5 mm

Figure 1.72 Slope with geogrid reinforcement

The slope is unstable without the geogrid reinforcement. The results for the unsupported case are shown in Figure 1.73. The factor of safety is calculated to be 0.96.

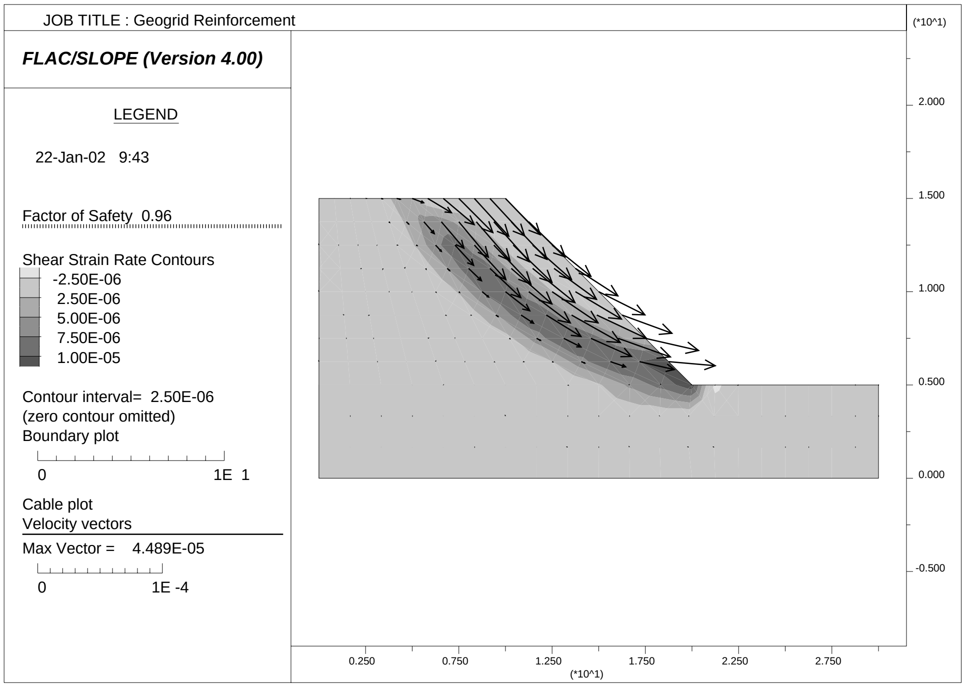
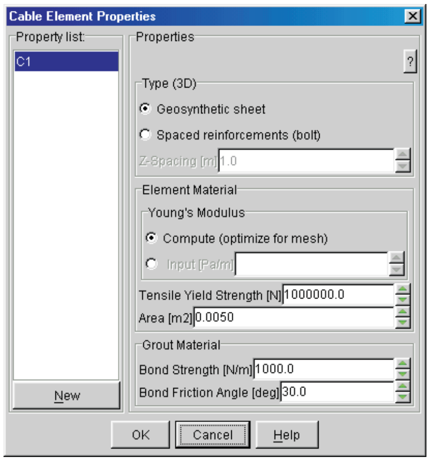


Figure 1.73 Factor-of-safety results for unsupported slope

The properties selected for the geogrid reinforcement are assigned in the *Cable Element Properties* dialog as shown in Figure 1.74.



Property list: C1

Properties

Type (3D)
● Geosynthetic sheet
○ Spaced reinforcements (bolt)
Z-Spacing [m] 1.0

Element Material

Young's Modulus
● Compute (optimize for mesh)
○ Input [Pa/m]

Tensile Yield Strength [N] 1000000.0
Area [m2] 0.0050

Grout Material
Bond Strength [N/m] 1000.0
Bond Friction Angle [deg] 30.0

Figure 1.74 Geogrid properties specified in Cable Element Properties dialog

The factor-of-safety calculation is run for this support in *Model 2*. The results are shown in Figure 1.75. The safety factor is now increased to 1.08.

The effect of the bonding resistance provided at the geogrid/soil interface can be seen when we increase the bond cohesion to 10,000 N/m. For this case, (*Model 3*) the calculated factor of safety is now 1.25 as shown in Figure 1.76.

