National Technical University of Athens (NTUA)

Department of Civil Engineering
Institute of Structural Analysis and Aseismic Research

MySpec
version 1.0

October 2003

Contents

1. Introduction

1.1 Scope

This document analyzes the features and usage of MySpec version 1.0. In addition, there is a brief description of the theory, together with examples that cover all different options.

1.2 Program requirements

The minimum requirements are:

- Operating System: Microsoft® Windows 95/98/ME/NT/2000/XP
- Visual Basic 6 Service Pack 5 runtime libraries.

1.3 Abbreviations

SDF:	Single Degree of Freedom
2DOF:	Two Degree of Freedom
ODE:	Ordinary Differential Equation

1.4 About MySpec

This program calculates the response of a SDF or a 2DOF system which is subjected to an excitation. The analysis may be linear elastic or non-linear. The response is calculated for several models.

The typical excitation corresponds to a ground motion specified by an earthquake. However, an arbitrary excitation force can be used as well, either alone or together with an earthquake.

All diagrams of interest can be plotted on the screen or sent to the printer. Also, the results can be exported in the form of ASCII text files. Moreover, such files can be easily inserted to a spreadsheet application.

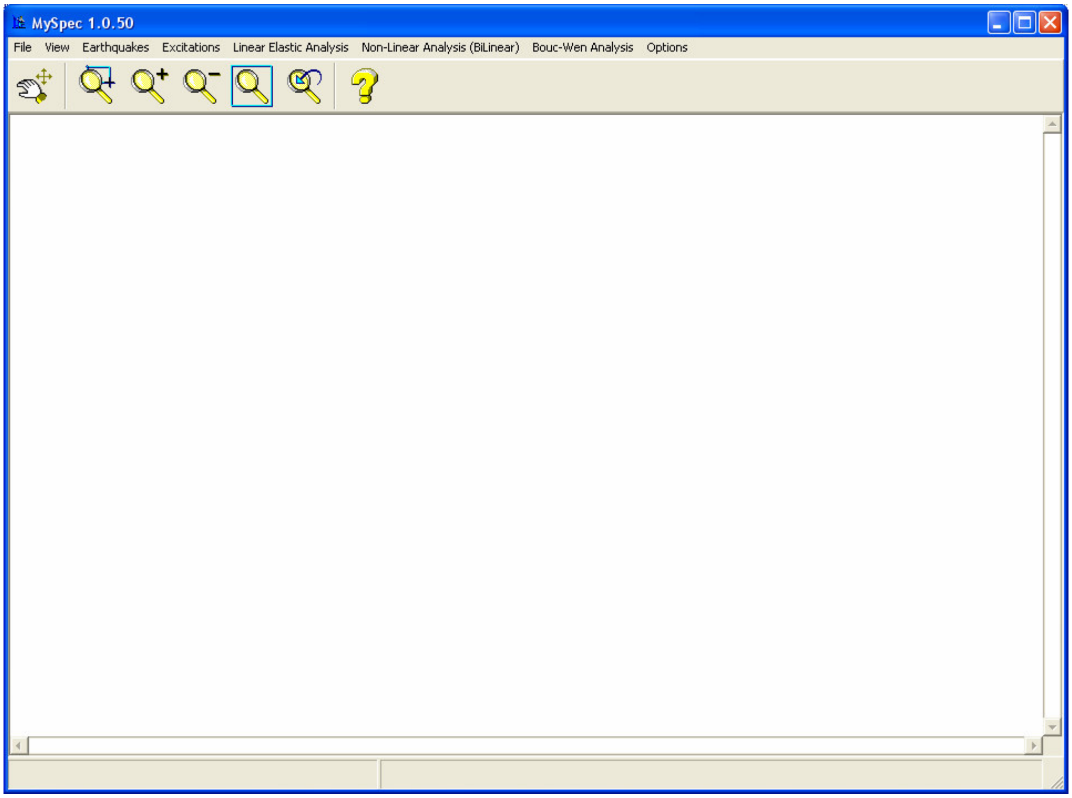
Apart from the diagrams, the program is capable of producing a simulation of the behaviour of the system in animated form and idealised as a lumped mass on the top of a column of specified stiffness.

All calculations are carried out using double precision arithmetics.

2. Interface

2.1 Main window

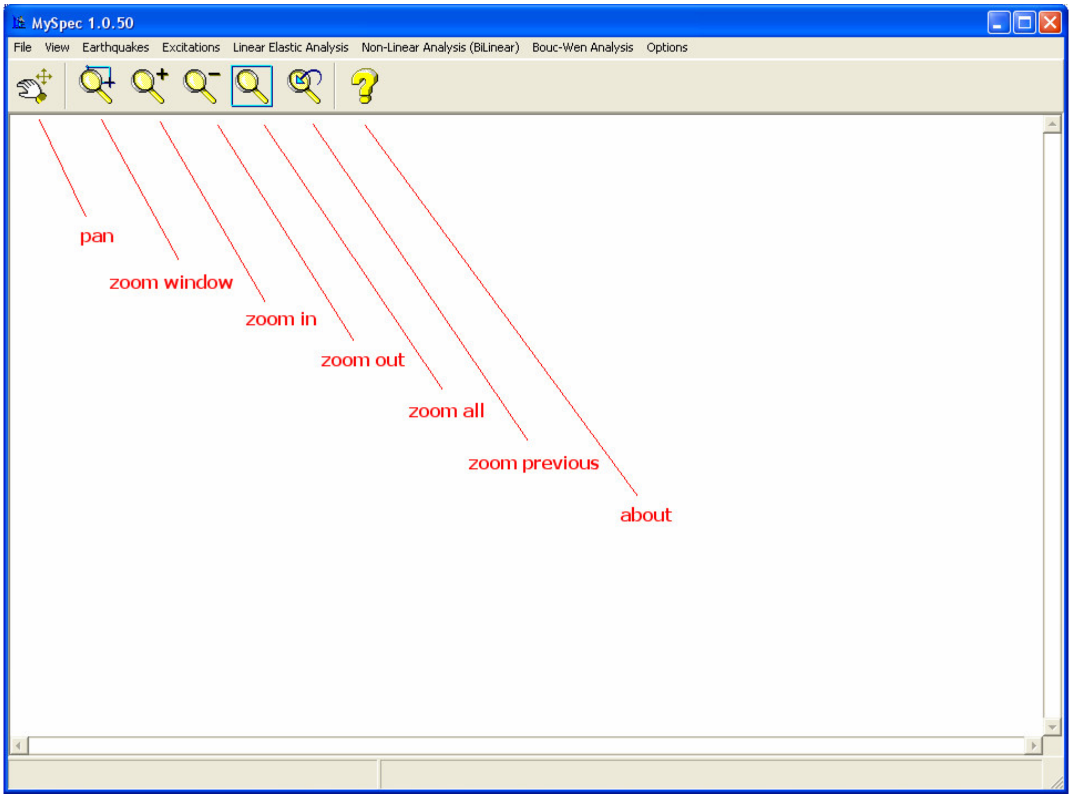
The main window of the program appears as follows:



There is a standard menu, the toolbar, the main drawing area and the status bar.

2.2 Toolbar

The toolbar contains the following buttons:



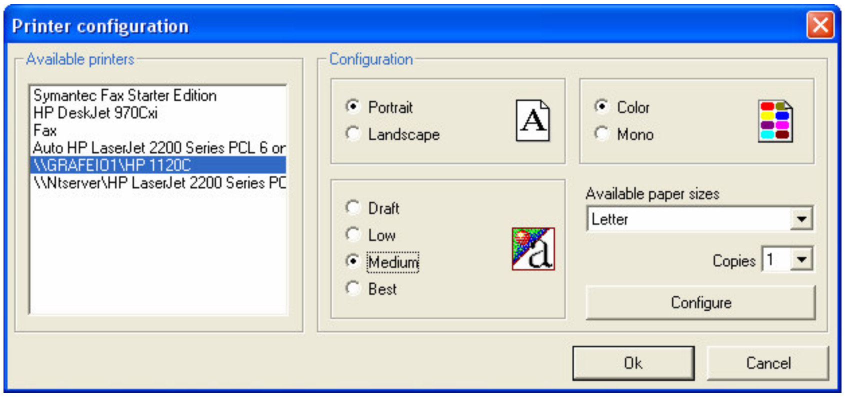
2.3 Drawing tools

Only one drawing can be shown at a time on the screen. The drawing tools can be found under the *"File"* and *"View"* menus. The most useful menus can be accessed directly from the associated buttons in the toolbar.

2.3.1 Print drawing

The current drawing can be printed directly to the printe, by selecting *"Print Drawing"* from the *"File"* menu.

If you haven't already selected a printer, a form displaying all printers will appear. You must select a printer in order to proceed. You can access the printer selection form later by selecting *"Configure Printer"* under the *"Tools"* menu:



2.3.2 Save image as .bmp file

You can save the current drawing as a bitmap, by selecting *"Save image as .bmp"* from the *"File"* menu. This file can be modified by all image editing programs.

2.3.3 Zoom all

You can "zoom all" i.e. display the entire drawing by selecting *"Zoom All"* from the *"View"* menu, or by clicking the associated button in the toolbar (see 2.2).

2.3.4 Zoom window

You can "zoom window" i.e. zoom to a part of the drawing by selecting *"Zoom Window"* from the *"View"* menu, or by clicking the associated button in the toolbar (see 2.2). You need to click twice on the drawing to specify the diagonal of the window.

2.3.5 Zoom previous

You can "zoom previous" i.e. return to the previous zoom by selecting *"Zoom Previous"* from the *"View"* menu, or by clicking the associated button in the toolbar (see 2.2).

2.3.6 Zoom in

You can "zoom in" i.e. zoom to a specified point by selecting *"Zoom In"* from the *"View"* menu, or by clicking the associated button of the toolbar (see 2.2). You need to click on the drawing to specify the point of zoom.

2.3.7 Zoom out

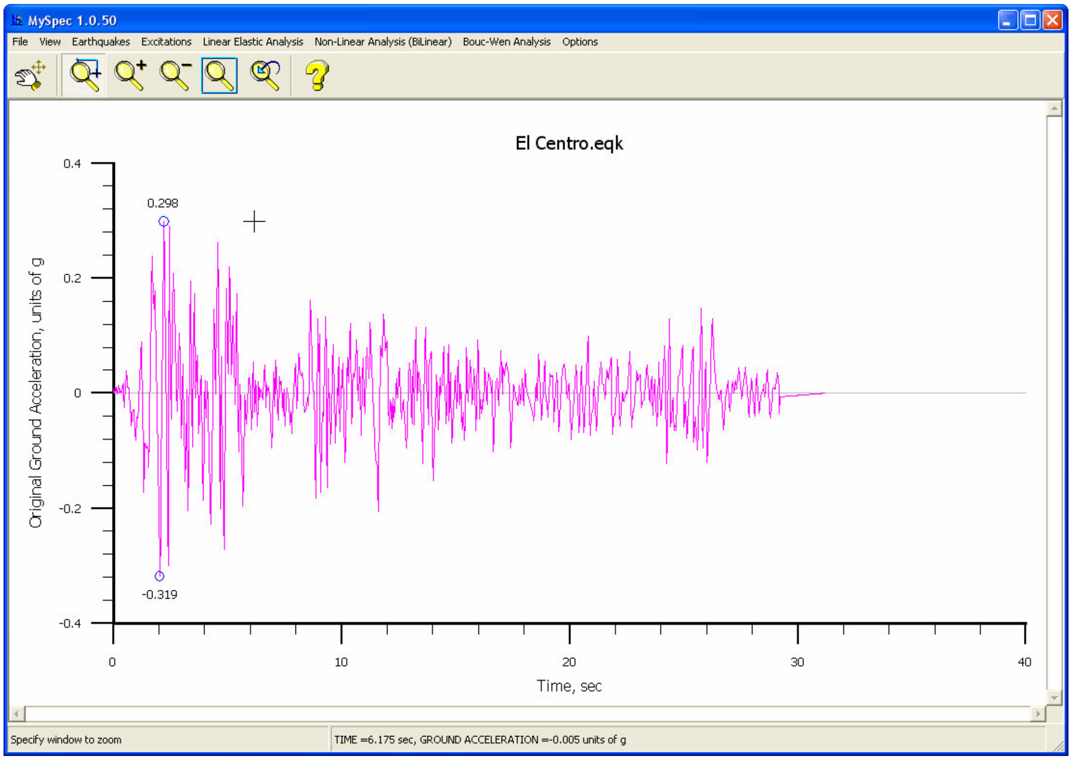
You can "zoom out" by selecting *"Zoom Out"* from the *"View"* menu, or by clicking the associated button of the toolbar (see 2.2). You need to click on the drawing to specify the point of zoom.

2.3.8 Pan

You can "*Pan*" i.e. move the drawing using the mouse, by selecting *"Pan"* from the *"View"* menu, or by clicking the associated button of the toolbar (see 2.2). You need to click and drag on the drawing in order to specify the move.

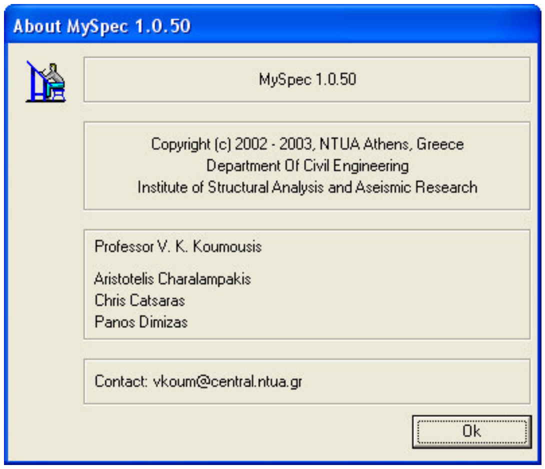
2.4 Status bar

The status bar displays information about both the active command and the current mouse position:



2.5 About box

You can access the about box by clicking the associated button of the toolbar (see 2.2):


About MySpec 1.0.50
MySpec 1.0.50
Copyright (c) 2002 - 2003, NTUA Athens, Greece
Department Of Civil Engineering
Institute of Structural Analysis and Aseismic Research
Professor V. K. Koumousis
Aristotelis Charalampakis
Chris Catsaras
Panos Dimizas
Contact: vkoum@central.ntua.gr
Ok

3. Earthquakes

3.1 General issues

All earthquake data is stored in an "earthquake library". When a certain model is used, the user can select the earthquake by selecting its name from a standard windows dropdown list box.

The earthquakes can be modified ("amplified" or "stretched") with respect to the original data. Also, a certain subset of the original data may be used.

Most functions related to earthquakes can be found under the *"Earthquakes"* menu.

In order to load an earthquake, you need to use a file with a certain format, described later in this chapter.

3.2 File Format

The file is a standard ASCII text file. You must use a dot "." as the decimal symbol.

The file format used is the following:

- Default extension: .EQK (this is not compulsory)
- First line: Earthquake title (string)
- Second line: Units. It can be "g" for units of g, "g/10" for tenths of g, "m/sec^2", "in/sec^2" etc.
- Third line: Time step in seconds, for example 0.02
- Forth line: First entry, ground acceleration for t = one time step. For t = 0, the acceleration is set to zero by default.
- Next lines: The rest entries. The last entry should be zero.

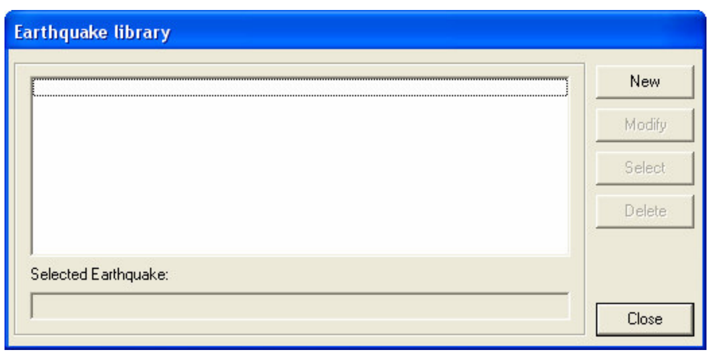
For example, the file might look like this:

```
North South ground acceleration, El Centro, May 18th, 1940.
G
0.02
  0.0063
```

```
 0.00364
 0.00099
 0.00428
 0.00758
 0.01087
 0.00682
 0.00277
-0.00128
 0.00368
 0.00864
  0.0136
…
 0.00032
-0.00025
-0.00019
-0.00013
-0.00006
       0
```

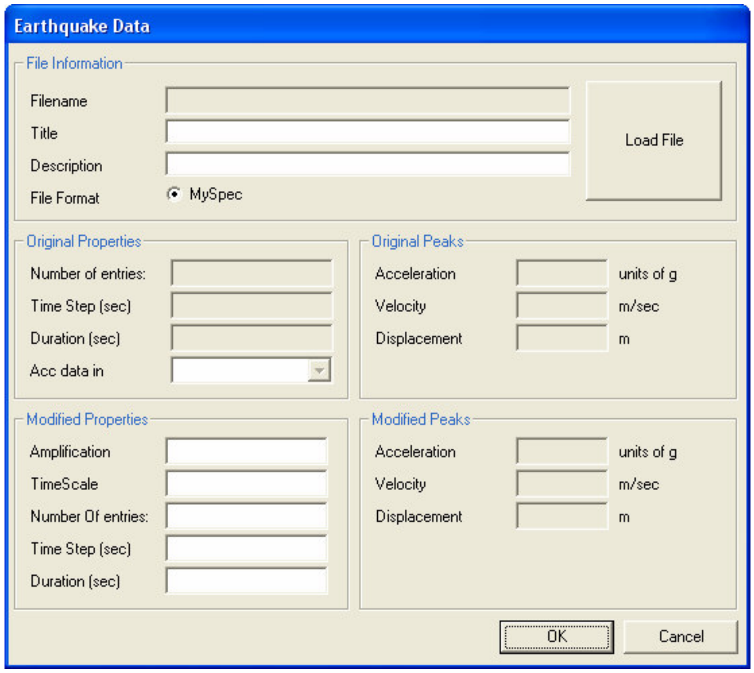
3.3 Earthquake library

To access the earthquake library, select *"Earthquake library"* under the *"Earthquakes"* menu. The following form will appear:

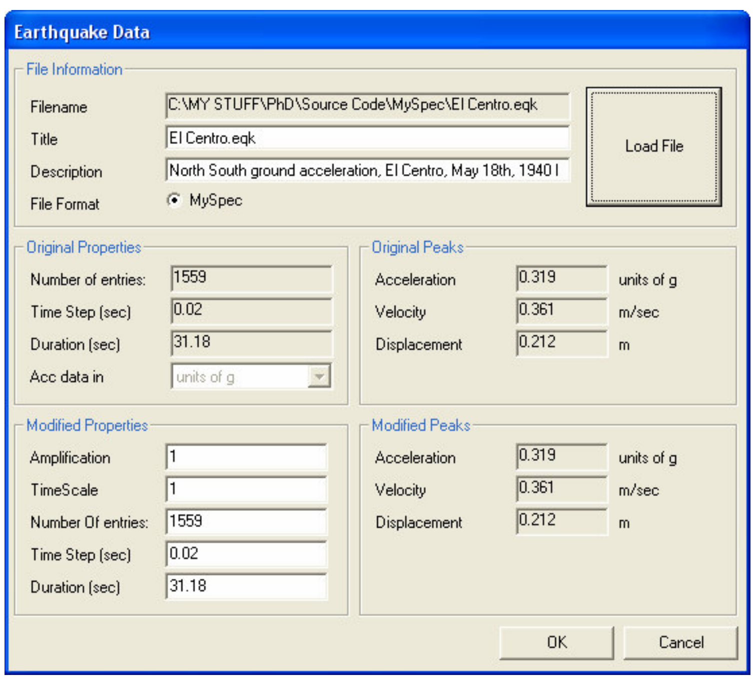


3.3.1 Add new earthquake

In order to add a new earthquake, click on the *"New"* button of the form of the earthquake library. The following form will appear:



Click on the *"Load File"* button on the top right corner and select a valid file. (Refer to *3.2 File Format* for further information). After having loaded the El Centro earthquake, the form may look like this:



In the *"File Information"* frame, the following data is displayed:

- Filename: The full path of the file.

- Title: The title of the earthquake, which will appear on the drawing. The filename is used as a default value.
- Description: The earthquake description found in the file. (Refer to *3.2 File Format* for further information). This information is printed in the reports.
- File Format: For the time being, only “MySpec” format is available.

In the *"Original Properties"* frame, which is read-only, the following data is displayed:

- Number of entries: Number of entries for the ground acceleration.
- Time step: The time step in seconds.
- Duration: The duration of the earthquake in seconds.
- Acc data in: The units of the ground acceleration data.

Based on this information, the program calculates automatically the acceleration, velocity, displacement peaks, which are displayed in *"Original Peaks"* frame. The units in which the results are displayed can be changed by selecting *"Units"* under the *"Options"* menu of the main form.

For more information on how the velocity and displacement data is calculated refer to *3.4 Earthquake calculations*.

In the *"Modified Properties"* frame, the following data is displayed:

- Amplification: The amplification factor, which can be set by the user. The default value is 1.
- Timescale: The time stretching factor, which can be set by the user. The default value is 1. A different value will consequently change the time step and the duration. Do not change this if you don’t have a specific reason.
- Number of entries: The subset of entries for the ground acceleration. Can be smaller than or equal to the counter of the original properties. The default value is that of the original properties. A different value will consequently change the timescale factor, the time step and the duration.
- Time step: The time step in seconds. The default value is that of the original properties. A different value will consequently change the timescale factor and the duration.

- ➢ Duration: The duration of the earthquake in seconds. The default value is that of the original properties. A different value will consequently change the timescale factor and the time step.

