IMDPlus User Manual and Worked Examples

LUSAS Version 15.1 : Issue 1

LUSAS
Forge House, 66 High Street, Kingston upon Thames,
Surrey, KT1 1HN, United Kingdom

Tel: +44 (0)20 8541 1999
Fax +44 (0)20 8549 9399
Email: info@lusas.com
http://www.lusas.com

Distributors Worldwide

Table of Contents

IMDPlus User Manual

The IMDPlus modal dynamics software option calculates the modal response of a system to a given input using calculated eigenmodes and eigenvectors from an eigenvalue analysis. (Note that the eigenvalue analysis must have been performed with mass normalised eigenmodes.)

IMDPlus Analysis Types

There are three types of analysis that can be performed in IMDPlus. These are as follows:

- **Seismic Analysis**
  A seismic analysis obtains the dynamic response of a structure to acceleration time histories of support motion.
- **Moving Load Analysis**
  A moving load analysis obtains the dynamic response of a structure to moving loads. In a moving load analysis the load can either be explicitly defined or a vehicle configuration (composite axle definition) can be used. In both cases the magnitude and configuration of the load remains constant throughout the analysis.
- **Moving Mass Analysis**
  A moving mass analysis obtains the dynamic response of a structure to moving masses. In a moving mass analysis spring-mass systems are used in a vehicle configuration. Each spring-mass system includes an unsprung mass and up to two sprung masses connected by springs and viscous dashpots. The configuration of the systems remains constant throughout the analysis but as they move across the structure, the dynamic response of the unsprung and sprung masses affects the forces acting on the underlying structure due to inertia effects.

All solutions from IMDPlus are carried out in the time domain.

IMDPlus Assumptions

The working assumptions for the IMDPlus modal dynamics facility are as follows:

- **Linear**

  The system is linear in terms of geometry, material properties and boundary conditions. Therefore geometrically nonlinear eigenvalue results are not applicable. Nor are nonlinear joint and slideline analyses suitable for this type of post-processing treatment.

- **No Cross-Coupling**

  There is no cross-coupling of modes caused by the damping matrix. This is reasonable for all but the most highly damped structures or applications.

- **Low Modes Dominant**

  The response is dominated by the lowest few modes.

- **Damping Ratios Are Below Critical**

  Damping ratios of 100% or more are not permitted due to the solution of the time domain response of the structure using either the Hilber–Hughes-Taylor (HHT) method or Duhamel's Integral.

- **Moving Mass Analysis**

  The working assumptions for the particular case of a moving mass analysis are as follows:

  - **No Loss of Contact**

    There is no loss of contact between the unsprung masses (wheels) of the spring–mass systems and the structure at any time during a moving mass analysis.

  - **Vertical Motion of Spring-Mass Systems Only**

    Only vertical motion of the spring-mass systems is considered in a moving mass analysis.

  - **Mass of Spring-Mass Systems Not Included in Eigenvalue Solution**

    The mass of the spring-mass systems have no effect on the natural frequencies of the structure.

  - **Recommended Limit on Total Mass of Spring-Mass Systems**

    The accuracy of the moving mass solution reduces for light-weight structures or structures where the magnitude of the moving mass is similar to the magnitude of the structure mass. A literature search recommends that for most solutions of this type, the total mass of all the spring-mass systems that are actively interacting with the structure at any given time should not exceed 10% of the total mass of the underlying structure.

Performing IMDPlus Calculations in LUSAS

The various stages of an IMDPlus analysis are initiated from the **Utilities → IMDPlus** menu or the IMDPlus Toolbar. The basic steps that need to be undertaken for each of the three analysis types are as follows:

Seismic Analysis

1. Select the acceleration time histories
2. Select the eigenmodes to include in the solution
3. Specify the damping for the eigenmodes
4. If required, modify the advanced solution options
5. Specify the frequency interpolation technique and if required, the time step
6. Specify the nodes or elements to process along with the output requirements

Moving Load Analysis

1. Generate the movement of a discrete load across the structure by using a number of distinct locations along the moving load path
2. Calculate the equivalent modal forces for each of these distinct locations
3. If the loading has not been explicitly defined in step 1, define moving load vehicle configuration(s) using the positions of the axles/bogies that form a vehicle together with the moving load factors to be applied at each position
4. If required, select the moving load vehicle configuration to be used in the analysis
5. Select the eigenmodes to include in the solution
6. Specify the damping for the eigenmodes
7. If required, modify the advanced solution options
8. Specify the speed ranges to analyse along with time stepping parameters
9. Specify the nodes or elements to process along with the output requirements

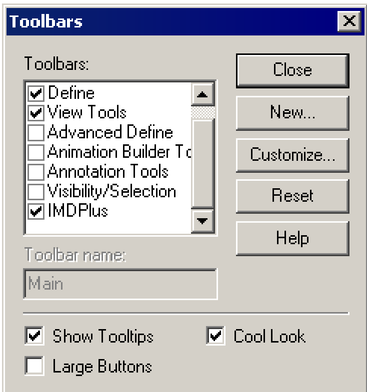
Moving Mass Analysis

1. Generate the movement of a discrete unit load across the structure by using a number of distinct locations along the moving load path
2. Calculate the equivalent modal forces for each of these distinct locations
3. Define spring-mass system property attribute(s)
4. Define moving mass vehicle configuration(s) using the positions of the axles/bogies that form a vehicle together with the spring-mass system property attribute that is to be used at each position
5. Select the moving mass vehicle configuration to be used in the analysis

6. If required, include a path surface profile that defines the surface irregularities along the length of the moving load path
7. Select the eigenmodes to include in the solution
8. Specify the damping for the eigenmodes
9. If required, modify the advanced solution options
10. If required, modify the path surface profile options
11. Specify the speed ranges to analyse along with time stepping parameters
12. Specify the nodes, elements or spring-mass systems to process and the output requirements

IMDPlus Toolbar

All of the IMDPlus dialogs can be launched from the **Utilities → IMDPlus** menu. Alternatively, the IMDPlus Toolbar can be activated from the **View > Toolbars...** menu, as shown in the following figure.



The Toolbar has been split into four sections as follows:

- ❑ **Analysis Type**

  This section allows the IMDPlus analysis type to be selected. Options available are as follows:

   **Seismic Analysis**

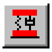
   **Moving Load Analysis**

   **Moving Mass Analysis**

  Only one of the analysis types can be selected at any time.

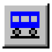
- **IMDPlus Vehicle Attributes**

  This section allows IMDPlus attributes to be defined prior to an IMDPlus analysis. Options available are as follows:



Spring-Mass System Properties

This button opens the spring-mass system properties dialog. It is only available when the analysis type is set to moving mass.



Vehicle Configurations

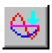
This button opens either the moving load or moving mass vehicle configuration dialog. The dialog launched depends on the analysis type selected. The button is only available when the analysis type is set to moving load or moving mass.

- **Moving Load Generation and Modal Force Calculator**

  This section allows the preliminary steps for a moving load or moving mass analysis to be performed. These are the generation of the moving load path and the calculation of the modal force history for use in the IMDPlus analysis, as follows:

Moving Load Generation

This button opens the moving load generation dialog. It is only available when the analysis type is set to moving load or moving mass.

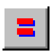


Modal Force Calculator

This button opens the modal force calculator dialog. It is only available when the analysis type is set to moving load or moving mass and the moving load generation step has already been carried out.

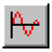
- **Running an IMDPlus Analysis**

  This section allows an IMDPlus analysis to be performed. Options available to the user are as follows.



Analysis Control

This button opens either the seismic, moving load or moving mass analysis control dialog. The dialog launched depends upon the analysis type selected. For both moving load and moving mass analysis types the button is only available when the modal force calculation has been carried out.

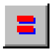


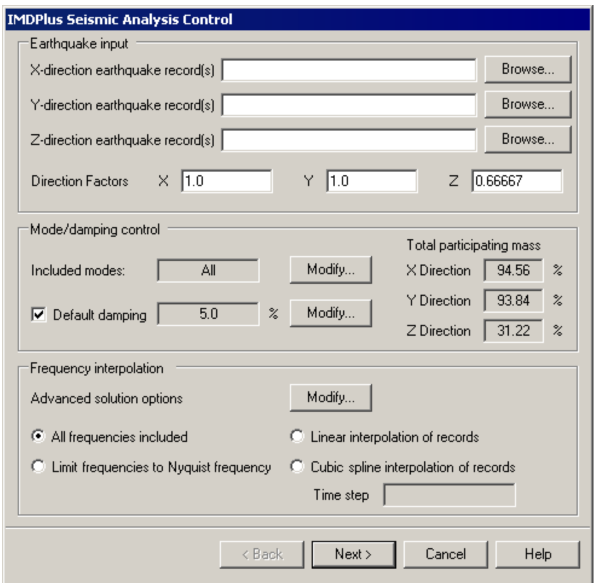
Output Control

This button opens either the seismic, moving load or moving mass output control dialog. The dialog launched depends upon the analysis type selected. It is only available when an IMDPlus analysis has been carried out and results are still available on disk for graphing.

Seismic Analysis

The seismic analysis option calculates the response of the structure to an acceleration time history applied at the support nodes. Both 2D and 3D structures can be analysed but they are restricted to having the acceleration inputs applied in the global X-, Y- directions and X-, Y- and Z-directions respectively. Only one acceleration time history can be specified for each excitation direction and therefore, all support nodes are accelerated with the same time history record in this direction. Different acceleration records can be applied to each global direction and these can be factored based on design code requirements. Up to seven earthquake combinations can be analysed in a single IMDPlus analysis.

The IMDPlus seismic analysis control dialog is accessed from the **Utilities > IMDPlus > Seismic...** menu or from the IMDPlus Toolbar. Ensure the  button is selected to choose a seismic analysis and then select the  button to launch the seismic analysis control dialog.



Earthquake Input

For two-dimensional structures, earthquake acceleration time history records can be applied in the global X- and Y-directions. For three-dimensional structures, an additional earthquake acceleration component in the global Z-direction can be applied. Permissible input formats for these acceleration records are space delimited, TAB delimited or Comma Separated Variable (CSV).

For all of these input formats, the time step of the acceleration records must be identical in all of the earthquake acceleration files. IMDPlus supports the solution of up to seven earthquakes at a time with enveloping / averaging of the Secondary Response Spectra (SRS). For space or TAB delimited files the number of time steps in each directional record for multiple record files must be identical across each of the earthquakes. For Comma Separated Variable files, the number of time steps in each directional record can vary but the number of time steps in individual earthquakes must be identical. If more than one earthquake is present in the record files, the IMDPlus analysis will process all valid combinations.

Sample Acceleration Time History Records

Space Delimited Single Combination

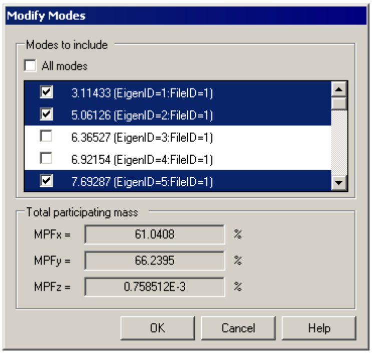
```
0.000  0.0000E+000
0.005 -2.7459E-002
0.010 -3.4127E-002
0.015 -4.0796E-002
0.020 -4.7464E-002
0.025 -5.4133E-002
```

Space Delimited Two Combinations

```
0.000  0.0000E+000  0.0000E+000
0.005 -2.7459E-002 -4.0070E-005
0.010 -3.4127E-002  8.2210E-004
0.015 -4.0796E-002  1.2618E-002
0.020 -4.7464E-002  1.4741E-002
0.025 -5.4133E-002  1.8604E-002
```

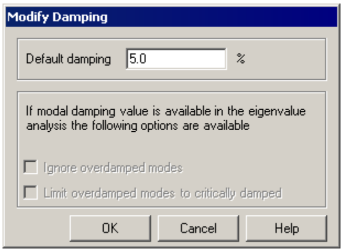
Mode / Damping Control

The modes to include in the analysis can be controlled either by the eigenmodes solved in the eigenvalue analysis or a subset defined in the mode control. The default option when first running an analysis is for all modes to be included in the analysis but individual modes can be included or excluded using the following dialog accessed via the **Modify…** button on the seismic analysis control dialog.



This dialog also calculates the total participating mass for the included modes. If a significant proportion of participating mass is missing, based on the selected modes of vibration, a warning will be issued when proceeding with the analysis. Omission of modes of vibration with significant mass contributions that can be excited by the input acceleration time histories will lead to unreliable solutions. Ideally, total mass participations in excess of 90% should be used unless it is guaranteed that the modes of vibration associated with any missing mass are at frequencies that cannot be excited.

The Default damping option allows the user to control the amount of damping used in the IMDPlus analysis. If Default damping is selected, all of the modes of vibration included in the analysis are forced to have the current default value. The default value is set, via the **Modify…** button on the seismic analysis control dialog.



The Default damping option can only be turned off when viscous damping has been included in the eigenvalue analysis. If Default damping is deselected the viscous damping present in the eigenvalue analysis is used in the IMDPlus analysis. When viscous damping is included in the eigenvalue solution additional options for the inclusion of overdamped modes of vibration in the IMDPlus analysis also become available. These options are as follows:

- **Ignore overdamped modes**
  Modes of vibration with a damping ratio of 100% or more are omitted from the IMDPlus solution.

- **Limit overdamped modes to critically damped**
  Modes of vibration with a damping ratio of 100% or more are limited to 99.9999% to allow them to be included in the analysis.

Frequency Interpolation

Four interpolation options are available for the interpretation of the acceleration time histories in IMDPlus. These are as follows:

- **All frequencies included**
  All modes of vibration are included in the analysis even if their frequencies are greater than the maximum frequency that may be reliably resolved based on the acceleration time history time step (the Nyquist frequency).

- ❑ **Limit frequencies to Nyquist frequency**
  All modes of vibration up to and including the Nyquist frequency are included in the analysis. Modes of vibration with frequencies in excess of this value are discarded.

- ❑ **Linear interpolation of records**
  Allows for linear interpolation of the earthquake records. The required time step is specified by the user for the analysis.

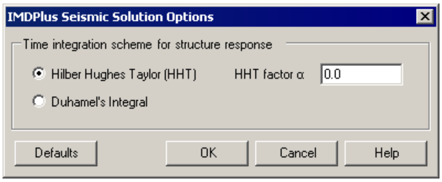
- ❑ **Cubic spline interpolation of records**
  Allows for cubic spline interpolation of the earthquake records. The required time step is specified by the user for the analysis.

Note. For rapidly varying input acceleration time histories, such as real acceleration records measured on site that often have large reversals in acceleration over one or two time steps, the cubic spline interpolation should generally not be used. For synthetic acceleration time history records, the use of cubic spline interpolation is usually valid since the Fourier series used to generate the synthetic records tend to produce smoother responses. It is however up to the user to ensure that the method of interpolation used is suitable for the analysis being carried out.

Note. The Nyquist frequency is equal to $1/(2*\delta t)$ where δt is the time step of the acceleration time history.

Advanced Seismic Solution Options

Advanced solution options can be set via the **Modify...** button on the seismic analysis control dialog.



The options available in a seismic analysis are as follows. The **Defaults** button resets all of the options to the default values.

- ❑ **Time integration scheme for structure response**
  The implicit time integration scheme used by IMDPlus to calculate the dynamic response of the structure.

  - ❑ **Hilber Hughes Taylor (HHT)**
    This is the default time integration scheme and is the same as the method used for an implicit transient dynamic analysis in LUSAS.

- **HHT factor α**

  The integration constant α used in the HHT integration scheme. For an unconditionally stable, second order accurate scheme, α must lie between the limits $-1/3 \le \alpha \le 0.0$. Values of α outside this range cannot be specified. The default value of α is 0.0, which reduces the method to the Newmark method. Further integration constants β and γ are computed automatically by IMDPlus using the following equations:

  $$\beta = (1-\alpha)^2 / 4 \quad , \quad \gamma = (1-2\alpha)/2$$

- **Duhamel's Integral**

  This is the time integration method used by IMDPlus in LUSAS 14.7 (and earlier versions of LUSAS) and is available as an alternative to the HHT integration scheme.

Moving Load Analysis

The moving load analysis option obtains the dynamic response of the structure to the passage of a moving load along a user defined path. Both 2D and 3D structures can be analysed and the magnitude and configuration of the load remains constant throughout the entire analysis. Two methods are available for the definition of the moving load configuration. These are:

- **Explicit definition**

  The complete load configuration is defined through a discrete load definition in Modeller. This can be used for analysing the passage of a single load configuration across the structure.

- **Composite axle definition**

  The complete load configuration consists of a unit load single axle configuration, defined as a discrete load definition in Modeller, and a vehicle configuration is that is defined separately. The vehicle configuration (composite axle definition) method is the recommended approach, as it allows multiple load configurations (such as multiple train rolling stock configurations) to be analysed rapidly without having to repeat initial steps for the IMDPlus solution.

Irrespective of the method used, two distinct steps are required to prepare the moving load input data for IMDPlus. These two steps are as follows:

1. Generation of the moving load path across the structure
2. Calculation of equivalent modal forces for the moving load path

Note. These two steps must be carried out before the first analysis but may be omitted from subsequent analyses, if and only if the moving load path, discrete load configuration and direction of the moving load do not change between analyses. If any of these are modified the moving load path and calculation of equivalent modal forces must be carried out again.

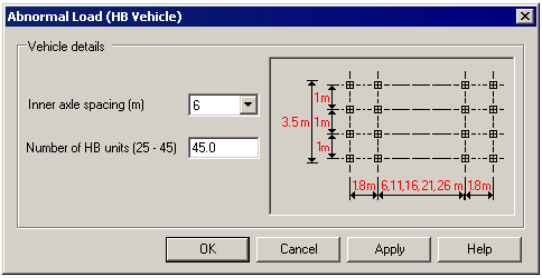
Moving Load Generation

The moving load generator is used to mimic the passage of a load across a structure by automatically setting up a number of static load cases at prescribed locations along a single line/arc/spline, or a collection of lines/arcs/splines, which define a single continuous path. These load cases can then be used to calculate the modal forces equivalent to the applied loading for import into IMDPlus. Before entering the moving load generator, the lines/arcs/splines defining the path must be selected and a discrete load representing the load configuration that is to be moved across the structure must have been defined.

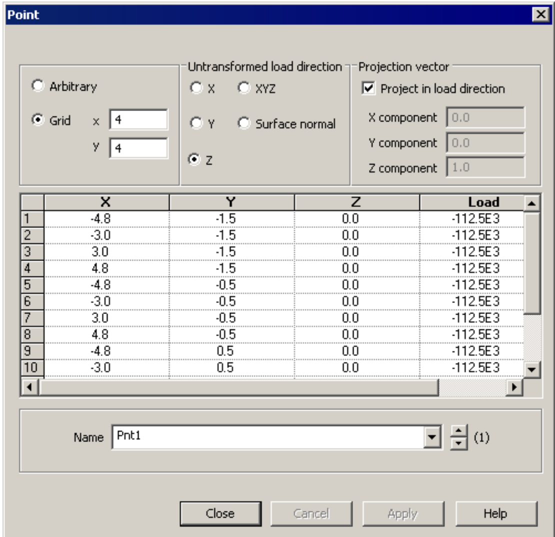
The definition requirements for this discrete load are governed by the type of moving load input that is going to be used in IMDPlus. If the discrete load is going to represent the whole load configuration (i.e. a vehicle configuration is not being used) then it must contain all of the loading associated with the configuration. However, if the composite axle definition method is being used in IMDPlus, this discrete load should represent a subset of the overall load configuration and will be used along with the vehicle configuration to build up the total load configuration in IMDPlus.

Explicit Definition of United Kingdom HB Vehicle

For the LUSAS Bridge option, HB vehicles can be defined from the **Bridge > Bridge Loading > United Kingdom...** menu entry.



After defining a vehicle load, either the initial definition dialog (as shown above) or the resulting attribute data can be viewed. This is done by right-clicking on the Attribute name in the Attributes Treeview and selecting either the **Edit Definition** or **Edit Attribute** menu item. Selecting the latter will show the following discrete point load data for the above vehicle selection.

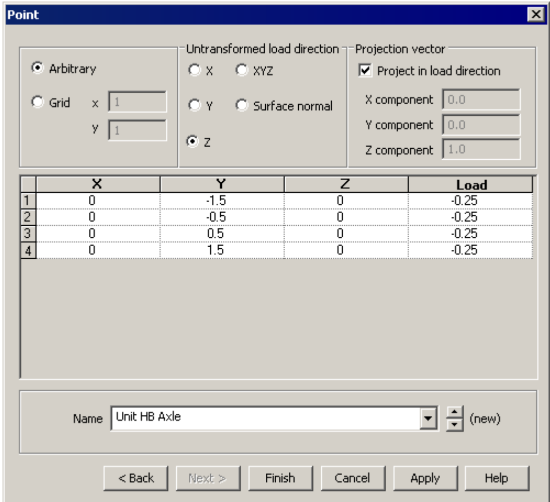


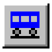
This discrete point load can now be used to define the passage of the HB vehicle across the structure.

Note. The front of the moving load is always the location with the maximum discrete load X co-ordinate.

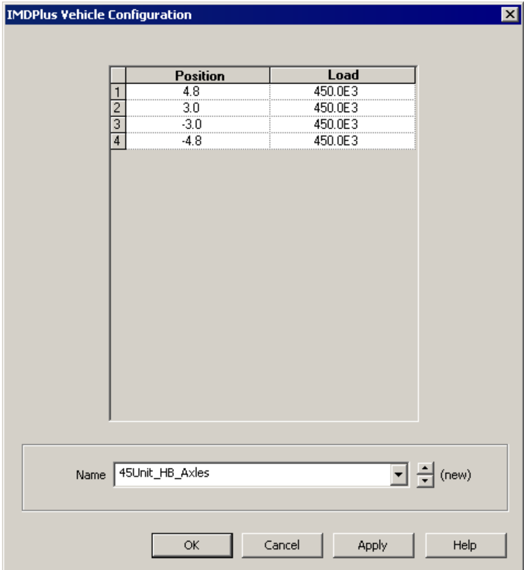
Composite Axle Definition of United Kingdom HB Vehicle using a Moving Load Vehicle Configuration

The equivalent composite axle definition of the above HB vehicle can be achieved by defining a unit axle of the across carriageway configuration and defining the vehicle configuration separately. The following discrete point load defines the unit axle for the HB vehicle.



The IMDPlus moving load vehicle configuration dialog is used to create or modify vehicle configuration attributes for use in a moving load analysis. It is accessed from the **Utilities > IMDPlus > Moving Load > Vehicle Configuration...** menu or from the IMDPlus Toolbar. Ensure the  button is selected to choose a moving load analysis and then select the  button to launch the moving load vehicle configuration dialog.

In a moving load vehicle configuration the position (relative to the origin / front of the vehicle) of each of the applied loads is defined along with the value of the moving load factor to be applied at each position. For the United Kingdom HB vehicle the axle positions and load factors shown in the following figure data should be entered into the dialog. A meaningful name should be entered into the Name field, for example, 45Unit_HB_Axles.



Note. Care should be taken when defining axle positions (X co-ordinates) in discrete load configurations. The path length should be sufficiently longer than the structure to ensure that the load configuration correctly arrives onto and departs from the structure. In the above example, the path should start at least 4.8 m before the structure and finish 4.8 m after the structure.

Note. The front of the moving load vehicle is always the position with the maximum co-ordinate. The rear of the vehicle is always the position with the minimum co-ordinate.

Note. The positions and loads that define the vehicle configuration can be copied and pasted from a Microsoft Excel spreadsheet. This method is demonstrated in the IMDPlus Worked Example: Train Induced Vibration of a Bridge.

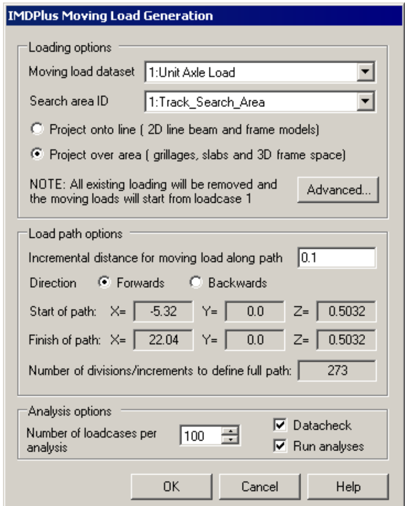
Note. Once created vehicle configurations will appear in the Utilities Treeview, from where they can be modified or deleted from the model. To modify a vehicle configuration either double-click or right-click on the Attribute name in the Utilities Treeview and select the **Edit Attribute...** menu item.

Note. Vehicle configurations can be easily transferred from one model to another using an attribute library file, accessed from the menu **File > Import/Export model data...**

Generation of the Moving Load Discrete Locations

If a valid continuous path has been selected and a valid discrete load defined then the moving load generator can be used to create the static load cases at prescribed locations along the path.

The IMDPlus moving load generation dialog is accessed from the **Utilities > IMDPlus > Moving Load > Moving Load Generation...** menu or from the IMDPlus Toolbar. Ensure the  button is selected to choose a moving load analysis and then select the  button to launch the moving load generation dialog.



If a search area has been defined then this can be used to correctly assign the moving load if there are multiple planes to which the discrete load could be assigned. **Project onto line** or **Project over area** can be selected depending upon the geometry defining the search area. For single deck models, the default search area which encompasses the whole structure can be used. Parameters for the inclusion of the load can be accessed through the **Advanced…** button which provides the full application options available for discrete loads. These options consist of the inclusion of loads outside the search area and the load factor to be applied.

Having defined the loading options, the load path options can be set based on the lines/arcs/splines selected to define the path. The incremental distance controls the separation of the discrete load locations used to define the passage of the load across the structure. This incremental distance should be positive and sufficiently small to capture the movement of the load.

Note. Due to the dynamic solution a sufficiently small distance should be used between the discrete locations defining the passage of the load to minimise oversampling for load locations between those defined. For static analyses, much larger distance increments can be used since the incremental movement does not need to be close to the time step of the analysis.

Ideally, the distance increment used for the movement of the load along the path should obey the following equation:

$$\delta \mathbf{Dist} \leq \mathbf{10} \times \mathbf{Minimum\ Speed} \times \delta \mathbf{t}$$

to give a maximum oversample of 10. δt is the minimum of the Nyquist time step and any user specified time step ($\delta t^{Nyquist} = 1/(2*f_{max})$ where f_{max} is the maximum frequency included in the analysis). Larger distance increments can be used but warnings will be issued by IMDPlus since the accuracy of the dynamic solution decreases with increasing oversampling ratio. Oversampling ratio reduces with increasing speed and therefore it is possible that higher oversampling ratios can be used for lower speeds where the dynamic excitation is reduced. It is however up to the user to ensure that use of higher oversampling ratios for any speed does not adversely affect the solutions to the dynamic analysis.

The moving load path forwards direction is defined by the order of selection of the lines, arcs or splines. The path starts at the first line that is connected to only one other selected line. If only a single line is selected, the forwards direction of the moving load path is defined by the line direction defined in Modeller (Line directions can be visualised through the Geometry layer in the Treeview). The direction that the moving load travels can be toggled between forwards and backwards relative to the original line selection for the generation of the discrete locations. The start and end of the path will be displayed along with the number of divisions / increments required for the full description.

The number of load cases per analysis controls the number of load cases for each LUSAS analysis used to obtain the loading information. By default this is set to 100 but can be increased up to 1000. Options are available to perform a datacheck analysis instead of a full static solution. The advantage of using this option is in both speed and the amount of disk space required for the LUSAS analyses. IMDPlus itself only requires the loading vector information from the datacheck analysis but if static solutions are also required for the passage of the moving load this option can be switched off. An option is also available not to run the analyses immediately. By default the LUSAS analyses will be carried out once the OK button has been clicked. If however the user wishes to run a large number of analyses resulting from the moving load definition overnight, this option can be switched off and a batch file generated so the analyses can be run separately. The original model, associated eigenvalue results and static load cases must be loaded prior to proceeding to the next stage of the moving load analysis.

This can be carried out by running the following two VBScript files which will have been generated in the directory **\<LUSAS Installation Folder>\Projects\<Model Name>\Associated Model Data\<Model Name>**

IMDPlus_Model_Reload.vbs

Reloads the model and eigenvalue results

IMDPlus_LoadVector_Load.vbs

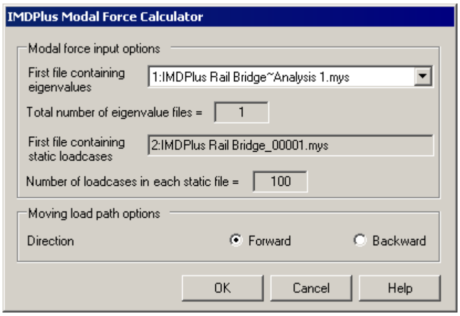
Loads the static results on top of the model and eigenvalue

Note. If for any reason the model and static load cases need to be loaded for the current moving load path and configuration in the future, these two VBScript files can be used.

Modal Force Calculator

The path and configuration of the moving load should have been defined using the moving load generation described previously. The discrete loads at distances along the path now need to be converted into equivalent modal forces for import into IMDPlus. This is carried out using the modal force calculator.

The IMDPlus modal force calculator dialog is accessed from the **Utilities > IMDPlus > Moving Load > Modal Force Calculator...** menu or from the IMDPlus Toolbar. Ensure the button is selected to choose a moving load analysis and then select the button to launch the modal force calculator dialog.



On entry, the eigenvalue and static results will be identified and placed into the dialog. Multiple eigenvalue results files are supported which allows the solution of structures with large numbers of eigenvalues over multiple analyses with frequency / eigenvalue ranges.

The moving load path options allow the user to change the direction of the load movement along the path but this option should not be used unless the load configuration is symmetrical.

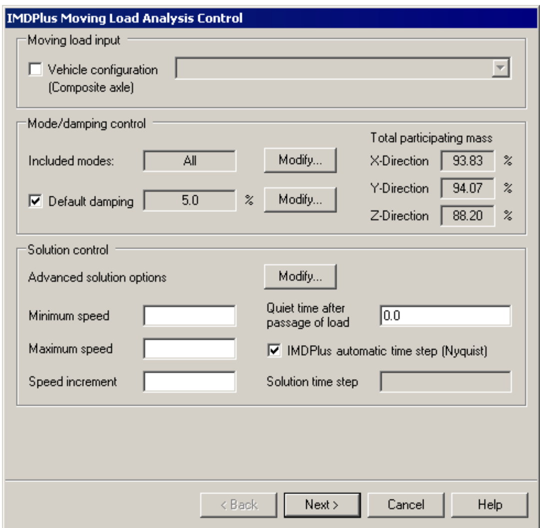
Note. Once the moving load generation and modal force calculation have been carried out in a moving load analysis, these two steps may be omitted from subsequent analyses, even if the analysis type is switched from moving load to moving mass but only if the moving load path, discrete load configuration and direction of the moving load do not change between analyses. If any of these are modified the moving load path and calculation of equivalent modal forces must be carried out again.

Running the Moving Load Analysis

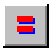
An IMDPlus moving load analysis can only be performed if the following steps have already been undertaken:

- The equivalent modal forces have been successfully calculated using the moving load generator and the modal force calculator
- If a composite axle definition is being used at least one moving load vehicle configuration has been defined in the model

The IMDPlus moving load analysis control dialog is shown in the following figure. On entering moving load analysis control for the first time the dialog will be filled with default information shown.

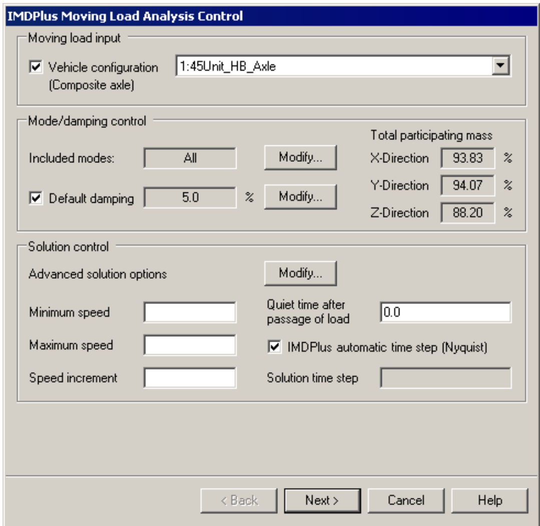


The IMDPlus moving load analysis control dialog is accessed from the **Utilities > IMDPlus > Moving Load > Moving Load Analysis...** menu or from the IMDPlus

Toolbar. Ensure the  button is selected to choose a moving load analysis and then select the  button to launch the moving load analysis control dialog.

Moving Load Input

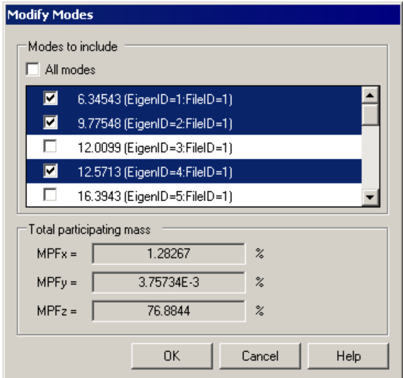
If a composite axle definition is being used in the analysis a vehicle configuration can be included by selecting the checkbox marked Vehicle configuration (Composite axle). This will enable the drop-down list that will contain all of the moving load vehicle configurations in the model. The required vehicle configuration can be selected for use in the IMDPlus analysis, as shown by the example in the following figure.



Note. Although a vehicle configuration (composite axle definition) is designed for internal construction of complex loading configurations from a single axle, for example defining a complete trainset from a single unit axle, it can also be used to model the passage of more than one load configuration across the structure. Each load configuration is restricted to having the same plan layout but can have different magnitudes governed by the load factors contained in the vehicle configuration.

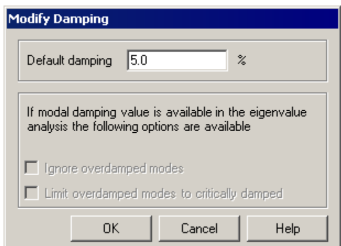
Mode / Damping Control

The modes to include in the analysis can be controlled either by the eigenmodes solved in the eigenvalue analysis or a subset defined in the mode control. The default option when first running an analysis is for all modes to be included in the analysis but individual modes can be included or excluded using the following dialog accessed via the **Modify…** button on the moving load analysis dialog.



This dialog also calculates the total participating mass for the included modes. If a significant proportion of participating mass is missing based on the selected modes of vibration a warning will be issued when proceeding with the analysis. Omission of modes of vibration with significant mass contributions that can be excited during the analysis will lead to unreliable solutions. Ideally, total mass participations in excess of 90% should be used unless it is guaranteed that the modes of vibration associated with any missing mass are at frequencies that cannot be excited.

The Default damping option allows the user to control the amount of damping used in the IMDPlus analysis. If Default damping is selected, all of the modes of vibration included in the analysis are forced to have the current default value. The default value is set, via the **Modify…** button on the moving load analysis control dialog.



The Default damping option can only be turned off when viscous damping has been included in the eigenvalue analysis. If Default damping is deselected the viscous damping present in the eigenvalue analysis is used in the IMDPlus analysis. When viscous damping is included in the eigenvalue solution additional options for the inclusion of overdamped modes of vibration in the IMDPlus analysis also become available. These options are as follows:

- **Ignore overdamped modes**

  Modes of vibration with a damping ratio of 100% or more are omitted from the IMDPlus solution.

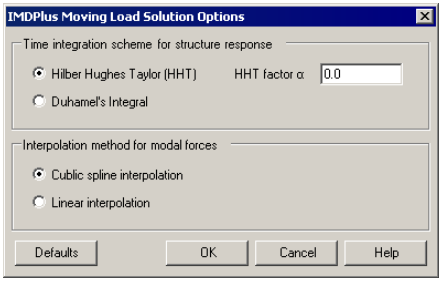
- **Limit overdamped modes to critically damped**
  Modes of vibration with a damping ratio of 100% or more are limited to 99.9999% to allow them to be included in the analysis.

Solution Control

The speeds and time stepping information are input via the solution control section. The input includes a range of speeds for the solution defined using a minimum speed, maximum speed and a speed increment. If only a minimum or maximum speed is specified then one speed, equal to the value entered, will be analysed for the moving load. A quiet time can be specified which allows for the inclusion of a user defined length of time after the passage of the load along the path. This enables decay of the structural vibration to be analysed. By default, IMDPlus determines the time step required for the analysis from the Nyquist time step. This is calculated from the maximum frequency included in the analysis. User control over the time step for the solution is available by disabling this feature.

Advanced Moving Load Solution Options

Advanced solution options can be set via the **Modify...** button on the moving load analysis control dialog.



The options available in a moving load analysis are as follows. The **Defaults** button resets all of the options to the default values.

- **Time integration scheme for structure response**
  The implicit time integration scheme used by IMDPlus to calculate the dynamic response of the structure.

  - **Hilber Hughes Taylor (HHT)**
    This is the default time integration scheme and is the same as the method used for an implicit transient dynamic analysis in LUSAS.

    - **HHT factor α**
      The integration constant α used in the HHT integration scheme. For an unconditionally stable, second order accurate scheme, α must lie between the limits $-1/3 \le \alpha \le 0.0$. Values of α outside this range cannot be

specified. The default value of α is 0.0, which reduces the method to the Newmark method. Further integration constants β and γ are computed automatically by IMDPlus using the following equations:

$$\beta = (1-\alpha)^2 / 4 \quad , \quad \gamma = (1-2\alpha)/2$$

- **Duhamel's Integral**

  This is the time integration method used by IMDPlus in LUSAS 14.7 (and earlier versions of LUSAS) and is available as an alternative to the HHT integration scheme.

- **Interpolation method for modal forces**

  The interpolation method used by IMDPlus to calculate the equivalent modal forces from the values obtained at the discrete path locations by the modal force calculator.

  - **Cubic spline interpolation**

    Equivalent modal forces are interpolated using a cubic spline interpolation technique. This is the default option.

  - **Linear interpolation**

    Equivalent modal forces are interpolated using a linear interpolation technique.

Moving Mass Analysis

The moving mass analysis option obtains the dynamic response of the structure to the passage of moving spring-mass systems along a user defined path. Both 2D and 3D structures can be analysed. The configuration of the spring-mass systems remains constant throughout the analysis but as they move across the structure, the dynamic response of the unsprung and sprung masses affects the forces acting on the underlying structure due to inertia effects.

It is important to note the following working assumptions for the case of a moving mass analysis:

- Only vertical motion of the spring-mass systems is considered
- There is no loss of contact between the unsprung masses (wheels) of the spring–mass systems and the structure at any time
- The mass of the spring-mass systems have no effect on the natural frequencies of the structure
- The accuracy of the moving mass solution reduces for light-weight structures or structures where the magnitude of the moving mass is similar to the magnitude of the structure mass. A literature search recommends that for most solutions of

this type, the total mass of all the spring-mass systems that are actively interacting with the structure at any given time should not exceed 10% of the total mass of the underlying structure.

The following two steps are required to define the moving spring-mass systems that are to be used in the analysis:

1. Define the spring-mass system property attribute(s)
2. Define the moving mass vehicle configuration(s) using the positions of the axles/bogies that form a vehicle together with the spring-mass system property attribute that is to be used at each position

Two further preliminary steps are required, in a similar manner to a moving load analysis, before a moving mass analysis can be undertaken. These generate equivalent modal forces by simulating the movement of a discrete unit load across the structure, using a number of distinct locations along a moving load path. The two steps are as follows:

3. Generation of the moving load path across the structure

   A discrete load representing the unit load configuration for the axles/bogies of the vehicle that is to be moved across the structure must be defined. The moving load generator is then used to mimic the path of this unit load across the structure by automatically setting up a number of static load cases at prescribed locations along the path that the vehicle configuration is to follow.

4. Calculation of equivalent modal forces for the moving load path

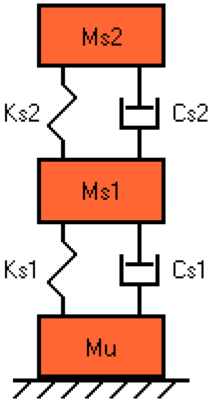
   The static load cases are used to calculate the modal forces equivalent to the applied unit loading for import into IMDPlus. The equivalent modal forces are used in IMDPlus together with the loading from the spring-mass systems of the vehicle to build up the complete loading configuration applied to the structure.

Note. Steps 3 and 4 must be carried out before the first analysis but may be omitted from subsequent analyses, if and only if the moving load path, discrete unit load configuration and direction of the moving load do not change between analyses. If any of these are modified the moving load path and calculation of equivalent modal forces must be carried out again.

Spring-Mass System Properties

A spring-mass system is represented by a single unsprung mass and up to two sprung masses, connected by springs and viscous dashpots, as shown in the following figure. A spring-mass system property attribute defines the masses, linear or nonlinear spring stiffnesses and viscous damping properties for a spring-mass system. Spring-mass system properties are assigned to different initial positions in a moving mass vehicle configuration in order to represent all of the axle/bogie locations of an entire vehicle or trainset. As these systems move across the structure, the motion of the unsprung and sprung masses affects the forces acting on the underlying structure due to inertia effects.

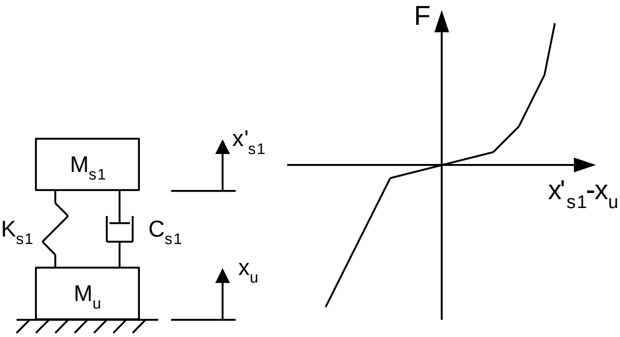
Note. A vehicle or trainset may contain several axle/bogie positions with the same spring-mass system properties, that is, masses, spring stiffnesses and viscous damping constants. A spring–mass system property attribute can therefore be assigned to multiple axle/bogie positions when defining a vehicle configuration.



Different spring behaviour can be modelled ranging from a simple linear elastic spring through to more complex nonlinear behaviour. Linear viscous dashpots can be optionally included for each of the sprung masses. Nonlinear spring behaviour allows for the inclusion of the following spring types.

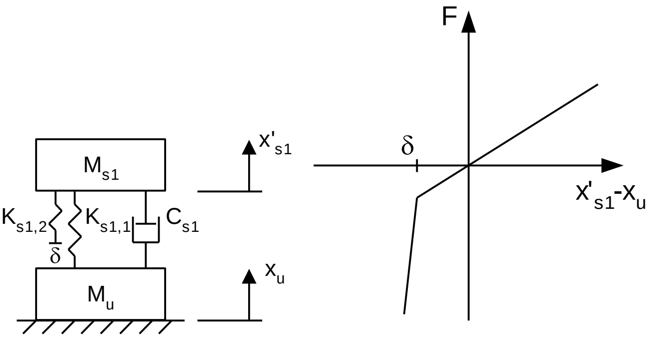
Piecewise linear springs

Piecewise linear springs can be used to model bilinear springs or more general nonlinear spring behaviour.



Piecewise-linear spring: Nonlinear spring behaviour

Piecewise linear springs can also be used to simulate stopper systems that limit the spring deflections through the inclusion of very high compressive spring stiffnesses after a given closure.



Piecewise-linear spring: Stopper system to limit deflections

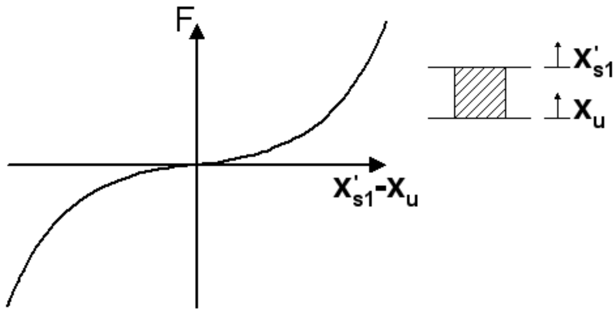
Polynomial Springs

Polynomial springs can be used to represent nonlinear spring behaviour using the following function:

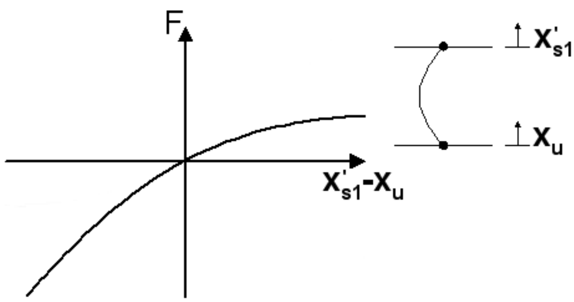
$$F = \sum_{a=1}^{N} C_a dx^a$$

The total spring force, F, is expressed in terms of the total spring deflection, dx, where N is the order of the polynomial and C_a are the polynomial constants. This allows for the special cases of rubber springs and 'C' springs, as shown in the following figures, along with more general spring definitions.

Note. The polynomial function always passes through the origin of the spring force-displacement curve. This ensures that the spring force is zero at zero spring displacement.

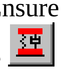


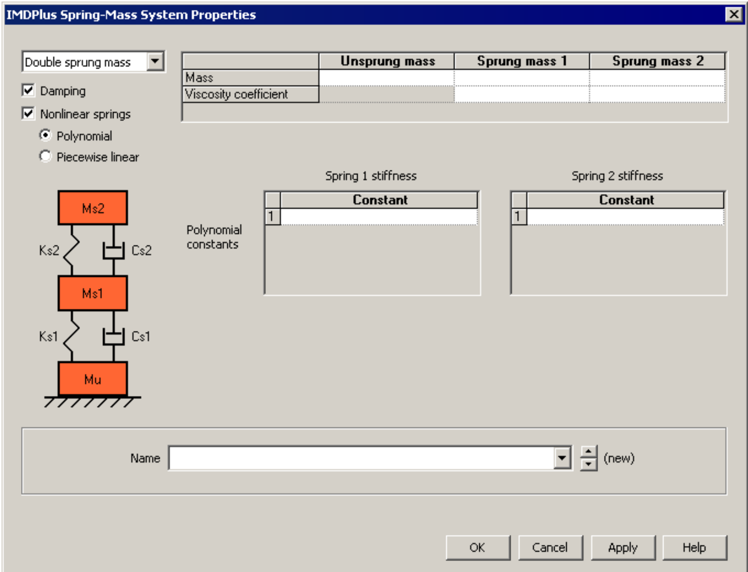
Polynomial spring: Rubber Spring $F = C_1(x'_{s1}-x_u) + C_3(x'_{s1}-x_u)^3$



Polynomial spring: 'C' Spring $F = C_1(x'_{s1}-x_u) + C_2(x'_{s1}-x_u)^2$

Spring Mass System Properties Dialog

The IMDPlus moving mass spring-mass system properties dialog is used to create or modify spring-mass system property attributes for use in a moving mass analysis. It is accessed from the **Utilities > IMDPlus > Moving Mass > Spring Mass Properties...** menu or from the IMDPlus Toolbar. Ensure the  button is selected to choose a moving mass analysis and then select the  button to launch the moving mass spring-mass system properties dialog.



Options for the spring-mass system are located in the upper left hand corner of the dialog. These define the number of sprung masses in the system and allow for the optional inclusion of damping and nonlinear spring behaviour. The number of sprung masses is controlled by selecting one of the following options from the drop-down list.

- ❑ **Unsprung mass only**
- ❑ **Single sprung mass**
- ❑ **Double sprung mass**

On the left hand side of the dialog a diagram of the chosen spring-mass system is displayed. All property data is input on the right hand side of the dialog as follows:

- ❑ **Unsprung and sprung masses**
  - Unsprung and sprung masses should be defined in the model mass units
  - If required, the unsprung mass can be set to 0.0 for a **Single sprung mass** or **Double sprung mass** system
  - If required, the 1st sprung mass can be set to 0.0 for a **Double sprung mass** system
- ❑ **Elastic spring stiffness**
  - Elastic spring stiffnesses are only required if **Nonlinear springs** has not been selected
  - Elastic spring stiffnesses should be defined in the model stiffness units

- **Viscosity coefficient**
  - Viscosity coefficients are only required if **Damping** has been selected
  - Viscosity coefficients should be defined in the model force*time/length units
  - If required, the viscosity coefficients can be set to 0.0
- **Nonlinear springs**

  If Nonlinear springs is selected the spring behaviour is defined using either polynomial constants or piecewise linear data.

