



## Modal Analysis of a Tuning Fork

---

In a natural frequency analysis the following assumptions are made:

- There is no applied load and vibration is due to the mass and stiffness of the structure alone.
- There is no damping.
- Vibration assuming sinusoidal displacements of the form  $a = A\sin(\omega t)$ .

The numerical solution produces a series of eigen pairs. The eigenvalues which indicate frequencies at which the vibration would naturally occur are output. The eigenvectors give the associated mode shape of vibration. It is important to note that the solved eigenvectors (and hence the resulting mode shape displacements) are normalised and hence may be arbitrarily scaled. Although displacement, strain and stress information may be plotted, these quantities are therefore only relative and cannot be used directly in the design process. It is common for the magnitudes of these quantities to be investigated by running subsequent modal analyses such as forced (harmonic) or spectral (seismic) response. In this case the resulting eigenvalues will be manipulated interactively during results processing using the Interactive Modal Dynamics (IMD) facility.

The default method for Eigenvalue Extraction, used here, is Subspace Iteration. This method has the following characteristics:

- All of the degrees of freedom in the model are used in the solution.
- An initial estimated solution is improved via subsequent iterations.

These characteristics make the method very accurate and robust.

### Keywords

**2D, Inplane, Plane Stress, Intersecting Features, Splitting Features, Surfaces by Joining, Mirroring, Natural Frequency, Eigenvalue, Eigenvalue Control, Interactive Modal Dynamics (IMD), Mode Shapes, Animation, Frequency Response Graphs.**

### Associated Files



- fork\_modelling.vbs** carries out the modelling of the fork.

## 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.

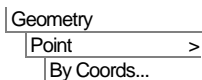
## Creating a new model

- Enter the file name as **fork**
- Use the default User-defined working folder.
- Ensure an analysis type of **Structural** is set.
- Set the analysis category as **2D Inplane**
- Select a **2D Plane stress** Startup template.
- Set the model units to **N,mm,t,s,C**
- Ensure the timescale units are **Seconds**
- Ensure the Layout grid is set as **None**
- Enter the title as **Tuning Fork - Frequency Analysis** and click the **OK** button.

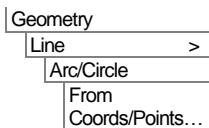


**Note.** Save the model regularly as the example progresses. Use the Undo button to correct any mistakes made since the last save was done.

## Feature Geometry



Define a Point at a coordinate of **(1.5, 0)** and click **OK**



- Enter coordinates of **(0, 0)**, **(1.5, 0)**, and **(1.5, 1.5)** to define the arc at the far left end of the fork. Select the coordinate **(1.5, 0)** as the **Centre** of the arc and click **OK**

## Modal Analysis of a Tuning Fork

- Drag a box to select the 2 Points shown.



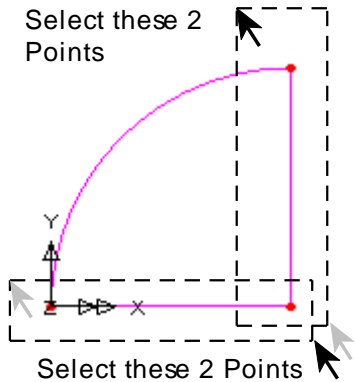
Use the New Line button to create the Line.

- Select the other 2 Points shown.



Use the New Line button to create the Line.

- Select the Arc, hold down the **Shift** key and then select the 2 straight Lines by individually clicking on each in turn.



Using the New Surface button, create a Surface from the Lines selected.

- Click on a blank part of the view window to clear the selection.

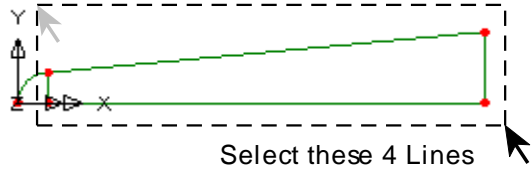


Define Points at coordinates of  $(23,0)$  and  $(23,3.5)$  and click **OK**

- Select pairs of Points and create new Lines between each pair.
- Drag a box around these Lines.



Create a Surface to form the next part of the base of the fork.