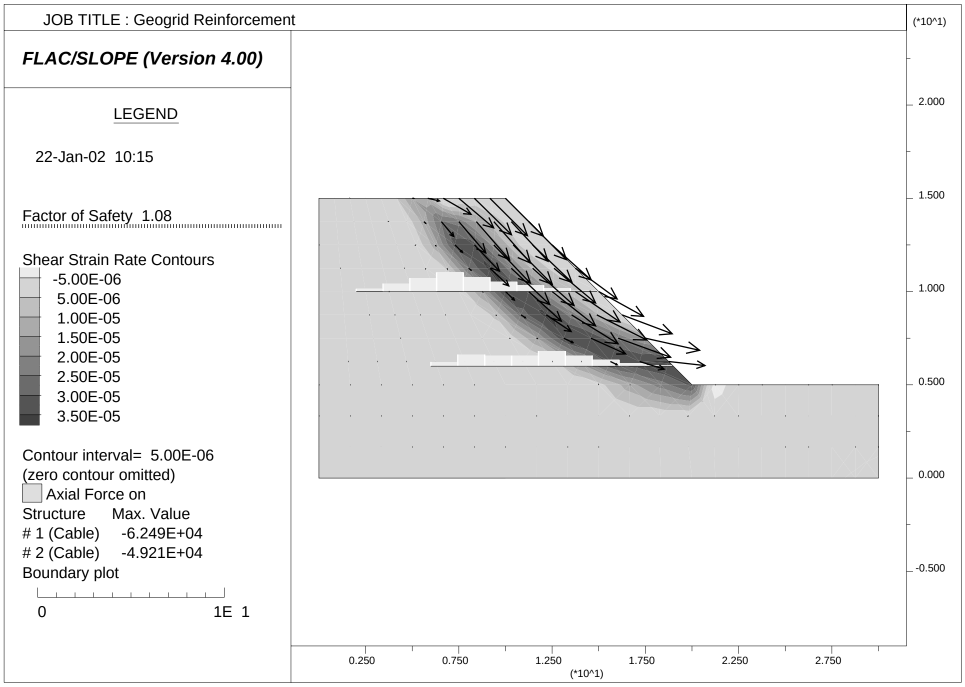


Figure 1.75 Factor-of-safety results for geogrid support with bond cohesion = 1000 N/m

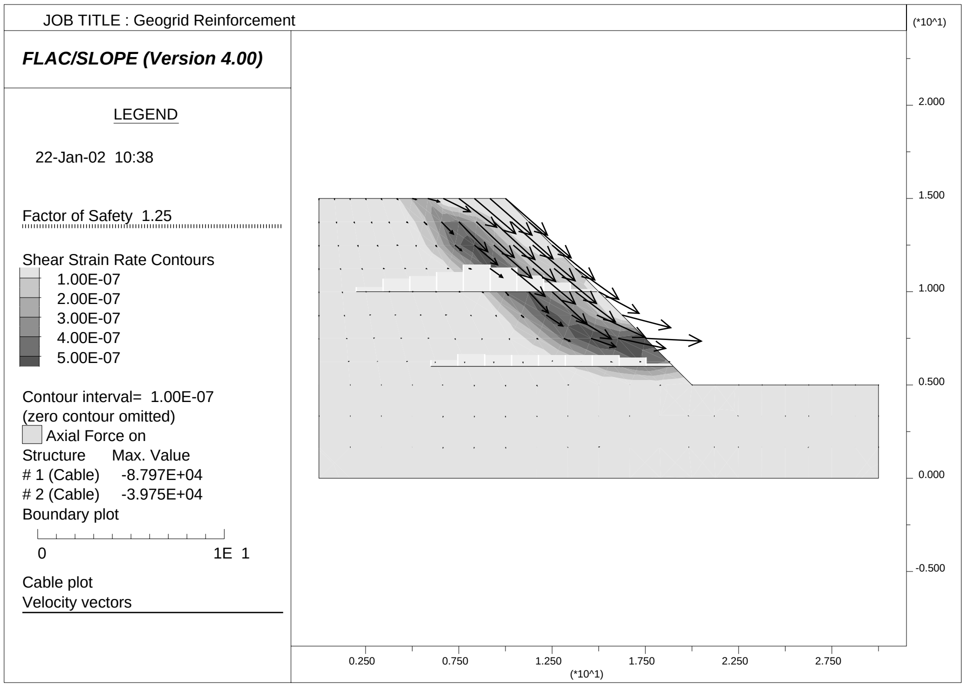


Figure 1.76 Factor-of-safety results for geogrid support with bond cohesion = 10,000 N/m

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 1.77 illustrates the geometry of the slope. This example is taken from Hoek and Bray (1981).

