USER MANUAL

PRE- AND POST-PROCESSING MODULES TO FACILITATE ANALYSIS WITH EAGD-84

Arnkjell Løkke,
Department of Structural Engineering
Norwegian University of Science and Technology

/ University of California, Berkeley

June 2013

PURPOSE

The modules described in this document are meant to be used as an addition to the original EAGD-84 program, providing users with the capabilities of pre-processing input and post-processing output from the program in the Matlab scripting language. They do not offer any new functionality to the computational part of the program EAGD-84.

It is assumed that the user is familiar with the use of EAGD-84. Additionally, the user should be familiar with the Matlab scripting language, in particular with numeric arrays and array operations, and with data structures and cell arrays.

All scripts are provided open-source, so the user can make modifications and/or additions to the code as required.

RUNNING THE PROGRAM

1. To run the GUI: Type GUI_EAGD in the Matlab command window, or open and run the script file GUI_EAGD.m. The use of the GUI should be self-explanatory.
2. To run modules in a script interface, see example files Example1.m or Example2.m.

SYSTEM REQUIREMENTS

All the modules and scripts described in this report have been written and tested in a Windows environment using Matlab release 2012a. The scripts should be compatible with older (and newer) versions of Matlab, but modifications to the code might be necessary for use in other environments. Because the scripts make use of basic Windows functionality, they cannot be used on systems running OS X, Linux or any other non-Windows operating systems.

LIMITATIONS

The application of these modules is first and foremost limited by the extent to which EAGD-84 can be used. For practical reasons, a few additional restrictions have been imposed on the possible user input. These should have little influence on a typical use of the program and are described further in the input data description.

EAGD-84 COMPUTER PROGRAM

EAGD-84 [Fenves and Chopra 1984] is a self-contained computer program that numerically evaluates the response of concrete gravity dams to earthquakes, including the effects of dam-water-foundation interaction, water compressibility and reservoir bottom absorption.

The dam monolith is idealized as a two-dimensional assemblage of planar, four-node non-conforming finite elements. Energy dissipation in the dam concrete is represented by constant hysteretic damping. The water impounded in the reservoir is idealized as a fluid domain of constant depth and infinite length in the upstream direction, and at the reservoir bottom, the absorptiveness of the reservoir bottom materials is characterized by a wave reflection coefficient. If the effects of dam-foundation interaction are to be included, the frequency-dependent dynamic stiffness matrix for the foundation region is defined with respect to the degrees-of-freedom of the nodal points at the dam base, computed from standard compliance data provided with the program. Earthquake excitation is defined by two components of free-field ground acceleration in a cross-sectional plane of the dam: the horizontal component transverse to the dam axis, and the vertical component.

Outputs from the program include hydrostatic loads; nodal point displacements and element stresses due to static loads; natural vibration frequencies and mode shapes of the dam (if the foundation is assumed to be rigid) or of an associated dam-foundation system (if dam-foundation interaction is included); complete response histories for stresses and displacements for each finite element; and the peak maximum and minimum principal stress in each finite element and the times at which they occur. The user is referred to the EAGD-84 user manual [Fenves and Chopra 1984] for additional details regarding the system idealization or description of input/output.

Recently, compliance data for the foundation region have been recomputed for an increased number of base nodal points, a finer range of dimensionless frequencies and a closely spaced set of constant hysteretic damping factors [Løkke and Chopra 2013], in particular η_f = 0.01, 0.02 0.03, 0.04, 0.05, 0.06, 0.07, 0.08, 0.09, 0.10, 0.12, 0.14, 0.16, 0.18, 0.20, 0.25, and 0.50; and NBASE ≤ 16. The new data is now provided with the program, and the source code has been updated to allow for the implementation of the extended data set.

Additionally, the EAGD-84 source code has been compiled to a running Windows executable; this file should work on most Windows computers, including 64-bit versions. The updated source code and the Windows executable can be run independently of the pre-and post-processor modules described in this user manual.

ORGANIZATION OF PRE- AND POST-PROCESSING MODULES

Pre-Processing and Program Execution

Use of the pre-processor scripts is organized in the three steps shown in Figure 1: (1) the user defines all the necessary input parameters; (2) the scripts pre-process the user input to compile a correctly formatted EAGD-84 input file; and (3) EAGD-84 is executed and the output is saved. The input parameters required to run an analysis are described in a later chapter.

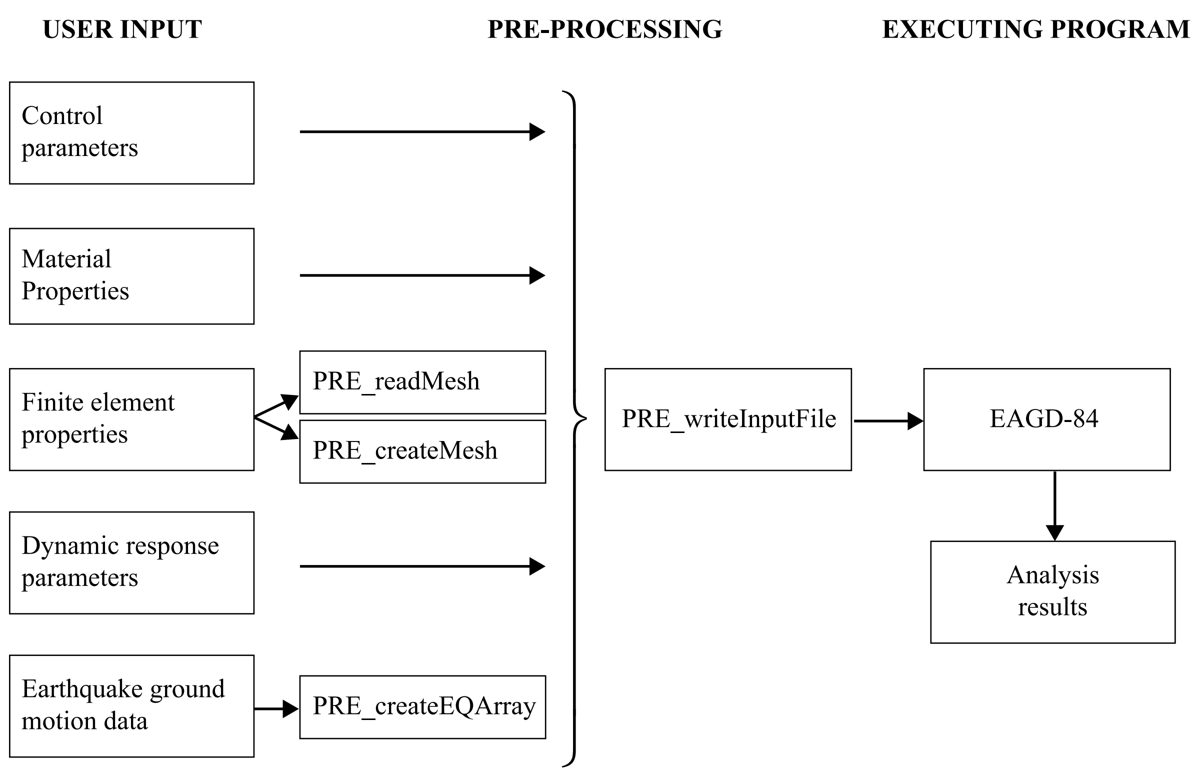

Figure 1 Organization of pre-processing and program execution.