  - **Polynomial springs**

    The spring force-displacement relationship is defined using a polynomial function

    - The polynomial constants C_a should be entered in the table where row 1 defines the linear constant C_1, row 2 defines the quadratic constant etc.
    - The polynomial function must have a positive value at a displacement of 0.0
    - At least one of the polynomial constants must be non zero
    - IMDPlus requires the spring stiffness to always be a positive value. With higher order functions the gradient of the force-displacement relationship will generally reach a peak and become negative at a limiting compressive and/or tensile displacement. In these cases the IMDPlus analysis will terminate if the spring displacements ever reach these limiting values.
  - **Piecewise linear springs**

    The spring stiffness-deformation relationship is defined using a piecewise linear curve

    - Compressive and tensile data is input separately to ensure that the resulting force-displacement curves always pass through the origin
    - **Stiffness**
      - Spring stiffnesses should be defined in the model stiffness units
      - Spring stiffnesses must always be defined as a positive value
    - **Deformation**
      - Spring deformations should be defined in the model length units
      - The deformation of the first data point (row 1) of each piecewise linear curve must be set to 0.0
      - In compression all deformations must be negative and must be monotonically decreasing
      - In tension all deformations must be positive and must be monotonically increasing

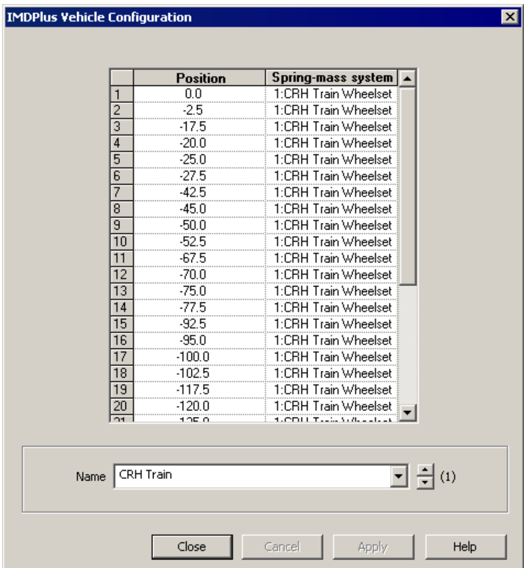
A meaningful name should be entered into the Name field.

Note. Once created spring-mass system properties will appear in the Utilities Treeview, from where they can be modified or deleted from the model. To modify a spring-mass system property either double-click or right-click on the Attribute name in the Utilities Treeview and select the **Edit Attribute...** menu item.

Note. Spring-mass system properties can be easily transferred from one model to another using an attribute library file, accessed from the menu **File > Import/Export model data...**

Moving Mass Vehicle Configuration

The IMDPlus moving mass vehicle configuration dialog is used to create or modify vehicle configuration attributes for use in a moving mass analysis. It is accessed from the **Utilities > IMDPlus > Moving Mass > Vehicle Configuration...** menu or from the IMDPlus Toolbar. Ensure the button is selected to choose a moving mass analysis and then select the button to launch the moving mass vehicle configuration dialog.



In an IMDPlus moving mass analysis a vehicle or trainset is represented by a number of spring-mass systems. A moving mass vehicle configuration includes the position (relative to the origin / front of the vehicle) of the axles/bogies that form the vehicle along with the spring-mass system attribute to be used at each position. Each row of the Spring-mass system column includes a drop-down list that contains all of the spring-mass system property attributes in the model. The spring-mass system properties required at each position can be selected for use in the vehicle configuration. A spring–mass system property attribute can be used at multiple positions in a vehicle. An example of a moving mass vehicle configuration is shown in the figure. A meaningful name should be entered into the Name field.

Note. Care should be taken with the definition of the spring-mass system positions. The path length should be sufficiently longer than the structure to ensure that the vehicle configuration correctly arrives onto and departs from the structure.

Note. The front of the moving mass vehicle is always the position with the maximum co-ordinate. The rear of the vehicle is always the position with the minimum co-ordinate.

Note. The positions and spring-mass system identifiers that define the vehicle configuration can be copied and pasted from a Microsoft Excel spreadsheet. This method is demonstrated in the IMDPlus Worked Example: High Speed Train Modelling through Sprung Masses.

Note. Once created vehicle configurations will appear in the Utilities Treeview, from where they can be modified or deleted from the model. To modify a vehicle configuration either double-click or right-click on the Attribute name in the Utilities Treeview and select the **Edit Attribute...** menu item.

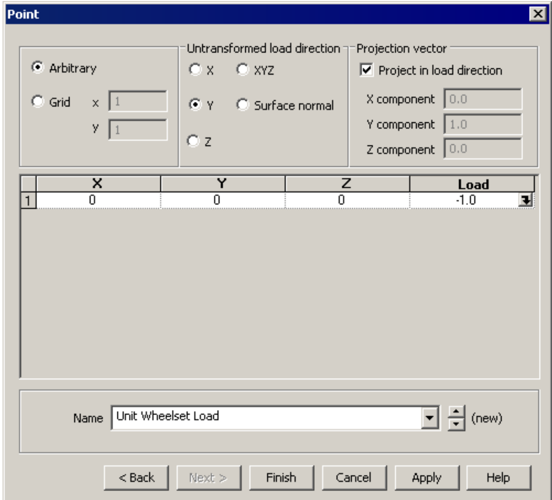
Note. Vehicle configurations can be easily transferred from one model to another using an attribute library file, accessed from the menu **File > Import/Export model data...**

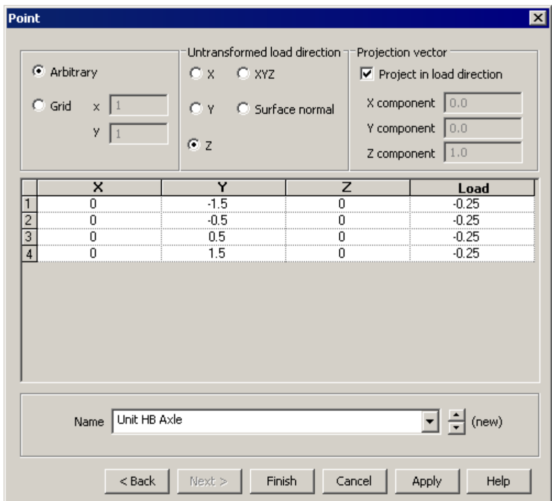
Moving Load Generation

In a moving mass analysis the moving load generator is used to mimic the path of a unit load across a structure by automatically setting up a number of static load cases at prescribed locations along a single line/arc/spline, or a collection of lines/arcs/splines, which define a single continuous path. These load cases are subsequently used to calculate the unit modal forces equivalent to the applied unit loading for import into IMDPlus, where the unit modal forces are used together with the vertical loading from each of the spring-mass systems to obtain the total modal force applied to the structure.

Note. In a moving mass analysis a discrete unit load must always be used in the moving load generator to ensure that the correct loads are applied to the structure during the IMDPlus analysis. The discrete unit load is used to calculate the unit modal forces that are applied to the structure as the discrete unit load moves along the moving load path. These unit modal forces are used in IMDPlus in conjunction with the vertical loading from the spring-mass systems of the vehicle configuration to calculate the total modal forces applied to the structure.

Before entering the moving load generator, the lines/arcs/splines defining the path must be selected and a discrete load representing the unit load configuration for the axles/bogies of the vehicle that is to be moved across the structure must have been defined. Examples of discrete unit loads are shown in the following figures.

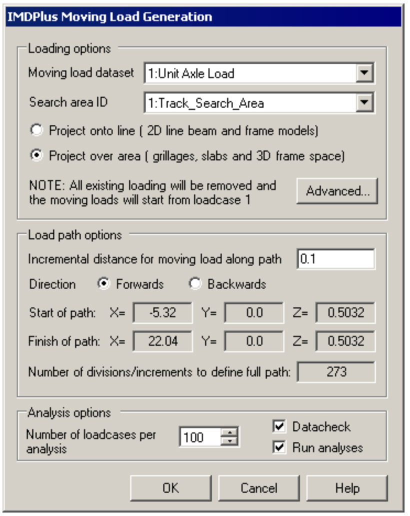




Generation of the Moving Load Discrete Locations

The IMDPlus moving load generation dialog is accessed from the **Utilities > IMDPlus > Moving Mass > Moving Load Generation...** menu or from the IMDPlus Toolbar. Ensure the  button is selected to choose a moving mass analysis and then select the  button to launch the moving load generation dialog.

If a valid continuous path has been selected and a valid discrete unit load defined then the moving load generator can be used to create the static load cases at prescribed locations along the path.



If a search area has been defined then this can be used to correctly assign the moving load if there are multiple planes to which the discrete unit load could be assigned. **Project onto line** or **Project over area** can be selected depending upon the geometry defining the search area. For single deck models, the default search area which encompasses the whole structure can be used. Parameters for the inclusion of the load can be accessed through the **Advanced…** button which provides the full application options available for discrete unit loads. These options consist of the inclusion of loads outside the search area and the load factor to be applied. The load factor for a moving mass analysis should always be set to 1.0.

Note. In a moving mass analysis the load factor defined under the **Advanced…** options should always be set to 1.0 as the moving load generator needs to define the passage of a discrete unit load across the structure. Any factor other than 1.0 can cause the incorrect static and dynamic loads to be calculated during a moving mass analysis.

Having defined the loading options, the load path options can be set based on the lines/arcs/splines selected to define the path. The incremental distance controls the separation of the discrete unit load locations used to define the passage of the unit load across the structure. This incremental distance should be positive and sufficiently small to capture the movement of the load.

Note. Due to the dynamic solution a sufficiently small distance should be used between the discrete locations defining the passage of the unit load to minimise oversampling for load locations between those defined.

Ideally, the distance increment used for the movement of the unit load along the path should obey the following equation:

$$\delta \mathbf{Dist} \leq \mathbf{10} \times \mathbf{Minimum\ Speed} \times \delta \mathbf{t}$$

to give a maximum oversample of 10. δt is the minimum of the Nyquist time step and any user specified time step ($\delta t^{Nyquist} = 1/(2*f_{max})$ where f_{max} is the maximum frequency included in the analysis). Larger distance increments can be used but warnings will be issued by IMDPlus since the accuracy of the dynamic solution decreases with increasing oversampling ratio. Oversampling ratio reduces with increasing speed and therefore it is possible that higher oversampling ratios can be used for lower speeds where the dynamic excitation is reduced. It is however up to the user to ensure that use of higher oversampling ratios for any speed does not adversely affect the solutions to the dynamic analysis.

The moving load path forwards direction is defined by the order of selection of the lines, arcs or splines. The path starts at the first line that is connected to only one other selected line. If only a single line is selected, the forwards direction of the moving load path is defined by the line direction defined in Modeller (Line directions can be visualised through the Geometry layer in the Treeview). The direction that the moving load travels can be toggled between forwards and backwards relative to the original line selection for the generation of the discrete locations. The start and end of the path will be displayed along with the number of divisions / increments required for the full description.

The number of load cases per analysis controls the number of load cases for each LUSAS analysis used to obtain the loading information. By default this is set to 100 but can be increased up to 1000. It is recommended that the option to perform a datacheck analysis instead of a full static solution is always selected as an IMDPlus moving mass analysis only requires the unit loading vector information from the datacheck analysis.

Note. In a moving mass analysis a full static solution will only take into account the loading from the discrete unit load and will not include the total static loading from the spring-mass systems of the vehicle configuration. The results of the static analysis will therefore not correspond to the total vehicle loading that is going to be applied to the structure.

An option is also available not to run the analyses immediately. By default the LUSAS analyses will be carried out once the OK button has been clicked. If however the user wishes to run a large number of analyses resulting from the moving load definition overnight, this option can be switched off and a batch file generated so the analyses can be run separately. The original model, associated eigenvalue results and static load cases must be loaded prior to proceeding to the next stage of the moving mass analysis. This can be carried out by running the following two VBScript files which will have been generated in the directory **\<LUSAS Installation Folder>\Projects\<Model Name>\Associated Model Data\<Model Name>**

IMDPlus_Model_Reload.vbs

Reloads the model and eigenvalue results

IMDPlus_LoadVector_Load.vbs

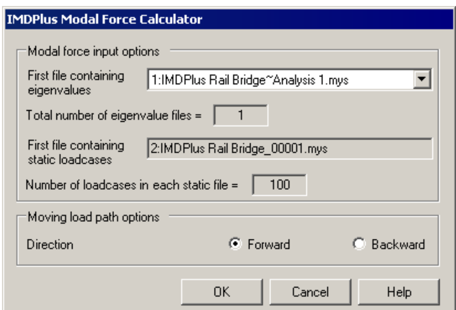
Loads the static results on top of the model and eigenvalue

Note. If for any reason the model and static load cases need to be loaded for the current moving load path and configuration in the future, these two VBScript files can be used.

Modal Force Calculator

The path and configuration of the moving unit load should have been defined using the moving load generation described previously. The discrete unit loads at distances along the path now need to be converted into equivalent modal forces for import into IMDPlus. This is carried out using the modal force calculator.

The IMDPlus modal force calculator dialog is accessed from the **Utilities > IMDPlus > Moving Mass > Modal Force Calculator...** menu or from the IMDPlus Toolbar. Ensure the button is selected to choose a moving mass analysis and then select the button to launch the modal force calculator dialog.



On entry, the eigenvalue and static results will be identified and placed into the dialog. Multiple eigenvalue results files are supported which allows the solution of structures with large numbers of eigenvalues over multiple analyses with frequency / eigenvalue ranges.

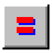
The moving load path options allow the user to change the direction of the unit load movement along the path but this option should not be used unless the load configuration is symmetrical.

Note. Once the moving load generation and modal force calculation have been carried out in a moving mass analysis, these two steps may be omitted from subsequent analyses, even if the analysis type is switched from moving mass to moving load but only if the moving load path, discrete load configuration and direction of the moving load do not change between analyses. If any of these are modified the moving load path and calculation of equivalent modal forces must be carried out again.

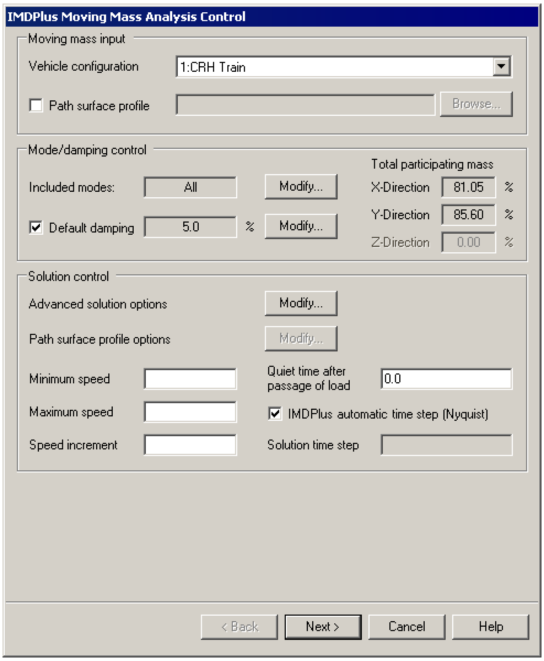
Running the Moving Mass Analysis

An IMDPlus moving mass analysis can only be performed if the following steps have already been undertaken:

- The equivalent modal forces have been successfully calculated using the moving load generator and the modal force calculator. A discrete load representing the unit load configuration for the axles/bogies of the vehicle that is to be moved across the structure must be used in the calculation of the modal forces to ensure that the correct loads are applied to the structure during the moving mass analysis.
- At least one moving mass vehicle configuration has been defined in the model

The IMDPlus moving mass analysis control dialog is accessed from the **Utilities > IMDPlus > Moving Mass > Moving Mass Analysis...** menu or from the IMDPlus Toolbar. Ensure the  button is selected to choose a moving mass analysis and then select the  button to launch the moving mass analysis control dialog.

On entering moving mass analysis control for the first time the dialog will be filled with default information, as shown by the example in the following figure.



Moving Mass Input

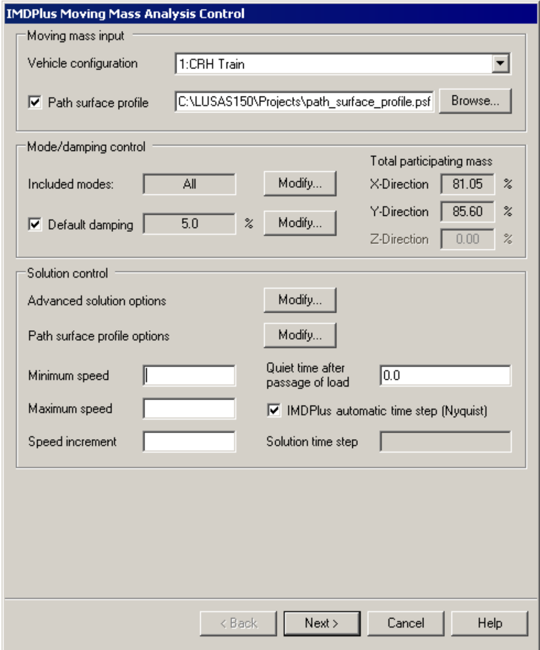
Vehicle Configuration

The vehicle configuration drop-down list will contain all of the moving mass vehicle configurations in the model. The required vehicle configuration can be selected for use in the IMDPlus analysis.

Path Surface Profile

The surface roughness of the structure along the path followed by the moving mass vehicle can be optionally included via a perturbation of the vertical displacement of the structure. This is termed the path surface profile displacement and it is added to the displacements of the unsprung masses during the analysis. The path surface profile velocity and acceleration vertical components are calculated during the IMDPlus analysis and also applied to the unsprung masses for dynamic consistency.

The path surface profile displacements are defined in a path surface profile file. Select the checkbox marked Path surface profile to include a path surface profile file in the IMDPlus moving mass analysis. This will enable the **Browse...** button so that the file can be selected, as shown by the example in the following figure.



The path surface profile file can contain either space, Comma Separated Variable (CSV) or TAB delimited data and must include a piecewise linear displacement vs. distance relationship that defines the surface irregularities along the path traversed by the moving mass vehicle. An example of a path surface profile is shown in the following figure. There should be a total of NPSP lines of data (NPSP defined below) in the file. All input data must be valid integer or real numbers and each line of the file should contain two data records as follows:

For i=1,NPSP

Dist_i PSP_i

End loop over NPSP

The path surface profile data is defined as follows:

- **NPSP**
  - The number of data points in the path surface profile
  - There must be a minimum of 2 data points in order to define a valid profile
- **Dist_i**

  The distance relative to the origin of the moving load path
  - Distances should be defined in the model length units
  - The distance of the first data point must be 0.0

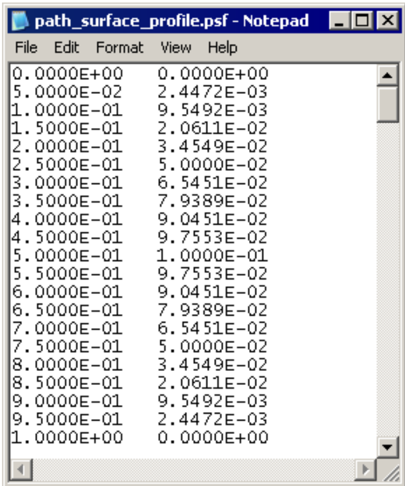
- The distances must be defined so that they are monotonically increasing
- Care should be taken to ensure that the last distance point in the file is equal to or greater than the length of the path. If this is not the case an additional point is automatically introduced during the IMDPlus analysis with a distance equal to the path length and a displacement of 0.0

❑ **$\mathtt{PSP_i}$**

The surface irregularity vertical displacement at the distance **$\mathtt{Dist_i}$**

- Displacements should be defined in the model length units
- The displacement of the first data point must be 0.0

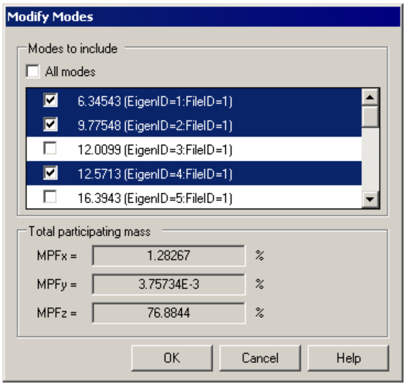
An example of a path surface profile file is shown in the following figure.




```
0.0000E+00   0.0000E+00
5.0000E-02   2.4472E-03
1.0000E-01   9.5492E-03
1.5000E-01   2.0611E-02
2.0000E-01   3.4549E-02
2.5000E-01   5.0000E-02
3.0000E-01   6.5451E-02
3.5000E-01   7.9389E-02
4.0000E-01   9.0451E-02
4.5000E-01   9.7553E-02
5.0000E-01   1.0000E-01
5.5000E-01   9.7553E-02
6.0000E-01   9.0451E-02
6.5000E-01   7.9389E-02
7.0000E-01   6.5451E-02
7.5000E-01   5.0000E-02
8.0000E-01   3.4549E-02
8.5000E-01   2.0611E-02
9.0000E-01   9.5492E-03
9.5000E-01   2.4472E-03
1.0000E+00   0.0000E+00
```

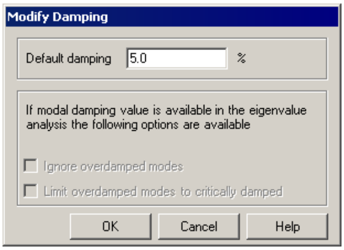
Mode / Damping Control

The modes to include in the analysis can be controlled either by the eigenmodes solved in the eigenvalue analysis or a subset defined in the mode control. The default option when first running an analysis is for all modes to be included in the analysis but individual modes can be included or excluded using the following dialog accessed via the **Modify…** button on the moving mass analysis dialog.



This dialog also calculates the total participating mass for the included modes. If a significant proportion of participating mass is missing based on the selected modes of vibration a warning will be issued when proceeding with the analysis. Omission of modes of vibration with significant mass contributions that can be excited during the analysis will lead to unreliable solutions. Ideally, total mass participations in excess of 90% should be used unless it is guaranteed that the modes of vibration associated with any missing mass are at frequencies that cannot be excited.

The Default damping option allows the user to control the amount of damping used in the IMDPlus analysis. If Default damping is selected, all of the modes of vibration included in the analysis are forced to have the current default value. The default value is set, via the **Modify…** button on the moving mass analysis control dialog.



The Default damping option can only be turned off when viscous damping has been included in the eigenvalue analysis. If Default damping is deselected the viscous damping present in the eigenvalue analysis is used in the IMDPlus analysis. When viscous damping is included in the eigenvalue solution additional options for the inclusion of overdamped modes of vibration in the IMDPlus analysis also become available. These options are as follows:

- **Ignore overdamped modes**

  Modes of vibration with a damping ratio of 100% or more are omitted from the IMDPlus solution.

- ❑ **Limit overdamped modes to critically damped**

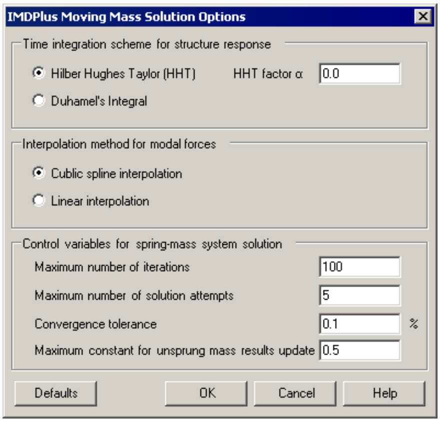
  Modes of vibration with a damping ratio of 100% or more are limited to 99.9999% to allow them to be included in the analysis.

Solution Control

The speeds and time stepping information are input via the solution control section. The input includes a range of speeds for the solution defined using a minimum speed, maximum speed and a speed increment. If only a minimum or maximum speed is specified then one speed, equal to the value entered, will be analysed for the vehicle configuration selected. A quiet time can be specified which allows for the inclusion of a user defined length of time after the passage of the vehicle along the path. This enables decay of the structural vibration to be analysed. By default, IMDPlus determines the time step required for the analysis from the Nyquist time step. This is calculated from the maximum frequency included in the analysis. User control over the time step for the solution is available by disabling this feature.

Advanced Moving Mass Solution Options

Advanced solution options can be set via the **Modify...** button on the moving mass analysis control dialog.



The options available in a moving mass analysis are as follows. The **Defaults** button resets all of the options to the default values.

- ❑ **Time integration scheme for structure response**

  The implicit time integration scheme used by IMDPlus to calculate the dynamic response of the structure.

  - ❑ **Hilber Hughes Taylor (HHT)**

    This is the default time integration scheme and is the same as the method used for an implicit transient dynamic analysis in LUSAS.

- **HHT factor α**

  The integration constant α used in the HHT integration scheme. For an unconditionally stable, second order accurate scheme, α must lie between the limits $-1/3 \le \alpha \le 0.0$. Values of α outside this range cannot be specified. The default value of α is 0.0, which reduces the method to the Newmark method. Further integration constants β and γ are computed automatically by IMDPlus using the following equations:

$$\beta = (1-\alpha)^2 / 4 \quad , \quad \gamma = (1-2\alpha)/2$$

- **Duhamel's Integral**

  This is the time integration method used by IMDPlus in LUSAS 14.7 (and earlier versions of LUSAS) and is available as an alternative to the HHT integration scheme.

Note. When Duhamel's Integral is selected the HHT factor α remains active and can still be modified. In moving mass analysis the HHT scheme is always used to calculate the dynamic response of the spring-mass systems, irrespective of the time integration scheme selected. The value of α will therefore always have an effect on a moving mass analysis.

- **Interpolation method for modal forces**

  The interpolation method used by IMDPlus to calculate the equivalent modal forces from the values obtained at the discrete path locations by the modal force calculator.

  - **Cubic spline interpolation**

    Equivalent modal forces are interpolated using a cubic spline interpolation technique. This is the default option.

  - **Linear interpolation**

    Equivalent modal forces are interpolated using a linear interpolation technique.

- **Control variables for spring-mass system solution**

  The variables used to control the solution of the spring-mass systems as they move across the structure.

  - **Maximum number of iterations**

    The maximum number if iterations allowed for the spring-mass system and structural solution. The default is 100 iterations per solution attempt. If convergence problems occur this value may need to be increased.

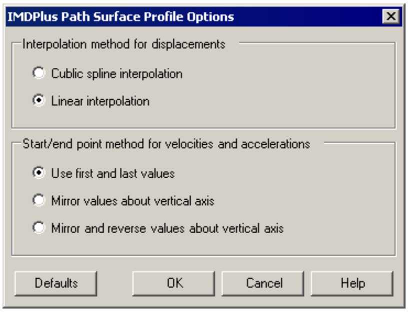
- **Maximum number of solution attempts**
  The maximum number of solution attempts allowed during the spring-mass system and structural solution for each time step of the analysis. The default is 5. If convergence problems occur this value may need to be increased.

- **Convergence tolerance**
  The convergence tolerance used for the spring-mass system and structural solution. The default is 0.1%. Increasing the convergence tolerance can reduce the time taken to perform an analysis but may produce inaccurate results.

- **Maximum constant for unsprung mass results update**
  The maximum constant, C_{USM}, used in the calculation of the unsprung mass results. The default is 0.5 and the constant must lie between the limits $0.0 < C_{USM} \leq 1.0$. Larger values of C_{USM} can be used to reduce the time taken to perform an analysis but may result in convergence problems. If convergence problems do occur this value may need to be reduced.

Path Surface Profile Options

If a path surface profile has been included in the analysis advanced path surface profile options can be set via the **Modify...** button on the moving mass analysis control dialog.



The path surface profile options are as follows. The **Defaults** button resets all of the options to the default values.

- **Interpolation method for displacements**
  The interpolation method used by IMDPlus to calculate the path surface profile displacements from the discrete values defined in the path surface profile file.

  - **Cubic spline interpolation**
    Path surface profile displacements are interpolated using a cubic spline interpolation technique.

- **Linear interpolation**

  Path surface profile displacements are interpolated using a linear interpolation technique. This is the default option.

- **Start/end point method for velocities and accelerations**

  IMDPlus calculates path surface profile velocities and accelerations from the displacement data defined in the path surface profile file. In order to calculate the complete velocity and acceleration profiles, displacement values beyond the extent of the user defined data are required. This option defines the method that is used to extrapolate the displacement profile at the start and end times of the analysis.

  - **Use first and last values**

    Displacement data is extrapolated using the path surface profile displacement values at the start and end of the analysis. This assumes that the displacement profile continues indefinitely with the first and last values. This is the default option and will be applicable in the majority of analyses.

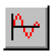
  - **Mirror values about vertical axis**

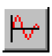
    Displacement data is extrapolated by mirroring it about the vertical axis at the start and end of the analysis. For example, this option would be applicable where the path surface profile is defined using a cosine function which has peak values at the start and end of the analysis.

  - **Mirror and reverse values about vertical axis**

    Displacement data is extrapolated by mirroring it about the vertical axis at the start and end of the analysis and then multiplying the extrapolated data by -1. For example, this option would be valid where the path surface profile is defined using a sine function with zero displacement at the start and end of the analysis.

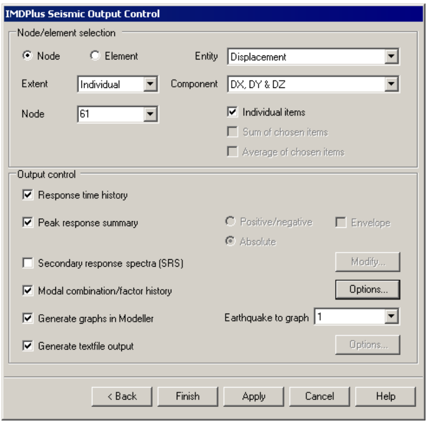
Visualising the Results

Results are extracted from IMDPlus through the output control dialog which is used for all of the seismic, moving load and moving mass analysis options. The IMDPlus output control dialog is accessed by selecting **Next>** from the seismic, moving load or moving mass analysis control dialogs or by selecting the  button from the IMDPlus Toolbar.

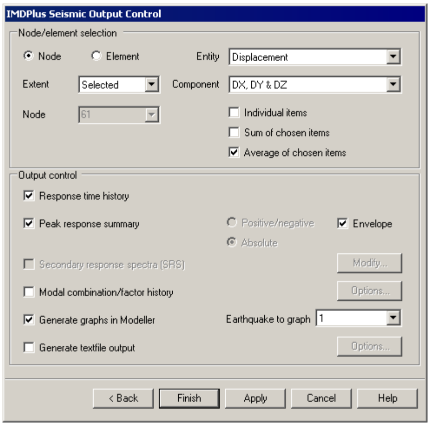
Note. The  button will only be available for selection when an IMDPlus analysis has been performed and the results are still available on disk for graphing.

For a seismic analysis the nodes or elements that are to be analysed are defined, together with the type of output that will be generated by IMDPlus. In addition, the

Earthquake to graph is selected using the drop-down list next to the **Generate graphs in Modeller** option. Examples of the appearance of the seismic output control dialog when processing an individual node or a selection of nodes are illustrated in the following figure.



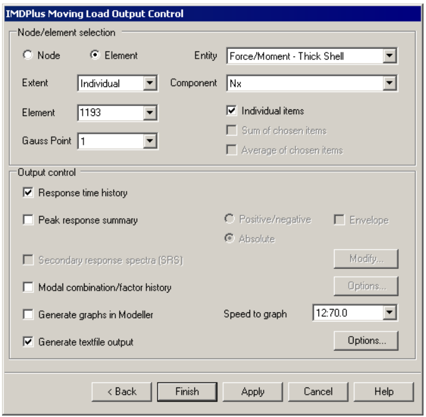
Individual Node

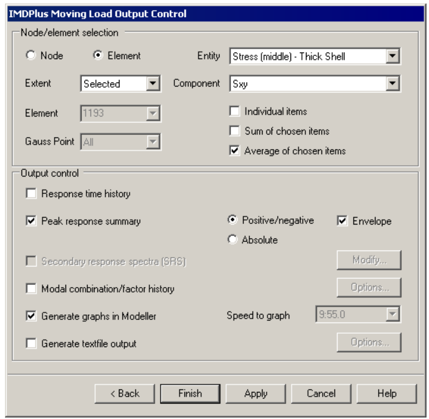


Selected Nodes

Seismic Analysis – Output Control Dialog

For a moving load analysis the nodes or elements that are to be analysed are defined, together with the type of output that will be generated by IMDPlus. In addition, the moving load **Speed to graph** is selected via the list next to the **Generate graphs in Modeller** option. Examples of the appearance of the moving load output control dialog when processing an individual element or a selection of elements are illustrated in the following figure.

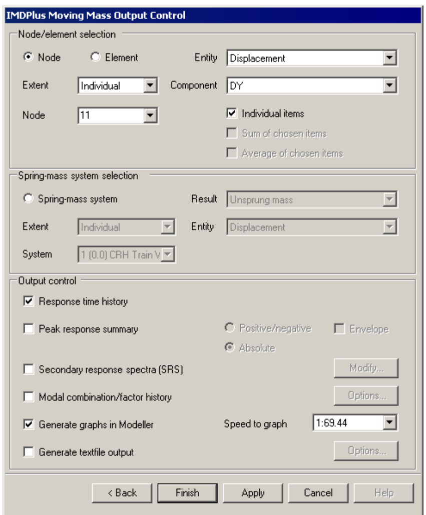


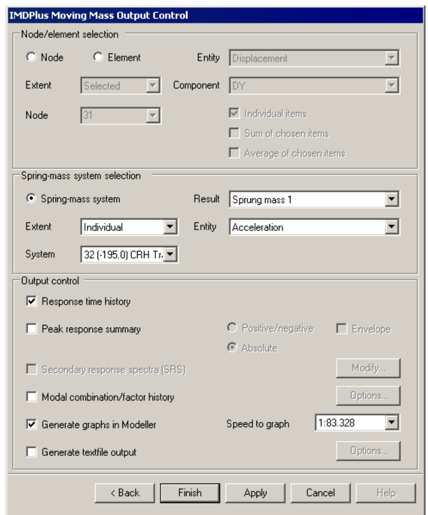


Individual Element **Selected Elements**

Moving Load Analysis – Output Control Dialog

For a moving mass analysis the nodes, elements or spring-mass systems that are to be analysed are defined, together with the type of output that will be generated by IMDPlus. In addition, the moving mass **Speed to graph** is selected via the list next to the **Generate graphs in Modeller** option. Examples of the appearance of the moving mass output control dialog when processing an individual node or an individual spring-mass system are illustrated in the following figure.





Individual Node **Individual Spring-Mass System**

Moving Mass Analysis – Output Control Dialog

Node / Element Selection

In this section of the output control dialog the nodes or elements that are to be analysed are defined, together with the result entity and component and the type of output that will be generated by IMDPlus.

Processing Individual, Selected or All Nodes or Elements

The **Extent** drop-down list includes the following options to determine whether individual or multiple, nodes or elements will be processed by IMDPlus. This enables results for a chosen set of nodes or elements to be obtained in a single IMDPlus analysis.

- **Individual**

  Process a single node or element in IMDPlus. This option is always available for both **Node** and **Element**. With **Individual** the **Node** or **Element** lists are enabled and filled with the node numbers or valid IMDPlus element numbers selected in the model. Any of the node or element numbers listed can be chosen for the analysis or alternatively a valid node or element number can be manually specified by the user.

- **Selected**

  Process the nodes or valid IMDPlus elements selected in the model. For **Node** this option will be available if nodes have been selected in the model. For **Element** the option will be available if valid IMDPlus elements have been selected in the model and they are all the same type.

- **All**

  Process all of the nodes or valid IMDPlus elements in the model. For **Node** this option is always available. For **Element** the option will only be available if all of the elements in the model are the same type.

Note. When analysing multiple elements, using **Selected** or **All**, only elements of the same type, that is, bar, thick beam, continuum, thick shell or thin shell elements, can be included in a set of IMDPlus elements. The element set can, however, contain elements with different numbers of Gauss points or nodes.

Note. When **Extent** is set to **Selected** or **All** the Node, Element and Gauss Point/Node/End drop-down lists will not be available for selection. In addition the element Gauss Point/Node/End is automatically set to **All** as multiple elements are going to be processed in a single analysis.

Note. Both **Generate graphs in Modeller** and **Generate textfile output** can be used when processing multiple node or element objects. With **Generate textfile output** a number of output file formats are supported by IMDPlus to allow import of results into graphing packages. These are discussed in the Output Control section presented later.

Entity and Component Output

If **Node** output is selected, result entities supported are displacements, velocities, accelerations, and reactions plus, for moving load and moving mass analyses, dynamic amplification factors. The available components placed into the list will depend on the nodes chosen for the analysis. In addition to the standard single component output, options are also available to output all translational or rotational displacements, velocities, accelerations, reactions and dynamic amplification factors (when applicable) to the same graph.

If **Element** output is selected, the result entities supported are governed by the element type chosen. For example, choosing a thick shell element allows selection of the Stress resultants (such as Nx, Mx and Sx), Stress (Top/Middle/Bottom) and Strain (Top/Middle/Bottom). Depending upon the element type either Gauss point, node or end results are available with the ability to select either a single location or All locations for an individual element. For a complete list of the output supported refer to the list of elements supported by IMDPlus.

Results for Individual items, Sum of chosen items and Average of chosen items

The **Individual items**, **Sum of chosen items** and **Average of chosen items** options determine the type of node or element output that will be generated by IMDPlus, see also **Response time history** and **Peak response summary** in the following section.

- **Individual items**

  This option produces individual results for each of the chosen nodes or element Gauss points/nodes/ends.

- **Sum of chosen items**

  This option adds together results from the chosen nodes or element Gauss points/nodes/ends to give summed node or element time histories from which peaks of the summed results are obtained.

- **Average of chosen items**

  This option obtains results by first summing the time history results from the chosen nodes or element Gauss points/nodes/ends. A simple average of this summed result is obtained to give the node or element average time histories from which the peak average results are obtained.

Sum of chosen items and **Average of chosen items** are only available for selection when **Extent** is set to **Selected** or **All**.

Note. Any combination of the options **Individual items**, **Sum of chosen items** and **Average of chosen items** can be used in an IMDPlus analysis, although using the **Individual items** option when **Extent** is set to **Selected** or **All** (nodes or elements) may produce a very large number of graphs, depending on the number of nodes or elements chosen for processing.

Note. When processing reactions for a chosen set of nodes, **Sum of chosen items** and **Average of chosen items** only take into consideration nodes with supported freedoms.

Note. When processing dynamic amplification factors only averaged results can be obtained for a chosen set of nodes as summed results are meaningless for this result entity.

Spring-Mass System Selection (Moving Mass Only)

This section of the output control dialog is only available in a moving mass analysis. It defines the spring-mass system, result and entity that will be processed by IMDPlus.

Note. Only one of the Node, Element or Spring-mass system radio buttons can be selected at any given time.

Processing Individual or All Spring-Mass Systems

The **Extent** drop-down list includes the following options to determine which spring-mass systems will be processed by IMDPlus.

- **Individual**
  Process a single spring-mass system in IMDPlus. With Individual the System list is enabled and will contain, in sequential order, all of the system identifiers, together with the initial positions for each system and the spring-mass system property set used at each position, for the vehicle configuration selected in the IMDPlus moving mass analysis control dialog. Any of the spring-mass systems listed can be chosen for the analysis.

- **All**
  Process all of the spring-mass systems for the vehicle configuration selected in the IMDPlus moving mass analysis control dialog.

Note. When **Extent** is set to **All** the System drop-down list will not be available for selection.

Note. Both **Generate graphs in Modeller** and **Generate textfile output** can be used when processing all of the spring-mass systems. With **Generate textfile output** a number of output file formats are supported by IMDPlus to allow import of results into graphing packages. These are discussed in the Output Control section presented later.

Result and Entity Output

The **Result** drop-down list contains the valid result types available for the system(s), selected using **Extent** and **System**. The result types available for selection depend on number of sprung masses in the system, that is, unsprung mass only, single sprung mass or double sprung mass and whether a path surface profile has been included in the analysis. Results are generated as the spring-mass system moves across the structure and are available for the range of speeds defined in the IMDPlus moving mass analysis control dialog.

- **Unsprung mass**
  The results for the unsprung mass of the spring-mass system(s).

- **Sprung mass 1**
  The results for the 1st sprung mass of the spring-mass system(s).

- **Sprung mass 2**
  The results for the 2nd sprung mass of the spring-mass system(s).

- **Contact point**
  The structural results at the contact point(s) between the spring-mass system(s) and the structure.

- **Path surface profile**
  The path surface profile followed by the system(s). These can be used to confirm that the user defined path surface profile has been interpreted correctly by IMDPlus.

The **Entity** drop-down list contains the valid entities for the **Result** type selected. The entities available for selection are as follows;

- **Displacement**
  The vertical component of displacement for the **Result** type of the spring-mass system(s).

- **Velocity**
  The vertical component of velocity for the **Result** type of the spring-mass system(s).

- **Acceleration**
  The vertical component of acceleration for the **Result** type of the spring-mass system(s).

- **Position**
  The horizontal position of the unsprung mass of spring-mass system(s), relative to the initial position of the origin / front of the vehicle. This entity type is only available when **Result** is set to **Unsprung mass**

- **Force**

The vertical force applied to the structure by the spring-mass system. This entity type is only available when **Result** is set to **Contact point**

Output Control

The output control section selects the types of results to visualise. Options are:

- ❑ **Response time history**

  Response time history outputs node or element time history results for the selected **Entity** and **Component** or spring-mass system time history results for the selected **Result** and **Entity**.

  Results available include displacements, velocities, accelerations, dynamic amplification factors and reactions for nodes, forces/moments, stresses/stress resultants and strains/strain resultants for elements and vertical displacement, velocity, acceleration, position and force for spring-mass systems.

  For nodes or elements the type(s) of output generated depends on the options selected in the node / element selection section of the dialog as follows.

  - ❑ **Individual items**

    Individual time history results for all of the chosen nodes or elements are generated.

  - ❑ **Sum of chosen items**

    Summed time history results for the chosen set of nodes or elements are generated.

  - ❑ **Average of chosen items**

    Averaged time history results for the chosen set of nodes or elements are generated.

  When processing spring-mass systems individual time history results for the chosen system(s) are always generated.

- ❑ **Peak response summary**

  Peak response summary outputs maximum, minimum and/or absolute peak results and times for the selected node / element **Entity** and **Component** or the selected spring-mass system **Result** and **Entity**.

  Results available include displacements, velocities, accelerations, dynamic amplification factors and reactions for nodes, forces/moments, stresses/stress resultants and strains/strain resultants for elements and vertical displacement, velocity, acceleration and force for spring-mass systems

  When processing nodes or elements an **Envelope** of the peak results can also be obtained for all components of the current result **Entity**. These highlight the individual nodes or element Gauss points/nodes/ends in the chosen set of nodes or elements where peak results occur for each of the earthquakes or speeds analysed.

For nodes or elements the type(s) of output generated depends on the options selected in the node / element selection section of the dialog as follows.

- **Individual output**
  Individual peak results for all of the chosen nodes or elements are generated.
- **Sum of chosen items**
  Peaks of the summed results for the chosen set of nodes or elements are generated. If **Envelope** has been selected an envelope of the individual peak results is also displayed in a Notepad application.
- **Average of chosen items**
  Peaks of the averaged results for the chosen set of nodes or elements are generated. If **Envelope** has been selected an envelope of the individual peak results is also displayed in a Notepad application.

When processing spring-mass systems individual peak summary results for the chosen system(s) are always generated.

Note. The options **Positive/Negative** and **Absolute** are not available for selection in a seismic analysis.

Note. Peak summaries are not available for spring-mass systems when **Result** is set to **Path surface profile** or **Entity** is set to **Position**

Note. The option **Envelope** is only available for selection when either **Sum of chosen items** or **Average of chosen items** is selected. **Envelope** is not available when spring-mass systems are being processed.

- **Secondary response spectra (SRS)**
  Secondary response spectra (SRS) outputs secondary response spectra based on the acceleration response for nodes. SRS output is not available for displacements, velocities, dynamic amplification factors or reactions at nodes and is not a valid option for elements or spring-mass systems. SRS output can only be used when both **Node** and **Individual items** are selected.
- **Modal combination/factor history**
  Modal combination/factor history outputs modal factors for the response of the structure at each time step of the analysis. The output takes the form of a VBScript file which can be imported into Modeller to define Modal Combinations of the modes of vibration.
- **Generate graphs in Modeller**

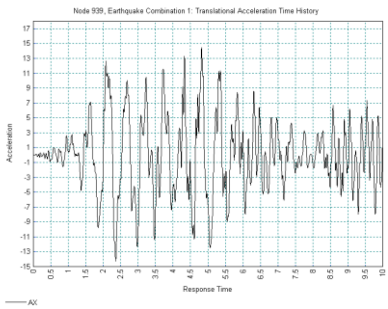
Generate graphs in Modeller presents the results from the IMDPlus analysis as graphs immediately after completion of the analysis. The earthquake or speed to graph is selected from the list of available records.

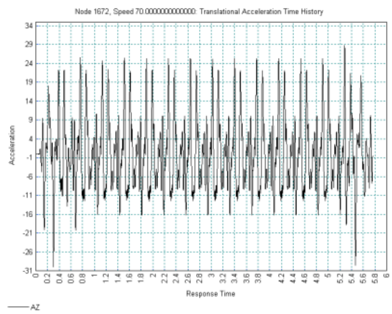
- ❑ **Generate textfile output**

  Generate textfile output saves the IMDPlus results to the **\<LUSAS Installation Folder>\Projects\<Model Name>\Associated Model Data\<Model Name>** directory in text format. These results can be archived or imported into additional graphing packages.

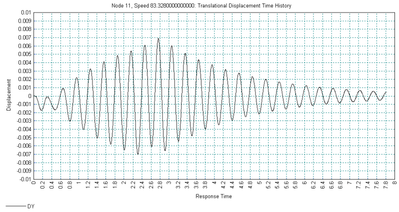
Response Time History

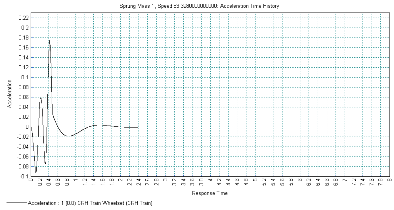
The time history results are computed by IMDPlus for the earthquake or speed specified in the output control dialogs shown in the preceding section. Examples of time history plots for an individual node and spring-mass system are shown in following figures.





Seismic Analysis Time History **Moving Load Analysis Time History**





Moving Mass Analysis Time Histories

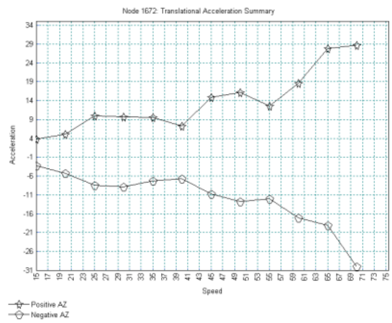
Peak Response Summary

The peak response summaries are computed by IMDPlus for all of the earthquakes or speeds included in the analysis. Examples of peak output for an individual node and spring-mass system are shown in following figures.

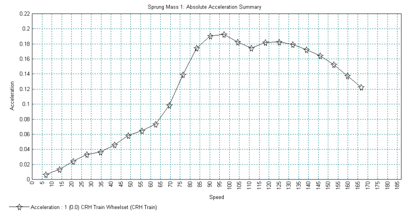
Note. Peak response summary graphs can only be generated in a moving load or moving mass analysis. For a seismic analysis the results are presented in tabular format.

```
#
#
# +---------------+
# | Record = 001 |
# +---------------+
#
# Entity                  Time              Value
Peak_Pos_AX          4.815000000        14.47622283
Peak_Neg_AX          2.355000000       -14.27035402
Abs_Peak_AX          4.815000000        14.47622283
Peak_Pos_AY          1.025000000        6.383296190
Peak_Neg_AY          3.410000000       -5.046086466
Abs_Peak_AY          1.025000000        6.383296190
Peak_Pos_RAZ         1.050000000        2.071596885
Peak_Neg_RAZ         1.030000000       -2.024839500
Abs_Peak_RAZ         1.050000000        2.071596885
Abs_Peak_RSLT        4.815000000        14.62696551
#
#
# +---------------+
# | Record = 002 |
# +---------------+
#
# Entity                  Time              Value
Peak_Pos_AX          9.360000000        8.342986674
Peak_Neg_AX          9.595000000       -7.757532327
Abs_Peak_AX          9.360000000        8.342986674
Peak_Pos_AY          9.435000000        1.720681106
Peak_Neg_AY          9.350000000       -1.439867728
Abs_Peak_AY          9.435000000        1.720681106
Peak_Pos_RAZ         9.610000000        0.4530301859
Peak_Neg_RAZ         9.370000000       -0.5866916056
Abs_Peak_RAZ         9.370000000        0.5866916056
Abs_Peak_RSLT        9.355000000        8.415340215
```

Peak Response Text Output For Seismic Analysis



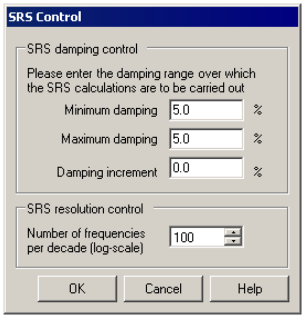
Peak Response Plot For Moving Load Analysis



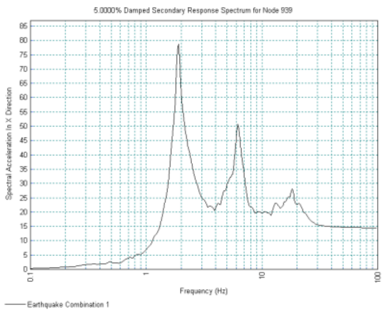
Peak Response Plots For Moving Mass Analysis

Secondary Response Spectra (SRS)

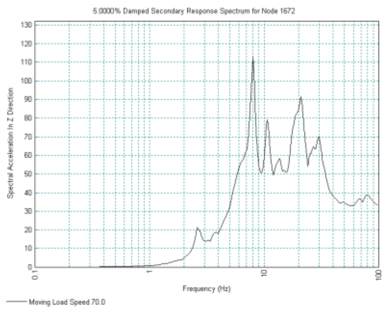
Secondary Response Spectra (SRS) are computed for the selected nodes from the time history acceleration responses. The damping ratio range for the SRS calculations along with the frequency resolution are accessed via the **Modify…** button. In this dialog the minimum and maximum damping ratios can be entered along with the damping ratio increment. Control is also provided over the resolution of the SRS results computed.