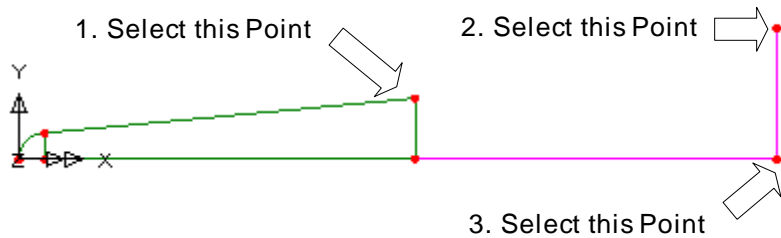
Click on a blank part of the view window to clear the selection and then Define Points at coordinates of  $(44,0)$  and  $(44,7.5)$  which form the ends of the radius handle of the fork.



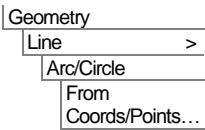
- Select pairs of Points and use the New Line button to create only the straight Lines shown.

Geometry  
Point >  
By Coords...

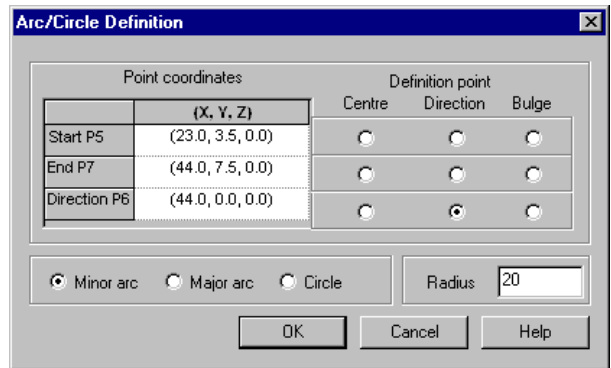
Geometry  
Point >  
By Coords...



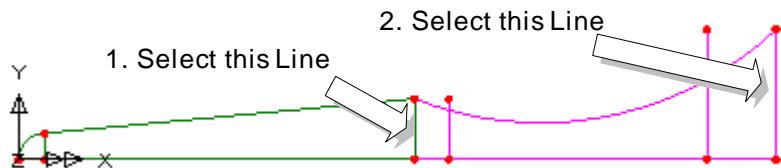
- To define the arc. Select the arc start and end Points, and the Point at the bottom right of the model.



- Specify that coordinate **(44,0)** is a **Direction Point**.
- Ensure that **Minor arc** is selected
- Enter an arc radius of **20**
- Click **OK** to draw the arc.



The Lines which mark the extent of the support conditions on the handle (that is the area over which the fork is assumed to be held) will now be defined. This will be done by copying existing Lines.



- Select the first Line shown.



Copy the Line once through a distance of **2** in the **X** direction and click **OK**

- Select the second Line shown

## Modal Analysis of a Tuning Fork

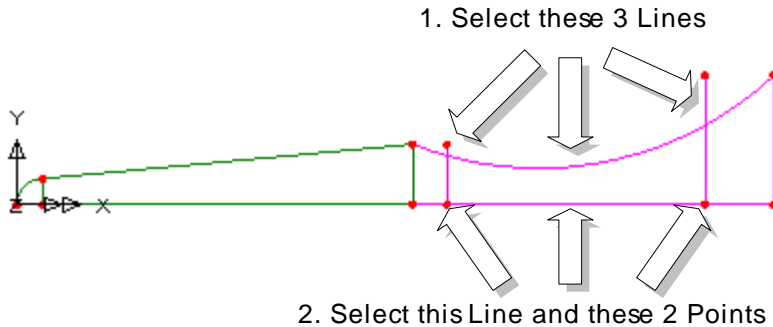
---



Copy the Line once through a distance of **-4** in the **X** direction and click **OK**

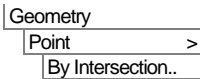


**Note.** Right-clicking the view window with geometry selected provides a menu with short-cuts to common commands, such as **Copy** and **Sweep**.



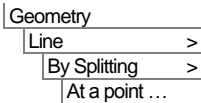
Points are now created at the intersections of the arc with the 2 new copied Lines.

- Select the 3 upper Lines shown using the **Shift** key to add to the initial selection.
- Create **Exact intersections only** and ensure the **Split intersecting lines** and **Delete geometry on splitting** options are selected. Click the **OK** button to create the new Points at the Line intersections and also split the selected Lines.

