ModalVIEW

ModalVIEW - Operating Tutorial

ABSignal

Version R2 (Rev 2012-02)

Information

Information in this document is subject to change without notice and does not represent a commitment on the part of ABSIGNAL.

Warranty
The media on which you receive ABSIGNAL software are warranted not to fail to execute programming instructions, due to defects in materials and workmanship. ABSIGNAL, at its option, repair or replace software media that do not execute programming instructions if ABSIGNAL receives notice of such defects. ABSIGNAL does not warrant that the operation of the software shall be uninterrupted or error free.

ABSIGNAL believes that the information in this document is accurate. The document has been carefully reviewed for technical accuracy. In the event that technical or typographical errors exist, ABSIGNAL reserves the right to make changes to subsequent editions of this document without prior notice to holders of this edition. The reader should consult ABSIGNAL if errors are suspected. In no event shall ABSIGNAL be liable for any damages arising out of or related to this document or the information contained in it.

Copyright
This publication may not be reproduced or transmitted in any form, electronic or mechanical, including photocopying, recording, storing in an information retrieval system, or translating, in whole or in part, without the prior written consent of ABSIGNAL. You may make copies of the software only for backup or archival purposes.

Support
Customers with valid software licenses will obtain free and unlimited email support for any kind of technical problem or question customer may have when using ABSIGNAL software. All support requests will be replied within two business days.

For technical support, contact ABSIGNAL at:
E-mail: support@absignal.com
Web: www.absignal.com/support/index.php

Table of Content

1. Browsing ModalVIEW

When ModalVIEW is launched for the first time, it must be activated before further operation. For the details about software activation, please refer to *Getting Started with ModalVIEW*.

1.1   Project Workbench

With a valid license, ModalVIEW can be normally started. When the ModalVIEW system is launched, the splash screen will show the information of version, copyright and license type.

The **project workbench** window, shown in figure, appears when the splash screen is closed. A default blank project is opened for you to start with. All work in ModalVIEW is done in an opened project. A project defines a group of structure file, measurement file, data file, mode shape file, report file and movie file, etc, which are generated during measurement and analysis operations.
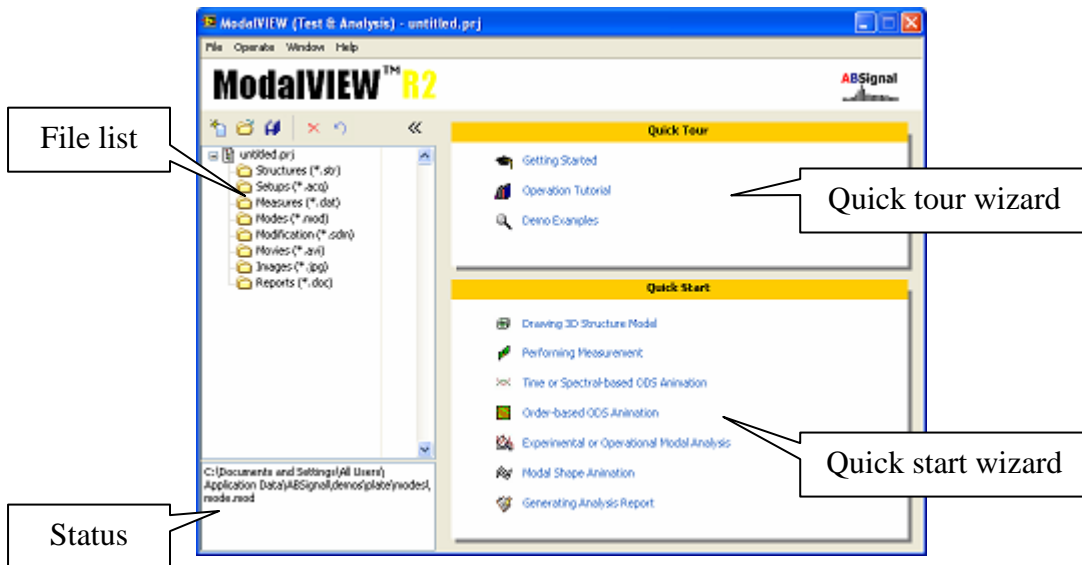

Figure 1-1 Project workbench window

There are four work areas in the **project workbench** window including file list, status, quick tour wizard and quick start wizard. The names of all the files that belong to current project will be shown in the tree list. In quick tour wizard of the project workbench, you can open help manuals and load demo examples by using wizard items. In quick start wizard, a series of wizard items guide you to perform modal testing and analysis step by step.

The **project workbench** window is minimized with quick tour and quick start work area being retracted, when other operating windows are opened, such as structure

view window, data view window, etc. The **project workbench** window is restored when you close all of other operating windows. When the project workbench window is restored, it is always floating above other operating windows.
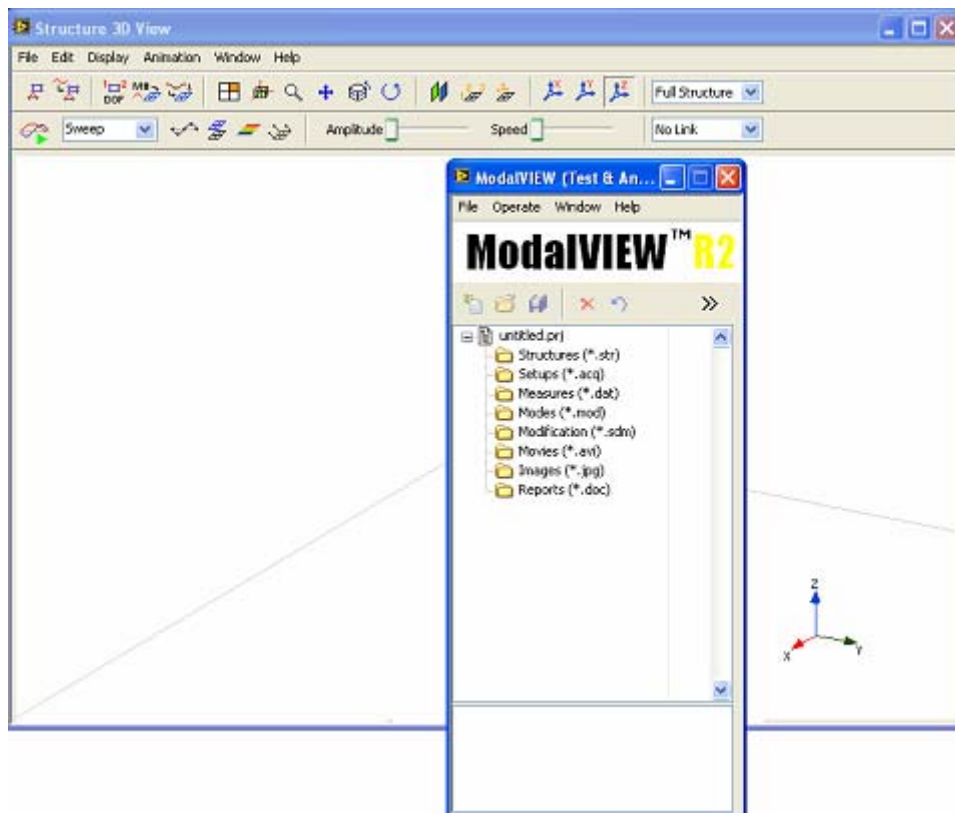

Figure 1-2 Floating project window

Project and file types

ModalVIEW uses a project to mange all kind of files used in modal testing and analysis process. File types contained in ModalVIEW project includes: structure file (*.str), measure config file(*.acq), data file(*.dat), mode shape file (*.mod), report file (*.doc) and movie file (*.avi), etc. For example, the following ModalVIEW project include a structure file (plate.str), a data file (freq response.dat) and a mode shape file (mode.mod).
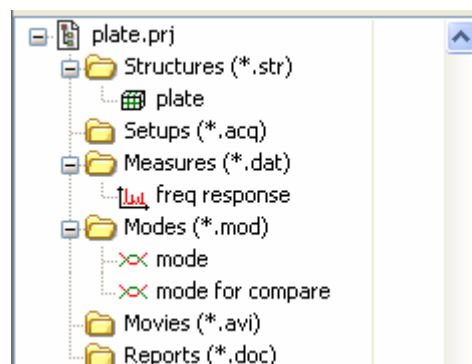
A ModalVIEW project consists of a project file, a group of files and directories on hard disk. As following figure shows, a project named 'plate' consists of a plate.prj file and a group of directories. All kinds of files in project are stored in corresponding directories.
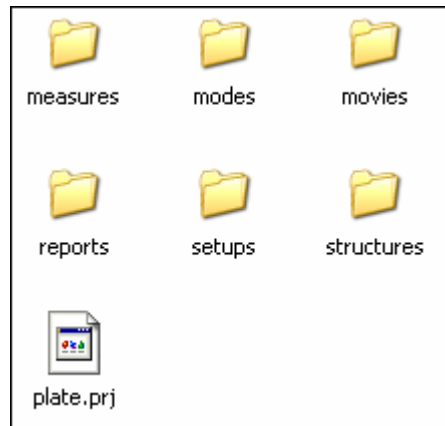
 NOTE: Do not manually delete or rename files in each directory. Please perform file operation in ModalVIEW.

Each ModalVIEW project should be stored in separate directory in order to avoid file confliction. For example, two projects (abc.prj and plate.prj) are stored in 'abc' and 'plate' directories, respectively.
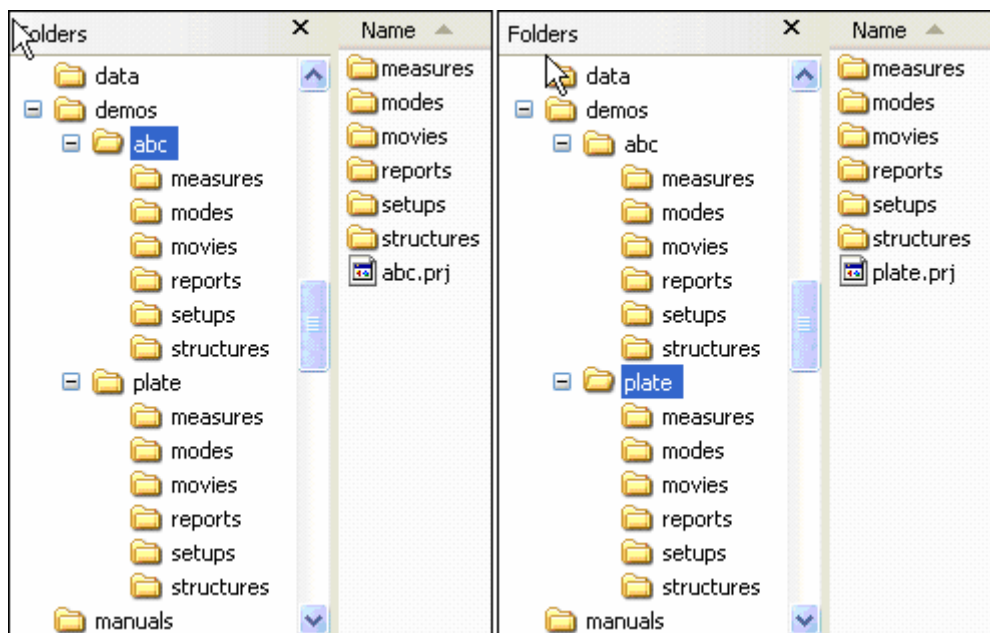
The status column in project tree displays the file status in project, as Figure 1-3 shows. **NA** represents the file don't exist on disk. **x** represents the file is marked as 'deleted'. ***** represents the file is newly added to current project.
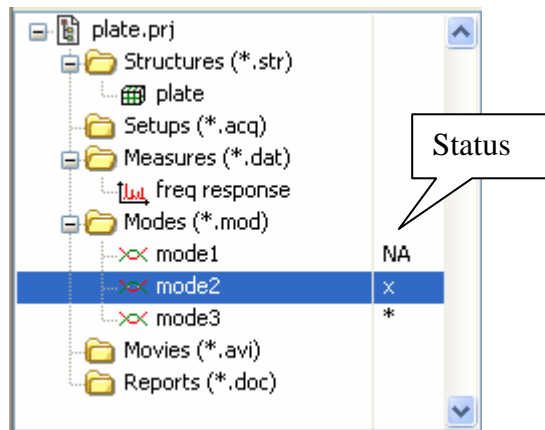
Figure 1-3 File status in project

When selecting a file in project and clicking remove button ![X], the selected file will be marked as 'deleted'. The file will still be kept on hard disk and can be restored by clicking button ![undo] before the project is saved.

After clicking project save button ![save], all the files with **NA** or **x** status will be permanently removed from current project and deleted from the hard disk, and newly added files will be copied to corresponding project directory on hard disk.

Create a new project

To create a new (empty) project

- Execute menu item **File | Project | New** in project workbench window.
- Or click **New Project** button ![new] on the toolbar.

An empty project is shown in project workbench window as following figure.
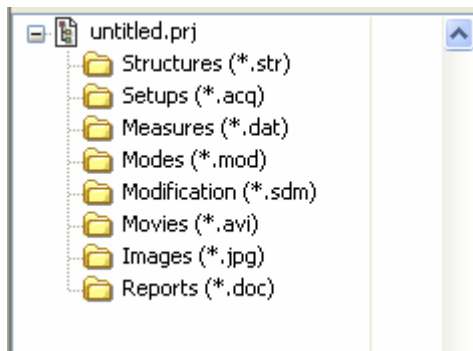
When the new project is initialized, ModalVIEW create a project file and a group of folders on the disk. The default project name is "untitled.prj". The folders are used to storing different data file such as structure file, measure config file, data file, mode shape file, report file and movie file.

Open a project

To open a previously saved project from disk

- Execute menu item **File | Project | Load…** in project workbench.
- Or click **Load Project** button ![icon] on the toolbar.

A file open dialog is displayed to prompt you to select a project file. When the project is opened, all data files belonging to current project are shown in the tree list, as Figure 1-4 shows.

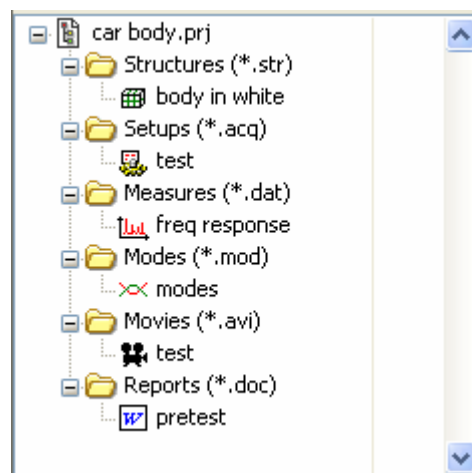**Figure 1-4 Files in project**

Open recently accessed projects

You can quickly load most recently opened ModalVIEW projects from menu items from **File | Recent Projects**, as Figure 1-5 shows.
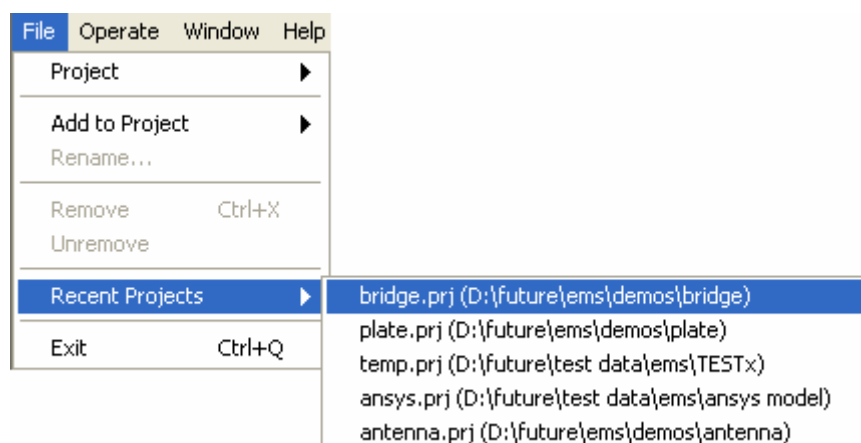

Figure 1-5 Recently opened projects

The records of five recently accessed projects are kept by default. The default number of accessed projects can be changed in the system settings of ModalVIEW.

Save current project

To save current project to its file to disk

- Execute menu item **File | Project | Save** in project workbench.
- Or click **Save Project** button ![icon] on the toolbar.

Save project operation only save dependencies for contained data files to project file. This operation does not save each data file to its disk file. If the project is newly created, ModalVIEW prompt you to save it as a new project file first.

To save current project to disk with a new project file name

- Execute menu item **File | Project | Save As…** in project workbench.

A file open dialog is displayed to prompt you to select file path and desired new file name. Type the file name in **File name** box and click **Save As** button. ModalVIEW create a new project file and a group of subfolders on the disk. All the data files contained in project are duplicated and saved in the corresponding folders.

> ![note icon] **NOTE: To avoid conflict of created subfolders for each project, create a separate folder for each project file on the hard disk.**

Add data file to project

Besides creating new data files using ModalVIEW, previously saved data files can be added to current project. Execute menu items as Figure 1-6 shows to add different data file to current project.
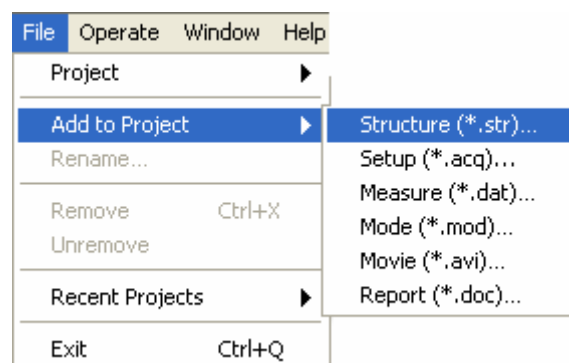

Figure 1-6 Add file to project menu

After data file is added to current project, the file name is displayed in tree list with an asterisk mark as Figure 1-7 shows. The newly added data file is not copied to corresponding subfolder associated with current project until current project file is saved and asterisk mark disappears.
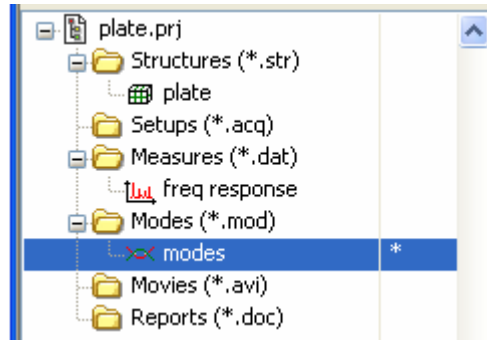
Figure 1-7 Status of newly added file

Rename data file in project

To rename data file in current project,

- Select corresponding file name in tree list, execute menu item **File | Rename…** in project workbench.

- Or click **Rename File in Project** button on the toolbar.

A dialog box is shown to prompt you to input a new file name. Type the new file name in **New File Name** box and click **OK** button, the data file name is changed accordingly.

> 🗒 **NOTE: The newly added files can not be renamed.**

Remove data file from project

To remove selected data file from project

- Select the file to be removed in the file tree list.
- Execute menu item **File | Remove** in project workbench.

- Or click **Remove** button on the toolbar.

After the file is moved from current project, the file name is displayed in tree list with an **x** mark as Figure 1-8 shows. The newly removed data file is not deleted from

corresponding subfolder associated with current project until current project file is saved and **x** mark disappears.
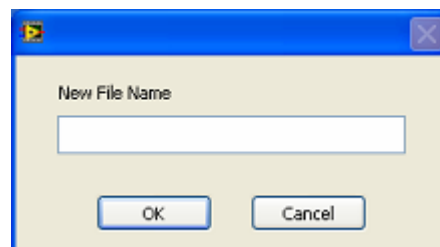

Figure 1-8 Status of deleted file

Before the project is saved, the removed data file can be recovered back to the current project.

 To recover removed data file back into current project

- Select the removed file in the file tree list.
- Execute menu item **File | Unremove** in project workbench.

- Or click **Unremove** button on the toolbar.

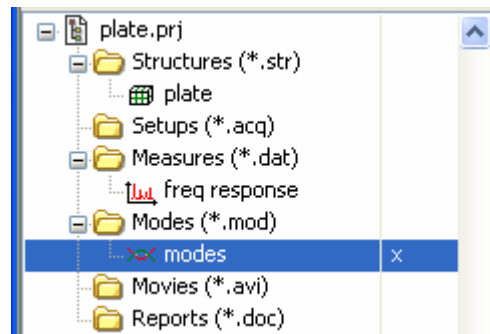## Open measurement window

To open the measurement window

- Execute menu item **Operate | Measurement…** in project workbench.

For operations in measurement window, please refer to section **Measurement Window**.

Open structure view window

To open the structure window

- Execute menu item **Operate | Structure View…** in project workbench.

You can also double click the structure file in the **Structures** folder in the tree list. The structure window will be opened with the selected structure file being loaded.

For operations about structure view window, please refer to section **Structure View Window**.

Open data block window

To open the data block window

- Execute menu item **Operate | Data View…** in project workbench.

You can also double click the data file in the **Measures** folder in the tree list. The data block window will be opened with the selected data file being loaded.

For operations about data view window, please refer to section **Data Block Window**.

Open mode view window

To open the mode view window

- Execute menu item **Operate | Mode View…** in project workbench.

You can also double click the mode file in the **Modes** folder in the tree list. The mode view window will be opened with the selected mode file being loaded.

For operations about mode view window, please refer to section **Mode View Window**.

Open modification view window

To open the structure modification view window

- Execute menu item **Operate | Modification View…** in project workbench.

You can also double click the structure modification file in the **Modification** folder in the tree list. The modification view window will be opened with the selected structure modification file being loaded.

For operations about modification view window, please refer to section **Modification View Window**.

Open image view window

To open the image view window

- Execute menu item **Operate | Image View…** in project workbench.

You can also double click the image file in the **Images** folder in the tree list. The image view window will be opened with the selected image file being loaded.

For operations about image view window, please refer to section **Image View Window**.

Open report generation window

To open the report generation window

- Execute menu item **Operate | Report Generation…** in project workbench.

For operations about report generation window, please refer to section **Report generation**.

Open report file

If Microsoft Word is installed, when double clicking the report file in the **Reports** folder of tree list, it will be opened in Microsoft Word.
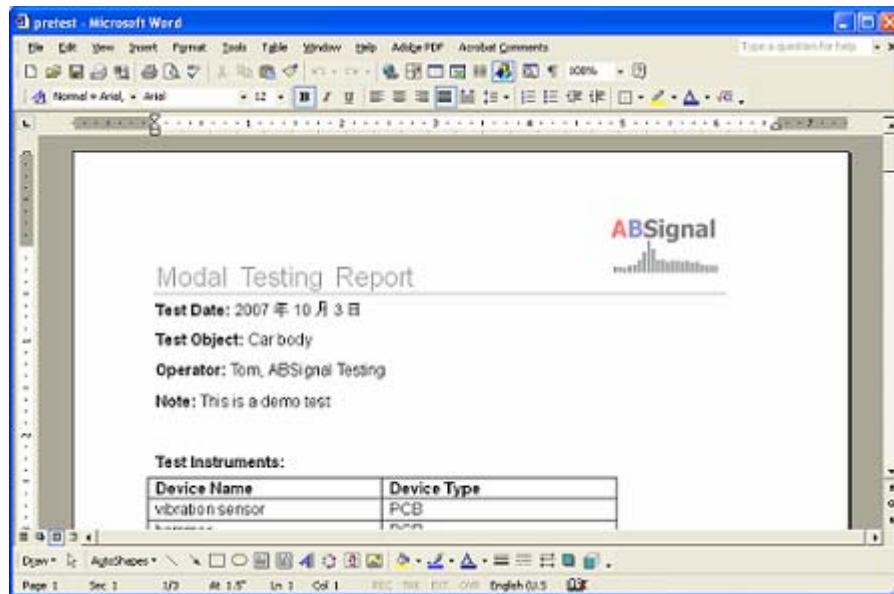
Open movie file

When double clicking the AVI file in the **Movies** folder of tree list, the AVI file will be opened in an associated media player.

Change system settings

To change system settings

- Execute menu item **Operate | Options…** in project workbench.

A dialog window is opened for you to change settings. Some settings do not take effect until you close ModalVIEW and start it again. For operations about system settings, please refer to section **System Configuration**.

Window arrangement

To change viewing of opened windows

- Execute menu items under **Window** menu in project workbench

There are four different viewing arrangements: for animation (tile left and right), tile windows, cascade windows, and center windows.

Use quick wizards

Project workbench provides some quick wizards to guide you through a series of operations, and finish modal testing experiment, analysis and report quickly and easily.
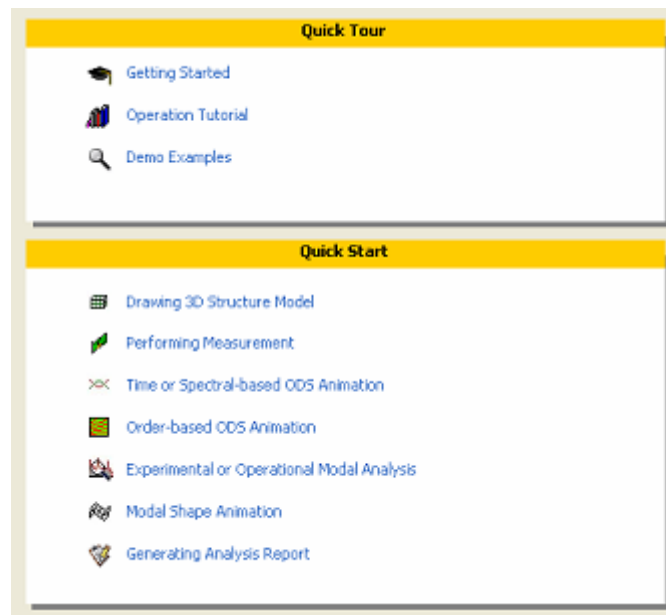
To execute each wizard operation, move cursor to wizard item and click it.

- **Getting Started** opens *Getting started with ModalVIEW* Manual in PDF reader.
- **Operational Tutorial** opens *ModalVIEW Operational Tutorial* Manual in PDF reader.
- **Demo Examples** opens example finder dialog window for you to select examples for demonstration, as Figure 1-9 shows.
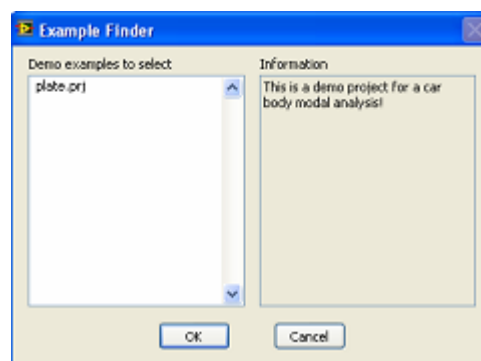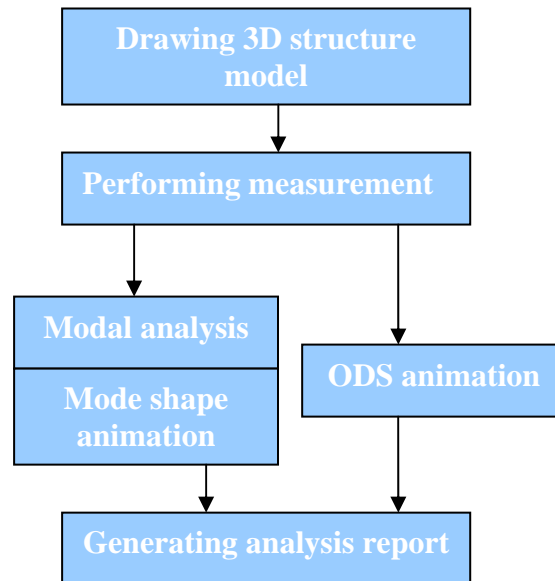

Figure 1-9 Example finder

Quick start wizard items provide user guidance for each step of modal testing and analysis process, as following flow chart shows.

1.2 Measurement Window

Measurement window is used to perform data acquisition from the hardware front end in modal testing experiment, as Figure 1-10 shows.
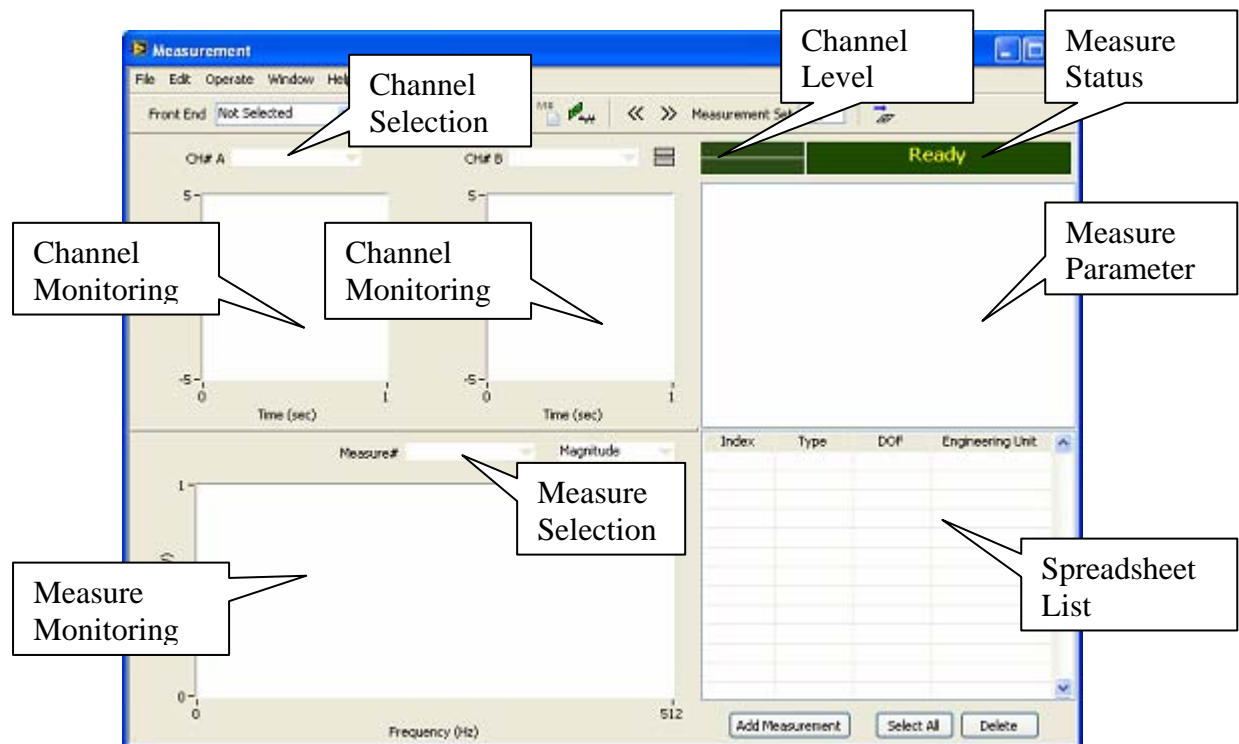

Figure 1-10 Measurement window

Measurement window contains displays of channel monitoring, measure monitoring, measure parameter, measure status and spreadsheet list. Channel monitoring is to display the data waveform acquired from selected channel. Measure monitoring is to display the time record or FRF data newly measured or selected from measure list. Double-hit detection displays the hammer impact status. Measure status displays the status of data measuring process in modal testing experiment. Spreadsheet list displays the measured time record or FRF data during testing experiment.
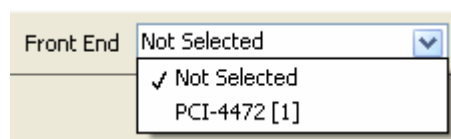
Connect acquisition front end

To refresh the available acquisition front ends in the **Front End** list

- Execute menu item **Operate | Refresh Front End**
- Click button on the toolbar.

To connect measurement window with acquisition front end

- Click **Front End** list box on the toolbar.

The available data acquisition hardware will be displayed in the list box and are ready to use. The digit in the square brackets indicates the number of installed hardware boards with same type.

 NOTE: ModalVIEW only support National Instruments DSA hardware.

- Select an acquisition front end in the list.

After the front end being connected, the menu item **Operate | DAQ Setup…** and

Operate | Test Setup… and the button on toolbar are enabled.

To disconnect measurement window from acquisition front ends

- Click **Front End** list box on the toolbar.
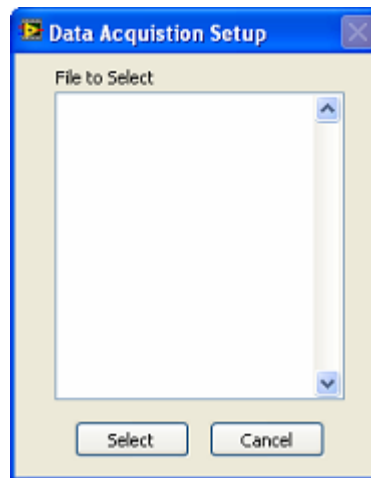- Select **Not Selected** item in the front end list box.

Load configuration file

To load previously saved configuration file for measurement

- Execute menu item **File | Load Config…**

A file open dialog is displayed to prompt you to select a configuration file in the project.

NOTE: If selected configuration file mismatch available front ends, a warning dialog will popup.
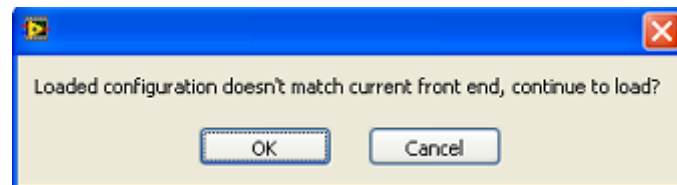
Save configuration file

To save current data acquisition and test settings to configuration file

- Execute menu item **File | Save Config**

If current configuration is newly created, ModalVIEW prompt you to save it as a new configuration file first.

To save current configuration to disk with a new configuration file name

- Execute menu item **File | Save Config As…**

A dialog box is displayed to prompt you to input desired new file name. Type the new file name in **File Name to Save** box and click **Save** button, the configuration file is saved accordingly.
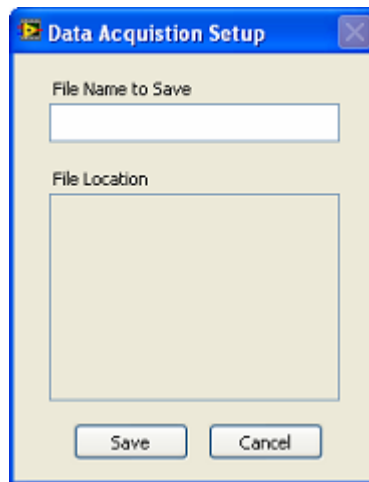
NOTE: If a duplicated file name exists in the project, a warning dialog is popup to prompt you to select another file name.
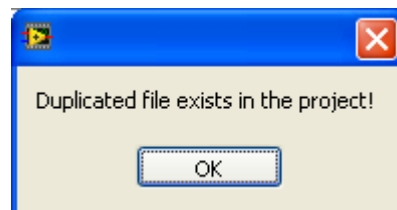
Copy channel settings to clipboard

To copy channel settings to Windows clipboard in spreadsheet format

- Execute menu item **File | Copy to Clipboard | Channel spreadsheet…**

The information of channel settings can be pasted from clipboard to other spreadsheet program, such as Microsoft Excel.

Copy settings to report generator

To copy channel settings to report generator

- Execute menu item **File | Copy to Report | Channel settings…**

A dialog box is displayed to prompt you to input desired report item name. Type the name in **Report item name** box and click **Ok** button, the channel settings is added to report item list accordingly.
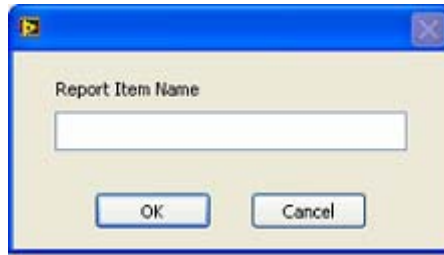
When report generator window being opened, the added report items are shown in **Available Report Items** list. Please refer to section **Report generation** for more details.
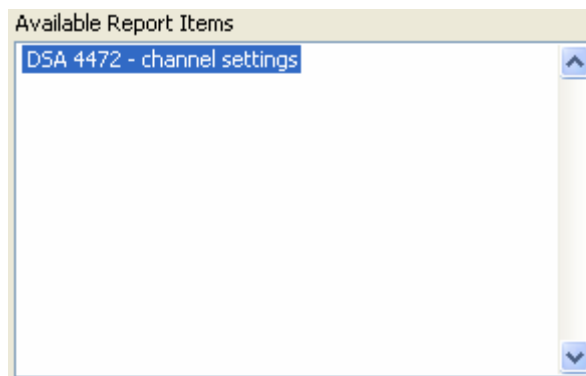
To copy test settings to report generator

• Execute menu item **File | Copy to Report | Test settings…**

Save measured data

To save measured time record or FRF data listed in the spreadsheet list to its file on disk

• Execute menu item **File | Save Measurement…**

A dialog box is displayed to prompt you to input desired new file name.

Type the new file name in **File Name to Save** box and click **Save** button, the measured data is saved accordingly.
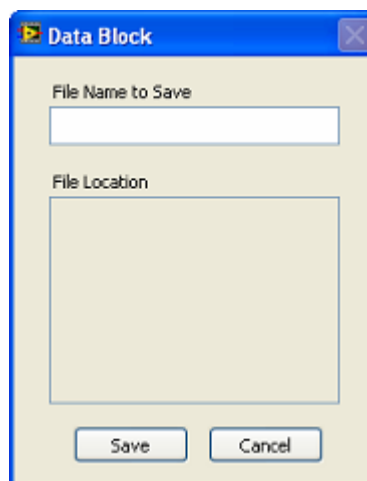
Export measured data

To export measured time record listed in the spreadsheet list into TDMS file

- Execute menu item **File | Export | TDMS…**

A file open dialog box will open for you to create a TDMS file.

> **NOTE: Only measured time record can be exported into TDMS file.**

Add measured data to spreadsheet

To save newly measured data into data spreadsheet list

- Execute menu item **Edit | Add Measurement**
- Or click button Add Measurement at the bottom of spreadsheet list.

When newly measured data is added to data spreadsheet list, the symbol "✚"for each row of data disappears, as following figure shows.

Index	Type	DOF	Engineering Unit
✚ 0	FRF	1Z : 1Z	V / V
✚ 1	FRF	2Z : 1Z	V / V

Index	Type	DOF	Engineering Unit
0	FRF	1Z : 1Z	V / V
1	FRF	2Z : 1Z	V / V

> **NOTE: If the newly measured data are not added to spreadsheet list, they will be overwritten by next measurement.**

Edit spreadsheet list of measured data
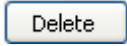
To select all measured data in spreadsheet list

- Execute menu item **Edit | Select All**
- Or click button **Select All** at the bottom of spreadsheet list.

To unselect all measured data in spreadsheet

- Execute menu item **Edit | Select None**

To delete measured data from spreadsheet list

- Select one or more items in spreadsheet list.
- Execute menu item **Edit | Delete**
- Or click button **Delete** at the bottom of spreadsheet list.

Properties of measured data

To view or edit properties of measured data

- Select one item in spreadsheet list
- Execute menu item **Edit | Properties…**
- Or double click one item in data spreadsheet

Data block properties window is opened as Figure 1-11 shows.

Figure 1-11 Data properties window

Configure data acquisition

After front end being connected, data acquisition parameters need to be properly configured. To setup data acquisition parameters

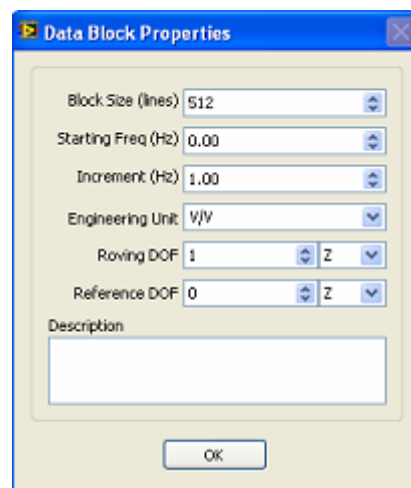- Execute menu item **Operate | DAQ setup…**
- Or click button on toolbar.

DAQ Setup window is opened, as Figure 1-12 shows. The window has three tab pages, **Channel**, **Acquisition** and **Calibration**.
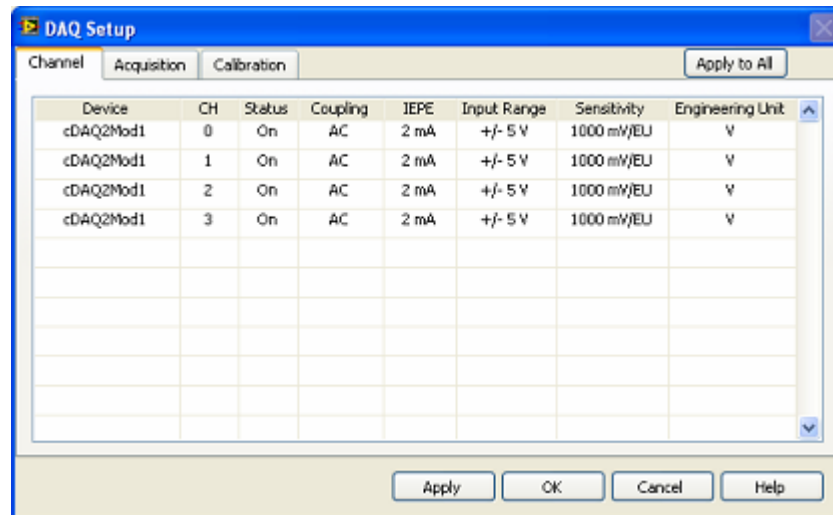

Figure 1-12 Data acquisition setup window

Channel tab contains channel spreadsheet. Each row contains all properties for a single channel. Each channel is defined by device name and channel index. The device name for front end is assigned by NI Measurement & Automation Explorer (MAX).

To change the channel properties

- Select a row in spreadsheet, double click corresponding property item to change.

To change the channel properties in batch

- Select a row in spreadsheet, click ⌊Apply to All⌋ button.

The properties of all the channels below the selected channel in the spreadsheet list will be automatically changed as the same as the properties of selected channel.

Acquisition tab contains acquisition settings for sampling rate, block size, trigger etc, as Figure 1-13 shows. Please refer to the chapter **Performing Measurement** for details about configuring these parameters.

Figure 1-13 Acquisition tab page

Calibration tab contains the spreadsheet of enabled channels. Each row contains a single channel for calibration. The calibrated sensitivity can be set into channel properties. Please refer to the section **Perform channel calibration** for details about the process of calibrating an input channel.

Figure 1-14 Calibration tab page

Setup testing experiment

Before starting modal testing experiment, measurement information such as testing type, node number, DOF range etc, data processing such as windowing, averaging etc, must be specified. According to specified testing information, ModalVIEW will automatically create measurement sets, which define connections between acquisition channels, sensors and measure points (DOFs) on the structure under test, to guide you to finish measurement in batches.

To setup testing parameters

- Execute menu item **Operate | Test setup…**
- Or click button ![M#] on toolbar.

Test Setup window is opened as Figure 1-15 shows. The window has three tab pages, **Measurement, Post-process** and **Measurement Sets**.
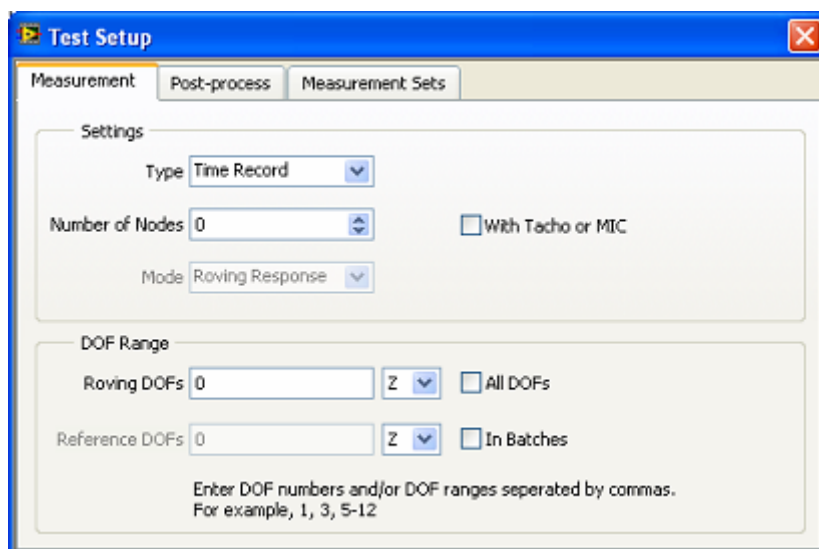
Figure 1-15 Test setup window

Measurement tab contains measurement settings for test type, node number, test mode, DOF range etc. Please refer to the chapter **Test setup** for details about configuring these parameters.

Post-process tab contains settings for windowing, data averaging etc, as Figure 1-16 shows. Please refer to the chapter **FRF Measurement** for details about configuring these parameters.

Figure 1-16 Post process tab page

Measurement Sets tab contains the spreadsheet list for generated measurement sets, as Figure 1-17 shows.
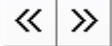
Figure 1-17 Measurement sets tab page

To generate measurement sets

- Click button [Generate] in measurement set tab, and click button [« »] to view different set.

By default, starting from first acquisition channel, ModalVIEW firstly assigns acquisition channels for measuring reference signals (from fixed excitation DOF or response DOF), then for measuring signals from roving points.
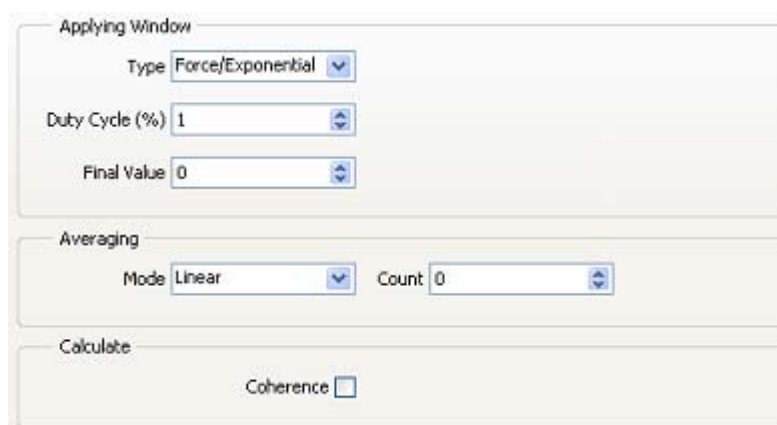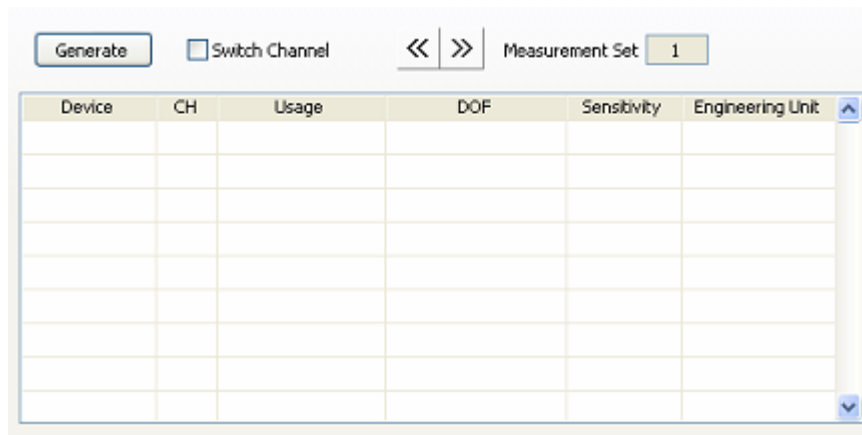
When [□ Switch Channel] is checked, ModalVIEW will assigns acquisition channels from last channel.

