

# Modal Response of a Sensor Casing

For LUSAS version:	16.0
For software product(s):	All (except LT versions)
With product option(s):	Plus

## Description

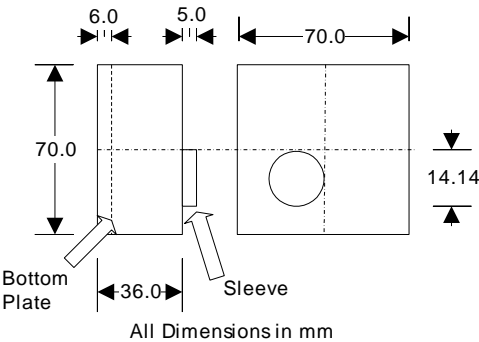
An aerospace sensor casing is to be assessed for dynamic stresses induced by vibration of the airframe to which it is attached.

The sensor casing is manufactured from steel plate with a uniform thickness of 0.8 mm.

The surfaces at the top of the sleeve are rigidly held. The loading is characterised by a random vibration at the supports of the airframe and is defined as an acceleration Power Spectral Density (PSD) specified by the airframe manufacturer.

The Interactive Modal Dynamics (IMD) facility is used to evaluate the response of the casing to this loading. A quarter model is initially defined with the sleeve modelled as if it were in the centre of the casing, and subsequently copied to create the full model. The sleeve is then repositioned to the location shown to show the associativity of features.

Units of N, m, kg, s, C are used for the analysis.



### Objectives

The following results plots are to be obtained:

- ☐ **Deformed Shape** A display of the deformed mesh for the first mode shape.
- ☐ **Frequency Response Function** FRF of a node using support motion excitation.
- ☐ **Power Spectral Density** PSD stress response at a node using a PSD excitation function.

### Keywords

3D, Shell, Linear, Eigenvalue, Scale Factor Transformation, Interactive Modal Dynamics (IMD), Default Attribute Assignment, Deformed Shape, Frequency Response Function (FRF), Power Spectral Density (PSD), Stress Contours.

### Associated Files

If results processing and not the actual modelling of an example is only of interest to you the VBS file(s) provided will allow you to quickly build a model for analysis. For more details refer to Creating a Model from the Supplied VBS files in the Introduction.



- ☐ **casing\_modelling.vbs** carries out the modelling of the example.

## Modelling

### Running LUSAS Modeller

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



**Note.** This example is written assuming a new LUSAS Modeller session has been started. If continuing from an existing Modeller session select the menu command **File>New** to start a new model file. Modeller will prompt for any unsaved data and display the New Model dialog.

### Creating a new model

- Enter the file name as **casing**
- Use the default User-defined working folder.
- Ensure an analysis type of **Structural** is set.

- Set the Analysis category as **3D**
- Set the model units as **N,m,kg,s,C**
- Ensure that timescale units are **Seconds**
- Select a **3D Beam/Shell** Startup template.
- Ensure the Layout grid is set as **None**
- Enter the title as **Modal Response of Sensor Casing**
- Click the **OK** button.

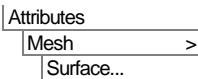


**Note.** Save the model regularly as the example progresses. Use the Undo button to correct any mistakes made since the last save was done.

## Default assignments

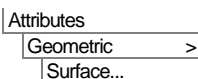
The material properties, plate thickness and mesh element type are uniform over the whole sensor casing. By using default attribute assignments, attribute datasets are automatically added to new features when they are created.

## Automatic Mesh Assignment




- Define a Surface mesh using **Thin shell**, **Quadrilateral** elements with **Linear** interpolation.
- Enter **Thin shell** for the mesh dataset name.
- Click the **OK** button to add the mesh dataset to the Treeview.
- To set this mesh dataset as the default surface mesh assignment click the right-hand mouse button on the **Thin shell** dataset name in the Treeview and select the **Set Default** option. The icon will change to indicating that **Thin shell** will automatically be assigned to all new Surfaces.

## Automatic Geometry Assignment




- Enter a Surface element thickness of **0.0008**
- The eccentricity can be left blank or set to zero as it is not required.
- Enter the dataset name as **Thickness of 0.0008** and click **OK** to add the dataset to the Treeview.

- To set this geometry dataset as the default geometry assignment click the right-hand mouse button on the **Thickness of 0.0008** dataset name in the  Treeview and select the **Set Default** option.

