Civil Engineering

UC UNIVERSITY OF CANTERBURY
Te Whare Wānanga o Waitaha
CHRISTCHURCH NEW ZEALAND

Ruaumoko Manual

Volume 4:
User Guide to Associated Programs

Author:
Athol J. Carr

UNIVERSITY OF CANTERBURY
Te Whare Wānanga o Waitaha
CHRISTCHURCH NEW ZEALAND

Department of Civil Engineering **COMPUTER PROGRAM LIBRARY**

Program name:– **DYNAPLOT**	Program type:– Post-Processor for RUAUMOKO	Program code:– ANSI Fortran77
Author:– Athol J Carr		Date:– April 28, 2007

DYNAPLOT

Result plotting for RUAUMOKO-2D and RUAUMOKO-3D

Purpose

This program is designed to plot the time history results or the hysteresis loops computer by the non-linear dynamic analysis programs RUAUMOKO-2D and RUAUMOKO-3D. The output is in the form of time-history plots of nodal displacements or forces or member actions or deformations or combinations of the above quantities. Alternatively hysteresis loops may be plotted for members or nodes.

Method of Analysis

The program generates each graph, one by one, requesting the control information for each graph then scanning the Post-processor file generated by RUAUMOKO-2D or RUAUMOKO-3D to select the required function to plot. The program can only plot the results from one non-linear analysis at a time.

Data for DYNAPLOT

All data for DYNAPLOT is prompted for by the program using the TinyCLIP command processor. For help on the input data type **$HELP** at any prompt. To get help on the use of the DYNAPLOT programme type **HELP** or **?** At any prompt.

Control Data for Post-Processor Files

The number of nodes and members with data stores in the **.RES** file is recorded at the beginning of the file together with the time-step between stored items are recorded as the first item in the **.RES** file. The structure geometry and mode shapes are also saved at the beginning of the file.
Running the post-processor program DYNAPLOT.

To run the program call the program by the method appropriate to your operating system. On a personal computer just type **DYNAPLOT** assuming that the files **DYNAPLOT.EXE** and **FQWIN.HLP** are in your current directory or path.

In Microsoft Windows operating systems another option is to create a shortcut on the desktop and for this purpose a suitable icon for DYNAPLOT, **Dynaplot.ico**, is supplied with the program.

The program prompts for responses to a series of questions. Default responses, where appropriate, are enclosed in square brackets, **[ ]**. File names must match the conventions of your operating system but file names, with paths where necessary, must not exceed 60 characters in length and must not contain blanks.

The first question asks for the name of the output file. The default is the computer console or terminal screen. This is normally the response for most users, however if an output file is created then the envelope values of each graph line drawn is output to that file.

The second question asks for the name of the DYNAPLOT post-processor file, do not supply the extension part of the name. This file has been given the extension **.RES** or **.RAS** by RUAUMOKO-2D or RUAUMOKO-3D and DYNAPLOT will attempt to open a file with the extension **.RES** but if that does not exist will attempt to open the file with the extension **.RAS**. If a file with the given name and path with either extension cannot be found the program will again prompt for the file name, and path. Once the file has been found DYNAPLOT will scan the file to determine whether it was created by RUAUMOKO-2D or RUAUMOKO-3D. The program DYNAPLOT will also scan the file to find the duration of the data stored in the file and if this is less that the duration **TIME** supplied in the RUAUMOKO data the user is informed of the duration that is available for plotting. The program will then ask if colour graphics is wanted. The program then prompt following the normal options in DYNAPLOT. Please see the manual pages for DYNAPLOT.

Hard copies of the plots.

In Microsoft Windows operating systems to get hard copies of the graphs use the pull down 'file' menu and select the Print or Save options to send the graph to the printer or to save the plot as a bitmap file (**.BMP)**.
On unix systems using GKS graphics select the Hard-copy option from the Choice window

Output from RUAUMOKO-2D post-processor file

The data that can be plotted consists of the data stored in the RUAUMOKO-2D post-processor file for which the default name is **RUAUMOKO.RES** or **RUAUMOKO.RAS**. The content of the file is controlled by the parameter **IOUT** in the program RUAUMOKO. At every time-step stored in the post processor file the following data is stored.

a. Analysis Data (17 items+Time)

1	=	X and Y ground accelerations	(non-zero only for accelerogram input)
2	=	X and Y ground velocities	(non-zero only for accelerogram input)
3	=	X and Y ground displacements	(non-zero only for accelerogram input)
4	=	Kinetic Energy	
5	=	Damping Work Done	
6	=	Strain Energy	
7	=	Applied Work	
8	=	Plastic Work	
9	=	Ductility	(only meaningful for Adaptive Pushover)
10	=	Effective X and Y displacement	(only meaningful for Adaptive Pushover)
11	=	Total X and Y Applied Forces	
12	=	Total Z Applied Moment	

b. For each node the following items are stored: (controlled by **IOUT** in RUAUMOKO-2D) (18 items)

1	=	X and Y displacements,	Z rotation
2	=	X and Y velocities,	Z rotational velocity
3	=	X and Y accelerations,	Z angular acceleration
4	=	X and Y applied forces,	Z applied moment.
5	=	X and Y damping forces,	Z damping moment
6	=	X and Y inertia forces,	Z inertia moment

c. For each member the following items are stored: (controlled by **IOUT** in RUAUMOKO-2D)

For Frame (Beam) members (17, 27 or 21 items)

1	=	Axial force,	Axial Strain,	% Stiffness,	Energy,	Damage Index
2	=	Moment,	Curvature,	% Stiffness,	Energy,	Damage Index (at End 1)
3	=	Moment,	Curvature,	% Stiffness,	Energy,	Damage Index (at End 2)
4	=	Shear-end1,	Shear-end2,			
5	=	Moment,	Curvature,	% Stiffness,	Energy,	Damage Index (at Hinge 3)
6	=	Moment,	Curvature,	% Stiffness,	Energy,	Damage Index (at Hinge 4)
7	=	Shear Dis1,	Shear Dis2,	% Shear Stiff,	Shear Energy	
8	=	Theta-1,	Theta-2,	% Flex-1,	% Flex-2	

Note: Groups 5 and 6 are for 4 Hinge Beam only.
Group 7 are for In-elastic Beam-Shear option only
Group 8 are for In-elastic Shear-Link model only.

For Spring members (15 items)

1	=	x and y forces,	z moment
2	=	x and y displacements,	z rotation
3	=	x and y % stiffnesses,	z % rotational stiffness
4	=	x and y energies,	z rotational energy
5	=	x and y damage indices	z rotational damage index

For Damper members (9 items)

1	=	x and y forces,	z moment
2	=	x and y velocities,	z rotational velocity
3	=	x and y % stiffnesses,	z % rotational stiffness

For Wall members (13 items)

1	=	Axial force and
2	=	Bending Moments at End 1 and End 2
3	=	Axial strain
4	=	Curvatures at End 1 and End 2
5	=	% axial stiffness
6	=	% flexural stiffness at End 1 and End 2
7	=	Shear Forces at End 1 and End 2
8	=	Neutral axis eccentricities at End 1 and End 2

For Tendon members (2 items)

1	=	Longitudinal or x Force
2	=	Longitudinal or x Deformation

For Contact members (5 items)

1	=	Contact or axial force
2	=	Damping Force
3	=	Friction force
4	=	Displacement in contact direction
5	=	% stiffness of spring

For Hybrid Quadrilateral finite elements (12 items)

1	=	σ-xx, σ-yy, τ-xy, at node I
2	=	σ-xx, σ-yy, τ-xy, at node J
3	=	σ-xx, σ-yy, τ-xy, at node K
4	=	σ-xx, σ-yy, τ-xy, at node L

For Masonry Panel Elements (15 items)

1	=	Force, Displacement and % Stiffness for Strut 1
2	=	Force, Displacement and % Stiffness for Strut 2
3	=	Force, Displacement and % Stiffness for Strut 3
4	=	Force, Displacement and % Stiffness for Strut 4
5	=	Force, Displacement and % Stiffness for Shear Strut

For Foundation Beam Elements (24 items)

1	=	Force, Deformation, % stiffness at Axial spring point 1
2	=	Force, Deformation, % stiffness at Axial spring point 1
3	=	Force, Deformation, % stiffness at Normal spring point 1
4	=	Force, Deformation, % stiffness at Normal spring point 2
5	=	Force, Deformation, % stiffness at Normal spring point 3
6	=	Force, Deformation, % stiffness at Normal spring point 4
7	=	Force, Deformation, % stiffness at Shear spring point 1
8	=	Force, Deformation, % stiffness at Shear spring point 2

Output from RUAUMOKO-3D post-processor file

The data that can be plotted consists of the data stored in the RUAUMOKO-3D post-processor file for which the default name is **RUAUMOKO.RES** or **RUAUMOKO.RAS**. The content of the file is controlled by the parameter **IOUT** in the program RUAUMOKO-3D. At every time-step stored in the post processor file the following data is stored.

a. Analysis Data (24 items+Time)

1	=	X, Y and Z ground accelerations	(non-zero only for accelerogram input)
2	=	X, Y and Z ground velocities	(non-zero only for accelerogram input)
3	=	X, Y and Z ground displacements	(non-zero only for accelerogram input)
4	=	Kinetic Energy	
5	=	Damping Work Done	
6	=	Strain Energy	
7	=	Applied Work	
8	=	Plastic Work	
9	=	Ductility	(only meaningful for Adaptive Pushover)
10	=	Effective X, Y and Z displacements	(only meaningful for Adaptive Pushover)
11	=	Total X, Y and Z Applied Forces	
12	=	Total X, Y and Z Applied Moments	

b. For each node the following items are stored: (controlled by **IOUT** in RUAUMOKO-3D) (36 items)

1	=	X, Y and Z displacements,	X, Y and Z rotations
2	=	X, Y and Z velocities,	X, Y and Z rotational velocities
3	=	X, Y and Z accelerations,	X, Y and Z angular accelerations
4	=	X, Y and Z applied forces,	X, Y and Z applied moments.
5	=	X, Y and Z damping forces,	X, Y and Z damping moments
6	=	X, Y and Z inertia forces,	X, Y and Z inertia moments

c. For each member the following items are stored: (controlled by **IOUT** in RUAUMOKO-3D)

For Frame (Beam) members (34 or 42 items)

1	=	Axial force,	Axial Strain,	% Stiffness,	Energy, Damage Index		
2	=	Moment-zz,	Curvature-zz,	% Stiffness,	Energy, Damage Index	(at End 1)
3	=	Moment-zz,	Curvature-zz,	% Stiffness,	Energy, Damage Index	(at End 2)
4	=	Torque,	Twist,	% Stiffness,	Energy, Damage Index	
5	=	Moment-yy,	Curvature-yy,	% Stiffness,	Energy, Damage Index	(at End 1)
6	=	Moment-yy,	Curvature-yy,	% Stiffness,	Energy, Damage Index	(at End 2)
7	=	Shear-y-1,	Shear-y-2,	Shear-z-1,	Shear-z-2	
8	=	γ-y at 1	γ-y at 2,	γ-z at 1,	γ-z at 2	
9	=	% Stiff-y,	%Stiff-z,	Energy-y,	Energy-z	
10	=	Theta-y-1,	Theta-y-2,	Theta-z-1,	Theta-z-2	
11	=	% Flex-y-1,	% Flex-y-2,	% Flex-z-1,	% Flex-z-2	

Note: Groups 8 and 9 are for the In-elastic Beam-Shear option only
Groups 10 and 11 are for the In-elastic Shear-Link model only.

For Spring members (30 items)

1	=	x, y and z forces,	x, y and z moments
2	=	x, y and z displacements,	x, y and z rotations
3	=	x, y and z % stiffnesses,	x, y and z % rotational stiffnesses
4	=	x, y and z energies,	x, y and z rotational energies
5	=	x, y and z damage indices	x, y and z rotational damage indices

For Damper members (18 items)

1	=	x, y and z forces,	x, y and z moments
2	=	x, y and z velocities,	x, y and z rotational velocities
3	=	x, y and z % stiffnesses,	x, y and z % rotational stiffnesses

For Tendon members (2 items)

1	=	Longitudinal or x Force
2	=	Longitudinal or x Deformation

For Contact members (18 items)

1	=	Contact or axial force,	y and z friction forces
2	=	Damping Force,	y and z spherical bearing forces
3	=	x or axial displacement	y and z lateral displacements
4	=	x or axial velocity	y and z lateral velocities
5	=	% x or contact stiffness	y and z % friction spring stiffnesses
6	=	Spherical bearing rise	y and z % spherical bearing stiffness

For Hybrid Quadrilateral finite elements, Forces and Moments per unit length.(24 items)

1	=	N-xx,	N-yy,	N-xy,	M-xx,	M-yy,	Mxy	at node I
2	=	N-xx,	N-yy,	N-xy,	M-xx,	M-yy,	Mxy	at node J
3	=	N-xx,	N-yy,	N-xy,	M-xx,	M-yy,	Mxy	at node K
4	=	N-xx,	N-yy,	N-xy,	M-xx,	M-yy,	Mxy	at node L

For Masonry Panel Elements (27 items)

1	=	Force, Displacement and % Stiffness for Strut 1
2	=	Force, Displacement and % Stiffness for Strut 2
3	=	Force, Displacement and % Stiffness for Strut 3
4	=	Force, Displacement and % Stiffness for Strut 4
5	=	Force, Displacement and % Stiffness for Shear Spring
6	=	Plate Bending Moments M-xx, M-yy, Mxy at node I
7	=	Plate Bending Moments M-xx, M-yy, Mxy at node J
8	=	Plate Bending Moments M-xx, M-yy, Mxy at node K
9	=	Plate Bending Moments M-xx, M-yy, Mxy at node L

For Foundation Beam Elements (24 items)

1	=	Force, Deformation, % stiffness	at Axial spring point 1
2	=	Force, Deformation, % stiffness	at Axial spring point 1
3	=	Force, Deformation, % stiffness	at Normal spring point 1
4	=	Force, Deformation, % stiffness	at Normal spring point 2
5	=	Force, Deformation, % stiffness	at Normal spring point 3
6	=	Force, Deformation, % stiffness	at Normal spring point 4
7	=	Force, Deformation, % stiffness	at Shear spring point 1
8	=	Force, Deformation, % stiffness	at Shear spring point 2

Control data for DYNAPLOT

DYNAPLOT prompts for the following information:

c. The name of the output file, the default being the output screen. If you wish the results to be output to a file then give the name (with path if necessary).
d. The name of the Post-processor file output by RUAUMOKO-2D or RUAUMOKO-3D, the default name of which is RUAUMOKO. The file must have the extension **.RES** or **.RAS**.
e. If you have a colour monitor you will be prompted to ask if you wish to use colour. If you are also going to hard-copy the plots and you have only a mono-chromatic printer then answer **N** or **NO** as all lines would appear as solid black lines. In monochromatic systems different line styles are used for each line.
f. What sort of graph do you want? Minimum of the first character required.)

ACCN	=>	Response Spectra from Nodal Accelerations
BASE	=>	Hysteresis plot of combined inputs
COMB	=>	Time history plot of combined inputs
DISP	=>	Snap-shots of deflected shapes
ENVL	=>	Envelope Nodal Results
FREE	=>	Plot mode shapes of free vibration
HYST	=>	Hysteresis plot
INPT	=>	Plot of input excitation
MOVE	=>	RUAUMOKO movie-like displaced shapes and plasticity
NODE	=>	Plot of Node Points in Mesh
PLOT	=>	Plot of Element Mesh (In 3D data can redefine view axes)
TIME	=>	Time history plot
VARY	=>	Time history plot of Inter-nodal data (i.e. inter-storey drifts)
WORK	=>	Time history of work or energies
KEEP	=>	Save last graph plotted to an ASCII file
OPEN	=>	Restart DYNAPLOT with a new (.RES) input data set
REDO	=>	Redraw the Last Plot on the screen
STOP	=>	Terminate DYNAPLOT

Each graph then asks for its own appropriate data.

The time ranges used in the plots must not exceed the time range stored in the post-processor file. If the range is exceeded then a message like “Ran out of RUAUMOKO results” will appear on the screen.

There is a maximum of ten lines in any time-history plot.
There is only one line in a hysteresis loop or Base option plot.
There is a maximum of five lines in an energy or work plot.

Graph titles which are to contain lower case letters or consist of more than one word must be enclosed in single or double quotes and in any case must not contain more than 32 characters, including blanks.

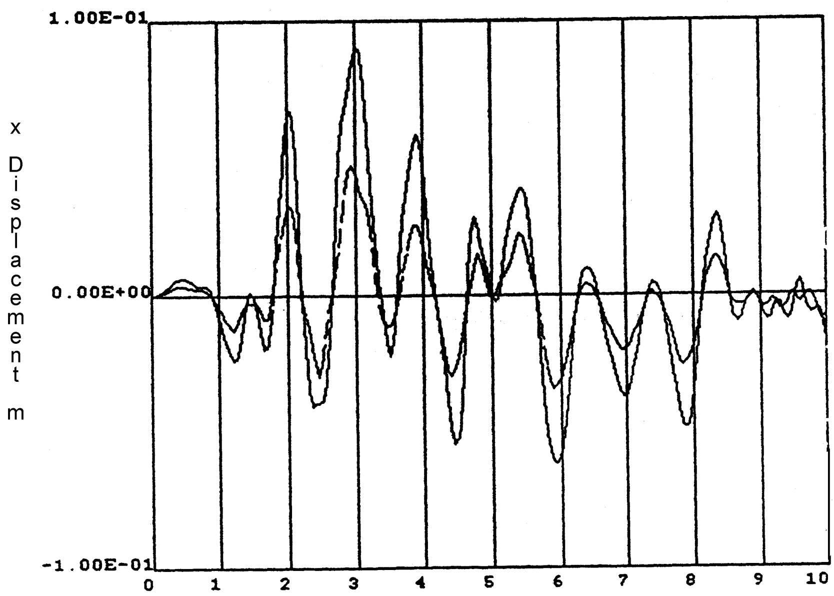
Note: Plots for which the horizontal axis is time will have the text “Time (Seconds)” appended to the title supplied by the user for the horizontal axis.

Time-History Plots

The program prompts for the following information:

a. Number of lines **N** to be plotted.
b. Whether Nodal data or Member data are to be plotted.
c. Which item of the modal or member data is to be plotted. (x, y or rotation etc.)
d. Which **N** nodes or members are to be plotted on the **N** lines.
e. Step interval for vertical grid lines.
f. First and last grid lines. If the grid step was 1.0 seconds and the graph was to plot from 0 to 10 seconds then the response would be **0 10**.
g. Graph vertical grid interval. If zero is selected then an automatic grid spacing is used.
h. The bottom and top horizontal grid line **IYN IYP**. The lowest graph value is **IYN*** (grid interval) and the top value is **IYP*** (grid interval). Note **IYN** MUST be less than **IYP**.
i. Graph height and width. Default values fit A4 sheet.
j. Title along bottom of plot.
k. Title along vertical axis at left of graph.

The example below plots two lines of nodal data with a time interval of **1.0** with the first and last grid lines being **1** and **10** and **IYN** and **IYP** being **-1** and **1** respectively.



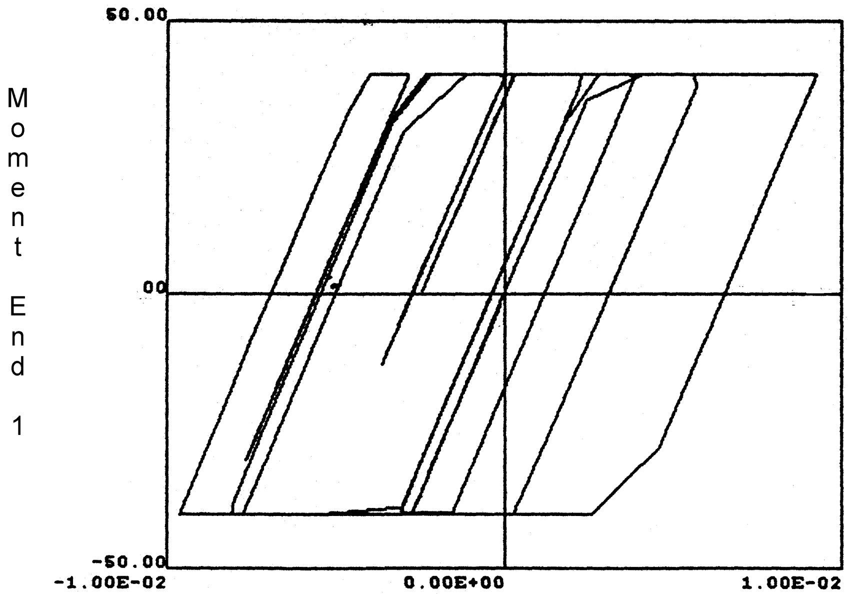
Nodes 9 and 3 - Time (Seconds)

Hysteresis Plots

The program prompts for the following information:

a. Whether Nodal data, or Member data is to be plotted.
b. Which item of the nodal or member data is to be plotted on the vertical axis. (Moment etc.)
c. Which item of the nodal or member data is to be plotted on the horizontal axis. (Curvature etc.)
d. Which node or member is to be plotted.
e. Start time for hysteresis plot.
f. Finish time for hysteresis plot.
g. Graph vertical grid interval. If zero is selected then an automatic grid spacing is used.
h. The bottom and top horizontal grid lines **IYN IYP**. The lowest graph value is **IYN*** (grid interval) and the top value is **IYP*** (grid interval). Note: **IYN** MUST be less than **IYP**.
i. Graph horizontal grid interval. If zero is selected then an automatic grid spacing is used.
j. The left and right horizontal grid lines **IXN IXP**. The left graph value is **IXN*** (grid interval) and the right value is **IXP*** (grid interval). Note: **IXN** MUST be less than **IXP**.
k. Graph height and width. Default values fit A4 sheet.
l. Title along bottom of plot.
m. Title along vertical axis at left of graph.

