18 Earthquake Loading of a Pile-Supported Wharf

18.1 Problem Statement

A seismic hazard concern in the design of pile-supported wharves at port water前后s is the structural stability of the wharf if earthquake-induced liquefaction occurs in the soils supporting the piles. The analysis of this type of problem is demonstrated using FLAC with the dynamic analysis option and liquefaction modeling facility. Calculations can be made with FLAC for both the deformation of the liquefiable soils and the displacements of the wharf structure that are induced by the earthquake motion. It is also possible to monitor various problem conditions during the seismic excitation, including the development of excess pore pressures in the soils and moments in the piles.

Figure 18.1 shows the problem conditions. The waterfront soils in this exercise consist of three layered deposits, denoted as Soils 1, 2 and 3. The upper two layers (Soils 2 and 3) are characterized as liquefiable silty clays. The thicknesses of the Soil 2 and Soil 3 layers are 6.5 m and 4.5 m, respectively. The top of Soil 1 is at elevation 11 m, and this soil extends to a significant depth beneath the wharf.

The wharf is constructed on a waterfront embankment that is 11 m high and has a slope angle of approximately $27^\circ$. The toe of the embankment is at elevation 11 m. The wharf is supported by two rows of piles that are 3 m apart and 16 m in length. The piles extend through Soils 2 and 3 and into Soil 1, as shown in Figure 18.1. Each row of piles has a spacing of 1 m along the length of the wharf.

Figure 18.1 Pile-supported wharf on layered embankment
The wharf is subjected to an earthquake motion with a peak acceleration of approximately 0.25 g and duration of 40 sec. Figure 18.2 shows the acceleration time history. This history is assumed to be recorded at a rock outcrop near the wharf site.

A fast Fourier transform analysis of the acceleration record (using “FFT.FIS” in Section 3 in the *FISH volume*) results in a power spectrum as shown in Figure 18.3. This figure indicates that the dominant frequency is approximately 1 Hz, the highest frequency component is less than 15 Hz, and most of the frequencies are less than 10 Hz.

### 18.2 Modeling Procedure

This example illustrates a recommended procedure to simulate this type of problem with *FLAC*. The analysis is divided into eight stages:

*Stage 1*: Estimate representative static and dynamic material properties.

*Stage 2*: Evaluate the seismic motion characteristics and determine the appropriate dynamic loading conditions.

*Stage 3*: Construct the *FLAC* model.

*Stage 4*: Calculate the static equilibrium state including the steady state water level and wharf structure at the time of the earthquake event.

*Stage 5*: Perform preliminary undamped runs to check model conditions and evaluate the necessity for additional damping in the model.

*Stage 6*: Apply the earthquake motion and monitor the wharf and soil response during the shaking period, assuming the soils do not liquefy. (This provides a base case for the simulations.)

*Stage 7*: Develop automatic rezoning functions to correct for any distorted mesh conditions that may develop during the large-strain dynamic simulation. (This is anticipated for the case that the soils can liquefy.)

*Stage 8*: Apply the earthquake motion and monitor the wharf and soil response during the shaking period, assuming the soils can liquefy.

The stages are described in sections Sections 18.2.1 through 18.2.8. The project file “WHARF.PRJ” contains a complete description of this example.
Figure 18.2  Horizontal acceleration time history

Figure 18.3  Power spectrum of input acceleration
### 18.2.1 Estimate Representative Static and Dynamic Material Properties

An effective-stress analysis is performed in this example. Effective-stress analyses are well-suited to coupled mechanical fluid-flow problems, especially those involving liquefaction. Mechanical volume change due to cyclic loading can produce a pore-pressure increase, and consequently an effective-stress decrease that can result in a liquefied state.

An effective-stress analysis in FLAC requires material property input under drained conditions. The following drained material properties are assigned to the soils.

#### Table 18.1 Drained properties for Soils 1, 2 and 3

<table>
<thead>
<tr>
<th>Property</th>
<th>Soil 1</th>
<th>Soil 2</th>
<th>Soil 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Dry density (kg/m$^3$)</td>
<td>2009</td>
<td>1813</td>
<td>1715</td>
</tr>
<tr>
<td>Young’s modulus (MPa)</td>
<td>610.9</td>
<td>163.7</td>
<td>163.7</td>
</tr>
<tr>
<td>Poisson’s ratio</td>
<td>0.3</td>
<td>0.3</td>
<td>0.3</td>
</tr>
<tr>
<td>Bulk modulus (MPa)</td>
<td>509.1</td>
<td>136.4</td>
<td>136.4</td>
</tr>
<tr>
<td>Shear modulus (MPa)</td>
<td>235.0</td>
<td>63.0</td>
<td>63.0</td>
</tr>
<tr>
<td>Cohesion (Pa)</td>
<td>4000</td>
<td>2000</td>
<td>2000</td>
</tr>
<tr>
<td>Friction angle (degrees)</td>
<td>40</td>
<td>35</td>
<td>30</td>
</tr>
<tr>
<td>Dilation angle (degrees)</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

The dynamic characteristics of all of the soils in this model are assumed to be governed by the modulus reduction factor ($G/G_{max}$) and damping ratio ($\lambda$) curves, as shown in Figures 18.4 and 18.5, and denoted by the “Shake91” legend. These curves are considered to be representative of clayey soils with an average mass density of 2000 kg/m$^3$, and an average shear modulus of 300 MPa; the data are derived from the input file supplied with SHAKE91 (for more information, see http://nisee.berkeley.edu/software/shake91/).