To manually change the setting of each channel in measurement set

- Select a row in spreadsheet list, double click a property item and change it.

Start & stop acquisition

To start data acquisition

- Click button [▶II] on toolbar or select menu item **Operate | Start**

To stop data acquisition

- Click button [▶II] on toolbar or select menu item **Operate | Stop**

NOTE: When data acquisition is started and in trigger-waiting state, you still can stop acquisition.

Data logging

Data logging is enabled when data acquisition is in scope mode and measurement type is Time Record. To continuously log measured data to disk after data acquisition is started

- Click button ![data logging icon] on toolbar

The logged data length in second is displayed in measurement status window.

View measurement set

To view previous measurement set

- Click button ![«] on toolbar or select menu item **Operate | Previous Set.**

To view next measurement set

- Click button ![»] on toolbar or select menu item **Operate | Next Set.**

The current measurement set index is displayed in ![Measurement Set 0] on the toolbar.

You can view status of current measurement set from status window, as Figure 1-18 shows.

```
-----------Measurement Status------------
Enabled Channels: 4
Sampling Rate: 1024 Hz
Block Size: 1024
Start Mode: Free Run
Measure Type: FRF - EMA
Excitation DOFs:
0Z
Response DOFs:
0Z  1Z  2Z
Average Count: 0 / 5
```

Figure 1-18 Measure status display

Modify monitor graph display

To change arrangement of channel monitor

- Click **Arrange Monitor View** button ![arrange icon] above channel monitor

The the view of channel monitors will be arranged in vertical or horizontal.

To modify the plot display of channel monitor

- Right click mouse button on the corresponding channel monitor, a floating popup menu will display as following

- Select menu item **Zoom In X**, press mouse down and drag it along X axis. The data plot will be zoomed in along X axis
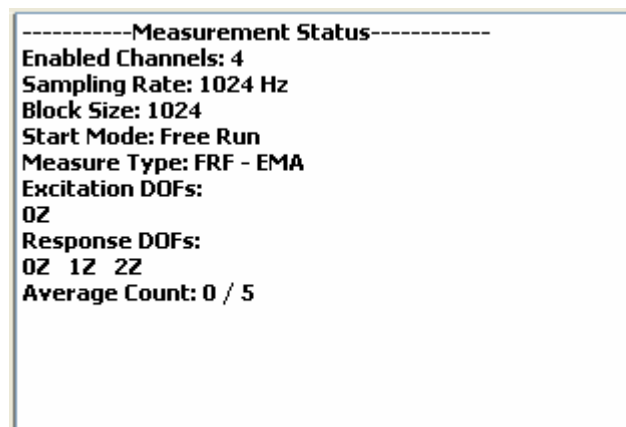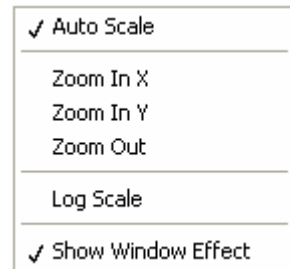.
- Select menu item **Zoom In Y**, press mouse down and drag it along Y axis. The data plot will be zoomed in along Y axis.

- Select menu item **Zoom Out**, the data plot will be zoomed out into its original state.

- Select menu item **Log Scale**, the mapping of data plot along Y axis will be changed from linear into logarithmic.

- Select menu item **Show Window Effect**, the data plot of measured signal with applied window will be displayed.

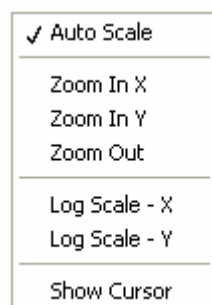To modify the plot display of measure monitor

- Right click mouse button on the corresponding measure monitor, a floating popup menu will display as following

- Select menu item **Zoom In X**, press mouse down and drag it along X axis. The data plot will be zoomed in along X axis
.
- Select menu item **Zoom In Y**, press mouse down and drag it along Y axis. The data plot will be zoomed in along Y axis.

- Select menu item **Zoom Out**, the data plot will be zoomed out into its original state.

- Select menu item **Log Scale - X**, the mapping of data plot along X axis will be changed from linear into logarithmic.

- Select menu item **Log Scale - Y**, the mapping of data plot along Y axis will be changed from linear into logarithmic.

- Select menu item **Show Cursor**, a vertical line cursor will displayed on the data plot.

To specify display mode to view measured data

- Select the listbox Magnitude above measure monitor. The real part, imaginary part, magnitude, phase and bode plot of measured FRF data can be displayed.

Generate stimulus signal

To generate stimulus signal from hardware front end

- Click button on toolbar or select menu item **Operate | Signal Generator...**

For details about signal generator, please refer to section **Excitation Window**.

Connect to structure model

To display DOFs on 3D structure model matched with acquisition channel during measurement

- Click button on toolbar, structure window is opened for you to load structure 3D model.

When measurement set being selected, the DOFs matched with acquisition channels will be indicated with measure axes as Figure 1-19 shows.
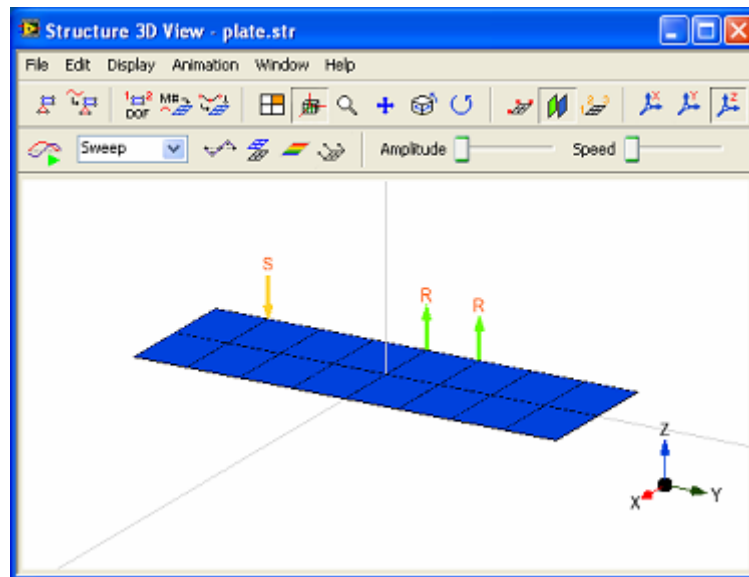
Figure 1-19 Structure measurement guide

1.3 Structure View Window

Structure view window is used to draw and display 3D structure model, and perform structure animation, as Figure 1-20 shows.
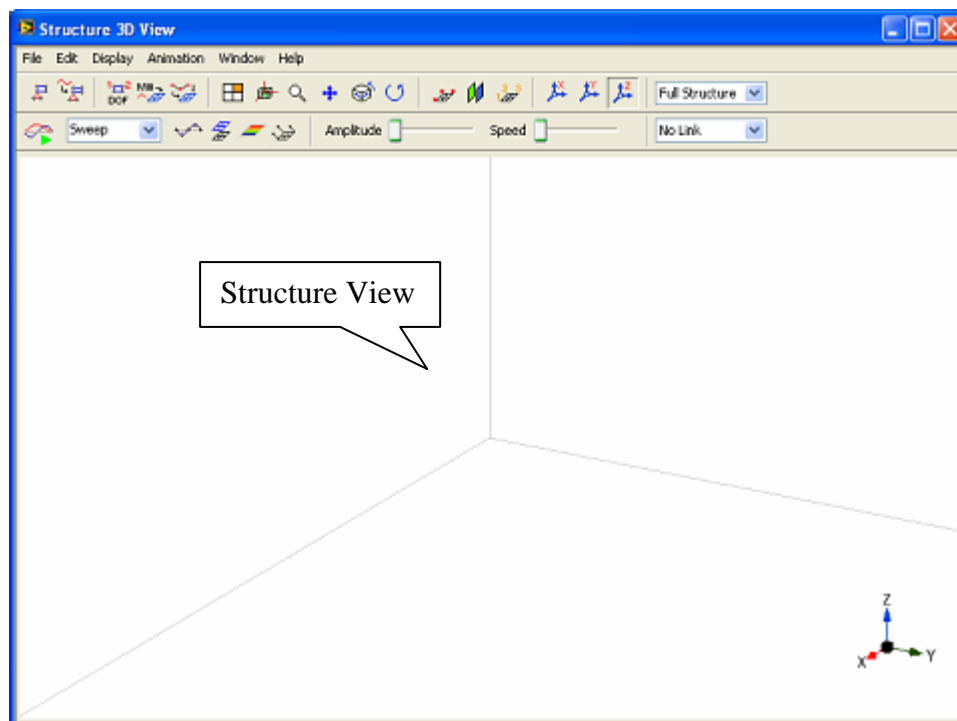
Structure View

Figure 1-20 Structure view window

Load structure file

To load previously saved structure file

- Execute menu item **File | Load…**

A file open dialog is displayed to prompt you to select a structure file in the project.
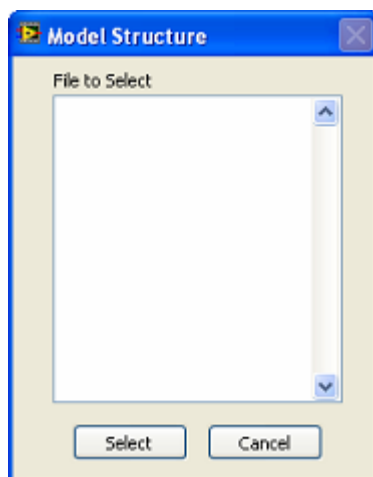
Select a file in the list and click **Select** button, 3D model of the structure will be loaded and displayed in the structure window.

Save structure model

To save structure model to its file on disk

- Execute menu item **File | Save**.

If the structure file is not saved before, ModalVIEW prompt you to save it as a new structure file first.

To save structure model to disk with a new file name

- Execute menu item **File | Save As…**.

A file open dialog is displayed to prompt you to select desired new file name.
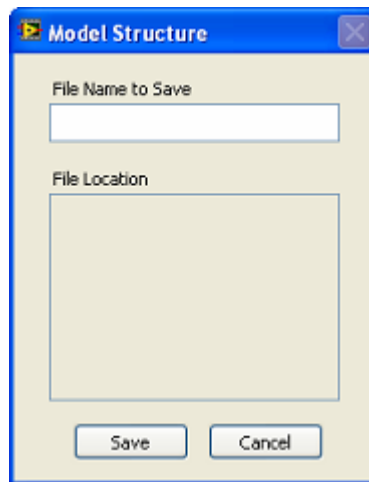
Type the desired file name into the **File Name to Save** box and click **Save** button.

NOTE: If there is a file with the same file name already in the project, a warning dialog box will be shown to prompt you to change the input file name.
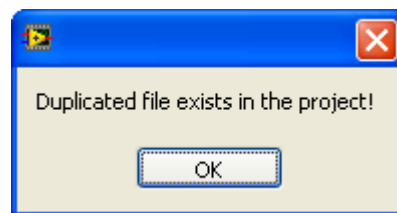
Import structure from external files

You can import the 3D structure model from other external files, show it in the structure view window and save the model data into ModalVIEW structure file.

To import structure data from ASCII file

- Execute menu item **File | Import | ASCII…**.

A file open dialog box will open for you to select ASCII structure file.

To import structure data from UFF file

- Execute menu item **File | Import | UFF…**.

A file open dialog box will open for you to select UFF structure file.

To import structure data from STL file

- Execute menu item **File | Import | STL…** in structure window.

A file open dialog box will open for you to select STL structure file.

To import structure data from SLP file

- Execute menu item **File | Import | SLP…** in structure window.

A file open dialog box will open for you to select SLP structure file.

To import structure data from Nastran bulk file

- Execute menu item **File | Import | BDF…** in structure window.

A file open dialog box will open for you to select Nastran structure file.

NOTE: Not all the data CARD defined in Nastran bulk file are supported.

For the details about UFF, ASCII, STL and SLP file format, please refer to section **Importing External Files**.

Export structure to external files

To export structure data to ASCII file

- Execute menu item **File | Export | ASCII…**.

A file open dialog box will open for you to create a new ASCII structure file.

To export structure data to UFF file

- Execute menu item **File | Export | UFF…**.

A file open dialog box will open for you to create a new UFF structure file.

Copy structure to clipboard

The 3D structure model displayed in the structure view window can be copied into Windows clipboard in Bitmap or spreadsheet data for other programs to use.

To copy 3D model into clipboard as a Bitmap

- Execute menu item **File | Copy to clipboard | Bitmap**.

To copy structure data into clipboard in tab delimited format

- Execute menu item **File | Copy to clipboard | spreadsheet**.

Copy structure to report generator

To copy the image of 3D structure model to report generator
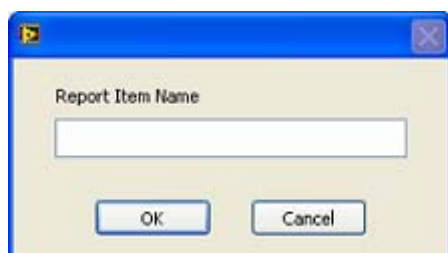
- Execute menu item **File | Copy to Report | Structure View…**

A dialog box is displayed to prompt you to input desired report item name. Type the name in **Report item name** box and click **OK** button, the 3D model view is added to report item list accordingly.

When report generator window being opened, the added report items are shown in **Available Report Items** list. Please refer to section **Report Generation Window** for more details.
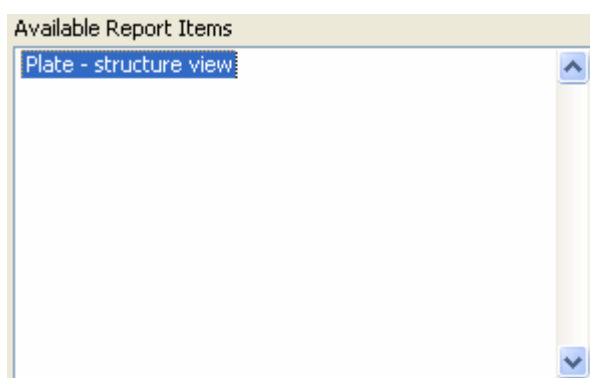
To select a group of modal shape views and copy it to report generator

- Execute menu item **File | Copy to Report | Shape View Arrangement…**

A dialog box is displayed to prompt you to insert current modal shape view in structure view window, as Figure 1-21 shows.

To add current view image into the group of mode shape images

- Input the name of shape view image, and click <u>Insert</u> button. The shape view will be added into shape view list.

To delete shape view from the shape view list

- Select one item from shape view list, and click [Delete] button. The shape view will be added into shape view list.

[Shapes per Row 3] decides how many shape views are arranged in a single row, when overall shape view image is generated for testing report.
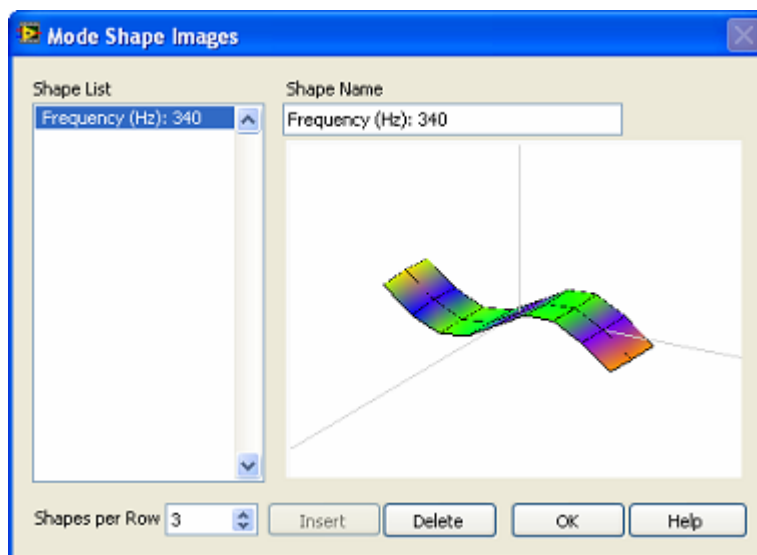
Figure 1-21 Shape view arrangement

To copy arranged mode shape images to report generator

- Execute menu item **File | Copy to Report | Mode View…**

A dialog box is displayed to prompt you to input desired report item name. Type the name in **Report item name** box and click **OK** button, the shape view is added to report item list accordingly.
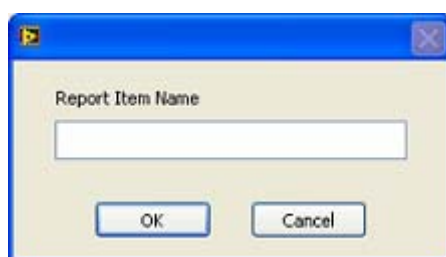
When report generator window being opened, the added report items are shown in **Available Report Items** list. Please refer to section **Report Generation Window** for more details.

Export animation movie

To create digital movie from structure view animation frames

- Execute menu item **File | Export Movie…**

A dialog box is displayed to prompt you to input desired file name. Type the name in **New File Name** box and click **OK** button.
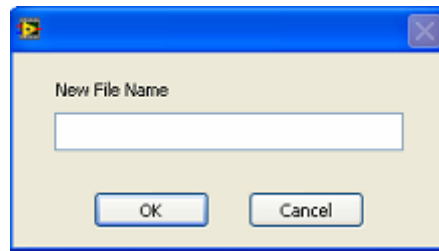
After the AVI movie is created, a message box will show to prompt the AVI movie is generated.
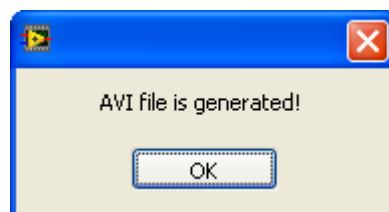
Open structure editor

To open structure editor window for 3D model drawing

- Execute menu item **Edit | Structure Editor…**
- Or click button on the toolbar

The structure editor window will be displayed at the right of structure view window. For the details about drawing 3D model structure, please refer to section **Structure Editor Window**.

Open measure properties

To open measure properties window to set measure properties for points on 3D structure model

- Execute menu item **Edit | Measure Properties…**
- Or click button on the toolbar

The measure properties window will be displayed at the right of structure view window. For the details about specifying measure properties of points on the 3D structure model, please refer to section **Measure Properties Window**.

Open structure group editor

To open structure group window to specify components and dummy substructures for 3D structure model

- Execute menu item **Edit | Structure Group…**

The structure group window will be displayed at the right of structure view window. For the details about grouping substructures in 3D structure model, please refer to section **Structure Group Window**.

Number point DOFs

To interactively number the nodes on the 3D structure model

- Execute menu item **Edit | Number DOFs…**
- Or click button ![DOF] on the toolbar

A floating dialog box **Number DOFs** is opened.
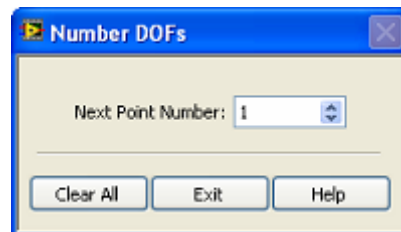
- Click one node on the 3D model, its DOF number is changed to the **Next Point Number** in the dialog box.

The **Next Point Number** is increased automatically. You can also manually change **Next Point Number** at any time.

To clear all the numbers assigned to nodes

- Click **Clear All** button on the **Number DOFs** dialog box.

To stop node numbering

- Click **Exit** button on the **Number DOFs** dialog box.

The nodes on the 3D structure model can also be automatically assigned DOF number in structure editor window. Please refer to section **Structure Editor Window** for details

Quick assign measurements

To match measurements of the opened animation source (time record, FRF or mode shape) to the nodes and its measurement direction on the 3D model

- Execute menu item **Edit | Quick Assign M#…**

- Or click button on the toolbar

A message box will show to display how many measurements are successfully assigned.

Interpolate unmeasured animation

To generate interpolated animation for the unmeasured nodes on 3D model

- Execute menu item **Edit | Interpolation…**
- Or click button on the toolbar

A dialog box will show to prompt you to specify interpolation range. That is how many nearest nodes are used to calculate interpolated animation.

 decides if interpolated motion of unmeasured nodes are computed from the measured nodes belonging to the same component.

 decides if interpolated motion of unmeasured nodes are partially computed only for specified DOFs.

Input number of nearest points in the dialog box and click **OK**. The interpolated animation is generated.

To enable display of interpolated animation during 3D model animation

- Execute menu item **Animation | Interpolation**

- Or click button ⟨icon⟩ on the toolbar

Open color map editor

To edit the color map of the defined color scale in the display of model animation

- Execute menu item **Edit | Color Map Editor…**

The color map editor window is displayed for you to specify color scale.
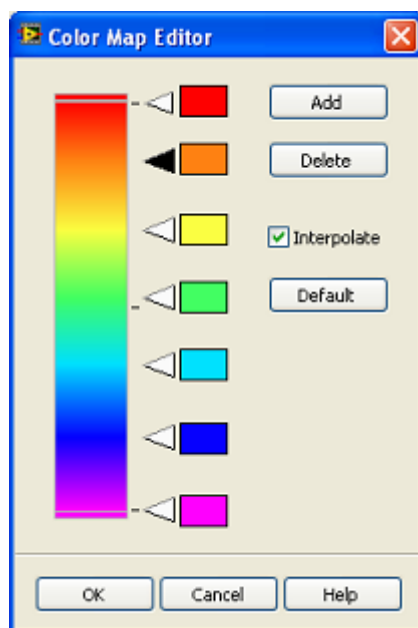
Change structure model display

To change the view mode of 3D model displayed in structure window

- Click button ⟨icon⟩ on the toolbar.
- Or select menu items in **Display | View Panel**

To display 3D model in the center of structure window

- Click button ⟨icon⟩ on the toolbar.
- Or select menu items in **Display | Center**

To zoom the 3D model in the view

- Click button ⟨icon⟩ on the toolbar, or select menu items in **Display | Zoom**
- Mouse left key down and drag cursor from top to bottom, the 3D model will zoom in.
- Mouse left key down and drag cursor from bottom to top, the 3D mode will zoom out.

- Or scroll down mouse center wheel to zoom in 3D model, and scroll up to zoom out 3D model.

To pan the 3D model in the view

- Click button [+] on the toolbar.
- Or select menu items in **Display | Pan**
- Drag mouse cursor in the view.

To rotate the 3D model in the view

- Click button [icon] on the toolbar or select menu items in **Display | View**
- Mouse left key down and drag cursor vertically.
- Or mouse down and drag cursor horizontally.

To restore the 3D model to its original view

- Click button [icon] on the toolbar.
- Or select menu items in **Display | Return**

To show points on the 3D model

- Click button [icon] on the toolbar.
- Or select menu items in **Display | Points**

To show surfaces on the 3D model

- Click button [icon] on the toolbar.
- Or select menu items in **Display | Surfaces**

To show point label of each node on the 3D model

- Click button [icon] on the toolbar.
- Or select menu items in **Display | Point Labels**

To show measurement axes of each node on the 3D model

- Select menu items in **Display | Measure Directions**

[icon] **NOTE: Point label and measurement axes can not be shown with surfaces on 3D model.**

To change the vertical direction of coordinate axis

- Click button on the toolbar.

To display dummy structures defined for the 3D model

- Select corresponding item from on the toolbar

For the details about dummy structure, please refer to chapter **Component & Dummy structure**.

Animate structure model
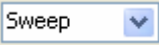
To start structure animation

- Click button on the toolbar.
- Or execute menu item **Animation | Run**

To stop structure animation

- Click button on the toolbar.
- Or execute menu item **Animation | Stop**

To change animation type

- Select type list on the toolbar.
- Or execute menu item **Animation | Mode | Sweep**
- Or execute menu item **Animation | Mode | Sine Dwell**
- Or execute menu item **Animation | Mode | Stationary**
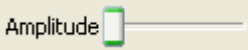
For the details about each animation type, please refer to section **Animation source**.

To change scale mode of structure deformation
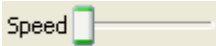
- Execute menu item **Animation | Scale | Auto**
- Or execute menu item **Animation | Scale | Relative**
- Or execute menu item **Animation | Scale | Fixed**

For the details about each scale mode, please refer to section **Deformation Scaling**.

To change the amplitude of animation

- Adjust amplitude value in on the toolbar.
- Or execute menu item **Animation | Amplitude | Increase**
- Or execute menu item **Animation | Amplitude | Decrease**

To change the speed of animation

- Adjust speed value in on the toolbar.
- Or execute menu item **Animation | Speed | Increase**
- Or execute menu item **Animation | Speed | Decrease**

To turn on / off interpolated animation of nodes on 3D model

- Click button on the toolbar.
- Or execute menu item **Animation | Interpolation**

To display deformed structure model

- Click button on the toolbar.
- Or execute menu item **Animation | Deformed**

To display color map on structure model

- Click button on the toolbar.
- Or execute menu item **Animation | Color Map**

To display arrow vector on structure model

- Click button on the toolbar.
- Or execute menu item **Animation | Arrow Vector**

To show the deformation ruler in the structure model display

- Execute menu item **Animation | Ruler | Deformed**

To show the color map of the defined color scale in the structure model display

- Execute menu item **Animation | Ruler | Color Map**

Change data display format

A few of display formats are provided to rendering scalar data on the structure model with color map.

To specify dB reference of scalar data

- Execute menu item **Animation | Format | dB Reference…**

To change scalar data between linear or dB scale

- Execute menu item **Animation | Format | Linear <> dB**

To display the magnitude part of scalar data with phase information

- Execute menu item **Animation | Format | Mag-Phase**

To display the magnitude part of scalar data

- Execute menu item **Animation | Format | Magnitude**

To display the phase part of scalar data

- Execute menu item **Animation | Format | Phase**

To display the real part of scalar data

- Execute menu item **Animation | Format | Real**

To display the imaginary part of scalar data

- Execute menu item **Animation | Format | Imaginary**

Change window arrangement

There are some predefined positions for structure view window: center, for edit, for animation and for DOF guide. In different state of analysis operation, structure view window is arranged to cooperate with other operating window.

To change the window arrangement

- Execute **Window** menu items in structure view window.

1.4 Data Block Window

Data block window is used to display and process time record or FRF measurements. This window contains data plot and spreadsheet for data properties, as Figure 1-22 shows.
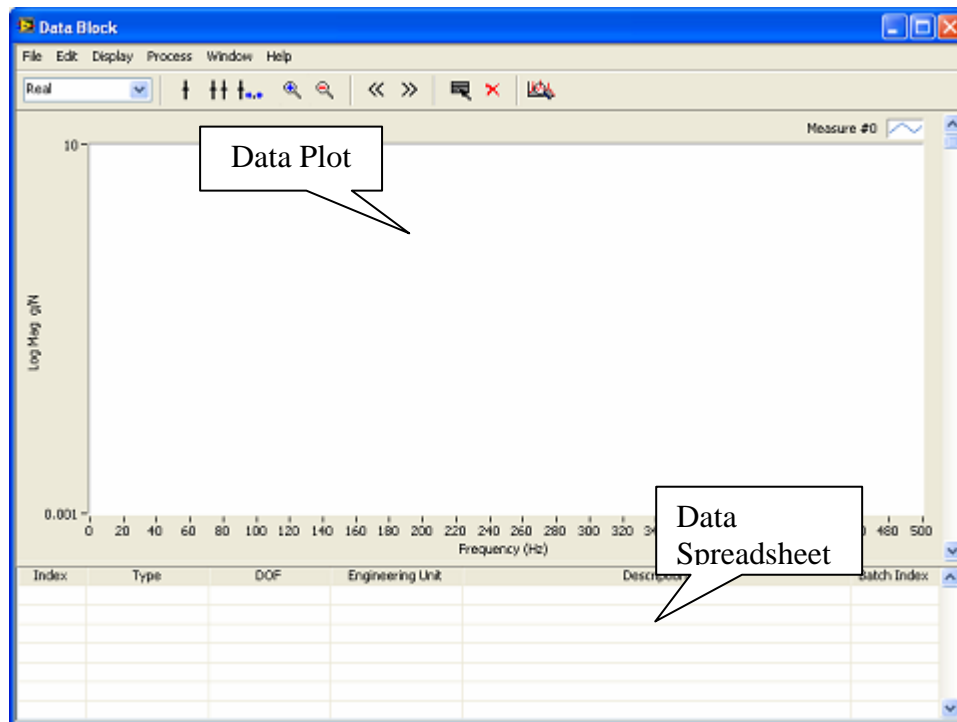
Figure 1-22 Data view window

Load data block file

To load previously saved data block file

- Execute menu item **File | Load…**

A file open dialog is displayed to prompt you to select a data block file in the project.

Select a file in the list and click **Select** button, data plots will be displayed in the data block window.

Save data block

To save data block to its file on disk

- Execute menu item **File | Save**.

If the data block file is not saved before, ModalVIEW prompt you to save it as a new data block file first.

To save data block file to disk with a new file name

- Execute menu item **File | Save As…**.

A file open dialog is displayed to prompt you to input desired new file name.
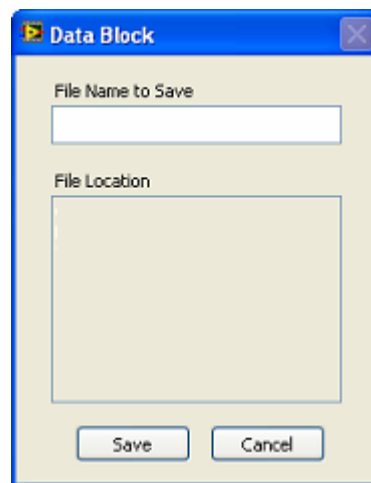
Type the desired file name into the **File Name to Save** box and click **Save** button.

> 📝 **Note: If there is a file with the same file name already in the project, a warning dialog box will be shown to prompt you to change the input file name.**
>
>
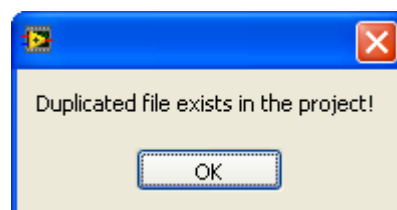
Import data block from external files

You can import data block from other external files, show the data block in the data block window and save the data block into ModalVIEW data block file.

To import data block from ASCII file

- Execute menu item **File | Import | ASCII…**.

A file open dialog box will open for you to select ASCII data file.

To import data block from multiple ASCII files in a file folder.

- Execute menu item **File | Import | ASCII File Folder…**.

A file open dialog box will open for you to select the file folder keeping data files.

To import data block from UFF file

- Execute menu item **File | Import | UFF…**.

A file open dialog box will open for you to select UFF data file.

To import data block from multiple UFF files in a file folder.

- Execute menu item **File | Import | UFF File Folder…**.

A file open dialog box will open for you to select the file folder keeping data files.

For the details about UFF and ASCII file format, please refer to section **Importing External Files**.

To import data block from TDMS file

- Execute menu item **File | Import | TDMS…**.

A file open dialog box will open for you to select UFF data file.

NOTE: Only measured time record can be imported from TDMS file.

Copy data to clipboard

To copy plot of the data block into clipboard as a Bitmap

- Execute menu item **File | Copy to clipboard | Bitmap**.

To copy properties of the data block into clipboard in tab delimited format

- Execute menu item **File | Copy to clipboard | Spreadsheet**.

To copy trace data of the data block into clipboard in tab delimited format

- Execute menu item **File | Copy to clipboard | Trace data**.

Edit data block

To select all traces in the data block

- Execute menu item **Edit | Select All**

To unselect all traces in the data block

- Execute menu item **Edit | Select None**

To select one or more traces in the data block

- Click items in the spreadsheet of data properties.
- Or hold down **Shift** key for selection of more items

To copy selected traces to data buffer in ModalVIEW

- Execute menu item **Edit | Copy**

> **NOTE: The traces kept in data buffer are valid even data block window is closed. You can paste these traces into another newly opened data block file.**

To paste traces from data buffer into current data block

- Execute menu item **Edit | Paste**

To merge data block from other file

- Execute menu item **Edit | Merge from File…**

A file open dialog is displayed to prompt you to select another data block file in the project.
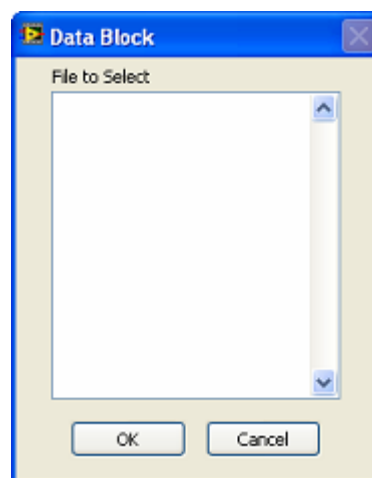
Select a file in the list and click **OK** button, the new data block will be merged into current data block.

> NOTE: The data type of merged data blocks should be the same.

To delete selected traces from the data block

- Execute menu item **Edit | Delete**

To clip part of selected trace data from the data block

- Show band cursor on the data plot
- Select the range of trace data to clip using band cursor
- Execute menu item **Edit | Clip Selected Data**

To sort measures in the data block

- Execute menu item **Edit | Sort…**

A dialog box will open to prompt you to select sorting method. Choose a sorting method and click **OK** button, traces in data block will be rearranged.
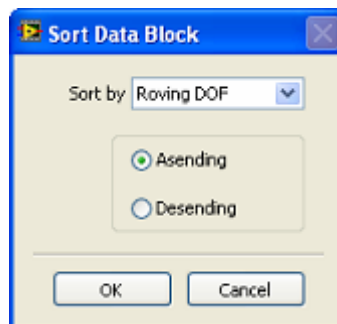
To perform mathematical operations on the data block

- Execute menu item **Edit | Math…**

A dialog box will open to prompt you to select math operations. Choose a math operation and click **OK** button to perform the operation.
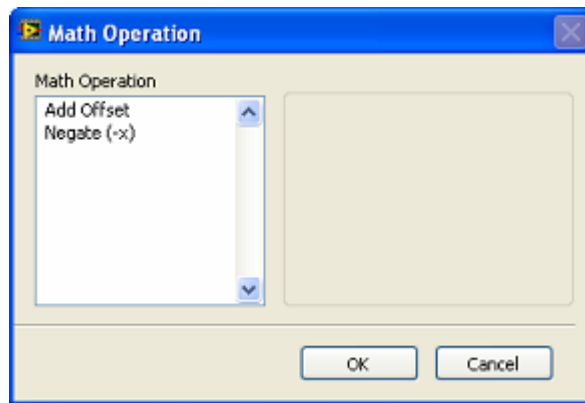
To automatically assign DOFs for measures in the data block

* Execute menu item **Edit | DOFs…**

DOF Generation dialog box will open, as Figure 1-23 shows.
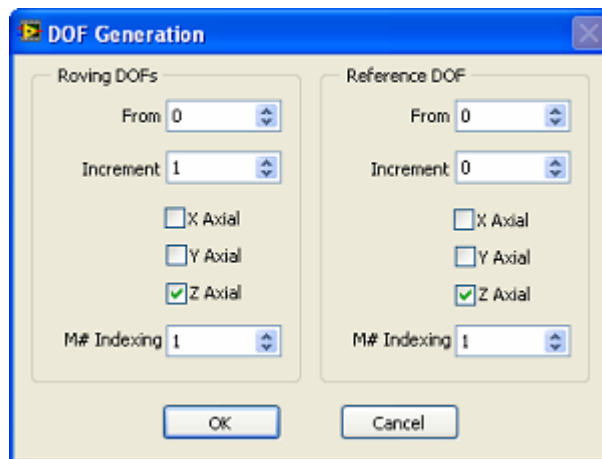
Figure 1-23 DOF generation window

Specify the parameters and click **OK** button, a series of DOFs are generated according to following scheme.
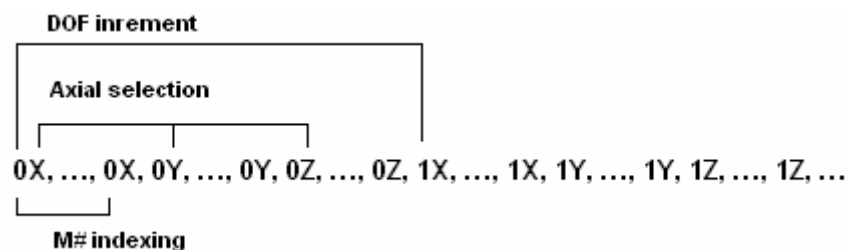
DOF information has two styles according to data type of the data block. DOF information for the data type of time record only requires roving DOF. DOF information for the data type of FRF requires roving DOF and reference DOF.

Data Type	DOF Form
Time record	Roving DOF (1X)

FRF	Roving DOF : Reference DOF (2X:0X)

For details about DOF information, please refer to section **Degree of Freedom (DOF)**.

To edit properties of selected measures in data block

- Execute menu item **Edit | Properties…**
- Or double click selected measures in spreadsheet of data properties.

The properties dialog box will open for you to edit measurement properties, as Figure 1-24 shows.
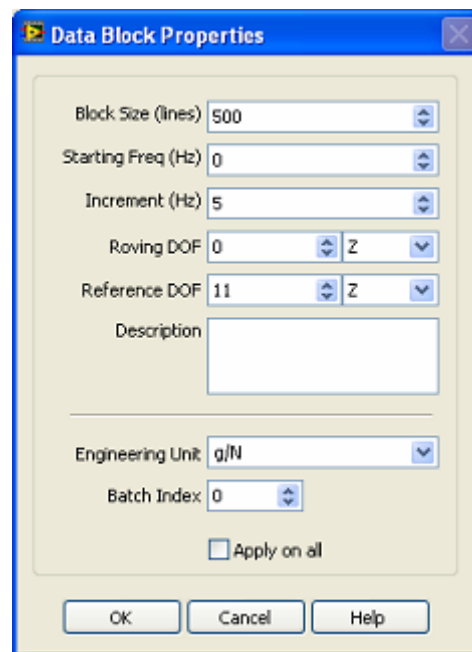
Figure 1-24 Data properties window

Configure data block display

Data view window can plot the trace of both real and complex data. The real part, imaginary part, magnitude, phase and bode plot of data can be displayed by selecting corresponding type in list box [Real ▼] on the toolbar.

To display trace plot of real data or real part of complex data
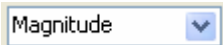
- Select **Real** in list box [Real ▼] on the toolbar
- Or execute menu item **Display | Real**

To display trace plot of the imaginary part of complex data

- Select **Imaginary** in list box [Imaginary ▼] on the toolbar

- Or execute menu item **Display | Imaginary**

To display trace plot of the magnitude of complex data

- Select **Magnitude** in list box [Magnitude ▼] on the toolbar
- Or execute menu item **Display | Magnitude**

To display trace plot of the phase of complex data

- Select **Phase** in list box [Phase ▼] on the toolbar
- Or execute menu item **Display | Phase**
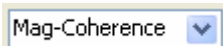
To display the plot of the real part and imaginary part of complex data

- Select **Re-Im** in list box [Re-Im ▼] on the toolbar,
- Or execute menu item **Display | Re-Im**

To display Bode plot of complex data

- Select **Bode** in list box [Bode ▼] on the toolbar,
- Or execute menu item **Display | Bode**

To display magnitude and coherence plot of FRF data

- Select **Coherence** in list box [Mag-Coherence ▼] on the toolbar
- Or execute menu item **Display | Mag-Coherence**

Select data block range

To show line cursor on the trace plot to select one point of data.

- Click button [|] on the toolbar
- Or execute menu item **Display | Cursor | Line**

To show band cursor on the trace plot to select one segment of data

- Click button [||] on the toolbar
- Or execute menu item **Display | Cursor | Band**

To show cursor value on the trace plot

- Click button [|#.#] on the toolbar
- Or execute menu item **Display | Cursor | Position**

To change line cursor position

- Place mouse pointer on cursor line.
- Mouse left key down and drag cursor line to the desired position on the trace graph.

To move band cursor position

- Place mouse pointer in the middle of band cursor.
- Mouse left key down and drag cursor line to the desired position on the trace graph.

To change the position of line cursor step by step

- Click button ⟪ or ⟫ on the toolbar

To zoom in the data plot

- Click button 🔍 on the toolbar
- Mouse left key down on the data plot and drag the mouse along abscissa of data plot

To zoom out the data plot

- Click button 🔍 on the toolbar

Data block preprocess

To perform preprocessing on time record

- Execute menu item **Process | Preprocess…**

Data preprocess window will open as Figure 1-25 shows.
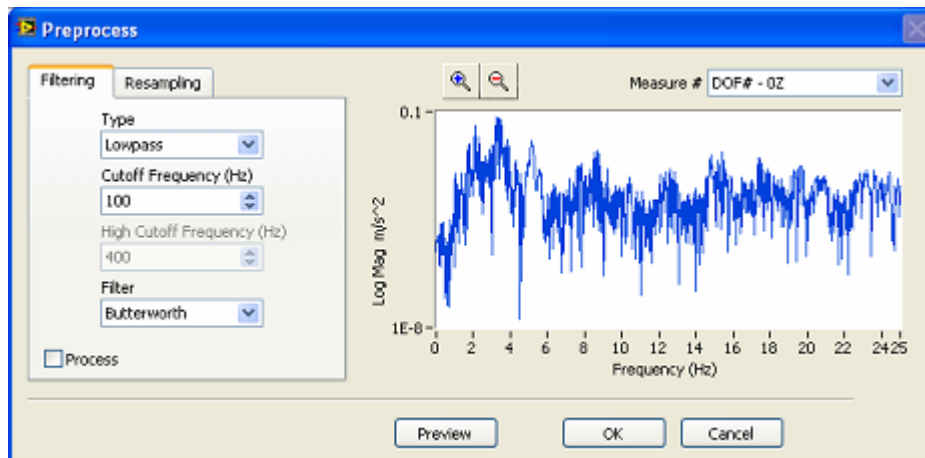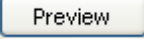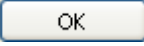
Figure 1-25 Data preprocess window

The power spectrum of time record is displayed on the right side. The spectrum can be zoom in or zoom out by selecting [🔍+] [🔍-] button.

The spectrum of time record from different channel can be viewed by selecting measure list Measure # [DOF# - 0Z ▼].

- Specify parameters in filter and resample tab page.
- Check corresponding process selection ☐ Process .
- Press [Preview] button to watch the spectrum of processed data.
- Repeat above steps until expected result is obtained.
- Press [OK] button, the time record will be processed as specified parameters. The plot of newly processed time record will be displayed in data view window.

Data block analysis

To check the integrity of FRF data

- Execute menu item **Process | Check FRF data…**

A dialog window will open to indicate if the FRF data of given reference DOF and response DOF exist or duplicated FRF data exist, as Figure 1-26 shows.

- Move mouse cursor to bar plot, the reference DOF and response DOF for each FRF data will be displayed.
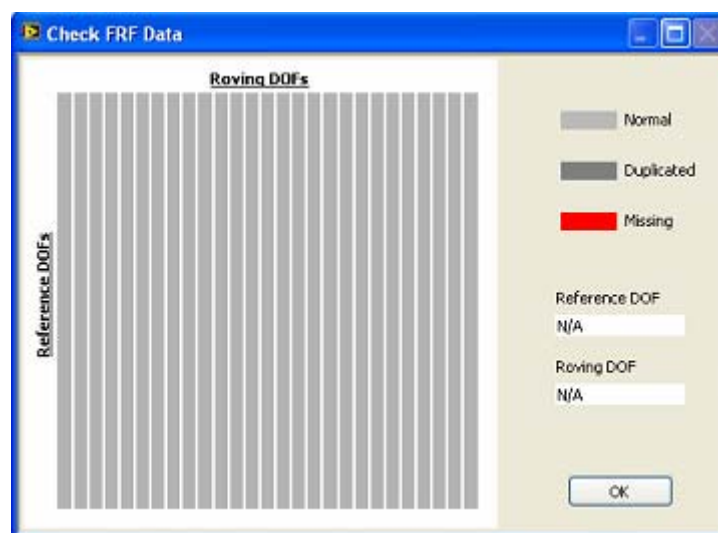
Figure 1-26 Check FRF data window

To change the unit of data block

- Execute menu item **Process | Unit Conversion…**

A dialog window is displayed to prompt you to select data unit. Select corresponding unit and press **OK**, the data unit will be converted into expected unit.
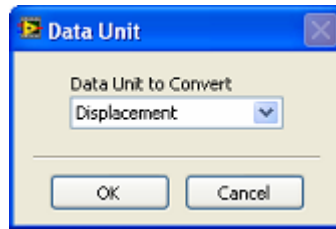
> **NOTE: ModalVIEW decides data unit according to unit string. If unit string contains invalid format, a warning dialog will display.**
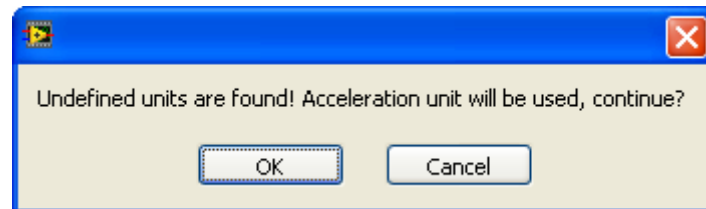
To scale ODS FRF data

- Execute menu item **Process | Scale ODS FRFs…**

A dialog window will show to select DOF of reference measure in ODS FRF data from different batches (the digit behind @ is the batch number). Press **OK** button, the magnitude of reference ODS FRF data will be scaled to unit power, other ODS FRF data in the same batch will be scaled in proportion.
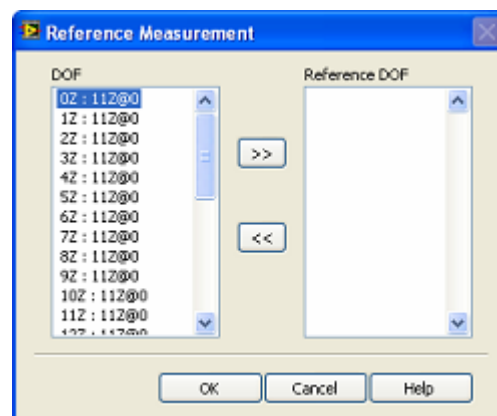
To estimate spectrum data from measured time record

- Execute menu item **Process | Spectra…**

Spectrum Estimation window will open. Specify parameters such as window, average etc., as Figure 1-27 shows

Figure 1-27 Spectrum estimation window

- Press **OK** button, the spectrum data is estimated. A dialog shows to prompt you to input the name of spectrum data file. The saved data file will be displayed in project tree.