Based on this information, the program calculates automatically the modified acceleration, velocity, displacement peaks, which are displayed in *"Modified Peaks"* frame. The units in which the results are displayed can be changed by selecting *"Units"* under the *"Options"* menu of the main form.

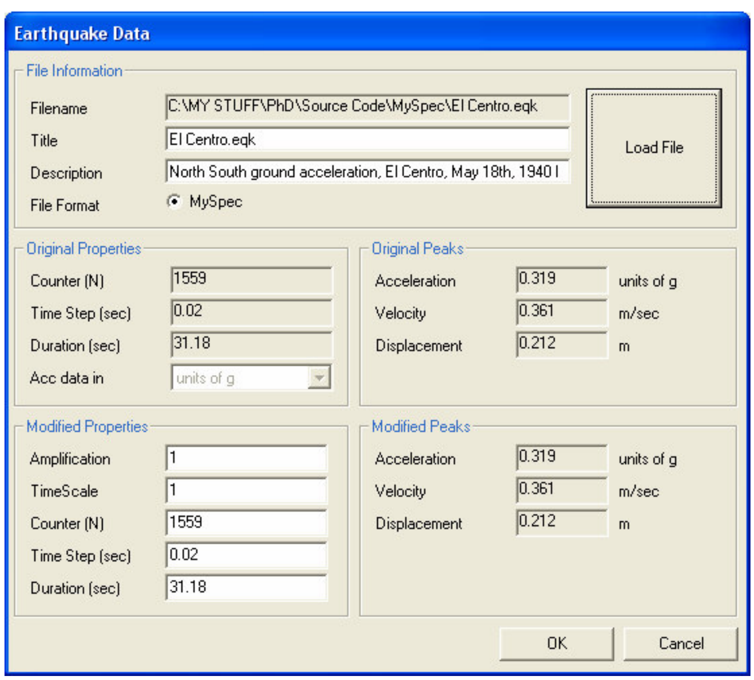
Note that all further calculations are carried out using the *modified earthquake data.* The original data is kept only for reference.

3.3.2 Delete earthquake

In order to delete an earthquake, select it from the list of the earthquake library form, then click on the *"Delete"* button. You must confirm the deletion, as this is permanent.

3.3.3 Modify earthquake

In order to modify an earthquake, select it from the list of the earthquake library form, then click on the *"Modify"* button. Trying to modify for example the El Centro earthquake, the following form will appear:

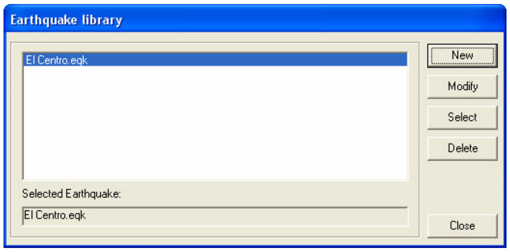


You can now modify the various factors available. Refer to *3.3.1 Add new earthquake* for further information.

3.3.4 Select active earthquake

In order to display the results of the calculations, you must first select the active (selected) earthquake. You can accomplish this by clicking on *"Select Earthquake"* under the *"Earthquake"* menu of the main form. The following form will appear:

You can select the active earthquake from the dropdown list box. The list is loaded with all earthquakes defined in the earthquake library:

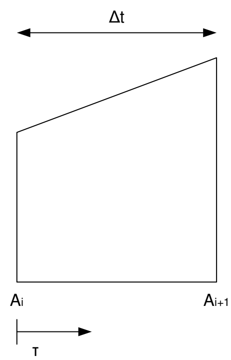


Alternatively, you can select the earthquake from the above list and hit the *"Select"* button on the right. The active (selected) earthquake is displayed at the bottom.

3.4 Earthquake calculations

All calculations are carried out using double precision arithmetic.

The ground velocity and displacement are calculated by assuming a linear variation of the ground acceleration within the time step.



$$A(\tau) = A_i + \frac{\tau \cdot (A_{i+1} - A_i)}{\Delta t}$$
$$A(0) = A_i$$
$$A(\Delta t) = A_{i+1}$$

Also:

$$V(t) = \int A(t)dt$$
$$D(t) = \int V(t)dt$$

Therefore, the ground velocity at all time steps is calculated as follows:

$$V_{i+1} = V_i + \frac{(A_{i+1} + A_i)}{2} \cdot \Delta t$$

The ground displacement is calculated as follows:

$$D_{i+1} = D_i + V_i \cdot \Delta t + \frac{A_{i+1} \cdot \Delta t^2}{6} + \frac{A_i \cdot \Delta t^2}{3}$$

Where:

Δt is the time step

A_i is the ground acceleration at time $t = i \cdot \Delta t$

V_i is the ground velocity at time $t = i \cdot \Delta t$

D_i is the ground displacement at time $t = i \cdot \Delta t$

3.5 Results

In order to display the results of the earthquake calculations, you must first select the active earthquake. (Refer to *3.3.4 Select active earthquake* for further information)

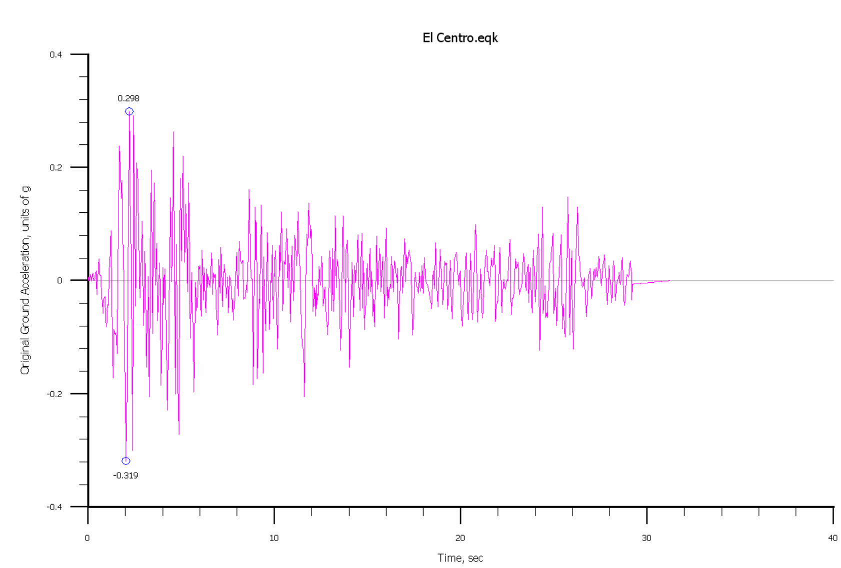
Having selected the active earthquake, you can display the results from the calculations. All graphs are available, both for the original earthquake and its modified form.

The units in which the results are displayed can be changed by selecting *"Units"* under the *"Options"* menu of the main form.

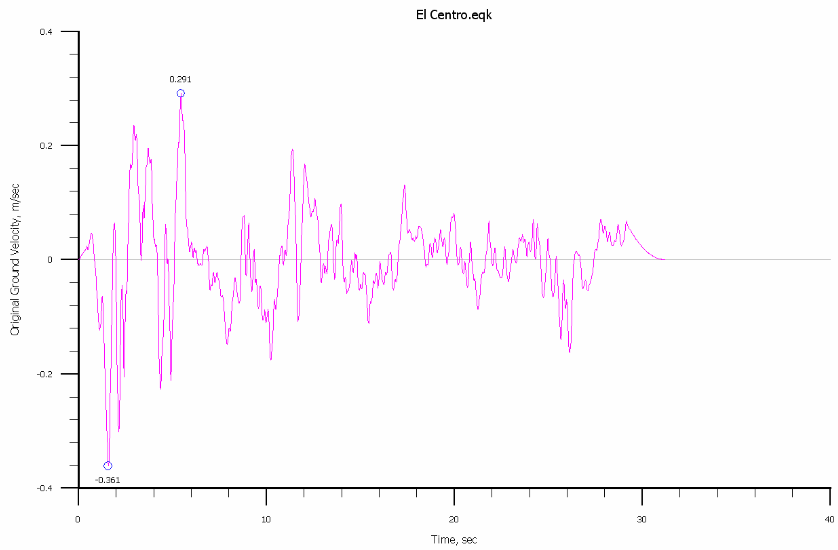
The results are displayed by selecting the appropriate graph from the *"Earthquakes > Plot"* menu.

Below there is a sample of the results for the El Centro earthquake.

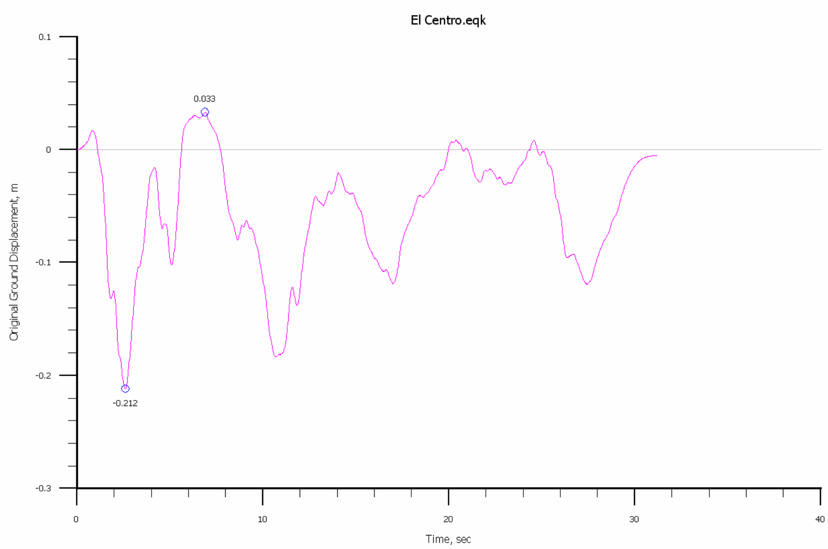
- Original ground acceleration in units of g:



- Original ground velocity in m/sec:



➢ Original ground displacement in m:

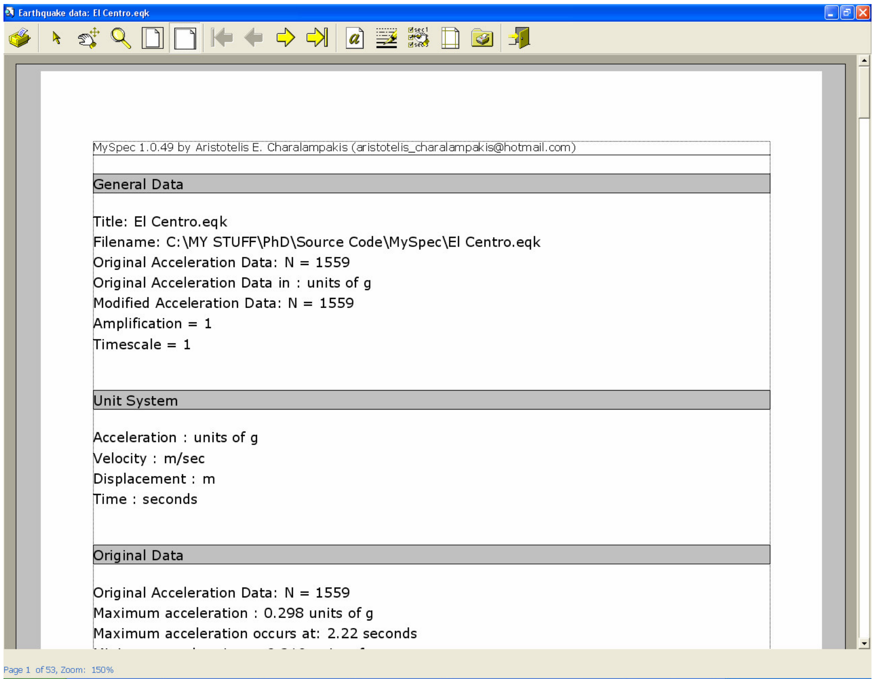


The modified earthquake results depend on various factors, such as the amplification and timescale factors.

3.6 Printing and exporting the results

3.6.1 Print results

You can print the results of the calculations directly, by selecting *"Print Analysis Report"* under the *"Earthquake"* menu. A full report will be prepared, and a stand alone print preview program will appear:

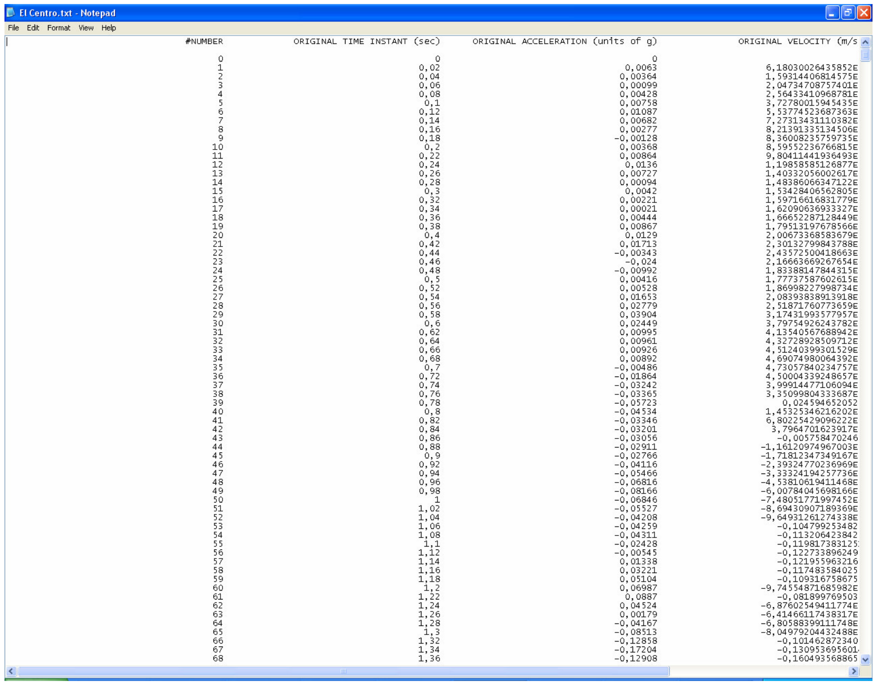


Most functions of this program are self-explanatory.

3.6.2 Export results

You can export the results in a simple ASCII text file, and use it in a spreadsheet application, for example. In order to create this file, select *"Export Results"* from the *"Earthquake"* menu of the main form.

The results are fixed - aligned in columns with a header for each column. You may need to modify the file, because both the original and the modified results are included in the same file.



El Centro.txt - Notepad

File Edit Format View Help

#NUMBER	ORIGINAL TIME INSTANT (sec)	ORIGINAL ACCELERATION (units of g)	ORIGINAL VELOCITY (m/s
0	0	0	
1	0,02	0,0063	6,1803002643585 2E
2	0,04	0,00364	1,59314406814575E
3	0,06	0,00099	2,04734708757401E
4	0,08	0,00428	2,56433410968781E
5	0,1	0,00758	3,72780015945435E
6	0,12	0,01087	5,53774523687363E
7	0,14	0,00682	7,27313431110382E
8	0,16	0,00277	8,21391335134506E
9	0,18	-0,00128	8,36008235759735E
10	0,2	0,00368	8,59552236766815E
11	0,22	0,00864	9,80411441936493E
12	0,24	0,0136	1,19858585126877E
13	0,26	0,00727	1,40332056002617E
14	0,28	0,00094	1,48386066347122E
15	0,3	0,0042	1,53428406562805E
16	0,32	0,00221	1,59716616831779E
17	0,34	0,00021	1,62090636933327E
18	0,36	0,00444	1,66652287128449E
19	0,38	0,00867	1,79513197678566E
20	0,4	0,0129	2,00673368583679E
21	0,42	0,01713	2,30132799843788E
22	0,44	-0,00343	2,43572500418663E
23	0,46	-0,024	2,16663669267654E
24	0,48	-0,00992	1,83388147844315E
25	0,5	0,00416	1,77737587602615E
26	0,52	0,00528	1,86998227998734E
27	0,54	0,01653	2,08393838913918E
28	0,56	0,02779	2,51871760773659E
29	0,58	0,03904	3,17431993577957E
30	0,6	0,02449	3,79754926243782E
31	0,62	0,00995	4,13540567688942E
32	0,64	0,00961	4,32728928509712E
33	0,66	0,00926	4,51240399301529E
34	0,68	0,00892	4,69074980064392E
35	0,7	-0,00486	4,73057840234757E
36	0,72	-0,01864	4,50004339248657E
37	0,74	-0,03242	3,99914477106094E
38	0,76	-0,03365	3,35099804333687E
39	0,78	-0,05723	0,024594652052
40	0,8	-0,04534	1,45325346216202E
41	0,82	-0,03346	6,80225429096222E
42	0,84	-0,03201	3,7964701623917E
43	0,86	-0,03056	-0,005758470246
44	0,88	-0,02911	-1,16120974967003E
45	0,9	-0,02766	-1,71812347349167E
46	0,92	-0,04116	-2,39324770236969E
47	0,94	-0,05466	-3,33324194257736E
48	0,96	-0,06816	-4,53810619411468E
49	0,98	-0,08166	-6,00784045698166E
50	1	-0,06846	-7,48051771997452E
51	1,02	-0,05527	-8,69430907189369E
52	1,04	-0,04208	-9,64931261274338E
53	1,06	-0,04259	-0,104799253482
54	1,08	-0,04311	-0,113206423842
55	1,1	-0,02428	-0,11981738312
56	1,12	-0,00545	-0,122733896249
57	1,14	0,01338	-0,121955963216
58	1,16	0,03221	-0,117483584025
59	1,18	0,05104	-0,109316758675
60	1,2	0,06987	-9,74554871685982E
61	1,22	0,0887	-0,081899769503
62	1,24	0,04524	-6,87602549411774E
63	1,26	0,00179	-6,41466117438317E
64	1,28	-0,04167	-6,80588399111748E
65	1,3	-0,08513	-8,04979204432488E
66	1,32	-0,12858	-0,101462872340
67	1,34	-0,17204	-0,130953695601
68	1,36	-0,12908	-0,160493568865

The units can be changed by selecting *"Units"* under the *"Options"* menu of the main form.

3.6.3 Print drawing

You can print directly the current drawing by selecting *"Print Drawing"* from the *"File"* menu, as described in *2.3.1 Print drawing*. The drawing is printed with the default printer settings, using the maximum available area.

3.6.4 Save drawing

You may save the current drawing in *.bmp format by selecting *"Save Image As Bmp"* from the *"File"* menu, as described in *2.3.2 Save Image as .bmp*.

4. Excitations

4.1 General issues

MySpec program is capable of using direct excitation data i.e. force instead of ground acceleration. All excitation data is stored in an "excitation library". When a certain model is used, the user can select the excitation by selecting its name from a standard windows dropdown list.

The excitations can be "amplified" or "stretched" with respect to the original data. Also, a certain subset of the original data may be used.

Most functions related to excitations can be found under the *"Excitations"* menu.

In order to load an excitation, you need to use a file with a certain format, described later in this chapter.

4.2 File Format

The file is standard ASCII text. You must use a period "." as the decimal symbol.

The file format used is the following:

- Default extension: .EXC (this is not compulsory)
- First line: Excitation title (string)
- Second line: Units. It can be "N" for Newtons, "KN", "Kips" etc.
- Third line: Time step in seconds, for example 0.1
- Fourth line: First entry, force for t = one time step. For t = 0, the force is zero by default.
- Next lines: The rest entries. The last entry should be zero.

For example, the file might look like this:

```
Half cycle sine pulse force

N

0.1

5

8.66
```

10

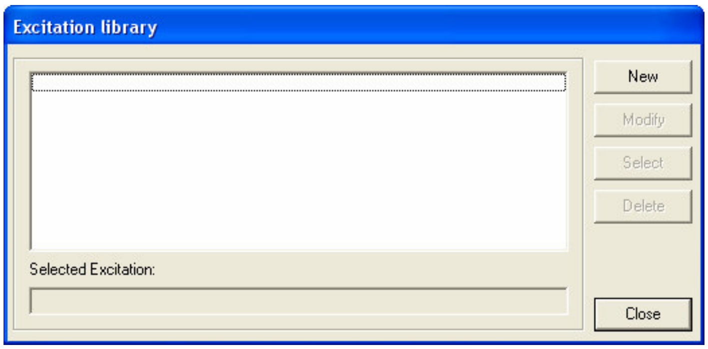
8.66

5

0

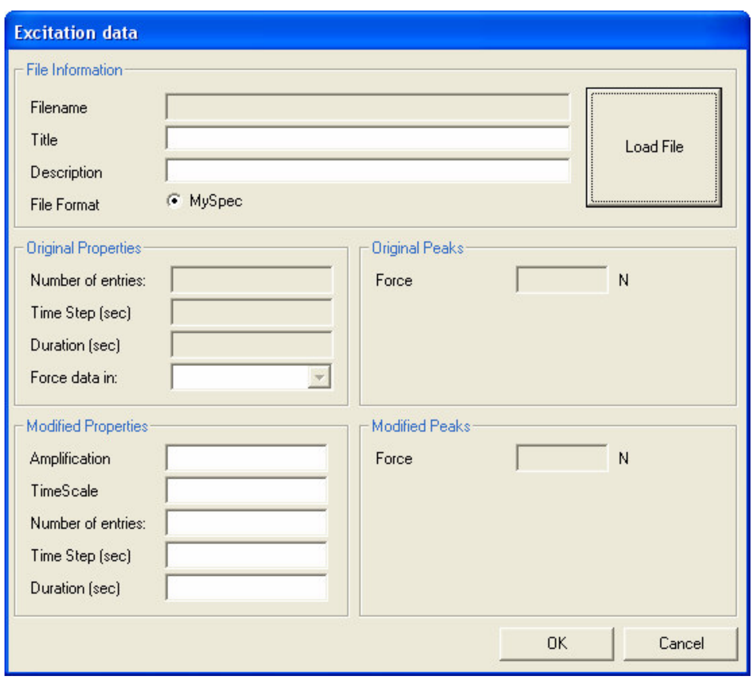
4.3 Excitation library

To access the excitation library, select *"Excitation library"* under the *"Excitations"* menu. The following form will appear:

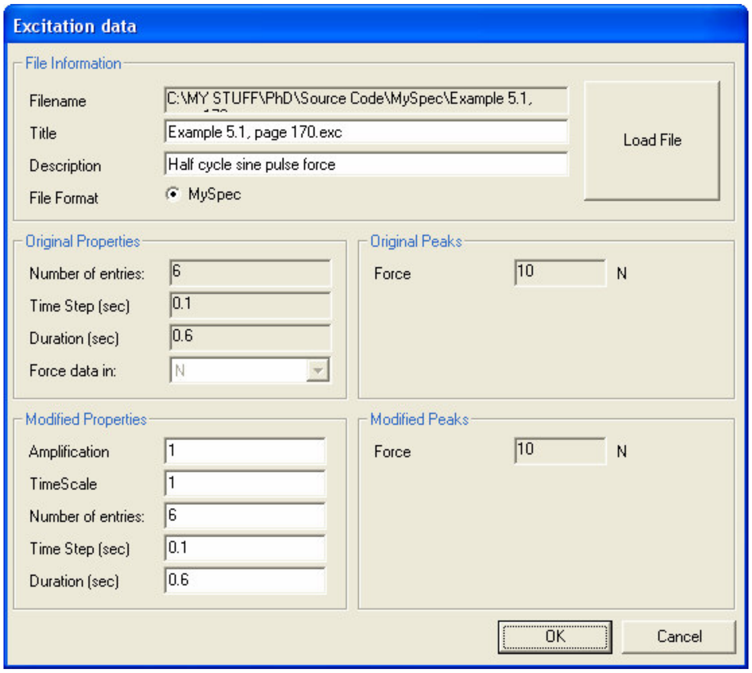


4.3.1 Add new excitation

In order to add a new excitation, click on the *"New"* button of the form of the excitation library. The following form will appear:



Click on the *"Load File"* button on the top right corner and select a valid file. (Refer to *4.2 File Format* for further information). After having loaded a valid file, the form may look like this:



In the *"File Information"* frame, the following data is displayed:

- Filename: The full path of the file.
- Title: The title of the excitation, which will appear on the drawing. The filename is used as a default value.
- Description: The excitation description found in the file. (Refer to *4.2 File Format* for further information). This information is printed in the reports.
- File Format: For the time being, only "MySpec" format is available.