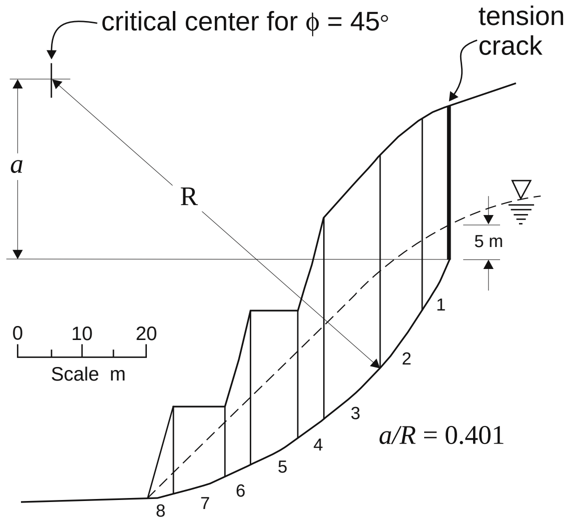


Figure 1.77 Failure surface solution from Bishop's method for a rock slope [Hoek and Bray (1981)]

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 1.77, 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 1.77, 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. The model also contains a water table at the position shown in Figure 1.77. The project save file for this example is "BENCH.PSL."

Figure 1.78 displays the failure plot for this model. The calculated factor of safety is 1.32. 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 a result of tensile softening (as identified from a plot of plasticity indicators). This progressive failure cannot be identified in a limit-equilibrium solution.

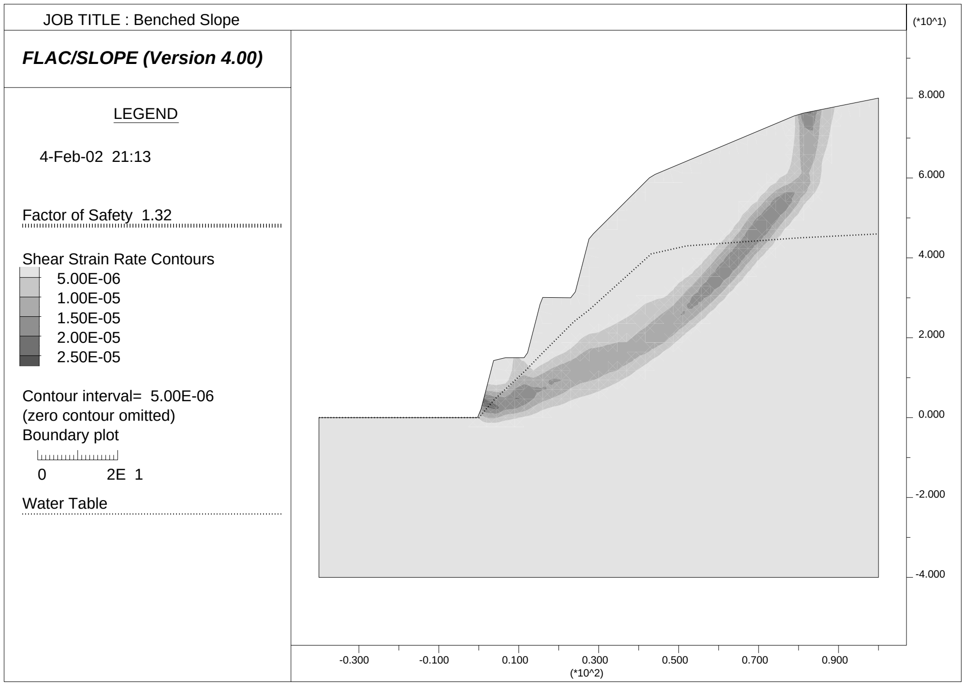


Figure 1.78 Failure plot for rock slope with benches

1.4.6 Slope Failure due to Surcharge Loads in Weightless Material

Consider a long slope at angle θ to the horizontal in uniform, weightless, Mohr-Coulomb material, and assume that there is a *linear failure surface* due to a uniform surcharge pressure, P, acting on a length, L, at an angle of α to the horizontal, as shown in Figure 1.79. At yield,

$$F_s = F_n \tan\phi + cD \tag{1.5}$$

where F_s and F_n are the shear and normal forces, respectively, acting on the slip surface, ϕ is the friction angle and c is the cohesion.

