

2 Slope Stability Analysis

This example addresses a common problem in soil mechanics: the stability of a soil slope. For this tutorial, we begin with a stable excavated slope, and then raise the water table, which causes slope failure to occur. The example is similar to that in [Section 1](#) in the **Examples volume** of the *FLAC* 4.0 manual, in which a groundwater flow calculation is performed first to raise the phreatic surface, and then an effective-stress calculation is performed to analyze the mechanical stability.

The model is created using *FLAC*'s graphical interface, the *GIIC*. We will perform this analysis using the groundwater flow mode (**CONFIG gw**), so we first select **GWFLOW** in the *Model Options* dialog box. We also select automatic adjustment of total stresses (**CONFIG ats**) by checking the **ADJUST TOT. STRESS** box because we will change the pore pressure boundary condition to simulate raising of the water table. We will perform this analysis using the SI system of units and record the data in the *Project Tree* record mode. (We check the **TREE** radio button in the *Project Record Format*.) We press to activate all of these options. Remember that these options can only be activated before the model is created. (Select **FILE/NEW PROJECT** to start a new model.)

We now set up a project save file by selecting the **FILE/SAVE PROJECT AS** menu item in the main menu. We select a directory in which to store files for this analysis and save the project as "SLOPE.PRJ." We are now ready to begin creating the model.

We generate the grid for the slope by pressing the modeling-tool tab and then click on the button in the toolbar. This tool allows us to divide our grid into separate regions (or blocks) of grid; each block can then be given different grid dimensions. We press in the dialog to select the default grid (two blocks by two blocks). An *Edit Block Grid* dialog will now appear (as shown in [Figure 2.1](#)), in which we can input the *x*- and *y*-coordinates, the number of zones and the grid ratios for each block. The parameters to define a coarsely zoned model of the slope are shown in [Figure 2.1](#).

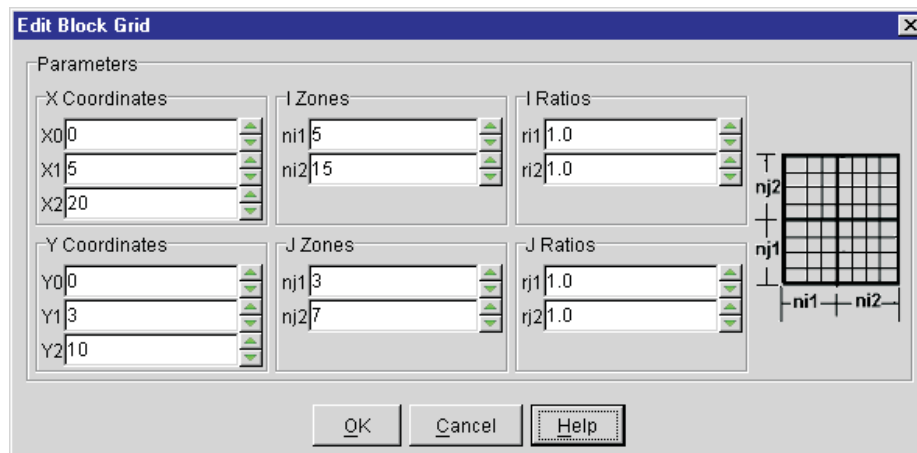


Figure 2.1 *Edit Block Grid dialog box*

After we press , a plot of the grid will appear along with a set of tools that we can use to manipulate the grid. Note that the grid has not yet been created by *FLAC*. It is a “virtual” grid that can be distorted and shaped in any manner we choose. Once we are satisfied, we can then create the *FLAC* grid by pressing the button. [Figure 2.2](#) shows the virtual grid with the block radio button active. Block corners are denoted by red squares. To create the slope, we move one of the block corners (Vertex #7). This is the middle block corner along the top boundary of the grid. (The vertex number is displayed when we right-click on a corner.) We can move the corner either by dragging it with the mouse, or by pressing the right mouse button, opening a dialog to type in the new coordinates. Here we move Vertex #7 to the position $x = 9.0$, $y = 10.0$.

We now wish to create a finer mesh within the slope block. Click the radio button. A red box will appear in the center of each block. We can drag the box in the slope block (the upper-right block in the grid) with the mouse to increase the mesh density in this block. The number of zones will be reported in the status bar. Alternatively, we can specify the number of zones by clicking the right mouse button to open a coordinate dialog. [Figure 2.3](#) shows the grid with the slope block increased to 30×20 zones. Likewise, we increase the bottom-right block to 30×8 zones, and the bottom-left block to 10×8 zones. Next, we click on the radio button and then click on the upper-left grid block to hide this block and prevent zones from being created.

We press to create the *FLAC* grid. Several commands will scroll by in the *Console* pane, and the *FLAC* grid will appear in the *Model-view* pane. The result is shown in [Figure 2.4](#). The commands created up to this point are listed in the *Record* pane, as shown in the figure. Note that we show the x - and y -axes for the model by clicking on the `VIEW/SHOWAXIS VALUES` menu item in the main menu.