Post-Processing

The output from EAGD-84 is organized in two files: (1) the file 'output', a formatted text file containing the majority of the output; and (2) 'fort.3', an unformatted binary file containing displacements and stresses for every time step in the analysis. This output is read into Matlab workspace by the two functions POST_readOutput and POST_readFort3, and can subsequently be easily accessed for the user to perform post-processing of the results. Figure 2 shows a schematic overview of the post-processing. Note that post-processing can be done independently of the pre-processor; its only requirement is that the two output files from EAGD-84 are present.

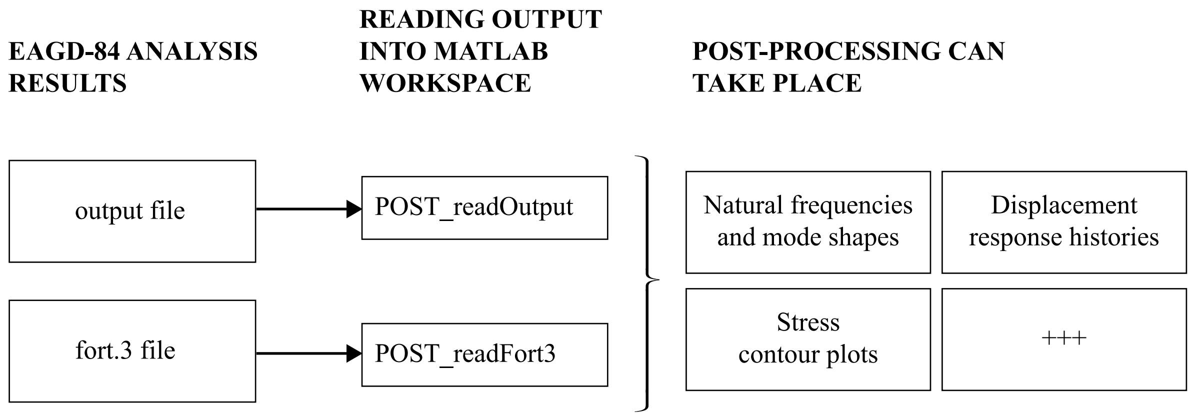

Figure 2 Organization of post-processing.

The majority of the analysis output is stored in the data structure 'Post', the fields comprising this data structure are described in Table 1.

Table 1 Description of fields in data structure 'Post'.

Post.Static	Contains displacements and stresses due to initial static loads, and the hydrostatic load vector.
Post.Vibration	Contains the vibration properties of the system, such as frequencies, mode shapes and frequency response functions.
Post.PrinStrOA	Contains the peak values of the maximum and minimum principal stresses, as well as their time of occurrence.
Post.Stress	Contains the stresses $\sigma_{xx}, \sigma_{yy}, \sigma_{xy}$, as well as the principal stresses σ_1 and σ_2, in each element at every time step of the computed response.
Post.Displacement	Contains the horizontal and vertical displacements of each nodal point at every time step of the computed response.

A set of functions that address basic post-processing tasks are provided, all the plots presented in the examples in this user manual are generated by these functions.

Table 2 Basic plot functions

Plot_Mesh	Plots the finite element idealization of the dam.
Plot_Disp	Plots the displacement response history for a given DOF.
Plot_Modeshape	Plots the mode shape for a given mode number.
Plot_Contour	Plots stress contours for a given stress distribution. Users can choose between filled contour plot in colors or contour lines in B/W.

Graphical User Interface

To further ease the accessibility of EAGD-84, a self-explanatory graphical user interface (GUI) has been developed. The GUI makes use of the functions described in the previous sections to create an EAGD-84 input file, execute EAGD-84, and perform basic post-processing of the results. An example showing the use of the GUI is presented later.

DESCRIPTION OF INPUT DATA

The input data necessary to run the pre-processor scripts are organized in the five categories described in this section. The user is referred to the EAGD-84 user manual for a more thorough description of the input parameters.

Control Parameters

The following input parameters for the control of program execution must be specified:

Table 3 Program control parameters.

IRES	= 0, compute the dynamic response due to earthquake ground motion. = 1, only perform static analysis and compute vibration properties.
ICOMB	= 0, Compute only dynamic response. = 1, Compute dynamic response and combine with response due to the static loads. This will automatically set IGRAV=1.
IGRAV	= 0, do not perform static analysis. = 1, perform static analysis due to weight of the dam and hydrostatic pressure of the impounded water.
IRIG	= 0, foundation rock is flexible, include dam-foundation rock interaction effects. = 1, foundation rock is rigid, exclude dam-foundation rock interaction effects.
PSP	= 0.0, dam and foundation are in generalized plane stress. = 1.0, dam and foundation are in plane strain.

Additionally, a set of control parameters will keep their default value unless they are changed directly in the code for the function PRE_writeInput: IOPR=0; IOPP=1; IGEN=1; ISEL=1; NUMMAT=1; NPRINT=100. The most important implication of this is that only a single material is allowed in the dam discretization.

Material Properties

The following material properties must be specified for the dam, foundation, and water. The user is referred to the EAGD-84 user manual for guidelines on the selection of material input parameters.

Table 4 Material input parameters.

Dam	EC	Young's modulus of elasticity, in ksf, of the dam concrete.
	POISC	Poisson's ratio of the dam concrete.
	DENSC	Mass density, in k-s^2/ft^4, of the dam concrete.
	DAMPC	Constant hysteretic damping factor for the dam concrete.
Foundation	EF	Young's modulus of elasticity, in ksf, of the foundation rock.
	DENSF	Mass density, in k-s^2/ft^4, of the foundation rock.
	DAMPF	Constant hysteretic damping factor for the foundation rock.
Water	ALPHA	Wave reflection coefficient α for the reservoir bottom materials, such as alluvium and sediments.