In FLAC, the hysteretic model function default is used to simulate the dynamic characteristics as defined by the curves in Figures 18.4 and 18.5. Runs are made with a one-zone model (see Example 1.8 in Dynamic Analysis) to calculate the default function values, $L_1$ and $L_2$, that provide a reasonable fit to both modulus reduction and damping ratio curves up to approximately 0.3% cyclic shear strain, as shown in the two figures.

A liquefaction condition is estimated for the upper two layers, Soils 2 and 3, in the vicinity of the embankment slope extending from the toe of the slope to $x = 95$ m. Liquefaction potential is quantified in terms of standard penetration test (SPT) results. A normalized standard penetration test value, $(N_1)_{60}$, of 10 is selected as representative for Soil 2 and Soil 3. This value is used to determine the parameters $C_1$ and $C_2$ in the Finn-Byrne liquefaction model in FLAC (selected by setting the property `ff_switch = 1` for the Finn-Byrne model). For a normalized SPT blow count of 10, the Finn-Byrne model parameters are $C_1 = 0.2452$ and $C_2 = 0.8156$. See Section 1.4.4.2 in Dynamic Analysis for a description of the formulation, and see Byrne (1991) for a discussion on the derivation of these parameters.
Figure 18.4  Modulus reduction curve for clayey soils (from SHAKE91 data)
FLAC default hysteretic damping with $L_1 = -3.156$

and $L_2 = 1.904$

Figure 18.5  Damping ratio curve for clayey soils (from SHAKE91 data)
FLAC default hysteretic damping with $L_1 = -3.156$

and $L_2 = 1.904$
The permeability, porosity and water bulk modulus are required for the groundwater phase of this analysis. A hydraulic conductivity of \(9.81 \times 10^{-7} \text{ m/sec}\) is assumed for all three soils. This corresponds to a mobility coefficient, required by FLAC, of \(10^{-10} \text{ m}^2 / (\text{Pa-sec})\). The porosity of the three soils is 0.3. The water bulk modulus is selected to be 200 MPa. This corresponds approximately to an air/water soil mixture at 99% saturation. (See Section 1.7.5.2 in Fluid-Mechanical Interaction.)

The structural properties for the wharf are listed in Tables 18.2 and 18.3. The properties listed in Table 18.2 are assigned to the wharf beam and pile elements, and the properties listed in Table 18.3 are assigned to represent the behavior at the pile-soil interface.

### Table 18.2 Structural properties for wharf

<table>
<thead>
<tr>
<th></th>
<th>Elastic Modulus (GPa)</th>
<th>Moment of Inertia ((\text{m}^4))</th>
<th>Cross Sect. Area ((\text{m}^2))</th>
<th>Mass Density ((\text{kg/m}^2))</th>
<th>Pile Perimeter (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Beams</td>
<td>20.0</td>
<td>(2.364 \times 10^{-3})</td>
<td>0.305</td>
<td>2000</td>
<td>—</td>
</tr>
<tr>
<td>Piles</td>
<td>20.0</td>
<td>(1.917 \times 10^{-4})</td>
<td>0.05</td>
<td>2000</td>
<td>0.785</td>
</tr>
</tbody>
</table>

### Table 18.3 Coupling spring properties for pile-soil interface

<table>
<thead>
<tr>
<th></th>
<th>Normal Stiffness ((\text{GN/m/m}))</th>
<th>Shear Stiffness ((\text{GN/m/m}))</th>
<th>Normal Cohesion ((\text{N/m}))</th>
<th>Shear Cohesion ((\text{N/m}))</th>
<th>Normal Friction (degrees)</th>
<th>Shear Friction (degrees)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Soil 1</td>
<td>0.01</td>
<td>0.01</td>
<td>4000</td>
<td>4000</td>
<td>40</td>
<td>40</td>
</tr>
<tr>
<td>Soils 2 &amp; 3</td>
<td>0.01</td>
<td>0.01</td>
<td>1000</td>
<td>1000</td>
<td>30</td>
<td>30</td>
</tr>
</tbody>
</table>

Material damping for the structural elements is not included in this exercise. Material damping of the soils, resulting primarily from material failure during liquefaction, is considered to provide sufficient damping. Note that structural damping can be added, if necessary, by specifying Rayleigh damping for the structures.
18.2.2 Evaluate Seismic Motion Characteristics

The characteristics of the input motion are evaluated first, before generating the wharf model. These characteristics are used to help select the appropriate mesh size, the locations for model boundaries, and any adjustments to the input wave record for application in this analysis. (See Section 1.4.2 in Dynamic Analysis for further information on the relation between wave propagation characteristics and mesh generation.)

