

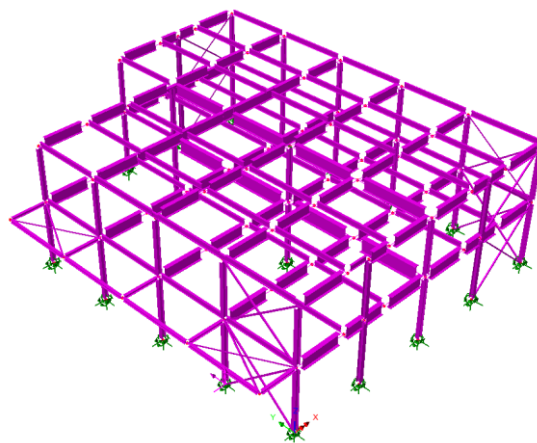
Seismic Analysis of a 3D Frame (Time Domain)

For LUSAS version:	17.0
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 2-storey 3D frame in the time domain. The geometry of the structure has been simplified to a wire-frame or stick representation with each of the members of the structure being represented by a line feature.

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



Objectives

The output requirements of the analysis are:

- ☐ Maximum deflections in the X-, Y- and Z-directions of a node at the top centre of the frame

- ☐ Average deflections and peak deflections in the X-, Y- and Z-directions from all of the unrestrained nodes in the frame
- ☐ Peak axial force and bending moment from all of the frame beams and columns
- ☐ Deformed mesh plots associated with the maximum deflections of a node at the top centre of the frame
- ☐ Envelopes of axial force and bending moment from modal combinations.

Keywords

3D, Frame, Seismic, Time Domain, Response, Interactive Modal Dynamics, IMD, Excitation, Eigenvalue, Modal Combination.

Associated Files



- ☐ **IMDPlus 3D Frame.mdl** Model of the structure.
- ☐ **e-w.prn, n-s.prn** and **up.prn** define the east-west horizontal, north-south horizontal and vertical earthquake accelerations for the 1940 El Centro earthquake.

Discussion

The mesh definition used in a dynamic analysis is somewhat different from that used in a static stress analysis. In a static analysis, and with experience, it is usually not too difficult to estimate where the high stresses are likely to occur. These estimates can then be used to develop a meshing strategy with a fine mesh in high stress locations and a coarse mesh in less critical locations. For a dynamic analysis the interaction between the stiffness and inertia forces will lead to deflected shapes which can be very different from those expected in a static analysis.

In a dynamic analysis both stiffness and mass distribution has to be considered. Generally, the best strategy for a dynamic analysis is to have a uniform mesh over the entire structure. If refinement is to be carried out a coarser mesh can sometimes be used in stiff regions. In regions that are more flexible, or where heavy masses are located, the mesh should be more refined.

In this example the global behaviour of the building is being considered for earthquake response. In this analysis the lower frequencies will be dominant and a relatively coarse mesh will suffice. If the higher frequencies are important, or if a local response for individual beams and columns is to be considered, a revised mesh with more elements would need to be used.

The seismic response analysis is performed in two distinct stages. A natural frequency analysis is performed first. This is used here to calculate the first 30 natural modes of vibration of the structure. The eigenvalues (frequencies) and eigenvectors (mode shapes) are stored and used in the subsequent seismic response analysis. In order to carry out a seismic analysis the eigenvectors must be normalised with respect to the

mass. Although natural frequencies are obtained from an eigenvalue analysis any information regarding the magnitudes of deformations or 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 support excitation governed by time histories of acceleration in the model global axes. In this example, the seismic excitation is applied directly to the bases of the columns using the first 2.5 seconds of the 1940 El Centro earthquake.

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.

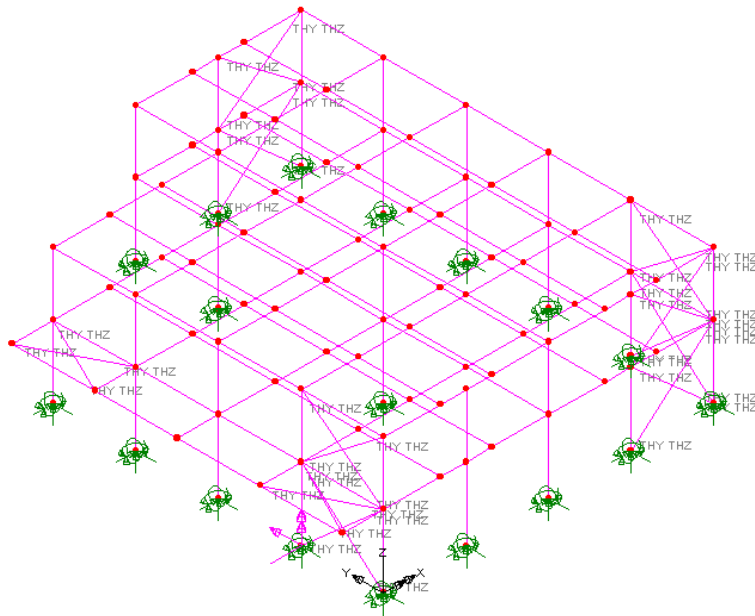
Building and loading the model

- To create the model, open the file **IMDPlus 3D Frame.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 isometric view of the model of the building will be displayed.

File
Open...



- In the \<LUSAS Installation Folder>\Projects\ folder create a new directory called **IMDPlus 3D Frame**
- Save the model into this new folder as **IMDPlus 3D Frame**. 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.

