

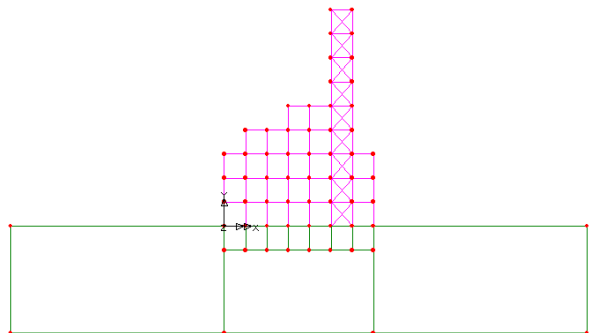
Seismic Analysis of a 2D Frame (Time Domain)

For LUSAS version:	16.0
For software product(s):	All
With product option(s):	IMDPlus
Note: The example exceeds the limits of the LUSAS Teaching and Training Version.	

Description

This example examines the seismic response of a 2D plane frame founded on an elastic medium. The model is an extension of the Seismic Response of a Plane Frame example.

Units used are N, m, kg, s, C throughout.



Objectives

The output requirements of the analysis are:

- ☐ Displacements, velocities and accelerations at the top of the tower
- ☐ Secondary response spectra at the top of the tower
- ☐ Forces and moments in a column at the base of the tower

- Average forces/moments and peak forces/moments from all of the columns at the base of the tower

Keywords

2D, Frame, Seismic, Time Domain, Response, Mass Participation, Interactive Modal Dynamics, Excitation, Eigenvalue, Secondary Response Spectra.

Associated Files



- **IMDPlus 2D Tower.mdl** Model file of the structure.
- **horizontal.prn** and **vertical.prn** define horizontal and vertical earthquake accelerations. These two files contain two earthquake records each; one for the 1940 El Centro earthquake and one for the 1994 Northridge earthquake.

Discussion

The worked example 'Seismic Response of a 2D Frame (Frequency domain)' is revisited in this example. The frame is now founded in a plane strain elastic medium and the response of the structure is evaluated in the time domain using IMDPlus.

The model is comprised of thick beam elements for the concrete columns, beam members, and steel diagonal bracing members, which have pinned end connections. The number of beam elements representing the components of the structure has been increased from 1 per line to 4 per line to provide greater definition of the deformed shapes. In addition to this modification, all support restraints have been removed from the frame and an additional 4.5 m length has been added to the base of each column to allow embedment of the column bases into the elastic medium.

The elastic medium is modelled as a 108m by 20m block which is fully restrained along its base and with cyclic translation constraints assigned to the sides. These constraints provide support to the sides of the elastic medium without the need to apply physical restraints, thus allowing direct and shear behaviours in the elastic medium.

The seismic response analysis is performed in two distinct stages. A natural frequency analysis is performed first. This is used to calculate the first 50 natural modes of vibration of the combined structure and elastic medium. The eigenvalues, frequencies and eigenvectors (mode shapes) are stored and used in the subsequent IMDPlus analysis. Although the natural frequencies are obtained from an eigenvalue analysis any information regarding the magnitudes of deformations or stresses / moments is non-quantitative.

The second phase of the analysis utilises the IMDPlus option which performs enhanced time domain solutions using Interactive Modal Dynamics (IMD). This is an alternative to performing a spectral response analysis and allows the excitation of the structure using acceleration time histories instead of spectral excitation curves.

In the IMDPlus solution, the structure is subjected to a support condition excitation governed by time histories of acceleration in the model global axes. In this example this is assumed to act along the base of the elastic medium in the form of horizontal and vertical accelerations. It should, however, be noted that no deconvolution of the records has been carried out to convert the surface responses recorded for these earthquakes to at-depth acceleration time histories to be input into the analysis. As a consequence, the ground level accelerations in the analysis will not correspond to the measured values. As stated previously, two earthquake records are used in the analysis, the first being the 1940 El Centro earthquake and the second being the 1994 Northridge earthquake. The two earthquake responses are computed during a single analysis. Only the first 10 seconds of each seismic event is analysed in this example.

Modelling

Running LUSAS Modeller

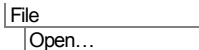
For details of how to run LUSAS Modeller see the heading *Running LUSAS Modeller* in the *Introduction to LUSAS Worked Examples* document.

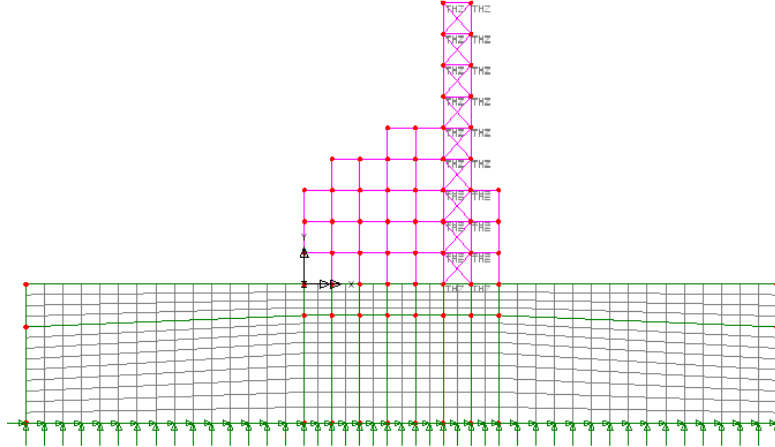
Building and loading the model

- To create the model, open the file **IMDPlus 2D Tower.mdl** located in the \<LUSAS Installation Folder>\Examples\Modeller directory.

Depending upon your access rights it may be necessary to copy this file to another folder before it can be opened, and remove the read-only protection before it can be saved to the new folder. Once opened:

After a short while the following view of the model of the building will be displayed.





File
Save As...

- In the \<LUSAS Installation Folder>\Projects\ folder create a new directory called **IMDPlus 2D Tower**
- Save the model into this new folder as **IMDPlus 2D Tower**. This helps keep all relevant IMDPlus created files separate from other analyses and is good practice.




Note. No static structural loading is required for this analysis because only the dynamic loading is considered during the results processing based on the results from the natural frequency analysis.

The modelling will now be completed by defining the controls necessary to extract the natural frequencies.

Defining Eigenvalue Controls

Eigenvalue controls are defined as properties of the loadcase.

- In the  Treeview expand **Analysis 1** then right-click on **Loadcase 1** and select **Eigenvalue** from the **Controls** menu.

The Eigenvalue dialog will appear.

- Set the **Number of eigenvalues** as **50**
- Ensure the **Shift to be applied** is set as **0**
- Ensure the **Type of eigensolver** is set as **Default**



Note. Eigenvalue normalisation is set to **Mass** by default. This is essential if the eigenvectors are to be used for subsequent IMD analysis.


- Click the **OK** button to finish.

File _____
Save _____



Save the model file.

Running the Analysis




- With the model loaded click the **Solve** button  and the **Solve Now** dialog will be displayed.
- Click the **OK** button to run the analysis.