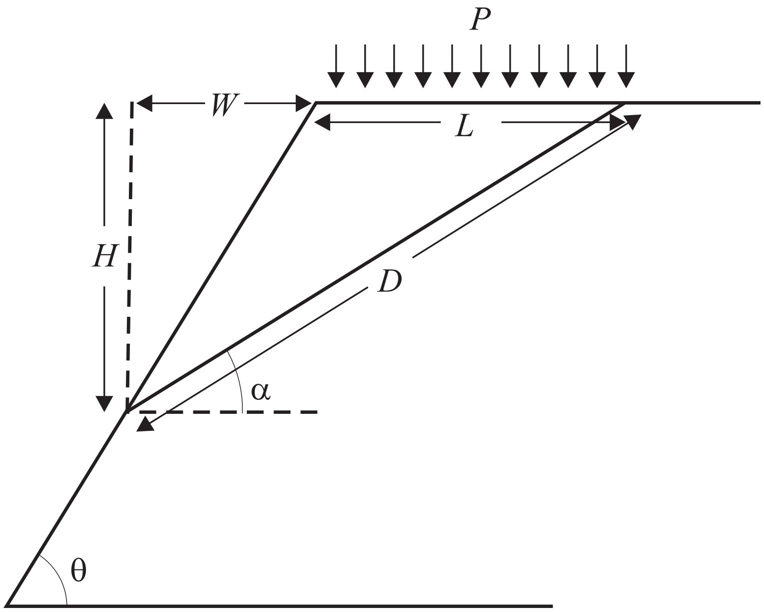


Figure 1.79 Slope with surcharge load

The total force acting on the slip surface is $F = PL$, and this is resolved into two components, $F_n = F\cos\alpha$ and $F_s = F\sin\alpha$. Substituting in Eq. (1.5),

$$PL(\sin\alpha - \cos\alpha\tan\phi) = cD \tag{1.6}$$

Noting that $H = W\tan\theta$ and $H = (L + W)\tan\alpha$, we obtain

$$D = \frac{L\tan\theta}{\tan\theta\cos\alpha - \sin\alpha} \tag{1.7}$$

Substituting in Eq. (1.6),

$$P = \frac{c \tan\theta}{(\sin\alpha - \cos\alpha\tan\phi)(\tan\theta\cos\alpha - \sin\alpha)} \tag{1.8}$$

Differentiating Eq. (1.8), and equating to zero, we obtain an expression for $\alpha = \alpha_m$ that minimizes the applied pressure, P:

$$\tan 2\alpha_m = \frac{\tan\theta + \tan\phi}{1 - \tan\theta\tan\phi} \tag{1.9}$$

Noting that $\tan(a+b) = (\tan a + \tan b)/(1 - \tan a \tan b)$, one solution of Eq. (1.9) is therefore:

$$\alpha_m = \frac{\theta + \phi}{2} \tag{1.10}$$

For given values of θ and ϕ, the "critical" value of α is found from Eq. (1.10), and then substituted into Eq. (1.8) to obtain the failure load.

This approach assumes that sliding on a single surface is the most critical mechanism. This is expected to be the case for $\theta = 90°$ (a vertical slope), but for lower slope angles, there is likely to be a more complicated mechanism — for example, sliding on two planes — that gives a lower factor of safety. Thus, the solution derived above should *only be used for nearly vertical slopes*. In the case of $\theta = 90°$, Eq. (1.8) degenerates to

$$P_{90} = \frac{c}{(\sin\alpha - \cos\alpha\tan\phi)\cos\alpha} \tag{1.11}$$

For example, for $\phi = 30°$, we obtain $\alpha_m = 60°$ and $P_{90} = 3.46c$.