In the *"Original Properties"* frame, which is read-only, the following data is displayed:

- Number of entries: The number of entries.
- Time step: The time step in seconds.
- Duration: The duration of the excitation in seconds.
- Force data in: The units of the force data.

Based on this information, the program calculates automatically the force peak, which is displayed in *"Original Peaks"* frame. The units in which the results are displayed can be changed by selecting *"Units"* under the *"Options"* menu of the main form.

In the *"Modified Properties"* frame, the following data is displayed:

- Amplification: The amplification factor, which can be set by the user. The default value is 1.
- Timescale: The time stretching factor, which can be set by the user. The default value is 1. A different value will consequently change the time step and the duration.
- Number of entries: The subset of entries for the force. Can be smaller than or equal to the counter of the original properties. The default value is that of the original properties. A different value will consequently change the timescale factor, the time step and the duration.
- Time step: The time step in seconds. The default value is that of the original properties. A different value will consequently change the timescale factor and the duration.
- Duration: The duration of the excitation in seconds. The default value is that of the original properties. A different value will consequently change the timescale factor and the time step.

Based on this information, the program calculates automatically the modified force peak, which is displayed in *"Modified Peaks"* frame. The units in which the results are displayed can be changed by selecting *"Units"* under the *"Options"* menu of the main form.

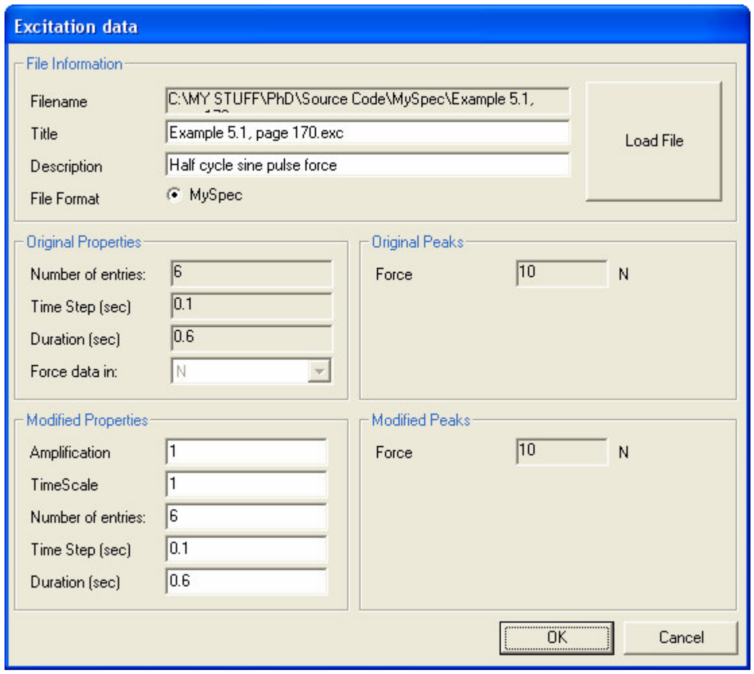
Note that all further calculations are carried out using the *modified excitation data.* The original data is kept only for reference.

4.3.2 Delete excitation

In order to delete an excitation, select it from the list of the excitation library form, then click on the *"Delete"* button. You must confirm the deletion, as this is permanent.

4.3.3 Modify excitation

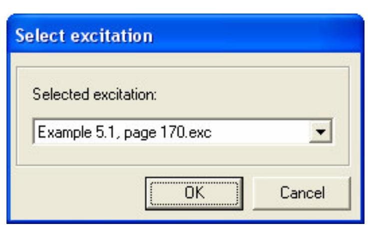
In order to modify an excitation, select it from the list of the excitation library form, then click on the *"Modify"* button. The following form may appear:



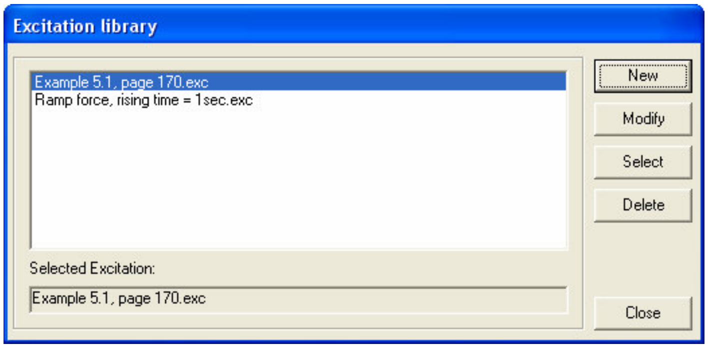
You can now modify the various factors available. Refer to *4.3.1 Add new excitation* for further information.

4.3.4 Select active excitation

In order to display the results of the calculations, you must first select the active (selected) excitation. You can accomplish this by clicking on *"Select Excitation"* under the *"Excitation"* menu of the main form. The following form will appear:



You can select the active excitation from the dropdown list. The list is loaded with all excitations defined in the excitation library:



Alternatively, you can select the excitation from the above list and hit the *"Select"* button on the right. The active (selected) excitation is displayed at the bottom.

4.4 Excitation calculations

All excitation data is stored in double precision arithmetic.

4.5 Results

In order to display the excitation data, you must first select the active excitation. (Refer to *4.3.4 Select active excitation* for further information)

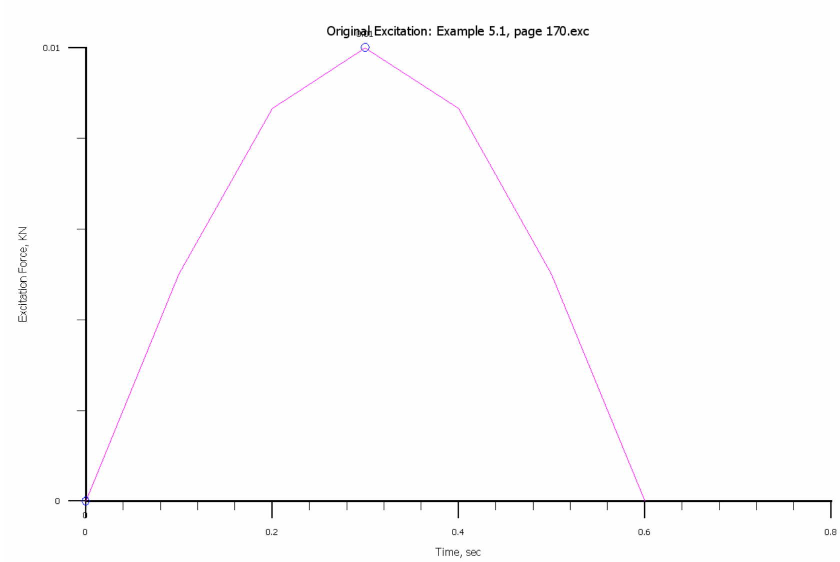
Having selected the active excitation, you can display the data both for the original excitation and its modified form.

The units in which the results are displayed can be changed by selecting *"Units"* under the *"Options"* menu of the main form.

The data is displayed by selecting the appropriate graph from the *"Excitations > Plot"* menu.

Below there is a sample of a simple excitation.

- Original excitation in N:

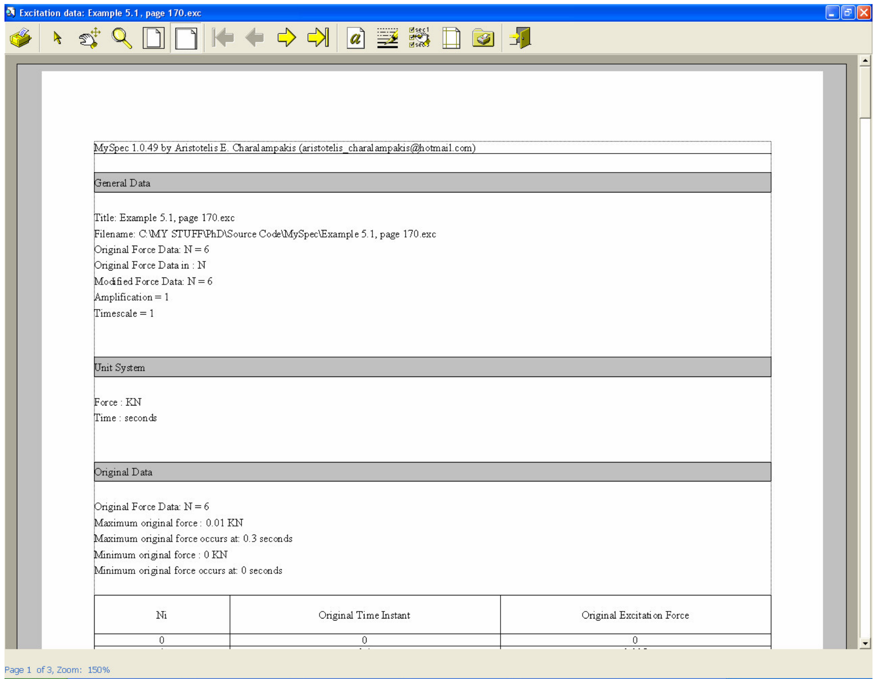


The modified excitation results depend on various factors, such as the amplification and timescale factors.

4.6 Printing and exporting the results

4.6.1 Print results

You can print the results of the calculations directly, by selecting *"Print Analysis Report"* under the *"Excitation"* menu. A full report will be prepared, and a stand alone print preview program will appear:

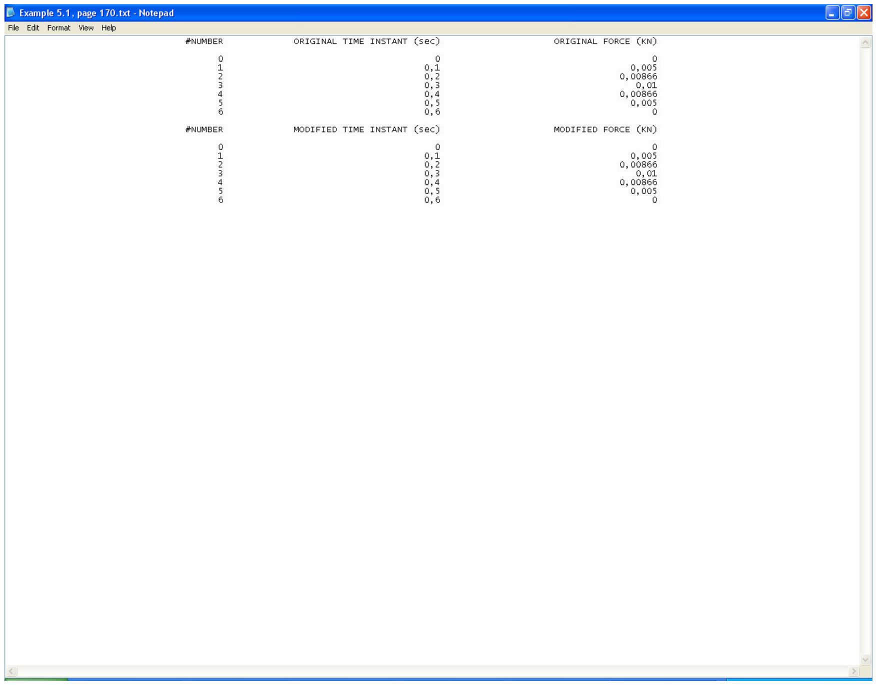


Most functions of this program are self-explanatory.

4.6.2 Export results

You can export the results in a simple ASCII text file, and use it in a spreadsheet, for example. In order to create this file, select *"Export Results"* from the *"Excitation"* menu of the main form.

The results are fixed - aligned in columns with a header for each column. You may need to modify the file, because both the original and the modified results are included in the same file.



Example 5.1, page 170.txt - Notepad

#NUMBER	ORIGINAL TIME INSTANT (sec)	ORIGINAL FORCE (KN)
0	0	0
1	0,1	0,005
2	0,2	0,00866
3	0,3	0,01
4	0,4	0,00866
5	0,5	0,005
6	0,6	0

#NUMBER	MODIFIED TIME INSTANT (sec)	MODIFIED FORCE (KN)
0	0	0
1	0,1	0,005
2	0,2	0,00866
3	0,3	0,01
4	0,4	0,00866
5	0,5	0,005
6	0,6	0

The units can be changed by selecting *"Units"* under the *"Options"* menu of the main form.

4.6.3 Print drawing

You can print directly the current drawing by selecting *"Print Drawing"* from the *"File"* menu, as described in *2.3.1 Print drawing*. The drawing is printed with the default printer settings, using the maximum available area.

4.6.4 Save drawing

You can save the current drawing in *.bmp format by selecting *"Save Image As Bmp"* from the *"File"* menu, as described in *2.3.2 Save Image as .bmp*.

5. Linear Elastic Analysis

5.1 General issues

MySpec calculates the linear elastic response of a stick model with one mass in one or two directions. The equations of the 2DOF model are uncoupled.

Furthermore, MySpec evaluates the linear elastic response spectrum of a SDF system.

Viscous damping may be used in all cases.

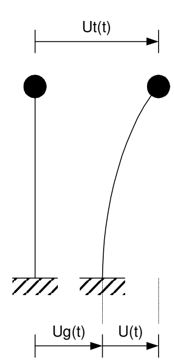
5.2 Calculations

The equation of motion for a viscously damped linear elastic SDF system is the following:

$$m \cdot \ddot{u} + c \cdot \dot{u} + k \cdot u = p(t)$$

Where u is the relative displacement of the system, $m \cdot \ddot{u}$ is the inertia force, $c \cdot \dot{u}$ is the damping force, c is the viscous damping coefficient, $k \cdot u$ is the spring force and $p(t)$ is the external dynamic force. (§1.5 Equation of motion: External force, Chopra [1])

In case of an earthquake, the external dynamic force is $p(t) = -m \cdot \ddot{u}_g(t)$, where $\ddot{u}_g(t)$ is the ground acceleration. The ground displacement $u_g(t)$ is a function of time:



The total (or absolute) displacement $u_t(t)$ is the sum of the ground displacement $u_g(t)$ and the relative displacement $u(t)$ of the system at all times.

The linear elastic response is calculated following the well-known Newmark's Method. (§5.4 Newmark's Method, Chopra [1])

In 1959, N. M. Newmark developed a family of time stepping methods, based on the following equations:

$$\dot{u}_{i+1} = \dot{u}_i + \left[(1-\gamma) \cdot \Delta t \right] \cdot \ddot{u}_i + (\gamma \cdot \Delta t) \cdot \ddot{u}_{i+1}$$
$$u_{i+1} = u_i + (\Delta t) \cdot \dot{u}_i + \left[(0.5-\beta) \cdot (\Delta t)^2 \right] \cdot \ddot{u}_i + \left[\beta \cdot (\Delta t)^2 \right] \cdot \ddot{u}_{i+1}$$

Typical selection for γ is $1/2$. For β a selection in the range $1/6 \le \beta \le 1/4$ is satisfactory.

The following set of values is used for the *average acceleration*:

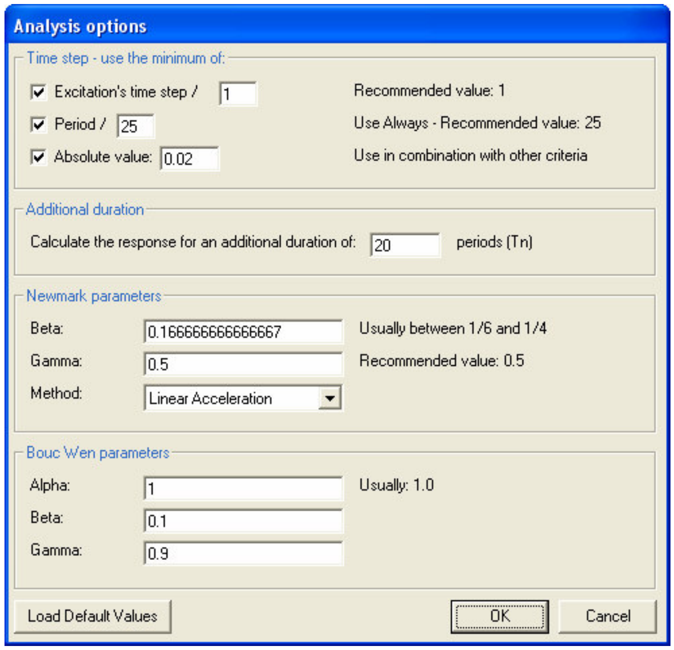
$$\gamma = 1/2$$
$$\beta = 1/4$$

Another set of values is used for the *linear acceleration*:

$$\gamma = 1/2$$
$$\beta = 1/6$$

These set of values are directly related to the assumption of the variation of the acceleration during the time step.

All parameters, such as β and γ of Newmark's method, can be modified by selecting *"Analysis"* under the *"Options"* menu of the main form. The following form will appear:



In the *"Time step – use the minimum of"* frame, the user can set restrictions on the maximum value of the time step used in the calculations. In general, the user should not modify these settings as this may result to diminished accuracy.

In the *"Additional duration"* frame, the user can select the additional duration for which the program should calculate the response. This period of time corresponds to free vibration after the end of the excitation. All graphs are plotted with magenta colour for the forced motion, followed by green colour for the duration of the free vibration.

In the "Newmark parameters" frame, the user can select the parameters β and γ of Newmark's method.

In the "Bouc Wen parameters" frame, the user can select the parameters A, β, γ of Bouc Wen model. These parameters refer to the Bouc-Wen model and are not used at this point.

If you modify any of these settings, you must re-calculate the solutions of all models.

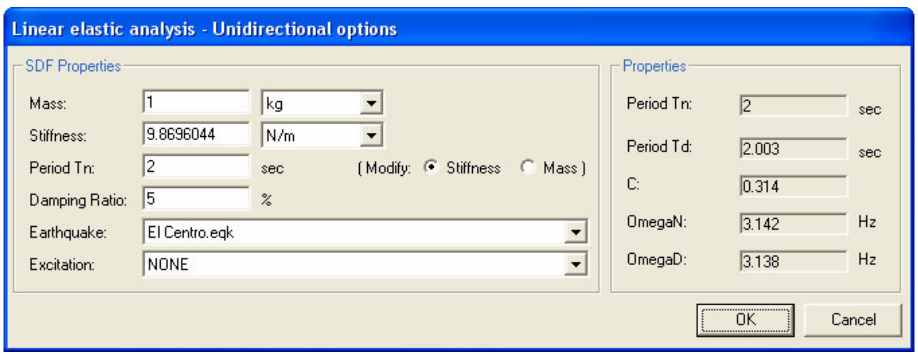
5.3 Single Unidirectional Model

5.3.1 Calculations

The response is calculated using Newmark's method. Refer to *5.2 Calculations* for more information.

5.3.2 Input data

In order to use the Single Unidirectional Model, select *"Single Unidirectional Model > Options"* from the *"Linear Elastic Analysis"* menu. The following form will appear:



In the *"SDF Properties"* frame, the following data is required:

- Mass: The mass of the SDF system. Make sure to select the correct units from the drop-down list.
- Stiffness: The stiffness of the SDF system. Make sure to select the correct units from the drop-down list.
- Period: The natural period T_n of the system in seconds, which is calculated as follows:

$$T_n = 2 \cdot \pi \cdot \sqrt{\frac{m}{k}}$$

If the user types a desired value of period in the text box, one of the previous text boxes (mass or stiffness) is changed accordingly, based on the option button *Modify : Mass Or Stiffness* on the right.