A LUSAS Datafile will be created from the model information. The LUSAS Solver uses this datafile to perform the analysis.

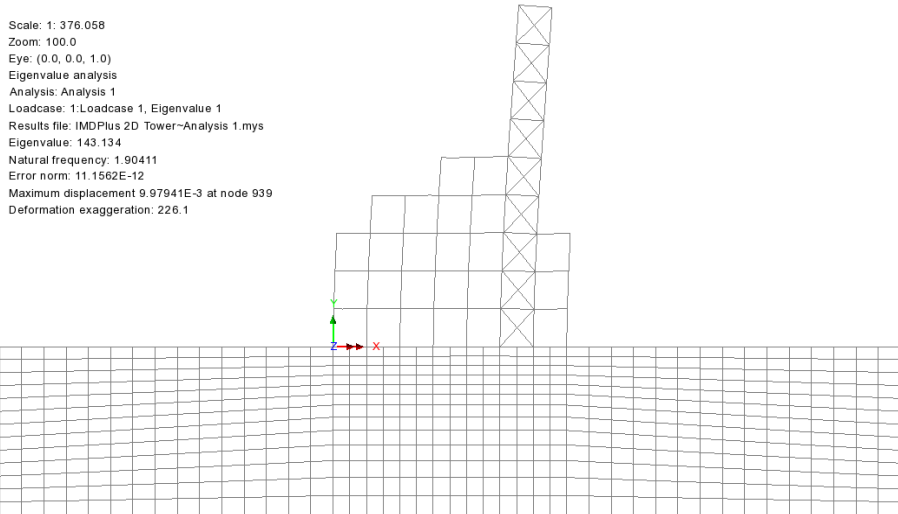
Viewing the Results

If the analysis was run from within LUSAS Modeller the results will be loaded on top of the current model and the loadcase results for each eigenvalue can be seen in the Loadcase panel of the Treeview. Eigenvalue 1 is set to be active by default.

Plotting Mode Shapes

- Turn off the display of the **Mesh**, **Geometry** and **Attributes** layers in the  Treeview.
- With no features selected click the right-hand mouse button in a blank part of the graphics window and select the **Deformed mesh** option to add the deformed mesh layer to the  Treeview. Select the **Visualise** tab and turn off the **Beam end releases** option. Click the **OK** button to accept the remaining default values and display the deformed mesh for Eigenvalue 1.
- At the bottom of the  Treeview select the **Window summary** option and click the **Details...** button. In the Window summary properties dialog set the position to **(50,-10)** and click the **OK** button to return to the graphics window.

Seismic Analysis of a 2D Frame (Time Domain)




Note. The mode shape may be inverted. This is because the sense is arbitrary since during vibration the deformed shape will appear in both directions.




Note. The window summary displays the values of the eigenvalue and the natural frequency and also a value for displacement at a node. It should be noted that the displacement value is non-quantitative and is related to the amount of mass in a particular mode using the mass normalisation technique. Therefore the only items that can be found using a basic eigenvalue analysis are the frequency and the mode shape.

By setting each Eigenvalue to be active the deformed mesh can be seen for all mode shapes.

- In the  Treeview right-click on **Eigenvalue 2** and select the **Set Active** option. The deformed mesh plot for Eigenvalue 2 will be displayed.

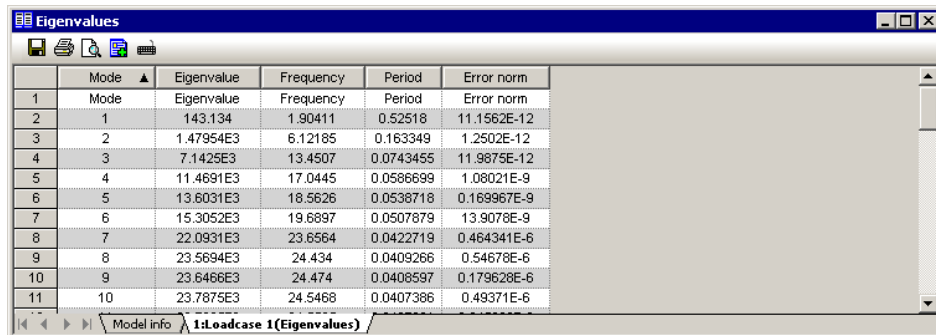
Printing Eigenvalue Results

Eigenvalue results for the whole structure can be displayed in the Text Output window.

- For loadcase 1, select results type **Eigenvalues**. In the available results panel select **Eigenvalues (Frequency)** and click **OK**. The Eigenvalue results will be displayed and a Print Results Wizard entry will be added to the Utilities  treeview.

Utilities
Print Results Wizard...

For inspection only the first 10 modes are shown here.



Mode	Eigenvalue	Frequency	Period	Error norm
1	143.134	1.90411	0.52518	11.1562E-12
2	1.47954E3	6.12185	0.163349	1.2502E-12
3	7.1425E3	13.4507	0.0743455	11.9875E-12
4	11.4691E3	17.0445	0.0586699	1.08021E-9
5	13.6031E3	18.5626	0.0538718	0.169967E-9
6	15.3052E3	19.6897	0.0507879	13.9078E-9
7	22.0931E3	23.6564	0.0422719	0.464341E-6
8	23.5694E3	24.434	0.0409266	0.54678E-6
9	23.6466E3	24.474	0.0408597	0.179628E-6
10	23.7875E3	24.5468	0.0407386	0.49371E-6



Note. The frequency in Hertz can be obtained by dividing the square root of the eigenvalue by 2π , and the period of vibration in seconds is obtained using the reciprocal of frequency ($1/\text{frequency}$). Values of error norm may vary from those shown.



Caution. The system eigenvectors have been normalised (in this case with respect to mass) therefore any derived quantities such as displacement and moment are also normalised and are not true design values.

- Close the print results window by selecting the close button in the top right hand corner of the window. It can be re-displayed by selecting the 'Show results' menu item from context menu of the PRW utility.

Checking the Mass Participation Factor



Note. In order to carry out a successful IMDPlus analysis you should ensure that a significant proportion of the total mass has been accounted for in the analysis. This requires checking that around 90% of the total mass has been achieved in the global directions. If less than 90% has been achieved no further modes need to be included, if and only if, the modes of vibration omitted cannot be excited by the input acceleration time histories or a significant proportion of the structure is restrained by support in these directions and therefore cannot participate in the modes of vibration. The acceptability of the included modes of vibration will vary from analysis to analysis but failure to check that a significant proportion of the total mass has been accounted for may lead to important modes being missed and subsequent errors in the analysis results.

