Seismic Response of a 3D Frame (Frequency Domain)

| For LUSAS version:	20.0
For software product(s):	All (except LT versions)
With product option(s):	None.
Note: This example exceeds the limits of the LUSAS Teaching and Training version.	

Description

This example examines the spectral response of a 2-storey 3D frame.

The geometry of the structure has been simplified to a wireframe or stick representation with each of the members of the structure being represented by a line feature only.

A fleshed view of the model is shown to the right.

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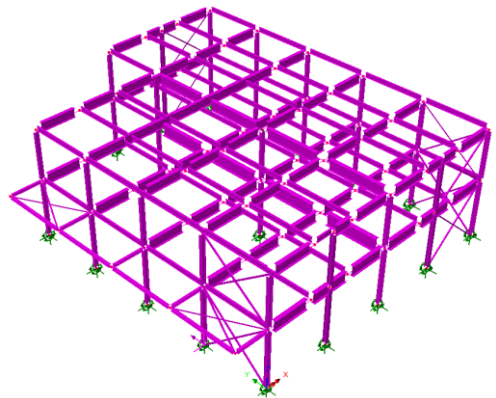
Objectives

The required output from the analysis consists of:

❑ Axial force and bending moment diagram from a CQC combination.

Keywords

3D, Frame, Seismic, Spectral, Response, Mass Participation, Interactive Modal Dynamics, Excitation, Eigenvalue, CQC Combination.

Associated Files

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

- ❑ **3D_frame.lvb** creates an initial model for further development.
- ❑ **3D_frame_spectrum_EC8.lvb** defines a typical spectral curve to Eurocode 8

Discussion

The mesh definition used on a dynamic analysis is somewhat different from that used on 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 have to be considered. Generally, the best strategy for a dynamic analysis is to have a uniform mesh over the entire structure but in stiff regions a coarser mesh can be used. In regions that are more flexible, or where heavy masses are located, the mesh can be more refined.

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

The spectral response analysis is performed in two distinct stages:

1. A natural frequency analysis is performed. This is used in this example 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 spectral response analysis. In order to carry out a spectral analysis the modes 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.

2. A spectral response calculation 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 Solver.

In a spectral response analysis, the structure is subjected to support excitation. In this example this is assumed to be the effect of seismic motion although any support motion could be envisaged. The excitation is specified as a spectral curve, in terms of period versus acceleration. Damping, which is implicitly included in the spectral curve, may also be specified at this stage.

From the eigen analysis, participation factors indicate, for each mode, the degree of structural response associated with an applied excitation. In the spectral analysis a mode combination is calculated from which a worst-case displacement/moment envelope may be obtained.

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
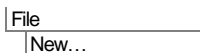
File
New…

- Enter a file name of **3D_frame**.

- Use the default **User-defined** working folder.

- Ensure an Analysis category of **3D** is set.
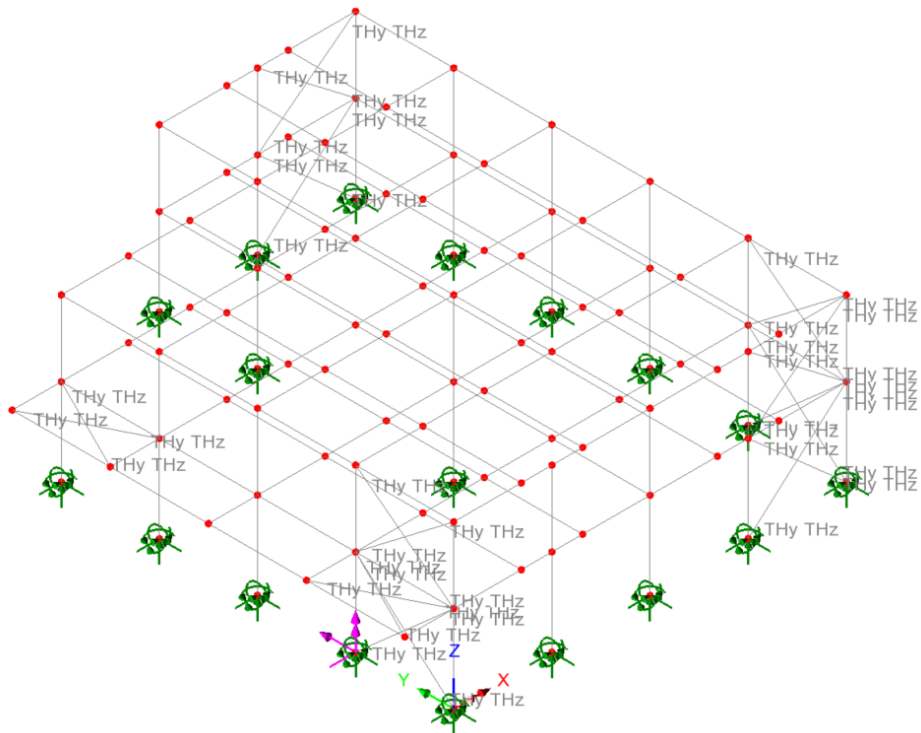
- Click the **OK** button.