Finite Element Properties

The user can choose between two options for defining the finite element properties: (1) use the automatic mesh generator PRE_createMesh that idealizes the dam using straight-line segments; or (2) provide a complete definition of the finite element idealization in a separate text file which is read by the function PRE_readMesh.

NOTE: The maximum number of nodal points at the base of the dam, NBASE, that can be selected in the program (if the foundation is assumed flexible) is limited by the maximum number of nodal points for which compliance data is available. With the current data set this is limited to NBASE ≤ 16, selecting a higher value for NBASE may give significantly erroneous results.

(1) Using PRE_createMesh

The function PRE_createMesh automatically generates a finite element idealization of the dam cross section defined by five straight line segments as shown in Figure 3a. The mesh is created top-down, using a constant number of elements along the breadth of the dam. Three examples of finite elements idealizations produced by PRE_createMesh are shown in Figure 3b-d.

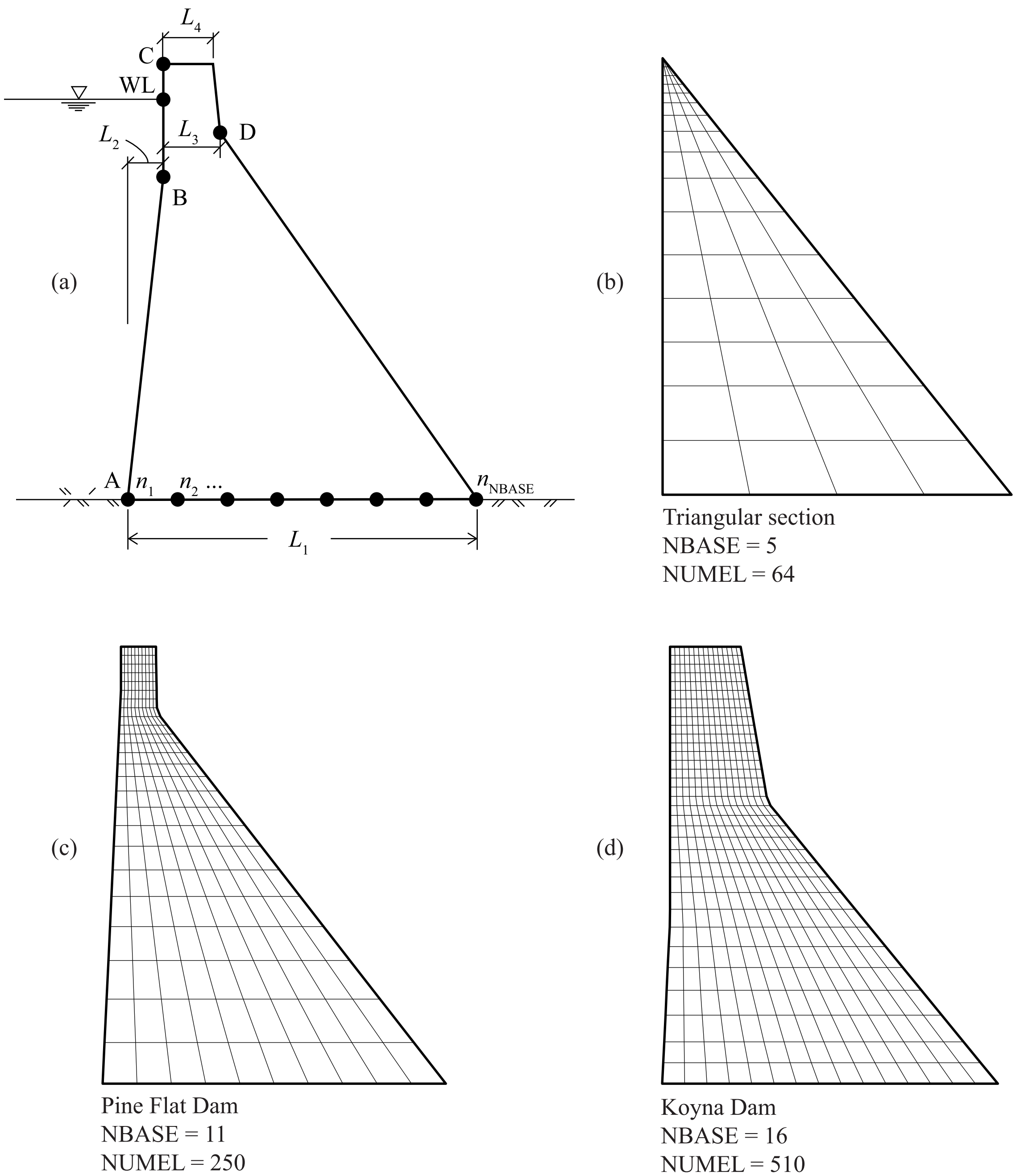

Figure 3 (a) Idealization of dam-cross section using straight-line segments; (b) - (d) examples of finite element idealizations for three different cross sections created by PRE_createMesh.

(2) Using PRE_readMesh

The use of an arbitrary, user-specified, finite element discretization of the dam is administered by the function PRE_readMesh. The following finite element parameters must be specified:

Table 5 Input for function PRE_readMesh.

NUMNP	Number of nodal points in the finite element idealization.
NUMEL	Number of elements in the finite element idealization.
NBASE	Number of nodal points at the dam base, in contact with the foundation. Available compliance data limits NBASE ≤ 16.
Spacing	Spacing, in ft., between the nodal points at the dam base.
WL	Elevation, in ft., of the free-surface of the impounded water.
NPP	Number of nodal points at the upstream face of the dam affected by the impounded water. NPP = 0 indicates an empty reservoir.
fName	File name, including extension, of the text file containing the finite element idealization. Example is 'Meshfile.txt'.

Nodal point coordinates, element definition, water nodal points and base nodal points of the finite element idealization are defined in a separate text file, see Table 6. This file must contain the equivalents of Card Sets E, F, G and H in the EAGD-84 card input (see EAGD-84 user manual). An example of a correct mesh input file is shown in Appendix A.

Table 6 Contents of mesh input file.

Nodal point coordinates	Defines the boundary condition and the *x*-, and *y*-coordinates, in ft., of every nodal point in the finite element idealization.
Element definition	Defines the element connectivity. Nodal points at each element must be numbered in counterclockwise direction
Water nodal points	Specifies the nodal points at the upstream face of the dam affected by the impounded water. If the free-surface water level is between two nodal points, both nodal points must be included.
Base nodal points	Specifies the nodal points at the base of the dam in contact with the flexible foundation. These nodal points must be equally spaced.

