User's Manual

1D Seismic Site Response Analysis Example

http://www.soilquake.net/ucsdsoilmodels/

University of California: San Diego

August 2, 2017

Table of Contents

USER'S MANUAL

TABLE OF CONTENTS

Page #

1 General Information 1

1.1. UCSDSAND3 Material Model 1

1.2. UCSDCLAY Material Model 1

2 Installation 3

2.1. FLAC 3

2.2. LS-DYNA 4

2.3. DIANA 4

2.4. Abaqus 4

3 Input 6

3.1. Program 6

3.2. Input File 6

3.3. Water Table 6

3.4. Rayleigh Damping 6

3.5. Analysis Parameter 7

3.6. Input Motion 7

3.7. Soil Properties 7

4 Using the Excel Sheet 9

4.1. FLAC 9

4.2. LS-DYNA 11

4.3. DIANA 11

4.4. Abaqus 12

1 GENERAL INFORMATION

The UCSD soil models **UCSDSAND3** and **UCSDCLAY** are three-dimensional (3D) elasto-plastic material models for the simulation of cohesionless and cohesive soils, respectively. These models has been evolving over the past two decades with extensive calibration through numerous sources including downhole-array records, laboratory tests, shake-table and centrifuge experiments. These soil models have been available in OpenSees since the early 2000s.

The UCSD soil models have now been implemented into a number of analysis packages including FLAC/FLAC3D, LS-DYNA, DIANA and Abaqus as user defined materials. For illustration purposes, one-dimensional (1D) site response analysis examples utilizing these soil models are provided on http://soilquake.net/ucsdsoilmodels/.

Note that the compiled files of these soil models provided in the above-mentioned examples are for the 1D site response analysis only. For 3D full version of the UCSD soil models, please contact Prof. Ahmed Elgamal (E-mail: elgamal AT ucsd DOT edu).

1.1. UCSDSAND3 Material Model

UCSDSAND3 material is an elasto-plastic material for simulating the essential response characteristics of pressure sensitive soil materials under general loading conditions. Such characteristics include dilatancy (shear-induced volume contraction or dilation) and non-flow liquefaction (cyclic mobility), typically exhibited in sands or silts during monotonic or cyclic loading.

In this model, plasticity is formulated based on the multi-surface (nested surfaces) concept, with a non-associative flow rule to reproduce dilatancy effect. The yield surfaces are of the Drucker-Prager type. This sand model reproduces conventional liquefaction triggering logic.

1.2. UCSDCLAY Material Model

UCSDCLAY material is an elasto-plastic material model which reproduces nonlinear hysteric shear behavior and accumulation of permanent shear deformation. Plasticity is exhibited only in the deviatoric stress-strain response (Elgamal et al. 2008)*. The volumetric stress-strain response is linear-elastic and independent of the deviatoric response. Nonlinear response is

formulated based on the multi-surface (nested surfaces) concept, with an associative flow rule. The yield surfaces are of the von Mises type.

This material is implemented to simulate monotonic or cyclic response of materials whose shear behavior is insensitive to the confinement change. Such materials include, for example, clay under fast (undrained) loading conditions.

*Elgamal, A., L. Yan, Z. Yang and J. P. Conte. (2008). "Three-dimensional seismic response of Humboldt Bay bridge foundation-ground System," *Journal of Structural Engineering*, ASCE, 134(7), 1165-1176.

2 INSTALLATION

UCSDSAND3 and **UCSDCLAY** have been implemented into a number of analysis packages including **FLAC/FLAC3D, LS-DYNA, DIANA** and **Abaqus** and as user defined materials (UDM). Below is information on how to install and use these materials.

2.1. FLAC

Note: The UCSD material files are currently available for 32-bit version of **FLAC**, and will not run for 64 bit.

Steps

1. Visit http://soilquake.net/ucsdsoilmodels/flac.html to download **FLAC_Example.rar** which contains the UCSD material models and corresponding examples files.

2. If you do not already have **FLAC 8**, please visit http://www.itascacg.com/software to download a 32-bit installation file. A free demo version which allows UDM can be found here. This installation may require a restart.

