

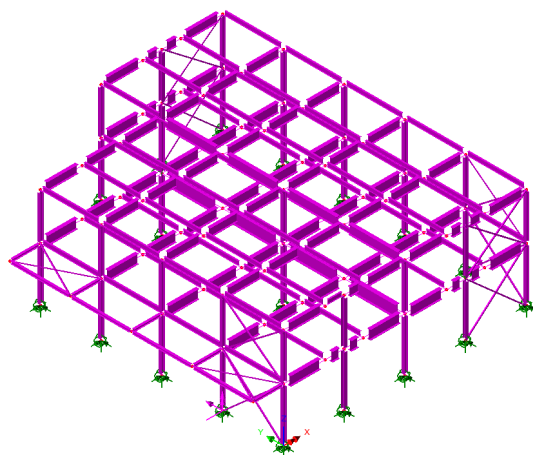
Seismic Analysis of a 3D Frame (Time Domain)

For LUSAS version:	22.0
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 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

Associated files can be downloaded from the user area of the LUSAS website.



- ☐ **IMDPlus_3D_frame.lvb** This file creates a model of the structure.
- ☐ **e-w.prn, n-s.prn** and **up.prn** These files 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.

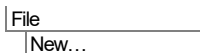
The second phase of the analysis utilises the IMDPlus option which performs enhanced time domain solutions using Interactive Modal Dynamics (IMD). This is an alternative to performing a spectral response analysis and allows the excitation of the structure using acceleration time histories instead of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

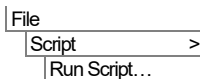
Creating the model



- Enter a file name of **IMDPlus 3D Frame**
- Use the default **User-defined** working folder.
- Ensure an Analysis category of **3D** is set.
- Click the **OK** button.

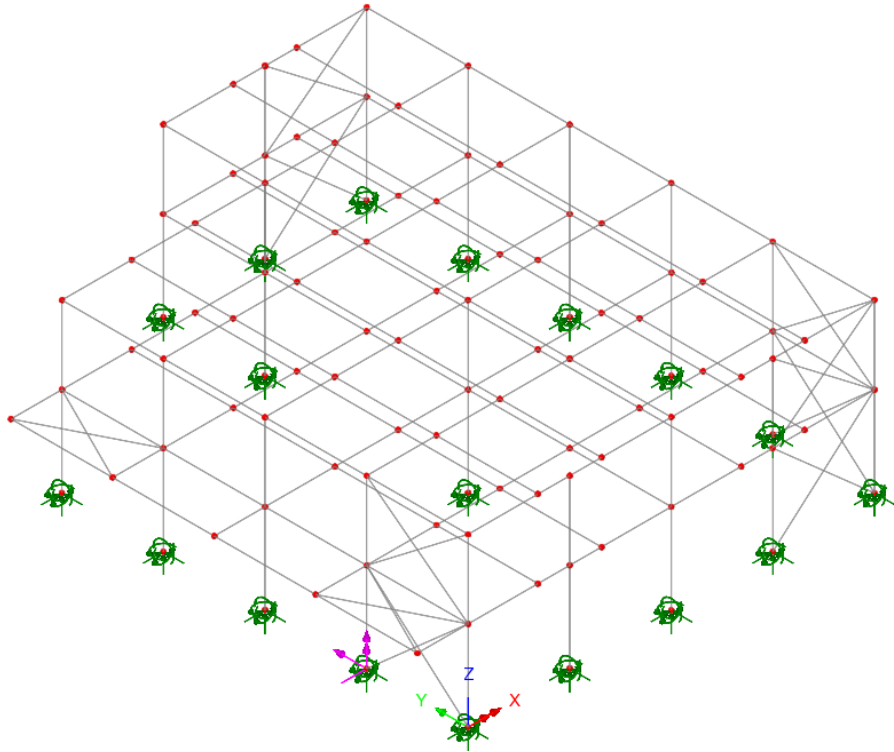


Note. There is no need to enter any other new model details when the intention is to run a script to build an initial model, since the contents of the script will overwrite any other settings made.



To create the model required for this example, open the script file **IMDPlus_3D_frame.lvb** that was downloaded and placed in a folder of your choosing.

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



File
Save As...



- Save the model into a new folder named **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

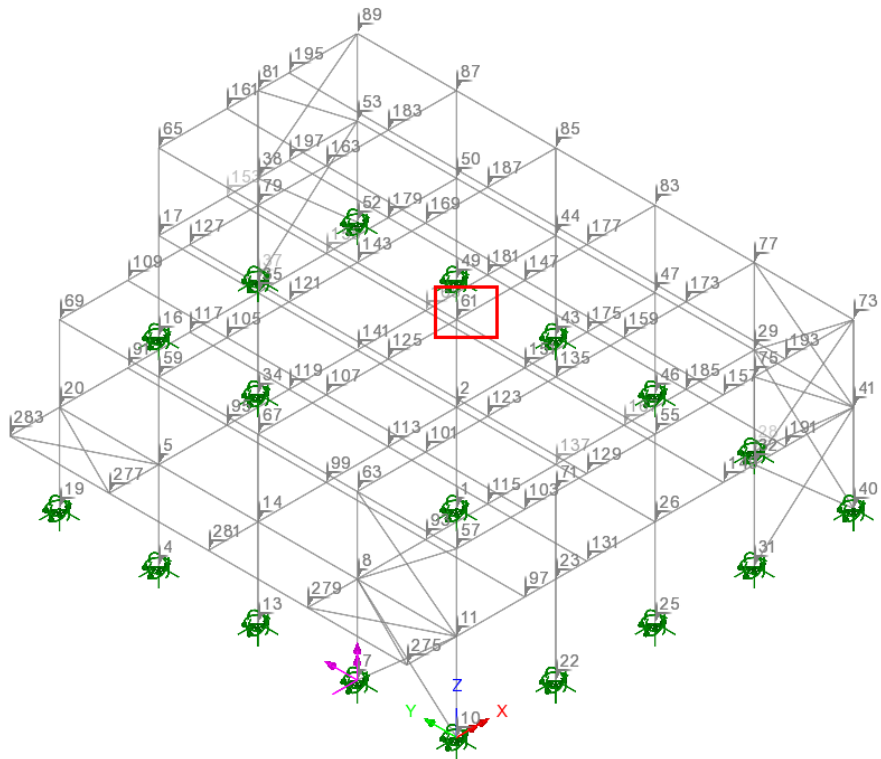
If the analysis was run from within LUSAS Modeller the results will be loaded on top of the current model and the loadcase results for each eigenvalue can be seen in the Loadcase panel of the  Treeview with Eigenvalue 1 set to be active by default.

