Reading Assignment
- Lecture Notes

Other Materials
- FLAC Manual

Homework Assignment #8
1. Complete CVEEN 7330 Modeling Exercise 10a (30 points)
2. Analyze a 30-m high triangular-shaped embankment (50 points)
   a. Assume the 8-m high embankment has 2H:1V slope and a crest of 30m. Note that you do not have to construct the model incrementally.
   b. Properties for embankment (prop density=2080 bulk=419E6 shear=193E6)
   c. Use FLAC's default hysteretic damping model for sand for embankment
   d. Use the Taft_Match.TXT acceleration time history record on the course website for the FLAC analysis.
   e. Provide all inputs and outputs including
      i. Input time history for FLAC analysis
      ii. FLAC model geometry
      iii. Acceleration time history at base of model
      iv. Acceleration time history at crest of model
      v. Shear Stress vs. shear strain at crest of embankment
      vi. FLAC code

Introduction
- Equivalent Linear Method (EQL)
  - QUAD4 and QUAD4M
  - Quake/W
- Nonlinear Finite Element Method
  - Quake/W
  - Plaxis?
- Nonlinear Finite Difference Method
  - FLAC

© Steven F. Bartlett, 2011
Total Vertical Stress from incremental building of model (homogenous case)

Total Vertical Stress from non-incremental building of model (homogenous case)

© Steven F. Bartlett, 2011
Total Vertical Stress from incremental building of model (heterogeneous case)

Total Vertical Stress from non-incremental building of model (heterogeneous case) (not exactly the same as above)

© Steven F. Bartlett, 2011
config dynamic
set dynamic off
grid 21 10
model elastic
prop density=1 bulk=1.33E7 shear=8E7
; note very low density assigned to this layer
;
model null i 1 j 2 10
group 'null' i 1 j 2 10
group delete 'null'
model null i 2 j 3 10
group 'null' i 2 j 3 10
group delete 'null'
model null i 3 j 4 10
group 'null' i 3 j 4 10
group delete 'null'
model null i 4 j 5 10
group 'null' i 4 j 5 10
group delete 'null'
model null i 5 j 6 10
group 'null' i 5 j 6 10
group delete 'null'
model null i 6 j 7 10
group 'null' i 6 j 7 10
group delete 'null'
model null i 7 j 8 10
group 'null' i 7 j 8 10
group delete 'null'
model null i 8 j 9 10
group 'null' i 8 j 9 10
group delete 'null'
model null i 9 j 10
group 'null' i 9 j 10
group delete 'null'
model null i 12 j 10
group 'null' i 12 j 10
group delete 'null'
model null i 13 j 9 10
group 'null' i 13 j 9 10
group delete 'null'
model null i 14 j 8 10
group 'null' i 14 j 8 10
group delete 'null'
model null i 15 j 7 10
group 'null' i 15 j 7 10
group delete 'null'
model null i 16 j 6 10
group 'null' i 16 j 6 10
group delete 'null'
model null i 17 j 5 10
group 'null' i 17 j 5 10
group delete 'null'
model null i 18 j 4 10
group 'null' i 18 j 4 10
group delete 'null'
model null i 19 j 3 10
group 'null' i 19 j 3 10
group delete 'null'
model null i 20 j 2 10
group 'null' i 20 j 2 10
group delete 'null'
model null i 21 j 1
group 'null' i 21 j 1
group delete 'null'
ini x 0.56119156 y 0.5621834 i 1 j 2
ini x 1.5523796 y 1.5533707 i 2 j 3
ini x 2.4985127 y 2.4995043 i 3 j 4
ini x 3.512227 y 3.5132189 i 4 j 5
ini x 4.5484686 y 4.5494604 i 5 j 6
ini x 5.494602 y 5.4955935 i 6 j 7
ini x 6.508317 y 6.5543623 i 7 j 8
ini x 7.49504 y 7.455494 i 8 j 9
ini x 8.50808 y 8.581791 i 9 j 10
ini x 9.54946 y 9.527925 i 10 j 11
ini x 10.518121 y 9.527925 i 11 j 12
ini x 11.576889 y 8.4691561 i 13 j 10
ini x 12.523023 y 7.4779687 i 14 j 9
ini x 13.536737 y 6.4642544 i 15 j 8
ini x 14.48287 y 5.4064751 i 16 j 7
ini x 15.6092205 y 4.3917713 i 17 j 6
ini x 16.487774 y 3.3132189 i 18 j 5
ini x 17.47896 y 2.5445583 i 19 j 4
ini x 18.515202 y 1.5083168 i 20 j 3
ini x 19.461334 y 0.5621834 i 21 j 2
;
fix x y j 1
set gravity=9.81
his 999 unbalanced
;
; heterogeneous case - layers 6 -10 are 10 x stiffer
prop density=1900 bulk=1.33E7 shear=8E7 solve
prop density=1900 bulk=1.33E7 shear=8E7 solve
prop density=1900 bulk=1.33E7 shear=8E7 solve
prop density=1900 bulk=1.33E7 shear=8E7 solve
prop density=1900 bulk=1.33E7 shear=8E7 solve
prop density=1900 bulk=1.33E7 shear=8E7 solve
prop density=1900 bulk=1.33E7 shear=8E7 solve
prop density=1900 bulk=1.33E7 shear=8E7 solve
Numerical Modeling