Examples of SRS plots are shown in the following figures:



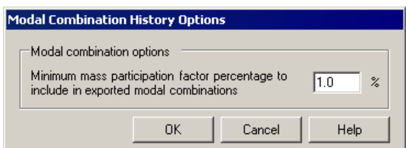
SRS Plot For Seismic Analysis



SRS Plot For Moving Load Analysis

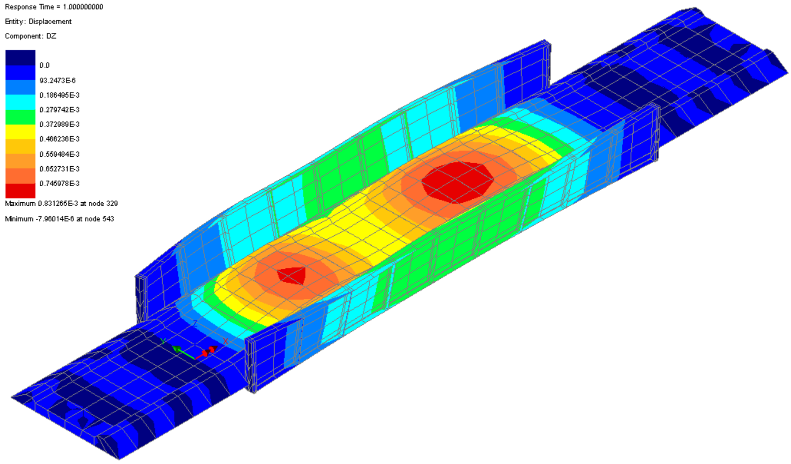
Modal Combination/Factor History

Modal combinaton/factor history outputs modal factors for the response of the structure at each time step of the IMDPlus analysis. The modal combination history options are accessed via the **Options…** button. In this dialog the minimum mass participation factor percentage to include in the exported modal combinations can be defined.

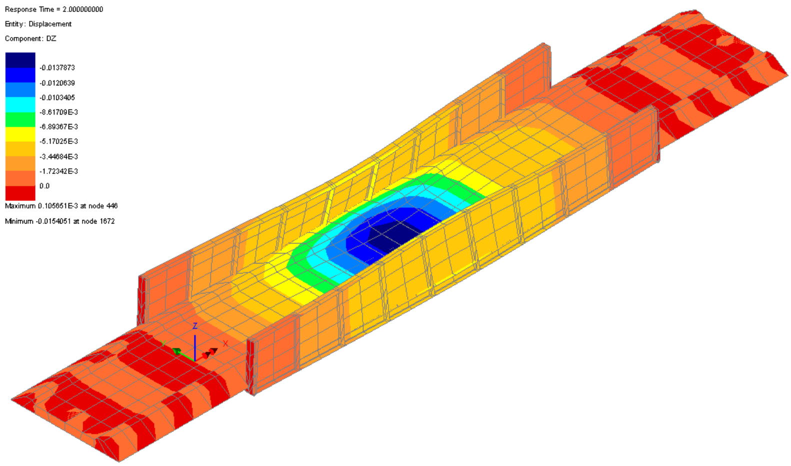


The output takes the form of a VBScript file which can be imported into Modeller to define modal combinations. These combinations can then be used to visualise the deformations and produce contour plots of entities other than velocities and accelerations, such as the examples shown in the following figures.

Note. Importing all modal combinations could take significant time. For inspecting key times during the analysis the required combination information can be extracted from the original VBScript file and placed into a user defined VBScript file.



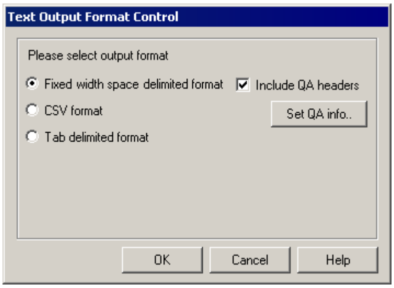
Vertical displacement plot for a time of 1.0s



Vertical displacement plot for a time of 2.0s

Generate Textfile Output

If the option **Generate textfile output** is selected the results from the IMDPlus analysis will remain in the **\<LUSAS Installation Folder>\Projects\<Model Name>\Associated Model Data\<Model Name>** directory after completion of the analysis. This allows the IMDPlus results to be archived and exported to additional graphing packages. If graphs are being generated in Modeller, the output format for these text files is the default IMDPlus format, which has all of the QA information placed in the header of the file and the results written in space delimited format. If however the option to **Generate graphs in Modeller** is switched off additional output formats become available via the **Options…** button which opens the following dialog.



The formats supported are:

- **Fixed width space delimited format** (IMDPlus default). Additional options are available to exclude the QA headers and set the QA information.
- **Comma Separated Variable (CSV) format.**
- **TAB delimited format**.

For all text output formats, the extensions of the files indicate the results entity. The following list describes the output file extensions used by IMDPlus:

***.dsp** - Displacements/Rotations

***.vel** - Velocities/Rotational Velocities

***.acc** - Accelerations/Rotational Accelerations

***.daf** - Dynamic Amplification Factors

***.rct** - Force/Moment Reactions

***.srs** - Secondary Response Spectra

***.fce** - Forces/Forces and Moments

***.str** - Stresses/Stress Resultants

***.stn** - Strains/Strain Resultants

***.pos** - Positions of Spring-Mass Systems

***.cfc** - Vertical Forces for Spring-Mass Systems

***.sum** - Peak response summary

Elements Supported By IMDPlus

Results for the following elements can be output and graphed directly from IMDPlus. Elements that are excluded from this list and are valid for 2D/3D eigenvalue analyses can still be used in an IMDPlus analysis but the results for these unsupported elements must be obtained through the modal combinations facility available in IMDPlus. All element output follows the default LUSAS Solver output directions (local or global) for the element types below. For further information please refer to the LUSAS Element Reference Manual.

- **Bar Elements** (Gauss Point Results)
  BAR2 - 2D 2-noded Bar
  BRS2 - 3D 2-noded Bar

- **Beam Elements** (End Results)
  BEAM - 2D Engineering Thick Beam
  BMS3 - 3D Engineering Thick Beam
  BTS3 - 3D Thick Beam
  BMI21 - 3D Thick Beam
  BMI22 - 3D Thick Beam
  BMI31 - 3D Thick Beam
  BMI33 - 3D Thick Beam

- **2D Continuum Elements** (Gauss Point Results)
  TPM3, TPM6, QPM4, QPM8 - 2D Plane Stress
  QPM4M - 2D Plane Stress (Enhanced Strain)
  TPK6, QPK8 - 2D Plane Stress Crack Tip
  TPN3, TPN6, QPN4, QPN8 - 2D Plane Strain
  QPN4M - 2D Plane Strain (Enhanced Strain)
  TNK6, QNK8 - 2D Plane Strain Crack Tip

- **3D Continuum Elements** (Gauss Point Results)
  TH4, TH10, PN6, PN12, PN15,
  HX8, HX16, HX20 - 3D Solid
  HX8M - 3D Solid (Enhanced Strain)

- **Thin Shell Elements** (Nodal Results)
  TS3, QSI4 - 3D Flat Thin Shell
  SHI4 - 3D Flat Thin Box Shell

- **Thick Shell Elements** (Gauss Point Results)

TTS3, TTS6, QTS4, QTS8 - 3D Thick Shell

- **Joint Elements** (Gauss Point Results)
  JNT3 - 2D Joint Element for Bars, Plane Stress and Plane Strain
  JPH3 - 2D Joint Element for Engineering Beams
  JNT4 - 3D Joint Element for Bars and Solids
  JSH4 - 3D Joint Element for Engineering Beams and Shells

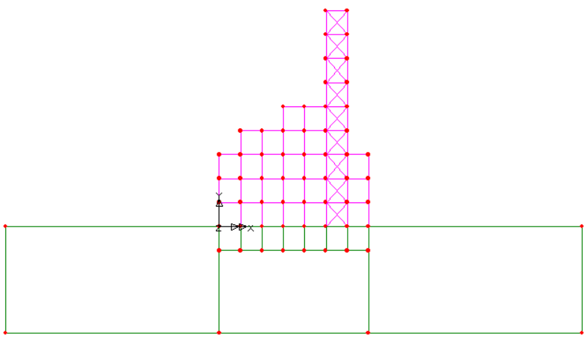
Seismic Analysis of a 2D Frame (Time Domain)

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 2D plane frame founded on an elastic medium. The model is an extension of the Seismic Response of a Plane Frame example.

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



Objectives

The output requirements of the analysis are:

- ❑ Displacements, velocities and accelerations at the top of the tower
- ❑ Secondary response spectra at the top of the tower
- ❑ Forces and moments in a column at the base of the tower
- ❑ Average forces/moments and peak forces/moments from all of the columns at the base of the tower

Keywords

Seismic, time domain, response, mass participation, interactive modal dynamics, excitation, eigenvalue, Secondary Response Spectra.

Associated Files

- **IMDPlus 2D Tower.mdl** Model file of the structure.
- **horizontal.prn** and **vertical.prn** define horizontal and vertical earthquake accelerations. These two files contain two earthquake records each; one for the 1940 El Centro earthquake and one for the 1994 Northridge earthquake.

Discussion

The Seismic Response of a Plane Frame is revisited in this example. The frame is founded in a plane strain elastic medium and the response of the structure is evaluated in the time domain using IMDPlus.

The model is comprised of thick beam elements for the concrete columns, beam members, and steel diagonal bracing members, which have pinned end connections. The number of beam elements representing the components of the structure has been increased from 1 per line to 4 per line to provide greater definition of the deformed shapes. In addition to this modification, all support restraints have been removed from the frame and an additional 4.5 m length has been added to the base of each column to allow embedment of the column bases into the elastic medium.

The elastic medium is modelled as a 108m by 20m block which is fully restrained along its base and with cyclic translation constraints assigned to the sides. These constraints provide support to the sides of the elastic medium without the need to apply physical restraints, thus allowing direct and shear behaviours in the elastic medium.

The seismic response analysis is performed in two distinct stages. A natural frequency analysis is performed first. This is used to calculate the first 50 natural modes of vibration of the combined structure and elastic medium. The eigenvalues, frequencies and eigenvectors (mode shapes) are stored and used in the subsequent IMDPlus analysis. Although the natural frequencies are obtained from an eigenvalue analysis any information regarding the magnitudes of deformations or stresses / 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 a support condition excitation governed by time histories of acceleration in the model global axes. In this example this is assumed to act along the base of the elastic medium in the form of horizontal and vertical accelerations. It should, however, be noted that no deconvolution of the records has been carried out to convert the surface responses recorded for these earthquakes to at-depth acceleration time histories to be input into the analysis. As a consequence, the ground level accelerations in the analysis will not correspond to the measured values. As stated previously, two earthquake records are used in the analysis, the first being the 1940 El Centro earthquake and the second being the 1994 Northridge earthquake. The two earthquake responses are computed during a single analysis. Only the first 10 seconds of each seismic event is analysed in this example.

Modelling

Running LUSAS Modeller

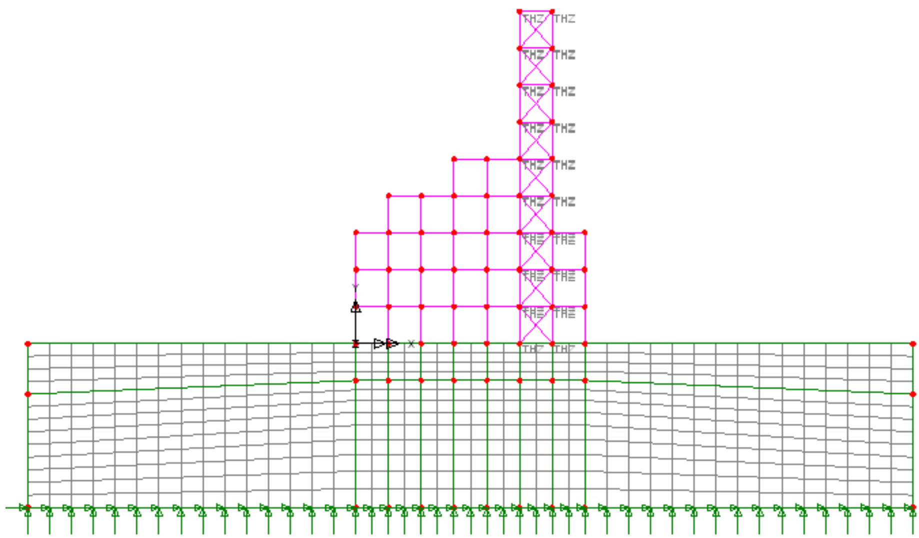
For details of how to run LUSAS Modeller see the heading ***Running LUSAS Modeller*** in the Examples Manual Introduction.

Building and loading the model

File
Open…

- To create the model, open the file **IMDPlus 2D Tower.mdl** located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory. Click the **OK** button if the Open File From Previous Version dialog appears.

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



File
Save As…

- In the **\<LUSAS Installation Folder>\Projects** folder create a new directory called **IMDPlus 2D Tower**
- Save the model into this new folder as **IMDPlus 2D Tower**. 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 based on the results from the natural frequency analysis.

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 **50**
- 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.

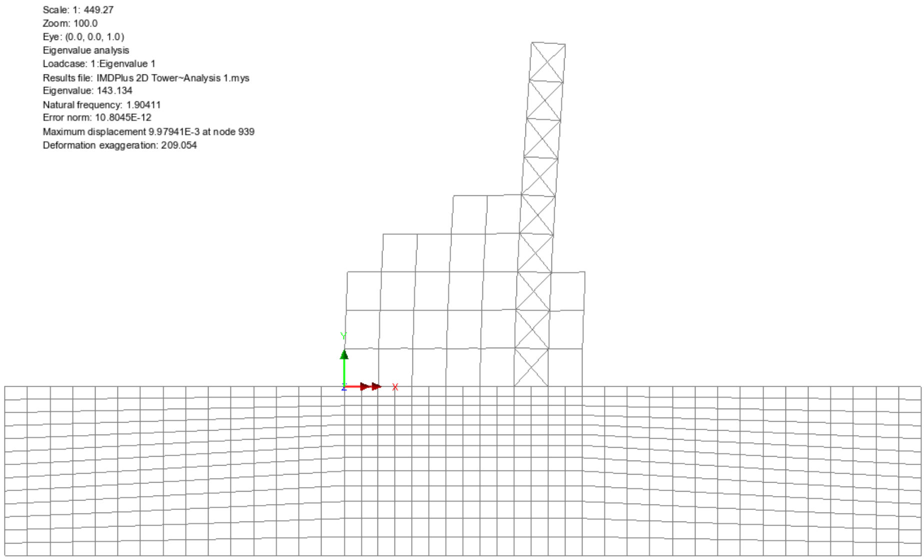
A LUSAS Datafile will be created from the model information. The LUSAS Solver uses this datafile to perform the analysis.

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. 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 a blank part of the graphics window and select the **Deformed mesh** option to add the deformed mesh layer to the Treeview. Select the **Visualise** tab and turn off the **Beam end releases** option. Click the **OK** button to accept the remaining default values and display the deformed mesh for Eigenvalue 1.
- At the bottom of the Treeview select the **Window summary** option and click the **Details...** button. In the Window summary properties dialog set the position to **(50,-10)** and click the **OK** button to return to the graphics window.



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

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.

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

- In the Treeview right-click on **Eigenvalue 2** and select the **Set Active** option. The deformed mesh plot for Eigenvalue 2 will be displayed.

Printing Eigenvalue Results

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

Utilities
Print Results Wizard…

- Select **Active** and press **Next**
- Select entity **None** of type **Eigenvalues** and click **Finish**

The Eigenvalue results will be printed to the Text Output window. For inspection only the first 10 modes are shown here. Error norms may vary.

MODE	EIGENVALUE	FREQUENCY	ERROR NORM
1	143.134	1.90411	0.108045E-10
2	1479.54	6.12185	0.402905E-06
3	7142.50	13.4507	0.142970E-09
4	11469.1	17.0445	0.834811E-09
5	13603.1	18.5626	0.131186E-08
6	15305.2	19.6897	0.147175E-06
7	22093.1	23.6564	0.464346E-06
8	23569.4	24.4340	0.139975E-06
9	23646.6	24.4740	0.179723E-06
10	23787.5	24.5468	0.627557E-06

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 text 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 IMDPlus 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 has been achieved in the global directions. If less than 90% has been achieved no further modes need to be included, if and only if, the modes of vibration omitted cannot be excited by the input acceleration time histories or a significant proportion of the structure is restrained by support in these directions and therefore cannot participate in the modes of vibration. The acceptability of the included modes of vibration will vary from analysis to analysis but failure to check that a significant proportion of the total mass has been accounted for may lead to important modes being missed and subsequent errors in the analysis results.

Utilities
Print Results Wizard…

- After selecting **Active**, ensure **Sum Mass Participation Factors** is displayed in the Type combo box and click **Finish.** The Sum Mass Participation Factors results will be printed to the Text Output window.

For inspection only modes 40 to 50 will be printed here. It can be seen that the 90% value has not been achieved and is discussed below.

MODE	SUM MASS X	SUM MASS Y	SUM MASS Z
40	0.812738	0.589605	0.00000
41	0.812738	0.668202	0.00000
42	0.812738	0.754026	0.00000
43	0.812739	0.754159	0.00000
44	0.812741	0.754160	0.00000
45	0.812744	0.757374	0.00000
46	0.812745	0.788122	0.00000
47	0.812747	0.788903	0.00000
48	0.812760	0.789672	0.00000
49	0.812760	0.789674	0.00000
50	0.812760	0.789731	0.00000

Note. In this analysis, a significant proportion of mass is restrained along the base of the elastic medium. For the 50 modes of vibration, 81.27% and 78.97% of the total mass is achieved in the horizontal and vertical directions. In order to achieve 90% total mass in both the horizontal and vertical directions 160 modes of vibration with an upper frequency equivalent to 97.2 Hz would actually be required. However, for the purposes of this example, the number of modes of vibration has been restricted to 50.

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

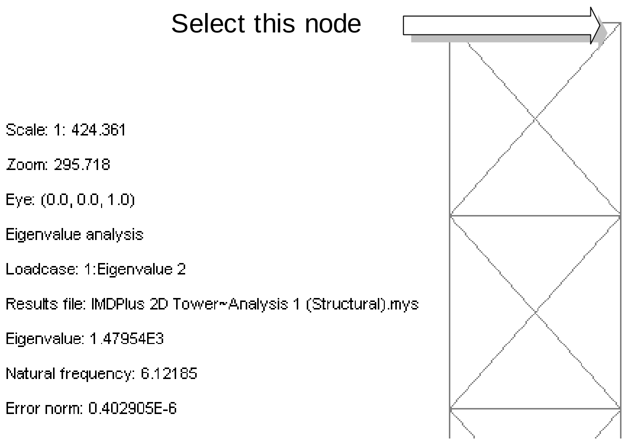
- Use the maximise button to increase the size of the graphics window.

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. Initially, in this example, the response of the node at the top-right corner of the tower will be investigated.

Note. 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.

- Turn off the display of the **Deformed Mesh** layer in the Treeview.
- In the Treeview double-click on the **Mesh** layer name. Select the **Visualise** tab and turn off the **Beam end releases** option. Click the **OK** button to accept all other settings. This will turn on the layer in the graphics window.
- Zoom-in to the top of the tower and select only the top-right node.



The main tower members have been meshed with 4 line mesh divisions so each line will have 3 nodes along its length in addition to one at either end.

Defining the Seismic Parameters

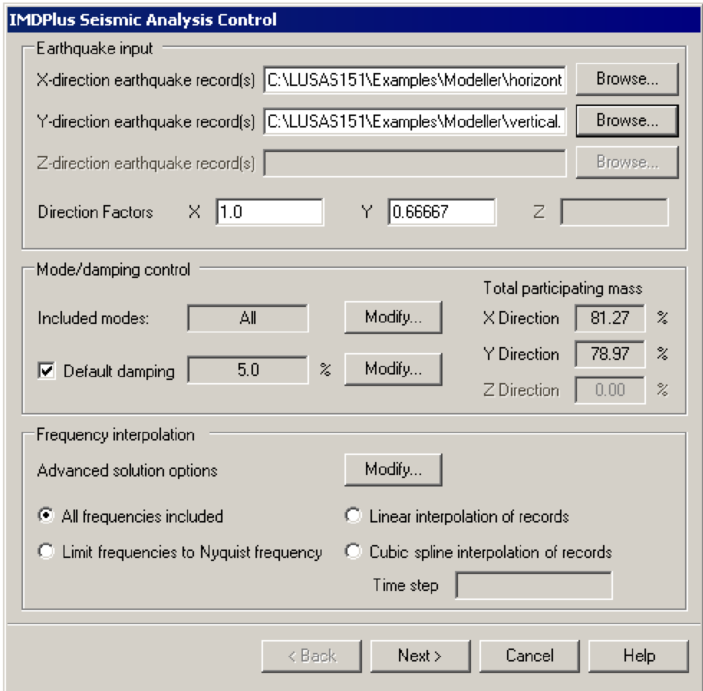
Note. To simplify the use of IMDPlus (and provide an alternative to picking IMDPlus menu selections from the Utilities menu) for the rest of the example it is recommended that the IMDPlus toolbar is enabled in LUSAS Modeller. To do this select the **View > Toolbars...** menu command and enable the **IMDPlus** 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.

Utilities
IMDPlus >
Seismic...

- Open the IMDPlus Seismic Analysis Control dialog through the menu or click on the button in the toolbar.

The input for the seismic analysis is defined on the IMDPlus Seismic Analysis Control dialog. The input consists of the earthquake acceleration records, included modes and damping along with interpolation options.



- In the Earthquake Input section, for the **X-Direction Earthquake Record(s)** browse to the \<**Lusas Installation Folder>\Examples\Modeller** directory and select **horizontal.prn**
- For the **Y-Direction Earthquake Record(s)** browse to the \<**Lusas Installation Folder>\Examples\Modeller** directory and select **vertical.prn**
- Ensure the **Direction Factors** in the **X** and **Y** directions are set to **1** and **0.66667**
- 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.
- Ensure the **All frequencies included** option is selected.
- Click the **Next** button to proceed to the IMDPlus Output Control dialog. When prompted that less than 90% total mass participation has been detected choose **Yes** since we are aware of this (see Checking the Mass Participation Factor).

The IMDPlus Output Control dialog will then be displayed.

Notes relating to the IMDPlus Seismic Analysis Control dialog

On the Seismic Analysis Control dialog the **Browse** buttons can be used instead of entering the directional earthquake file names. As each file is loaded, the contents are checked to ensure that they are in the correct format.

Under the **Modify** dialog for the modes, individual modes can be included or excluded from the seismic analysis and total mass participations for the excitation directions are reported.

Under the **Modify** dialog for the damping, control over the inclusion of over-damped modes can be adjusted if viscous modal damping has been included in the original eigenvalue analysis. To include viscous modal damping, Rayleigh damping parameters must be defined in the material properties assigned to the model or a separate damping attribute defined and assigned to the model. This allows direct mirroring of step-by-step dynamic solutions and the inclusion of frequency dependent damping.

Under the **Modify** dialog for the advanced solution options, the time integration scheme used in the IMDPlus analysis can be selected. Either a Hilber Hughes Taylor (HHT) scheme or Duhamel's integral can be utilised.

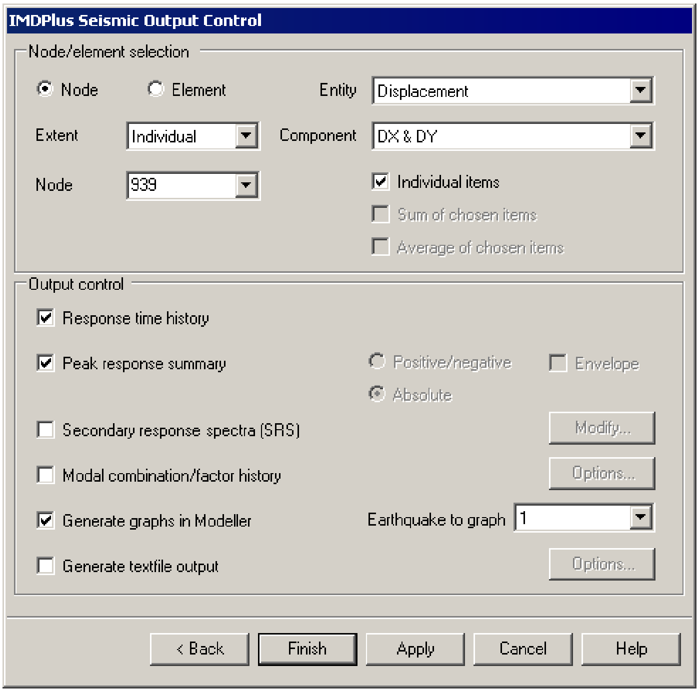
Four methods of interpolation are available in IMDPlus. The default All frequencies included will solve the seismic analysis using all of the frequencies included in the eigenvalue analysis. Limit frequencies to Nyquist frequency will limit the included frequencies to the Nyquist frequency which is governed by the time step of the earthquake records and defined as $1/2\delta t$. Linear interpolation of records allows the specification of an alternative time step for the analysis where the earthquake records are linearly interpolated to this time step. Cubic spline interpolation of records allows the specification of an alternative time step for the analysis where the earthquake records are interpolated with a cubic spline to this time step. This last form of interpolation should generally not be used for measured earthquake records as it can smooth the peak responses but is often more appropriate for synthesised earthquake records.

Displacement Plots

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

- Select **Node** and select **Extent** as **Individual**
- Choose **Node** number **939** from the drop-down list (or your node number if different). This is the node at the top-right of the tower.
- Select results entity **Displacement** and component **DX & DY**
- Ensure **Individual items** is selected. **Sum of chosen items** and A**verage of chosen items** will not be available as an individual node is being processed.
- Ensure **Response time history** is selected.
- Select **Peak response summary**
- Ensure **Generate graphs in Modeller** is selected and specify the **Earthquake to graph** as **1**

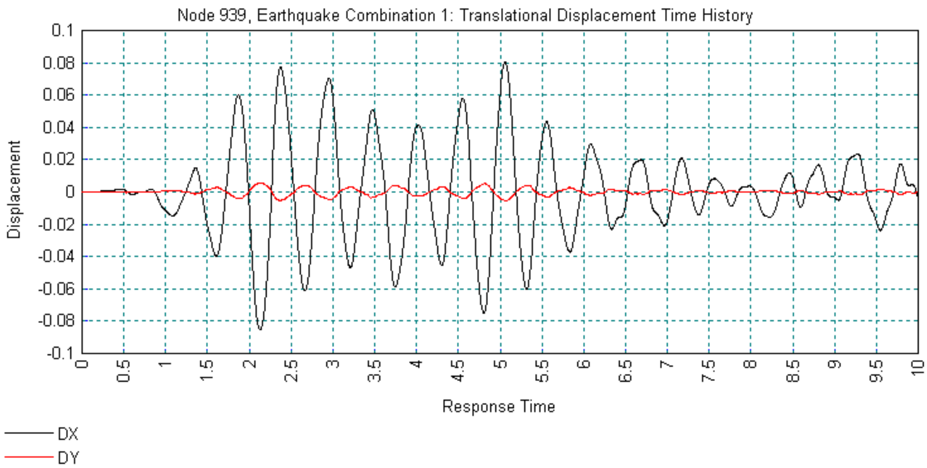
The dialog should look like this:



- Click the **Apply** button to proceed.

An IMDPlus analysis will run and after a short while the horizontal (DX) and vertical (DY) displacements for the point at the top of the tower are displayed on the graph for the first 10 seconds of the 1940 El Centro earthquake. (Earthquake 1)

Peak displacements are also output to Notepad and indicate absolute peak displacements of 0.085 m in the X-direction, 0.006 m in the Y-direction, a peak rotation of 0.003 radians and an absolute peak resultant displacement of 0.086 m (Record 1).

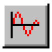



```
#
#
# +---------------+
# | Record = 001 |
# +---------------+
#
# Entity                    Time              Value
Peak_Pos_DX              5.060000000       0.8045374752E-001
Peak_Neg_DX              2.130000000      -0.8537246821E-001
Abs_Peak_DX              2.130000000       0.8537246821E-001
Peak_Pos_DY              2.125000000       0.5614023646E-002
Peak_Neg_DY              5.075000000      -0.5391873880E-002
Abs_Peak_DY              2.125000000       0.5614023646E-002
Peak_Pos_THZ             2.130000000       0.2572687303E-002
Peak_Neg_THZ             5.065000000      -0.2500171174E-002
Abs_Peak_THZ             2.130000000       0.2572687303E-002
Abs_Peak_RSLT            2.130000000       0.8555481793E-001
```

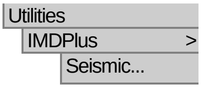
Note. Displacements are relative to the supports.

- Close the Notepad application and graph window.

If the IMDPlus toolbar has been enabled,

- Click on the  button in the toolbar to open the IMDPlus Output Control dialog

If the IMDPlus toolbar is not enabled,



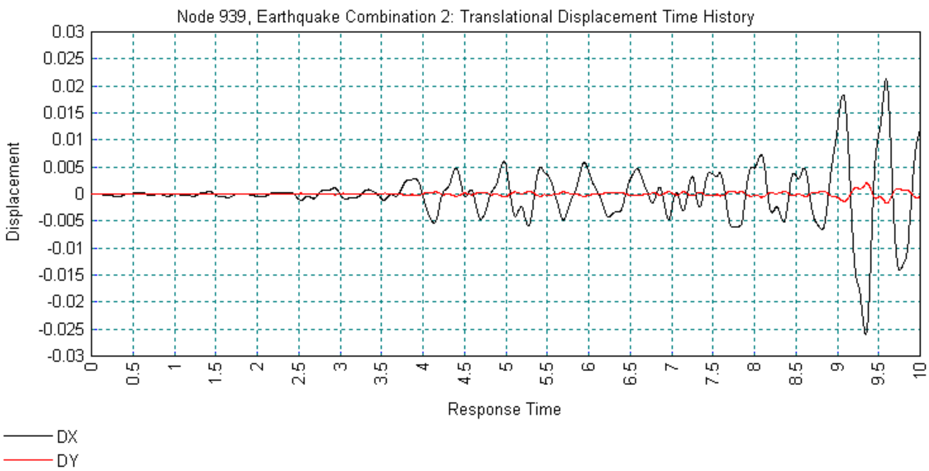
- Open the IMDPlus Seismic Analysis Control dialog through the menu and click **Next** > to keep the existing analysis control settings and open the IMDPlus Output Control dialog.

  In the IMDPlus analysis we have analysed two earthquake combinations at the same time. We are therefore also interested in the displacement response of the top of the tower for the second earthquake.

  Enter the following information into the IMDPlus Output Control dialog:

- Select earthquake **2** in the **Earthquake to graph** list box.
- Click the **Apply** button to proceed.

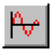
The horizontal (DX) and vertical (DY) displacements for the point at the top of the tower are displayed on the graph for the first 10 seconds of the 1994 Northridge earthquake. Peak displacements are also output to Notepad and indicate absolute peak displacements of 0.026 m in the X-direction, 0.002 m in the Y-direction, a peak rotation of 0.001 radians and an absolute peak resultant displacement of 0.026 m (Record 2).



```
#
#
# +---------------+
# | Record = 002 |
# +---------------+
#
# Entity                    Time              Value
Peak_Pos_DX             9.590000000       0.2129361383E-001
Peak_Neg_DX             9.345000000      -0.2602101143E-001
Abs_Peak_DX             9.345000000       0.2602101143E-001
Peak_Pos_DY             9.355000000       0.2024933080E-002
Peak_Neg_DY             9.600000000      -0.1693798327E-002
Abs_Peak_DY             9.355000000       0.2024933080E-002
Peak_Pos_THZ            9.355000000       0.9072047996E-003
Peak_Neg_THZ            9.600000000      -0.7763811724E-003
Abs_Peak_THZ            9.355000000       0.9072047996E-003
Abs_Peak_RSLT           9.345000000       0.2609322984E-001
```

- Close the Notepad application and graph window.

If the IMDPlus toolbar has been enabled,

- Click on the  button in the toolbar to open the IMDPlus Output Control dialog

If the IMDPlus toolbar is not enabled,

Utilities
IMDPlus >
Seismic...

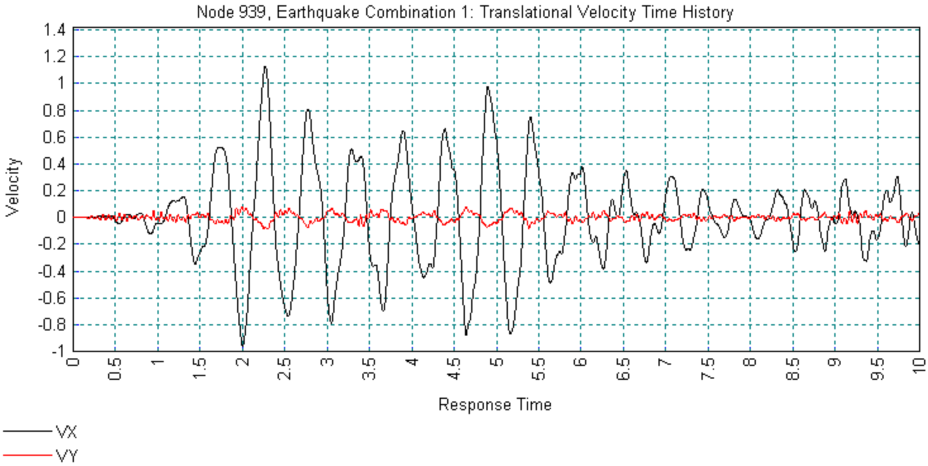
- Open the IMDPlus Seismic Analysis Control dialog through the menu and click **Next** > to keep the existing analysis control settings and open the IMDPlus Output Control dialog.

Velocity Plots

In addition to the displacement response of the top of the tower we are also interested in the velocities experienced by the tower. Enter the following information into the output control dialog.

- Select **Velocity** results of **VX & VY**
- Select earthquake **1** in the **Earthquake to graph** list box
- Click **Apply** to proceed

The horizontal (VX) and vertical (VY) velocities of the top of the tower are displayed on the graph for the first 10 seconds of the 1940 El Centro earthquake. Peak velocities are also output to Notepad and indicate absolute peak velocities of 1.13 m/s in the X-direction, 0.085 m/s in the Y-direction, a peak rotational velocity of 0.039 rad/s and an absolute peak resultant velocity of 1.13 m/s (Record 1).

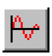



```
#
#
# +--------------+
# | Record = 001 |
# +--------------+
#
# Entity                  Time                Value
Peak_Pos_VX           2.265000000          1.130035452
Peak_Neg_VX           2.005000000         -0.9632754105
Abs_Peak_VX           2.265000000          1.130035452
Peak_Pos_VY           1.995000000          0.8042521467E-001
Peak_Neg_VY           2.295000000         -0.8502032703E-001
Abs_Peak_VY           2.295000000          0.8502032703E-001
Peak_Pos_RVZ          2.020000000          0.3271606639E-001
Peak_Neg_RVZ          2.270000000         -0.3916539671E-001
Abs_Peak_RVZ          2.270000000          0.3916539671E-001
Abs_Peak_RSLT         2.265000000          1.132369132
```

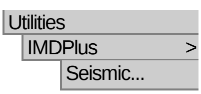
Note. Velocities are relative to the supports.

- Close the Notepad application and graph window.

If the IMDPlus toolbar has been enabled,

- Click on the  button in the toolbar to open the IMDPlus Output Control dialog

If the IMDPlus toolbar is not enabled,



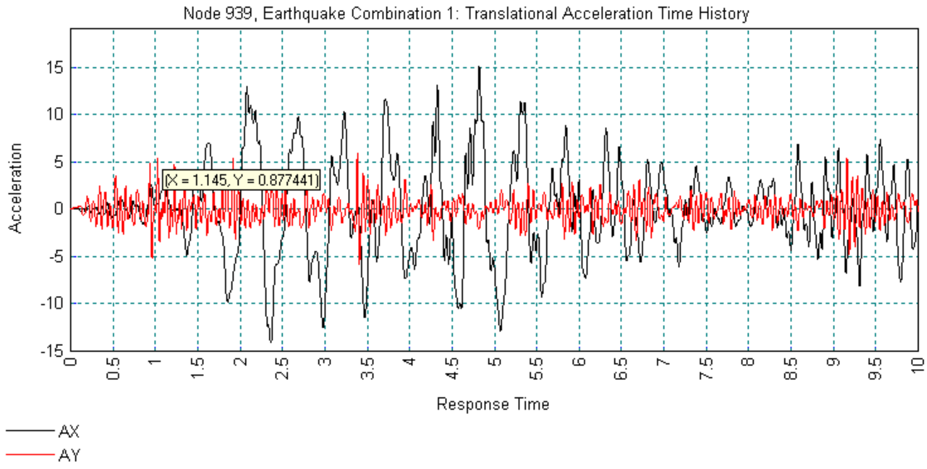
- Open the IMDPlus Seismic Analysis Control dialog through the menu and click **Next** > to keep the existing analysis control settings and open the IMDPlus Output Control dialog.

Acceleration and Secondary Response Spectra Plots

The acceleration response of the top of the tower is now required. Enter the following information into the output control dialog.

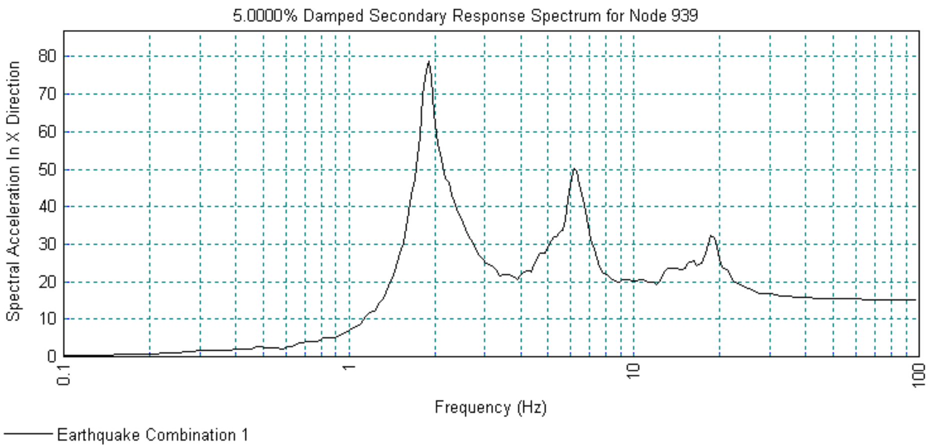
- Select **Acceleration** results of **AX & AY**
- Select **Secondary Response Spectra (SRS)**
- Click on the **Modify…** button to bring up the SRS Control dialog.
- On the SRS Control dialog set the **Minimum damping** and **Maximum damping** to **5.0** and the **Damping increment** to **0.0**
- Click the **OK** button and **Yes** to confirm that only a single damping ratio is to be used
- Click the **Apply** button to proceed.
- When the IMDPlus analysis is finished select **No** when asked whether to show average, maximum and minimum SRS envelopes.

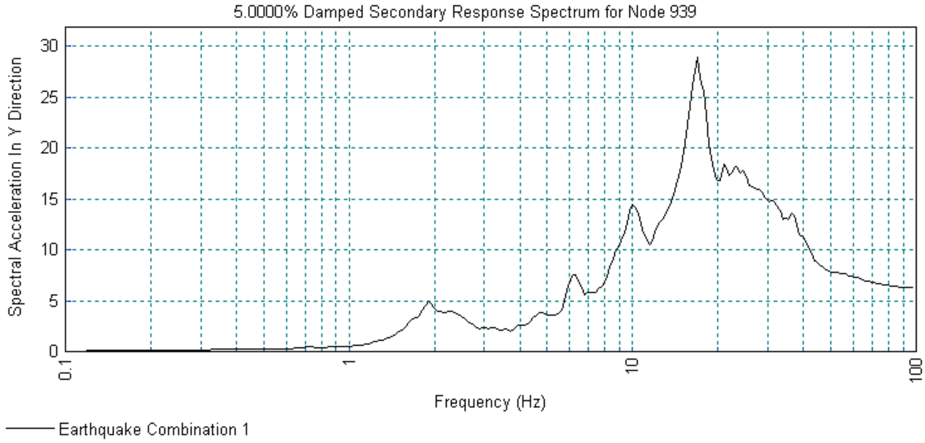
The horizontal (AX) and vertical (AY) accelerations of the top of the tower are displayed on the graph for the first 10 seconds of the 1940 El Centro earthquake. Peak accelerations are also output to Notepad and indicate absolute peak accelerations of 15.089 m/s^2 in the X-direction, 5.937 m/s^2 in the Y-direction, a peak rotational acceleration of 1.56 rad/s^2 and an absolute peak acceleration of 15.24 m/s^2 (Record 1).



```
#
#
# +---------------+
# | Record = 001 |
# +---------------+
#
# Entity                   Time            Value
Peak_Pos_AX           4.820000000      15.08907716
Peak_Neg_AX           2.365000000     -14.07786654
Abs_Peak_AX           4.820000000      15.08907716
Peak_Pos_AY           3.385000000      5.936610684
Peak_Neg_AY           3.410000000     -5.754584426
Abs_Peak_AY           3.385000000      5.936610684
Peak_Pos_RAZ          3.480000000      1.559971790
Peak_Neg_RAZ          1.235000000     -1.457953105
Abs_Peak_RAZ          3.480000000      1.559971790
Abs_Peak_RSLT         4.820000000      15.24035142
```

Secondary Response Spectra are also generated for the horizontal and vertical directions in two additional graphs. These graphs represent the peak acceleration responses of a range of single degree of freedom oscillators with known natural frequencies and damping should they be attached to the node at the top of the tower.





- Close the Notepad application and graph windows.

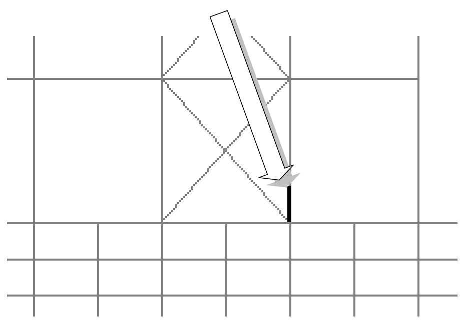
Displaying Individual Element Results

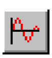
In the previous section the displacement, velocity and acceleration response of the top of the tower were investigated. Now, forces and moments in one of the columns of the structure at ground level will be evaluated.

 In the LUSAS graphics window re-size the display to show the whole tower.

 Zoom-in to the base of the tower and select the element immediately above ground level in the second column from the right.

The main tower members have been meshed with 4 line mesh divisions so each line will have 4 elements along its length.



- Click on the  button in the toolbar to open the IMDPlus Output Control dialog

If the IMDPlus toolbar is not enabled,

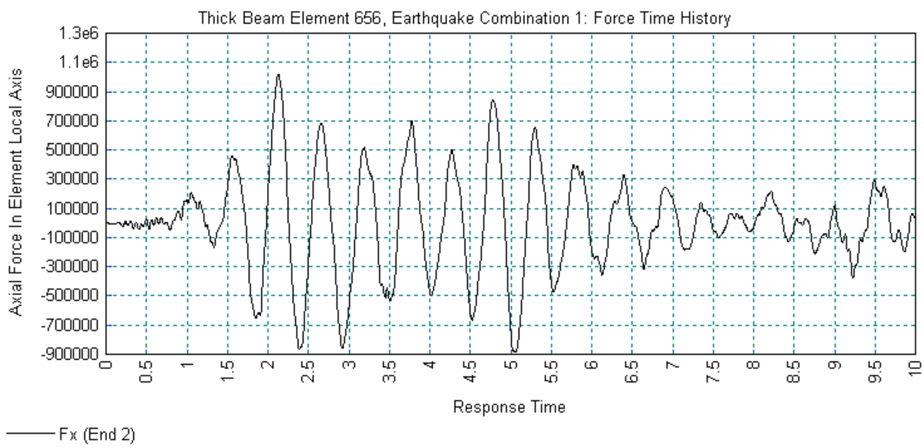
Utilities
IMDPlus >
Seismic...

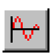
- Open the IMDPlus Seismic Analysis Control dialog through the menu and click **Next >** to keep the existing analysis control settings and open the IMDPlus Output Control dialog.

Enter the following information into the IMDPlus Output Control dialog:

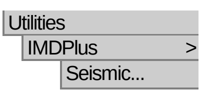
- Select **Element** and ensure **Extent** remains selected as **Individual**
- Choose **Element** number **656** from the drop-down list (or your element number if different).
- Select element **End** number **2** which corresponds to the ground level. (Note that line directions can be viewed to ascertain which is end 1 or end 2 of a line).
- Select **Force/Moment - Thick 2D Beam** results of **Fx**
- Ensure **Individual items** remains selected. **Sum of chosen items** and **Average of chosen items** will not be available as an individual element is being processed.
- Deselect **Peak response summary**
- Click the **Apply** button to proceed.

The time history of the axial force (in N) in the column is displayed on the graph for the first 10 seconds of the 1940 El Centro earthquake. Similar graphs can also be generated for the shear force and bending moment by selecting Fy (in N) and Mz (in Nm) from the output control dialog.



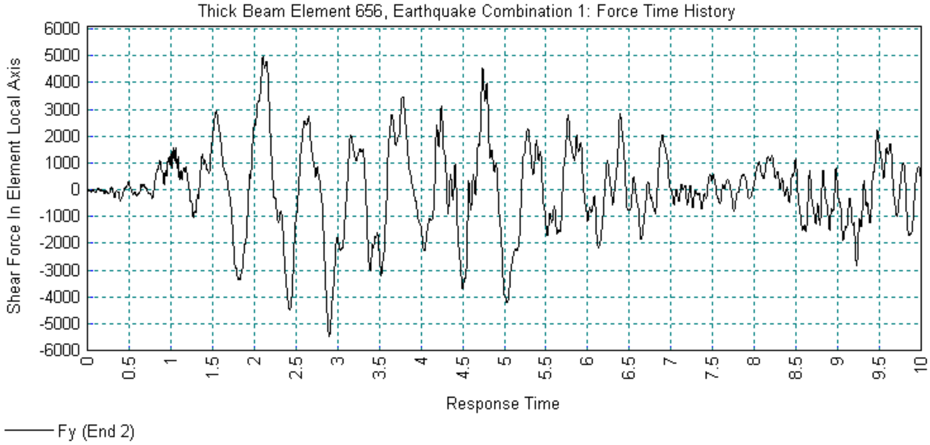
- Close the graph window.
- Click on the  button in the toolbar to open the IMDPlus Output Control dialog

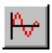
If the IMDPlus toolbar is not enabled,



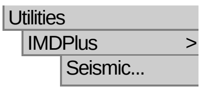
- Open the IMDPlus Seismic Analysis Control dialog through the menu and click **Next** > to keep the existing analysis control settings and open the IMDPlus Output Control dialog.

- Select **Force/Moment - Thick 2D Beam** results of **Fy**
- Click the **Apply** button to proceed.