The mesh size for the model should be selected to provide accurate wave transmission. Based upon the elastic properties listed in Table 18.1, Soil 2 has the lowest shear wave speed (173 m/sec, for a shear modulus of 63 MPa and saturated density of 2113 kg/m³). If the largest zone in the model is set to approximately 1.0 m (in order to provide reasonable runtimes for this example), then the maximum frequency that can be modeled accurately is

\[ f = \frac{C_s}{10 \Delta l} \approx 17 \text{ Hz} \]  \hspace{1cm} (18.1)

The given acceleration time history contains a very small amount of frequency components above 10 Hz, as shown in Figure 18.3. Therefore, it is not necessary to filter the input wave for this example.*

The acceleration time history requires some adjustment before application in the dynamic analysis. The evaluation and adjustment is performed using the [Utility/Seismic] tool. When the tool is active, the window shown in Figure 18.6 appears.

To import the horizontal acceleration time history for the wharf model, press the “file open” button in the tool, select the file named “ACC1.HIS” and then press the [Select] button. Choose “history” as the file format, select “Acceleration” as the ground motion type with the input unit g, then press the [Next] button for filtering.

* If filtering is required, the [Seismic] tool, described in Section 1.2.5.7 of the GUI Reference, can be used to filter the wave.
SI units are used in the wharf example. To convert the units from $g$ to SI units, right-click on the plot and select **Convert Units** on the pop-up menu, as shown in Figure 18.7. The maximum frequency that can be modeled accurately for the mesh size is 17 Hz. From the spectrum plot as shown in Figure 18.7, the given acceleration time history contains a very small amount of frequency components above 10 Hz. Therefore, it is not necessary to filter the input wave for this example.
Press the button in the baseline correction window, as shown in Figure 18.8. Right-click on the plot window, select \textit{Show Zero Line}, and the zero lines will also be shown on the plots. In this example, we use moving average to remove the displacement drift. After some adjustments, we find that for this record, calculating 20 iterations of the running average over 2 seconds does a very good job at getting the static offset to be close to zero. If you run once, the static offset is basically 0, as shown in Figure 18.9.
Figure 18.8  Waveform before baseline correction

Figure 18.9  Waveform after baseline correction
The processed data can be saved by pressing the save button. Name the file “VEL2.TAB”, press the Select button, choose “Velocity” as the motion type and “table” as the file format and then press the Export button to export the data as shown in Figure 18.10.

For this exercise, the motion in “VEL2.TAB” is assumed to correspond to the motion at the base of the FLAC model. The base is selected at elevation -9 m, or 20 m beneath the toe of the waterfront embankment, which is considered to be at a sufficient distance from the embankment to minimize any boundary effects. (See Section 18.2.5.) A deconvolution analysis is not performed in this case.

Note that a SHAKE deconvolution analysis can be used to determine an input acceleration at the depth it is applied in a model, accounting for propagation of the wave from the location where it is recorded. See Section 1.4.1.7 in Dynamic Analysis for further discussion on earthquake deconvolution for FLAC models.

Figure 18.10 Waveform after baseline correction
18.2.3 Construct FLAC Model

An effective-stress analysis in FLAC is a fully coupled mechanical fluid-flow analysis that requires the selection of the groundwater-flow calculation mode in addition to the mechanical, dynamic calculation mode (CONFIG gwflow dynamic) in the FLAC Model options dialog. Also, the interface options for structural elements and advanced constitutive models are activated, and ten extra grid variables are selected in the dialog.

The problem geometry is generated in FLAC using the [Generate/Simple] tool, which begins the model creation based upon a rectangular mesh with dimensions of \( 0 \leq x \leq 100 \text{ m} \) and \(-9 \text{ m} \leq y \leq 22 \text{ m}\). Note that the lateral boundaries are located outside the region of liquefiable soils. The [Virtual/Edit] tool is then used to add and move points (0,11), (26.73,11) (48,22) and (100,22) to create the slope surface, as shown in Figure 18.11. Boundary conditions are assigned: roller boundaries on the sides and pinned boundary on the base, and a \(100 \times 40\) quadrilateral-zone mesh is prescribed for the model, as shown in Figure 18.12. The maximum zone size is 1.0 m.