### Automatic Material Assignment

The material required for this example is provided by default when using the startup templates specified when creating the model.

- Set the material **Steel (Mild Steel Steel - Structural)** as the default material assignment for the sensor by clicking the right-hand mouse button on the dataset in the  Treeview and selecting the **Set Default** option.

### Feature Geometry

In a natural frequency analysis, consistent units such (N, m, kg, sec) must be used. It is however more convenient to define the geometry in millimetres and scale the model when the geometry input is complete.

### Defining the sleeve of the sensor



**Note.** The sleeve is defined initially as if it were positioned centrally on the casing. Later in the example it will be moved to its actual position to show the associativity of features.



Define a Line on the sleeve from (0,0) to (-5,-5) and click the **OK** button.

- Select the Line just drawn.

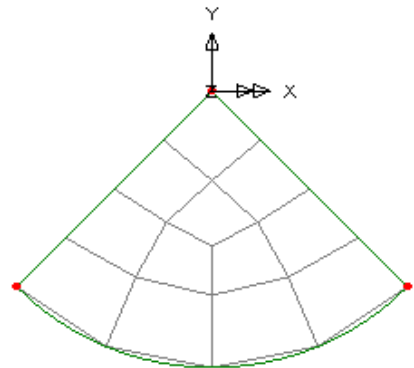


Sweep the line into a surface by choosing the **Rotate** option and entering an angle of **90** degrees to rotate the line about the **Z-axis** about an origin of **0,0,0**


- Click the **OK** button to complete the Sweep operation.



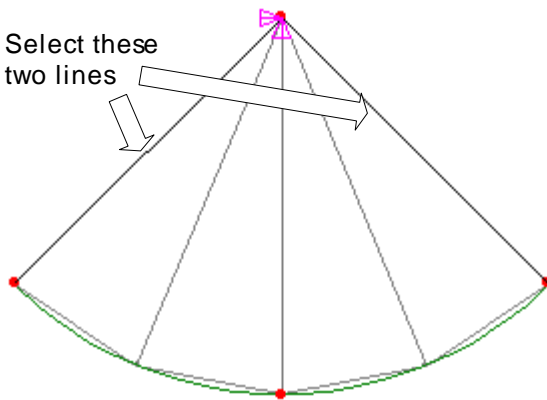
Select the fleshing on/off button to turn-off geometric property visualisation.



The number of elements modelling the end of the sleeve will be reduced.

- Select the two radial Lines and adjust the mesh by assigning Line mesh **Null, div = 1** from the  Treeview.

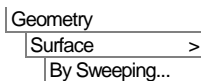
Select these two lines



## Defining the mesh for the sleeve

The arc will now be swept to create a Surface.

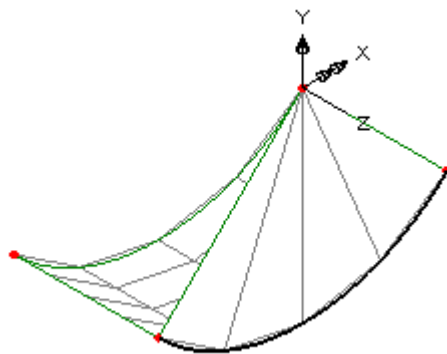
- Select the arc



Sweep the arc into a surface by selecting the **Translate** option and enter a translation of **-5** in the **Z** direction. Click the **OK** button to create the surface.



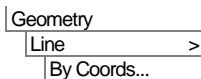
Use the Isometric view button to view the Surface created.



The sides of the sensor will now be defined.



**Note.** To simplify the modelling the sleeve will be defined at the centre of the plate and moved when the full model has been generated.



Enter coordinates of **(-35, -35, -5)** and **(35, -35, -5)** to define a Line representing the bottom edge of the housing. Click **OK** to generate the Line.

A new Surface can be created by joining this new Line to the previously defined arc to form a quarter of the housing plate.

## Modal Response of a Sensor Casing

- Select the arc shown right and add the Line at the bottom edge of the sensor to the selection by hold down the **Shift** key.



A Surface will be created.

The Line at the bottom edge of the sensor casing is now to be swept to create the bottom Surface of the casing.

- Select the bottom edge Line of the housing.



Choose the **Translate** option and define a translation of **-30** in the **Z** direction. Click the **OK** button to create the new Surface.