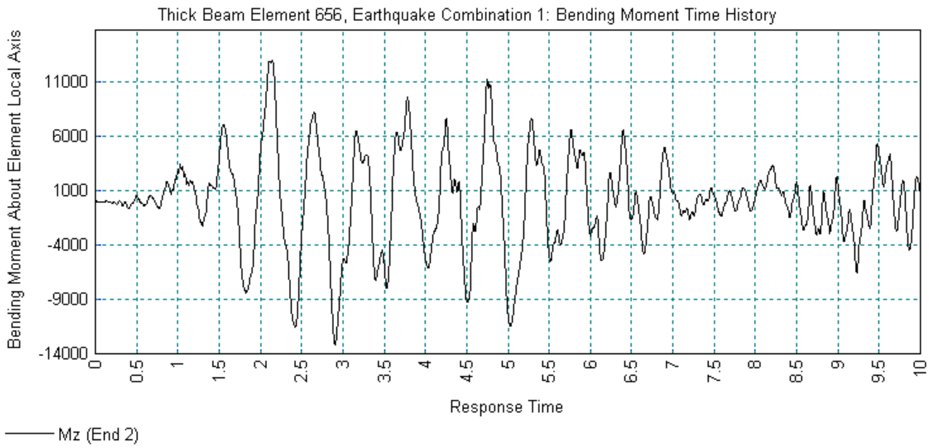
- Close the graph window.
- Click on the  button in the toolbar to open the IMDPlus Output Control dialog

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.

- Select **Force/Moment - Thick 2D Beam** results of **Mz**
- Click the **Finish** button to finish output processing of the individual element results.
- Click **Yes** when asked whether to free up disk space by deleting the temporary files created by IMDPlus.



- Close the graph window.

Displaying Results for a Selected Set of Elements

The example analyses presented in the previous sections have all utilised individual nodes or elements of the structure. In this section a set of beam elements will be selected and analysed in order to evaluate the average forces/moments and peak forces/moments from the bases of all the columns of the tower.

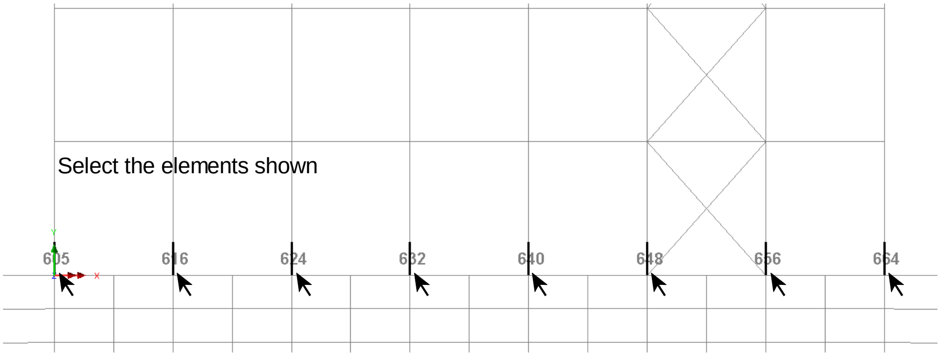
In the LUSAS graphics window re-size the display to show the whole tower.

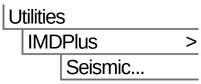
- With no features selected click the right-hand mouse button in a blank part of the graphics window and select the **Labels** option to add the labels to the Treeview.
- On the Properties dialog select the **Element** > **Name** check box and the **Label selected items only** check box.
- Click the **Font...** button and select Font: **Arial,** Font style: **Bold,** Size: **20** and click **OK.**
- Click **OK** to return to the graphics window.

Zoom-in to the base of the frame and select the beam elements immediately above ground level for all of the columns in the structure, as shown in the following figure (your element numbers may be different from those shown in the diagram).

Note. Use the **Ctrl** or **Alt** keys to select multiple elements in the model.

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.





- Open the IMDPlus Seismic Analysis Control dialog through the menu or click on the button in the toolbar
- On the IMDPlus Seismic Analysis Control dialog click the **Next** button to accept the previously defined values. When prompted about the total mass participation click the **Yes** button to continue.

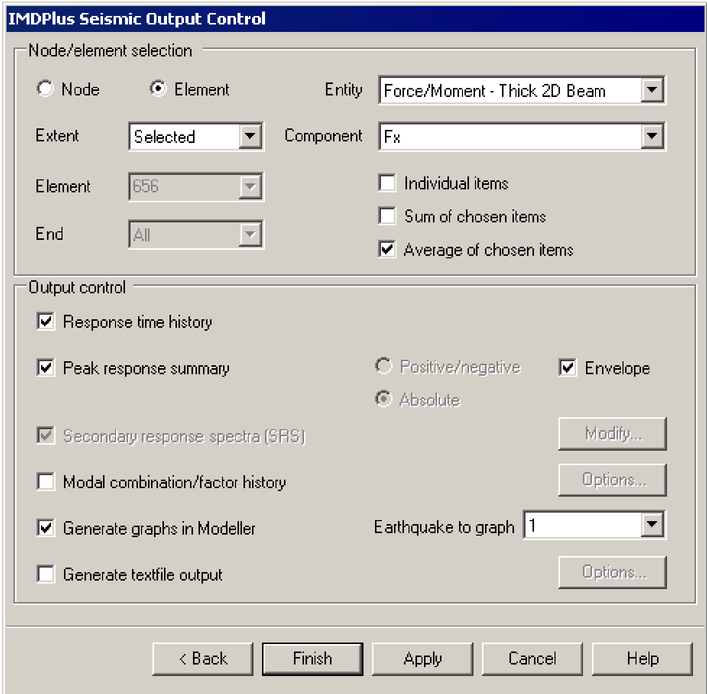
Enter the following information into the IMDPlus Output Control dialog.

- Select **Element** and change **Extent** to **Selected** using the drop-down list. This chooses the eight beam elements at the base of the frame for processing. These beams were selected in the preceding step.

Note. As **Extent** has been set to **Selected** 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 2D Beam** of component **Fx**
- Deselect **Individual items** and select **Average of chosen items** as the averaged results for the eight beam elements are going to be investigated.
- Ensure **Response time history** is selected
- Select **Peak response summary** and select **Envelope**
- Ensure **Generate graphs in Modeller** is selected and **Earthquake to graph** is set to **1**

The dialog should look like this:

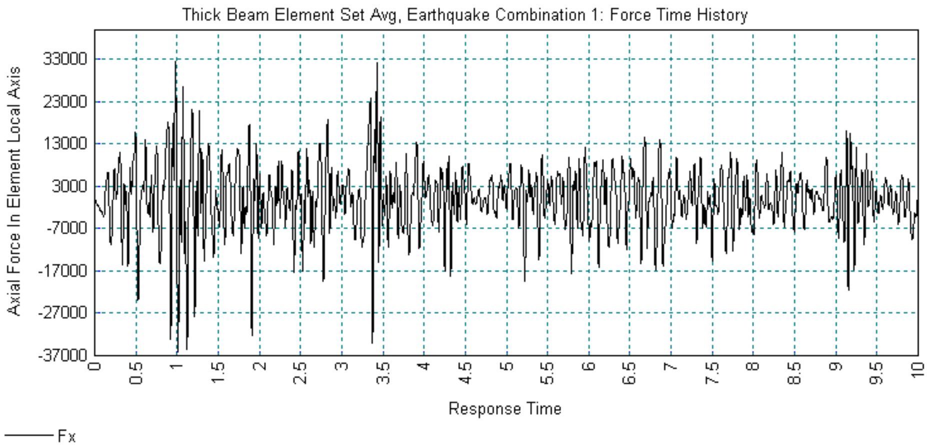


- Click the **Apply** button to proceed. Click **Yes** when asked whether you want to process all of the selected elements.

Note. Any combination of the options **Individual items**, **Sum of chosen items** and **Average of chosen items** can be used 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 16 beam ends of the selected elements. A simple average of this summed result is obtained to give the element average time history of axial force from which the peak average results are obtained.

The IMDPlus analysis will run and after a short while a graph showing the average axial (Fx) force obtained from the selected beam elements will be displayed for the first 10 seconds of the 1940 El Centro earthquake (Earthquake 1). The average axial force in the beam elements is shown in the following figure.



Peak results for the averaged element forces and moments are displayed in Notepad in file peak_forceSet_Avg1.sum. These indicate a peak positive average axial (Fx) force of 32.7kN at solution time 0.98 seconds and a peak negative average axial (Fx) force of -36.1kN at 1.01 seconds for the 1940 El Centro earthquake (Record 1), as shown in the following figure.

```
#
#
# +--------------+
# | Record = 001 |
# +--------------+
#
# Entity                      Time              Value
Peak_Pos_Fx             0.9800000000        32659.62368
Peak_Neg_Fx              1.010000000       -36075.22195
Abs_Peak_Fx              1.010000000        36075.22195
Peak_Pos_Fy              2.105000000        5585.413286
Peak_Neg_Fy              2.905000000       -5997.747790
Abs_Peak_Fy              2.905000000        5997.747790
Peak_Pos_Mz              2.145000000        8359.900914
Peak_Neg_Mz              2.905000000       -8270.496031
Abs_Peak_Mz              2.145000000        8359.900914
```

An envelope of the peak results from the selected beam elements is displayed in Notepad in file peak_forceSet1.sum. Results for the 1940 El Centro earthquake (Record 1) are shown in the following figure. A peak positive axial (Fx) force of 1019.1kN occurs at time 2.13 seconds at end 1 of element 656. A peak negative axial (Fx) force of -956.0kN occurs at time 2.135 seconds at end 1 of element 648.

Note. The envelope of the peak results is obtained in IMDPlus by examining the individual results from all 16 ends of the chosen beam elements. This helps to quickly identify the locations where the peak results occur in the selected set of elements.

```
#
#
# +--------------+
# | Record = 001 |
# +--------------+
#
# Entity                  Time            Value          Element      End
Peak_Pos_Fx          2.130000000      1019065.962          656          1
Peak_Neg_Fx          2.135000000     -955968.1579          648          1
Abs_Peak_Fx          2.130000000      1019065.962          656          1
Peak_Pos_Fy          2.105000000      6684.561177          640          1
Peak_Neg_Fy          2.905000000     -6985.728050          640          1
Abs_Peak_Fy          2.905000000      6985.728050          640          1
Peak_Pos_Mz          2.145000000      15472.76490          640          2
Peak_Neg_Mz          2.905000000     -15552.26423          640          2
Abs_Peak_Mz          2.905000000      15552.26423          640          2
```

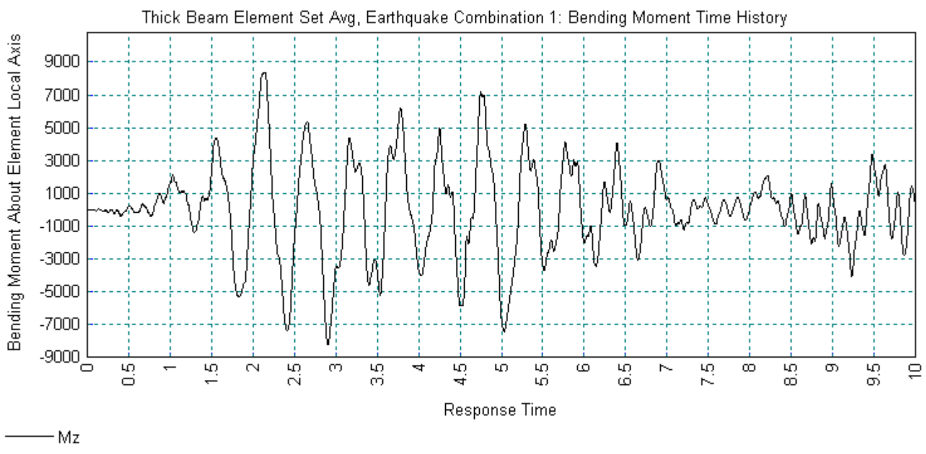
- Close the Notepad applications and graph windows.
- Click on the button in the toolbar to open the IMDPlus Output Control dialog

If the IMDPlus toolbar is not enabled,

Utilities
IMDPlus >
Seismic...

- 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.
- Select **Force/Moment - Thick 2D Beam** results of **Mz**
- Click the **Finish** button to finish output processing of the element results. Click **Yes** when asked whether you want to process all of the selected elements.
- Click **Yes** when asked whether to free up disk space by deleting the temporary files created by IMDPlus.

A graph of the average (Mz) moment obtained from the selected elements will be displayed, as shown in the following figure.



All of the peak averaged results displayed in Notepad will be the same as those shown when the axial (Fx) force was analysed earlier. These indicate an average absolute peak (Mz) moment of 8.4kNm at time 2.145 seconds (Record 1). The envelope of the peak results indicates an absolute peak (Mz) moment of 15.6kNm occuring at time 2.905 seconds in end 2 of element 640 (Record 1).

- Close the Notepad applications and graph windows.

Save the model

File
Save…

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

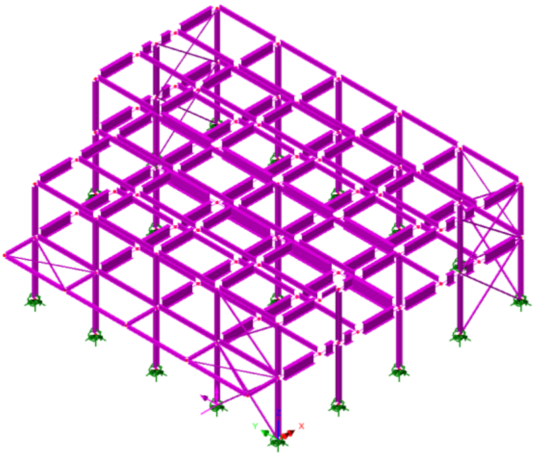
This completes the example.

Seismic Analysis of a 3D Frame (Time Domain)

For software product(s):	All
With product option(s):	IMDPlus
Note: The example exceeds the limits of the LUSAS Teaching and Training Version.	

Description

This example examines the seismic response of a 2-storey 3D frame in the time domain. The geometry of the structure has been simplified to a wire-frame or stick representation with each of the members of the structure being represented by a line feature.



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

Objectives

The output requirements of the analysis are:

- ❑ Maximum deflections in the X-, Y- and Z-directions of a node at the top centre of the frame
- ❑ Average deflections and peak deflections in the X-, Y- and Z-directions from all of the unrestrained nodes in the frame

- Peak axial force and bending moment from all of the frame beams and columns
- Deformed mesh plots associated with the maximum deflections of a node at the top centre of the frame
- Envelopes of axial force and bending moment from modal combinations.

Keywords

Seismic, time domain, response, interactive modal dynamics, excitation, eigenvalue, modal combination.

Associated Files

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

Discussion

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

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

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

The seismic response analysis is performed in two distinct stages. A natural frequency analysis is performed first. This is used here to calculate the first 30 natural modes of vibration of the structure. The eigenvalues (frequencies) and eigenvectors (mode shapes) are stored and used in the subsequent seismic response analysis. In order to carry out a seismic analysis the eigenvectors must be normalised with respect to the mass. Although natural frequencies are obtained from an eigenvalue analysis any information regarding the magnitudes of deformations or moments is non-quantitative.

The second phase of the analysis utilises the IMDPlus option which performs enhanced time domain solutions using Interactive Modal Dynamics (IMD). This is an alternative to performing a spectral response analysis and allows the excitation of the structure using acceleration time histories instead of spectral excitation curves. In the IMDPlus solution, the structure is subjected to support excitation governed by time histories of acceleration in the model global axes. In this example, the seismic excitation is applied directly to the bases of the columns using the first 2.5 seconds of the 1940 El Centro earthquake.

Modelling

Running LUSAS Modeller

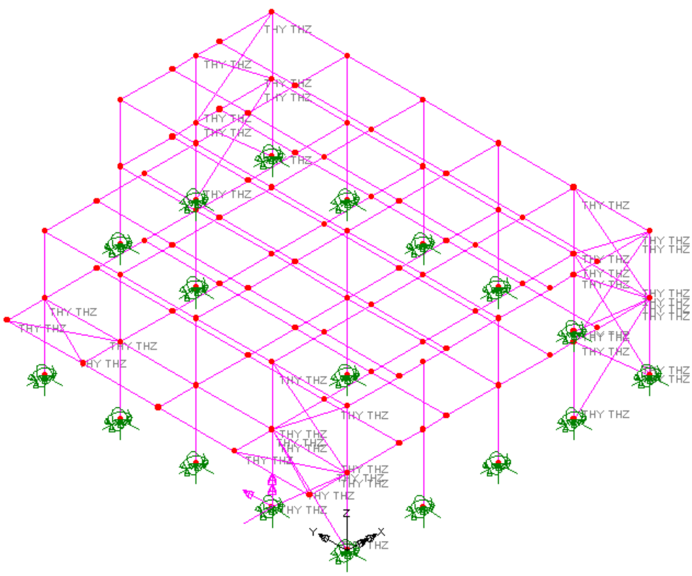
For details of how to run LUSAS Modeller see the heading ***Running LUSAS Modeller*** in the Examples Manual Introduction.

Building and loading the model

File
Open…

- To create the model, open the file **IMDPlus 3D Frame.mdl** located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory. Click the **OK** button if the Open File From Previous Version dialog appears.

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



File
Save As…

- In the **\<LUSAS Installation Folder>\Projects** folder create a new directory called **IMDPlus 3D Frame**
- Save the model into this new folder as **IMDPlus 3D Frame**. This helps keep all relevant IMDPlus created files separate from other analyses and is good practice.

Note. No static structural loading is required for this analysis because only the dynamic loading is considered during the results processing.

The modelling will now be completed by defining the controls necessary to extract the natural frequencies.

Defining Eigenvalue controls

Eigenvalue controls are defined as properties of the loadcase.

- In the Treeview expand **Analysis 1** then right-click on **Loadcase 1** and select **Eigenvalue** from the **Controls** menu.

The Eigenvalue dialog will appear.

- Set the **Number of eigenvalues** as **30**
- Ensure the **Shift to be applied** is set as **0**
- Ensure the **Type of eigensolver** is set as **Default**

Note. Eigenvalue normalisation is set to **Mass** by default. This is essential if the eigenvectors are to be used for subsequent IMD analysis.

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

File
Save

Save the model file.

Running the Analysis

- With the model loaded click the **Solve** button and the **Solve Now** dialog will be displayed.
- Click the **OK** button to run the analysis.

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

Viewing the Results

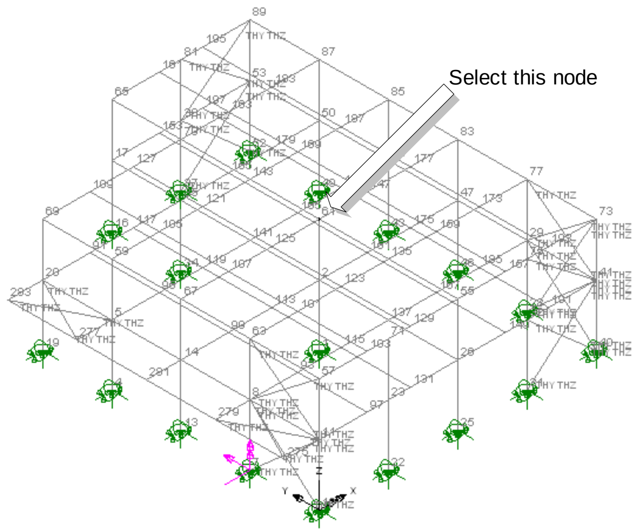
If the analysis was run from within LUSAS Modeller the results will be loaded on top of the current model and the loadcase results for each eigenvalue can be seen in the Loadcase panel of the Treeview with Eigenvalue 1 set to be active by default.

Seismic Response Analysis

Seismic response calculations are performed using the IMDPlus (Interactive Modal Dynamics) facility. This involves defining the excitation and specifying the results required. With a seismic response analysis, additional damping information may also be set. Unlike results from a natural frequency analysis, the output values obtained from a seismic response are design values.

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

- Turn off the **Geometry** layer in the Treeview.
- With no features selected click the right-hand mouse button in a blank part of the graphics window and select the **Labels** option to add the labels to the Treeview.
- On the Properties dialog select the **Node** > **Name** check box and click **OK**
- Select the node at the top and in the centre of the frame, node **61** as shown in the following diagram



Defining the Seismic Parameters

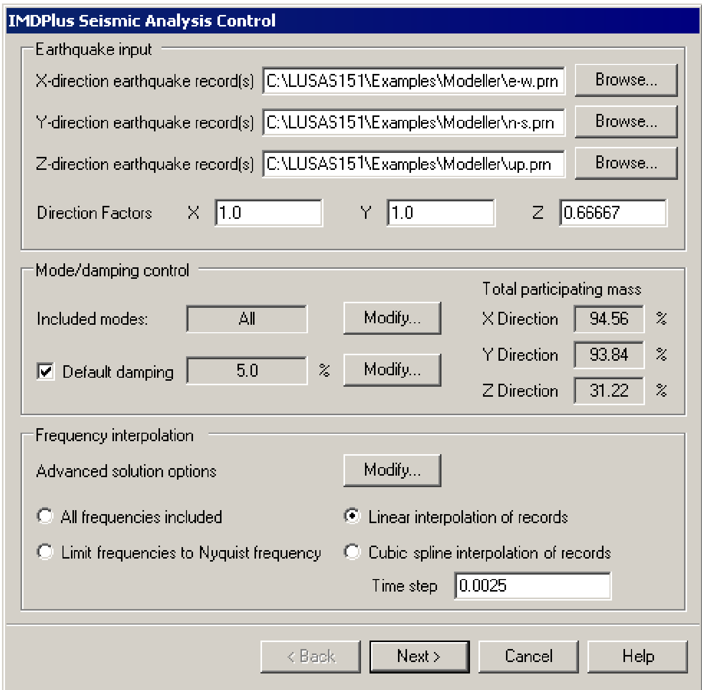
Note. To simplify the use of IMDPlus (and provide an alternative to picking IMDPlus menu selections from the Utilities menu) for the rest of the example it is recommended that the IMDPlus toolbar is enabled in LUSAS Modeller. To do this select the **View** > **Toolbars...** menu command and enable the **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.

IMDPlus > Seismic...

- Open the IMDPlus Seismic Analysis Control dialog through the menu or click on the button in the toolbar.

The input for the seismic analysis is defined on the IMDPlus Seismic Analysis Control dialog. The input consists of the earthquake acceleration records, included modes and damping along with interpolation options.



- In the Earthquake Input section, for the **X-Direction Earthquake Record(s)** browse to the \<**Lusas Installation Folder>\Examples\Modeller** directory and select **e-w.prn**

- For the **Y-Direction Earthquake Record(s)** browse to the \<**Lusas Installation Folder>\Examples\Modeller** directory and select **n-s.prn**
- For the **Z-Direction Earthquake Record(s)** browse to the \<**Lusas Installation Folder>\Examples\Modeller** directory and select **up.prn**
- Ensure that the **Direction Factors** are set to **1.0** in the X-Direction, **1.0** in the Y-Direction and **0.66667** in the Z-Direction
- Ensure **Included modes** is set to **All**. If this is not the case, click on the **Modify…** button and turn the **All modes** option on.
- Ensure **Default damping (5%)** is selected. If a different damping is displayed, click on the **Modify…** button and set the **Default damping** to **5**
- In the Frequency Interpolation section, click on the **Modify...** button to change the advanced solution options. Click the **Defaults** button to set the default options and click the **OK** button.
- Select the **Linear interpolation of records** option and enter a time step of **0.0025** (Half the time step of the original earthquake records)
- Click the **Next** button to proceed. When prompted about significant missing total mass choose **Yes** as only 31.22% is included in the Z-Direction (vertical) for 30 modes.

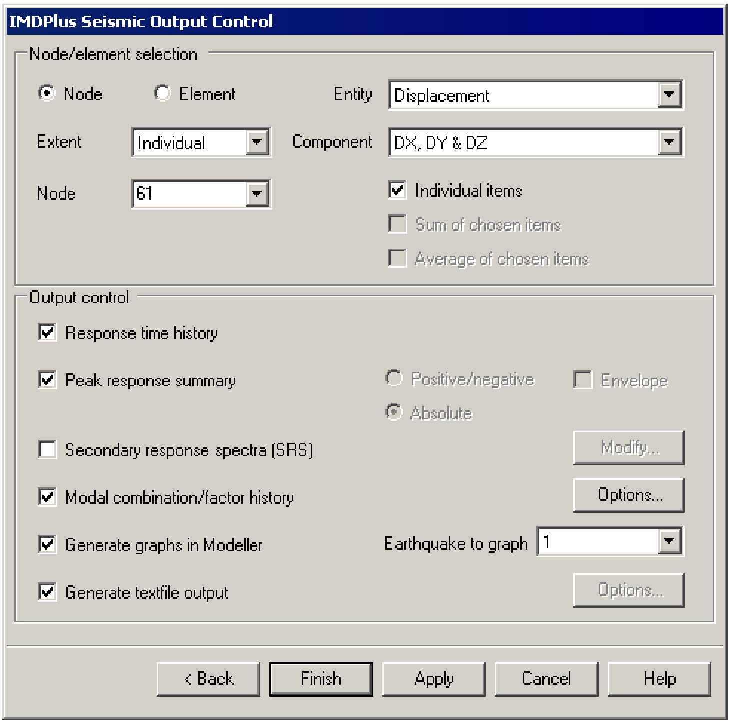
Individual Node Displacements

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

- Select **Node** and select **Extent** as **Individual**
- Choose **Node** number **61** from the drop-down list.
- Select **Displacement** results of **DX, DY & DZ**
- Ensure **Individual items** is selected. **Sum of chosen items** and **Average of chosen items** will not be available as an individual node is being processed.
- Select **Response time history**
- Select **Peak response summary**
- Select **Modal combination/factor history** and click on the **Options…** button.
- Set the **Minimum mass participation factor percentage to include** to **1.0** and click on **OK**

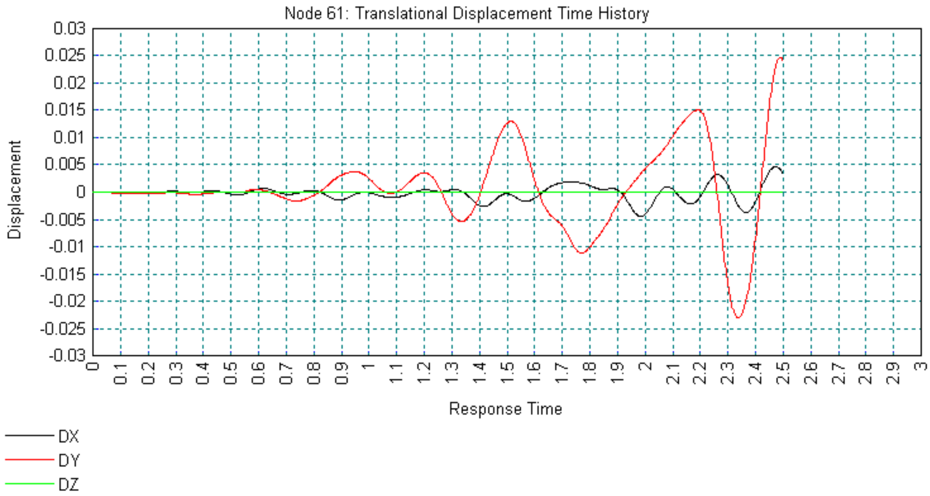
- Select **Generate textfile output**

The dialog should now look like this:



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

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



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

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

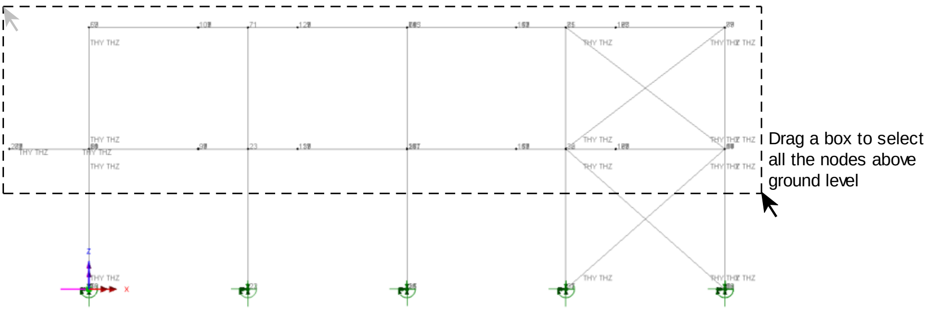
- Close the Notepad application and graph window.

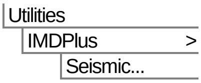
Average and Peak Displacements for a Selection of Nodes

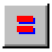
This section investigates results for a chosen set of nodes containing all of the unrestrained nodes in the model, that is, all of the nodes above ground level. The average displacements and peak displacements of the selected nodes will be obtained.

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


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





- Open the IMDPlus Seismic Analysis Control dialog through the menu or click on the  button in the toolbar
- On the IMDPlus Seismic Analysis Control dialog click the **Next** button to accept the previously defined values. When prompted about the total mass participation click the **Yes** button to continue.

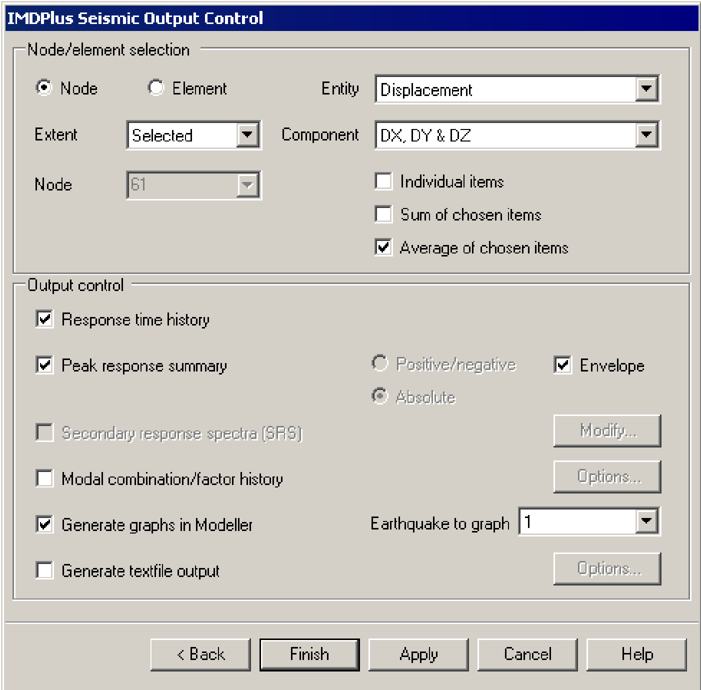
Enter the following information into the IMDPlus Output Control dialog.

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

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

- Select result entity **Displacement** and component **DX, DY & DZ**
- Deselect **Individual items** and select **Average of chosen items** as the averaged results of the selected nodes are going to be investigated.
- Ensure **Response time history** and **Generate graphs in Modeller** remain selected.
- Ensure **Peak response summary** remains selected and also select **Envelope**
- Deselect **Modal combination/factor history** and **Generate textfile output**

The dialog should now look like this:

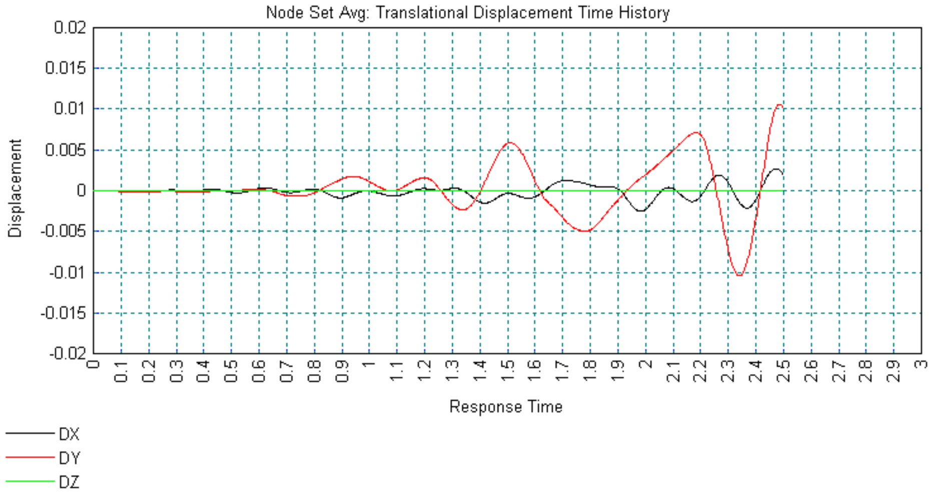


- Click the **Apply** button to perform the analysis. Click **Yes** when asked whether you want to process all of the selected nodes.

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

Note. **Average of chosen items** obtains results by first summing the results from the 89 nodes selected. A simple average of these summed results is obtained to give the averaged node time histories from which the peak average results are obtained.

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



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

```
#
#
# +---------------+
# | Record = 001 |
# +---------------+
#
# Entity                        Time              Value
Peak_Pos_DX                2.472500000       0.2711970446E-002
Peak_Neg_DX                1.982500000      -0.2545200708E-002
Abs_Peak_DX                2.472500000       0.2711970446E-002
Peak_Pos_DY                2.485000000       0.1059843682E-001
Peak_Neg_DY                2.340000000      -0.1046089938E-001
Abs_Peak_DY                2.485000000       0.1059843682E-001
Peak_Pos_DZ                0.9900000000      0.4570291210E-004
Peak_Neg_DZ                1.015000000      -0.8529296567E-004
Abs_Peak_DZ                1.015000000       0.8529296567E-004
Peak_Pos_THX               2.337500000       0.3003976055E-003
Peak_Neg_THX               2.492500000      -0.3044003645E-003
Abs_Peak_THX               2.492500000       0.3044003645E-003
Peak_Pos_THY               2.475000000       0.4385270496E-004
Peak_Neg_THY               1.990000000      -0.4369077358E-004
Abs_Peak_THY               2.475000000       0.4385270496E-004
Peak_Pos_THZ               2.340000000       0.2687725883E-003
Peak_Neg_THZ               2.482500000      -0.3000152315E-003
Abs_Peak_THZ               2.482500000       0.3000152315E-003
Abs_Peak_RSLT              2.482500000       0.1156838441E-001
```

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

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

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

- Close all of the Notepad applications and graph windows.

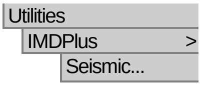
Peak Axial Forces from All of the Elements in the Model

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

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

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

- Click on the button in the toolbar to open the IMDPlus Output Control dialog



If the IMDPlus toolbar is not enabled,

- Open the IMDPlus Seismic Analysis Control dialog through the menu and click **Next >** to keep the existing analysis control settings and open the IMDPlus Output Control dialog.

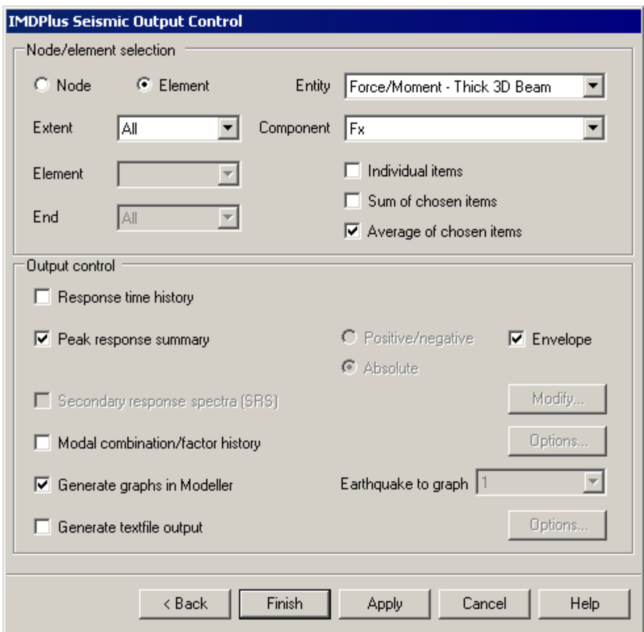
Enter the following information into the IMDPlus Output Control dialog.

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

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

- Select results entity **Force/Moment – Thick 3D Beam** and component **Fx**
- Ensure **Average of chosen items** remains selected and **Individual items** and **Sum of chosen items** deselected as only the average results of the chosen elements are going to be investigated.
- Deselect **Response time history**
- Ensure **Peak response summary** and **Envelope** remain selected.

The dialog should now look like this:



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

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

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

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

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

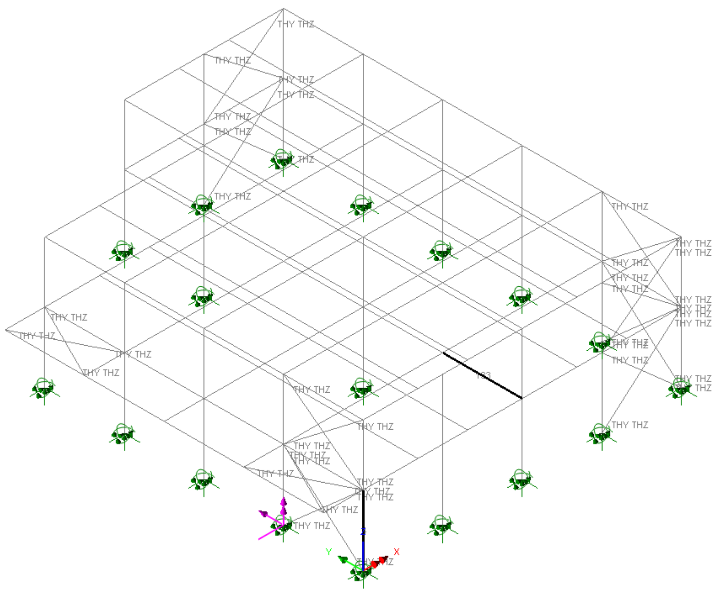
```
#
#
# +--------------+
# | Record = 001 |
# +--------------+
#
# Entity               Time           Value          Element    End
Peak_Pos_Fx     2.340000000     34152.50443           10      1
Peak_Neg_Fx     2.342500000    -30970.35591            4      1
Abs_Peak_Fx     2.340000000     34152.50443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

- Close the Notepad applications.

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

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

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



Displaying Deformed Mesh Plots from Modal Combinations

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

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

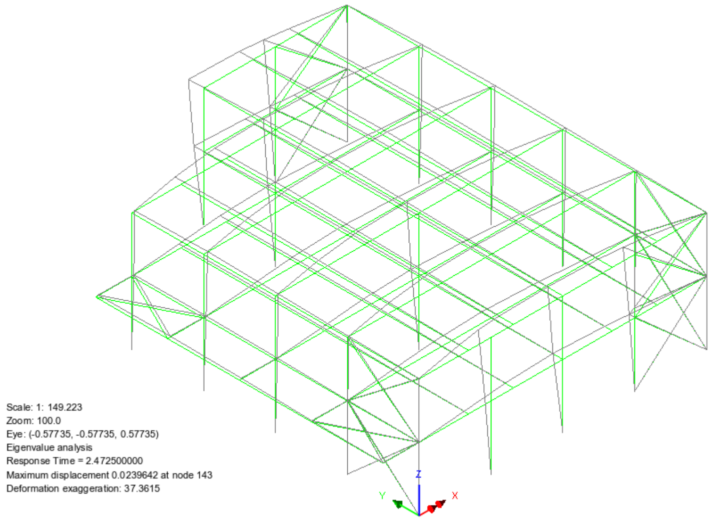
File
Script >
Run Script...

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

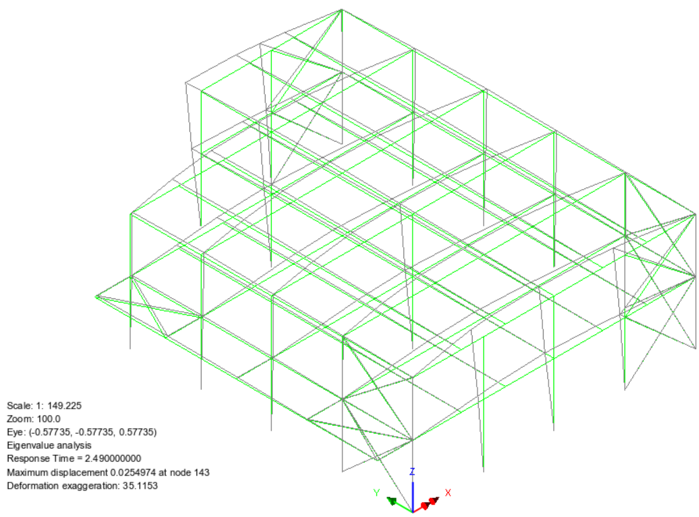
The combinations will appear under the original model in the Treeview.

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

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



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



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

Displaying Envelopes for the Whole Earthquake

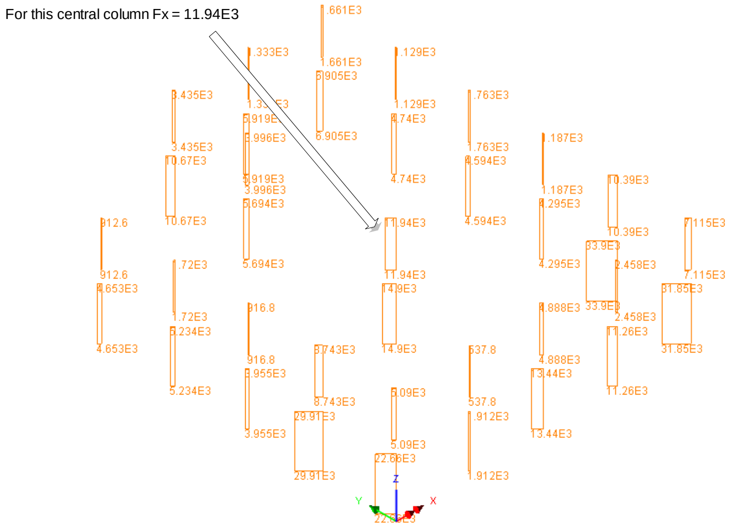
Plots showing the maximum and minimum axial force and bending moment in the columns for the entire earthquake duration are to be displayed.

Analyses
Envelope…

Create an envelope

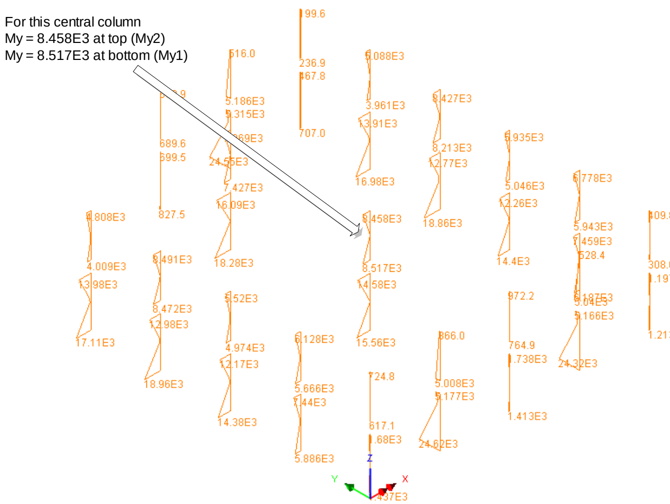
- Enter **Whole Earthquake** for the name
- Select **model** data from the drop-down list
- Hold down the shift key and select combinations **2** to **1002** in the list and click on the >> '**Add to**' button to add all combinations to the envelope
- Click the **OK** button to create the max and min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

The envelope will now be assembled over the 1000 modal combinations generated by IMDPlus. On modern computers this will take under 5 minutes.



- In the Treeview right-click on the **Whole Earthquake (Max)** envelope and select the **Set Active** option to set this envelope active. Select entity **Force/Moment – Thick 3D Beam** of component **My** and click the **OK** button. When informed that the component is different to the one used in the drawing layers click the **Yes** button to update the diagrams layer.

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



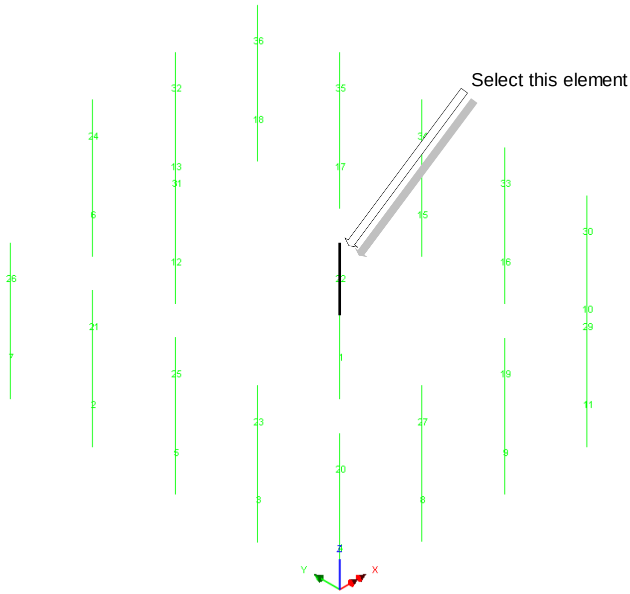
Displaying results for a selected member

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

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

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

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



Utilities
IMDPlus >
Seismic...

- 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 select **Element** change **Extent** to **Individual**

- Select **Element** number **22** from the drop-down list and ensure **End** is set to **All**
- Choose **Force/Moment – Thick 3D Beam** and component **Fx**.
- Select **Individual items**. **Sum of chosen items** and **Average of chosen items** will not be available as an individual element is being processed.
- Ensure **Peak response summary** is selected.
- Click the **Finish** button and click the **Yes** button to confirm that both ends of the beam element are to be processed (because All ends has been previously set).
- Click **Yes** when asked whether to free up disk space by deleting the temporary files created by IMDPlus.

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

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

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

- Close the Notepad application.

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

Save the model

File
Save…

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

This completes the example.

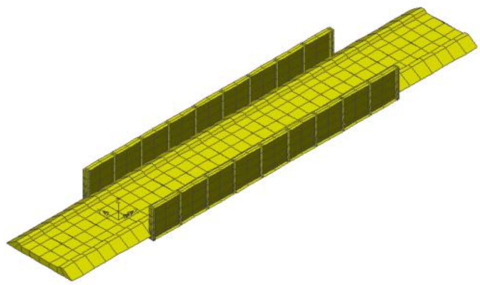
Train Induced Vibration of a Bridge

For software product(s):	Any Plus version
With product option(s):	IMDPlus
Note: The example exceeds the limits of the LUSAS Teaching and Training Version.	

Description

This example examines the response of a steel rail bridge to the passage of a train.

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



Objectives

The output requirements of the analysis are:

- ❑ Deflections in the X-, Y- and Z-directions for a train speed of 15 m/s
- ❑ Accelerations in the vertical direction for a train speed of 70 m/s
- ❑ Peak deflections and accelerations in the vertical direction for a speed range of 15 m/s to 70 m/s in 5 m/s intervals
- ❑ Stress resultants and peak stress resultants in main girder web.
- ❑ Averaged and peak vertical displacements and resultant dynamic amplification factors from all of the nodes of the bridge structure

- Total summed reactions and peak reactions from all of the supported nodes of the bridge structure
- Averaged and peak stresses for a side panel of the bridge structure

Keywords

Moving load, time domain, response, interactive, modal, dynamics, IMDPlus, eigenvalue.

Associated Files

- **IMDPlus Rail Bridge.mdl** Model of the structure.
- **ec1-3 Type3.lmd** Attribute library that contains the Eurocode ENV 1991-3 Type 3 moving load train definition for the example.
- **ec1-3 Type3.xls** Microsoft Excel spreadsheet that contains the axle positions and loads of the Eurocode ENV 1991-3 Type 3 train set.

Modelling

Running LUSAS Modeller

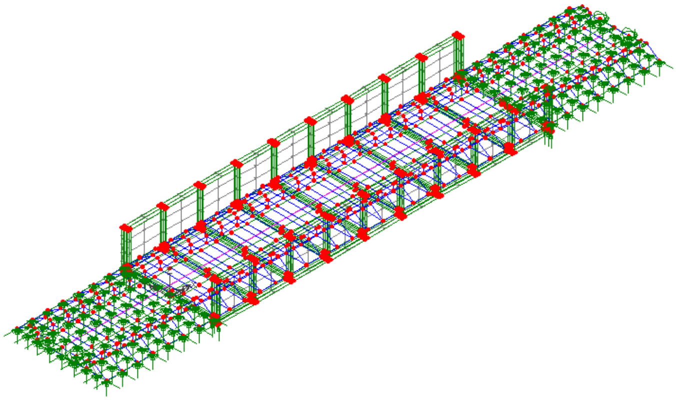
For details of how to run LUSAS Modeller see the heading ***Running LUSAS Modeller*** in the Examples Manual Introduction.

Building and loading the model

File
Open…

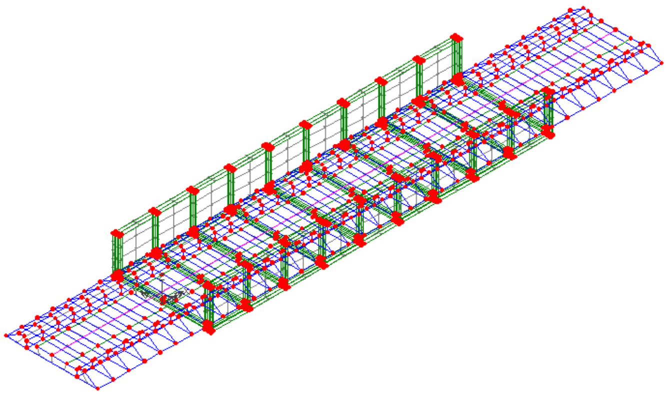
To create the model, open the read-only file **IMDPlus Rail Bridge.mdl** located in the **\<LUSAS Installation Folder>\Examples\Modeller** directory. Click the **OK** button if the Open File From Previous Version dialog appears.

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



File
Save As…

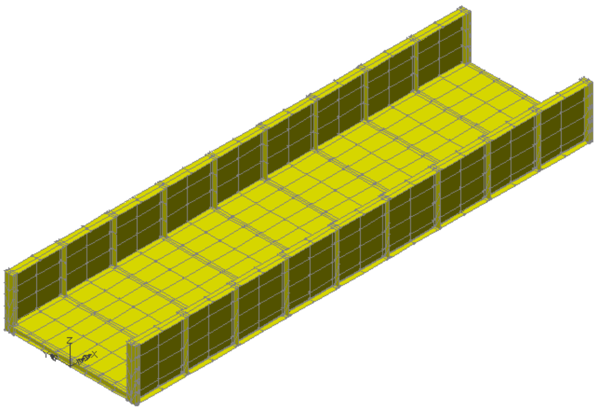
- In the \<**LUSAS Installation Folder**>**\Projects** folder create a new directory called **IMDPlus Rail Bridge**
- Save the model into this new folder as **IMDPlus Rail Bridge**. This helps keep all relevant IMDPlus created files separate from other analyses and is good practice.
- Turn off the display of the **Attributes** layer in the Treeview.



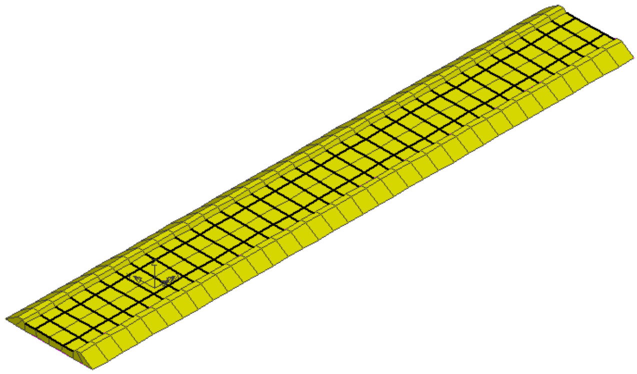
Note. No static structural loading is required for this analysis because only the dynamic loading is considered during the results processing, which is defined later.

Modelling Discussion

The bridge is approximately 16.5m long and 4.75m wide and carries a single track with ballast and concrete sleepers. The primary structure is constructed from steel and is modelled using shell elements. For this analysis, modelling of the ballast/track-bridge interaction has been carried out using a full 3D solid representation of the ballast with the sleepers modelled with thick beam elements laid along the top of the ballast. The rails of the track have been modelled using thick beam elements spanning between the sleepers and the whole ballast/track model is connected to the bridge deck using a sliding-only slideline. For the purposes of this example the bridge and ballast model has been represented with a coarse mesh and therefore the results from this analysis will be inaccurate. For analysing actual structures it is recommended that a finer mesh be used which can capture both the vibration modes of the bridge and the movement of the load across the bridge more accurately.

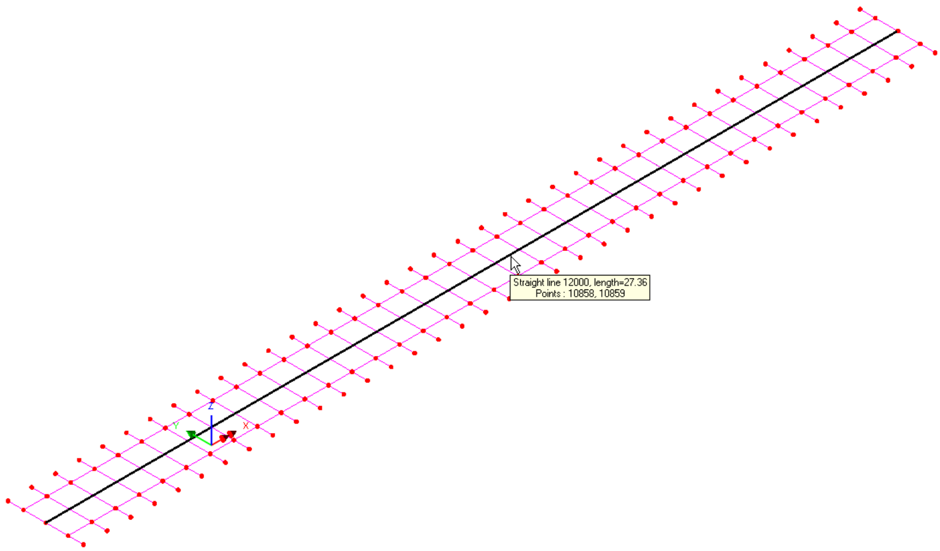


Bridge representation



Ballast representation

To allow the loading of the rails to be distributed to the 3D solid elements using discrete point loads the track modelling uses weak, weightless shell elements which span between the beam elements representing the rails. The track representation is shown in the following figure.



Track (sleepers and rails) representation

Note that the line down the centre of the track (Line 12000) is used to define the path of the train across the bridge and will be used later in this example. In order to avoid any adverse dynamic behaviour of the weak weightless shell elements, only one division is used to span between the two rails thus providing full support to the shells without adding stiffness to the model.

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 **Eigenvalues required** to **Range**
- Ensure the **Range** is set to **Frequency**
- Set the **Minimum frequency** as **0**
- Set the **Maximum frequency** as **35**
- Set the **Number of eigenvalues** to **0** (this solves for ALL frequencies in the range)
- 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 layer. Eigenvalue 1 is set to be active by default.

Checking the Mass Participation Factor

Note. In order to carry out a successful IMDPlus 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 has been achieved in the global directions. If less than 90% has been achieved no further modes need be included, if and only if, the modes of vibration omitted cannot be excited by the dynamic input or a significant proportion of the structure is restrained by support in these directions and therefore cannot participate in the modes of vibration. The acceptability of the included modes of vibration will vary from analysis to analysis but failure to check that a significant proportion of the total mass has been accounted for may lead to important modes being missed and subsequent errors in the analysis results.

Utilities
Print Results Wizard…

- Select **Active** and click **Next**.
- Select Entity **None** and from the Type drop-down menu select **Sum mass participation factors** and then click **Finish.** The Sum Mass Participation Factors results will be printed to the Text Output window.

It can be seen that the 90% value has almost been achieved in all directions for this analysis. This is discussed in the note below.

MODE	SUM MASS X	SUM MASS Y	SUM MASS Z
1	0.110940E-01	0.117572E-04	0.707920
2	0.121970E-01	0.133845E-04	0.708029
3	0.125612E-01	0.182964	0.708274
4	0.131909E-01	0.182989	0.769089
5	0.290473E-01	0.352634	0.770999
6	0.430279E-01	0.813529	0.771960
7	0.177903	0.890132	0.806536
8	0.178201	0.892043	0.831905
9	0.188464	0.893014	0.832302
10	0.207253	0.893215	0.833539
11	0.864564	0.893542	0.861563
12	0.901000	0.938795	0.862830
13	0.901132	0.939785	0.862830
14	0.920871	0.940199	0.864118

15	0.920872	0.940360	0.881332
16	0.938299	0.940366	0.882002
17	0.938309	0.940637	0.882002
18	0.938347	0.940642	0.882003
19	**0.938366**	**0.940708**	**0.882003**

Note. In this analysis we are only including modes of vibration with frequencies up to and including 35 Hz with frequencies higher than this value considered insignificant for the analysis. In this analysis the printed results show that 93.8% of the total mass is achieved in the X-direction, 94.1% is achieved in the Y-direction and 88.2% is achieved in the Z-direction.

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

Utilities
Print Results Wizard…

- Select **Active** and click **Next**.
- Select Entity **None** and from the Type drop-down menu select **Mass participation factors** and then click **Finish.** The Mass Participation Factors results will be printed to the Text Output window.

MODE	MASS PF X	MASS PF Y	MASS PF Z
1	0.110940E-01	0.117572E-04	**0.707920**
2	0.110297E-02	0.162728E-05	0.109262E-03
3	0.364226E-03	**0.182951**	0.245424E-03
4	0.629662E-03	0.241889E-04	0.608148E-01
5	0.158564E-01	**0.169645**	0.191047E-02
6	0.139805E-01	**0.460895**	0.960755E-03
7	0.134875	0.766029E-01	0.345759E-01
8	0.298085E-03	0.191048E-02	0.253685E-01
9	0.102626E-01	0.971036E-03	0.397193E-03
10	0.187893E-01	0.200908E-03	0.123754E-02
11	0.657310	0.327165E-03	0.280234E-01
12	0.364360E-01	0.452530E-01	0.126713E-02
13	0.132603E-03	0.990173E-03	0.130638E-07
14	0.197387E-01	0.413466E-03	0.128847E-02
15	0.797027E-06	0.161819E-03	0.172134E-01
16	0.174272E-01	0.517541E-05	0.670045E-03
17	0.100738E-04	0.271287E-03	0.727634E-06
18	0.383935E-04	0.473477E-05	0.479303E-07

