

# Seismic Response of a 2D Frame (Frequency Domain)

For LUSAS version:	19.0
For software product(s):	All (except LT versions)
With product option(s):	None.

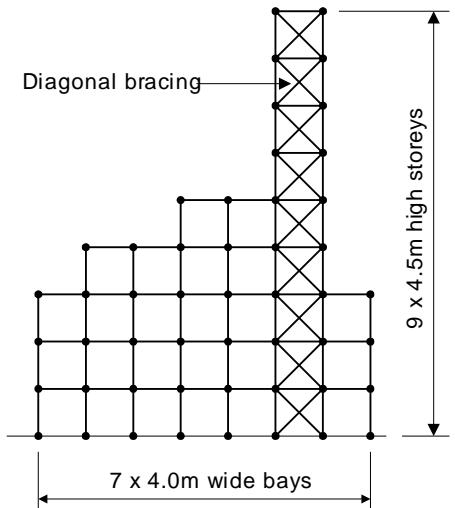
## Description

This example examines the Spectral Response analysis of a 2D braced tower frame. The geometry is simplified to a wire-frame or stick representation, with each structural member being represented by Point and Line features only.

The model is comprised of thick beam elements for the concrete column, beam members, and steel diagonal bracing members which have pinned end connections. The structure is fully restrained against displacement and rotation at ground level.

Since the global response of the structure is required, the model of the tower is further simplified by meshing each Line with a single element. This will effectively avoid the extraction of local panel modes for individual beams and columns. These local modes could be investigated independently in a more detailed analysis.

The spectral response analysis is performed in three distinct stages. Firstly a natural frequency analysis is performed. This is used here to calculate the first 10 natural



modes of vibration of the structure. The eigenvalues, frequencies and eigenvectors (mode shapes) are stored and used in the subsequent spectral response analysis. Although the 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 is the spectral response calculation, in which the spectral response computation is performed interactively as a results processing operation using the Interactive Modal Dynamics (IMD) facility. This is an alternative to performing a non-interactive spectral response analysis in LUSAS.

In a spectral response analysis, the structure is subjected to a support condition excitation. In this example this is assumed to be the effect of Seismic/Earthquake motion although any support motion could be envisaged. The excitation is specified as a spectral curve, in terms of displacement, velocity or acceleration and represents an envelope of these effects over a measured frequency range. Damping may also be specified at this stage.

From the spectral analysis participation factors are obtained which indicate, for each mode, the degree of structural response associated with this applied excitation.

The last phase of the analysis considers self weight in conjunction with the spectral response and to this end two load combinations are defined from which worst case force/moment envelopes may be obtained.

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

### Keywords

**2D, Inplane, Frame, Beam, End Releases, Fleshing, Natural Frequency, Eigenvalue, Spectral Response, Seismic, Axial Force, Shear Force, Bending Moment Diagrams, Participation Factors, Modal Dynamics, Interactive Modal Dynamics (IMD), Modal Response, Global Response, Graphing, Animation, Load Combination.**

### Associated Files



- tower\_modelling.vbs** carries out the example modelling.
- tower\_spectrum.vbs** defines a typical spectral curve excitation.

## 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.



**Note.** This example is written assuming a new LUSAS Modeller session has been started. If continuing from an existing Modeller session select the menu command **File>New** to start a new model file. Modeller will prompt for any unsaved data and display the New Model dialog.

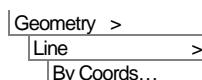
## Creating a new model

- Enter a File name of **tower**
- Use the default User-defined working folder.
- Ensure an Analysis type of **Structural** is set.
- Select an Analysis Category of **2D Inplane**
- Set Model units of **N,m,kg,s,C**
- Leave the Timescale units as **Seconds**
- Select a Startup template of **2D Frame**
- Ensure the Layout grid is set as **None**
- Enter a Title of **Tower**
- Click the **OK** button.



**Note.** Save the model regularly as the example progresses. Use the Undo button to correct any mistakes made since the last save was done.

## Defining the Column and Beam Members



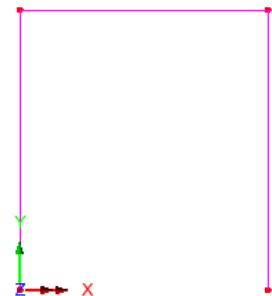
 Enter coordinates of **(0, 0)**, **(0, 4.5)**, **(4, 4.5)** and **(4, 0)** and click **OK** to define the first bay of the ground level storey.



**Note.** Sets of coordinates must be separated by commas or spaces unless the 'Grid Style' method is chosen. The **Tab** key is used to create new entry fields. The arrow keys are used to move between entries.

### Meshing - Columns and Beams

As the global response of the structure is required only one beam element need be used along each line. This will prevent LUSAS from picking up any panel modes caused by local vibration of the columns and beams.



- Select **Thick Beam, Linear** elements.
- Enter the number of divisions as **1**
- Enter the attribute name as **Thick beam, linear order, div = 1** and click **OK**
- With the whole model selected, drag and drop the Line mesh attribute Thick Beam from the  Treeview onto the selected features.

### Geometric Properties - Columns and Beams

Attributes  
Geometric >  
Line...

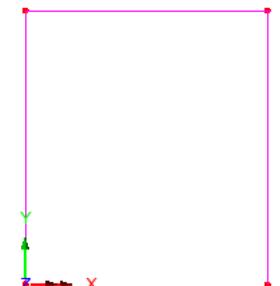
- From the Usage section select **2D Thick Beam (any beam)** from the drop down list and change the values on the dialog to match those shown in the table for a column member.
- Enter the attribute name as **Concrete Column** and select the **Apply** button to create the attribute and re-use the dialog.
- Change the values on the dialog to match those shown in the table for a beam member.
- Change the attribute name to **Concrete Beam** and click **OK**

Geometric property	Column Member	Beam Member
Cross-sectional area	0.1	0.08
Second moment of area about z axis Izz)	0.833e-3	0.533e-3
Effective Shear Area in Y Direction (Asy)	0.1	0.08
Offset in y direction (Ry)	0	0



**Note.** Whilst in practice column sizes would vary with height and beam sizes may vary throughout the structure, in this example, for simplicity, all of the concrete columns and beam sections are of the same size and assumed to be of solid square section.