This example is run in *FLAC/Slope* for the case of a 20 m high vertical slope with a surcharge load of 100,000 Pa applied at the top of the slope over a length of 10 m. The cohesion of the slope material is 100,000 Pa and the friction angle is 30°. (The tensile strength is set to a high value, and gravitational acceleration is set to zero, in order to correspond to the above solution.) The project save file for this example is "SURF.PSL."

The result for this case is shown in Figure 1.80; the calculated factor of safety is 3.46, which matches the solution obtained above. Note that the friction angle is excluded from the factor-of-safety calculation because we wish to determine the collapse load, which is proportional to the cohesion, c (see Eq. (1.8)), which is varied in the safety factor calculation to obtain collapse. The friction angle is held constant, for Eq. (1.8) to apply in this case. The failure mode is fairly well-defined in Figure 1.80, by the shear strain contours and velocity vectors, to be approximately 60°.

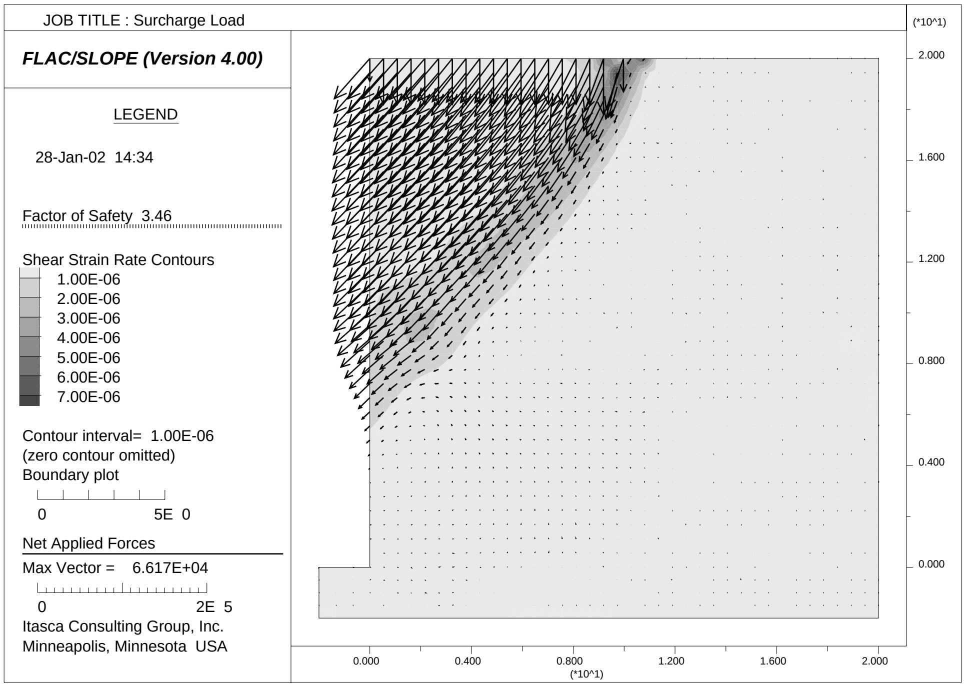


Figure 1.80 Factor-of-safety result for slope with surcharge load

For a slope with weight, the solution may be expected to lie partway between the weightless solution (derived above) and the zero-surcharge solution, which may involve a slip surface that intersects the toe of the slope. As the surcharge increases (in proportion to the weight of the sliding mass), the critical slip surface can be expected to migrate from the toe of the slope to partway up the slope face.

Note that the solution above is of the Limit Equilibrium type. Therefore, there is no assurance that it bounds the true solution.

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:

$$c^{trial} = \frac{1}{F^{trial}}\, c \tag{1.12}$$

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

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. In *FLAC/Slope*, a bracketing approach is used similar to that proposed by Dawson, Roth and Drescher (1999). The procedure in *FLAC/Slope* is as follows. First the code finds a "representative number of steps" (denoted by N_r), which characterizes the response time of the system. N_r is found by setting the cohesion to a large value, 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.

---

* 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 in Section 3.8 in the **User's Guide** volume of the full *FLAC* manual for more information.

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. Billaux, 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.