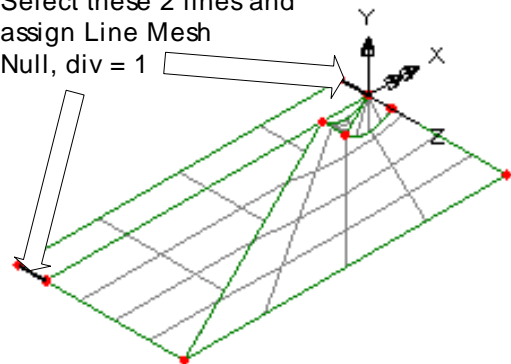
To define the portion of casing beyond the bottom plate, select the Line shown.



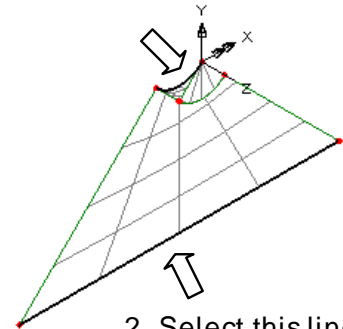
Define a translation of **-6** in the **Z** direction and click **OK** to sweep a new Surface.

- Now adjust the mesh on the new surface by selecting the 2 Lines of the casing indicated in the diagram and drag and drop the Line mesh **Null, div = 1** from the Treeview onto the selected features.

Select these 2 lines and assign Line Mesh  
Null, div = 1

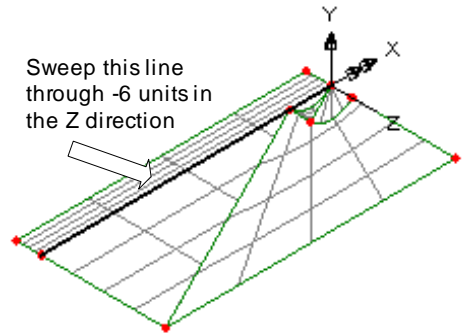


1. Select this arc



2. Select this line

Sweep this line  
through -6 units in  
the Z direction



This completes the definition of the features of the quarter model.


## Supports



**Note.** Supports are to be assigned to the quarter model before copying is done to save time assigning the support dataset to the equivalent copied Surfaces.

- Select the Surface at the end of the sleeve. If necessary, zoom in and cycle through the displayed features by clicking the left-hand mouse button until the Surface required is highlighted.

Selected: Surface 1 (4 items cyclable)

- Drag and drop the support dataset **Pinned** from the  Treeview onto the selected surface and click **OK**

The full housing model can now be created from the quarter model

- Using the **Ctrl** and **A** keys together, select the whole model.



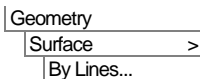
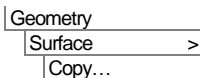
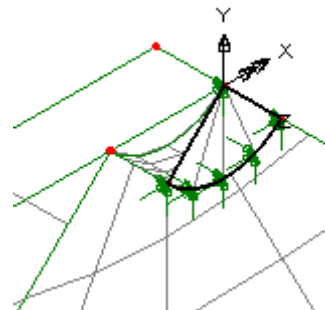
Use the copy button to **Rotate** the selected features through **90** degrees about the **Z-axis** and create **3** copies of the original selection.

- Click **OK** and Modeller will create and display the extra features.

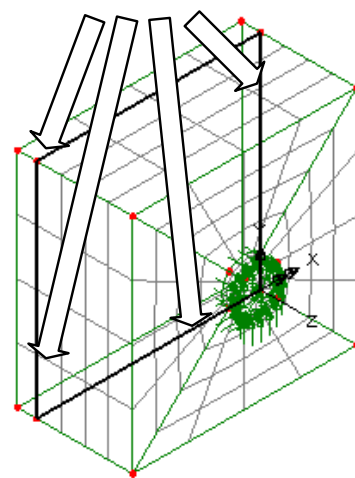
- To generate the bottom plate select the 4 Lines shown.



The Surface defining the bottom plate will be created.