- Select the 2 vertical lines representing the column members. (Hold the **Shift** key down to add to the initial selection).
- Drag and drop the geometry attribute **Concrete Column** from the  Treeview onto the selected features.
- Select the horizontal line representing the Beam.
- Drag and drop the geometry attribute **Concrete Beam** from the  Treeview onto the selected line.



Geometric property assignments are visualised by default. In general the diagrams in this example will not show geometric visualisations.

## Material Properties - Columns and Beams

An ungraded concrete material attribute is provided in the  Treeview.

- Select the whole model by either using the **Ctrl** and **A** keys to select all features or by dragging a box around the model.
- Drag and drop the material attribute **Concrete (Ungraded | Concrete)** from the  Treeview onto the selected features.

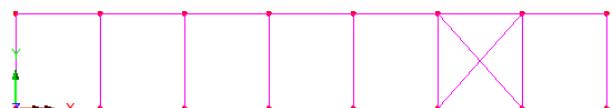
## Copying the Column and Beam Members

Now that one bay has been fully defined it will be copied to form the other 6 bays of the ground storey of the tower. With the whole model selected:

Geometry  
Line  
Copy...

 Enter a translation in the **X** direction of **4** metres.

- Enter the number of copies required as **6**
- Click the **OK** button to create 6 additional bays.



Bay 6 of the structure requires diagonal bracing to be added with the ends of both members being free to rotate about the **Z** direction in order to model a pinned connection.

### Defining the Bracing Members

- Select 2 diagonally opposite points in bay 6. (Hold the **Shift** key down to add to the selection).



The line will be drawn.

- Repeat for the other pair of diagonally opposite Points.

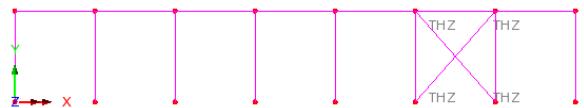
### Meshing - Bracing Members

Only one beam element is to be used along each bracing member. However, the ends of the bracing members must be free to rotate about the Z direction in order to model a pinned connection, therefore the existing mesh dataset (as used for the beam and column members) cannot be used. A new mesh dataset must now be created from the existing one.

- In the Treeview double-click the **Thick beam, linear order, div = 1** attribute name.
- On the line mesh end conditions dialog press the **End conditions** button and for the start of the line select the **Rotation about Z** option. Ensure that 'End of line' is set to 'Same as the start of line' and click the **OK** button to return to the Line mesh dialog.
- Change the dataset name to **Thick Beam (Pinned Ends)** and click the **OK** button.
- With the 2 lines representing the bracing members selected, drag and drop the line mesh attribute **Thick Beam (Pinned Ends)** from the Treeview onto the selected features.



**Note.** The visualisation of the beam end releases is shown by default. If it is required to turn-off their visualisation select the Treeview and double click on **Mesh**, select the Visualise tab and deselect the **Beam end releases** option.

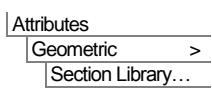


### Geometric Properties - Bracing members

For the purposes of this example the bracing members are to be defined as Universal Beams.

The standard sections dialog will appear.

- From the **UK Sections** library select the **Universal Beams (BS4)** section type and select the **254x146x43kg UB** section.



- Enter an attribute name of **Bracing Member** and click the **OK** button to add the Universal Beam attribute to the  Treeview.
- Select the 2 diagonal lines representing the bracing members.
- Drag and drop the geometry attribute **Bracing member (254x146x43kg UB major z)** from the  Treeview onto the selected features.

 Select the fleshing on/off button to turn-off the geometric visualisation. If at any time during the example you wish to visualise the geometry select this button.

## Material Properties - Bracing Members

Attributes  
Material >  
Material Library...

- Select material **Steel - Structural** from the drop down list, select grade **Mild steel** and click **OK** to add the material attribute to the  Treeview.
- With the 2 bracing members selected, drag and drop the Material attribute **Iso2 (Mild Steel Ungraded)** from the  Treeview onto the selected features.

The first bay of the ground storey of the tower is now completely defined. The support conditions are not defined or assigned yet because this bay will be copied upwards to create other storeys where supports will not be required.

## Copying the Model Features

The other storeys of the tower can now be generated by copying selected features through specified distances in the Y direction.

- Select the whole model using the **Ctrl** and **A** keys together.

Geometry  
Line >  
Copy...

 Enter a translation in the **Y** direction of **4.5** metres. Enter the attribute name as **Y=4.5**. Click the **Save** button to save the attribute for re-use. Enter the number of copies required as **2** and click the **OK** button to finish and generate two additional storeys.



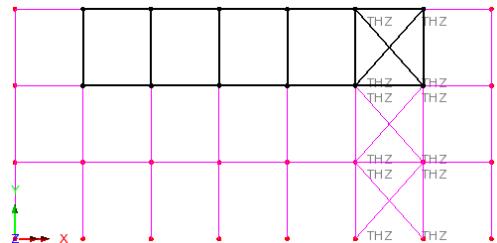
**Note.** All attributes assigned to the features on the ground level storey, for example, line mesh, geometric and material properties will be duplicated on the other storeys automatically when the new features are created by copying.

## Seismic Response of a 2D Frame (Frequency Domain)

---

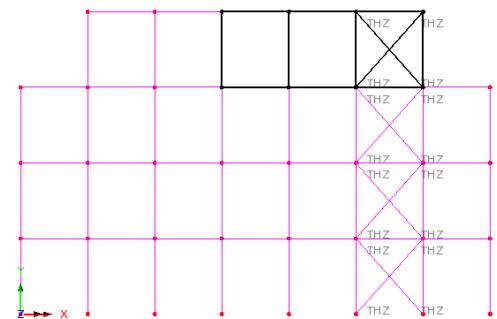
- Drag a box around the 5 bays shown highlighted to create the next level of the structure.