![Figure 18.11 The slope surface in defined using the Blocks stage of the Virtual/Edit tool](image-url)
The virtual grid is executed to create the FLAC model by pressing the virtual/execute button. The three soil layers are now defined in the utility/table tool. Three closed tables are created to circumscribe each soil layer. Figure 18.13 shows the creation of table 2, which circumscribes the Soil 2 layer. The three soil materials and their properties can now be assigned using the table range in the material/assign tool, as shown by Figure 18.14.

The properties in Table 18.1 are entered into a material database by clicking the database button in the lower-right corner of the material/assign tool. The three material types, Soil 1, Soil 2 and Soil 3, are created in a material class named wharf, and their properties are assigned by editing the dialog for each material. The materials are then stored in a separate database file, named “WHARF.GMT,” which can be accessed at any time in subsequent analyses. The three materials are made available for the present model by clicking the execute button in the Material List dialog; the materials will then be listed in the Material List in the material/assign tool. By clicking on the table radio button, then highlighting each material and clicking on one zone in each of the three table regions of the model plot, the selected region of zones will change color, corresponding to that of the selected material. Once all three materials have been assigned, the execute button is pressed to send the commands to FLAC. The resulting model with the assigned materials is shown in Figure 18.15.

Note that the dynamic calculation phase will be performed using the large-strain mode in FLAC. By using the virtual meshing tool, and by using the closed-table range to assign materials, the model will consist entirely of quadrilateral-shaped zones. This will help prevent the development of badly distorted zones along the slope face during the large-strain calculation.
Figure 18.13 Table 2 is created to circumscribe Soil 2, in the Utility/Table tool.

Figure 18.14 Soil materials are assigned using the Table range in the Material/Assign tool.
18.2.4 Calculate Static Equilibrium State

This analysis requires an estimate of the state of stress at the wharf site at the time of the earthquake loading. One important consideration, particularly in a liquefaction analysis, is the representation of the initial static shear-stress distribution in the wharf embankment. The initial, static shear stress can affect the triggering of liquefaction.

For this exercise, the wharf embankment construction is not modeled directly. If information on the embankment construction stages is available, the stages should be simulated in the model in order to provide a more realistic representation of the initial stress state. For this example, the static equilibrium calculation is performed in three steps.

First, the model is brought to an equilibrium state, assuming dry material conditions. Gravity is prescribed in the Settings/Mechanical tool, groundwater flow is turned off in the Settings/GW tool, and the dynamic analysis mode is turned off in the Settings/Dynamic tool. The model is then brought to a static mechanical-equilibrium state by selecting the Run/Solve tool and then checking the Solve initial equilibrium as elastic model box.

Next, the water level is raised and the equilibrium state of the submerged embankment is calculated. This is performed as an uncoupled, fluid flow-mechanical calculation. First, the steady-state flow condition is achieved by applying a pore-pressure gradient using the WATER table command. The water table is defined by a TABLE command, and the level is located at elevation $y = 20$ m. Groundwater properties, porosity and permeability, are assigned through the Material/GWProp tool.
Groundwater flow is turned on, the fluid density and bulk modulus are assigned, and the water table ID is set in the [Settings/GW] tool. The mechanical calculation is turned off in the [Settings/Mechanical] tool, and the steady-flow state is calculated.

The weight of the reservoir water above the embankment is included when the WATER table command is executed. The effect of the weight of the water in the reservoir and within the embankment on the stress state in the embankment is calculated by turning the mechanical calculation mode back on (in the [Settings/Mechanical] tool), and turning the groundwater flow mode off and setting the water bulk modulus to zero (in the [Settings/GW] tool). The model is now solved for the mechanical equilibrium state of the submerged embankment.

In the third step, the wharf structure is added using the [Structure/Beam] tool to create the wharf deck and the [Structure/Pile] tool to create the supporting piles. Note that each pile is divided into 16 segments. This ensures that at least one pile node is located within each zone along the length of the pile. The wharf beam and piles share the same nodes at their intersection. This provides a rigid connection between the wharf and piles. The structural properties for the wharf structure, as listed in Tables 18.2 and 18.3, are specified using the [Structure/Prop] tool. Note that the 1 m spacing is entered with the pile properties. The different pile-soil interface properties listed in Table 18.3 are assigned by specifying a structural property ID number for the 5 pile segments within Soil 1 that is different from the one assigned for the 11 segments within Soils 2 and 3.

The model is brought to an equilibrium state with the wharf in place. Figure 18.16 shows the model geometry with the wharf structure. The initial pore pressure contours are also plotted. Figure 18.17 shows the total vertical stress contours and axial forces in the piles at the equilibrium state.

Figure 18.16 Pore-pressure contours at equilibrium state, including wharf structure
### 18.2.5 Perform Preliminary Evaluation Runs

Before running full nonlinear simulations, preliminary runs are made to assess the effect of model boundary locations, and to estimate maximum levels of cyclic shear strain, natural frequency ranges and extent of plastic failure. These runs also help evaluate the necessity for additional material damping in the model.