The example below plots the hysteresis curve for moment versus curvature. Note that although the member is bi-linear the results file has only stored data at every 5 time-steps and hence some loops appear to have had the corners cut off



Curvature End 1 - Member 9

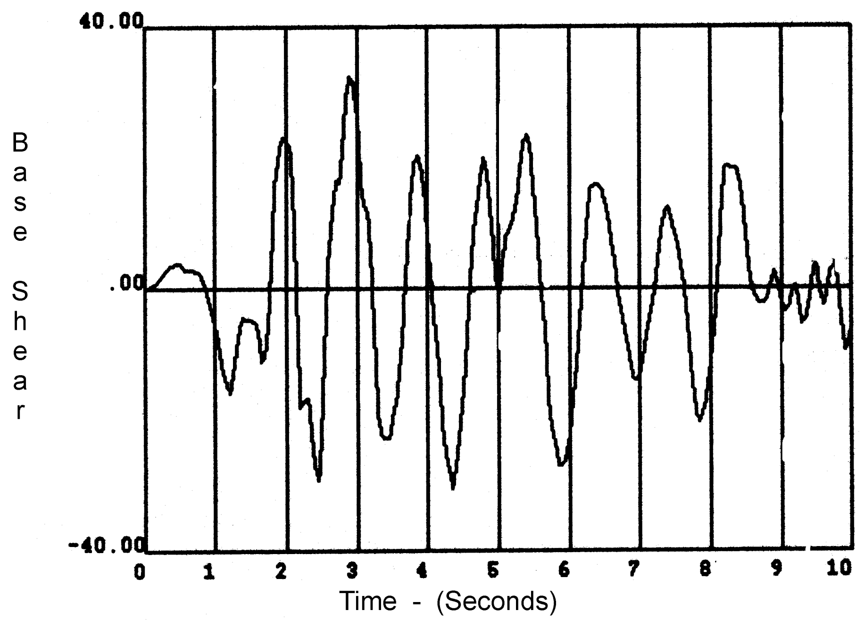
Combination Time-History Plots

This option enables combinations of node and/or member results to be combined into one plot line. Other aspects are identical to that of the Time-History plots. The maximum number of contributions to any one plot line is 50.

The program prompts for the following information:

a. Number of Lines on Plot?
b. For each line:
 1: Number of Contributions?
 2: For each Contribution:
  i Which node or member contributes to line?
  iii Is it a Node or Member?
  iv Which node or member item contributes to the line?
  v Multiplier for contribution.
c. Step interval for vertical grid lines
d. First and last grid lines. If the grid step was 1.0 seconds and the graph was to plot from 0 to 10 seconds then the response would be **0 10**
e. Graph vertical grid interval. If zero is selected than an automatic grid spacing is used.
f. The bottom and top horizontal grid lines **IYN IYP**. The lowest graph value is **IYN*** (grid interval) and the top value is **IYP*** (grid interval). Note **IYN** MUST be less than **IYP**.
g. Graph height and width. Default values fit A4 sheet.
h. Title along bottom of plot.
i. Title along vertical axis at left of graph.

In the example below the Base Shear is obtained by summing the shears in the two ground floor columns.



Base Shear Combination Hysteresis Plots

This option enables combinations of node and/or member results to be combined into one plot line and to be plotted in the form of a Hysteresis plot by plotting the combined results against a second set of combined results. An example would be the plot of base-shear (a combined input) versus the interstorey drift (a second combined input). In the example below the Base Shear is plotted against the top floor displacement. The maximum number of contributions to each axis value is 50.

The program prompts for the following information:

a. For the Vertical axis:
   1 Number of Contributions?
   2 For each Contribution:
      i Which node or member contributes to line?
      ii Is it a Node or Member?
      iii Which node or member item contributes to the line?
      iv Multiplier for contribution.

b For the Horizontal axis:
   1 Number of Contributions?
   2 For each Contribution:
      i Which node or member contributes to line?
      ii Is it a Node or Member?
      iii Which node or member item contributes to the line?
      ii Multiplier for contribution.

c Graph vertical grid interval. If zero is selected than an automatic grid spacing is used.

d The bottom and top horizontal grid lines **IYN IYP**. The lowest graph value is **IYN*** (grid interval) and the top value is **IYP*** (grid interval). Note **IYN** MUST be less than **IYP**.

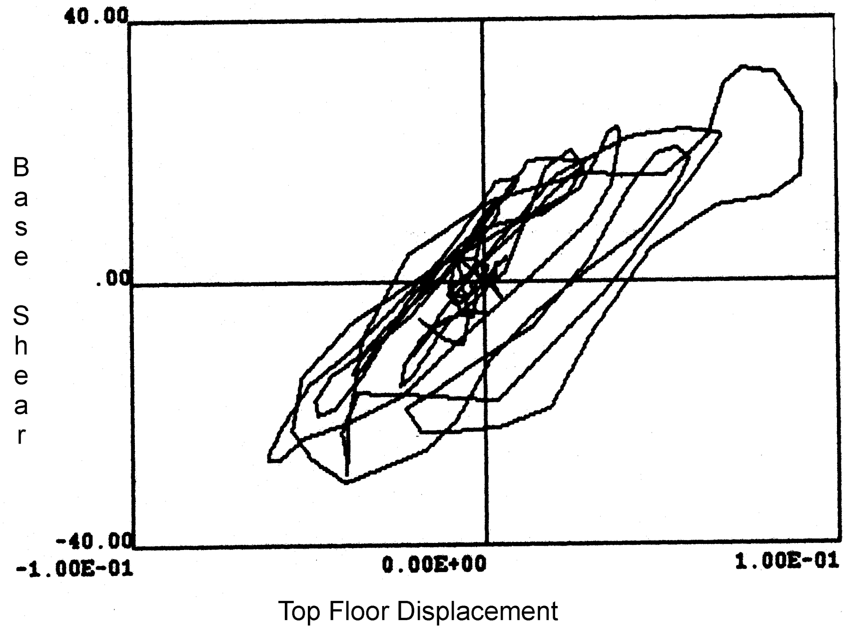
e Graph horizontal grid interval. If zero is selected then an automatic grid spacing is used.

f The left and right horizontal grid lines **IXN IXP**. The left graph value is **IXN*** (grid interval) and the right value is **IXP*** (grid interval). Note **IXN** MUST be less than **IXP**.

g Graph height and width. Default values fit A4 sheet.

h Title along bottom of plot.

i Title along vertical axis at left of graph.



Work-Energy History Plots

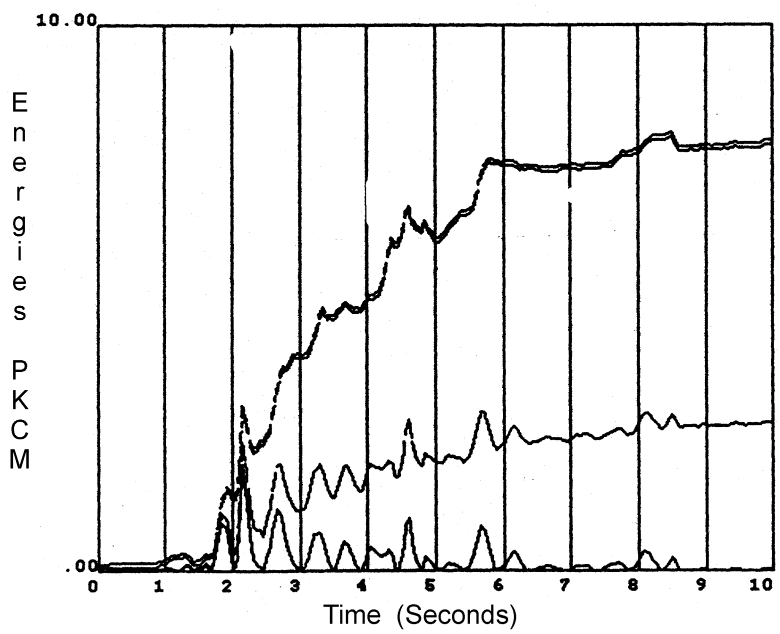
This option enables the Time-history of work or energies to be plotted. An option is to plot rate-of-change of work or "power" to be plotted.

The program prompts for the following information:

- a Whether Work-Energy or Power is to be plotted.
- b Number of lines to be plotted.
- c The five energy contributions multiplied by the factors below and are summed for each line:
  - 1 Five multipliers (numbers) for each line are:
    - i Multiplier for Kinetic Energy
    - ii Multiplier for Damping Work done
    - iii Multiplier for Strain Energy including Plastic work
    - iv Multiplier for External Applied Work
    - v Multiplier for Plastic Work?
- d Time Step interval for vertical grid lines.
- e First and last grid lines. If the grid step was 1.0 seconds and the graph was to plot from 0 to 10 seconds then the response would be **0 10**
- f Graph vertical grid interval. If zero is selected then an automatic grid spacing is used.
- g The bottom and top horizontal grid lines **IYN IYP**. The lowest graph value is **IYN*** (grid interval) and the top value is **IYP*** (grid interval). Note **IYN** MUST be less than **IYP**.
- h Graph height and width. Default values fit A4 sheet.
- i Title along bottom of plot.
- j Title along vertical axis at left of graph.

The example below plots four lines, the bottom line is the kinetic energy, the second line is the kinetic energy and damping work done, the third line is the sum of the kinetic energy, the damping work done and the strain energy. The fourth line is the External Applied Work. The vertical grid spacing is **1.0** seconds and the first and last grids being **0** and **10** respectively. The vertical spacing was the default **0.0** and **IYN** and **IYP** are **0** and **1** respectively.

If one selects the **Default** option then the following energy plot is produced without the necessity of specifying all the data items above.



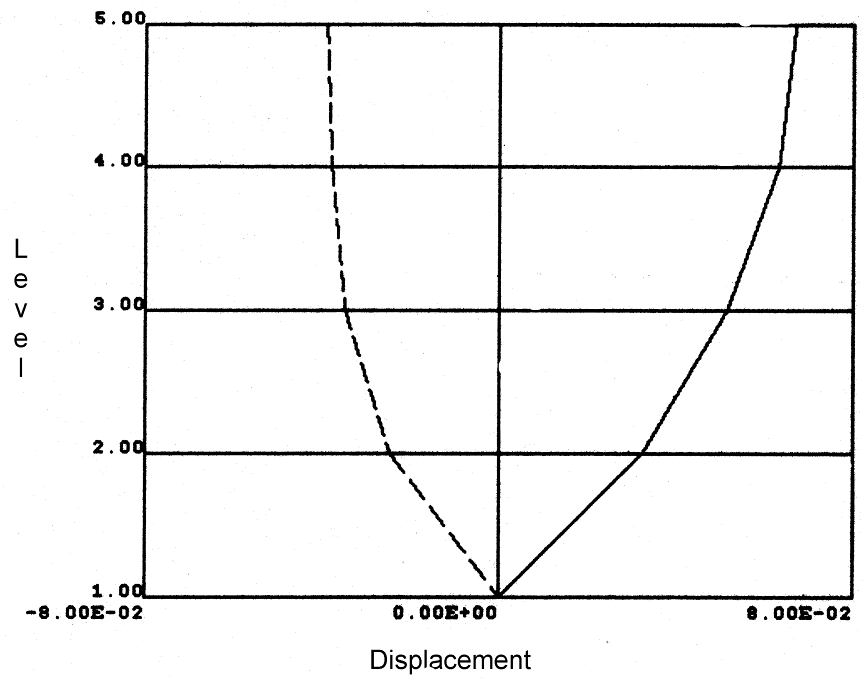
Snap-shots of Deflected Shapes

This option enables the displaced shape of the structure to be plotted at various time-steps.

The program prompts for the following information:

- a The item to be plotted. For a building, usually the x-displacement. See table **b** in the section on 'Output from RUAUMOKO post-processor file'.
- b Number of storeys or levels in plot. The number of storeys, or levels, in building plus one.
- c List of nodes, one required for each level. If the list cannot be fitted on a single input line the line may be continued by placing an isolated ampersand (**&**) at the end of the line and the program will prompt for the remaining nodes.
  If the first node is zero then the displacement for that node will be taken as zero.
  If any other nodes are zero then linear interpolation will be used for that level using the nearest available nodes from the adjoining levels.
  The node for the last level must be provided.
- d Graph horizontal grid interval. If zero is selected then an automatic grid spacing is used.
- e The left and right vertical grid lines **IYN IYP**. The left graph value is **IYN*** (grid interval) and the right value is **IYP*** (grid interval). Note **IYN** MUST be less than **IYP**.
- f Graph height and width. Default values fit A4 sheet.
- g Title along bottom of plot. Usually **Displacement**.
- h Title along vertical axis at left of graph. Usually **Level**
- i The number of displaced shapes to be shown on the one plot (maximum of 10).
- j The times at which the deflected shapes are to be plotted. These must be in ascending time sequence and the minimum separation is the time-step at which results are stored in the **.RES** file. If the interval is too small the program will increment the steps automatically. The actual times used will be the first available step on or after the time specified.

The example below plots two lines, the solid line is the deflected shape at 2.0 seconds, the dashed line is the deflected shape at 2.5 seconds. The data was stored for nodes 1, 3, 5, 7 and 9.



Movie-like Presentation of Deflected Shapes and Plasticity

The option presents the movie-like displays of deflected shapes and plasticity. This show the same deformed mesh pictures that are able to be displayed in RUAUMOKO as it carries out the time-history analysis. However, in DYNAPLOT they may be displayed picture by picture so that hard copies of the plots may be produced.

The program prompts for the following information:

a. Whether continuous or stepped effects
b. Start time for the movie-like motion
c. Finish time for the movie-like motion.

If continuous effects are selected then no further action is required until the sequence finishes except that on some systems a key-press will stop the sequence and a second key-press will then terminate the plot. For stepped effects press the **Enter** key to proceed to the next picture. In the later case the sequence must proceed to the end of the sequence to terminate. The user is then asked if a further sequence is to be shown.

This feature is only available if **all** nodes and **all** members have had their results saved to the **.RES** post-processor file.

Mode Shapes of Free Vibration

The option presents the movie-like displays of the modes shapes of free vibration. This is similar to the displays in RUAUMOKO except that here if the number of cycles is **S** (upper or lower case) rather than the number of cycles then the mode shape is displayed only in its maximum plotted position so that a hard copy may be produced.

This feature is only available if **all** nodes and **all** members have had their results saved to the **.RES** post-processor file.

Plotting the Member or Element Mesh.

This produces the plot of the mesh as is shown in Ruaumoko. However, in this case you are asked for the range of members, or elements that are to be shown in the plot. The default range is for all elements. For three-dimensional structures, i.e. the data has come from Ruaumoko3D, you are also asked if you wish to change that plot view axes.

This feature is only available if **all** nodes and **all** members have had their results saved to the **.RES** post-processor file.

Plotting the Nodal Point, or Joint, Mesh.

This produces the plot of thenodal points or joints in the mesh. As in the case for plotting the element mesh, case you are asked for the range of nodes or joints that are to be shown in the plot. The default range is for all nodes.

This feature is only available if **all** nodes and **all** members have had their results saved to the **.RES** post-processor file.

Redrawing of the Last Plot

This option enables the user to redraw the last graph plotted. This is normally used where the **Enter** key has been pressed before the user obtained a hard-copy. This enables the graph to be redrawn without going through the selection of the data items again.

Keeping the Last Graph

This enables to user to save the graph data into an ASCII file so that it may be retrieved into the program COMBPLOT or into a spreadsheet etc. for combining with other graphical information. For Hysteresis data each output line saves the coordinates of the points on the plot. For Time-history plots the first item on the line for each point is the time and this is followed by the values of each line at that time. The last two items saved are the x and y titles for the graph. There are several choices for the output format, one fitting an 80 column line another a 132 column line as well as formats suitable for spreadsheet input such as for Microsoft Excel.

There is an option for a single line Time-history graph and that is to save the file as a Caltech Format accelerogram. This is particularly useful if the graph is for the total acceleration for node in the structure and if one wishes to analyze a substructure located at that point in the structure or if one wants a Response Spectra for the motion at that point. The resulting file may be input to RUAUMOKO or SPECTRA as a Caltech format accelerogram.

Restarting with a new post-processor file.

If one selects the **OPEN** option from the main menu, DYNAPLOT restarts and prompts for the name of a new post-processor (**.RES** or **.RAS**) file.

Running the program DYNAPLOT in Batch Mode

Dynaplot has always had the option of not actually plotting the graphs that it generates, though they may still be saved for spreadsheet use with the **Keep** option.

To turn the graphics off follow the name of the post-processor file (the **.res** or **.ras** file) with the word **no** in upper or lower case (or mixed case).

The user can also run the program using the **$add** option of the command processor of having responses already prepared for production analyses. These are most easily generated using the **$log** option of the command processor whilst running **Dynaplot** in the conventional *interactive* mode. The resulting file can then be edited, with any text editor such as **Notepad** etc. to form the input for later runs. Please see the example **Dynaplot.txt** file in the **Results** directory on the cdrom.

Dynaplot now has an option to be run in *batch* mode. This is useful for researchers running large numbers of parameter studies and who wish to use some features of the post-processing. In the batch mode the graphics is automatically turned off.

To run the **Dynaplot** in batch mode the command Dynaplot must be followed by at least 3 arguments, i.e.

Dynaplot Output.wri Example Input.txt

where

Output.wri is the name of the output file. This file contains the maximum and minimum values of every line drawn in every plot drawn by **Dynaplot**. Any extension may be used, i.e. **.wri** or **.txt** etc. This file MUST NOT exist or the program will terminate.

Example is the name of the postprocessor file. This file MUST exist. The file will have the extension **.res** or **.ras** depending on whether it is binary or ASCII post-processor file. The extension is NOT supplied with the name.

Input.txt is the name of the data file containing all the command input for Dynaplot, which is very similar to the example file **Dynaplot.txt** in the Results directory on the Ruaumoko cdrom. This file MUST exist.

The input data file is probably best generated during a conventional *interactive* use of **Dynaplo**t saving a copy of all the commands and data with the **$log** option of the command processor. The only difference is that in the *batch* mode there is only one output file and the **Open** option in **Dynaplot** works in a different manner. In the *interactive* mode in **Dynaplot** when the **Open** option is used, the existing output file (if not **screen**) is closed and the user is prompted for a new output file name. The user is then prompted for a new post-processor file name. In the *batch* mode there is only one output file, that given in the command line. If the **Open** option is invoked, the command **Ope**n must be followed (on the same line) by the name of the new post-processor file. The file name has the **.res** or **.ras** implied and the extension therefore MUST NOT be applied.

UNIVERSITY OF CANTERBURY
Te Whare Wānanga o Waitaha
CHRISTCHURCH NEW ZEALAND

Department of Civil Engineering **COMPUTER PROGRAM LIBRARY**

Program name:– **COMBPLOT**	Program type:– Time-History Output for RUAUMOKO	Program code:– ANSI Fortran77
Author:– Athol J Carr		Date:– April 28, 2007

COMBPLOT

Combining DYNAPLOT Time-History Plots

The program COMBPLOT is to combine the time-history outputs from different analyses on to a single graph. After using DYNAPLOT to plot the time-history plot the KEEP option of DYNAPLOT is used to put the graph information into a file. This is done for all the analyses of the different structures or the analyses of the same structure to different excitations.

The requirement is that the time length is the same for all the plots and that you can remember the names given to the output files. Each KEEP call in DYNAPLOT should use a new file name.