3. Extract all the files in **FLAC_Example.rar** to a desired location.

4. If you have:
   a. FLAC 7
      Move modelucsdclaydemo003.dll and modelucsdsand3demo003.dll to C:\Program Files (x86) \Itasca\FLAC700\Exe32\plugins\models (default location). If you specified a different location for **FLAC**, please move .dll files to the models folder.

   b. FLAC 8
      Move modelucsdclaydemo005.dll and modelucsdsand3demo005.dll to C:\Program Files (x86) \Itasca\FLAC800\Exe32\plugins\models (default location). If you specified a different location for **FLAC**, please move the .dll files to the models folder.

5. Update **FLAC_location.txt** with the location of the FLAC installation.

Input

2.2. LS-DYNA

Steps

1. Visit http://soilquake.net/ucsdsoilmodels/ls_dyna.html to download **LS-DYNA_Example.rar** which contains the UCSD material models and corresponding examples files.

2. If you do not already have **LS-DYNA**, please visit http://www.lstc.com/download/ to obtain the analysis package.

3. Extract all the files in **LS-DYNA_Example.rar** to a desired location.

4. Move **lsdyna_ucsddemosoilmodels.exe** to the main folder of the **LS-DYNA** installation directory.

5. Update **LS_DYNA_location.txt** with the location of the **lsdyna.exe**.

2.3. DIANA

Steps

1. *Coming Soon!*

2.4. Abaqus

Steps

1. Visit http://soilquake.net/ucsdsoilmodels/abaqus.html to download **Abaqus_Example.rar** which contains the UCSD material models and corresponding examples files.

2. If you do not already have **Abaqus**, please visit https://www.3ds.com/ to obtain the analysis package.

3. Extract all the files in **Abaqus_Example.rar** to a desired location.

Input

4. Please move **explicitU-D.dll** (explicit analysis) and **standardU.dll** (implicit analysis) to the **userlib** folder inside the **Abaqus** installation folder. (ex. C:\SIMULIA\Abaqus\userlib). If the path was changed, please move to appropriate folder.

5. Add usub_lib_dir="C:\SIMULIA\Abaqus\userlib" (same path as step 4) to abaqus_v6.env, which can be found in the site folder. (ex. C:\SIMULIA\Abaqus\6.14-2\SMA\site)

6. Update **Abaqus_location.txt** with the location of the Abaqus installation.

3 INPUT

After you download and moved the UCSD material model, you can make use of the **1D_Seismic_Site_Response_Example.xlsm** to perform a site response analysis. A large majority of the of the process has been automated, which includes generation of the input file and initiating the analysis. Below is a description of the necessary inputs for the excel sheet.

3.1. Program

- **Program Name**: Name of the analysis package that should be used with this excel sheet. (Do not change.)

3.2. Input File

- **File Name**: Enter a file name of the project file that will be created. Should be entered without the extension.

3.3. Water Table

- **Height (m)**: Enter the desired water table height. (0 is for ground surface.)

3.4. Rayleigh Damping

For Abaqus/LS-DYNA:

- **Stiffness Proportional**: Enter the stiffness proportional damping term to be used. (Currently stiffness proportional damping is not available for Abaqus.)

- **Mass Proportional**: Enter the mass proportional damping term to be used.

For FLAC:

- **Fraction of Critical Damping**: Enter the damping at the center frequency.

- **Center Frequency**: Enter the frequency with the minimum damping.

3.5. Analysis Parameter

- **Type**: Select for implicit or explicit analysis. This option should be explicit for **FLAC**, and **LS-DYNA**. Implicit should be used for **Abaqus.**

- **Newmark - Beta**: Enter the beta parameter for the Newton method algorithm. (Only available for implicit analysis.)

- **Newmark -** Enter the gamma parameter for the Newton method algorithm. (Only available for implicit analysis.)

3.6. Input Motion

- **File Name**: Select an input motion from a drop-down list.

- **Scale Factor**: The scale factor used to amplify the input motion.