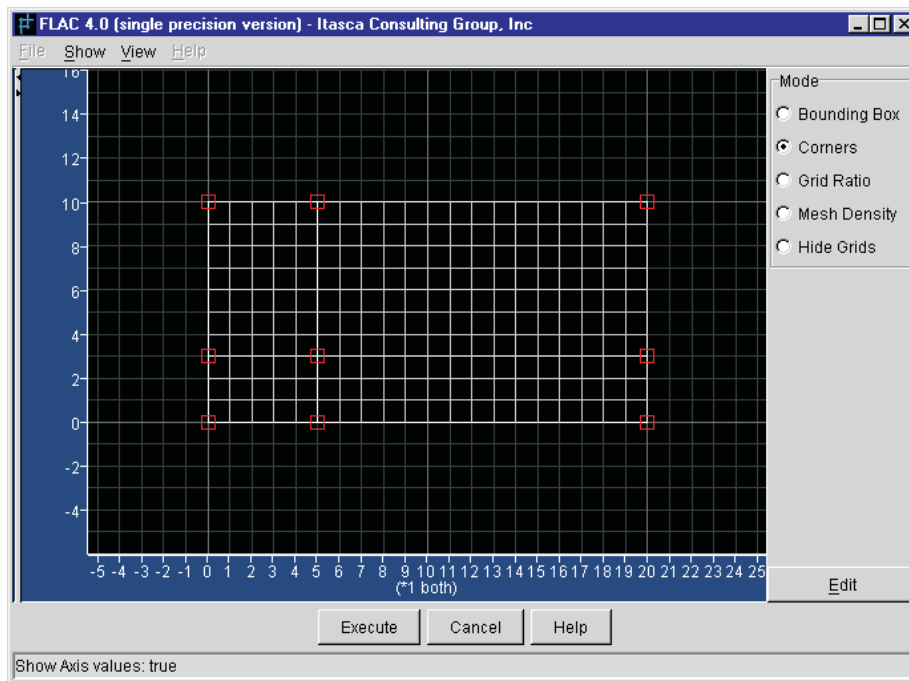


Figure 2.2 *GHIC virtual grid with button active*

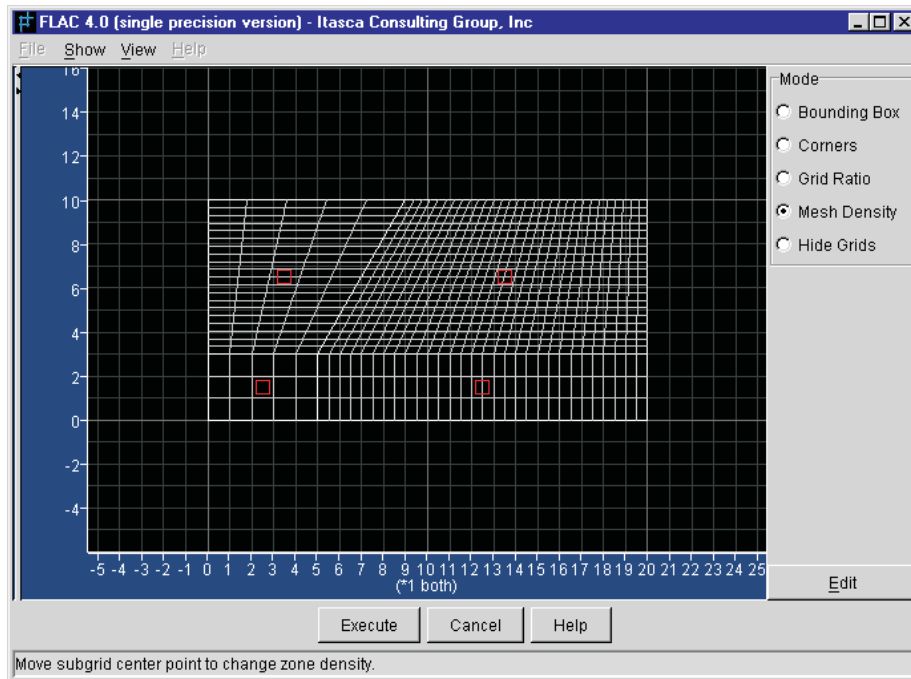


Figure 2.3 **MESH DENSITY** tool creates finer mesh in slope block

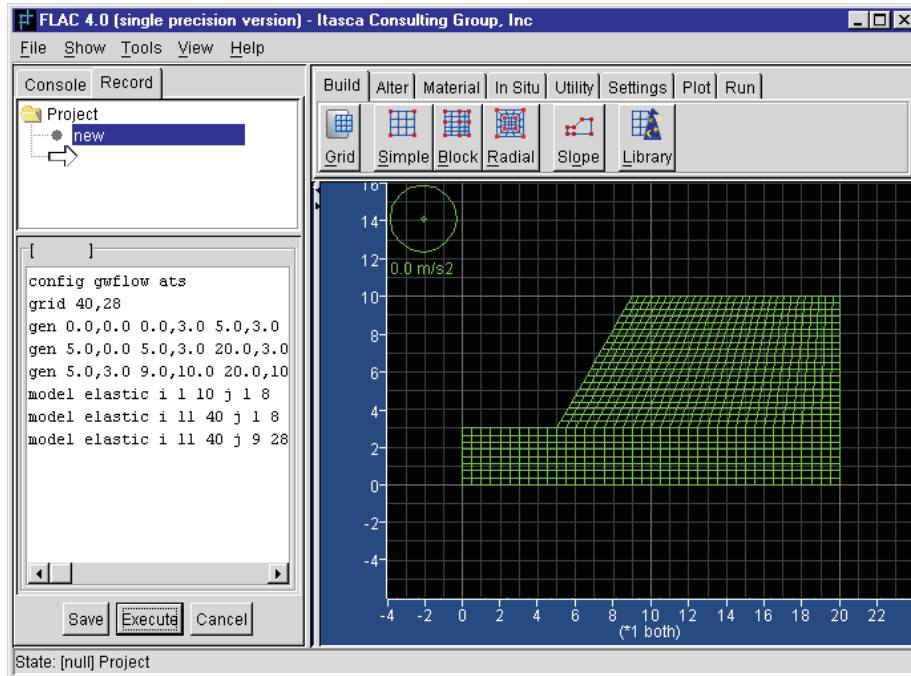


Figure 2.4 **FLAC grid for slope example**

We skip the **ALTER** tab in the modeling-stage tab bar because we do not need to make any further alterations to the grid.

We assign a material model and properties via the **ASSIGN** button from the **MATERIAL** tab toolbar. First we create a material by pressing the **CREATE** button to open a *Material* dialog. [Figure 2.5](#) shows the dialog box with the material we have selected for this example (*silty clay*) and its associated properties. We press **OK** to create this material. The material is then entered in the material list shown in the *Assign* pane.