Numerical model such as FLAC offers these advantages over Limit Equilibrium methods:

- **Any failure mode develops naturally**; there is no need to specify a range of trial surfaces in advance.
- **No artificial parameters** (e.g., functions for inter-slice angles) need to be given as input.
- **Multiple failure surfaces** (or complex internal yielding) evolve naturally, if the conditions give rise to them.
- **Structural interaction** (e.g., rock bolt, soil nail or geogrid) is modeled realistically as fully coupled deforming elements, not simply as equivalent forces.
- **Solution consists of mechanisms that are feasible kinematically.**

There are a number of methods that could have been employed to determine the factor of safety using FLAC. The FLAC **shear strength reduction (SSR)** method of computing a factor of safety performs a series of computations to bracket the range of possible factors of safety. During SSR, the program lowers the strength (friction angle) of the soil and computes the maximum unbalanced force to determine if the slope is moving. If the force unbalance exceeds a certain value, the strength is increased and the original stresses returned to the initial value and the deformation analyses recomputed. This process continues until the force unbalance is representative of the initial movement of the slope and the angle for this condition is compared to the angle available for the soil to compute the **factor of safety**.
Figure 1.1  Grid plot of initial slope

A Mohr-Coulomb constitutive model is assigned to all zones (assumed because no range is given) with the following properties:

- density: 1500 kg/m³
- shear modulus: $0.3 \times 10^8$ Pa
- bulk modulus: $10^8$ Pa
- friction angle: 20°
- cohesion: $10^{10}$ Pa
- tensile strength: $10^{10}$ Pa

Note that a high cohesion and tensile strength are assigned to prevent slope failure during the initialization of gravitational stresses in the model (see below).
Generating the slope

The first GENERATE command defines the base of the slope, and the second GENERATE command creates the slope. Note that the zones are aligned with the angle of the slope so that the zones along the slope face are all quadrilateral-shaped. This is recommended because all zones are then composed of two overlaid sets of triangular elements. These zones are well-suited for plasticity analysis (see Section 1.3.3.2 in Theory and Background). It is also possible to create a slope using the GENERATE line command. However, with this command, single triangular zones will be created along the slope face; these zones are not as accurate for plasticity analysis.