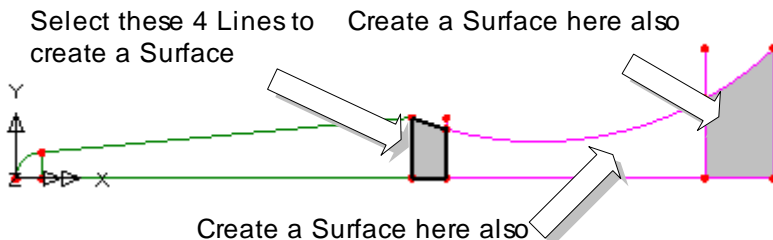


In order to create 3 separate Surfaces the Line along the axis of symmetry has to be split into 3 new Lines.

- Select the Line and 2 Points shown in the previous diagram.
- Ensure that **Delete features on splitting** is selected and click **OK** to create 3 new Lines.



Three Surfaces are now defined using the Lines previously created.



- Select the 4 Lines (remembering to hold the **Shift** key down after the first line is selected to add to the selection) to form the boundary of the Surface shown.

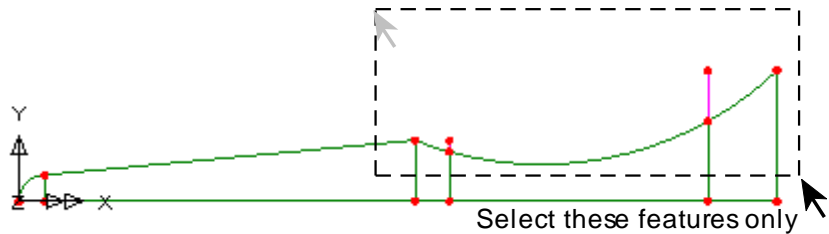
Geometry  
Surface >  
By Lines...



Use the New Surface button to create a Surface.

- Repeat, selecting each set of 4 Lines to define the remaining 2 Surfaces

Lines and Points that are left over from the previous operations can be deleted to tidy-up the model.



- Drag a box around the features shown, ensuring that no Surfaces are selected.

Edit  
Delete



To delete the unwanted Lines.

- Select **Yes** to delete Lines.
- Select **Yes** to delete Points.



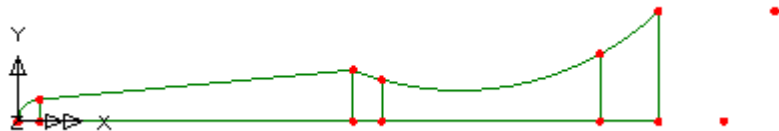
**Note.** Only those Point and Line features that are not used to define any Surfaces will be deleted. Pressing **Yes to All** will delete all lower order features without prompting.

- Click on a blank part of the view window to clear the selection.

Geometry  
Point >  
By Coords...

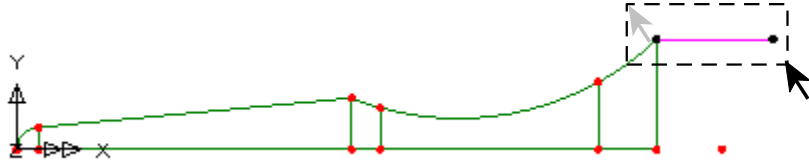


Enter coordinates of **(48.5,0)** and **(52,7.5)** and click **OK** to define the points.



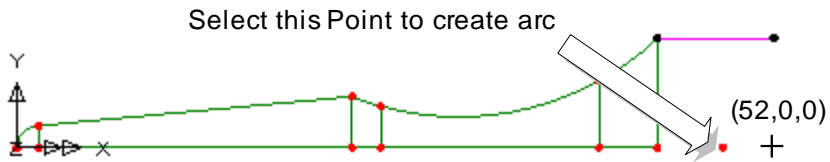
## Modal Analysis of a Tuning Fork

A horizontal Line will now be created.




- Select the two Points and create the horizontal straight Line as shown above.

Now define the arc at the start of the arm section of the fork.



- Select the single Point.

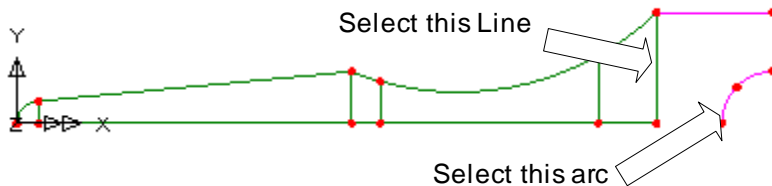
 Select the **Rotate** option to sweep the point through an angle of **-90** degrees about the **Z-axis** around an origin Point of **(52, 0, 0)**


- Click **OK** to sweep the Point and create an Arc.