To estimate FRF data from measured time record

- Execute menu item **Process | MIMO FRF Estimation…**

MIMO FRF Estimation window will open. Select time records for stimulus/reference and time records for response, and specify parameters such as window, average etc., as Figure 1-28 shows
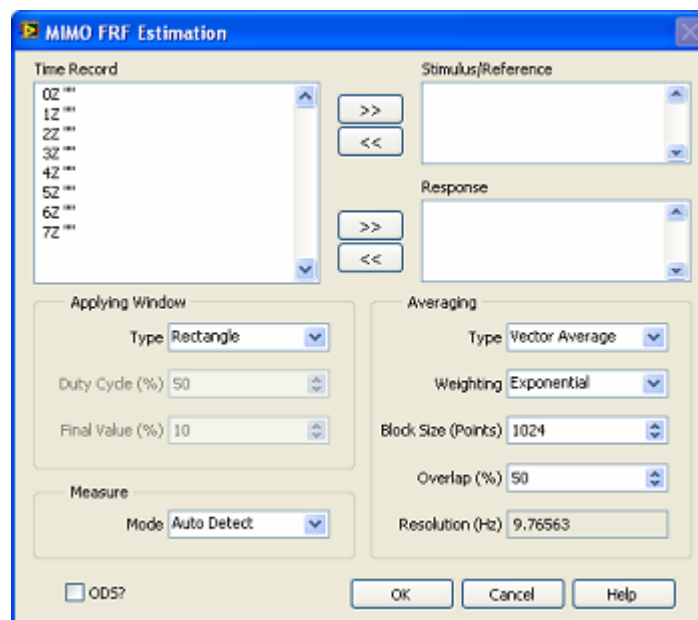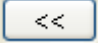
Figure 1-28 MIMO FRF estimation window

- Select [ ] ODS? to decide if using response as a reference to estimate ODS FRF.

- Use add and remove button $\boxed{>>}$, $\boxed{<<}$ to select time records for stimulus/reference and response.
- Press **OK** button, the FRF data is estimated. A dialog shows to prompt you to input the name of FRF data file. The saved data file will be displayed in project tree.

For the details about FRF estimation from measured time records, please refer to the section **FRF Calculation**.

To estimate modal parameters from FRF data block

- Click button ![icon] on the toolbar
- Or execute menu item **Process | Mode Estimation…**

Mode Estimation window will open for you to estimate modes from current data block. Please refer to section **Mode Estimation Window** for the details about its operation.

Change window arrangement

There are some predefined positions for data view window: center, for animation and for modal analysis. In different state of analysis operation, data view window is arranged to cooperate with other operating window.

To change the window arrangement

- Execute **Window** menu items in data view window.

1.5   Operational Order Window

Operational order window is used to display order spectral map of measured data, and extract time waveform or ODS FRF data from the specified order. This window contains order spectral map and spreadsheet for data properties, as Figure 1-29 shows.
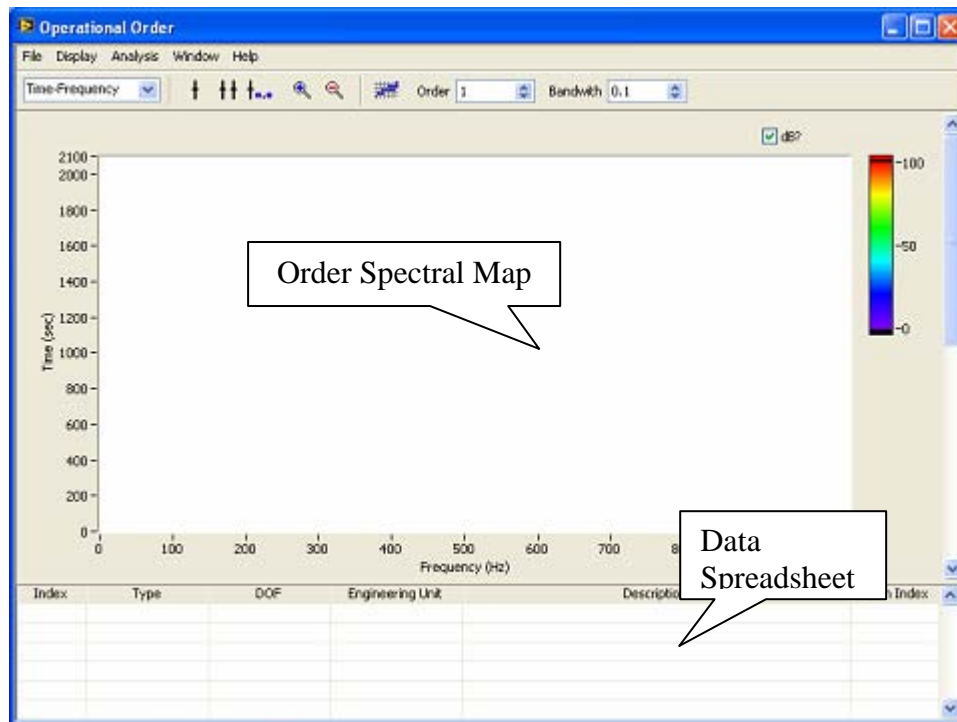
Figure 1-29 Operational order window

Load data block file

To load previously saved data block file

- Execute menu item **File | Load…**

A file open dialog is displayed to prompt you to select a data block file in the project.
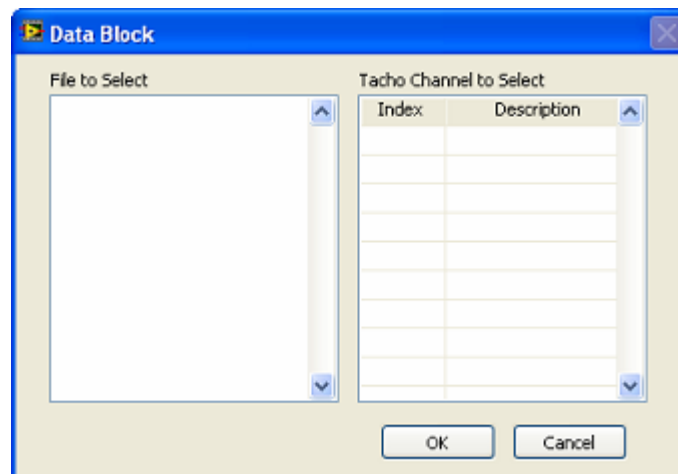
Select a file in the list and click **OK** button, order spectral map in Time-Frequency type will be displayed in the operational order window.

If data block contains tacho signal for order tracking, tacho channel can be specified by clicking corresponding row in **Tach Channel to Select list.**
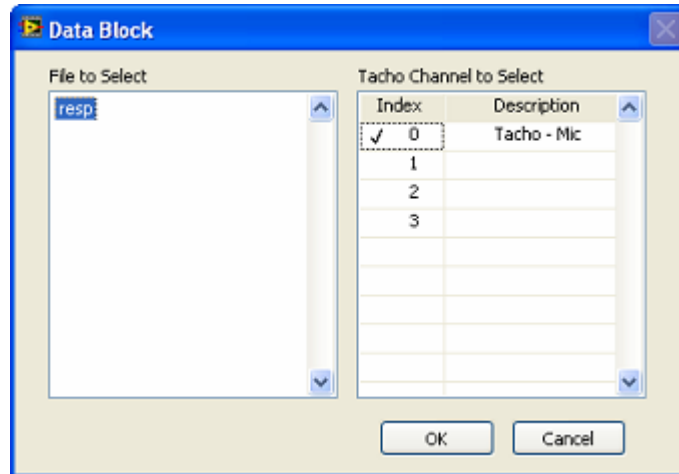
After data block being loaded, ModalVIEW automatically extracts tacho signal from data block according to the specified tacho channel. If tacho signal being loaded, order spectral map in RPM-Frequency type can be displayed in the operational order window.

Load tacho signal

Tacho signal can also be seperately loaded. To load previously saved tacho signal from data block file

• Execute menu item **File | Import tacho…**

A file open dialog is displayed to prompt you to select a data block file in the project.
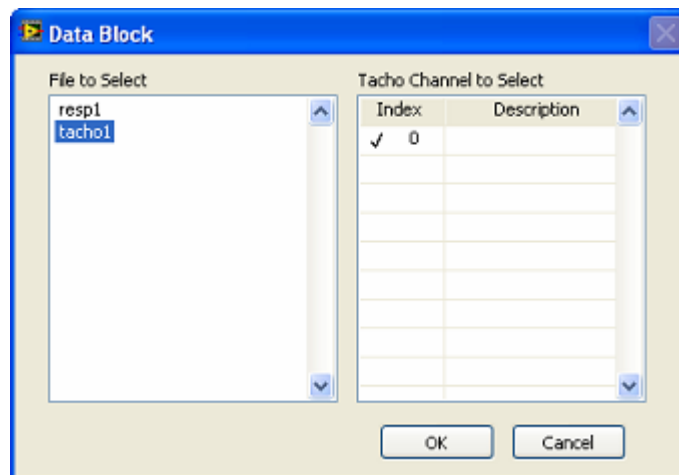
Select a file in the list and specify the channel containing tacho signal, click **OK** button. After tacho signal being loaded, order spectral map in RPM-Frequency type can be displayed in the operational order window.

Copy spectral map to clipboard

To copy order spectral map into clipboard as a Bitmap

- Execute menu item **File | Copy to clipboard | Bitmap**.

Specify spectral map type

Order spectral map can be displayed in Time-Frequency or RPM-Frequency, to specify display type

- Select corresponding item from Time-Frequency ∨ on the toolbar.
- Or select menu item **Display | Time-Frequency**
- Or select menu item **Display | RPM-Frequency**

> **NOTE: To display RPM-Frequency spectral map, the tacho signal must be loaded.**

Select position in spectral map

To show line cursor on the order spectral map for specifying a single position

- Click button ┃ on the toolbar
- Or execute menu item **Display | Cursor | Line**

To show band cursor on the order spectral map for specifying a range

- Click button ┃┃ on the toolbar
- Or execute menu item **Display | Cursor | Band**

To show cursor value on the order spectral map

- Click button ┃#.# on the toolbar

To move line cursor position

- Place mouse pointer on cursor line.
- Mouse left key down and drag mouse pointer to the desired position on the spectral map.

To move band cursor position

- Place mouse pointer in the middle of band cursor.
- Mouse left key down and drag mouse pointer to the desired position on the spectral map.

Select order in spectral map

To specify the order to extract in order spectral map

- Specify order index Order [1   ▲▼] on the toolbar.
- Specify order bandwidth Bandwith [0.1   ▲▼] on the toolbar.

To display the mask range of selected order in order spectral map

- Click button [▦] on the toolbar
- Or execute menu item **Display | Order Mask**

Specify speed profile settings

To specify the settings for speed profile generation from tacho signal

- Execute menu item **Analysis | Speed Profile…**

Speed profile window will open as Figure 1-30 shows.
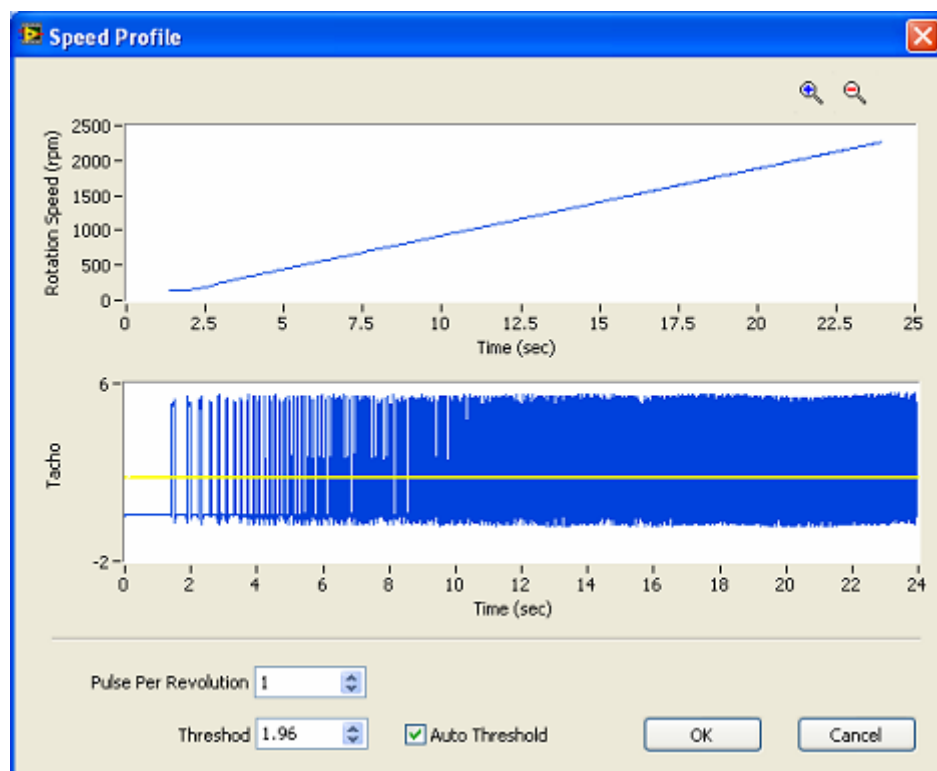
Figure 1-30 Speed profile generation from tacho signal

When specifying the parameters of threshold, pulse per revolution, the speed profile will be generated from measured tacho signal accordingly. When **Auto Threshold** is checked, ModalVIEW will automatically select proper threshold for tacho signal to generate speed profile.

Specify spectral map settings

To specify the settings of spectral map

- Execute menu item **Analysis | Order Settings…**

The order settings window will be displayed。 When the display type of spectral map is Time-Frequency, the order setting window is displayed.
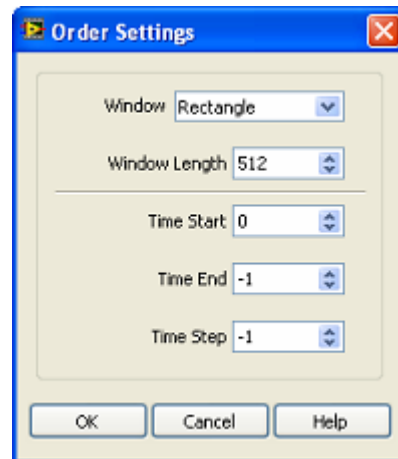
When the display type of spectral map is RPM-Frequency, the order setting window is displayed as
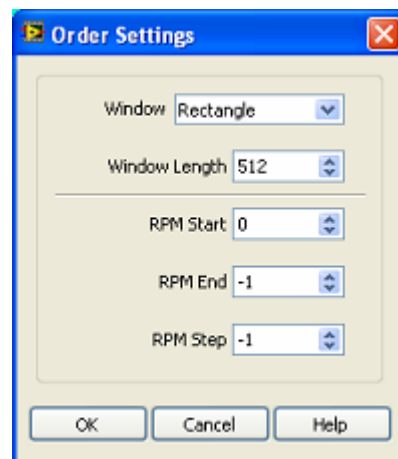
Order Analysis

To generate ODS FRF data from selected order

- Specify order index and order bandwidth.
- Execute menu item **Analysis | ODS FRF…**

A parameter settings window will be displayed.
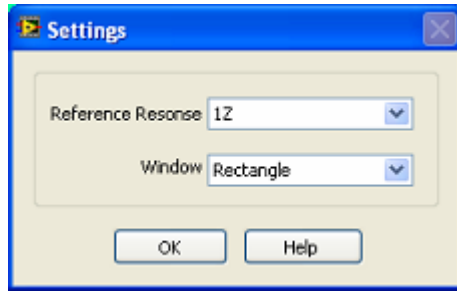
Specify the DOF of reference response and window type, click **OK** button. The generated ODS FRF data from selected order will be displayed in data viewer window.

To generate time waveform from selected order

- Specify order index and order bandwidth.
- Execute menu item **Analysis | Order Waveform…**

The generated time waveform from selected order will be displayed in data viewer window.

1.6 Mode View Window

Mode view window is used to display, edit and analyze mode shape table, as Figure 1-31 shows.
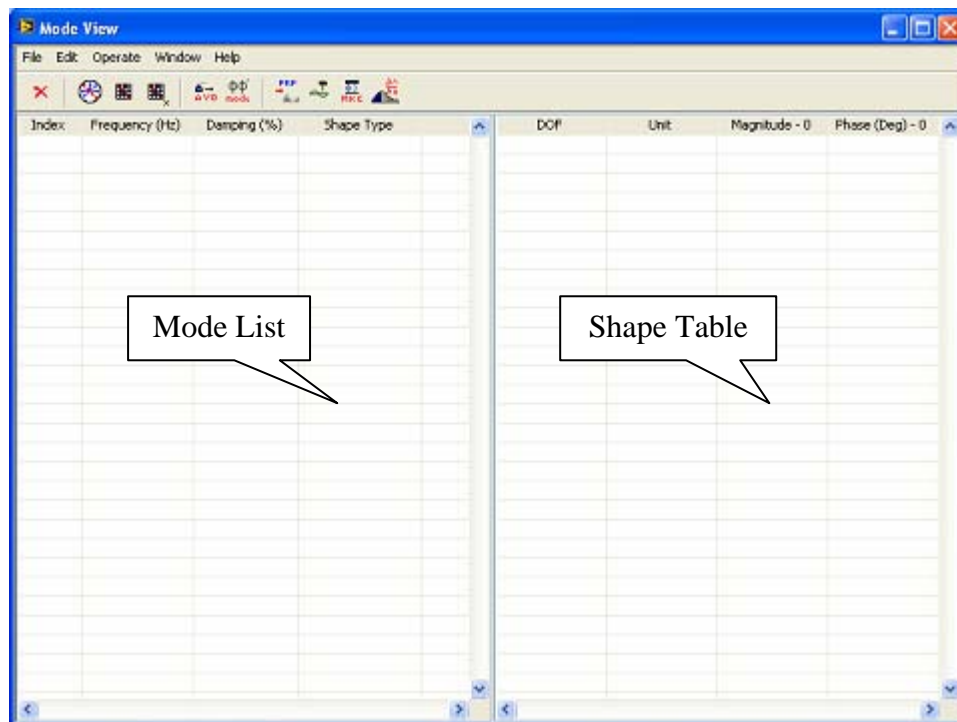
Figure 1-31 Mode view window

Load mode shape file

To load previously saved mode shape file

- Execute menu item **File | Load…**

A file open dialog is displayed to prompt you to select a mode shape file in the project.
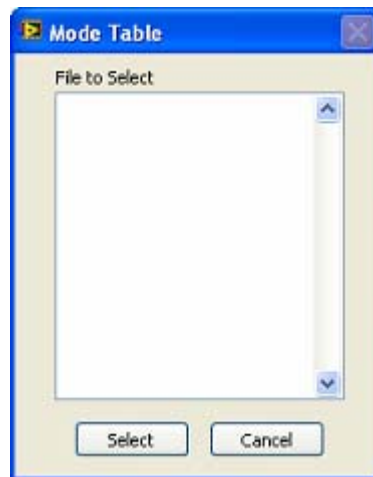
Select a file in the list and click **Select** button, mode shape table will be displayed in the mode view window.

Save mode shape

To save mode list and shape table to its file on disk

- Execute menu item **File | Save**.

If the mode shape file is not saved before, ModalVIEW prompt you to save it as a new mode shape file first.

To save mode shape file to disk with a new file name

- Execute menu item **File | Save As…** in mode view window.

A file open dialog is displayed to prompt you to select desired new file name.
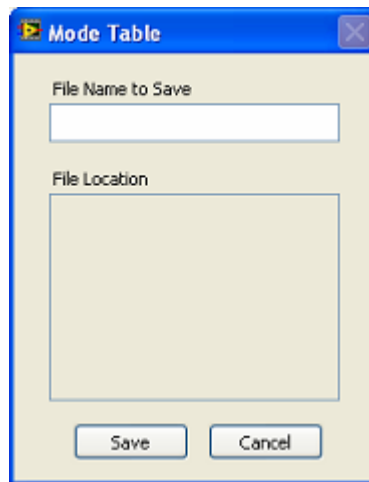
Type the desired file name into the **File Name to Save** box and click **Save** button.

Note: If there is a file with the same file name already in the project, a warning dialog box will be shown to prompt you to change the input file name

---
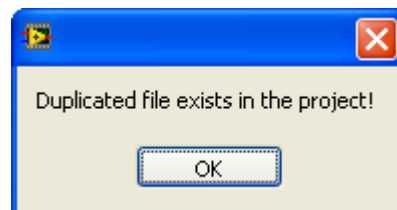
Import mode shape from external files

You can import mode shape from other external files, show the shape table in the mode view window and save the shape table into ModalVIEW mode shape file.

To import mode shape data from UFF file

- Execute menu item **File | Import | UFF file…**.

A windows file open dialog box will open for you to select UFF mode shape file.

For the details about UFF format, please refer to section **Importing External Files**.

Copy mode shape to clipboard

To copy mode list into clipboard in tab delimited format

- Execute menu item **File | Copy to clipboard | Mode List**

To copy mode shape data into clipboard in tab delimited format

- Execute menu item **File | Copy to clipboard | Spreadsheet**

Copy mode shape to report generator

To copy mode list displayed in mode view window to report generator

- Execute menu item **File | Copy to Report | Mode List…**

A dialog box is displayed to prompt you to input desired report item name. Type the name in **Report item name** box and click **OK** button, the mode list is added to report item list accordingly.
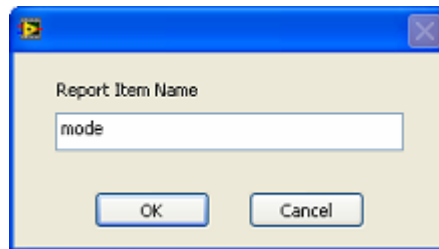
When report generator window being opened, the added report items are shown in **Available Report Items** list. Please refer to section **Report Generation Window** for more details.
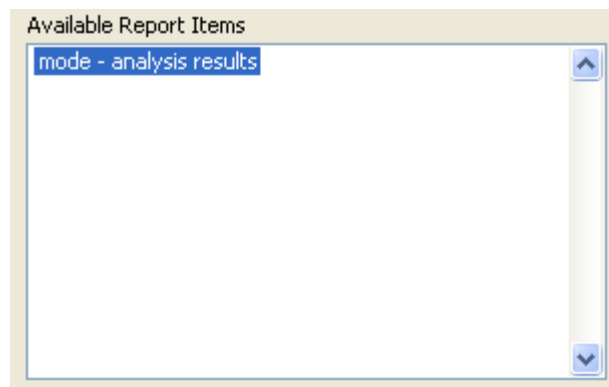
Edit mode shape table

To select all modes in the mode list

- Execute menu item **Edit | Select All**

To unselect all modes in the model list

- Execute menu item **Edit | Select None**

To copy selected modes to data buffer in ModalVIEW

- Execute menu item **Edit | Copy**

> 📝 **NOTE: The modes kept in data buffer are valid even mode view window is closed. You can paste these modes into another newly opened mode shape file.**

To paste modes from data buffer into current mode table

- Execute menu item **Edit | Paste**

To merge modes from other mode shape file

- Execute menu item **Edit | Merge from File…**

A file open dialog is displayed to prompt you to select another mode shape file in the project.

Select a file in the list and click **Select** button, the new modes will be merged into current mode table.

> 📝 **NOTE: The DOF information of merged modes must be the same.**

To delete selected modes from the mode table

- Execute menu item **Edit | Delete**

To sort modes in the mode table according to frequency

- Execute menu item **Edit | Sort Modes | Ascending**
- Or execute menu item **Edit | Sort Modes | Descending**

To sort shapes in the shape table according to DOF

- Execute menu item **Edit | Sort Shapes…**

A dialog box will open to prompt you to select sorting scheme. Choose a sorting method and click **OK** button, mode shapes will be rearranged.

To automatically assign DOFs to mode shape

• Execute menu item **Edit | DOFs…**

DOF Generation dialog box will open, as Figure 1-32 shows.
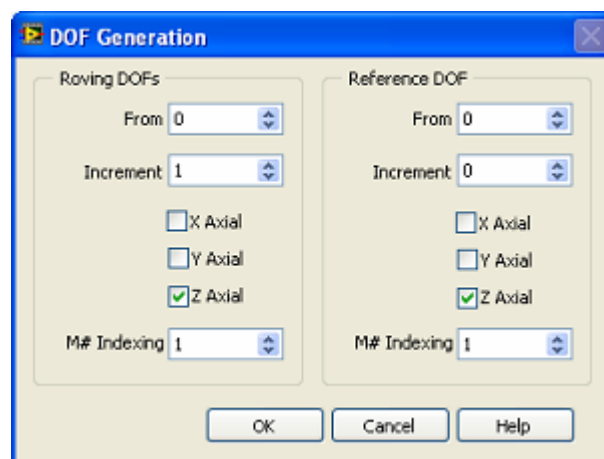
Figure 1-32 DOF generation window

Specify the parameters and click **OK** button, a series of DOFs are generated according to following scheme.
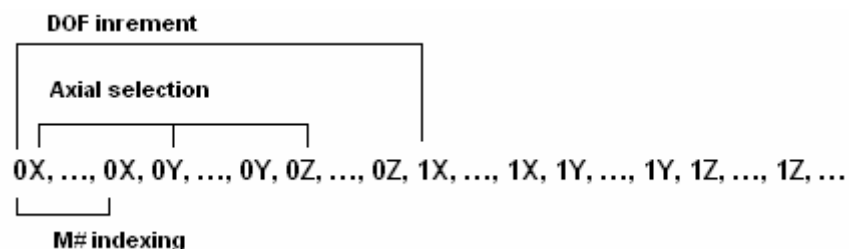
The DOF for mode shape has following style.

Roving DOF: Reference DOF (2X: 0X)

For details about DOF information, refer to section **Degree of Freedom (DOF)**.

To edit properties of selected mode in mode table

- Execute menu item **Edit | Properties…**
- Or double click selected mode in mode list.

The properties dialog box will open for you to edit mode properties, as Figure 1-33 shows.
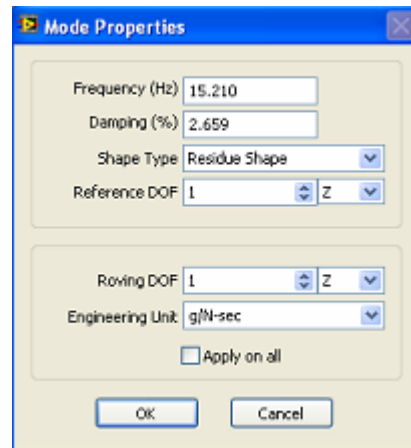
Figure 1-33 Mode properties window

Analyze mode parameter

To display complexity plot of the selected mode

- Click button on the toolbar
- Or execute menu item **Operate | Complexity Plot…**

A mode plot window will open and complexity plot is displayed, as Figure 1-34 shows. The blue lines in the plot indicate the vector of mode shape. Select different mode in mode list, the complexity plot changes accordingly.
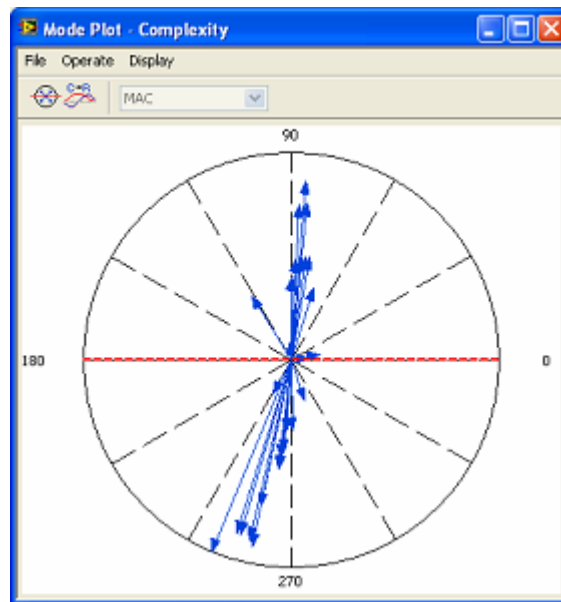
Figure 1-34 Complexity plot

To convert complex mode to normal mode

- Click button [image] on the toolbar in mode plot window
- Or execute menu item **Operate | Normalize Mode**

The red dashed line is used as a phase reference to convert complex mode into a normal mode. Phase of mode shape on one side of phase reference line is changed to 0, and on the other side is changed to 180.

- Click button [image] on the toolbar in mode plot window
- Or execute menu item **Operate | Complex to Real**

For the details about complex mode and normal mode, please refer to section **Normal and Complex**.

To compute values of modal assurance criterion (MAC)

- Click button [image] on the toolbar
- Or execute menu item **Operate | MAC…**

A mode plot window will open and MAC plot is displayed, as Figure 1-35 shows. Select different mode in mode table, the MAC plot changes accordingly.
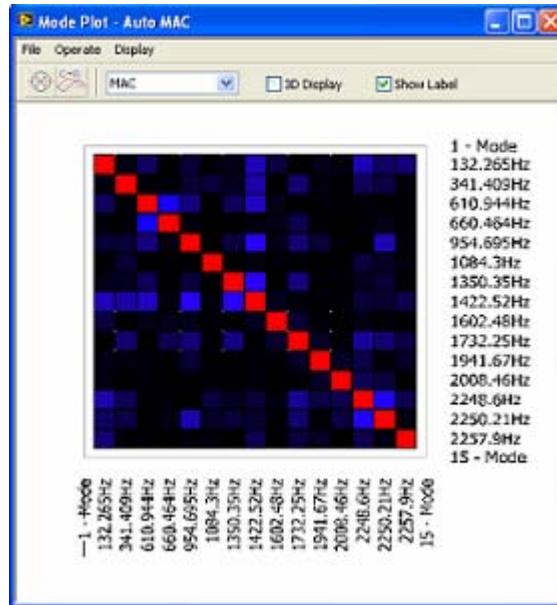
Figure 1-35 MAC plot

- Select corresponding item in control ⟨CoMAC ▾⟩ to display MAC or CoMAC plot, or execute menu item **Display | MAC** or **Display | CoMAC** in mode plot window.
- Check ☐ 3D Display , the MAC plot is displayed in 3D bar plot.
- Check ☐ Show Label , the plot label of mode frequency is displayed.
- Check ☐ Show Table , the MAC value in spreadsheet is displayed.

To compute values of cross modal assurance criterion (Cross MAC)

- Click button ▦ on the toolbar
- Or execute menu item **Operate | Cross MAC…**

A file open dialog is displayed to prompt you to select another mode shape file in the project. Select a file in the list and click **Select** button, the modes will be loaded for comparison.

For the details about MAC, CoMAC and Cross MAC, please refer to section **Modal assurance criteria (MAC)**.

To copy MAC or Complexity plot to clipboard in a Bitmap

- Execute menu item **File | Copy to Clipboard | Bitmap**

To copy MAC spreadsheet to clipboard

- Execute menu item **File | Copy to Clipboard | Spreadsheet**

Modify mode parameter
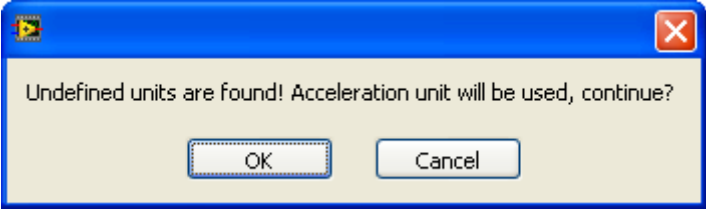
To change the unit of mode shape

- Click button [icon] on the toolbar
- Or execute menu item **Operate | Acc, Vel, Disp…**

A dialog window is displayed to prompt you to select shape unit. Select corresponding unit and press **OK**, the shape unit will be converted into expected unit.
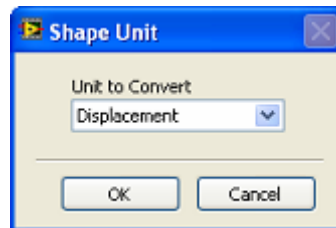
  NOTE: ModalVIEW decides shape unit according to unit string. If unit string contains invalid format，a warning dialog will display.

For details about shape unit, please refer to section **Mode shape unit**.

To change shape type from residue shape to scaled shape

- Click button [icon] on the toolbar
- Or execute menu item **Operate | Scale Mode Shape**

For details about residue shape and scaled shape, please refer to section **Mode shape type**.

Utilize mode parameters

To synthesize FRF data from modes

- Click button [icon] on the toolbar
- Or execute menu item **Operate | FRF Synthesis…**

A dialog window is displayed to prompt you to specify FRF parameters, as Figure 1-36 shows.
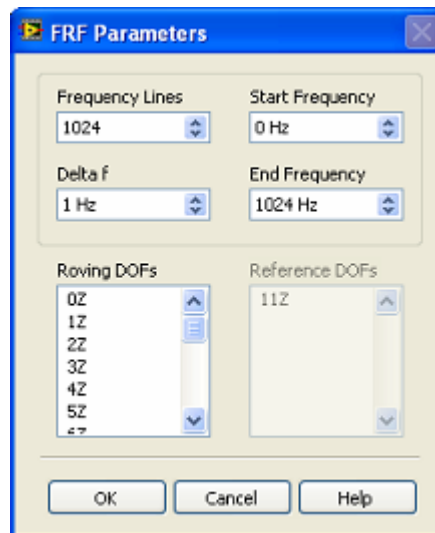
Figure 1-36 FRF parameters window

After specifying parameters, press **OK** button. The synthesized FRF data plot will be displayed in opened data view window.

To compute forced response shape from modes

- Click button ⬛ on the toolbar
- Or execute menu item **Operate | Forced Response…**

A dialog window is displayed to prompt you to specify parameters, as Figure 1-37 shows.
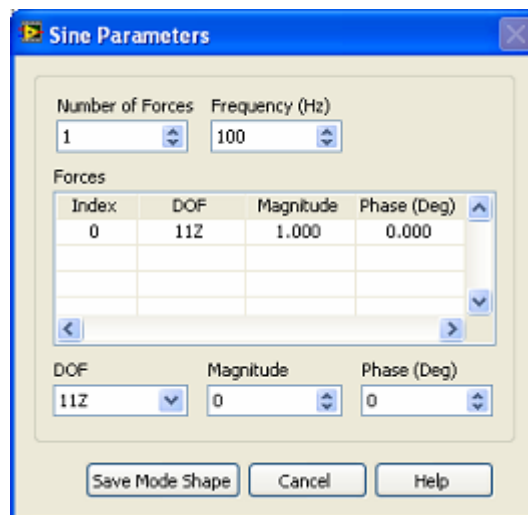
Figure 1-37 Sine parameters window

After specifying parameters, press **Save Mode Shape** button. A file open dialog is displayed to prompt you to save the mode shape from sine excitation into mode shape file.
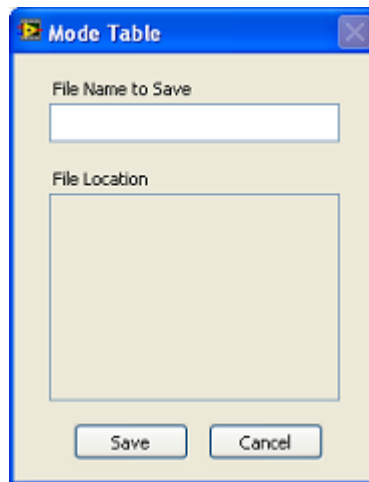
Type the desired file name into the **File Name to Save** box and click **Save** button. The generated mode shape is saved for structure animation.

For more information about FRF synthesis and forced sine response, please refer to section **Mode** .

To compute mass, stiffness, and damping matrix from estimated modes

*   Click button on the toolbar
*   Or execute menu item **Operate | M, K, C Matrix…**

A dialog window is displayed to display mass, stiffness and damping matrix, as Figure 1-38 shows.
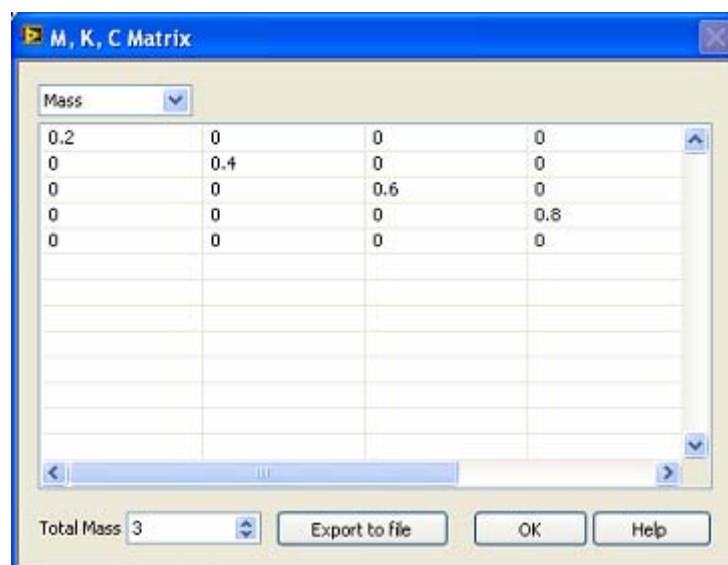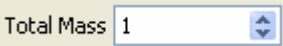
Figure 1-38 Mass, stiffness and damping window

*   Specify the total mass of the structure in
*   Select item in to display corresponding matrix values.

- Press button ![Export to file] to export mass, stiffness and damping matrix into ASCII text data file.

For more information about mass, stiffness and damping matrix, please refer to section **Mass, stiffness and damping matrix.**

To perform structure sensitivity analysis using estimated modes

- Click button ![icon] on the toolbar
- Or execute menu item **Operate | Sensitivity…**

A sensitivity matrix plot window will open, as Figure 1-39 shows. Select different mode in mode list, the sensitivity matrix plot is updated accordingly, which indicates how structure modification influences the natural frequency of selected mode.
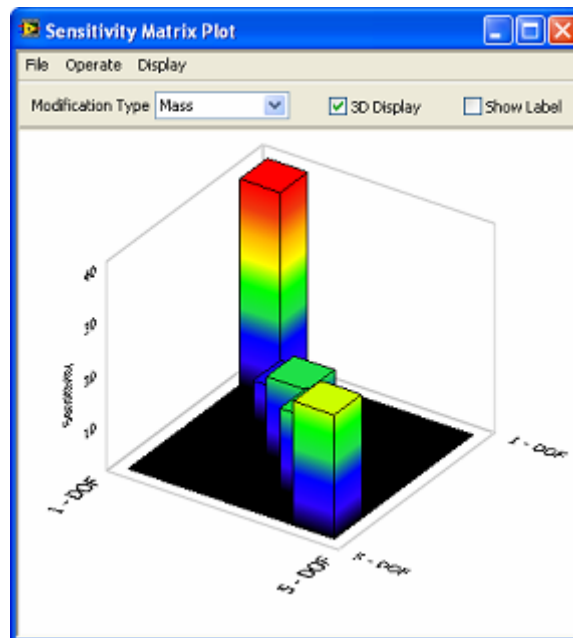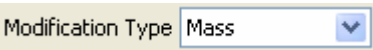
Figure 1-39 Sensitivity matrix plot

- Select items in ![Modification Type Mass] to specify the type of structure modification: Mass, Stiffness or Damping.
- Check ![3D Display], the sensitivity matrix plot is displayed in 3D bar plot.
- Check ![Show Label], the plot label of mode DOF is displayed.

To copy sensitivity matrix plot to clipboard in a Bitmap

- Execute menu item **File | Copy to Clipboard | Bitmap**

To copy sensitivity matrix data to clipboard in spreadsheet

- Execute menu item **File | Copy to Clipboard | Spreadsheet**

For more information about sensitivity analysis, please refer to section **Sensitivity analysis**.

1.7 Modification View Window

Modification view window is used to edit changes in the mass, stiffness or damping properties of a structure, and perform structure modification prediction by evaluating the effect of mass, spring-damper and tuned absorber modifications, as Figure 1-40 shows.
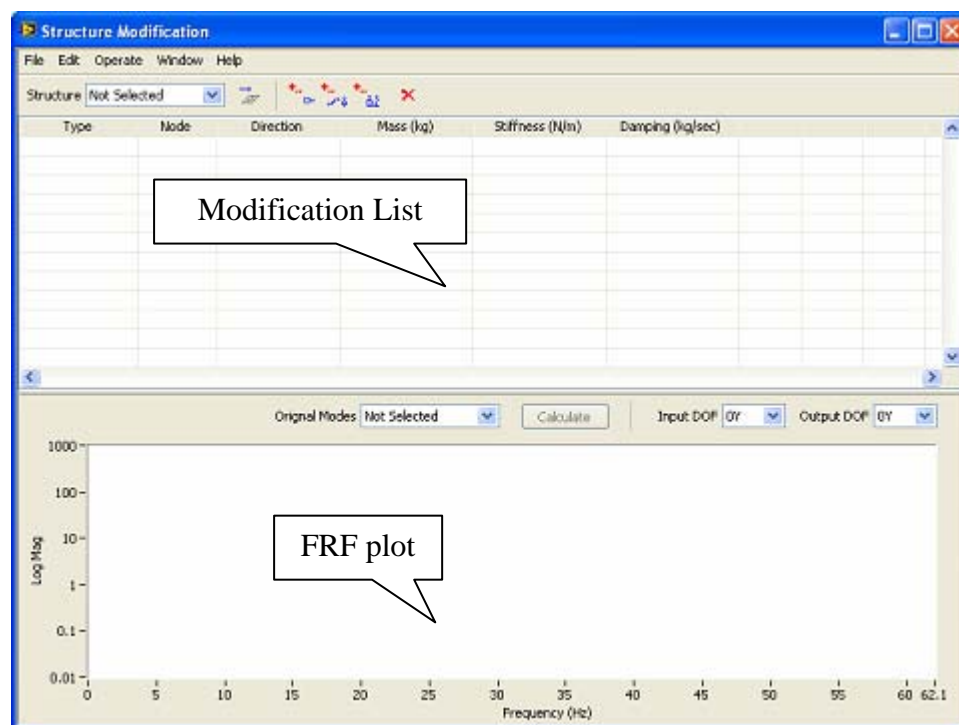
Figure 1-40 Modification view window

Load modification file

To load previously saved structure modification file

- Execute menu item **File | Load…**

A file open dialog is displayed to prompt you to select a structure modification file in the project.
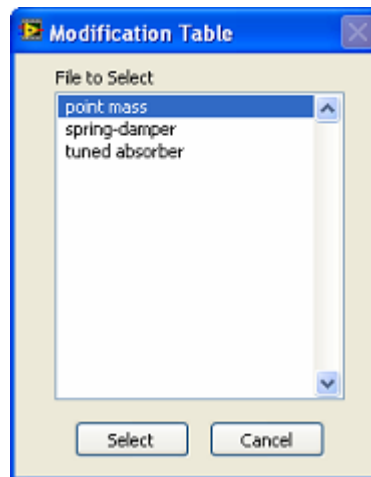
Select a file in the list and click **Select** button, modification table will be displayed in the modification view window.

Save modification file

To save structure modification table to its file on disk

- Execute menu item **File | Save**.

If the structure modification file is not saved before, ModalVIEW prompt you to save it as a new structure modification file first.

To save structure modification file to disk with a new file name

- Execute menu item **File | Save As…**.

A file open dialog is displayed to prompt you to input desired new file name.
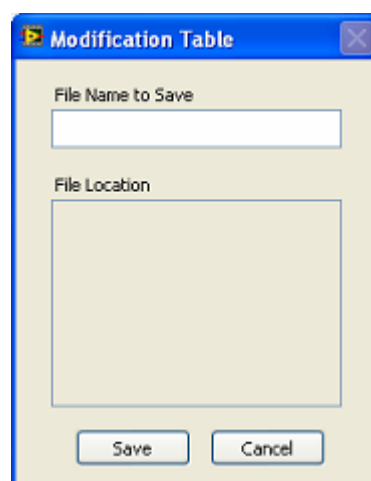
Type the desired file name into the **File Name to Save** box and click **Save** button.

Note: If there is a file with the same file name already in the project, a warning dialog box will be shown to prompt you to change the input file name.
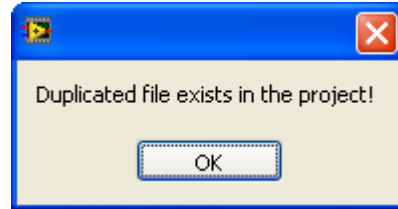
Copy to clipboard

To copy structure modification table into clipboard as a spreadsheet

- Execute menu item **File | Copy to clipboard | Modification Table**.

To estimated FRF plot into clipboard as a Bitmap

- Execute menu item **File | Copy to clipboard | FRF Plot**.

Save modified mode

To save the modes of modified structure to its file on disk

- Execute menu item **File | Save Modified Modes…**.

A file open dialog is displayed to prompt you to input desired new file name.
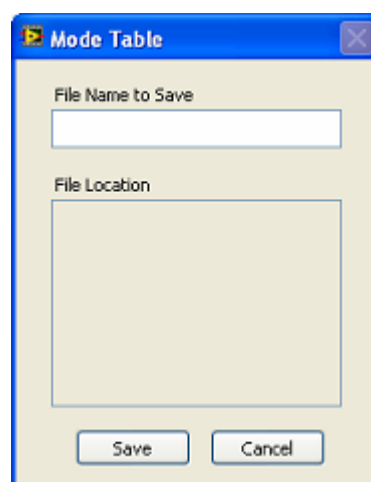
Type the desired file name into the **File Name to Save** box and click **Save** button.

Edit modification table

To add a mass on the single point of the structure

- Execute menu item **Edit | Add Mass…**
- Or click button ![icon] on the toolbar

A dialog window is displayed to prompt you to specify point mass to add. After specifing the parameters, click **OK** button.
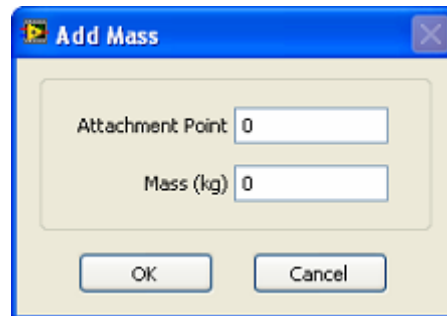
To add a spring-damper between two points of the structure

- Execute menu item **Edit | Add Spring-damper…**
- Or click button ![icon] on the toolbar

A dialog window is displayed to prompt you to specify point mass to add. After specifing the parameters, click **OK** button.