The area directly to the left of the slope face is excavated by declaring the appropriate zones as null. This is done by creating a “region” (i.e., the grid is divided into two regions separated by a boundary) that is defined by “marking” selected gridpoints as boundaries between regions. The following commands mark the boundary of the excavated region and then null the zones within that region:

```
mark i = 1,6  j = 4
mark i = 6  j = 4,11
model null region 1,10
```

The marked boundaries can be verified by issuing the PRINT mark command. The MODEL null command will delete zones in the region containing zone (1,10). Figure 1.1 shows the resulting FLAC grid.
config ats
grid 20,10
;Mohr-Coulomb model
mm

; soil properties --- note large cohesion to force initial elastic
; behavior for determining initial stress state. This will prevent
; slope failure when initializing the gravity stresses
prop s=.3e8 b=1e8 d=1500 fri=20 coh=1e10 ten=1e10

; warp grid to form a slope :
gen 0,0 0,3 20,3 20,0 i 1,4
gen same 9,10 20,10 same i 6 21 j 4 11
mark i=1,6 j=4
mark i=6 j=4,11
model null region 1,10
; displacement boundary conditions
fix x i=1
fix x i=21
fix x y j=1
; apply gravity
set grav=9.81
; displacement history of slope
his ydis i=10 j=10
; solve for initial gravity stresses
solve
;
; reset displacement components to zero
ini xdis=0 ydis=0
; set cohesion to 0
; this is done to explore the failure mechanism in the cohesionless slope
prop coh=0
; use large strain logic
set large
step 1200; comment this line out to calculate factor of safety of undeformed slope
solve fos
save dry_slope.sav 'last project state'
At step 1200

Factor of safety = 0.27 (However, this is surficial slip is not of particular interest. This slip surface will be eliminated, see next page.)

© Steven F. Bartlett, 2010
Note that the surficial failure at the top of the slope can be prevented by slightly increasing the cohesive strength of the soil at the slope face. This often done to explore deeper failure surfaces in the soil mass.

The last part of the FLAC code has been modified to look like this:

; set cohesion to 0
prop coh=0
group 'Soil-Clay:low plasticity' i 6 j 4 10
model mohr group 'Soil-Clay:low plasticity'
prop density=1900.0 bulk=1.33E6 shear=8E5 cohesion=100e3 friction=30.0 dilation=0.0 tension=0.0
group 'Soil-Clay:low plasticity'
; use large strain logic
set large
;step 1200
solve fos
Factor of safety = 0.58

(This is the true factor of safety of the slope for a rotation, slump failure.)
A variety of finite element and finite difference computer programs are available for use in two-dimensional seismic site response analyses. The computer program QUAD4, originally developed by Idriss and his co-workers (Idriss et al., 1973) and recently updated as QUAD4M by Hudson et al. (1994), is among the most commonly used computer programs for two-dimensional site response analysis. QUAD4M uses an equivalent-liner soil model similar to the model used in SHAKE. Basic input to QUAD4M includes the two-dimensional soil profile, equivalent-linear soil properties, and the time history of horizontal ground motion. Time history of vertical ground motion may also be applied at the base of the soil profile. The base can be modeled as a rigid boundary, with design motions input directly at the base, or as a transmitting boundary which enables application of ground motions as hypothetical rock outcrop motions. With respect to the input soil properties, QUAD4M is very similar to SHAKE91. However, the ability to analyze two-dimensional geometry and the option for simultaneous base excitation with horizontal and vertical acceleration components make QUAD4M a more versatile analytical tool than SHAKE91.

A major difference between the QUAD4M and SHAKE91 equivalent-linear models is that the damping ratio in QUAD4M depends on the frequency of excitation or rate of loading. In QUAD4M, the equivalent-linear viscous damping ratio is used to fix the frequency dependent damping curve at the natural frequency of the soil deposit in order to optimize the gap between model damping and the damping ratio. A major drawback of QUAD4M is its limited pre- and post-processing capabilities. These limited capabilities make finite element mesh generation and processing and interpretation of the results difficult and time consuming. QUAD4M is available from the National Information Service for Earthquake Engineering (NISEE) at University Of California at Berkeley for a nominal cost.

Similar software is available commercially and can be purchased such as QUAKE/W. Generalized material property functions allow you to use any laboratory or published data. Three constitutive models are supported: a Linear-Elastic model, an Equivalent Linear model, and an effective stress Non-Linear model.

