

IMDPlus User Manual

LUSAS Version 14.2 : 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

Copyright ©1982-2008 LUSAS

All Rights Reserved.

Table of Contents

IMDPlus Assumptions	1
Performing IMDPlus Calculations in LUSAS	2
Seismic Analysis	2
Moving Load Analysis	2
Seismic Analysis	2
Earthquake Input	3
Mode and Damping Control	3
Frequency Interpolation	4
Moving Load Analysis.....	5
Setup Moving Load Path	5
Setup Modal History Information	10
Running the Moving Load Analysis	11
Visualising the Results	13
Node / Element Selection.....	14
Output Control	15
Elements Supported By IMDPlus.....	21

IMDPlus User Manual

The IMDPlus modal dynamics facility 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.)

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

- ❑ **Seismic Analysis** Allows the analysis of the seismic response of a structure to acceleration time histories of support motion.
- ❑ **Moving Load Analysis** Allows the analysis of the moving load response of a structure where the magnitude and configuration of the load remains constant throughout the analysis.

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 Duhamel's Integral.

Performing IMDPlus Calculations in LUSAS

The IMDPlus command is initiated from the **Utilities** menu. The basic steps for the two analysis types are as follows:

Seismic Analysis

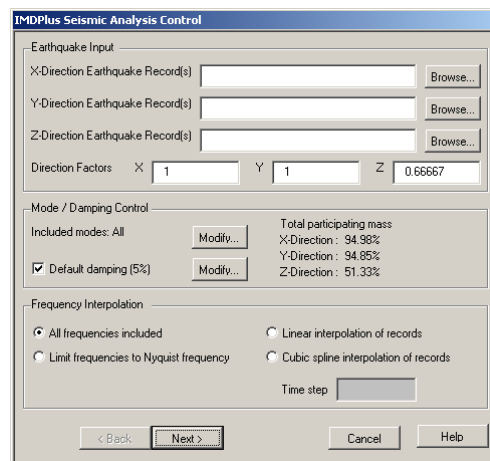
1. Select the acceleration time histories
2. Select the eigenmodes to include in the solution
3. Specify the damping for the eigenmodes
4. Specify the frequency interpolation technique
5. Specify the node / element to process and output requirements

Moving Load Analysis

1. Specify the movement of the load across the structure at discrete locations and the equivalent modal forces
2. Specify the load configuration if this has not been carried out explicitly in the previous step
3. Select the eigenmodes to include in the solution
4. Specify the damping for the eigenmodes
5. Specify the speed ranges to analyse along with time stepping parameters
6. Specify the node / element to process and output requirements

Seismic Analysis

The seismic analysis option analyses the response of the structure to an acceleration time history input to the support nodes. 2D and 3D structures can be analysed but are restricted to having the acceleration inputs applied in the global X-, Y-directions for a 2D structure and in the global X-, Y- and Z-directions for a 3D structure. 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 for this direction. Different acceleration records can be



applied to each global direction and factored based on design code requirements. Up to seven earthquake combinations can be analysed in a single IMDPlus analysis.

Earthquake Input

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

For all formats, the time step of the acceleration records must be identical in all 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
```

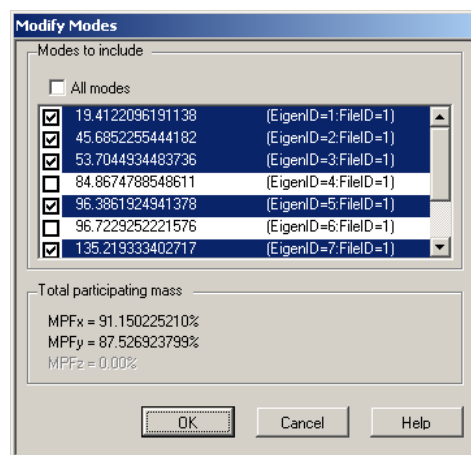
Space Delimited Two Combinations