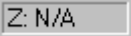
Select these 4 lines




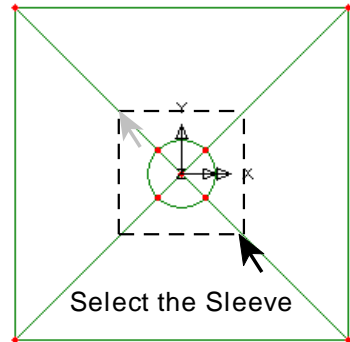
## Modal Response of a Sensor Casing


---

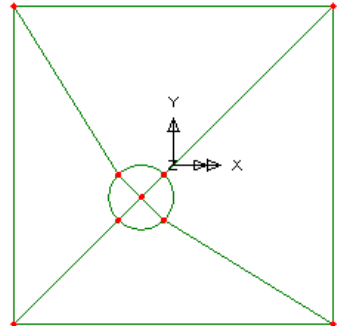
Finally the sleeve must be moved off centre.

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

- For clarity delete the **Mesh** and **Attributes** layers from the  Treeview.
- Box select the features which make up the sleeve.




 Move the sleeve to the required position by entering a translation of **-7.07** in the X direction and **-7.07** in the Y direction and click **OK**



Finally the geometry must be scaled so the units are metres.

- Use the **Ctrl + A** keys together to select the full model.

 Select the **Scale** option and enter an X, Y and Z scale factor of **0.001** about an origin of 0,0,0.


- Click **OK** to scale the geometry from millimetres to metres.



## Eigenvalue Analysis Control

To carry out results processing using the Interactive Modal Dynamics facility, an eigenvalue analysis must be performed. The results from the eigenvalue analysis normalised to global mass will be used to perform the Frequency Response Function calculations.

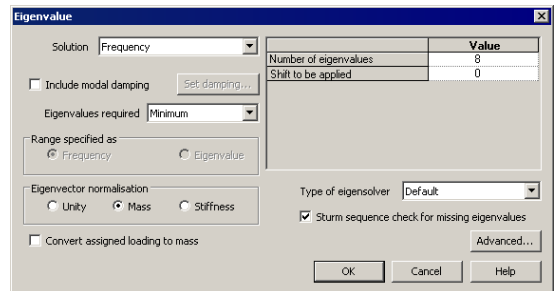
The eigenvalue analysis control properties are applied as a function of the load case.

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

The Eigenvalue dialog will appear.

The following parameters need to be specified to perform a frequency analysis with the minimum number of eigenvalues.

- Set the **Number of eigenvalues** required as **8**
- Set the **Shift to be applied** as **0**
- Leave the type of eigensolver as **Default**



The Eigenvalue dialog box is shown with the following settings:

- Solution:** Frequency
- Include modal damping:** ☐ (Set damping... button)
- Eigenvalues required:** Minimum
- Range specified as:** ☒ Frequency ☐ Eigenvalue
- Eigenvector normalisation:** ☐ Unity ☒ Mass ☐ Stiffness
- Convert assigned loading to mass:** ☐
- Type of eigensolver:** Default
- Sturm sequence check for missing eigenvalues:** ☒
- Advanced...** button
- OK**, **Cancel**, and **Help** buttons.

	Value
Number of eigenvalues	8
Shift to be applied	0



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

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

The model is now complete.


## Running the Analysis



Open the **Solve Now** dialog and press **OK** to run the analysis.

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

### If the analysis is successful...

Analysis loadcase results are added to the  Treeview.

In addition, 2 files will be created in the Associated Model Data directory where the model file resides:



- ☐ **casing.out** this output file contains details of model data, assigned attributes and selected statistics of the analysis.
- ☐ **casing.mys** this is the LUSAS results file which is loaded automatically into the Treeview to allow results processing to take place.

### If the analysis fails...

If the analysis fails, information relating to the nature of the error encountered can be written to an output file in addition to the text output window. Any errors listed in the text output window should be corrected in LUSAS Modeller before saving the model and re-running the analysis.

### Rebuilding a Model

If it proves impossible for you to correct the errors reported a file is provided to enable you to re-create the model from scratch and run an analysis successfully.



- ☐ **casing\_modelling.vbs** carries out the modelling of the example.



Start a new model file. If an existing model is open Modeller will prompt for unsaved data to be saved before opening the new file.

- Enter the file name as **casing** and click **OK**

To recreate the model, select the file **casing\_modelling.vbs** located in the \<LUSAS Installation Folder>\Examples\Modeller directory.



Rerun the analysis to generate the results

File  
New...


File  
Script >  
Run Script...