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 **30**
- 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

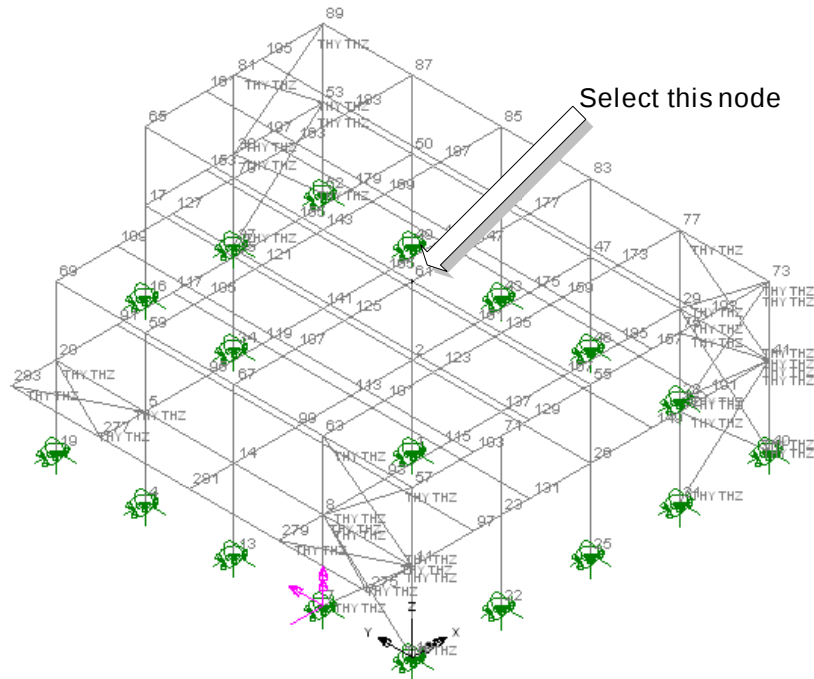
If the analysis was run from within LUSAS Modeller the results will be loaded on top of the current model and the loadcase results for each eigenvalue can be seen in the Loadcase panel of the  Treeview with Eigenvalue 1 set to be active by default.

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. 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.

Initially, in this example, the response of a node at the top and in the centre of the frame will be investigated. Node numbers will be added to the mesh display to allow selection.

- Turn off the **Geometry** layer in the  Treeview.
- 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 **Node > Name** check box and click **OK**
- Select the node at the top and in the centre of the frame, node **61** as shown in the following diagram



Defining the Seismic Parameters



Note. To simplify the use of IMDPlus (and provide an alternative to picking IMDPlus menu selections from the Analyses > IMDPlus menu item) 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 through the menu 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)

Y-direction earthquake record(s)

Z-direction earthquake record(s)

Direction Factors X Y Z

Mode/damping control

Included modes:

☒ Default damping %

Total participating mass

X Direction %

Y Direction %

Z Direction %

Frequency interpolation

Advanced solution options

☐ All frequencies included ☒ Linear interpolation of records

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

Time step

- In the Earthquake Input section, for the **X-Direction Earthquake Record(s)** browse to the \<Lusas Installation Folder>\Examples\Modeller directory and select **e-w.prn**
- For the **Y-Direction Earthquake Record(s)** browse to the \<Lusas Installation Folder>\Examples\Modeller directory and select **n-s.prn**
- For the **Z-Direction Earthquake Record(s)** browse to the \<Lusas Installation Folder>\Examples\Modeller directory and select **up.prn**
- Ensure that the **Direction Factors** are set to **1.0** in the X-Direction, **1.0** in the Y-Direction and **0.66667** in the Z-Direction
- 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.
- Select the **Linear interpolation of records** option and enter a time step of **0.0025** (Half the time step of the original earthquake records)
- Click the **Next** button to proceed. When prompted about significant missing total mass choose **Yes** as only 31.22% is included in the Z-Direction (vertical) for 30 modes.

Individual Node Displacements

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

- Select **Node** and select **Extent** as **Individual**
- Choose **Node** number **61** from the drop-down list.
- Select **Displacement** results of **DX, DY & DZ**
- 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**
- Select **Modal combination/factor history** and click on the **Options...** button.
- Set the **Minimum mass participation factor percentage to include** to **1.0** and click on **OK**
- Select **Generate textfile output**

The dialog should now look like this:

IMDPlus Seismic Output Control

Node/element selection

☒ Node ☐ Element Entity: Displacement

Extent: Individual Component: DX, DY & DZ

Node: 61

☒ Individual items
☐ Sum of chosen items
☐ Average of chosen items

Output control

☒ Response time history

☒ Peak response summary ☐ Positive/negative ☐ Envelope
☒ Absolute

☐ Secondary response spectra (SRS) Modify...

☒ Modal combination/factor history Options...

☒ Generate graphs in Modeller Earthquake to graph: 1

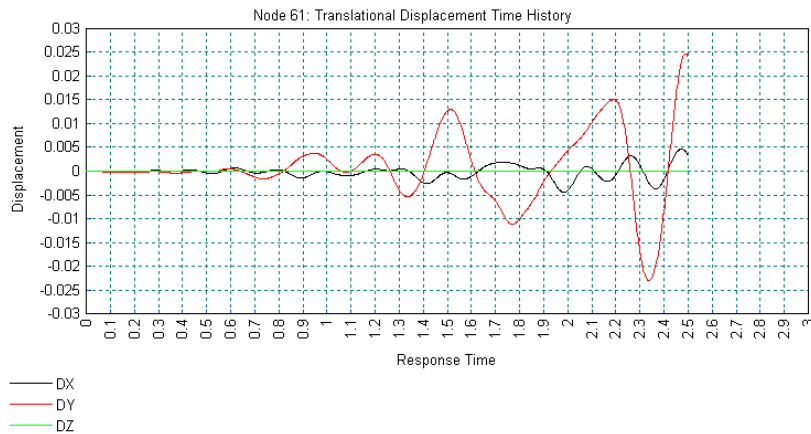
☒ Generate textfile output Options...