 Select the attribute **Y=4.5** from the drop down list and click the **OK** button to create 1 copy of the features selected.

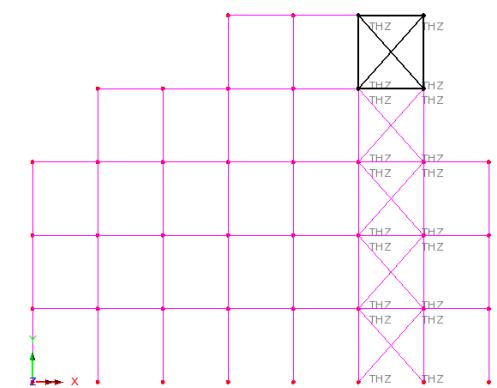


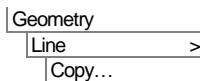
- Next, drag a box around the 3 bays highlighted to create the next level of the structure.

 Select the attribute **Y=4.5** from the drop down list and click the **OK** button to create 1 copy of the features selected.



- Next, drag a box around the single bay highlighted. This will be used to create the remaining levels of the structure.





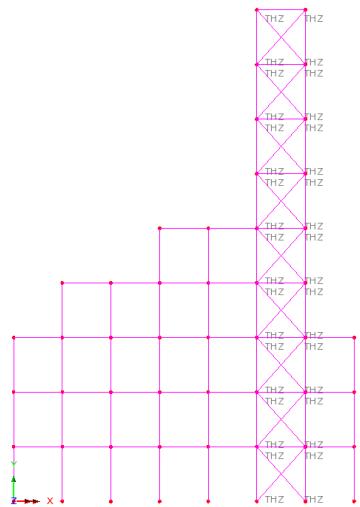
Select the attribute **Y=4.5**

- Change the number of copies to **4**
- Click the **OK** button to create the remaining levels.

## Supports

LUSAS provides the more common types of support by default. These can be seen in the Treeview.

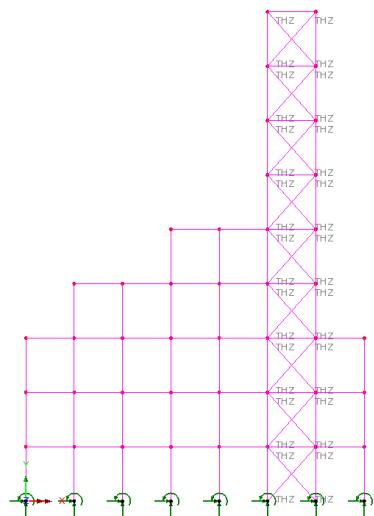
The tower must be fully restrained at the base, therefore 3 degrees of freedom (U, V and rotation about the Z axis) must be restrained. A fully fixed support type is therefore required.



## Assigning the Supports

- Drag a box around the 8 Points along the base of the structure.
- From the Treeview drag and drop the support attribute **Fully Fixed** onto the selected features, ensure the **All analysis loadcases** option is selected and click **OK**

The model geometry is now complete.



## Loading



**Note.** No static structural loading is required for the first part of this analysis because the spectral combination is carried out during results processing using the results from the natural frequency analysis which is independent of applied loading. The modelling will now be completed by defining the controls necessary to extract the natural frequencies.

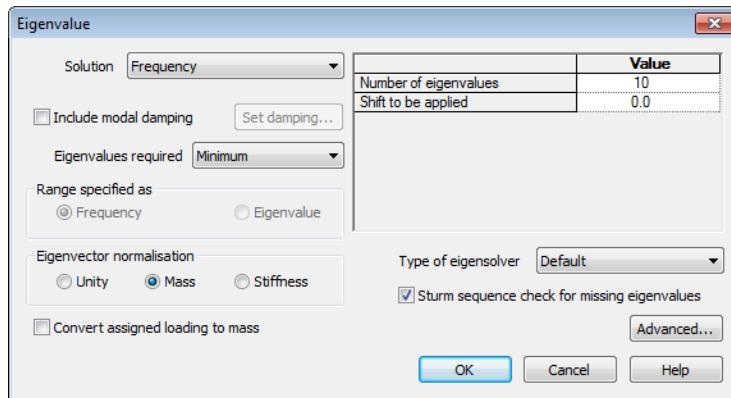
### Eigenvalue Analysis Control

Since the first stage of a spectral response analysis involves a computation of the natural modes of vibration an initial eigenvalue extraction analysis must be carried out. The solution parameters for eigenvalue analysis are specified using an eigenvalue control dataset. In this example the first 10 natural modes of vibration of the tower are computed.

Eigenvalue analysis control properties 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.



The following parameters need to be specified:

- Set the **Number of eigenvalues** as **10**
- 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 when results processing.

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

### Saving the model



## Running the Analysis

With the model loaded:



Open the **Solve Now** dialog. Ensure **Analysis 1** is selected and press **OK** to run the analysis.

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

### If the analysis is successful...

Analysis loadcase results are added to the Treeview

In addition, 2 files will be created in the new Associated Model Data directory where the model file resides:



- tower.out** this output file contains details of model data, assigned attributes and selected statistics of the analysis.
- tower.mys** this is the LUSAS results file which is loaded automatically into the Treeview to allow results processing to take place.

### If the analysis fails...

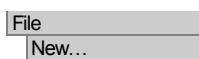
If the analysis fails, information relating to the nature of the error encountered can be written to an output file in addition to the text output window. Any errors listed in the text output window should be corrected in LUSAS Modeller before saving the model and re-running the analysis.

### Rebuilding a Model

If it proves impossible for you to correct the errors reported a file is provided to enable you to re-create the model from scratch and run an analysis successfully.

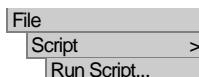


- tower\_modelling.vbs** carries out the modelling of the example.



Start a new model file. If an existing model is open Modeller will prompt for unsaved data to be saved before opening the new file.