- **Time Step**: Enter the desired time step for the dynamic analysis. Can be set to zero, to allow the time step to be automatically specified.

- **Number of Cycles**: If SineWave is selected, you can enter the number of cycles for the SineWave input.

- **Frequency (Hz.)**: Enter the desired frequency for the SineWave input.

3.7. Soil Properties

Under the section titled soil properties, 4 different soil layers can be defined. The parameters that need to be supplied are as follows:

- **Soil Type**: Select from a drop down list the desired material type. The corresponding properties will populate the table. Additional materials can be added in the material properties sheet.

- **Layer Thickness (m)**: Enter the thickness of the soil layer.

- **Number of Elements**: Enter number of elements for soil layer.

- **Density (Mg/m3)**: Enter the desired density of the soil layer.

Input

- **Shear Modulus (kPa)**: Enter the low-strain shear modulus of the soil layer.
- **Bulk Modulus (kPa)**: Enter the low-strain bulk modulus of the soil layer.
- **Friction Angle (°)**: Enter friction angle for the soil layer.
- **Cohesion (kPa)**: Enter the cohesion for the soil layer.
- **Peak Shear Strain (%)**: Enter the shear strain where the shear strength is reached.
- **Number of Yield Surface**: Enter the number of yield surfaces for the stress-strain backbone curve.
- **Reference Pressure (kPa)**: Enter the confinement at which the shear modulus and bulk modulus are defined.
- **contrac1/contrac2/contrac3**: A non-negative constant defining the rate of shear-induced volume decrease (contraction) or pore pressure buildup. A larger value corresponds to faster contraction rate.
- **dilat1/dilat2/dilat3**: Non-negative constants defining the rate of shear-induced volume increase (dilation). Larger values correspond to stronger dilation rate.
- **liquefac1/liquefac2**: Parameters controlling the mechanism of liquefaction-induced perfectly plastic shear strain accumulation, i.e., cyclic mobility.
- **mType**: Specify 0, 1 or 2 to specify if the shear strength should match triaxial compression (0), triaxial extension (1), or simple shear (2).
- **b1/b2/b3/b4**: Additional contraction parameters.

4 USING THE EXCEL SHEET

Steps

1. If prompted, please enable macros or active content. This Excel sheet makes use of macros to generate the input files and run the analysis using the selected analysis program.

2. Enter desired parameters as explained in the previous section titled "**Input**".

3. Click "**Run**" to create an input file according to entered parameters and start the analysis. This process is completely automated. Please do not click or type until the analysis is started. If you wish to view previously completed runs, please use the **"View Results"** button. For additional information please reference the sections below for the corresponding analysis package.

4.1. FLAC

Note: If this is your first time using **FLAC**, you may be prompted to copy application data. In this case you will need to select an appropriate location and click "**OK**". After this the analysis, can start. You may need to close **FLAC** and press "**Run**" again.

1. When the analysis is completed, you should be able to click the tabs as shown below. Clicking the various tabs will update the screen with the corresponding plots as indicated by the title of the tab.

2. Additionally, you can right click the tab and select edit to change the plot. This will allow you to view responses that are not shown by default (Figure 2). After you select edit you will see various items that can be selected. The currently available histories are shear stress (Effective SXY), shear strain (shearstrain_z), acceleration, displacement, effective vertical stress (Effective SYY), effective horizontal stress (Effective SXX), pore pressure (Grid-point pp) and confinement (conf). The lower most element and nodes are labeled as 1, and increases by 1 as you approach ground surface. This numbering convention is used in the names of the time histories.