To add a spring-damper between two points of the structure

- Execute menu item **Edit | Add Tuned Absorber…**
- Or click button ![icon] on the toolbar

A dialog window is displayed to prompt you to specify point mass to add. After specifing the parameters, click **OK** button.
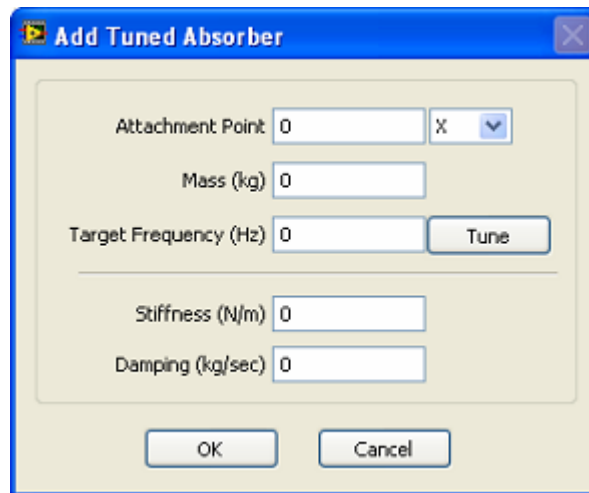
For more information about structure modification prediction, please refer to section **Modification Prediction**.

To select all items in the modification list

- Execute menu item **Edit | Select All**

To unselect all items in the modification list

- Execute menu item **Edit | Select None**

To delete selected items from the modification list

- Click items in the modification list.
- Execute menu item **Edit | Delete**.

To modify selected item in the modification list

- Double click the item in the modification list.
- Or select one item in the list, and execute menu item **Edit | Modify**.

A dialog window is displayed to prompt you to adjust corresponding parameters.

Connect to structure model

To display modification guide on 3D structure model matched with the structure modification list.

- Select the structure model from the listbox on the toolbar.
- Click button on toolbar.

The selected structure 3D model will be displayed in structure window. The corresponding structrure modification will be displayed on the structure model, as Figure 1-41 shows.
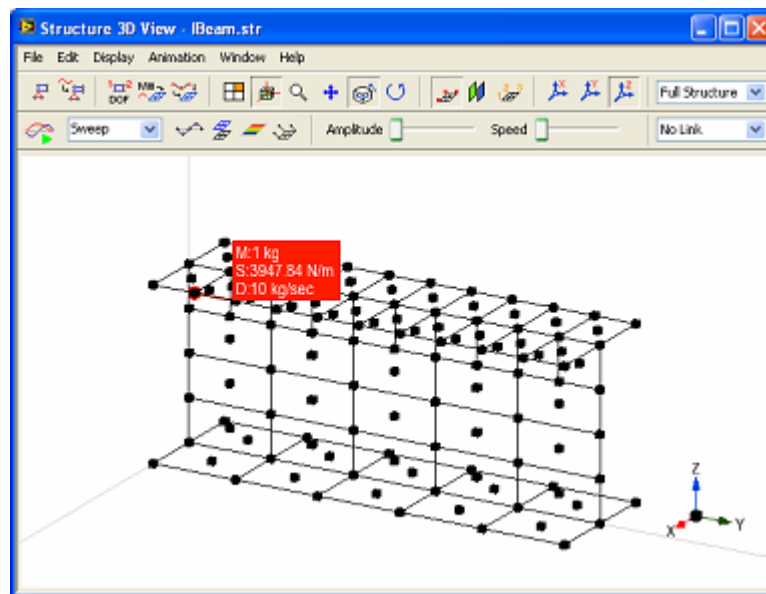
Figure 1-41 Structrue Modification Guide

Perform Modification Prediction

To specify the settings for FRF synthesis after structure modification

- Execute menu item **Operate | FRF Settings…**

A dialog window is displayed to prompt you to specify the parameters for FRF computation.
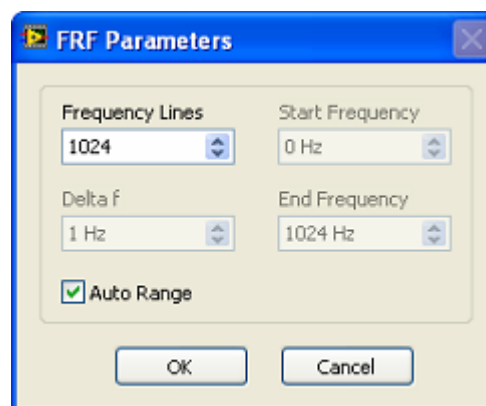
To predict FRF plot and new modes of the structure after modification

- Select the structure model from the Structure Not Selected listbox on the toolbar.

- Select the structure's original modes from the `Orignal Modes` `Not Selected` listbox.
- Click `Calculate` button or execute menu item **Operate | Compute Modification**

To view FRF plot of the corresponding input and output DOFs

- Select input DOF and output DOF from the `Input DOF` `0Y` listbox and the `Output DOF` `0Y` listbox.

Change window arrangement

There are some predefined positions for modification view window.

To change the window arrangement

- Execute corresponding **Window** menu items in modification view window.

1.8 Image View Window

Image view window is used to display structure's digital photo, as Figure 1-42 shows. From the digital photo, the structure's 2D profile can be extracted and be used to quickly generate 3D structure model.
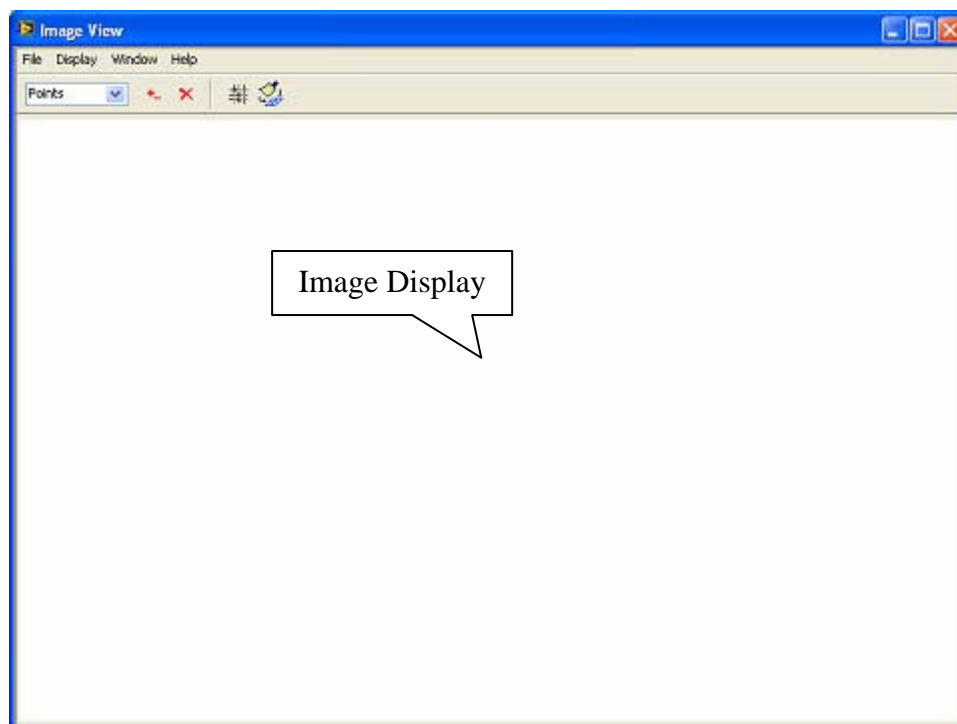
Figure 1-42 Image view window

Load image file

To load previously saved image file

- Execute menu item **File | Load…**

A file open dialog is displayed to prompt you to select an image file in the project. Select a file in the list and click **Select** button, the image will be displayed in the image view window.

Copy image to clipboard

The structure image and 2D profile in image view window can be copied into window clipboard in Bitmap for other programs to use.

To copy the image file into clipboard as a Bitmap

- Execute menu item **File | Copy to Clipboard | Bitmap**.

Save 2D structure profile

To save depicted 2D structure profile into a structure file

- Execute menu item **File | Save 2D Profile…**

A file open dialog is displayed to prompt you to select desired new file name. Type the desired file name into the **New File Name** box and click OK button, a new structure file is saved and added into current project.

Set image display mode

To lock image's aspect ratio

- Execute menu item **Display | Lock Aspect Ratio**

Change window arrangement

There are some predefined positions for image view window.

To change the window arrangement

- Execute **Window** menu items in image view window.

Edit 2D structure profile

The 2D structure profile is defined by nodes and lines.

To depict node of 2D structure profile on the structure image

- Specify object type [Points ▼] in toolbar as **Points**

- Click button [+..] to enable the add operation.
- Click on the appropriate place using mouse left button to add a new node on the structure image.

To add line between nodes to form 2D structure profile

- Specify object type [Points ▼] in toolbar as **Lines**

- Click button [+..] to enable the add operation.
- Click two nodes using mouse cursor sequentially and add a line between two nodes.

To delete nodes or lines on the 2D profile

- Specify corresponding object type [Points ▼] in toolbar
- Click to select the node or line to be deleted.

- Click button [×] to delete the selected object.

To clear depicted 2D profile

- Click button [🖉] on the toolbar.

To automatically generate grids in 2D profile

- Click button [#] on the toolbar.

The depicted 2D structure profile can be saved into a structure file. For more information about 2D structure profile generation, please refer to the section **Generate 3D model from image**.

1.9 Report Generation Window

Report generation window is used to specify modal testing parameters such as operators, instruments, and report item, as Figure 1-43 shows. According to specified report information, a test report in Microsoft Word file format is automatically generated from the predefined report template.
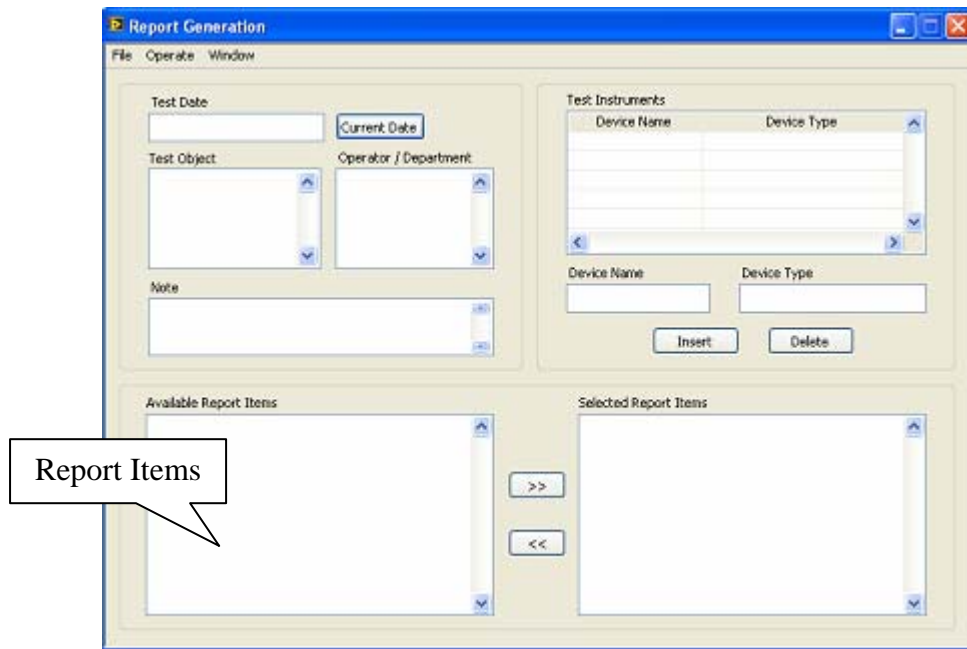
Figure 1-43 Report generation window

Specify report information

To specify test date, test object, operator and note for the testing experiment

- Type the desired content into the **Test Date, Test Object, Operator / Department** and **Note** box.

To obtain current test date

- Click button Current Date , the current date will be inserted into **Test Date** box.

To add new device to the **Test Instruments** list

- Type the device name and type into **Device Name** and **Device Type** box.
- Click button Insert , the specified device will be added.

To delete test device from the **Test Instruments** list

- Select one item in the test instrument list.
- Click button Delete .

Available Report Items list contains the report items which can be exported to test report.
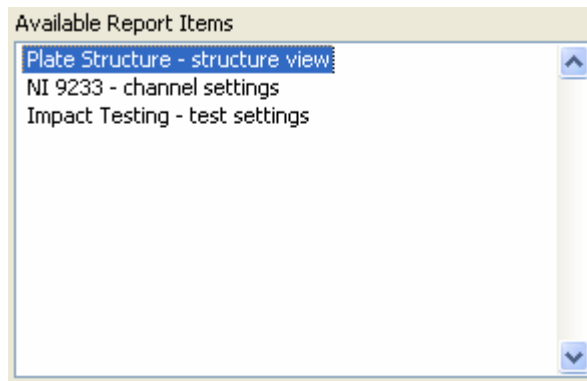
To select report item and export it to test report

- Select one item in the **Available Report Items** list.
- Click button [>>], the item will be added into **Selected Report Items** list.

To remove report item from **Selected Report Items** list

- Select one item in the **Selected Report Items** list.
- Click button [<<], the item will be removed.

Clear available report items

To delete the selected report item from the list

- Select one item in the **Available Report Items** list.
- Execute menu item **Operate | Delete Selected Item**.

To clear all the available report item in the list

- Execute menu item **Operate | Clear All Report Items**.

Generate test report

To generate test report after specifying all the report information,

- Execute menu item **File | Generate Report...**.

A file open dialog is displayed to prompt you to input desired new report file name.
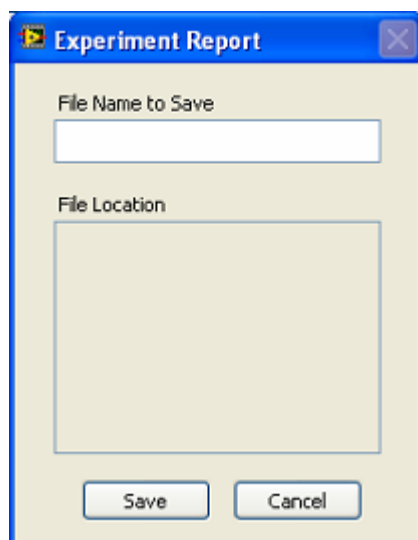
Type the desired file name into the **File Name to Save** box and click **Save** button.

NOTE: If there is a file with the same file name already in the project, a warning dialog box will be shown to prompt you to change the input file name.

When test report is successfully generated, a dialog box will show as following
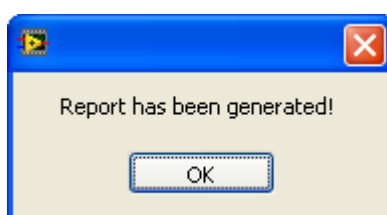
NOTE: Please close other opened word documents before generating report.

1.10 Structure Editor Window

Structure editor window is used for 3D model drawing, as Figure 1-44 shows.
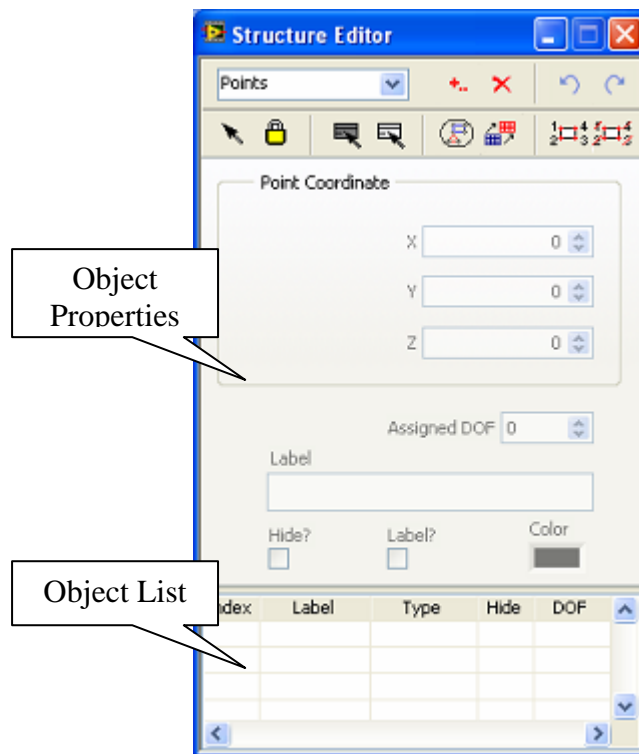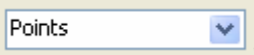
Figure 1-44 Structure editor window

Edit structure object

Structure editor provides a series of objects, point, line, surface (triangle), surface (quad) and substructure, to build 3D structure model.

To change object type to edit

- Choose object list on the toolbar.

The display page of object properties will change accordingly.

To select a object

- Choose the object type.
- Select the object in spreadsheet
- Or press button on the toolbar to enable click select operation, select object on 3D model by clicking mouse cursor on structure model viewer window

To select all objects

- Press button on the toolbar

To unselect all objects

- Press button ![icon] on the toolbar

To edit properties of a object

- Select the object in object list.
- Specify information in properties page.

To add a object to 3D structure model

- Choose the object type to add.

- Press button ![icon] on the toolbar to enable add operations

If the object type is substructure, a dialog box will display for you to select a new substructure.
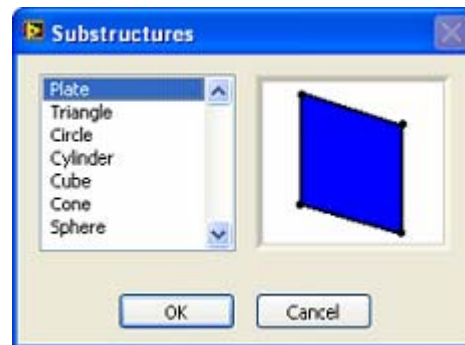
Select a corresponding substructure and click OK, a new substructure will be added to 3D model.

If the object type is point or line, add the object to 3D mode by clicking mouse cursor on structure model view window. For details, please refer to chapter **Creating Geometry Model**.

To delete a object from 3D model

- Select the object.

- Press button ![icon] on the toolbar.

To add a pre-defined model in file to 3D model as a new substructure

- Press button ![icon] on the toolbar.

A file open dialog is displayed to prompt you to select another structure file in the project.
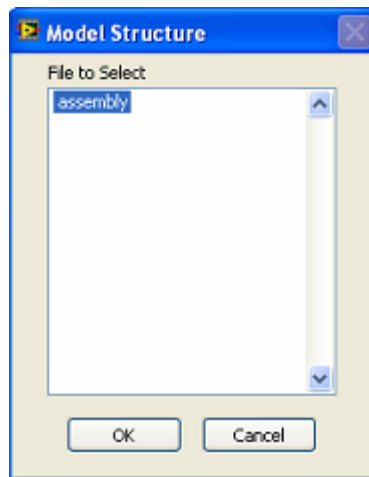
- Select a structure file, press OK button.

To extrude a 2D structure profile into a 3D structure model

- Press button on the toolbar.

A dialog box displays to prompt you to input extrude parameters include extrude direction and distance.
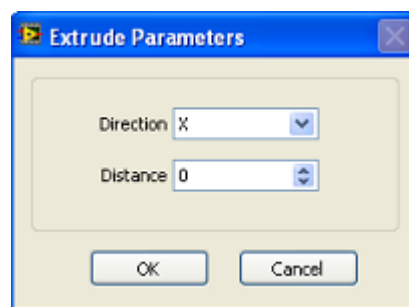
- Specify appropriate parameters, press OK button.

To restore the 3D model to last editing state

- Press button on the toolbar.

To redo the restored editing state

- Press button on the toolbar.

Note: The restore operation can be executed repeatedly until all the stored editing state being restored.

To move a object in 3D model by dragging it with the mouse cursor

- Press button ![arrow icon] on the toolbar to enable click select operation.
- Press button ![lock icon] on the toolbar to unlock object move.
- Click to select an object in structure view window and drag it to a new position.

Number point DOF

To automatically assign DOF number to each node on 3D model

- Press button ![DOF icon] on the toolbar.

To clear assigned DOF number of each node on 3D model

- Press button ![DOF icon] on the toolbar.

1.11 Measure Properties Window

Measure properties window is used to specify measure axes direction, measure type and assigned measurement for points on 3D structure model, as Figure 1-45 shows.
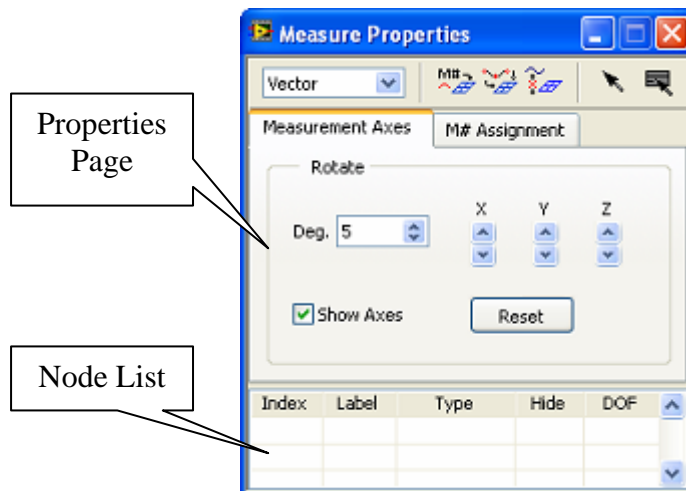
Figure 1-45 Measure properties window

Select data type

To specify the type of measured data associated with on the structure model

- Select the list box ![Vector list box] in the toolbar.

Select node

To select a node on 3D structure model

- Select the node in spreadsheet
- Or press button ![cursor] on the toolbar to enable click select operation, select a node on 3D model by clicking mouse cursor on structure model viewer window

To select all nodes

- Press button ![select all] on the toolbar

Specify measurement axes

To change direction of measurement axes of selected node

- Select a node in spreadsheet.
- Set rotational increment Deg. 5 in degree.
- Check ☐ Show Axes to display measurement axes on 3D model.

- Click rotate control X ▲▼ to rotate measurement axes along X axial.

- Click rotate control Y ▲▼ to rotate measurement axes along Y axial.

- Click rotate control Z ▲▼ to rotate measurement axes along Z axial.

To reset measurement axes to default direction

- Press button Reset .

Assign measurement

To change measure type for each measure direction of selected node

- Select a node in spreadsheet.
- Switch tab page to **M# Assignment**.
- Double click the item in **Equation Type** column and select measure type of corresponding measurement direction.

To manually assign measurement index for each node

- Select a node in spreadsheet.
- Switch tab page to M# Assignment.
- Double click the item in **Measure #** column and set measurement index for corresponding measurement direction.

> ![note icon] **NOTE: Measurement index is assigned only when measure type is 'measured'.**

To automatically assign measurement index for each node

- Click button ![M# icon] on the toolbar.

A dialog box will show to prompt you to specify the range of selected nodes, as Figure 1-46 shows. After specifying point selection, click **OK** button. Measurement index will be incrementally assigned to each measurement direction of selected nodes.
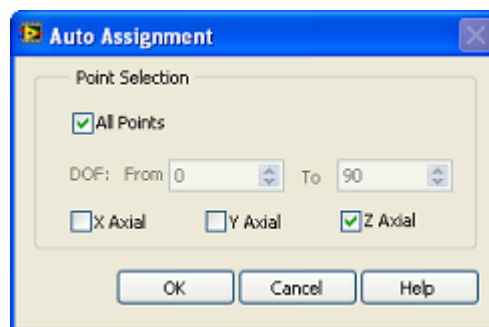
Figure 1-46 Auto assignment window

To generate interpolated animation for unmeasured nodes on 3D model
- Click button ![interpolate icon] on the toolbar.

A dialog box will show to prompt you to specify interpolation range. That is how many nearest nodes are used to calculate interpolated animation.
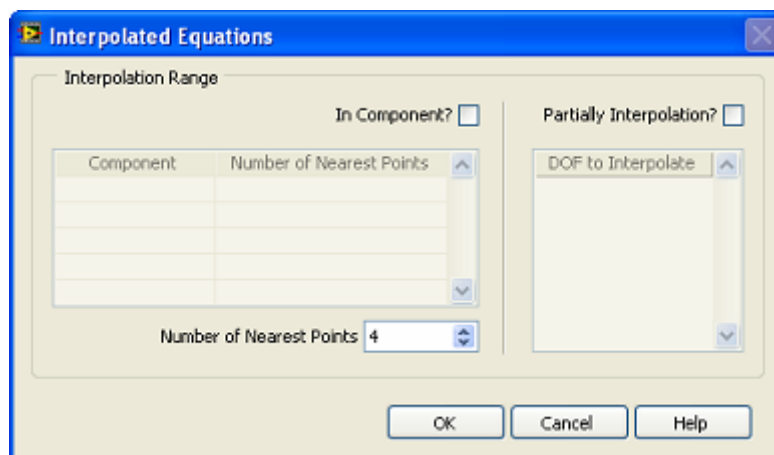
![In Component?] decides if interpolated motion of unmeasured nodes are computed from the measured nodes belonging to the same component.

Partially Interpolation? ☐ decides if interpolated motion of unmeasured nodes are partially computed only for specified DOFs.

Input number of nearest points in the dialog box and click **OK**. The interpolated animation is generated.

To clear specified measure type and assigned measurement for selected node

- Select nodes in object list.

- Click button [icon] on the toolbar.

For more information about specifying measurement properties for each node, please refer to chapter **Data assignment**.

1.12 Structure Group Window

Structure group window is used to define components and dummy structures in 3D model structure, so that the 3D model structure can be processed or animated at component level, as Figure 1-47 shows.
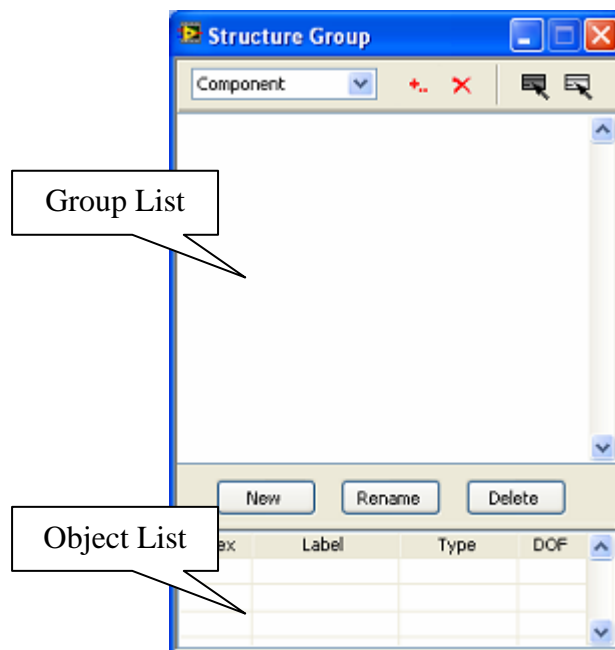
Figure 1-47 Structure group window

Edit Structure Group

To change group type to edit

- Choose group type [Component ▾] on the toolbar.

The object list will change accordingly. When the group type is component, the points of 3D model structure will be listed. When the group type is dummy structure, the created components will be listed.

To add a new group

- Press button [New] under group list

A dialog box will display for you to input group name.
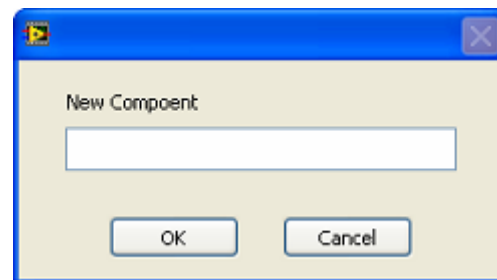
Type the desired group name and click **OK** button. The newly added group will be displayed in group tree list.

To rename a group

- Click to select a group in group list
- Press button [Rename] under group list

To delete a group from group list

- Click to select a group in group list
- Press button [Delete] under group list

To add items to a group

- Click to select a group in tree list
- Click to select items in object list

- Press button [+..] on the toolbar

The added items will be displayed in tree list as children of the group.
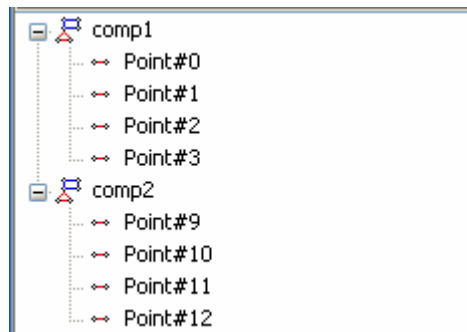
To delete items from a group

- Click to select children items of the group
- Press button on the toolbar

To select all items in object list

- Press button on the toolbar

To select none item in object list

- Press button on the toolbar

For more information about component and dummy structure, please refer to chapter **Component & Dummy structure.**

1.13 Mode Estimation Window

Mode estimation window is used to extract modes from time record or FRF data block, and save estimated modes into mode shape file, as Figure 1-48 shows.
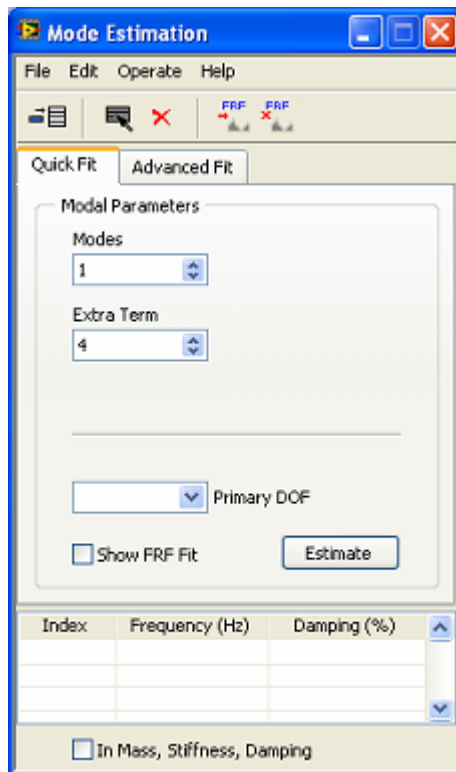
Figure 1-48 Mode estimation window

Quick fit modes

To quickly estimate modes from FRF data block

- Select FRF data block range with band cursor in the data block window.
- Select tab page **Quick Fit** as Figure 1-49 shows.
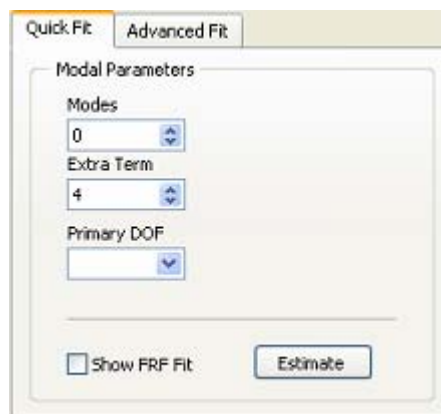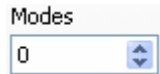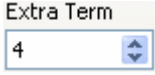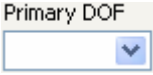
Figure 1-49 Quick fit tab page

- Specify mode number to fit in control box .

- Specify extra term in control box .

- Select primary DOF control box .

- Press button , the estimated modes will be displayed in spreadsheet

To check if the fitted FRF match its original FRF data block

- Select check box to display the fitted FRF on original FRF data block in data block window.

To add current estimated modes into mode list

- Press button on the toolbar.

The newly estimated modes will be added into mode list. the symbol " ✿ " before corresponding items in mode list will disappear, as Figure 1-50 shows.

Index	Frequency (Hz)	Damping (%)
✿ 0	341.408	0.054

⬇

Index	Frequency (Hz)	Damping (%)
0	341.408	0.054

Figure 1-50 Add modes to mode list

📝 **NOTE: If the newly estimated modes are not added to mode list, they will be overwritten by next estimation.**

For more information about quickly mode estimation, please refer to section **Quick fit**.

Advanced fit modes

To fit modes from time record or FRF data block with advanced process

- Preprocess time record in the data view window
- Or specify FRF data block range with band cursor in the data view window.
- Select tab page **Advanced Fit** as Figure 1-51 shows.
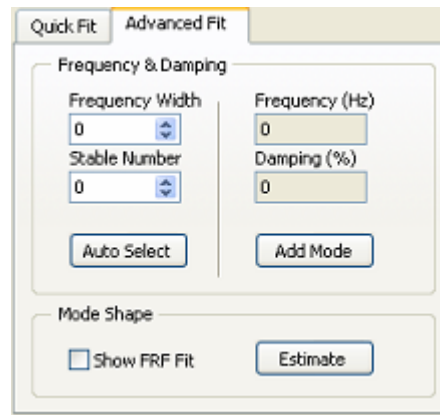
Figure 1-51 Advanced fit tab page
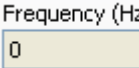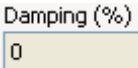
The stabilization chart window will open. For details about stabilization chart, please refer to section **Stabilization chart**.
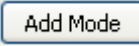
- Specify stabilization criteria and press button [Generate] in stabilization chart window**.**

There are two methods to select stable modes from stabilization chart: manual select and auto select.
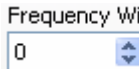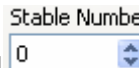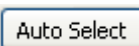
To manually select modes from stabilization chart

- Move the mouse pointer to mode character on stabilization chart, and click the left mouse button**.**
- The values of frequency and damping ratio of selected mode will be displayed in control box [Frequency (Hz)] and [Damping (%)].
- Press button [Add Mode] to add selected mode into spreadsheet.
- Repeat above steps to add more stable modes.
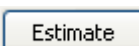
To automatically select modes from stabilization chart

- Specify frequency width in control [Frequency Width].
- Specify stable number of modes in control [Stable Number].
- Press button [Auto Select] to select stable modes into spreadsheet.

When stable modes are selected into spreadsheet list

- Press button [Estimate] to estimate mode shape for selected stable modes

To check if fitted FRF match its corresponding FRF data block

- Select check box to display fitted FRF on FRF data block in data block window.

To add current estimated modes into mode list

- Execute menu item **Edit | Add Fit Modes**
- Press button on the toolbar.

For more information about advanced mode estimation, please refer to section **Advanced fit**.

Change mode representation

To change mode display from Frequency-Damping into Mass-Stiffness-Damping

- Check at the bottom of mode list.
- Specify **Reference Mass** in quick fit tab page.

Edit mode list

To select all modes in spreadsheet list

- Execute menu item **Edit | Select All**
- Or press button on the toolbar.

To delete selected mode in spreadsheet list

- Select a mode in spreadsheet list
- Execute menu item **Edit | Delete**
- Or press button on the toolbar.

To sort modes in frequency ascending order

- Execute menu item **Edit | Sort | Ascending**

To sort modes in frequency descending order

- Execute menu item **Edit | Sort | Descending**

Synthesize FRF

To synthesize FRF from selected modes

- Select modes in spreadsheet
- Execute menu item **Operate| Synthesize FRF**
- Or press button in the toolbar

The synthesized FRF plot will be displayed on the FRF data block in data block window.

To delete synthesized FRF

- Execute menu item **Operate| Delete Synthesized FRF**
- Or press button in the toolbar

Save Mode List

To save mode list to mode shape file

- Execute menu item **File | Save Mode Table…**

A file open dialog is displayed to prompt you to input desired new file name.
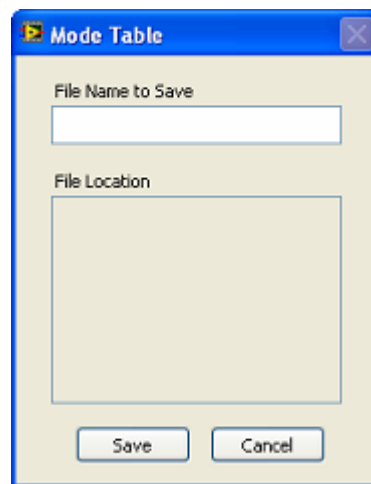
Type the desired file name into the **File Name to Save** box and click **Save** button.

📝 **Note: If there is a file with the same file name already in the project, a warning dialog box will be shown to prompt you to change the input file name.**
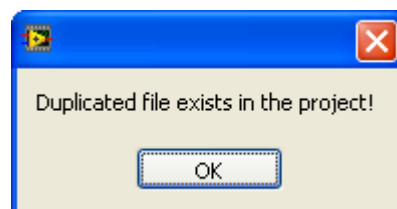
Copy to report generator

To copy modal analysis settings to report generator

- Execute menu item **File | Copy to Report | Analysis Settings…**

A dialog box is displayed to prompt you to input desired report item name. Type the name in **Report item name** box and click **Ok** button, the analysis settings is added to report item list accordingly.

When report generator window being opened, the added report items are shown in **Available Report Items** list. Please refer to section **Report generation** for more details.
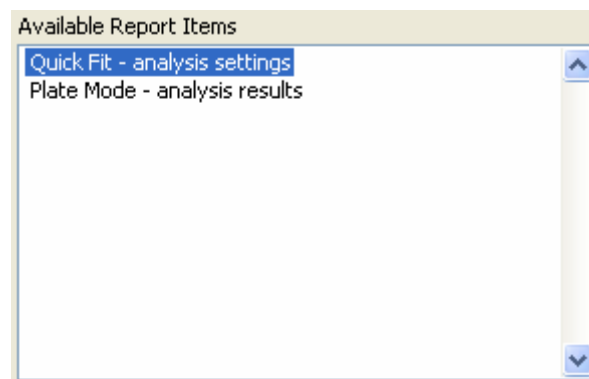
To copy mode list to report generator

- Execute menu item **File | Copy to Report | Mode List…**

1.14 Stabilization Chart Window

Stabilization chart window is used to generate and display stabilization chart. It helps for modes selection in advanced mode fit process, as Figure 1-52 shows.
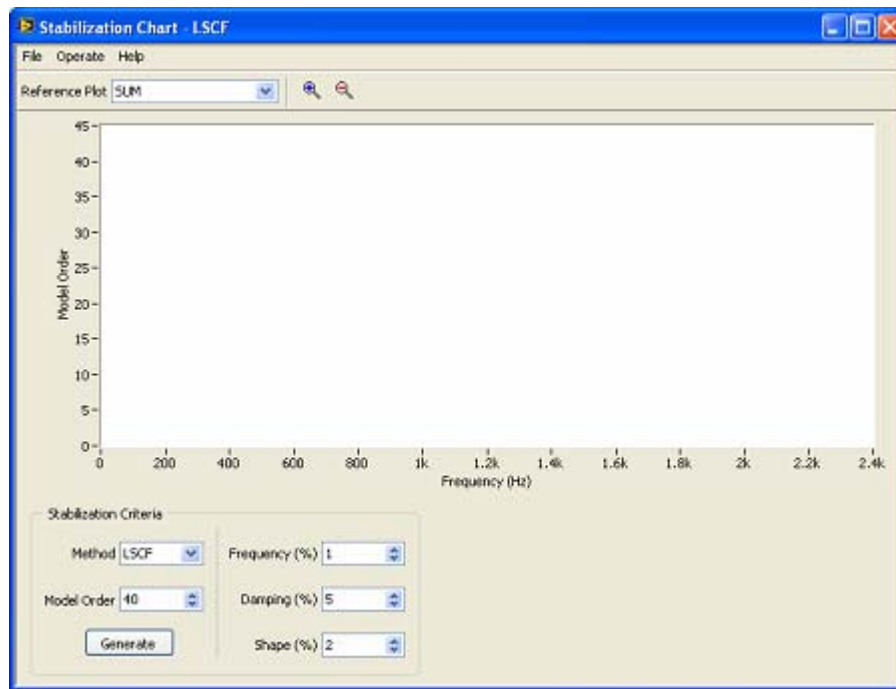
Figure 1-52 Stabilization chart window

Generate stabilization chart

To generate stabilization chart
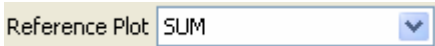
- Specify stabilization criteria in control box: **Model order**, **Frequency (%)**, **Damping (%)**, and **Shape (%)**.

- Execute menu item **Operate | Generate**, or press button Generate.

The generated stabilization chart is represented using a group of symbols, s, v, d, f and o. Each symbol represents an estimated mode. The symbols indicate the estimated mode is stable or not.

Symbol	Frequency	Damping ratio	Mode shape
s	yes	yes	yes
v	yes	no	yes
d	yes	yes	no
f	yes	no	no
o	no	no	no

Select measure to display

To select FRF data block to show in stabilization chart

- Select corresponding items in ring control Reference Plot SUM

98

The selection of **NONE** will show no FRF data block in stabilization chart.

The selection of **ALL** will show all FRF data block in stabilization chart.

The selection of **SUM** will show the summation of all FRF data block in stabilization chart.

The selection of **MMIF** will show the multivariate mode indicator function plot in stabilization chart.

The selection of **CMIF** will show the complex mode indicator function plot in stabilization chart.

Change stabilization plot display

To zoom in the data plot

- Click button on the toolbar.
- Mouse left key down on the data plot and drag the mouse along abscissa of stabilization plot.

To zoom out the data plot

- Click button on the toolbar.

Copy to clipboard

To copy stabilization chart into clipboard as a Bitmap

- Execute menu item **File | Copy to Clipboard | Bitmap**.

1.15 Excitation Window

Excitation window is used to generate the stimulus signal from hardware front end, as Figure 1-53 shows.
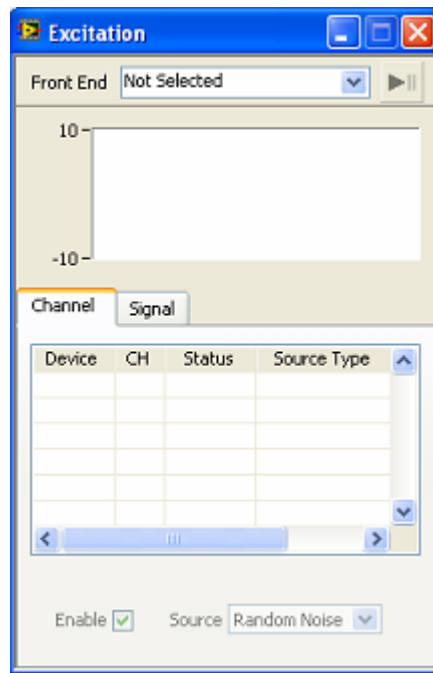
Figure 1-53 Excitation generation window

To connect to front end for signal generation

- Click **Front End** list box on the toolbar in **Excitation** window.

Signal generation hardware will be displayed in the list box and are ready to use. The digit in the square brackets indicates the number of same hardware boards installed.
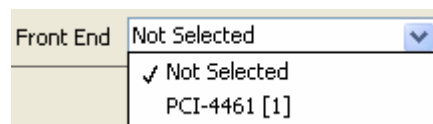
📝 **NOTE: ModalVIEW only support signal generation hardware of National Instruments.**

- Select a signal generation front end in the list.
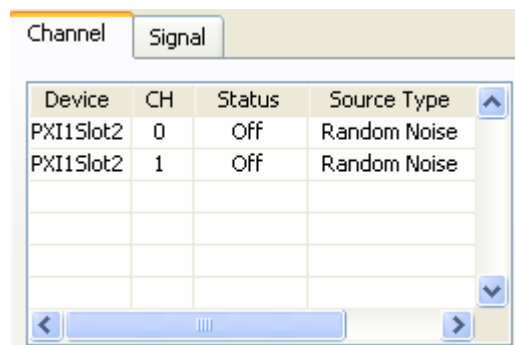
After front end being connected, the start button ▶❚❚ is enabled.

To disconnect acquisition window from front end

- Click Front End [PXI-4461 [1]] list box on the toolbar.
- Select **Not Selected** item in the list box.

Specify output channel
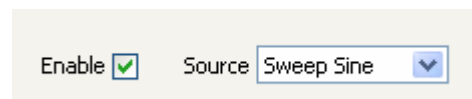
After the front end being connected, the available channels for signal output are list in spreadsheet on **Channel** tab.

To change the channel properties
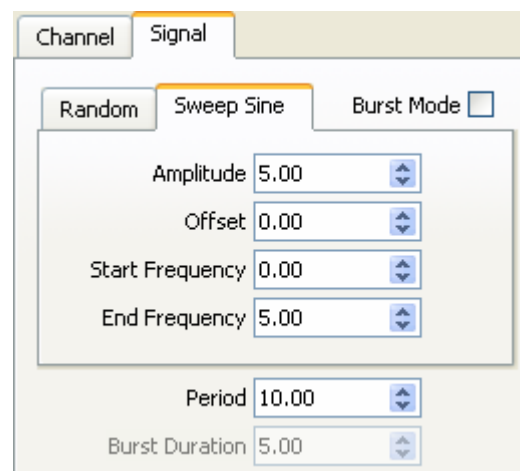
- Select a row in spreadsheet, property items under spreadsheet are enabled to change.

Specify output signal

Signal tab contains the parameters for output signal. Please refer to section **Excitation output** for details about configuring output signal.

Perform generation

To start & stop signal generation

- Click button on toolbar.

1.16 System Configuration

System settings window is used to configure running attributes of ModalVIEW, as Figure 1-54 shows.
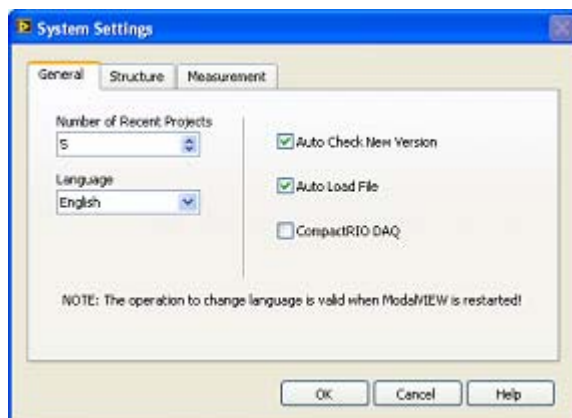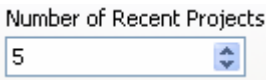
Figure 1-54 System settings window

Recently opened project

ModalVIEW automatically records the file paths of recently opened project, and list the file paths on menu **File | Recent Projects** in project window. By clicking corresponding menu item, the project file can be opened directly.

To specify the number of recorded projects to keep

- Specify [Number of Recent Projects 5] on general tab page in system settings window.

Language interface

ModalVIEW supports two language interfaces: English and Chinese.

To specify the language interface

- Specify [Language English] on general tab page in system settings window.

> **Note: The operation to change language is valid when ModalVIEW is restarted.**

For more information about settings in structure and measurement tab page, please refer to corresponding sections, **Digital movie** and **Double hit detection**.

2. Creating Geometry Model

Before actually taking modal testing, a 3D representation for structure under test must be prepared. The 3D structure model is so called a wire-frame model. It comprises of points and lines connecting the points. The arrangement information for sensors and shakers are also included in the 3D model. The points on the wire-frame model define positions where the measurement will be taken or the artificial stimulus will be applied. A local coordinate system associated with each point defines measurement or stimulus directions.

2.1 Coordinate system

There are two coordinate systems used in ModalVIEW to define geometry model, global coordinate system and local coordinate system. Both of them are defined as right handed Cartesian coordinate system.