Dynamic Response Parameters

Since EAGD-84 computes dynamic response by use of Fourier transformations, a set of dynamic response parameters must be selected:

Table 7 Dynamic response parameters.

NEV	Number of generalized coordinates (i.e. modes) included in the response computation. A general rule is to include all vibration modes that significantly contribute to the dynamic response.
NEXP	Compute the complex frequency response function for the generalized coordinates at N = 2^{NEXP} harmonic excitation frequencies; the response history of the dam is computed at N time intervals.
DT	Time interval, in seconds, for which the response history is computed. Also determines the maximum excitation frequency represented in the response.

The parameter DT determines the maximum excitation frequency *F*, in Hz, represented in the response:

$$F = \frac{1}{2\mathrm{DT}} \tag{1}$$

To ensure that the program computes accurate dynamic response, this frequency should be (i) greater than the frequencies of all the significant harmonics represented in the ground motion, and (ii) large enough to include the range of frequencies over which the dam has significant dynamic response; the latter criterion is met if $F > f_{\mathrm{NEV}}$, where f_{NEV} is the vibration frequency, in Hz, of the highest vibration mode included in the analysis. The parameters DT and NEXP also need to satisfy the two conditions:

$$\mathrm{DT} \cdot 2^{\mathrm{NEXP}} \geq \frac{1}{f_1} \max\left\{25, \frac{1.5}{\eta_s}\right\} \tag{2}$$

$$\mathrm{DT} \cdot 2^{\mathrm{NEXP}} \geq \mathrm{DUR} \tag{3}$$

where f_1 is the fundamental vibration period, in Hz, of the dam-foundation rock system; η_s is the constant hysteretic damping factor of the dam concrete; and DUR is the duration of response history computation determined by the earthquake ground motion data (see Table 8).

The user is referred to the EAGD-84 user manual for a more comprehensive discussion of the selection of dynamic response parameters.

Earthquake Ground Motion Data

The horizontal and vertical components of earthquake ground motions are read and converted into the correct EAGD-84 format by the function PRE_createEQArray. The following ground motion parameters must be defined:

Table 8 Earthquake ground motion parameters.

IHV	= 0, Compute response due to the horizontal component, only, of the ground motion. = 1, Compute response due to the vertical component, only, of the ground motion. = 2, Compute response due to the horizontal and vertical components, simultaneously, of the ground motion.
NUMREC	Number of ordinates in the ground motion record(s). The number of ordinates must be the same for both horizontal and vertical ground motion records.
dt	Time step of the ground motion record(s). The time step must be constant and the same for both horizontal and vertical ground motion records. Together, dt and NUMREC determines duration of response history computation: $\mathrm{DUR} = \mathrm{NUMREC} \cdot \mathrm{dt}$.
SFAC	Scale factor for ground motion records. The scale factor must be the same for both horizontal and vertical ground motion records.
hName	File name, including extension, of the horizontal ground motion file. Example input is 'horzacc.txt'.
hNumHead	Number of header lines (rows containing non-acceleration values) in the horizontal ground motion record.
vName	File name, including extension, of the vertical ground motion file. Example input is 'vertacc.txt'.
vNumHead	Number of header lines (rows containing non-acceleration values) in the vertical ground motion record.

The acceleration files must contain acceleration values *only* (i.e. any time intervals must not be present in the file), and the acceleration values must be in units of g (acceleration due to gravity) and have a constant time step. An example of a correct acceleration file is shown in Figure 4, downloaded from the PEER Ground Motion Database. Note that the acceleration file is not limited to having any specific number of columns, i.e., even a file containing a single vector of acceleration values can be used.

```
  PEER NGA Rotated Accelerogram (November 1, 2007)
 H1 for rotation: PARKFIELD 06/28/66 04:26, CHOLAME #12, 050
 rotation angle - clockwise   181.1
     4411   0.01000 NPTS, DT
-0.8415830E-03 -0.9355882E-03 -0.5400591E-03  0.1997773E-04 -0.1220169E-03
-0.1507867E-03 -0.1669972E-03 -0.1494014E-03 -0.9529132E-04 -0.3359684E-04
-0.8635408E-05 -0.4068283E-04 -0.1043561E-03 -0.1503497E-03 -0.1442841E-03
-0.9426178E-04 -0.3950542E-04 -0.1102707E-04 -0.1263175E-04 -0.4120144E-04
-0.1083362E-03 -0.2044567E-03 -0.2776437E-03 -0.2968528E-03 -0.2720504E-03
-0.2117007E-03 -0.1153847E-03  0.1810423E-04  0.1784976E-03  0.3069022E-03
 0.3158494E-03  0.1869128E-03  0.1013242E-04 -0.7304514E-04  0.2191915E-04
 0.2212701E-03  0.3267031E-03  0.2219068E-03  0.4100883E-04  0.3318371E-04
 0.2173917E-03  0.3488348E-03  0.3288004E-03  0.3233982E-03  0.3729555E-03
 (...)
```

Number of header lines = 4

Acceleration values, in g

Figure 4 Example of acceleration input file.

EXAMPLES

Example 1: Using Scripts to Analyze Idealized Dam

The following example presents the necessary steps for performing a dynamic analysis of an idealized concrete gravity dam cross-section subjected to the horizontal component of Taft ground motion by using the modules directly in a Matlab script file.

The script file shown in Figure 6 is used to create the EAGD-84 input file, execute the program, and load all output into the Matlab workspace. This file contains all the necessary input for the program to run. The mesh generated by the automatic mesh generator is shown in Figure 5.