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.

File
Script   >
Run Script…
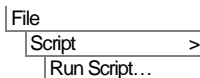
- To create the model, open the read-only file **3D_frame.lvb** that was downloaded and placed in a folder of your choosing.

⊕ If necessary, select the isometric button to view the frame in 3D.

⚡ Toggling the Fleshing button on and off will show the steelwork arrangement.

✎ **Note.** No static structural loading is required for this analysis because a 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.

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.

The following parameters need to be specified:

- Set the Number of eigenvalues to be **30**

- Ensure the Shift to be applied is set as **0**

- Ensure the Type of eigensolver is set as **Default**

Note. Eigenvector normalisation is set to **Mass** by default. This is essential if the eigenvectors are to be used for subsequent IMD analysis in results processing as they are in this case.

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

File
  Save

Save the model file.

Running the Analysis

With the model loaded:

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

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

Viewing the Results

Analysis loadcase results for each eigenvalue can be seen in the Treeview. Eigenvalue 1 is set to be active by default.

### Plotting Mode Shapes

- Turn off the display of the **Mesh, Geometry** and **Attributes** layers in the Treeview.

- With no features selected, click the right-hand mouse button in the view window and select the **Deformed mesh** option to add the deformed mesh layer to the Treeview. Select the **Visualise** tab and, if it is already selected, deselect the **Beam end releases** option. Click the **OK** button to accept the remaining default values and display the deformed mesh for **Eigen mode 1.**

Tools
  Annotation   >
    Window Summary

- Show the **Window Summary** for an overview of the results.

Scale: 1: 159.832
Zoom: 100.0
Eye: (-0.57735, -0.57735, 0.57735)
Eigenvalue analysis
Analysis: Analysis 1
Loadcase: 1:Loadcase 1, 1:Mode 1 Frequency = 3.11433
Results file: 3D_frame~Analysis 1.mys
Eigenvalue: 382.904
Natural frequency: 3.11433
Error norm: 1.57978E-12
Maximum displacement 9.64901E-3 at node 99
Deformation exaggeration: 99.3876
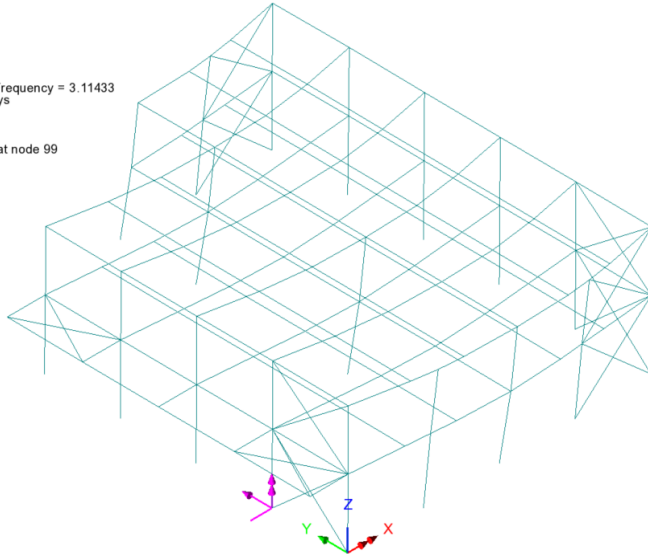
Note. The window summary displays the values of the eigenvalue and the natural frequency and also a value for displacement at a node. It should be noted that the displacement value is non-quantitative and is related to the amount of mass in a particular mode using the mass normalisation technique. Therefore, the only items that can be found using a basic eigenvalue analysis are the frequency and the mode shape.