< Back Finish Apply Cancel Help

- Click the **Finish** button to perform the analysis.
- Click **Yes** when asked whether to free up disk space by deleting the temporary files created by IMDPlus.

The horizontal (DX and DY) and vertical (DZ) displacement of the top of the column are displayed on the graph for the first 2.5 seconds of the 1940 El Centro earthquake.

Seismic Analysis of a 3D Frame (Time Domain)



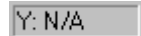
Peak displacements are also output to Notepad and indicate absolute peak displacements of 0.0047 m in the X-direction, 0.0248 m in the Y-direction along with additional output for the Z-direction and rotations about each of these axes. From this output we can identify times of 2.4725 seconds and 2.49 seconds which correspond to the absolute peak displacements in the X and Y directions respectively. This information will be used in a later section to visualise the deformed shapes using modal combinations.

```
#
#
# +-----+
# | Record = 001 |
# +-----+
#
# Entity      Time      Value
Peak_Pos_DX   2.472500000  0.4661208719E-002
Peak_Neg_DX   1.985000000 -0.4450961391E-002
Abs_Peak_DX   2.472500000  0.4661208719E-002
Peak_Pos_DY   2.490000000  0.2475375967E-001
Peak_Neg_DY   2.337500000 -0.2302464293E-001
Abs_Peak_DY   2.490000000  0.2475375967E-001
Peak_Pos_DZ   1.082500000  0.6606589245E-004
Peak_Neg_DZ   1.015000000 -0.9454571681E-004
Abs_Peak_DZ   1.015000000  0.9454571681E-004
Peak_Pos_THX  2.327500000  0.5261706667E-004
Peak_Neg_THX  2.500000000 -0.5465704918E-004
Abs_Peak_THX  2.500000000  0.5465704918E-004
Peak_Pos_THY  2.470000000  0.1598727930E-004
Peak_Neg_THY  2.002500000 -0.1607368682E-004
Abs_Peak_THY  2.002500000  0.1607368682E-004
Peak_Pos_THZ  2.380000000  0.4129028145E-003
Peak_Neg_THZ  2.480000000 -0.4321706513E-003
Abs_Peak_THZ  2.480000000  0.4321706513E-003
Abs_Peak_RSLT 2.490000000  0.2508876353E-001
```

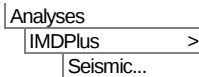
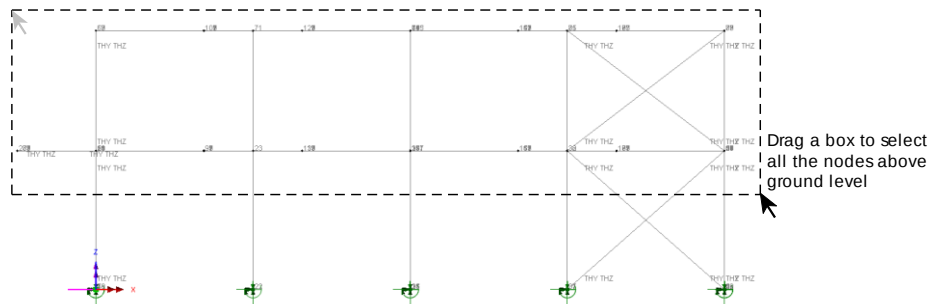
- Close the Notepad application and graph window.

Average and Peak Displacements for a Selection of Nodes

This section investigates results for a chosen set of nodes containing all of the unrestrained nodes in the model, that is, all of the nodes above ground level. The average displacements and peak displacements of the selected nodes will be obtained.

 Rotate the model to view along the Y axis by clicking the status bar at the bottom of the Modeller window.

 Using Select Nodes drag a box to select all of the 89 nodes above ground level, as shown in the following figure.



- 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 **Node** and change **Extent** to **Selected** using the drop-down list. This chooses all 89 nodes above ground level that were selected in the preceding step.



Note. As **Extent** has been set to **Selected** the Node drop-down list will not be available for selection.

- Select result entity **Displacement** and component **DX, DY & DZ**
- Deselect **Individual items** and select **Average of chosen items** as the averaged results of the selected nodes are going to be investigated.
- Ensure **Response time history** and **Generate graphs in Modeller** remain selected.

- Ensure **Peak response summary** remains selected and also select **Envelope**
- Deselect **Modal combination/factor history** and deselect **Generate textfile output**

The dialog should now look like this:

IMDPlus Seismic Output Control

Node/element selection

☒ Node ☐ Element Entity: Displacement

Extent: Selected Component: DX, DY & DZ

Node: 61

☐ Individual items
☐ Sum of chosen items
☒ Average of chosen items

Output control

☒ Response time history
☒ Peak response summary
☐ Secondary response spectra (SRS)
☐ Modal combination/factor history
☒ Generate graphs in Modeller
☐ Generate textfile output

☐ Positive/negative ☒ Envelope
☒ Absolute

Modify... Options...
Options...

Earthquake to graph: 1

< Back Finish Apply Cancel Help

- Click the **Apply** button to perform the analysis. Click **Yes** when asked whether you want to process all of the selected nodes.