To avoid the creation of a 5-sided Surface at the start of the arm section the arc will be split into two.

- Select the newly created arc.
- Enter **2** for the number of divisions. Ensure **Delete original lines after splitting** is selected and click **OK** to replace the arc with two new arcs.

To complete the junction section of the fork two new Surfaces will be defined.

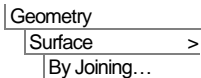


 Zoom in to the working area.



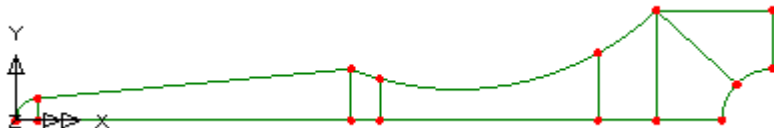


Revert to standard cursor mode and select one of the Arcs. Hold down the **Shift** key and select the Line on the opposite side as shown above.



Create the first Surface.

- Repeat the previous procedure to create the second Surface as shown below.



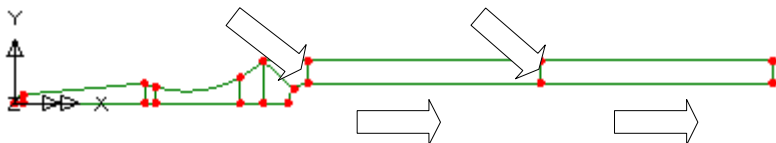
The remainder of the fork will be created by sweeping the vertical Line at the right-hand end of the model through a distance in the X direction.



Resize the model so all features are in view.

1. Select this vertical Line to create Surface

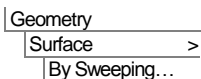
2. Select this vertical Line to create Surface



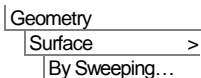
- Select the vertical Line shown.



Enter a distance of **41.25** in the **X** direction.



- Click **OK** to create the Surface.
- Select the new Line at the right-hand end of the model.



Use the Sweep Feature button to sweep the Line through a distance of **41.25** in the **X** direction to create the Surface.

The geometry of the half-model of the tuning fork is now complete. Attribute data such as mesh, loading and supports will now be added to the half-model before copying and mirroring to create a full model for analysis.


### Meshing

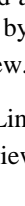
A frequency analysis can use a relatively coarse mesh, since stress output is not required from the analysis. With this in mind, a series of Line meshes will be used to control the density of the Surface mesh.

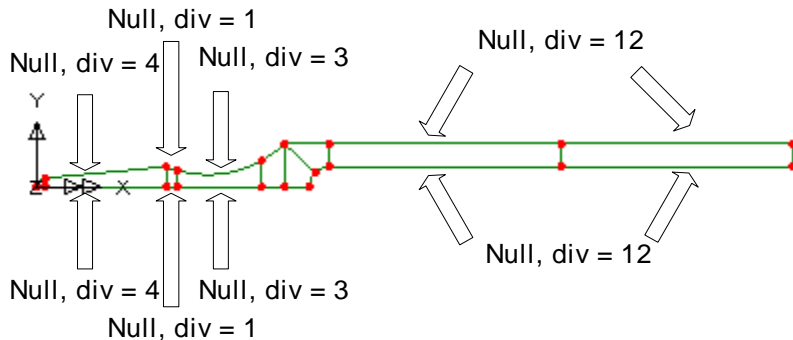
#### Line meshes

As the majority of the Lines require only 2 divisions the default number of mesh divisions will be reset.


- Select the **Meshing** tab, set the default number of divisions to **2** and click **OK** to return to the view window. Any Lines to which a line mesh dataset is not assigned will adopt this as a default.

Selected Lines will be assigned a number of Line divisions. LUSAS provides a limited number of Line Mesh datasets by default in the 2D Plane stress Startup template. These can be found in the  Treeview.

- With the relevant sets of Lines selected, drag and drop the appropriate Line mesh datasets from the  Treeview onto the selected features. Use the Zoom in button as necessary.



#### Surface mesh

- Define a Surface mesh using **Plane Stress, Quadrilateral, Quadratic** elements. Name the dataset **Plane stress (QPM8)** and click **OK**
- Using the **Ctrl** and **A** keys together Select the half model of the fork.
- Drag and drop the Surface mesh dataset **Plane stress (QPM8)** from the  Treeview onto the selected features.