To combine the plots COMBPLOT will ask for the number of lines to be plotted on the one time-history plot. Then for each line it will ask for the name of the file which contains the line that you wish to plot and which of the lines in the original plot that you wish to use. There will be at least one line in the plot but the original graph could have had up to 10 lines. When all lines have been input then COMBPLOT will request information on grid intervals and titles in a manner similar to those asked in DYNAPLOT for Time-History plots.

The program can also take in the list outputs from SPECTRA to pull ONE spectra line from ONE damping level from each of the (up to 10) SPECTRA runs for the different earthquakes. The proviso is that the period range and period intervals are the same for all the SPECTRA analyses and that you can remember the names of the output files created in each run with SPECTRA.

All data is prompted for and is reasonably self-explanatory.

Running the post-processor program COMBPLOT.

To run the program call the program by the method appropriate to your operating system. On a personal computer just type **COMBPLOT** assuming that the files **COMBPLOT.EXE** and **FQWIN.HLP** are in your current directory or path.

In Microsoft Windows operating systems another option is to create a shortcut on the desktop and for this purpose a suitable icon for COMBPLOT, **Combplot.ico**, is supplied with the program.

The program prompts for responses to a series of questions. Default responses, where appropriate, are enclosed in square brackets, **[ ]**. File names must match the conventions of your operating system but file names, with paths where necessary, must not exceed 60 characters in length and must not contain blanks.

The first question asks for the name of the output file. The default is the computer console or terminal screen. This is normally the response for most users, however if an output file is created then the envelope values of each graph line drawn is output to that file.

To get hard copies of the plots.

In Microsoft Windows operating systems to get hard copies of the graphs use the pull down 'file' menu and select the Print or Save options to send the graph to the printer or to save the plot as a bitmap file (**.BMP)**.
On unix systems using GKS graphics select the Hard-copy option from the Choice window.

End

UNIVERSITY OF CANTERBURY
Te Whare Wānanga o Waitaha
CHRISTCHURCH NEW ZEALAND

Department of Civil Engineering **COMPUTER PROGRAM LIBRARY**

Program name:– **PQUAKE**	Program type:– Response Spectrum Analysis	Program code:– ANSI Fortran77
Author:– Athol J Carr		Date:– April 28, 2007

PQUAKE

Plotting of Earthquake Accelerograms

Purpose

This program is designed to an input earthquake accelerogram. The input formats for the accelerograms are the same as those formats accepted by the program RUAUMOKO.

Data for PQUAKE

All data for PQUAKE is prompted for by the program using the TinyCLIP command processor. For help on the input data or on the use of the SPECTRA program type **HELP** or **?** At any prompt. To get help on the command processor type **$HELP** at any prompt.

Running the program PQUAKE.

To run the program call the program by the method appropriate to your operating system. On a personal computer just type **PQUAKE** assuming that the files **PQUAKE.EXE** and **FQWIN.HLP** are in your current directory or path.

In Microsoft Windows operating systems another option is to create a shortcut on the desktop and for this purpose a suitable icon for PQUAKE, **Pquake.ico**, is supplied with the program.

The program prompts for responses to a series of questions. Default responses, where appropriate, are enclosed in square brackets, **[ ]**. File names must match the conventions of your operating system but file names, with paths where necessary, must not exceed 60 characters in length and must not contain blanks.

The first question asks for the name of the output file. The default is the computer console or terminal screen.

To get hard copies of the plots.

In Microsoft Windows operating systems to get hard copies of the graphs use the pull down 'file' menu and select the Print or Save options to send the graph to the printer or to save the plot as a bitmap file (**.BMP**).
On unix systems using GKS graphics select the Hard-copy option from the Choice window
As part of the initial information, following the name of the output file, the user is prompted for the name (and path) for the file containing the earthquake accelerogram. The program will attempt to determine the format of the accelerogram from the filename extension. If the program is not able to do so because the extension is not one of the standard forms then the program will prompt for the file type. This must be one of the standard formats, BERG, CALTECH, NCEER, FREE, CSMIP, EXCEL, PEER or SAC as used by Ruaumoko.

If the file type is CALTECH, NCEER, CSMIP, PEER or SAC then the user will be prompted for the time-step **DELTAT** at which the accelerogram is digitized**.**

The user will also be prompted for the divisor **ASCALE** required to convert the accelerogram to the units of the acceleration of gravity. For example, if the acceleration record is in units of cm/sec/sec the **ASCALE** is 981 If the acceleration record is already in the units of the acceleration of gravity the **ASCALE** is 1.0 unless the user also wishes to scale the record to some other magnitude.

The user is also prompted for which line, **ISTART**, in the earthquake accelerogram record the analysis is to start. The default value is 1, i.e. the record starts at the first line of the data but in some cases, particularly where the input is a recorded shake-table record there may be a long lead-in with very little happening, i.e. between switching on the recording equipment and the start of the shake-table motion , and this gives a means of omitting this from the calculations.

DELTAT Excitation data interval
BERG The default value is the time-step for the integration DT.
CALTECH Format records the record time-interval
(usually 0.025, 0.02, 0.01 or 0.005 seconds)
NCEER Format records the record time-interval
(usually 0.02 seconds)
FREE Format records (see BERG Format records)
CSMIP Format records the record time-interval
(usually 0.02 seconds)
EXCEL Format records (see BERG Format records)
PEER Format records the record time-interval
(usually 0.004 seconds)
SAC Format records the record time-interval
(usually 0.020 seconds)

ASCALE 1/(Scale factor) for the record.
BERG, FREE or EXCEL Format records omit or set equal to 1.0
CALTECH records are usually in units of acceleration multiplied by a constant as the F6.0 format is usually an Integer I6 format and the decimal point is missing with about 5 integer digits for the largest numbers. The values must be converted units of the acceleration of gravity . If the record is in the units of mm/sec/sec and if g = 9.81 m/sec/sec. then **ASCALE** = 9810.0.
NCEER and CSMIP records are usually in cm/sec/sec and **ASCALE** = 981.0 to bring the record to units of the acceleration of gravity.

Accelerogram Data

a. Accelerogram flag.

One input line with the word **STAR, START** or **DATA:** (the colon is mandatory) starting in column 1 and the word must be in upper case. This **START** line may be preceded by as many header lines as desired. This **START** line is not used for PEER or SAC Format records as these records start with 4or 2 lines of header information respectively.

START

b. Accelerogram

The remainder of the input is the acceleration record itself. The record is in the form of a series of lines each of which starts with a *Line Sequence Number* (which must be in an ascending order) followed by either (i) a group of 4 or 1 successive time-acceleration points (BERG, FREE or EXCEL Format), or (ii) a sequence of 10, 8, 5 or 6 uniformly spaced acceleration values at **DELTAT.**

Note that the NCEER, CSMIP or PEER records do not have a line sequence number. The analysis acceleration record will begin at the first time on or implied by the beginning of the accelerogram line **ISTART** and there must then be sufficient lines remaining to span the analysis time-history length.

The record must be on one of the following formats. The FORTRAN format is provided in parentheses for each case.

(1) BERG FORMAT **(I3,4(F8.4,F9.6))** (default filename extension is **.eqb**)

ISEQ T1 G1 T2 G2 T3 G3 T4 G4

ISEG	Line sequence number	**I 3**
Ti	Time of point on accelerogram (seconds)	**F 8.4**
Gi	Acceleration (decimal fraction of gravity)	**F 9.6**

If the line sequence number is greater than 999 it is not read or checked by the program.

(2) CALTECH FORMAT **(I4,6X,10F6.0)** or more precisely **(I4,6X,10I6)** (default filename extension is **.eqc**)

ISEQ G1 G2 G3 G4 G5 G6 G7 G8 G9 G10

ISEQ	Line sequence number	**I 4**
Gi	Acceleration (multiplied by **ASCALE**) at intervals of **DELTAT**	**I 6**

If the line sequence number is greater than 9999 it is not read or checked by the program.

(3) NCEER FORMAT **(10F8.2)** (default filename extension is **.eqn**)

G1 G2 G3 G4 G5 G6 G7 G8 G9 G10

Gi	Acceleration (multiplied by **ASCALE)** at intervals of **DELTAT**	**F 8.2**

(4) FREE FORMAT **(*)** (default filename extension is **.eqf**)

ISEQ T1 G1

ISEQ	Line sequence number	**I**
T1	Time of point on accelerogram (seconds)	**F**
G1	Acceleration (decimal fraction of gravity)	**F**

The three items may be placed anywhere on the line and separated by at least one blank column. The lines must be in consecutive order with **ISEQ** starting at 1 and increasing line by line. This format is particularly useful where the excitation record has been generated on a spreadsheet.

(5) CSMIP FORMAT **(8F10.3)** (default filename extension is **.eqs**)

G1 G2 G3 G4 G5 G6 G7 G8

Gi	Acceleration (multiplied by **ASCALE)** at intervals of **DELTAT**	**F 10.3**

(6) EXCEL FORMAT **(*)** (default filename extension is **.eqe**)

T1 G1

T1	Time of point on accelerogram (seconds)	**F**
G1	Acceleration (decimal fraction of gravity**)**	**F**

The three items may be placed anywhere on the line and separated by at least one blank column. The lines must be in consecutive order. This format, which is similar to the FREE format except without the sequence numbers is particularly useful where the excitation record has been generated on a spreadsheet.

(6) PEER FORMAT **(5E15.7)** (default filename extension is **.eqp**)

G1 G2 G3 G4 G5

Gi	Acceleration (multiplied by **ASCALE)** at intervals of **DELTAT**	**E 15.7**

Note, The accelerogram time-step **DELTAT** is usually 0.004 seconds

(7) **SAC** FORMAT **(6E13.5)** (default filename extension is **.eqk**)

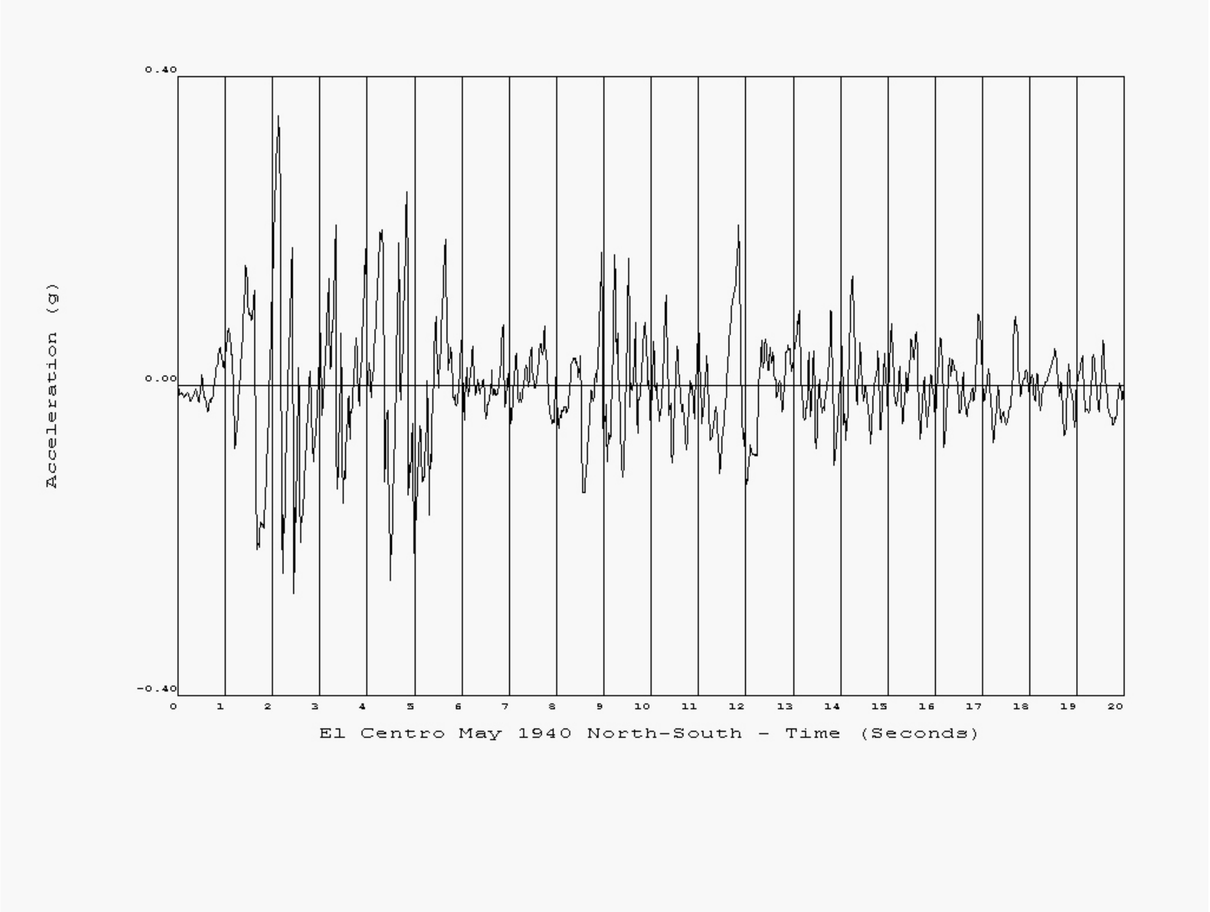
G1 G2 G3 G4 G5 G6

Gi Acceleration (multiplied by **ASCALE**) at intervals of **DELTAT** **E 13.5**

Note, The accelerogram time-step **DELTAT** is usually 0.020 seconds

Example

The page following the information on the accelerograms gives the graphic output for the El Centro May 1940 North-South component from the file **EL40NSC.EQB** using the default inputs except for 20 seconds duration and for monochromatic plotting.



UNIVERSITY OF CANTERBURY
Te Whare Wānanga o Waitaha
CHRISTCHURCH NEW ZEALAND

Department of Civil Engineering **COMPUTER PROGRAM LIBRARY**

Program name:– **SPECTRA**	Program type:– Response Spectrum Analysis	Program code:– ANSI Fortran77
Author:– Athol J Carr		Date:– April 28, 2007

SPECTRA

Response Spectra Computation

Purpose

This program is designed to compute and plot the response spectra for an input earthquake accelerogram. The program computes the Spectral Displacement SD, the Spectral Velocity SV, The Pseudo Spectral Velocity PSV, the Spectral Acceleration SA and the Pseudo Spectral Acceleration PSA. The program also produces an Energy Spectra which is defined as the square root of the integral over the duration of the record of the velocity of the mass multiplied by the ground acceleration. The output is both tabular and graphical. The input formats for the accelerograms are the same as those formats accepted by the program RUAUMOKO.

Method of Analysis

The program asks for information on the range of natural periods, period interval and damping ratios before reading and digitizing the accelerogram. It then, for each damping ratio, computes the responses of single degree of freedom oscillators where the natural period is varied over the period range specified. The program has no user manual as all data is prompted for and there are help files available in the program

Running the post-processor program SPECTRA.

To run the program call the program by the method appropriate to your operating system. On a personal computer just type **SPECTRA** assuming that the files **SPECTRA.EXE** and **FQWIN.HLP** are in your current directory or path.

In Microsoft Windows operating systems another option is to create a shortcut on the desktop and for this purpose a suitable icon for SPECTRA, **Spectra.ico**, is supplied with the program.

The program prompts for responses to a series of questions. Default responses, where appropriate, are enclosed in square brackets, **[ ]**. File names must match the conventions of your operating system but file names, with paths where necessary, must not exceed 60 characters in length and must not contain blanks.

The first question asks for the name of the output file. The default is the computer console or terminal screen

To get hard copies of the plots.

In Microsoft Windows operating systems to get hard copies of the graphs use the pull down 'file' menu and select the Print or Save options to send the graph to the printer or to save the plot as a bitmap file (**.BMP)**.
On unix systems using GKS graphics select the Hard-copy option from the Choice window

Data for SPECTRA

All data for SPECTRA is prompted for by the program using the TinyCLIP command processor. For help on the input data or on the use of the SPECTRA program type **HELP** or **?** At any prompt. To get help on the command processor type **$HELP** at any prompt.

Note: In the following user guide, each line of required data is indicated by a box containing the data items. Below each box is a description of the data items. The data items on each line may be separated by commas or blank spaces. The format for the items are indicated by the letter at the end of each descriptive line with **A** indicating a character string, **I** indicating an integer value and **F** indicating a floating point number. A floating point number may or may not have a decimal point and may also take a scientific or exponent form such as 1.5E6 which could also be expressed as 1500000.0. Character strings will be upper-cased unless enclosed in double or single quotes and will terminate at the first blank space unless the string is enclosed in quotes.

As part of the initial information, following the name of the output file, the user is prompted for the name (and path) for the file containing the earthquake accelerogram. The program will attempt to determine the format of the accelerogram from the filename extension. If the program is not able to do so because the extension is not one of the standard forms then the program will prompt for the file type. This must be one of the standard formats, BERG, CALTECH, NCEER, FREE, CSMIP, EXCEL, PEER or SAC as used by Ruaumoko.

If the file type is CALTECH, NCEER, CSMIP, PEER or SAC then the user will be prompted for the time-step **DELTAT** at which the accelerogram is digitized.

The user will also be prompted for the divisor **ASCALE** required to convert the accelerogram to the units of the acceleration of gravity. For example, if the acceleration record is in units of cm/sec/sec the ASCALE is 981 If the acceleration record is already in the units of the acceleration of gravity the **ASCALE** is 1.0 unless the user also wishes to scale the record to some other magnitude.

The user is also prompted for which line, **ISTART**, in the earthquake accelerogram record the analysis is to start. The default value is 1, i.e. the record starts at the first line of the data but in some cases, particularly where the input is a recorded shake-table record there may be a long lead-in with very little happening, i.e. between switching on the recording equipment and the start of the shake-table motion , and this gives a means of omitting this from the calculations.

DELTAT Excitation data interval

BERG The default value is the time-step for the integration **DT**.

CALTECH Format records the record time-interval
(usually 0.025, 0.02, 0.01 or 0.005 seconds)

NCEER Format records the record time-interval
(usually 0.02 seconds)

FREE Format records (see BERG Format records)

CSMIP Format records the record time-interval
(usually 0.02 seconds)

EXCEL Format records (see BERG Format records)

PEER Format records the record time-interval
(usually 0.004 seconds)

SAC Format records the record time-interval
(usually 0.020 seconds)

ASCALE 1/(Scale factor) for the record.

BERG, FREE or EXCEL Format records omit or set equal to 1.0

CALTECH records are usually in units of acceleration multiplied by a constant as the **F6.0** format is usually an Integer **I6** format and the decimal point is missing with about 5 integer digits for the largest numbers. The values must be converted units of the acceleration of gravity . If the record is in the units of mm/sec/sec and if **g** = 9.81 m/sec/sec. then **ASCALE** = 9810.0.

NCEER and CSMIP records are usually in cm/sec/sec and **ASCALE** = 981.0 to bring the record to units of the acceleration of gravity.

Accelerogram Data

a. Accelerogram flag.

One input line with the word **STAR**, **START** or **DATA:** (the colon is mandatory) starting in column 1 and the word must be in upper case. This **START** line may be preceded by as many header lines as desired. This **START** line is not used for PEER or SAC Format records as these records start with 4 or 2 lines of header information respectively.

START

b. Accelerogram

The remainder of the input is the acceleration record itself. The record is in the form of a series of lines each of which starts with a *Line Sequence Number* (which must be in an ascending order) followed by either (i) a group of 4 or 1 successive time-acceleration points (BERG, FREE or EXCEL Format), or (ii) a sequence of 10, 8 or 4 uniformly spaced acceleration values at **DELTAT** time intervals apart, the (CALTECH, NCEER, CSMIP or PEER Formats).

Note that the NCEER, CSMIP, EXCEL, PEER or SAC records do not have a sequence number. The analysis acceleration record will begin at the first time on or implied by the beginning of the accelerogram line **ISTART** and there must then be sufficient lines remaining to span the analysis time-history length **TIME**.

The record must be on one of the following formats. The program will normally determine the accelerogram format from the extension for the file name. If it is not one of the following extensions then the program will prompt for the user to state the format type. The FORTRAN format is provided in parentheses for each case. The **I3** format implies 3 characters for the number which is right justified, **6X** implies 6 blank characters. The **F8.4** implies 8 characters for the number and if omitted the decimal point is located in front of the 4th to last character in the 8 character set which is assume to be right justified, if the decimal point is provided the number may be located anywhere in the 8 character space. A number preceding the format, i.e. **10F8.4** means that there are 10 numbers each having an 8 character field. Each format starts on a new line.