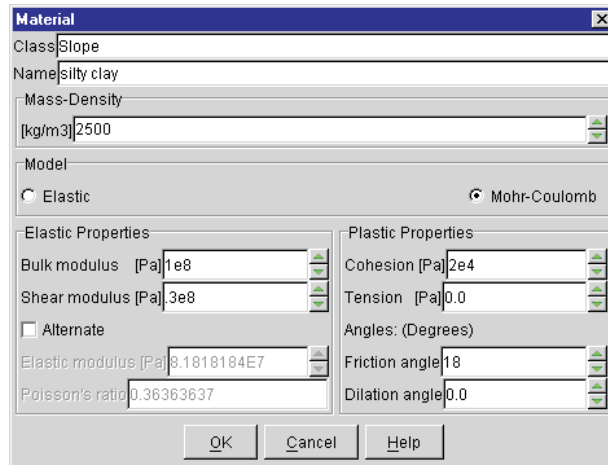


Figure 2.5 *Material properties dialog in the **ASSIGN** tool*

Next, we assign the material to zones in the grid. It is possible to assign different materials and properties to different zones or to different marked regions of the grid using this tool. In our example, all the zones are assigned the same material with uniform properties. In this case, we only need to highlight the *Slope: silty clay* material in the list and then press the **SETALL** button. This assigns the material to all the non-null zones in the grid. The corresponding **GROUP**, **MODEL** and **PROPERTY** commands are listed in the *Changes* sub-pane when we press **SETALL**. The assigned material is indicated in [Figure 2.6](#). We then press **EXECUTE** to send these commands to *FLAC*.

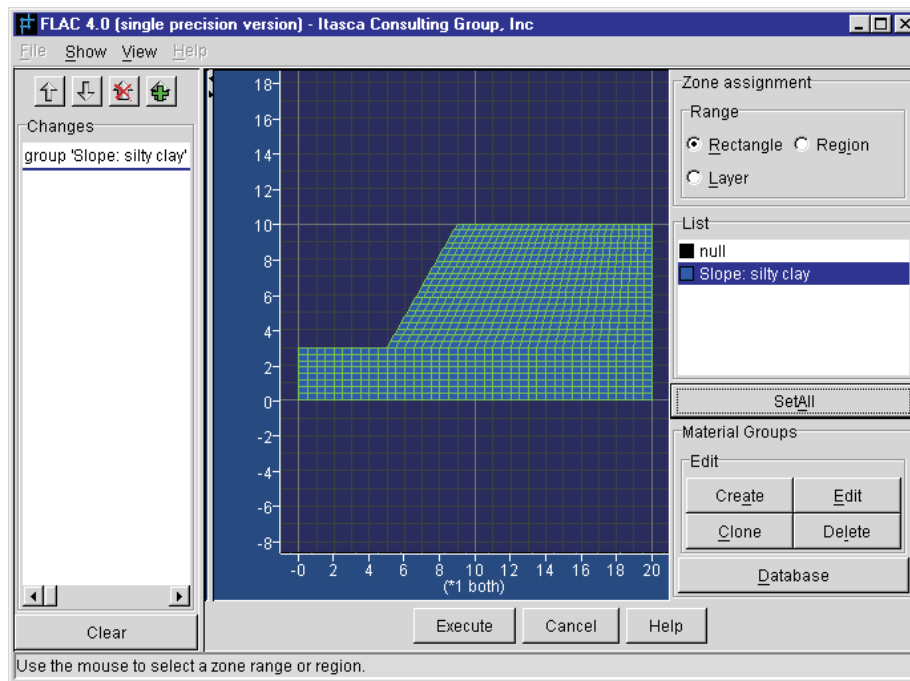


Figure 2.6 *Material assigned in the Assign pane*

The boundary conditions for our model are specified via the `IN SITU` tab. We press the `FIX` button and can now use the mouse to assign the fixity condition for the boundaries. By default, the radio button for `BOTH X&Y` fixed gridpoint velocity is active. We hold down the left mouse button and drag the mouse along the bottom boundary of the grid. Gridpoints are marked and, when we release the button, a letter denoting the fixity condition is printed at the selected gridpoints. In addition, the corresponding `FIX` command is listed in the *Changes* sub-menu. We then click on the `X` fixed gridpoint radio button and drag the mouse along the left and right boundaries of the model. The results are shown in [Figure 2.7](#). Press `EXECUTE`, and the `FIX` commands are sent to *FLAC*.

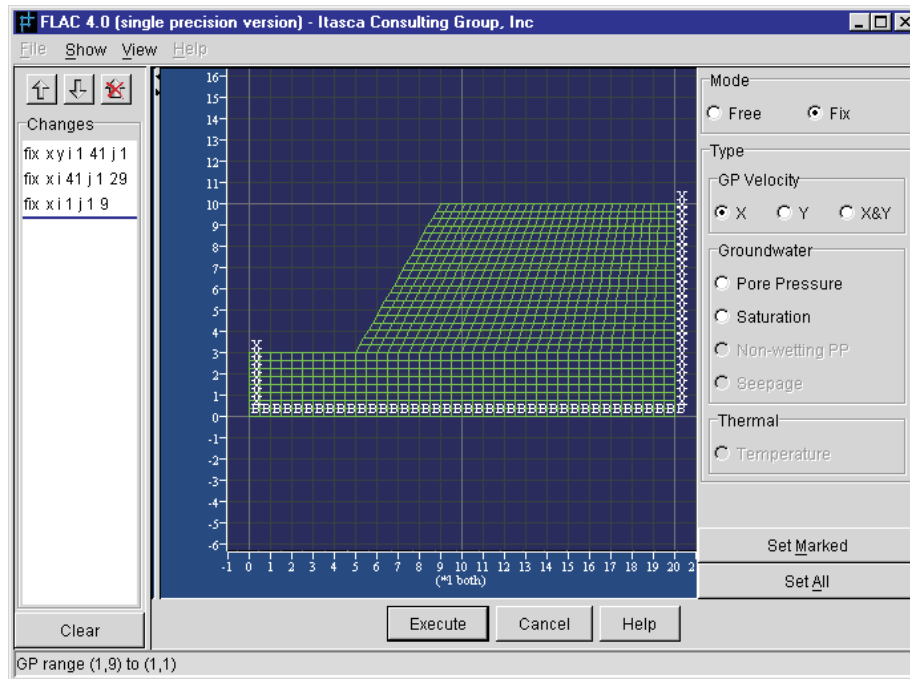


Figure 2.7 **FIX** tool to set boundary conditions

The next modeling-stage tool, accessed from the **UTILITY** tab, allows us to monitor variables in our model during the calculation. We click on the **HISTORY** button to open the *History* pane, and then click on the **GP** mode radio button. We select the *x*-displacement history from the *History Information* sub-menu. We then point at the gridpoint we wish to monitor. When we click on the gridpoint, a **HISTORY** command is created to monitor the *x*-displacement of the selected gridpoint. Figure 2.8 shows the **HISTORY** pane. Press **EXECUTE** and the **HISTORY** command is sent to *FLAC*.