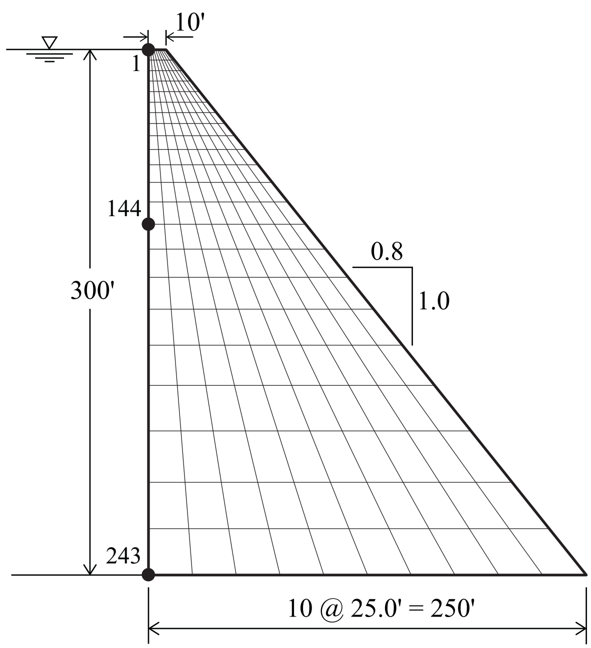

Figure 5 Mesh for idealized dam generated by PRE_createMesh, consisting of 220 (22 x 10) quadrilateral four-node elements.

```
%-------------------------------------------------------------------------%
% EXAMPLE 1: DYNAMIC ANALYSIS OF IDEALIZED CROSS SECTION
%-------------------------------------------------------------------------%
clc; clear all; close all

%% DEFINE INPUT

% 1. Define control parameters
IRES=0; ICOMB=0; IGRAV=0; IRIG=0; PSP=0.0;

% 2. Define material properties
EC=5.76e5; DENSC=4.8e-3; POISC=0.20; DAMPC=0.04; % Dam
EF=5.76e5; DENSF=5.1e-3;             DAMPF=0.04; % Foundation
ALPHA=0.75;                                      % Water

% 3. Define FE geometry using automatic mesh-generator
L1=250; L2=0; L3=10; L4=10;                        % lengths, in ft
elA=0; elB=300; elC=300; elD=300; elWL=300;        % elevations, in ft
NBASE=11;
ft=1.0;                                            % Conversion factor to ft

% 4 Define dynamic response parameters
NEV=12; NEXP=12; DT=0.01;

% 5. Define EQ ground motion data
IHV=0; NUMREC=3000; dt=0.01; SFAC=1;
hName='Taft_horz.txt'; hNumHead=1;

%% CREATE INPUT FILE

% Create finite element idealization
[COORD Element WL Spacing NUMNP NUMEL NBASE,
 NPP WatNodes BaseNodes] = PRE_createMesh(L1,L2,L3,L4, ...
 elA,elB,elC,elD,elWL,NBASE,ft);

% Create earthquake array
[EQArrayH EQArrayV] = PRE_createEQArray(IHV, NUMREC, dt, SFAC, hName, hNumHead);

% Create input file
PRE_writeInput(IRES,ICOMB,IGRAV,IRIG,PSP,EC,POISC,DENSC,DAMPC, ...
 EF,DENSF,DAMPF,ALPHA,NUMNP,NUMEL,NBASE,Spacing,WL,NPP,COORD,Element, ...
 WatNodes,BaseNodes,NEV,NEXP,DT,IHV,NUMREC,dt,EQArrayH,EQArrayV);

%% RUN EAGD-84
RUN_E1A

%% READ OUTPUT TO WORKSPACE
clear all;

% Read output file
[NUMNP NUMEL NBASE COORD Element Post] = POST_readOutput;

% Read fort.3 file
[NUMNP NUMEL Post] = POST_readFort3(Post);
```

Figure 6 Matlab script file to create input file, run EAGD-84, and read output data into the Matlab workspace.

Once the script file has been run and the output data is read into the Matlab workspace, results can easily be accessed and post-processed using the utility functions (Table 2) provided with the post-processor scripts, in addition to available built-in Matlab utility functions. A few examples of such post-processing of the program output are presented below,

- The first four mode shapes, including the corresponding vibration periods, of the dam on flexible foundation are plotted in Figure 7.
- The horizontal and vertical displacements, relative to the free-field ground motion, at three levels on the upstream face of the dam (nodal points 1, 144 and 243, see Figure 5) due to the horizontal component of Taft ground motion are shown in Figure 8.
- The distribution of envelope values of the maximum principal stresses in the dam, excluding stresses due to static loads, is plotted in Figure 9.

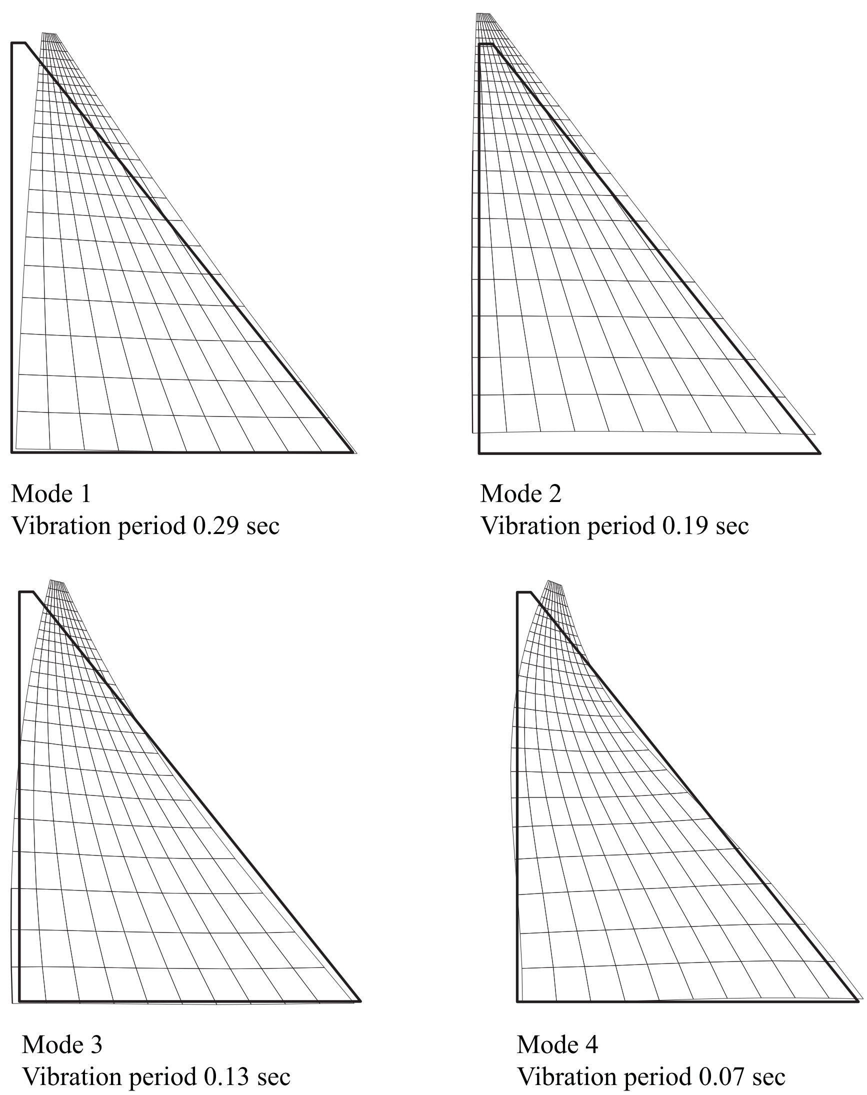

Figure 7 First four vibration modes for the associated dam-foundation system.