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

- In the Treeview, in the eigenvalue results, right-click on **Mode 2 Frequency = <value>** and select the **Set Active** option. The deformed mesh plot for Mode 2 will be displayed.
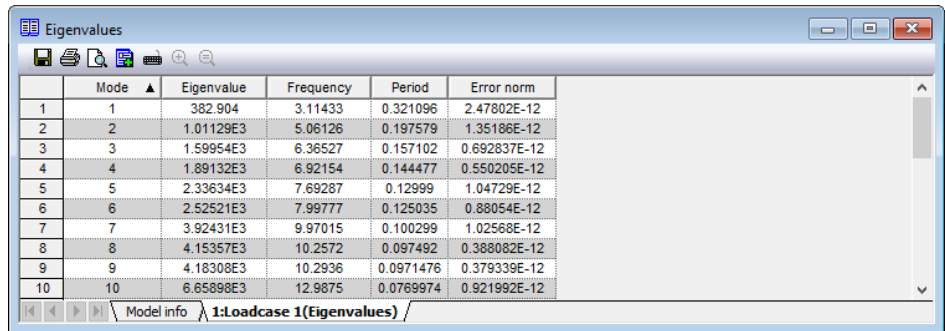
By setting each Eigenmode to be active the deformed mesh can be seen for all mode shapes.

Printing Eigenvalue Results

Utilities
  Print Results Wizard

- Select results type **Eigenvalues**. For loadcase 1, in the available results panel select **Eigenvalues (Frequency)**, ensure the **Display now** option is selected, and click **OK.** The Eigenvalue results will be displayed, and a Print Results Wizard entry will be added to the Utilities treeview.

For inspection only the first 10 modes will be shown here.

	Mode ▲	Eigenvalue	Frequency	Period	Error norm
1	1	382.904	3.11433	0.321096	2.47802E-12
2	2	1.01129E3	5.06126	0.197579	1.35186E-12
3	3	1.59954E3	6.36527	0.157102	0.692837E-12
4	4	1.89132E3	6.92154	0.144477	0.550205E-12
5	5	2.33634E3	7.69287	0.12999	1.04729E-12
6	6	2.52521E3	7.99777	0.125035	0.88054E-12
7	7	3.92431E3	9.97015	0.100299	1.02568E-12
8	8	4.15357E3	10.2572	0.097492	0.388082E-12
9	9	4.18308E3	10.2936	0.0971476	0.379339E-12
10	10	6.65898E3	12.9875	0.0769974	0.921992E-12

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

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

- Close the print results window by selecting the close button in the top right-hand corner of the window.

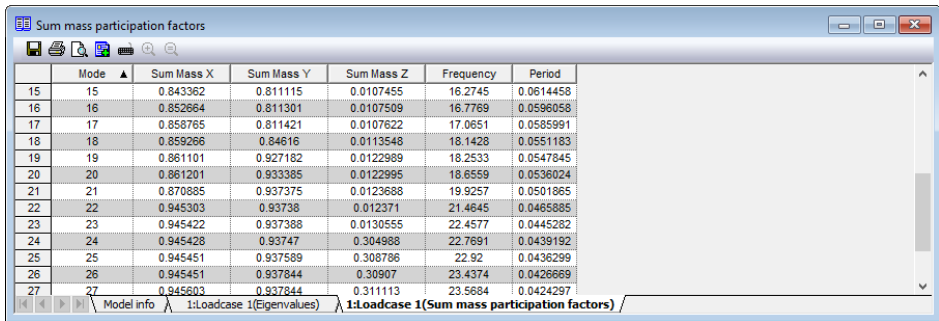## Checking the Mass Participation Factor

Note. In order to carry out a successful response analysis you should ensure that a significant proportion of the total mass has been accounted for in the analysis. This requires checking that around 90% of the total mass is in the global x and y directions. Failure to check that a significant proportion of the total mass has been accounted may lead to important modes being missed and subsequent errors in the resulting response analysis results.