- Enter the file name as **tower**
- To recreate the model, select the file **tower\_modelling.vbs** located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory.





Rerun the analysis to generate the results.

## Viewing the Results

Eigenvalue loadcase results are present in the

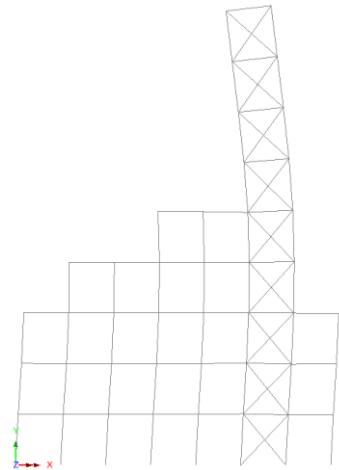
 Treeview, with Eigenvalue 1 set to be active by default.

### Plotting Mode Shapes

- Turn off the display of the **Mesh**, **Geometry** and **Attributes** layers in the Treeview.
- By default the **Deformed mesh** layer is visible.  
To see other mode shapes:
- In the Treeview right-click on **Mode 2 Frequency = <value>** and select the **Set Active** option. The deformed mesh plot for Eigenvalue 2 will be displayed.
- By setting each Eigenvalue to be active the deformed mesh can be seen for all mode shapes.



**Note.** The mode shape for eigenvalue 1 may be inverted. This is because the sense is arbitrary since during vibration deformed shape will appear in both directions.



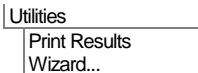
### Printing Eigenvalue Results

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

---

- Select results type **Eigenvalues**.
- In the Results panel, for **Loadcase 1** 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...



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

	Mode	Eigenvalue	Frequency	Period	Error norm
1	1	147.282	1.9315	0.517732	1.24145E-12
2	2	1.52716E3	6.2196	0.160782	0.178321E-12
3	3	7.48077E3	13.7655	0.0726451	0.215481E-12
4	4	12.4645E3	17.7688	0.0562784	0.240758E-12
5	5	16.4644E3	20.4218	0.0489674	0.121916E-12
6	6	26.0145E3	25.6701	0.0389558	0.577943E-12
7	7	31.9302E3	28.4395	0.0351624	5.17683E-12
8	8	33.871E3	29.291	0.0341402	15.8402E-9
9	9	34.9382E3	29.7489	0.0336147	0.107775E-6
10	10	35.9895E3	30.1931	0.0331201	0.501761E-6


**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.

## Spectral Response Analysis

Spectral response calculations are performed using the IMD (Interactive Modal Dynamics) facility. Setting up the spectral response results involves the following steps:

- Define the Spectral Excitation curve.
- Define the dynamic excitation type (support motion acceleration in this example), direction, and specify the type of results required using a CQC (Combined Quadratic Combination) for spectral response.
- Set the IMD loadcase to be active.

The spectral combination results can be displayed using a deformed shape plot or force and moment diagrams.



**Note.** Additional damping information may also be set. Unlike results from a natural frequency analysis, the output values obtained from a spectral combination are design values.

## Stage 1 - Defining a Spectral Excitation Curve

Values for the spectral excitation curve need to be specified. For this analysis Acceleration versus Period values are to be used with a damping value of 5%.



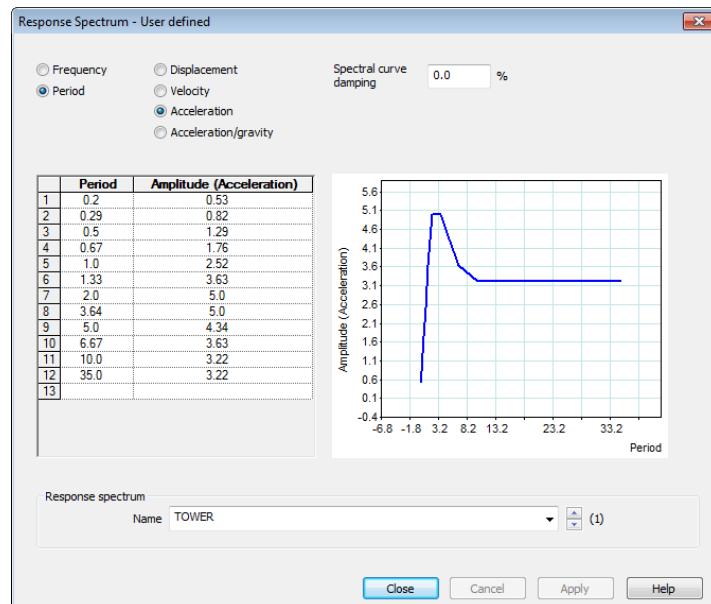
**Note.** The Utilities > Response Spectrum menu item would normally be used to define response spectra data. However, to simplify entering the required data, the full response spectrum definition for this example is contained in a command file.

## Seismic Response of a 2D Frame (Frequency Domain)

- In the `\<LusasInstallationFolder>\Examples\Modeller` directory select the file `tower_spectrum.vbs` and click OK

A response spectra dataset **TOWER** and corresponding graph datasets containing the spectral data for period and acceleration will be created in the  Treeview.

- Double-click on the **TOWER** response spectra entry to view the curve definition of the response spectrum.

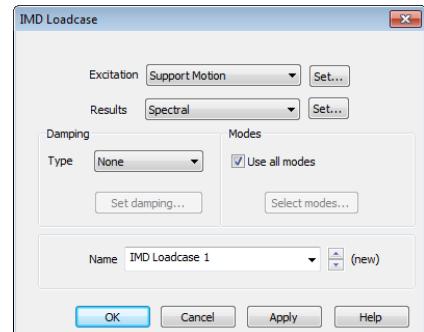


- Click **Close** after viewing

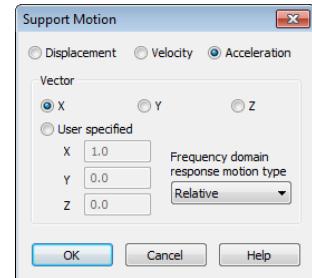
## Stage 2 - Defining the Dynamic Excitation

- Select **Support Motion** from the Excitation drop down list and **Spectral** from the Results drop down list. Ensure the **Use all modes** option is selected.