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 view 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 **2** 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.

- 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 by selecting the menu item or click on the  button in the toolbar.

The input required on this dialog consists of the earthquake acceleration records, included modes and damping along with interpolation options.

Analyses
IMDPlus >
Seismic...

IMDPlus Seismic Analysis Control

Earthquake input

X-direction earthquake record(s) C:\Users\paul_b\Documents\LUSAS_assi Browse...

Y-direction earthquake record(s) C:\Users\paul_b\Documents\LUSAS_assi Browse...

Z-direction earthquake record(s) C:\Users\paul_b\Documents\LUSAS_assi Browse...

Direction Factors X 1.0 Y 1.0 Z 0.66667

Mode/damping control

Included modes: All Modify...

☒ Default damping 5.0 % Modify...

Total participating mass

X Direction	94.56	%
Y Direction	93.84	%
Z Direction	31.22	%

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 0.0025

< Back Next > Cancel Help

In the Earthquake input section:

- For the **X-direction earthquake records(s)**, browse to select the associated example file **e-w.prn** that was downloaded and placed in a folder of your choosing.
- For the **Y-direction earthquake records(s)**, browse to select the associated example file **n-s.prn** that was downloaded and placed in a folder of your choosing
- For the **Z-direction earthquake records(s)**, browse to select the associated example file **up.prn** that was downloaded and placed in a folder of your choosing
- 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

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:

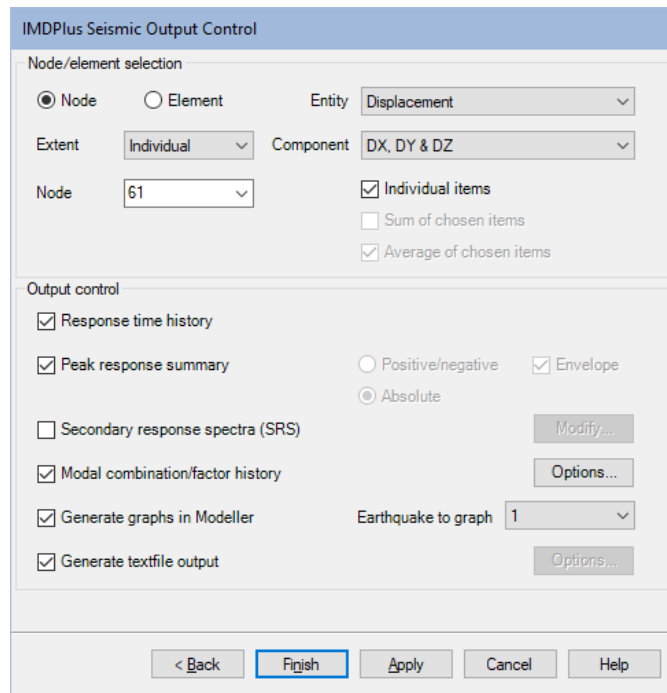
- Click on the **Modify...** button to change the advanced solution options. Click the **Defaults** button to set the default options and click the **OK** button.
- Select the **Linear interpolation of records** option and enter a time step of **0.0025** (Half the time step of the original earthquake 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. The options ‘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**
- Ensure **Generate graphs in Modeller** is selected
- Select **Generate textfile output**

The dialog should now look like this:

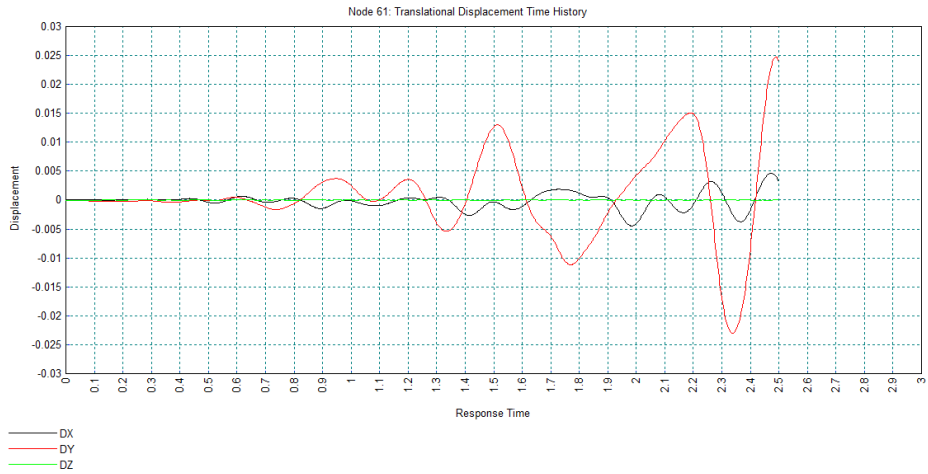


The image shows the 'IMDPlus Seismic Output Control' dialog box. It is divided into two main sections: 'Node/element selection' and 'Output control'. In the 'Node/element selection' section, 'Node' is selected with radio buttons, 'Extent' is set to 'Individual', 'Entity' is 'Displacement', 'Component' is 'DX, DY & DZ', and 'Node' is '61'. Checkboxes for 'Individual items', 'Sum of chosen items', and 'Average of chosen items' are present. The 'Output control' section has checkboxes for 'Response time history', 'Peak response summary', 'Secondary response spectra (SRS)', 'Modal combination/factor history', 'Generate graphs in Modeller', and 'Generate textfile output'. There are also radio buttons for 'Positive/negative' and 'Absolute', and a checked 'Envelope' checkbox. Buttons for 'Modify...', 'Options...', and 'Options...' are available. At the bottom, there are buttons for '< Back', 'Finish' (highlighted with a blue border), 'Apply', 'Cancel', and 'Help'.