- In the Utilities 🔧 treeview, double click the **PRW1** entry

- Ensure results type **Eigenvalues**. For loadcase 1, in the available results panel select **Sum mass participation factors** and click **OK.** The Sum Mass Participation Factors results will be added to those of the Eigenvalues (frequency) results.

- In the Utilities 🔧 treeview, right-click on the **PRW1** entry and select **Show results**.

The results will be displayed.

- Select the **Sum mass participation factors** tab appropriate tab. For inspection only modes 20 to 25 will be printed here. It can be seen that a greater than 90% participation value has been achieved in mode 22.

	Mode ▲	Sum Mass X	Sum Mass Y	Sum Mass Z	Frequency	Period
15	15	0.843362	0.811115	0.0107455	16.2745	0.0614458
16	16	0.852664	0.811301	0.0107509	16.7769	0.0596058
17	17	0.858765	0.811421	0.0107622	17.0651	0.0585991
18	18	0.859266	0.84616	0.0113548	18.1428	0.0551183
19	19	0.861101	0.927182	0.0122989	18.2533	0.0547845
20	20	0.861201	0.933385	0.0122995	18.6559	0.0536024
21	21	0.870885	0.937375	0.0123688	19.9257	0.0501865
22	22	0.945303	0.93738	0.012371	21.4645	0.0465885
23	23	0.945422	0.937388	0.0130555	22.4577	0.0445282
24	24	0.945428	0.93747	0.304988	22.7691	0.0439192
25	25	0.945451	0.937589	0.308786	22.92	0.0436299
26	26	0.945451	0.937844	0.30907	23.4374	0.0426669
27	27	0.945603	0.937844	0.311113	23.5684	0.0424297

Model info \ 1:Loadcase 1(Eigenvalues) \ **1:Loadcase 1(Sum mass participation factors)** /

- Close the print results window.

Spectral Response Analysis

Spectral response calculations are performed using the IMD (Interactive Modal Dynamics) facility. This involves defining the spectral curve and excitation and specifying the results required in an IMD loadcase. The spectral results can then be interrogated by setting the IMD loadcase active.

With spectral response analysis 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.

3D_frame_spectrum_EC8.lvb defines response spectrum to Eurocode EC8.

File
  Script   >
    Run Script…

- Select the file **3D_frame_spectrum_EC8.lvb** that was downloaded and placed in a folder of your choosing. Click **OK**

- A response spectra dataset 3D_frame_spectrum containing the spectral data for period and acceleration will be created in the ⚒ treeview.

- Double-click on the **3D_frame_spectrum response spectra** entry to view the curve definition of the response spectrum.

- Define a spectral curve damping value of **5** (%) because that is the assumed damping for the curves of EN1998-1 Figure 3.2 and 3.3
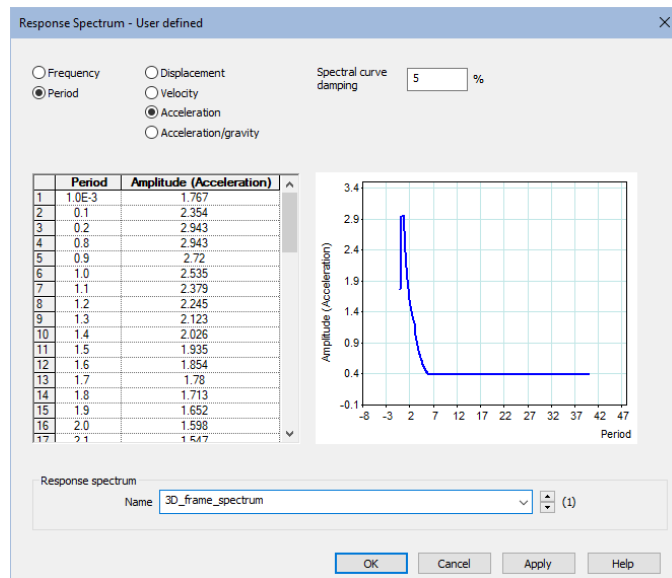
- Click the **OK** button.

Stage 2 : Defining the Dynamic Excitation

Analyses
   IMD Loadcase…

- From the Excitation drop down list select **Support Motion**. Press the **Set** button alongside and ensure the **Acceleration** support motion option is set to a **User specified** direction which is **Relative.** In the box next to the X direction enter the value of **1.0** and in the box next to the Y direction enter the value of **0.6**. Click the **OK** button to return to the IMD loadcase dialog.