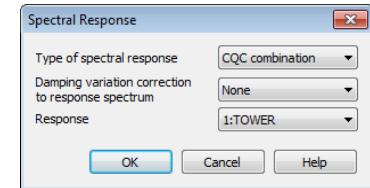
]



- Click the Excitation Set... button.
- On the Support Motion dialog select **Acceleration** support motion option in the **X** direction which is **Relative**, and click the **OK** button to return to the IMD loadcase dialog.



- On the IMD loadcase dialog click the Results Set... button.
- On the Spectral Response dialog set the type of spectral response to **CQC Combination** with the damping variation correction set as **None**. The response spectrum **TOWER** read in from the command file will already be selected in the drop down list. Click the **OK** button to return to the IMD loadcase dialog.
- On the IMD loadcase dialog change the IMD loadcase name to be **IMD Spectral Response**, and click the **OK** button to finish.



## Selecting the IMD Results Loadcase

- In the  Treeview right-click on the **IMD Spectral Response** loadcase name and select the **Set Active** option.

The spectral combination results are now activated. Any subsequent results plots, such as deformed shapes or moments, will be for the spectral combination results only.

## Using Page Layout Mode

The model was created using a Working Mode view which allows a model of any size to be created. Whilst previous results have been viewed using this mode of operation, in order to allow additional information to be added without obscuring the model, Page Layout Mode will be used instead.



The view window will resize to show the mesh layer on an A4 size piece of paper.



- Set the page layout to **Portrait** and click **OK**

### Displaying Results for Spectral Combination

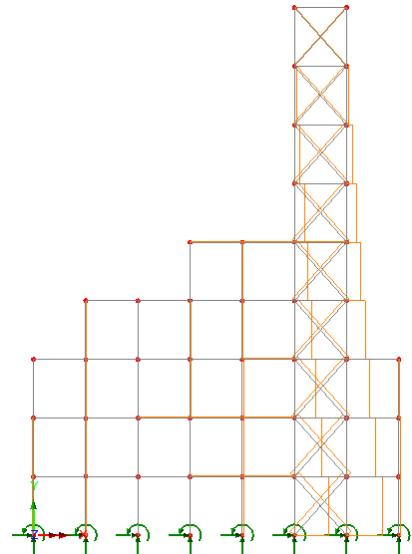
A plot showing the tower deformed shape, bending moment and axial force for the spectral combination is to be displayed.

- Turn-off the display of the **Deformed Mesh** and **Geometry** layers from the  Treeview.
- Turn-on the **Mesh** layer in the  Treeview.

### Axial Force Diagram

- With no features selected, click the right-hand mouse button in a blank part of the View window area 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 and choose the **Label only if selected** option.
- Click the **OK** button to finish.

Results will be calculated for the active IMD spectral response loadcase and an axial force diagram of stresses in each member will be displayed.



## Displaying results on selected members

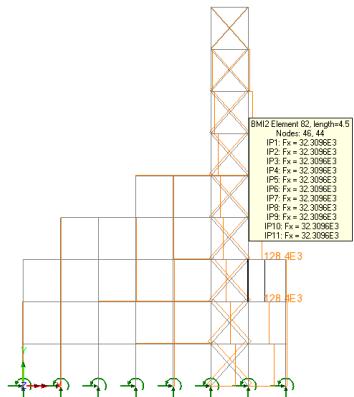
- Hovering over an element with the cursor will display previously chosen results information for gauss points along that member.
- Selecting a member will display results for that member.



**Note.** Results for more than one member can be displayed by holding down the **Shift** key, or by dragging a box around the members of interest with the cursor.



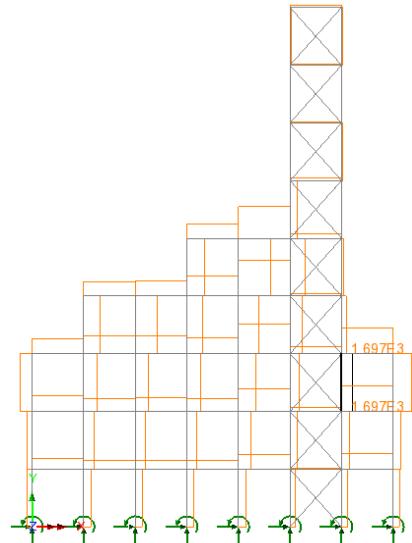
 If necessary, use the Zoom in button to view results information.



These types of element results can also be plotted in isolation on selected model features. This is covered later in the example.

## Shear Force Diagram

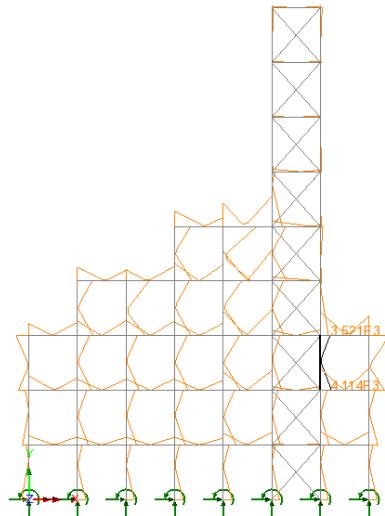
- In the  Treeview double-click on the **Diagrams** layer. The diagram properties will be displayed.
- Select **Force/Moment** results of shear force **Fy** in the members.
- Click the **OK** button to finish.
- A shear force diagram of stresses in each member will be displayed.



### Bending Moment Diagram

- In the  Treeview double-click on the **Diagrams** layer. The diagram properties will be displayed.
- Select **Force/Moment** results of bending moment **Mz** in the members.
- Click the **OK** button to finish.

A bending moment diagram of stresses in each member will be displayed.



### Printing Results



**Note.** Text and border annotation may be added to the display from the Utilities> Annotation menu.

This adds a border to the display.