(1) **BERG** FORMAT **(I3, 4(F8.4, F9.6))** (default extension is **.eqb**)

ISEQ T1 G1 T2 G2 T3 G3 T4 G4

ISEG	Line sequence number	**I 3**
Ti	Time of point on accelerogram (seconds)	**F 8.4**
Gi	Acceleration (decimal fraction of gravity)	**F 9.6**

If the line sequence number is greater than 999 it is not read or checked by the program.

(2) **CALTECH** FORMAT **(I4, 6X, 10 I6)** (default extension is **.eqc**)

ISEQ G1 G2 G3 G4 G5 G6 G7 G8 G9 G10

ISEQ	Line sequence number	**I 4**
Gi	Acceleration (multiplied by **ASCALE**) at intervals of **DELTAT**	**I 6**

If the line sequence number is greater than 9999 it is not read or checked by the program.

(3) **NCEER** FORMAT **(10 F8.2)** (default extension is **.eqn**)

G1 G2 G3 G4 G5 G6 G7 G8 G9 G10

Gi	Acceleration (multiplied by **ASCALE**) at intervals of **DELTAT**	**F 8.2**

(4) **FREE** FORMAT **(*)** (default extension is **.eqf**)

ISEQ T1 G1

ISEQ	Line sequence number	**I**
T1	Time of point on accelerogram (seconds)	**F**
G1	Acceleration (decimal fraction of gravity)	**F**

The three items may be placed anywhere on the line and separated by at least one blank column. The lines must be in consecutive order with **ISEQ** starting at 1 and increasing line by line. This format is particularly useful where the excitation record has been generated on a spreadsheet.

(5) **CSMIP** FORMAT **(8 F10.3)** (default extension is **.eqs**)

G1 G2 G3 G4 G5 G6 G7 G8

Gi	Acceleration (multiplied by **ASCALE**) at intervals of **DELTAT**	**F 10.3**

(6) **EXCEL** FORMAT **(*)** (default extension is **.eqe**)

T1 G1

T1	Time of point on accelerogram (seconds)	**F**
G1	Acceleration (decimal fraction of gravity)	**F**

The three items may be placed anywhere on the line and separated by at least one blank column. The lines must be in consecutive order. This format, which is similar to the FREE format except without the sequence numbers is particularly useful where the excitation record has been generated on a spreadsheet.

(7) **PEER** FORMAT **(5E15.7)** (default extension is **.eqp**)

G1 G2 G3 G4 G5

Gi	Acceleration (multiplied by **ASCALE**) at intervals of **DELTAT**	**E 15.7**

Note: The acclerogram time-step **DELTAT** is usually 0.004 seconds.

(8) **SAC** FORMAT **(6 E13.5)** (default extension is **.eqk**)

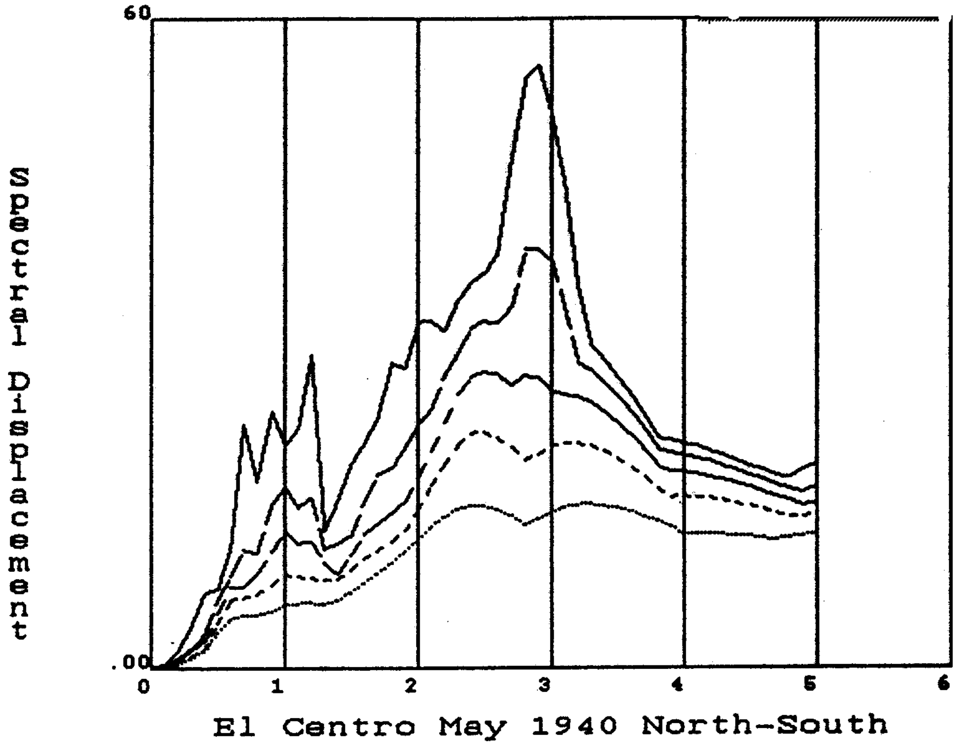
G1 G2 G3 G4 G5 G6

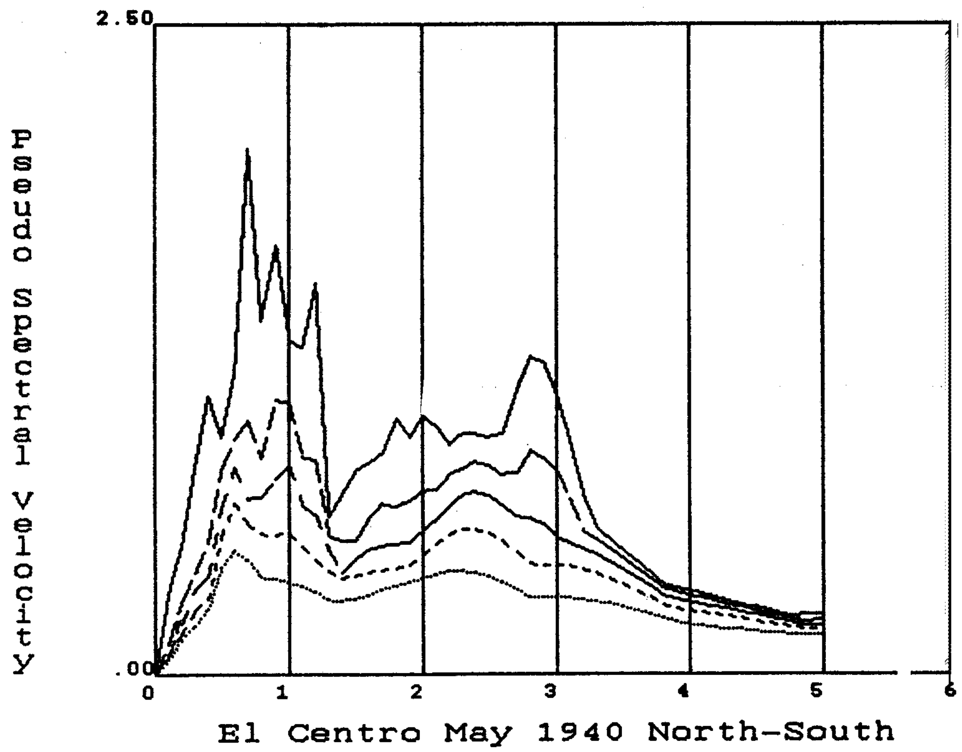
Gi	Acceleration (multiplied by **ASCALE**) at intervals of **DELTAT**	**E 13.5**

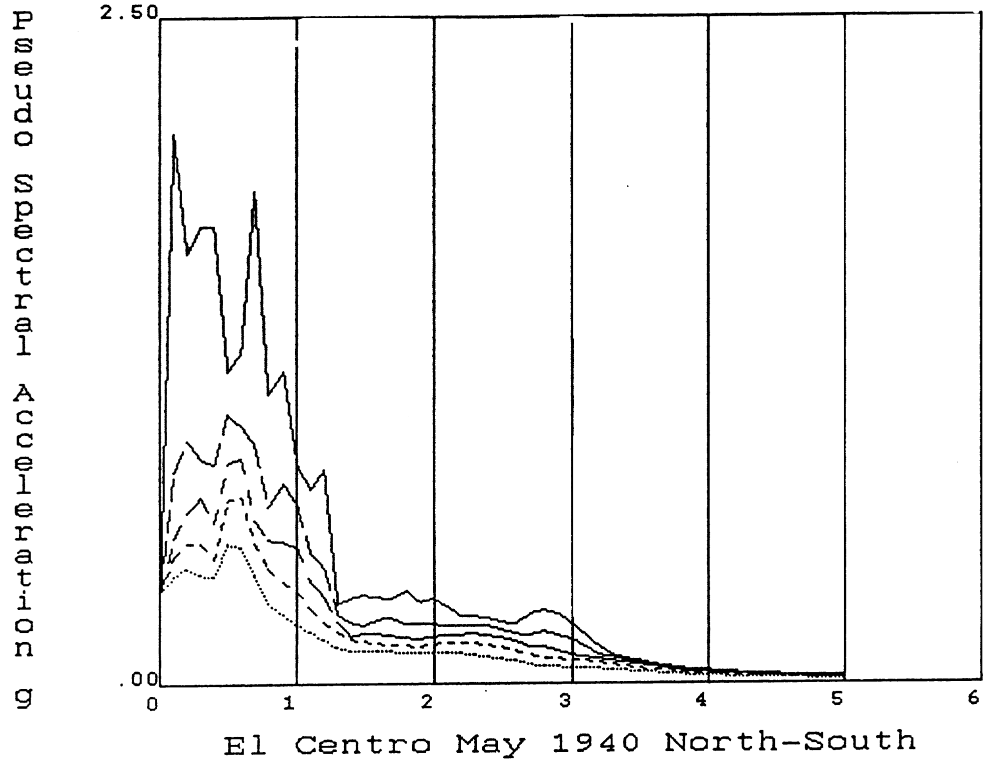
Note: The acclerogram time-step **DELTAT** is usually 0.020 seconds.

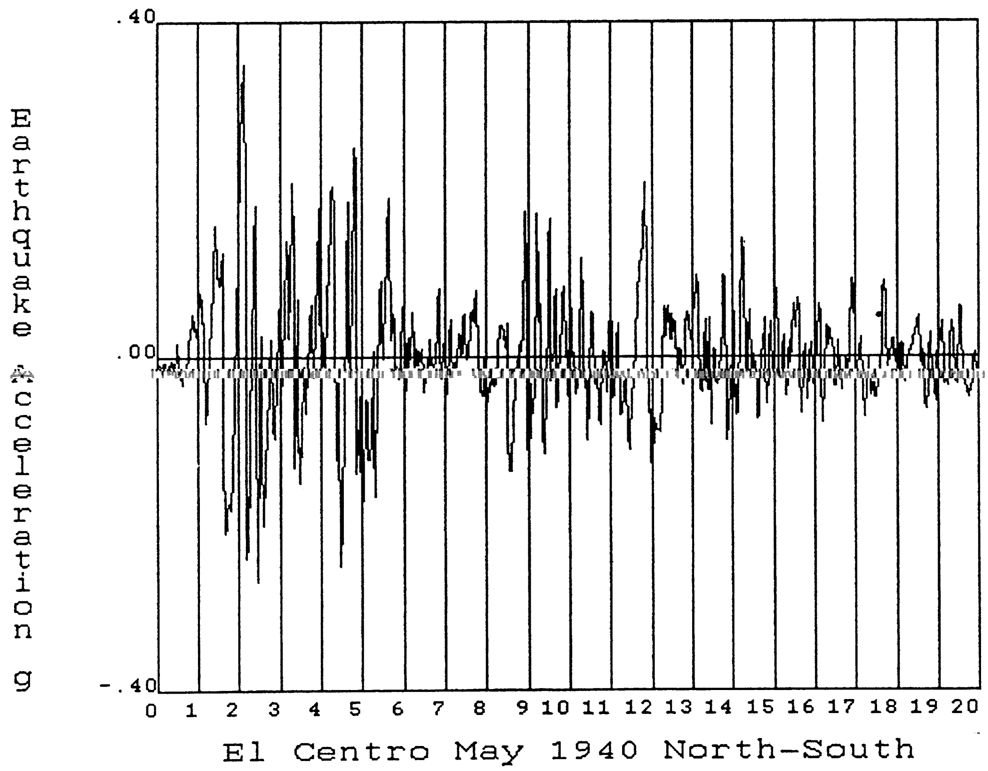
Example

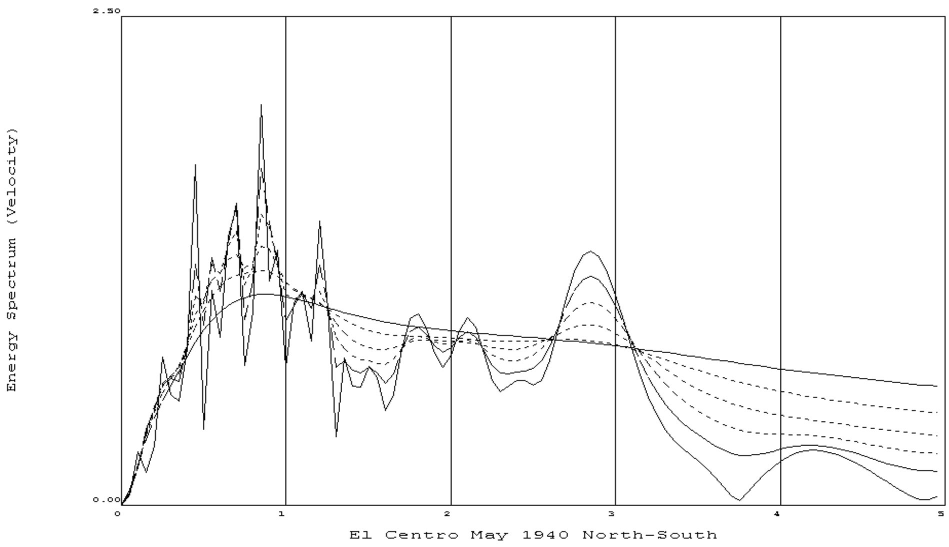
The following pages show the graphic output for the El Centro May 1940 North-South component **EL40NSC.EQB** using the default inputs except for 20 seconds duration and monochromatic plotting.






Pseudo Spectral Acceleration g
2.50
.00
0
1
2
3
4
5
6
El Centro May 1940 North-South


Earthquake Acceleration g
.40
.00
-.40
0 1 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17 18 19 20
El Centro May 1940 North-South


2.50
0.00
Energy Spectrum (Velocity)
0
1
2
3
4
5
El Centro May 1940 North-South

End.

This page is intentionally left blank.

UNIVERSITY OF CANTERBURY
Te Whare Wānanga o Waitaha
CHRISTCHURCH NEW ZEALAND

Department of Civil Engineering **COMPUTER PROGRAM LIBRARY**

Program name:– **SIMQKE**	Program type:– Fortran Format Printing	Program code:– ANSI Fortran77
Author:– Athol J Carr		Date:– April 28, 2007

SIMQKE

A PROGRAM FOR ARTIFICIAL MOTION GENERATION

The program **SIMQKE** has these major capabilities:

It computes a power spectral density function from a specified smooth response spectrum.
It generates statistically independent artificial acceleration time histories and tries, by iteration, to match the specified response spectrum.
It performs a baseline correction on the generated motion to ensure zero final ground velocity.
It calculates response spectra with the time histories as input.

BRIEF DESCRIPTION OF THE MOTION GENERATION PROCEDURE

An explanation of the input to **SIMQKE** and complete flowchart (Fig. 1) are given.

The method used by the program for artificial ground motion generation is based on the fact that any periodic function can be expanded into a series of sinusoidal waves:

$$X(t) = \sum_n A_n \sin(\omega_n t + \phi_n) \qquad (1)$$

where A_n is the amplitude and ϕ_n is the phase angle of the n^{th} contributing sinusoid. By fixing an array of amplitudes and generating different arrays of phase angles, one obtains different motions with the same general appearance but different details. The computer program uses a random number generator to produce strings of phase angles with uniform likelihood in the range between 0 and 2π.

The amplitudes A_n are related to the (one-sided) spectral density function $G(\omega)$ in the following way:

$$G(\omega_n)\Delta\omega = \frac{A_n^2}{2} \quad (2)$$

Since the total power may be expressed as:

$$\Sigma A\frac{n^2}{2} = \Sigma\, G(\omega_n)\Delta\omega \rightarrow \int_o^\infty G(\omega)\,d\omega \quad (3)$$

where $G(\omega_n)\,\Delta\omega$ may be interpreted as the contribution to the total power of the motion from the sinusoid with frequency ω_n. Allowing the number of sinusoids in the motion to become very large, the total power will become the area under the continuous curve $G(\omega)$.

The power of the motion produced by using equation 1 does not vary with time. To simulate the transient character of real earthquakes, the steady-state motions are multiplied by a deterministic envelope function I(t). The artificial motion Z(t) then becomes:

$$Z(t) = I(t)\sum_n A_n \sin(\omega_n t + \phi_n) \quad (4)$$

The resulting motion is stationary in frequency content with a peak acceleration close to the target peak acceleration. This program has incorporated three different intensity envelope functions such as the "Trapezoidal" envelope (Hou, 1968), the "Exponential" envelope (Lio, 1969) and the "Compound" envelope (Jennings, 1968) functions as shown in Fig. 2. The program artificially raises or lowers the generated peak acceleration to match exactly the target peak acceleration.

The response spectra corresponding to the motion (4) are then computed. The response spectrum for one chosen damping value is called the "target" response spectrum which the programme will attempt to "match" the input spectrum.

To smoothen the calculated spectrum and to improve the matching, an iterative procedure is implemented. In each cycle of the iteration, the calculated response is compared with the target at a set of control frequencies (the user specifies the number of control frequencies). The ratio of the desired response to the computed response is obtained at each control frequency and the corresponding value of the power spectral density is modified in proportion to the square of this ratio, i.e., at any cycle i:

$$G(\omega)_{i+1} = G(\omega)_i \left(\frac{S_v^{(\omega)}}{S_{v(\omega)}^{(i)}} \right)^2 \quad (5)$$

where S_v is the target spectral value. With the modified spectral density function a new motion is generated and a new response spectrum is calculated. The procedure should not be expected to be convergent at all control frequencies; the response at a control frequency is dependent not only on the spectral density function value for that frequency, but also on other values at frequencies close to the frequency of interest as well. Usually, it is not productive to iterate for more than about 4 cycles. If an adequate level of agreement cannot be reached, the user is advised to "start fresh" by generating an entirely new motion (with a new set of random phase angles). For more elaborate explanation of some features of the programme, the reader is referred to Gasparini and Vanmarcke (1976)*.

*Gasparini, D. and Vanmarcke, E.H., "Simulated Earthquake Motions Compatible with Prescribed Response Spectra", M.I.T. Department of Civil Engineering Research Report R76-4, Order No. 527, January 1976.

Running the program SIMQKE.

To run the program call the program by the method appropriate to your operating system. On a personal computer just type **SIMQKE** assuming that the files **SIMQKE.EXE** and **FQWIN.HLP** are in your current directory or path.

In Microsoft Windows operating systems another option is to create a shortcut on the desktop and for this purpose a suitable icon for SIMQKE, **Simqke.ico**, is supplied with the program.

The program prompts for responses to a series of questions. Default responses, where appropriate, are enclosed in square brackets, **[ ]**. File names must match the conventions of your operating system but file names, with paths where necessary, must not exceed 60 characters in length and must not contain blanks.

The first question asks for the name of the output file. The default is the computer console or terminal screen. This is normally the response for most users, however if an output file is created then the envelope values of each graph line drawn is output to that file.

To get hard copies of the plots.

In Microsoft Windows operating systems get hard copies of the graphs use the pull down 'file' menu and select the Print or Save options to send the graph to the printer or to save the plot as a bitmap file (**.BMP)**.
On unix systems using GKS graphics select the Hard-copy option from the Choice window

Note: In the following user guide, each line of required data is indicated by a box containing the data items. Below each box is a description of the data items. The data items on each line may be separated by commas or blank spaces. The format for the items are indicated by the letter at the end of each descriptive line with **A** indicating a character string, **I** indicating an integer value and **F** indicating a floating point number. A floating point number may or may not have a decimal point and may also take a scientific or exponent form such as 1.5E6 which could also be expressed as 1500000.0. Character strings will be upper-cased unless enclosed in double or single quotes and will terminate at the first blank space unless the string is enclosed in quotes.