Global conditions for the model are specified through the **SETTINGS** tab. We set the gravitational acceleration by pressing the **GRAVITY** button, and a *Gravity Settings* dialog box opens. We specify gravity as 10.0 m/sec² to simplify this example. (If we press the globe icon, the value of 9.81 m/sec², assuming SI units, will be selected.)

We also wish to run the model in large-strain mode, so we set this mechanical condition by pressing the **MECH** button from the **SETTINGS** tab, which opens a *Mechanical Settings* dialog. We click on the **LARGE-STRAIN** radio button. Press **EXECUTE** to send this command to *FLAC*.

We are running this analysis in the groundwater-flow mode; however, for the initial compaction stage, we turn fluid flow off. Press the **GW** button to open the *Groundwater Flow Settings* dialog box. Uncheck the **GROUNDWATER FLOW CALCULATION** item, and press **EXECUTE** to send the **SET flow off** command to *FLAC*. The *Groundwater Flow Settings* dialog box is shown in Figure 2.9.

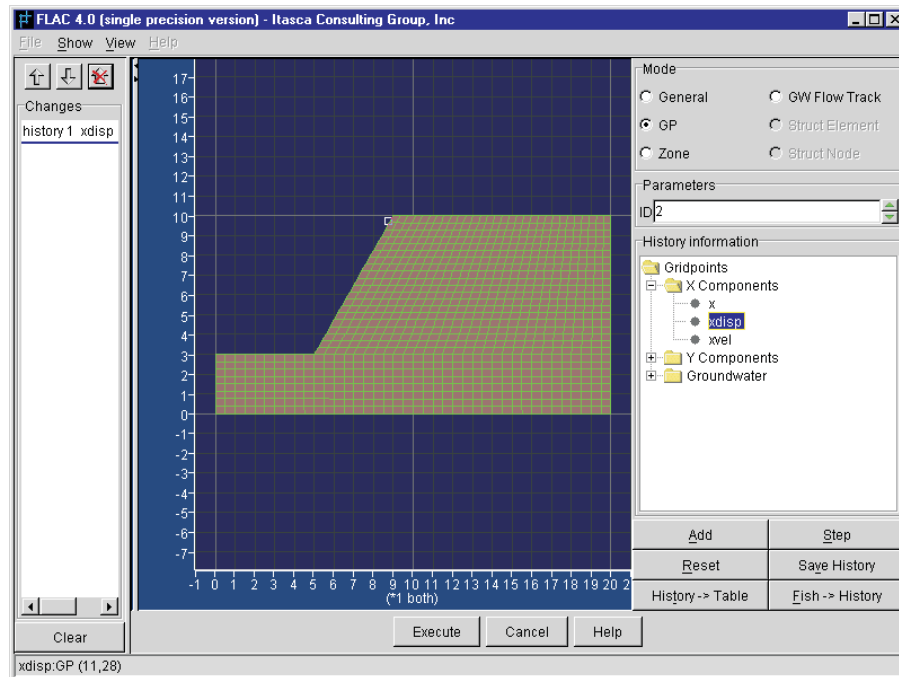


Figure 2.8 **HIST** tool to select model variables to monitor

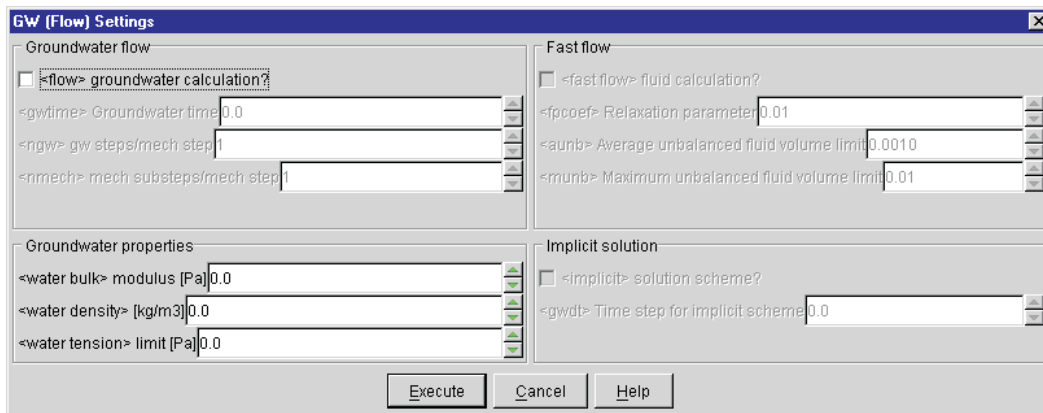


Figure 2.9 Groundwater Flow Settings dialog box

The **PLOT** tab gives us access to all plotting facilities in *FLAC*. We can use the **MODEL** button to open a *FLAC Plot items* pane that accesses tools to create a new plot view containing various plot items. When executed, this will send a **PLOT** command to *FLAC* that includes keywords that correspond to the selected plot items. We will use this tool later in our analysis. There is also a **QUICK** button that allows us to save plot views that we may wish to view often. For example, the grid plot and unbalanced force history plot are listed in the **QUICK** menu by default; if we click on **GRID**, a grid plot-view will be created.

We can make a hardcopy plot of the grid by clicking on the FILE/PRINT PLOT menu item in the main menu. If the current Windows default printer is connected to the LPT1 port, we can send the plot directly to the printer by clicking this menu item. The FILE/PRINT PLOT SETUP menu item can be used to change the printer device settings. [Figure 2.10](#) shows the grid plot-view and the highlighted FILE/PRINT PLOT menu item.