Now view how the results would print.

This shows how the bending moment results for the whole model will appear on printed output.

- Click the **Close** button to exit the print preview window.

Removes the border from the display.



### Plotting Results for a Selected Part of the Model

The view window will resize to show the model in normal working mode.



**Note.** Changing from Page Layout to Working Mode will resize and reposition any displayed annotation. Page layout mode should be used when preparing a view for printing. Working Mode should be used when viewing on the screen.

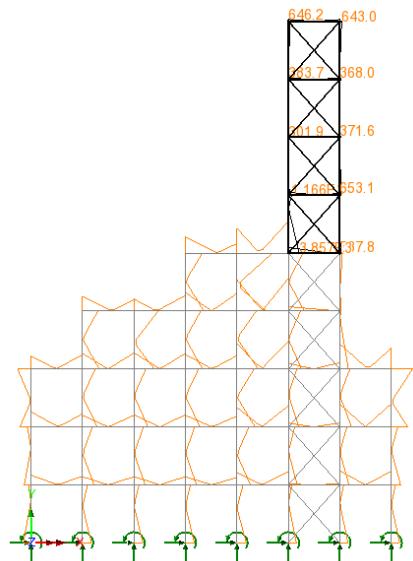
If results are to be plotted for a selected part of the structure only, a Group may be created of the area of interest. For example, to plot results of just the top 4 storeys of the tower:

- Drag a box around the upper 4 storeys of the tower as highlighted in the adjacent image.

Geometry  
Group >  
New Group

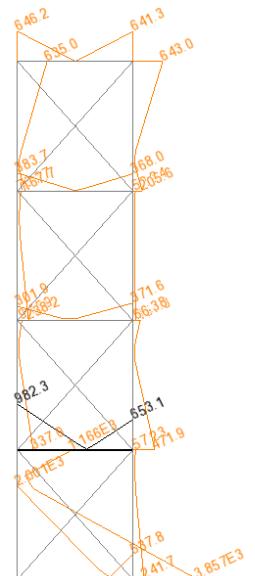
 Enter **Upper 4 storeys** for the group name. Click the **OK** button to complete creation of the group.

- Click the right-hand mouse button on the **Upper 4 storeys** group in the  Treeview, and select the **Set as Only Visible** option to turn on the display of this group only.



The results for just the upper 4 storeys will be displayed.

- Double click on the **Diagrams** layer in the  Treeview and select the **Diagram Display** tab.
- Deselect the **Peaks only** and the **Label only if selected** options.
- Set the number of significant figures to **4**
- Choose the **Label Font** button and change the size to **11** and click **OK**
- Change the Angle to **25**
- Select the **Scale** tab and set the magnitude to **2**
- Select an element and the values of Mz (as previously selected) for that element will be highlighted.
- Turn off the Diagrams layer



## Re-display the Whole Model

To re-display the whole model in working mode:

- Select the  Treeview and click the right-hand mouse button on **tower.mdl**, and select the **Visible** option to turn on the display of all groups.
- Select **Yes** to act on subgroups as well.

### Frequency Versus Participation Factor Graphs

A graph of Participation factors (for the X and Y directions) versus Frequency is to be plotted. For the current spectral combination, where the applied ground motion is in the X direction, the graph will show to what extent each structural mode is excited in the X and Y directions. A large modal participation factor indicates a significant response in that direction.

### Using the Graph Wizard

The graph wizard provides a step-by-step means of selecting which results are to be plotted on the X and Y axes of the graph. The X axis is always defined first.

- Ensure the **Time history** option is selected and click the **Next** button.
- Select the **Named** option and click the **Next** button.
- Select **Natural Frequency** from the drop down list and click the **Next** button.

The Y axis results to be graphed are now defined.

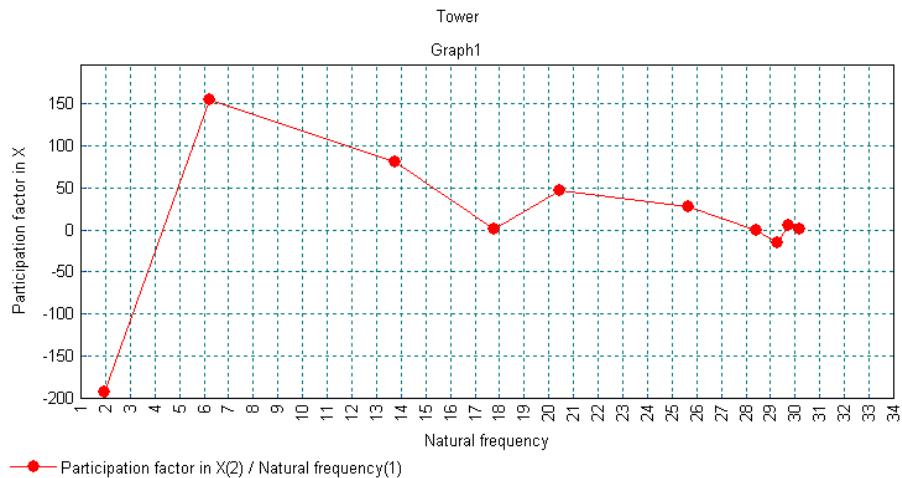
- Select **Named** results and click the **Next** button.
- Select **Participation Factor X** and click the **Next** button.

Additional information for the graph may now to be added.

- Enter the Y axis title as **Participation Factor in X**
- Ensure that the **Show symbols** button is selected.
- Click the **Finish** button to display the graph.

 **Note.** If no graph or axis titles are entered default names will be used.

 **Note.** The sign of the participation factor displayed on the graph is arbitrary as the mode shape from which it is derived can have a positive or negative amplitude.



A graph will be created and the values plotted are displayed in an adjacent table. To see the graph at the best resolution enlarge the window to a full size view.

From the graph it can be seen that the first 3 modes are dominated by the motion in the X direction.

A graph of Participation factor in the Y direction will now be plotted on top of the existing graph.