```
    19    0.185724E-04    0.667088E-04    0.664934E-07
```

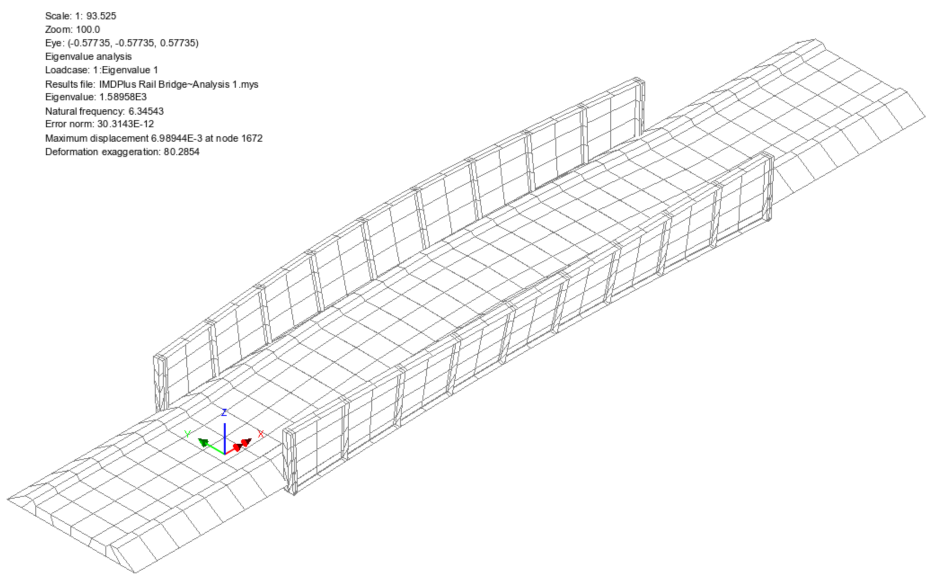
From these mass participation factors the major modes of vibration of the bridge can be seen to be mode 1 in the Z-direction (vertical) and modes 3, 5 and 6 in the Y-direction (lateral).

- Close the text window by selecting the close button in the top right hand corner of the window.
- Use the maximise button to increase the size of the graphics window.

Plotting Mode Shapes

- Turn off the display of the **Mesh,** and **Geometry** layers in the Treeview.
- With no features selected click the right-hand mouse button in a blank part of the Graphics area and select the **Deformed mesh** option to add the deformed mesh layer to the Treeview. Click the **OK** button to accept the default values and display the deformed mesh for Eigenvalue 1.
- In the panel at the bottom of the Treeview select the **Window summary** option and click the **Details...** button. In the Window summary properties dialog set the position to **(50,-10)** and click the **OK** button to return to the graphics window.

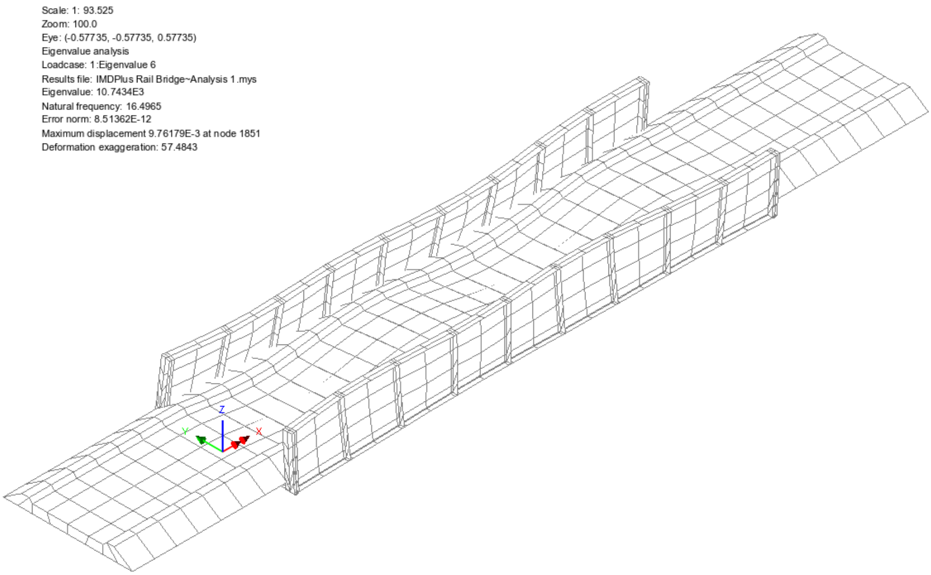
This mode of vibration is the primary mode in the vertical direction as determined in the Checking the Mass Participation Factor section.



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

- In the Treeview right-click on **Eigenvalue 6** and select the **Set Active** option. The deformed mesh plot for Eigenvalue 6 will be displayed.

This mode of vibration has the highest participating mass in the lateral (Y) direction for the whole bridge as determined in the Checking the Mass Participation Factor section.



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. This is because the sense is arbitrary since during vibration the deformed shape will appear in both directions.

Selecting individual nodes and elements of interest

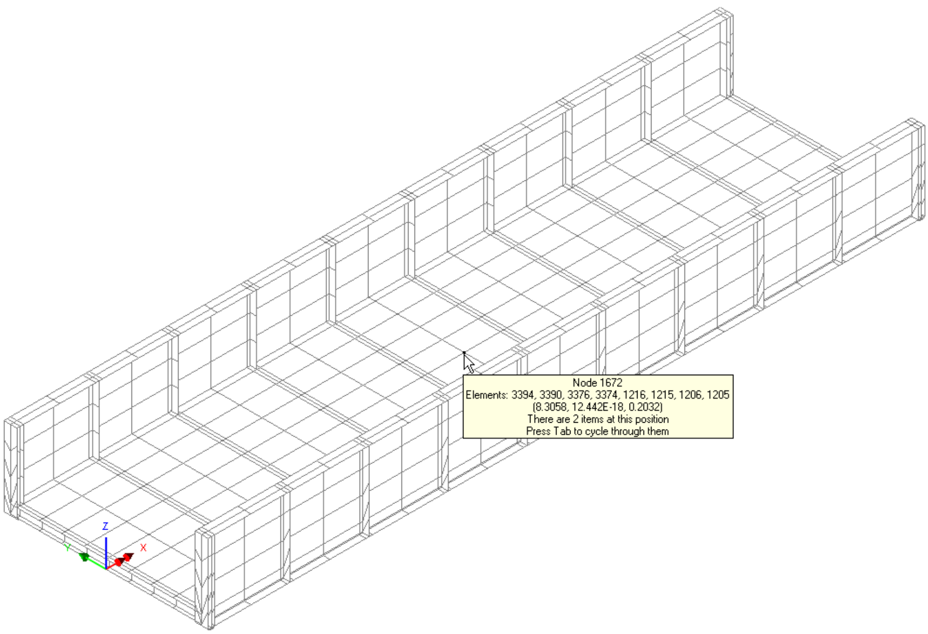
Prior to running an IMDPlus analysis, individual node and element numbers for the locations of the structure that will be assessed should be ascertained. This can be done by selecting the locations of interest with the cursor and noting down the numbers of the node and elements concerned.

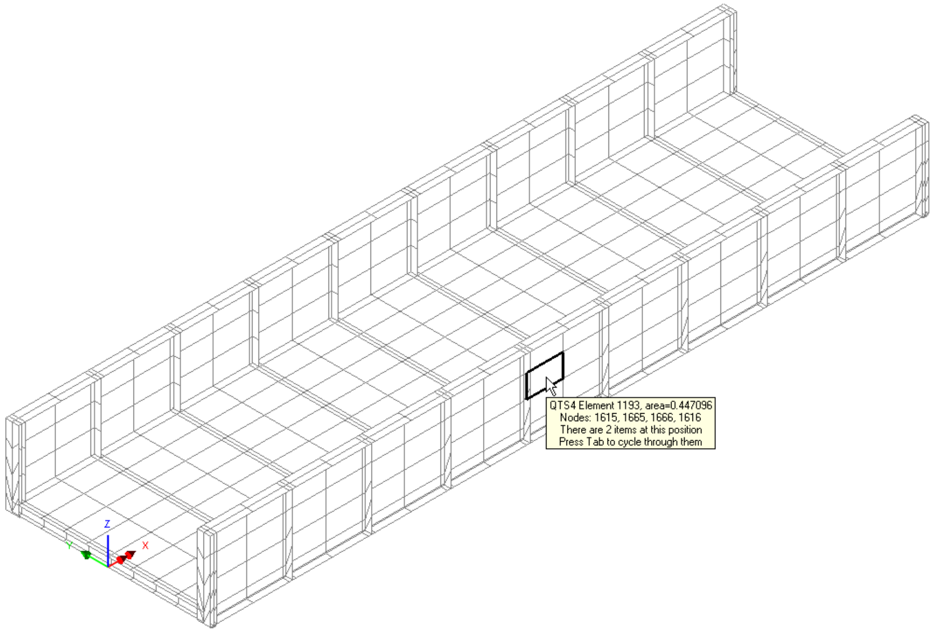
- Turn off the display of the **Deformed mesh** and **Annotation** layers in the Treeview.
- In the Treeview double-click on the **Mesh** layer name and click the **OK** button to accept the default settings. This will turn on the layer.

To view just the bridge structure without the ballast and track:

- In the Treeview click the right-hand mouse button on the group name **Bridge_Structure** and select the **Set as Only Visible** option from the drop-down menu.

By selecting or hovering the cursor over the node and element shown, the node or element number of interest can be obtained. For the first stage of this analysis we are interested in the node at the centre of the bridge deck and the element in the middle of the side girder, as shown in the following figures.





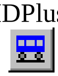
Defining the Eurocode ENV 1991-3 Type 3 Trainset

Before the analysis can be performed we need to define the moving load representation of the Eurocode ENV 1991-3 Type 3 trainset in LUSAS Modeller.

Note. To simplify the use of IMDPlus for the rest of the example it is recommended that the IMDPlus toolbar is enabled in LUSAS Modeller. To do this select the **View > Toolbars...** menu command and enable the **IMDPlus** option in the list and click **Close**.

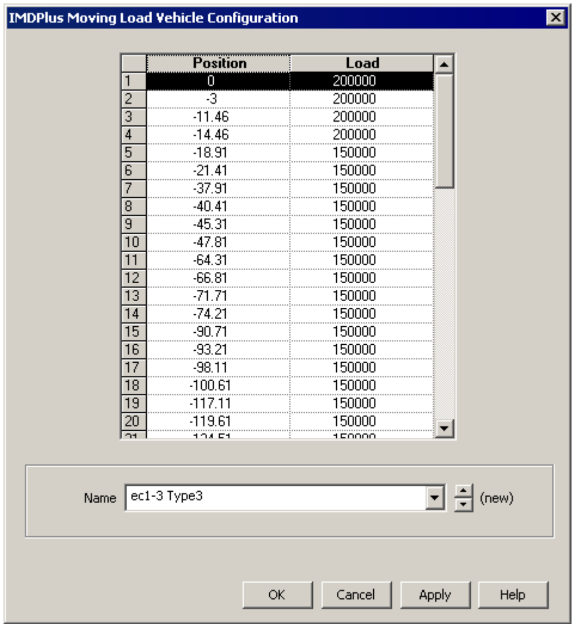
- If the IMDPlus toolbar has been enabled then click on the  Moving Load analysis button in the toolbar to select a moving load analysis and enable the toolbutton shortcuts to the dialogs.

Utilities
IMDPlus >
Moving Load >
Vehicle Configuration...

- Open the IMDPlus Moving Load Vehicle Configuration dialog through the menu or click on the  button in the toolbar. The moving load vehicle is now defined using the positions of the axles relative to the front of train together with the moving load factors to be applied at each position. To simplify the definition of the vehicle the positions and loads are included in a Microsoft Excel spreadsheet so they can be copied and pasted into the dialog.

- Locate the **ec1-3 Type3.xls** spreadsheet in the **\<LusasInstallationFolder>\Examples\Modeller** folder and open it.
- Select all of the **60** positions and loads in the first and second columns and copy these to the clipboard using **Ctrl+C**
- Select both the **Position** and **Load** headers of the grid in the dialog and paste these into the grid using **Ctrl+V**
- Enter the name as **ec1-3 Type3**

Once finished the dialog should appear as shown.



- Click **OK** to create the vehicle configuration.

Note. Alternatively each **Position** and **Load** in the table can be entered manually.

Note. The positions of the axles should be defined as negative values to ensure that the whole trainset passes over the structure. Negative values place the axles behind the front of the train which is at a position of 0. The movement of the front of the train will be calculated from the start of the path to the end of the path with additional time included in the IMDPlus solution to allow all of the axles with negative positions to pass along the path.

If any axles are defined with positive values then these will already be the distance down the path equivalent to the position entered in the vehicle configuration. Those axles with positive positions may already be over the structure at the start of the IMDPlus analysis if the start of the path has not been defined to account for this. If any positive positions are used then always ensure that the lines defining the beginning of the path allow sufficient additional length in order to model the transition of the whole of the train onto the structure correctly.

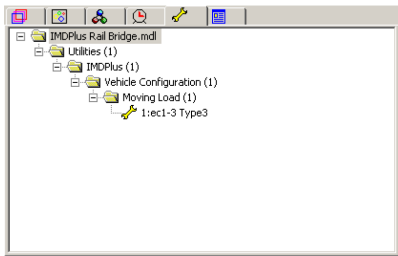
Note. Once a trainset or vehicle has been defined it can be saved in a library so it can be imported and used in future analyses without the need to go through all of the definition process above. This is achieved through the **Library Browser** accessed through the **File > Import/Export Model Data...** menu command.

Importing a Vehicle

If it proves time consuming or you are unable to obtain the same results shown later in the example the vehicle definition can be imported from the library provided in the **\<LusasInstallationFolder>\Examples\Modeller** folder.

- ❑ The **ec1-3 Type3.lmd** contains the vehicle library for the example.

- Select the **Import from library to model** option
- Click on the **Choose file...** button and browse to the **ec1-3 Type3.lmd** file in the **\<LusasInstallationFolder>\Examples\Modeller** folder.
- Select the **IMDPlus (1)** entries in the tree
- Click the **OK** button.



After vehicle configuration attribute has been defined it will appear in the Utilities treeview.

To edit the vehicle configuration attribute simply double-click with the left mouse button or right-click and choose **Edit Attribute...** on the name of the attribute to bring up the original dialog.

Moving Load Analysis

Moving load calculations are performed using the IMDPlus (Interactive Modal Dynamics) facility. In order to carry out the moving load analysis of the train travelling across the bridge we need to carry out three stages:

1. Define and set-up the path along which the moving loads will travel using a unit load defined as either a discrete point or patch load. For this example the unit axle load has already been defined as a discrete point load called **Unit Axle Load** which acts vertically
2. Convert the loading along this path from this unit load into modal forces that are applied in the IMDPlus moving load analysis

3. Run an IMDPlus moving load analysis to calculate the response of the bridge

Note. This process is similar to the IMDPlus moving mass analysis procedure but with the trainset / vehicle defined by constant forces rather than spring-mass systems.

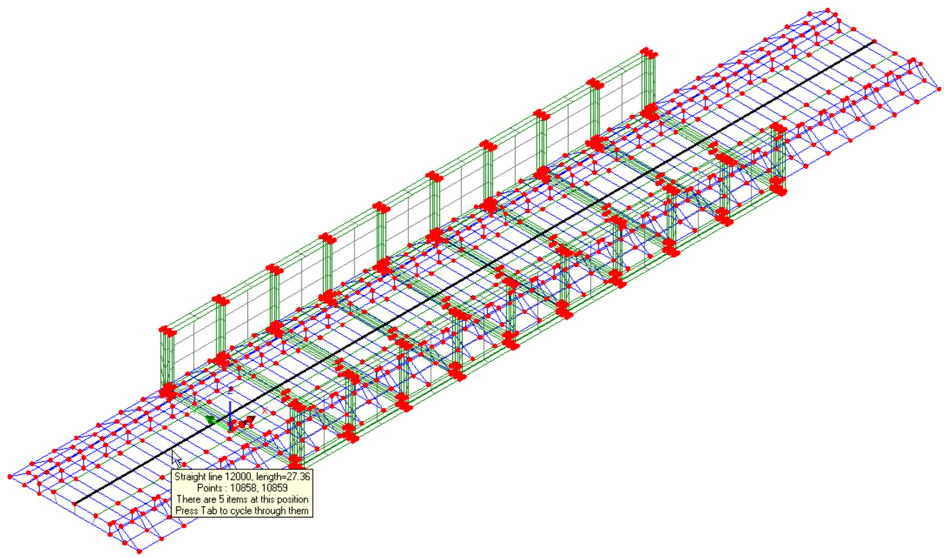
Note. Before an IMDPlus analysis can be carried out the load that is going to pass over the structure must be defined using either discrete point or patch loads. For this example this has already been carried out with a single axle of unit load defined as a discrete point load. Defining a single axle allows multiple load configurations to be analysed through the composite axle definition method in IMDPlus without the need to carry out the path and modal force stages for each layout. For railways the axle lengths remain constant over all of the train set and this method would normally be used. For moving loads where axles are of different widths the full definition of the load must be carried out with the path and modal force stages carried out for each layout.

Defining the Moving Load Path

To solve for the passage of the train across the bridge the path for the moving load must be defined. Line 12000 (the line representing the path of the moving load) will be set to be the current selection.

To view the complete model again:

- In the Treeview click the right-hand mouse button on the group name **IMDPlus Rail Bridge.mdl** and select the **Visible** option from the drop-down menu. Click **Yes** when asked whether to act on subgroups as well.
- In the Treeview double-click on the **Geometry** layer name and click the **OK** button to accept the default settings. This will also turn on the layer.
- Select the **line** shown (line 12000)



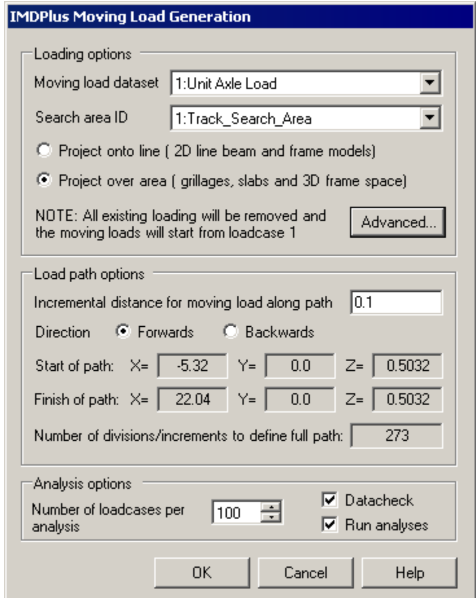
Note. The path can be built from multiple lines and arcs but these must form a continuous path without branching.

Utilities
IMDPlus >
Moving Load >
Moving Load Generation...

- Open the IMDPlus Moving Load Generation dialog through the menu or click on the button in the toolbar.

On startup of the IMDPlus Moving Load Generation dialog, all valid discrete loads and search areas will be made available in the loading options along with information about the path defined by the current selection.

In this example a single discrete load called Unit Axle Load which defines the unit loading from a single axle of the train is present along with a search area that is assigned to the weak weightless shells.



- Ensure **1:Unit Axle Load** is selected from the **Moving load dataset** list.
- Ensure **1:Track_Search_Area** is selected from the **Search area ID** for the assignment of the discrete loading.
- Ensure **Project over area** is selected.
- Click on the **Advanced** button to adjust the inclusion of load characteristics. On the Moving Load Advanced Options dialog choose the **Include Full Load** option for loads outside the search area and click the **OK** button.
- On the IMDPlus Moving Load Generation dialog set the **Incremental distance** to **0.1**

Note. Using search areas targets the application of the loading to the required features as described in the Modeller Reference Manual.

Note. By default the incremental distance is set to one tenth of the length of the line along which the load moves.

- Click the **OK** button to proceed and choose **Yes** to accept the warnings and save the current model.

The program will now generate the loading information for the 274 locations of the unit axle along the path before returning control back to LUSAS Modeller. This process will take only a minute or so on a modern computer and does not need to be repeated unless the lateral configuration of the load changes. For a railway this will not happen but it may be required for highway analyses where the axle lengths and tyre configurations could vary.

Note. The discrete loading locations defined by this dialog will be tabulated into three datafiles with a maximum of 100 loadcases each and the analyses will be performed automatically. These analyses will use the same file basename as the original model with a numeric indicator appended to it (e.g. _00001, _00002, etc). They are required for the modal force calculation stage.

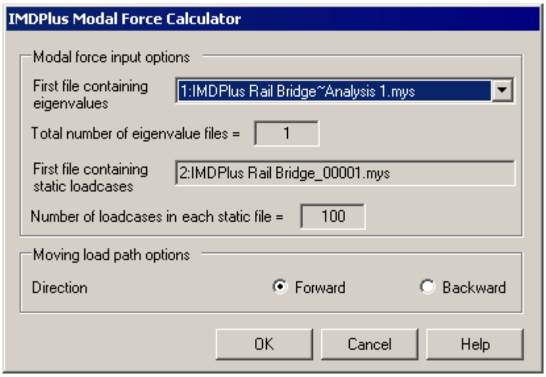
Generating the Modal Force History for the Moving Load

In the previous stage the passage of the train axle across the structure has been defined. The modal forces for the IMDPlus solution now need to be calculated using the Modal Force Calculator.

Utilities
IMDPlus >
Moving Load >
Modal Force Calculator...

- Open the IMDPlus Modal Force Calculator dialog through the menu or click on the button

- Click the **OK** button to accept the default information and proceed.



Note. This process will take only a minute or so on a modern computer and does not need to be repeated unless the moving load path or configuration in Stage 1 is changed.

Defining the Moving Load Parameters

All of the basic moving load information has now been defined for the IMDPlus analysis. The next stage is to define the included modes, damping and speed parameters.

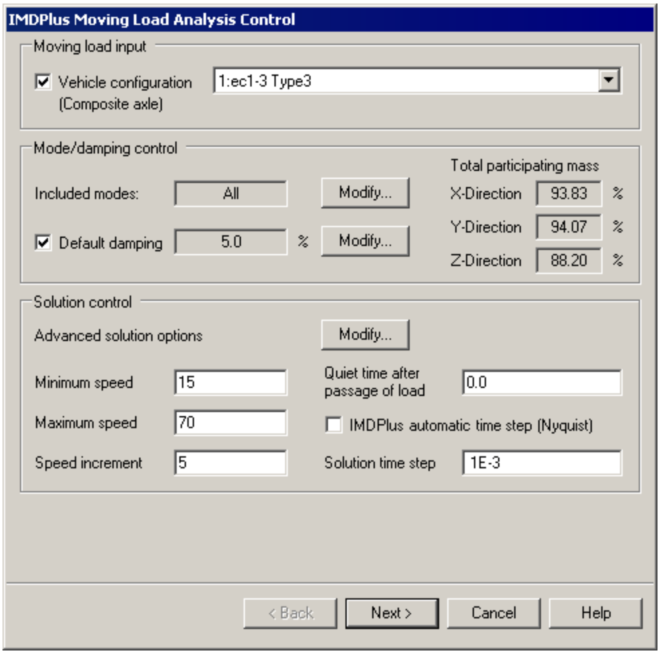
Utilities
IMDPlus >
Moving Load >
Moving Load Analysis...

- Open the IMDPlus Moving Load Analysis Control dialog through the menu or click on the button in the toolbar.

- Click on the **Vehicle Configuration (Composite axle)** option since we are using a unit load axle definition.

- Ensure the Vehicle configuration contains **1:ec1-3 Type3**

- Ensure that **Included modes** is set to **All**. If this is not the case, click on the **Modify…** button and ensure the **All modes** option is selected.



- Ensure **Default damping** of **5.0** is selected. If a different damping is displayed, click on the **Modify…** button and set the **Default damping** to **5.0**

- In the Solution control 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.

- Enter the **Minimum Speed** as **15**, the **Maximum Speed** as **70** and the **Speed Increment** as **5**

- Deselect the **IMDPlus determining time step (Nyquist)** option so we can specify the required time step

- Enter the **Solution time step** as **1E-3**

- Click the **Next** button to proceed. When prompted about significant missing total mass choose **Yes** to continue. For this analysis we are only interested in the contributions of modes of vibration up to and including 35Hz which means that we are not going to achieve the 90% total mass target.

The information entered above will analyse the passage of a Eurocode ENV 1991-3 Type 3 train (with axle details as held in the file ec1-3 Type3.xls) across the bridge for a speed range of 15 m/s to 70 m/s in increments of 5 m/s (or 54 kph to 252 kph in increments of 18 kph). The quiet time allows for the decay of the response of the bridge after the train has passed across and the solution time step forces the time step to be used in the analyses.

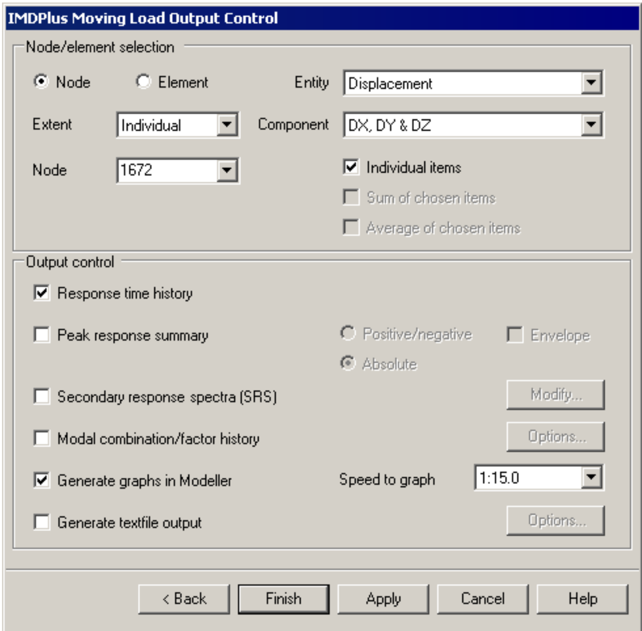
Displaying Individual Node and Element Results for Moving Load Analysis

The IMDPlus Output Control dialog will appear. This controls the results output for the model.

Displacement and Acceleration Graphs

The response of the mid-span of the bridge for the range of speeds selected will be investigated. Initially we will look at the displacements of the mid-span for a single speed of 15 m/s (or 54 kph).

Enter the following information into the output control dialog:

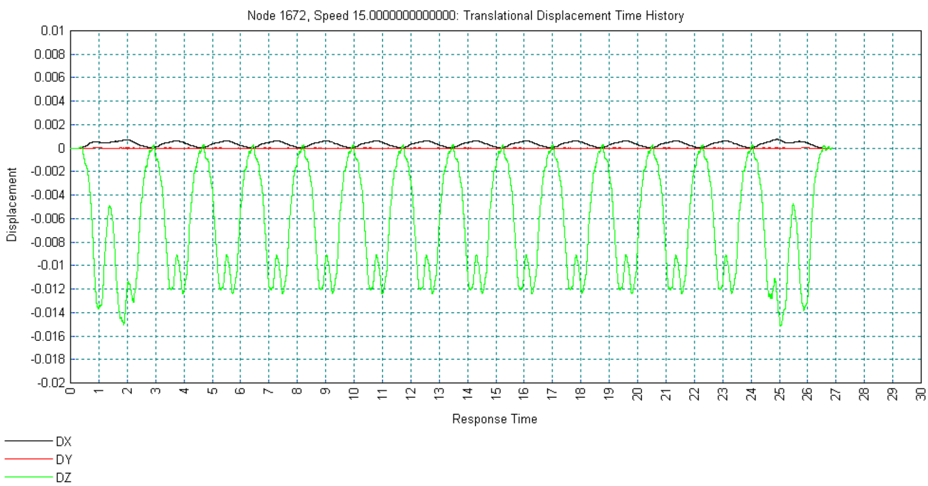


- Choose **Node** and select **Extent** as **Individual**
- Enter **Node** number **1672** (This is the node in the centre of the bridge deck at the mid-span of the bridge)
- Select **Displacement** results of **DX, DY & DZ**
- Ensure **Individual items** is selected. **Sum of chosen items** and **Average of chosen items** will not be available as an individual node is being processed.
- Ensure **Response time history** is selected.
- Ensure **Generate graphs in Modeller** is selected.

- Set the **Speed to graph** as **1:15.0** which indicates that the first speed of 15 m/s is being processed
- Click the **Apply** to proceed.

Note. Clicking the Apply button instead of the Finish button keeps the IMDPlus Control Dialog accessible for subsequent graph plotting.

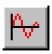
The IMDPlus analysis will now run.



- When the graph has been displayed, close the graph

The vertical acceleration response of the mid-span for a single speed of 15 m/s (or 54 kph) will now be investigated.

If the IMDPlus toolbar has been enabled,

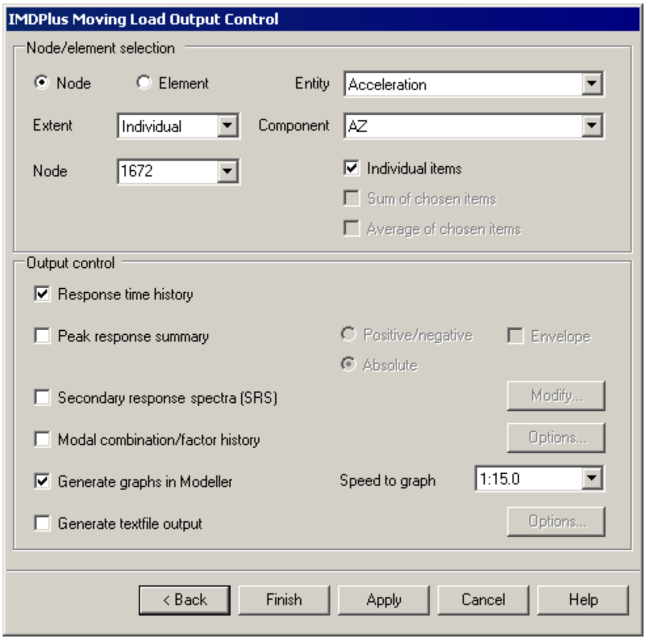
- Click on the  button in the toolbar to open the IMDPlus Output Control dialog

If the IMDPlus toolbar is not enabled,

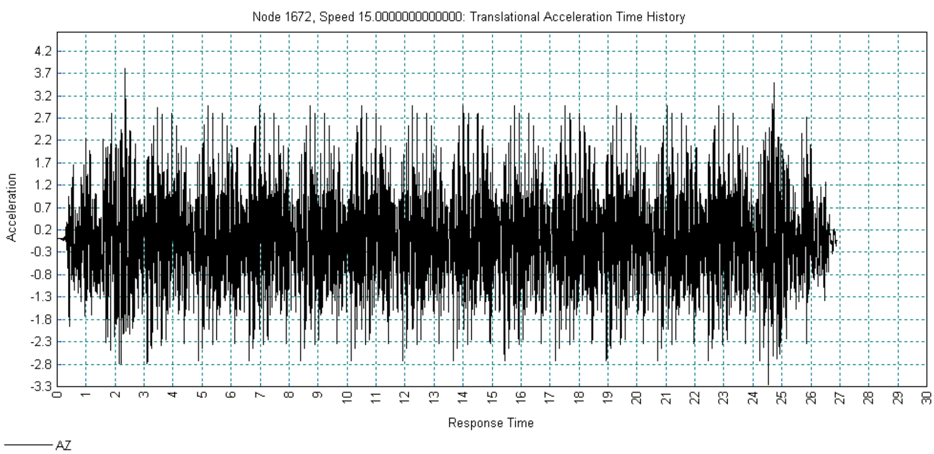
Utilities
IMDPlus >
Moving Load >
Moving Load Analysis...

- Open the IMDPlus Moving Load Analysis Control dialog through the menu and click **Next** > to keep the existing analysis control settings and open the IMDPlus Output Control dialog.

- On the IMD Output Control dialog select **Acceleration** results of **AZ** for the vertical component only
- Click the **Apply** button and not the Finish button to proceed.



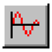
Note. IMDPlus does not need to be rerun since the acceleration results were computed at the same time as the displacement results.



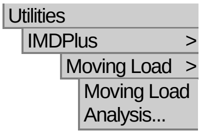
- When the graph has been displayed, close the graph

Previously we have investigated the displacement and acceleration response of the mid-span of the bridge deck for a single train speed. We will now look at the peak positive and negative vertical displacement and acceleration responses of the mid-span over the speed range of 15 m/s to 70 m/s as specified previously in the moving load analysis control dialog.

If the IMDPlus toolbar has been enabled,

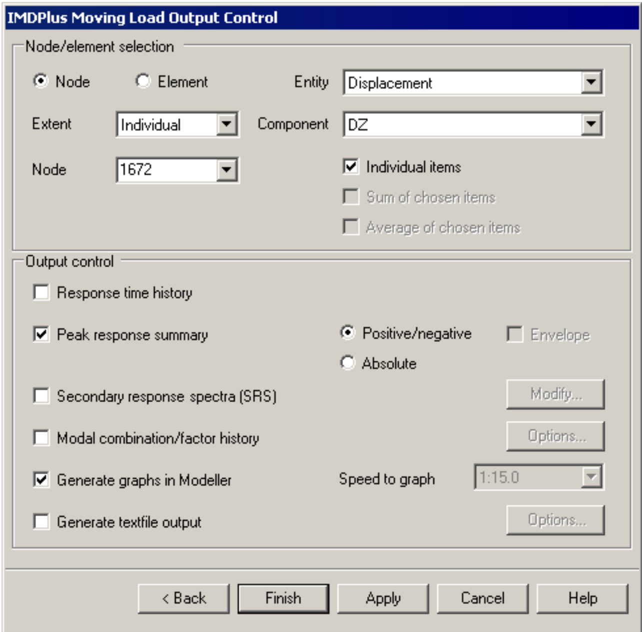
- Click on the  button in the toolbar to open the IMDPlus Output Control dialog

If the IMDPlus toolbar is not enabled,

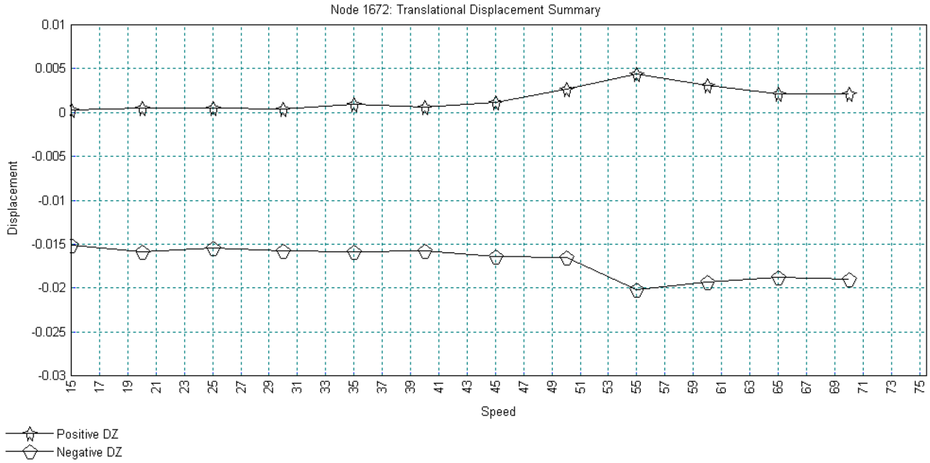


- Open the IMDPlus Moving Load Analysis Control dialog through the menu and click **Next >** to keep the existing analysis control settings and open the IMDPlus Output Control dialog.

- On the IMDPlus Output Control dialog select **Displacement** results of **DZ** for the vertical component only.
- Deselect the **Response time history** option
- Select the **Peak response summary** option and select **Positive/negative**
- Click the **Apply** button to proceed.

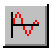


The IMDPlus analysis will now run.

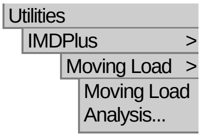


- When the graph has been displayed, close the graph

If the IMDPlus toolbar has been enabled,

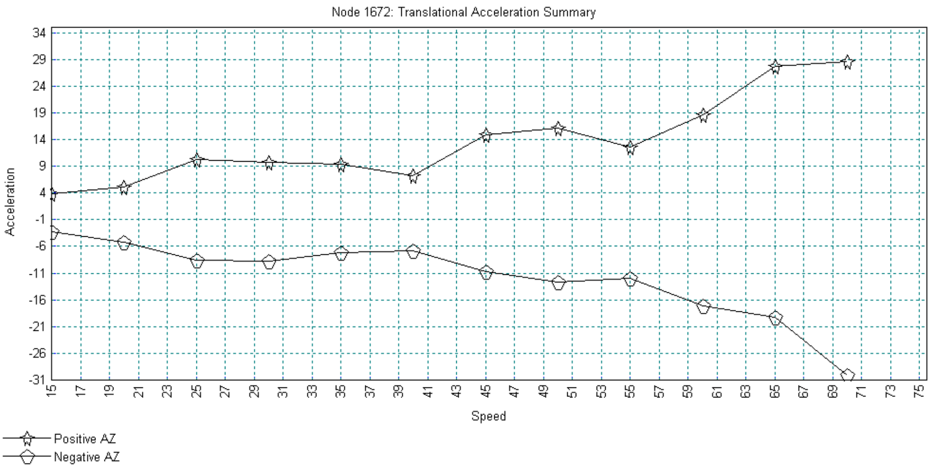
- Click on the  button in the toolbar to open the IMDPlus Output Control dialog

If the IMDPlus toolbar is not enabled,



- Open the IMDPlus Moving Load Analysis Control dialog through the menu and click **Next** > to keep the existing analysis control settings and open the IMDPlus Output Control dialog.

- Select **Acceleration** results of **AZ** for the vertical component only
- Click the **Apply** button to display a graph of acceleration versus speed at the mid-span.

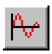


- When the graph has been displayed, close the graph

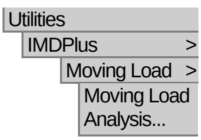
Stress Resultant Graphs for Shells

In the previous section the displacement and acceleration responses at the mid-span of the bridge were investigated. We will now look at the stress resultants in the web of one of the main girders. Results for element number 1193 in the centre of the web of the nearside girder will be investigated.

If the IMDPlus toolbar has been enabled,

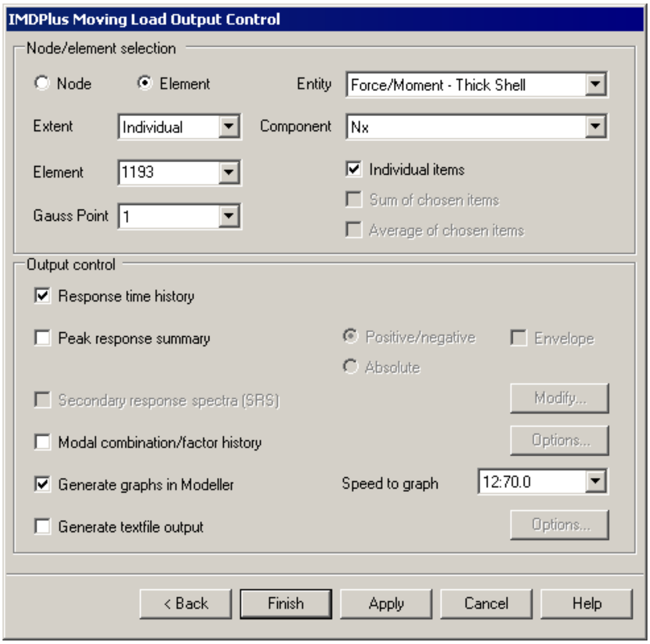
- Click on the  button in the toolbar to open the IMDPlus Output Control dialog

If the IMDPlus toolbar is not enabled,



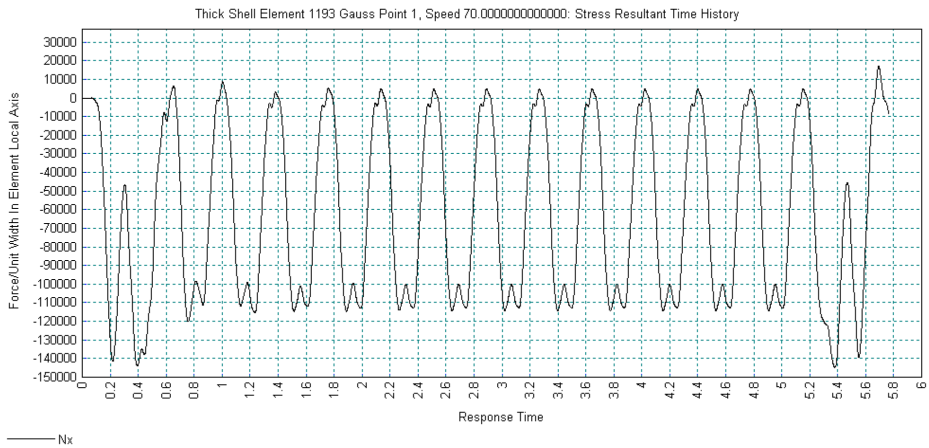
- Open the IMDPlus Moving Load Analysis Control dialog through the menu and click **Next >** to keep the existing analysis control settings and open the IMDPlus Output Control dialog.

Enter the following information into the IMDPlus Output Control dialog.



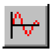
- Select **Element** and ensure **Extent** remains set as **Individual**
- Enter **Element** number **1193** and set **Gauss Point** to **1**
- Select **Force/Moment – Thick Shell** results of **Nx** which is the stress resultant in the longitudinal bridge direction.
- Ensure **Individual items** is selected. **Sum of chosen items** and **Average of chosen items** will not be available as an individual element is being processed.
- Ensure the **Response time history** option is selected.
- Deselect the **Peak response summary** option.
- Set the **Speed to graph** as **12:70.0** which indicates that the twelfth and last speed of 70 m/s is being processed.
- Click the **Apply** button to display a graph of Nx versus time for gauss point 1 of element 1193.

The IMDPlus analysis will now run and a graph of Nx versus time for gauss point 1 of element 1193 will be displayed.

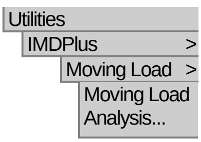


- When the graph has been displayed, close the graph

If the IMDPlus toolbar has been enabled,

- Click on the  button in the toolbar to open the IMDPlus Output Control dialog

If the IMDPlus toolbar is not enabled,

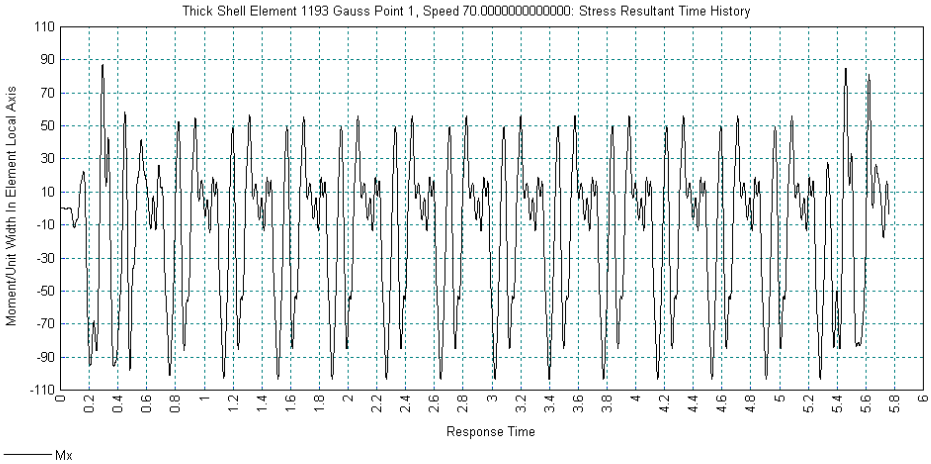


- Open the IMDPlus Moving Load Analysis Control dialog through the menu and click **Next** > to keep the existing analysis control settings and open the IMDPlus Output Control dialog.

- Select **Force/Moment – Thick Shell** results of **Mx** which is the moment in the along bridge direction

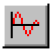
- Click the **Apply** button to proceed.

After a short pause a graph of Mx versus time for gauss point 1 of element 1193 will be created.

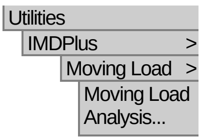


- When the graph has been displayed, close the graph

If the IMDPlus toolbar has been enabled,

- Click on the  button in the toolbar to open the IMDPlus Output Control dialog

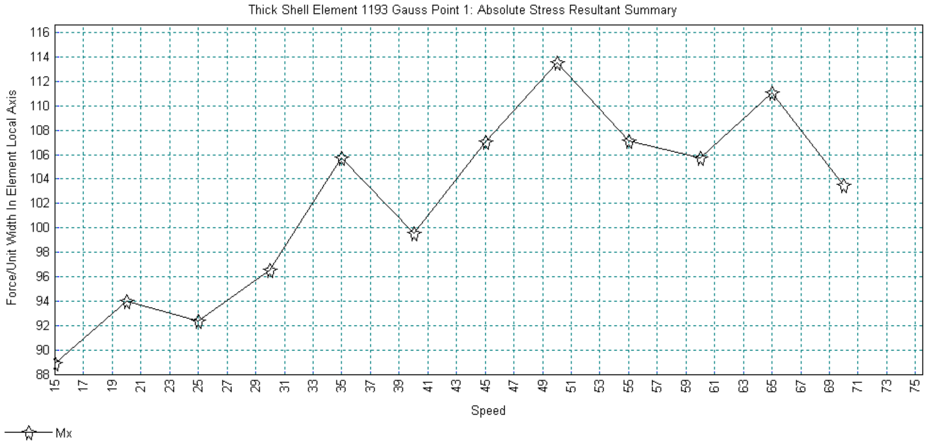
If the IMDPlus toolbar is not enabled,



- Open the IMDPlus Moving Load Analysis Control dialog through the menu and click **Next** > to keep the existing analysis control settings and open the IMDPlus Output Control dialog.

- Deselect the **Response time history** option
- Select the **Peak response summary** option and select **Absolute**
- Click the **Finish** button.
- Click **Yes** when asked whether to free up disk space by deleting the temporary files created by IMDPlus.

A graph of absolute peak Mx versus speed for gauss point 1 of element 1193 will be generated.



- Close the graph window.

Displaying Results for a Selection of Nodes and Elements

In the preceding sections the analyses utilised individual nodes and elements of the structure. In the following sections chosen sets of nodes and elements will be used in order to investigate the following results for the bridge structure:

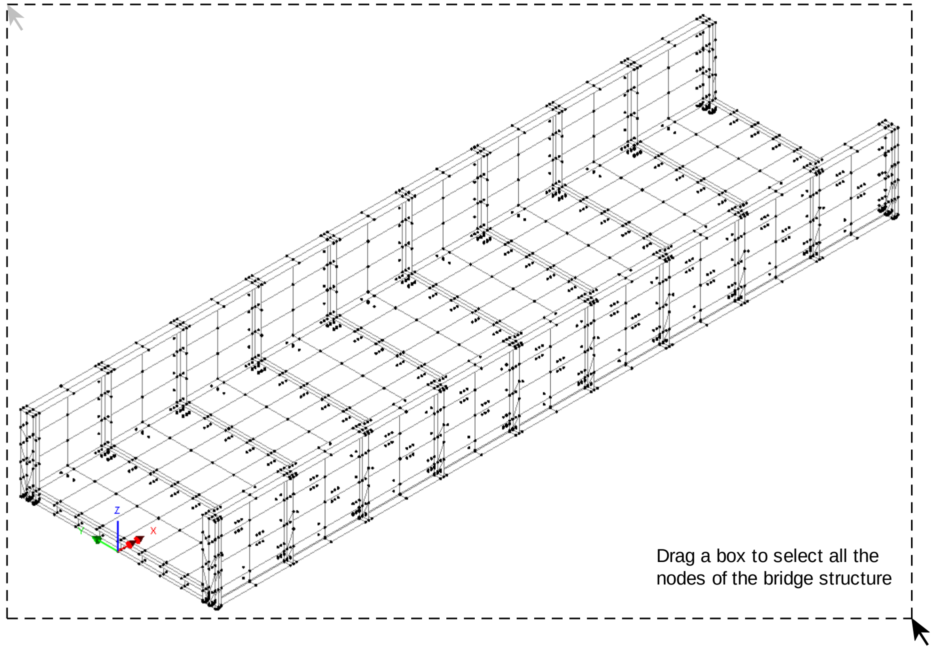
- ❑ Averaged and peak displacements and dynamic amplification factors
- ❑ Total summed reactions and peak reactions
- ❑ Averaged and peak stresses in a side panel

Averaged and Peak Displacements and Dynamic Amplification Factors for the Bridge Structure

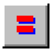
In order to examine the averaged and peak displacements and dynamic amplification factors, a selection of nodes needs to be created that contains all of the nodes of the bridge structure.

- Turn off the **Geometry** layer in the Treeview.
- In the Treeview click the right-hand mouse button on the group name **Bridge_Structure** and select the **Set as Only Visible** option from the drop-down menu.

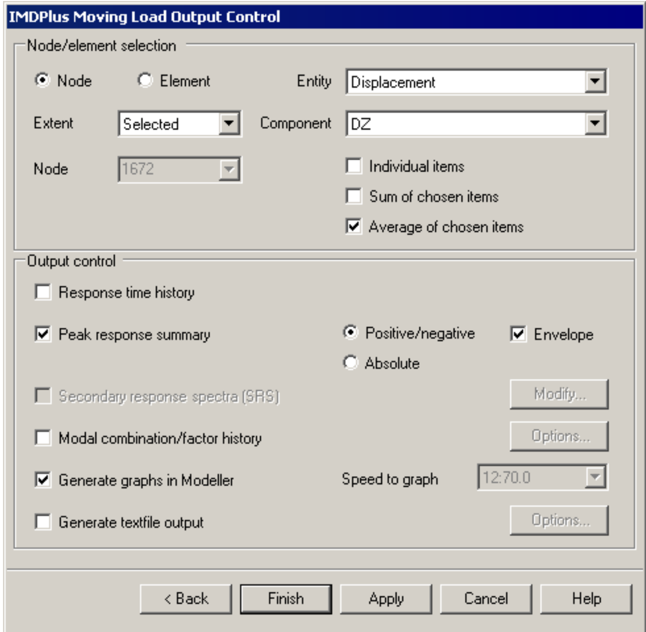
Using Select Nodes drag a box to select all of the 1665 nodes of the bridge structure, as shown in the following figure.



Utilities
IMDPlus >
Moving Load >
Moving Load Analysis...

- Open the IMDPlus Moving Load Analysis Control dialog through the menu or click on the  button in the toolbar.
- On the IMDPlus Moving Load 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.



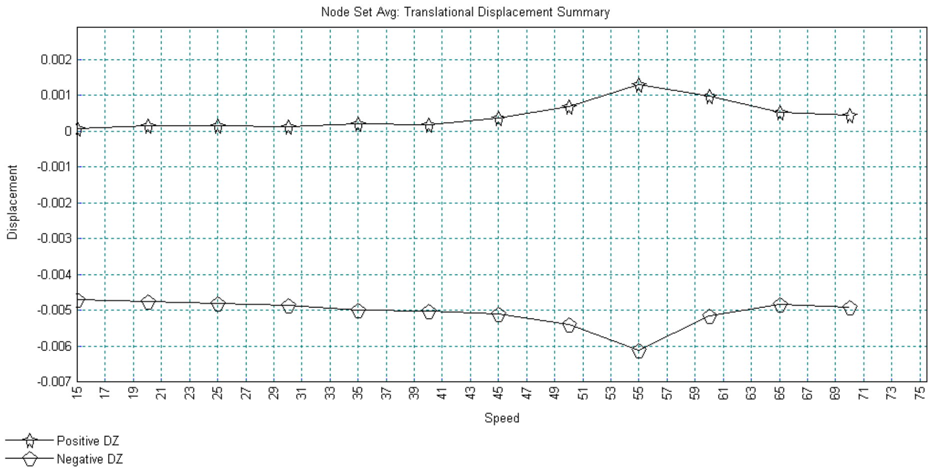
- Select **Node** and change **Extent** to **Selected** using the drop-down list. This chooses all 1665 nodes of the bridge structure that were selected in the preceding step.
- Select result entity **Displacement** and component **DZ**
- Deselect **Individual items** and select **Average of chosen items** as only the averaged results for the selected nodes are going to be investigated.
- Ensure **Peak response summary** remains selected, then select **Positive/negative** and **Envelope**
- Ensure **Generate graphs in Modeller** remains selected
- Click the **Apply** button to perform the analysis. Click **Yes** when asked whether you want to process all of the selected nodes.

The IMDPlus analysis will now run. On modern computers this will take under 15 minutes.

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

Note. **Average of chosen items** obtains results by first summing the results from the selected nodes. A simple average of this summed result is obtained to give the averaged node time histories from which the peak average results are obtained.

A graph of peak positive and negative results for the averaged vertical (DZ) displacements, for all of the speeds analysed, 15m/s to 70m/s, is displayed as shown in the following figure. It can be seen that the largest peaks occur at a speed of 55m/s, with positive and negative peak values of 0.0013m and -0.0061m respectively.



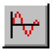
An envelope of the peak displacements and rotations for the selected nodes is displayed in Notepad in file peak_dspSet1.sum. By investigating all of the speeds analysed it can be seen that the individual peak vertical (DZ) displacements also occur at a moving load speed of 55m/s. These are shown in the following figure and both positive and negative peak Z-displacements occur at node 1624 with respective values of 0.0044m at time 4.656 seconds and -0.0204m at time 6.826 seconds.

Note. The envelope of the peak results is obtained in IMDPlus by examining the individual results from all of the nodes in the bridge structure. This enables the locations of the nodes with maximum and minimum results to be quickly identified.