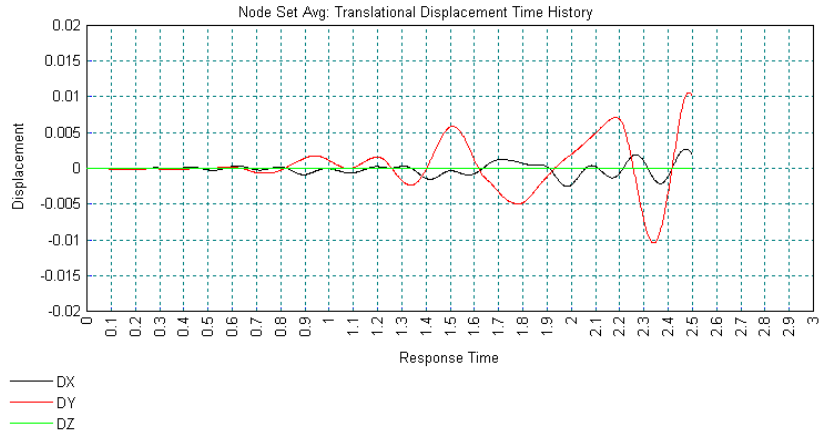


Note. Any combination of the options **Individual items**, **Sum of chosen items** and **Average of chosen items output** can be used together in an IMDPlus analysis, although using the **Individual items** option when **Extent** is set to **Selected** or **All** (nodes or elements) may produce a large 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 89 nodes selected. A simple average of these summed results is obtained to give the averaged node time histories from which the peak average results are obtained.

A graph showing the average horizontal (DX and DY) and vertical (DZ) displacements for all of the selected nodes is displayed for the 1940 El Centro earthquake. The average node displacements are shown in the following figure.



The peak results for the averaged node displacements are displayed in Notepad in file peak_dspSet_Avg1.sum. These indicate absolute peak average displacements of 0.0027m at time 2.4725 seconds in the X-direction and 0.0106m at time 2.485 seconds in the Y-direction.

```
#
#
# +-----+
# | Record = 001 |
# +-----+
#
# Entity      Time      Value
Peak_Pos_DX   2.472500000  0.2711970446E-002
Peak_Neg_DX   1.982500000  -0.2545200708E-002
Abs_Peak_DX   2.472500000  0.2711970446E-002
Peak_Pos_DY   2.485000000  0.1059843682E-001
Peak_Neg_DY   2.340000000  -0.1046089938E-001
Abs_Peak_DY   2.485000000  0.1059843682E-001
Peak_Pos_DZ   0.99000000000  0.4570291210E-004
Peak_Neg_DZ   1.015000000  -0.8529296567E-004
Abs_Peak_DZ   1.015000000  0.8529296567E-004
Peak_Pos_THX  2.337500000  0.3003976055E-003
Peak_Neg_THX  2.492500000  -0.3044003645E-003
Abs_Peak_THX  2.492500000  0.3044003645E-003
Peak_Pos_THY  2.475000000  0.4385270496E-004
Peak_Neg_THY  1.990000000  -0.4369077358E-004
Abs_Peak_THY  2.475000000  0.4385270496E-004
Peak_Pos_THZ  2.340000000  0.2687725883E-003
Peak_Neg_THZ  2.482500000  -0.3000152315E-003
Abs_Peak_THZ  2.482500000  0.3000152315E-003
Abs_Peak_RSLT 2.482500000  0.1156838441E-001
```

An envelope of the peak results for the chosen nodes is displayed in Notepad in file peak_dspSet1.sum and is shown in the following figure. The peak X-displacement of 0.0068m occurs at node 143 at time 2.745 seconds and the peak Y-displacement of 0.0248m occurs at node 135 at time 2.49 seconds.



Note. The envelope of the peak results is obtained in IMDPlus by examining the individual results from the 89 unrestrained nodes. This helps to quickly identify the nodes of the model where the maximum and minimum displacements occur.

```
#
#
# +-----+
# | Record = 001 |
# +-----+
#
# Entity              Time              Value              Node
Peak_Pos_DX           2.475000000          0.6821753781E-002      143
Peak_Neg_DX           2.372500000          -0.5593165797E-002     143
Abs_Peak_DX           2.475000000          0.6821753781E-002      143
Peak_Pos_DY           2.490000000          0.2475797788E-001      135
Peak_Neg_DY           2.337500000          -0.2302854973E-001     135
Abs_Peak_DY           2.490000000          0.2475797788E-001      135
Peak_Pos_DZ           2.477500000          0.9252464285E-003      277
Peak_Neg_DZ           1.995000000          -0.8662857720E-003     277
Abs_Peak_DZ           2.477500000          0.9252464285E-003      277
Peak_Pos_THX          2.342500000          0.3661382546E-002      23
Peak_Neg_THX          2.485000000          -0.3739308469E-002     23
Abs_Peak_THX          2.485000000          0.3739308469E-002      23
Peak_Pos_THY          2.477500000          0.4783863731E-003      277
Peak_Neg_THY          1.995000000          -0.4772056675E-003     277
Abs_Peak_THY          2.477500000          0.4783863731E-003      277
Peak_Pos_THZ          2.485000000          0.5087373266E-002      69
Peak_Neg_THZ          2.342500000          -0.5074594480E-002     69
Abs_Peak_THZ          2.485000000          0.5087373266E-002      69
Abs_Peak_RSLT         2.487500000          0.2551992385E-001      143
```

- Close all of the Notepad applications and graph windows.

Peak Axial Forces from All of the Elements in the Model

In this section the peak results for a set of elements containing all of the 201 beam elements in the model will be investigated.



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.



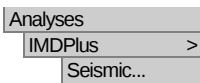
Note. As all of the elements in the model are going to be processed there is no need to pre-select the elements in the graphics window prior to running the IMDPlus analysis. Instead the **All** option from the **Extent** drop down list will be utilised on the IMDPlus Output Control dialog.

- 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 change **Extent** to **All** using the drop-down list. This chooses all of the elements in the model.



Note. As **Extent** has been set to **All** 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 3D Beam** and component **Fx**
- Ensure **Average of chosen items** remains selected and **Individual items** and **Sum of chosen items** deselected as only the average results of the chosen elements are going to be investigated.
- Deselect **Response time history**
- Ensure **Peak response summary** and **Envelope** remain selected.