- Click the **Finish** button to perform the analysis.
- On completion, choose **No** to the question about saving disk space.

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

Seismic Analysis of a 3D Frame (Time Domain)



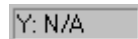
Peak displacements are also output to the Notepad application on screen and indicate absolute peak displacements of 0.0047 m in the X-direction, 0.0248 m in the Y-direction along with additional output for the Z-direction and rotations about each of these axes. From this output we can identify times 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    0.2475375968E-001
Peak_Neg_DY       2.337500000    -0.2302464294E-001
Abs_Peak_DY       2.490000000    0.2475375968E-001
Peak_Pos_DZ       1.082500000    0.6606589285E-004
Peak_Neg_DZ       1.015000000    -0.9454571680E-004
Abs_Peak_DZ       1.015000000    0.9454571680E-004
Peak_Pos_THX      2.327500000    0.5261706717E-004
Peak_Neg_THX      2.500000000    -0.5465704935E-004
Abs_Peak_THX      2.500000000    0.5465704935E-004
Peak_Pos_THY      2.470000000    0.1598728151E-004
Peak_Neg_THY      2.002500000    -0.1607368862E-004
Abs_Peak_THY      2.002500000    0.1607368862E-004
Peak_Pos_THZ      2.380000000    0.4129028139E-003
Peak_Neg_THZ      2.480000000    -0.4321706509E-003
Abs_Peak_THZ      2.480000000    0.4321706509E-003
Abs_Peak_RSLT     2.490000000    0.2508876354E-001
```

- Close the Notepad application and graph window.
- On completion, choose **No** to the question about saving disk space.

Average and Peak Displacements for a Selection of Nodes

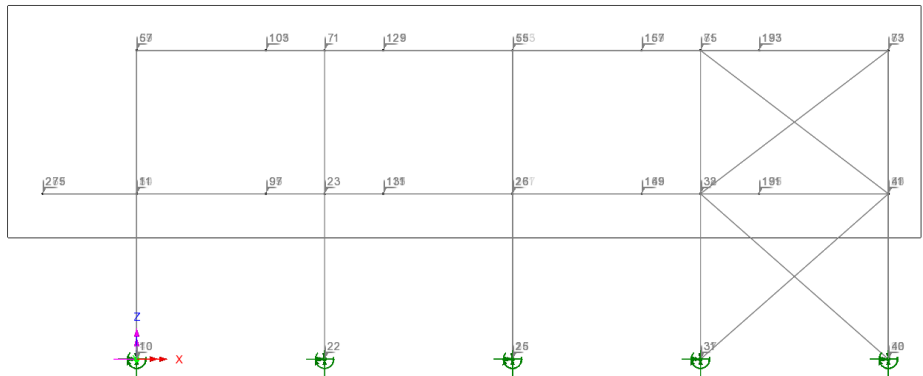
This section investigates results for a chosen set of nodes containing all of the unrestrained nodes in the model, that is, all of the nodes above ground level. The average displacements and peak displacements of the selected nodes will be obtained.

 Y: N/A

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



Using the **Select Nodes** cursor, drag a box to select all nodes above ground level as shown in the following figure. 89 nodes will be selected (but this is not reported).



Analyses
IMDPlus >
Seismic...

-  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**

Seismic Analysis of a 3D Frame (Time Domain)

- 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 ensure the **Envelope** option is selected also.
- 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 ☐ 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 **Apply** button to perform the analysis. Click **Yes** when asked whether you want to process all of the selected nodes.

•

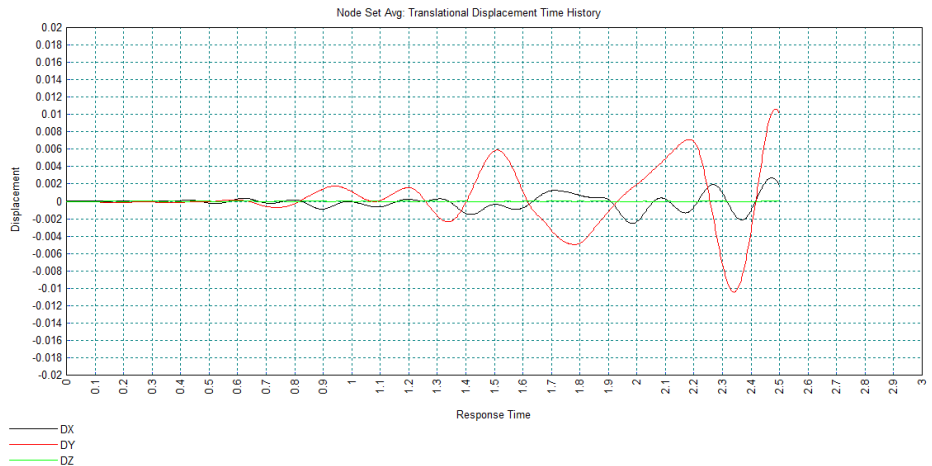


Note. Any combination of the options of **Individual items**, **Sum of chosen items** and **Average of chosen items output** can be used together in an IMDPlus analysis, although using the **Individual items** option when **Extent** is set to **Selected** or **All** (nodes or elements) may produce a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

Seismic Analysis of a 3D Frame (Time Domain)

```
#
# +-----+
# | Record = 001 |
# +-----+
#
# Entity          Time          Value
Peak_Pos_DX       2.472500000    0.2711970454E-002
Peak_Neg_DX       1.982500000    -0.2545200709E-002
Abs_Peak_DX       2.472500000    0.2711970454E-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       0.990000000    0.4570291161E-004
Peak_Neg_DZ       1.015000000    -0.8529296654E-004
Abs_Peak_DZ       1.015000000    0.8529296654E-004
Peak_Pos_THX      2.337500000    0.3003976047E-003
Peak_Neg_THX      2.492500000    -0.3044003640E-003
Abs_Peak_THX      2.492500000    0.3044003640E-003
Peak_Pos_THY      2.475000000    0.4385270549E-004
Peak_Neg_THY      1.990000000    -0.4369077447E-004
Abs_Peak_THY      2.475000000    0.4385270549E-004
Peak_Pos_THZ      2.340000000    0.2687725883E-003
Peak_Neg_THZ      2.482500000    -0.3000152317E-003
Abs_Peak_THZ      2.482500000    0.3000152317E-003
Abs_Peak_RSLT     2.482500000    0.1156838441E-001
```

An envelope of the peak results for the chosen nodes is displayed in the Notepad file **peak_dspSet1.sum** and is shown in the following figure. The peak X-displacement of 0.0068m occurs at node 143 at time 2.475 seconds and the peak Y-displacement of 0.02475m occurs at node 135 at time 2.49 seconds.