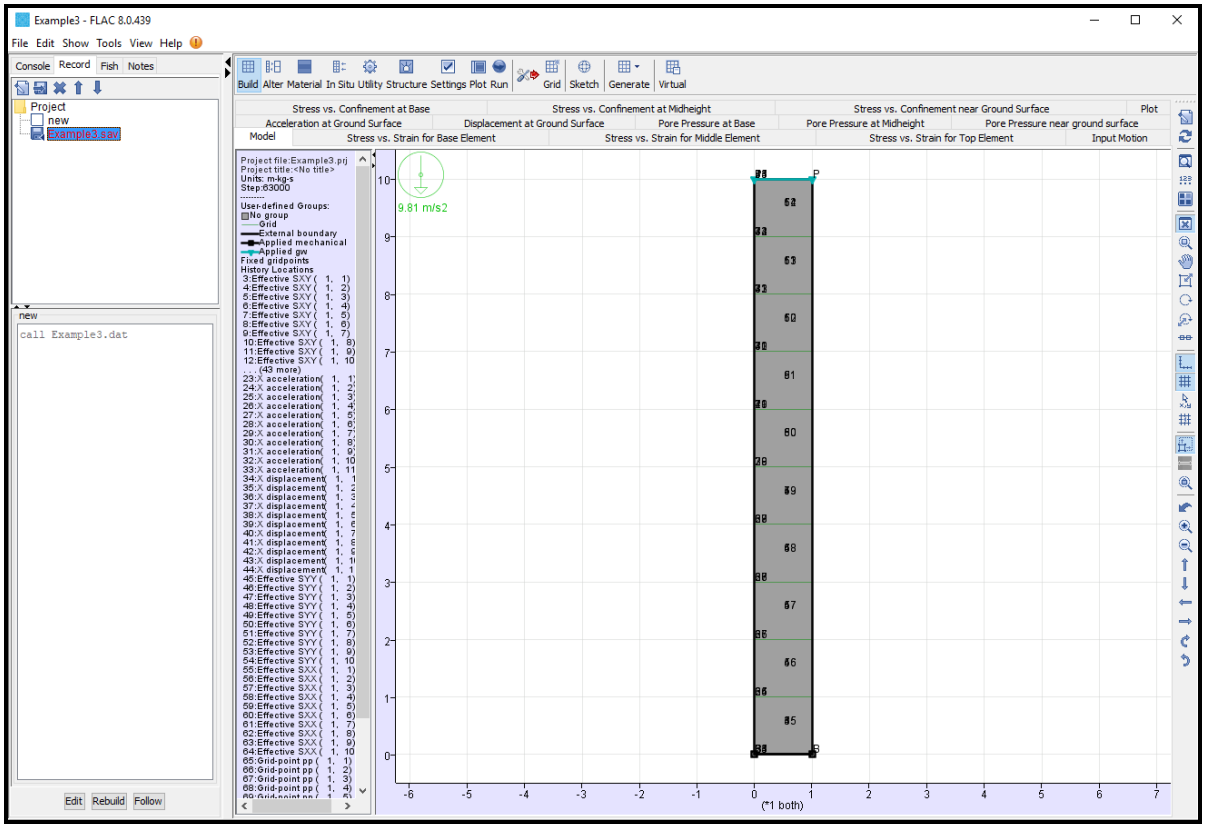

Figure 1 FLAC GUI

Note: To view the data as time histories, you want to change the item in the first column and leave the "Dynamic time" selected in the second.