- Damping ratio: The damping ratio ζ of the system, in percentage (%). The damping ratio (or fraction of critical damping) is defined as follows:

$$\zeta = \frac{c}{c_{cr}} = \frac{c}{2 \cdot m \cdot \omega_n}$$

The critical damping coefficient is defined as follows:

$$c_{cr} = 2 \cdot m \cdot \omega_n$$
$$\omega_n = \sqrt{\frac{k}{m}}$$

Where ω_n is the natural frequency. The critical damping coefficient is used in viscously damped vibrations. For example, the equation governing viscously damped free vibration of a SDF system is the following:

$$\ddot{u} + 2 \cdot \zeta \cdot \omega_n \cdot \dot{u} + \omega_n^2 \cdot u = 0$$

- Earthquake: In addition to or separately from the excitation, you can select the desired earthquake from the drop-down list. Note that the *modified* form of the earthquake is used in the calculations.
- Excitation: In addition to or separately from the earthquake, you can select the desired excitation from the drop-down list. Note that the *modified* form of the excitation is used in the calculations.

In the *"Properties"* frame, the following results are displayed:

- Period T_n: The natural period of the system in seconds, which is calculated as follows:

$$T_n = 2 \cdot \pi \cdot \sqrt{\frac{m}{k}}$$

- Period T_d: The natural period of damped vibration in seconds, which is related to T_n by the following equation:

$$T_d = \frac{T_n}{\sqrt{1 - \zeta^2}}$$

- C: The damping coefficient, which is related to the damping ratio ζ by the following equation:

$$c = 2 \cdot m \cdot \omega_n \cdot \zeta$$

- OmegaN: The natural frequency ω_n of the SDF system, which is given by the following equation:

$$\omega_n = \sqrt{\frac{k}{m}}$$

- OmegaD: The natural frequency of damped vibration ω_d of the SDF system, which is related to the undamped natural frequency ω_n by the following equation:

$$\omega_d = \omega_n \cdot \sqrt{1-\zeta^2}$$

When you have successfully entered all data, click on the *"OK"* button. The response is calculated automatically.

5.3.3 Results

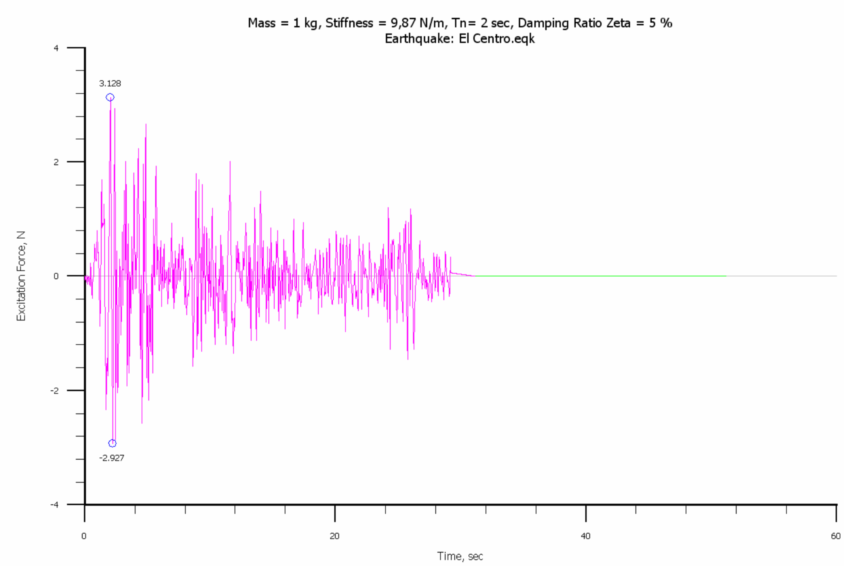
In order to display the results, select the appropriate graph by clicking on *"Single Unidirectional Model > Plot …"* from the *"Linear Elastic Analysis"* menu of the main form.

The units in which the results are displayed can be changed by selecting *"Units"* under the *"Options"* menu of the main form.

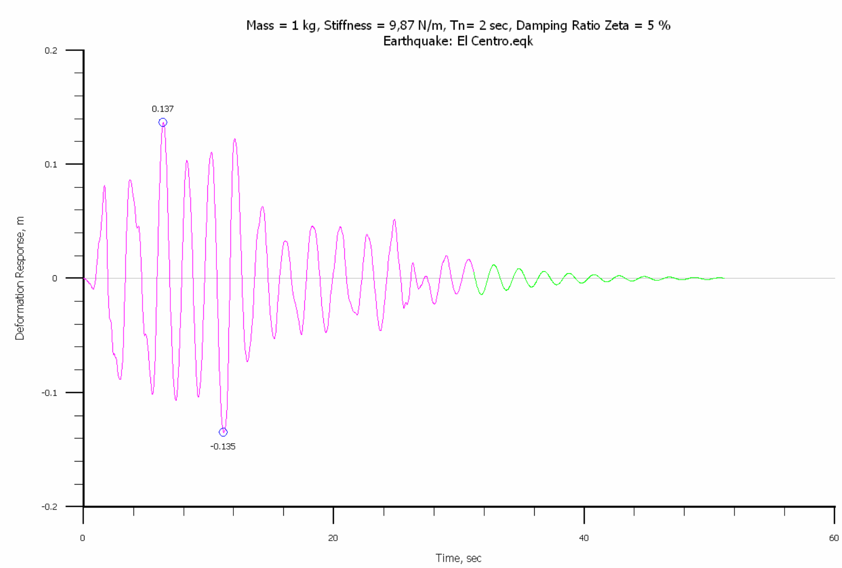
Note that the forced vibration graph is represented by the magenta colour; the free vibration graph is represented by the green colour.

The results for the example shown in *5.3.2 Input data* and the El Centro earthquake are the following:

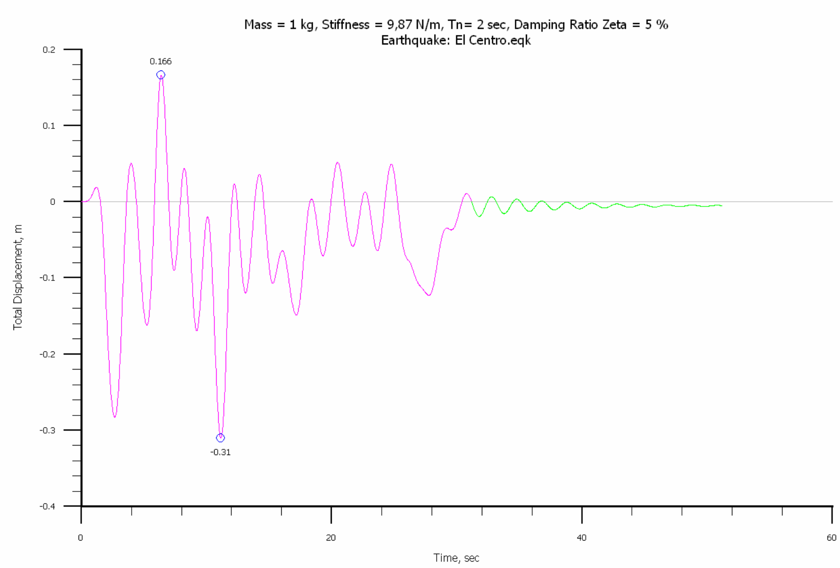
- Excitation force:



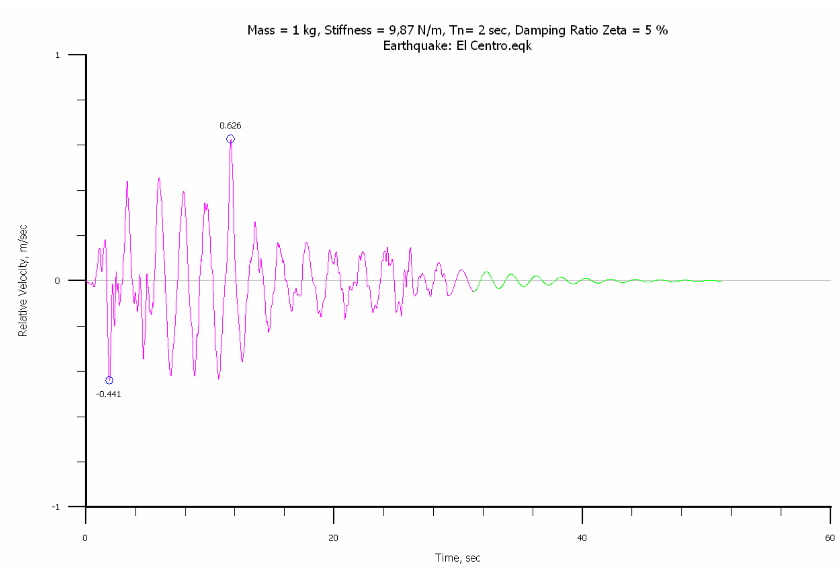
➢ Relative displacement:



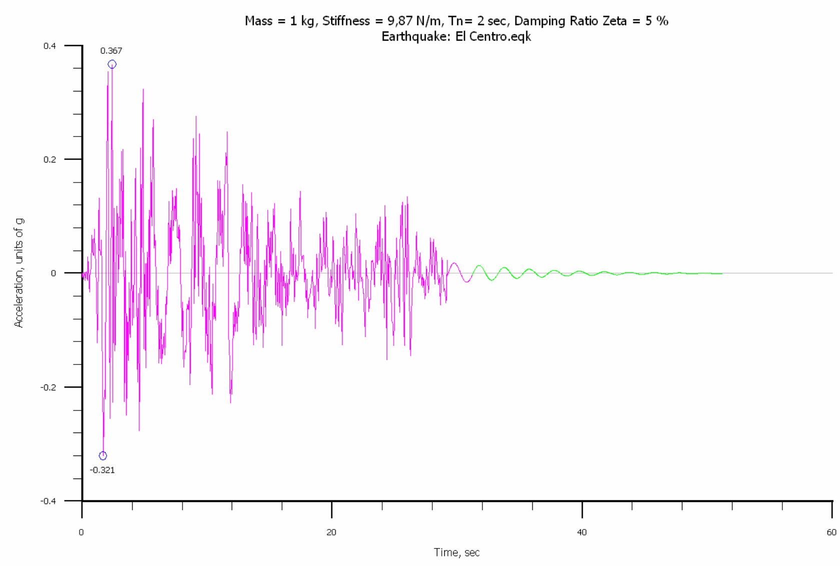
➢ Total displacement (This is the sum of the relative displacement plus the ground displacement, if any):



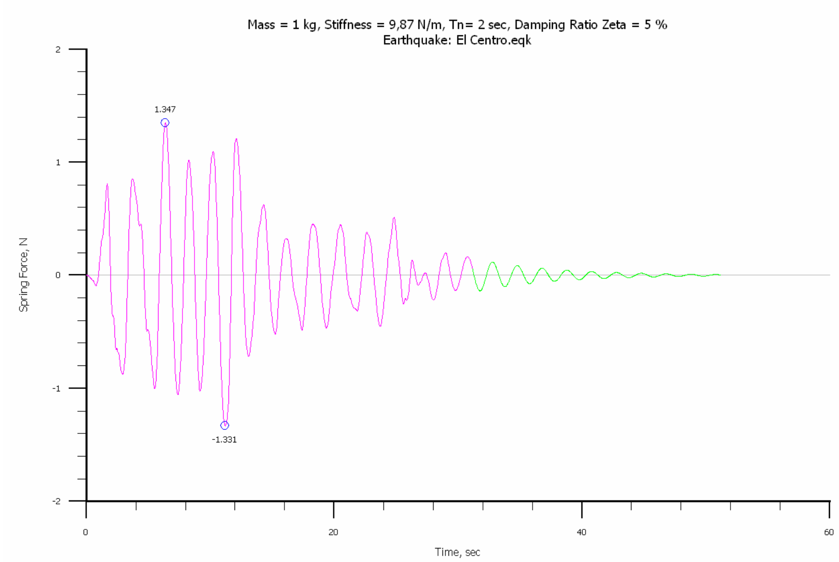
➢ Relative velocity:



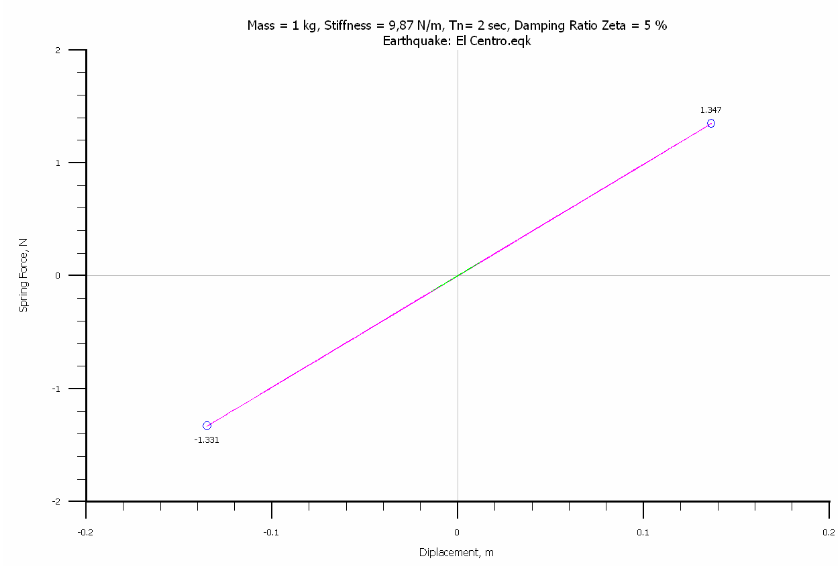
➢ Acceleration:



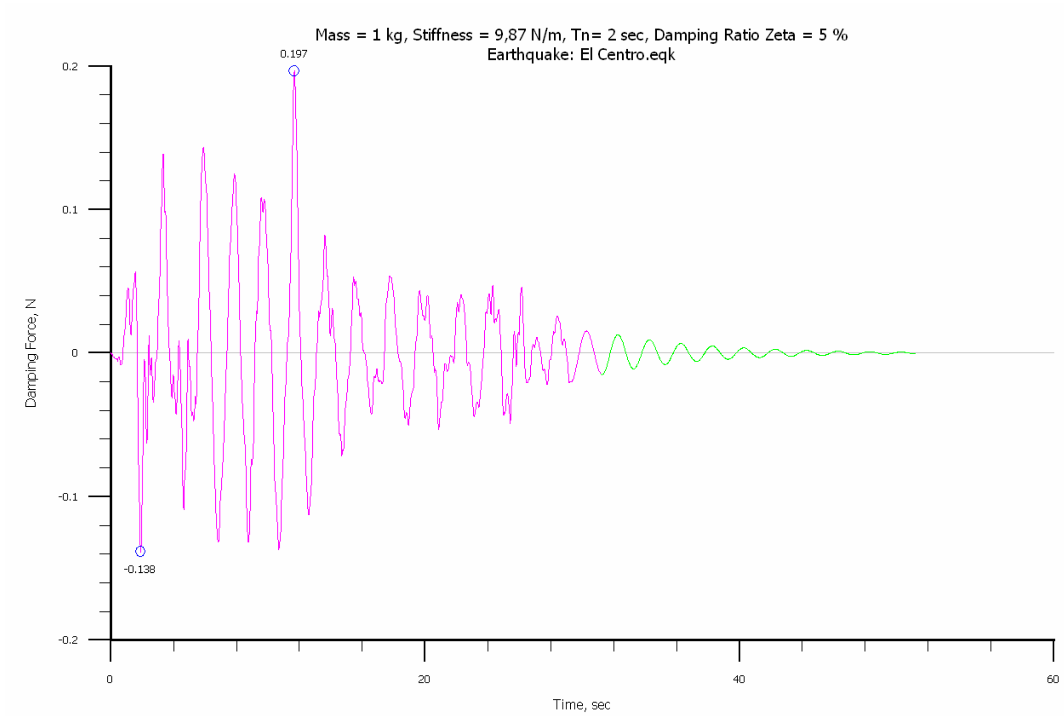
➢ Spring force vs time: (In linear elastic analysis, this graph is the deformation response graph multiplied by the stiffness, i.e. $F_s = k \cdot u$)



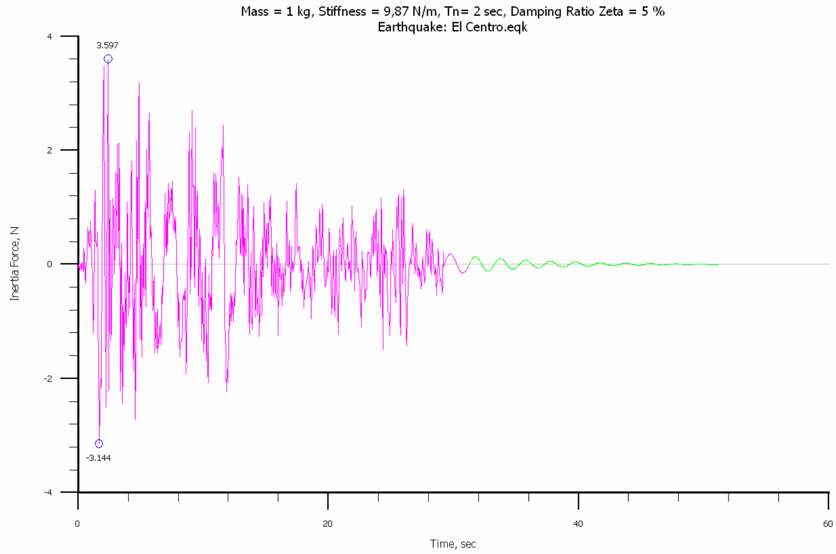
➢ Spring force vs displacement: (In linear elastic analysis, all points of this graph lie on the same line, because $F_s = k \cdot u$ and there is no yielding)



➢ Damping force: (This graph is the relative velocity graph multiplied by the damping coefficient)

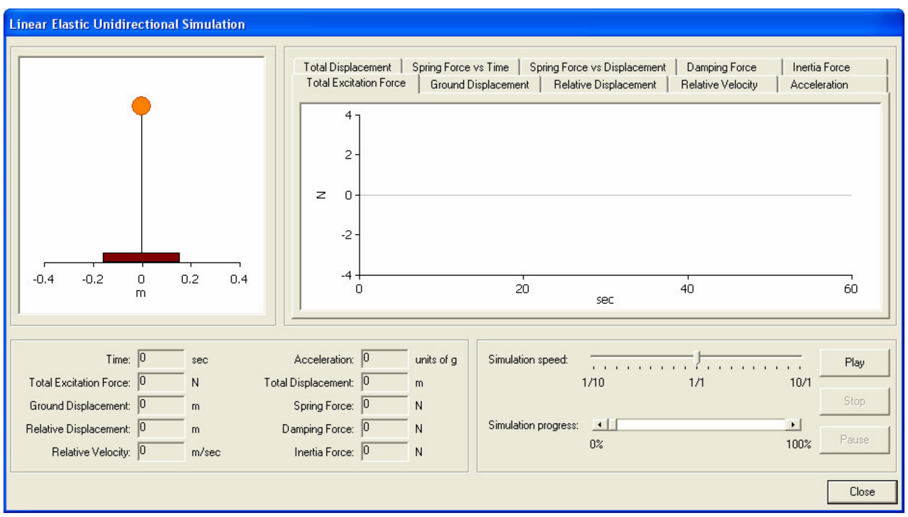


➢ Inertia force: (This graph is the acceleration graph multiplied by the mass)



5.3.4 Simulation

In order to simulate the response of the SDF system and present it in animated form, select *"Single Unidirectional Model > Response simulation"* from the *"Linear Elastic Analysis"* menu of the main form. The following form will appear:



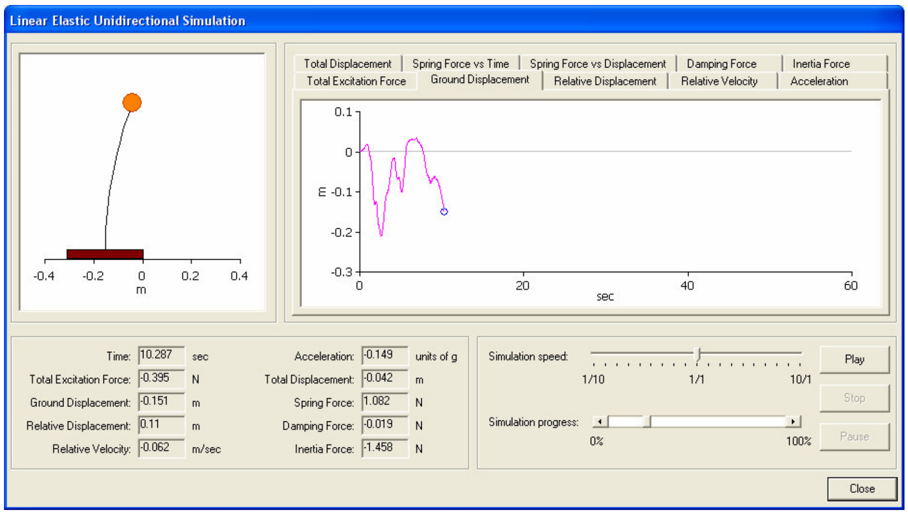
You can simulate the response by clicking on the *"Play"* button of the bottom right frame. In the same frame, you can adjust the speed of the simulation by using the slider and you can jump to a specific point of the simulation by using the horizontal progress bar.

At each time instant all data is displayed in the bottom left frame.

In the top left frame, you can see the main simulation panel which depicts an ideal SDF system: a lumped mass on top of a massless supporting structure (column), which, in turn, is fixed on the ground, which moves under an earthquake.

Finally, you can pick one of the diagrams available in the top right frame and watch its progress.

During the simulation the form may look like this:



The units in which the results are displayed can be changed by selecting *"Units"* under the *"Options"* menu of the main form.