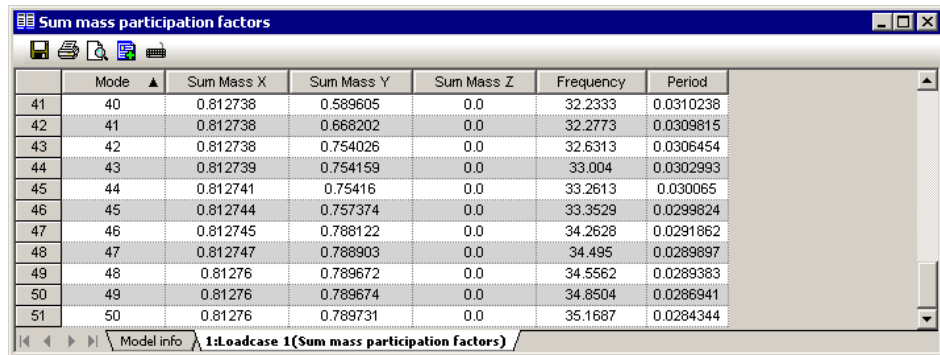
- Select results type **Eigenvalues**. For loadcase 1, in the available results panel deselect **Eigenvalues (Frequency)**, then select **Sum mass participation factors** and click **OK**. The Sum Mass Participation Factors results will be printed to the print results window.

For inspection only modes 40 to 50 will be printed here. It can be seen that the 90% value has not been achieved and is discussed below.

Utilities

Print Results
Wizard...

Seismic Analysis of a 2D Frame (Time Domain)



	Mode	Sum Mass X	Sum Mass Y	Sum Mass Z	Frequency	Period
41	40	0.812738	0.589605	0.0	32.2333	0.0310238
42	41	0.812738	0.668202	0.0	32.2773	0.0309815
43	42	0.812738	0.754026	0.0	32.6313	0.0306454
44	43	0.812739	0.754159	0.0	33.004	0.0302993
45	44	0.812741	0.75416	0.0	33.2613	0.030065
46	45	0.812744	0.757374	0.0	33.3529	0.0299824
47	46	0.812745	0.788122	0.0	34.2628	0.0291862
48	47	0.812747	0.788903	0.0	34.495	0.0289897
49	48	0.81276	0.789672	0.0	34.5562	0.0289383
50	49	0.81276	0.789674	0.0	34.8504	0.0286941
51	50	0.81276	0.789731	0.0	35.1687	0.0284344

Model info 1: loadcase 1(Sum mass participation factors)



Note. In this analysis, a significant proportion of mass is restrained along the base of the elastic medium. For the 50 modes of vibration, 81.27% and 78.97% of the total mass is achieved in the horizontal and vertical directions. In order to achieve 90% total mass in both the horizontal and vertical directions 160 modes of vibration with an upper frequency equivalent to 97.2 Hz would actually be required. However, for the purposes of this example, the number of modes of vibration has been restricted to 50.



- Close the print results window. Use the maximise button to increase the size of the graphics window.

Seismic Response Analysis

Seismic response calculations are performed using the IMDPlus (Interactive Modal Dynamics) facility. This involves defining the excitation and specifying the results required. Initially, in this example, the response of the node at the top-right corner of the tower will be investigated.



Note. With a seismic response analysis, additional damping information may also be set. Unlike results from a natural frequency analysis, the output values obtained from a seismic response are design values.

- Turn off the display of the **Deformed Mesh** layer in the  Treeview.
- In the  Treeview double-click on the **Mesh** layer name. Select the **Visualise** tab and turn off the **Beam end releases** option. Click the **OK** button to accept all other settings. This will also turn on the layer in the graphics window.

- Zoom-in to the top of the tower and select only the top-right node.

The main tower members have been meshed with 4 line mesh divisions so each line will have 3 nodes along its length in addition to one at either end.

Select this node

Scale: 1: 424.361

Zoom: 295.718

Eye: (0.0, 0.0, 1.0)

Eigenvalue analysis

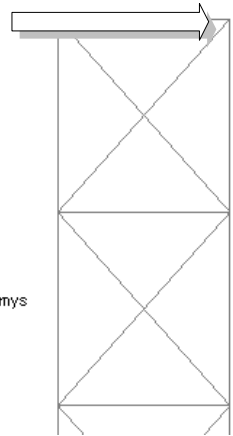
Loadcase: 1:Eigenvalue 2

Results file: IMDPlus 2D Tower~Analysis 1 (Structural).mys

Eigenvalue: 1.47954E3

Natural frequency: 6.12185



Error norm: 0.402905E-6

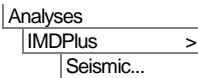


Defining the Seismic Parameters



Note. To simplify the use of IMDPlus (and provide an alternative to picking IMDPlus menu selections from the Utilities menu) for the rest of the example it is recommended that the IMDPlus toolbar is enabled in LUSAS Modeller. To do this select the **View > Toolbars...** menu command, select the **Toolbars** tab, enable the **IMDPlus** option in the list and click **Close**.

- If the IMDPlus toolbar has been enabled then click on the  Seismic analysis button to enable the other relevant toolbar buttons (the seismic tool button) shortcuts to the IMDPlus dialogs.
- Open the IMDPlus Seismic Analysis Control dialog using the menu item, or click on the  button in the toolbar.



The input for the seismic analysis is defined on the IMDPlus Seismic Analysis Control dialog. The input consists of the earthquake acceleration records, included modes and damping along with interpolation options.

IMDPlus Seismic Analysis Control

Earthquake input

X-direction earthquake record(s) C:\LUSAS151\Examples\Modeller\horizontal Browse...

Y-direction earthquake record(s) C:\LUSAS151\Examples\Modeller\vertical Browse...

Z-direction earthquake record(s) Browse...

Direction Factors X 1.0 Y 0.66667 Z

Mode/damping control

Included modes: All Modify...

☒ Default damping 5.0 % Modify...

Total participating mass

X Direction 81.27 %

Y Direction 78.97 %

Z Direction 0.00 %

Frequency interpolation

Advanced solution options Modify...

☒ All frequencies included ☐ Linear interpolation of records

☐ Limit frequencies to Nyquist frequency ☐ Cubic spline interpolation of records

Time step

< Back Next > Cancel Help

- In the Earthquake Input section, for the **X-Direction Earthquake Record(s)** browse to the \<Lusas Installation Folder>\Examples\Modeller directory and select **horizontal.prn**
- For the **Y-Direction Earthquake Record(s)** browse to the \<Lusas Installation Folder>\Examples\Modeller directory and select **vertical.prn**
- Ensure the **Direction Factors** in the **X** and **Y** directions are set to **1** and **0.66667**
- In the Mode/Damping Control section, ensure **Included modes** is set to **All**. If this is not the case, click on the **Modify...** button and turn the **All modes** option on.
- Ensure **Default damping (5%)** is selected. If a different damping is displayed, click on the **Modify...** button and set the **Default damping** to **5**
- In the Frequency Interpolation section, click on the **Modify...** button to change the advanced solution options. Click the **Defaults** button to set the default options and click the **OK** button.
- Ensure the **All frequencies included** option is selected.
- Click the **Next** button to proceed to the IMDPlus Output Control dialog. When prompted that less than 90% total mass participation has been detected choose **Yes** since we are aware of this (see Checking the Mass Participation Factor).