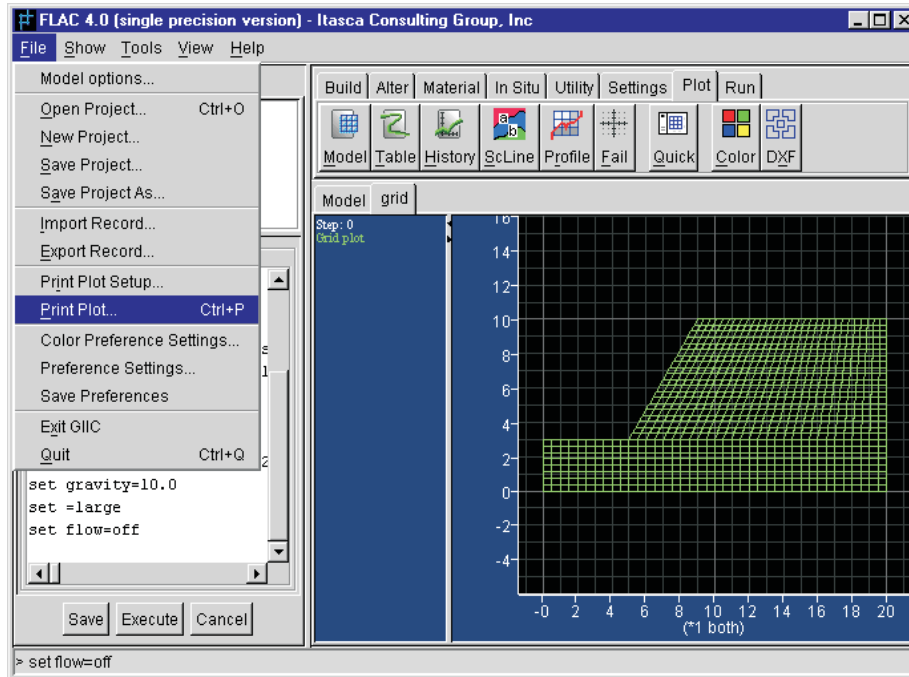


Figure 2.10 PLOT/PRINT menu to create a hardcopy plot

We are now ready to perform the initial calculation to bring the model to a static equilibrium state under gravitational loading. We click the **RUN** tab and press the **SOLVE** button to solve for the static solution. A *Solve options* dialog box will open, showing that we have selected the mechanical calculation mode. The dialog is shown in [Figure 2.11](#). We select the **SOLVE INITIAL EQUILIBRIUM AS ELASTIC MODEL** box; this invokes the **SOLVE elastic** command, which will perform the initial equilibrium calculation in two steps. First, the cohesion and tensile strengths of the material will be adjusted to high values in order to bring the model to elastic equilibrium, and then the strengths will be reset to their actual values and the solution continued to reach the final equilibrium state. (This approach to develop the initial stress state is discussed in [Section 2.2.1.2](#) in the **User's Guide**.)

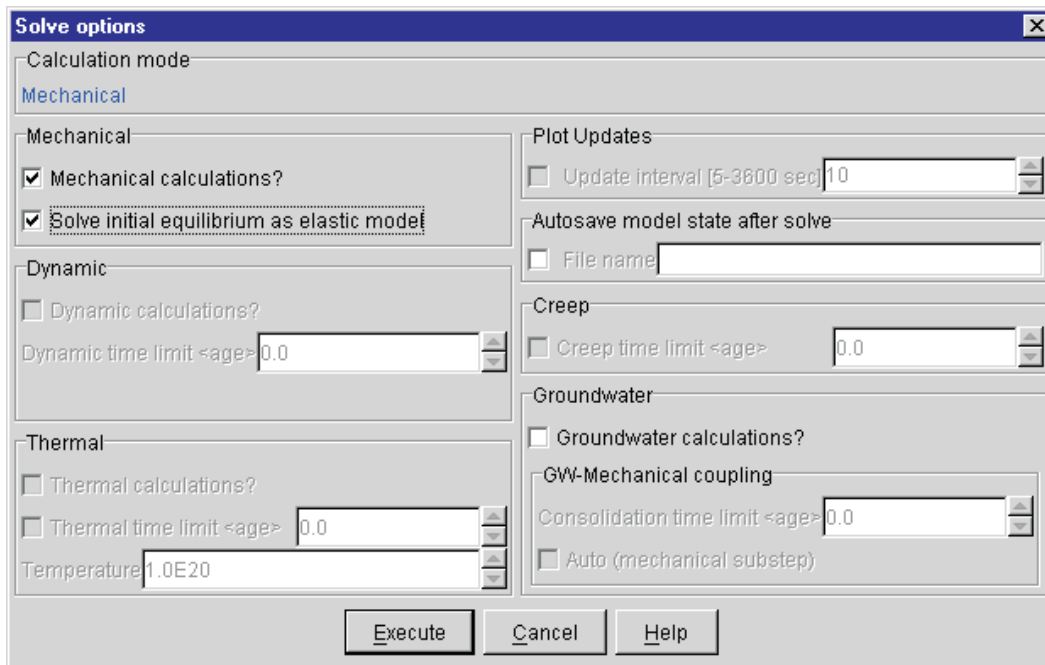


Figure 2.11 Solve options dialog

When we press **EXECUTE**, a *FLAC cycling...* dialog box will appear with calculation step number, maximum unbalanced force and equilibrium ratio values updated in the box while the calculation is processing. (See [Section 2.6.4](#) of the *FLAC 4.0 User's Guide* for an explanation of these solution values.) The dialog will close when the static solution is reached. We check the maximum unbalanced force by clicking on the **QUICK** button from the **PLOT** tab. Click on the **UNBALANCED FORCE** item to produce the plot view, as shown in [Figure 2.12](#). The maximum unbalanced force is approaching zero, indicating that a stable equilibrium state is reached. Note that the sudden jump in maximum unbalanced force during cycling, as indicated on this plot, occurs as a result of the adjustment in cohesion and tensile strength from high values to actual values during the **SOLVE elastic** solution.