```
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 and Damping Control

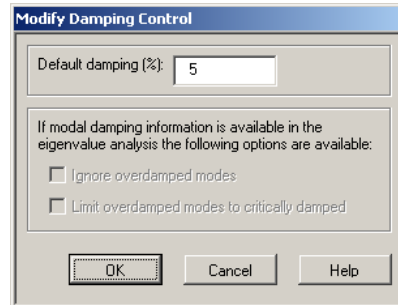
The modes to include in the analysis can be controlled either by the eigenmodes solved in the eigenvalue analysis or a subset defined in the mode control. The default option when first running an analysis is for all modes to be included in the analysis but individual modes can be included or excluded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

The 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 force all modes of vibration included in the analysis to have the current default value. This default damping is set via the Modify... button on the seismic analysis dialog. The default damping option cannot be turned off unless viscous damping has been included in the eigenvalue analysis.



If viscous damping is present in the eigenvalue analysis, additional options for the inclusion of overdamped modes of vibration in the IMDPlus analysis 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 the linear interpolation of the records. Required time step is specified by the user for the analysis.
- Cubic spline interpolation of records** Allows the cubic spline interpolation of the records. 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.

Moving Load Analysis

The moving load analysis option analyses the response of the structure to the passage of a moving load along a user defined path. Only 3D structures can be analysed and the magnitude and configuration of the load remains constant during the whole analysis. Two methods are available for the definition of the moving load configuration. These are:

- Explicit definition of the configuration through a discrete load definition in Modeller.
- Composite axle definition where a unit load single axle configuration is defined as a discrete load definition in Modeller and the axle configuration is defined separately.

For analysing the passage of a single load configuration across the structure, the explicit definition method is generally used but where multiple load configurations are to be analysed (such as multiple train rolling stock configurations) the composite axle definition allows these to be analysed rapidly without having to repeat initial steps for the IMDPlus solution.

Two distinct steps are required to prepare the moving load input data for IMDPlus. These two steps are as follows:

1. Definition 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 and load configuration has not changed between analyses. If either of these have been modified the moving load path and calculation of equivalent modal forces must be carried out again.

Setup Moving Load Path

The moving load generator is used to mimic the path of a load across a structure by automatically setting up a number of static load cases at prescribed locations along a single line/arc or a collection of lines/arcs which define a single continuous path.

IMDPlus User Manual

These load cases can then be used to calculate the modal forces equivalent to the applied loading for import into IMDPlus.

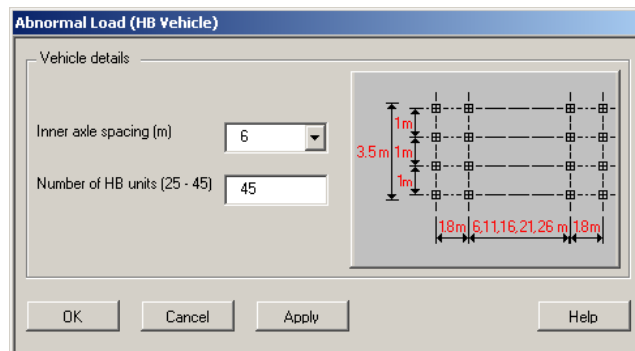
Before entering the moving load generator, the lines/arcs defining the path must be selected and a discrete load representing the load configuration to be moved across the structure must have been defined. The definition requirements for this discrete load are governed by the type of moving load input that is going to be used in IMDPlus. If the discrete load is going to represent the whole load configuration (i.e. the composite axle definition method is not being used) then this discrete load contains all of the loading associated with the configuration. If however 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 composite axle definition file to build up the complete configuration in IMDPlus.



Note. The front of the moving load is always the location with the maximum discrete load X coordinate.

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. This generates a discrete point load as indicated below.



Point

Projection vector

X component: 0.0

Y component: 0.0

Z component: 1.0

Arbitrary

Grid

x: 4

y: 4

Untransformed load direction

X

Y

Z

Surface normal

	X Coordinate	Y Coordinate	Z Coordinate	Load
1	-4.8	-1.5	0.0	-112.5E3
2	-3.0	-1.5	0.0	-112.5E3
3	3.0	-1.5	0.0	-112.5E3
4	4.8	-1.5	0.0	-112.5E3
5	-4.8	-0.5	0.0	-112.5E3
6	-3.0	-0.5	0.0	-112.5E3
7	3.0	-0.5	0.0	-112.5E3
8	4.8	-0.5	0.0	-112.5E3
9	-4.8	0.5	0.0	-112.5E3
10	-3.0	0.5	0.0	-112.5E3

Attribute: HB 6m spacing 45 units (1)

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

Composite Axle Definition of United Kingdom HB Vehicle

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 axle spacings separately in a text file. The following discrete point load defines the unit axle for the HB vehicle.