```
#
# +-----+
# | Record = 001 |
# +-----+
#
# Entity          Time          Value          Node
Peak_Pos_DX       2.475000000    0.6821753781E-002    143
Peak_Neg_DX       2.372500000    -0.5593165795E-002    143
Abs_Peak_DX       2.475000000    0.6821753781E-002    143
Peak_Pos_DY       2.490000000    0.2475797789E-001    135
Peak_Neg_DY       2.337500000    -0.2302854975E-001    135
Abs_Peak_DY       2.490000000    0.2475797789E-001    135
Peak_Pos_DZ       2.477500000    0.9252463708E-003    277
Peak_Neg_DZ       1.995000000    -0.8662857635E-003    277
Abs_Peak_DZ       2.477500000    0.9252463708E-003    277
Peak_Pos_THX      2.342500000    0.3661382540E-002    23
Peak_Neg_THX      2.485000000    -0.3739308466E-002    23
Abs_Peak_THX      2.485000000    0.3739308466E-002    23
Peak_Pos_THY      2.477500000    0.4783863332E-003    277
Peak_Neg_THY      1.995000000    -0.4772056608E-003    277
Abs_Peak_THY      2.477500000    0.4783863332E-003    277
Peak_Pos_THZ      2.485000000    0.5087373279E-002    69
Peak_Neg_THZ      2.342500000    -0.5074594498E-002    69
Abs_Peak_THZ      2.485000000    0.5087373279E-002    69
Abs_Peak_RSLT     2.487500000    0.2551992386E-001    143
```



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.

- 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.



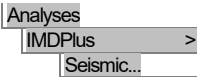
Note. Only elements of the same type, for example, thick beam elements, continuum elements or thick shell elements can be used in a set of IMDPlus elements. However, the element set may contain elements with different numbers of Gauss points or nodes.



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

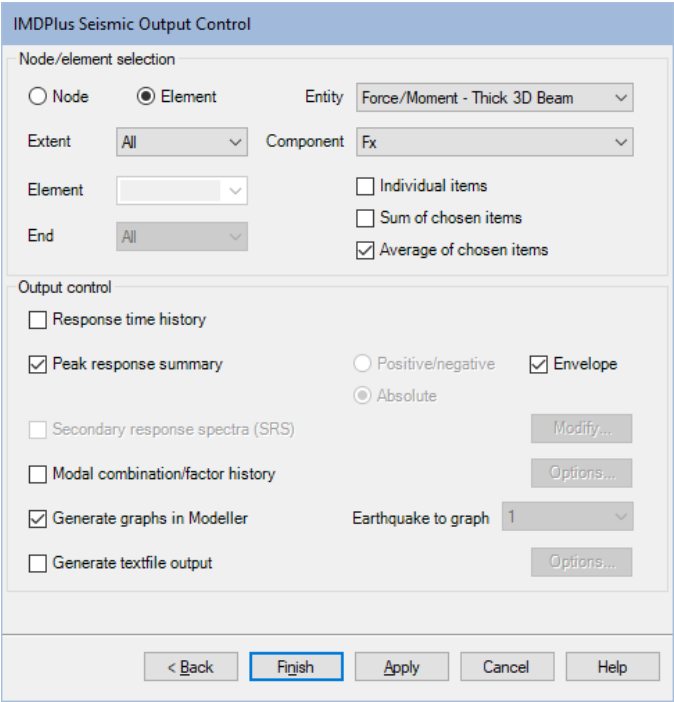


Note. As **Extent** has been set to **All** the drop-down lists **Element** and **End** are not available for selection. In addition the element **End** is automatically set to **All** as multiple elements are going to be processed in a single analysis.

- 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** are 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:

Seismic Analysis of a 3D Frame (Time Domain)



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, 'Element' is selected with a radio button, and the 'Entity' is set to 'Force/Moment - Thick 3D Beam'. The 'Extent' is set to 'All' and the 'Component' is set to 'Fx'. There are also fields for 'Element' and 'End' (set to 'All'). Checkboxes for 'Individual items', 'Sum of chosen items', and 'Average of chosen items' are present, with 'Average of chosen items' being checked. The 'Output control' section has checkboxes for 'Response time history' (unchecked), 'Peak response summary' (checked), 'Secondary response spectra (SRS)' (unchecked), 'Modal combination/factor history' (unchecked), 'Generate graphs in Modeller' (checked), and 'Generate textfile output' (unchecked). There are also radio buttons for 'Positive/negative' and 'Absolute', with 'Absolute' selected. A checked 'Envelope' checkbox is also present. Buttons for 'Modify...', 'Options...', and 'Options...' are available. At the bottom, there are buttons for '< Back', 'Finish' (highlighted with a blue border), 'Apply', 'Cancel', and 'Help'.

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

Positive/negative ☐ Absolute ☒ Envelope ☒

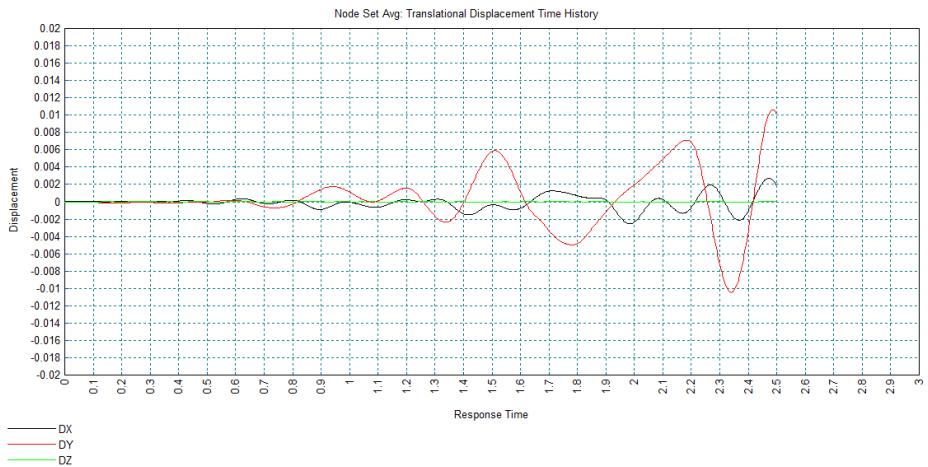
Modify... Options... Options...

Earthquake to graph: 1

< Back Finish Apply Cancel Help

- Click the **Finish** button to perform the analysis. Click **Yes** when asked whether you want to process all of the elements in the model.
- 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 a Notepad 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 (F_x) of -30.97kN occurs at end 1 of element 4 at time 2.3425 seconds. The peak bending moment (M_y) 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.

Seismic Analysis of a 3D Frame (Time Domain)