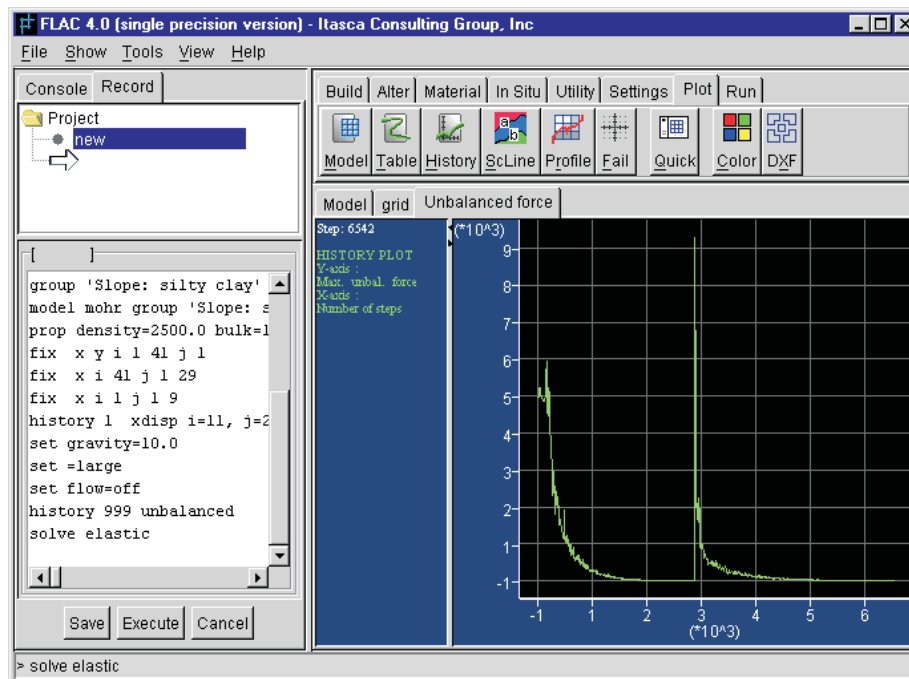


Figure 2.12 Unbalanced force plot view from the **QUICK** button

We can save the model state at this stage via the **SAVESTATE** button in the **RUN** toolbar. A *Save File (model state)* dialog box will open, and we can select a title for the saved state. Note that the default extension is “.SAV.” We save this file as “SLOPE1.SAV.”; the file name will be shown in the *Project Tree* record pane. This state can be restored at any time either with the **RESTORESTATE** button in the **RUN** toolbar, or by clicking on the file name in the project tree.

At this stage we wish to raise the water level in the slope to evaluate this effect on stability. There are several ways in which this can be done. (See [Section 1.7.2](#) in **Fluid-Mechanical Interaction** of the *FLAC 4.0* manual for a discussion on different ways in which coupled hydromechanical analyses can be performed.) We are only interested in the steady-state response, so we can uncouple the analysis and perform a flow-only calculation first to find the phreatic-surface location, then perform a mechanical-only calculation to investigate the effect on slope stability. We press the **SETTINGS** tab and the **MECH** button, and then turn off the **PERFORM MECHANICAL CALCULATIONS?** selection. Using the **GW** button to open the *Groundwater flow settings* dialog, we turn on the **GROUNDWATER FLOW CALCULATION** selection.

Also, groundwater properties must be specified. We select a water bulk modulus of $1e4$ Pa and a water density of 1000 kg/m³. We specify a low value for water bulk modulus to speed convergence to the steady state solution because we are not interested in the transient behavior. (Note that there is a lower limit for the water bulk modulus to satisfy numerical stability — see [Section 1.10.4.2](#) in **Fluid-Mechanical Interaction**.)

We must also specify the grid material properties related to groundwater flow. We click on the **GWPROP** button from the **MATERIAL** toolbar, then click on the **SETALL** button. This opens the *Model*

Groundwater properties dialog box. We set the porosity to 0.3 and the isotropic permeability to 10^{-10} (m/s)/(Pa/m) and click . We press to send these properties to *FLAC*.

We now apply pore pressure boundary conditions to raise the water level to 3 m at the left boundary and 10 m at the right. We press the tab to access the tool. We select *GROUNDWATER/PP* from the *B.C. types* sub-menu, then point at the bottom-right corner of the model and drag the mouse upwards along the right boundary while pressing the left mouse button. The boundary will be marked by a red line indicating the portion of the boundary along which this boundary condition is applied. Now, we click the button and a *Apply value* dialog box opens. [Figure 2.13](#) shows the pane and the *Apply value* dialog box. We enter the values shown in the dialog to apply a pore pressure variation along this boundary. We calculate pore pressure variation using [Eq. \(3.2\)](#) in the **User's Guide**. Note that pore pressure is positive in compression, so for water density = 1000 kg/m³ and gravity = 10 m/sec², pore pressure, p , varies in the y -direction, as given by

$$p = 100,000 - 100,000y \quad (2.1)$$

along the right boundary. An arrow will be displayed along the boundary indicating the direction of the increasing magnitude for this pore pressure variation. If we enter the direction incorrectly, we can start over by pressing the button. When we press in the dialog box, the corresponding **APPLY** command is entered in the *Changes* sub-menu. We follow this same procedure to apply the pore pressure variation along the left boundary. We also apply a zero pore pressure, to allow flow, along the slope (from gridpoint $i = 1, j = 9$ to gridpoint $i = 11, j = 29$). Press to send these commands to *FLAC*. The resulting **APPLY** commands and the marked boundaries are shown in [Figure 2.14](#). The boundary conditions will be displayed on the *Model*-view pane.

You will note in [Figure 2.14](#) that only the commands created after the model was saved as "SLOPE1.SAV" are recorded in the *Record* pane. Only these commands are associated with this "branch" of the project tree.

We perform a flow-only calculation to find the steady flow state by pressing the button in the toolbar. The groundwater calculation mode is now active as indicated in the *Solve options* dialog. Note that flow time is reported in the *FLAC cycling...* dialog box; however, this time is not meaningful in this case because the diffusivity of the system was changed to speed convergence to the steady state.