Two types of preliminary runs are made: (1) undamped elastic-material runs to monitor shear strains and velocity levels throughout the model during the dynamic excitation; and (2) undamped Mohr-Coulomb material runs to identify the approximate extent of material failure resulting from the dynamic excitation.

For these runs, we begin the dynamic loading stage by turning on the dynamic calculation mode from the Settings/Dynamic tool. The input velocity, as described previously in Section 18.2.2, is read into FLAC as table 103.

The dynamic boundary conditions are assigned in the In Situ/Apply tool. The free-field boundary is set first for the side boundaries by selecting the Free-field button. (Note that conditions within the grid along the sides of the model must not be changed after the free-field boundary is applied, because these conditions will not be transferred to the free field.)
Next, the dynamic input is assigned along the bottom boundary. The wharf material, Soil 1, is assumed to extend to a significant depth beneath the dam. Therefore, it is necessary to apply a quiet (viscous) boundary along the bottom of the model to minimize the effects of reflected waves at the bottom.

Quiet boundary conditions are assigned in both the \( x \)- and \( y \)-directions by first selecting the \text{QUIET} button and dragging the mouse along the bottom boundary, and then selecting the \text{QUIET} button and repeating the procedure.

In order to apply quiet boundary conditions along the same boundary as the dynamic input, the dynamic input must be applied as a stress boundary, because the effect of the quiet boundary will be nullified if the input is applied as an acceleration (or velocity) wave. The velocity record (in table 103) is converted into a shear stress boundary condition using a two-step procedure:

1. Convert the velocity wave into a shear stress wave using the formula

\[
\sigma_s = \text{factor} \times (\rho \ C_s) \ v_s
\]  

(18.2)

where

- \( \sigma_s \) = applied shear stress;
- \( \rho \) = mass density of the material at the boundary;
- \( C_s \) = speed of \( s \)-wave propagation through the medium at the boundary; and
- \( v_s \) = input shear particle velocity.

Note that the \( \text{factor} \) in Eq. (18.2) is normally equal to 2 to account for the input energy dividing into downward and upward propagating waves. (See Section 1.4.1 and Eq. (1.14) in Dynamic Analysis.)

2. Monitor the \( x \)-velocity at the bottom of the model during the dynamic run to compare this velocity to the input motion shown in Figure 18.10. Some adjustment to the \( \text{factor} \) for the input stress wave may be required in order to produce a velocity at the bottom of the model that corresponds to the input velocity.

This two-step procedure is applied as follows to prescribe the dynamic wave as a shear stress boundary condition along the base for this example.

First, the \text{STRESS/SXY} boundary condition type is selected in the \text{IN SITU/APPLY} tool, and the mouse is dragged from the bottom-left corner of the model to the bottom-right corner. The \text{ASSIGN} button is pressed, which opens the \text{Apply value} dialog. The velocity record, in table 103, is considered a multiplier, \( v_s \), for the applied value. The velocity record is applied by checking the \text{TABLE} radio button, and selecting table number 103 as the multiplier.

The applied value for \text{MM} in the \text{Apply value} dialog is set to \( 2 \rho_s \ C_s \) (from Eq. (18.2)), in which \( \rho_s \) and \( C_s \) correspond to the saturated density (2309 kg/m\(^3\)) and shear wave speed (319 m/sec) for wharf Soil 1.
We monitor histories of selected model variables during the dynamic calculation. These are chosen using the [Utility/History] tool. Velocities and accelerations are monitored at the bottom and top of the model in order to evaluate the transmission of the wave through the model. Shear strains and shear stresses are also monitored during the dynamic loading.

Velocity and acceleration histories are calculated at specified x- and y-coordinate locations using \textit{FISH} function “VEL\_ACC\_HIST.FIS.” Shear stress and shear strains are monitored at selected locations via \textit{FISH} function “STRESS\_STRAIN\_HIST.FIS,” and shear strains are monitored throughout the model using \textit{FISH} function “MON\_EX.FIS.”

Several runs are made in order to determine an appropriate value for \textit{factor}. A value of 1.1 is found to produce a reasonable comparison between calculated and input velocity histories, as shown in Figure 18.18. The reason that this value provides a better match than the value of 2 is because the base of the model is within the range of the velocity doubling effect of the free surface.\textsuperscript{*}

The effect of velocity doubling is also evident by comparing the velocity at the model base to that at the top of the model. As shown in Figure 18.19, there is only a small amplification of the velocity wave at the free surface compared to the base, indicating that the base is within the range of velocity doubling.

\begin{center}
\textsuperscript{*} The extent of velocity doubling can be estimated based upon the dominant frequency of the input wave (0.5 Hz) and the shear wave speed of saturated Soil 1 (319 m/sec). In this case, velocity doubling is estimated to extend approximately 160 m below the ground surface. (See Section 1.4.1 in \textit{Dynamic Analysis}.)
\end{center}
Figure 18.18 Comparison of velocity histories at the model base
– undamped – elastic material

Figure 18.19 Comparison of velocity histories at the base and top of the model
– undamped – elastic material
Velocity histories recorded along the base of the model are also compared, to check whether the dynamic motion is applied uniformly along the base. Figure 18.20 compares three histories along the base for the undamped run with Mohr-Coulomb material, and indicates that the motion is uniform.