```
#
# +-----+
# | Record = 001 |
# +-----+
#
# Entity      Time      Value      Element      End
Peak_Pos_Fx   2.340000000  34149.26344    10           1
Peak_Neg_Fx   2.342500000 -30971.16846    4            1
Abs_Peak_Fx    2.340000000  34149.26344    10           1
Peak_Pos_Fy   2.482500000  15452.76533     1            1
Peak_Neg_Fy   2.342500000 -14411.27529     1            1
Abs_Peak_Fy    2.482500000  15452.76533     1            1
Peak_Pos_Fz   2.490000000  9383.87646    133           1
Peak_Neg_Fz   2.472500000 -10137.26444    12            1
Abs_Peak_Fz    2.472500000  10137.26444    12            1
Peak_Pos_Mx   2.485000000   174.0392983    47            1
Peak_Neg_Mx   2.485000000 -172.9316869    58            1
Abs_Peak_Mx    2.485000000   174.0392983    47            1
Peak_Pos_My   2.337500000  43015.97259    133           1
Peak_Neg_My   2.490000000 -46422.42600    133           1
Abs_Peak_My    2.490000000  46422.42600    133           1
Peak_Pos_Mz   2.482500000  27225.59879     1            2
Peak_Neg_Mz   2.482500000 -27322.66281     1            1
Abs_Peak_Mz    2.482500000  27322.66281     1            1
```

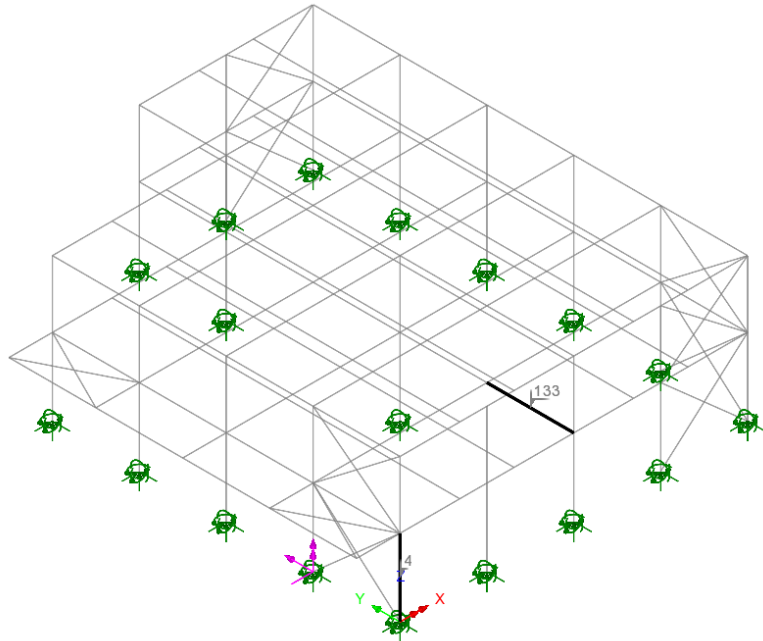
- Close the Notepad applications.

Identifying elements in the model

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

-  Click on the Isometric view button.
- Click the left-hand mouse button in a blank part of the view 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 deselect the **Node > Name** check box, then 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 view window.
- With no features selected click the right-hand mouse button in a blank part of the view window and select the **Advanced Selection...** option.
- For **Type and Name**, select **Element** from the drop-down list and enter element number **4**.
- Ensure **Add to selection** is chosen and click the **Apply** button to accept all other settings. Beam element 1 (the vertical member at the origin where the view axis are) will be highlighted in the view window.

- Enter element number **133** and click the **OK** button. Beam element 42 will also be displayed in the view window, as shown in the following figure.

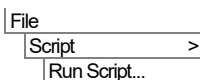


Displaying Deformed Mesh Plots from Modal Combinations

When the IMDPlus analysis of node 61 at the top and in the centre of the frame was carried out in an earlier section, the Modal combination/factor history option was selected and caused IMDPlus to generate a command file 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 your **\Documents\LUSAS220\Projects\IMDPlus 3D Frame\LUSASFiles\IMDPlus 3D Frame** directory.