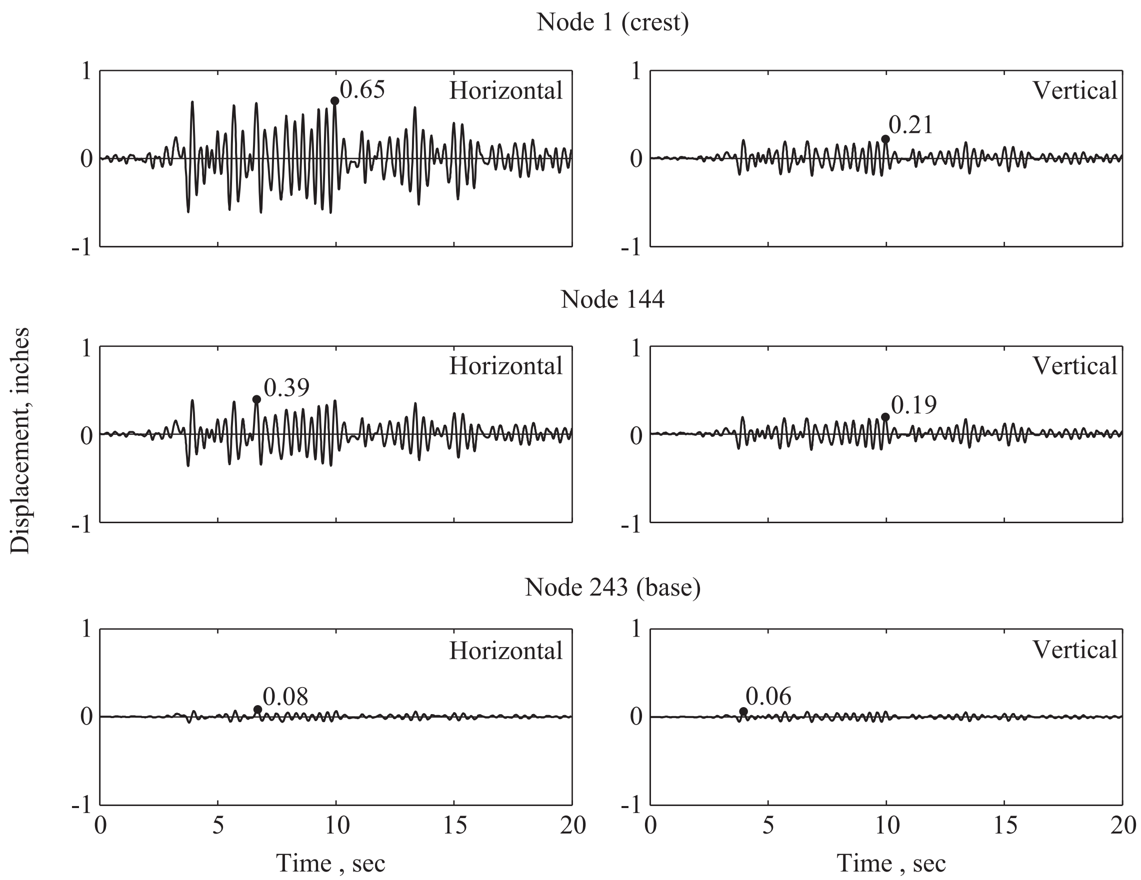

Figure 8 Displacement response histories for nodal points 1, 144 and 243.

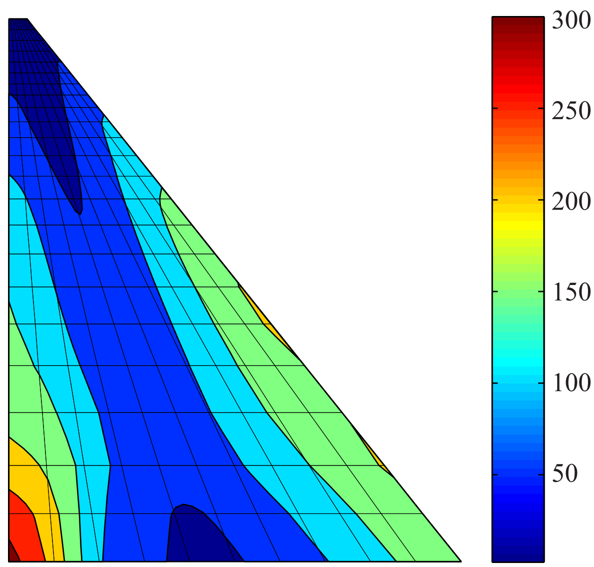

Figure 9 Filled contour plot showing envelope values of maximum principal stresses; initial static stresses are excluded.

Example 2: Using GUI to Analyze Pine Flat Dam

The following example shows how the GUI can be used to perform a dynamic analysis of Pine Flat Dam subjected to the horizontal and vertical components of Taft ground motion, simultaneously. The mesh used in the analysis is produced by the automatic mesh generator and is shown in Figure 3c. The input parameters are chosen to be the same as the example analysis presented in Fenves and Chopra (1984).

Figure 11 shows the GUI with the input parameters for Pine Flat Dam, units are in ft., ksf, and k-s^2/ft^4. The earthquake ground motion parameters and the names of the files containing acceleration values are defined in the separate window shown in Figure 10. These two windows define all input parameters necessary to run the analysis.

The GUI is run in three steps: (1) the EAGD-84 input file is created, (2) EAGD-84 is executed, and (3) the results from EAGD-84 are read into the Matlab workspace and post-processed. The response histories for horizontal and vertical displacement of nodes 1, 166 and 276, corresponding to locations at the crest, at 2/3 height, and at the base of the dam, respectively, are plotted in Figure 12; the envelope of the maximum principal stresses in the dam over the duration of the ground motion is plotted in Figure 13.