![Figure 18.20 Comparison of velocity histories along the model base – undamped – Mohr-Coulomb material](image)

The shear strains are monitored throughout the model during the 40 second dynamic loading, and peak strains are determined using FISH function “MON_EX.FIS.” Figure 18.21 shows a contour plot of the maximum shear strains for the undamped, elastic-material model. This figure indicates that maximum elastic shear strains are smaller than 0.1% throughout almost the entire model. This range of shear strains is considered appropriate for inclusion of hysteretic damping based upon the dynamic characteristics of the soils, as shown in Figures 18.4 and 18.5. (Shear modulus reduction is in the range of 0 to 60%, and damping ratio is in the range of 0 to 10%).

The frequency range for the natural response of the elastic materials is calculated to be relatively uniform throughout the model, with a dominant frequency of approximately 0.5 Hz. Figure 18.22 displays a typical power spectrum recorded in Soil 3. This is comparable to the dominant frequency of the input velocity.
**Figure 18.21** Maximum shear strain contours – undamped – elastic material

**Figure 18.22** Power spectrum of velocity at $x = 50$, $y = 21$ in Soil 3 – undamped – elastic material
The locations of the model boundaries relative to the wharf embankment are selected after monitoring the development of the failure surface within the embankment slope during the dynamic run with the undamped, Mohr-Coulomb material model. The failure surface is estimated based upon the development of shear-strain concentration bands within the shear-strain contour plots. Figure 18.23 displays the contour plot for the undamped, Mohr-Coulomb material model. Based upon experience, the model boundaries are located approximately two to three times the extent of the failure surface, as defined by the shear bands, away from the failed region. The locations of the lateral boundaries are also selected to be beyond the extent of the liquefiable soils. Note that boundaries may need to be extended farther out if a significant extension of the failure region is shown from the subsequent liquefaction simulations.

The model is also checked for base rotation. The bottom boundary is a quiet boundary and free-field boundaries are applied along the sides of the model. Therefore, it is possible for rotation of the model base to develop. (See Section 1.4.1.5 in Dynamic Analysis.) Vertical displacements are monitored at the bottom corners of the model. After 40 seconds, a constant counterclockwise rotation of the model is evident from the displacement histories, see Figure 1.9 in Section 1.4.1.5 in Dynamic Analysis. The rotation starts after approximately 2 seconds of earthquake loading, and has a fairly small magnitude after 40 seconds. This rotation has a fairly minor impact on the model results. However, for illustrative purposes, a correction is made by adding the SET corr_ffrot on command before the APPLY ff command. Figure 18.24 illustrates the effect of this correction on preventing base rotation.
Figure 18.23 Shear-strain contour plot indicating failure surface as defined by concentration of shear contours – undamped – Mohr-Coulomb material

Figure 18.24 Histories of vertical displacement at bottom corners of model – undamped – Mohr-Coulomb material with SET corr_frot on
18.2.6 Perform Earthquake Simulation Assuming No Liquefaction

A fully coupled nonlinear seismic analysis is performed using the Mohr-Coulomb model to represent the wharf embankment soils, with additional hysteretic damping applied to simulate the dynamic characteristics of the soils.

The default hysteretic damping function with the selected parameters ($L_1 = -3.156$ and $L_2 = 1.904$) produces curves that provide a reasonable match to the shear modulus reduction curve and damping ratio curve up to approximately 0.3%, as shown in Figures 18.4 and 18.5. This is considered appropriate to cover the range of elastic strains as indicated from the undamped elastic material run shown in Figure 18.21.

Hysteretic damping is assigned in the In Situ/Initial tool. The dialog shown in Figure 18.25 is opened by selecting the Zones type, checking the Hysteretic Damping menu item, and then Assign, to assign the same values for all zones in the model.

![Hysteretic damping parameters](image)

Hysteretic damping does not completely damp high frequency components, so a small amount of stiffness-proportional Rayleigh damping is also applied. (See Section 1.4.3 in Dynamic Analysis.) A value of 0.2% at the dominant frequency (0.5 Hz) is assigned in the Dynamic damping parameters dialog shown in Figure 18.26. Rayleigh damping is applied by selecting the GPs type, and then Dynamic Damping in the In Situ/Initial tool.