Point

Projection vector

X component: 0.0

Y component: 0.0

Z component: 1.0

Arbitrary

Grid

x: 0

y: 0

Untransformed load direction

X

Y

Z

Surface normal

	X Coordinate	Y Coordinate	Z Coordinate	Load
1	0.0	-1.5	0.0	-0.25
2	0.0	-0.5	0.0	-0.25
3	0.0	0.5	0.0	-0.25
4	0.0	1.5	0.0	-0.25
5				

Attribute: Unit HB Axle (1)

The axle spacing is defined as follows:

4.8 450000
3.0 450000
-3.0 450000
-4.8 450000

and would be saved in a text file with a meaningful name, for example 45Unit_HB_Axles.prn .



Note. Care should be taken with the definition of discrete load configurations with positive and negative along path offsets (X Coordinates). 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 explicit example, the path should start at least 4.8 m before the structure and finish 4.8 m after the structure.

Generation of the Moving Load Discrete Locations

The screenshot shows the 'IMDPlus Moving Load Generation' dialog box. It is divided into three sections: 'Loading options', 'Load path options', and 'Analysis options'.
- 'Loading options': 'Moving load dataset' is set to '1:Unit Axle Load' and 'Search area ID' is set to '1:Track_Search_Area'. A 'NOTE' states: 'All existing loading will be removed and the moving loads will start from loadcase 1'. An 'Advanced...' button is present.
- 'Load path options': 'Incremental distance for moving load along path' is set to '0.05'. 'Direction' is set to 'Forwards'. 'Start of path' is X = -5.32, Y = 0, Z = 0.5032. 'Finish of path' is X = 22.04, Y = 0, Z = 0.5032. 'Number of divisions/increments to define full path = 547'.
- 'Analysis options': 'Number of load cases per analysis' is set to '100'. 'Datacheck' and 'Run analyses' are both checked.
Buttons for 'OK', 'Cancel', and 'Help' are at the bottom.

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. 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. 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, the number of divisions in the patch load x and y directions and the load factor to be applied.

Having defined the loading options, the load path options can be set based on the lines/arcs 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:

$$\delta \text{Dist} \leq 3 \times \text{Minimum Speed} \times \delta t$$

to give a maximum oversample of 3 where δt is the minimum of the Nyquist time step and any user specified time step ($\delta t^{\text{Nyquist}} = 1/(2 * f_{\text{max}})$ 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 moving load 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 speeds 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 and arcs with the start being at the first line with only one other connected line selected. 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.

IMDPlus User Manual

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 the batch file generated and echoed to the user can be run separately. The original model, associated eigenvalue results and the static load cases must however 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 current working directory:



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.

Setup Modal History Information

The path and configuration of the moving load should have been defined using the moving load generator described previously. The discrete loads at distances along the path now need to be converted into equivalent modal forces which are imported into IMDPlus. This is carried out using the modal force calculator accessed from the Setup Modal History Information button on the main IMDPlus control form.

The screenshot shows the 'IMDPlus Modal Force Calculator' dialog box. It is divided into three main sections: 'Modal force input options', 'Modal force output options', and 'Moving load path options'.
- **Modal force input options:** 'First file containing eigenvalues' is set to '1:IMDPlus Rail Bridge.mys'. 'Total number of eigenvalue files = 1'. 'First file containing static load cases' is set to '2:IMDPlus Rail Bridge_00001.mys'. 'Number of load cases in each static file = 100'.
- **Modal force output options:** 'Basename for modal force output file' is 'IMDPlus Rail Bridge'.
- **Moving load path options:** 'Start load case number' is '1:Distance = 0 (Unit Axle Load)'. 'End load case number' is '548:Distance = 27.35 (Unit Axle Load)'. 'Load case increment' is '1'. The 'Forwards' radio button is selected, and the 'Backwards' radio button is disabled.
At the bottom are 'OK', 'Cancel', and 'Help' buttons.

