

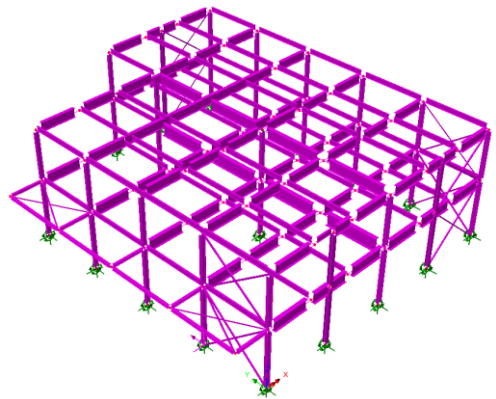
Seismic Response of a 3D Frame (Frequency Domain)

For LUSAS version:	19.0
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 wire-frame 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 diagram from a CQC combination.

Keywords

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

Associated Files



- ❑ **3D_frame.mdl** Model of the structure.
- ❑ **3D_frame_spectrum_EC8.vbs** 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 has 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 more coarse 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, 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.

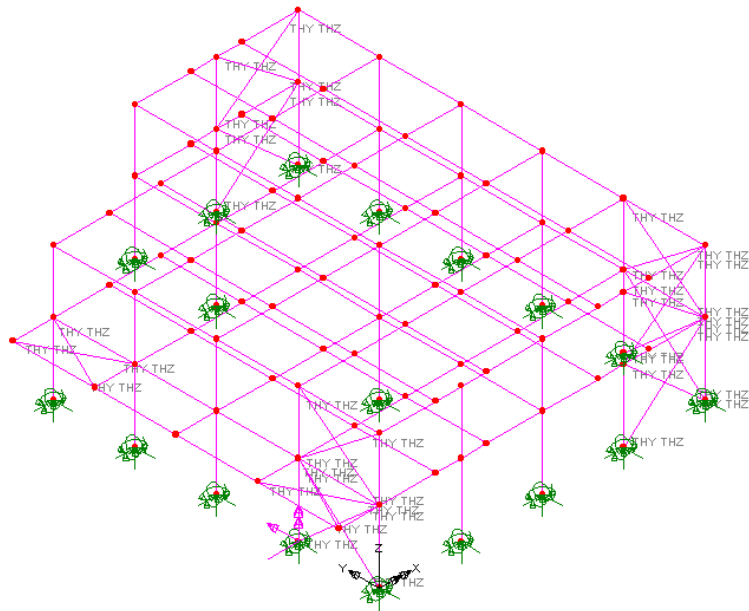
Building and loading the model

For this example a model file is provided:

- To create the model, open the read-only file **3d_frame.mdl** located in the \<LUSAS Installation Folder>\Examples\Modeller directory.

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

File
Open...



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



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

Save the model



Save the model file to the \<Lusas Installation Folder> \Projects directory.

File
Save As




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**.

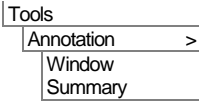
A LUSAS Datafile will be created from the model information. The LUSAS Solver uses this datafile to perform the analysis.

Viewing the Results

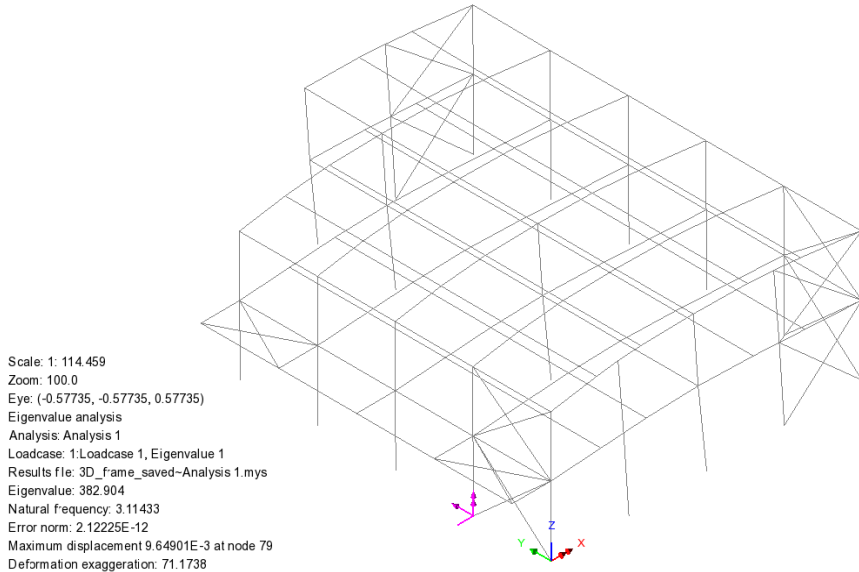
Analysis loadcase results for each eigenvalue can be seen in the  Treeview. Eigenvalue 1 is set to be active by default.