The dialog should now look like this:

IMDPlus Seismic Output Control

Node/element selection

☐ Node ☒ Element Entity **Force/Moment - Thick 3D Beam**

Extent **All** Component **Fx**

Element

End **All**

☐ Individual items

☐ Sum of chosen items

☒ Average of chosen items

Output control

☐ Response time history

☒ Peak response summary ☐ Positive/negative ☒ Envelope

☐ Absolute

☐ Secondary response spectra (SRS) **Modify...**

☐ Modal combination/factor history **Options...**

☒ Generate graphs in Modeller Earthquake to graph **1**

☐ Generate textfile output **Options...**

< Back Finish Apply Cancel Help

Seismic Analysis of a 3D Frame (Time Domain)

- Click the **Finish** button to perform the analysis. Click **Yes** when asked whether you want to process all of the elements in the model. On modern computers this will take under 5 minutes.
- Click **Yes** when asked whether to free up disk space by deleting the temporary files created by IMDPlus.

Peak average results for the chosen set of elements are output to Notepad in file peak_forceSet_Ave2.sum.



Note. **Average of chosen items** obtains the element results by first summing the results from all of the beam ends in the model. A simple average of these summed results is obtained to give the average response time histories from which the peak average results are obtained.

An envelope of the peak results for all of the elements in the model is displayed in Notepad in file peak_forceSet2.sum and is shown in the following figure. It can be seen that the peak compressive axial force (Fx) of -30.97kN occurs at end 1 of element 4 at time 2.3425 seconds. The peak bending moment (My) occurs at end 1 of element 133 at time 2.49 seconds and has a value of -46.42kNm.



Note. The envelope of the peak results is obtained in IMDPlus by examining the individual results for all of the beam elements in the model. This enables the locations of the elements with maximum and minimum results to be quickly identified.

```
#
#
# +-----+
# | Record = 001 |
# +-----+
#
# Entity      Time      Value      Element      End
Peak_Pos_Fx   2.340000000    34152.50443    10            1
Peak_Neg_Fx   2.342500000   -30970.35591    4             1
Abs_Peak_Fx   2.340000000    34152.50443    10            1
Peak_Pos_Fy   2.482500000    15453.07531    1             1
Peak_Neg_Fy   2.342500000   -14410.67957    1             1
Abs_Peak_Fy   2.482500000    15453.07531    1             1
Peak_Pos_Fz   2.490000000    9383.017514    133           1
Peak_Neg_Fz   2.472500000   -10137.43335    12            1
Abs_Peak_Fz   2.472500000    10137.43335    12            1
Peak_Pos_Mx   2.485000000     174.0245661    47            1
Peak_Neg_Mx   2.485000000   -172.9160181    58            1
Abs_Peak_Mx   2.485000000    174.0245661    47            1
Peak_Pos_My   2.337500000    43014.11544    133           1
Peak_Neg_My   2.490000000   -46420.96736    133           1
Abs_Peak_My   2.490000000    46420.96736    133           1
Peak_Pos_Mz   2.482500000     27226.15924    1             2
Peak_Neg_Mz   2.482500000   -27323.19660    1             1
Abs_Peak_Mz   2.482500000    27323.19660    1             1
```

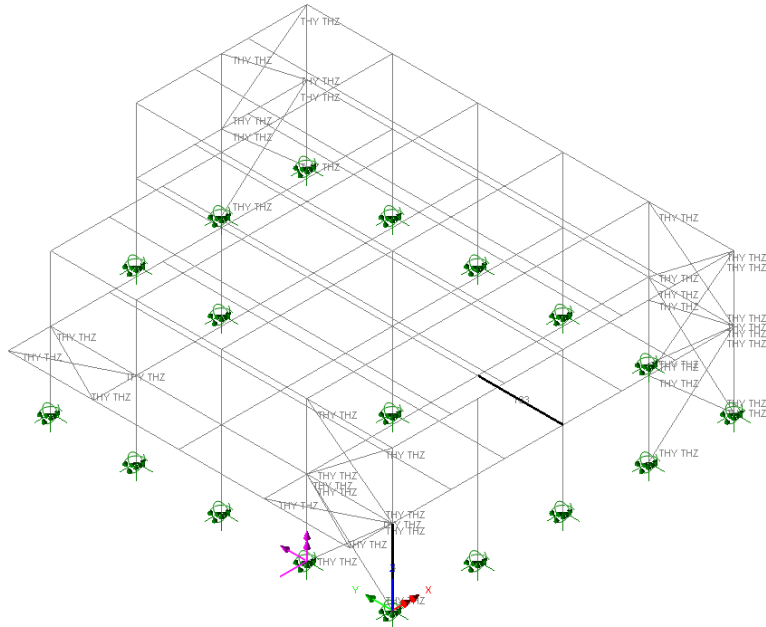
- Close the Notepad applications.

The elements that contain the peak compressive axial force and peak bending moment will now be located in the model.



In the LUSAS graphics window click on the Isometric view button.

- Click the left-hand mouse button in a blank part of the graphics window to remove the node selection created in the preceding sections.
- Double-click on the **Labels** layer name in the  Treeview.
- On the Properties dialog de-select the **Node > Name** check box, select the **Element > Name** check box and select the **Label selected items only** check box.
- Click the **OK** button to accept all other settings. This will turn on the layer in the graphics window.
- With no features selected click the right-hand mouse button in a blank part of the graphics window and select the **Advanced Selection...** option.
- Select **Type and Name**, select **Element** from the drop-down list and enter element number **4**
- Select **Add to selection** and click the **Apply** button to accept all other settings. Beam element 4 will be highlighted in the graphics window.
- Enter element number **133** and click the **OK** button. Beam element 133 will also be displayed in the graphics window, as shown in the following figure.