- From the Results drop-down list select **Spectral**. Press the **Set** button alongside and ensure the type of spectral response is **CQC Combination** with the damping variation correction set as **Eurocode**, which is an appropriate method for correcting an assumed curve damping (5% as above) to suit the structure damping (which will be specified shortly as 0.8% below). The response spectrum **3D_frame_spectrum** read in from the supplied VBS file will already be selected in the drop-down list. Click the **OK** button to return to the IMD loadcase dialog.

- In the Damping type section, select **Specified values**. Press the **Set Damping** button and specify a Viscous damping percentage of **0.8.** This is as per EN1991-1-4 Table F.2, for a steel frame.

- In the Modes section, ensure the **Use all modes** option is selected.

- Ensure that **IMD Loadcase 1** is defined for the name.

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

Note. Using the user-specified support motion option a single earthquake response can be applied in multiple directions at the same time. In this case 100% of the earthquake response is being applied in the X direction at the same time as 60% of the earthquake response is being applied in the Y direction. Note that specifying a user direction re-orientates and factors the input motion according to a vector, which unless adjusted, will amplify (scale) the motion.

Note. There are a number of different methods of carrying out modal combinations:

1. If the natural frequencies are well separated the square root of the sum of the squares (SRSS) has been shown to produce acceptable results.

2. For a situation where the modes have close separation of natural frequencies then an absolute sum (AbsSum) is taken on the assumption that they can peak in phase with each other.

3. Alternatively, a complete quadratic combination (CQC) method can be used (as in this example) which effectively combines both the SRSS and AbsSum methods as it accounts for the separation between modes.

Note. If the defined damping differs from that inherent in the spectral curve a correction may be applied. There is a number of damping variation correction options available for use. For more details on these see the *Theory Manual*.

Stage 3: Selecting the IMD Results Loadcase

- In the 🕐 Treeview right-click on the **IMD Loadcase 1** loadcase name and select the **Set Active** option.

Scale: 1: 160.199
Zoom: 100.0
Eye: (-0.57735, -0.57735, 0.57735)
Spectral response analysis
CQC combination
Spectrum: 3D_frame_spectrum
Loadcase: 2:IMD Loadcase 1
Maximum displacement 0.012328 at node 92
Deformation exaggeration: 77.9684
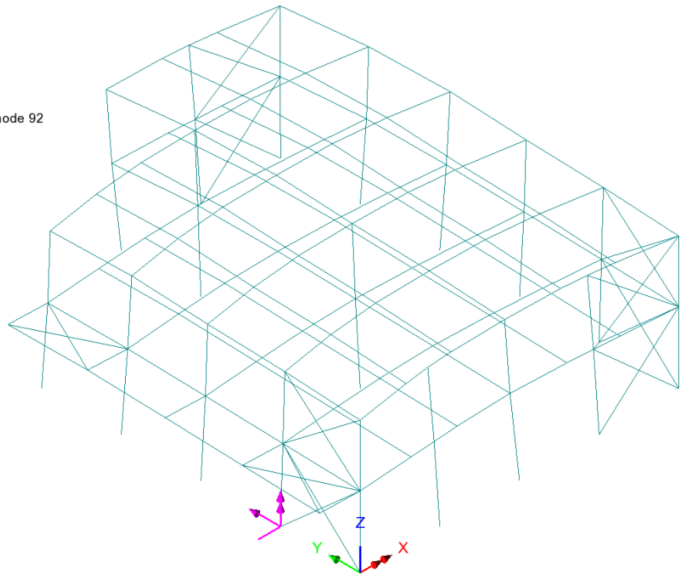
Stage 4 : Displaying the Spectral Combination Results

Results plots showing the axial force and bending moments in the members for the spectral combination are to be displayed.

- 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 **Deformed Mesh** layer in the Treeview .

- With no features selected, click the right-hand mouse button in a blank part of the Graphics 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 and ensure the **Label values** option is selected. Plot values on **80%** of the element length. Click the **OK** button to finish.