The IMDPlus Output Control dialog will then be displayed.



Notes relating to the IMDPlus Seismic Analysis Control dialog

On the Seismic Analysis Control dialog the **Browse** buttons can be used instead of entering the directional earthquake file names. As each file is loaded, the contents are checked to ensure that they are in the correct format.

Under the **Modify** dialog for the modes, individual modes can be included or excluded from the seismic analysis and total mass participations for the excitation directions are reported.

Under the **Modify** dialog for the damping, control over the inclusion of over-damped modes can be adjusted if viscous modal damping has been included in the original eigenvalue analysis. To include viscous modal damping, Rayleigh damping parameters must be defined in the material properties assigned to the model or a separate damping attribute defined and assigned to the model. This allows direct mirroring of step-by-step dynamic solutions and the inclusion of frequency dependent damping.

Under the **Modify** dialog for the advanced solution options, the time integration scheme used in the IMDPlus analysis can be selected. Either a Hilber Hughes Taylor (HHT) scheme or Duhamel's integral can be utilised.

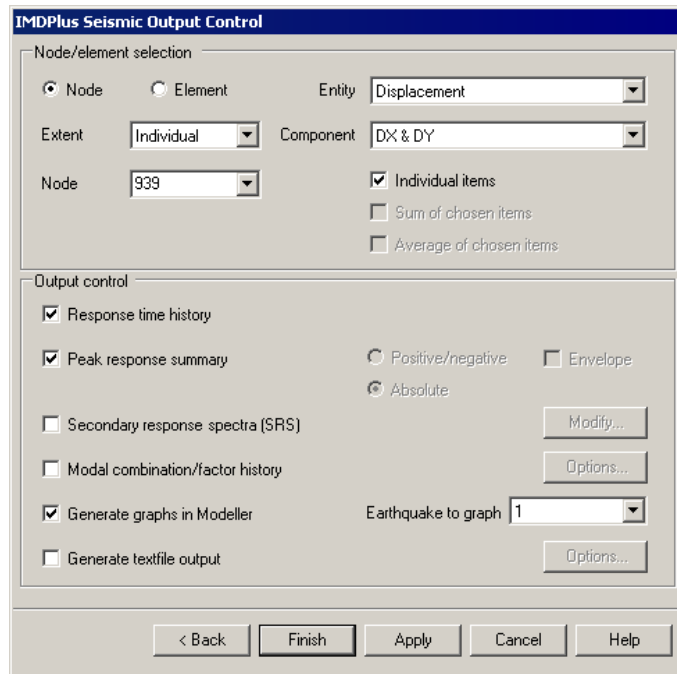
Four methods of interpolation are available in IMDPlus. The default All frequencies included will solve the seismic analysis using all of the frequencies included in the eigenvalue analysis. Limit frequencies to Nyquist frequency will limit the included frequencies to the Nyquist frequency which is governed by the time step of the earthquake records and defined as $1/2\delta t$. Linear interpolation of records allows the specification of an alternative time step for the analysis where the earthquake records are linearly interpolated to this time step. Cubic spline interpolation of records allows the specification of an alternative time step for the analysis where the earthquake records are interpolated with a cubic spline to this time step. This last form of interpolation should generally not be used for measured earthquake records as it can smooth the peak responses but is often more appropriate for synthesised earthquake records.

Displacement Plots

The displacement of the top of the tower will be investigated initially. Enter the following information into the IMDPlus Output Control dialog.

- Select **Node** and select **Extent** as **Individual**
- Choose **Node** number **939** from the drop-down list (or your node number if different). This is the node at the top-right of the tower.
- Select results entity **Displacement** and component **DX & DY**
- Ensure **Individual items** is selected. **Sum of chosen items** and **Average of chosen items** will not be available as an individual node is being processed.
- Ensure **Response time history** is selected.
- Select **Peak response summary**
- Ensure **Generate graphs in Modeller** is selected and specify the **Earthquake to graph** as **1**

The dialog should look like this:

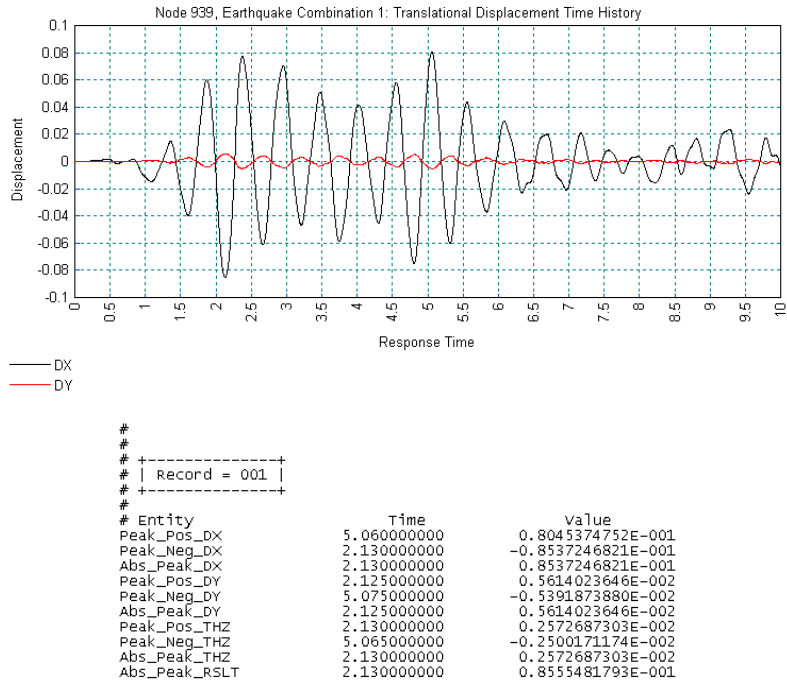


The image shows the 'IMDPlus Seismic Output Control' dialog box. It is divided into two main sections: 'Node/element selection' and 'Output control'. In the 'Node/element selection' section, 'Node' is selected with a radio button, 'Extent' is set to 'Individual', 'Node' is set to '939', 'Entity' is 'Displacement', and 'Component' is 'DX & DY'. The 'Individual items' checkbox is checked, while 'Sum of chosen items' and 'Average of chosen items' are unchecked. In the 'Output control' section, 'Response time history' and 'Peak response summary' are checked. 'Positive/negative' and 'Envelope' are unchecked, while 'Absolute' is selected. 'Secondary response spectra (SRS)' and 'Modal combination/factor history' are unchecked. 'Generate graphs in Modeller' is checked, and 'Earthquake to graph' is set to '1'. 'Generate textfile output' is unchecked. There are 'Modify...' and 'Options...' buttons next to 'Secondary response spectra (SRS)', 'Modal combination/factor history', and 'Generate textfile output'. At the bottom, there are buttons for '< Back', 'Finish', 'Apply', 'Cancel', and 'Help'.

- Click the **Apply** button to proceed.