Displaying Deformed Mesh Plots from Modal Combinations

When the IMDPlus analysis of node 61 was carried out in an earlier section, the Modal combination/factor history option was selected and caused IMDPlus to generate a command file in the \<LUSAS Installation Folder>\Projects\IMDPlus 3D Frame\Associated Model Data\IMDPlus 3D Frame directory called **Modal_Combinations-rec001.vbs**. These combinations can now be generated in Modeller by loading this command file.



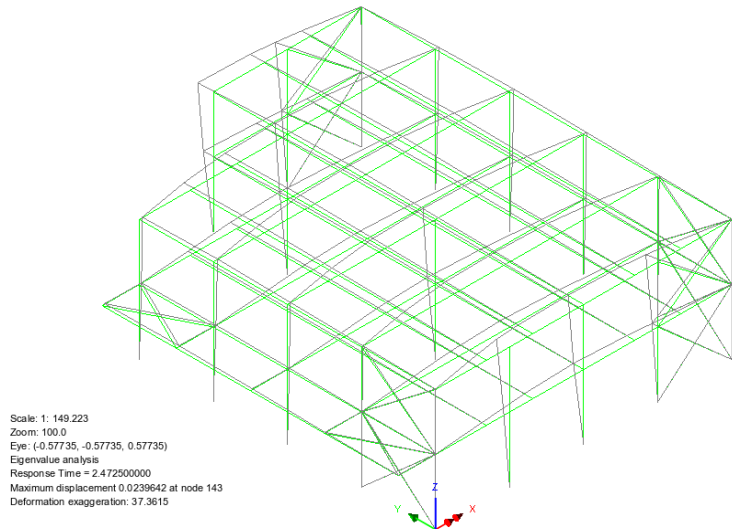
Modal_Combinations-rec001.vbs defines the modal combinations for all time steps in the seismic analysis.

- Open the file **Modal_Combinations-rec001.vbs** located in the \<LUSAS Installation Folder>\Projects\IMDPlus 3D Frame\Associated Model Data\IMDPlus 3D Frame directory.

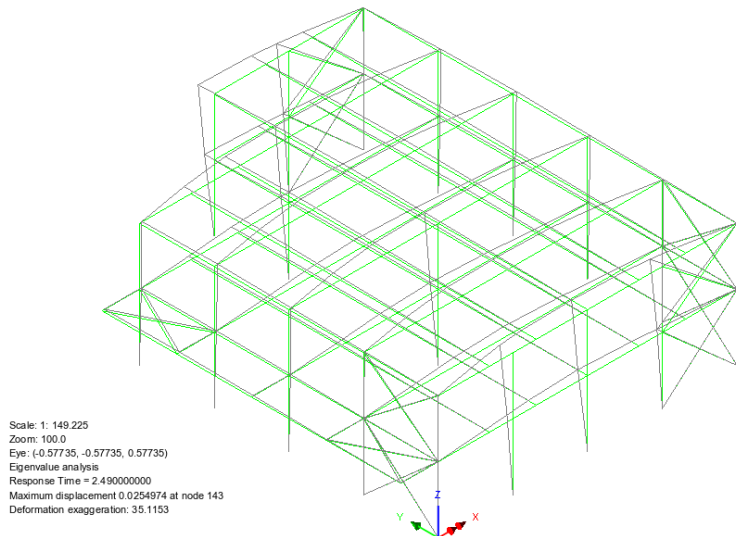
The combinations will appear under the original model in the  Treeview.

- Click the left-hand mouse button in a blank part of the graphics window to remove the element selection created in the previous section.
- Turn off the display of the **Labels** and **Attributes** layers in the  Treeview.
- In the  Treeview click the right-hand mouse button on the **Local Coordinates 1** entry and deselect **Show definition**
- In the  Treeview click the right-hand mouse button on the **Mesh** and select **Properties**. Select the **Choose pen...** button and select a **Green** colour for the mesh. Select the **Visualise** tab and turn off the **Beam end releases** option. Click the **OK** button.
- With no features selected click the right-hand mouse button in a blank part of the Graphics area 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.
- 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 **(50.0,-120.0)** and click the **OK** button to return to the graphics window.
- In the  Treeview right-click on the **991:Response Time = 2.472500000** dataset name and select the **Set Active** option to set this combination active. This is the time at which the absolute peak displacement in the X-direction occurred at node

61. The deformed mesh will now be displayed for this instance in time as shown below.



- In the  Treeview right-click on the **998:Response Time = 2.490000000** dataset name and select the **Set Active** option to set this combination active. This is the time at which the absolute peak displacement in the Y-direction occurred at node 61. The deformed mesh will now be displayed for this instance in time as shown below.





Note. Standard post-processing such as displaying contours can be performed on these modal combinations to allow the global behaviours at instances in time to be investigated.