Note that the 'Orientate by element axes' and 'Orientate flat to screen/page' options are greyed-out because they have no relevance for axial stress diagrams. A 'flat to screen/page' plot is obtained by default.
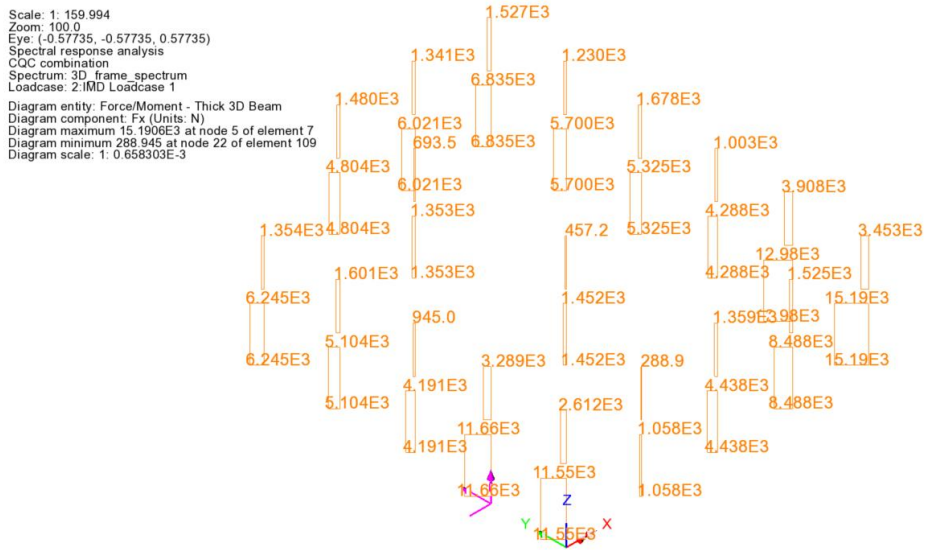
Results will be calculated for the active IMD load case and an axial force diagram of force in each member will be displayed. If necessary, use the Zoom in button to view results information. These types of results can also be plotted in isolation on selected model features.

- In the ▣ Treeview double-click the **Diagram** layer to view its properties.

- Select **Force/Moment** results of axial force **My** in the members. Click the **OK** button to finish.

Scale: 1: 159.994
Zoom: 100.0
Eye: (-0.57735, -0.57735, 0.57735)
Spectral response analysis
CQC combination
Spectrum: 3D_frame_spectrum
Loadcase: 2:IMD Loadcase 1

Diagram entity: Force/Moment - Thick 3D Beam
Diagram component: My (Units: N.m)
Diagram maximum 21.1142E3 at node 14 of element 20
Diagram minimum 5.36694 at Gauss point 6 of element 3
Diagram scale: 1: 0.473616E-3
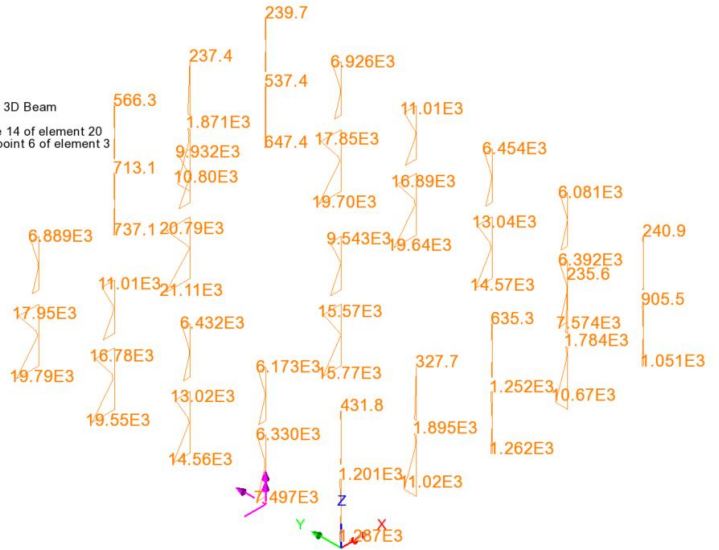
Note that 'Orientate by element axes' is the default diagram display option for all components other than Fx. This can be changed by selecting the Diagram display tab and selecting 'Orientate flat to screen / page' as necessary.

File
   Save

Save the model file.

This completes the example.