Moving Load Analysis

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 basename for the modal force output defaults to the filename of the original model loaded but can be changed.

The moving load path options allow the user to choose a subset of the original path defined previously using the moving load generator. By default, all of the static load cases are utilised but the start, end and load case increments can be adjusted should the user wish. Options are also available to change the direction of the load movement along the path but this option should not be used unless the load configuration is symmetrical.

Running the Moving Load Analysis

If the modal forces have been calculated using the moving load generator and modal force calculator the moving load analysis can then be solved. On entering the moving load analysis control for the first time the dialog will be filled with the default information.

The screenshot shows the 'IMDPlus Moving Load Analysis Control' dialog box. It is divided into several sections:

- Moving Load Input:** A text field for 'Modal forces associated with passage' contains 'IMDPlus Rail Bridge.mfh'. A 'Browse...' button is to its right.
- Composite Axle Definition:** An unchecked checkbox 'Use composite axle definition method' is present. Below it is a text field for 'Composite axle configuration file' with a 'Browse...' button.
- Mode / Damping Control:** 'Included modes: All' has a 'Modify...' button. 'Default damping (5%)' is checked and has a 'Modify...' button. To the right, 'Total participating mass' is listed: X-Direction: 0.22%, Y-Direction: 53.68%, Z-Direction: 58.75%.
- Solution Control:** Three text fields for 'Minimum speed', 'Maximum speed', and 'Speed increment' are on the left. On the right, 'Quiet time after passage of the moving load' is 0.0. A checked checkbox 'IMDPlus determining time step (Nyquist)' is present, along with a 'Solution time step' text field.

At the bottom, there are navigation buttons: '< Back', 'Next >', 'Cancel', and 'Help'.

Moving Load Input

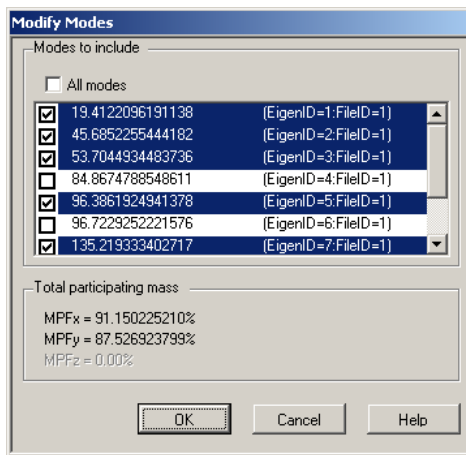
If the default modal force history file is not being used or an alternative file is going to be used then this should be input into the dialog. A composite axle definition can also be specified if this form of load configuration is to be used in IMDPlus.



Note. Although the composite axle definition method 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 composite axle file.

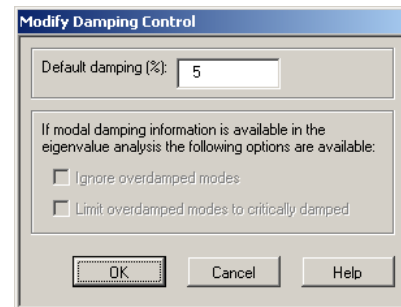
Mode And Damping Control

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.



The 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 force all modes of vibration included in the analysis to have the current default value. This default damping is set via the Modify... button on the seismic analysis dialog. The default damping option cannot be turned off unless viscous damping has been included in the eigenvalue analysis.



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

Visualising the Results

Results are extracted from IMDPlus through the output control dialog which is used by both the seismic and moving load analysis options. The appearance of the output control dialog is illustrated in the following figures for a seismic and moving load analysis. For a seismic analysis, the options for peak response summaries of Positive/Negative or Absolute are not available and the earthquake response to graph is selected through the list next to the Generate graphs in Modeller option. For a moving load analysis, the options for peak response summaries of Positive/Negative or Absolute are available and the moving load speed to graph is selected through the list next to the Generate graphs in Modeller option.