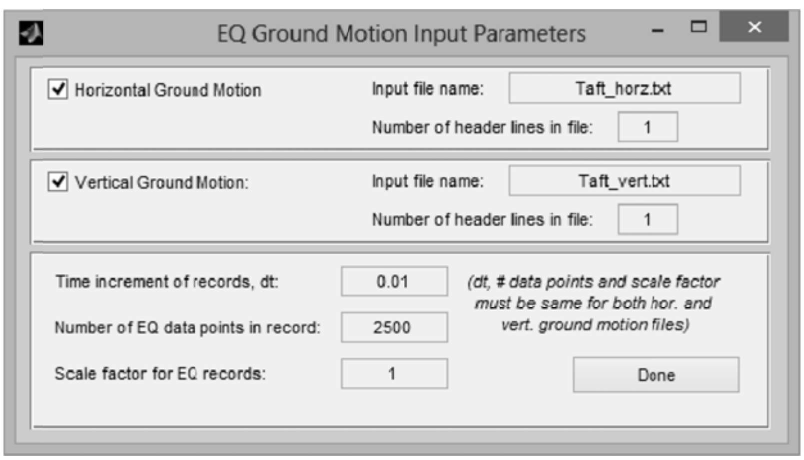

Figure 10 Window for defining earthquake ground motion in GUI.

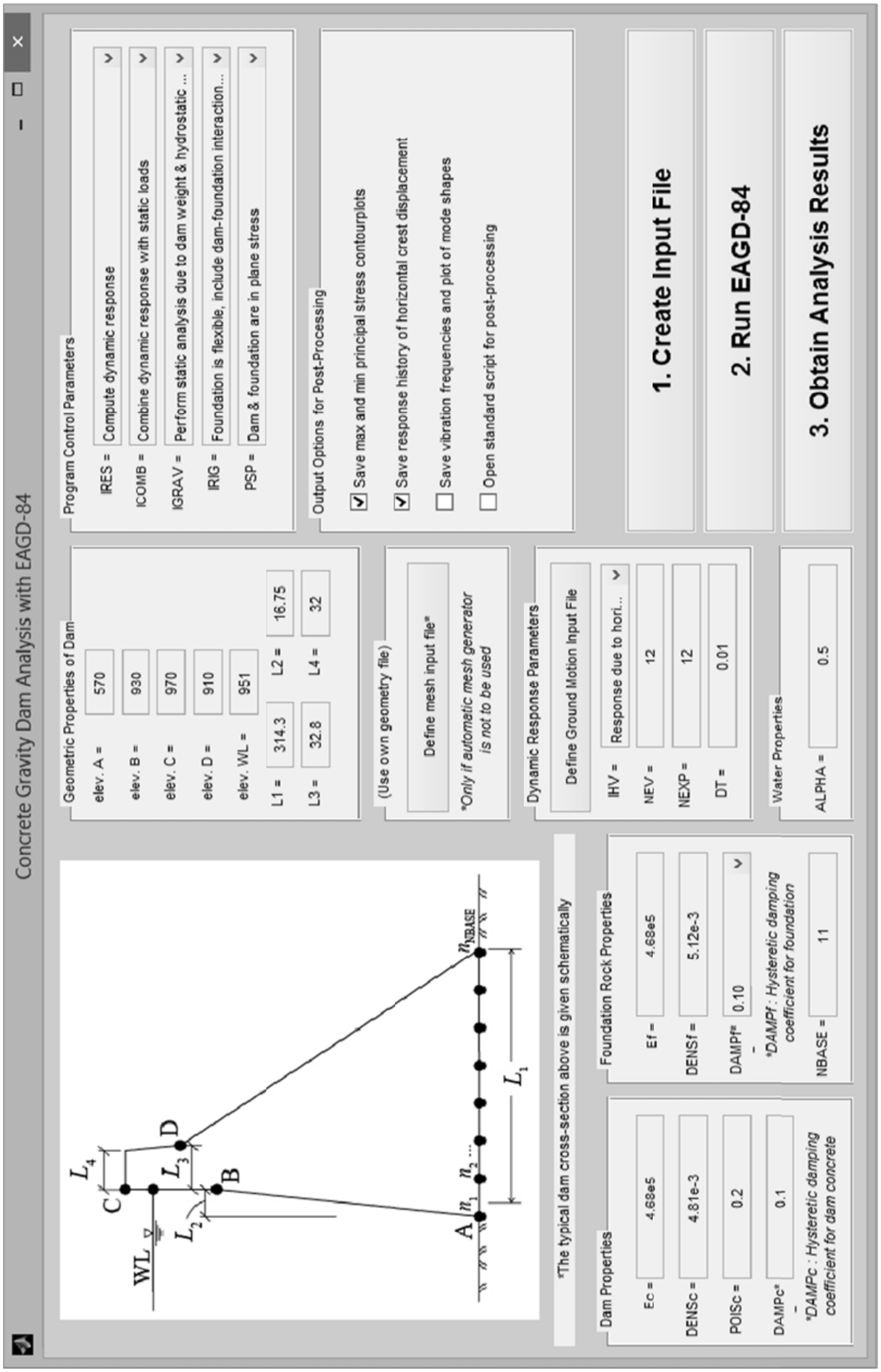

Figure 11 Main user window with input for Pine Flat Dam.