The Figure 2-1illustrates the coordinate systems used in ModalVIEW.
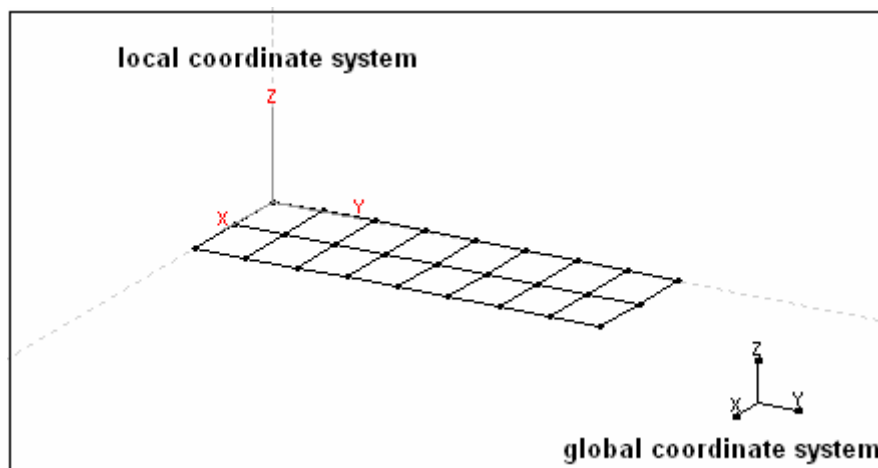
Figure 2-1 Coordinate system

Global coordinate system

The global coordinate system is the overall reference frame in which the geometry model is defined. The location of each node on the geometry model is defined by a set of real numbers, the coordinates defined in global coordinates system.

Local coordinate system

Local coordinate system is used to define the measurement directions for each node. Original point of each local coordinate system is superposed with each node position. The default axis orientation of local coordinate system is consistent with the global. A local coordinate system is also used to define separate substructures.

2.2   Model components

ModalVIEW uses nodes, lines and substructures to describe a wire-frame geometry model as Figure 2-2 shows.
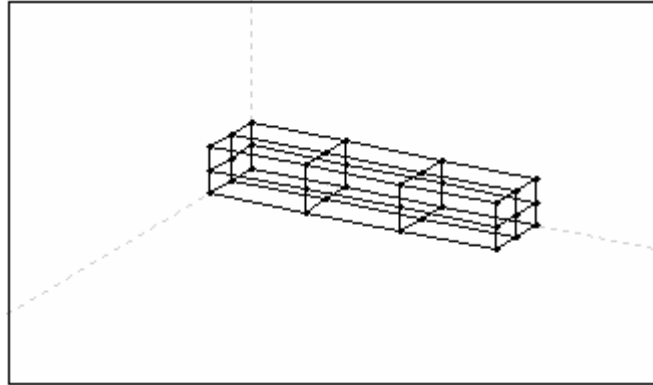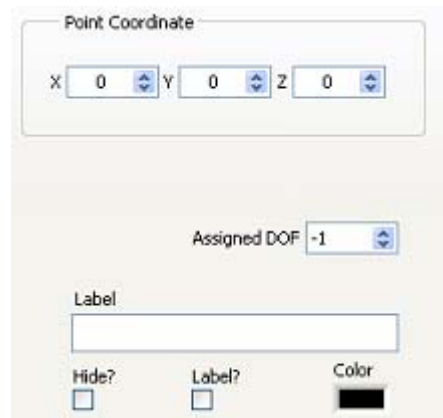
Figure 2-2 Wire-frame geometry model

Node

Node is the basic element in the wire-frame model. In ModalVIEW, each node is specified by its label, location, and assigned number to specify structure degree of freedom (DOF). In structure editor window, you can specify these values in tab page of point properties.

Line

Line is connected between different nodes to give more visibility to geometry model. Each line is specified by its label and indices of connected points. In structure editor window, you can specify these values in tab page of line properties.

Surface

Surface is used for visual representation of the structure. There are triangular surface and quadrangular surface defined. Triangular surface is specified by its label and indices of connected three points. Quadrangular surface is specified by its label and indices of connected four points. In structure editor window, you can specify these values in tab page of surface properties.

Substructure

Substructure is an assembly of nodes and lines. It is convenient to create and manipulate a complex structure by using several substructures. Each substructure is specified by its label, geometry type & dimension parameter, and local coordinate system. In structure editor window, you can specify these values in tab page of substructure properties.
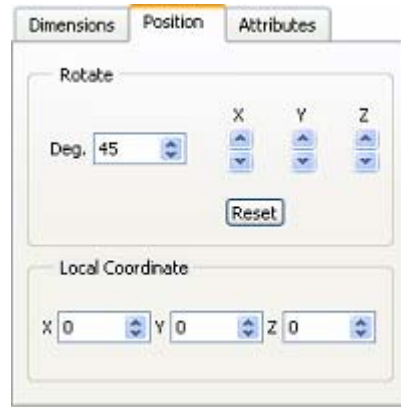
Component & Dummy structure

Component is defined as a group of nodes including connected lines and surfaces. Each node can belong to only one component. Dummy structure is defined as a group of components. Each component can be part of multiple dummy structures, as Figure 2-3 shows.
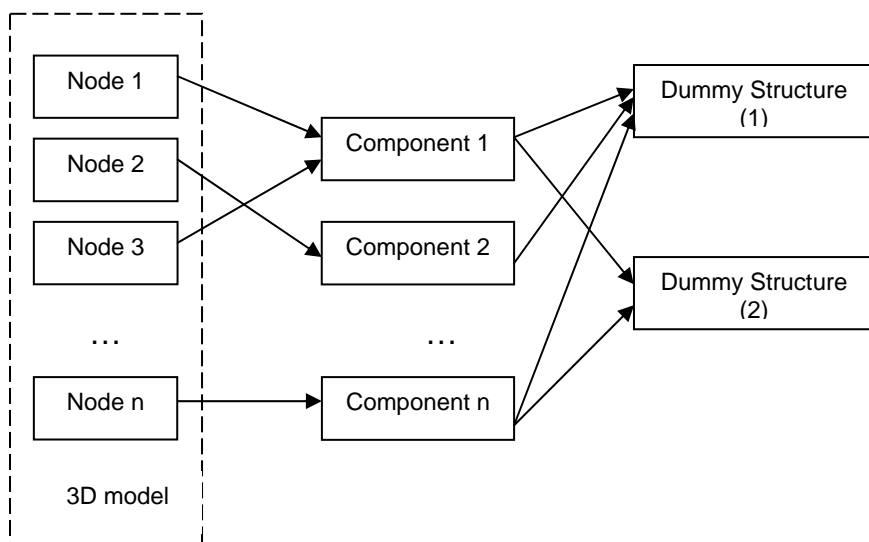
Figure 2-3 Component and dummy structure

By grouping a complex structure into a series of components and dummy structure with physical meaning, the motion of unmeasured nodes can be accurately interpolated at component level. The dummy structures can also be displayed and animated separately.

2.3 Degree of Freedom (DOF)

From a practical point of view, degree of freedom is defined as the independent motion of the physical structure at given location and direction. Motion at a node on the geometry model for a direction is named a degree of freedom, or DOF. The total

number of DOFs for a given geometry model can be computed as the number of nodes times the number of movement directions at each nodes. For example, the accelerations of x, y, z directions are measured at each of 100 nodes on a general deformable body. The number of DOFs would be equal to 300.

In ModalVIEW, DOF for a node is represented by a node number followed by axial label of its local coordinates system. For example, DOF '1Z' represents a DOF with node number being '1' and direction axial being 'Z'.

The DOF numbers are used as mapping reference when connecting sensors to channels of data acquisition hardware in model testing experiment. They are also used as mapping reference when assigning measurement data to nodes on 3D model in structure animation. In ModalVIEW, it is required to assign a unique DOF number to each node which is actually measured with a sensor.

2.4   Measurement properties

Measurement properties for each node define the relationship between node motion and measurement data during structure animation. The measurement properties for each node include **Measure direction** and **Measure type**.

Each node on 3D model has a local coordination. The X, Y, Z axes of local coordination defines three measurement directions. The directions of X, Y, Z axes coincide with global coordination of the 3D model by default. In case the actual measurement direction of the sensor mounted on physical object does not coincide with default directions of measurement axes, you can adjust the measurement axes of each node in measurement properties window.

There are three measure types defined for the measurement directions of each node on 3D model: **Measured**, **Interpolated** and **Fixed**. The default measure type for each measurement directions is interpolated.

Measured nodes on 3D model correspond to the measurement points on physical object, where measurements are actually made. Generally, there are much more nodes used to create 3D model structure than the number of actually measured points on physical object, where the sensors are fixed on. **Interpolated** and **fixed** nodes have no corresponding measurement data with them. During structure animation, the motion of measured nodes is determined by assigned measurement data (time data, FRF data, or mode shape). The motion of interpolated nodes is computed according to the motion of neighboring measured nodes. The fixed nodes are not animated.

2.5   Building a new model

There are a few methods for you to create a 3D model structure in ModalVIEW.

- Add nodes to the drawing, and edit coordinate of each node. Then, add lines between the nodes.

- Add substructures to the drawing, and modify them as required.
- Import 3D model from ASCII or UFF file, which contains information of node coordinates and line connections.

If the structure's photo or design drawing is available, ModalVIEW also provides the 3D structure model construction from 2D structure profile, which is depicted from structure image or design drawing. Please refer to the section **Generate 3D model from image** for details.

Before drawing a 3D model, structure viewer window and structure editor window need to be opened as Figure 2-4 shows.

- Click quick start item **Drawing 3D Structure Model** in project window.
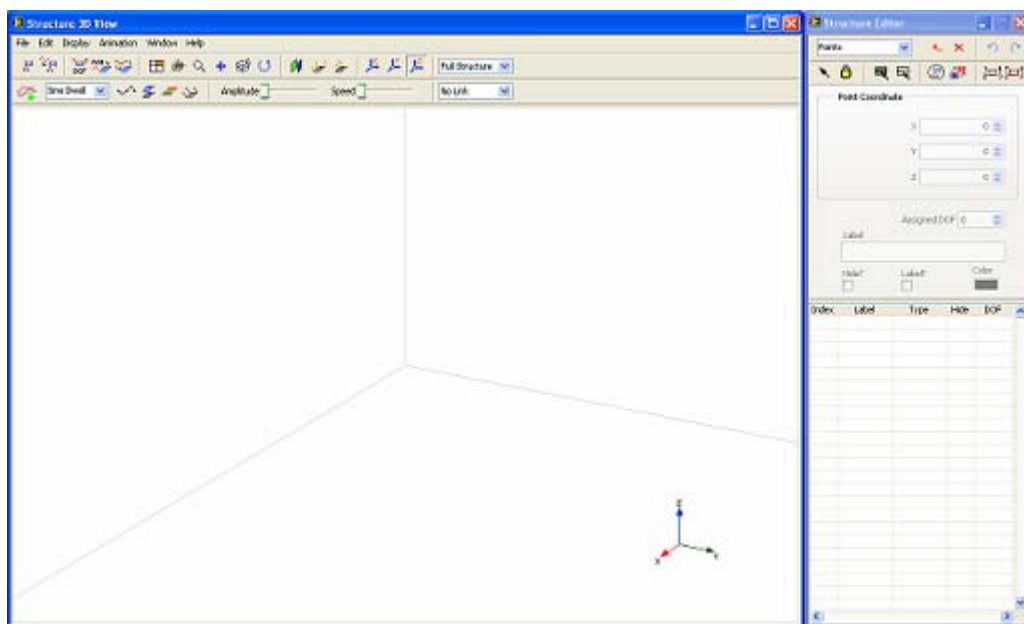
Figure 2-4 Interface for 3D model drawing

As an example, how to build a plate structure with different methods are illustrated in following sections.

Build model with nodes and lines

To add nodes to the model

In structure editor window:

- Select object type as **Points** on the toolbar

- Press button ![button] on the toolbar to enable add operation.

In structure view window:

- Click button ![button] to change window into Quad view.

- Click mouse cursor in structure view window to add four nodes, as Figure 2-5 shows.
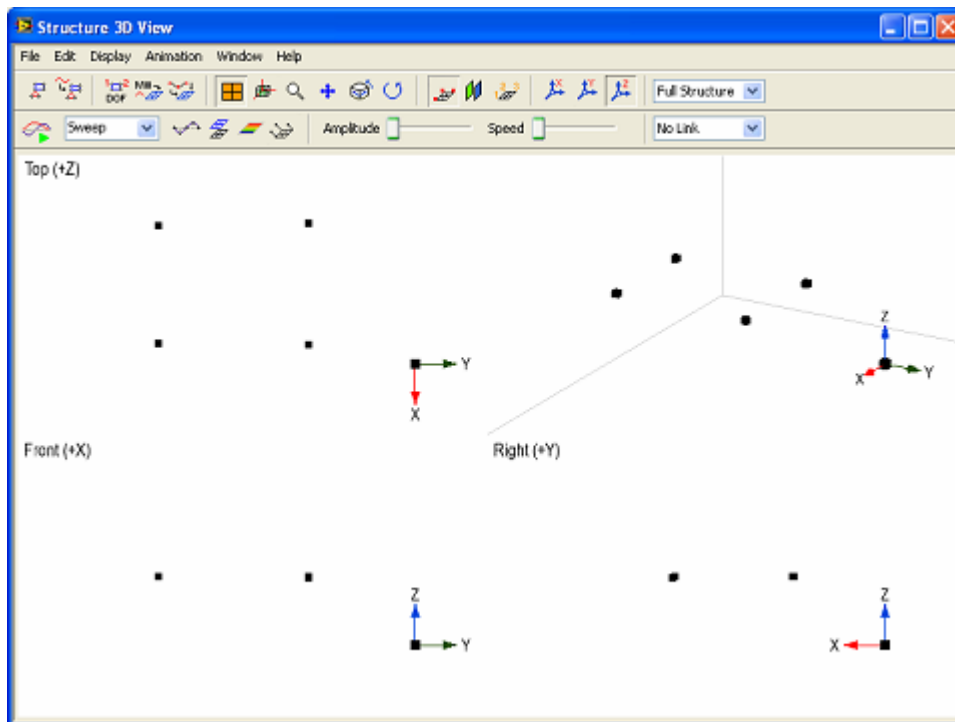
Figure 2-5 Quad view display

To add lines to the model

In structure editor window:

- Select object type as **Lines** on the toolbar
- Press button on the toolbar to enable add operation.

In structure view window:

- Click a node in structure 3D view.
- Move mouse cursor to another node and click it.

A line will be added between two nodes, as Figure 2-6 shows.
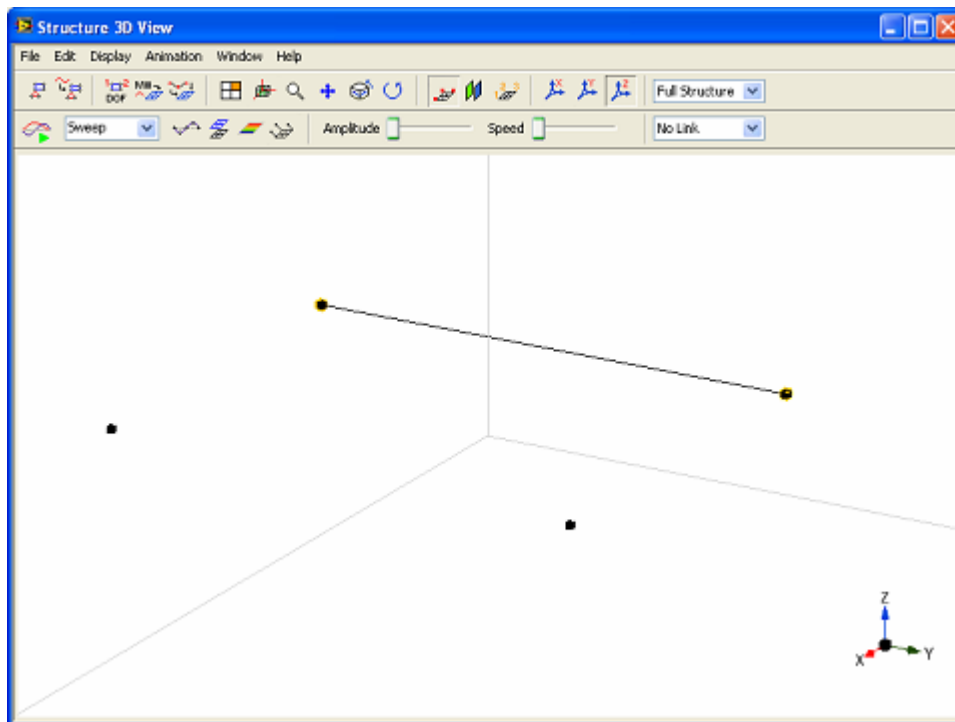
Figure 2-6 Operation to add line between points

- Repeat above steps to add other three lines.

Build model with substructures

To create a plate structure with predefined substructures

In structure editor window:

- Select object type as **Substructures** on the toolbar
- Press button ![icon] on the toolbar.

A dialog box will display for you to select a new substructure.
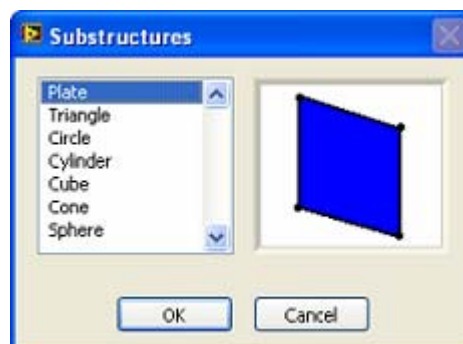
Select substructure **Plate** and click OK, a plate structure will be added to 3D model.

- Select the new added plate structure in the spreadsheet

- In the **Position** property page, press down arrow of rotation control **Y** to change plate structure into a horizontal one.

Import model from files

Model structure in UFF, ASCII and STL file format can be imported into structure view window. You can edit a model structure by using tools from third party and save the structure into UFF file or ASCII spreadsheet.

For details about UFF, ASCII and STL file format import for structure model, please refer to section **Importing External Files**.

Assign DOF number to node

There are three methods to assign DOF number:

1. Automatically assigning
2. Manually assigning
3. Interactively assigning

Automatically assigning is used when all the nodes are measured points and need to be assigned DOF numbers. Manually assigning and interactively assigning are used when only part of nodes need to be assigned DOF number.

To automatically assign DOF number

- Click button on the toolbar in model editor window

To manually assign DOF number to node

- Open model editor window
- Choose the object type as **Points**.
- Select one node in spreadsheet list
- Specify in node property page.

To interactively assign DOF number to node

- Click button on the toolbar in structure view window. A floating dialog box **Number DOFs** is opened.
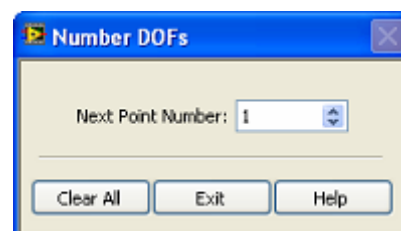
- Click nodes on plate structure one by one.

The **Next Point Number** in dialog box is automatically increased. The node will be assigned different DOF number as Figure 2-7 shows.
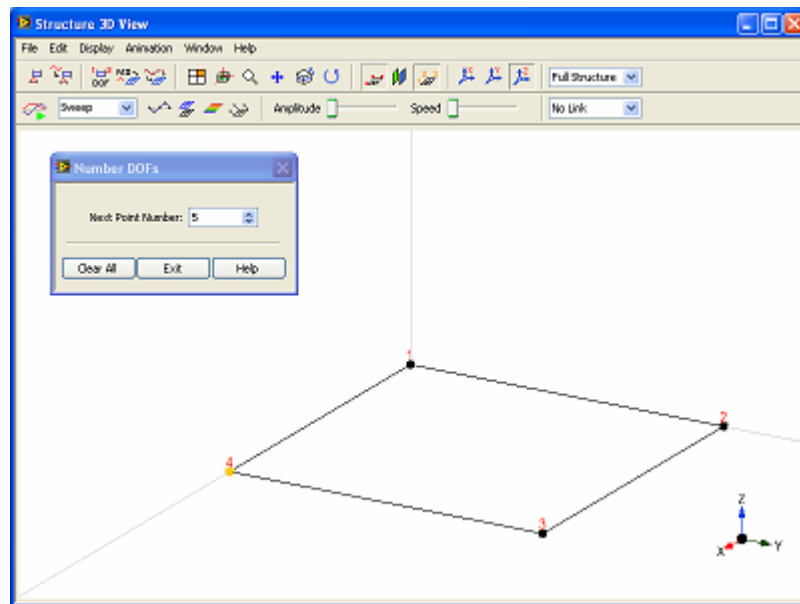
Figure 2-7 Node DOF assignment

Set measurement properties

To specify measurement properties of nodes

- Open measure properties window
- Select one node in spreadsheet
- In the **Measurement Axes** property page, press rotation control, **X**, **Y**, or **Z** to change the measurement directions of the selected node.
- In the **M# Assignment** property page, specify measure type for each measurement direction defined by X, Y, Z axes.
- Repeat above steps to specify measurement properties of other nodes.

At this time, the index of measurement data is not assigned to each measurement direction. You can assign it later when measurement data is ready.

 NOTE: It is not necessary to specify measurement properties for the unmeasured nodes without containing measurement directions of 'measured' or 'fixed' type.

After DOF number being assigned and measurement properties being specified for each node on 3D model, the new model structure is finished and is ready for use.

Generate 3D model from image

For an example, the following introduce the generation of 3D structure model for a gearbox from its design drawing.

To generate 2D structure profile from gearbox design drawing

- In project window, execute menu item **Operate | Image View…** to open image view window.
- In image view window, execute menu item **File | Load…** to load gearbox design drawing, as Figure 2-8 shows.

To depict nodes of the 2D structure profile

- Specify object type Points in toolbar as **Points**

- Click button to enable the add operation.
- Click on the appropriate place using mouse left button to add a new node on the structure image.
- Repeat above step to add other points, as Figure 2-9 shows.
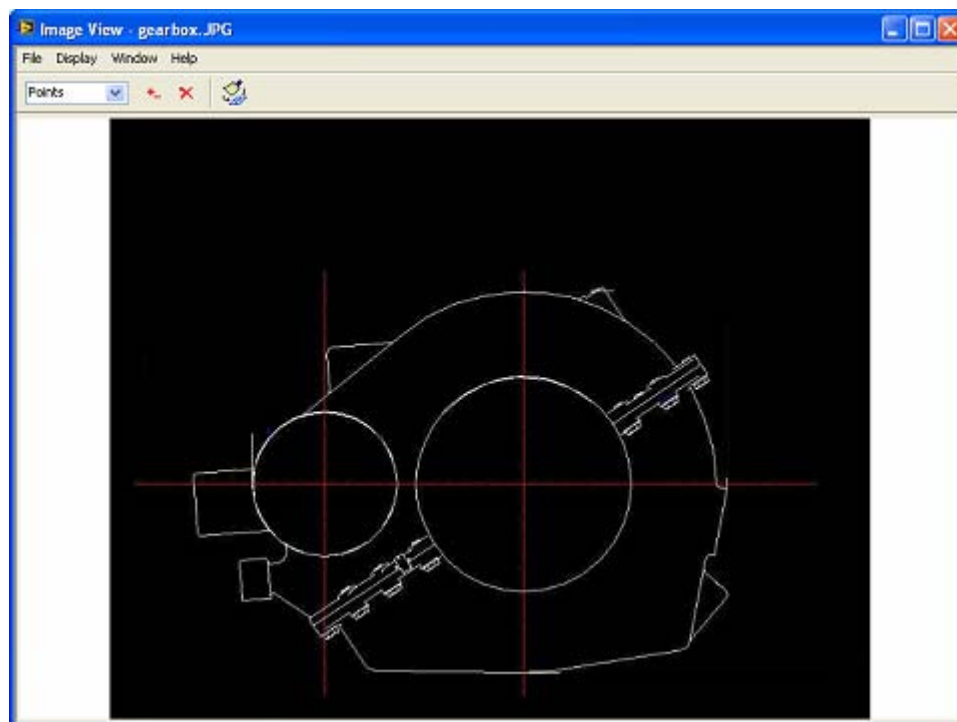
Figure 2-8 Design drawing of gearbox
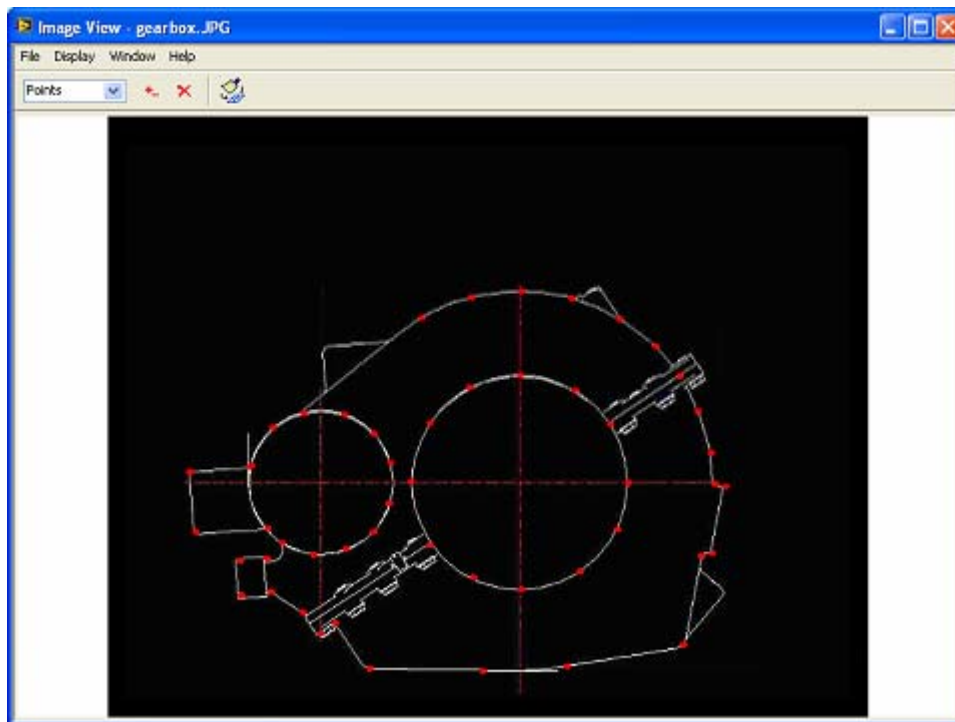
Figure 2-9 Points of 2D structure profile

To add line between nodes on 2D structure profile

- Specify object type `Points ▼` in toolbar as **Lines**
- Click button `+.` to enable the add operation.
- Click two nodes using mouse cursor sequentially and add a line between two nodes.
- Repeat above step to add other lines, as Figure 2-10 shows.
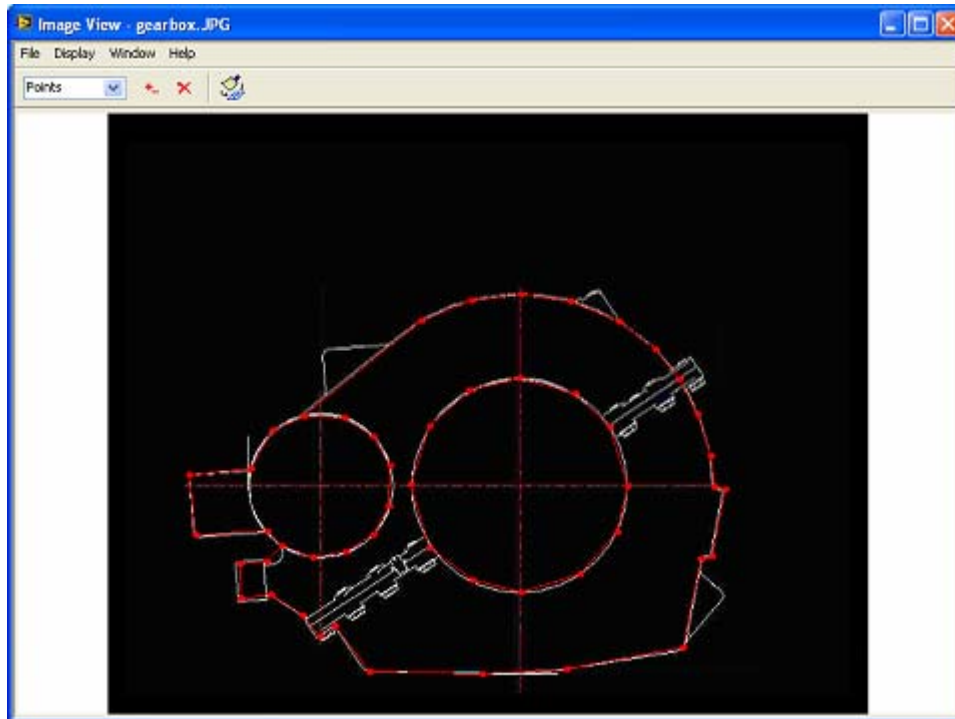
Figure 2-10 Points and lines of 2D structure profile

To save depicted 2D structure profile of the gearbox into a structure file

- Execute menu item **File | Save 2D profile…**

A file open dialog is displayed to prompt you to select desired new file name. Type the desired file name into the **New File Name** box and click OK button, a new structure file is saved and added into current project.

To generate 3D structure model from created 2D structure profile

- Click quick start item **Drawing 3D Structure Model** in project window.
- In structure view window, load previously saved 2D structure profile.
- In structure editor window, click button in the toolbar to extrude 2D structure profile.

A dialog box displays to prompt you to input extrude parameters include extrude direction and distance.
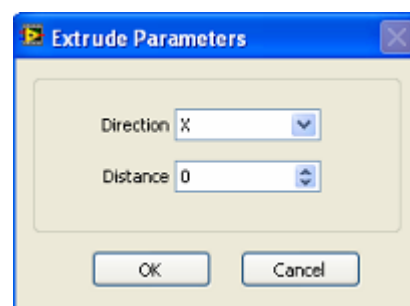
- Specify the extrude direction and distance as X axial and 300, respectively, click OK button. The 3D structure model of the gearbox is automatically generated from the 2D profile, as Figure 2-11 shows.
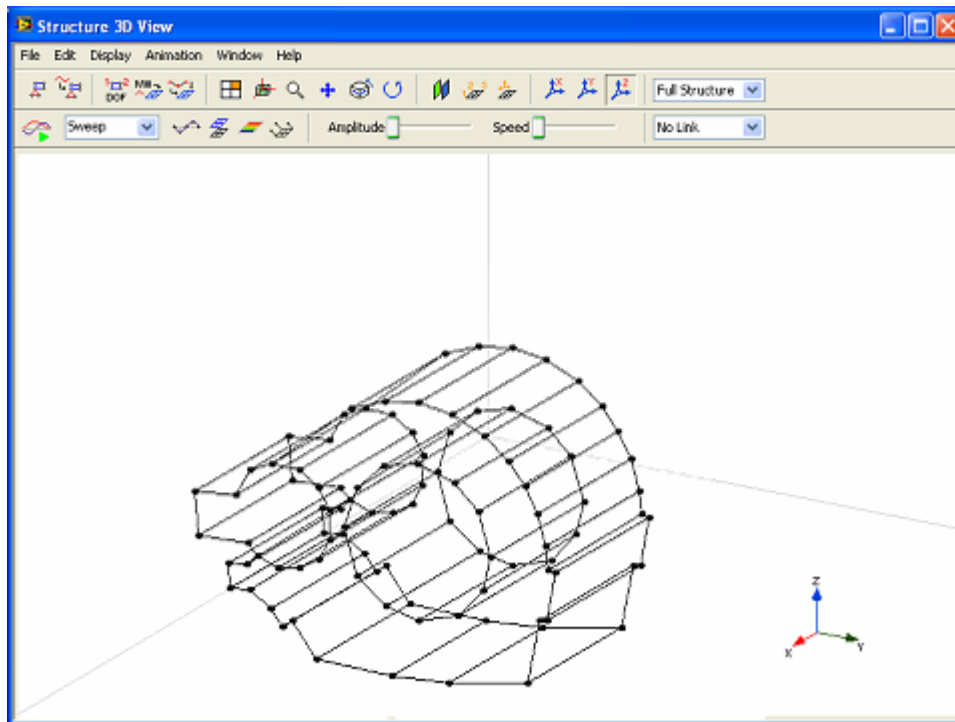
Figure 2-11 3D structure model of the gearbox

3.Performing Measurement

After 3D structure model being created, measurement experiment is performed to obtain vibration data of the structure. The vibration data can be measured as accelerations, velocities and displacements, which are acquired from corresponding sensors for each measured DOF. To display spatial deformation of the structure with time, the vibration data from each measured DOF must be simultaneously acquired, which needs a multi-channel data acquisition hardware system.

3.1 Measurement type

There are two types of measurement data used to analyze vibration of a structure: time domain operating data and frequency response function (FRF) data. The measured data can be directly used as animation source to display time-based or frequency based operating deflection shape (ODS) of the structure. Mode parameters (natural frequency, damping ratio and mode shape) of the structure can be further exacted from time record or FRF measurements. The structure can be animated with extracted mode shapes.

ModalVIEW supports three measurement selections: Time Record, FRF – EMA, and FRF –OMA.

Time Record is to simultaneously measure time domain data from sensors fixed on the structure, which is in operating state. The measured operating data can be used directly to visualize ODS of the structure.

FRF – EMA is to measure frequency response function (FRF) between vibration response from the structure and reference excitation force to the structure. A set of FRF measurements (FRF matrix) is commonly used to extract modal parameters of the structure in experimental modal analysis (EMA). It can also be used for ODS animation.

FRF – OMA is to measure FRF data synthesized from auto-spectrum of measured responses with phase information relative to a reference response. It is used to extract modal parameters of the structure in operational modal analysis (OMA), where ambient excitation force is unmeasured.

3.2 FRF Measurement

FRF defines the relationship, as a function of frequency, between excitation signal and response signal from a pair of DOFs on a structure. A set of FRF measurements from all pairs of DOFs on the structure creates a FRF matrix defined as Figure 3-1 shows.

$$\begin{Bmatrix} Y_1 \\ Y_2 \\ Y_3 \\ \vdots \\ Y_n \end{Bmatrix} = \begin{bmatrix} H_{11} & H_{12} & H_{13} & \cdots & H_{1n} \\ H_{21} & H_{22} & H_{23} & \cdots & H_{2n} \\ H_{31} & H_{32} & H_{33} & \cdots & H_{3n} \\ \vdots & \vdots & \vdots & \ddots & \vdots \\ H_{n1} & H_{n2} & H_{n3} & \cdots & H_{nn} \end{bmatrix} \begin{Bmatrix} X_1 \\ X_2 \\ X_3 \\ \vdots \\ X_n \end{Bmatrix}$$

Figure 3-1 FRF matrix

Y_i is the spectrum of response signal at DOF i, X_j is the spectrum of excitation signal at DOF j, and H_{ij} is the FRF between DOF j and DOF i. The FRF are estimated from the measured auto- and cross-spectrum of and between input and output (There are also some other methods to compute FRF). Each row of FRF matrix has the same DOF for response and different DOFs for excitation. Each column of FRF matrix has the same excitation DOF and different DOFs for excitation.

Single-reference testing

Theoretically, FRF matrix is symmetric. Each row of FRF matrix has the same FRF measurements as its corresponding column. And one row or column of FRF measurements contains sufficient information to estimate modal parameters of the structure. In practice, a set of FRFs are measured with a fixed excitation (roving response) or a fixed response (roving excitation), which is called single-reference modal testing.

In the fixed excitation test, a shaker is used as an excitation source, which is fixed at one reference DOF (point and direction) on the structure. Multiple responses are measured at different DOFs by roving sensors (unless there are available sensors for all response DOFs). The calculated FRFs correspond to one column of the FRF matrix, as Figure 3-2 shows.

Figure 3-2 FRF measurement with fixed excitation

In the fixed response test, an impact hammer is used as an excitation source, which applies excitation at different DOFs. Single response is measured at a reference DOF by a sensor fixed on the structure. The calculated FRFs correspond to one row of the FRF matrix, as Figure 3-3 shows.

$$\begin{Bmatrix} Y_1 \\ Y_2 \\ Y_3 \\ \vdots \\ Y_n \end{Bmatrix} = \begin{bmatrix} H_{11} & H_{12} & H_{13} & \cdots & H_{1n} \\ H_{21} & H_{22} & H_{23} & \cdots & H_{2n} \\ H_{31} & H_{32} & H_{33} & \cdots & H_{3n} \\ \vdots & \vdots & \vdots & \ddots & \vdots \\ H_{n1} & H_{n2} & H_{n3} & \cdots & H_{nn} \end{bmatrix} \begin{Bmatrix} X_1 \\ X_2 \\ X_3 \\ \vdots \\ X_n \end{Bmatrix}$$
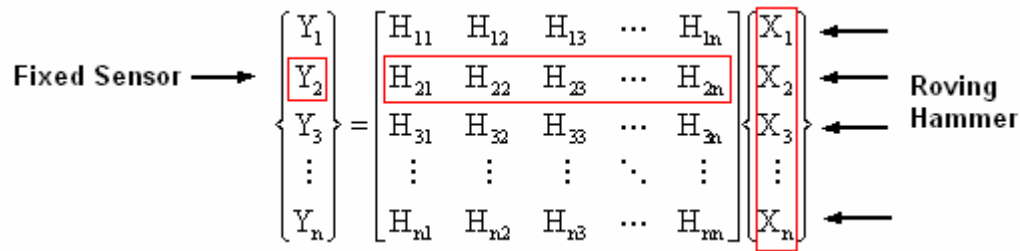
Figure 3-3 FRF measurement with fixed response

From above, the single reference testing is performed with a roving response (fixed excitation) or a roving excitation (fixed response). When performing FRF measurement, ModalVIEW provides two selections of measurement mode: **Roving Response** and **Roving Excitation**. In roving response mode, one acquisition channel is reserved for measuring excitation at reference DOF. The left channels are used for measuring roving responses. In roving excitation mode, one acquisition channel is reserved for measuring response at reference DOF. The left channels are used for measuring roving excitations.

Multi-reference testing

There are some situations require FRF measurements with more than one reference DOF. That is to say FRFs are measured with multiple fixed excitations or multiple fixed responses, which is called multi-reference modal testing. The calculated FRFs correspond to more columns or rows of the FRF matrix.

Multiple-reference testing can be used in following cases:

- A single reference DOF is not sufficient to find all modes.
- For some symmetrical structure, more mode shapes are corresponding to the same natural frequency.
- The structure can not obtain adequate excitation from one reference DOF.

The advantage of multiple excitations is that the energy can be uniformly distributed over the structure. It is especially necessary for the testing of large and complex structures.

ModalVIEW supports multi-reference testing. You just need to specify two or more reference DOFs in measurement settings. Acquisition channels will be reserved for measuring multiple fixed excitations or multiple fixed responses.

FRF Calculation

For single-reference testing, a single FRF measurement can be calculated as following methods:

$$H_1 = S_{XY}/S_{XX} \text{ or } H_2 = S_{YY}/S_{YX}$$

S_{XX} and S_{YY} are the auto spectrum of excitation and response signal, respectively. S_{XY} is the cross spectrum between excitation and response. S_{YX} is the cross spectrum between response and excitation. S_{XY} is conjugate with S_{YX}.

Windowing

Fast Fourier transform (FFT) is used to compute auto spectrum and cross spectrum. When the FFT of non-periodic excitation is computed, the resulting spectrum suffers from 'leakage'. Leakage is the smearing of computed spectrum. It leads to a distorted and inaccuracy FRF measurement. To reduce the leakage effect, a special windowing function is applied on the measured time domain data before FFT operation on it.

For different excitation signal, ModalVIEW provides several very commonly used windows. You can select proper window to use according to Table 3-1.

Excitation Type	Window Type
Periodic signal	Rectangular
Broad band random signal	Hanning
Narrow band or sinusoid signal	Flat top
Impulse (transient) signal	Force / Exponential

Table 3-1 Window type selection

Two extra parameters need to be specified for force / exponential window: **Duty cycle (%)** and **Final value (%)**.
Duty cycle (%) is to specify the percentage of time the force window has a value of "1" prior to and during the impulse signal versus "0" following the impulse.

Final value (%) decides how much the exponential window damps the value at the end of response signal in percentage. The default value is 10%. That imply at the end of response signal, the exponential window has attenuated the response signal by 90%.

NOTE: If the response signal decays to zero before the end of sampling due to the special excitation of impact or burst random, no window is required.

Averaging

In FRF calculation, spectrum average is used due to following reasons:

• Random noise in the measured signal.
• Non-linearity from the structure dynamics.

To improve measurement accuracy, FRF is obtained by averaging repeated measurements at the same excitation DOF and response DOF. Commonly used averaging types include **RMS averaging** and **vector averaging**. RMS averaging is used for random signal excitation. Vector averaging is used for impulse excitation. **Linear weighting** and **exponential weighting** are typically used weighting mode. In

a linear average, all measurements have the same weight to compute mean value. In exponential average, the last measurement has the most weight and the first measurement has the least weight.

By averaging, H_1 method can reduce the fluctuation due to the influence of random noise at the response measurement. H_2 method can reduce the influence of random noise at the excitation measurement.

In modal testing, the noise disturbance is mostly from response measurement. ModalVIEW selects the method H_1 and H_2 for FRF calculation. Average mode and average count can also be specified in measurement setting.

Coherence

In FRF measurement, coherence function is to check the correlation between stimulus signal and measured response signal. It is used as a quality assessment of measured FRF data. The coherence function is defined as following

$$\gamma = \left|S_{XY}\right|^2 / (S_{XX}S_{yy})$$

Coherence is always less than or equal to one. The low coherence indicates there is uncorrelated noise measured FRF data.

Synthesized FRF for OMA

Due to its physical size or boundary condition, sometimes it is impossible or difficult to excite the structure with external forces. Operational modal analysis (OMA) is used to identify modal parameters of the structure by utilizing only response measurement of the structure in operational condition. Without measuring excitation signal, FRF is synthesized from auto spectrum of roving responses combined with the phase of the cross spectrum between the roving response and a reference response.

ModalVIEW provides measurement type **FRF – OMA**. Synthesized FRF can be generated from measured responses and used to exact modal parameters for operational modal analysis.

3.3 Data acquisition

ModalVIEW uses dynamic signal acquisition (DSA) hardware of National Instruments as data acquisition front-ends enabling applications from 4 to hundreds of input channels. ModalVIEW can simultaneously acquire data directly from multi-channel DSA hardware. The installed DSA hardware can be identified and listed in front end list control in ModalVIEW measurement window.

The most commonly used NI DSA hardware includes:

NI 923x	4 channel
NI 446x	2 channel

NI 447x	8 channel
NI 449x	16 channel
NI 443x	5 channel

For more information about NI DSA hardware, please refer to National Instruments Corp, http://www.ni.com.

Setup measure channels

To setup measure channels involves following steps:

- Open measurement window
- Select DSA hardware from **Front End** list box

For an example, NI 4472 is selected
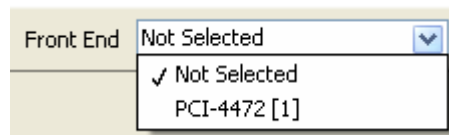
The digit in the square brackets indicates the number of same hardware boards identified. ModalVIEW will connect to all the same type of DSA hardware and automatically synchronize them.

- Click button on toolbar to configure input channels

In **DAQ setup** window, eight input channels are displayed in channel spreadsheet, as Figure 3-4 shows. The properties for each channel include: Status, Coupling, IEPE, Input Range, Sensitivity and Engineering Unit.

Device	CH	Status	Coupling	IEPE	Input Range	Sensitivity	Engineering Unit
PXI1Slot2	0	On	AC	4 mA	+/- 0.5 V	1000 mV/EU	V
PXI1Slot2	1	On	AC	4 mA	+/- 0.5 V	1000 mV/EU	V
PXI1Slot2	2	On	AC	4 mA	+/- 0.5 V	1000 mV/EU	V
PXI1Slot2	3	On	AC	4 mA	+/- 0.5 V	1000 mV/EU	V
PXI1Slot2	4	On	AC	4 mA	+/- 0.5 V	1000 mV/EU	V
PXI1Slot2	5	On	AC	4 mA	+/- 0.5 V	1000 mV/EU	V
PXI1Slot2	6	On	AC	4 mA	+/- 0.5 V	1000 mV/EU	V

Figure 3-4 Channel spreadsheet

Coupling is for selecting either AC or DC coupling on hardware channel. AC coupling removes the DC offset from the input signal. DC coupling allow measuring the entire signal. In vibration measurement, AC coupling is often selected.

IEPE is for specifying channel to generate the required current for IEPE sensors. Accelerometers for vibration measurement often need IEPE excitation. Please refer to sensor specification for IEPE selection.
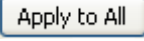
Input Range is for specifying the minimum and maximum analog signal levels for input channel. A smaller range can improve the measurement precision. The input range can not be specified arbitrarily, please refer to hardware specification for details.

Sensitivity is for specifying the sensitivity of the sensor in volts per engineering unit. Please refer to the sensor documentation or perform channel calibration to determine this value.

Engineering Unit is for specifying the units to use to return channel measurements.

For vibration measurement using NI 4472 front end and IEPE accelerometers, the following table shows one of typically used channel settings.

Coupling	IEPE	Input Range	Sensitivity	Engineering Unit
AC	4mA	+/- 10V	100 mV/EU	g

You can select one channel in channel spreadsheet and press button Apply to All , the settings of all the following channels will be updated into the same one as selected channel.

Perform channel calibration

In practice, it is necessary to check if the sensors are working properly as expected and to calibrate the sensitivity of sensors or the whole measurement chain. The values of calibration can be set back into channel sensitivity settings.

ModalVIEW uses AC calibration method to compute sensor/channel sensitivity. In this method, a sinusoidal load (reference signal) with known amplitude and frequency is applied to the sensor. The data acquisition front end measures the voltage signal from the sensor/channel to calibrate. The sensitivity of the sensor can be calculated from the amplitude of measured signal.

$$\text{Sensitivity (mV/EU)} = A_{measure} / A_{ref}$$

$A_{measure}$ is the amplitude of the measured signal in RMS.
A_{ref} is the amplitude of the reference signal in RMS and expressed in engineering unit.

To calibrate an input channel involves the following steps:

In **Calibration** tab page of **DAQ Setup** window

• Select the channels to be calibrated by checking the checkbox of each channel.

- Specify the frequency, amplitude and duration of the reference signal.

- Start the reference signal generation.

 NOTE: The generated reference signal must have a frequency value close to the specified reference frequency. The amplitude must be reasonably stable.

- Press button to start the aquistion for calibration, and the indicator will flicker unitl the calibration process ends.

Once the calibration process finish, the calibrated sensitivity will be displayed. Compare the measured sensitivity to the sensor documentation. If the difference of sensitivity lies within a reasonable limit, the new sensitivity value will be considered as good value. Or repeat the calibration process again.

- Press button to newly calibrated sensitivity into channel sensitivity settings.

Specify acquisition parameters

After measurement channels being configured, the acquisition parameters are to be specified on **Acquisition** tab including sampling rate, block size and triggering.

Sampling Rate (Hz) is specified above 2.5 times of interested maximum frequency component in measurement signal. For an example, if interested maximum frequency in measurement signal is 400Hz, the sampling rate are generally set about 1000Hz.

For measurement of period signal, **Block Size (Points)** is specified according to expected frequency resolution.

Frequency resolution = **Sampling Rate / (2 *Block Size)**

For measurement of transient signal, **Block Size (Points)** is specified to insure the signal decay to zero before the end of the acquisition.