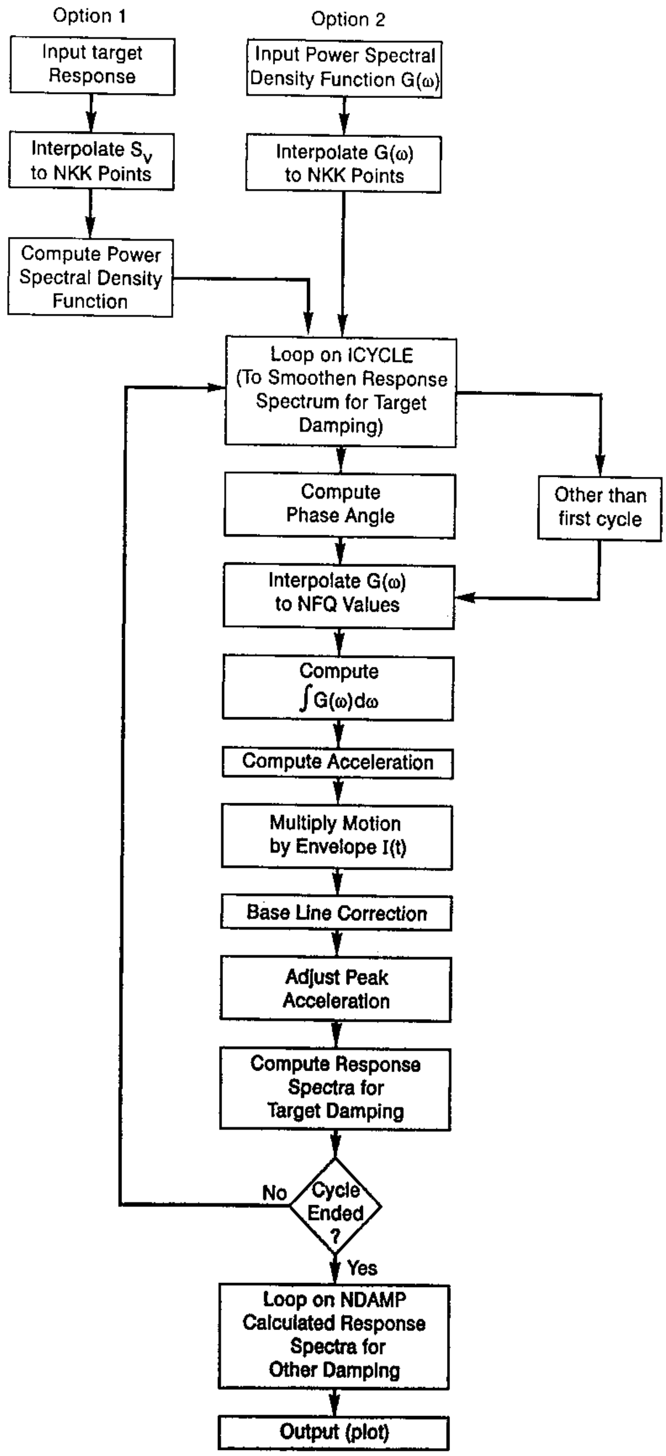


Figure 1. Flowchart for SIMQKE

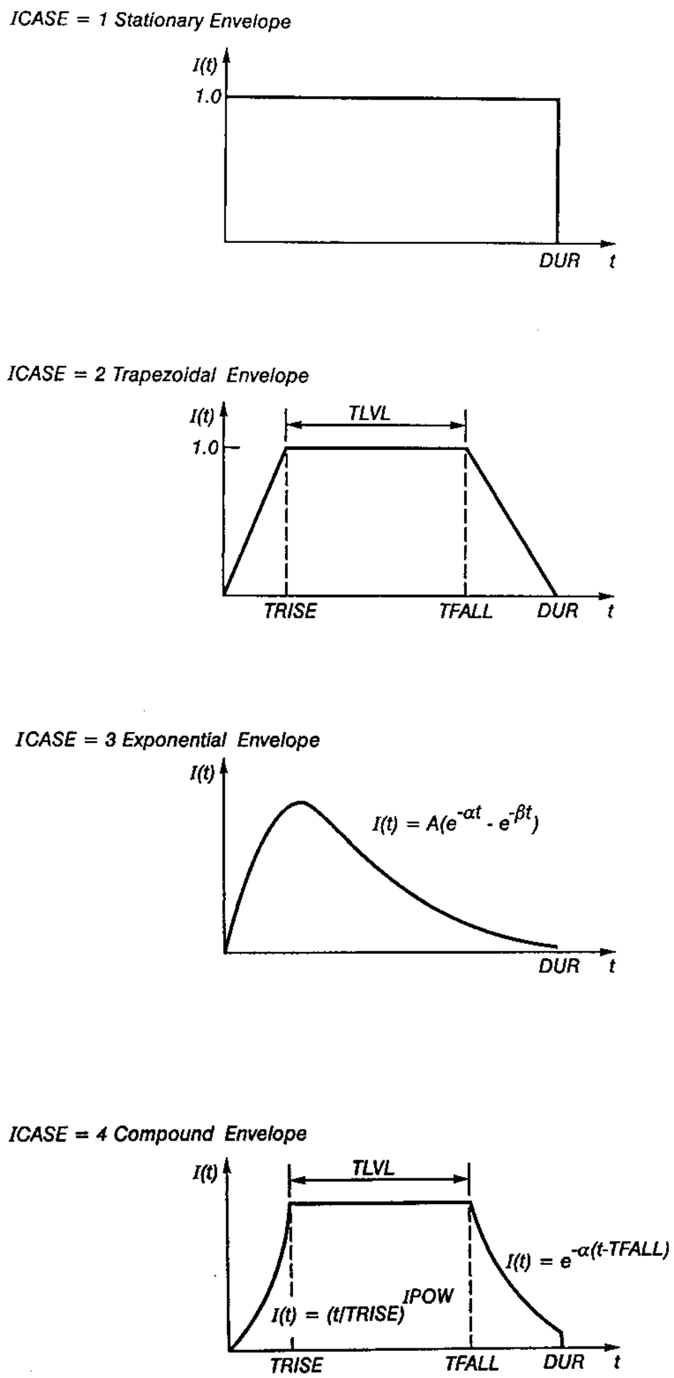


Figure 2. Intensity Envelopes

1. **Name for the Artificial Record.**
This name may be more explanatory than the file name requested by the opening routines.

NAME

NAME	Name for the Artificial Earthquake (Maximum of 72 characters)	**A**

Note: If the name contains blank spaces or lower case characters enclose the response in single or double quotes. e.g. "Artificial NZS 4203"

2. **Frequency Envelope Parameters.**

TS TL TMAX TMIN GRAV ISPEC

TS	Shortest Period of desired Response Spectrum	(seconds) **(>0.0)**	**F**
TL	Longest Period of desired Response Spectrum	(seconds)	**F**
TMIN	Minimum Period for range of Simulation **TMIN** $\geq$ **TS**. If = 0.0 then **TMIN=TS**	(seconds)	**F**
TMAX	Maximum Period for range of Simulation **TMAX** $\leq$ **TL**. If = 0.0 then **TMAX=TL**	(seconds)	**F**
GRAV	Acceleration of Gravity	(length/sec/sec)	**F**
ISPEC	=0: Psuedo Spectral Velocity =1: Spectral Acceleration	(length/sec) (gravity units)	**I**

Note: The parameter **TS** must be greater than zero as logarithmic interpolation is used in the calculations.

3. **Acceleration Time Envelope Parameters.**

ICASE DUR TRISE TLVL A ALFA BETA IPOW

ICASE	=1; Stationary Envelope Parameters =2; Trapezoidal Envelope Parameters =3; Exponential Envelope Parameters =4; Compound Envelope Parameters		**I**
DUR	Accelerogram Duration	(Seconds)	**F**
TRISE	Accelerogram Rise Time	(Seconds) (**ICASE** 2 or 4)	**F**
TLVL	Accelerogram Level Time	(Seconds) (**ICASE** 2 or 4)	**F**
A	Exponential Factor	(**ICASE** 3)	**F**
ALFA	Exponential Factor	(**ICASE** 3 or 4)	**F**
BETA	Exponential Factor	(**ICASE** 3)	**F**
IPOW	Exponential Power Factor	(**ICASE** 4)	**I**

Note: See Figure 2 for the definition of terms and the implied time envelopes. Supply zero values for those items not used for a specific **ICASE**.

4. Accelerogram Control Data.

DELT AGMX RAN NDAMP NCYC NKK NRES NGWK IPCH

DELT	Time Step for Accelerogram (Seconds) (Usually 0.02 or 0.01)	**F**
AGMX	Maximum Acceleration (Gravity)	**F**
RAN	Random Number for Seed purposes (Odd number)	**I**
NDAMP	Number of Damping Ratios for final Spectrum ($1 \leq$ **NDAMP** ≤ 5)	**I**
NCYC	Number of Cycles of Iteration to smoothen spectra match. Experience has shown little improvement for **NCYC** > 5. **NCYC**=1 implies no iteration.	**I**
NKK	Number of Response Spectral Points at equal intervals between **TMIN** and **TMAX** (Maximum is 300, Usually 200 to 300).	**I**
NRES	Number of points to define input Target Spectrum or Spectral Density Function	**I**
NGWK	=0; Target Spectrum is a Response Spectrum Either an Acceleration Spectrum or Psuedo Velocity Spectrum. See **ISPEC** above. =1; Target is Power Spectral Density Function. (**NCYC** is reset to 1)	**I**
IPCH	=0; Accelerogram is **NOT** written to Accelerogram File. Normally used in test runs to see if Spectra generated etc. make sense. =1; Accelerogram is written to Specified File.	**I**

Note: The value of **AGMX** should be slightly larger than the peak acceleration expected for the given Target Spectrum. The peak acceleration value will be scaled to match this value but such an isolated spike in the accelerogram will have minimal effect on the final Spectrum and can be remove manually later. If a too small a values is specified then the whole record will be scaled down so that the peak value matches **AGMX** and the Target Spectrum cannot be matched. In theory it is not realistic to specify both the Peak Acceleration and a Target Response Spectrum. This value is used to control the output if a Power Spectral Density Function is input as the Target.

5. Damping Values. The **NCYC** damping values are input.

DAMP1 DAMP2 DAMP3 DAMP4 DAMP5

DAMP1	First % Critical Damping Ratio. The input Target Spectrum is assumed to be for this % of Critical Damping and this is used for control of the iteration to match Resultant and Target Spectra.	**F**
DAMP2	Second % Critical Damping Ratio.	**F**
DAMP3	Third % Critical Damping Ratio.	**F**
DAMP4	Fourth % Critical Damping Ratio.	**F**
DAMP5	Fifth % Critical Damping Ratio.	**F**

Note: **DAMP2, DAMP3, DAMP4** and **DAMP5** are only used in the generation of the final Response Spectrum for the accelerogram.

6. **Input Target Spectrum**. One line for each of the **NRES** points.

Either

6a. Response Spectrum. NGWK=0

PERIOD VALUE

PERIOD	Natural Period (Seconds)	**F**
VALUE	Spectral Value (Acceleration or Velocity, see **ISPEC** above)	**F**

Note: The Natural Periods must be in ascending numerical order and the shortest period must be less than or equal **TS** and the longest Period must equal or exceed **TL**
Units for variable **VALUE** are dependant on the type of target spectrum. An Acceleration Spectra is (Gravity) units and a Velocity Spectra is in (length/second) units.

Or

6b. Power Spectral Density Function. NGWK=1

OMEGA VALUE

OMEGA	Circular Frequency (Radians/second)	**F**
VALUE	Power Spectral Density.(Length/second)	**F**

Note: The Natural Frequencies must be in ascending numerical order.

End.

UC
UNIVERSITY OF CANTERBURY
Te Whare Wānanga o Waitaha
CHRISTCHURCH NEW ZEALAND

Department of Civil Engineering **COMPUTER PROGRAM LIBRARY**

Program name:– **HYSTERES**	Program type:– Displacement History	Program code:– ANSI Fortran77
Author:– Athol J Carr		Date:– April 28, 2007

HYSTERES

Hysteresis Loop Verification or Tuning

Purpose

The program HYSTERES takes a displacement history and computes the associated hysteresis loop for a specified stiffness, yield strength and post-yield behaviour. Virtually all of the hysteresis models in RUAUMOKO may be specified.

This program may be used to see how a particular rule works but may also be used to determine the best choice of loop parameters to obtain the most suitable hysteresis loop for use in a RUAUMOKO analysis.

The choice of input displacement histories is available:

All data is prompted for and is reasonably self-explanatory.

In Windows95, Windows98 or WindowsNT etc. to get hard copies of the graphs use the pull down 'file' menu and select the Print or Save options to send the graph to the printer or to save the plot as a bitmap file (**.BMP**). On unix systems using GKS graphics select the Hard-copy option from the Choice window.

Running the program HYSTERES.

To run the program call the program by the method appropriate to your operating system. On a personal computer just type **HYSTERES** assuming that the files **HYSTERES.EXE** and **FQWIN.HLP** are in your current directory or path.

In Microsoft Windows operating systems another option is to create a shortcut on the desktop and for this purpose a suitable icon for HYSTERES, **Hysteres.ico**, is supplied with the program.

The program prompts for responses to a series of questions. Default responses, where appropriate, are enclosed in square brackets, **[ ]**. File names must match the conventions of your operating system but file names, with paths where necessary, must not exceed 60 characters in length and must not contain blanks.

The first question asks for the name of the output file. The default is the computer console or terminal screen.

To get hard copies of the plots.

In Microsoft Windows operating systems to get hard copies of the graphs use the pull down 'file' menu and select the Print or Save options to send the graph to the printer or to save the plot as a bitmap file (**.BMP**).
On unix systems using GKS graphics select the Hard-copy option from the Choice window

Input data for Hysteresis.

Note: In the following user guide, each line of required data is indicated by a box containing the data items. Below each box is a description of the data items. The data items on each line may be separated by commas or blank spaces. The format for the items are indicated by the letter at the end of each descriptive line with **A** indicating a character string, **I** indicating an integer value and **F** indicating a floating point number. A floating point number may or may not have a decimal point and may also take a scientific or exponent form such as 1.5E6 which could also be expressed as 1500000.0. Character strings will be upper-cased unless enclosed in double or single quotes and will terminate at the first blank space unless the string is enclosed in quotes.

1 Output file name: supply any suitable file name.

2 Section Properties

STIFF BILIN YP YN

STIFF	Stiffness		**F**
BILIN	Bilinear factor < 1.0	(or Ramberg Osgard function > 1.0)	**F**
YP	Positive Yield Force	(> 0.0)	**F**
YN	Negative Yield Force	(< 0.0)	**F**

3 Hysteresis Choice

IHYST

IHYST	Hysteresis Rule choice, see RUAUMOKO manual for choices **Note:** That Rule 22 cannot be run from HYSTERES	**I**

Some hysteresis rules will require extra data.

3a **IHYST=23** BOUC

DT

DT	Equivalent Time Step	**F**

3b **IHYST = 24** (Remennikov)

LENGTH E A

LENGTH	Member Length	**F**
E	Elastic Modulus	**F**
A	Cross-sectional Area	**F**

3c **IHYST = 33** Masonry Strut

LENGTH

LENGTH	Strut Length	**F**

4 Strength Degradation (only if rules allow degradation – see Appendix B in RUAUMOKO manuals)

ILOS

ILOS	= 0	No strength degradation. See Appendix A of RUAUMOKO manual	**I**
	= 1	Strength reduction in each direction based on its ductility factor	
	= 2	Strength reduction based on number of cycles	
	= 3	Strength reduction based on maximum ductility	

At this point the program will prompt for data on strength degradation if **ILOS > 0** and for hysteresis rule data if required (see RUAUMOKO manuals Appendices A and B).

5 Graph Vertical Grid Division

GRID

GRID	= Number of horizontal grid lines on plots, range is 1 to 5 (Default = 1)	**I**

6 Initial Displacement

DINIT

DINIT	= Initial displacement (Default = 0.0)	**F**

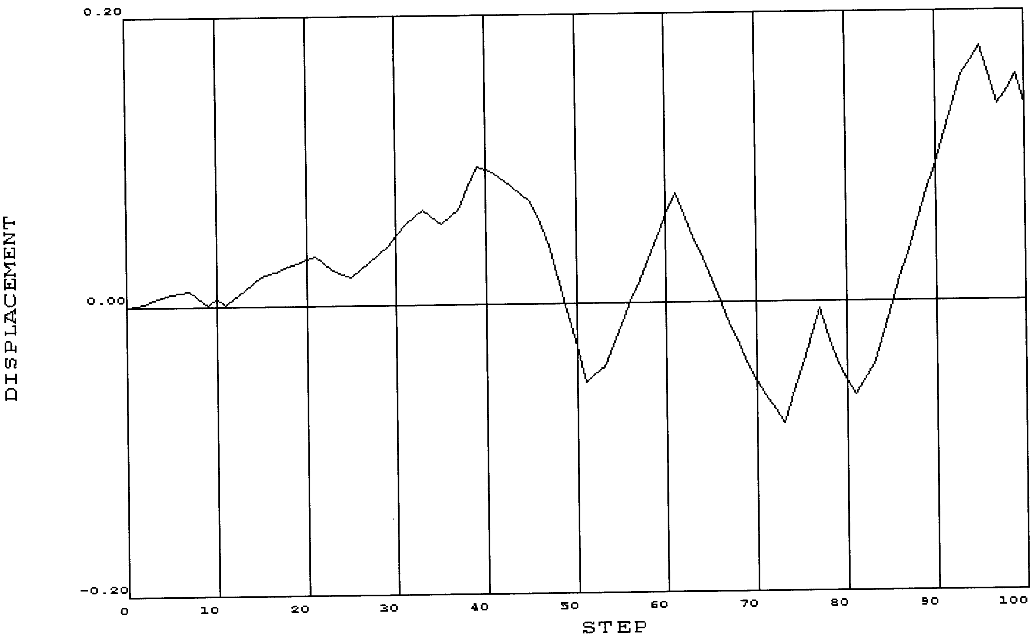
7 Displacement History Choice

IHIST SCALE

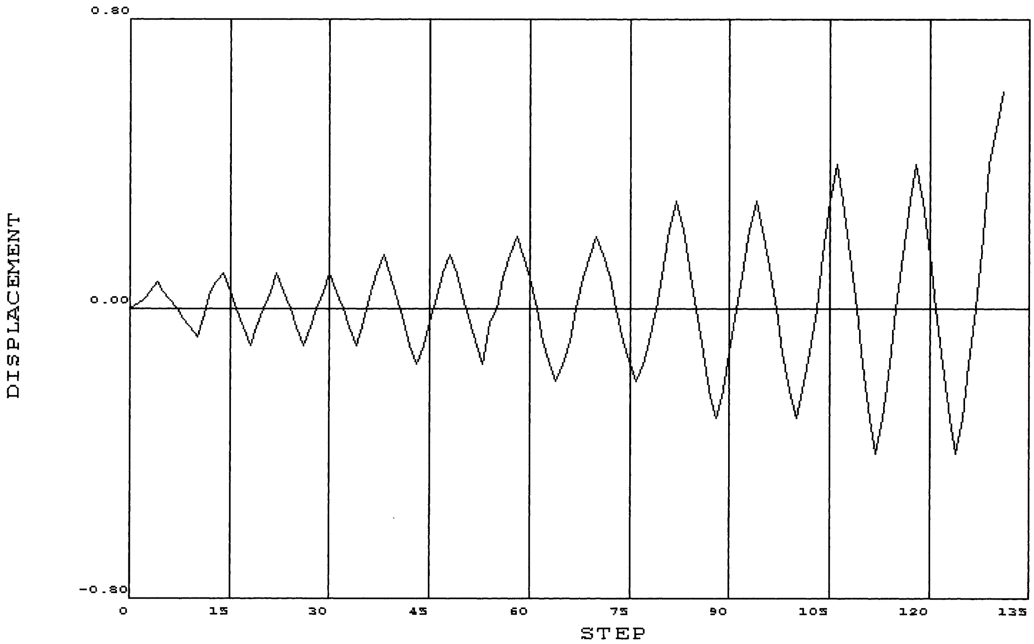
IHIST	= 0	Built-in Displacement History	**I**
	= 1	Built-in Laboratory-Test-like Displacement History	
		1 cycle ductility 0.75	
		3 cycles ductility 1.0	
		2 cycles ductility 1.5	
		2 cycles ductility 2.0	
		2 cycles ductility 3.0	
		2 cycles ductility 4.0	
		then to ductility 6.0	
	= 2	Built-in Sine Wave Displacement History	
	= 3	User Supplied Incremental Displacement History	
	= 4	User Supplied Displacement History	
	= 5	User Supplied Displacement History with Measured Force	
	= 6	User Supplied Displacement History with Measured Force	
	= 7	CUREe Abbreviated Cyclic Displacement History	
	= 8	CUREe Near-Field Cyclic Displacement History	
	= 9	CUREe Standard Cyclic Displacement History	
	= 10	ISO Deformation Protocol	
	= 11	SPD Deformation Protocol	
	= 12	ATC-24 Displacement Protocol	
SCALE	Multiplier on displacement and experimental forces (Default value = 1.0)		**F**
	(Not used for **IHIST** = 7, 8, 9, 10, 11 or 12)		

Notes:

1. The CUREe (California Universities for Research in Earthquake Engineering) Displacement Histories were developed for the Wood-Frame project. They are considered appropriate for the displacement testing of wood structures and components.
2. The ISO Protocol (1997) was prepared by Group 7 of the ISO Technical Committee on Timber Structures, originally for joint testing but later considered appropriate for testing wood-framed shear-walls. The loading history is based on the displacement at ultimate load.
3. The SPD (Sequential Phased Displacement) protocol was developed in 1987 by the Technical Coordinating Committee on Masonry Research (TCCMAR) and this has been modified and accepted by the Structural Engineers Association of Southern California (SEAOSC). The protocol is based on what is called the First Major Event (FME) which can generally be considered as the displacement corresponding to the yield state of the specimen.
4. The ATC-24 Protocol(1992) was developed by Krawinkler for the testing of steel components. The displacement history is controlled by the yield displacement of the specimen.