- Ensure the **Time history** option is selected and click the **Next** button.
- Select the **Previously Defined** option. Click the **Next** button.
- Select **Natural Frequency** results. Click the **Next** button.

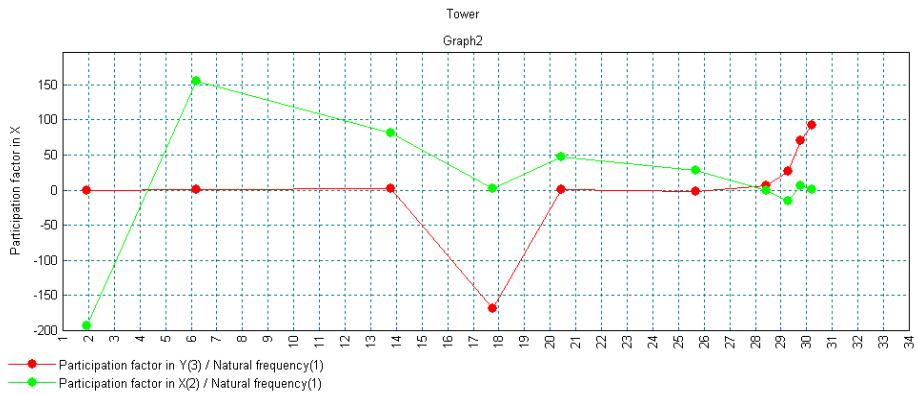
The Y axis results to be graphed are now defined.

- Select **Named** results and click the **Next** button.
- Select **Participation Factor in Y**. Click the **Next** button.

The results are to be plotted on top of the existing graph, so no additional title information is to be added, but the graph to be included must be chosen:

- Include **Graph 1** by ticking the check box inside the Include existing graphs panel of the dialog.
- Click the **Finish** button.

## Seismic Response of a 2D Frame (Frequency Domain)



The Participation factors for the Y axis are superimposed on the graph.

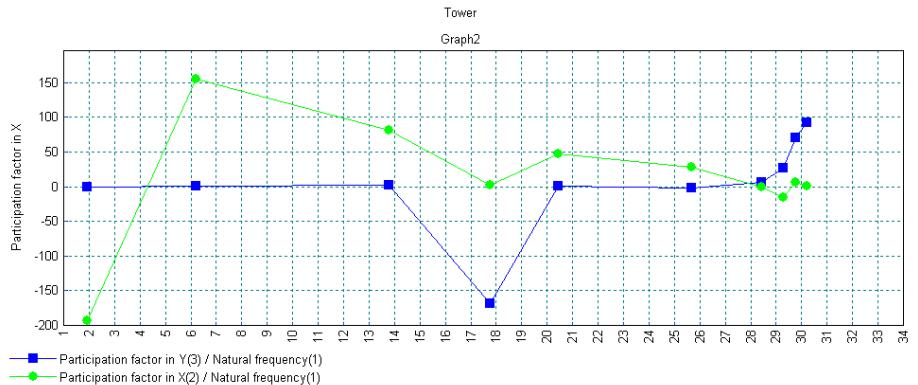
From the graph it can be seen that the fourth mode shape is dominated by motion in the Y direction.



**Note.** The graph properties may be modified by clicking the right-hand mouse button in the graph display area and selecting the **Edit Graph Properties** option.

To change the symbols on the second graph dataset:

- Select the **Curves** tab and the second dataset **Participation Factor Y / Natural Frequency** from the dropdown list.
- Change the symbol to **Square**, select the **Symbol Start Colour** and select **dark blue**
- Select the **Line Colour** and select the same **dark blue**
- Click **OK** to action the change.



## Creating Animations of Mode Shapes

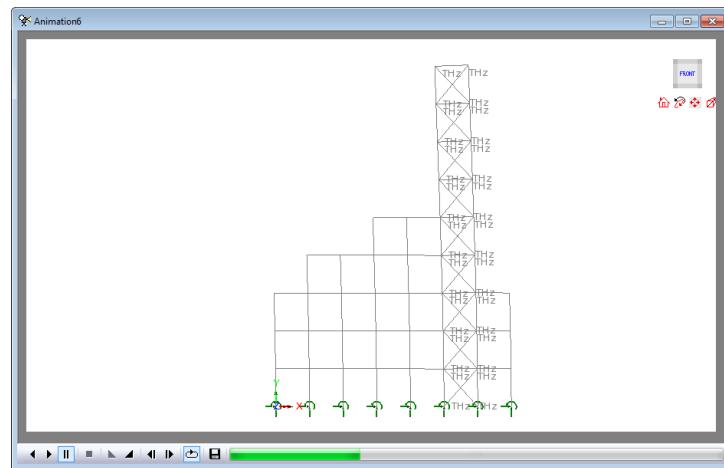
The dominant mode shapes (Eigenvalue modes 1, 2, 3 and 4) caused by the applied response spectra will now be animated. To see the deformations in the structure the deformed mesh layer must be displayed.

- Re-select the view window containing the model data and if using a ‘Multiple documents’ layout
- Turn off the display of the **Mesh**, **Diagram** and **Annotation** layers in the  Treeview.
- Add or turn-on the **Deformed mesh** layer and **OK** the dialog properties.
- In the  Treeview right-click on the Eigenvalue results file **Mode 1 Eigenvalue = <value>** and select the **Set Active** option.

The deformed shape for Mode 1 will now be displayed.

Tools  
Animation Wizard...

- Ensure the **Active loadcase** button is selected and select the **Next** button.
- Use a **Sine** deformation with **16** frames. Set the deformation magnitude of **6** mm for the range **0 to 1**
- Click **Finish** to build and display the animation.



- To see the animation at the best resolution enlarge the window to a full size view.

The buttons at the bottom of the window may be used to slow-down, speed-up, pause, step through frame by frame, or stop the animation.

### Saving Animations

Animations may be saved for replay in LUSAS at any time or saved for display in other windows animation players.