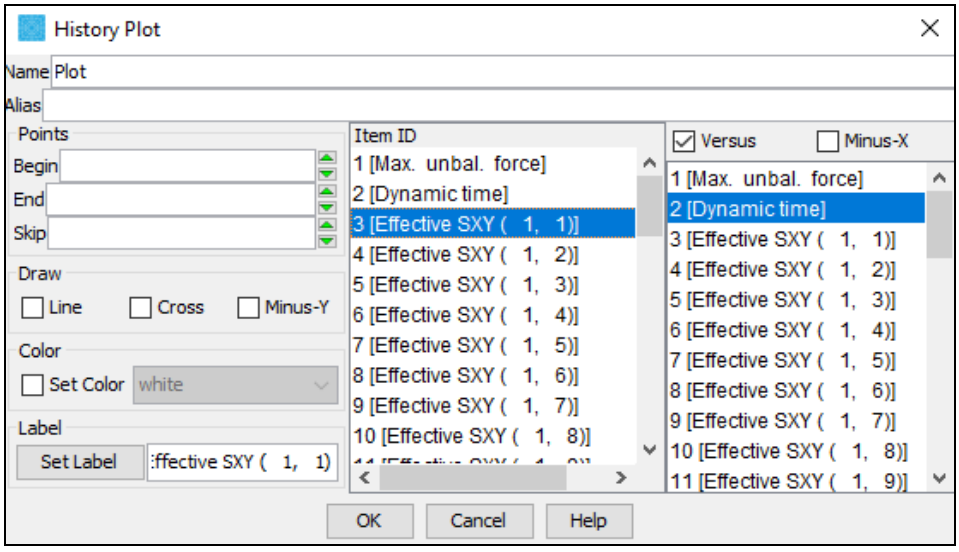

Figure 2 Updating history plot

4.2. LS-DYNA

Steps

1. When the analysis is completed, you can click "**View Results**" to open the **LS-DYNA** interface. The different time histories can be accessed from the nodal and element option. The nodal option includes acceleration and displacement time histories, while the element option contains the stress, strain and pore water pressure time histories (history var#1 and history var#2 for total and excess respectively).

2. After choosing the desired time history you can click a location on the model to view the selected time history for that area.