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 Eigenvalue 1.




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



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 Eigenvalue = <value>** and select the **Set Active** option. The deformed mesh plot for Mode 2 will be displayed.

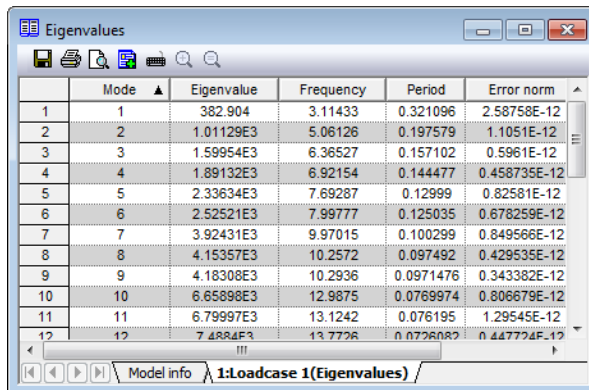
By setting each Eigenmode to be active the deformed mesh can be seen for all mode shapes.

Printing Eigenvalue Results

- 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	382.904	3.11433	0.321096	2.58758E-12
2	1.01129E3	5.06126	0.197579	1.1051E-12
3	1.59954E3	6.36527	0.157102	0.5961E-12
4	1.89132E3	6.92154	0.144477	0.458735E-12
5	2.33634E3	7.69287	0.12999	0.82581E-12
6	2.52521E3	7.99777	0.125035	0.678259E-12
7	3.92431E3	9.97015	0.100299	0.849566E-12
8	4.15357E3	10.2572	0.097492	0.429535E-12
9	4.18308E3	10.2936	0.0971476	0.343382E-12
10	6.65898E3	12.9875	0.0769974	0.806679E-12
11	6.79997E3	13.1242	0.076195	1.29545E-12
12	7.4884E3	13.7726	0.0726082	0.447724E-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 ($1/\text{frequency}$). Values of error norm may vary from those shown.





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

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

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.

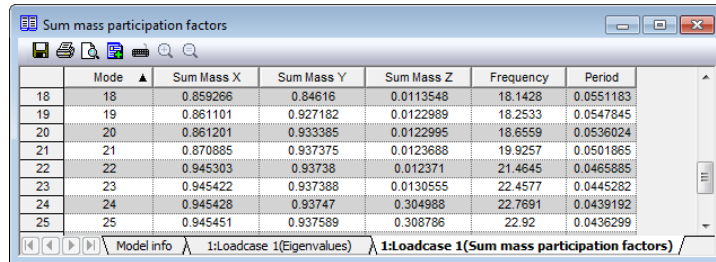
- In the Utilities  treeview, double click the **PRW1** entry
- Ensure results type **Eigenvalues**. For loadcase 1, in the available results panel select **Sum mass participation factors** and click **OK**. The Sum Mass Participation Factors results will be added to those of the Eigenvalues (frequency) results.
- In the Utilities  treeview, right-click on the **PRW1** entry and select **Show results**.

The results will be displayed.

Utilities
Print Results
Wizard...

Seismic Response of a 3D Frame (Frequency Domain)

- 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
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