An IMDPlus analysis will run and after a short while the horizontal (DX) and vertical (DY) displacements for the point at the top of the tower are displayed on the graph for the first 10 seconds of the 1940 El Centro earthquake. (Earthquake 1)


Peak displacements are also output to Notepad and indicate absolute peak displacements of 0.085 m in the X-direction, 0.006 m in the Y-direction, a peak rotation of 0.003 radians and an absolute peak resultant displacement of 0.086 m (Record 1).



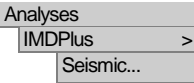
Note. Displacements are relative to the supports.

- Close the Notepad application and graph window.

If the IMDPlus toolbar has been enabled,

- Click on the  button in the toolbar to open the IMDPlus Output Control dialog

If the IMDPlus toolbar is not enabled,



- Open the IMDPlus Seismic Analysis Control dialog through the menu and click **Next >** to keep the existing analysis control settings and open the IMDPlus Output Control dialog.

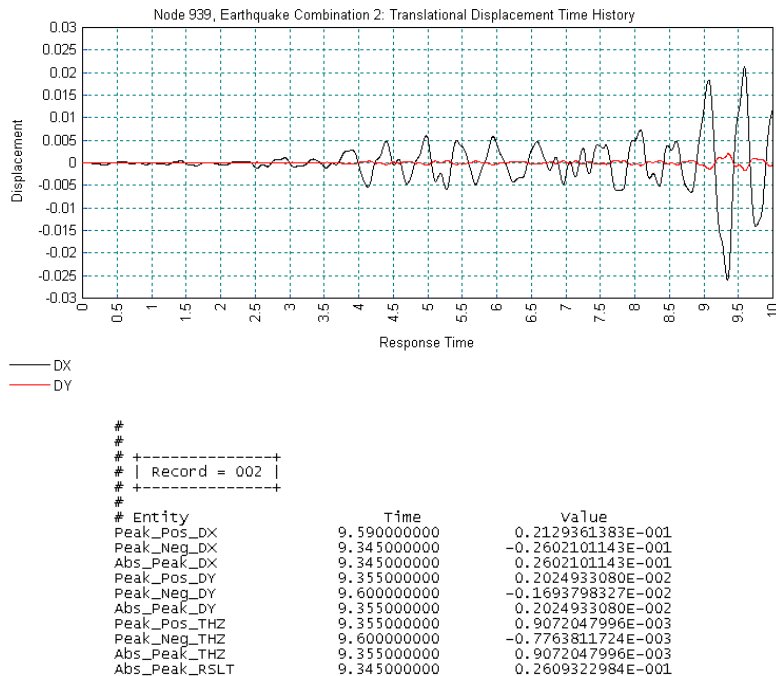
Seismic Analysis of a 2D Frame (Time Domain)

In the IMDPlus analysis we have analysed two earthquake combinations at the same time. We are therefore also interested in the displacement response of the top of the tower for the second earthquake.

Enter the following information into the IMDPlus Output Control dialog:


- Select earthquake **2** in the **Earthquake to graph** list box.
- Click the **Apply** button to proceed.

The horizontal (DX) and vertical (DY) displacements for the point at the top of the tower are displayed on the graph for the first 10 seconds of the 1994 Northridge earthquake. Peak displacements are also output to Notepad and indicate absolute peak displacements of 0.026 m in the X-direction, 0.002 m in the Y-direction, a peak rotation of 0.001 radians and an absolute peak resultant displacement of 0.026 m (Record 2).

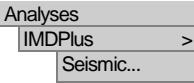


- Close the Notepad application and graph window.

If the IMDPlus toolbar has been enabled,

- Click on the  button in the toolbar to open the IMDPlus Output Control dialog

If the IMDPlus toolbar is not enabled,



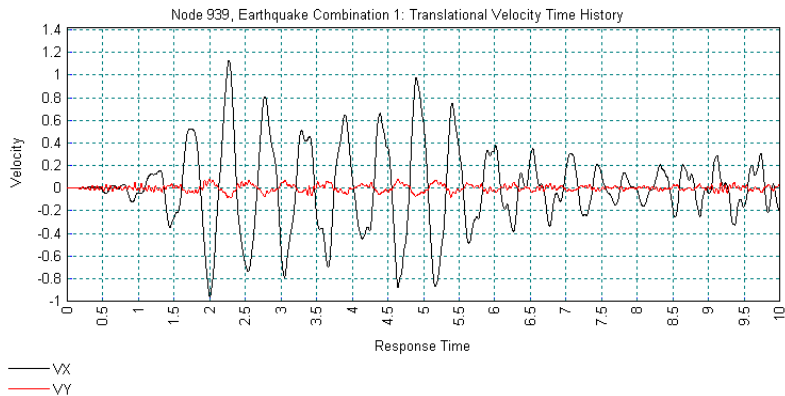
- Open the IMDPlus Seismic Analysis Control dialog through the menu and click **Next >** to keep the existing analysis control settings and open the IMDPlus Output Control dialog.

Velocity Plots

In addition to the displacement response of the top of the tower we are also interested in the velocities experienced by the tower. Enter the following information into the output control dialog.

- Select **Velocity** results of **VX & VY**
- Select earthquake **1** in the **Earthquake to graph** list box
- Click **Apply** to proceed

The horizontal (VX) and vertical (VY) velocities of the top of the tower are displayed on the graph for the first 10 seconds of the 1940 El Centro earthquake. Peak velocities are also output to Notepad and indicate absolute peak velocities of 1.13 m/s in the X-direction, 0.085 m/s in the Y-direction, a peak rotational velocity of 0.039 rad/s and an absolute peak resultant velocity of 1.13 m/s (Record 1).




#		
#		
#	-----	
#	Record = 001	
#	-----	
#		
#	Entity	Time Value
	Peak_Pos_VX	2.265000000 1.130035452
	Peak_Neg_VX	2.005000000 -0.9632754105
	Abs_Peak_VX	2.265000000 1.130035452
	Peak_Pos_VY	1.995000000 0.8042521467E-001
	Peak_Neg_VY	2.295000000 -0.8502032703E-001
	Abs_Peak_VY	2.295000000 0.8502032703E-001
	Peak_Pos_RVZ	2.020000000 0.3271606639E-001
	Peak_Neg_RVZ	2.270000000 -0.3916539671E-001
	Abs_Peak_RVZ	2.270000000 0.3916539671E-001
	Abs_Peak_RSLT	2.265000000 1.132369132



Note. Velocities are relative to the supports.

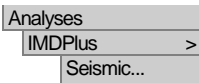
- Close the Notepad application and graph window.

If the IMDPlus toolbar has been enabled,

- Click on the  button in the toolbar to open the IMDPlus Output Control dialog

If the IMDPlus toolbar is not enabled,

- Open the IMDPlus Seismic Analysis Control dialog through the menu and click **Next >** to keep the existing analysis control settings and open the IMDPlus Output Control dialog.