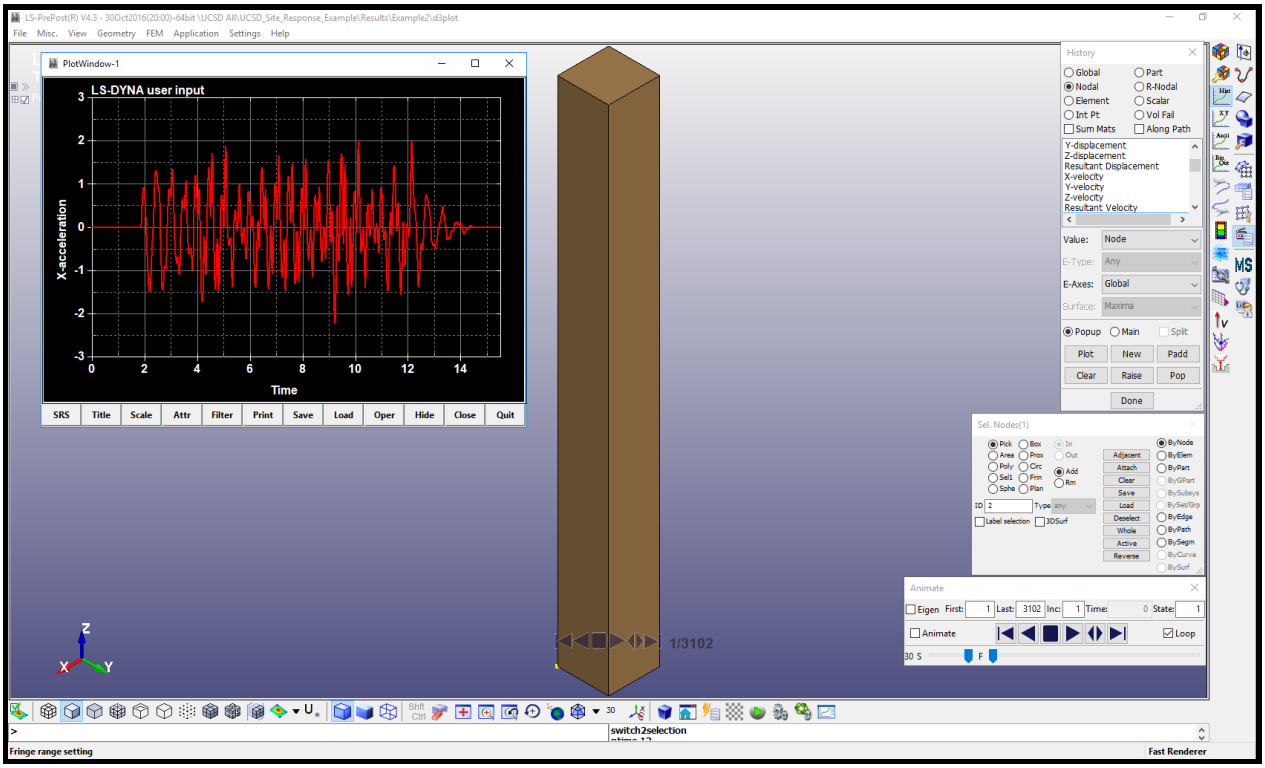

Figure 3 LS-DYNA GUI

4.3. DIANA

Steps

1. *Coming Soon!*

4.4. Abaqus

Steps

1. When the analysis is completed, you can click "**View Results**" to open the **Abaqus** interface. After loading the interface, the acceleration, displacement, strain and stress should be readily available as shown below. Clicking the desired time history and selecting plot will update the **Abaqus** interface.

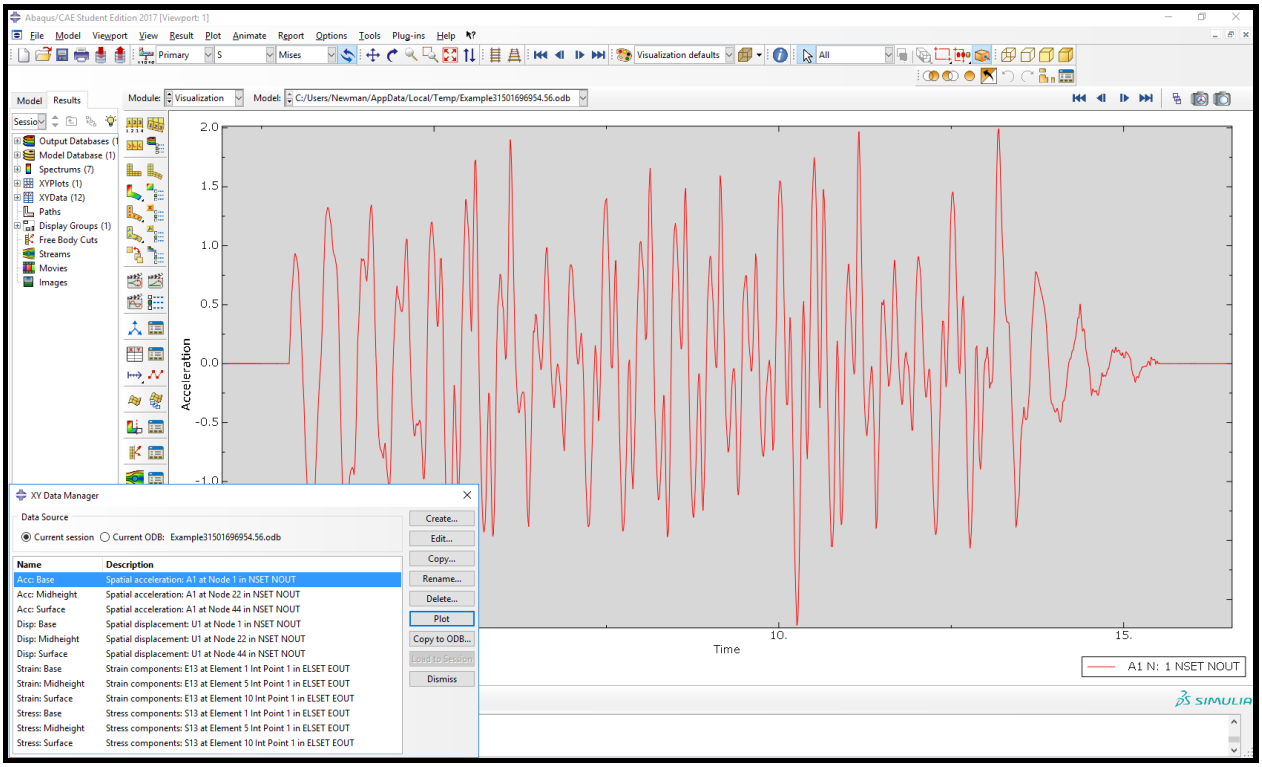

Figure 4 Abaqus GUI