## Viewing the Results






This section outlines some typical results processing operations for a natural frequency and Interactive Modal Dynamics (IMD) analysis. The following interactive results processing operations are performed:

- ☐ **Mode Shape Plots** Displaying mode shapes from the natural frequency analysis.
- ☐ **Frequency Response Function (IMD)** Graphing of Acceleration due to support motion vs. Frequency for a selected node (all frequencies) using a linear scale.
- ☐ **Power Spectral Density Response (IMD)** Displaying the PSD stress response for a node on the top plate due to a PSD acceleration input at the supports.

### Selecting a Results Loadcase

Analysis loadcase results are present in the  Treeview, and for eigenvalue analysis the loadcase results the first Eigenvalue is set active by default.

### Displaying the 1st Mode Shape

- Turn off all layers in the  Treeview.
- Ensure **Eigenvalue 1** is set active in the  Treeview.
- If not already visible, turn enable the **Deformed mesh layer**: With no features selected click the right-hand mouse button in a blank part of the graphics window and select **Deformed mesh** to add the deformed mesh layer to the  Treeview.
- In the panel at the bottom of the  Treeview, press the **Deformations...** button  and change the specified magnitude to **20** and click the **OK** button to display the deformed mesh for eigen mode 1.



Use the Dynamic Rotation button to rotate the model to a similar view to that shown.



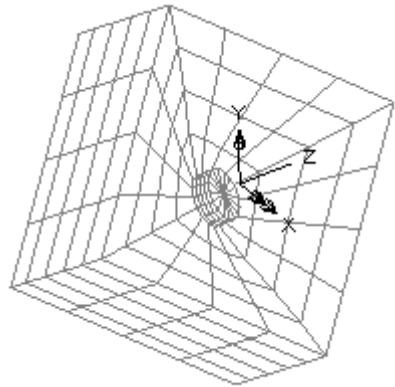
Return to normal cursor mode.




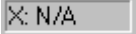
**Note.** For more complex mode shapes it may be beneficial to animate the mode shape so it can be seen in more detail.



**Note.** In some cases the mode shape may appear inverted. This is because the Eigenmode represents the shape of the vibrating body and the displacements may be multiplied by -1.



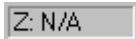
### Stresses as Filled Contours

- Turn on the **Mesh** and turn off the **Deformed mesh** layers in the  Treeview
- Transform the view to visualise the model from the X direction by clicking on the  in the status bar at the bottom of the Modeller window.
- Select an area enclosing the top plate of the housing as shown.



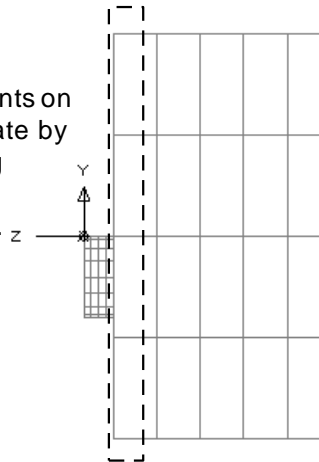
Create a new group consisting of the elements forming the top plate.


- Enter **Top Plate** as the group name and click **OK**.





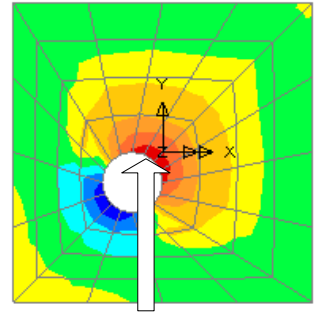
Rotate the model to view along the Z axis by clicking in the status bar at the bottom of the Modeller window.

Select elements on top plate by boxing region shown.



- Make the top plate the only visible part of the model by clicking with the right-hand mouse button on **Top Plate** in the  Treeview and selecting **Set as Only Visible**

- Ensure the results for **Eigenvalue 1** are set active in the  Treeview.
- With no features selected right-click on a blank part of the screen and select **Contours**
- Select the **Stress (top) – Thin Shell** entity and maximum absolute stress component **Sabs**
- Click the **OK** button to display contours of maximum absolute stress.
- To display the mesh on top of the contours select **Mesh** in the  Treeview and drag and drop it on top of the **Contours** layer name.



Highest stressed node.

The node where the maximum absolute stress occurs is noted at the bottom of the contour key.

- **Make a note of this node number** as it will be used later in the example.