IHIST = 0 Built-in Displacement History



IHIST = 1 Built-in Displacement History
(In this example the yield displacement is ± 0.08.)

8 **Displacement History** (only if **IHIST** $\geq$ 2)

8a if **IHIST** = 2

AMP OMEGA DT DUR

AMP	=	Amplitude of Displacement Wave	**F**
OMEGA	=	Period (seconds)	**F**
DT	=	Time Each Step (seconds)	**F**
DUR	=	Duration (seconds)	**F**

8b **IHIST** = 3

DR1 DR2 ... DRN

DR1	=	First displacement increment	**F**
DR2	=	Second displacement increment	**F**
...			
DRN	=	Nth displacement increment	**F**

Use as many lines as required, minimum of one displacement increment per line.
End with word **STOP** as last displacement increment.

8c **IHIST** = 4

R1 R2 ... RN

R1	=	First displacement	**F**
R2	=	Second displacement	**F**
...			
RN	=	Nth displacement	**F**

Use as many lines as required, minimum of one displacement per line.
End with word **STOP** as last displacement.

8d **IHIST** = 5

Ri Fi

Ri	=	Displacement	**F**
Fi	=	Experimental Force	**F**

One line per displacement step.
End with line where **Ri** is word **STOP**

8e **IHIST** = 6

Ni Ri Fi

Ni	=	Line number	**I**
Ri	=	Displacement	**F**
Fi	=	Experimental Force	**F**

One line per displacement step.
End with line where **Ni** is word **STOP**

The program will compute hysteresis rule behaviour outputting Force, Displacement stiffness and plot flag values at each step.

After **STOP** is encountered in the input the program will plot

a. The displacement history
b. The force history compiled by the hysteresis rule
c. The hysteresis loop generated.

If **IHIST** is 5 or 6 above the last two plots will show the experimental force history and experimental loop in red.

8f **IHIST** = 7, 8 or 9 CUREe Displacement History.
(California Universities for Research in Earthquake Engineering)

DREF

DREF	Reference deformation. This is usually taken as 60% of the deformation where the load carrying ability of the system has fallen to 80% of the maximum load sustained by the system.	**F**

8g **IHIST** = 10 ISO Deformation Protocol.

DMAX

DMAX	Delta Maximum.	**F**

8h **IHIST** = 11 SPD Deformation Protocol.

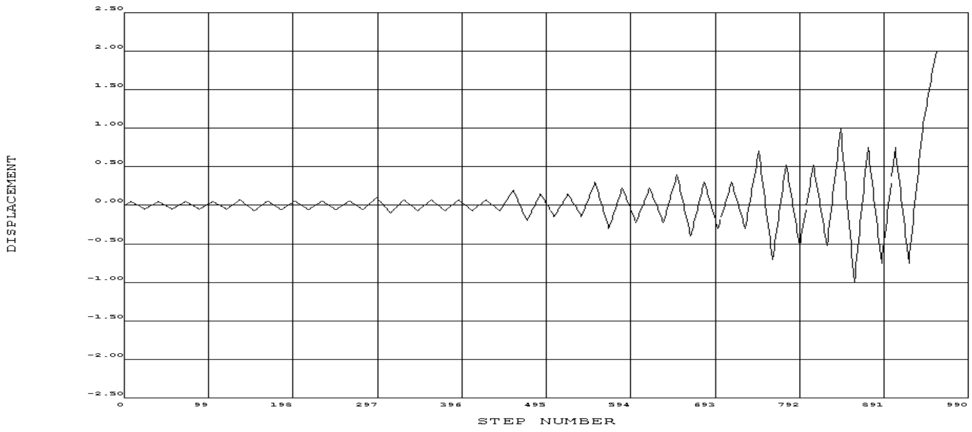
DFME

DFME FME Displacement (Default = 0.75). **F**

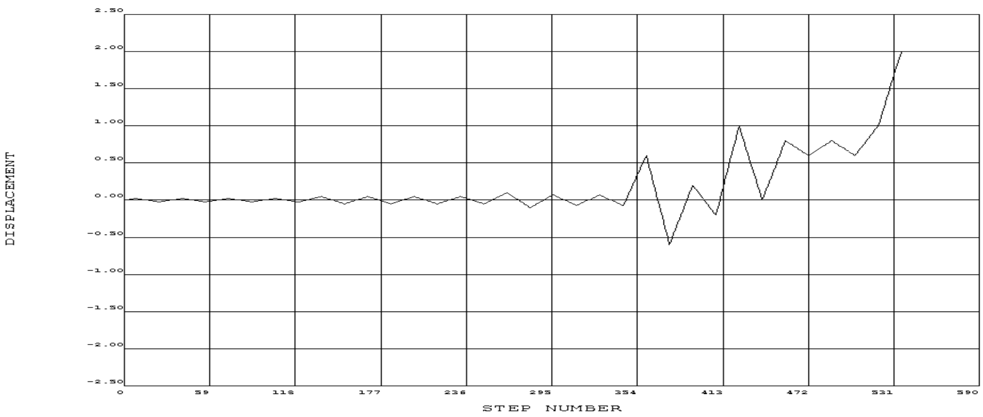
8i **IHIST** = 12 ATC-24 Deformation Protocol.

DYIELD

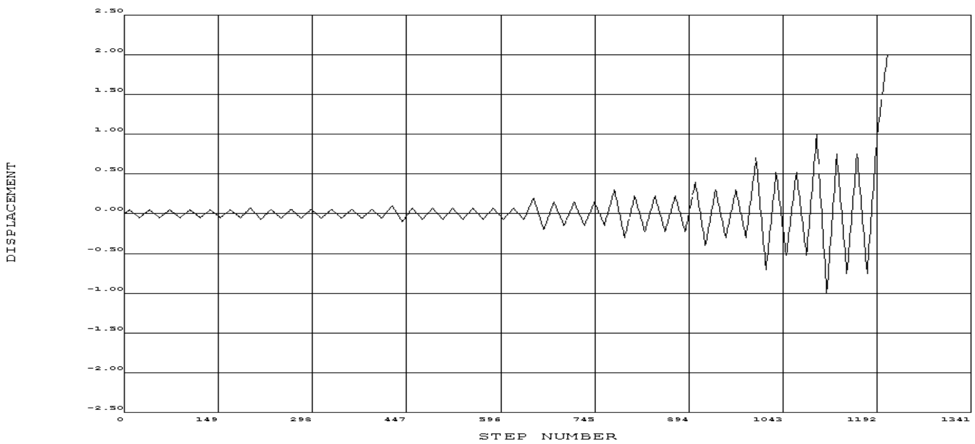
DYIELD Yield Displacement **F**



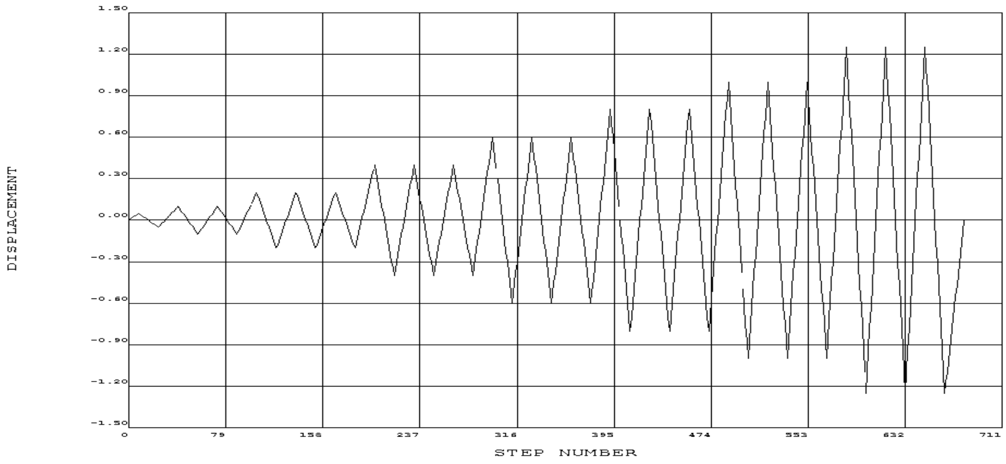
CUREe Abridged Displacement History (to be multiplied by DREF)



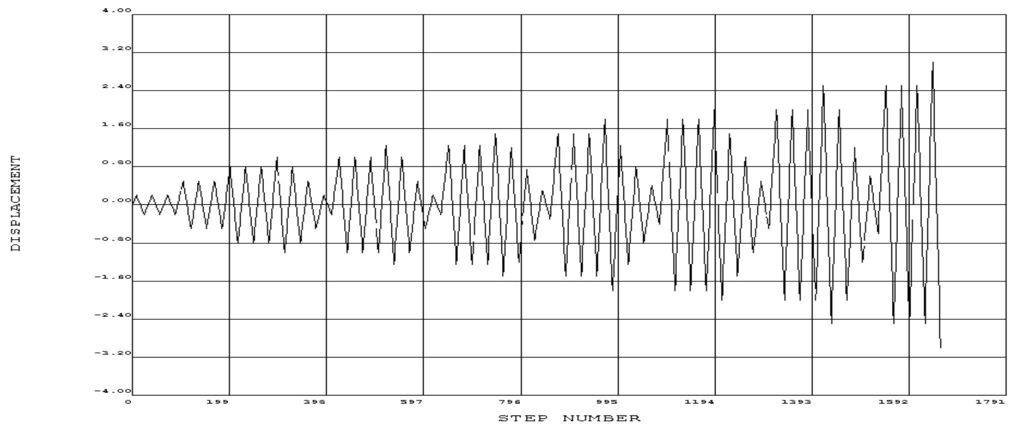
CUREe Near-Field Displacement History (to be multiplied by DREF)



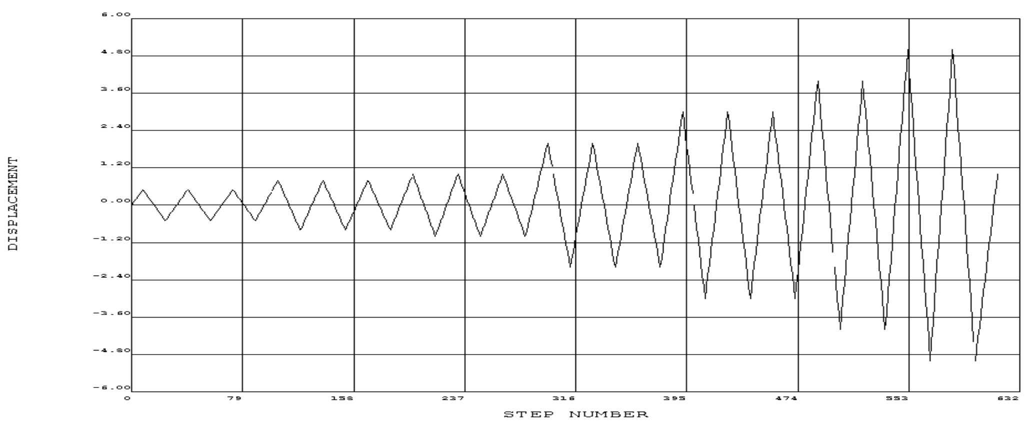
CUREeE Standard Displacement History (to be multiplied by DREF)



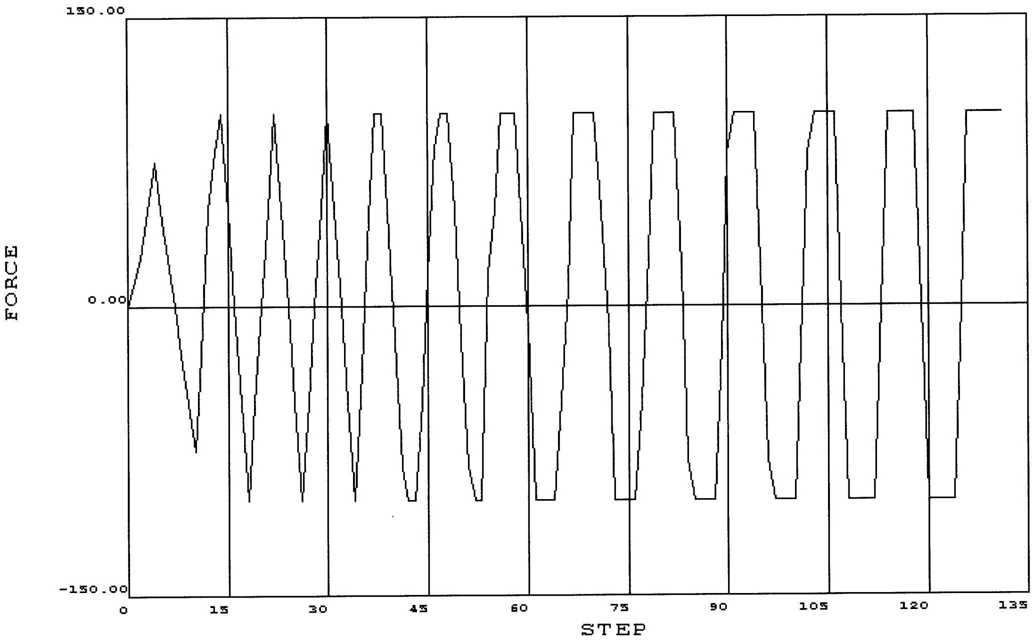
ISO Displacement Protocol (to be multiplied by DMAX)



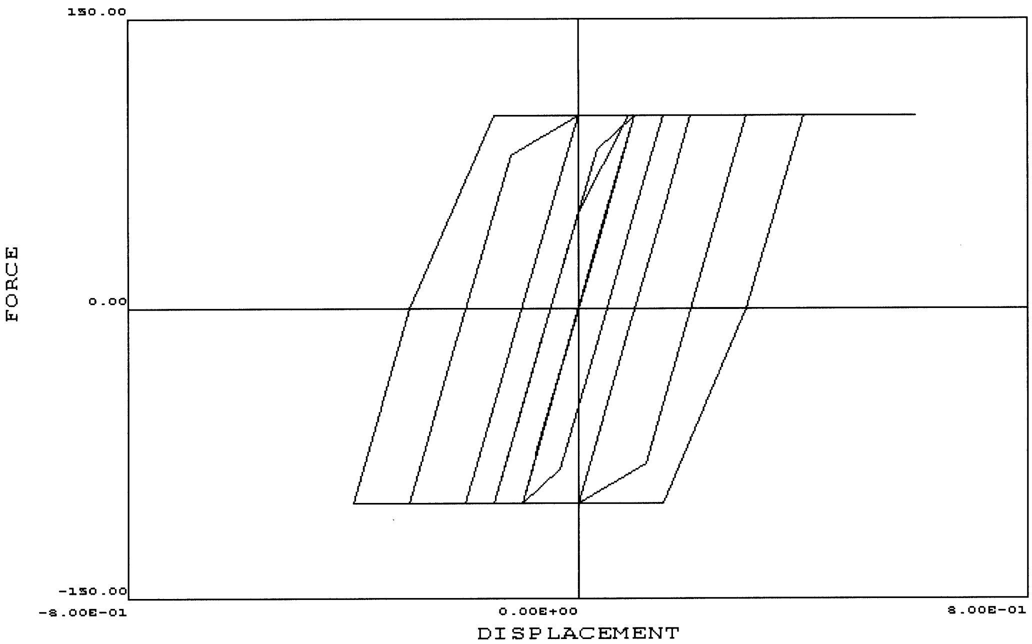
SPD Displacement Protocol (to be multiplied by DFME)



ATC-24 Displacment Protocol (to be multiplied by DYIELD)



Example of Force Versus Step Output
for system following **IHIST** = 1 Built-in Displacement History



Example of Bi-linear Hysteresis Loop
for system following **IHIST** = 1 Built-in Displacement History

This page is intentionally left blank.

Department of Civil Engineering **COMPUTER PROGRAM LIBRARY**

Program name:– **INSPECT**	Program type:– Response Spectrum Analysis	Program code:– ANSI Fortran77
Author:– Athol J Carr		Date:– September 27, 2011

INSPECT

In-elastic Response Spectra Computation

Purpose

This program is designed to compute and plot the inelastic response spectra for an input earthquake accelerogram and for most of the hysteresis rules available in the program RUAUMOKO . The program computes the Spectral Displacement, the Spectral Acceleration, the yield force spectrum, the elastic and plastic work as well as the residual displacement spectra for the given The output is both tabular and graphical. The input formats for the accelerograms are the same as those formats accepted by the program RUAUMOKO.

Method of Analysis

The program asks for information on the range of natural periods, period interval and damping ratios before reading and digitizing the accelerogram. It then, for each ductility ratio, computes the responses of single degree of freedom in-elastic oscillators where the natural period of free vibration is varied over the period range specified. The program iterates with the choice of yield force, up to 200 cycles, to obtain the target ductility values.

Running the program INSPECT.

To run the program call the program by the method appropriate to your operating system. On a personal computer just type **INSPECT** assuming that the files **INSPECT.EXE** and **FQWIN** .HLP are in your current directory or path.

In Microsoft Windows operating systems another option is to create a shortcut on the desktop and for this purpose a suitable icon for INSPECT, **Inspect.ico**, is supplied with the program.

The program prompts for responses to a series of questions. Default responses, where appropriate, are enclosed in square brackets, **[ ]**. File names must match the conventions of your operating system but file names, with paths where necessary, must not exceed 60 characters in length and must not contain blank characters.

The first question asks for the name of the output file. The default is the computer console or terminal screen. This is normally the response for most users, however if an output file is created then the envelope values of each graph line drawn is also output to that file.

The second question is whether the user wishes to create an ASCII post-processor file as well as the conventional output file. This file, if it is output, has for each ductility a line for each natural period which contains the program graph output data:

- The natural period,
- The maximum displacement
- The maximum acceleration
- Elastic Energy
- Plastic work
- Yield force to give the required ductility.
- Residual displacement.
- Ductility Match
- Reduction Ratio (seismic coefficient as a function of the elastic acceleration spectra)

The default answer is No. If the answer is Yes, then the user is prompted for the file name, the default having the same name as the output file but with the extension **.SAS**

The next question is for the name of the input earthquake accelerogram. This data is the same as asked for by SPECTRA. If the accelerogram does not have time-acceleration pairs of data as in the Caltech, NCEER, CSMIP and PEER format accelerogram the user is prompted for the accelerogram time-step **deltat**. The user is then prompted for the accelerogram scale factor **ascale** required to convert the accelerations to the acceleration of gravity. The final question is the line number **istart** at which the record is to start. The default value is line 1.

The program will then prompt for the first line of the data file. If this already exists then one may respond with

$add <datafilename>

or the user may supply the data as requested by each prompt. The data is described in the following sections.

To get hard copies of the plots.

In Microsoft Windows operating systems, to get hard copies of the displayed graphs use the pull down **file** menu and select the **Print** or **Save** options to send the graph to the printer or to save the plot as a bitmap file (**.bmp)**. If the plot is to be sent directly to the printer it is recommended that **landscape** be the chosen orientation and unless a colour printer is being used then the graphics option should have been **monochromatic** and not **colour**.

On a Microsoft Windows operating system the bitmap plots can be converted to jpeg files (**.jpg**) by opening the bitmap file in Microsoft **Paint** (right-click on file name and select **open with** and then select **Paint**). When In Paint select **Save As** option and select file type **jpeg**. The resulting jpeg files are a fraction of the size of the bitmap files.

On unix operating systems using the GKS graphics system select the Hard-copy option from the Choice window.

Input data for INSPECT.