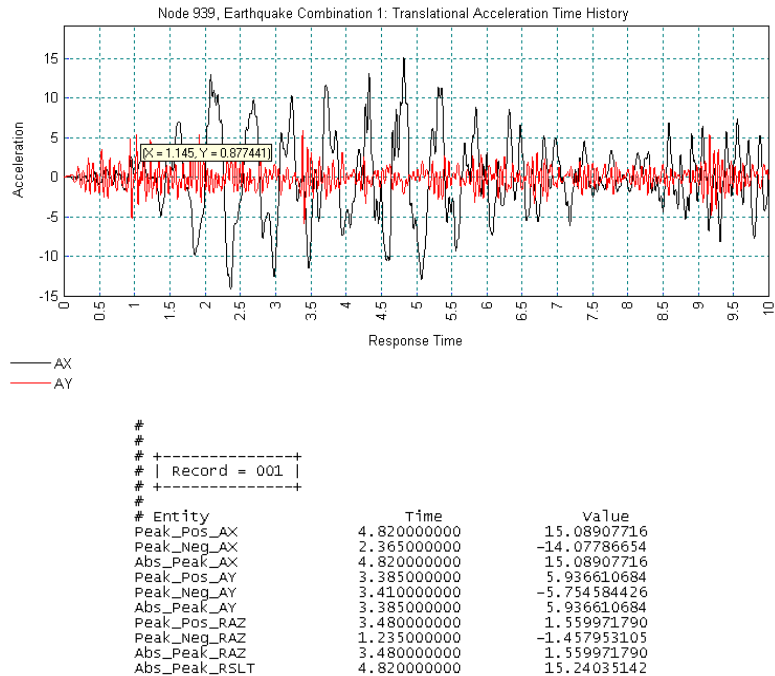


Acceleration and Secondary Response Spectra Plots

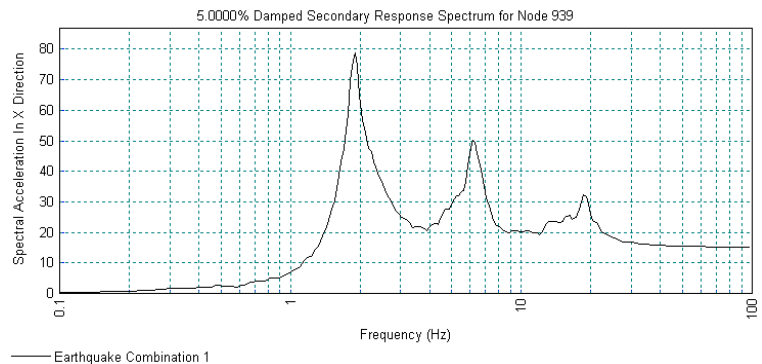
The acceleration response of the top of the tower is now required. Enter the following information into the output control dialog.

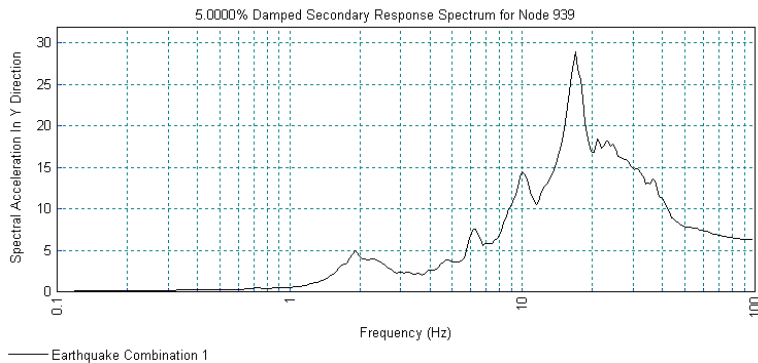
- Select **Acceleration** results of **AX & AY**
- Select **Secondary Response Spectra (SRS)**
- Click on the **Modify...** button to bring up the SRS Control dialog.
- On the SRS Control dialog set the **Minimum damping** and **Maximum damping** to **5.0** and the **Damping increment** to **0.0**
- Click the **OK** button and **Yes** to confirm that only a single damping ratio is to be used
- Click the **Apply** button to proceed.
- When the IMDPlus analysis is finished select **No** when asked whether to show average, maximum and minimum SRS envelopes.

The horizontal (AX) and vertical (AY) accelerations of the top of the tower are displayed on the graph for the first 10 seconds of the 1940 El Centro earthquake. Peak accelerations are also output to Notepad and indicate absolute peak accelerations of 15.089 m/s^2 in the X-direction, 5.937 m/s^2 in the Y-direction, a peak rotational acceleration of 1.56 rad/s^2 and an absolute peak acceleration of 15.24 m/s^2 (Record 1).



Secondary Response Spectra are also generated for the horizontal and vertical directions in two additional graphs. These graphs represent the peak acceleration responses of a range of single degree of freedom oscillators with known natural frequencies and damping should they be attached to the node at the top of the tower.





- Close the Notepad application and graph windows.

Displaying Individual Element Results

In the previous section the displacement, velocity and acceleration response of the top of the tower were investigated. Now, forces and moments in one of the columns of the structure at ground level will be evaluated.



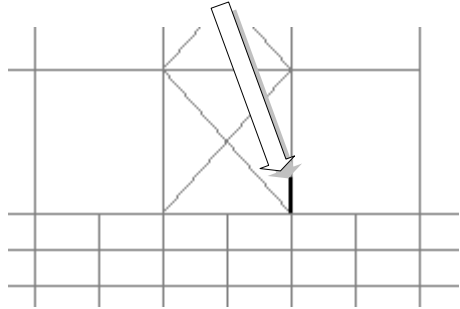
In the LUSAS graphics window re-size the display to show the whole tower.




Zoom-in to the base of the tower and select the element immediately above ground level in the second column from the right.

The main tower members have been meshed with 4 line mesh divisions so each line will have 4 elements along its length.

Select this element



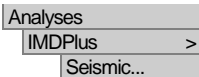
- Click on the  button in the toolbar to open the IMDPlus Output Control dialog

If the IMDPlus toolbar is not enabled,

- Open the IMDPlus Seismic Analysis Control dialog through the menu and click **Next >** to keep the existing analysis control settings and open the IMDPlus Output Control dialog.

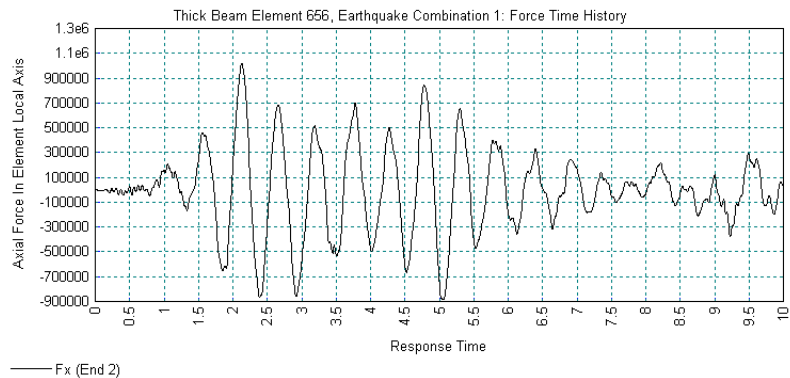
Enter the following information into the IMDPlus Output Control dialog:


- Select **Element** and ensure **Extent** remains selected as **Individual**



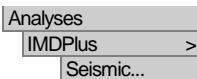
- Choose **Element** number **656** from the drop-down list (or your element number if different).
- Select element **End** number **2** which corresponds to the ground level. (Note that line directions can be viewed to ascertain which is end 1 or end 2 of a line).
- Select **Force/Moment - Thick 2D Beam** results of **Fx**
- Ensure **Individual items** remains selected. **Sum of chosen items** and **Average of chosen items** will not be available as an individual element is being processed.
- Deselect **Peak response summary**
- Click the **Apply** button to proceed.