**Note.** The contours are relative stress contours which have no quantitative meaning.

## Interactive Modal Dynamics



**Note.** A FRF (frequency response function) is a transfer function in the frequency domain. In general terms, the transfer function indicates how much of the input excitation is transferred to the selected output point.

Compute the modal acceleration response in the Z direction at the node in the centre of the bottom surface due to a harmonically varying acceleration applied to the structural supports.

- Delete the **Contours** layer from the  Treeview.
- Redisplay the full model by selecting the **casing.mdl** in the  Treeview with the right-hand mouse button and selecting **Set as Only Visible**. Select **Yes** to act on sub-group as well.

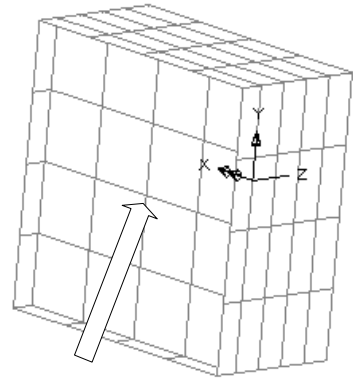


Rotate the model as shown



Return to normal cursor mode.

- Select the node at the centre of the bottom plate.



Select this node at centre of back plate

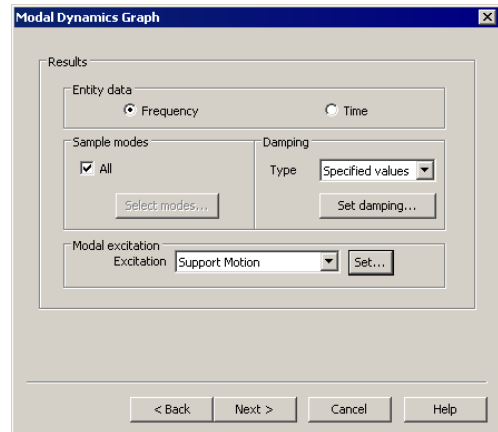
Utilities

Graph Wizard...

- Select the **Modal Expansion** option and click the **Next** button.
- On the Modal Dynamics Graph dialog choose the **Frequency** entity.
- Under the damping section choose **Specified values** from the drop down list.
- Select the **Set damping** button and on the Damping Values dialog set the viscous damping to **0** and the structural damping to **2.8**

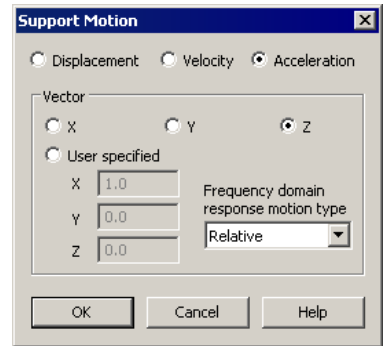


**Note.** If damping values are specified for the first mode only, the other modes will take the same values.

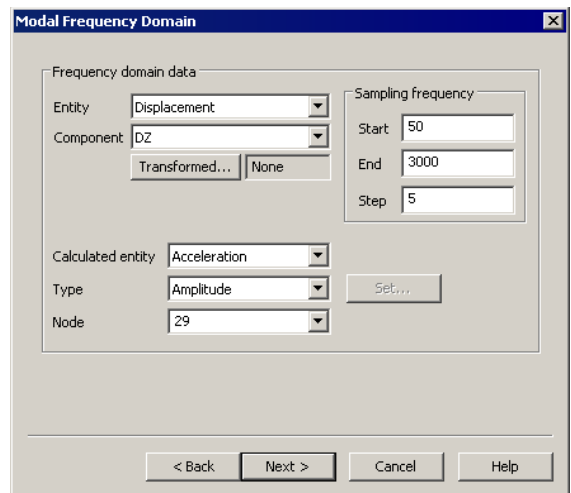


- Click the **OK** button to return to the Modal Dynamic Graph dialog.
- On the main dialog, from the Modal Excitation drop down list select **Support Motion**

- Click on the Modal excitation **Set** button and on the Support Motion dialog specify **Acceleration** in the **Z** direction.
- Click the **OK** button to return to the Model Dynamics Graph dialog and click **Next**