- Loading of the modal combinations may take some time. Click **Yes** to load all time steps.

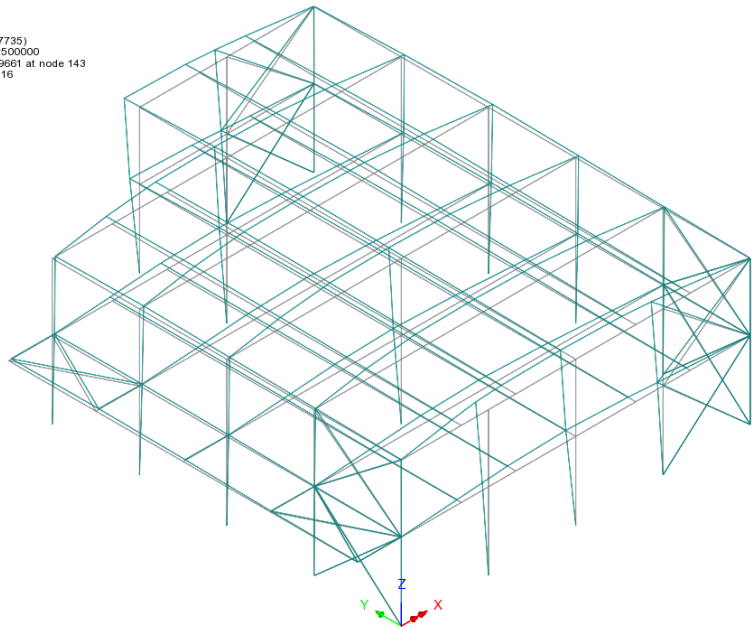
The combinations will appear under the original model in the  Treeview.

Seismic Analysis of a 3D Frame (Time Domain)

- Click the left-hand mouse button in a blank part of the view 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**
- With no features selected click the right-hand mouse button in a blank part of the View window and select the **Deformed Mesh** option to add the deformed mesh layer to the  Treeview. Click the **OK** button.
- 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 **(15,-30)** and click the **OK** button to return to the view 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 at the top and in the centre of the frame. The deformed mesh will now be displayed for this instance in time as shown below.

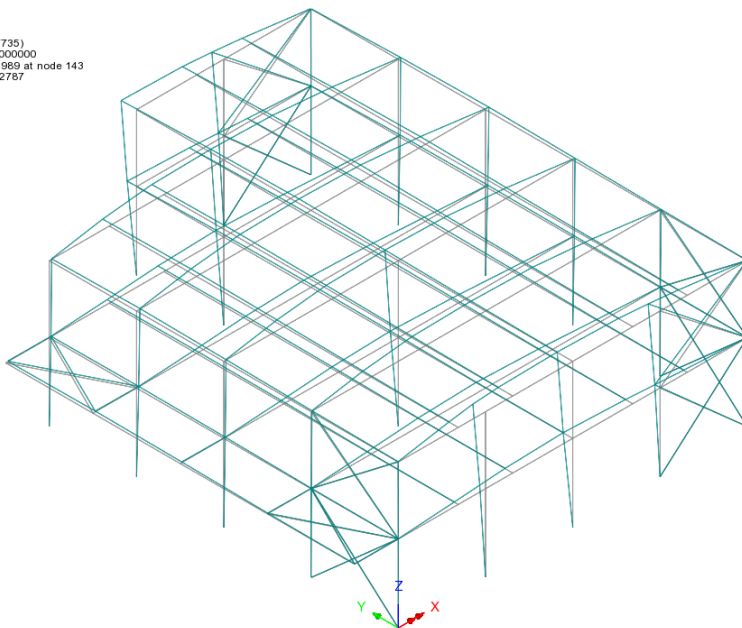
Scale: 1: 124.464
Zoom: 100.0
Eye: (-0.57735, -0.57735, 0.57735)
IMG> Response Time = 2.472500000
Maximum displacement 0.0239661 at node 143
Deformation exaggeration: 31.16



- 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 at the top and in the centre of the frame. The deformed mesh will now be displayed for this instance in time as shown below.

Scale: 1: 124.429
Zoom: 100.0
Eye: (-0.57735, -0.57735, 0.57735)
IMD+ Response Time = 2.490000000
Maximum displacement 0.0264969 at node 143
Deformation exaggeration: 29.2787



Note. Results 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.

Analyses

Envelope...

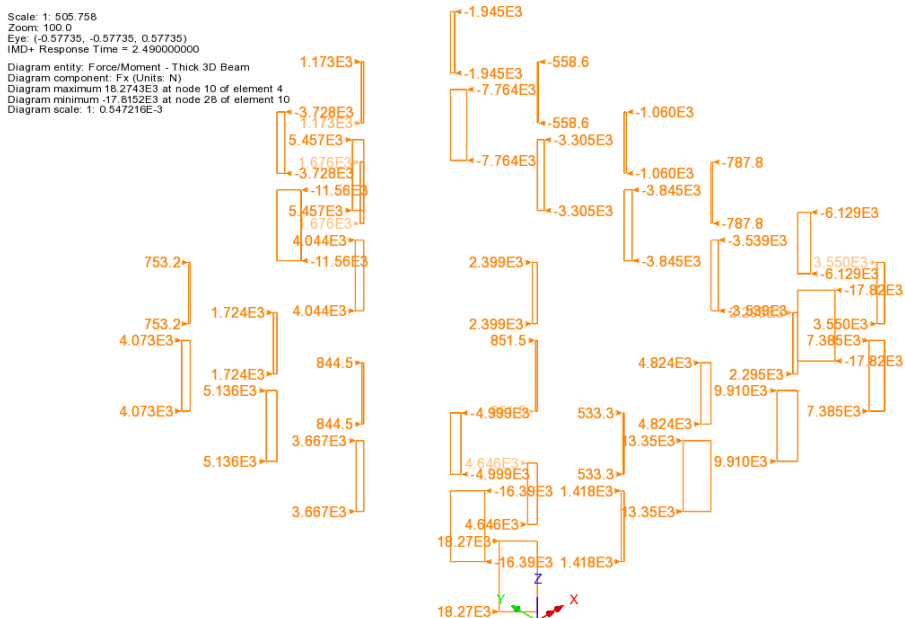
This will allow you to create an envelope.

- 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.

Seismic Analysis of a 3D Frame (Time Domain)

- 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 the view window, click the right-hand mouse button and select the **Diagrams** option to add the Diagrams layer to the  Treeview.
- Select **Force/Moment** results of axial force **Fx** in the members. Select the **Diagram display** tab, ensure the **Label values** option is selected. Plot values on **80%** of the element length.
- Click the **OK** button.

This shows the axial forces in the columns for the current loadcase (Response Time = 2.490000000)

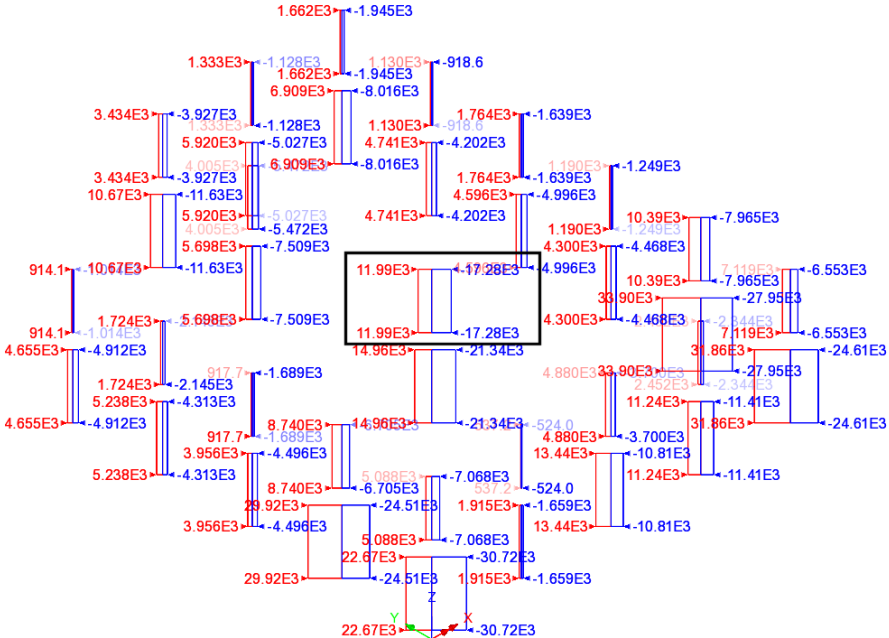


- In the  Treeview right-click on the **Whole Earthquake (Max)** envelope and select the **Set Active** option to set this envelope active.