```
#
#
# +--------------------------+
# | Speed = 55.0000000000000 |
# +--------------------------+
#
# Entity                     Time              Value                Node
Peak_Pos_DX           6.807000000     0.3177041183E-002         1028
Peak_Neg_DX           6.815000000    -0.1450325103E-002         2392
Peak_Pos_DY           6.830000000     0.1432152932E-001         1617
Peak_Neg_DY           6.834000000    -0.1436008318E-001         1696
Peak_Pos_DZ           4.656000000     0.4432708343E-002         1624
Peak_Neg_DZ           6.826000000    -0.2040436034E-001         1624
Peak_Pos_THX          6.837000000     0.1278003970E-001         1694
Peak_Neg_THX          6.837000000    -0.1272787014E-001         1673
Peak_Pos_THY          6.974000000     0.5659962656E-002          950
Peak_Neg_THY          0.5750000000   -0.6138300538E-002         2285
Peak_Pos_THZ          0.4680000000    0.4499018200E-002         1861
Peak_Neg_THZ          7.150000000    -0.4332889472E-002         2277
```

- Close the Notepad application and graph windows.

If the IMDPlus toolbar has been enabled,

- Click on the  button in the toolbar to open the IMDPlus Output Control dialog

If the IMDPlus toolbar is not enabled,

Utilities
IMDPlus >
Moving Load >
Moving Load Analysis...

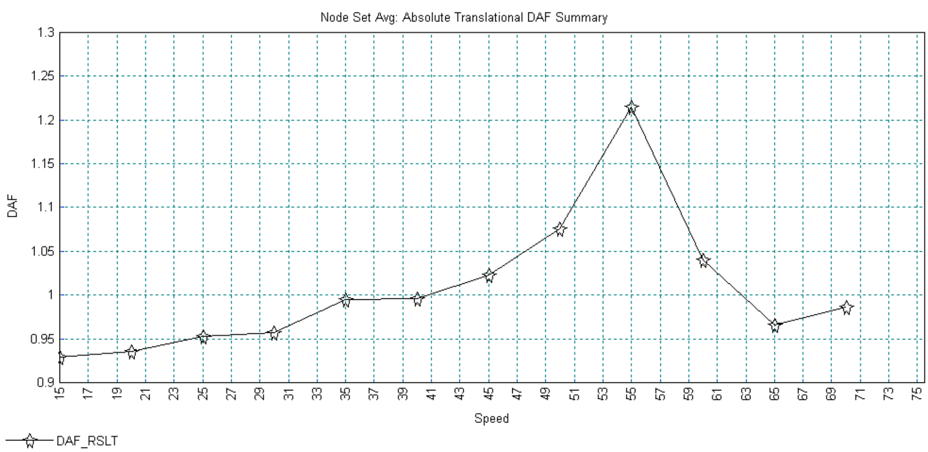
- Open the IMDPlus Moving Load Analysis Control dialog through the menu and click **Next** > to keep the existing analysis control settings and open the IMDPlus Output Control dialog.

Enter the following information into the IMDPlus Output Control dialog.

- Select result entity **Dynamic Amplification Factor** and component **DAF_RSLT**
- Ensure **Peak response summary** and **Envelope** remain selected and select **Absolute**
- Click the **Finish** button. Click **Yes** when asked whether you want to process all of the selected nodes.
- Click **Yes** when asked whether to free up disk space by deleting the temporary files created by IMDPlus.

A graph showing the absolute peak of the average resultant dynamic amplification factor against speed is displayed, as shown in the following figure. This shows that a maximum averaged DAF of 1.215 occurs at speed 55m/s.

Note. Only averaged dynamic amplification factor results can be obtained for a set of nodes as summed results are meaningless for this result entity.



An envelope of the peak dynamic amplification factors for the selected nodes is displayed in Notepad in file Abs_peak_dafSet1.sum is. These results indicate that the individual peak resultant DAF occurs at a speed of 60m/s. A value of 1.71 is calculated at a solution time of 0.529 seconds at node 34.

```
#
#
# +-------------------------+
# | Speed = 60.0000000000000 |
# +-------------------------+
#
# Entity                   Time           Value           Node
Abs_Peak_DAF_DX         6.284000000    1.965105582        1057
Abs_Peak_DAF_DY         0.6690000000   3.252813452        1582
Abs_Peak_DAF_DZ         6.367000000    1.578463432        2412
Abs_Peak_DAF_THX        0.6710000000   4.548161937        1624
Abs_Peak_DAF_THY        6.351000000    1.652702690        1296
Abs_Peak_DAF_THZ        6.423000000    2.023591492        1112
Abs_Peak_RSLT           0.5290000000   1.712660195          34
```

- Close the Notepad application and graph window.

Total and Peak Reactions for the Bridge Structure

The total sum of the reactions and the peak reactions that act on the bridge structure for each of the moving load speeds will now be investigated. The set of nodes from the preceding section could be utilised for this purpose. However, in order to improve the efficiency of the IMDPlus solution, the chosen selection of nodes will be reduced so that it only includes the geometric features that contain the supported nodes of the bridge structure.

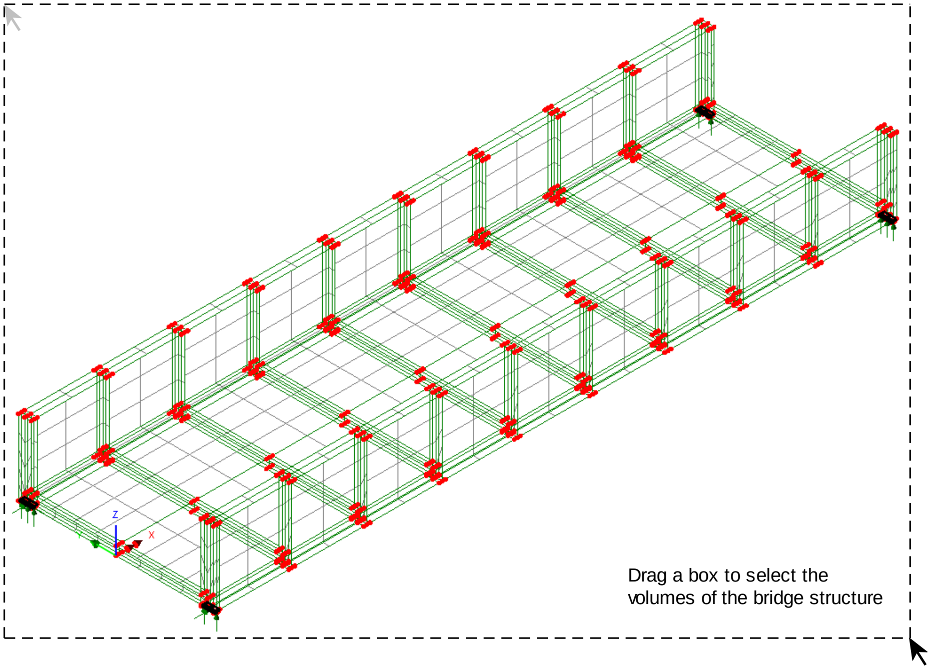
- Click the left-hand mouse button in a blank part of the graphics window to remove the node selection created in the preceding section.

In the LUSAS graphics window click on the Supports on/off button to show the supported nodes in the model.

- In the Treeview click the right-hand mouse button on the group name **Bridge_Structure** and select the **Set as Only Visible** option from the drop-down menu.

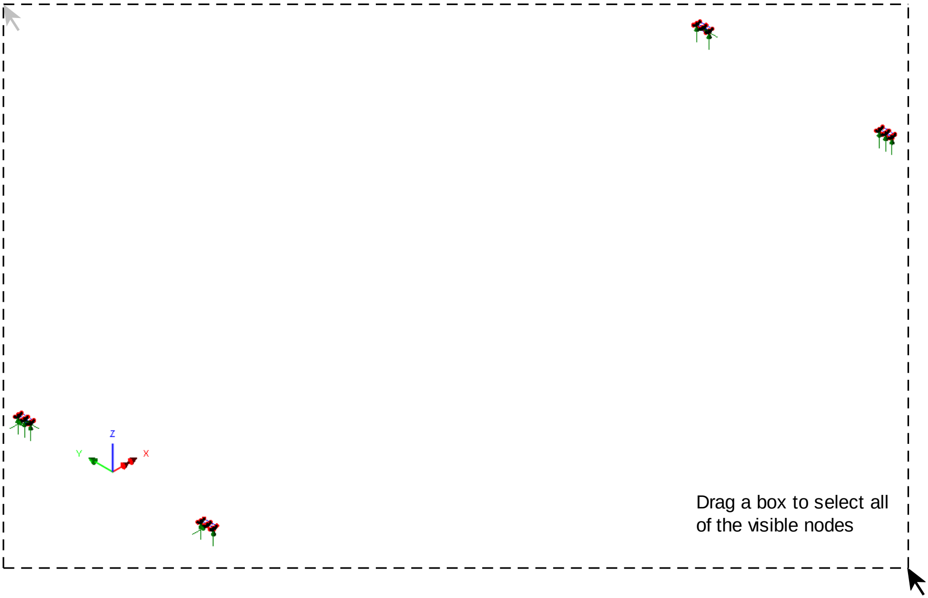
- Turn on the **Geometry** layer in the Treeview.

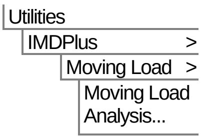
Using Select Volumes drag a box around the entire bridge structure to select the volumes immediately above the bridge supports, as shown in the following figure.



- Click the right-hand mouse button in a blank part of the graphics window and select the **Keep as Only Visible** option. Only the selected volumes will remain visible.

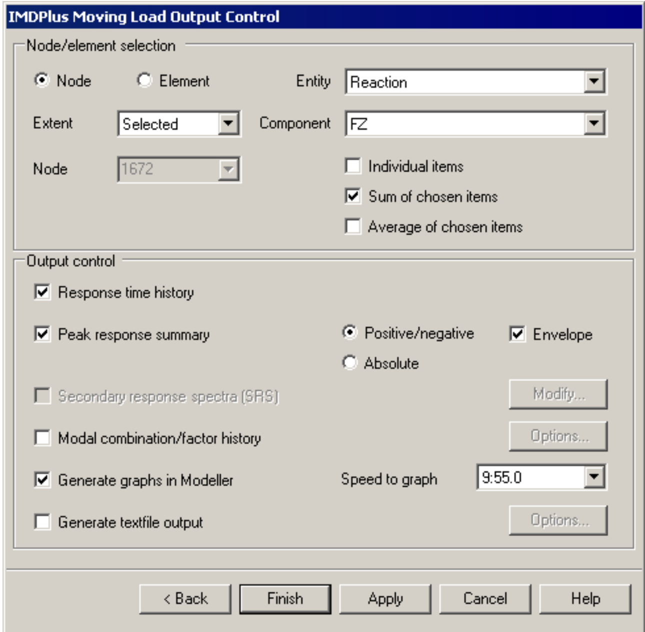
 Using Select Nodes drag a box to select the 120 visible nodes of the bridge structure, as shown in the following figure.





- Open the IMDPlus Moving Load Analysis Control dialog through the menu or click on the  button in the toolbar.
- On the IMDPlus Moving Load 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.

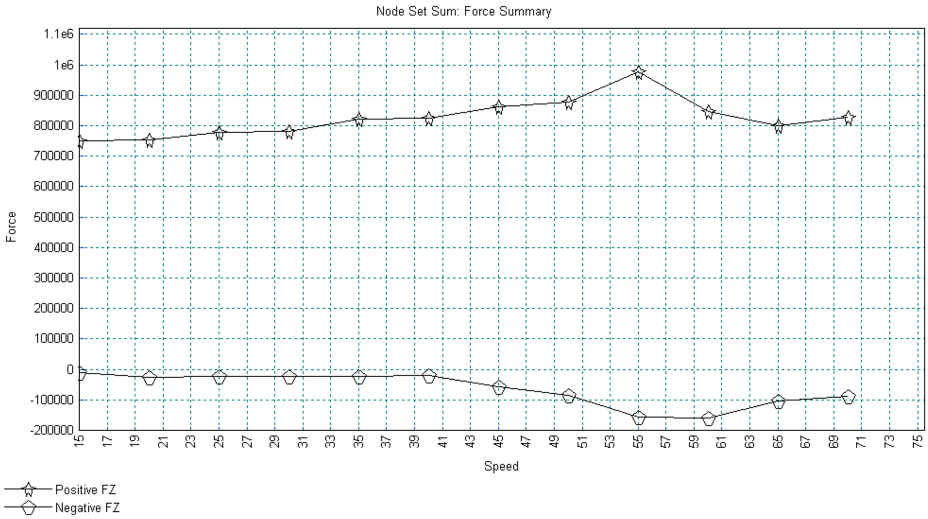


- Select **Node** and ensure **Extent** remains set to **Selected**. This chooses the 120 visible nodes of the bridge structure that were selected in the preceding step.
- Select result entity **Reaction** and component **FZ**
- Deselect **Average of chosen items** and select **Sum of chosen items**. Ensure **Individual items** remains deselected.
- Select **Response time history**
- Ensure **Peak response summary** and **Envelope** remains selected and select **Positive/negative**
- Ensure **Generate graphs in Modeller** remains selected.
- Set the **Speed to graph** as **9:55.0** which indicates that the ninth and speed of 55 m/s is being processed
- Click the **Finish** button to perform the analysis of the node set. Click **Yes** when asked whether you want to process all of the selected nodes.
- Click **Yes** when asked whether to free up disk space by deleting the temporary files created by IMDPlus.

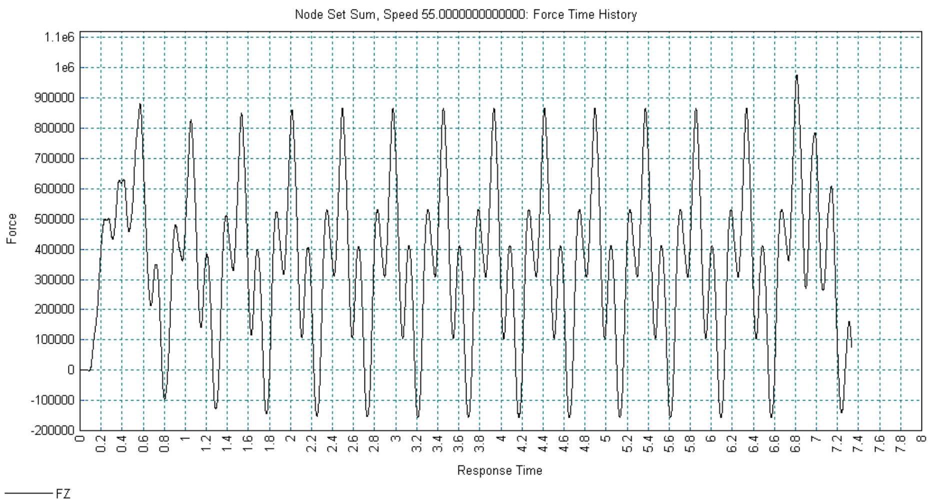
Note. **Sum of chosen items** adds together the results from the selected nodes to give the summed nodal time histories from which the peaks of the summed results are obtained.

Note. **Average of chosen items** obtains results by first summing the results from the selected nodes. A simple average of this summed result is obtained to give the averaged node time histories from which the peak average results are obtained.

The summed nodal results that give the total vertical reactions acting on the bridge structure are displayed. The graph of the peak positive and negative summed reaction in the Z-direction for all of the speeds analysed is shown in the following figure. Examining the results for each speed shows a positive peak reaction of 976.6kN at a train speed of 55m/s and a negative holding down peak reaction of -159.0kN at a speed of 60 m/s.



In addition, the following figure shows the time history of the summed FZ reactions for the analysis speed of 55m/s. For this speed it can be seen that the positive peak value of 976.6kN occurs at an analysis time of 6.81 seconds.



The envelope of the peak reactions from file peak_rctSet1.sum are displayed in a Notepad application. These give a summary of the peak reactions obtained from all of the individual nodes of the bridge structure for each of the moving load speeds analysed. The vertical (FZ) component of interest shows consistently that the maximum positive reaction (from a downward force) occurs at node 82 and the maximum negative reaction (from an upward force) occurs at node 112. Results for speeds of 55m/s and 60m/s are shown in the following figure.

```
#
#
# +--------------------------+
# | Speed = 55.0000000000000 |
# +--------------------------+
#
# Entity                      Time            Value            Node
Peak_Pos_FX            6.756000000      52731.24971               9
Peak_Neg_FX            4.576000000     -18589.20194               9
Peak_Pos_FY            7.019000000      22881.92033              69
Peak_Neg_FY            6.994000000     -19169.69107               9
Peak_Pos_FZ     ==>   0.5760000000 ==>  259862.3328    ==>       82
Peak_Neg_FZ            6.943000000     -135238.8224             112
Peak_Pos_MX                    N/A              N/A             N/A
Peak_Neg_MX                    N/A              N/A             N/A
Peak_Pos_MY                    N/A              N/A             N/A
Peak_Neg_MY                    N/A              N/A             N/A
Peak_Pos_MZ                    N/A              N/A             N/A
Peak_Neg_MZ                    N/A              N/A             N/A
#
#
# +--------------------------+
# | Speed = 60.0000000000000 |
# +--------------------------+
#
# Entity                      Time            Value            Node
Peak_Pos_FX            6.196000000      48130.35311               9
Peak_Neg_FX            6.722000000     -17547.90507              32
Peak_Pos_FY           0.6700000000      23428.89095               9
Peak_Neg_FY           0.7000000000     -22936.61174               9
Peak_Pos_FZ           0.5340000000      257957.9082              82
Peak_Neg_FZ     ==>    6.367000000 ==> -145133.7444    ==>      112
Peak_Pos_MX                    N/A              N/A             N/A
Peak_Neg_MX                    N/A              N/A             N/A
Peak_Pos_MY                    N/A              N/A             N/A
Peak_Neg_MY                    N/A              N/A             N/A
Peak_Pos_MZ                    N/A              N/A             N/A
Peak_Neg_MZ                    N/A              N/A             N/A
```

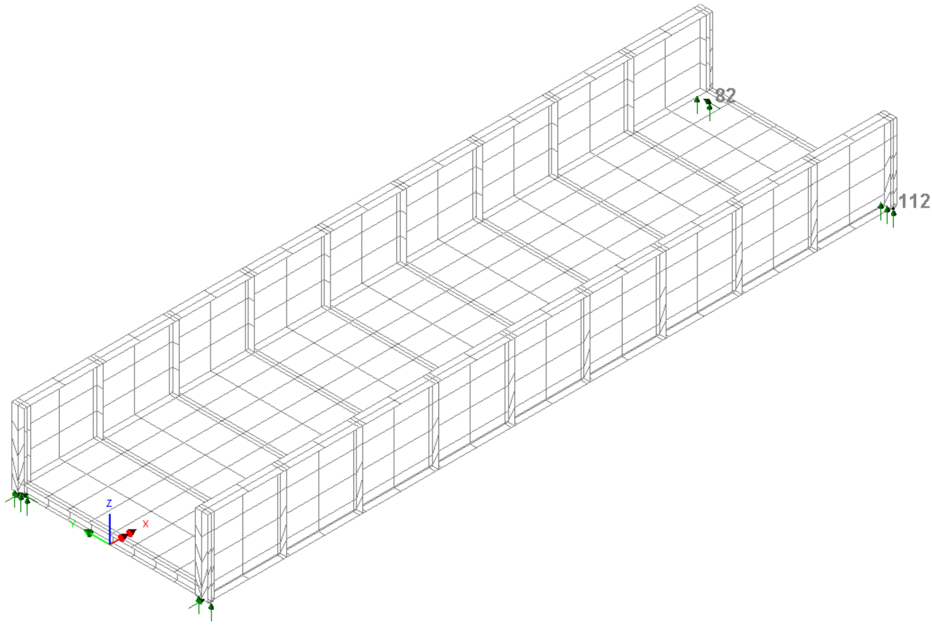
The maximum positive reaction of 259.9kN occurs at a time 0.576 seconds for speed 55m/s and the maximum negative reaction of -145.1kN occurs at a time 6.367 seconds for speed 60m/s.

- Close the Notepad application and graph windows.

The nodes 82 and 112 representing two of the vertical supports for the bearings of the bridge will now be located in the model.

- Click the left-hand mouse button in a blank part of the graphics window to remove the node selection created in the preceding section.

- In the Treeview click the right-hand mouse button on the group name **Bridge_Structure** and select the **Set as Only Visible** option from the drop-down menu.
- Turn off the **Geometry** layer in the Treeview.
- With no features selected click the right-hand mouse button in a blank part of the graphics window and select the **Labels** option to add the labels to the Treeview.
- On the Properties dialog select the **Name/Node** and **Label selected items only** check boxes.
- Click the **Font...** button and select Font: **Arial,** Font style: **Bold,** Size: **20** and click **OK.**
- Click the **OK** button to accept all other settings. This will turn on the layer in the graphics window.
- With no features selected click the right-hand mouse button in a blank part of the graphics window and select the **Advanced Selection...** option.
- Select **Type and Name**, select **Node** from the drop-down list and enter node number **82**
- Select **Add to selection** and click the **Apply** button to accept all other settings. Node 82 will be highlighted in the graphics window.
- Enter node number **112** and click the **OK** button. Node 112 will also be displayed in the graphics window, as shown in the following figure.



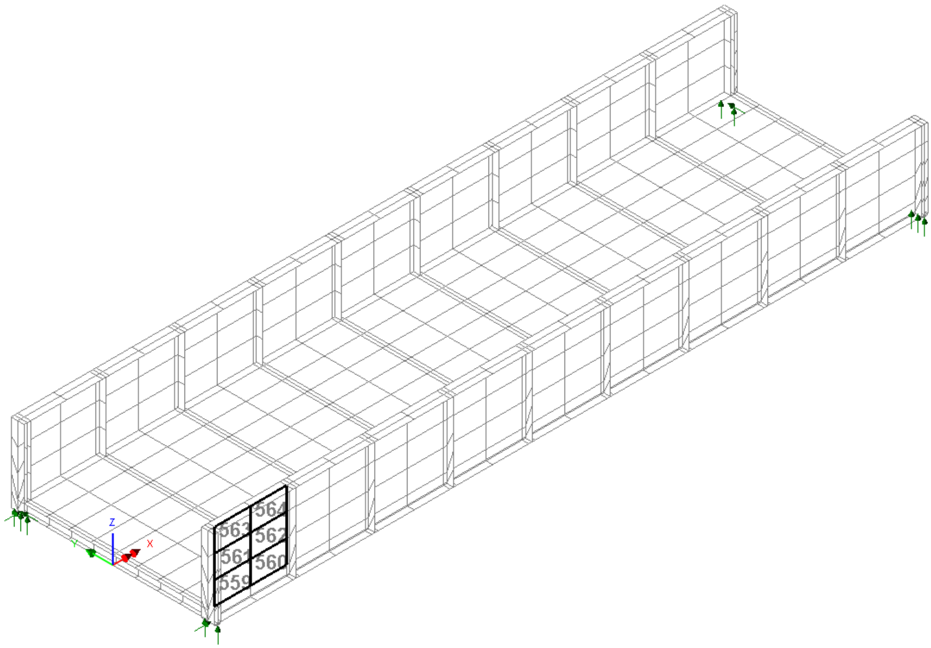
- Click the left-hand mouse button in a blank part of the graphics window to remove the node selection.

Average and Peak Stresses in a Side Panel of the Bridge Structure

Finally, the average peak and total peak in-plane shear stresses in a side panel of the bridge structure will be investigated.

- Double-click on the **Labels** layer name in the Treeview.
- On the Properties dialog de-select the **Name/Node** check box and select the **Name/Element** check box. Click the **OK** button.

Select the six elements that form the side panel of the bridge structure shown in the following figure.

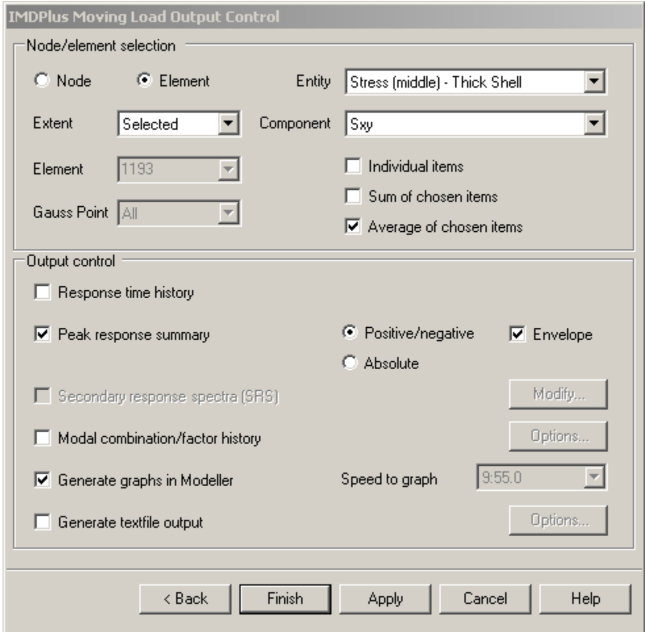


Note. Only elements of the same type, for example, thick beam elements, continuum elements or thick shell elements can be used in a set of IMDPlus elements. However, the element set may contain elements with different numbers of Gauss points or nodes.

Utilities
IMDPlus >
Moving Load >
Moving Load Analysis...

- Open the IMDPlus Moving Load Analysis Control dialog through the menu or click on the button in the toolbar.
- On the IMDPlus Moving Load 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 **Element** and set **Extent** to **Selected** from the drop-down list. This chooses the 6 elements of the bridge structure that were selected in the preceding step.

- Select result entity **Stress (middle) – Thick Shell** and component **Sxy**

- Deselect **Sum of chosen items** and select **Average of chosen items**. Ensure **Individual Output** remains deselected.

- Deselect **Response time history**

- Ensure **Peak response summary**, **Positive/negative** and **Envelope** all remain selected.

- Ensure **Generate graphs in Modeller** remains selected.

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

- Click the **Finish** button to perform the analysis of the element set. Click **Yes** when asked whether you want to process all of the selected elements.

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

The graph shown in the following figure is displayed. It shows the peak positive and negative, average middle shear stress, Sxy, for the side panel of the bridge structure. The largest positive and negative peaks occur at a speed of 55m/s with values of 2.05Mpa and -10.2MPa respectively.

Note. **Average of chosen items** obtains the element results by first summing the results from all of the Gauss points of the selected thick shell elements. A simple average of these summed results is obtained to give the average element response time histories from which the peak average results are obtained.

An envelope of the peak element stresses for each of the moving load speeds is displayed in Notepad in file peak_stressSet1.sum. This gives a summary of the peak stresses and stress resultants obtained by examining all of the individual Gauss point results of the selected elements. The results for speed 55m/s are shown in the following figure. The maximum positive and negative middle in-plane shear (Sxy) stress occurs at Gauss point 2 of element 560. Values of 2.69MPa at time 7.224 seconds and -13.6MPa at time 6.986 seconds are observed.