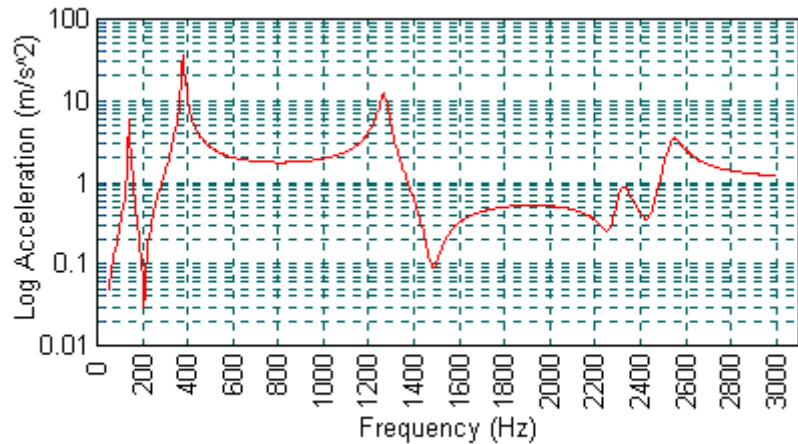
- On the Modal Frequency Domain dialog select the **Displacement** entity in the **DZ** direction.
- Set the calculated entity as **Acceleration** of Type **Amplitude**. The selected node number at the centre of the back plate will be displayed in the drop down node list.
- Set the sampling frequencies start, end and step entries to **50, 3000** and **5** respectively.



- Click **Next**
- On the Display Graph dialog ensure that the **Show symbols** option is not selected.
- In the Y Scale section select the **Use logarithmic scale** option.
- Click on the **Finish** button and a graph of log acceleration against frequency will be plotted.
- To specify axis labels and title right click on the graph and select **Edit graph properties**
- On the **General** tab set the graph title to **FRF in Z Direction at Centre of Back Plate (2.8% Structural Damping)**
- Click on the **X Axis Style** tab and set the axis name to **Frequency (Hz)**
- Click on the **Y Axis Style** tab and axis name to **Log Acceleration (m/s^2)**

- Click **OK** to update the graph display.

FRF in Z Direction at Centre of Back Plate (2.8% Structural Damping)



AMPLITUDE / FREQUENCY —



Close the graph window

### Power Spectral Density definition

A Power Spectral Density (PSD) force input defines the frequency content of a random loading, such as turbulent pressure acting on an aircraft component. A PSD analysis is, therefore, useful when broadband random dynamic forces excite structural vibrations.

### Graph of PSD Stress Response

- Click the left-hand mouse button in a blank part of the graphics window to clear the current selection.
- Click the right-hand mouse button in a blank part of the graphics window and select the **Advanced Selection** option.
- With the **Type and name** option selected, choose **Node** from the drop down list and **enter the node number** at which the maximum principal stress occurs (as noted down at an earlier stage in this example).
- Click **OK** to add the node to the selection.
- To compute a PSD stress response select the **Modal expansion** option and click **Next**



- On the Modal Dynamics Graph dialog choose the **Frequency** entity.
- Under the damping section choose **Specified values** from the drop down list. Select the **Set damping** button and set the viscous damping to **0** and the structural damping to **2.8** and click **OK**
- From the Excitation drop down list select **Support Motion**
- Click on the **Set** button and under support motion specify **Acceleration** in the **Z** direction and click the **OK** button to return to the Model Dynamics Graph dialog.
- Click **Next** to move to the next dialog.

- On the Modal Frequency Domain dialog select entity **Stress (top) – Thin Shell** with component **S1**
- Select **PSD Response** from the Type drop down list and click the **PSD set** button.

The image shows the 'Modal Frequency Domain' dialog box. It contains the following fields and buttons:

- Frequency domain data** section:
  - Entity: Stress (top) - Thin Shell
  - Component: S1
  - Buttons: Transformed..., None
- Sampling frequency** section:
  - Start: 50
  - End: 3000
  - Step: 5
- Calculated entity**: Acceleration
- Type**: PSD Response
- Node**: 140
- Buttons**: Set..., < Back, Next >, Cancel, Help