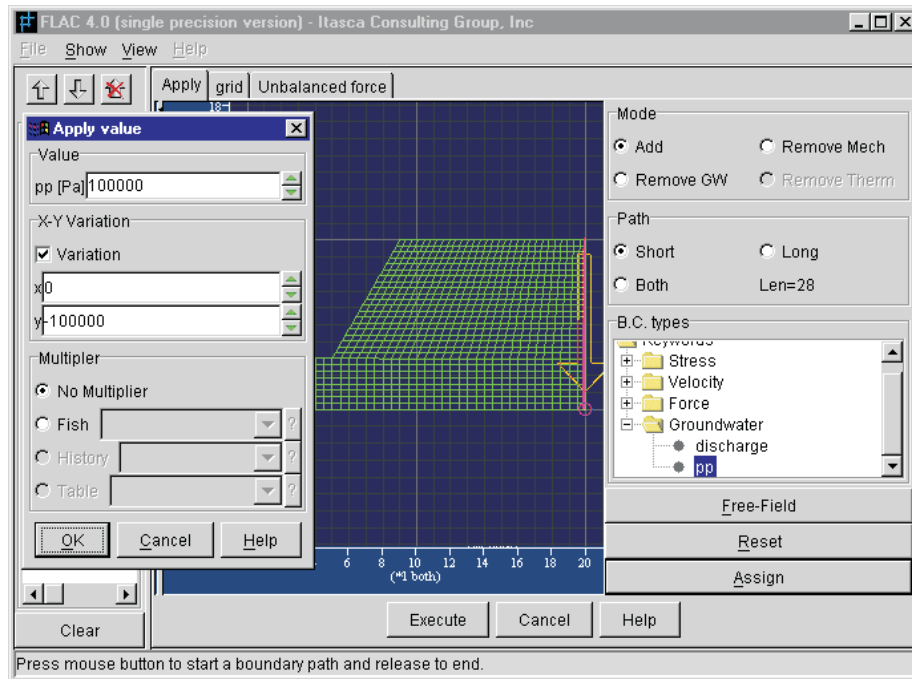


Figure 2.13 **APPLY** pane with the *Apply value* dialog box to specify the pore pressure boundary condition for the right boundary

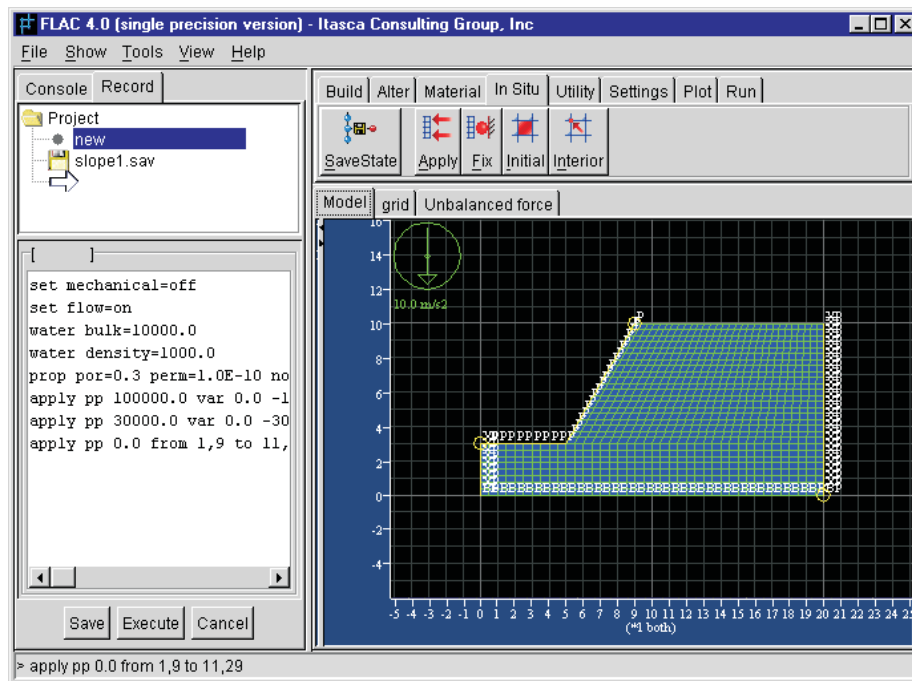


Figure 2.14 **APPLY** commands created for the specified boundary conditions

When the calculation stops, we view the steady flow state by creating a plot of pore pressure contours and flow vectors. We click on the **MODEL** button in the **PLOT** toolbar. We select the **GROUNDWATER/PP** plot item and the **GROUNDWATER/FLOW** plot item from the *Plot Items* tree and add these to the *Add Plot Items* list. Note that the plot items can be edited by pressing the **EDIT** button. For example, we can click on the **ZERO** radio button to suppress the zero contour. Press **OK** to create the plot view. [Figure 2.15](#) shows the resulting plot which displays the steady-state flow profile. (We turn off the resources views from the **SHOW/RESOURCES** menu item and the modeling-stage tab bar from the **SHOW/TOOLS** menu item in order to make the plot view full-screen in [Figure 2.15](#).)

We save the model state at this stage as “SLOPE2.SAV.”

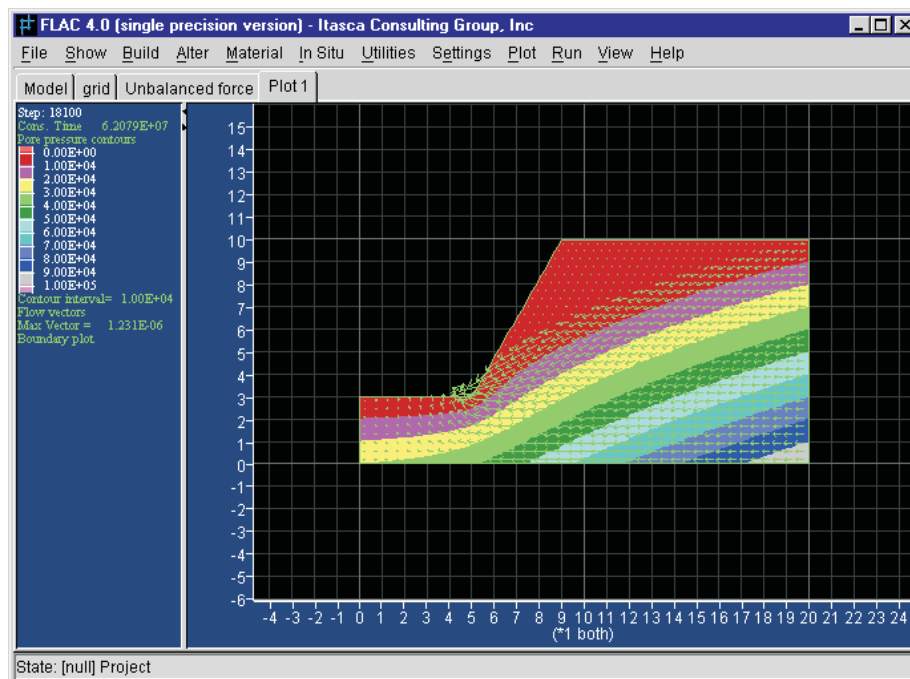


Figure 2.15 Steady-state flow plot view

Now, we run the model in mechanical-only mode by turning on the mechanical calculation from the *Mechanical Settings* dialog, and turning off the flow calculation from the *Groundwater Flow Settings* dialog. Note that we also set the water bulk modulus to zero in this dialog box, because we do not wish to allow pore pressure generation as a result of mechanical deformation.