All data for INSPECT is prompted for by the program using the TinyCLIP command processor. For help on the input data or on the use of the INSPECT program type **HELP** or **?** at any prompt. To get help on the command processor type **$HELP** at any prompt.

Note: In the following user guide, each line of required data is indicated by a box containing the data items. Below each box is a description of the data items. The data items on each line may be separated by commas or blank spaces. The format for the items are indicated by the letter at the end of each descriptive line with **A** indicating a character string, **I** indicating an integer value and **F** indicating a floating point number. A floating point number may or may not have a decimal point and may also take a scientific or exponent form such as 1.5E6 which could also be expressed as 1500000.0. Character strings will be upper-cased unless enclosed in double or single quotes and will terminate at the first blank space unless the string is enclosed in quotes.

1 Output File name. (Not used in **Batch** mode, as this is part of the command line)

NAME

NAME Output file name (including path if necessary). Maximum 60 characters **A**

The default name is SCREEN which means that output is only put to the output screen and no electronic copy is made. The name will be upper-cased and will terminate at the first blank character unless the name (and path) is enclosed in quotes (single ' or double "). On Windows systems an extension **.wri** (WordPad) or **.txt** (NotePad) extension is recommended

2 Secondary output file. (Not used in **Batch** mode, as this is part of the command line)

SAS SAZ

SAS	**Yes/No**	Is a secondary output text file required?	(default= **No**)	**A**
SAZ	**Yes/No**	Is a Tracinf output text file required?	(default= **No**)	**A**

This file contains the output from the program with all data for a given ductility arranged in columns that may be easily input to a spreadsheet ..
If **SAZ** is Yes then you are later prompted for the period at which the displacement, acceleration and force are recorded, plotted and a hysteresis plot is also made. The three time-history values are output in the **SAZ** file

2a SAS file name**. (**Only if **SAS** in line 2 above is **Y** or **YES)**
(Not used in **Batch** mode, as this is part of the command line)

SASNAME

SASNAME Output file name (including path if necessary). Maximum 60 characters **A**

The default name is the same as that of the output file in part 1 but with the extension **.SAS**. The default name will be over-ridden by any supplied name.

2b **SAZ** file name**.** **(**Only if **SAZ** in line 2 above is **Y** or **YES)**
(Not used in **Batch** mode, as this is part of the command line)

SAZNAME

SAZNAME	Output file name (including path if necessary). Maximum 60 characters	**A**

The default name is the same as that of the output file in part 1 but with the extension **.SAZ.** The default name will be over-ridden by any supplied name.

3 **Earthquake Accelerogram name** (Not used in **Batch** mode, as this is part of the command line)

EQNAME

EQNAME	File name (including path if necessary) for earthquake accelerogram file Maximum 60 characters	**A**

The name will be upper-cased and will terminate at the first blank character unless the name (and path) is enclosed in quotes (single ’ or double ”) This must be one of the standard formats, BERG, CALTECH, NCEER, FREE, CSMIP, EXCEL, PEER or SAC as used by Ruaumoko.

The program recognizes the following file extensions as denoting the following accelerogram formats:

.eqb	Berg format records
.eqc	Caltech format records
.eqn	NCEER format records
.eqf	Free format records
.eqs	CSMIP format records
.eqe	EXCEL format records
.eqp	PEER format records
.eqk	SAC format records

For details of the formats see the section on accelerogram later in the manual

3a **Earthquake Accelerogram File Type (**only if file type is not recognized from the file extension**)**
(Not used in **Batch** mode, as this is part of the command line)

TYPE

TYPE	**B** or **Berg**	Berg format record	**A**
	C or **Caltech**	Caltech format record	
	N or **Nceer**	NCEER format record	
	F or **Free**	Free format record	
	S or **CSmip**	CSMIP format record	
	E or **Excel**	Excel format record	
	P or **Peer**	PEER format record	
	K or **SAc**	SAC format record	

The **TYPE** may upper or lower case and sufficient latters supplied to be distinctive (ie**.C** or **CS** above)

.

4. Earthquake Accelerogram **DELTAT** (only if file type is not Berg, Free or Excel Format above**)**
(Not used in **Batch** mode, as this is part of the command line)

DELTAT

DELTAT Time-step at which accelerogram is digitized **F**

The default answer is that most commonly used for that type record.

CALTECH Format records the record time-interval	(usually 0.025, 0.02, 0.01 or 0.005 seconds)
NCEER Format records the record time-interval	(usually 0.02 seconds.) (Note NZ GNS records are at 0.005 seconds)
CSMIP Format records the record time-interval	(usually 0.02 seconds)
PEER Format records the record time-interval	(usually 0.004 seconds)
SAC Format records the record time-interval	(usually 0.020 seconds)

5. Earthquake Accelerogram scale factor **ASCALE** (only if file type is not Berg, Free or Excel Format)
(Not used in **Batch** mode, as this is part of the command line)

ASCALE

ASCALE Scale factor (Divisor to convert to fractions of acceleration of gravity "g") **F**

The default answer is that most commonly used for that type record.
Divisor **ASCALE** is required to convert the accelerogram to the units of the acceleration of gravity. For example, if the acceleration record is in units of cm/sec/sec the ASCALE is 981 If the acceleration record is already in the units of the acceleration of gravity the **ASCALE** is 1.0 unless the user also wishes to scale the record to some other magnitude.
BERG, FREE or EXCEL Format records omit or set equal to 1.0
CALTECH records are usually in units of acceleration multiplied by a constant as the **F6.0** format is usually an Integer **I6** format and the decimal point is missing with about 5 integer digits for the largest numbers. The values must be converted units of the acceleration of gravity .
If the record is in the units of mm/sec/sec and if **g** = 9.81 m/sec/sec. then **ASCALE** = 9810.0.
NCEER and CSMIP records are usually in cm/sec/sec and **ASCALE** = 981.0 to bring the record to units of the acceleration of gravity. (Note, NZ GNS records are in mm/sec/sec so **ASCALE** = 9810.0)

6 First line in record to start **ISTART** (Not used in **Batch** mode, as this is part of the command line)

ISTART

ISTART First line in record to start calculuation (Default =1) **I**

The user is also prompted for which line, **ISTART**, in the earthquake accelerogram record the analysis is to start. The default value is 1, i.e. the record starts at the first line of the data but in some cases, particularly where the input is a recorded shake-table record there may be a long lead-in with very little happening, i.e. between switching on the recording equipment and the start of the shake-table motion , and this gives a means of omitting this from the calculations.

7. Hysteresis Choice

IHYST

IHYST Hysteresis Rule choice, see RUAUMOKO manuals for choices **I**

Note: That Rules 22 and 24 cannot be run from INSPECT

8. Bi-linear factor or Ramberg-Osgood r factor. Only if **IHYST** is greater than 1.

R

R Bi-linear factor or Ramberg-Osgood r factor **F**

9. Strength degradation Choice

ILOS

ILOS = 0 No strength degradation **I**
= 1 Strength reduction in each direction based on ductility in that direction
= 2 Strength reduction based on number of inelastic cycles
= 3 Strength reduction based on maximum ductility

10. Strength Degradation Parameters (only if **ILOS** greater than 0, see Appendix A in RUAUMOKO manuals). (only if rules allow degradation – see Appendix B in RUAUMOKO manuals)

11. Hysteresis Parameters (Only if **IHYST** greater than 0 and only if the hysteresis rule required further data, see Appendix B) Note that nay forces, such as cracking forces, are taken as a fraction of the yield force and any displacements are taken as a fraction of the yield displacement.

Some hysteresis rules will require extra data, (For required data description please see Appendix B of Ruaumoko manuals)
for example
.
11a IHYST = 33 Masonry Strut

LENGTH

LENGTH Strut Length **F**

12. Number of Ductilities

ND TOL PDELTA IOP

ND	Number of extra Ductilities. (1 ≤ **ND** ≤ 5) The first ductility is elastic = 1.0	**I**
TOL	% Convergence tolerance for ductility match. (0.1% ≤ **TOL** ≤ 10.0%) (If left blank or 0.0 supplied then **Default** = 0.5%)	**F**
PDELTA	No or blank, P-Delta effects are not included Yes, P-Delta effects are included	**A**
IOP	= 0 or blank: Exponential function used for ducitility convergence (**Default**) = 1; Linear interpolation used for ductility convergence (problematic) = 2; Binary Chop procedure used for ductility convergence.	**I**

13. Ductility Ratios. (ND numbers)

D2 D3 D4 D5 D6

Di	Ductility ratios	**F**

Note: The first ductility value **D1** is always 1.0 (elastic)
The default value of **D2** if **ND** = 1 is 2.0
The default values of **D2** and **D3** if **ND** = 2 are 2.0 and 4.0
The default values of **D2**, **D3** and **D4** if **ND** = 3 are 2.0, 4.0 and 6.0
The default values of **D2**, **D3**, **D4** and **D5** if **ND** = 4 are 2.0, 4.0, 6.0 and 8.0
The default values of **D2**, **D3**, **D4**, **D5** and **D6** if **ND** = 5 are 2.0, 3.0, 4.0, 6.0 and 8.0

14. Length and Weight. (Only if **PDELTA** on line 6 above is **Y** or **YES**)

L P

L	Column Length	**F**
P	Vertical Load on Column	**F**

Note: if either **L** or **P** is less than or equal to zero then the **PDelta** option is reset to **NO**

14. % Damping ratio.

DAMP IDAMP KAPPA

DAMP	% damping ratio	**F**
IDAMP	0 = ; (or blank) Constant Damping coefficient retains initial elastic value (**Default**) 1 = ; Secant Damping Ratio remains constant 2 = ; Tangent Damping Ratio remains Constant 3 = ; Rayleigh Secant Damping using Tangent Stiffness 4 = ; Rayleigh Tangent damping using Tangent Stiffness	**I**
KAPPA	Fraction of Rayleigh Damping from Mass term (Constant contribtion) (0.0 to 1.0) (Only used for **IDAMP** = 3 or 4)	**F**

Note: Secant Damping means that damping force = Damping*velocity
Tangent Damping means that the increment of the damping force = damping*increment in velocity

15. Number of Natural Periods of Free-Vibration.

NP

NP	Number of Natural periods of free vibration (Default=100, Maximum=500)	**I**

16. Natural Period of Free-vibration Step. (Seconds).

DPE

DPE	Natural period of free-vibration step (seconds)	**F**

17. Accelerogram Integration Time-step. (Seconds)

DT

DT	Integration time step (seconds) (**Default** = Min^{m} (0.01 seconds, **DPE**/10, **DELTAT**)	**F**

18. Duration of Integration. (Seconds)

TIME

TIME Duration of integration (seconds) **F**

Note: If the duration exceeds the length of the accelerogram then the remaining part of the integration is free-vibration.

19 Duration of Free Vibration following Earthquake Integration. (Seconds)

FREE

FREE Duration of free vibration following earthquake integration (seconds) **F**

Note: This is only of significance if there is any likelihood that the systems are still non-linear at the end of the earthquake integration duration or if accurate residual displacements are required.

20. Acceleration of Gravity.

GRAV

GRAV Acceleration of gravity (**Default** = 9.81) **F**

20a. Trace Period. (Only if **SAZ** is **Y, YE** or **YES)**

NTRACE

NTRACE Period number at which trace information is saved and plotted (**Default** = 21) **I**
The period is **(NTRACE-1)** times the period step **DPE. If DPE** is 0.05 seconds and **NTRACE** is 21 then the data is stored for the 1 second period

21. Are graphs to be plotted. (Not used in **Batch** mode)

IPLOT

IPLOT Graphs plotted (**Yes/No**, the default is **Yes**) **A**

22. **Earthquake Title**. Only if **IPLOT** is **Yes**. (Not used in **Batch** mode)

TITLE

TITLE	Title for graphs. Maximum of 30 characters	**A**

Note: If the title contains blank characters or the case is to be maintained then enclose the title in single or double quotation marks.

23. **Graph Scale factors.** Only if **IPLOT** is **Yes**. (Not used in **Batch** mode)

SD SA SE SP YF RD

SD	Maximum value for Spectral Displacement	**F**
SA	Maximum value for Spectral Acceleration	**F**
SE	Maximum value for Spectral Elastic Work	**F**
SP	Maximum value for Spectral Plastic Work	**F**
YF	Maximum value for Spectral Yield Force	**F**
RD	Maximum value for Spectral Residual Displacement	**F**

Note: If zero values are supplied then an automatic scaling is used for the plots.

24. **Number of Grid Lines up the Plots.** Only if **IPLOT** is **Yes.** (Not used in **Batch** mode)

NG

NG	Number of grid spaces in the vertical axis in the spectra plots. (Default = 5)	**I**

Accelerogram Data

a. Accelerogram flag.

One input line with the word **STAR**, **START** or **DATA:** (the colon is mandatory) starting in column 1 and the word must be in upper case. This **START** line may be preceded by as many header lines as desired. This **START** line is not used for PEER or SAC format records as these records start with a fixed 4 or 2 lines of header information respectively.

START

b. Accelerogram

The remainder of the input is the acceleration record itself. The record is in the form of a series of lines each of which starts with a *Line Sequence Number* (which must be in an ascending order) followed by either (i) a group of 4 or 1 successive time-acceleration points (BERG, FREE or EXCEL Format), or (ii) a sequence of 10, 8 or 4 uniformly spaced acceleration values at **DELTAT** time intervals apart, the (CALTECH, NCEER, CSMIP, PEER or SAC Formats).

Note that the NCEER, CSMIP, EXCEL, PEER or SAC records do not have a sequence number. The analysis acceleration record will begin at the first time on or implied by the beginning of the accelerogram line **ISTART** and there must then be sufficient lines remaining to span the analysis time-history length **TR**.

The record must be on one of the following formats depending on the value of **IBERG** provided in the earlier data. The FORTRAN format is provided in parentheses for each case.

The **I3** format implies 3 characters for the number which is right justified, **6X** implies 6 blank characters. The **F8.4** implies 8 characters for the number and if omitted the decimal point is located in front of the 4th to last character in the 8 character set which is assume to be right justified, if the decimal point is provided the number may be located anywhere in the 8 character space. A number preceding the format, i.e. **10F8.4** means that there are 10 numbers each having an 8 character field. Each format starts on a new line.

(1) **BERG** FORMAT **(I3,4(F8.4,F9.6))** (Default filename extension is **.eqb**)

ISEQ T1 G1 T2 G2 T3 G3 T4 G4

ISEG	Line sequence number	**I 3**
Ti	Time of point on accelerogram (seconds)	**F 8.4**
Gi	Acceleration (decimal fraction of gravity)	**F 9.6**

If the line sequence number is greater than 999 it is not read or checked by the program.

(2) **CALTECH** FORMAT **(I4,6X,10F6.0)** or more precisely **(I4,6X,10I6)**(Default filename extension is **.eqc**)

ISEQ G1 G2 G3 G4 G5 G6 G7 G8 G9 G10

ISEQ	Line sequence number	**I 4**
Gi	Acceleration (multiplied by **ASCALE**) at intervals of **DELTAT**	**I 6**

If the line sequence number is greater than 9999 it is not read or checked by the program.

(3) **NCEER** FORMAT **(10F8.2)** (Default filename extension is **.eqn**)

G1 G2 G3 G4 G5 G6 G7 G8 G9 G10

Gi	Acceleration (multiplied by **ASCALE**) at intervals of **DELTAT**	**F 8.2**

(4) **FREE** FORMAT **(*)** (Default filename extension is **.eqf**)

ISEQ T1 G1

ISEQ	Line sequence number	**I**
T1	Time of point on accelerogram (seconds)	**F**
G1	Acceleration (decimal fraction of gravity)	**F**

The three items may be placed anywhere on the line and separated by at least one blank column. The lines must be in consecutive order with **ISEQ** starting at 1 and increasing line by line. This format is particularly useful where the excitation record has been generated on a spreadsheet.

(5) **CSMIP** FORMAT **(8F10.3)** (Default filename extension is **.eqs**)

G1 G2 G3 G4 G5 G6 G7 G8

Gi	Acceleration (multiplied by **ASCALE**) at intervals of **DELTAT**	**F 10.3**

(6) **EXCEL** FORMAT **(*)** (Default filename extension is **.eqe**)

T1 G1

T1	Time of point on accelerogram (seconds)	**F**
G1	Acceleration (decimal fraction of gravity)	**F**

The three items may be placed anywhere on the line and separated by at least one blank column. The lines must be in consecutive order. This format, which is similar to the FREE format except without the sequence numbers is particularly useful where the excitation record has been generated on a spreadsheet.

(6) **PEER** FORMAT **(5E15.7)** (Default filename extension is **.eqp**)

G1 G2 G3 G4 G5

Gi	Acceleration (multiplied by **ASCALE**) at intervals of **DELTAT**	**E15.7**

Note: The accelerogram time-step **DELTAT** is usually 0.004 seconds.

(7) **SAC** FORMAT **(6E13.5)** (Default filename extension is **.eqk**)

G1 G2 G3 G4 G5 G6

Gi	Acceleration (multiplied by **ASCALE**) at intervals of **DELTAT**	**E 13.5**

Note: The accelerogram time-step **DELTAT** is usually 0.020 seconds.

Running INSPECT in batch mode.

The program may be run in a batch mode as well as the interactive mode described above. In the batch mode the graphics is disabled.

The **.BAT** file has the command line as follows:

INSPECT outputfile inputfile quake1file format deltat ascale istart save tsave

In Windows95 or Windows98 operating systems the command **INSPECT** should be preceded by **START/w** to prevent the system from attempting to multi-task following command lines.

If the **INSPECT** executable file **INSPECT.EXE** is not in your path then the path should be part of the command.

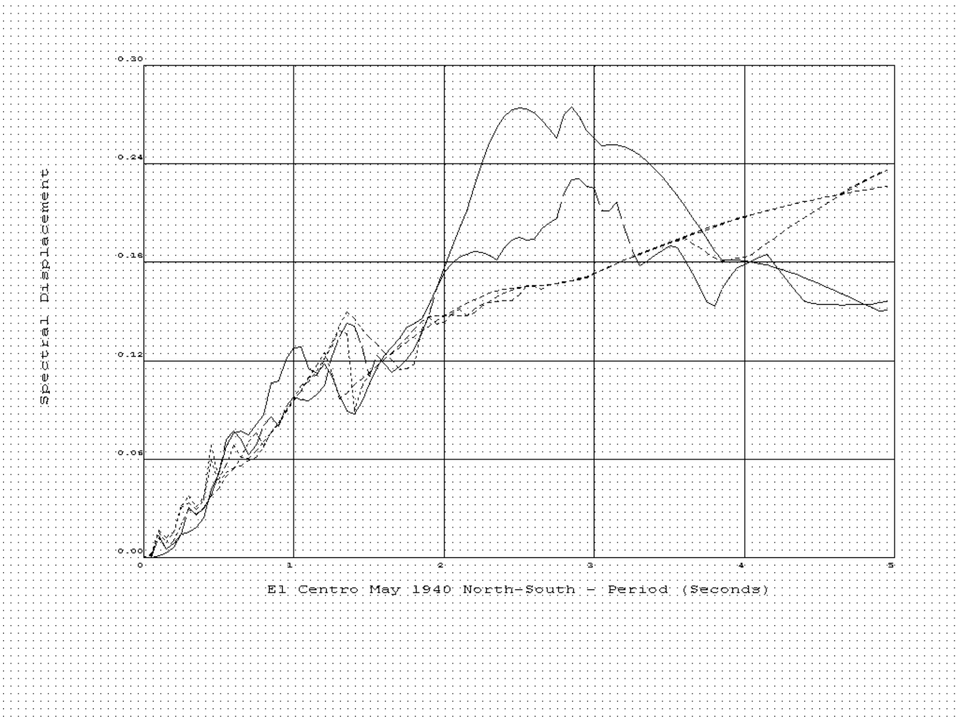
1. The **outputfile** is the name (including the path) of the output file. This file must NOT exist. If the file does exist the analysis will terminate immediately.
2. The **inputfile** is the name (including path) of the input data file (in the form required in the following part of this manual) and the file MUST exist. If the file cannot be found, because the name or path is incorrect then the analysis will terminate.
3. The file **quake1file** is the name (including path) of the excitation file. This file MUST exist or the analysis will be terminated.
4. The **format** is the word **Berg**, **Caltech**, **NCEER**, **CSMIP**, **Free**, **Excel, Peer** or **SAC**, or at least the sufficient letters of the word to uniquely define the accelerogram format.
5. The **deltat** is the time-step of the accelerogram. If Berg or Free format etc. supply 1.0
6. The **ascale** is the scale factor required to convert the acceleration values to the acceleration of gravity. If no scaling then **ascale=1.0**
7. The **istart** is the number of the first line of the accelerogram to use.
8. The **Save** is the word SAVE, in upper case if the user wishes to create the ASCII post-processor file which will have the same root as the name of the outputfile but with the extension **.SAS**.
9. The **Tsave** is the word SAVE, in upper case if the user wishes to create the ASCII Trace data file which will have the same root as the name of the outputfile but with the extension **.SAZ**.