- On the Set Active dialog select **Force/Moment** results of axial force **Fx** and click the **OK** button to finish.

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

- In the  Treeview, double-click on the **Diagrams** option, and from the Envelopes and Combinations panel select **Show minimum too**.

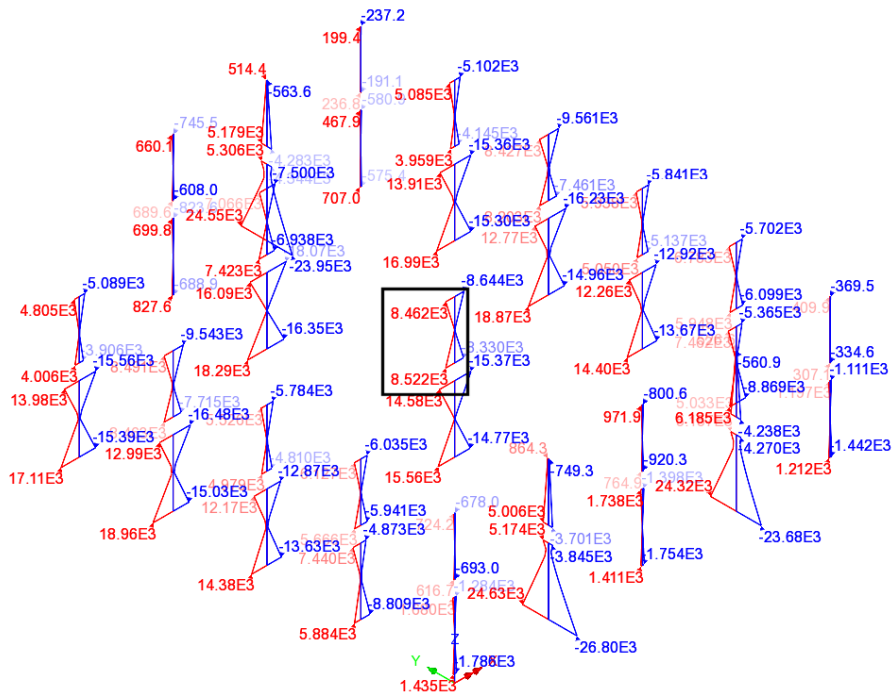
It should be seen that, for the central column F_x (Max) = 11.99e3 and F_x (Min) = -17.28e3.



- 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** 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 should be seen that, for the central column My (Max) at top = 8.462e3 and My (Max) at bottom = 8.522e3.



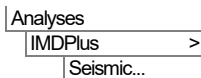
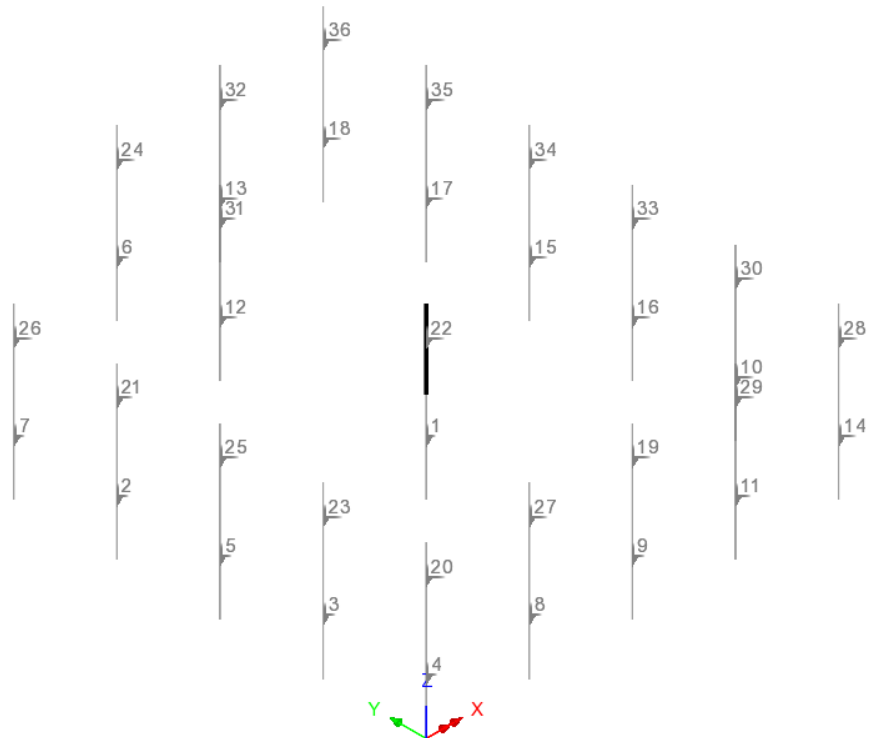
Displaying results for a selected member

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.

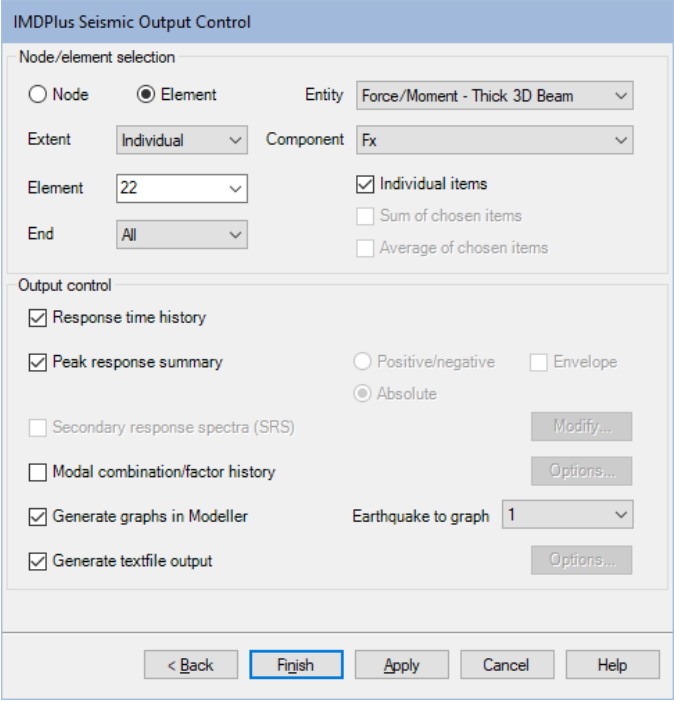
- Double-click on the **Labels** layer name in the  Treeview. On the Properties dialog select the **Element name** check box and click **Close**.
-  Select the element in the centre of the frame as shown in the following diagram.



- 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.

Seismic Analysis of a 3D Frame (Time Domain)

- Ensure **Generate graphs in Modeller** and **Generate textfile output** remain selected.



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, 'Element' is selected with a radio button, 'Entity' is set to 'Force/Moment - Thick 3D Beam', 'Extent' is 'Individual', 'Component' is 'Fx', 'Element' is '22', and 'End' is 'All'. Under 'Individual items', 'Individual items' is checked, while 'Sum of chosen items' and 'Average of chosen items' are unchecked. The 'Output control' section has 'Response time history' and 'Peak response summary' checked. 'Secondary response spectra (SRS)' and 'Modal combination/factor history' are unchecked. For 'Peak response summary', 'Absolute' is selected with a radio button, while 'Positive/negative' and 'Envelope' are unchecked. There are 'Modify...' and 'Options...' buttons next to these options. 'Generate graphs in Modeller' and 'Generate textfile output' are both checked. 'Earthquake to graph' is set to '1'. There are 'Options...' buttons next to these two options. At the bottom, there are buttons for '< Back', 'Finish' (which is highlighted with a blue border), 'Apply', 'Cancel', and 'Help'.

- 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).
- 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 file in the format indicated below.