The dynamic loading and boundary conditions are applied in the same manner as described in Section 18.2.5. Note that the damping must be assigned before the free-field boundary condition is applied. Otherwise, the damping parameters will not be prescribed in the free-field grid.

The response of this model is only slightly different from that for the undamped, Mohr-Coulomb model run described previously in Section 18.2.5. The velocity at the top of the model is approximately 20% higher than that at the base, which is comparable to the undamped run. See Figure 18.27.
The movement of the embankment essentially stops when the earthquake loading ends at 40 seconds. The deformation of the embankment is illustrated in Figures 18.28 and 18.29. The crest has settled approximately 0.2 ft and translated approximately 0.5 ft after 40 seconds, as indicated in Figure 18.28. The extent of failure within the embankment slope is shown by the shear band in Figure 18.29.

![Rayleigh damping parameters used with hysteretic damping](image)

**Figure 18.26 Rayleigh damping parameters used with hysteretic damping**

![x-velocity histories at top and base of wharf embankment model](image)

**Figure 18.27 x-velocity histories at top and base of wharf embankment model**
Earthquake Loading of a Pile-Supported Wharf

**Figure 18.28** x- and y-displacement histories recorded at wharf embankment crest – Mohr-Coulomb material

**Figure 18.29** Shear strain contours at 40 seconds – Mohr-Coulomb material
18.2.7 Automatic Rezoning Functions

When a simulation is running in large-strain mode, the geometry of some zones can become extremely distorted, such that the simulation has to halt. In FLAC, a “bad geometry” error message will be issued to indicate that the model has run into such a situation. In order to have the simulation continue, a new, regular mesh is needed to replace the old, distorted mesh, and grid-dependent data need to be transferred from the old mesh to the new mesh. Automatic rezoning is used in FLAC to perform this operation and avoid a bad-geometry interruption during cycling. See Section 4 in Theory and Background for further information on the automatic rezoning logic.

For this exercise, FISH functions are used to generate a new mesh, transfer model variables from the old mesh to the new mesh, and reassign boundary conditions. The functions are accessed in “WHARF_REZONE.FIS”. The rezoning is initiated via the function _rezdyn. The rezoning operation is performed in six steps:

1. Remove the applied pressure conditions from the surface boundary.

2. Select a zone range in which to perform the rezoning. By default, the entire grid is rezoned. The rezone region can be limited with the REZONE set range i1i2j1j2 command. In this exercise, the rezoning region is restricted to the embankment slope area from i-zone 14 to 93 and j-zone 21 to 40. This includes the anticipated extent of liquefiable soils. (Note that only one constitutive model can exist within the region selected for automatic rezoning.) The region must also completely include the structural elements that represent the wharf structure, because the structure is located within the slope area that will be rezoned.

3. Store the surface profile (the coordinates of the gridpoints) of the current (old) mesh, over the region to be rezoned, in a lookup table (table 12).

4. Call in the function responsible for generating a new mesh (_newmesh), within which a new rectangular mesh is created and fit along the top boundary to the slope surface defined by table 12. The FISH function “TABTOP.FIS” available from the FISH library (accessed from the Utility/FishLib tool at the GridGeneration/gentabletop menu item) is used to fit the zoning within the surface defined by table 12.

5. The FLAC built-in command REZONE invokes _newmesh to create the new mesh and automatically transfer the data from the old mesh to the new mesh.

6. The water pressure is reapplied along the boundary of the new mesh. The ending gridpoint for the applied pressure is found via the FISH function _seekwtgp.

The commands SET geometry 0.3 and SET rez_func_rezdyn are given so that the rezoning will occur automatically whenever any zone in the FLAC model distorts such that the ratio of subzone area to total zone area falls below 0.3.
18.2.8 Perform Earthquake Simulations Assuming Soils Can Liquefy

The liquefaction simulation is performed by changing Soils 2 and 3 to Finn-Byrne materials. The Finn-Byrne model is assigned to zones within the spatial range of $x = 10$ m to $x = 95$ m, and $y = 0$ to $y = 22$ m. This range also includes a portion of Soil 1 material. See Figure 18.30. This is necessary to allow the range selected for automatic rezoning to include zones at the toe of the slope that may potentially experience excessive distortion. The spatial range also includes all the structural elements representing the wharf structure. As mentioned previously in Section 18.2.7, the region defined for automatic rezoning can only contain one constitutive model.

![Figure 18.30 Finn-Byrne model assigned to zones within region of potentially liquefiable soils](image)

The tool is used to assign Finn-Byrne material. The Byrne (1991) liquefaction model is selected for the soils (see Eq. (1.93) in Section 1.4.4.2 in Dynamic Analysis), and properties are prescribed corresponding to a normalized SPT blow count of 10. For example, Figure 18.31 displays the dialog to enter properties for Soil 2. Note that the latency property is set to a high value at this stage. This is done to make sure that the model is still at equilibrium when changing the selected zones from Mohr-Coulomb to Finn-Byrne material. When is issued, only a few steps are taken, which ensures that the model is still in equilibrium.