As many such command lines as desired may be arranged in a **.bat** file.

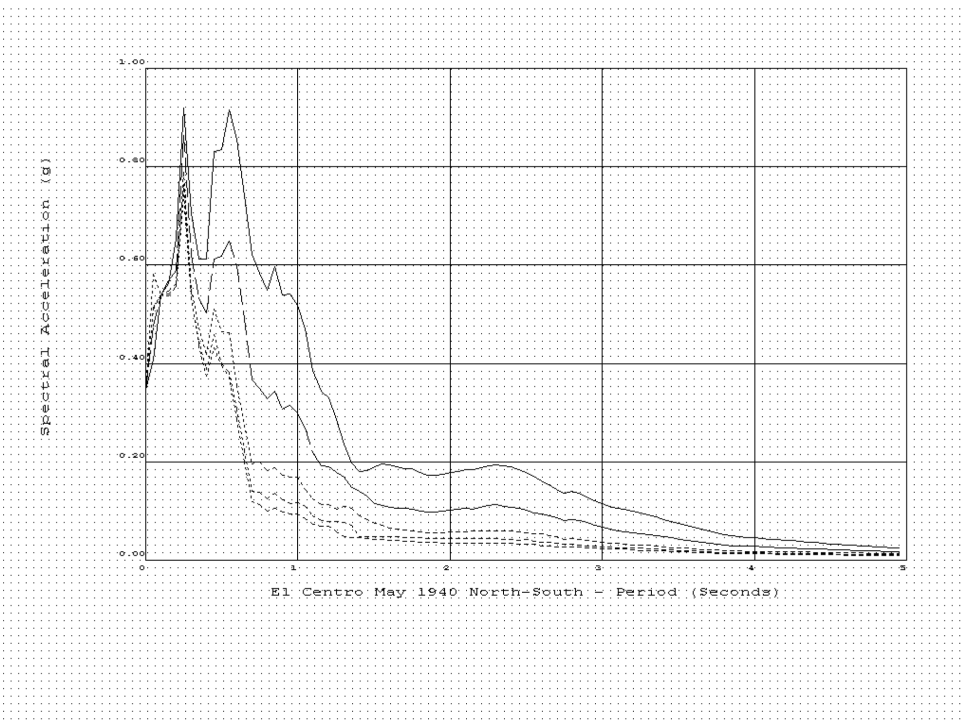
The input data file is arranged as described below for the interactive version except that lines 1 to 6 and lines 21 to 24 are to be omitted.

Example

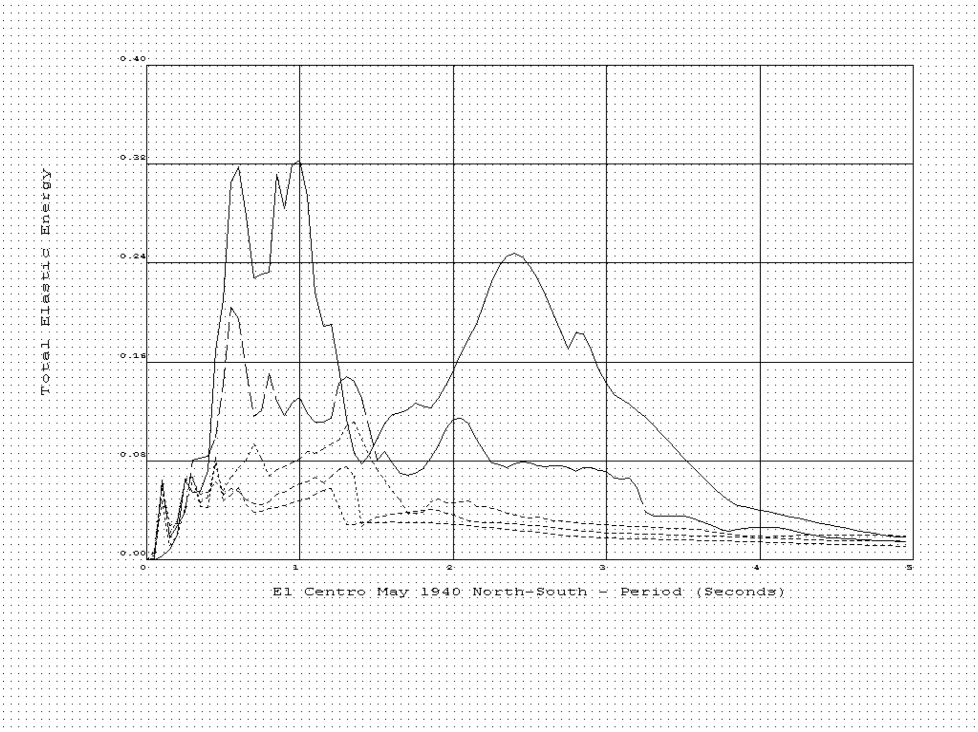
The following pages give the graphic output for the El Centro May 1940 North-South component **EL40NSC.EQB** using the default inputs except for a bi-linear hysteresis with a bi-linear factor of 0.05 for 4 ductility values of 1, 2, 4, 6 and 8 and using a 20 seconds duration and monochromatic plotting. The elastic (ductility 1.0) line is the solid line and the dashed line is for ductility 2.0 etc.



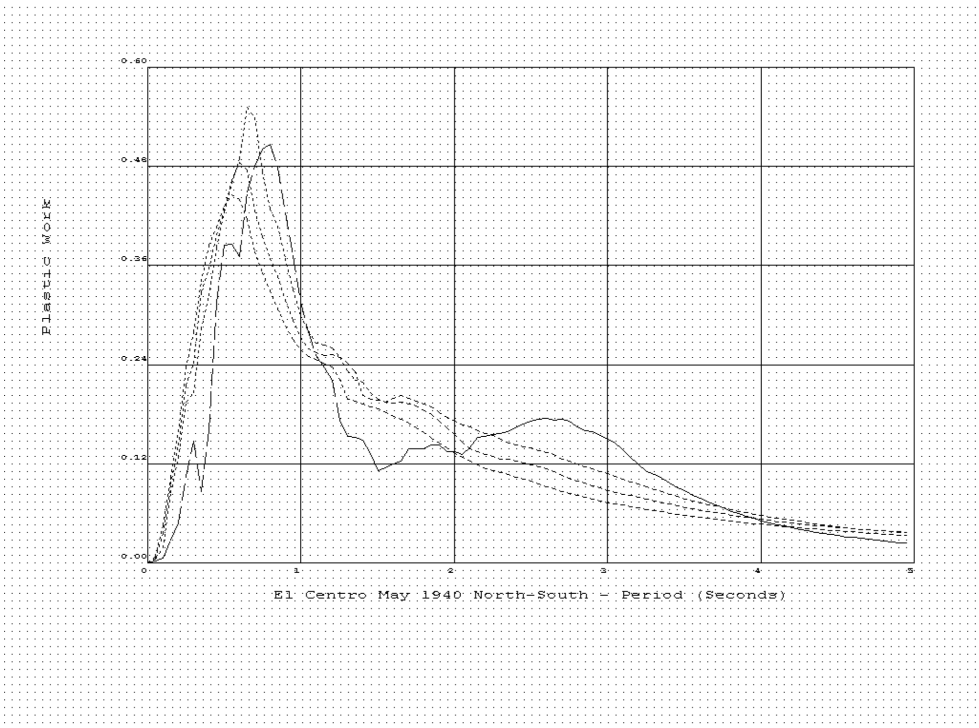
Spectral Displacement



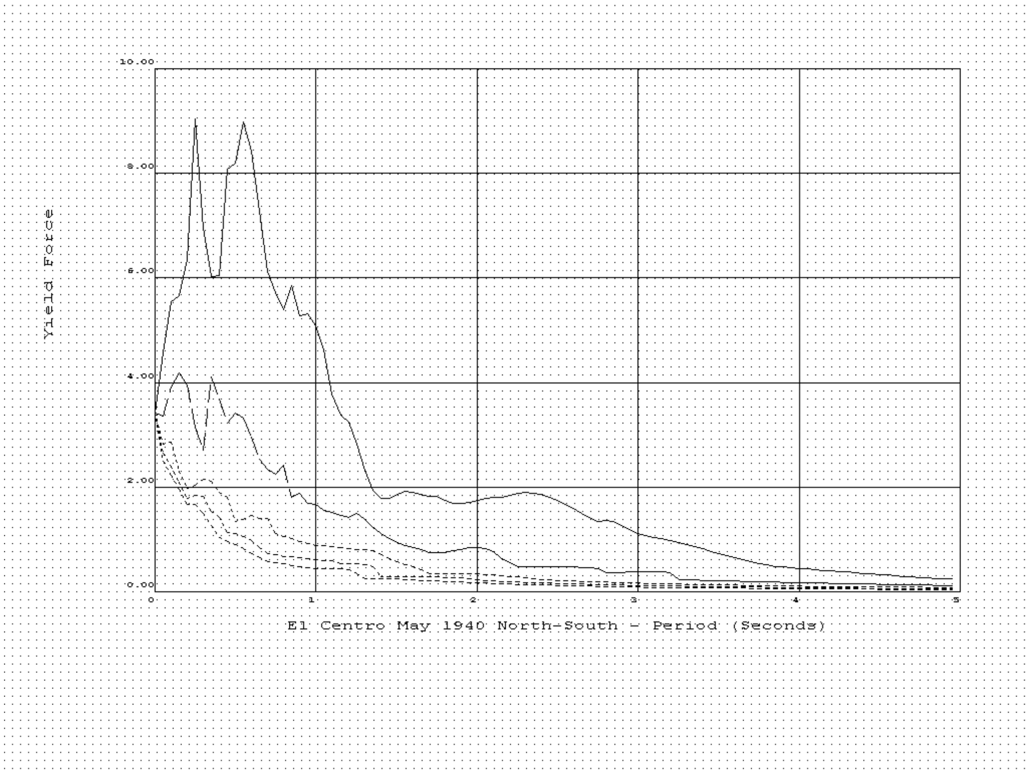
Spectral Acceleration



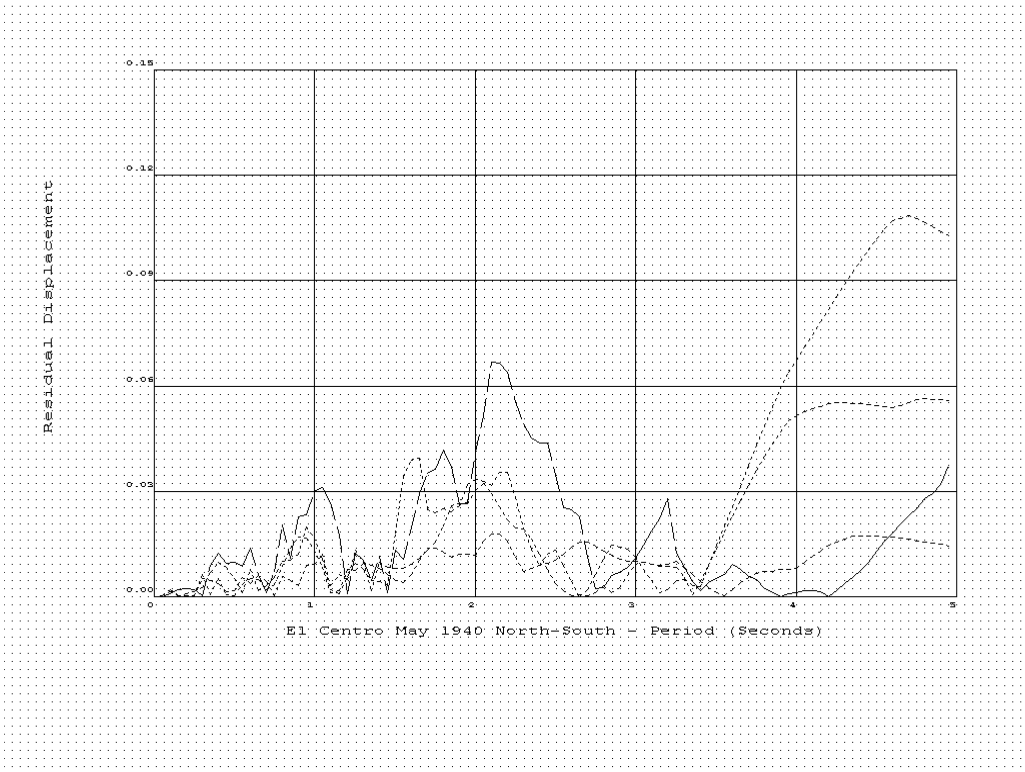
Elastic Energy Spectra



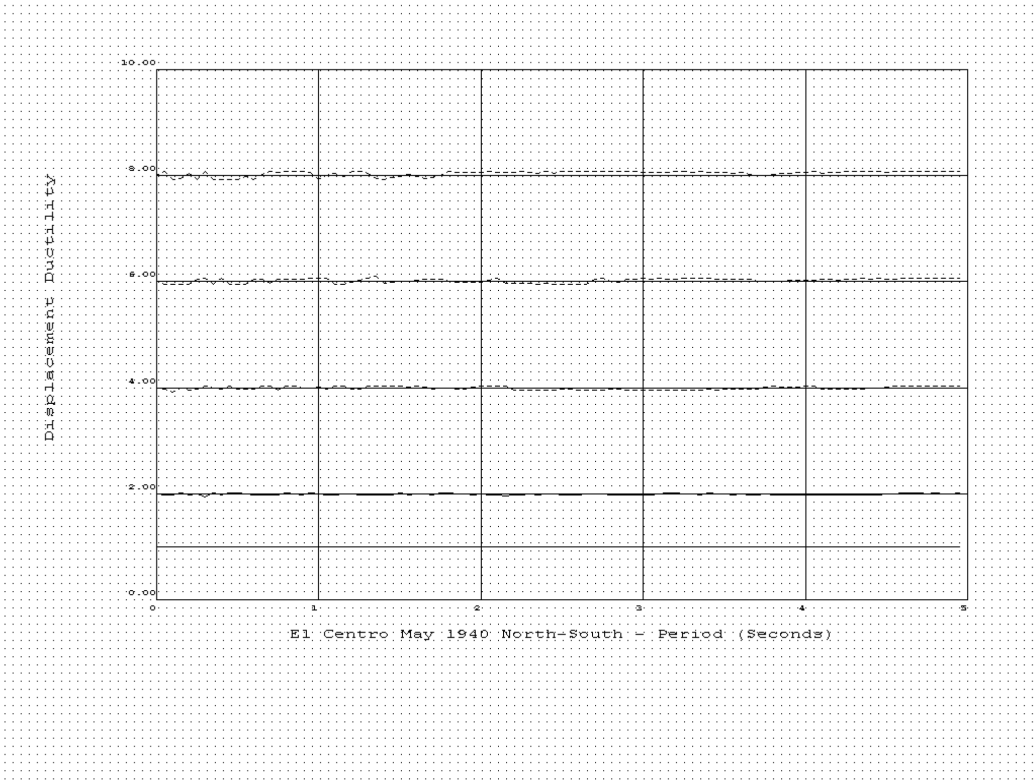
Plastic Work Spectra



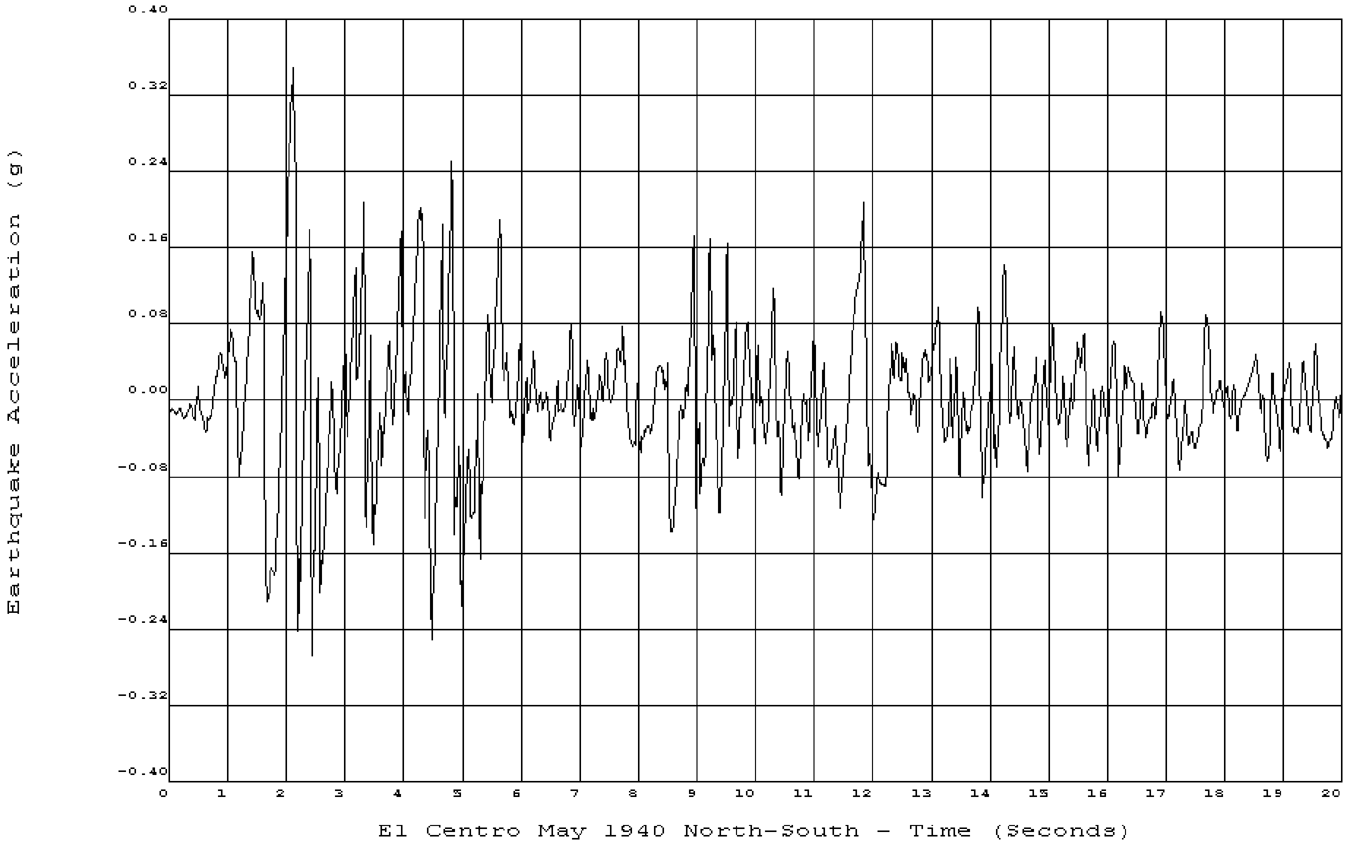
Yield Force Spectra



Residual Displacement Spectra



Computed versus Target Ductility Spectra



Earthquake Acceleration (g)
0.40
0.32
0.24
0.16
0.08
0.00
-0.08
-0.16
-0.24
-0.32
-0.40
0 1 2 3 4 5 6 7 8 9 10 11 12 13 14 15 16 17 18 19 20
El Centro May 1940 North-South - Time (Seconds)

.

End.

UNIVERSITY OF CANTERBURY
Te Whare Wānanga o Waitaha
CHRISTCHURCH NEW ZEALAND

Department of Civil Engineering **COMPUTER PROGRAM LIBRARY**

Program name:– **FPRINT**	Program type:– Fortran Format Printing	Program code:– ANSI Fortran77
Author:– Athol J Carr		Date:– April 28, 2007

FPRINT

Printing Files with FORTRAN Carriage Control

Output files using FORTRAN carriage control have the printer carriage control as the first character on the line to be printed. The number 1 (one) as the first character on the line means move to a new page, a 0 (zero) implies skip a line before printing and a blank character as the first character of the line implies printing on the next line of the printed output. Most current computer printers, and operating systems, do not recognize these carriage controls unless the output is sent directly to the printer and not to a file.

The program **FPRINT** will send a file directly to the printer on the PC with the command

FPRINT <filename>

where **<filename>** is the name of the file. The printed output will use the ANSI FORTRAN standard carriage controls.

To run the program call the program by the method appropriate to your operating system from the command line or by setting up shortcuts to run the program. For the command line usage on a personal computer this assumes that the file **FPRINT.EXE** is in your current directory or path.

This page is intentionally left blank.