Dynamic Loading and Boundary Conditions

FLAC models a region of material subjected to external and/or internal dynamic loading by applying a dynamic input boundary condition at either the model boundary or at internal gridpoints. Wave reflections at model boundaries are minimized by specifying either quiet (viscous), free-field or three-dimensional radiation-damping boundary conditions. The types of dynamic loading and boundary conditions are shown schematically in Figure 3.4; each condition is discussed in the following sections.

Application of Dynamic Input

In FLAC, the dynamic input can be applied in one of the following ways:

(a) an acceleration history;
(b) a velocity history;
(c) a stress (or pressure) history; or
(d) a force history.

Dynamic input is usually applied to the model boundaries (i.e., exterior) with the APPLY command. Accelerations, velocities and forces can also be applied to interior gridpoints by using the INTERIOR command.
Note that the free-field boundary, shown in Figure 3.4, is not required if the dynamic source is only within the model (i.e., applied interior to the model.)

Note that the quiet boundary shown on the sides of Figure 3.4 is not required if the dynamic source is applied at the base or top (i.e., applied exterior to the model.)
Free field boundary condition

![Diagram showing free field boundary condition with seismic wave input.]

**Figure 3.7  Model for seismic analysis of surface structures and free-field mesh**

Numerical analysis of the seismic response of surface structures such as dams requires the discretization of a region of the material adjacent to the foundation. The seismic input is normally represented by plane waves propagating upward through the underlying material. The boundary conditions at the sides of the model must account for the free-field motion which would exist in the absence of the structure. In some cases, elementary lateral boundaries may be sufficient. For example, if only a shear wave were applied on the horizontal boundary, it would be possible to fix the boundary along the sides of the model in the vertical direction only (see the example in Section 3.6.3 in FLAC manual). These boundaries should be placed at sufficient distances to minimize wave reflections and achieve free-field conditions. For soils with high material damping, this condition can be obtained with a relatively small distance (Seed et al., 1975).

However, when the material damping is low, the required distance may lead to an impractical model. An alternative procedure is to “enforce” the free-field motion in such a way that boundaries retain their non-reflecting properties — i.e., outward waves originating from the structure are properly absorbed. This approach was used in the continuum finite-difference code NESSI (Cundall et al., 1980). A technique of this type was developed for FLAC, involving the execution of a one-dimensional free-field calculation in parallel with the main-grid analysis.
Instructions for free field boundaries

The following conditions are required in order to apply the free-field boundary condition.