IMDPlus Output Control

Node / Element Selection

Select node for displacement/velocity/acceleration calculation, element for all other quantities

Node 61

Element

GP / Node

Entity Displacement

Component DX, DY & DZ

Output Control

Response time history

Peak response summary Positive/Negative Absolute

Generate Secondary Response Spectra (SRS)

Modal combination/factor history

Generate graphs in Modeller Earthquake to graph 1

Generate textfile output

< Back Finish **Apply** Cancel Help

IMDPlus Output Control

Node / Element Selection

Select node for displacement/velocity/acceleration calculation, element for all other quantities

Node 1672

Element

GP / Node

Entity Displacement

Component DX, DY & DZ

Output Control

Response time history

Peak response summary Positive/Negative Absolute

Generate Secondary Response Spectra (SRS)

Modal combination/factor history

Generate graphs in Modeller Speed to graph 1:15.0

Generate textfile output

< Back Finish **Apply** Cancel Help

Node / Element Selection

The node or element to analyse in IMDPlus is selected in this section. If nodes or valid IMDPlus elements are selected then the node and element lists will be filled with these nodes and/or elements. Two additional options will also be available if nodes are selected. These are to process all of the selected nodes or all of the nodes in the model. If either of these options is selected for the IMDPlus analysis then the option

for generating the graphs in Modeller will be disabled and results will be saved to disk through the Generate textfile output option. A number of output file formats are supported by IMDPlus for import of these results into graphing packages which are discussed under the output control section below.

If node output is selected, results entities supported are displacements, velocities and accelerations. Depending upon the node chosen for analysis, the components available will be placed into the list. In addition to the standard single component output, options are also available to output all translational or rotational displacements, velocities and accelerations to the same graph.

If element output is selected, the results entities supported are governed by the element type chosen. For example, choosing a thick shell element allows selection of the Stress (Stress resultants such as N_x , M_x and S_x), 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 the element. For a complete list of the output supported refer to the list of element supported by IMDPlus.

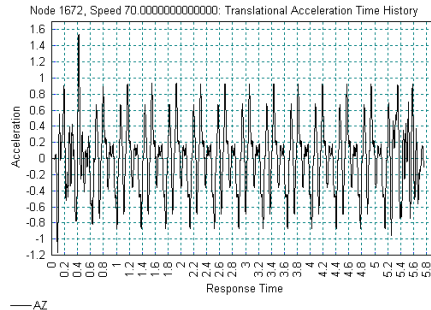
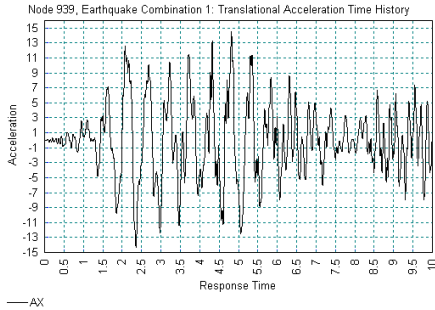
Output Control

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

- Response time history outputs displacement / velocity / acceleration time histories for nodes or outputs force / stress / strain time histories for elements.
- Peak response summary outputs maxima / minima / absolute peak responses and times of displacements / velocities / accelerations for nodes or forces / stresses / strains for elements.
- Generate Secondary Response Spectra outputs Secondary Response Spectra (SRS) based on the acceleration response for nodes. SRS output is not available for displacements or velocities at nodes and are not valid for elements.
- 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 command file which can be imported into Modeller to define Modal Combinations of the modes of vibration.
- Generate graphs in Modeller presents the results from the IMDPlus analysis as graphs immediately after completion of the analysis. The earthquake / speed to graph is selected from the list of available records.
- Generate textfile output saves the IMDPlus results to the current working 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 all earthquakes / speeds specified in the previous dialog.