5.3.5 Export results

You can export the results in a simple ASCII text file, and use it for example in a spreadsheet application. In order to create this file, select *"Single Unidirectional Model > Export Results"* from the *"Linear Elastic Analysis"* menu of the main form.

The results are fixed - aligned in columns with a header for each column.

5.4 Single Bidirectional Model

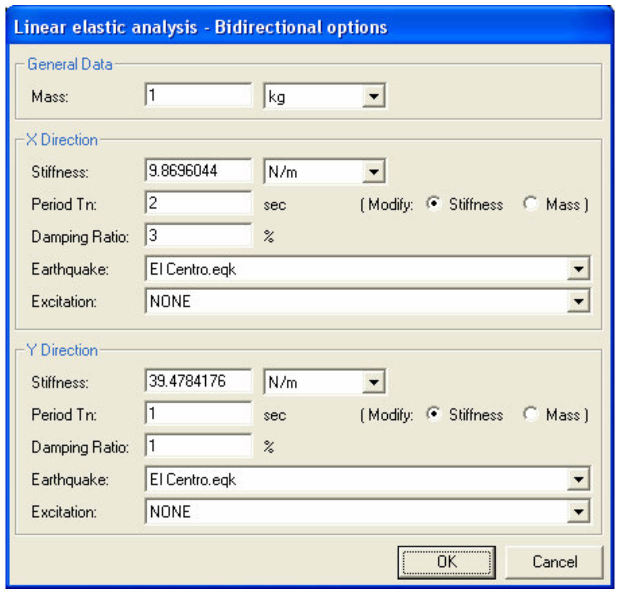
5.4.1 Calculations

The response is calculated using Newmark's method. Refer to *5.2 Calculations* for more information.

Note that the two degrees of freedom are uncoupled. Therefore, for the same input as in the single unidirectional model, the results are the same in both directions.

5.4.2 Input data

In order to use the Single Bidirectional Model, select *"Single Bidirectional Model > Options"* from the *"Linear Elastic Analysis"* menu. The following form will appear:



In each of the X, Y directions, the following data is required:

- Mass: The mass of the system. Make sure to select the correct units from the drop-down list box.
- Stiffness: The stiffness of the system. Make sure to select the correct units from the drop-down list box.
- Period: The natural period T_n of the system in seconds, which is calculated as follows:

$$T_n = 2 \cdot \pi \cdot \sqrt{\frac{m}{k}}$$

  If the user types a desired value of period in the text box, one of the previous text boxes (mass or stiffness) is changed accordingly, based on the option button *Modify : Mass Or Stiffness* on the right.

- Damping ratio: The damping ratio ζ of the system, in percentage (%). The damping ratio (or fraction of critical damping) is defined as follows:

$$\zeta = \frac{c}{c_{cr}} = \frac{c}{2 \cdot m \cdot \omega_n}$$

The critical damping coefficient is defined as follows:

$$c_{cr} = 2 \cdot m \cdot \omega_n$$

$$\omega_n = \sqrt{\frac{k}{m}}$$

Where ω_n is the natural frequency. The critical damping coefficient is used in viscously damped vibrations. For example, the equation governing viscously damped free vibration of a SDF system is the following:

$$\ddot{u} + 2 \cdot \zeta \cdot \omega_n \cdot \dot{u} + \omega_n^2 \cdot u = 0$$

- Earthquake: In addition to or separately from the excitation, you can select the desired earthquake from the drop-down list. Note that the *modified* form of the earthquake is used in the calculations.
- Excitation: In addition to or separately from the earthquake, you can select the desired excitation from the drop-down list. Note that the *modified* form of the excitation is used in the calculations.

When you have successfully entered all data, click on the *"OK"* button. The response is calculated automatically.

5.4.3 Results

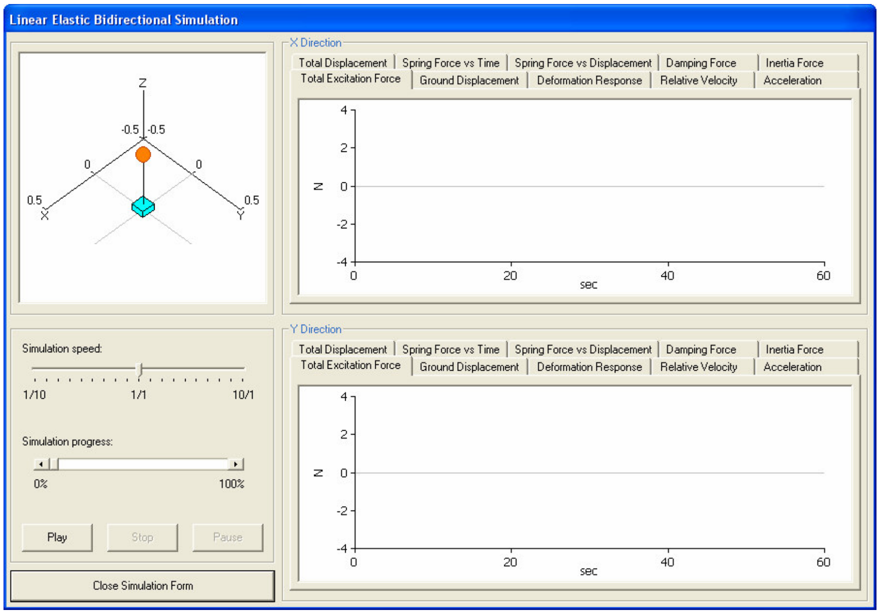
In order to display the results, select the appropriate graph by clicking on *"Single Bidirectional Model > Plot X (or Y) > …"* from the *"Linear Elastic Analysis"* menu of the main form.

The units in which the results are displayed can be changed by selecting *"Units"* under the *"Options"* menu of the main form.

Note that the forced vibration period is displayed by magenta colour; the free vibration is displayed by green colour.

5.4.4 Simulation

In order to simulate the response of the system in animated form, select *"Single Bidirectional Model > Response simulation"* from the *"Linear Elastic Analysis"* menu of the main form. The following form will appear:

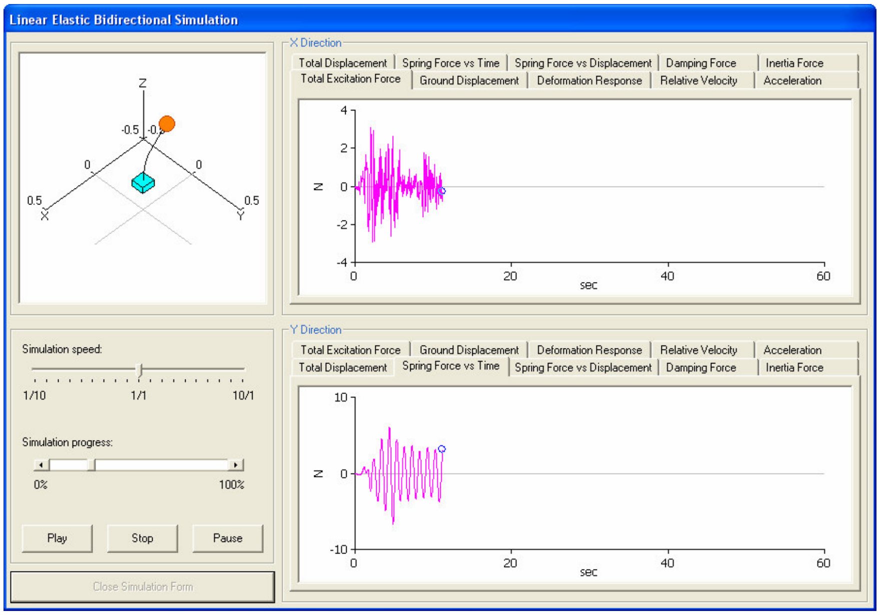


You can simulate the response by clicking on the *"Play"* button of the left frame. In the same frame, you can adjust the speed of the simulation by using the slider and you can jump to a specific point of the simulation by using the horizontal progress bar.

In the top left frame, you can see the main simulation panel which depicts an ideal 2DOF system: a lumped mass on top of a massless supporting structure (column), which, in turn, is fixed on the ground which moves in case of an earthquake.

Finally, you can pick one of the diagrams available in the top (for the X direction) and the bottom (for the Y direction) right frame and watch its progress.

During the simulation the form may look like this:



The units in which the results are displayed can be changed by selecting *"Units"* under the *"Options"* menu of the main form.

5.4.5 Export results

You can export the results in a simple ASCII text file, and use it in a spreadsheet, for example. In order to create this file, select *"Single Bidirectional Model > Export Results"* from the *"Linear Elastic Analysis"* menu of the main form.

The results are fixed - aligned in columns with a header for each column.

5.5 Elastic Response Spectrum

5.5.1 Calculations

Given a range of periods for a SDF system and the time step, MySpec can produce the elastic response spectrum of various response parameters for a specific excitation and / or earthquake. The response is calculated using Newmark's method for each period (refer to *5.2 Calculations* for more information).

The mass of the SDF system is needed in case of force excitations. Also, viscous damping may be taken into account.

The program may trim the very small periods or use a minimum time step because the iterative process is time consuming; for each period the full response of the SDF system must be calculated so that the peak values can be stored.

MySpec can produce response spectra for the following quantities (Refer to §6.5 Response spectrum concept, Chopra [1]):

- Deformation: The peak value of deformation D for each period.
- Pseudo velocity: The peak value of the pseudo velocity V which is defined as follows:

  $$V = \omega_n \cdot D$$

  Where ω_n is the natural frequency and D is the peak deformation of the *same* system.

  Pseudo velocity has units of velocity and it is related to the peak value of the strain energy E_{s0} stored in the system during the excitation:

  $$E_{s0} = m\frac{V^2}{2}$$

  (§6.6.2 Pseudo velocity Response Spectrum, Chopra [1])
- Pseudo acceleration: The peak value of the pseudo acceleration A which is defined as follows:

  $$a = \omega_n^{\,2} \cdot D$$

  Where ω_n is the natural frequency and D is the peak deformation of the *same* system.

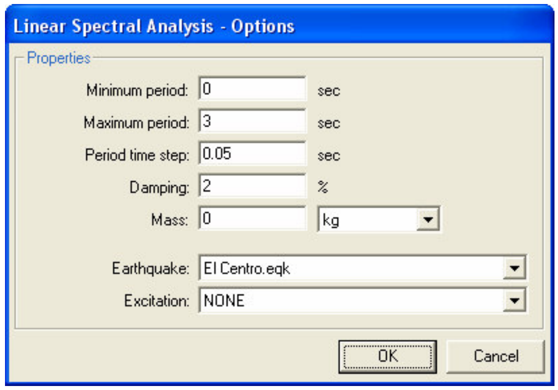
  Pseudo acceleration has units of acceleration and it is related to the peak value of base shear V_{b0} or the peak value of the equivalent static force f_{s0}:

  $$V_{b0} = f_{s0} = m \cdot A$$

  (§6.6.3 Pseudo acceleration Response Spectrum, Chopra [1]),
- Relative Velocity: The peak value of the relative velocity.
- Acceleration: The peak value of the acceleration.

5.5.2 Input data

In order to produce the elastic response spectrum, select *"Response Spectrum > Options"* from the *"Linear Elastic Analysis"* menu. The following form will appear:



The following data is required:

- Minimum period: The minimum period of the range, in seconds.
- Maximum period: The maximum period of the range, in seconds.
- Period time step: The time step, in seconds.
- Damping: The damping ratio ζ of the system, in percentage (%). The damping ratio (or fraction of critical damping) is defined as follows:

  $$\zeta = \frac{c}{c_{cr}} = \frac{c}{2 \cdot m \cdot \omega_n}$$

  The critical damping coefficient is defined as follows:

  $$c_{cr} = 2 \cdot m \cdot \omega_n$$
  $$\omega_n = \sqrt{\frac{k}{m}}$$

  Where ω_n is the natural frequency.

- Mass: The mass of the SDF system. This is required in case of force excitations. Make sure to select the correct units from the drop-down list.
- Earthquake: In addition to or separately from the excitation, you can select the desired earthquake from the drop-down list. Note that the *modified* form of the earthquake is used in the calculations.

➢ Excitation: In addition to or separately from the earthquake, you can select the desired excitation from the drop-down list. Note that the *modified* form of the excitation is used in the calculations.

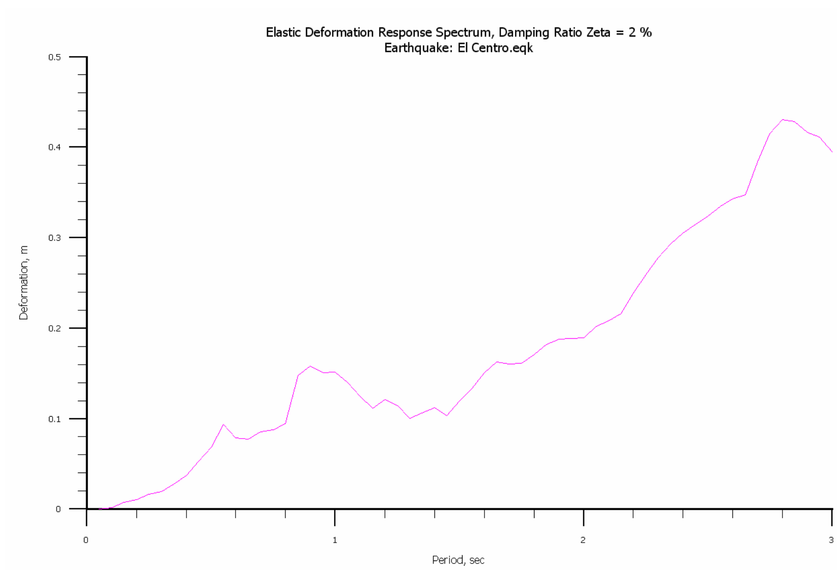
5.5.3 Results

In order to display the results, select the appropriate graph by clicking on *"Response Spectrum > Plot …"* from the *"Linear Elastic Analysis"* menu of the main form.

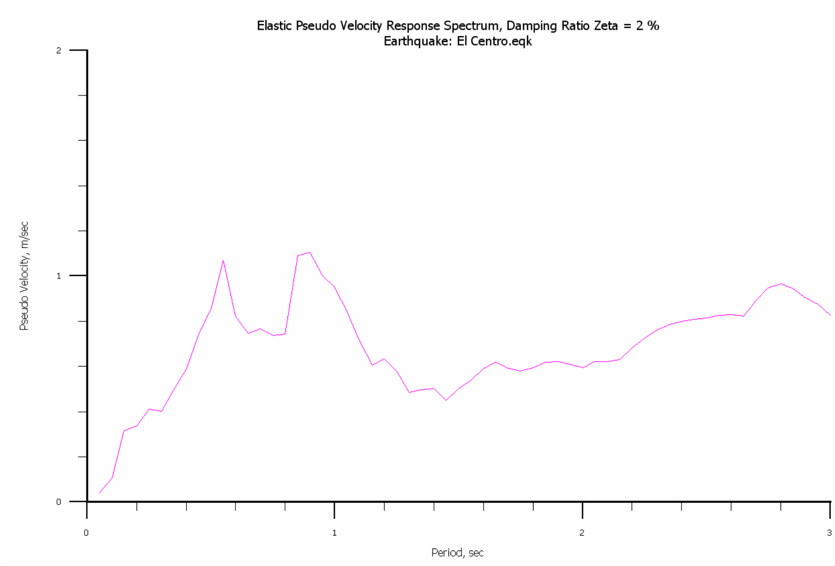
The units in which the results are displayed can be changed by selecting *"Units"* under the *"Options"* menu of the main form.

The results for the example shown in *5.5.2 Input data* and the El Centro earthquake are the following:

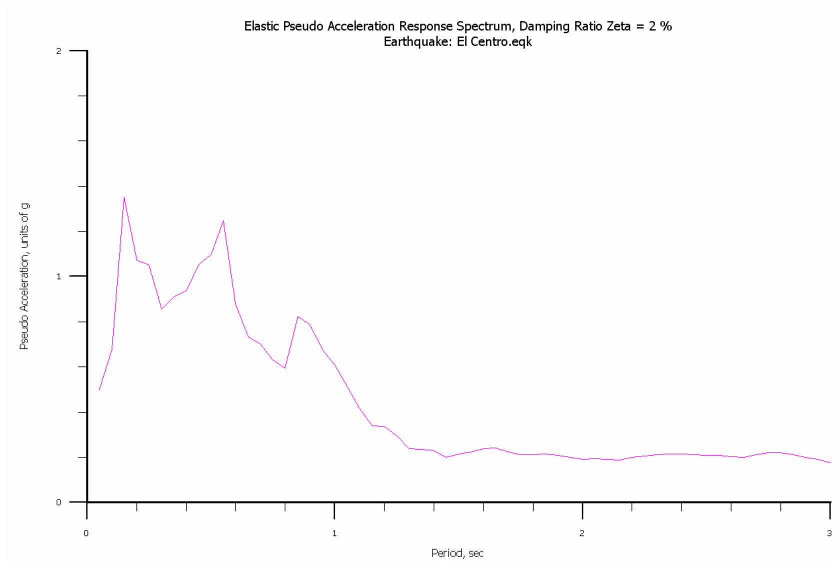
➢ Deformation response spectrum:



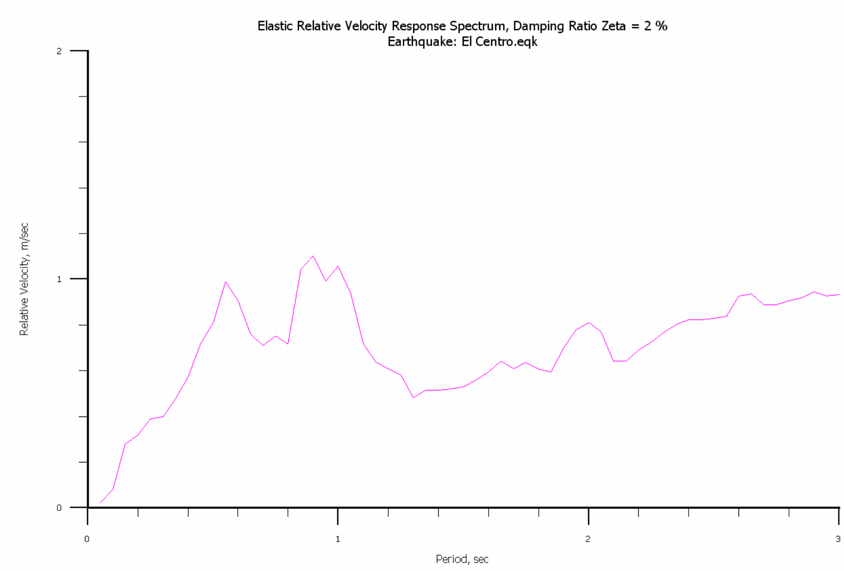
➢ Pseudo velocity response spectrum:



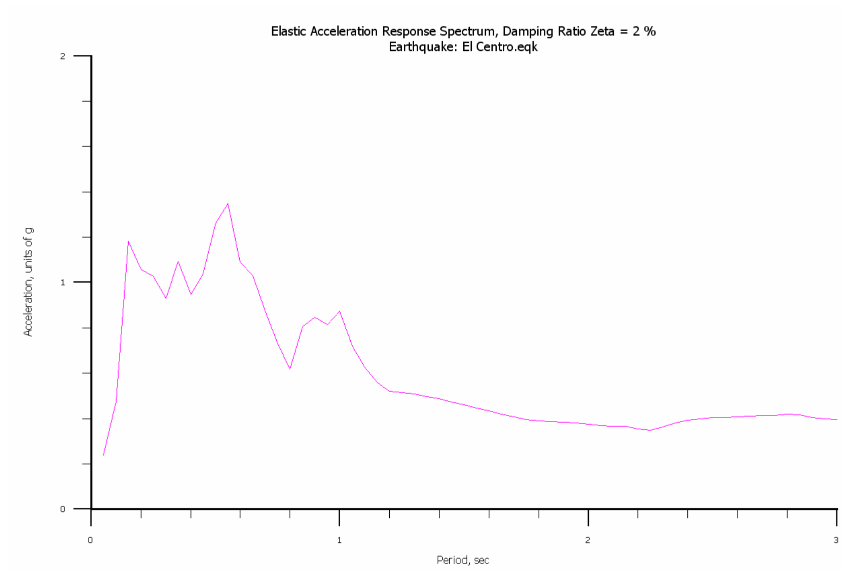
➢ Pseudo acceleration response spectrum:



➢ Relative velocity response spectrum:



- Acceleration response spectrum:



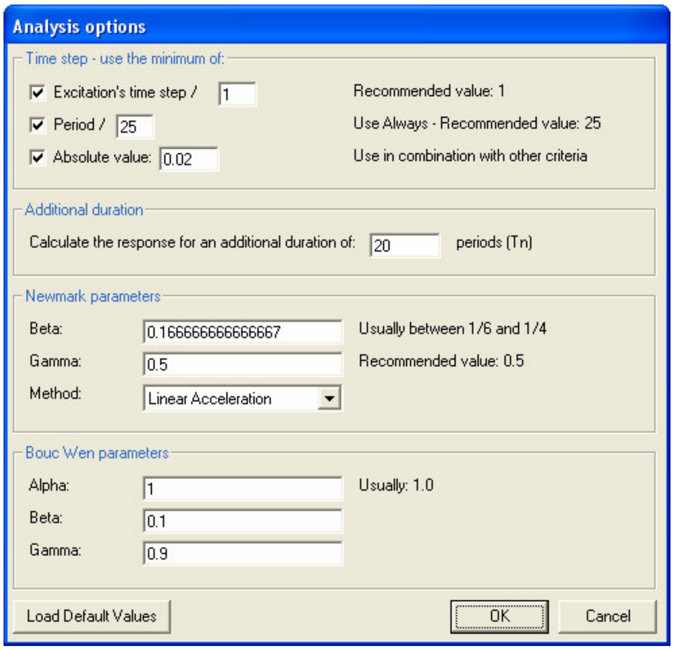
6. Non Linear Analysis – Bilinear Model

6.1 General issues

MySpec evaluates the non linear response based either on a generic bilinear model or a Bouc – Wen hysteretic model.