```

#
# +-----+
# | Record = 001 |
# +-----+
#
# Entity                Time                Value
Peak_Pos_Fx1            1.082500000            12278.77265
Peak_Neg_Fx1            1.015000000            -17680.15871
Abs_Peak_Fx1            1.015000000            17680.15871
Peak_Pos_Fx2            1.082500000            12278.77265
Peak_Neg_Fx2            1.015000000            -17680.15871
Abs_Peak_Fx2            1.015000000            17680.15871
Peak_Pos_Fy1            2.500000000            14770.86991
Peak_Neg_Fy1            2.330000000            -13276.07028
Abs_Peak_Fy1            2.500000000            14770.86991
Peak_Pos_Fy2            2.500000000            14770.86991
Peak_Neg_Fy2            2.330000000            -13276.07028
Abs_Peak_Fy2            2.500000000            14770.86991
Peak_Pos_Fz1            1.982500000            5579.077650
Peak_Neg_Fz1            2.475000000            -5871.017302
Abs_Peak_Fz1            2.475000000            5871.017302
Peak_Pos_Fz2            1.982500000            5579.077650
Peak_Neg_Fz2            2.475000000            -5871.017302
Abs_Peak_Fz2            2.475000000            5871.017302
Peak_Pos_Mx1            2.285000000            1.401093705
Peak_Neg_Mx1            2.195000000            -0.9637896459
Abs_Peak_Mx1            2.285000000            1.401093705
Peak_Pos_Mx2            2.285000000            1.401093705
Peak_Neg_Mx2            2.195000000            -0.9637896459
Abs_Peak_Mx2            2.285000000            1.401093705
Peak_Pos_My1            2.475000000            8924.422607
Peak_Neg_My1            1.982500000            -8469.979247
Abs_Peak_My1            2.475000000            8924.422607
Peak_Pos_My2            1.982500000            8601.998361
Peak_Neg_My2            2.475000000            -9040.890337
Abs_Peak_My2            2.475000000            9040.890337
Peak_Pos_Mz1            2.330000000            20325.83032
Peak_Neg_Mz1            2.500000000            -22609.55755
Abs_Peak_Mz1            2.500000000            22609.55755
Peak_Pos_Mz2            2.500000000            22589.30438
Peak_Neg_Mz2            2.330000000            -20298.94472
Abs_Peak_Mz2            2.500000000            22589.30438

```

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.