Time History Plot For Seismic Analysis

Time History Plot For Moving Load Analysis

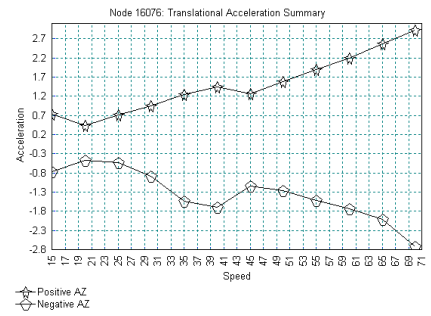
Peak Response Summary

The peak response summaries are computed by IMDPlus for all earthquakes / speeds specified in the previous dialog.



Note. Graphs can only be generated for a moving load analysis. For a seismic analysis the results are presented in tabular format

#	Entity	Time	Value	Time	Value
		Record 1		Record 2	
1	Peak_Pos_AX	4.815000000	14.48218000	9.360000000	8.351762150
2	Peak_Neg_AX	2.355000000	-14.25790481	9.595000000	-7.371060000
3	Abs_Peak_AX	4.815000000	14.48218000	9.360000000	8.351762150
4	Peak_Pos_AY	1.825000000	6.398925195	9.405000000	1.722979810
5	Peak_Neg_AY	2.118000000	-5.86476652	9.350000000	-1.941150800
6	Abs_Peak_AY	1.825000000	6.398925195	9.405000000	1.722979810
7	Peak_Pos_RRZ	1.825000000	-2.482118902	9.410000000	0.462090272
8	Peak_Neg_RRZ	1.825000000	-2.482118902	9.370000000	-0.5864685794
9	Abs_Peak_RRZ	1.825000000	2.482118902	9.370000000	0.5864685794

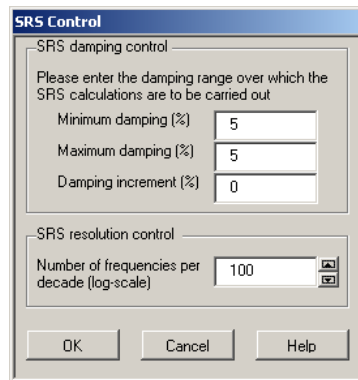


Peak Response Text Output For Seismic Analysis

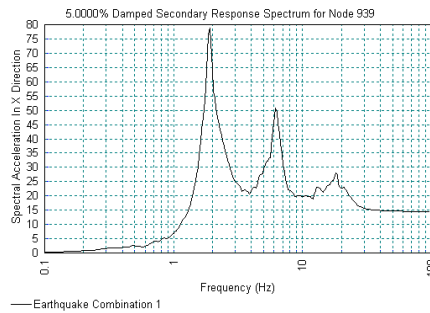
Peak Response Plot For Moving Load Analysis

Generate Secondary Response Spectra

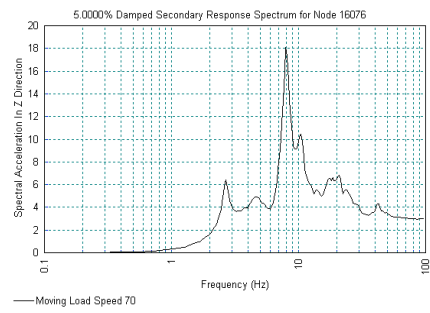
The Secondary Response Spectra (SRS) are computed for the selected node from the time history acceleration response. The damping ratio range for the SRS calculations along with the frequency resolution are accessed via the Modify... button. In this dialog the minimum and maximum damping ratios can be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



SRS plots appear as follows:



SRS Plot For Seismic Analysis



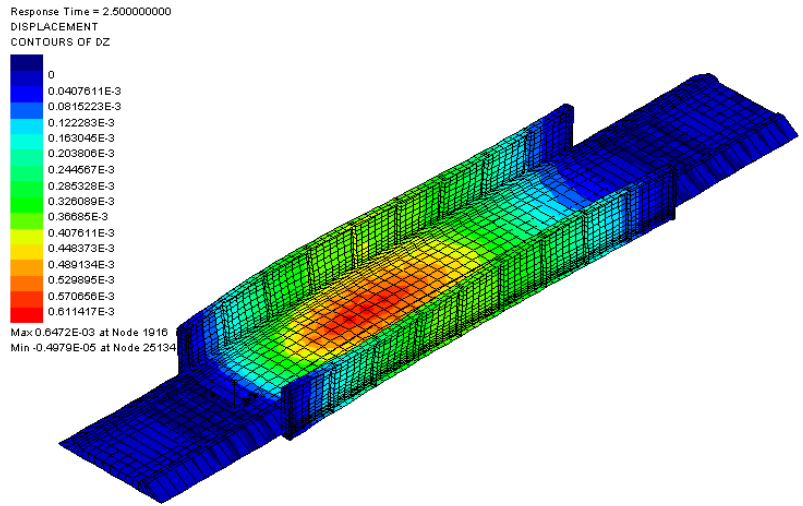
SRS Plot For Moving Load Analysis

Modal Combination/Factor History

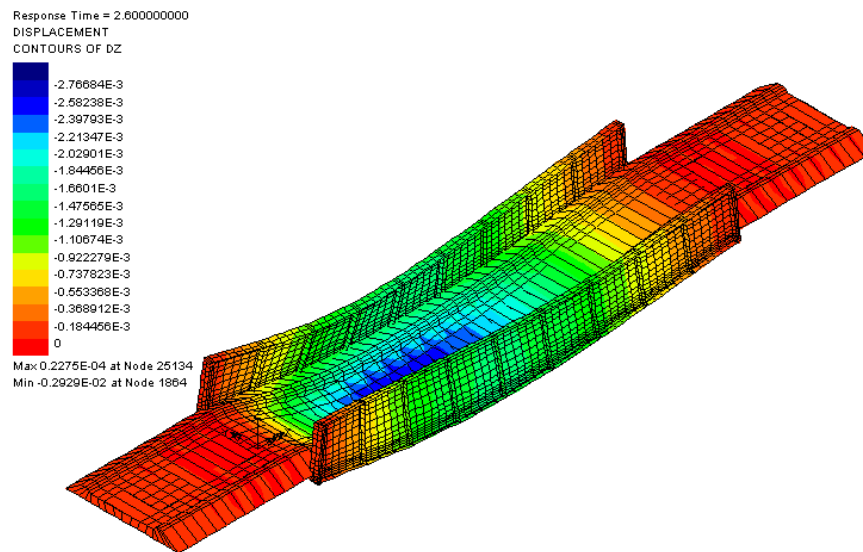
Modal combination/factor history outputs modal factors for the response of the structure at each time step of the IMDPlus analysis. The output takes the form of a VB Script file which can be imported into Modeller to define modal combinations. These combinations can then be used to visualise the deformations and produce contour plots of entities other than velocities and accelerations.



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.



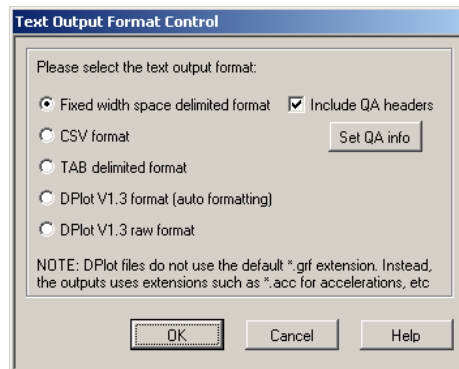
Vertical displacement plot for a time of 2.5s



Vertical displacement plot for a time of 2.6s

Generate Textfile Output

If the option is selected to generate textfile output, the results from the IMDPlus analysis will remain in the current working 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 in the default IMDPlus format which consists of all 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.**
- DPlot V1.3 format** with automatic graph formatting. (Default DPlot *.grf extension not used)
- DPlot V1.3 raw format** with minimal graph formatting. (Default DPlot *.grf extension not used)

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
- *.srs** - Secondary Response Spectra
- *.fce** - Forces/Forces and Moments
- *.str** - Stresses/Stress Resultants
- *.stn** - Strains/Strain Resultants
- *.sum** - Peak response summary



Note. The current version of DPlot is available from www.dplot.com. Alternatively, DPlot Viewer can be downloaded from this site which allows DPlot files to be viewed and printed.

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.

- 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
- 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