6.2 Analysis options

All analysis parameters can be inserted or modified by selecting *"Analysis"* under the *"Options"* menu of the main form. The following form will appear:



In the *"Time step – use the minimum of"* frame, the user can set restrictions on the maximum value of the time step used in the calculations. In general, the user should not modify these settings as this may result to diminished accuracy.

In the *"Additional duration"* frame, the user can select the additional duration for which the program should calculate the response. This period of time corresponds to free vibration after the end of the excitation. All graphs display the response during this period of time with green colour, while for the duration of the forced vibration the response is displayed with magenta.

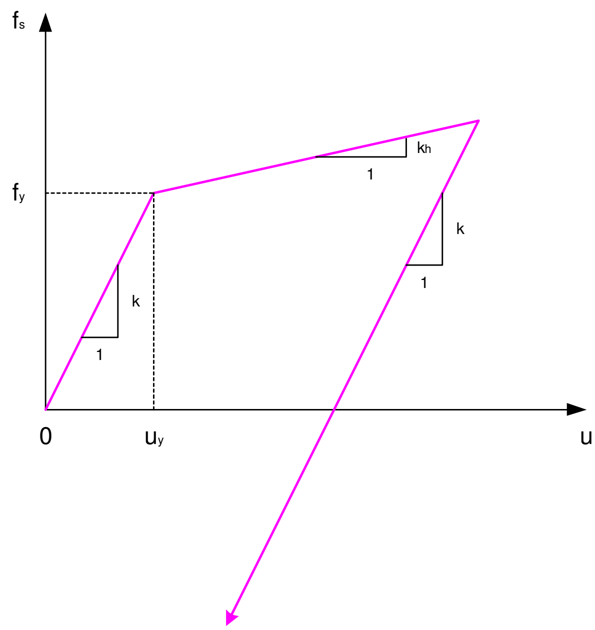
In the "Newmark parameters" frame, the user can select the parameters β and γ of Newmark's method.

In the "Bouc Wen parameters" frame, the user can select the parameters A, β, γ of Bouc Wen model.

If you modify any of these settings, you must re-calculate the solutions of all models.

6.3 Calculations

MySpec evaluates the nonlinear response based on a generic bilinear model. A typical bilinear force - displacement model is shown below:



The system is linearly elastic with stiffness k up to the yield force. For displacement bigger than u_y the system responds with a constant hardening stiffness k_h.

In the unloading branch, the system regains its initial stiffness k.

The yield strength f_y and the hardening stiffness k_h may be different for the positive and negative directions of loading. Also, if the system has yielded and is unloaded, the new elastic branch has the same yield strength *range* as the initial values i.e. $f_{y.pos} + \left| f_{y.neg} \right|$ but may be displaced in the positive or the negative direction because of the hardening stiffness.

The response is calculated with the well-known Newmark Method. (§5.7 Analysis of Nonlinear response: Newmark's Method, Chopra [1]).

In general, Newmark's method is very satisfactory in terms of accuracy. Since the time step is constant, two are the main sources of error:

- The tangent stiffness is used instead of the (actual) secant stiffness in the calculation of the incremental resisting force. The secant stiffness cannot be used because it is not known.
- The detection of the transitions in the force – deformation relationship is inaccurate. This inevitably leads to error accumulation.

MySpec addresses these errors by using a small time step (this can be modified by the user) and by *modifying the time step* at the transitions from the elastic to inelastic branch and vice versa.

In the first case i.e. the transition from the elastic to the inelastic branch, the condition is that the resisting force equals the yield strength: $f_s = f_y$.

In the second case i.e. the transition from the loading to the unloading branch, the condition is that the velocity is zero: $\dot{u} = 0$.

When a transition is detected, the time step is continuously divided by two in order to detect the transition point as accurately as possible. When the corresponding condition is met to certain accuracy, the algorithm continues to the next time step.

As mentioned earlier, N. M. Newmark's method is based on the following equations:

$$\dot{u}_{i+1} = \dot{u}_i + \left[(1-\gamma) \cdot \Delta t \right] \cdot \ddot{u}_i + (\gamma \cdot \Delta t) \cdot \ddot{u}_{i+1}$$
$$u_{i+1} = u_i + (\Delta t) \cdot \dot{u}_i + \left[(0.5-\beta) \cdot (\Delta t)^2 \right] \cdot \ddot{u}_i + \left[\beta \cdot (\Delta t)^2 \right] \cdot \ddot{u}_{i+1}$$

Typical selection for γ is $1/2$. For β a selection in the range $1/6 \le \beta \le 1/4$ is satisfactory.

The following set of values is used for the *average acceleration*:

$$\gamma = \frac{1}{2}$$
$$\beta = \frac{1}{4}$$

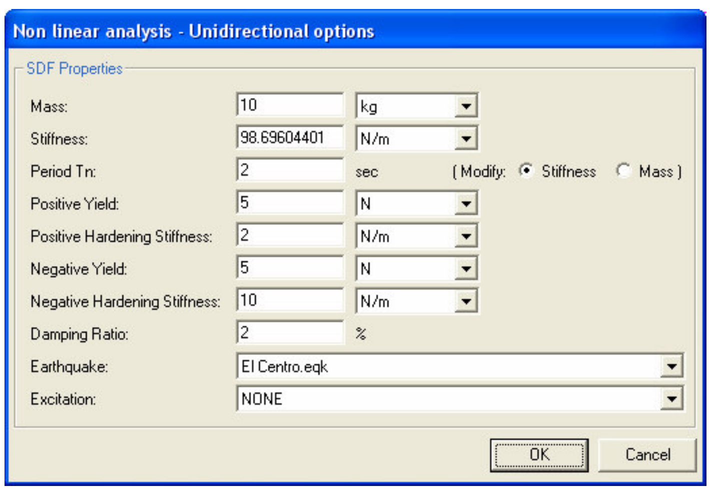
Another set of values is used for the *linear acceleration*:

$$\gamma = \frac{1}{2}$$
$$\beta = \frac{1}{6}$$

These set of values are directly related to the assumption of the variation of the acceleration during the time step.

6.4 Input data

In order to use the bilinear model, select *"Single Unidirectional (Bilinear Model) > Options"* from the *"Non Linear Analysis"* menu. The following form will appear:



In the *"SDF Properties"* frame, the following data is required:

- Mass: The mass of the SDF system. Make sure to select the correct units from the drop-down list box.
- Stiffness: The stiffness of the SDF system. Make sure to select the correct units from the drop-down list box.
- Period: The natural period T_n of the system in seconds, which is calculated as follows:

$$T_n = 2 \cdot \pi \cdot \sqrt{\frac{m}{k}}$$

If the user types a desired value of period in the text box, one of the previous text boxes (mass or stiffness) is changed accordingly, based on the option button *Modify : Mass Or Stiffness* on the right.

- Positive yield: The positive yield strength of the system. Make sure to select the correct units from the drop-down list box.
- Positive hardening stiffness: The positive hardening stiffness of the system. Make sure to select the correct units from the drop-down list box.
- Negative yield: The negative yield strength of the system (use positive values). Make sure to select the correct units from the drop-down list box.
- Negative hardening stiffness: The negative hardening stiffness of the system (use positive values). Make sure to select the correct units from the drop-down list box.
- Damping ratio: The damping ratio ζ of the system, in percentage (%). The damping ratio (or fraction of critical damping) is defined as follows:

$$\zeta = \frac{c}{c_{cr}} = \frac{c}{2 \cdot m \cdot \omega_n}$$

The critical damping coefficient is defined as follows:

$$c_{cr} = 2 \cdot m \cdot \omega_n$$
$$\omega_n = \sqrt{\frac{k}{m}}$$

Where ω_n is the natural frequency.

- Earthquake: In addition to or separately from the excitation, you can select the desired earthquake from the drop-down list box. Note that the *modified* form of the earthquake is used in the calculations.
- Excitation: In addition to or separately from the earthquake, you can select the desired excitation from the drop-down list box. Note that the *modified* form of the excitation is used in the calculations.

When you have successfully entered all data, click on the *"OK"* button. The response is calculated automatically.

6.5 Results

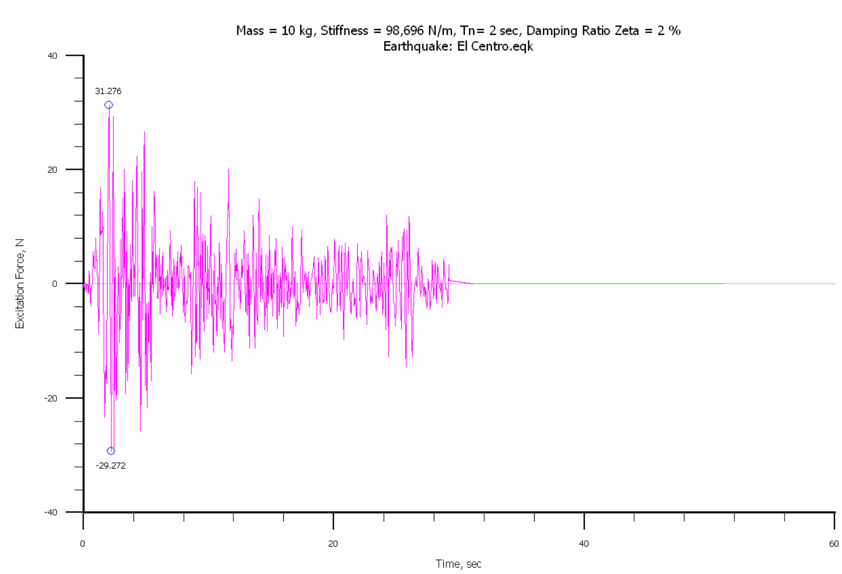
In order to display the results, select the appropriate graph by clicking on *"Single Unidirectional (Bilinear Model) > Plot …"* from the *"Non Linear Analysis"* menu of the main form.

The units in which the results are displayed can be changed by selecting *"Units"* under the *"Options"* menu of the main form.

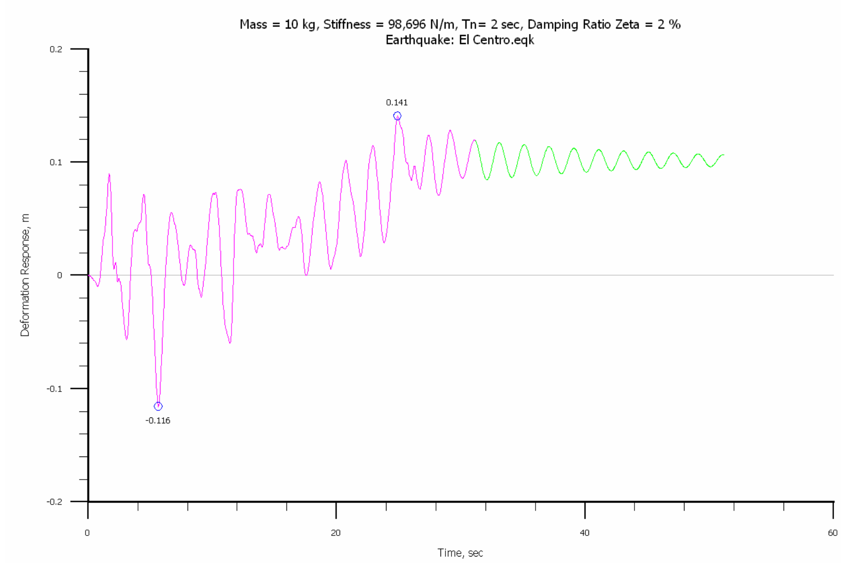
Note that the forced vibration period is represented by magenta colour; the free vibration is represented by green colour.

The results for the example shown in *6.3.2 Input data* and the El Centro earthquake are the following:

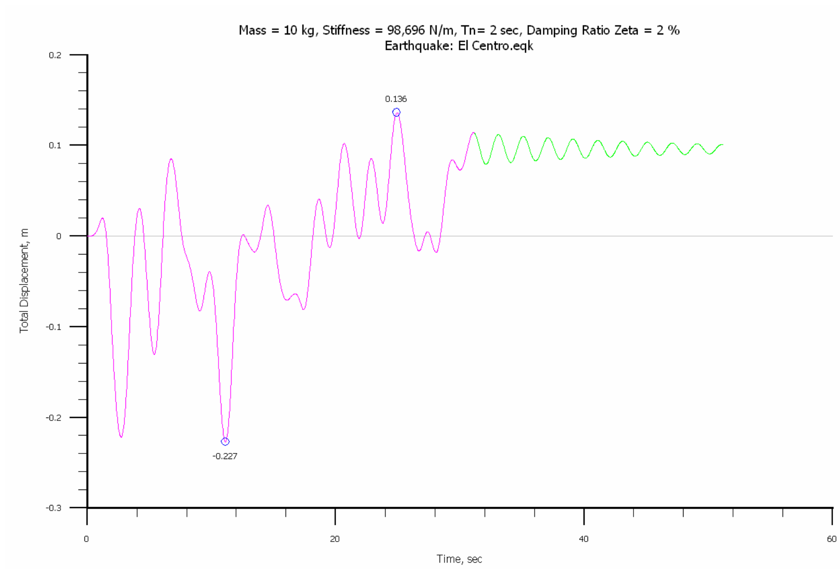
- Excitation force:



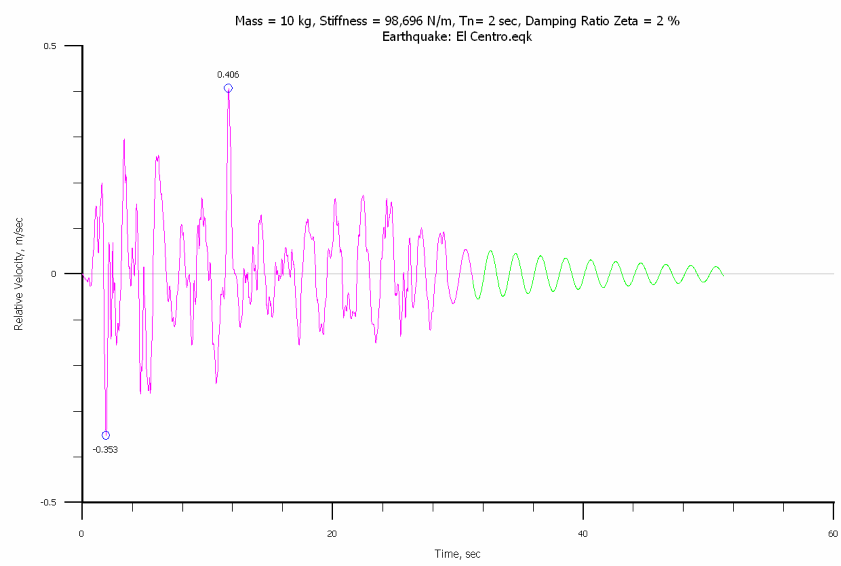
- Relative Displacement:



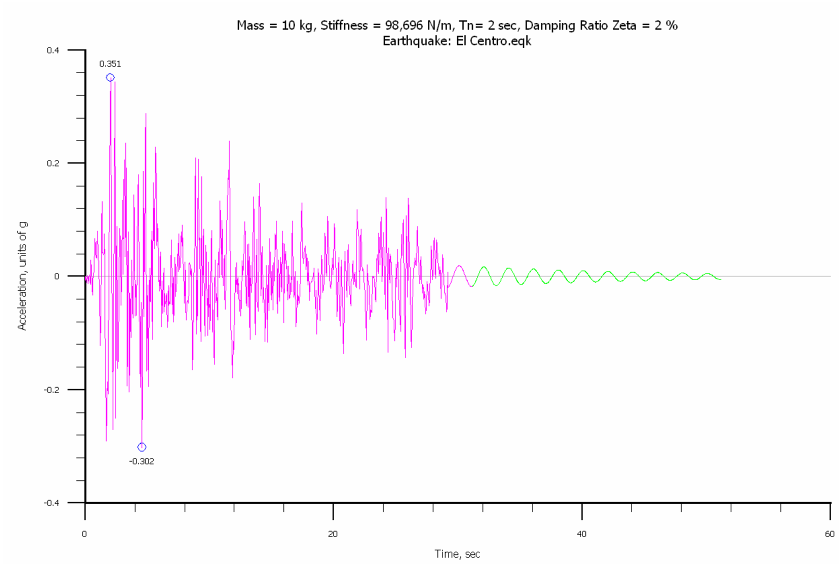
- ➢ Total displacement:



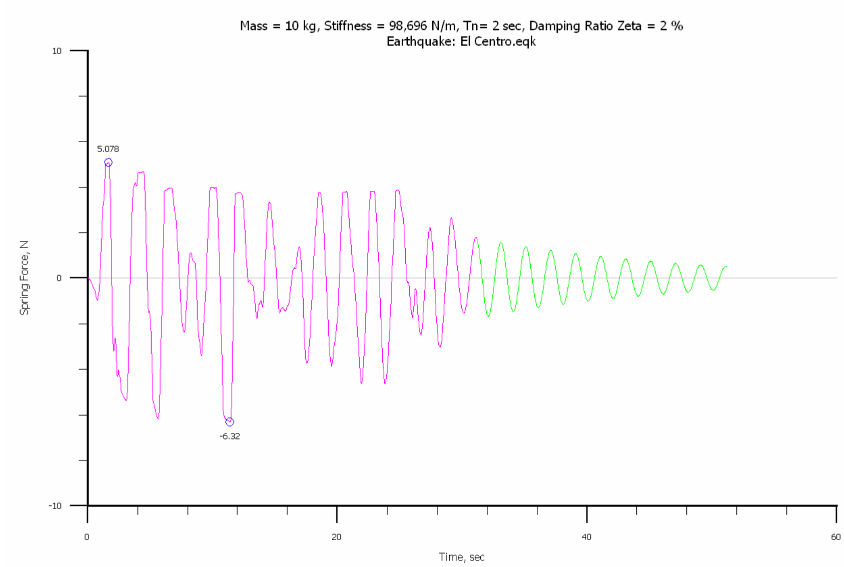
- ➢ Relative velocity:



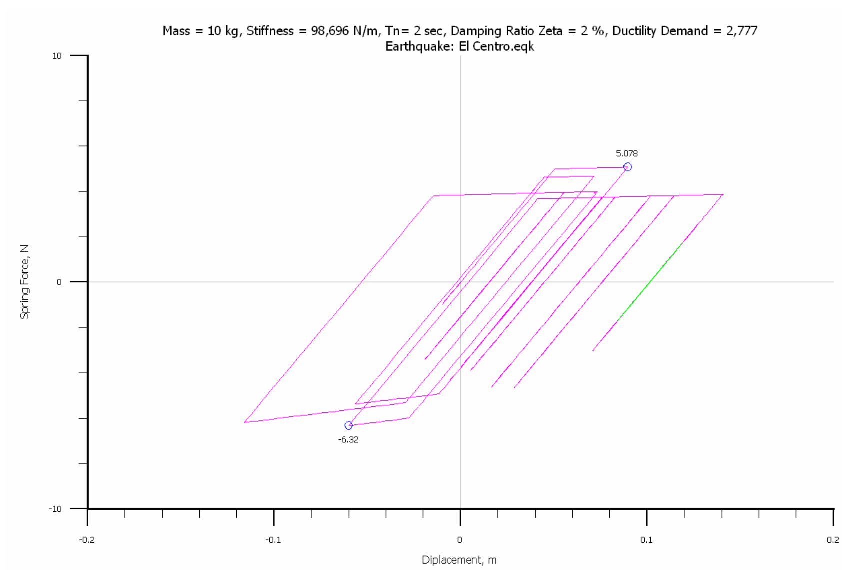
➢ Acceleration:



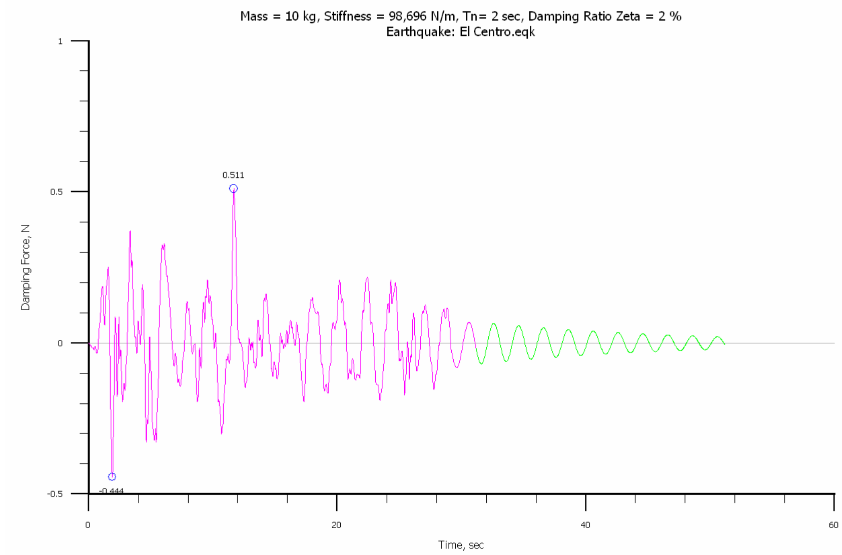
➢ Spring force vs time:



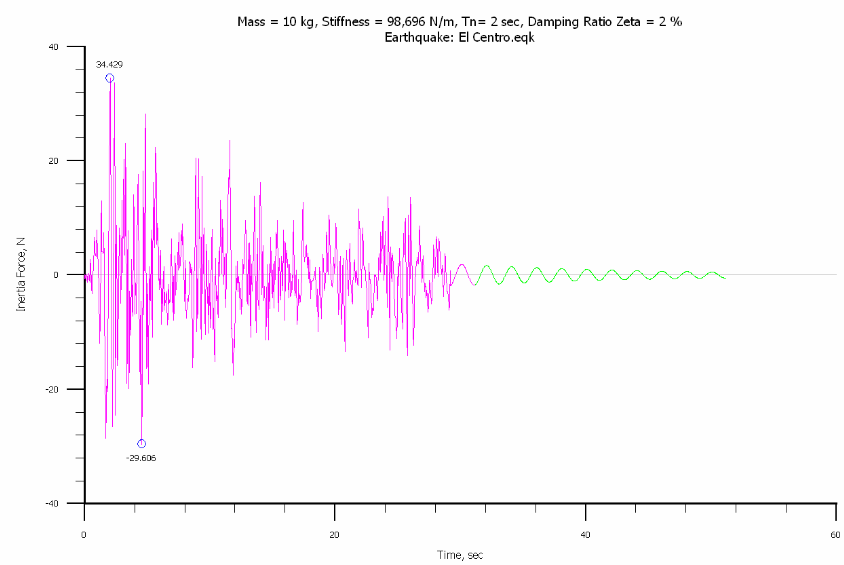
➢ Spring force vs displacement:



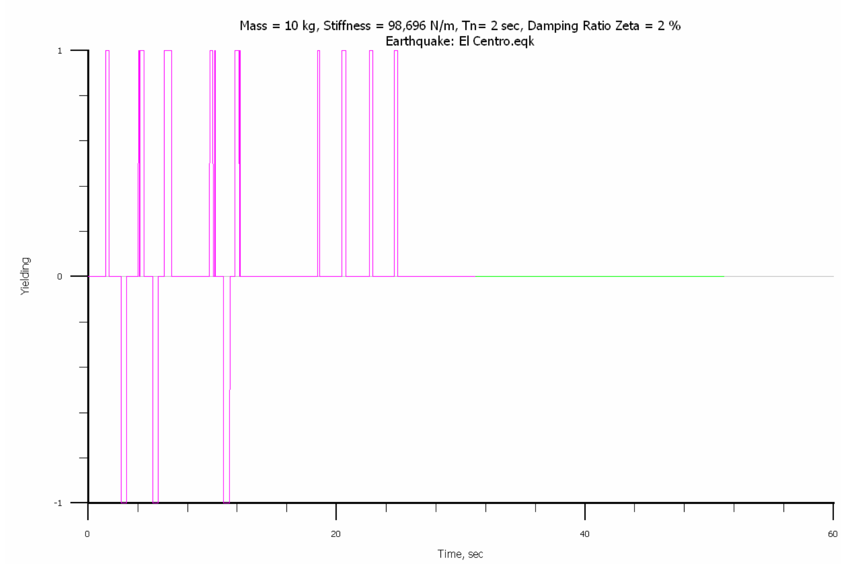
➢ Damping force:



➢ Damping force:

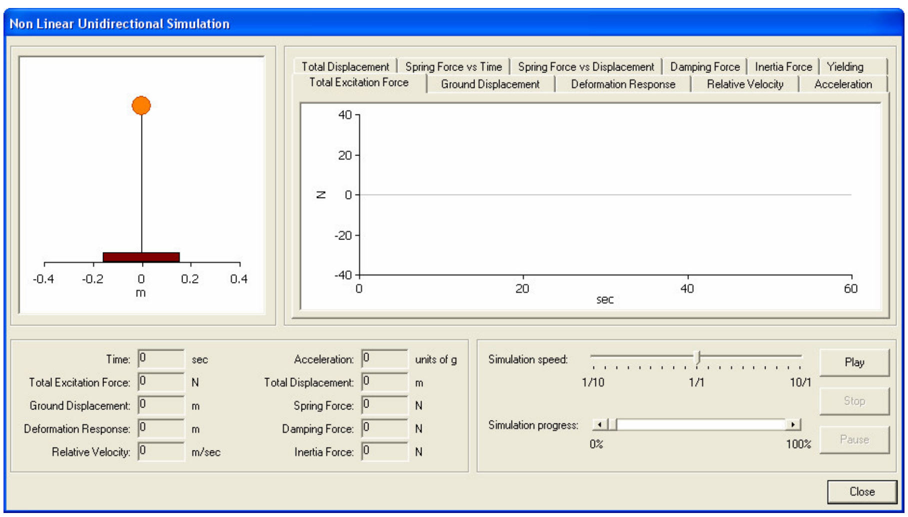


➢ Yielding (1 for positive yielding, 0 for the elastic branch, -1 for negative yielding):



6.6 Simulation

In order to simulate the response of the system in animated form, select *"Single Unidirectional (Bilinear Model) > Response simulation"* from the *"Non Linear Analysis"* menu of the main form. The following form will appear:



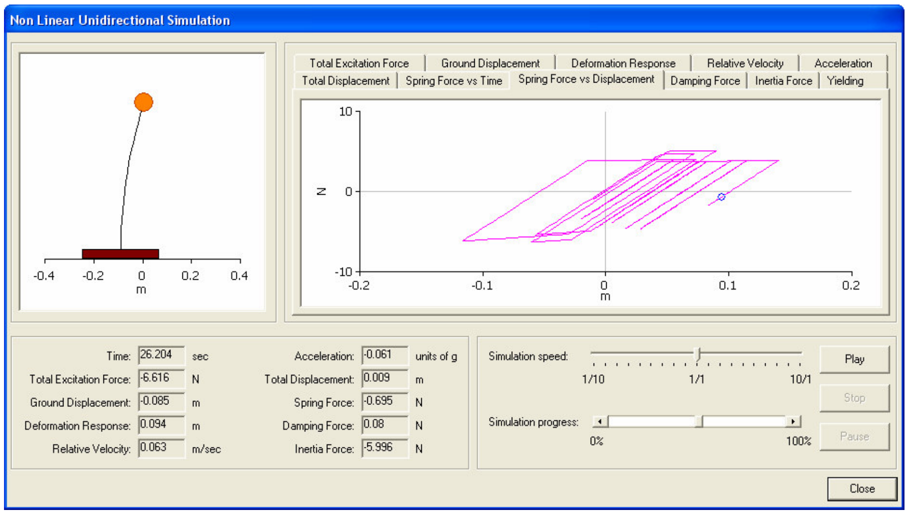
You can simulate the response by clicking on the *"Play"* button of the bottom right frame. In the same frame, you can adjust the speed of the simulation by using the slider and you can jump to a specific point of the simulation by using the horizontal progress bar.

At each time instant all data is displayed in the bottom left frame.

In the top left frame, you can see the main simulation panel which depicts an ideal SDF system: a lumped mass on top of a massless supporting structure (column), which, in turn, is fixed on a rectangular base. This base represents the ground and it moves in case of earthquakes.

Finally, you can pick one of the diagrams available in the top right frame and watch its progress.

During the simulation the form may look like this:



The units in which the results are displayed can be changed by selecting *"Units"* under the *"Options"* menu of the main form.

6.7 Export results

You can export the results in a simple ASCII text file, and use it in a spreadsheet, for example. In order to create this file, select *"Single Unidirectional Model > Export Results"* from the *"Non Linear Analysis"* menu of the main form.

The results are fixed - aligned in columns with a header for each column.

7. Non Linear Analysis – Bouc Wen Model

7.1 General issues

MySpec evaluates the nonlinear response based on a generic Bouc Wen Model.

This hysteresis or memory – dependent model is very popular because of its versality and simplicity; it is a very concise model governed by a single differential equation. This was first introduced by Bouc in 1967 [2]. In 1976, Wen [3], extended the model and demonstrated its versality by producing a variety of hysteretic patterns.

7.2 Calculations

For a SDF system the restoring force can be written as:

$$F(t) = a \cdot \frac{F_y}{u_y} \cdot u(t) + (1-a) \cdot F_y \cdot z(t) \tag{7.2.1}$$

where, F_y is the yield force, u_y is the yield displacement, a is the ratio of post-yield to pre-yield (elastic) stiffness and $z(t)$ is a dimensionless hysteretic parameter obeying a single differential equation:

$$\dot{z}(t) = \frac{1}{u_y}\left[A - \left|z(t)\right|^n \cdot \left(\gamma \cdot sign\left(\dot{u}(t) \cdot z(t)\right) + \beta\right)\right] \cdot \dot{u}(t) \tag{7.2.2}$$

where, A, β, γ, n are dimensionless quantities controlling the shape of the hysteresis loop.

The equation of motion for a SDF system with external viscous damping c is given as:

$$m \cdot \ddot{u}(t) + c \cdot \dot{u}(t) + F(t) = f(t) \tag{7.2.3}$$

where, $u(t)$ is the displacement, $F(t)$ is the restoring force, $f(t)$ is the excitation force. Substituting (7.2.1) into (7.2.3) we obtain:

$$m \cdot \ddot{u}(t) + c \cdot \dot{u}(t) + a \cdot \frac{F_y}{u_y} \cdot u(t) + (1-a) \cdot F_y \cdot z(t) = f(t) \tag{7.2.4}$$

Equations (7.2.2) and (7.2.4) are transformed into a state-space form as follows:

$$\begin{Bmatrix} x_1(t) = u(t) \\ x_2(t) = \dot{u}(t) \\ x_3(t) = z(t) \end{Bmatrix} \tag{7.2.5}$$

$$\begin{Bmatrix} \dot{x}_1(t) \\ \dot{x}_2(t) \\ \dot{x}_3(t) \end{Bmatrix} = \begin{Bmatrix} x_2(t) \\ -\frac{1}{m}\cdot\left[c\cdot x_2(t) + a\cdot\frac{F_y}{u_y}\cdot x_1(t) + (1-a)\cdot F_y\cdot x_3(t) - f(t)\right] \\ \frac{1}{u_y}\cdot\left[\left(A - \left|x_3(t)\right|^n\cdot\left(\gamma\cdot sign\left(x_2(t)\cdot x_3(t)\right)+\beta\right)\right)\cdot x_2(t)\right] \end{Bmatrix} \tag{7.2.6}$$

The above system of three first order non-linear ODEs is solved numerically following Livermore stiff ODE integrator which is based on a "predictor-corrector" method [4].

Extending the above equations for a two degree of freedom system, the equations of motion in two directions x and y are two ODEs of second order in time.

$$\begin{aligned} &\begin{bmatrix} m & 0 \\ 0 & m \end{bmatrix}\cdot\begin{Bmatrix} \ddot{u}_x(t) \\ \ddot{u}_y(t) \end{Bmatrix} + \begin{bmatrix} c_x & 0 \\ 0 & c_y \end{bmatrix}\cdot\begin{Bmatrix} \dot{u}_x(t) \\ \dot{u}_y(t) \end{Bmatrix} + \\ &\begin{bmatrix} a_x\cdot F_{y,x}/u_{y,x} & 0 \\ 0 & a_y\cdot F_{y,y}/u_{y,y} \end{bmatrix}\cdot\begin{Bmatrix} u_x(t) \\ u_y(t) \end{Bmatrix} + \\ &\begin{bmatrix} (1-a_x)\cdot F_{y,x} & 0 \\ 0 & (1-a_y)\cdot F_{y,y} \end{bmatrix}\cdot\begin{Bmatrix} z_x(t) \\ z_y(t) \end{Bmatrix} = \begin{Bmatrix} f_x(t) \\ f_y(t) \end{Bmatrix} \end{aligned} \tag{7.2.7}$$

These equations are coupled through the dimensionless hysteretic variables $z_x(t)$, $z_y(t)$ which are governed by a system of non-linear equations:

$$\begin{aligned} &\begin{Bmatrix} \dot{z}_x(t)\cdot u_{y,x} \\ \dot{z}_y(t)\cdot u_{y,y} \end{Bmatrix} = A\cdot\begin{Bmatrix} \dot{u}_x(t) \\ \dot{u}_y(t) \end{Bmatrix} - \\ &\begin{bmatrix} z_x^{\,2}\cdot\left(\gamma\cdot sign\left(\dot{u}_x(t)\cdot z_x(t)\right)+\beta\right) & z_x\cdot z_y\cdot\left(\gamma\cdot sign\left(\dot{u}_y(t)\cdot z_y(t)\right)+\beta\right) \\ z_x\cdot z_y\cdot\left(\gamma\cdot sign\left(\dot{u}_x(t)\cdot z_x(t)\right)+\beta\right) & z_y^{\,2}\cdot\left(\gamma\cdot sign\left(\dot{u}_y(t)\cdot z_y(t)\right)+\beta\right) \end{bmatrix}\cdot\begin{Bmatrix} \dot{u}_x(t) \\ \dot{u}_y(t) \end{Bmatrix} \end{aligned} \tag{7.2.8}$$

These equations were developed by Park et al. [5]. For the above system of equations, $n = 2$.

Equations (7.2.7) and (7.2.8) are converted in state space form by introducing additional equations as follows:

$$\left\{\begin{matrix} x_1(t) = u_x(t) \\ x_2(t) = \dot{u}_x(t) \\ x_3(t) = z_x(t) \\ x_4(t) = u_y(t) \\ x_5(t) = \dot{u}_y(t) \\ x_6(t) = z_y(t) \end{matrix}\right\} \tag{7.2.9}$$

$$\left\{\begin{matrix} \dot{x}_1(t) \\ \dot{x}_2(t) \\ \dot{x}_3(t) \\ \dot{x}_4(t) \\ \dot{x}_5(t) \\ \dot{x}_6(t) \end{matrix}\right\} = \left\{\begin{matrix} x_2(t) \\ -\dfrac{1}{m}\cdot\left[c_x \cdot x_2(t) + a_x \cdot \dfrac{F_{y,x}}{u_{y,x}} \cdot x_1(t) + (1-a_x)\cdot F_{y,x} \cdot x_3(t) - f_x(t)\right] \\ \dfrac{1}{u_{y,x}}\cdot\begin{bmatrix} A \cdot x_2(t) - x_3^{\,2}(t)\cdot\left(\gamma \cdot sign\left(x_2(t)\cdot x_3(t)\right)+\beta\right)\cdot x_2(t) - \\ x_3(t)\cdot x_6(t)\cdot\left(\gamma \cdot sign\left(x_5(t)\cdot x_6(t)\right)+\beta\right)\cdot x_5(t) \end{bmatrix} \\ x_5(t) \\ -\dfrac{1}{m}\cdot\left[c_y \cdot x_5(t) + a_y \cdot \dfrac{F_{y,y}}{u_{y,y}} \cdot x_4(t) + (1-a_y)\cdot F_{y,y} \cdot x_6(t) - f_y(t)\right] \\ \dfrac{1}{u_{y,y}}\cdot\begin{bmatrix} A \cdot x_5(t) - x_3(t)\cdot x_6(t)\cdot\left(\gamma \cdot sign\left(x_2(t)\cdot x_3(t)\right)+\beta\right)\cdot x_2(t) - \\ x_6^{\,2}(t)\cdot\left(\gamma \cdot sign\left(x_5(t)\cdot x_6(t)\right)+\beta\right)\cdot x_5(t) \end{bmatrix} \end{matrix}\right\}$$

(7.2.10)

The above system of six first order non-linear ODEs is solved numerically following Livermore stiff ODE integrator which is based on a "predictor-corrector" method [4].

7.3 Applications

This model is extremely useful for the investigation of the dynamic behaviour of hysteretic Lead Rubber Bearing (LRB) and Friction Pendulum Systems (FPS); it provides a unified base for the analyses of both types of isolators (Koumousis V. K. [6]).

For a Lead Rubber Bearing (LRB) isolator, the restoring force is given by equation (7.2.1). The behaviour of the mass isolator in two directions is given by equation (7.2.7), (7.2.8).

On the other hand, a Coulomb friction sliding system requires multiple stick – slip conditions that result into a complicated system of equations. However, a modified viscoplasticity model leads to a convenient formulation that describes accurately the behaviour of a sliding system, especially for Teflon – steel interfaces, where the coefficient of friction increases with velocity.

In this case, the friction force is determined as:

$$F(t) = \mu_s \cdot m \cdot g \cdot z(t) \tag{7.3.1}$$

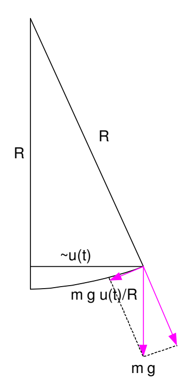
with:

$$\mu_s = f_{\max} - \Delta f \cdot \exp\left(-a \cdot \left|\dot{u}(t)\right|\right) \tag{7.3.2}$$

where, $f_{\max}$ is the coefficient at a large velocity of sliding, Δf is the difference between the coefficient of friction at a large and a very low velocity of sliding and a is a constant. Parameters $f_{\max}$ and Δf are generally dependent on bearing pressure, whereas a is nearly independent of pressure.

The dimensionless quantity $z(t)$ follows again equation (7.2.2) and controls the stick – slip conditions. For slip conditions, $z(t) = \pm z_{\max}$, while for stick conditions (elastic behaviour) $\left|z(t)\right| < z_{\max}$. $z_{\max}$ is dependent on the Bouc Wen parameters A, β, γ and should be equal to ± 1, as described in the next section.

Considering sliding on a spherical surface alters the above behaviour by adding the pendulum effect in the second equation of state space ODE system. For small values of $u(t)/R$ the restoring force of the pendulum effect may be expressed by the term, $-\dfrac{m \cdot g \cdot u(t)}{R}$:



7.4 Behaviour of the hysteretic parameter Z for a SDF system

For an SDF system, the restoring force is given by equation (7.2.1), while the hysteretic parameter $z(t)$ obeys equation (7.2.2). The hysteretic parameter $z(t)$ takes its maximum and minimum value when the system yields in the positive and negative direction respectively. In order to find the extreme values of $z(t)$, we set:

$$\dot{z}(t)=0 \tag{7.4.1}$$

From equation (7.4.2), we get:

$$\left\{\begin{array}{l} \dot{u}(t)=0 \\ or \\ A-\left|z(t)\right|^{n}\cdot\left(\gamma\cdot sign\left(\dot{u}(t)\cdot z(t)\right)+\beta\right)=0 \end{array}\right\} \tag{7.4.2}$$

In the general case, the second equation yields:

$$z_{\max}=\left(\frac{A}{\gamma\cdot sign\left(\dot{u}(t)\cdot z(t)\right)+\beta}\right)^{\frac{1}{n}} \tag{7.4.3}$$

However, when the system yields and $z(t)$ is either maximum or minimum, both the velocity $\dot{u}(t)$ and the hysteretic parameter $z(t)$ share the same sign; for example, while yielding in the positive direction, both $\dot{u}(t)$ and $z(t)$ are positive. When the velocity diminishes and reaches zero, this is the condition that signifies the reversal of the movement towards the negative direction and the end of yielding. Similarly, while yielding in the negative direction, both $\dot{u}(t)$ $z(t)$ are negative. In all cases:

$$sign\left(\dot{u}(t)\cdot z(t)\right)=1 \tag{7.4.4}$$

Equation (7.4.5) then becomes:

$$z_{\max}=\left(\frac{A}{\beta+\gamma}\right)^{\frac{1}{n}} \tag{7.4.5}$$

Note that $\frac{A}{\beta+\gamma}$ should be positive. For an SDF system, the parameters A,β,γ should be chosen in such a way that $z_{\max,\min}=\pm1$. This becomes apparent in an elastoplastic system, where the ratio a of post-yield to pre-yield (elastic) stiffness is zero. Equation (7.2.1) then becomes:

$$F(t) = F_y \cdot z(t) \qquad (7.4.6)$$

It is obvious that, while yielding, the restoring force F should be equal to the yield force F_y, therefore, $z_{\max,\min} = \pm 1$. In particular, $A = 1, \beta = 0.1, \gamma = 0.9$ are suggested by Constantinou et al [7]. The model exhibits greater sensitivity to the relative value of β with respect to γ and vice versa; the shape of the hysteresis loop can be modified by these two parameters.

7.5 Behaviour of the hysteretic parameter Z for a 2DOF system

For a two degree of freedom system, the equations of motion in two directions x and y are given by (7.2.7). The behaviour of the hysteretic parameters is controlled by equations (7.2.8).