Displaying Envelopes for the Whole Earthquake

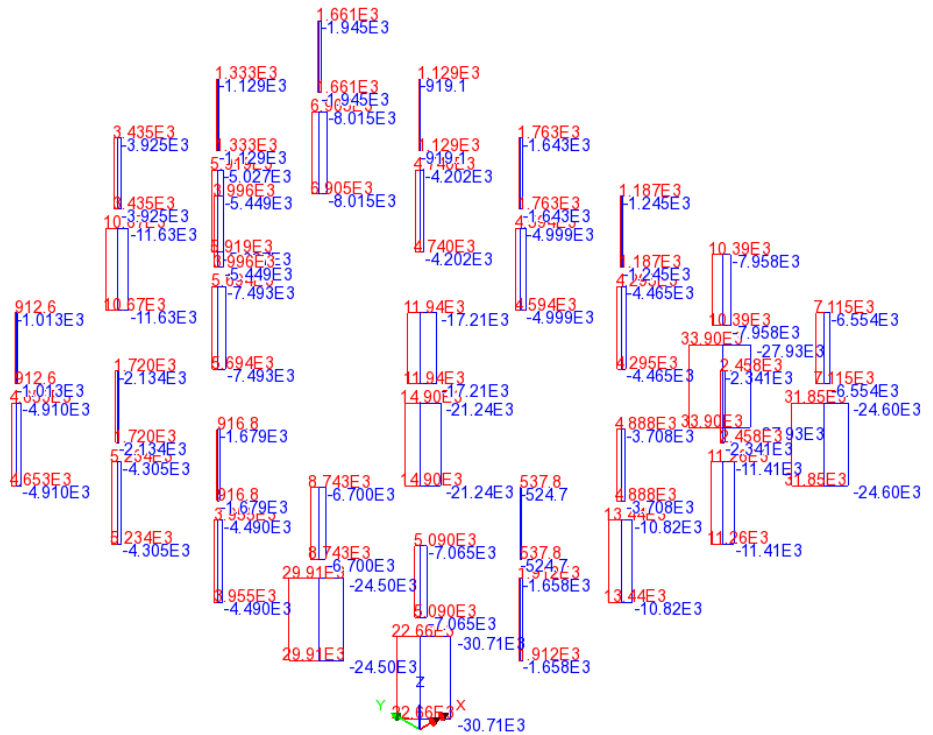
Plots showing the maximum and minimum axial force and bending moment in the columns for the entire earthquake duration are to be displayed.

Create an envelope

- Enter **Whole Earthquake** for the name
- Hold down the shift key and select combinations **2** to **1002** in the list and click on the >> '**Add to**' button to add all combinations to the envelope
- Click the **OK** button to create the max and min envelopes.
- In the  Treeview right-click the group named **Columns** and **Set as Only Visible** to only view the column members.
- Turn off the display of the **Mesh** and **Deformed Mesh** layers in the  Treeview.
- In a blank part of the Graphics area, click the right-hand mouse button and select the **Diagrams** option to add the Diagrams layer to the  Treeview.
- Select **Force/Moment – Thick 3D Beam** results of axial force **Fx** in the members. For envelopes and smart combinations ensure that **Show minimum too** is selected. Select the **Diagram display** tab, ensure the **Label values** option is selected, and deselect the **Peaks Only** option. Plot values on **80%** of the element length.
- Click the **OK** button.
- In the  Treeview right-click on the **Whole Earthquake (Max)** envelope and select the **Set Active** option to set this envelope active.
- On the Set Active dialog select **Force/Moment – Thick 3D Beam** results of axial force **Fx** and click the **OK** button to finish.

The envelope will now be assembled over the 1000 modal combinations generated by IMDPlus. On modern computers this will take a minute or so..

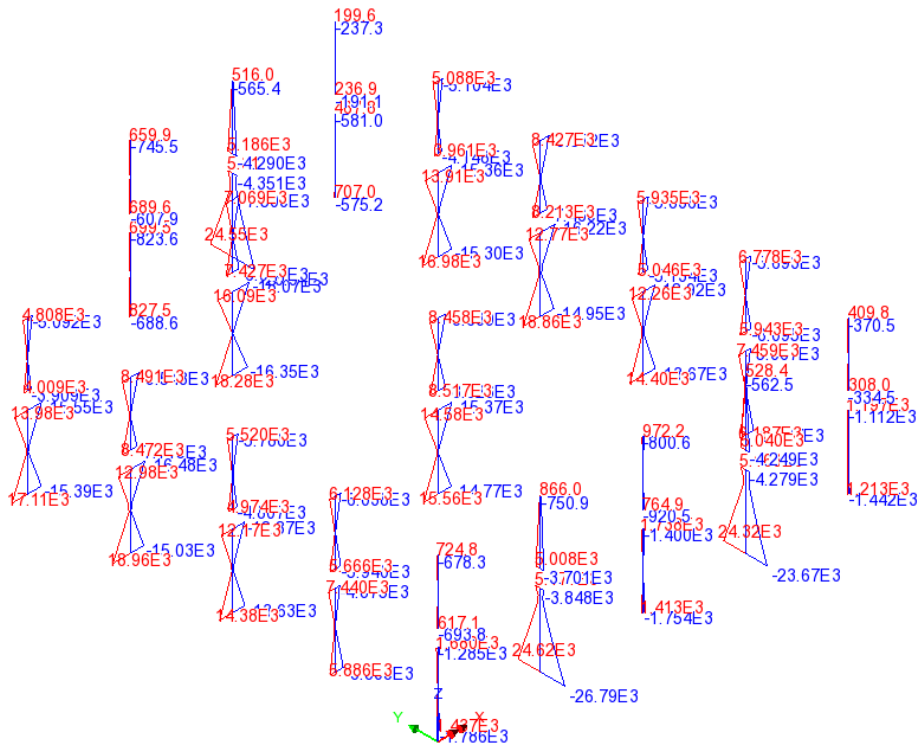
Analyses
Envelope...



It can be seen that, for the central column F_x (Max) = 11.94e3 and F_x (Min) = -17.21e3.

- In the  Treeview right-click on the **Whole Earthquake (Max)** envelope and select the **Set Active** option to set this envelope active. Select entity **Force/Moment** – **Thick 3D Beam** of component **My**. Ensure that **Change existing drawing layers to above component** is selected, then click the **OK** button.

The envelope will now be assembled over the 1000 modal combinations generated by IMDPlus.



It can be seen that, for the central column M_y (Max) at top = $8.458e3$ and M_y (Max) at bottom = $8.517e3$.