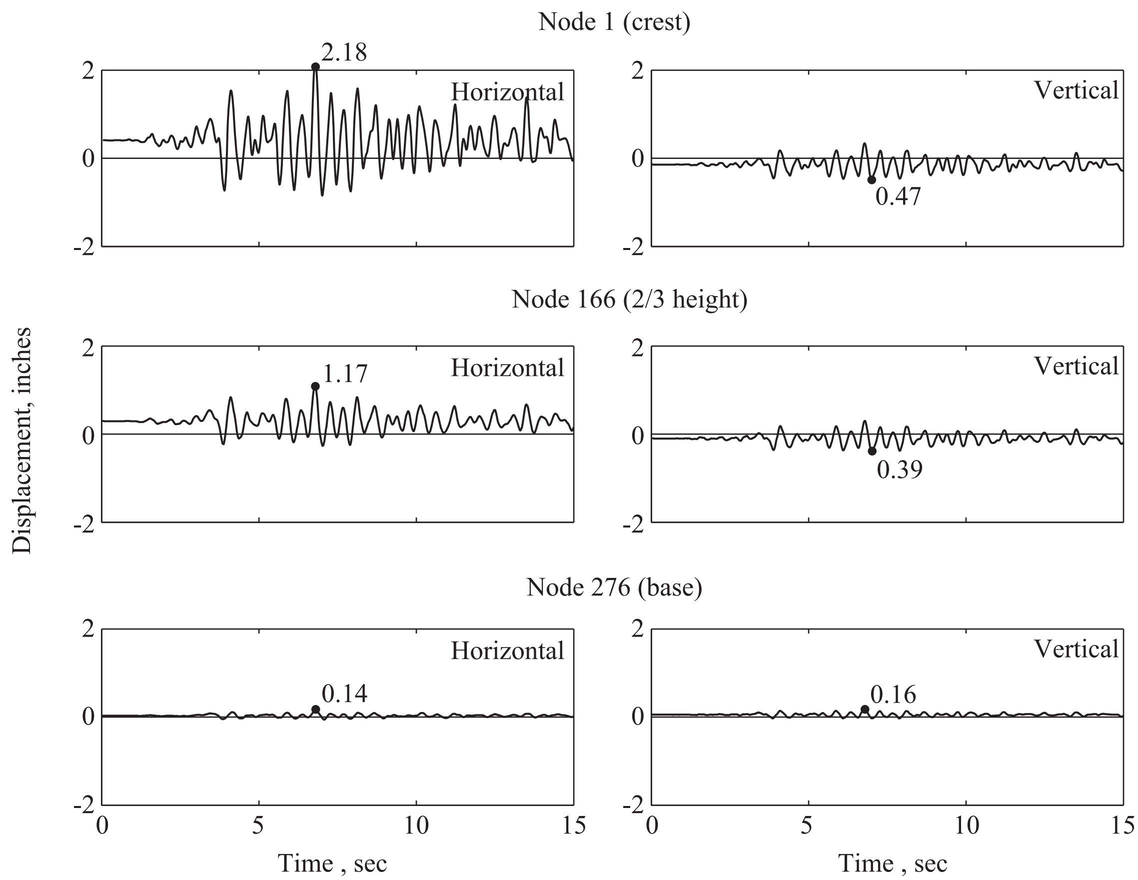

Figure 12 Displacement response of Pine Flat Dam due to horizontal and vertical components, simultaneously, of Taft ground motion; initial static displacements are included.

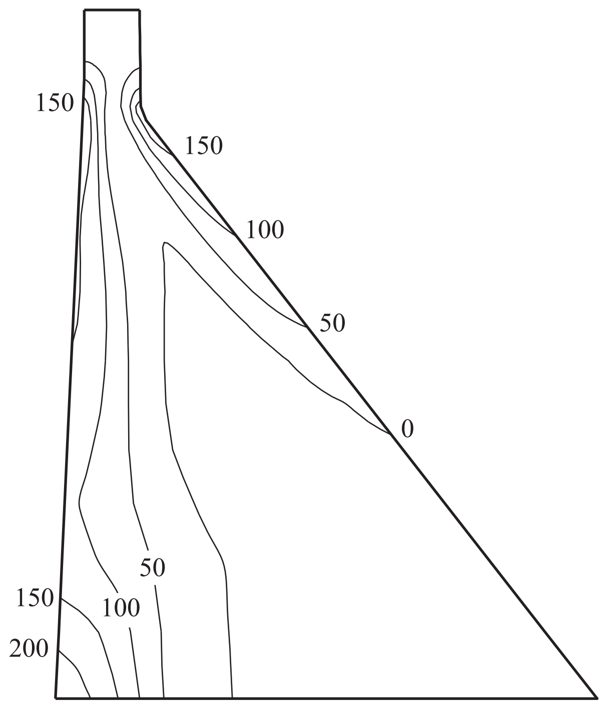

Figure 13 B/W contour plot showing envelope values of maximum principal stresses, in psi, in Pine Flat Dam due to horizontal and vertical components, simultaneously, of Taft ground motion; initial static stresses are included.

By comparing the results presented in Figures 12 and 13 with the results presented in Fenves and Chopra (1984), it is apparent – as expected – that the results are essentially identical except for minor differences due to small variances in the mesh used to compute the results.

REFERENCES

Fenves, G., and A.K. Chopra (1984). EAGD-84: A computer program for earthquake response analysis of concrete gravity dams, *Report No. UCB/EERC-84/11,* Earthquake Engineering Research Center, University of California, Berkeley, Calif., 78 pgs.

Løkke, A. and A. K. Chopra, Response spectrum analysis of concrete gravity dams including dam-water-foundation interaction, *Submitted for publication,* Pacific Earthquake Engineering Research Center, University of California, Berkeley, 2013.

APPENDIX A: EXAMPLE OF MESH INPUT FILE

Definition of nodal points:

```
      1      0.00    16.750   400.000
      2      0.00    20.750   400.000
      3      0.00    24.750   400.000
      4      0.00    28.750   400.000
      5      0.00    32.750   400.000
      6      0.00    36.750   400.000
      7      0.00    40.750   400.000
      8      0.00    44.750   400.000
      9      0.00    48.750   400.000
     10      0.00    16.750   383.000
     11      0.00    20.750   383.000

(...)

    152      0.00   253.390    32.000
    153      0.00   289.360    32.000
    154      0.00     0.000    -0.000
    155      0.00    39.290    -0.000
    156      0.00    78.580    -0.000
    157      0.00   117.870    -0.000
    158      0.00   157.160    -0.000
    159      0.00   196.450    -0.000
    160      0.00   235.740    -0.000
    161      0.00   275.030    -0.000
    162      0.00   314.320    -0.000
```

Definition of element connectivity:

```
    1       1     10     11     2            1
    2       2     11     12     3            1
    3       3     12     13     4            1
    4       4     13     14     5            1
    5       5     14     15     6            1
    6       6     15     16     7            1
    7       7     16     17     8            1
    8       8     17     18     9            1
    9      10     19     20    11            1
   10      11     20     21    12            1

(...)

  126     141    150    151   142            1
  127     142    151    152   143            1
  128     143    152    153   144            1
  129     145    154    155   146            1
  130     146    155    156   147            1
  131     147    156    157   148            1
  132     148    157    158   149            1
  133     149    158    159   150            1
  134     150    159    160   151            1
  135     151    160    161   152            1
  136     152    161    162   153            1
```

Definition of nodal points in contact with water:

```
  10   19   28   37   46   55   64   73   82   91  100  109  118  127  136
 145  154
```

Definition of nodal points at the dam base:

```
 154  155  156  157  158  159  160  161  162
```

Figure 14 Example (excerpts) of correct mesh input file. In this file: NUMNP=162; NUMEL=136; NBASE=9; Spacing=39.29; WL=381.0; NPP=17.