- 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

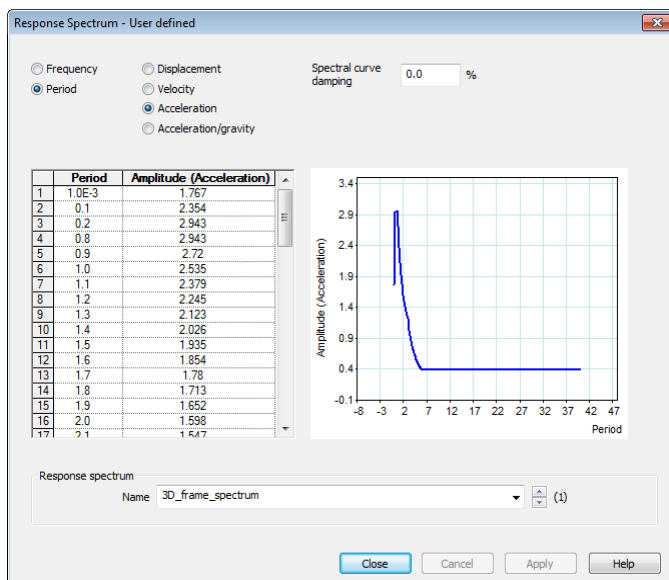
Values for the spectral excitation curve need to be specified. For this analysis Acceleration versus Period values are to be used.



3d_frame_spectrum_ec8.vbs defines response spectrum to Eurocode EC8.

- Select the file **3d_frame_spectrum_ec8.vbs**, which is located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory. 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.

File
Script >
Run Script...



- Click the **Close** 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 **None**. 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 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

Seismic Response of a 3D Frame (Frequency Domain)

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:

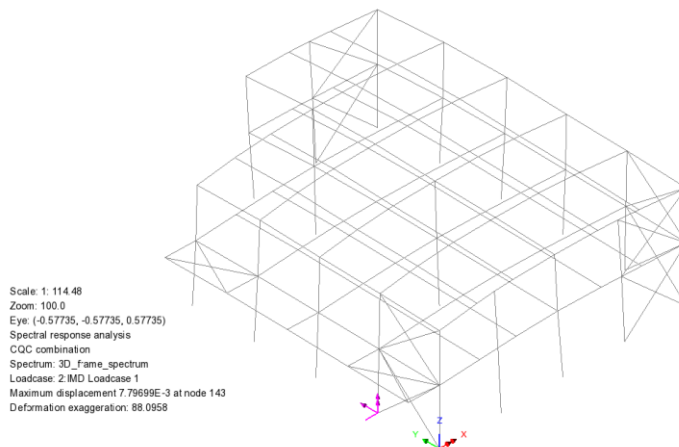
1. If the natural frequencies are well separated the square root of the sum of the squares (SRSS) has been shown to produce acceptable results.
2. For a situation where the modes have close separation of natural frequencies then an absolute sum (AbsSum) is taken on the assumption that they can peak in 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.

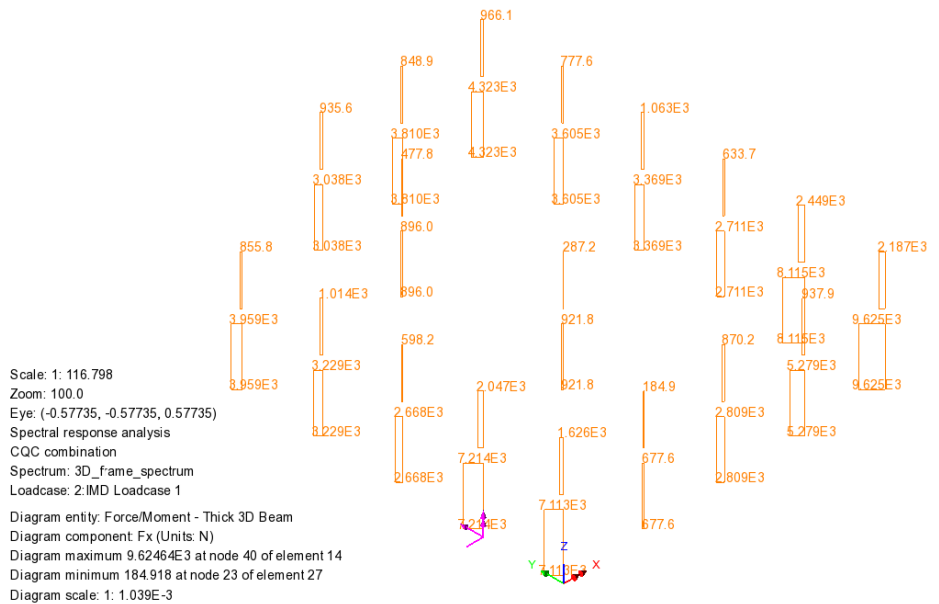


Stage 4 : Displaying the Spectral Combination Results


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.