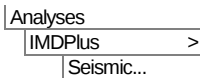
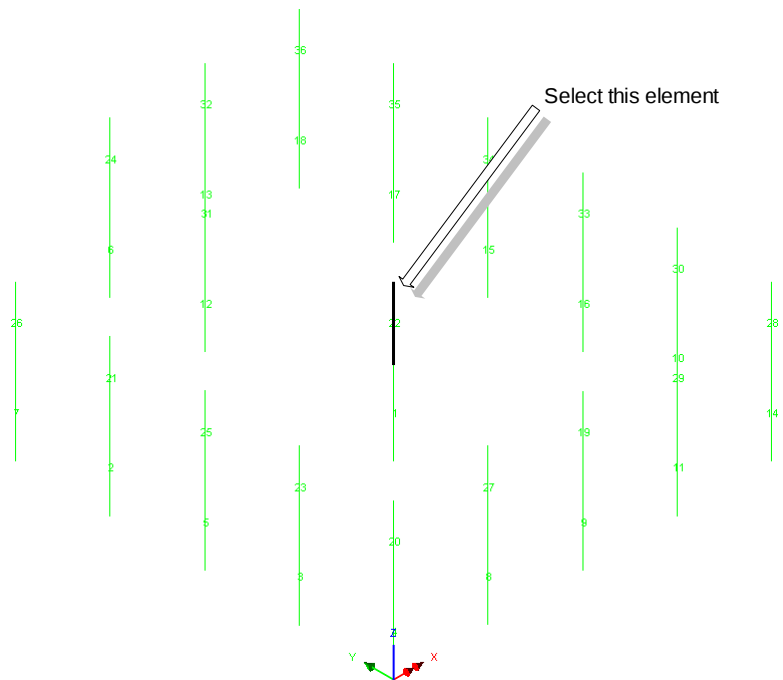
Displaying results for a selected member

To check these previous results for F_x and M_y we will utilise IMDPlus again to check the response of the member in centre of the frame.

- Turn off the display of the **Diagrams** and **Annotation** layers in the  Treeview.
- Turn on the display of the **Mesh** layer in the  Treeview.

Element numbers will be added to the mesh display to allow selection.

- Double-click on the **Labels** layer name in the  Treeview. On the Properties dialog de-select the **Label selected items only** check box and click **OK**. (This turns on the display of a turned-off layer.)
-  Select the element in the centre of the frame as shown in the following diagram.



- 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.
- On the IMDPlus Output Control dialog ensure **Element** is selected, and change **Extent** to **Individual**
- Select **Element** number **22** from the drop-down list and ensure **End** is set to **All**
- Choose **Force/Moment – Thick 3D Beam** and component **Fx**.
- Select **Individual items**. **Sum of chosen items** and **Average of chosen items** will not be available as an individual element is being processed.
- Ensure **Peak response summary** is selected.
- Click the **Finish** button and click the **Yes** button to confirm that both ends of the beam element are to be processed (because All ends has been previously set).

Seismic Analysis of a 3D Frame (Time Domain)

- Click **Yes** when asked whether to free up disk space by deleting the temporary files created by IMDPlus.

The peak forces and moments in element 22 will be output to a Notepad application in the format indicated below.

```
#
#
# +-----+
# | Record = 001 |
# +-----+
#
# Entity      Time      value
Peak_Pos_Fx1 1.082500000 12244.70438
Peak_Neg_Fx1 1.015000000 -17622.48885
Abs_Peak_Fx1 1.015000000 17622.48885
Peak_Pos_Fx2 1.082500000 12244.70438
Peak_Neg_Fx2 1.015000000 -17622.48885
Abs_Peak_Fx2 1.015000000 17622.48885
Peak_Pos_Fy1 2.500000000 14769.22560
Peak_Neg_Fy1 2.330000000 -13277.01107
Abs_Peak_Fy1 2.500000000 14769.22560
Peak_Pos_Fy2 2.500000000 14769.22560
Peak_Neg_Fy2 2.330000000 -13277.01107
Abs_Peak_Fy2 2.500000000 14769.22560
Peak_Pos_Fz1 1.982500000 5578.155101
Peak_Neg_Fz1 2.475000000 -5870.701372
Abs_Peak_Fz1 2.475000000 5870.701372
Peak_Pos_Fz2 1.982500000 5578.155101
Peak_Neg_Fz2 2.475000000 -5870.701372
Abs_Peak_Fz2 2.475000000 5870.701372
Peak_Pos_Mx1 2.285000000 1.401136532
Peak_Neg_Mx1 2.195000000 -0.9631674992
Abs_Peak_Mx1 2.285000000 1.401136532
Peak_Pos_Mx2 2.285000000 1.401136532
Peak_Neg_Mx2 2.195000000 -0.9631674992
Abs_Peak_Mx2 2.285000000 1.401136532
Peak_Pos_My1 2.475000000 8923.919641
Peak_Neg_My1 1.982500000 -8468.623770
Abs_Peak_My1 2.475000000 8923.919641
Peak_Pos_My2 1.982500000 8600.530840
Peak_Neg_My2 2.475000000 -9040.426557
Abs_Peak_My2 2.475000000 9040.426557
Peak_Pos_Mz1 2.330000000 20327.29239
Peak_Neg_Mz1 2.500000000 -22606.97454
Abs_Peak_Mz1 2.500000000 22606.97454
Peak_Pos_Mz2 2.500000000 22586.85579
Peak_Neg_Mz2 2.330000000 -20300.36149
Abs_Peak_Mz2 2.500000000 22586.85579
```

Comparison of the results for Fx and My for the central column show good agreement between the envelope and IMDPlus results. Exact agreement would have been obtained if all modes were included in the modal combinations used for the envelope instead of the 1% minimum participating mass selected earlier.

- Close the Notepad application.



Note. For an IMDPlus seismic analysis selecting Peak response summary will tabulate results for all components and not just the one selected (Fx). If Response time history is selected only the results for the selected component will be graphed.

Save the model

File

Save...



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

This completes the example.