Constantinou et al. [7] have shown that when the system yields, equations (7.2.8) have the following solution (provided that $\frac{A}{\beta + \gamma} = 1$):

$$\begin{aligned} z_x(t) &= \cos(\theta) \\ z_y(t) &= \sin(\theta) \end{aligned} \qquad (7.5.1)$$

where θ is the angle of the direction of motion at each time instance:

$$\theta = ArcTan\left(\frac{\dot{u}_y}{\dot{u}_x}\right) \qquad (7.5.2)$$

We can investigate the special case of a fully symmetrical 2DOF system i.e. a 2DOF system with the same properties in both directions, subjected to the same excitation in both directions. Because of the symmetrical equations (7.2.7), (7.2.8), we expect the same results for both directions. In this case, $z_x(t)$ and $z_y(t)$ should have the same values at all times; therefore they should obtain their (common) maximum or minimum values simultaneously. Similarly to the SDF system, the hysteretic parameter $z(t)$ takes its extremes values when the system yields, in either direction. In order to find the extreme values of $z_x(t)$, we set:

$$\dot{z}_x(t) = 0 \tag{7.5.3}$$

From equation (7.2.8), we get:

$$\left\{ \begin{array}{l} \dot{u}_x(t) = 0 \\ or \\ A - z_{x.\max}{}^2 \cdot \left(\gamma \cdot sign\left(\dot{u}_x(t) \cdot z_x(t)\right) + \beta\right) - z_{x.\max} \cdot z_y \cdot \left(\gamma \cdot sign\left(\dot{u}_y(t) \cdot z_y(t)\right) + \beta\right) = 0 \end{array} \right\}$$

(7.5.4)

The general case is given by the second equation. When the system yields and $z_x(t)$ is either maximum or minimum, both the velocity $\dot{u}_x(t)$ and the hysteretic parameter $z_x(t)$ share the same sign; for example, while yielding in the positive direction of x axis, both the velocity $\dot{u}_x(t)$ and the hysteretic parameter $z_x(t)$ are positive. When the velocity diminishes and reaches zero, this is the condition that signifies the reversal of the movement towards the negative direction and the end of yielding. Similarly, while yielding in the negative direction of x axis, both $\dot{u}_x(t)$ and $z_x(t)$ are negative. In all cases:

$$sign\left(\dot{u}_x(t) \cdot z_x(t)\right) = 1 \tag{7.5.5}$$

Because of the symmetry, the same equation is valid for direction y:

$$sign\left(\dot{u}_y(t) \cdot z_y(t)\right) = 1 \tag{7.5.6}$$

Equation (7.5.4) becomes:

$$A - z_{x.\max}{}^2 \cdot (\beta + \gamma) - z_{x.\max} \cdot z_{y.\max} \cdot (\beta + \gamma) = 0 \tag{7.5.7}$$

We expect:

$$z_{x.\max} = z_{y.\max} \tag{7.5.8}$$

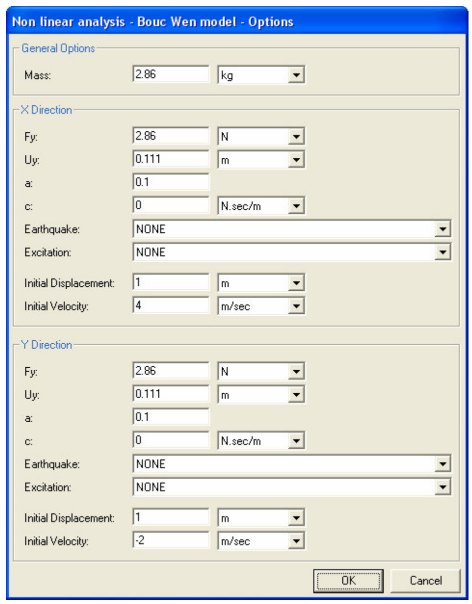
Combining equations (7.5.6) and (7.5.7) we obtain:

$$z_{\max} = \sqrt{\frac{A}{2 \cdot (\beta + \gamma)}} \tag{7.5.9}$$

Equations (7.5.1) provide the same result as equation (7.5.9), since the angle of the direction of motion is 45^0 (due to symmetry) and $\cos\left(45^0\right)=\sin\left(45^0\right)=\frac{1}{\sqrt{2}}$.

7.6 Input data

In order to use the Bouc Wen model, select *"Bouc Wen Analysis > Options"*. The following form will appear:



In the *"General Data"* frame, the following data is required:

- Mass: The mass of the system. Make sure to select the correct units from the drop-down list box.

Both in the *"X direction"* and the *"Y direction"* frame, the following data is required:

- F_y: The yield force of the system. Make sure to select the correct units from the drop-down list box.
- U_y: The yield displacement of the system. Make sure to select the correct units from the drop-down list box.
- a: The ratio of post-yield to pre-yield (elastic) stiffness. Dimensionless.
- c: The viscous damping. Make sure to select the correct units from the drop-down list box.

- Earthquake: In addition to or separately from the excitation, you can select the desired earthquake from the drop-down list box. Note that the *modified* form of the earthquake is used in the calculations.
- Excitation: In addition to or separately from the earthquake, you can select the desired excitation from the drop-down list box. Note that the *modified* form of the excitation is used in the calculations.
- Initial displacement: The initial displacement of the system. . Make sure to select the correct units from the drop-down list box.
- Initial velocity: The initial velocity of the system. Make sure to select the correct units from the drop-down list box.

Note that the usage of an earthquake or an excitation is not compulsory; you can use an initial displacement or velocity instead.

Also note that the rest of Bouc-Wen's parameters i.e. A, β, γ can be modified by selecting *"Analysis"* under the *"Options"* menu of the main form. Refer to *6.2 Analysis Options* for more information. Because of the equations (7.2.10) which are used for the calculations, parameter n is fixed ($n = 2$).

When you have successfully entered all data, click on the *"OK"* button. The response is calculated automatically.

7.7 Results

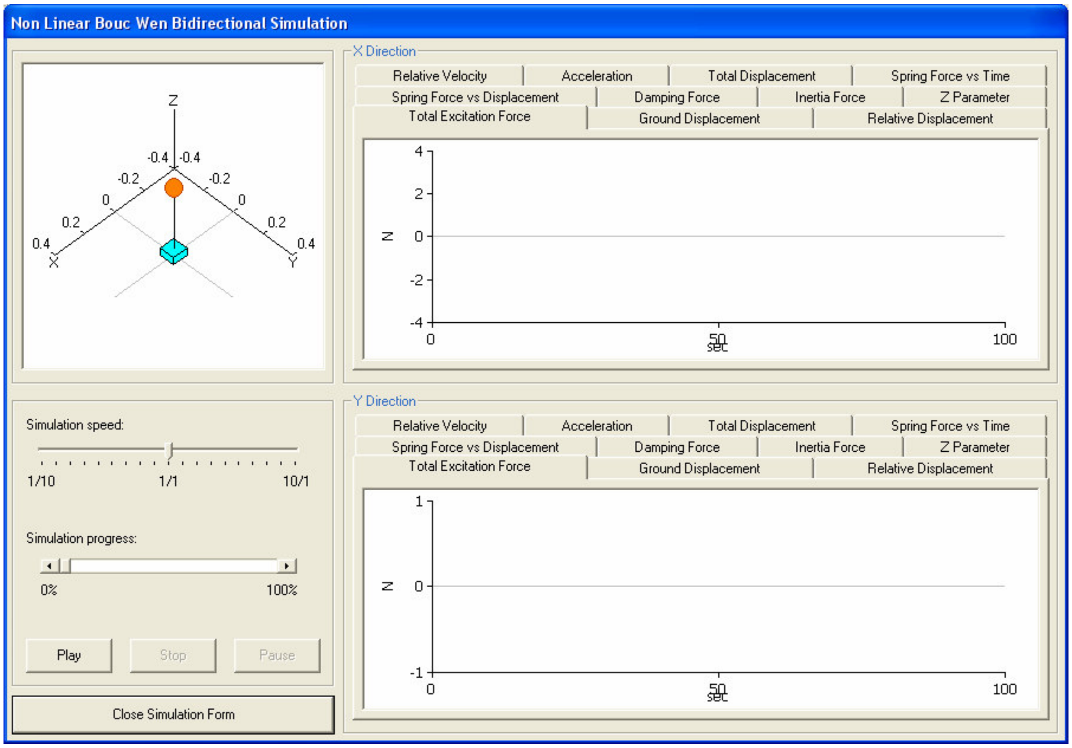
In order to display the results, select the appropriate graph by clicking on *"Bouc Wen Analysis > Plot X (or Y) > …"* menu of the main form.

The units in which the results are displayed can be changed by selecting *"Units"* under the *"Options"* menu of the main form.

Note that the forced vibration period is displayed by magenta colour; the free vibration is displayed by green colour.

7.8 Simulation

In order to simulate the response of the system in animated form, select *"Bouc Wen Analysis > Response simulation"*. The following form will appear:

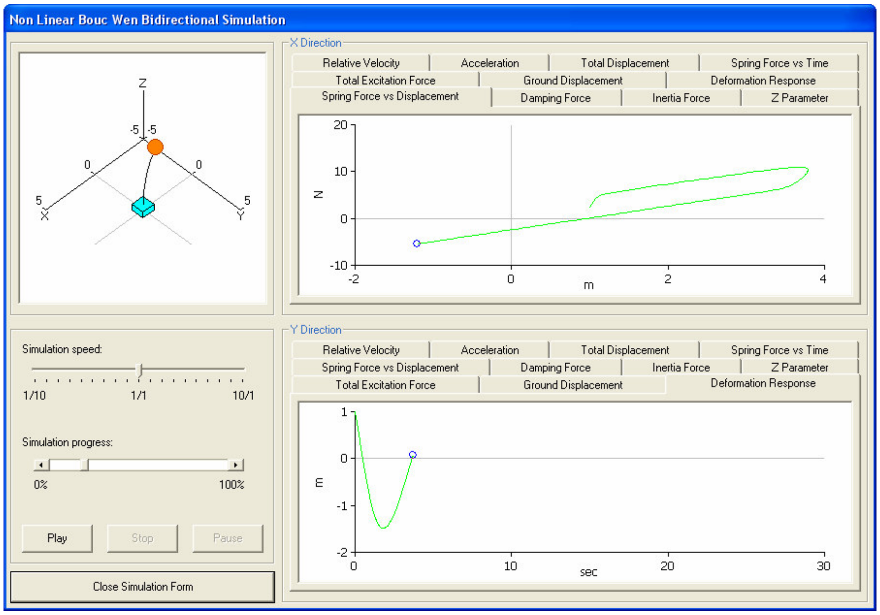


You can simulate the response by clicking on the *"Play"* button of the left frame. In the same frame, you can adjust the speed of the simulation by using the slider and you can jump to a specific point of the simulation by using the horizontal progress bar.

In the top left frame, you can see the main simulation panel which depicts an ideal 2DOF system: a lumped mass on top of a massless supporting structure (column), which, in turn, is fixed on the ground which moves in case of an earthquake.

Finally, you can pick one of the diagrams available in the top (for the X direction) and the bottom (for the Y direction) right frame and watch its progress.

During the simulation the form may look like this:



The units in which the results are displayed can be changed by selecting *"Units"* under the *"Options"* menu of the main form.

7.9 Export results

You can export the results in a simple ASCII text file, and use it in a spreadsheet, for example. In order to create this file, select *"Bouc Wen Analysis > Export Results"*.

The results are fixed - aligned in columns with a header for each column.

7.10 Validation

In order to validate the performance of MySpec in estimating the response of a hysteretic system under known excitation, the following problem is considered:

7.10.1 Problem formulation

Assume that we have a SDF hysteretic oscillator that obeys the Bouc - Wen model excited by a sinusoidal excitation of the form:

$$f(t) = \sin((0.03 \cdot t + 0.2) \cdot t) \quad (7.10.1)$$

We select this kind of excitation because its Power Spectral Density (PSD) resembles to the one caused by an earthquake. The properties of the oscillator are summarized in the following table:

m =	1
c =	0.632455532
ω =	3.162278
ζ =	10%
u_Y =	0.1
F_Y =	1

Based on other researcher's work (Constantinou et al. [7]), the values of the Bouc Wen parameters are set as follows:

A =	1
β =	0.1
γ =	0.9
n =	2

Tsai et al. [8], showed that the latter problem can be solved in an incremental form analytically. Therefore, solving this problem with MySpec and comparing the results with these from the analytical model we are able to validate the results. The method used for analytical modeling of the Bouc - Wen model is shown in the next paragraph.

7.10.2 Analytical Solution of the Bouc - Wen model

If we consider a small displacement increment du, the Bouc - Wen model can be described as follows:

$$\frac{dz(t)}{dt} \cdot u_Y = A \cdot \frac{du(t)}{dt} - |z|^n \left(\gamma \cdot sign\left(\frac{du(t)}{dt} \cdot z(t) \right) + \beta \right) \cdot \frac{du(t)}{dt} \qquad (7.10.2)$$

By virtue of the same sign between $\frac{du(t)}{dt}$ and $du(t)$ the latter equation can be rewritten as:

$$dz(t) \cdot u_Y = A \cdot du(t) - |z|^n \left(\gamma \cdot sign\left(du(t) \cdot z(t) \right) + \beta \right) \cdot du(t) \qquad (7.10.3)$$

One can set:

$$y(t) = \gamma \cdot sign\left(du(t) \cdot z(t)\right) + \beta \tag{7.10.4}$$

We can also express hysteretic parameter $z(t)$ z as a function of the value of the previous time step plus a small increment dz:

$$z = z_0 + dz \tag{7. 10.5}$$

Substitution of equations (7.10.4) and (7.10.5) into equation (7.10.3) results in an n-degree polynomial:

$$| z_0 + dz |^n \cdot y(t) \cdot du(t) + dz \cdot u_Y - A \cdot du(t) = 0 \tag{7. 10.6}$$

For our case, where $n = 2$, one can evaluate the exact solution of dz for each time step from:

$$dz = -\frac{1}{2 \cdot y(t) \cdot du(t)} \cdot \left(u_Y + 2 \cdot z_0 \cdot y(t) \cdot du(t) \pm \sqrt{u_Y{}^2 + 4 \cdot u_Y \cdot z_0 \cdot y(t) \cdot du(t) + 4 \cdot du^2(t) \cdot y(t) \cdot A}\right)$$

(7.10.7)

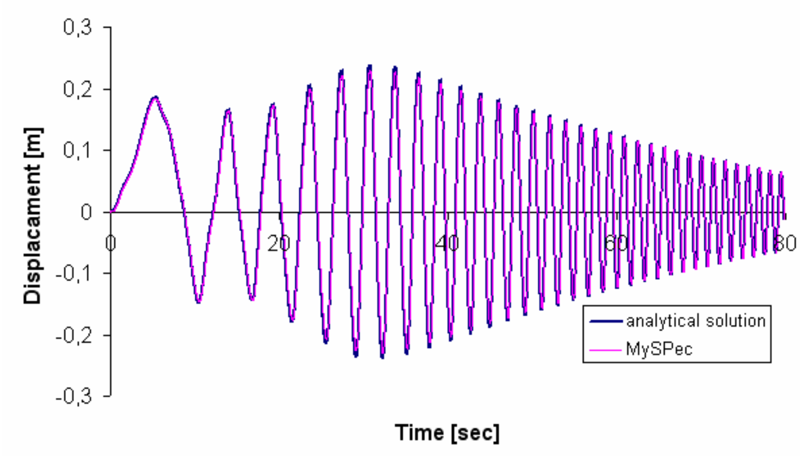
Combining the above equation with the method of linear acceleration, one is able the compute response of a hysteretic oscillator analytically in an incremental form.

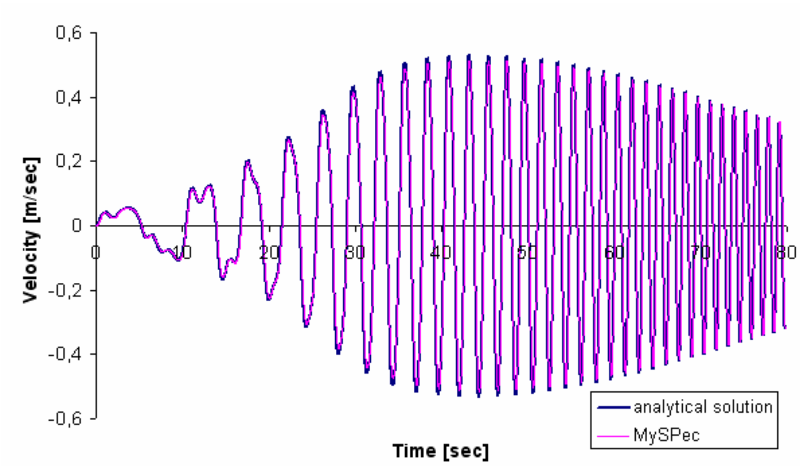
7.10.3 Comparison and results

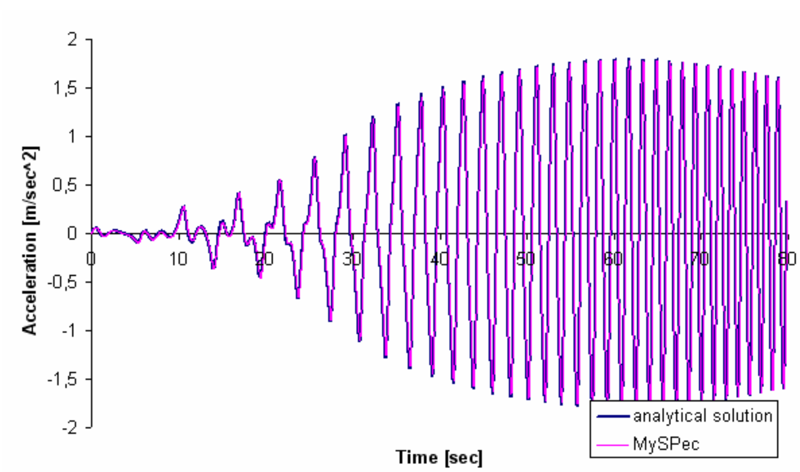
The assumed problem was solved with the proposed analytical method as well as with MySpec. As can be shown from the following figure, the two methods provide almost identical results. The mean squared errors for each parameter are summarized in the following table table 2.

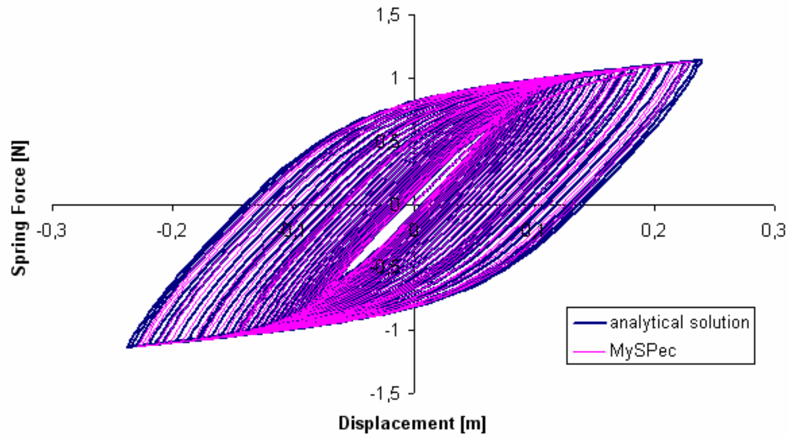
Parameter	Displacement	Velocity	Acceleration	Z
Mean squared error	$2 \cdot 10^{-5}$	10^{-4}	$3 \cdot 10^{-4}$	$3 \cdot 10^{-4}$

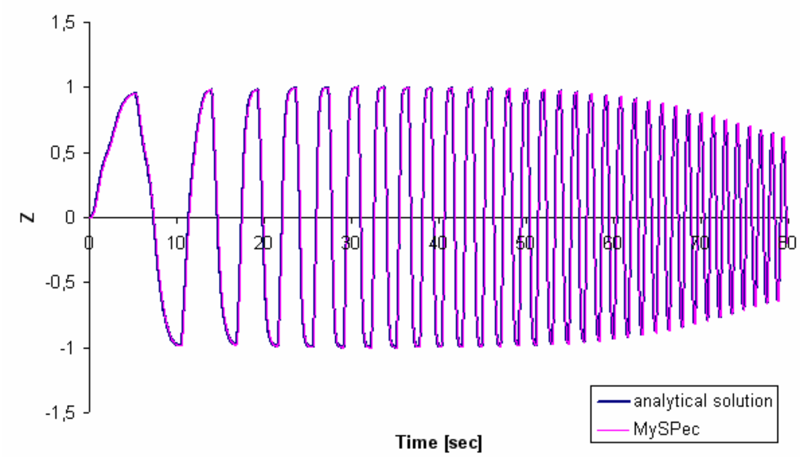
From the above table it becomes obvious that the mean global error of the numerical used by MySpec is of the order 10^{-4}.


0,3
0,2
0,1
0
-0,1
-0,2
-0,3
Displacament [m]
0
20
40
60
80
analytical solution
MySPec
Time [sec]


0,6
0,4
0,2
0
-0,2
-0,4
-0,6
Velocity [m/sec]
0
10
20
30
40
50
60
70
80
analytical solution
MySPec
Time [sec]


2
1,5
1
0,5
0
-0,5
-1
-1,5
-2
Acceleration [m/sec^2]
0
10
20
30
40
50
60
70
80
analytical solution
MySPec
Time [sec]


Spring Force [N]
Displacement [m]
1,5
1
0,5
0
-0,5
-1
-1,5
-0,3
-0,2
-0,1
0
0,1
0,2
0,3
analytical solution
MySPec


Z
Time [sec]
1,5
1
0,5
0
-0,5
-1
-1,5
0
10
20
30
40
50
60
70
80
analytical solution
MySPec

References

1. Anil K. Chopra, Dynamics of Structures, Theory and applications to earthquake engineering, second edition.
2. R. Bouc. Forced vibration of mechanical systems with hysteresis. Abstract, *Proceedings of the Fourth Conference on Non-linear oscillation,* Prague, Czechoslovakia (1967).
3. Y. K. Wen. Method for random vibration of hysteretic systems. *J. Eng. Mech. ASCE* **102**, 249-263 (1976).
4. Alan C. Hindmarsh Scientific Computing. *ODEpack, a Systemized Collection of ODE solvers,* R. S. Stepleman et al. (eds.) North-Holland, Amsterdam, 1983.
5. Park, Y., J., We,, Y. K. and Ang, A.H.S., *Random vibration of hysteretic systems under bidirectional ground motion,* Earthquake engineering and structural dynamics, 14, 4, pp. 543-557, 1986
6. Koumousis V. K., (2001), *Non-Linear Dynamic Behaviour of Base-Isolators*, Proceedings of the 3rd HSTAM Conference, Thessaloniki 19-21 July.
7. Constantinou, M. C., Mokha, A. and Reinhorn, A.M. *Teflon bearings in base isolation II: Modeling*, 1990, J. Struct. Engrg. ASCE, 116(2), 455-474.
8. Tsai, C. S., Tsu-Cheng Chiang, Bo-Jen Chen and Shih-Bin Lin, *An advanced analytical model for high damping rubber bearings*, Earthquake Engrg. Struct. Dyn., 2003, Volume 32, Issue 9, 1373-1387.