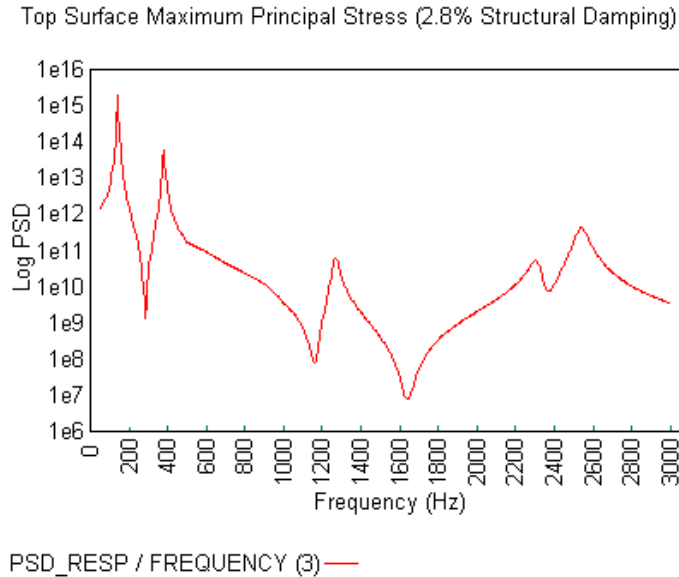
- On the RMS response for Power Spectral density dialog ensure the **Linear/Linear** scale option is selected and define a frequency PSD dataset using the frequency and amplitude values shown in the table on the right (Use the **Tab** key to create a new line in the table).
- Label the dataset **Frequency PSD** and click **OK**
- The selected node number will be displayed in the drop down node list.
- Click the **Next** button.

Linear Frequency	Linear Amplitude
15	8
100	14
125	30
500	31
600	61
900	108
1000	79
3000	47

- On the **Display Graph** dialog deselect the **Show grid** option. Deselect the **Show symbols** option and select the **Use Logarithmic scale** option for the Y Scale.

- Click the **Finish** button to plot a graph of log PSD against frequency.

The RMS value of the input PSD and response PSD are written to the message window when the graph is plotted.



To specify axis labels and title right-click on the graph and select **Edit graph properties**

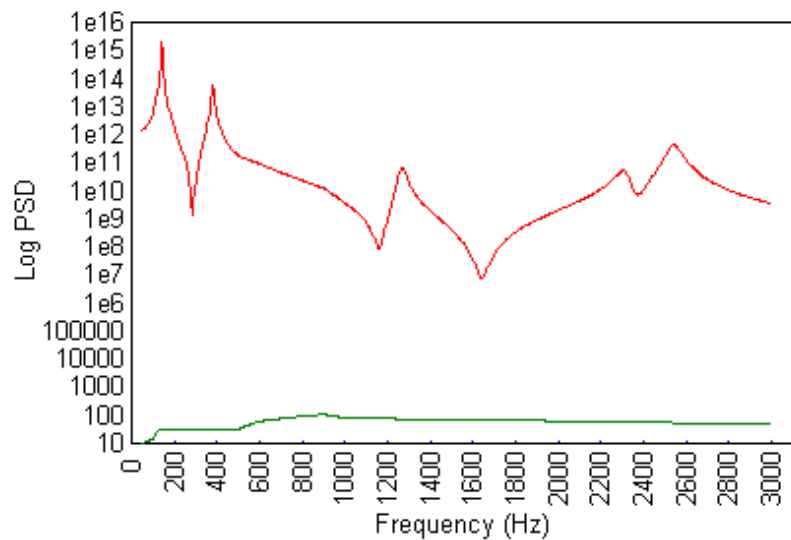
- On the **General** tab set the graph title to **Top Surface Maximum Principal Stress (2.8% Structural Damping)**
- Click on the **X Axis Style** tab and set the axis name to **Frequency (Hz)**
- Click on the **Y Axis Style** tab and axis name to **Log PSD**
- Click **OK** to update the graph.
- Close the graph. It will be subsequently be used to plot the PSD input values.

### PSD Input

The PSD input data used in the acceleration of the supports can be plotted on the same graph by following the above procedure until the Modal Frequency Domain dialog is reached.

- With the node at which maximum principal stress occurs still selected, follow the steps in creating the PSD Response graph starting with the **Utilities > Graph Wizard** menu option but, when the Model Frequency Domain dialog is displayed, instead of selecting PSD Response select **PSD Input** in the **Type** drop down list.
- When the Display Graph dialog is reached, from the **Include existing graphs** list include **Graph 2** and ensure that the **Show symbols** is not selected
- Click **Finish** to update the graph.

Top Surface Maximum Principal Stress (2.8% Structural Damping)



PSD\_RESP / FREQUENCY(3) —

PSD\_INPUT / FREQUENCY(5) —

This completes the example.