- Ensure the animation window is the active window.
- Save the animation to your projects folder and enter **tower\_1** for the animation file name. An **.avi** file extension is automatically appended to the file name.
- Animations can be compressed to save disk space by altering the **Compression Quality** slider.



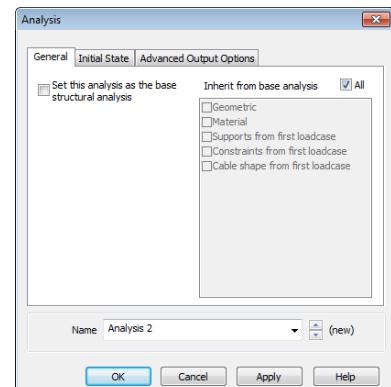
Save the model file.

### Viewing the effects of self weight and spectral response

Spectral loadcases can be combined with other loadcases to provide results which include dead and live loads but since spectral loadcases are computed from an eigenvalue analysis the sign of the spectral displacements is always positive. The most adverse results can, however, be obtained using load combinations.

This adds a new structural analysis to the model

- Ensure **Set this analysis as the base structural analysis** is not selected,
- Ensure that **Inherit from base analysis** is selected.
- Name the analysis **Analysis 2** and press **OK**.



Loading from self weight can be automatically added to any loadcase by enabling gravity as a property of a loadcase in the Treeview:

- In the Treeview expand **Analysis 2** and then right-click on **Loadcase 2** and rename it to be **Self Weight**
- In the Treeview right-click on **Self Weight** and select **Gravity**

## Running the Analysis



On the **Solve Now** dialog, ensure that **Analysis 2** is selected and press **OK**.

## Viewing the Results

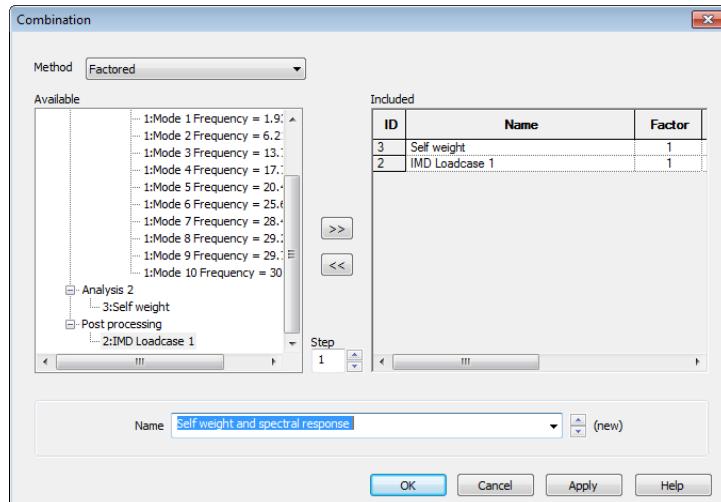
After a successful analysis the self weight results file will be added to the Treeview.

### Defining Self weight and Spectral load combinations

Two basic load combinations will now be created to view the combined effects of the self weight and spectral results; one for self weight plus the spectral results and one for self weight minus the spectral results.

Analyses  
Basic Combination...

The combination properties dialog will appear.



- From the available loadcases, select **Self Weight** and click the button to add the loadcase to the load combination. Leave the load factor set as **1**
- Then, select **IMD Spectral Results** and click the button to add the Spectral results loadcase to the load combination. Leave the load factor set as **1**
- Enter **Self weight and spectral response** for the combination name
- Click the **OK** button to finish.



**Note.** The IMD Spectral Results loadcase references the eigenvalues held in the Analysis 1 results file.

## Seismic Response of a 2D Frame (Frequency Domain)

---

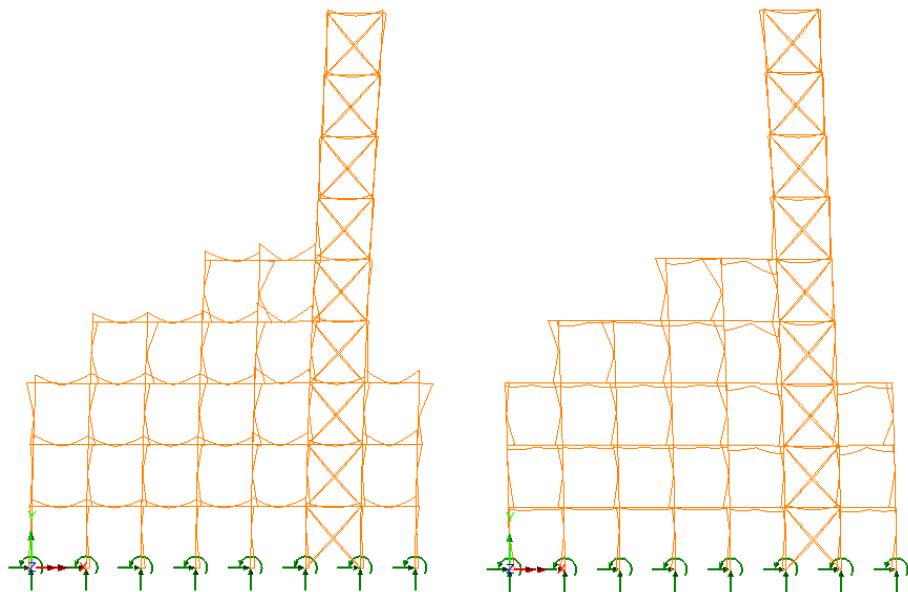
The previous procedure is repeated to create a second basic load combination.

- From the available loadcases, select **Self Weight** and click the  button to add the loadcase to the load combination. Leave the load factor set as **1**
- Then, select **IMD Spectral Results** and click the  button to add the Spectral results loadcase to the load combination. Change the load factor to be **-1**
- Enter **Self Weight - Spectral Response** for the combination name
- Click the **OK** button to finish.



Save the model file. This also saves any load combinations with the model.

By setting the Self Weight +/- Spectral Response load combinations active, shear force and bending moments can be viewed for each combination. Use or modify the Diagrams layer to view values on the frame.



This completes the example.