To verify configuration of measurement channels and acquisition, you can check **Scope Mode**. In scope mode, signal are continually acquired from enabled channels and shown in the upper channel monitor area of measurement window. ModalVIEW provides two monitoring types: time record and power spectrum. You can select it in **Monitoring Type**.

Trigger box need to be checked when perform hammer impact testing. In trigger mode, the data acquisition will be held until trigger condition is met.

Trigger Source is to specify the measurement channel from which the signal is used as trigger source. The measurement channel for trigger source is commonly used for impact force acquisition.

Type specifies on which slope of the signal to start acquiring, positive slope or negative slope.

Level (%) specifies at what threshold level to start acquiring. The level is defined as percentage of maximum channel input.

Pre-trigger (%) specifies samples of data to acquire before the trigger occurs. Pre-trigger insures that entire impact signal is captured. The pre-trigger sample is defined as percentage of total block size, as Figure 3-5 shows.
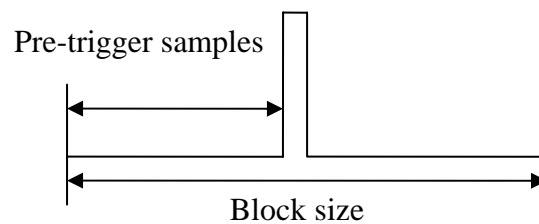
Figure 3-5 Pre-trigger diagram

The trigger level and pre-trigger are highlighted on channel monitor graph with a cross line, as Figure 3-6 shows. The horizontal line indicates the trigger level, and vertical line indicates the pre-trigger position.

Figure 3-6 Pre-trigger indication

After above data acquisition related parameters are specified, close DAQ setup window and return to measurement window.

125

3.4   Excitation output

For shaker excitation in testing experiment, a signal generator is used to generate excitation signal to drive the shaker. The commonly used excitation types include random, and sweep sin (Chirp). They all can be generated by burst mode. In burst mode, the excitation signal is continuously generated followed by a short period of no excitation. This makes windowing being unnecessary before FFT computation. ModalVIEW can generate uncorrelated random excitation signals for multi-reference testing.

ModalVIEW uses dynamic signal generation hardware of National Instruments as signal generator to drive the shaker. The installed dynamic signal generation hardware can be identified and listed in front end list control in ModalVIEW excitation window.

The most commonly used NI dynamic signal generation hardware includes:

NI 4461	2 channel
NI 4431	1 channel

For more information about NI dynamic signal generation hardware, please refer to National Instruments Corp, http://www.ni.com.

To setup excitation output involves following steps:

- Click button ![icon] on toolbar in measurement window.

In opened excitation window

- Select dynamic signal generation hardware from **Front End** list box

For an example, NI 4461 is selected
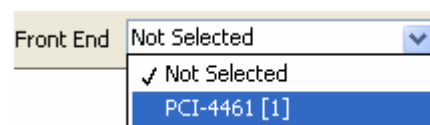
Signal generation hardware will be displayed in the list box and are ready to use. The digit in the square brackets indicates the number of same hardware boards installed in computer.

Setup output channels

After front end being connected, the available channels for signal output are list in spreadsheet on **Channel** tab. Each channel can be respectively specified signal type for output.

The properties for each channel include: Status, Source Type.

Status is to enable channel for output excitation.
Source Type is for selecting excitation type including random noise and sweep sine.

Specify excitation signal

After output channels are configured, the parameters for excitation signal are to be configured on **Signal** tab.

Amplitude specifies the peak value of excitation profile in voltage. **Offset** specifies the DC offset of excitation profile in voltage.

Start Frequency specifies the beginning of the sine sweep range in Hertz. **End Frequency** specifies the ending of the sine sweep range in Hertz.

Burst Mode is to select if excitation signal is generated in burst mode.

Period specifies the total time used to generate one single excitation profile in second. **Burst Duration** specifies the time of no excitation in burst mode. The duration is included in the total time for one period, as Figure 3-7 shows.
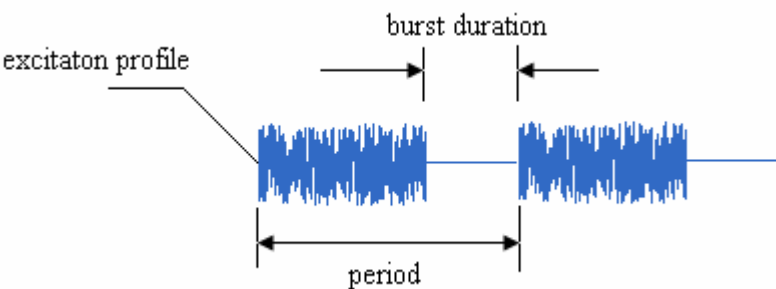
Figure 3-7 Burst mode diagram

Before starting excitation output, check the connection between dynamic signal generation hardware and the shaker amplifier.

3.5   Test setup

In test setup, measurement information such as testing type, number of DOFs to measure etc, and data post-processing such as windowing, averaging etc, are specified. According to test information, ModalVIEW will automatically create measurement sets, which define connections between acquisition channels, sensors and measure points on structure under test, to guide you to finish measurement in batches. The information of roving points or reference points for each measurement set can be highlighted on 3D model, which indicate where to arrange sensors on the structure and where to give excitation.

To setup testing experiment involves following steps:

- Click button on toolbar to open **Test Setup** window.

Specify measurement

To define a measurement, there are three parameters for you to select in measurement tab page: **type**, **number of node** and **mode**.

For different testing experiments, you can select **type** and **mode** according to the following table.

Testing Experiment		Selection	
		Type	Mode
Operational excitation	Time-based ODS	Time record	N/A
	FRF-based ODS	FRF - OMA	Roving response
	Modal analysis	FRF - OMA	Roving response
Roving hammer impact	FRF-based ODS	FRF - EMA	Roving excitation
	Modal analysis	FRF - EMA	Roving excitation
Fixed hammer impact	FRF-based ODS	FRF - EMA	Roving response
	Modal analysis	FRF - EMA	Roving response
Fixed shaker excitation	Modal analysis	FRF - EMA	Roving response

Operational excitation is used when it is difficult to apply artificial excitation the structure due to its physical size, boundary conditions or testing environment.

Number of Node is to specify how many nodes on structure model are consecutively assigned DOFs and to be measured as response points.

Select DOF range

DOF range is to specify which DOFs (nodes and directions) on structure model are to be measured in testing experiment. The roving DOFs and reference DOFs are specified respectively. For time record, only the range of roving DOF is required to specify. For measure type of FRF - EMA and FRF – OMA, the range of reference DOF is also required.

To specify the range of **Roving DOFs**

- Enter DOF number into [Roving DOFs 0] or check box [☐ All DOFs]
- Select measurement axial in [Z ▾]

The DOF number of **Roving DOFs** can be entered in number or in ranges separated by commas. For example: 1, 3, 5-12.

If **All** is selected for measurement axial, X, Y and Z axes will all be included.

If **All DOFs** is checked, Roving DOF numbers will be disabled and be automatically specified as 0, 1, 2, …, **Number of Nodes**.

To specify the range of **Reference DOFs**

- Enter DOF number into [Reference DOFs 0]
- Select measurement axial in [Z ▾]

The DOF number of **Reference DOFs** can be entered in number or in ranges separated by commas. For example: 1, 3, 5-12.

The total number of measurements is decided by the number of Roving DOFs. One measurement channel is required for each DOF.

For example, the setting as Figure 3-8 will generate 8*3 = 24 DOFs. The total number of measurements is 24.

Figure 3-8 Time record measure (24 DOFs)

The setting as Figure 3-9 will generate 11*3 = 33 DOFs. The total number of measurements is 33.

Figure 3-9 Time record measure (33 DOFs)

Acquire speed or sound signal

When performing model testing of the structures such as rotating machine, engines etc, it usually requires simultaneous acquisition of speed or sound signal besides vibration signal. In measurement tab page, if ☐ With Tacho or MIC is enabled, a dedicated acquisition channel will be reserved for tachometer or microphone when generating measurement sets.

Specify post-process

When the measure type is selected as FRF – EMA or FRF – OMA, some parameters for FRF calculation from time domain data are to be specified.

Window type is selected according to excitation type as in Table 3-1.

When impact excitation is used, average count is specified according to noise level in response measurement. The commonly specified value is 3-5. When random excitation is used, a large average count is needed.

When coherence is checked, the coherence plot will be displayed in measurement window during FRF measurement

Generate measurement set
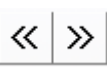
When the total number of measurements is greater than the number of available channels in DSA hardware, the measurements can be acquired in multiple measurement sets. ModalVIEW can automatically generate measurement sets according specified measurement type, DOF range and available measurement channels.

To generate measurement set

- Click button [Generate] on tab page **Measurement Sets**.
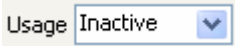
The measurement set will be listed in spreadsheet

Device	CH	Usage	DOF	Sensitivity
PXI1Slot2	0	Excitation	0Z	1000 mV/EU
PXI1Slot2	1	Response	1Z	1000 mV/EU
PXI1Slot2	2	Response	2Z	1000 mV/EU
PXI1Slot2	3	Response	3Z	1000 mV/EU
PXI1Slot2	4	Response	5Z	1000 mV/EU
PXI1Slot2	5	Response	6Z	1000 mV/EU
PXI1Slot2	6	Response	7Z	1000 mV/EU

- Click button [« »] to view different measurement set.

The usage and DOF assigned to acquisition channel can be manually changed in each measurement set.

To manually change the assigned measurement for acquisition channel

- Select the acquisition channel in measurement set.
- Change the usage in Usage [Inactive ▼].
- Select DOF number and measurement direction in DOF [0 ↕] [Z ▼].

After measurement sets for testing experiment are generated, close Test setup window and return to measurement window.

3.6 Testing process

After measurement sets are generated, it's time to start testing experiment. The testing process is shown in the diagram below.

```
          ┌─────────────────────────┐
     ┌───▶│  Select measurement set │
     │     └─────────────────────────┘
     │                │
     │                ▼
     │     ┌─────────────────────┐
     │     │  Data Acquisition   │◀──────────┐
     │     └─────────────────────┘           │
     │                │                      │
     │                ▼                      │
     │            ╱───────────╲          No  │
     │          ╱   Data is     ╲────────────┘
     │          ╲   valid?      ╱
     │            ╲───────────╱
     │                │ Yes
     │                ▼
     │     ┌─────────────────────┐
     │     │  Save measurement   │
     │     └─────────────────────┘
     │                │
     │                ▼
     │            ╱───────────╲
     │   No     ╱     All       ╲
     └─────────╱  measurement    ╲
               ╲     sets?       ╱
                ╲───────────────╱
                       │ Yes
                       ▼
                 ┌──────────┐
                 │   End    │
                 └──────────┘
```
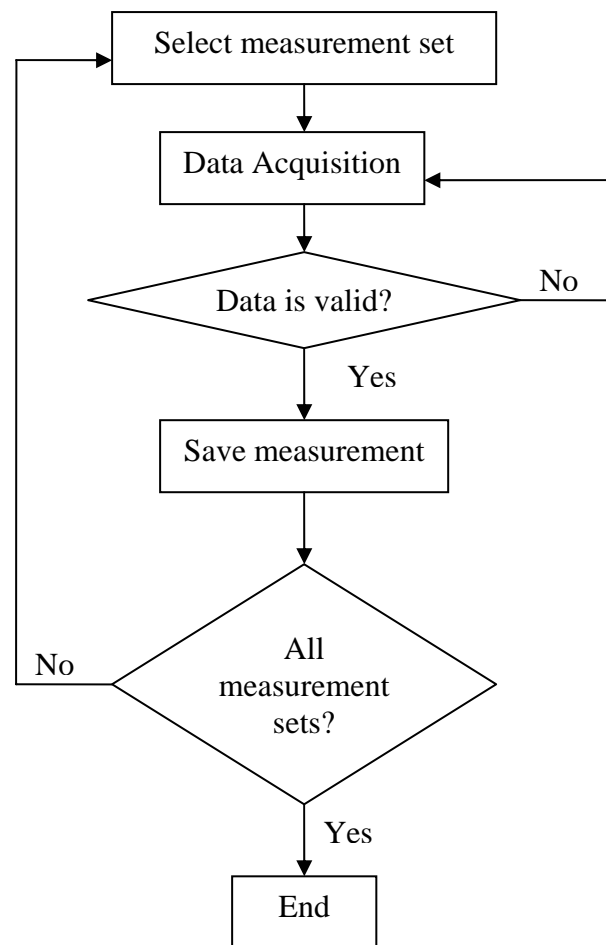
Measurement guidance

During experiment process, the information of data acquisition, testing setup, excitation and response DOFs for current measurement set, average count etc. will be displayed in measurement window.

```
-----------Measurement Status------------
Enabled Channels: 8
Sampling Rate: 1024 Hz
Block Size: 1024
Start Mode: Trigger (PXI1Slot2/CH0)
Measure Type: FRF - EMA
Excitation DOFs:
0Z
Response DOFs:
0Z  1Z  2Z  3Z  4Z  5Z  6Z
Average Count: 0 / 0
```

According to displayed measurement status, you can arrange sensors to corresponding DOFs on 3D model. Average count will tell how many measurements are performed and averaged for current measurement set.

The DOFs of excitation or responses for current measurement set can be displayed on 3D model structure to guide you to arrange sensors or apply excitation.

To display measurement guidance on 3D model

- Click button [icon] on toolbar in measurement window.
- Load 3D model structure into structure view window.

When measurement set is selected, the corresponding guidance will display on 3D model. Label 'S' indicates DOFs for excitation. Label 'R' indicates DOFs for response.
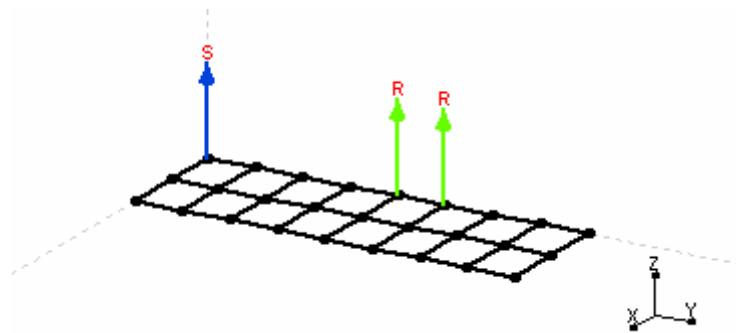
Double hit detection

In impact testing experiment using impact hammer to hit the structure under test, double-hits occur when the hammer contacts the rebound structure second time. The measured excitation signal is shown as in Figure 3-10.
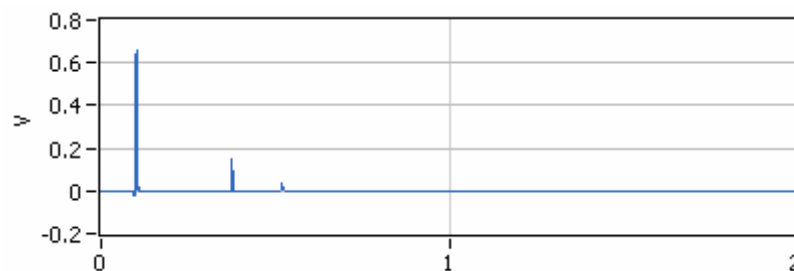
Figure 3-10 Excitation signal with double hit

The quality of measured FRF data will be deteriorated due to double hit of impact hammer. The operator should discard current FRF measurement. By default, ModalVIEW enables the double hit detection. When double hit is detected in excitation signal, a warning dialog window will be shown to prompt operator to discard current measurement.
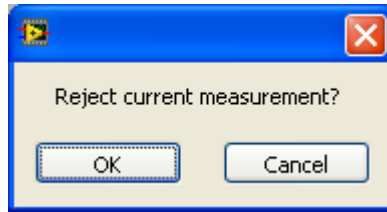
To setup double hit detection

- In project window, execute menu item **Operate | Option…**.

The system settings window is opened, as Figure 3-11 shows.
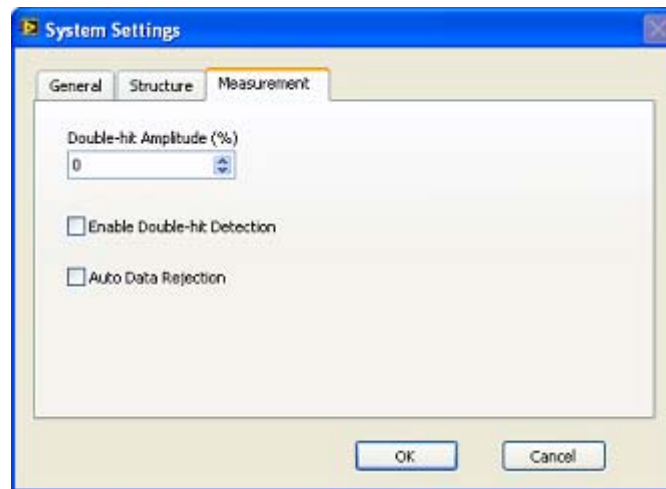
Figure 3-11 System settings window

In **Measurement** tab page

- Change into a new one. The default value is 60%. Double-hit is detected according to if the amplitude of second pulse exceeds the specified percentage of the amplitude of main pulse in measured impact signal.
- Check to enable double hit detection.
- Check to automatically discard measured FRF data when double-hit occurs.

Measurement steps

You can finish testing process as the following steps.

In measurement window

- Click button on toolbar

The structure view window is opened for the display of DOF guidance. This step is not required.

- Click button to select a measurement set.
- According to DOFs guidance displayed in measurement status, fix sensors to corresponding positions on structure and connect them with corresponding acquisition channels.
- Click button on toolbar to start data acquisition for current measurement set.
- Apply multiple excitations to the structure until average count being reached.

A dialog box will show to prompt you to keep current measurement to measurement list.
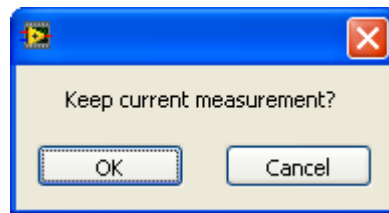
If the measured data is valid, click OK button to keep current measurement. Or click

Cancel button and restart data acquisition again by clicking button .

- Repeat above steps until all measurement sets are finished.
- Execute menu item **File | Save Measurement…** to save measured data into data file for further use.

3.7 Merge multiple test

When the measurement sets for the testing experiment are too many to be finished in one time, you can finish measurement sets in batches. The measured data for each time can be saved in a separated data file.

To merge separated data files into a single data file

- Open one of data files to merge in data view window.
- Execute menu item **Edit | Merge from File…**

A data block selection window is opened for you to select data file to merge. Select one data file and click OK, the selected data file will be merged into current opened data file.
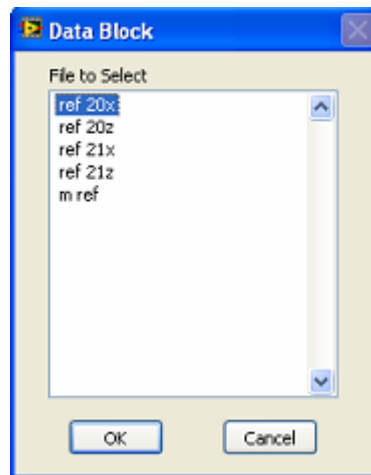
- Repeat above step until all the data files are merged in.
- Save the current opened data file as a new one.

The newly saved data file will contain all the measurement data merged from separated data file.

4. Modal Estimation

After a set of FRF measurements is obtained in testing experiment, modal parameters of the structure can be identified by curve fitting FRF data.

4.1 Methods

In modal parameters, frequency and damping are global characteristics of a structure. Theoretically, they can be found in any FRF measurement taken from different DOFs. To estimate mode shapes, the entire data set of FRF measurements from separated DOFs is required. According to how modal estimation is performed from the data set of FRF measurements, curve fitting methods for modal estimation can be classified as following:

SDOF vs. MDOF

SDOF method estimates modal parameters of a single mode at a time. MDOF method is to simultaneously estimate one or more modes from FRF measurements.

SDOF method is valid when the structure is simple and contains lightly coupled modes. In another word, the modes of the structure are well separated in frequency, and the frequency response at one frequency is lightly influenced by neighbouring modes. It is necessary to use MDOF method when the complex structure contains heavily coupled modes. That is the frequency response at any frequency is a combination of many modes.

Modal coupling can be indicated from the peaks in FRF plot, as Figure 4-1, Figure 4-2 shows.
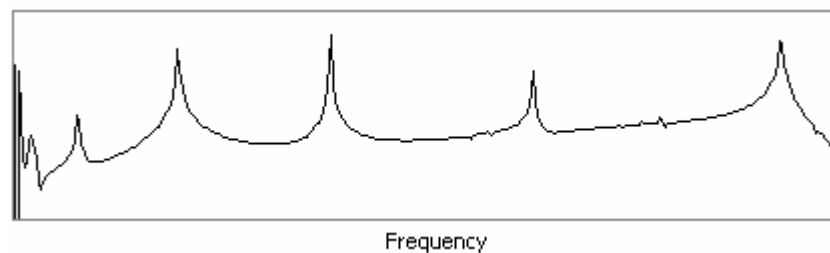
Figure 4-1 Lightly coupled modes
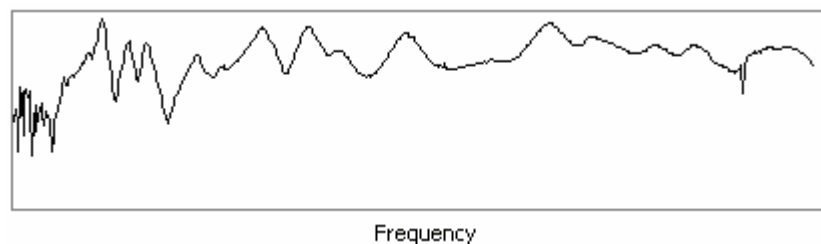
Figure 4-2 Heavily coupled modes

Local vs. global

Local method estimates modal parameters of frequency and damping from individual FRF measurement. Global method is to estimate frequency and damping from a full set of FRF measurements at a time.

Compared to local method, global method yields better estimates of frequency and damping, especially when local modes exist in FRF measurements. In another word, resonance peaks corresponding to local modes wouldn't appear in some FRF plots, as Figure 4-3 shows.
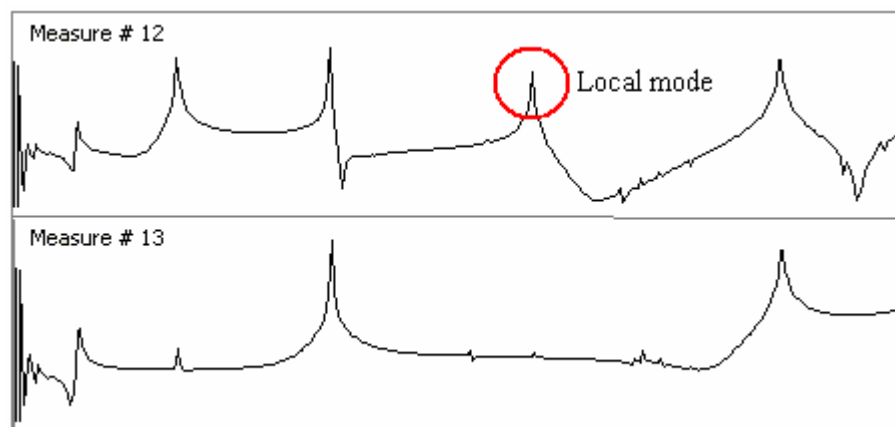
Figure 4-3 Local mode

Single reference vs. poly reference

Single reference method estimates modes from FRF measurements with only one reference DOF, which correspond to one column or row of the FRF matrix. Poly reference method estimates modes from FRF data with more than one reference DOF, which correspond to more columns or rows of the FRF matrix.
Poly reference method is used in cases where a structure has closely coupled modes. This is often referred as repeated roots, from which modal parameters of frequency and damping are extracted.

Out-of-band modes

In practice, modal parameters are estimated in an interested frequency band. The FRF data in the selected frequency band contains effects of modes outside of the band as Figure 4-4 shows.
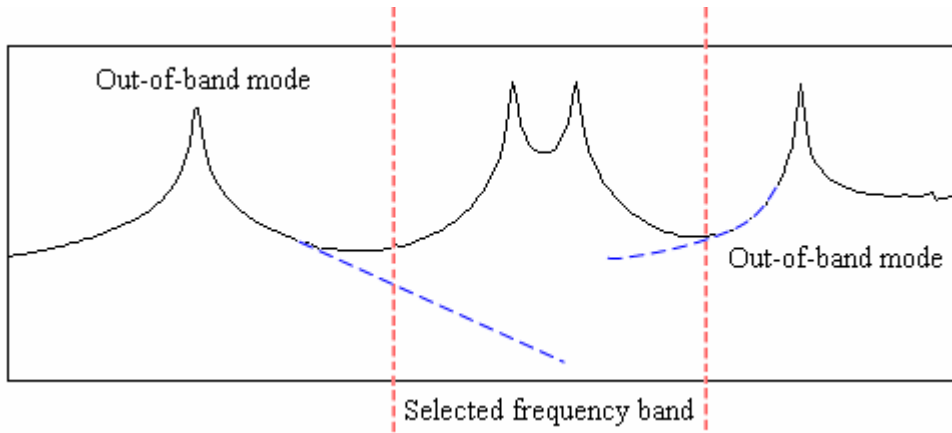
Figure 4-4 Out-of-band modes

To obtain accurate modal parameters, the effects from out-of-band modes must be compensated in curve fitting method. For the MDOF method of polynomial curving fitting, the effects can be compensated by specifying extra terms for numerator polynomial in curve fitting model. The curving fitting model for FRF measurement is defined as following:

$$H(\omega) = \frac{N(\omega)}{D(\omega)} + E(\omega) = \frac{\prod\limits_{i=1}^{m}(j\omega + b_i)}{\prod\limits_{i=1}^{n}(j\omega + a_i)}$$

The polynomial order m for numerator is greater than the order n for denominator. The order for extra terms is defined as $m - n$.

Stabilization chart

Stabilization chart is used to find physically meaningful modes from a set of computed modes by curve fitting. The stabilization chart is created with horizontal axis being frequency and vertical axis being model order. The computed modes for a range of model orders (number of modes to estimate) are displayed on the chart. Theoretically, the physical modes of the structure always appear at a nearly identical frequency, this is an indication that physical modes are stable, while mathematical or fake modes scatter at different frequency, as Figure 4-5 shows.

139
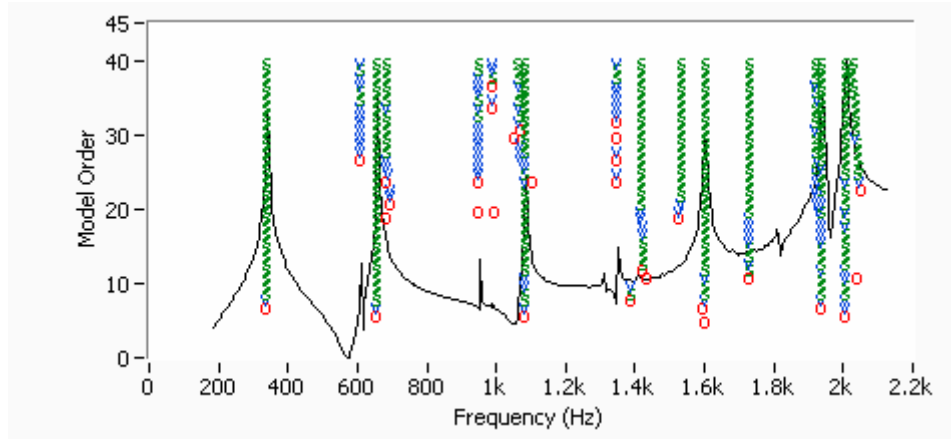
Figure 4-5 Mode stabilization chart

The mode stability is decided by the stability of its three modal parameters, frequency, damping and modal shape. The stability criteria are defined as the changing ratio of modal parameters from a lower model order to a higher model order as following:

$$\frac{\left|p_n - p_{n-1}\right|}{p_{n-1}} \times 100\%$$

The typical values for stability criterion:

- Frequency stability (1%)
- Damping ratio stability (5%)
- Modal shape stability (2%).

The computed modes on stabilization chart are represented using a group of symbols, s, v, d, f and o. Each symbol represents an estimated mode. The symbols indicate the estimated mode is stable or not.

Symbol	Frequency	Damping ratio	Mode shape
s	yes	yes	yes
v	yes	no	yes
d	yes	yes	no
f	yes	no	no
o	no	no	no

Select stable modes

ModalVIEW provides two methods, auto select and manual select, to select stable modes from stabilization chart.

For auto selection, frequency width and stable number are specified. ModalVIEW picks stable modes from stabilization chart by judging the number of stable modes in given frequency width. If the number of stable modes is greater than specified stable number, the mode in corresponding frequency range is regarded as stable mode.

For manual selection, you can manually pick stable modes from stabilization chart using mouse cursor according to the indication from stability symbols.

> **NOTE: The stability chart is used for obtaining frequency & damping estimates in the advanced fit method.**

4.2 Quick fit

Quick fit in ModalVIEW is a rational polynomial fit (RPF) method to extract modal parameters from a set of FRF measurements. It is a global fitting, MDOF methods. It also supports FRF measurements from multiple-reference testing, which are used to estimate closely coupled modes or repeated modes of a structure.

> **NOTE: Quick fit method works best to extract sparsely-spaced, light-damped modes in a narrow frequency band.**

Quick fit involves several steps：

1. Select frequency band for mode estimation.
2. Specify the number of modes to extract in the frequency band.
3. Specify polynomial extra term to compensate effects from out-of-band modes.
4. Select primary reference DOF if multiple-reference FRF measurements are used.
5. Perform modes estimation for selected frequency band.
6. Overlap fitted FRF plot with FRF measurements to verify fit quality.
7. Save the estimated modes into a mode file.

A quick fit example

- Select "plate" demo project in example finder.

The structure file "plate" and data file "freq response" are contained in the project.

- Double click data file "freq response" in project tree.

The data file will be loaded and displayed in data view window, as Figure 4-6 shows.
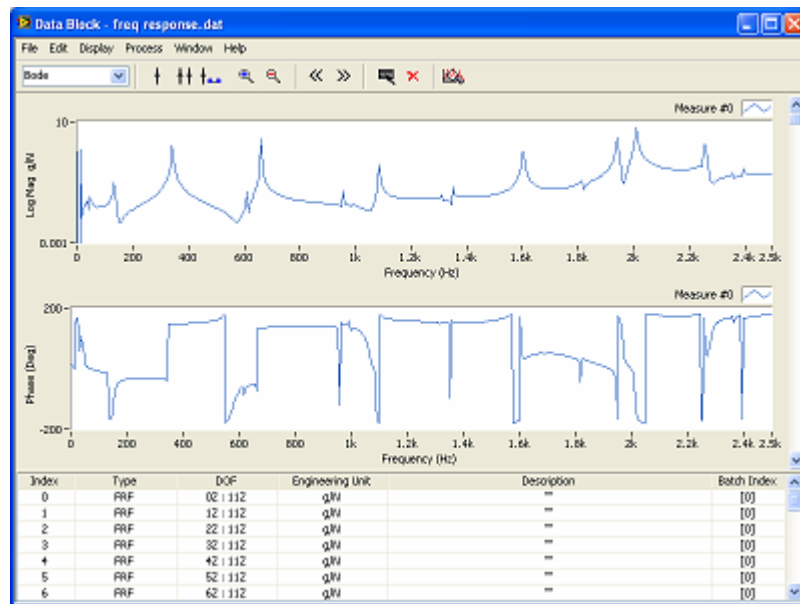
Figure 4-6 Measured FRF data view

In data view window

- Press ![band cursor button] and ![cursor values button] buttons in the data viewer window to show the band cursor and cursor values in the data graph.
- Use band cursor to specify frequency band you want to estimation modal parameters. In this example, a frequency range from 215Hz to 835Hz is selected, as Figure 4-7 shows.
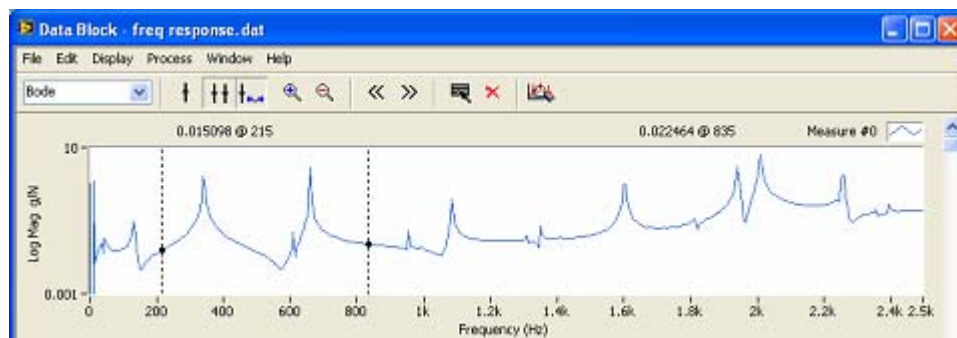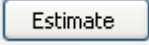
Figure 4-7 Frequency band selection

- Press button ![mode estimation button] in the data viewer window to open mode estimation window.

In mode estimation window

- On the **Quick Fit** tab, set the number of **Modes** to fit in the specified frequency band. Each peak in FRF measurements indicates at least one mode. There are two peaks in the selected frequency range. The number of **Modes** is set two.
- Keep the default value 4 of **Extra Term** to compensate out of band modes.
- The used FRF data has only one reference DOF. There is one selection for **Primary DOF**: 11Z.

- Click ![Estimate] button, the estimated modes will be listed in the modal parameter spreadsheet.
- Check **Show FRF FIT** box, the red synthesized FRFs from estimated modes are computed and overlaid with original FRFs. The synthesized FRFs should match original FRFs well if estimated modes are accurate enough, as Figure 4-8 shows.
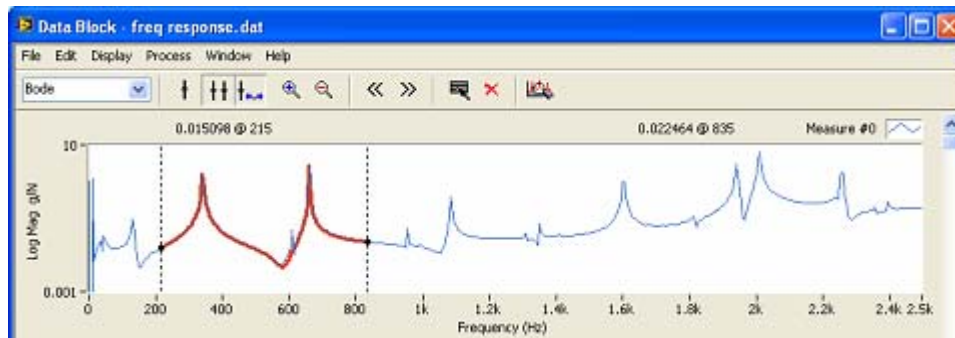
Figure 4-8 Comparison of synthesized FRF and original FRF

- Click ![button] button, to keep the estimated modes into modes list.

You can repeat above steps and continue to fit modes of interest in other frequency bands. After finishing estimation for all modes of interest, execute menu command File | Save Modes… in the mode estimation window, a dialog prompt file name to save. Input file name "modes" and click OK. The mode table file "modes" will be saved in estimated modes and displayed in ModalVIEW project tree.

4.3   Advanced fit

Advanced fit in ModalVIEW is a least-square complex frequency (LSCF) method to extract modal parameters from poly-reference FRF measurements. It is a global fitting, MDOF method.

> **NOTE: Advanced fit method works best to extract closely-spaced modes in a wide frequency band.**

Advanced fit involves several steps：

- Select frequency band for mode estimation.
- Generate mode stabilization plot.
- Select physical modes from stabilization plot.
- Select primary reference DOF if multiple-reference FRF measurements are used.
- Estimate mode shape for selected modes.
- Overlap fitted FRF plot with FRF measurements to verify fit quality.
- Save the estimated modes into a mode file.

An advanced fit example

- Select "plate" demo project in example finder.
- Double click data file "freq response" in project tree.
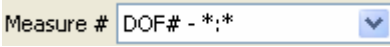
In data view window

- Press and buttons in the data viewer window to show the band cursor and cursor values in the data graph.
- Use band cursor to specify frequency band you want to estimation modal parameters. In this example, a frequency range from 185Hz to 2185Hz is selected.
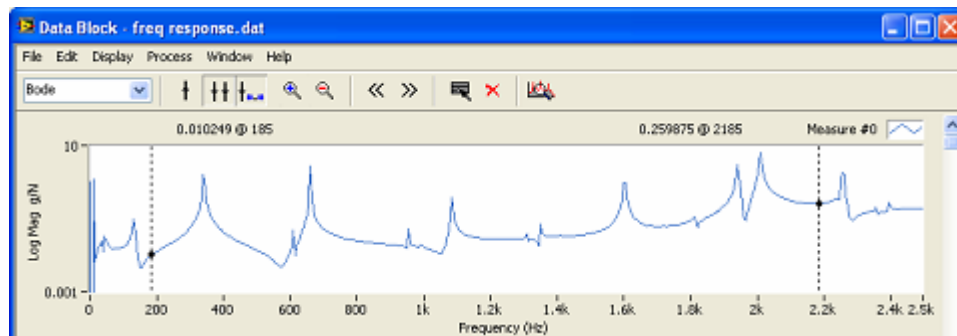
- Press button in the data viewer window to open mode estimation window.
- Select the **Advanced Fit** tab in mode estimation window, the stabilization chart window is opened.

In stabilization chart window

- Specify maximum model order to compute modes in control box **Model order**. In this example, default value 40 is used.
- Specify stabilization criteria in **Frequency (%)**, **Damping (%)**, and **Shape (%)**. The default value is used as 1%, 5% and 2% respectively.
- Press button Generate to generate stabilization chart.
- Select Measure # DOF# - *:* to overlap FRF measurements with stabilization chart. Item "DOF# - 0Z:11Z" is selected to display the FRF measurement of 0Z:11Z on stabilization chart.
- Select physical (stable) modes from stabilization chart.

To automatically select stable modes:

In mode estimation window

- Specify **Frequency Width** and **Stable Number** being 1(Hz) and 5 respectively.
- Click button Auto Select .

The selected stable modes will be added into mode spreadsheet, as Figure 4-9 shows.

Index	Frequency (Hz)	Damping (%)
0	341.408	0.054
1	660.465	0.037
2	954.676	0.035
3	1084.302	0.133
4	1421.760	2.735
5	1602.482	0.076
6	1732.261	0.053
7	1941.651	0.065
8	2008.446	0.093

Figure 4-9 Auto selected modes

To manually select stable modes:

- Click mode symbol with mouse cursor in stabilization chart, the selected symbol will be circled with a red ring. The frequency and damping values of selected mode will also be displayed in control **Frequency (Hz)** and **Damping (%)** in mode estimation window, as Figure 4-10 shows.
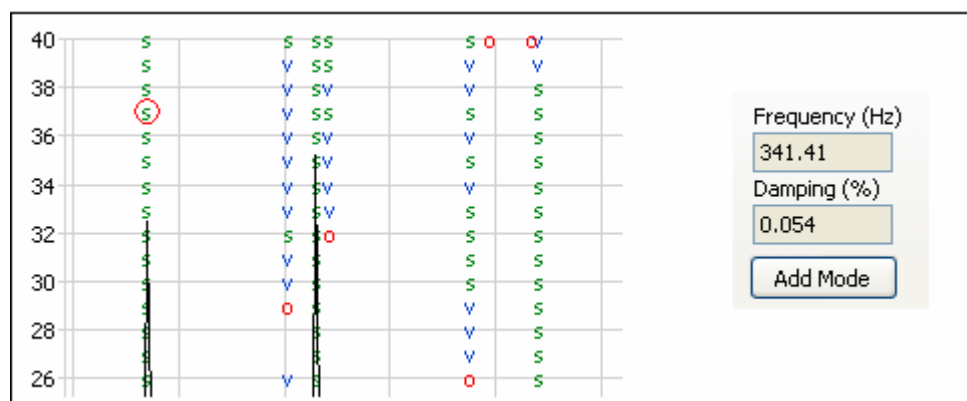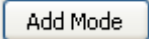
Figure 4-10 Manually selected stable modes

- Click button [Add Mode] to add selected mode into mode spreadsheet.
- Repeat above steps until all the stable modes to be selected are added into mode spreadsheet.

To delete selected stable modes from mode spreadsheet:

- Select the mode in the spreadsheet.

- Click button [×]

After all the physical modes are selected, the mode shapes for selected physical modes can be estimated.

- The used FRF data has only one reference DOF. There is one selection for **Primary DOF**: 11Z.

- Click [Estimate] button, the mode shapes are estimated.

- Check **Show FRF FIT** box, the red synthesized FRFs from estimated modes are computed and overlaid with original FRFs. The synthesized FRFs should match original FRFs well if estimated modes are accurate enough, as Figure 4-11 shows.

Figure 4-11 Comparison of synthesized FRF and original FRF

After finishing estimation for all modes of interest, execute menu command **File | Save Modes…** in the mode estimation window, a dialog prompt file name to save. Input file name "modes" and click **OK**. The mode table file "modes" will be saved in estimated modes and displayed in ModalVIEW project tree.

4.4   Operational Modal Analysis

ModalVIEW also provides stochastic subspace identification (SSI) method to extract modal parameters from a group of time records. It is a global fitting, MDOF method for operational modal analysis (OMA).

SSI method involves several steps：

- Preprocess time records using filtering and down sampling.
- Generate mode stabilization plot.
- Select physical modes from stabilization plot.
- Estimate mode shape for selected modes.
- Save the estimated modes into a mode file.

An OMA example

- Select "bridge" demo project in example finder.
- Double click data file "bridge - 14" in project tree.

The data file will be loaded and displayed in data view window.
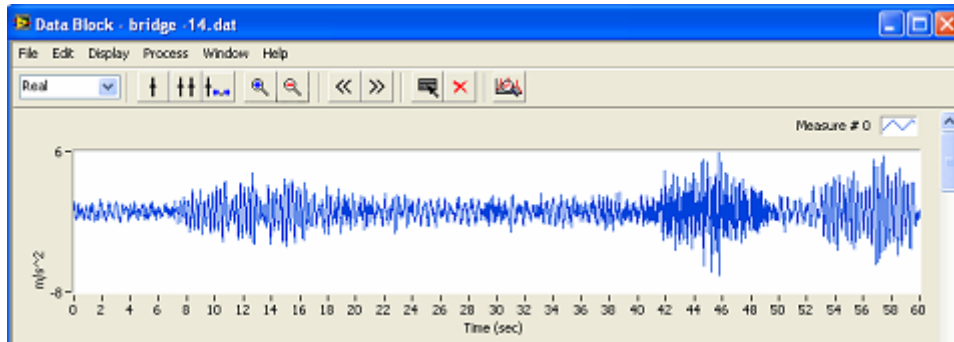
- Execute menu item **Process | Preprocess…**, the data preprocess window is opened.

In data preprocess window
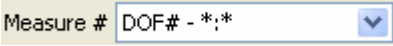
- Specify the Decimation Factor being 10. The sampling rate of time records is changed from 50 Hz to 5 Hz.

The time records after down sampling are shown as following figure.
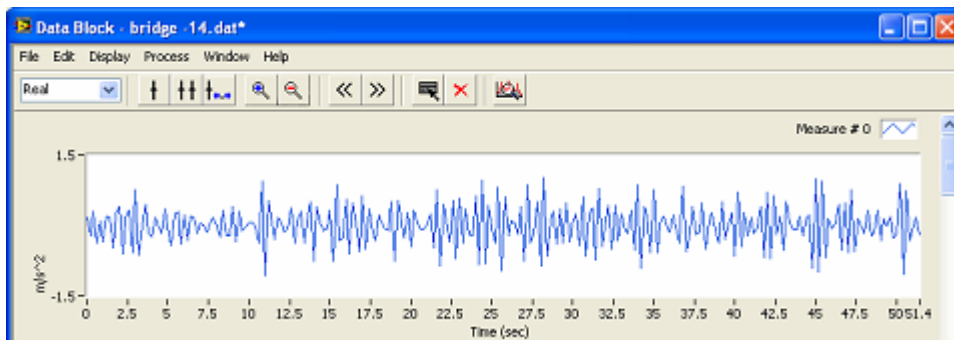
- Specify the Decimation Factor being 10. The sampling rate of time records is changed from 50 Hz to 5 Hz.
- Click button on the toolbar, the mode estimation window and stabilization window are opened.

In stabilization window

- Specify maximum model order to compute modes in control box **Model order**. In this example, the value 80 is used.
- Specify stabilization criteria in **Frequency (%)**, **Damping (%)**, and **Shape (%)**. The default value is used as 1%, 5% and 2% respectively.
- Press button to generate stabilization chart.
- Select to overlap FRF measurements with stabilization chart. Item "DOF# - 5Z" is selected to display the FRF measurement of 5Z on stabilization chart.
- Select physical (stable) modes from stabilization chart.

To automatically select stable modes:

In mode estimation window
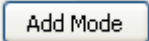
- Specify **Frequency Width** and **Stable Number** being 0.1(Hz) and 5 respectively.
- Click button [Auto Select].

The selected stable modes will be added into mode spreadsheet, as Figure 4-12 shows.

Index	Frequency (Hz)	Damping (%)
0	0.367	0.781
1	0.503	0.747
2	0.634	0.249
3	0.780	0.630
4	0.999	0.669
5	1.036	0.345
6	1.265	0.615
7	1.388	0.386
8	1.682	3.325
9	1.756	1.160
10	1.844	0.565
11	1.955	0.867
12	2.081	1.560
13	2.130	0.490
14	2.280	0.573
15	2.413	1.033

Figure 4-12 Auto selected modes
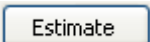
To manually select stable modes:

- Click mode symbol with mouse cursor in stabilization chart, the selected symbol will be circled with a red ring. The frequency and damping values of selected mode will also be displayed in control **Frequency (Hz)** and **Damping (%)** in mode estimation window.
- Click button [Add Mode] to add selected mode into mode spreadsheet.
- Repeat above steps until all the stable modes to be selected are added into mode spreadsheet.

To delete selected stable modes from mode spreadsheet:

- Select the mode in the spreadsheet.
- Click button [×]

After all the physical modes are selected, the mode shapes for selected physical modes can be estimated.

- Click [Estimate] button, the mode shapes are estimated.

After finishing estimation for all modes of interest, execute menu command **File | Save Modes…** in the mode estimation window, a dialog prompt file name to save.

Input file name "modes" and click **OK**. The mode table file "modes" will be saved in estimated modes and displayed in ModalVIEW project tree.

4.5 Mode Validation

Mode validation is to evaluate the validity of estimated modes. A few of mode properties can be used for mode validation.