1. The lateral boundaries of the grid must be vertical and straight.
2. The free field boundaries may be applied to the whole grid or to a sub-grid, starting at (1,1), with the left-hand boundary being \( i = 1 \). The right-hand boundary corresponds to the last-encountered non-null zone, scanning along \( j = 1 \) with increasing \( i \) numbers. Any other disconnected sub-grids are not considered when the free-field boundaries are created. Therefore, if sub-grids are used in a simulation that requires free-field boundaries to the main grid, this grid must be the “first” one—i.e., its left and bottom sides must be lines \( i = 1 \) and \( j = 1 \), respectively. The optional keyword \( \text{ilimits} \) forces the free field to be applied on the outer \( i \) limits of the grid (as specified in the \( \text{GRID} \) command). This keyword should be used if null zones exist on the \( j = 1 \) row of zones. It is advisable to perform PLOT apply to verify that the free field is applied to the correct boundary before starting a dynamic simulation.
3. The bottom zones (\( j = 1 \)) at \( i = 1 \) and \( i = \text{imax} \) must not be null.
4. The model should be in static equilibrium before the free-field boundary is applied.
5. The free-field condition must be applied before changing other boundary conditions for the dynamic stage of an analysis. Damping properties must be declared before issuing the free field command.
6. The free-field condition can only be applied for a plane-strain or plane-stress analysis. It is not applicable for axisymmetric geometry.
7. Both lateral boundaries of the grid must be included in the free field because the free field is automatically applied to both boundaries when the \( \text{APPLY ff} \) command is given.
8. The free field can be specified for a groundwater flow analysis (CONFIG gw). A one-dimensional fluid flow model will also be created when \( \text{APPLY ff} \) is issued, and pore pressures will be calculated in the free field.
9. Interfaces and attach-lines do not get transferred to the free-field grid. Thus, an INTERFACE or ATTACH condition should not extend to the free-field boundary. The effect of an interface can be reproduced with a layer of zones having the same properties of the interface.
10. The use of 3D damping when the free field is derived from the sides of a subgrid may not work.
Modeling of Slope Using FLAC without and with free field boundary

```plaintext
config dynamic
set dynamic off
grid 20,10
model elastic
; fill material
  group 'Soil-Sand:uniform - coarse' j 4 10
model mohr group 'Soil-Sand:uniform - coarse'
  prop density=1600.0 bulk=1.67E8 shear=1E8 friction = 35 cohesion=10e3 group 'Soil-Sand:uniform - coarse'
;
; foundation
  group 'Soil-Sand:uniform - coarse - elastic' j 1 3
model elastic group 'Soil-Sand:uniform - coarse - elastic'
  prop density=1600.0 bulk=1.67E8 shear=1E8 group 'Soil-Sand:uniform - coarse - elastic'
;
model null i 1 5 j 4 10
  group 'null' i 1 5 j 4 10
  group delete 'null'
model null i 6 5 10
  group 'null' i 6 5 10
  group delete 'null'
model null i 7 6 10
  group 'null' i 7 6 10
  group delete 'null'
model null i 8 7 10
  group 'null' i 8 7 10
  group delete 'null'
model null i 9 8 10
  group 'null' i 9 8 10
  group delete 'null'
model null i 10 9 10
  group 'null' i 10 9 10
  group delete 'null'
model null i 11 10
  group 'null' i 11 10
  group delete 'null'
ini x 5.534771 y 3.5359905 i 6 j 5
ini x 6.535177 y 4.536397 i 7 j 6
ini x 7.462383 y 5.4880033 i 8 j 7
ini x 8.56039 y 6.56161 i 9 j 8
ini x 9.536397 y 7.5620165 i 10 j 9
ini x 10.488003 y 8.489223 i 11 j 10
ini x 11.56161 y 9.538429 i 12 j 11
```

© Steven F. Bartlett, 2014
; fix x y j 1
fix x i 21
set gravity = 9.81
solve
set dynamic on
set large
ini dy_damp hyst default -3.325 0.823; sand
;
; BOUNDARY CONDITIONS (OPTION 1 or OPTION 2)
;
; free x i 21; OPTION 1 - FIX IN Y ONLY
; fix y i 1; OPTION 1 - FIX IN Y ONLY
; fix y i 21; OPTION 1 - FIX IN Y ONLY
;
apply ff; OPTION 2 - free field
free x i 21; OPTION 2 - free field
fix x y j 1; OPTION 2 - free field
;
his read 100 TAFT_FLAC.acc
; apply xacc 9.81 his 100 j 1; acceleration in m/s^2; OPTION 1 - FIX in Y ONLY
; apply xacc 9.81 his 100 i 1 j 2 4; acceleration in m/s^2; OPTION 1 - FIX in Y ONLY
; apply xacc 9.81 his 100 i 21 j 2 11; acceleration in m/s^2; OPTION 1 - FIX in Y ONLY
apply xacc 9.81 his 100 j 1; acceleration in m/s^2; OPTION 2 - free field
;
apply yvel 0 j 1; keeps base of model from moving
;
def strain1;
deltay = 1.0; one m vertical spacing between nodes
strain1 = (xdisp(7,5) - xdisp(7,4))/deltay; shear strain at toe
end
;
his 2 dytime
his 3 sxy i 11 j 9
his 4 strain1
his 5 xdisp i 13 j 11; crest
his 6 xacc i 13 j 11; crest
his 7 xacc i 1 j 4; free field
his 8 xdisp i 6 j 5; toe
ini xdisp = 0
ini ydisp = 0
set dytime 2
solve dytime 17
;
set hisfile flac-0001.his
his write 7 vs 2 ; accn

© Steven F. Bartlett, 2014
acceleration applied to sides of model and base

acceleration applied to base with free field boundary on sides

© Steven F. Bartlett, 2014