The soil zones that are assigned the Finn-Byrne material model are given the group names Soil 1f, Soil 2f and Soil 3f in order to distinguish these soils from the original soil groups.
Preliminary runs with the Finn-Byrne model produced significant deformations along the lateral (free-field) boundaries of the model. As discussed in Section 1.4.1.4 in Dynamic Analysis, the free field boundary performs a small-strain calculation even though the main grid is executing in large-strain mode. In order to reduce the mismatch between large-strain and small-strain calculations, a five-zone wide column of high strength zones is located at the left and right boundaries to minimize the large deformation. The high-strength zones are sufficiently far from the slope that they do not significantly affect model response. Alternatively, the lateral boundaries could be moved farther out. However, this will increase the simulation time.

The dynamic calculation is run in the same manner as described previously. The latency value is set to 50 for Soils 2f and 3f. (Latency remains at a high value for Soil 1f so that this soil will perform as a Mohr-Coulomb material.) Note that the free-field boundary condition must be applied after changes to the material models are made, to ensure that these changes are transferred to the free field.

The automatic rezoning functions in “WHARF.REZONE.FIS” are called into the FLAC model at this point, before beginning the dynamic loading. The command SET rez_func rezdyn is also given so that rezoning will be performed automatically when a zone distortion limit (set by the SET geometry 0.3 command) is reached.

The normalized excess pore-pressure ratio (or cyclic pore-pressure ratio), $u_e/\sigma'_c$, can be used to identify the region of liquefaction in the model, where $u_e$ is the excess pore pressure and $\sigma'_c$ is the initial effective confining stress. A liquefaction state is assumed to occur when $u_e/\sigma'_c = 1$. The excess pore-pressure ratio is calculated in FISH function “GETEXCESSPP.FIS,” and the maximum value is stored in FISH extra array ex.6.

Figures 18.32 through 18.34 show the progressive development of liquefaction and failure in the model. These plots illustrate the initiation of liquefaction, as shown by the contour line for $u_e/\sigma'_c = 1$, and the growth of the failure surface in the slope, as defined by the concentration of shear strain contours. At 2 seconds, the slope begins to fail in a manner similar to the failure calculated for the Mohr-Coulomb material (compare Figure 18.32 to Figure 18.29). By 3 seconds, a contour line for $u_e/\sigma'_c = 1$ has developed and the failure surface, as indicated by the shear strain contour, has extended into the liquefied region, as shown in Figure 18.33. By 10 seconds, the slope failure is well-defined, as shown in Figure 18.34.
Figure 18.32 Shear strain contours and excess pore pressure ratio = 1 contour at 2 seconds – Finn-Byrne material

Figure 18.33 Shear strain contours and excess pore pressure ratio = 1 contour at 3 seconds – Finn-Byrne material
The permanent movements of the embankment and wharf structure are illustrated in Figures 18.35 and 18.36. The crest of the embankment settles approximately 0.7 ft. and moves horizontally roughly 1.6 ft., as shown in the displacement history plot in Figure 18.35. The slope movement at 40 seconds is evident in the shear strain contour plot in Figure 18.36. Note that a badly distorted grid is not produced in this simulation, and an automatic rezoning calculation is not performed.

For this simulation, the shear strength parameters of the liquefiable soils do not change. It has been observed (e.g., see Olson et al. 2000) that if effective stress goes to zero, the shear strength reduces to a “strain-mobilized (liquefied) shear strength,” which implies a residual strength. In order to approximate this loss in strength, an additional simulation is made with Finn-Byrne material, in which Soils 2f and 3f have their frictional strength reduced to $5^\circ$ if $u_e/\sigma'_c \geq 1$ in a zone during the dynamic loading. (See “GETEXCESSPP.FIS”.) This strength reduction is arbitrary, but is intended to produce a pronounced loss in strength and increase in deformation in order to also illustrate the application of the automatic rezoning logic.

This time the embankment moves horizontally approximately 6.0 ft., as shown in the displacement history plot in Figure 18.37. A significant distortion of the grid is produced near the toe of the embankment, as illustrated in Figure 18.38. An automatic rezoning operation is performed at roughly 6 seconds, when the zone distortion limit of 0.3 is reached.
Figure 18.35 x- and y–displacement histories recorded at wharf embankment crest – Finn-Byrne material

Figure 18.36 Shear strain contours at 40 seconds – Finn-Byrne material
**Figure 18.37** x- and y-displacement histories recorded at wharf embankment crest – Finn-Byrne material with residual strength

**Figure 18.38** Shear strain contours at 40 sec – Finn-Byrne material with residual strength
18.3 References