Linearly independent

Theoretically, each row or column vector of modal shape matrix should be linear independent or orthogonal to all other vectors. Modal shape matrix combined from estimated modes is defined as following:

$$
\text{DOF number} \downarrow \quad \overset{\text{Mode shape number} \longrightarrow}{\begin{bmatrix} \phi_{11} & \phi_{12} & \cdots & \phi_{1n} \\ \phi_{21} & \phi_{22} & \cdots & \phi_{2n} \\ \vdots & \vdots & \ddots & \vdots \\ \phi_{m1} & \phi_{m2} & \cdots & \phi_{mn} \end{bmatrix}}
$$

Row index m is the DOF number. Column index n is the modal shape number.

Normal and Complex

Modes can be divided into two types, normal modes and complex modes. The mode shape of normal modes can be described as a sign valued real number. The mode shape of complex modes is complex valued and cannot be described as a real number. In other words, with normal mode, all DOF points of the structure are moving in phase or 180 degree out of phase with each other. With complex mode, the different DOF points of the structure are moving in some general phase relationship with each other, as Figure 4-13 shows.

149
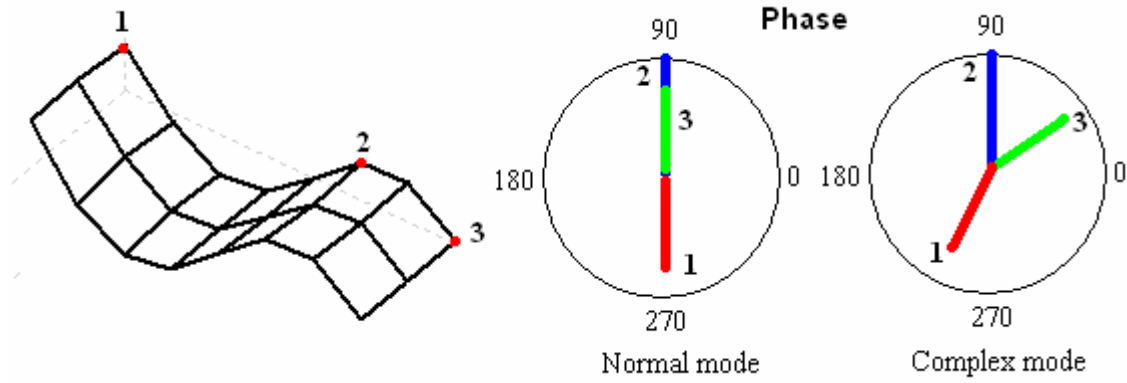
Figure 4-13 Normal mode and complex mode

The damping property of a structure decides whether the modes will be normal or complex. For lightly or proportionally damped structures, the estimated modes should approximate normal. In practice, the normal modes on the structure could be estimated as complex modes due to the distorted FRF measurements, curve fitting error or some other inaccuracies.

Modal assurance criteria (MAC)

MAC and CoMAC quantitatively compare column and row vector pairs of modal shape matrix respectively, and check their orthogonality.

MAC is defined as following:

$$MAC = \frac{\left|X_i^* X_j\right|}{\left|X_i\right|\left|X_j\right|} \qquad i, j = 0,...,n$$

X_i is the column vector of modal shape matrix $\begin{bmatrix} \phi_{1i} & \phi_{2i} & \cdots & \phi_{mi} \end{bmatrix}$

CoMAC is defined as following:

$$CoMAC = \frac{\left|X_i^* X_j\right|}{\left|X_i\right|\left|X_j\right|} \qquad i, j = 0,...,m$$

X_i is the row vector of modal shape matrix $\begin{bmatrix} \phi_{i1} & \phi_{i2} & \cdots & \phi_{in} \end{bmatrix}$

The value range of MAC and CoMAC is from 0, representing linear independent, to 1, representing linear dependent.

ModalVIEW provide MAC and CoMAC plot to show the value between any two of shape vectors.

For example, to compute MAC and CoMAC

- Select "plate" demo project in example finder.

In mode view window

- Execute menu **File | Load…** to load mode shape file "mode".

- Click button on the toolbar, MAC plot will be displayed.
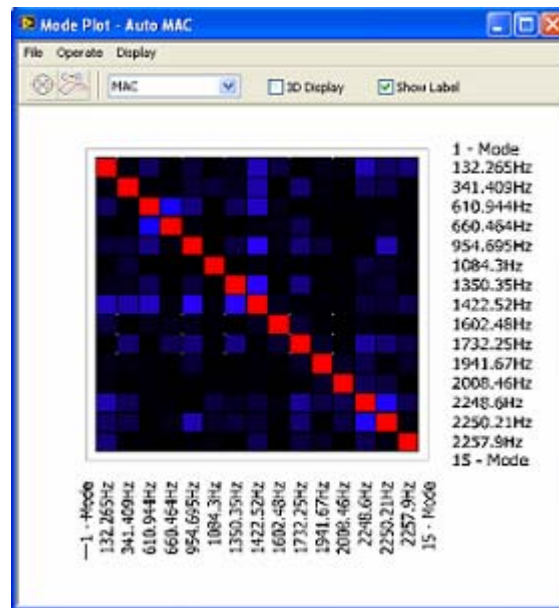
Check to change plot view into 3D style. Check to select if the frequency of each mode is displayed on the plot.

- Select CoMAC in control to display CoMAC plot.

MAC plot and values can be exported to windows clipboard and pasted into other software for further process.

- Execute menu item **File | Copy to Clipboard | Bitmap**
- Or execute menu item **File | Copy to Clipboard | Spreadsheet**

In a more general way, MAC can be extended to two arbitrary mode shape vectors. This is useful in comparing the mode shapes from different analysis process, for example, experimental modal analysis (EMA) and finite element analysis (FEA). The finite element model for the structure can be verified according to the similarity of corresponding modal shapes.

ModalVIEW provides Cross MAC plot to show the value between two arbitrary mode shape vectors from different mode shape files.

For example, to compute Cross MAC

- Select "plate" demo project in example finder.

In mode view window

- Execute menu **File | Load…** to load mode shape file "mode" into the left spreadsheet.
- Click button on the toolbar

A dialog box is displayed to prompt you to select another mode table for comparison.
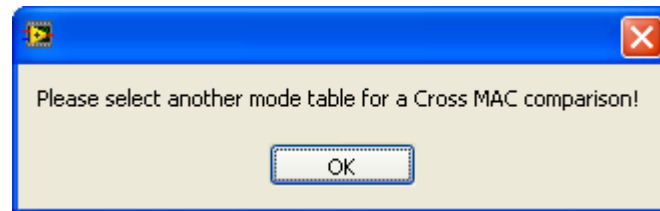
Click **OK** button, a file open dialog is displayed for you to select another mode shape file in the project. Select a file in the list and click **OK** button, the modes will be loaded for comparison. Cross MAC plot will be displayed.

Move cursor on the plot, the value of Cross MAC will be displayed at the bottom of the window.

Mode complexity

ModalVIEW provides mode complexity plot to display the magnitude and phase of estimated mode shapes. According to whether shape vector stand along a straight line on complexity plot or not, normal or complex mode type of the estimated modes can be decided.

For example, to display complexity plot

- Select "plate" demo project in example finder.

In mode view window

- Execute menu **File | Load…** to load mode shape file "mode".

- Click button on the toolbar, complexity plot will be displayed.
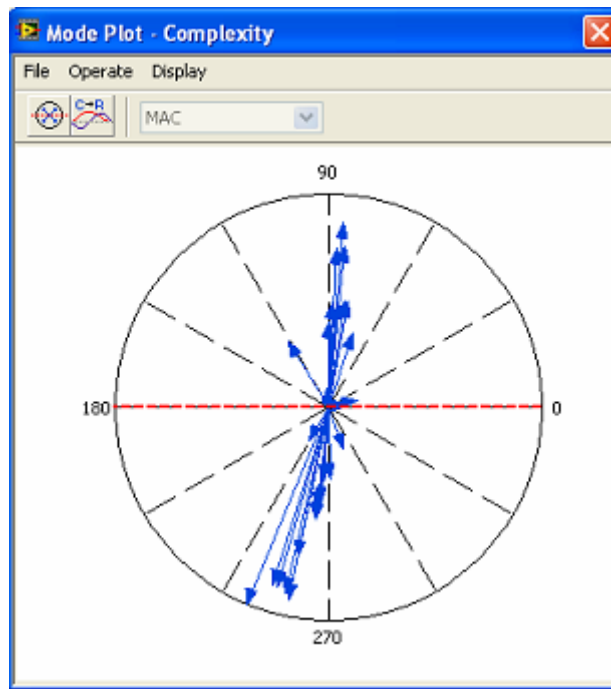
- Click to select different mode in spreadsheet to view its complexity plot.

Mode normalization

The resulting complex modes from mode estimation may be due to errors from the data measurement and mathematical computation, which can be normalized into real modes. Mode normalization can rectify the errors and simplify the mode shape for structure animation.

A red dashed line on complexity plot is used as the phase reference line to convert complex modes into normal ones. A complex mode is normalized with magnitude being reserved and phase being changed to 0 or 180 degree. The phase of mode shape vectors mostly near to one side of the phase reference line are changed into 0 degree and near to the other side into 180 degree.

Mode normalization involves a few steps:

1. Select the mode for normalization in mode table window.
2. Display the complexity plot of selected mode.
3. Specify phase reference line on complexity plot.
4. Convert complex mode into normal mode.

To change the position of phase reference line

In mode view window

- Drag to rotate the reference line to expected position with mouse cursor.

- Or press button on toolbar in mode plot window, the reference line will be automatically rotated to an optimal position.

After the position of phase reference being specified, press button on toolbar to normalize complex mode. The resulting normal modes can be saved into a new mode shape file.

4.6 Mode Conversion

In order to make estimated modal parameters from vibration measurement compatible with structure dynamic model, ModalVIEW provides some operations used for mode parameter modification.

Mode shape unit

Vibration measurement can be made using different sensors to measure acceleration, velocity and displacement. The unit of estimated mode shape depends on the unit of response measurement (acceleration, velocity and displacement). Unit of measurement can be converted using integration of the time waveform. In frequency domain, the equivalent operation can be down according to Laplace transform, as following equation.

$$H_a(s) = H_v(s) \times s = H_d(s) \times s^2$$

$H_a(s)$, $H_v(s)$ and $H_d(s)$ are the transfer functions in acceleration, velocity and displacement unit respectively.

Since mode shape is obtained from residues in partial fraction expansion of a transfer function, the residues have following relationship in different unit.

$$R_a(k) = R_v(k) \times p_k = R_d(k) \times p_k^2, k = 1,...,n$$

$R_a(k)$, $R_v(k)$ and $R_d(k)$ are the residues in acceleration, velocity and displacement unit respectively. p_k is the pole decided by frequency and damping ratio of the k_{th} mode.

Correspondingly, the unit of mode shape can be converted according to above residue relationship. In ModalVIEW, the unit of mode shape is defined in engineering unit string. When the unit is converted among acceleration, velocity and displacement, the engineering unit string is changed correspondingly. ModalVIEW can identify engineering unit strings as following:

Unit	Engineering Unit String
Acceleration	(m/s^2)/N-sec 或 g/N-sec
Velocity	(m/s)/N-sec
Displacement	m/N-sec

Mode shape type

ModalVIEW defines three type of mode shape: **Residue shape**, **Scaled shape** and **ODS shape**. Residue mode shape is obtained by curve fitting measured FRF data. The mode with residue shape can be used for FRF synthesis or forced response simulation. ODS shape is obtained from time record or cross spectrum in operational modal analysis (OMA). The mode with ODS shape can only be used in structure animation and not for FRF synthesis or forced response simulation. Scaled shape is converted from residue shape using the relationship as following equation.

$$[r(k)] = A_k [u_k][u_k]^T, k = 1,...,n$$

$[r(k)]$ is the residue matrix for k_{th} mode. $[u_k]$ is the vector of scaled mode shape. A_k is the scaling factor. In ModalVIEW, A_k is specified as 1. The modes with scaled shape can also be used to generate residue shapes for any reference DOF.

For example, to convert mode shape unit

*   Select "plate" demo project in example finder.

In mode view window

*   Execute menu **File | Load…** to load mode shape file "mode".

*   Click button on the toolbar.
*   Or execute menu item **Operate | Scale Mode Shape**

The mode shape type is converted from residue shape to scaled shape. The DOF form is changed from "Roving DOF : Reference DOF" to "Roving DOF" correspondingly.

4.7   Mode Analysis

The estimated modes in modal analysis can be used for different structure dynamic simulation studies. FRF synthesis, forced response simulation, structure dynamic matrix and sensitivity analysis are provided in ModalVIEW.

FRF synthesis

In FRF synthesis, estimated modes are used to compute FRF at given stimulus and response DOF for comparison with experimentally measured FRFs.

When mode type is residue mode, FRFs can be synthesized only for DOFs of residue modes. The reference DOFs can not be specified. The list box for reference DOFs selection in FRF parameters window is disabled as following shows.
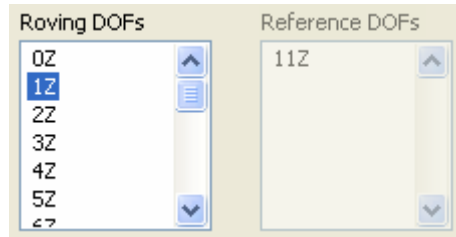
When mode type is scaled mode, FRFs can be synthesized between any two roving DOFs of the modes. The reference DOFs can be specified. The list box for reference DOFs selection in FRF parameters window is enabled for selection as following shows.
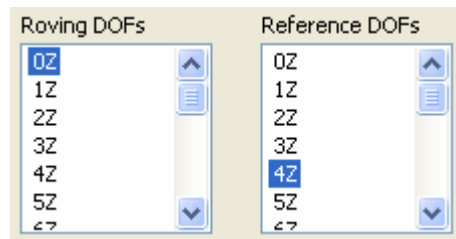
For an example, to synthesize FRF data from estimated modes.

- Select "plate" demo project in example finder.

In mode view window

- Execute menu **File | Load…** to load mode shape file "mode".

- Click button on the toolbar
- Or execute menu item **Operate | FRF Synthesis…**
- Specify FRF parameters as Figure 4-14 shows.
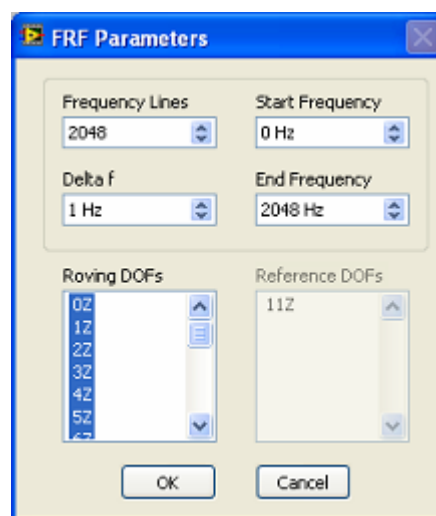
Figure 4-14 FRF parameters window

156

- Press **OK** button. The synthesized FRF data plot will be displayed in opened data view window, as Figure 4-15 shows.

Figure 4-15 Synthesized FRF data

Forced response simulation

In ModalVIEW, forced response simulation is to compute structure's frequency response with sine excitation at given reference DOFs. The sine excitation is defined by its magnitude, phase and applied DOF on structure. The DOF for sine excitation decides where to give sine excitation on structure. The frequency response of structure is represented by forced response shape, a vector of magnitude and phase at each DOF with the same frequency as the sine excitation. The forced response shape can be used for structure animation as a mode shape.

When mode type is residue mode, only the reference DOFs in residue shape can be selected as DOFs of sine excitation. When mode type is scaled mode, sine excitation can be applied on structure at any roving DOF in scaled shape.

For example, compute forced response shape from estimated modes.

- Select "plate" demo project in example finder.

In mode view window

- Execute menu **File | Load…** to load mode shape file "mode".

- Click button ![icon] on the toolbar

- Or execute menu item **Operate | Forced Response…**

- Specify sine excitation parameters as Figure 4-16 shows.
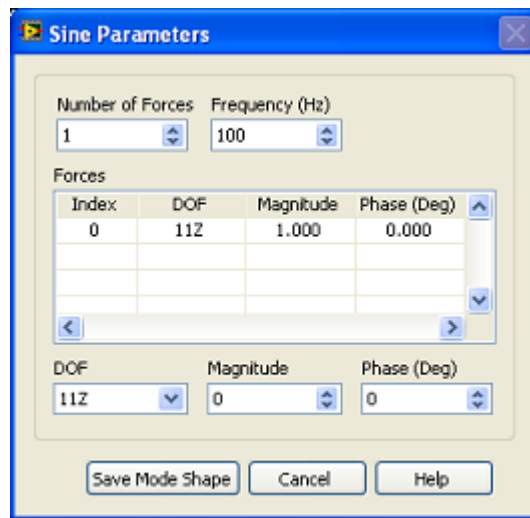
Figure 4-16 Sine parameters window

- Press **Save Mode Shape** button. The mode shape from sine excitation will be saved for structure animation.

Mass, stiffness and damping matrix

The estimated mode parameters, natural frequency, damping ratio and mode shape, from measured vibration data, can be transformed into mass, stiffness and damping matrix. These matrixes define structure dynamic characteristics, which can be further used for finite element model validation and structural dynamic modification.

Taking Laplace transforms of structure dynamic equation yields following equation:

$$\Phi[p^2] + M^{-1}C[p] + M^{-1}K\Phi = 0$$

Φ is mode vector matrix. p are poles, which decide natural frequency and damping. M, K, C are mass, stiffness and damping matrix, respectively.

ModalVIEW compute mass, stiffness and damping matrix according to following assumptions:

- The stiffness matrix is symmetric
- The mass matrix can be approximated by a diagonal matrix
- The total mass of the structure is known and meet following equation:

$$\sum m_i = M$$

m_i is the discrete mass in mass matrix. M is the total mass.

> 📝 **NOTE: When estimating mass, stiffness and damping matrix from mode parameters, the mode shape type must be 'scale mode shape'.**

For an example, compute mass, stiffness and damping matrix from estimated mode parameters.

- Select "plate" demo project in example finder.

In mode view window

- Execute menu **File | Load…** to load mode shape file "mode".

- Click button ![MKC] on the toolbar
- Or execute menu item **Operate | M, K, C Matrix…**

M, K, C matrix window is opened. Specify the total mass of the structure in ![Total Mass 1], the matrix data will be displayed in the list. Change list box ![Mass] to display the different matrix data.

Sensitivity analysis

The estimated mode parameters can be further used to analyze the effect of structure's change of mass, stiffness and damping on the structure dynamics, which can be used as guidance to structural dynamic modification.

The ratio between the change of natural frequency and the changes of structure mass, stiffness and damping is described as following formula:

$$\frac{\partial \omega_i}{\partial M_k} = -\lambda_i^2 \frac{\psi_{ki}^2}{a_i}$$

$$\frac{\partial \omega_i}{\partial C_{kl}} = -\lambda_i \frac{(\psi_{ki} - \psi_{li})^2}{a_i}$$

$$\frac{\partial \omega_i}{\partial K_{kl}} = -\frac{(\psi_{ki} - \psi_{li})^2}{a_i}$$

ω_i is the *ith* natural frequency of the structure. M_k is the local mass of the structure at DOF k. C_{kl} is the damping between DOF k and l. K_{kl} is the stiffness between DOF k and l. a_i is the scaling factor of mode vector matrix.

For an example, perform sensitivity analysis using estimated mode parameters.

- Select "sdm" demo project in example finder.

In mode view window

- Execute menu **File | Load…** to load mode shape file "mode".

- Click button on the toolbar
- Or execute menu item **Operate | Sensitivity…**

Sensitivity matrix plot window is opened. Select different mode in mode table, the plot is updated accordingly, which indicates how structure modification influences the natural frequency of selected mode.

- Select items in Modification Type Mass to specify the type of structure modification: Mass, Stiffness or Damping.
- Check ☐ 3D Display, the sensitivity matrix plot is displayed in 3D bar plot.
- Check ☐ Show Label, the plot label of mode DOF is displayed.

Modification Prediction

The estimated mode parameters can be further converted into the modal model, which is defined by mass, stiffness and damping matrix. The physical modification of structure can be represented by changes in mass, stiffness and damping matrix. ModalVIEW allows you to evaluate the effect of applied point mass, spring-damper and tuned absorber on the structure's dynamic characteristics.

Point mass modificaiton is defined by the amount of mass and node number to add on the structure.
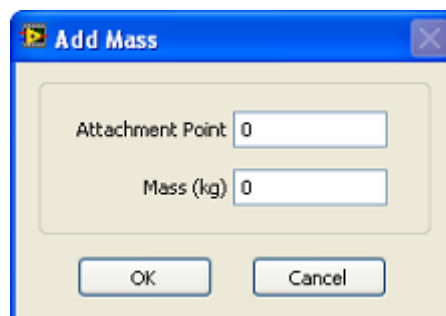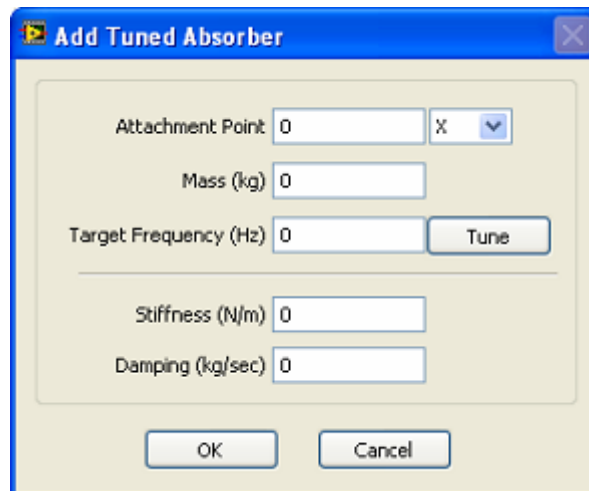
Spring-damper modificaiton is defined by the values of stiffness and damping, the node number between which the spring-damper is to be applied on the structure.

Tuned absorber modification is a single degree of freedom system consisting of a rigid mass connected by a spring and a damper to the structure. It is defined by the amount of mass, the value of stiffness and damping, and the node number and direction to which the tuned absorber is coupled. The value of stiffness and damping can be automatically computed by specifing target frequency to be tuned.

ModalVIEW converts modificatoin elements into changes in mass, stiffness and damping matrix of the modal model, and compute structure dynamics according to the following equation.

$$(p(\begin{bmatrix} 0 & M \\ M & C \end{bmatrix} + \begin{bmatrix} 0 & \Delta M \\ \Delta M & \Delta C \end{bmatrix}) + (\begin{bmatrix} -M & 0 \\ 0 & K \end{bmatrix} + \begin{bmatrix} -\Delta M & 0 \\ 0 & \Delta K \end{bmatrix}))|Y| = |F|$$

M, K, C are the original mass, stiffness and damping matrix. $\Delta M, \Delta K, \Delta C$ are the matrix changes due to structure modification. p is the system eigenvalue. $|Y|, |F|$ are displacement and force vector, respectively.

NOTE: When performing modification prediction, the mode shape type must be 'scaled mode shape'.

161

For an example, to perform structure modification

- Select "sdm" demo project in example finder.

In structure modification window

- Execute menu item **File | Load…**, open modification file 'point mass'
- Select 'truss' structure file in the Structure Not Selected listbox on the toolbar.
- Select 'mode' mode file in the Orignal Modes Not Selected listbox.
- Click Calculate button, the original FRF plot and newly synthesized FRF plot for the modified structure are displayed and overlapped together, as Figure 4-17 shows.
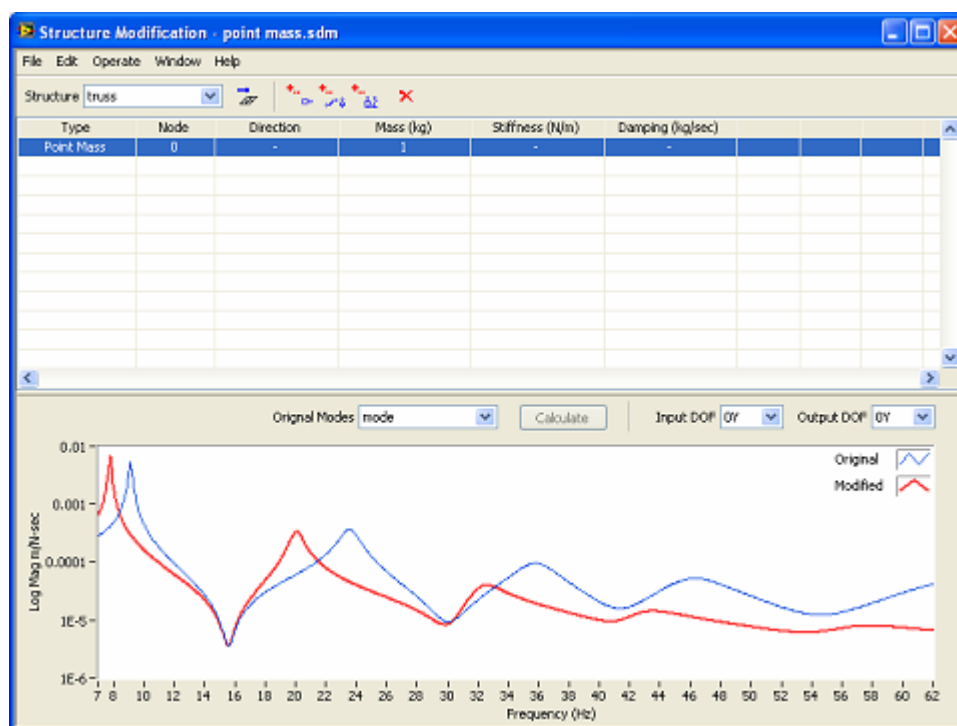
Figure 4-17 Modification Prediction

- Select different input DOF and output DOF from the Input DOF OY listbox and the Output DOF OY listbox to watch FRF plot of the corresponding input and output DOFs.

From the newly synthesized FRF plot, the structure's vibration frequency of the first mode decreases after point mass being added on the structure.

- Execute menu item **File | Save Modifed Modes…**.

A file open dialog is displayed to prompt you to input desired new file name. Type the desired file name into the **File Name to Save** box and click **Save** button. The mode parameters of modified structure will be created for the further analysis.

5. Model Animation

By mapping measured time- or frequency- domain data or estimated mode shape on the node motion of created 3D model of a structure, vibration behavior and dynamic properties of the structure under test can be displayed and investigated.

Generally, there are much more nodes used to create 3D model structure than the number of actually measured points on physical object, where the sensors are fixed on. During structure animation, the motion of measured nodes is determined by animation source (time data, FRF data, or mode shape). The motion of unmeasured nodes is interpolated according to the motion of neighboring measured or fixed nodes. Refer to section **Measurement properties** for details.

5.1 Animation source

ModalVIEW supports three types of animation display: time-based ODS, frequency-based ODS and mode shape, which requires different animation source.

Time record

Time record is the data source used in time-based ODS animation. ModalVIEW supports three animation modes for time-based ODS: sweep, sine dwell and stationary dwell. The relationship between motion of each node on 3D model and time record is defined as following:

Sweep animation	$d = a \times s(t),\ \ t = 0,...,n$
Sine dwell animation	$d = a \times s_i \times \sin(\theta),\ \ \ \theta = 0,...,2\pi$
Stationary dwell animation	$d = a \times s_i$

d is the change of node position along one measurement direction. a is amplitude factor. $s(t)$ is the time record corresponding to the node DOF. s_i is the value of time record at a time.

For sweep animation, the value of time record is continuously changed with time from beginning to the end in a loop.

FRF measurement

FRF measurement is the data source used in frequency-based ODS animation. Frequency-based ODS also have three animation modes: sweep, sine dwell and stationary dwell. The relationship between motion of each node on 3D model and FRF measurement is defined as following:

Sweep animation	$d = a \times r(\omega) \times \sin(\theta(\omega)),\ \ \ \omega = \omega_0,...,\omega_n$
Sine dwell animation	$d = a \times r(\omega_i) \times \sin(\theta + \theta(\omega_i)),\ \ \ \theta = 0,...,2\pi$
Stationary dwell animation	$d = a \times r(\omega_i) \times \sin(\theta(\omega_i))$

d is the change of node position along one measurement direction. *a* is amplitude factor. $r(\omega)$ and $\theta(\omega)$ are the magnitude and phase of FRF measurement corresponding to the node DOF. $r(\omega_i)$ and $\theta(\omega_i)$ are the magnitude and phase of FRF at frequency ω_i.

For sweep animation, the frequency value of FRF is continuously changed from start frequency to the end frequency in a loop.

Mode shape

Mode shape is the data source used in mode shape animation. Mode shape animation supports three animation modes: sweep, sine dwell and stationary dwell. The relationship between motion of each node on 3D model and mode shape is defined as following:

Sweep animation	$d = a \times r(v) \times \sin(\theta(v)),\quad v = v_0,...,v_n$
Sine dwell animation	$d = a \times r(v_i) \times \sin(\theta + \theta(v_i)),\quad \theta = 0,...,2\pi$
Stationary dwell animation	$d = a \times r(v_i) \times \sin(\theta(v_i))$

d is the change of node position along one measurement direction. *a* is amplitude factor. $r(v)$ and $\theta(v)$ are the magnitude and phase of complex element in mode shape v, which is corresponding to node DOF. $r(v_i)$ and $\theta(v_i)$ are the magnitude and phase of mode shape v_i.

For sweep animation, the mode shape in the list is continuously changed one by one in a loop.

5.2 Data assignment

ModalVIEW animates each node on 3D model of the structure according to its measurement properties and assigned animation source, which defines the relationship between node motion and animation data during structure animation.

The animation data is assigned by specifying its index in spreadsheet into measurement properties of the corresponding node. The index for each data is displayed in the first column of data spreadsheet, as Figure 5-1 shows.

time record

Index	Type	DOF	Engineering Unit
0	Time Record	0Z	m/s^2
1	Time Record	1Z	m/s^2
2	Time Record	2Z	m/s^2
3	Time Record	3Z	m/s^2
4	Time Record	4Z	m/s^2

FRF measurement

Index	Type	DOF	Engineering Unit
0	FRF	0Z : 11Z	g/N
1	FRF	1Z : 11Z	g/N
2	FRF	2Z : 11Z	g/N
3	FRF	3Z : 11Z	g/N
4	FRF	4Z : 11Z	g/N
5	FRF	5Z : 11Z	g/N

mode shape

Index	Frequency (Hz)	Damping (%)	Shape Type
0	132.265	0.532	Residue Shape
1	341.409	0.054	Residue Shape
2	610.944	0.019	Residue Shape
3	660.464	0.037	Residue Shape

Figure 5-1 Data index

In measurement properties, three measure types, measured, interpolated and fixed, are defined for the X, Y, Z measurement directions of each node on 3D model. Only measured nodes is to be assigned an index of measurement data or mode shape. During structure animation, the motion of measured nodes is determined by assigned measurement data (time data, FRF data, or mode shape).

In ModalVIEW, data assignment for model animation can be performed manually or automatically.

Manual assignment

For example, to manually assign animation data

- Select "plate" demo project in example finder.

In structure view window

- Execute menu **File | Load…** to load structure file "plate".
- Press button ![icon] on toolbar to open measure properties window.

In measure properties window

- Select one node in spreadsheet.

The measurement axial of selected node will be displayed on 3D model structure, as Figure 5-2 shows.
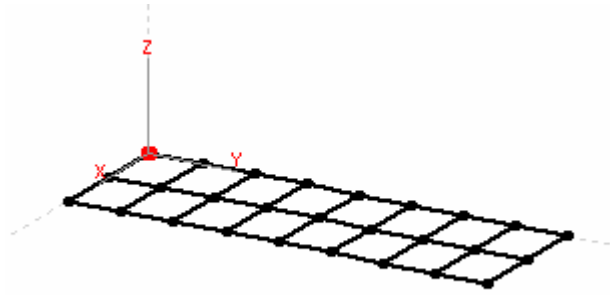
Figure 5-2 Measurement axial

On tab page **M# Assignment**, select corresponding measure type for each measurement axial. When 'Measured' measure type is selected for an axial, the measurement index for corresponding axial is enabled for input as Figure 5-3 shows.

Figure 5-3 M# assignment tab page

According to the DOF information of selected node, you can input the measurement index of measurement data "freq response" with the same DOF information.

- Repeat above steps until all nodes with 'Measured' measure type are assigned measurement index.

Automatic assignment

For example, to automatically assign measurements

- Select "plate" demo project in example finder.
- Double click "plate" in project tree to open it in structure view window.

According to animation source selection, measurement data or mode shape.

- Open data view window and load data block file "freq response".
- Or open mode view window and load mode shape file "mode".

In structure view window

- Press button on toolbar.

ModalVIEW will automatically assign opened animation source, measurement data or mode shape, to node on 3D model according to matched DOF information. After measurements being assigned, a dialog box will open to report the number of assigned measurements.
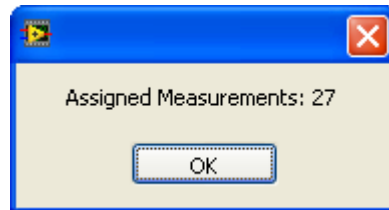
📝 **NOTE: In order to use automatic assignment, the animation data, measurement data or mode shape, and nodes on 3D model should contain related DOF information.**

DOF forms for different animation source are listed as following. Roving DOF is used to match node DOF for automatic assignment of measurement index.

Animation Source	DOF Form
Time record	Roving DOF (1X)
FRF	Roving DOF : Reference DOF (2X:0X)
Mode shape	Roving DOF : Reference DOF (2X:0X)

5.3   Animation operation

After 3D model being assigned animation source, the model is ready to display animation shape.

To start model animation

- Make sure the structure view, data view or mode shape window are opened. The corresponding files are loaded.

In structure view window

- Select proper animation type.
- Click button on toolbar.

Deformation Scaling

During structure animation, ModalVIEW provides three methods to scale structure deformation: auto, relative and fixed.

For auto scale, deformation value for each DOF of the structure is scaled relative to maximum magnitude of each shape.

For relative scale, deformation value for each DOF of the structure is scaled relative to maximum magnitude of all shapes in connected data source.

For fixed scale, deformation value for each DOF of the structure is scaled with a fixed scale factor. The fixed scale factor is used to scale all shapes during animation in following equation.

$$DOF\ deformation = Fixed\ scale\ factor * Shape\ value$$
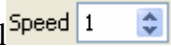
The fixed scale factor can be decided to a suitable one according to the largest structure coordinate.

Speed & amplitude

During structure animation, animation speed can be changed by adjusting speed factor. The value range of speed factor is from 1 to 20, which is corresponding to slowest speed and fastest speed, respectively.

- The speed factor is specified in control Amplitude 1 .

The deformation of model animation is also modified by amplitude factor. The value range of amplitude factor is from 1 to 20, which is corresponding to smallest deformation and largest deformation, respectively.

- The amplitude factor is specified in control Speed 1 .

Interpolation

During structure animation, the motion of measured nodes is determined by assigned measurement data (time data, FRF data, or mode shape). The motion of interpolated nodes is computed according to the motion of neighboring measured or fixed nodes.

After measurement properties for each node on 3D model is defined and animation source is assigned, the motion of interpolated nodes can be computed. To generate interpolated animation for unmeasured nodes.

- Click button ⬚ on the toolbar

A dialog box will show to prompt you to specify interpolation range. That is how many measured or fixed nodes nearest in geometry are used to calculate interpolated animation.

In Component? ☐ decides if interpolated motion of unmeasured nodes are computed from the measured nodes belonging to the same component.

Partially Interpolation? ☐ decides if interpolated motion of unmeasured nodes are partially computed only for specified DOFs.
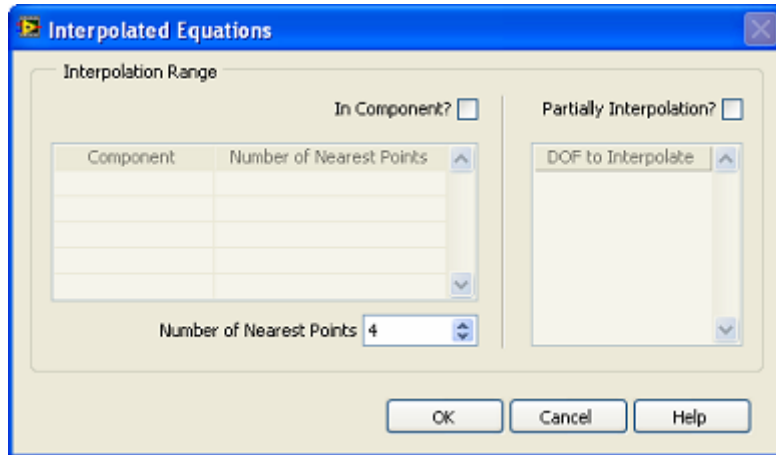
Input number of nearest points in the dialog box and click **OK** button. The interpolated animation is generated.

During 3D model animation, to enable interpolated motion of unmeasured nodes

- Click button on the toolbar

Time-based ODS

For an example, to perform structure animation of time-based ODS

- Select "beam" demo project in example finder.
- Click quick start "ODS Animation"

The structure view and data view window are opened and tiled in left and right, as Figure 5-4 shows.

- Load structure file "beam" into structure view window.
- Load data block file "time record" into data view window.
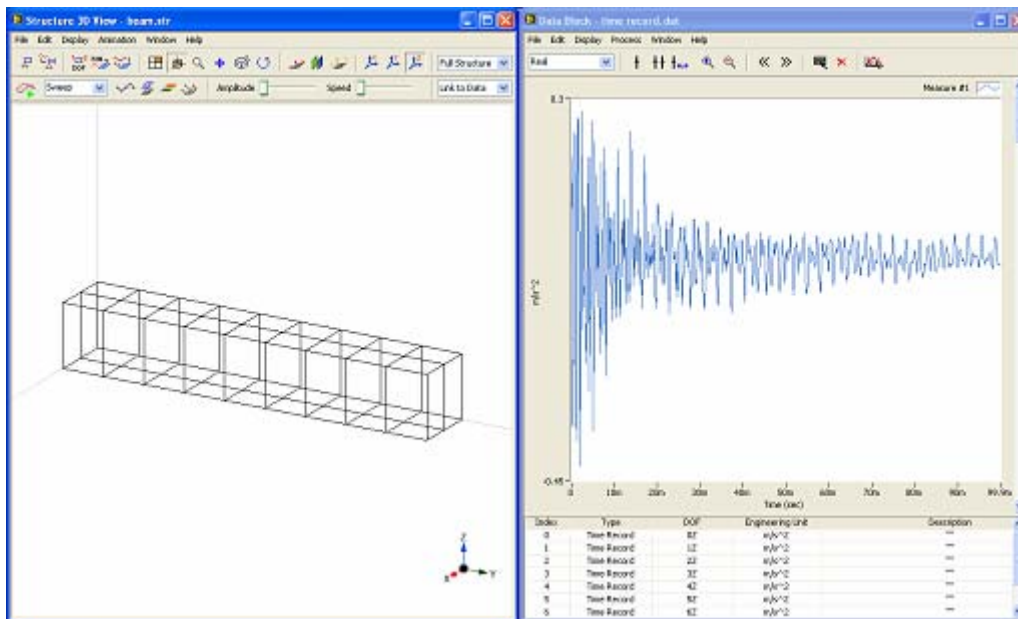
Figure 5-4 Time-based ODS

In structure view window

- Click button on toolbar.

Eight time records in data view window will be automatically assigned to eight measured nodes on beam structure according to matched DOF information. A dialog box will open to report the number of assigned measurements.
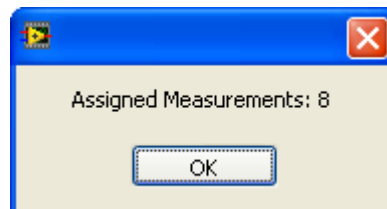
- Click button on the toolbar

A dialog box will show to prompt you to specify interpolation range.
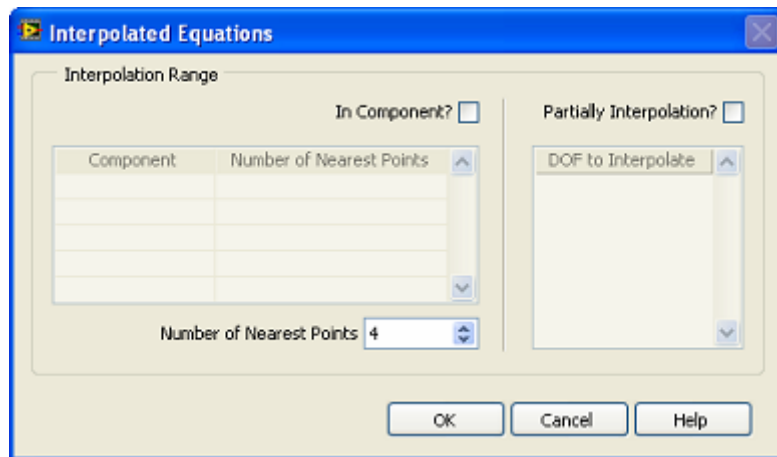
The number of nearest points is specified as 4 in the dialog box and click **OK**. The interpolated animation is generated.

It is ready to start structure animation of time-based ODS now. The default animation type is sweep. You can change the animation type before or during animation.

- Click button on toolbar to start animation.
- Adjust and to proper values. In this example, amplitude and speed are specified as 10 and 5, respectively.

During animation, you can view different animation displays for a good understanding of structure's time-based ODS.

To enable / disable animation of interpolated nodes

- Click button on the toolbar

To overlap un-deformed structure with animation

- Click button on the toolbar

To change animation type

- Select corresponding item in control on the toolbar

During sweep animation, the line cursor on data graph in data view window moves from left to right to indicate current data time. You can dynamically adjust the position of moving cursor to a new position.

To change the position of moving cursor

- Drag the cursor to the new position and release it. The cursor will keep moving from the new position.

During sine dwell and stationary animation, the cursor will remain its current position. You can drag the cursor to the new time position to view the different ODS.

To stop structure animation

- Click button ![button] on toolbar.

Frequency-based ODS

For an example, to perform structure animation of frequency-based ODS

- Select "plate" demo project in example finder.
- Click quick start "ODS Animation"

The structure view and data view window are opened and tiled in left and right, as Figure 5-5 shows.

- Load structure file "plate" into structure view window.
- Load data file "freq response" into data view window.
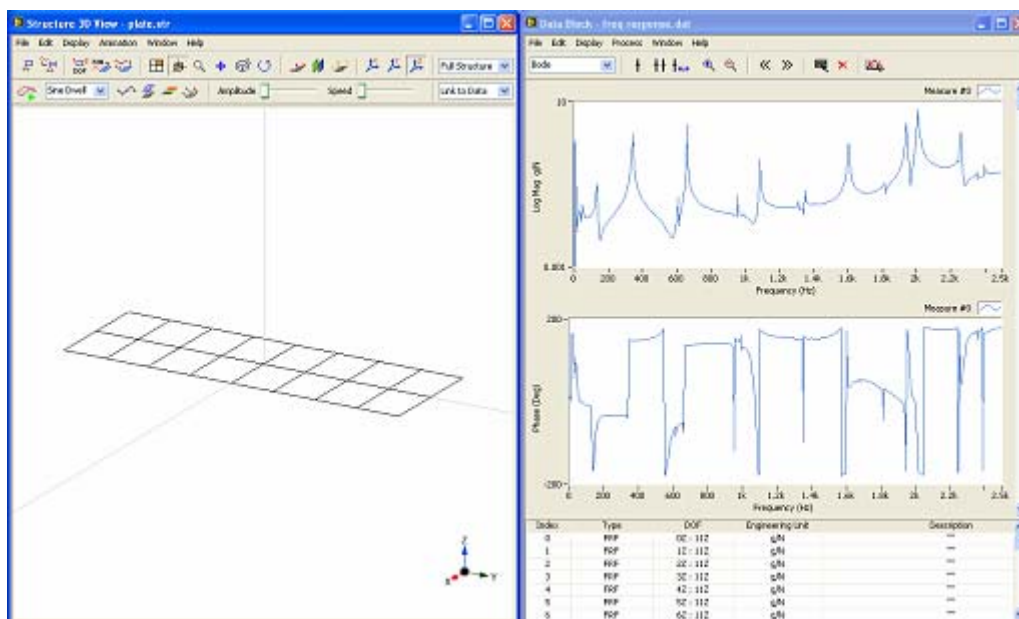
Figure 5-5 Frequency-based ODS

In structure view window

- Click button ![M# button] on toolbar.

Twenty-seven FRF measurements in data view window will be automatically assigned to twenty-seven measured nodes on beam structure according to matched DOF information. A dialog box will open to report the number of assigned measurements.

All nodes on plate structure are measured in this example. Animation interpolation is not needed.

It is ready to start structure animation of frequency-based ODS now.

- Click button on toolbar to start animation.
- Adjust and to proper values. In this example, amplitude and speed are specified as 10 and 5, respectively.

During animation, you can view different animation displays for a good understanding of structure's frequency-based ODS.

- Click button on the toolbar to overlap un-deformed structure with animation

To change animation type

- Select corresponding item in control on the toolbar

During sweep animation, the line cursor on data graph in data view window moves from left to right to indicate current data time. You can dynamically adjust the position of moving cursor to a new position.

To change the position of moving cursor

- Drag the cursor to the new position and release it. The cursor will keep moving from the new position.

During sine dwell and stationary animation, the cursor will remain its current position. You can drag the cursor to peaks on the FRF plot to view different ODS, as Figure 5-6 shows.
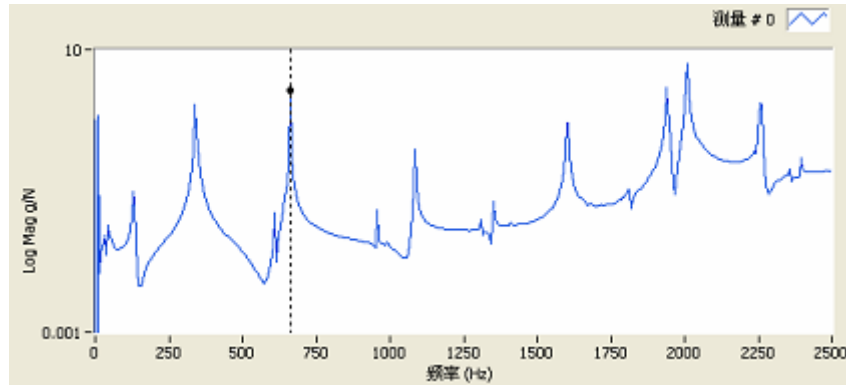
Figure 5-6 FRF peaks selection

To stop structure animation

- Click button on toolbar.

Mode shape

For an example, to perform structure animation of mode shape

- Select "plate" demo project in example finder.
- Click quick start "Modal Shape Animation"

The structure view and mode view window are opened and tiled in left and right, as Figure 5-7 shows.