We do not know if the slope will be stable or not. Therefore, we use the **CYCLE** button in the **RUN** toolbar so that we can monitor the model response as the calculation progresses. In this mode, we can refresh the active plot view at a selected time interval.

We first create a plot view to monitor during cycling: *x*-displacement contours and displacement vectors. We click on the **MODEL** button in the **PLOT** toolbar, select the **CONTOUR-GP/XDISP** plot item and the **VECTOR/DISPLACEMENT** plot item from the *Plot Items* tree and add these to the *Add Plot*

Items list. We edit the x -displacement contour plot, specifying a minimum contour value of -0.44, a maximum contour value of 0.0, and a contour interval of 0.05. By specifying these settings, the fill colors will not change as the plot is refreshed during cycling. We press to create the plot view.

We now press the button in the toolbar, select a duration for stepping of 5000 steps, and select 5 seconds as the update interval to refresh the plot. We press and the currently active plot is refreshed every 5 seconds during the calculation. The resulting plot indicates that a slip surface develops beginning at the slope toe, the slope fails and a well-defined failure wedge is formed. [Figure 2.16](#) shows the plot view when the cycling stops.

We save the model state at this stage as “SLOPE3.SAV.” We also update the project tree by pressing the FILE/SAVE PROJECT menu item. The entire project can now be restored at a later time via the FILE/OPEN PROJECT menu item.

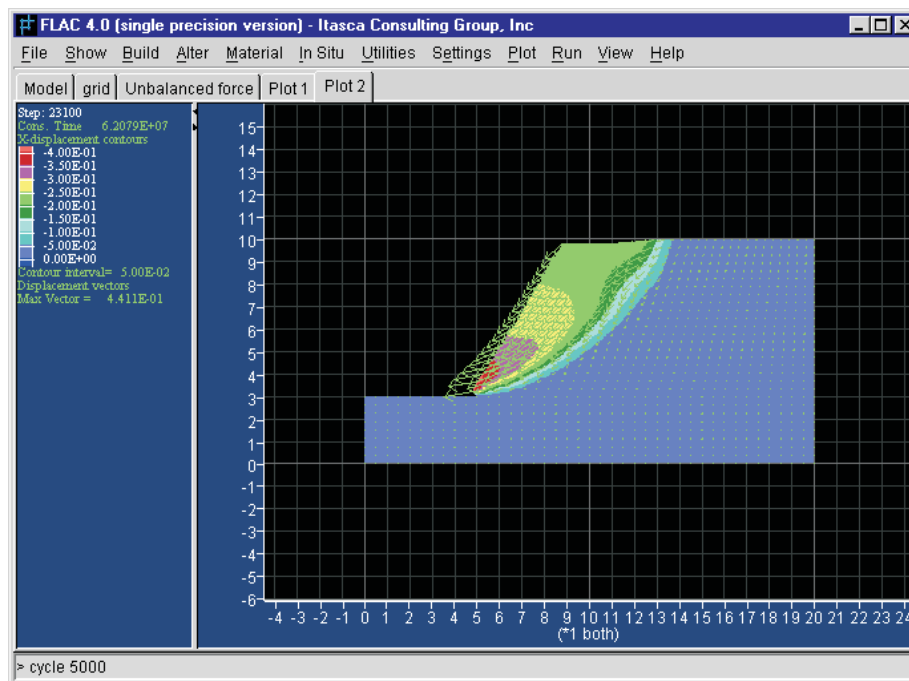


Figure 2.16 Displacement vectors and x -displacement contours in failed slope

We recommend that you now try variations of this example to become more familiar with the *GIIC* operation. For example, begin with the “SLOPE1.SAV” model state and raise the water table in stages (using the tool) to determine the water level at which slope failure occurs. Try adding structural support (e.g., soil nails) to stabilize the slope. You can add structural elements via the *Model Options* dialog after restoring a model state (e.g., “SLOPE2.SAV”) and then perform the mechanical-only calculation to study the effect of support on stability.

We can save the *FLAC* commands we generated for this model by pressing the FILE/EXPORT RECORD menu item. We save the record as “SLOPE.DAT”; the file is listed in [Example 2.1](#).

Example 2.1 “SLOPE.DAT”

```
;Project Record save
new
;Branch 1: slopel.sav
config gwflow ats
grid 40,28
gen 0.0,0.0 0.0,3.0 5.0,3.0 5.0,0.0 i 1 11 j 1 9
gen 5.0,0.0 5.0,3.0 20.0,3.0 20.0,0.0 i 11 41 j 1 9
gen 5.0,3.0 9.0,10.0 20.0,10.0 20.0,3.0 i 11 41 j 9 29
model elastic i 1 10 j 1 8
model elastic i 11 40 j 1 8
model elastic i 11 40 j 9 28
group 'Slope: silty clay' notnull
model mohr group 'Slope: silty clay'
prop density=2500.0 bulk=1.00000008E8 shear=3.0E7 cohesion=20000.0 &
friction=18.0 dilation=0.0 tension=0.0 group 'Slope: silty clay'
fix x y i 1 41 j 1
fix x i 41 j 1 29
fix x i 1 j 1 9
history 1 xdisp i=11, j=28
set gravity=10.0
set =large
set flow=off
history 999 unbalanced
solve
save slopel.sav
;Branch 2: slope2.sav
set mechanical=off
set flow=on
water bulk=10000.0
water density=1000.0
prop por=0.3 perm=1.0E-10 notnull
apply pp 100000.0 var 0.0 -100000.0 from 41,1 to 41,29
apply pp 30000.0 var 0.0 -30000.0 from 1,1 to 1,9
apply pp 0.0 from 1,9 to 11,29
solve
save slope2.sav
;Branch 3: slope3.sav
set mechanical=on
set flow=off
water bulk=0.0
cycle 5000
save slope3.sav
```