```
#
#
# +-------------------------+
# | Speed = 55.0000000000000 |
# +-------------------------+
#
# Entity                    Time            Value          Element    GP
Peak_Pos_Top_Sx          6.804000000      6178832.782        560       1
Peak_Neg_Top_Sx          6.981000000     -17724493.52        564       4
Peak_Pos_Top_Sy          6.812000000      3238750.965        560       1
Peak_Neg_Top_Sy          6.797000000     -5028203.725        559       4
Peak_Pos_Top_Sxy         7.224000000      2745090.347        560       3
Peak_Neg_Top_Sxy         6.981000000     -11816861.23        560       2
Peak_Pos_Top_Syz         6.812000000     0.1396187750E-023   560       2
Peak_Neg_Top_Syz         6.803000000    -0.1716039574E-023   559       2
Peak_Pos_Top_Szx         6.906000000     0.7440377432E-024   560       1
Peak_Neg_Top_Szx         6.824000000    -0.1110811094E-023   559       2
Peak_Pos_Mid_Sx          6.985000000      5728221.278        560       2
Peak_Neg_Mid_Sx          6.986000000     -16401763.30        564       4
Peak_Pos_Mid_Sy          6.980000000      3243441.794        560       2
Peak_Neg_Mid_Sy          6.796000000     -5224766.419        559       4
Peak_Pos_Mid_Sxy         7.224000000      2689334.309        560       2
Peak_Neg_Mid_Sxy         6.986000000     -13610114.52        560       2
Peak_Pos_Mid_Syz         0.1350000000     236128.4546        560       2
Peak_Neg_Mid_Syz         6.914000000     -226209.2933        562       2
Peak_Pos_Mid_Szx         6.814000000      341032.6462        560       1
Peak_Neg_Mid_Szx         6.913000000     -322802.1144        560       1
Peak_Pos_Bot_Sx          6.980000000      6474320.454        560       2
Peak_Neg_Bot_Sx          6.803000000     -16534341.68        564       4
Peak_Pos_Bot_Sy          0.3340000000     4167596.897        562       2
Peak_Neg_Bot_Sy          6.980000000     -5722445.497        559       4
Peak_Pos_Bot_Sxy         7.223000000      3133893.260        559       2
Peak_Neg_Bot_Sxy         6.801000000     -16182311.93        560       2
Peak_Pos_Bot_Syz         6.801000000     0.2213546367E-023   560       2
Peak_Neg_Bot_Syz         0.3410000000   -0.2010308949E-023   559       2
Peak_Pos_Bot_Szx         0.5380000000    0.4221235670E-023   559       2
Peak_Neg_Bot_Szx         6.840000000    -0.1863288508E-023   560       1
Peak_Pos_Nx              6.985000000      72748.41024        560       2
Peak_Neg_Nx              6.986000000     -208302.3939        564       4
Peak_Pos_Ny              6.980000000      41191.71079        560       2
Peak_Neg_Ny              6.796000000     -66354.53352        559       4
Peak_Pos_Nxy             7.224000000      34154.54573        560       2
Peak_Neg_Nxy             6.986000000     -172848.4544        560       2
Peak_Pos_Mx              7.011000000      91.80102937        562       1
Peak_Neg_Mx              6.909000000     -138.1163838        562       1
Peak_Pos_My              6.873000000      77.03755272        562       4
Peak_Neg_My              6.907000000     -100.7634410        562       1
Peak_Pos_Mxy             6.974000000      88.52683013        559       2
Peak_Neg_Mxy             6.906000000     -35.90092843        560       2
Peak_Pos_Sx              6.814000000      2887.409738        560       1
Peak_Neg_Sx              6.913000000     -2733.057902        560       1
Peak_Pos_Sy              0.1350000000     1999.220915        560       2
Peak_Neg_Sy              6.914000000     -1915.238683        562       2
```

- Close the Notepad application and graph windows.

Save the model

File
  Save…

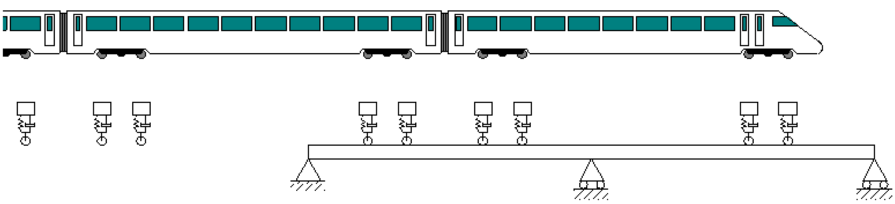
Save the model.

This completes the example.

High Speed Train Modelling through Sprung Masses

For software product(s):	Any Plus version
With product option(s):	IMDPlus
Note: The example exceeds the limits of the LUSAS Teaching and Training Version and requires approximately 4 GB of disk space.	

Description



This example examines the response of a two-span continuous beam to the passage of a Chinese CRH high speed train modelled as sprung masses based on the paper by Wang *et al*, Latin Journal of Solids and Structures 7 (2010).

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

Objectives

The output requirements of the analysis are:

- Vertical mid-span deflection response of the two spans for a train speed of 250 kph
- Peak mid-span vertical deflections in the Y-direction for train speeds between 25 kph and 600 kph in 25 kph increments

- ❑ Vertical mid-span deflection response of the two spans for a train speed of 300 kph, the critical velocity exciting the first resonant frequency of the continuous beam
- ❑ Displacement time histories of the body and wheel of the first and last spring-mass systems for a train speed of 300 kph and acceleration time histories of the bodies
- ❑ Peak acceleration time histories of all locomotive and passenger bodies over the speed range of 25 kph to 600 kph

Keywords

Moving mass, time domain, response, interactive modal dynamics, IMDPlus, eigenvalue.

Associated Files

- ❑ **CRH Train.lmd** Attribute library that contains the CRH train definition for the example.
- ❑ **CRH_Wheelset_Positions.xls** Microsoft Excel spreadsheet that contains the wheel positions and spring-mass system identifiers of the CRH train for the example.

Modelling

Running LUSAS Modeller

For details of how to run LUSAS Modeller see the heading ***Running LUSAS Modeller*** in the Examples Manual Introduction.

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 the file name as **IMDPlus CRH High Speed Train**
- Choose User-defined and browse to the **\<LUSAS Installation Folder>\Projects** folder, click on the **Make New Folder** button and enter **IMDPlus CRH Analysis** as the folder name and click the **OK** button
- Enter the title as **IMDPlus Sprung Mass Modelling of CRH High Speed Train**
- Ensure the model units are **N,m,kg,s,C**

- Ensure the timescale units are **Seconds**
- Ensure the analysis type is **Structural**
- Ensure the Startup template is **None**
- Select the **Vertical Y axis** option and click **OK**

Feature Geometry

Geometry
Line >
Coordinates…

Enter coordinates of (**0, 0**), (**20,0**) and (**40, 0**) to define the lines representing the main features of the two-span continuous beam.

This completes the geometry for the beam that the train will pass across. A further line must also be defined which will be used by IMDPlus to define the path that the train passes along since this line cannot form part of the physical structure.

Geometry
Line >
Coordinates…

Enter coordinates of (**0, 10**) and (**40, 10**) to define the line above the two-span continuous beam that represents the path of the train (line is defined above for ease of selecting, the train loading will be projected from this line vertically onto the structure).

Meshing

The Line features representing the two-span continuous beam are to be meshed using two-dimensional beam elements.

Attributes
Mesh >
Line...

- Select **Thick beam, 2 dimensional, Linear** elements.
- Set the number of divisions to be **20**
- Enter the attribute name as **Thick Beam** then click **OK**
- Select the lines defining the two-span continuous beam.
- Drag and drop the mesh attribute **Thick Beam** from the Treeview onto the selected model Lines.

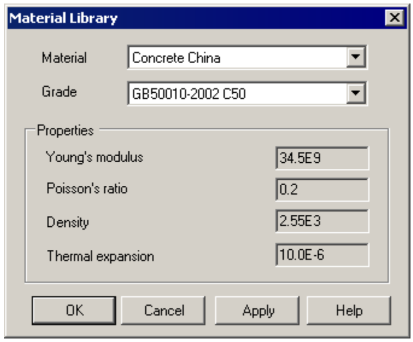
40 BEAM elements will be created along the lines.

Material Properties

The Line features representing the two-span continuous beam are to be assigned C50 concrete in accordance with the China National Standard: Code for Design of Concrete Structures.

Attributes
Material >
Material Library …

- Select **Concrete China** material.
- Select **GB50010-2002 C50** grade.
- Click **OK** to define the material.
- With the lines defining the two-span beam still selected, drag and drop the material attribute **Iso1 (Concrete China GB50010-2002 C50)** from the Treeview onto the selected model Lines.

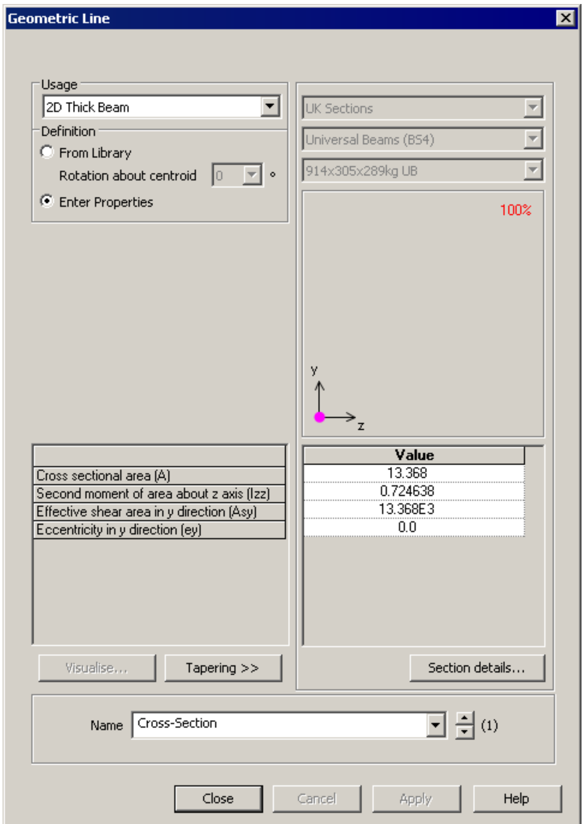


Geometric Properties

The Line features representing the two-span continuous beam are to be assigned with a custom section equivalent to the properties used in the paper by Wang *et al*. In this paper the flexural rigidity EI value of 2.5E10 N/m^2 and mass per unit length of 3.4088E4 kg/m equate to the properties entered below.

Attributes
Geometric >
Line …

- Select the usage as **2D Thick Beam**
- Ensure the definition is set to **Enter Properties**
- Set the Cross sectional area (A) as **13.368**
- Set the Second moment of area about z axis (Izz) as **0.724638**
- Set the Effective shear area in y direction (Asy) as **13.368E3**
- Ensure the Eccentricity in y direction (ey) is set to **0.0**
- Enter the attribute name as **Cross-Section** and click **OK**
- With the lines defining the two-span beam still selected, drag and drop



the geometric attribute **Cross-Section** from the  Treeview onto the selected model Lines.

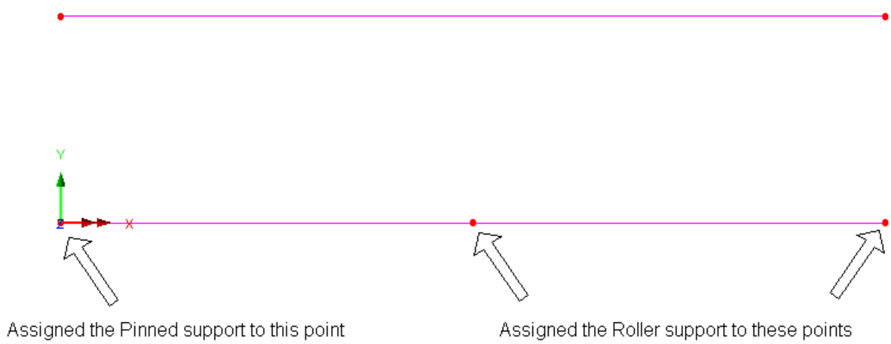
Note. The effective shear area has been set to 1000 times the area of the section to effectively remove shear deformation from the calculations of the beams. The paper uses Bernoulli-Euler beams which do not account for shear deformations.

Supports

The continuous beam is supported with a pinned support at the left-hand end and is supported by roller supports at the middle and right-hand end of the beam.

Attributes
Support…

- Set the Translation in X as **Fixed**
- Set the Translation in Y as **Fixed**
- Enter the attribute name as **Pinned** and click **Apply**
- Set the Translation in X as **Free**
- Leave the Translation in Y as **Fixed**
- Enter the attribute name as **Roller** and click **OK**
- Assign the Pinned support to the left-hand point, and the Roller support to the other two points on the model as indicated below, clicking OK to accept the defaults when the Assign Support dialog appears.

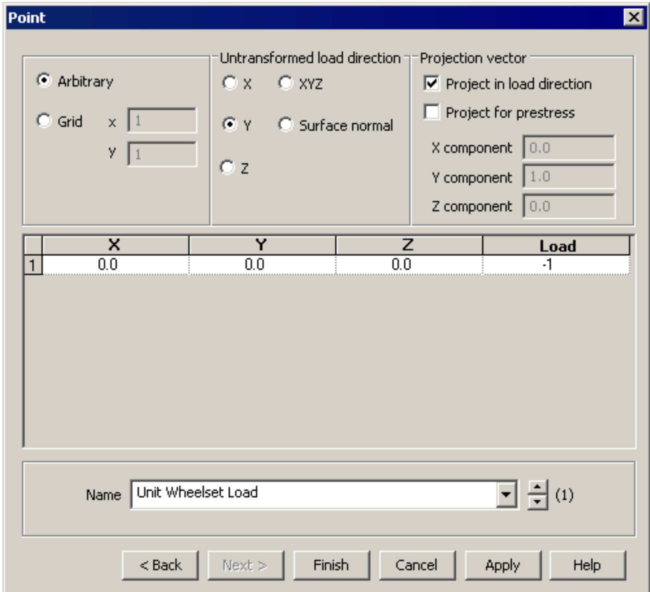


Loading

To allow the IMDPlus moving mass analysis to be carried out a unit load representing the action of the spring-mass system on the beam needs to be included in the model so this can be passed across the beam to calculate the modal forces.

- Click the left-hand mouse button in a blank part of the graphics window to remove all items from the selection.

Attributes
Loading…



- On the Structural Loading dialog select the **Point** type from the Discrete loading section and click the **Next** button
- Ensure **Arbitrary** is selected
- Ensure that the Untransformed load direction is set to **Y**
- Ensure the **Project in load direction** is selected
- Enter a single load into the grid with the **X** set to **0.0** , **Y** set to **0.0** , **Z** set to **0.0** and the **Load** set to **-1** so it acts downwards in the global Y axis direction
- Enter the attribute name as **Unit Wheelset Load** and click **Finish**

Search Area

When the discrete load representing the single unit wheelset is passed across the structure it is best practice to target the action of this load to the model using a search area.

Attributes
Search Area…

- Set the attribute name as **Continuous Beam** and click **OK**
- Select the lines defining the two-span continuous beam.

- Drag and drop the search area attribute **Continuous Beam** from the Treeview onto the selected model Lines.
- Click the left-hand mouse button in a blank part of the graphics window to remove all items from the selection.

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:

- Ensure the Solution is set to **Frequency**
- Set the Eigenvalues required to **Range**
- Ensure the Range specified as is set to **Frequency**
- Set the Minimum frequency as **0**
- Set the Maximum frequency as **50**
- Set the Number of eigenvalues as **0** to solve for all modes in the range
- 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 analyses.

- 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 Treeview. Eigenvalue 1 is set to be active by default.

Checking the Frequencies

The Wang *et al* paper lists only the first two frequencies but the other vertical frequencies can be calculated from the equations in the paper. These frequencies of the vertical modes below 50 Hz of the Bernoulli-Euler beams are listed below:

f_1 = 3.363 Hz, f_2 = 5.255 Hz, f_3 = 13.452 Hz, f_4 = 17.025 Hz, f_5 = 30.267 Hz, f_6 = 35.522 Hz

The frequencies obtained from the analysis are obtained from the Print Results Wizard.

Utilities
Print Results Wizard…

- Select **Active** and click **Next**.
- Select Entity **None** and from the Type drop-down menu select **Eigenvalues** and then click **Finish.** The eigenvalue / frequency results will be printed to the Text Output window.

It can be seen that LUSAS has obtained 7 modes of vibration in the range of 0 to 50 Hz. All of the modes of vibration listed above for the Bernoulli-Euler beam are closely matched but with an additional mode of vibration, mode 5, at 22.990 Hz which was not captured in the paper. This will be shown to be a horizontal mode of vibration below which is not considered by Wang *et al*.

MODE	EIGENVALUE	FREQUENCY	ERROR NORM
1	445.894	3.36075	0.100680E-10
2	1087.93	5.24953	0.414174E-11
3	7105.85	13.4162	0.828539E-12
4	11376.9	16.9759	0.470560E-12
5	20866.7	22.9904	0.843402E-12
6	35737.0	30.0870	0.219183E-09
7	49189.1	35.2984	0.466222E-06

Checking the Mass Participation Factor

Note. In order to carry out a successful IMDPlus analysis you should ensure that a significant proportion of the total mass has been accounted for in the analysis. Ideally this requires checking that around 90% of the total mass has been achieved in the global directions. If less than 90% has been achieved no further modes need be included if the modes of vibration omitted cannot be excited by the dynamic input or a significant proportion of the structure is restrained by support in these directions and therefore cannot participate in the modes of vibration. The acceptability of the included modes of vibration will vary from analysis to analysis but failure to check that a significant proportion of the total mass has been accounted for may lead to important modes being missed and subsequent errors in the analysis results.

Note. The mode shapes should also be checked as a mode can have a low mass participation factor and still be significant in the analysis. This often occurs with 'symmetrical modes' where equal mass is moving in opposite directions in the mode shape. This is the case for the first mode in this analysis which is a simple sinusoid shape with one span moving up with the other of equal length and mass moving down. This mode has a zero mass participation factor in the Y-direction but is an important mode in the analysis (see the resonance of the beam to the train speed later in the example). A certain amount of engineering judgement is therefore required for the assessment of the importance of modes of vibration and this should be used in addition to the simple checks on the sum of the mass participation included in the analysis.

Utilities
Print Results Wizard…

- Select **Active** and click **Next**.
- Select Entity **None** and from the Type drop-down menu select **Sum mass participation factors** and then click **Finish.** The Sum Mass Participation Factors results will be printed to the Text Output window.

In can be seen that the 90% value has almost been achieved in both the X- and Y-directions and is discussed in the note below.

MODE	SUM MASS X	SUM MASS Y	SUM MASS Z
1	0.175061E-36	0.439835E-26	0.00000
2	0.135185E-34	0.738434	0.00000
3	0.178691E-30	0.738434	0.00000
4	0.338629E-29	0.745294	0.00000
5	0.810569	0.745294	0.00000
6	0.810569	0.745294	0.00000
7	0.810569	0.856006	0.00000

Note. In this analysis we are only including the first 7 modes of vibration with frequencies up to and including 50 Hz. These 7 modes include the 6 vertical modes considered by Wang *et al* and also an additional horizontal mode, mode 5, with 81.06% mass participation in the X-direction. If more than 7 modes of vibration were included then the sum mass participation would approach the 90% in all directions (solving from 0 to 100 Hz will give 12 modes of vibration which will give 90.06% in the X-direction and 89.97% in the Y-direction but these higher modes may not be excited by the passage of the trainset and may not significantly contribute to the vibration response of the structure).

- Close the text windows by selecting the close button in the top right hand corner of the windows.
- Use the maximise button to increase the size of the graphics window.

Note. To simplify the use of IMDPlus for the rest of the example it is recommended that the IMDPlus toolbar is enabled in LUSAS Modeller. To do this select the **View > Toolbars...** menu command and enable the **IMDPlus** option in the list and click **Close**.

- If the IMDPlus toolbar has been enabled, click on the  Moving Mass analysis button in the toolbar to select a moving mass analysis and enable the toolbutton shortcuts to the dialogs.

Defining the CRH Trainset

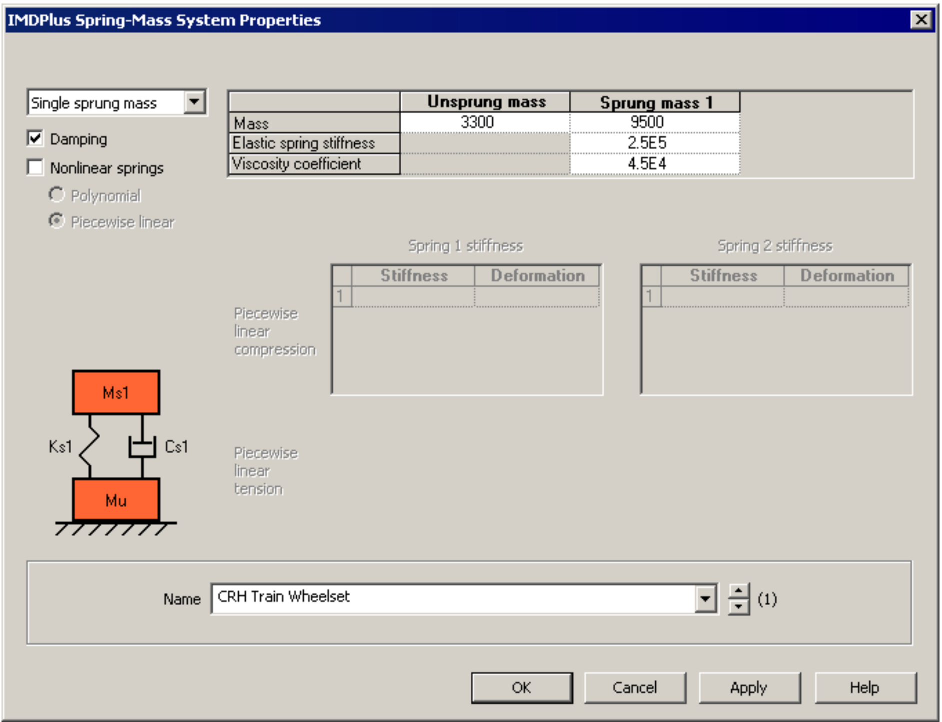
Before the analysis can be performed we need to define the sprung mass representation of the CRH Chinese high speed trainset in LUSAS Modeller. This is a two stage process where

1. the individual spring-mass definitions for the wheelsets / bogies / bodies need to be defined,
2. the complete trainset configuration is defined.

The spring-mass representation from the paper will be used for the analysis which includes only a single spring-mass definition that is used for all of the wheelsets of the trainset.

Utilities
IMDPlus >
Moving Mass >
Spring Mass Properties...

- Open the IMDPlus Spring-Mass System Properties dialog through the menu or click on the  button in the toolbar. The wheelsets / bodies of the CRH train will be defined with the wheelset as the unsprung mass and the body as the first spring-mass.



- Select **Single sprung mass** from the drop-down list.
- Ensure **Damping** is selected.
- Enter the Mass of the Unsprung mass as **3300**
- Enter the Mass of Sprung mass 1 as **9500**
- Enter the Elastic spring stiffness of Sprung mass 1 as **2.5E5**
- Enter the Viscosity coefficient of Sprung mass 1 as **4.5E4**
- Enter the Name as **CRH Train Wheelset**
- Click the **OK** button.

The Chinese CRH trainset in the paper consists of a total of eight cars. Using the dimensions shown in the paper results in the following spring-mass systems to define the total trainset.

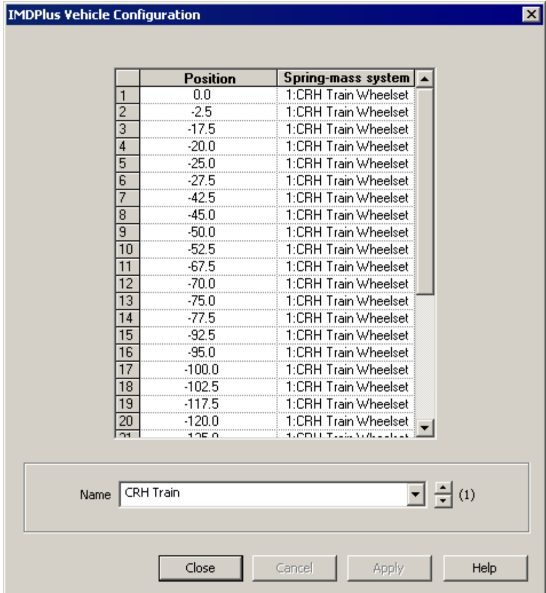
Car	Distance from Previous Axle (m)	Distance from front of train (m)
Locomotive	0	0.0
Locomotive	2.5	-2.5
Locomotive	15	-17.5
Locomotive	2.5	-20.0
Passenger 1	5	-25.0
Passenger 1	2.5	-27.5
Passenger 1	15	-42.5
Passenger 1	2.5	-45.0
Passenger 2	5	-50.0
Passenger 2	2.5	-52.5
Passenger 2	15	-67.5
Passenger 2	2.5	-70.0
Passenger 3	5	-75.0
Passenger 3	2.5	-77.5
Passenger 3	15	-92.5
Passenger 3	2.5	-95.0
Passenger 4	5	-100.0
Passenger 4	2.5	-102.5
Passenger 4	15	-117.5
Passenger 4	2.5	-120.0
Passenger 5	5	-125.0
Passenger 5	2.5	-127.5
Passenger 5	15	-142.5
Passenger 5	2.5	-145.0
Passenger 6	5	-150.0
Passenger 6	2.5	-152.5
Passenger 6	15	-167.5
Passenger 6	2.5	-170.0
Locomotive	5	-175.0
Locomotive	2.5	-177.5
Locomotive	15	-192.5
Locomotive	2.5	-195.0

Utilities
IMDPlus >
Moving Mass >
Vehicle Configuration...

- Open the IMDPlus Moving Mass Vehicle Configuration dialog through the menu or click on the  button in the toolbar.

The moving mass vehicle is now defined using the Distance from front of train in the table above and the spring-mass system. To simplify the definition of the vehicle the positions and identifiers of the spring-mass systems are included in a Microsoft Excel spreadsheet so they can be copied and pasted into the dialog.

- Locate the **CRH_Wheelset_Positions.xls** spreadsheet in the **\<LusasInstallationFolder>\Examples\Modeller** folder and open it.
- Select all of the **32** positions and spring-mass system identifiers in the first and second columns and copy these to the clipboard using **Ctrl+C**
- Select both the **Position** and **Spring-mass system** headers of the grid in the dialog and paste these into the grid using **Ctrl+V**
- Enter the Name as **CRH Train**



Once finished the dialog should appear as shown.

- Click **OK** to create the vehicle configuration.

Note. Alternatively each **Position** in the table can be entered manually and the **Spring-mass system** associated with each of the wheelset/body positions can be selected from the drop-down lists containing the available spring-mass system property attributes.

Note. The positions of the wheelsets should be defined as negative values to ensure that the whole trainset passes over the structure. Negative values place the wheelsets behind the front of the train which is at a position of 0. The movement of the front of the train will be calculated from the start of the path to the end of the path with additional time included in the IMDPlus solution to allow all of the wheelsets with negative positions to pass along the path.

If any wheelsets are defined with positive values then these will already be the distance down the path equivalent to the position entered in the trainset definition. Those wheelsets with positive positions may already be over the structure at the start of the IMDPlus analysis if the start of the path has not been defined to account for this. If any positive positions are used then always ensure that the lines defining the beginning of the path allow sufficient additional length in order to model the transition of the whole of the train onto the structure correctly.

Note. Once a trainset or vehicle has been defined it can be saved in a library so it can be imported and used in future analyses without the need to go through all of the definition process above. This is achieved through the **Library Browser** accessed through the **File > Import/Export Model Data...** menu command.

Importing a Vehicle

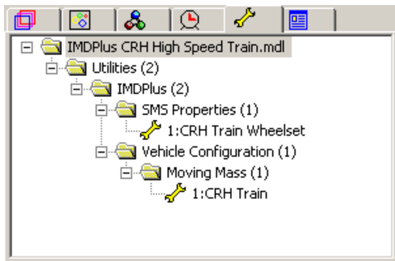
If it proves time consuming or you are unable to obtain the same results shown later in the example the spring-mass system and vehicle definition can be imported from the library provided in the **\<LusasInstallationFolder>\Examples\Modeller** folder.

- **CRH Train.lmd** contains the vehicle library for the example.

File
Import/Export Model Data…

- Select the **Import from library to model** option
- Click on the **Choose file...** button and browse to the **CRH Train.lmd** file in the **\<LusasInstallationFolder>\Examples\Modeller** folder.
- Select the **IMDPlus (2)** entries in the tree
- Click the **OK** button.

After the spring-mass systems and vehicle attributes have been defined they will appear in the Utilities treeview.



To edit the spring-mass system or vehicle attributes simply double-click with the left mouse button or right-click and choose **Edit Attribute...** on the name of the attribute to bring up the original dialog.

Moving Mass Analysis

Moving mass calculations are performed using the IMDPlus (Interactive Modal Dynamics) facility. In order to carry out the moving mass analysis of the train travelling across the structure we need to carry out three stages:

4. Define and set-up the path along which the spring-mass systems will travel using a unit load defined as either a discrete point or patch load. For this example the unit wheelset load has already been defined as a discrete point load called **Unit Wheelset Load** which acts vertically

5. Convert the loading along this path from this unit load into modal forces that are applied in the IMDPlus moving mass analysis, to compute the dynamic effects of the spring-mass systems on the response of the structure

6. Run an IMDPlus moving mass analysis to calculate the response of the structure and the spring-mass systems

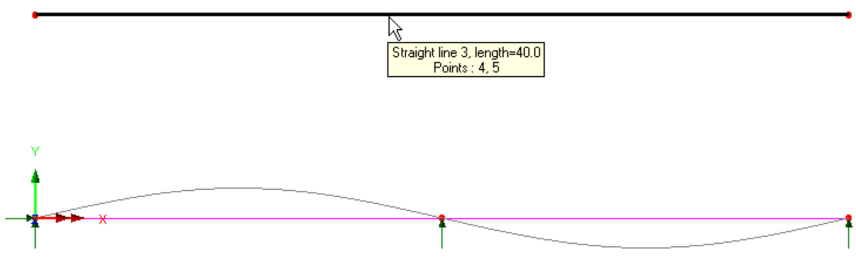
Note. This process is similar to the IMDPlus moving load analysis procedure but with the trainset / vehicle defined by the spring-mass systems rather than constant forces.

Defining the Moving Mass Path

To solve for the passage of the CRH train across the structure the path for the moving mass must be defined. Line 3 (the line representing the path of the moving mass) will be set to be the current selection.

To ensure the whole model is visible:

- In the Treeview click the right-hand mouse button on the group name **IMDPlus CRH High Speed Train.mdl** and select the **Visible** option from the drop-down menu.
- In the Treeview click the right-hand mouse button and choose the **On/Off** option if it has been turned off or alternatively double-click on the **Geometry** layer name and click the **OK** button to accept the default settings if it was turned off.
- Select the **line** shown (line 3)



Note. The path can be built from multiple lines and arcs but these must form a continuous path without branching.

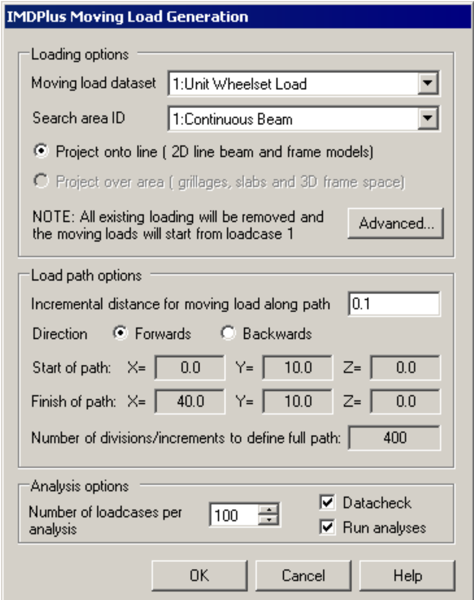
Utilities
IMDPlus >
Moving Mass >
Moving Load Generation...

- Open the IMDPlus Moving Load Generation dialog through the menu or click on the button in the toolbar.

On startup of the IMDPlus Moving Load Generation dialog, all valid discrete loads and search areas will be made available in the loading options along with information about the path defined by the current selection.

In this example a single discrete load called Unit Wheelset Load which defines the unit loading from a single wheelset of the train is present along with a search area that is assigned to the bridge structure.

- Ensure **1:Unit Wheelset Load** is selected from the Moving load dataset list.
- Ensure **1:Continuous Beam** is selected from the Search area ID for the assignment of the discrete loading.
- Ensure **Project onto line** is selected (the analysis is 2D so this should be the only option).
- Click on the **Advanced** button to adjust the inclusion of load characteristics. On the Moving Load Advanced Options dialog choose the **Exclude all load** option for loads outside the search area, ensure the load factor is **1.0** and click the **OK** button.



- On the IMDPlus Moving Load Generation dialog set the Incremental distance to **0.1**

Note. Using search areas targets the application of the loading to the required features as described in the Modeller Reference Manual.

Note. By default the incremental distance is set to one tenth of the length of the line along which the load moves.

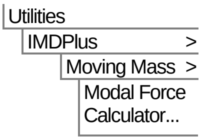
- Leave all other settings as their defaults and click the **OK** button to proceed and choose **Yes** to save the current model.

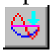
The program will now generate the loading information for the 401 locations of the unit wheelset along the path before returning control back to LUSAS Modeller.

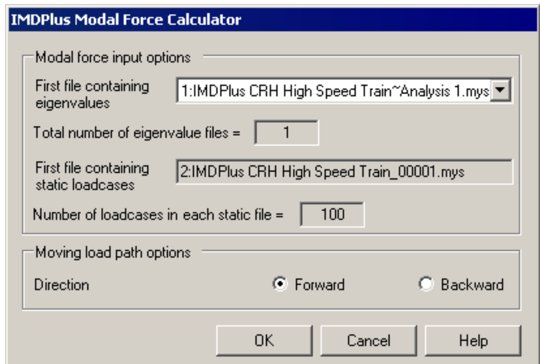