- Load structure file "plate" into structure view window.
- Load mode shape file "mode" into mode view window.
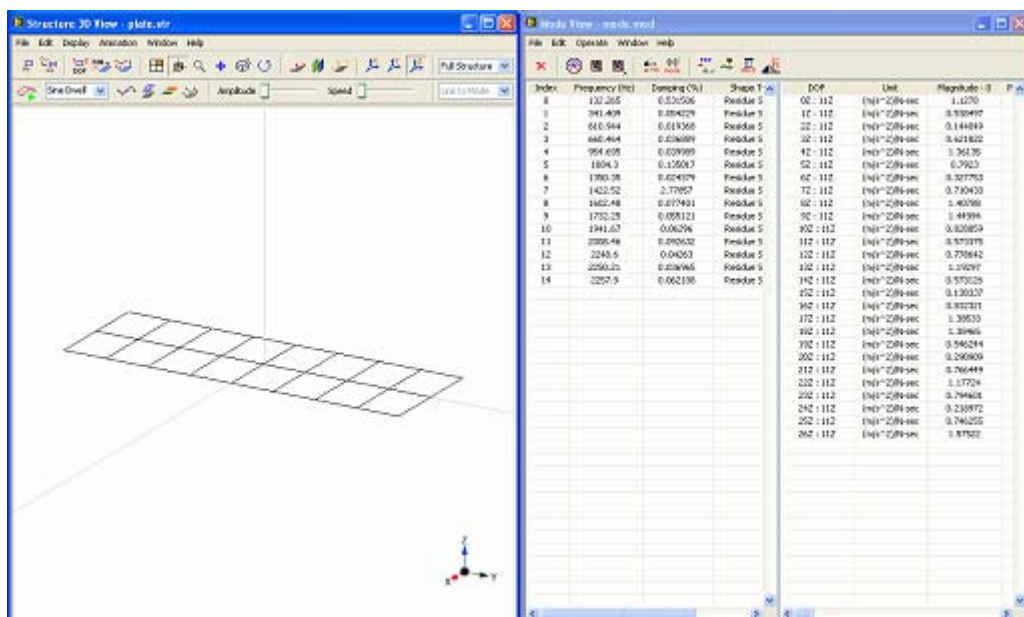
Figure 5-7 Mode shape animation

In structure view window

- Click button on toolbar.

Mode shapes with 27 DOF in mode view window will be automatically assigned to twenty-seven measured nodes on plate structure according to matched DOF information. A dialog box will open to report the number of assigned measurements.
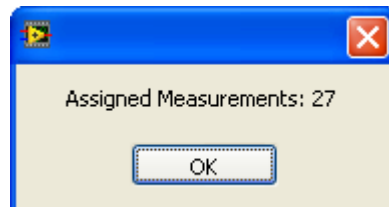
All nodes on 3D model are measured in this example. Animation interpolation is not needed.

It is ready to start structure animation of mode shape now. The default animation type is sweep. You can change the animation type before or during animation.

- Click button on toolbar to start animation.
- Adjust and to proper values. In this example, amplitude and speed are specified as 10 and 5, respectively.

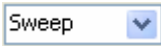During animation, you can view different animation displays for a good understanding of structure's mode shape.
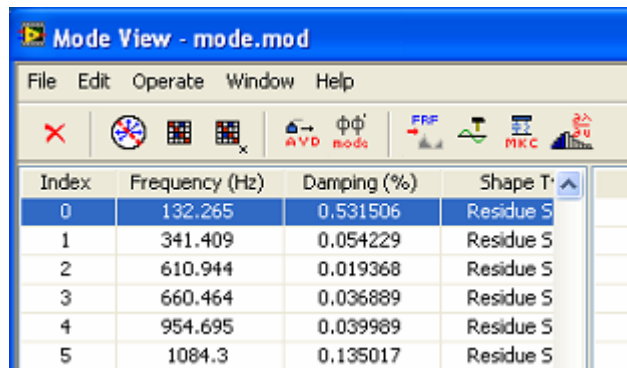
- Click button on the toolbar to overlap un-deformed structure with animation

To change animation type

- Select corresponding item in control on the toolbar

During sweep animation, the mode shape in mode view window will be displayed one by one.

During sine dwell and stationary animation, the selected mode shape is displayed. You can click to select a new mode in mode spreadsheet for mode shape display.

To stop structure animation

- Click button on toolbar.

5.4 Digital movie

ModalVIEW can document structure animation process by saving animation frame into AVI file. The video can be played back using Windows Media Player for presentation.

Frame rate

Frame rate decides how fast are the digital movie played back. The default frame rate for created structure animation movie is 5 frames per second.

To change the default frame rate

- Execute menu item **Operate | Option…**.

The system settings window is opened as Figure 5-8.
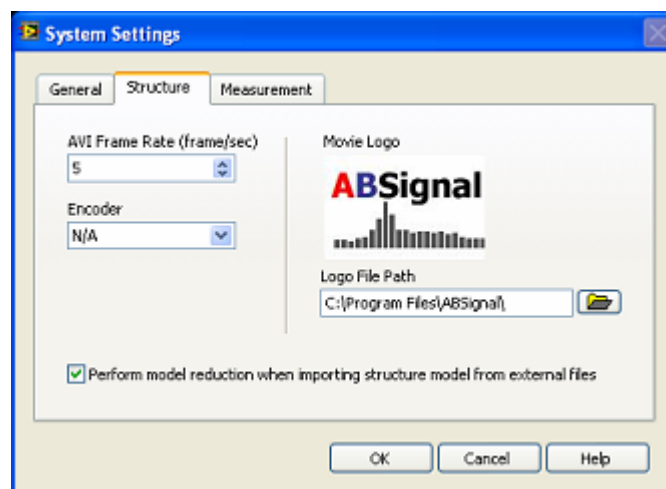
Figure 5-8 System settings window

In structure tab page

- Change the value of **AVI Frame Rate** into a new one.

AVI Codec

In order to reduce file size of created structure animation video, ModalVIEW supports two kinds of compression formats, MSCV and DIVX. Before using compression method, make sure the corresponding compression Codec is installed on the computer. To be able to play the created AVI video in other computer, the computer must have the same decompression Codec that was used to compress the AVI file originally.

To change the compression method

- In project window, execute menu item **Operate | Option…**.

The system settings window is opened.

In structure tab page

- Change the method of **Encoder** into a new one. The default method is **N/A**, which means no compression method is used.

Movie logo

Sometimes it's necessary to emphasize the source of origin of generated animation movie. You can use your logo on surface of video clip. ModalVIEW automatically adds the selected movie logo stored in JPEG format to the upper-right corner of generated video clip.

To change the moive logo

- Specify the logo file path in structure tab page.
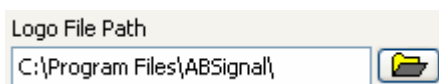
Movie generation

For example, to create digital movie of structure animation

- Load project file 'plate' from example finder.

In this example, ODS animation is used.

- Click quick start "ODS Animation".

The structure view window and data view window will be tiled left and right.

- Load model structure "plate" into structure view window.
- Load data block "freq response" into data view window.

In structure view window

- Select proper animation type.
- Click button on toolbar to review model animation process.
- Adjust animation amplitude to a proper value.
- Execute menu **File | Export to Movie…**.

A file open dialog is displayed to prompt you to input desired new file name.
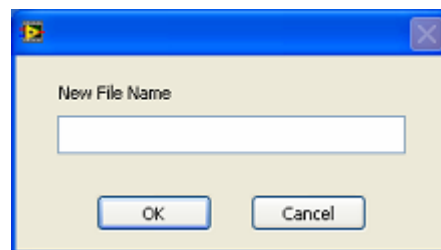
Type the desired file name into the **New File Name** box and click OK button. The process to generate AVI movie will start.
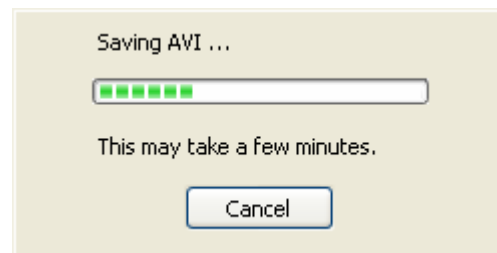
 NOTE: When AVI compression codec is not used，it may take long time to generate AVI file. And the generated file size may be large.

When the process finishes, a dialog box will display to indicate the AVI file is generated.
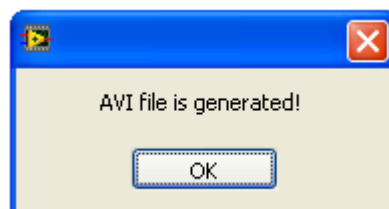
- Double click newly generated AVI file in project tree.

The AVI Movie is displayed in Windows Media Player.

6. Report generation

Test report is a very important document for modal testing experiment. ModalVIEW provides a standard report template for users to set report content about test object, operator, instruments, mode list, model shape diagram etc. Word format report can be automatically generated with the specified content.

6.1 Report items

ModalVIEW supports following pre-defined report contents:

Test date
Test object
Operator / Department
Note
Test Instruments (Device name and device type)
Structure View
Channel Settings
Test Settings
Analysis Settings
Analysis Results
Shape View

Test information

The report information of test date, test object, operator, note and test instruments are specified in report generation window, as Figure 6-1 shows.
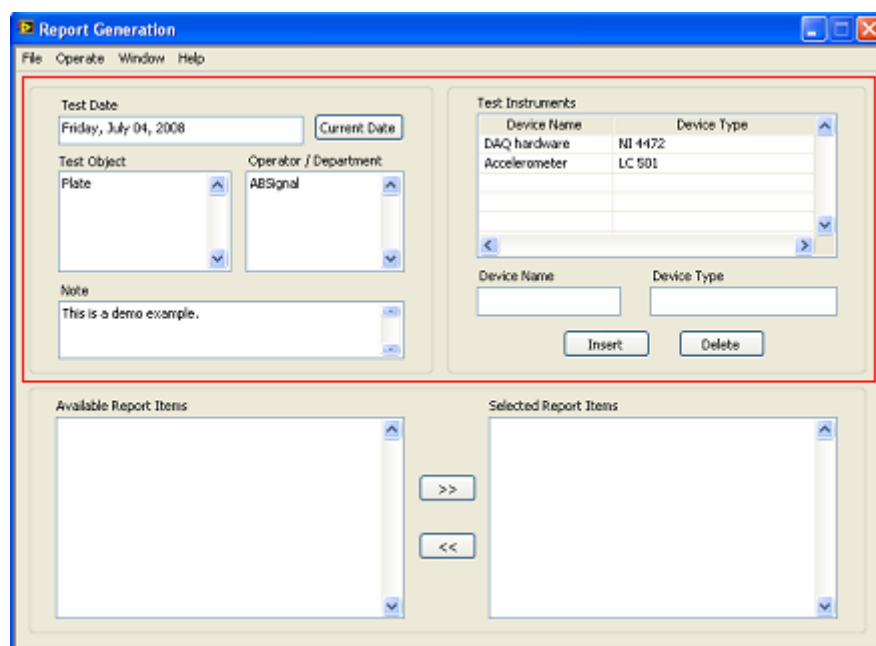
Figure 6-1 Report information settings

The following content is added to generated report.

Test Date: Friday, July 04, 2008	
Test Object: Plate	
Operator: ABSignal	
Note: This is a demo example.	
Test Instruments:	
Device Name	**Device Type**
DAQ hardware	NI 4472
Accelerometer	LC 501

Structure view

The content of structure view is specified in structure view window.

To add structure view to report content

* Execute menu item **File | Copy to Report | Structure View…**

A dialog box is displayed to prompt you to input desired report item name. Type "plate" in **Report item name** box and click **OK** button, the current view of structure model is added to report item list accordingly.

In report generator window, the report item "plate – structure view" will be displayed in the **Available Report Items** list.

* Click button , add "plate – structure view" into **Selected Report Items** list.

The following content is added to generated report.

Structure View:

Channel settings

The content of channel settings is specified in measurement window.

To add channel settings to report content

- Execute menu item **File | Copy to Report | Channel Settings…**

A dialog box is displayed to prompt you to input desired report item name. Type "NI 4472" in **Report item name** box and click **OK** button, the current channel setting for measurement is added to report item list accordingly.

In report generator window, the report item "NI 4472 – channel settings" will be displayed in the **Available Report Items** list.

- Click button [>>], add "NI 4472 – channel settings" into **Selected Report Items** list.

The following content is added to generated report.

Channel Settings:

CH	Input Range	Sensitivity	Engineering Unit
0	+/- 5 V	1000 mV/EU	V
1	+/- 5 V	1000 mV/EU	V
2	+/- 5 V	1000 mV/EU	V
3	+/- 5 V	1000 mV/EU	V
4	+/- 5 V	1000 mV/EU	V
5	+/- 5 V	1000 mV/EU	V
6	+/- 5 V	1000 mV/EU	V
7	+/- 5 V	1000 mV/EU	V

Test settings

The content of test settings is specified in measurement window.

To add test settings to report content

- Execute menu item **File | Copy to Report | Test Settings…**

A dialog box is displayed to prompt you to input desired report item name. Type "hammer test" in **Report item name** box and click **OK** button, the current channel setting for measurement is added to report item list accordingly.

In report generator window, the report item "hammer – test settings" will be displayed in the **Available Report Items** list.

- Click button , add "hammer test – channel settings" into **Selected Report Items** list.

The following content is added to generated report.

> **Test Settings:**
>
> Modal Testing Method: Roving Excitation - FRF
>
> Roving DOFs: 0Z 1Z 2Z 3Z 4Z 5Z 6Z 7Z 8Z 9Z 10Z 11Z 12Z 13Z 14Z 15Z 16Z 17Z 18Z 19Z 20Z 21Z 22Z 23Z 24Z 25Z 26Z
>
> Reference DOFs: 0Z
>
> Pre-process: Rectangle window, Linear average# 3
>
> Sampling Rate (Hz): 1024.000

Analysis settings

The content of analysis settings is specified in mode estimation window.

To add analysis settings to report content

- Execute menu item **File | Copy to Report | Analysis Settings…**

A dialog box is displayed to prompt you to input desired report item name. Type "quick fit" in **Report item name** box and click **OK** button, the current analysis setting for mode estimation is added to report item list accordingly.

In report generator window, the report item "quick fit – analysis settings" will be displayed in the **Available Report Items** list.

- Click button [>>], add "quick fit – analysis settings" into **Selected Report Items** list.

The following content is added to generated report.

> **Analysis Settings:**
>
> Analysis Method: Quick Fit
>
> Frequency Band (Hz): 195.000 - 890.000

Analysis results

The content of analysis result is specified in mode estimation window.

To add analysis result to report content

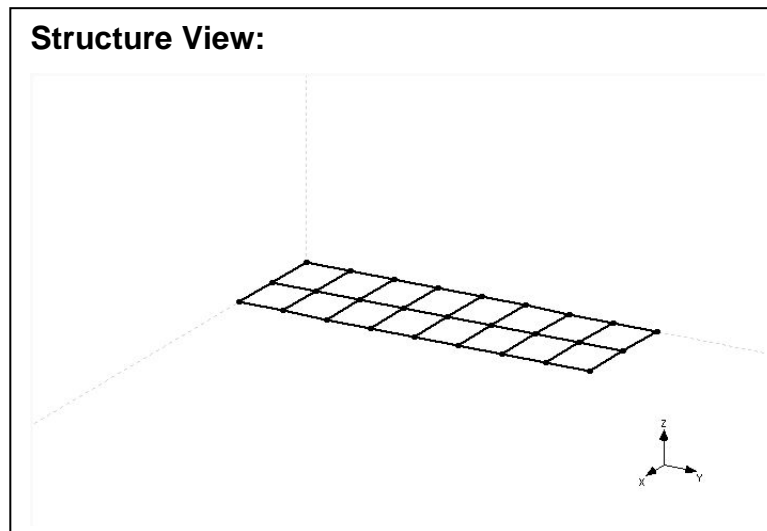- Execute menu item **File | Copy to Report | Mode List…**

A dialog box is displayed to prompt you to input desired report item name. Type "low freq modes" in **Report item name** box and click **OK** button, the estimated mode list is added to report item list accordingly.

In report generator window, the report item "low freq modes – analysis results" will be displayed in the **Available Report Items** list.

- Click button [>>], add "low freq modes – analysis results" into **Selected Report Items** list.

The following content is added to generated report.

Analysis Results:

Frequency (Hz)	Damping (%)
660.459979	0.035915
341.410791	0.054639

Shape view

The content of structure shape views is specified in structure view window. During stationary dwell, select to display different shape views in structure window.

To select a group of modal shape views and copy it to report generator

- Execute menu item **File | Copy to Report | Shape View Arrangement…**

A dialog box is displayed to prompt you to insert current modal shape view in structure window.
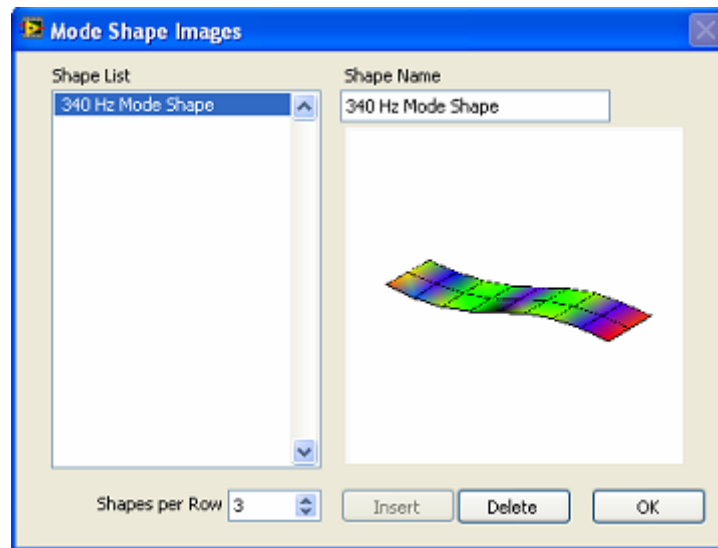
- Input the name of shape view image, and click [Insert] button. The shape view will be added into shape view list.
- Click **OK** button, close

- Repeat above steps to different mode shape view.

In the window, [Shapes per Row 3] decides how many shape views are arranged in a single row, when overall shape view image is generated for report.

To copy selected mode shape images to report generator

- Execute menu item **File | Copy to Report | Mode Shape View…**

A dialog box is displayed to prompt you to input desired report item name. Type "shapes" in **Report item name** box and click **OK** button, the view of mode shapes is added to report item list accordingly.

In report generator window, the report item "shapes – shape view" will be displayed in the **Available Report Items** list.

- Click button [>>], add "shapes – shape view" into **Selected Report Items** list.

The following content is added to generated report.

Mode Shape View:
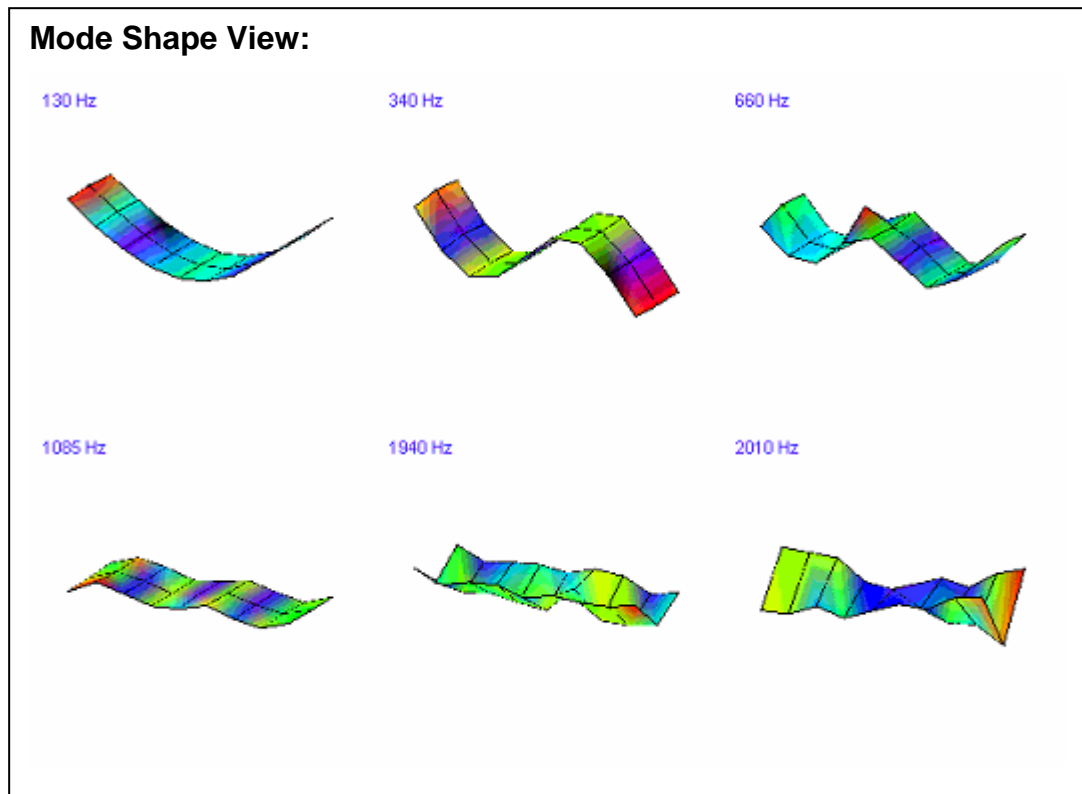
All the report contents available to be added into the test report are listed in **Available Report Items** list in report generator window.

To delete the selected report item from the list

- Select one item in the **Available Report Items** list.
- Execute menu item **Operate | Delete Selected Item**.

To clear all the available report item in the list

- Execute menu item **Operate | Clear All Report Items**.

Generate test report

To generate test report after specifying all the report information,

- Execute menu item **File | Generate Report...**.

A file open dialog is displayed to prompt you to select desired new report file name.
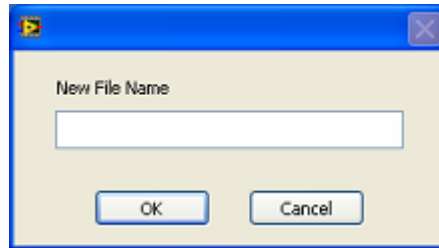
Type the desired file name into the **New File Name** box and click **OK** button.

When test report is successfully generated, a dialog box will show as following
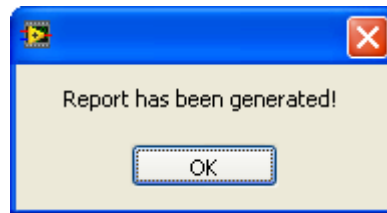
NOTE: Please close other opened word documents before generating report. Or testing report may not be generated normally, a warning dialog window will display.
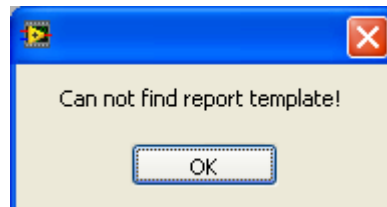
• Double click newly generated report file in project tree.

The report file is displayed in Microsoft Word.
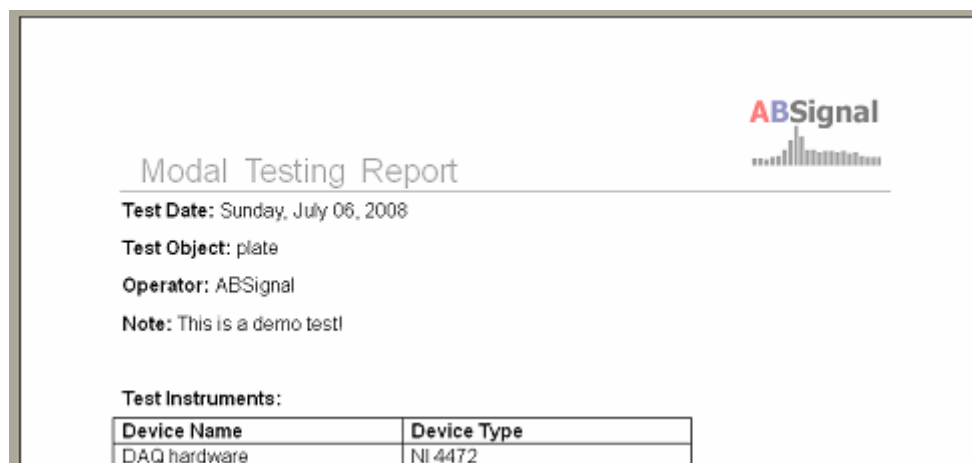
6.2 Clipboard copy

ModalVIEW provides a series of operations to copy data spreadsheet, graphs from different ModalVIEW windows into Windows Clipboard. The copied contents can be pasted into other software tools to create test report.

The following contents can be copied to Windows clipboard:

Channel spreadsheet
Structure bitmap
Structure spreadsheet
Data plot bitmap
Data spreadsheet
Data trace
Mode spreadsheet
Stabilization chart bitmap
Complexity plot bitmap
MAC plot bitmap
MAC spreadsheet

Channel spreadsheet

The channel settings for data acquisition can be copied into clipboard in ASCII text spreadsheet, as following.

Device	CH	Status	Coupling	IEPE	Input Range	Sensitivity	Engineering Unit	
PXI1Slot2	0	On	AC	4 mA	+/- 5 V	1000 mV/EU	V	
PXI1Slot2	1	On	AC	4 mA	+/- 5 V	1000 mV/EU	V	
PXI1Slot2	2	On	AC	4 mA	+/- 5 V	1000 mV/EU	V	
PXI1Slot2	3	On	AC	4 mA	+/- 5 V	1000 mV/EU	V	
PXI1Slot2	4	On	AC	4 mA	+/- 5 V	1000 mV/EU	V	
PXI1Slot2	5	On	AC	4 mA	+/- 5 V	1000 mV/EU	V	
PXI1Slot2	6	On	AC	4 mA	+/- 5 V	1000 mV/EU	V	
PXI1Slot2	7	On	AC	4 mA	+/- 5 V	1000 mV/EU	V	

To copy the content into clipboard in measurement window

- Execute menu item **File | Copy to Clipboard | Channel Spreadsheet**

Structure bitmap

The bitmap of displayed structure model can be copied into clipboard, as following.
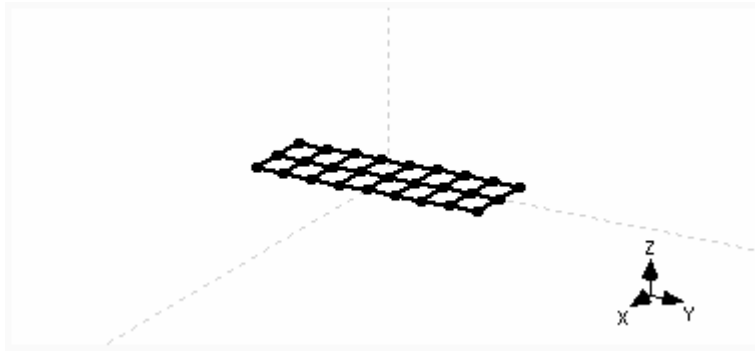
To copy the content into clipboard in structure view window

• Execute menu item **File | Copy to Clipboard | Bitmap**

Structure spreadsheet

The information nodes and lines of displayed structure model can be copied into clipboard in ASCII text spreadsheet, as following.

```
[points]
0       -50.000     -150.000     0.000
1       -50.000     -112.500     0.000
2       -50.000     -75.000      0.000
3       -50.000     -37.500      0.000
4       -50.000     0.000   0.000
5       -50.000     37.500 0.000
6       -50.000     75.000 0.000
7       -50.000     112.500      0.000
…
[lines]
0       9       0
1       10      1
2       11      2
3       12      3
4       13      4
…
```

In section [points], the first column is the point index, the column 2-4 are the x, y, z coordination. In section [lines], the first column is the line index, the column 2-3 are the index of connected points.

To copy the content into clipboard in structure view window

• Execute menu item **File | Copy to Clipboard | Spreadsheet**

Data plot bitmap

The graph of data plot displayed in data view window can be copied into clipboard in bitmap, as following.
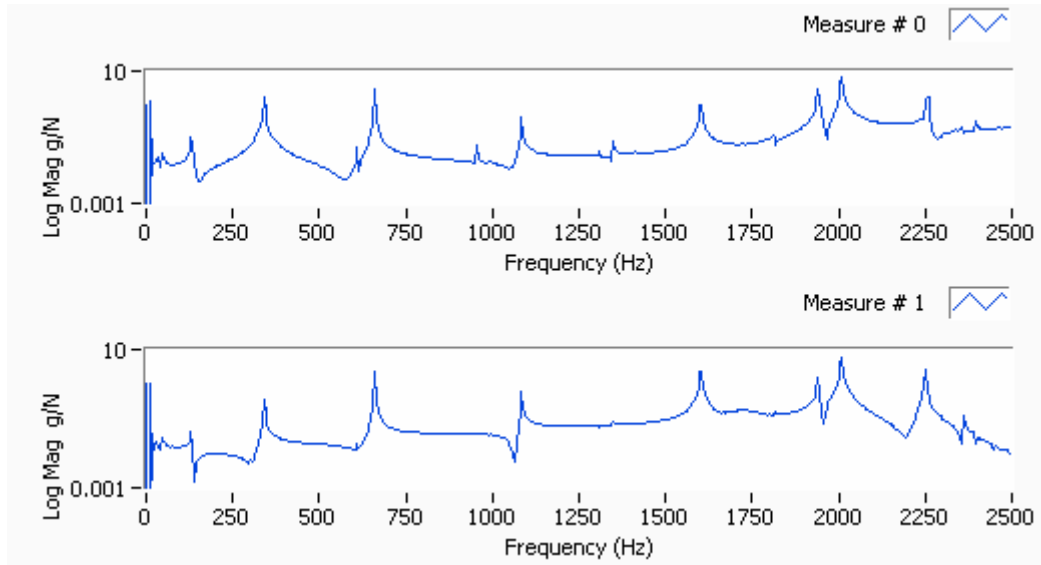
To copy the content into clipboard in data view window

- Execute menu item **File | Copy to Clipboard | Bitmap**

Data spreadsheet

The data spreadsheet in data view window can be copied into clipboard in ASCII text spreadsheet, as following.

Index	Type	DOF	Engineerin	Description
0	FRF	0Z : 11Z	g/N	
1	FRF	1Z : 11Z	g/N	
2	FRF	2Z : 11Z	g/N	
3	FRF	3Z : 11Z	g/N	
4	FRF	4Z : 11Z	g/N	
5	FRF	5Z : 11Z	g/N	
6	FRF	6Z : 11Z	g/N	
7	FRF	7Z : 11Z	g/N	
8	FRF	8Z : 11Z	g/N	
9	FRF	9Z : 11Z	g/N	

To copy the content into clipboard in data view window

- Execute menu item **File | Copy to Clipboard | Spreadsheet**

Data trace

The data spreadsheet in data view window can be copied into clipboard in ASCII text spreadsheet, as following.

Measurem	Frequency	Response						
Engineerin	g/N			g/N			g/N	
DOF	0Z : 11Z			1Z : 11Z			2Z : 11Z	
Index	Frequency	Mag	Phase	Frequency	Mag	Phase	Frequency	
0	0	1	0.190516	0	1	0.012539	0
1	5	0	0.193086	5	0	0.01281	5
2	10	1.185106	0	10	1.035565	0	10
3	15	0.006946	2.76216	15	0.001668	2.782121	15
4	20	0.013009	2.892504	20	0.008926	2.897676	20
5	25	0.015893	2.220853	25	0.015875	1.990117	25

To copy the content into clipboard in data view window

- Execute menu item **File | Copy to Clipboard | Trace Data**

Mode spreadsheet

The mode spreadsheet in mode view window can be copied into clipboard in ASCII text spreadsheet, as following.

Index	Frequency (Hz)	Damping (%)	Shape Type
0	132.265	0.532	Residue Shape
1	341.409	0.054	Residue Shape
2	610.944	0.019	Residue Shape
3	660.464	0.037	Residue Shape
DOF	Magnitude	Phase (Deg)	Unit
0Z : 11Z	1.1	-101.5	
1Z : 11Z	0.5	-88.4	
2Z : 11Z	0.1	94.9	
3Z : 11Z	0.6	89.8	
4Z : 11Z	1.4	85.4	
5Z : 11Z	0.8	81.8	
6Z : 11Z	0.3	-71.6	

To copy the content into clipboard in mode view window

- Execute menu item **File | Copy to Clipboard | Spreadsheet**

Stabilization chart bitmap

The stabilization plot displayed in stabilization chart window can be copied into clipboard in bitmap, as following.
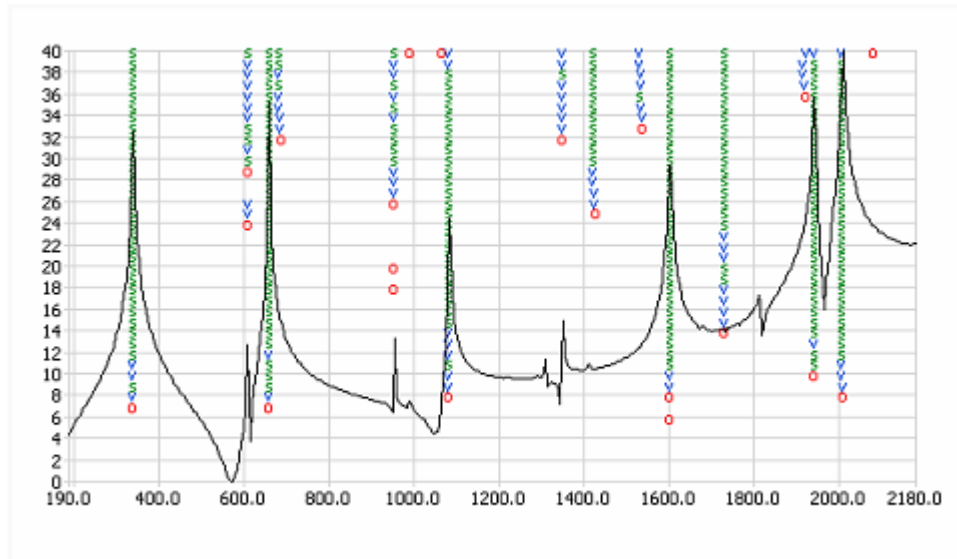
To copy the content into clipboard in stabilization char window

- Execute menu item **File | Copy to Clipboard | Bitmap**

Complexity plot bitmap

The complexity plot displayed in mode plot window can be copied into clipboard in bitmap, as following.
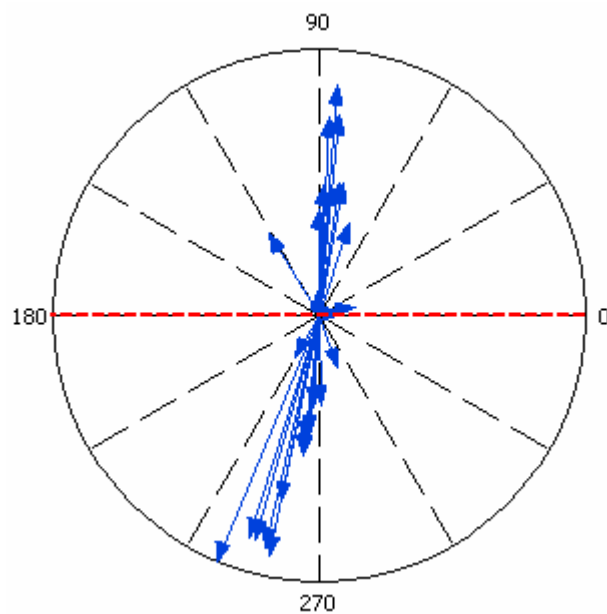
To copy the content into clipboard in mode plot window

- Execute menu item **File | Copy to Clipboard | Bitmap**

MAC plot bitmap

The MAC plot displayed in mode plot window can be copied into clipboard in bitmap, as following.
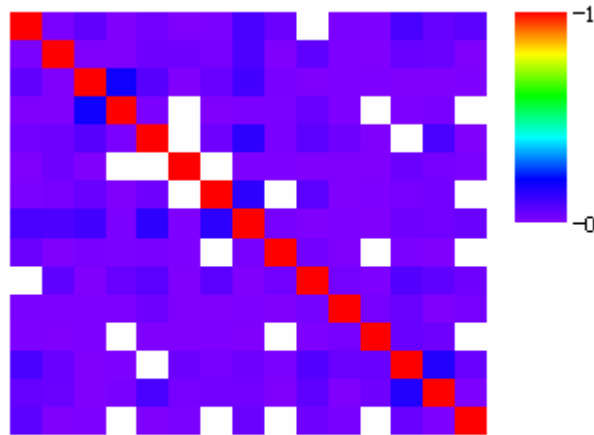
To copy the content into clipboard in mode plot window

- Execute menu item **File | Copy to Clipboard | Bitmap**

MAC spreadsheet

The valued of MAC plot in mode view window can be copied into clipboard in ASCII text spreadsheet, as following.

	Mode 0	Mode 1	Mode 2	Mode 3	Mode 4	Mode 5
Mode 0	1	0.01	0.05	0.005	0.027	0.005
Mode 1	0.01	1	0.006	0.005	0.031	0.03
Mode 2	0.05	0.006	1	0.18	0.065	0.003
Mode 3	0.005	0.005	0.18	1	0.011	0.001
Mode 4	0.027	0.031	0.065	0.011	1	0
Mode 5	0.005	0.03	0.003	0.001	0	1

To copy the content into clipboard in mode plot window

- Execute menu item **File | Copy to Clipboard | Spreadsheet**

7.Importing External Files

ModalVIEW can import measured data, mode shape and structure model stored in UFF or ASCII text format.

7.1   UFF file format

UFF file format is commonly used in area of modal analysis for data exchange. There are several existing data formats. ModalVIEW can import following data formats.

Data Format	Data Type
UFF 15	Structure model (points)
UFF 82	Structure model (lines)
UFF 55	Mode shape
UFF 58	Measurement data

UFF 15 defines the position information of nodes on 3D model. UFF 82 defines the information of connected nodes for lines on 3D model.

To import structure model from a UFF file in structure view window

- Execute menu item **File | Import | UFF File…**

A windows file open dialog box will open for you to select UFF structure file.

> **NOTE: Node information (UFF 15 data set) and line information (UFF 82 data set) must be contained in a single UFF structure file.**

UFF 55 defines the information of mode shapes. To import mode shapes from a UFF file in mode view window

- Execute menu item **File | Import | UFF File…**

A windows file open dialog box will open for you to select UFF mode shape file.

UFF 58 defines the information of measured data including time record or FRF measurement. To import measured data from a UFF file in data view window

- Execute menu item **File | Import | UFF File…**

A windows file open dialog box will open for you to select UFF data file.

7.2   ASCII text file format

ModalVIEW can import measured data (time record and FRF measurement) and structure model from files in ASCII text format.

Structure model

The format of ASCII text file for structure model is defined as following:

[points]
point index<tab>x<tab>y<tab>z
point index<tab>x<tab>y<tab>z
…
point index<tab>x<tab>y<tab>z

[lines]
line index<tab>point index<tab>point index
line index<tab>point index<tab>point index
…
line index<tab>point index<tab>point index

[surfaces]
surface index<tab>point index<tab>point index<tab>point index<tab>point index
surface index<tab>point index<tab>point index<tab>point index<tab>point index
…
surface index<tab>point index<tab>point index<tab>point index<tab>point index

[points], [lines] and **[surfaces]** are keywords. They can not be modified.

In section **[points]**, each row defines one node on 3D model including point index and x, y, z coordination. Each value is separated by a single tab character.

In section **[lines]**, each row defines one connecting line including line index, start point index and end point index. Each point index is separated by a single tab character.

In section **[surfaces]**, each row defines one structure surface including surface index, 4 point indexes. 4 points define a quadrangular surface. If the last point index equals to -1, the first 3 points define a triangular surface. Each point index is separated by a single tab character.

For an example, the following content defines a plate structure. You can copy the content into Windows Notepad, save it into a text file and import it into structure view window.

```
[points]
0       0       0       0
1       0       50      0
2       0       0       50
3       0       50      50
[lines]
0       0       2
1       1       3
2       0       1
3       2       3
[surfaces]
0       0       1       2       3
```

To import structure model from an ASCII text file in structure view window

- Execute menu item **File | Import | ASCII…**

A windows file open dialog box will open for you to select ASCII structure file.

Measured data

The format of ASCII text file for measured data is defined as following:

[type]
time/freq
[initial]
data<tab>data<tab>…data
[interval]
data <tab> data <tab>…data
[data]
data<tab>data<tab>…data
data<tab>data<tab>…data
…
data<tab>data<tab>…data

[type], **[initial], [interval] and [data]** are keywords. They can not be modified.

In section **[type]**, the followed value must be **time** or **freq**, which defines the type (time record or FRF measurement) of measurements stored in the ASCII text file.

In section **[initial]**, the followed values define the initial time or frequency for the time record or FRF measurement. The values are in one row and separated by a single tab character. The number of values is decided by the number of measurements stored in the file.

In section **[interval]**, the followed values define the time interval or frequency increment for the time record or FRF measurement for each measurement. The values are in one row and separated by a single tab character.

In section **[data]**, data format is decided by stored measurement type. For time record, each column of data is for one record. For FRF measurement, every two columns are for one measurement, which are magnitude and phase, respectively. Each row of data is the value at a time or frequency. The values in one row are separated by a single tab character.

For an example, the following content is a time record with 3 measurements. You can copy the content into Windows Notepad, save it into a text file and import it into data view window.

```
[type]
time
[initial]
0       0       0
[interval]
0.01    0.01    0.01
[data]
0.000   1.000   0.000
0.383   0.924   -0.383
0.707   0.707   -0.707
0.924   0.383   -0.924
1.000   0.000   -1.000
0.924   -0.383  -0.924
0.707   -0.707  -0.707
0.383   -0.924  -0.383
0.000   -1.000  0.000
-0.383  -0.924  0.383
-0.707  -0.707  0.707
-0.924  -0.383  0.924
-1.000  0.000   1.000
-0.924  0.383   0.924
-0.707  0.707   0.707
-0.383  0.924   0.383
```

To import measured data from an ASCII text file in data view window

• Execute menu item **File | Import | ASCII…**

A windows file open dialog box will open for you to select ASCII data file.

ModalVIEW also supports data import from multiple ASCII data file, in which the time record or FRF data from a single channel or DOF is stored. When the data type is time record, there is one column data in a single file. When the data type is FRF, there are two column data in a single file, one for magnitude and one for phase in radian. The two values in one row are separated by a single tab character.

> 📝 **NOTE: The default sampling rate or frequency interval for imported time record or FRF is one; please manually change the data properties to the correct value after importing the data.**

To import data block from multiple ASCII files in a file folder.

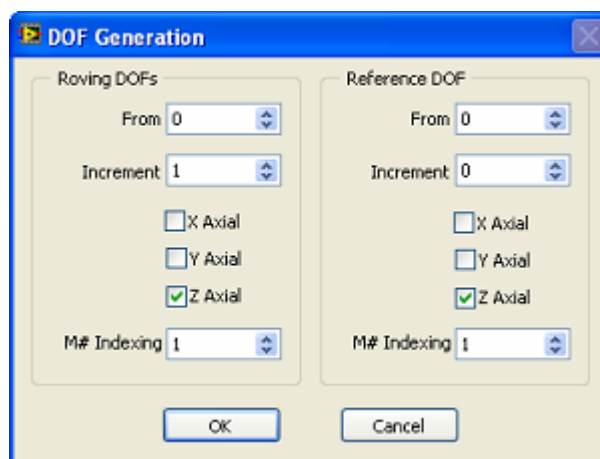- Execute menu item **File | Import | ASCII File Folder…** in data block window.

A file open dialog box will open for you to select the file folder keeping data files.
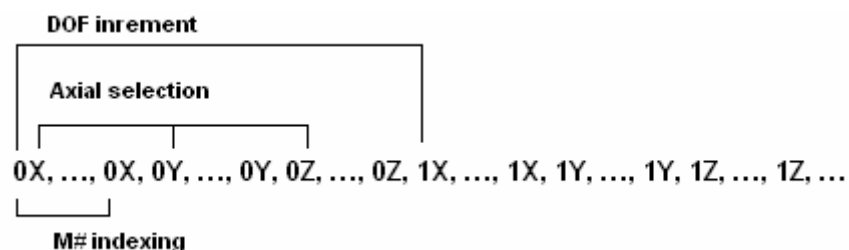
DOF Generation

The imported time record or FRF measurement contains no DOF information. To automatically create DOF information for imported data

- Execute menu item **Edit | DOFs…**

DOF Generation dialog box will open.

Specify the parameters and click **OK** button, a series of DOFs are generated according to following scheme.

DOF information for different data type is generated as following.

Data Type	DOF Form
Time record	Roving DOF (1X)

FRF	Roving DOF : Reference DOF (2X:0X)

DOF information can also be assigned to imported data by changing properties of each measurement. To change the property of selected measurement

- Execute menu item **Edit | Properties…**
- Or double click selected measurement in properties spreadsheet.

7.3   STL file format

STL file is a widely used standard data transmission format for 3D structure model in computer aided design (CAD) field. STL format specifies both ASCII and binary representations. ModalVIEW can import 3D model strucutre in STL file generated by other CAD software tools.

To import 3D structure model from an STL file in structure view window

- Execute menu item **File | Import | STL file…**

A windows file open dialog box will open for you to select STL file. The 3D structure model will be displayed in data view window, as Figure 7-1 shows.
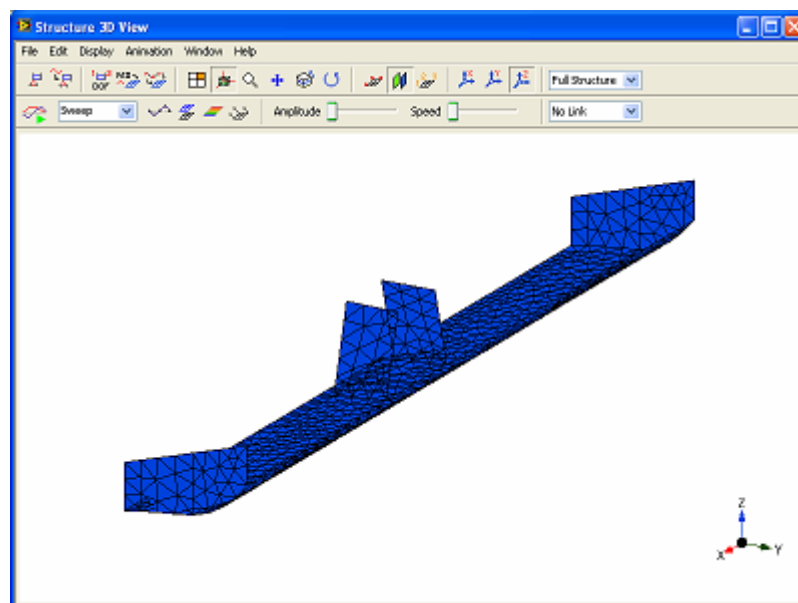

Figure 7-1 Import STL file

- Execute menu item **File | Save**, to **s**ave current imported structure into ModalVIEW structure file.