The time history of the axial force (in N) in the column is displayed on the graph for the first 10 seconds of the 1940 El Centro earthquake. Similar graphs can also be generated for the shear force and bending moment by selecting Fy (in N) and Mz (in Nm) from the output control dialog.



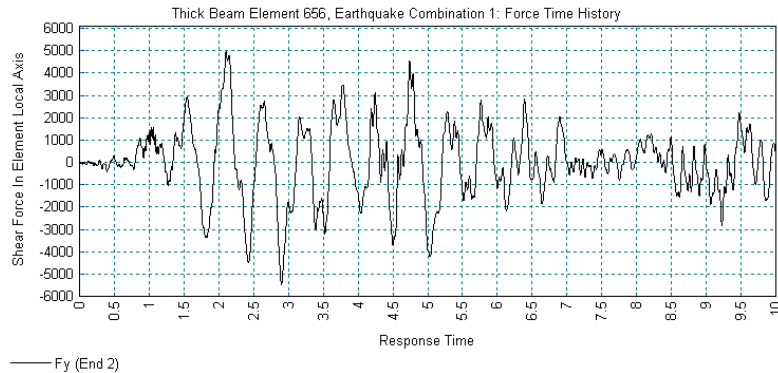
- Close the graph window.
- Click on the  button in the toolbar to open the IMDPlus Output Control dialog


If the IMDPlus toolbar is not enabled,



- Open the IMDPlus Seismic Analysis Control dialog through the menu and click **Next >** to keep the existing analysis control settings and open the IMDPlus Output Control dialog.
- Select **Force/Moment - Thick 2D Beam** results of **Fy**
- Click the **Apply** button to proceed.

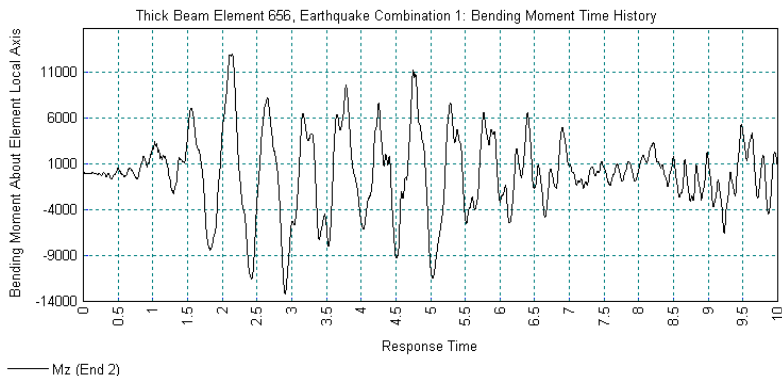
Seismic Analysis of a 2D Frame (Time Domain)



- Close the graph window.
- Click on the  button in the toolbar to open the IMDPlus Output Control dialog

If the IMDPlus toolbar is not enabled,

- Open the IMDPlus Seismic Analysis Control dialog through the menu and click **Next >** to keep the existing analysis control settings and open the IMDPlus Output Control dialog.
- Select **Force/Moment - Thick 2D Beam** results of **Mz**
- Click the **Finish** button to finish output processing of the individual element results.
- Click **Yes** when asked whether to free up disk space by deleting the temporary files created by IMDPlus.




- Close the graph window.

Displaying Results for a Selected Set of Elements

The example analyses presented in the previous sections have all utilised individual nodes or elements of the structure. In this section a set of beam elements will be selected and analysed in order to evaluate the average forces/moments and peak forces/moments from the bases of all the columns of the tower.



In the LUSAS graphics window re-size the display to show the whole tower.

- With no features selected click the right-hand mouse button in a blank part of the graphics window and select the **Labels** option to add the labels to the  Treeview.
- On the Properties dialog select the **Element > Name** check box and the **Label selected items only** check box.
- Click the **Font...** button and select Font: **Arial**, Font style: **Bold**, Size: **20** and click **OK**.
- Click **OK** to return to the graphics window.



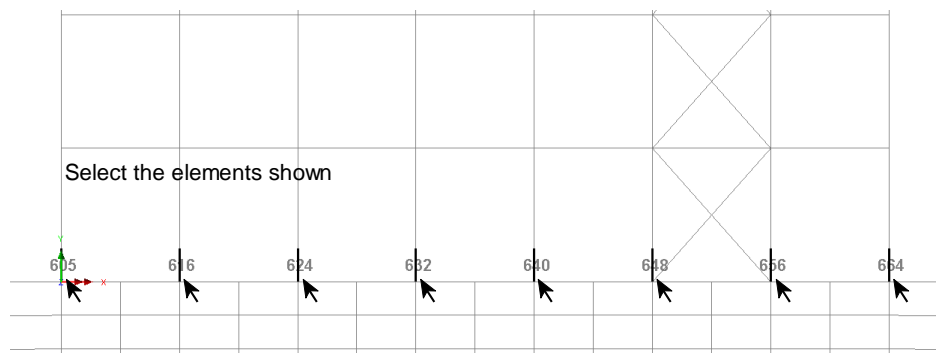
Zoom-in to the base of the frame and select the beam elements immediately above ground level for all of the columns in the structure, as shown in the following figure (your element numbers may be different from those shown in the diagram).



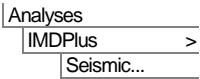
Note. Use the **Ctrl** or **Alt** keys to select multiple elements in the model.




Note. Only elements of the same type, for example, thick beam elements, continuum elements or thick shell elements can be used in a set of IMDPlus elements. However, the element set may contain elements with different numbers of Gauss points or nodes.



Seismic Analysis of a 2D Frame (Time Domain)



- Open the IMDPlus Seismic Analysis Control dialog through the menu or click on the  button in the toolbar
- On the IMDPlus Seismic Analysis Control dialog click the **Next** button to accept the previously defined values. When prompted about the total mass participation click the **Yes** button to continue.

Enter the following information into the IMDPlus Output Control dialog.

- Select **Element** and change **Extent** to **Selected** using the drop-down list. This chooses the eight beam elements at the base of the frame for processing. These beams were selected in the preceding step.