Note. The discrete loading locations defined by this dialog will be tabulated into five datafiles with a maximum of 100 loadcases each and the analyses will be performed automatically. These analyses will use the same file basename as the original model with a numeric indicator appended to it (e.g. _00001, _00002, etc). They are required for the modal force calculation stage.

Generating the Modal Force History for the Moving Mass

In the previous stage the passage of the unit train wheelset across the structure has been defined. The modal forces for the IMDPlus solution now need to be calculated using the Modal Force Calculator.

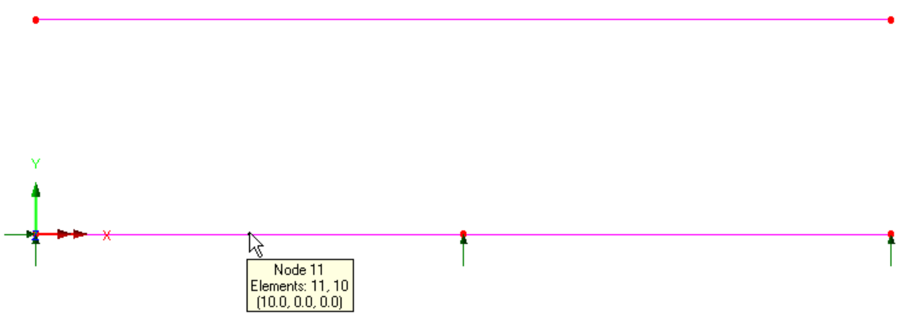


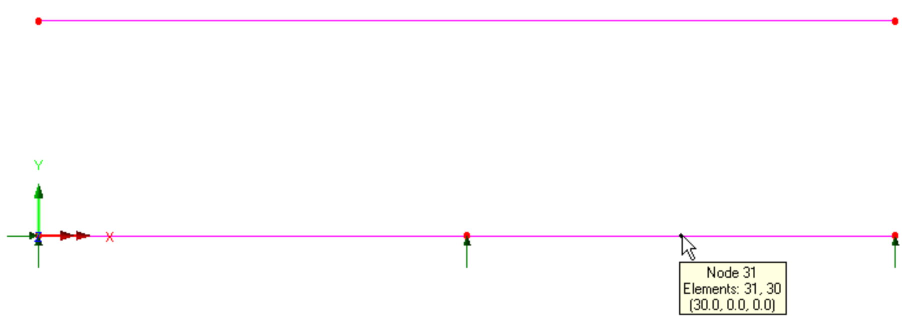
- Open the IMDPlus Modal Force Calculator dialog through the menu or click on the  button in the toolbar.
- Click the **OK** button to accept the default information and proceed.



Identification of Output Nodes

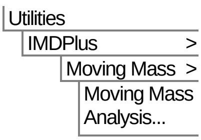
In this example all of the output relates to the mid-spans of the two-span continuous beam model. If the model is identical then these node numbers should be node 11 for the first span and node 31 for the second span as shown below. These nodes will be entered into the IMDPlus dialogs later.

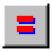


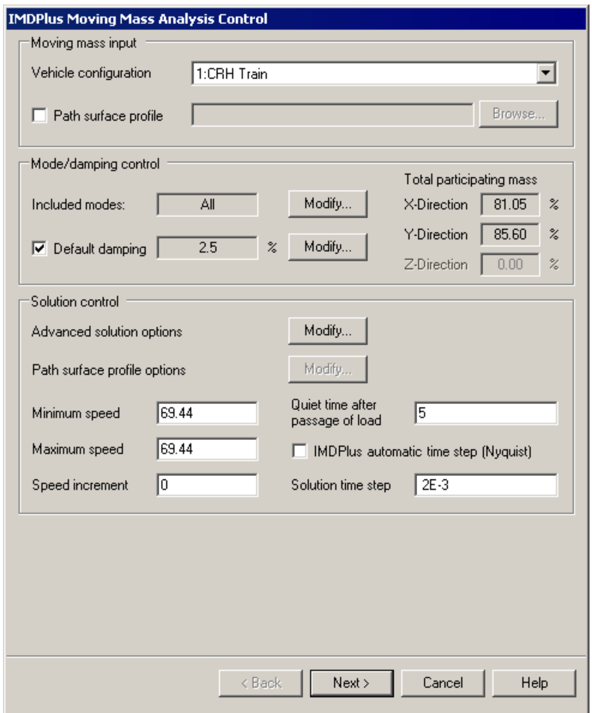


Defining the Moving Mass Parameters

All of the basic moving mass information has now been defined for the IMDPlus analysis. The next stage is to define the included modes, damping and speed parameters.



- Open the IMDPlus Moving Mass Analysis Control dialog through the menu or click on the  button in the toolbar.



- Ensure the Vehicle configuration contains **1:CRH Train**
- Ensure that the **Path surface profile** is not selected.
- Ensure that Included modes is set to **All**. If this is not the case, click on the **Modify…** button and ensure the **All modes** option is selected.
- Click on the **Modify...** button to change the Default damping. Enter **2.5** for the default damping and click the **OK** button.
- In the Solution control 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.

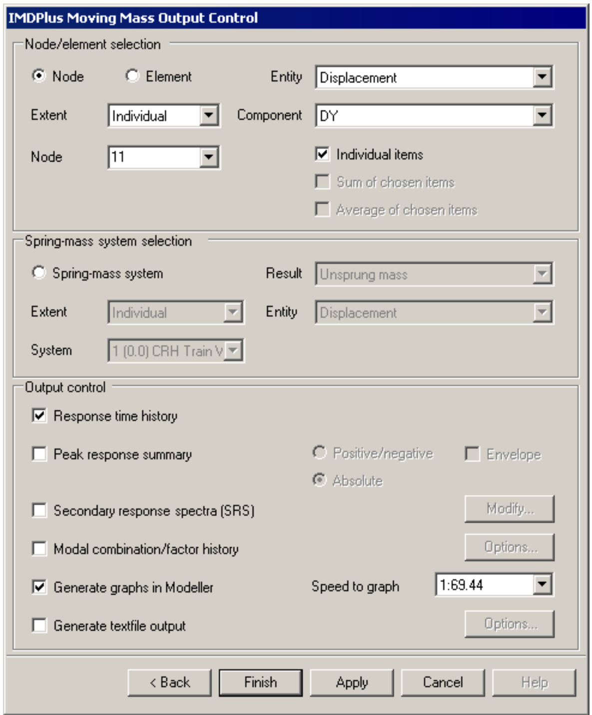
- Enter the Minimum Speed as **69.44**, the Maximum Speed as **69.44** and the Speed Increment as **0** to analyse a single speed of 69.44 m/s which is equivalent to 250 kph.
- Enter the Quiet time after passage of load as **5**
- Deselect the **IMDPlus determining time step (Nyquist)** option so we can specify the required time step.
- Enter the Solution time step as **2E-3**
- Click the **Next** button to proceed. When prompted about missing total mass due to mass participation factors below 90% choose **Yes** to continue. For this analysis we are only interested in the contributions of modes of vibration up to and including the first 7 modes (0 to 50 Hz) which means that we are not going to achieve the 90% total mass target.

Vertical Mid-Span Displacement Responses

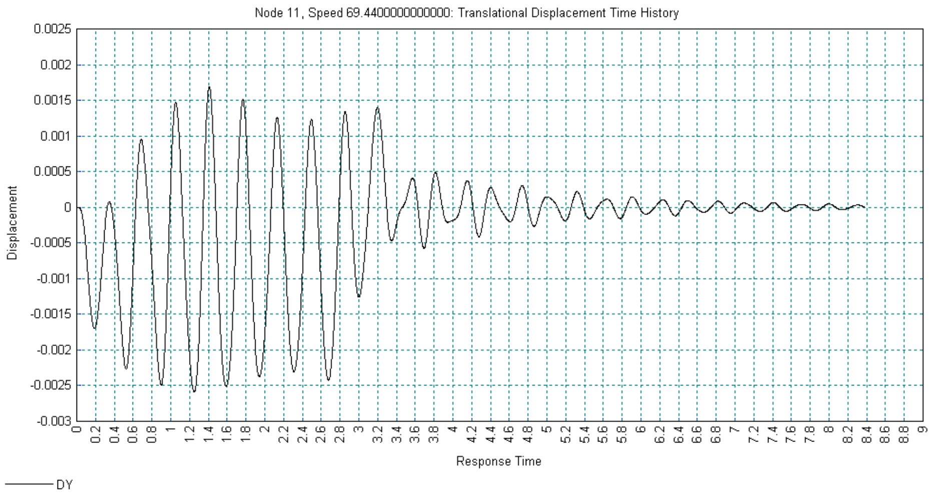
The IMDPlus Output Control dialog will appear. This controls the results output for the model. The vertical displacement response at the mid-span of the two spans of the continuous beam, nodes 11 and 31, for a speed of 250 kph will be investigated. The first span, node 11 is investigated first.

- Choose **Node** and select Extent as **Individual**

- Enter Node number **11** (This is the node in the mid-span of the first span)
- Select **Displacement** results of **DY**
- Ensure **Individual items** is selected. **Sum of chosen items** and **Average of chosen items** will not be available as an individual node is being processed.
- Ensure **Response time history** is selected.
- Ensure **Generate graphs in Modeller** is selected.
- Click the **Apply** to proceed with the analysis.



Note. Clicking the Apply button instead of the Finish button keeps the IMDPlus session active for subsequent graph plotting.

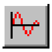


The graph generated shows the vertical displacement response at the mid-span of the first span of the continuous beam for the passage of the CRH trainset at 250 kph. The peak downwards deflection of the beam is approximately 2.6 mm and the peak upwards deflection approximately 1.7 mm. The quiet time after the passage of the trainset allows the decay of the response to be observed with most of the dynamic response damped out at around 8.5 seconds after the trainset first enters the continuous beam. The response matches the response of the first span in the Wang *et al* paper, noting that the sign conventions are opposite to the paper which has a positive vertical displacement in the downwards direction.

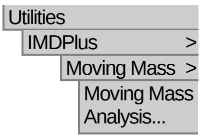
- When the graph has been displayed, close the graph

The vertical displacement response at the mid-span of the second span of the continuous beam for the speed of 250 kph will now be investigated.

If the IMDPlus toolbar has been enabled,

- Click on the  button in the toolbar to open the IMDPlus Output Control dialog

If the IMDPlus toolbar is not enabled,

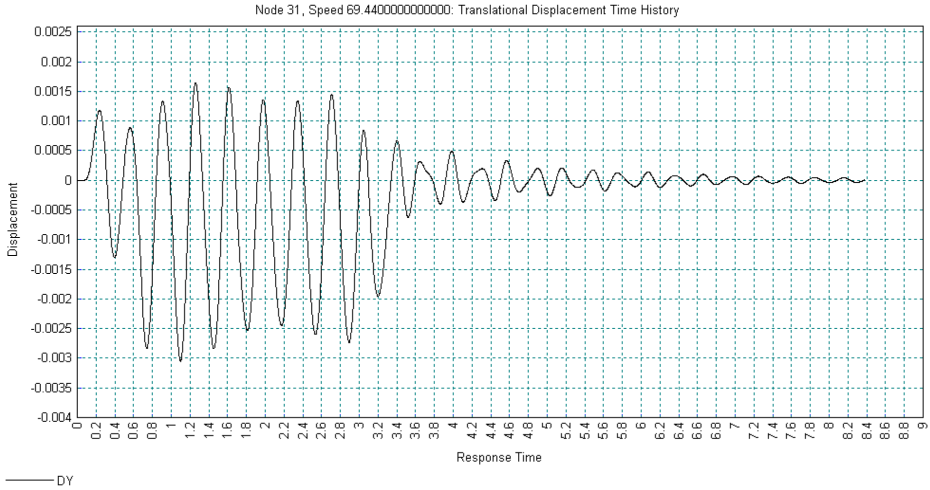


- Open the IMDPlus Moving Mass Analysis Control dialog through the menu and click **Next >** to keep the existing analysis control settings and open the IMDPlus Output Control dialog.

- Ensure **Node** and Extent as **Individual** are still selected
- Enter Node number **31** (This is the node in the mid-span of the second span)
- Ensure all other dialog input is unchanged.
- Click the **Finish** to proceed.

The IMDPlus will now proceed for the second span. The solution time for this analysis will be quicker than for the first node because of the presence of binary results databases that are saved to the disk.

- Click **No** when asked whether to free up disk space by deleting the temporary files created by IMDPlus. Choosing no will save data extraction times for subsequent analyses at the cost of some disk space.



The graph generated shows the vertical displacement response at the mid-span of the second span of the continuous beam for the passage of the CRH trainset at 250 kph. The peak downwards deflection of the beam is approximately 3 mm and the peak upwards deflection approximately 1.6 mm. The response again matches the response of the second span in the Wang *et al* paper, noting that the sign conventions are opposite to the paper which has a positive vertical displacement in the downwards direction.

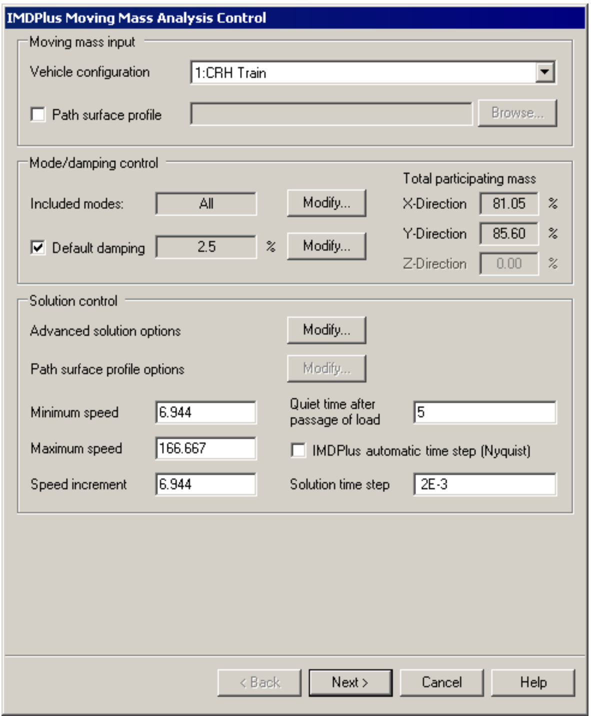
- When the graph has been displayed, close the graph

Peak Vertical Displacement of Mid-Spans for Multiple Speeds

The response of the two spans of the continuous beam were investigated for a single speed of 250 kph. The response of these two spans for a range of speeds between 25 kph and 600 kph (in 25 kph increments) is now investigated to determine the worst speeds for the response of the beam.

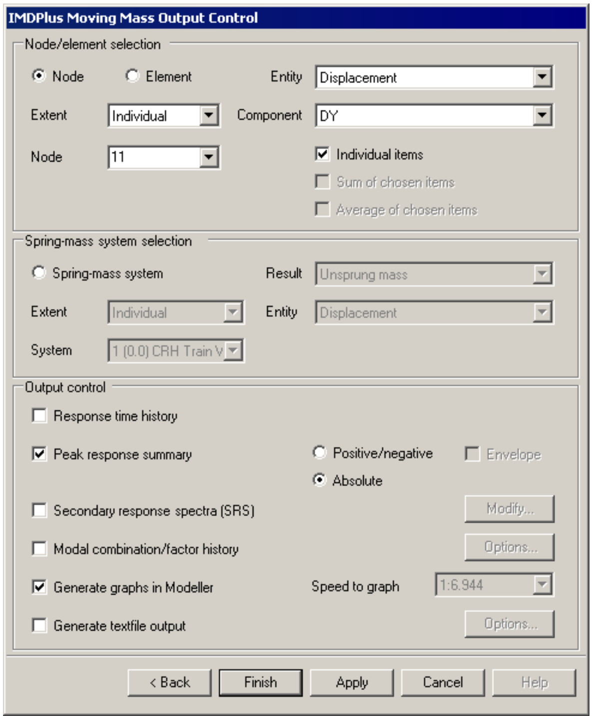
Utilities
IMDPlus >
Moving Mass >
Moving Mass Analysis...

- Open the IMDPlus Moving Mass Analysis Control dialog through the menu or click on the button in the toolbar.



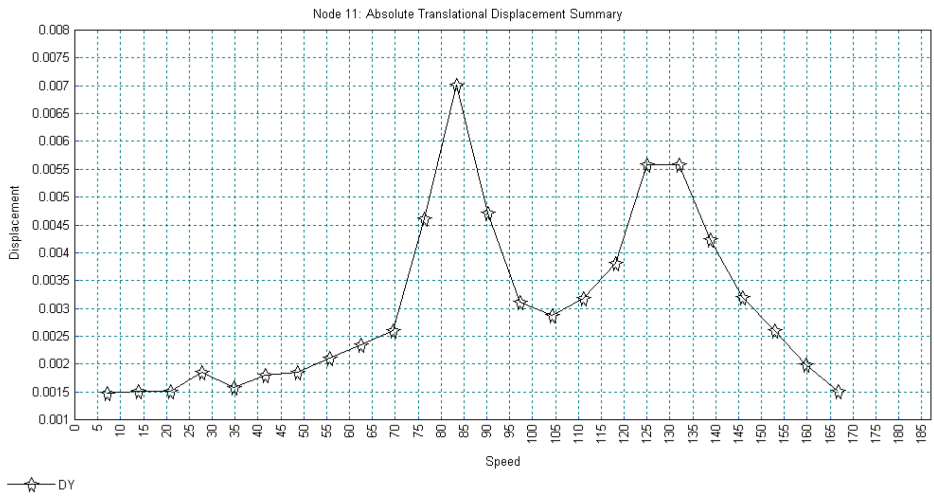
- Ensure that the Moving mass input and Mode/damping control sections of the dialog are unchanged from the previous analyses
- Enter the Minimum Speed as **6.944**, the Maximum Speed as **166.667** and the Speed Increment as **6.944** to analyse a range of speeds between 6.944 m/s (25 kph) and 166.667 m/s (600 kph) in increments of 6.944 m/s (25 kph).
- Ensure the Quiet time after passage of load is **5**
- Ensure the **IMDPlus determining time step (Nyquist)** option is deselected and the Solution time step is **2E-3**
- Click the **Next** button to proceed. When asked whether the speed range is correct choose **Yes** to continue. When prompted about missing total mass due to mass participation factors below 90% choose **Yes** to continue.

- Ensure **Node** and Extent as **Individual** are still selected
- Enter Node number **11**
- Ensure **Displacement** results of **DY** is selected.
- Ensure **Individual items** is selected.
- Deselect **Response time history** .
- Select **Peak response summary** and **Absolute**
- Ensure **Generate graphs in Modeller** is selected.
- Click the **Apply** to proceed.



IMDPlus will now calculate the sprung-mass and continuous beam interaction behaviours for all 24 of the speeds chosen in the 25 kph to 600 kph range.

Note. Due to the complexity of the equilibrium calculations of the compatible responses of the 32 spring-mass systems and the continuous beam as they pass along the path for the 24 speeds, the first analysis will take extra time to complete. Once this is done, so long as no changes to the modal information or analysis options are made, all subsequent solutions will be considerably faster due to the presence of binary results databases on disk.

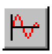


The graph shows that the worst vertical displacement response of the first span occurs around 83.328 m/s which is equivalent to a train speed of 300 kph. At this speed the peak vertical displacement response is approximately 7 mm. A second peak is observed between speeds of 124.992 m/s (450 kph) and 131.936 m/s (475 kph). Both of these peaks agree with the Wang *et al* paper which states that the speeds where resonance is induced in the beam for the first two modes of vibration are 302.67 kph and 472.98 kph. The magnitudes of the two peaks are also in agreement with Wang *et al*.

- When the graph has been displayed, close the graph

The peak vertical displacement response at the mid-span of the second span of the continuous beam for the same speed range will now be investigated.

If the IMDPlus toolbar has been enabled,

- Click on the  button in the toolbar to open the IMDPlus Output Control dialog

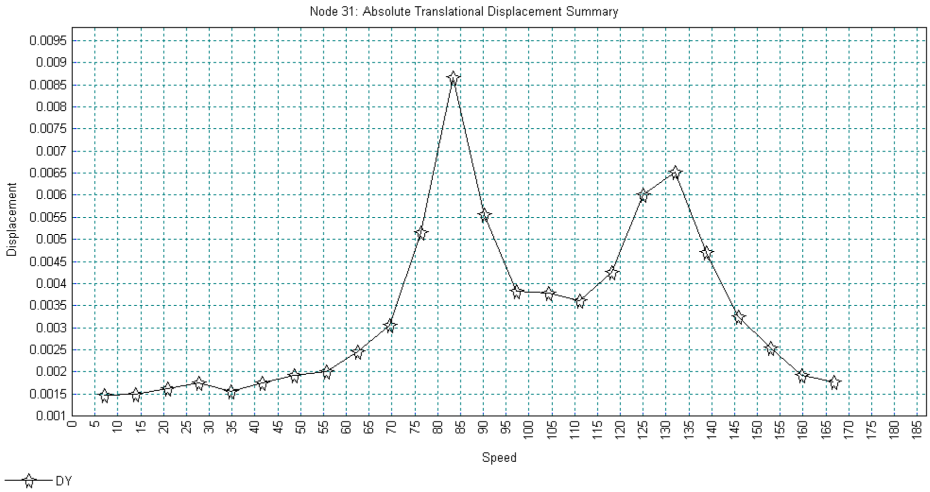
If the IMDPlus toolbar is not enabled,

Utilities
IMDPlus >
Moving Mass >
Moving Mass Analysis...

- Open the IMDPlus Moving Mass Analysis Control dialog through the menu and click **Next** > to keep the existing analysis control settings and open the IMDPlus Output Control dialog.

- Ensure **Node** and Extent as **Individual** are still selected
- Enter Node number **31**
- Ensure all other dialog input is unchanged.
- Click the **Apply** to proceed.

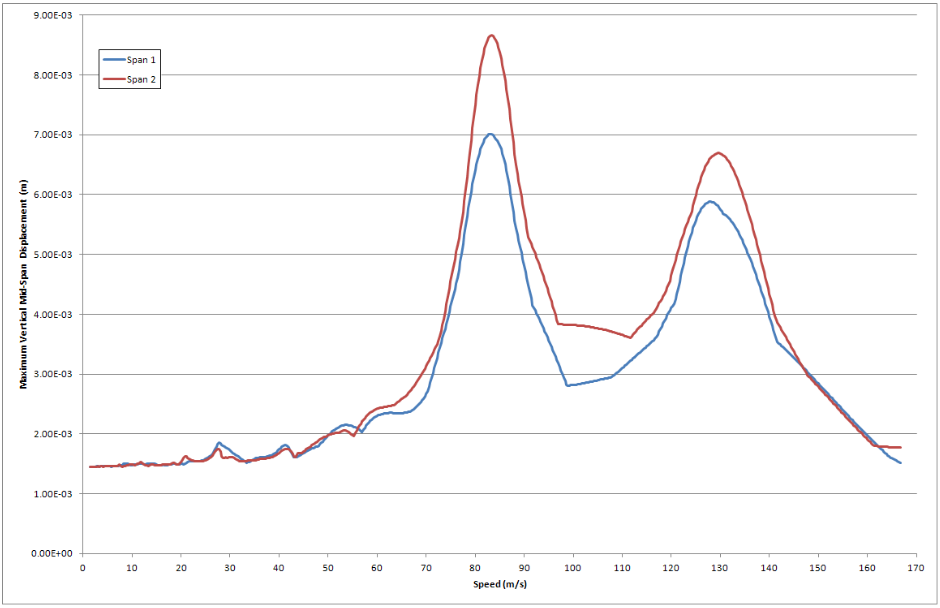
The IMDPlus analysis will now proceed for the second span. The solution time for this analysis over the range of speeds will be much quicker than for the first node because of the presence of binary results databases that are saved to the disk.



The graph shows that the worst vertical displacement response of the second span occurs again around 83.328 m/s which is equivalent to a train speed of 300 kph. At this speed the peak vertical displacement response is approximately 8.7 mm. A second peak is again observed between speeds of 124.992 m/s (450 kph) and 131.936 m/s (475 kph). Both of these peaks agree with the Wang *et al* paper and a more refined speed range would provide the same smooth curves in the paper.

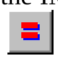
- When the graph has been displayed, close the graph

Note. The Wang *et al* paper considered speeds between 5 kph and 600 kph in very small increments and therefore appears much smoother than the two graphs above. If this analysis is conducted with a similar resolution of speed increment then the following graphs are obtained from IMDPlus which agree with those in the paper (plotted together in Microsoft Excel for a minimum speed of 5 kph, a maximum speed of 600 kph and a speed increment of 1 kph).

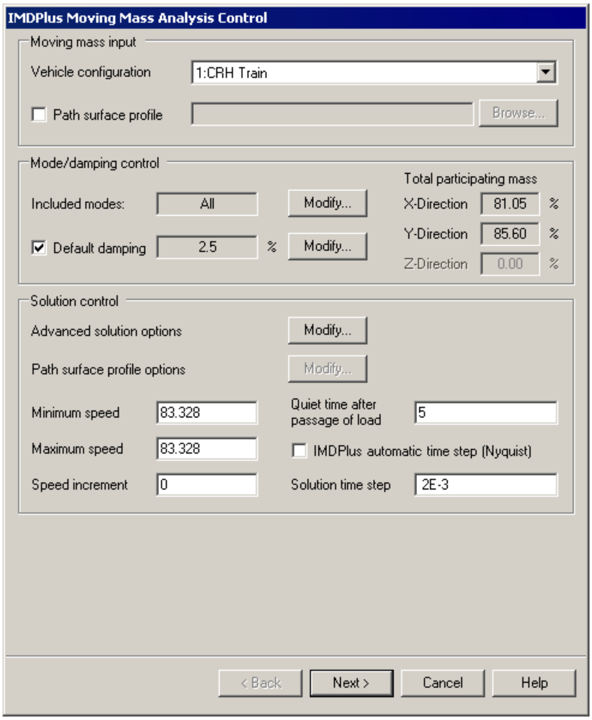


Resonant Response of Mid-Spans at 300 kph

The train speed of 300 kph (83.328 m/s) showed that there was a resonant response in the continuous beam at around this speed. The time history response of the two mid-span nodes of the continuous beam will now be investigated at this speed.

- Click on a blank part of the model window to deselect the line (if it is still selected)
- Right click on a blank part of the model window and choose **Advanced Selection...**
- Set the Type and name as **Node** and enter **11,31** in the textbox
- Click on **OK** to select both mid-span nodes
- Open the IMDPlus Moving Mass Analysis Control dialog through the menu or click on the  button in the toolbar.

Utilities
IMDPlus >
Moving Mass >
Moving Mass Analysis...



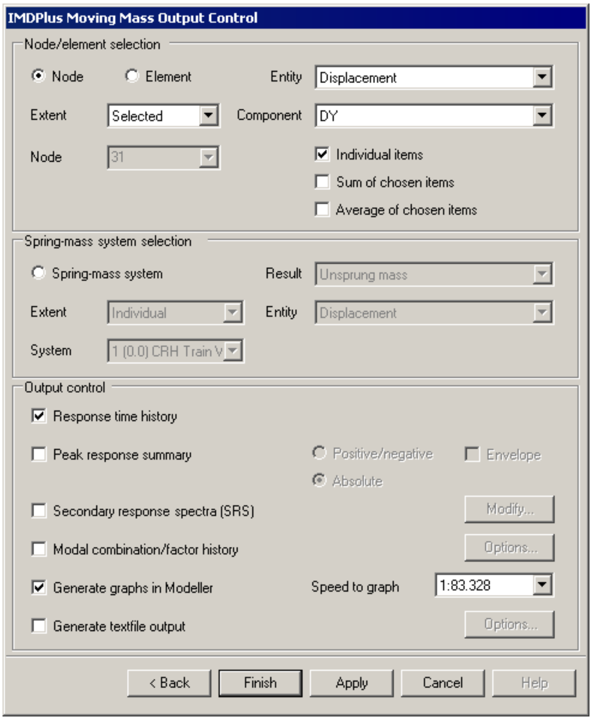
- Ensure that the Moving mass input and Mode/damping control sections of the dialog are unchanged from the previous analyses

- Enter the Minimum Speed as **83.328**, the Maximum Speed as **83.328** and the Speed Increment as **0** to analyse a single speed of 83.328 m/s which is equivalent to 300 kph.

- Ensure the Quiet time after passage of load is **5**

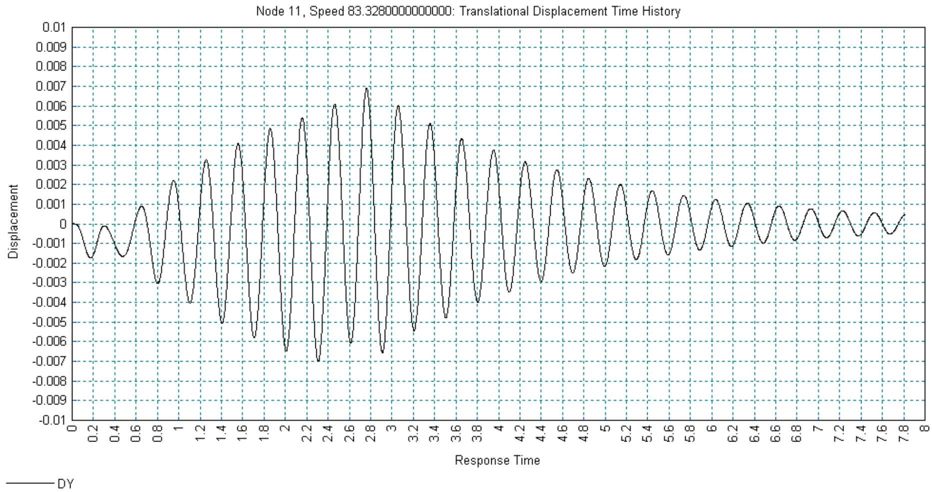
- Ensure the **IMDPlus determining time step (Nyquist)** option is deselected and the Solution time step is **2E-3**

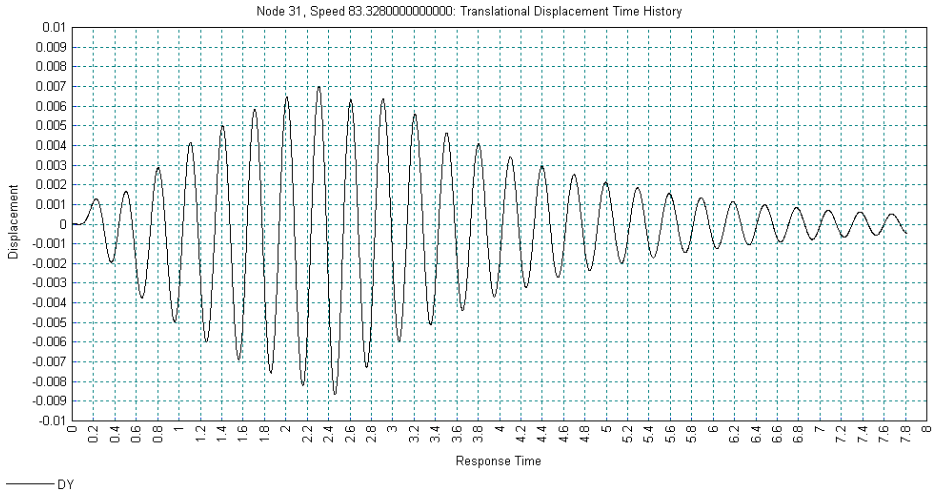
- Click the **Next** button to proceed and click **Yes** when informed about the analysis having a mass participation below 90%.



- Choose **Node** and Extent as **Selected**
- Ensure **Displacement** results of **DY** is selected
- Ensure **Individual items** is selected
- Ensure **Sum of chosen items** and **Average of chosen items** are deselected
- Select **Response time history**
- Deselect **Peak response summary**
- Ensure **Generate graphs in Modeller** is selected
- Ensure the Speed to graph is set to **1:83.328** (equivalent to 300 kph)
- Click the **Apply** to proceed
- Choose **Yes** when asked whether you wish to process all selected nodes

The responses of the two mid-span nodes are shown below, firstly for the first span (node 11) and secondly for the second span (node 31). In both figures the resonance of the continuous beam is observed until the departure of the last wheelset of the train from the structure after 2.82 seconds. After this time the damping causes the decay of the vertical response of the beam which will eventually return to a static state.



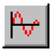


- When the graphs have been displayed, close the graphs

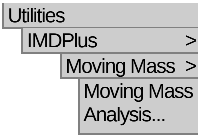
Displacement Time Histories of First and Last Spring-Mass Systems

In the previous analyses attention was focused on the vertical displacement response of the continuous beam from the passage of the CRH trainset over a range of speeds. The response of the actual train is now investigated.

If the IMDPlus toolbar has been enabled,

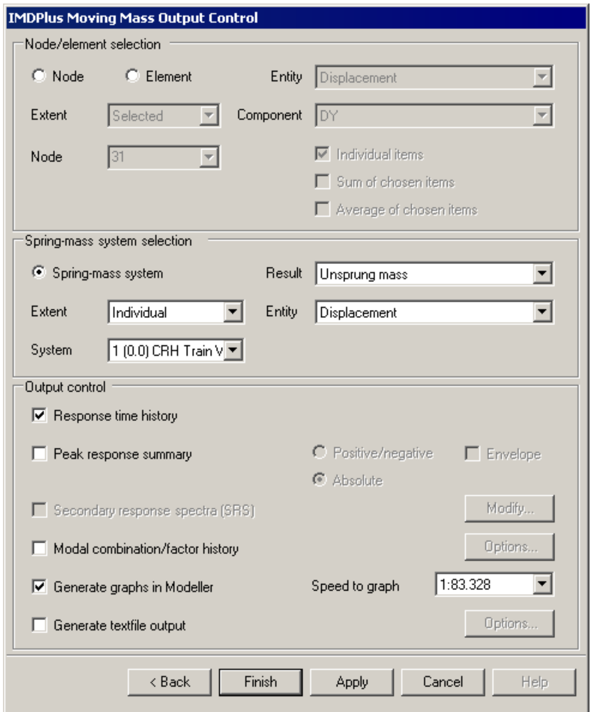
- Click on the  button in the toolbar to open the IMDPlus Output Control dialog

If the IMDPlus toolbar is not enabled,

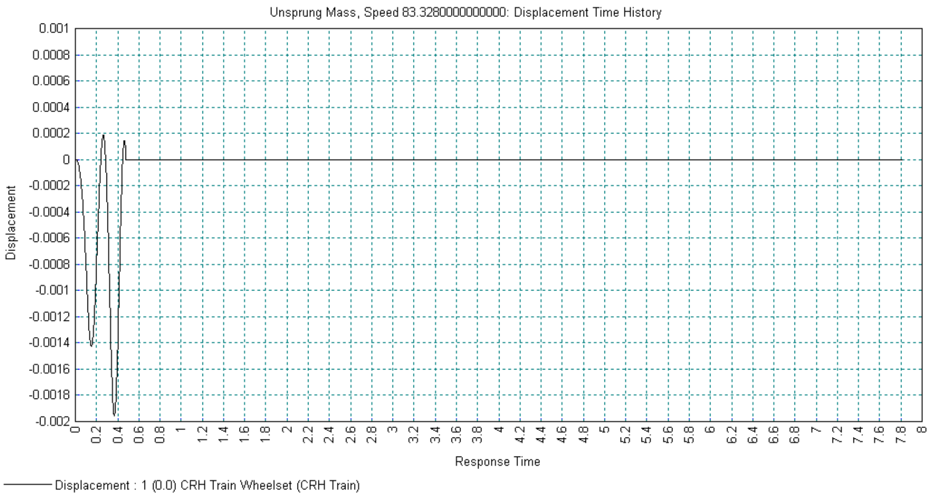


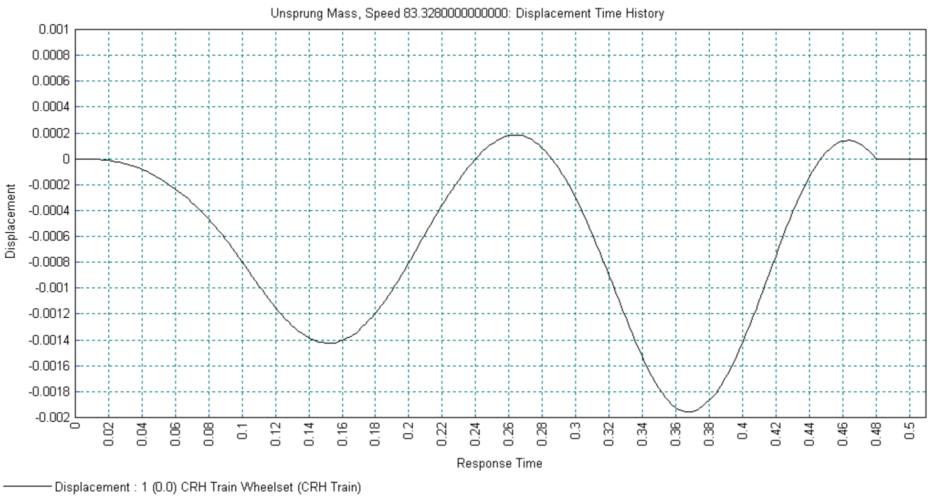
- Open the IMDPlus Moving Mass Analysis Control dialog through the menu and click **Next** > to keep the existing analysis control settings and open the IMDPlus Output Control dialog.

- Select **Spring-mass system** and Extent as **Individual**
- Set the System to **1 (0.0) CRH Train Wheelset**
- Set the Result to **Unsprung mass**
- Set the Entity to **Displacement**
- Ensure all of the selections in the **Output control** are identical to the previous analysis
- Click the **Apply** to proceed.



The plotting of the results will now proceed. IMDPlus should not need to resolve the analysis since the spring-mass system results should already be available.

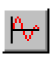




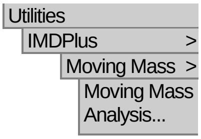
The second graph shows the curve after zooming into the first 0.5 seconds of the time history. This indicates that the displacement of the unsprung mass returns to zero (i.e. the first spring-mass system has left the structure) after about 0.48 seconds. This can be confirmed by dividing the total length of the continuous beam by the speed, 40 / 83.328 = 0.48 seconds.

- When the graph has been displayed, close the graph

If the IMDPlus toolbar has been enabled,

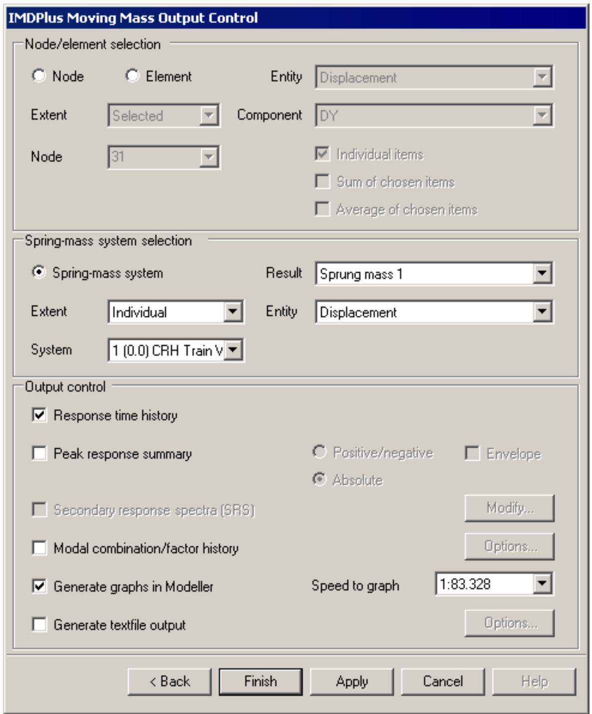
- Click on the  button in the toolbar to open the IMDPlus Output Control dialog

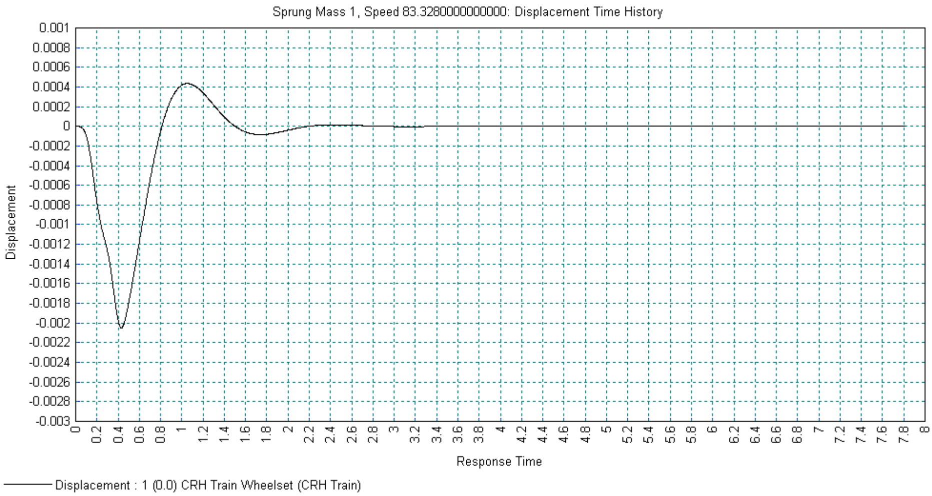
If the IMDPlus toolbar is not enabled,



- Open the IMDPlus Moving Mass Analysis Control dialog through the menu and click **Next** > to keep the existing analysis control settings and open the IMDPlus Output Control dialog.

- Ensure **Spring-mass system** is selected
- Ensure Extent is set to **Individual**
- Ensure the System is set **1 (0.0) CRH Train Wheelset**
- Set the Result to **Sprung mass 1**
- Ensure the Entity is set to **Displacement**
- Ensure all of the selections in the **Output control** are identical to the previous analysis
- Click the **Apply** to proceed.





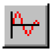
The graph shows that the sprung mass representing the body of the first wheelset of the train displaces just over 2 mm from its static position 0.43 seconds after it first enters the continuous beam and the viscous damping of the spring-mass system quickly damps out the body motion within 2 to 2.5 seconds after the wheelset leaves the structure (which occurs at 0.48 seconds).

- When the graph has been displayed, close the graph

Note. All results for the sprung masses are with respect to their static equilibrium positions. Static displacements of the sprung masses are summarised in the IMDPlus.log file at the beginning of the analysis.

The last spring-mass system is 195 m behind the first one and therefore the continuous beam will have received dynamic input from 31 spring-mass systems by the time it enters the beam. As a result, the vertical displacement of the unsprung and sprung masses are expected to be different from the graphs presented above.

If the IMDPlus toolbar has been enabled,

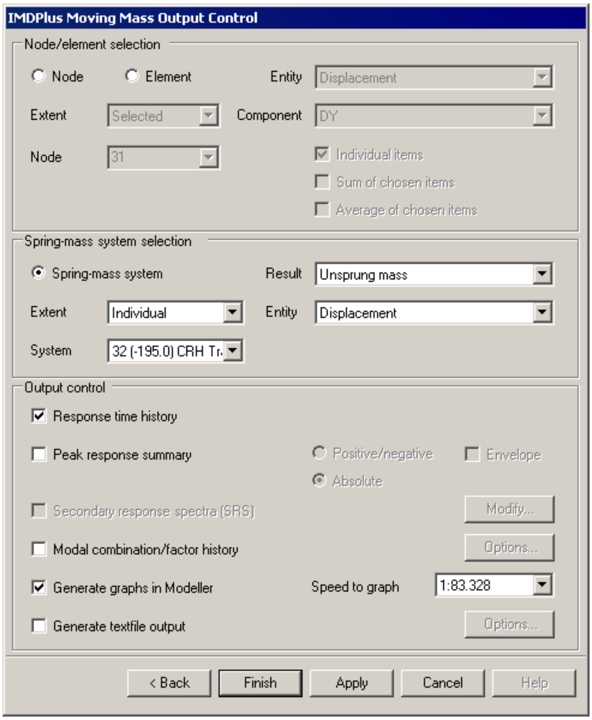
- Click on the  button in the toolbar to open the IMDPlus Output Control dialog

If the IMDPlus toolbar is not enabled,

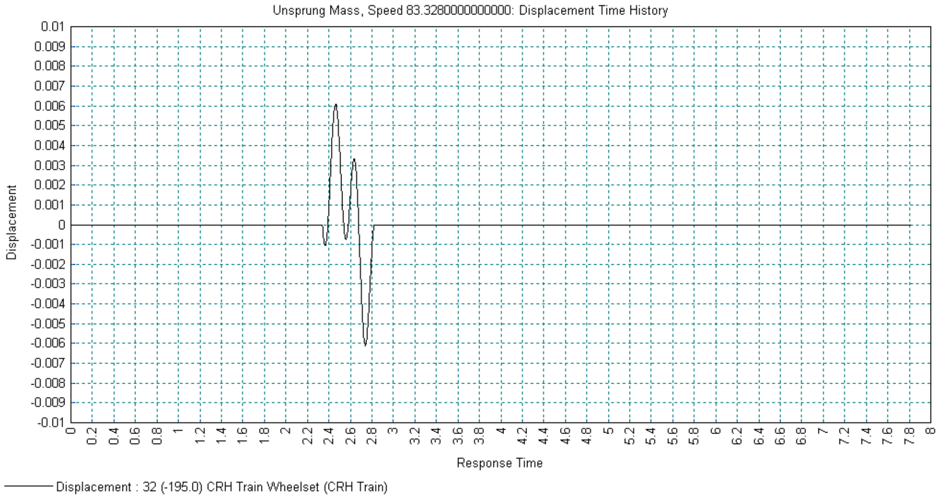
Utilities
IMDPlus >
Moving Mass >
Moving Mass Analysis...

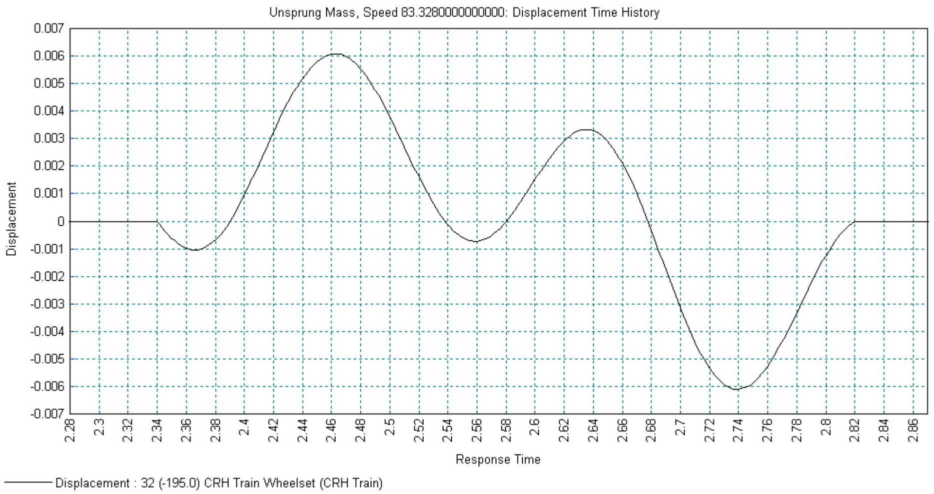
- Open the IMDPlus Moving Mass Analysis Control dialog through the menu and click **Next** > to keep the existing analysis control settings and open the IMDPlus Output Control dialog.

- Ensure **Spring-mass system** is selected
- Ensure Extent is set to **Individual**
- Set the System to **32 (-195.0) CRH Train Wheelset**
- Set the Result to **Unsprung mass**
- Ensure the Entity is set to **Displacement**
- Ensure all of the selections in the **Output control** are identical to the previous analysis
- Click the **Apply** to proceed.



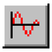
The graphs show the vertical displacement of the last sprung mass for the whole time history and having zoomed into the region where the dynamic behaviour is observed. This indicates that the displacement of the unsprung mass is zero until it enters the structure at about 2.34 seconds and returns to zero after it has left the structure at about 2.8 seconds. Comparison of the vertical displacement response while the spring-mass system is over the beam with the first unsprung mass system above shows how the existing vibration of the continuous beam has altered the unsprung behaviour.





- When the graph has been displayed, close the graph

If the IMDPlus toolbar has been enabled,

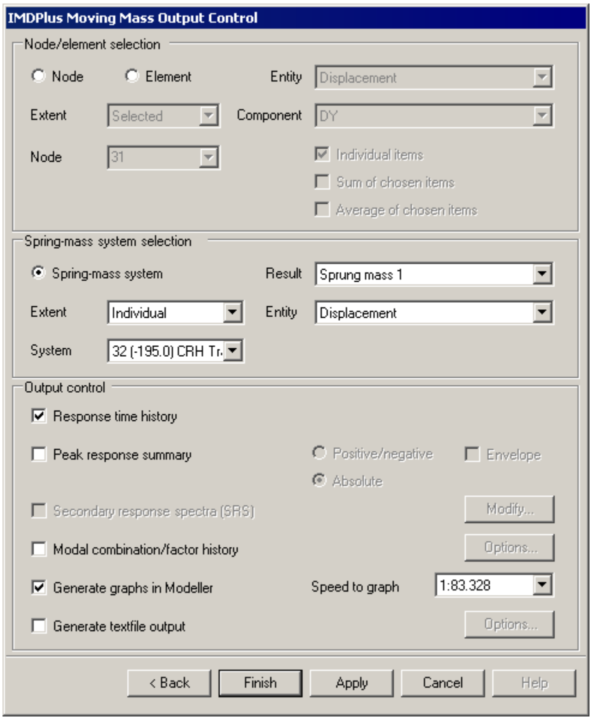
- Click on the  button in the toolbar to open the IMDPlus Output Control dialog

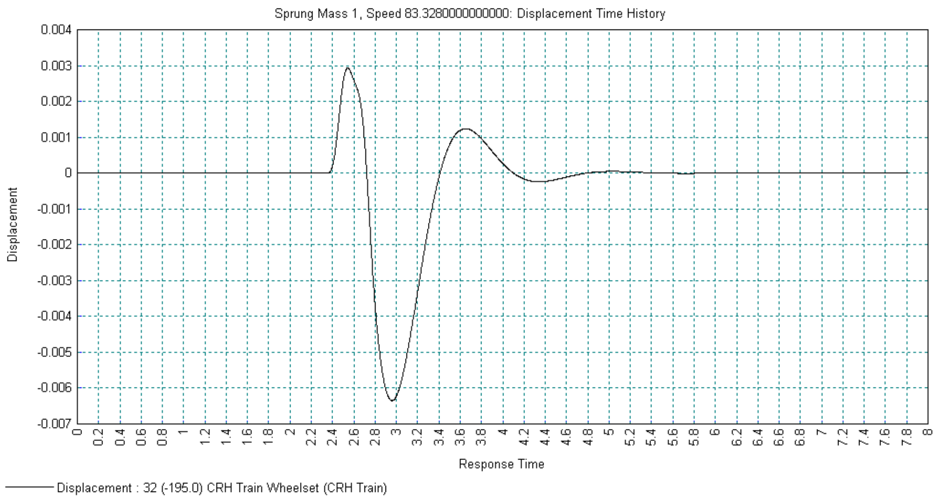
If the IMDPlus toolbar is not enabled,

Utilities
IMDPlus >
Moving Mass >
Moving Mass Analysis...

- Open the IMDPlus Moving Mass Analysis Control dialog through the menu and click **Next** > to keep the existing analysis control settings and open the IMDPlus Output Control dialog.

- Ensure **Spring-mass system** is selected
- Ensure Extent is set to **Individual**
- Ensure the System is set **32 (-195.0) CRH Train Wheelset**
- Set the Result to **Sprung mass 1**
- Ensure the Entity is set to **Displacement**
- Ensure all of the selections in the **Output control** are identical to the previous analysis
- Click the **Apply** to proceed.





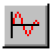
The graph shows that the sprung mass representing the body of the last wheelset of the train displaces almost 6.5 mm from its static position while it is over the continuous beam before the damping of the spring-mass system quickly damps out the body motion after the wheelset leaves the structure.

- When the graph has been displayed, close the graph

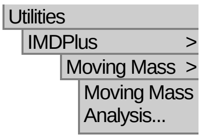
Acceleration Time Histories of First and Last Spring-Mass System Bodies

The acceleration response of the sprung masses representing the bodies of train locomotives and passenger cars can be used to assess the comfort level that would be experienced by passengers. The accelerations of the sprung mass for the first and last sprung mass systems will be investigated.

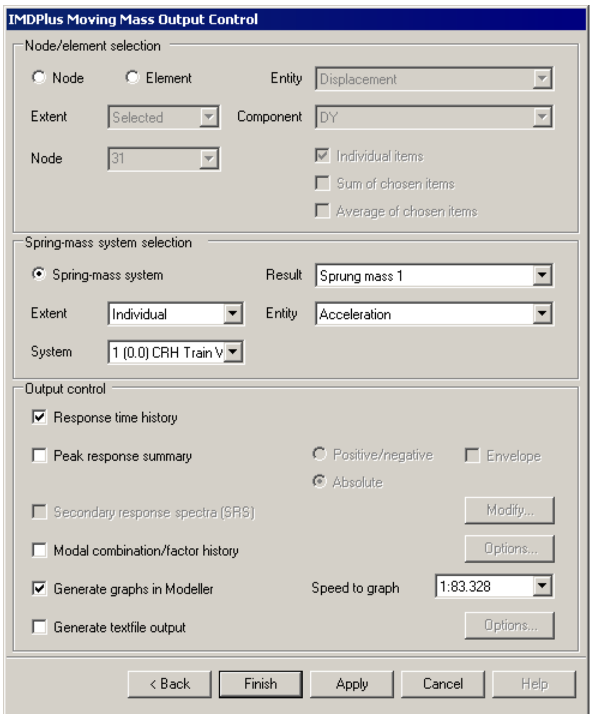
If the IMDPlus toolbar has been enabled,

- Click on the  button in the toolbar to open the IMDPlus Output Control dialog

If the IMDPlus toolbar is not enabled,

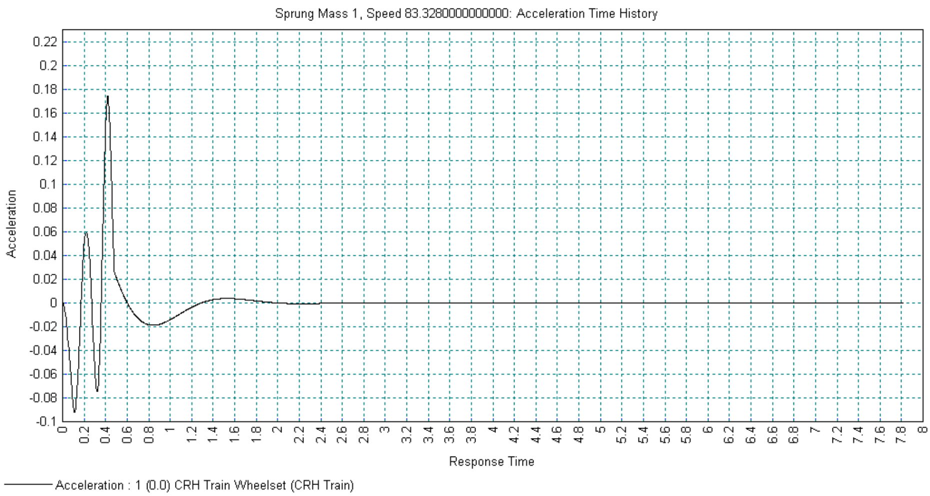


- Open the IMDPlus Moving Mass Analysis Control dialog through the menu and click **Next >** to keep the existing analysis control settings and open the IMDPlus Output Control dialog.



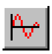
- Ensure **Spring-mass system** is selected
- Ensure Extent is set to **Individual**
- Set the System to **1 (0.0) CRH Train Wheelset**
- Ensure the Result is set to **Sprung mass 1**
- Set the Entity to **Acceleration**
- Ensure all of the selections in the **Output control** are identical to the previous analysis
- Click the **Apply** to proceed.

The graph shows the vertical acceleration response of the body of the first wheelset which experiences a maximum acceleration of +0.174 m/s^2 or 0.018 g as it is about to leave the continuous beam. This peak acceleration for the first wheelset occurs while the dynamic response of the continuous beam is still increasing and therefore a higher peak acceleration might be expected for subsequent wheelsets of the train.



- When the graph has been displayed, close the graph

If the IMDPlus toolbar has been enabled,

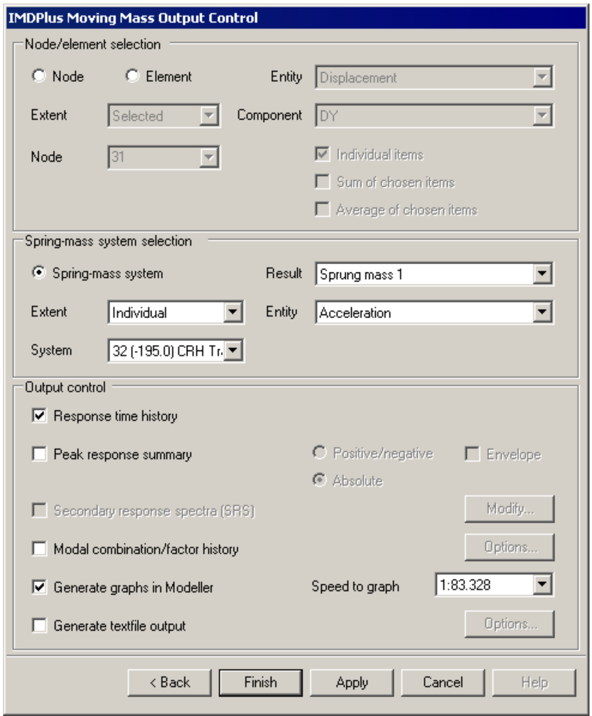
- Click on the  button in the toolbar to open the IMDPlus Output Control dialog

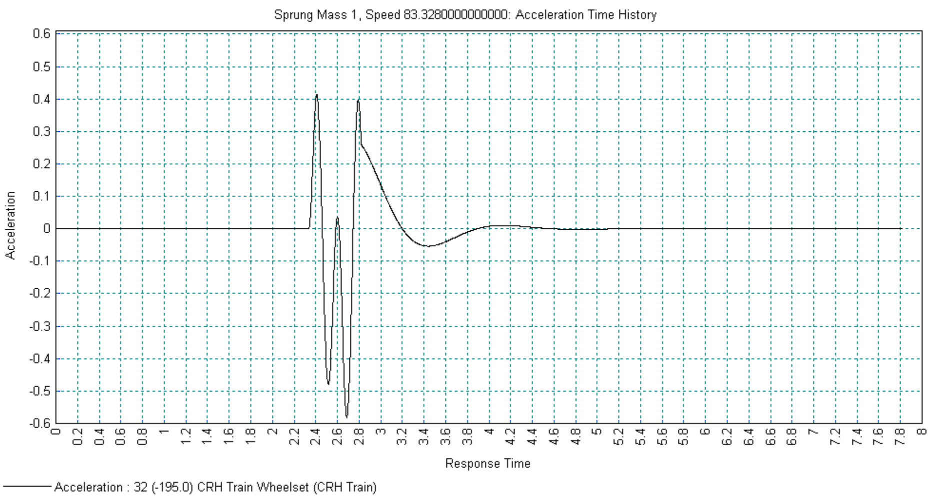
If the IMDPlus toolbar is not enabled,

Utilities
IMDPlus >
Moving Mass >
Moving Mass Analysis...

- Open the IMDPlus Moving Mass Analysis Control dialog through the menu and click **Next** > to keep the existing analysis control settings and open the IMDPlus Output Control dialog.

- Ensure **Spring-mass system** is selected
- Ensure Extent is set to **Individual**
- Set the System to **32 (-195.0) CRH Train Wheelset**
- Ensure the Result is set to **Sprung mass 1**
- Ensure the Entity is set to **Acceleration**
- Ensure all of the selections in the **Output control** are identical to the previous analysis
- Click the **Finish** to proceed. Click **No** when asked whether to free up disk space.



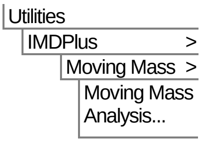


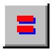
The graph shows the vertical acceleration response of the body of the last wheelset and indicates that it experiences a maximum acceleration of -0.582 m/s^2 or -0.059 g while it is about to leave the continuous beam. This peak acceleration for the last wheelset is significantly higher than the first wheelset as might be expected for subsequent wheelsets of the train since the beam has already experienced excitation from all of the previous wheelsets.

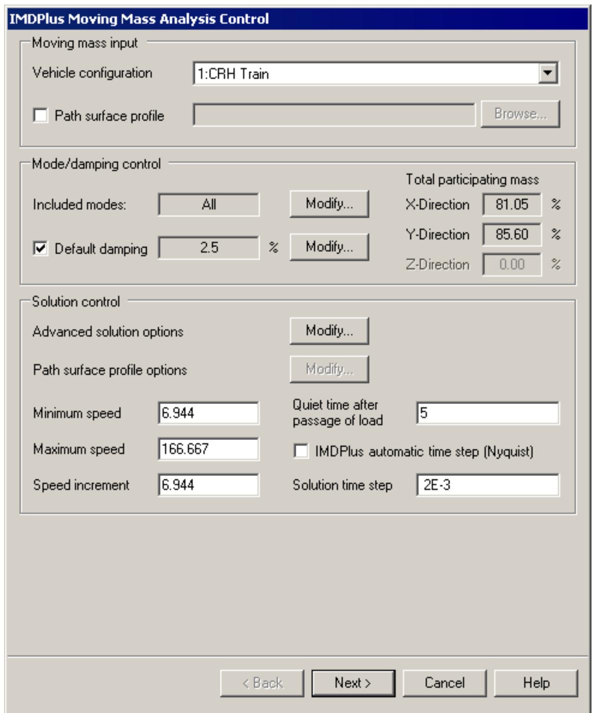
- When the graph has been displayed, close the graph

Peak Acceleration Response of All Train Bodies

In the previous graphs the acceleration response of the sprung masses representing the bodies of the first and last wheelsets representing the train were inspected. The peak vertical acceleration response of all of the sprung masses representing the bodies of the train is now inspected for the 25 kph to 600 kph speed range.



- Open the IMDPlus Moving Mass Analysis Control dialog through the menu or click on the  button in the toolbar.



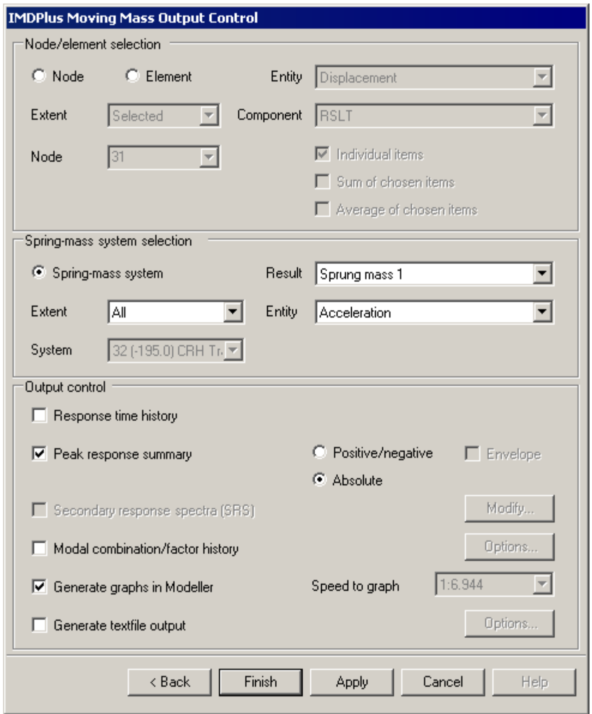
- Ensure that the Moving mass input and Mode/damping control sections of the dialog are unchanged from the previous analyses

- Enter the Minimum Speed as **6.944**, the Maximum Speed as **166.667** and the Speed Increment as **6.944** to analyse a range of speeds between 6.944 m/s (25 kph) and 166.667 m/s (600 kph) in increments of 6.944 m/s (25 kph).

- Ensure the Quiet time after passage of load is **5**

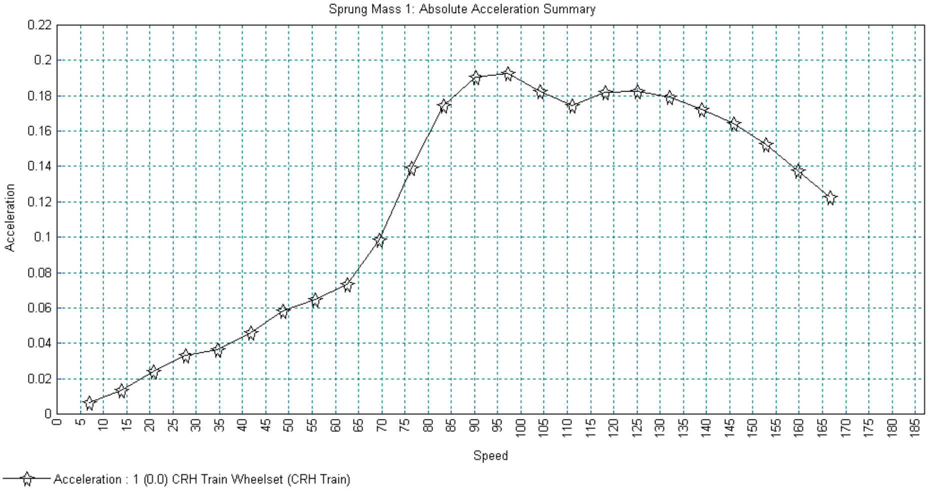
- Ensure the **IMDPlus determining time step (Nyquist)** option is deselected and the Solution time step is **2E-3**

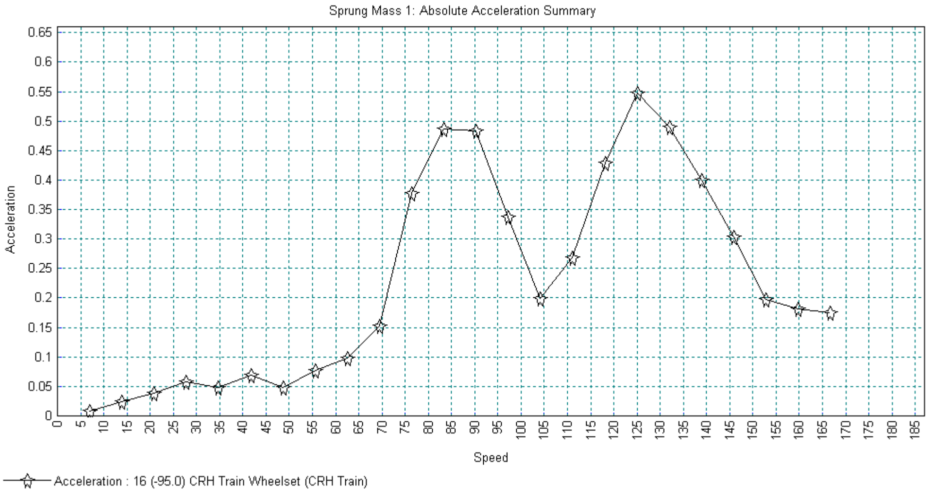
- Click the **Next** button to proceed. When asked whether the speed range is correct choose **Yes** to continue. When prompted about missing total mass due to mass participation factors below 90% choose **Yes** to continue.

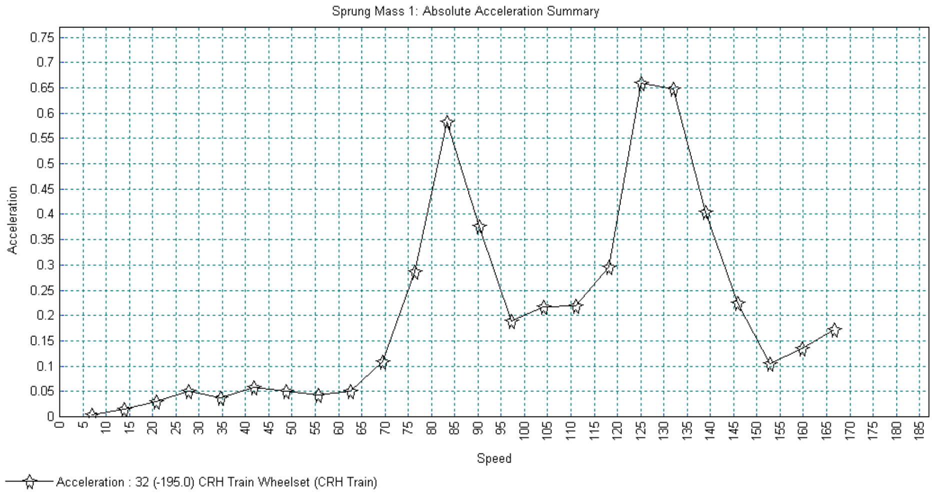


- Ensure **Spring-mass system** is selected
- Set Extent to **All**
- Ensure the Result is set to **Sprung mass 1**
- Ensure the Entity is set to **Acceleration**
- Deselect **Response time history**
- Select **Peak response summary** and **Absolute**
- Ensure all of the selections in the **Output control** are identical to the previous analysis
- Click the **Finish** to proceed.
- Click **Yes** when asked whether to free up disk space by deleting the temporary files created by IMDPlus.

IMDPlus will perform the analysis and the graphs for all 32 of the spring-mass systems will be displayed and can be printed out. A sample of the graphs are shown below demonstrating the different behaviours of each of the sprung bodies of the train (first spring-mass system, sixteenth spring-mass system and the last spring-mass system).

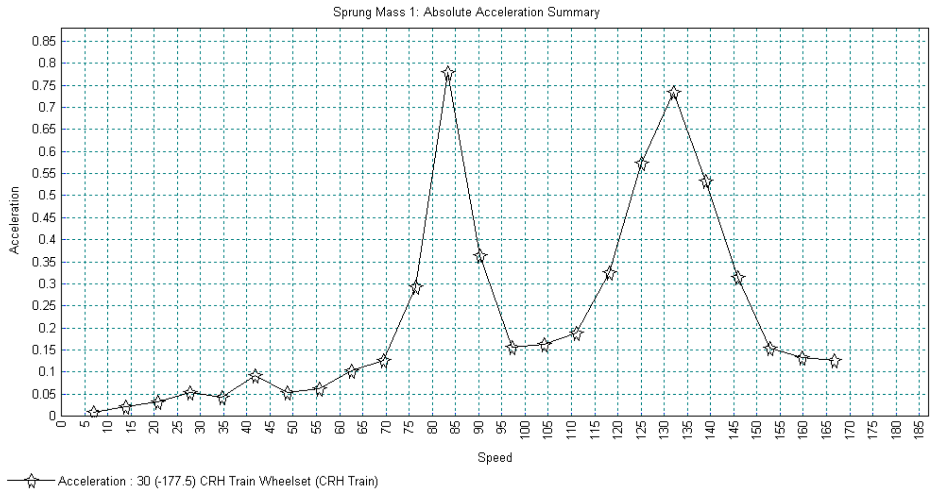

Sprung Mass 1: Absolute Acceleration Summary
Acceleration
Speed
Acceleration : 1 (0.0) CRH Train Wheelset (CRH Train)


Sprung Mass 1: Absolute Acceleration Summary
Acceleration
Speed
Acceleration : 16 (-95.0) CRH Train Wheelset (CRH Train)



The graphs of the absolute peak vertical acceleration of the sprung bodies over the speed range appears to indicate that the peak acceleration increases as the number of wheelset spring-mass systems which have already passed along the beam increases. This is roughly true but from inspection of each wheelset spring-mass system it can be seen that the absolute peak acceleration out of the whole 32 wheelsets of the train actually occurs at the 30th wheelset, indicated in the graph below.

For the 30th wheelset the absolute peak acceleration is observed at the first resonant frequency equivalent to a train speed of 300 kph and is approximately 0.78 m/s^2 or almost 0.08 g.



- Close all of the graph windows.

Save the model

File
Save…

Save the model.

This completes the example.