Note. As **Extent** has been set to **Selected** the drop-down lists **Element** and **End** are not available for selection. In addition the element **End** is automatically set to **All** as multiple elements are going to be processed in a single analysis.

- Select results entity **Force/Moment – Thick 2D Beam** of component **Fx**
- Deselect **Individual items** and select **Average of chosen items** as the averaged results for the eight beam elements are going to be investigated.
- Ensure **Response time history** is selected
- Select **Peak response summary** and select **Envelope**
- Ensure **Generate graphs in Modeller** is selected and **Earthquake to graph** is set to **1**

The dialog should look like this:

- Click the **Apply** button to proceed. Click **Yes** when asked whether you want to process all of the selected elements.



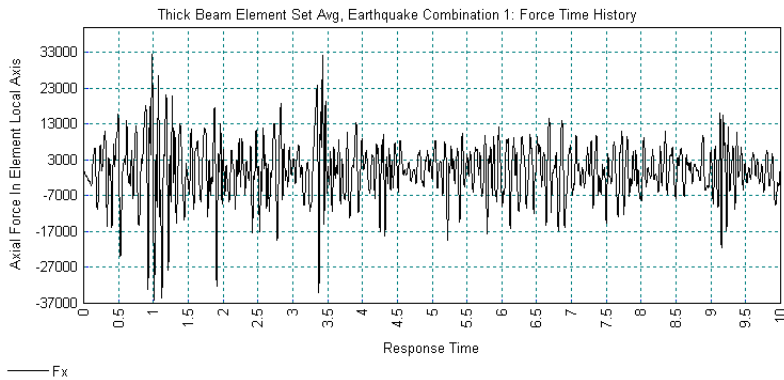
Note. Any combination of the options **Individual items**, **Sum of chosen items** and **Average of chosen items** can be used in an IMDPlus analysis, although using the **Individual items** option when **Extent** is set to **Selected** or **All** (nodes or elements) may produce a large number of graphs, depending on the number of nodes or elements chosen for processing.



Note. **Average of chosen items** obtains results by first summing the results from the 16 beam ends of the selected elements. A simple average of this summed result is obtained to give the element average time history of axial force from which the peak average results are obtained.

The IMDPlus analysis will run and after a short while a graph showing the average axial (Fx) force obtained from the selected beam elements will be displayed for the first 10 seconds of the 1940 El Centro earthquake (Earthquake 1). The average axial force in the beam elements is shown in the following figure.

Seismic Analysis of a 2D Frame (Time Domain)



Peak results for the averaged element forces and moments are displayed in Notepad in file `peak_forceSet_Avg1.sum`. These indicate a peak positive average axial (Fx) force of 32.7kN at solution time 0.98 seconds and a peak negative average axial (Fx) force of -36.1kN at 1.01 seconds for the 1940 El Centro earthquake (Record 1), as shown in the following figure.


```
#
#
# +-----+
# | Record = 001 |
# +-----+
#
# Entity              Time              Value
Peak_Pos_Fx          0.9800000000          32659.62368
Peak_Neg_Fx          1.0100000000         -36075.22195
Abs_Peak_Fx          1.0100000000          36075.22195
Peak_Pos_Fy          2.1050000000          5585.413286
Peak_Neg_Fy          2.9050000000         -5997.747790
Abs_Peak_Fy          2.9050000000          5997.747790
Peak_Pos_Mz          2.1450000000          8359.900914
Peak_Neg_Mz          2.9050000000         -8270.496031
Abs_Peak_Mz          2.1450000000          8359.900914
```

An envelope of the peak results from the selected beam elements is displayed in Notepad in file `peak_forceSet1.sum`. Results for the 1940 El Centro earthquake (Record 1) are shown in the following figure. A peak positive axial (Fx) force of 1019.1kN occurs at time 2.13 seconds at end 1 of element 656. A peak negative axial (Fx) force of -956.0kN occurs at time 2.135 seconds at end 1 of element 648.

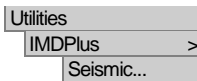


Note. The envelope of the peak results is obtained in IMDPlus by examining the individual results from all 16 ends of the chosen beam elements. This helps to quickly identify the locations where the peak results occur in the selected set of elements.


```
#
#
# +-----+
# | Record = 001 |
# +-----+
#
# Entity           Time           value           Element      End
Peak_Pos_Fx       2.130000000          1019065.962           656           1
Peak_Neg_Fx       2.135000000         -955968.1579          648           1
Abs_Peak_Fx       2.130000000          1019065.962           656           1
Peak_Pos_Fy       2.105000000          6684.561177          640           1
Peak_Neg_Fy       2.905000000         -6985.728050          640           1
Abs_Peak_Fy       2.905000000          6985.728050          640           1
Peak_Pos_Mz       2.145000000          15472.76490          640           2
Peak_Neg_Mz       2.905000000         -15552.26423          640           2
Abs_Peak_Mz       2.905000000          15552.26423          640           2
```

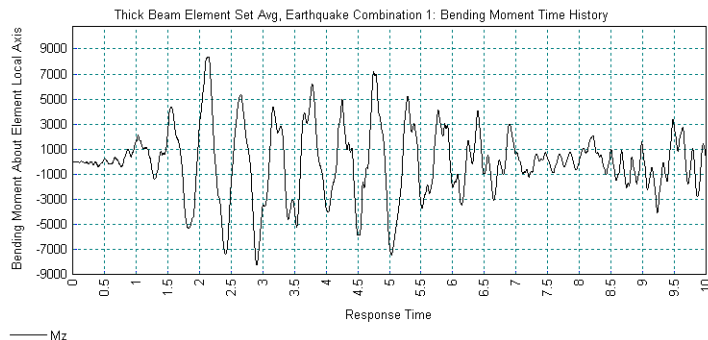
- Close the Notepad applications and graph windows.
- Click on the  button in the toolbar to open the IMDPlus Output Control dialog

If the IMDPlus toolbar is not enabled,



- Open the IMDPlus Seismic Analysis Control dialog through the menu and click **Next >** to keep the existing analysis control settings and open the IMDPlus Output Control dialog.
- Select **Force/Moment - Thick 2D Beam** results of **Mz**
- Click the **Finish** button to finish output processing of the element results. Click **Yes** when asked whether you want to process all of the selected elements.
- Click **Yes** when asked whether to free up disk space by deleting the temporary files created by IMDPlus.

A graph of the average (Mz) moment obtained from the selected elements will be displayed, as shown in the following figure.



All of the peak averaged results displayed in Notepad will be the same as those shown when the axial (Fx) force was analysed earlier. These indicate an average absolute peak (Mz) moment of 8.4kNm at time 2.145 seconds (Record 1). The envelope of the peak

Seismic Analysis of a 2D Frame (Time Domain)

results indicates an absolute peak (Mz) moment of 15.6kNm occurring at time 2.905 seconds in end 2 of element 640 (Record 1).

- Close the Notepad applications and graph windows.

Save the model



Save the model. This saves all load combinations that have been defined during the results processing operations.

This completes the example.

File
Save...