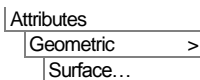



**Note.** Since all of the Surfaces are 4 (or 3) sided, a regular mesh pattern is created.

At any time the mesh (and other layers) displayed in the view window may be hidden or redisplayed. With no features selected click the right-hand mouse button in a blank part of the view window and select **Mesh**. If a mesh was previously displayed it will be hidden. If previously hidden, it will be displayed. This facility can be used to simplify the display when it is required.

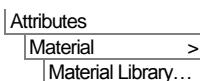
- Remove the **Mesh** from the display as described in the previous note.


## Geometric Properties




- Specify a thickness of **4** and leave the eccentricity blank.
- Enter the dataset name as **Thickness** and click **OK**.
- With the whole model selected (Using the **Ctrl** and **A** keys together) drag and drop the geometry dataset **Thickness** from the  Treeview onto the selected features.

## Material Properties



- Select material **Steel – Structural**, Region **None**, and Grade **Stainless steel**, and click **OK**
- With the whole model selected, drag and drop the material dataset **Iso3 (Stainless Steel | Steel – Structural)** from the  Treeview onto the selected features.
- Ensure the **Assign to surfaces** option is selected and click **OK**

## Support Conditions


When using the 2D Plane stress Startup template, LUSAS provides the more common types of support by default. These can be seen in the  Treeview. To model the holding of the tuning fork the support dataset **Fixed in XY** will be used.

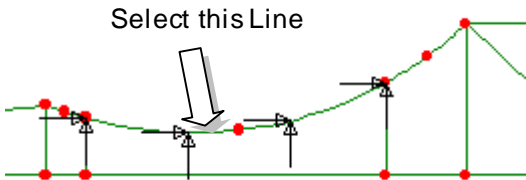



**Note.** 2D plane stress elements only have X and Y degrees of freedom therefore a restraint in the Z direction is not necessary.

## Modal Analysis of a Tuning Fork

---

- With the arc shown in the diagram selected, drag and drop the support dataset **Fixed in XY** from the  Treeview onto the selected Line.



- Ensure the **Assign to lines** and **All analysis loadcases** options are selected and click **OK**
- If supports are not visualised when expected they can be visualised for each support condition by right-clicking on the support name in the supports section of the  Treeview and selecting **Visualise Assignments**.



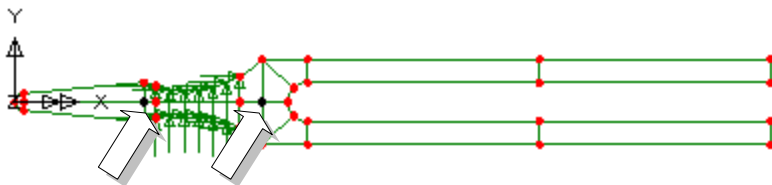
**Note.** In practice this support would not be a rigid support since it is hand-held but this should not significantly affect the frequencies obtained.

### Loading

No loading is required for a natural frequency analysis.



**Note.** The fork is symmetrical about its centre-line, therefore only half of the structure has so far been created. For static structural analysis it would be common to apply a symmetry support condition to the centreline, so that only half of the structure need be analysed. However, in a frequency analysis the use of symmetry in this way is less common since this will force the analysis to only solve for the symmetric modes of vibration ignoring any anti-symmetric modes. Generally, both symmetric and unsymmetric vibration modes are of significance, therefore the model will be mirrored to form the complete model.



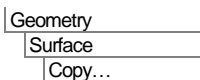
Select these 2 Points to define mirror plane

- Select the 2 Points on the centreline of the tuning fork as shown.

The Points are stored in memory.

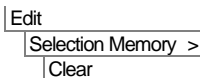
- Select the whole model using the **Ctrl** and **A** keys together.

Edit  
Selection Memory >  
Set



Select **Mirror - Points 5 7** (note point numbers may differ between models) from the drop-down list and click the **Use** button on the dialog to use the Mirror Points stored in memory.

- Click **OK** to create the full model.



This will remove the Points from the selection memory.



**Note.** The model is 2 Dimensional, therefore only 2 Points are required to define the mirror plane (the Z direction is assumed as the screen Z plane). In addition, the selection of Points as far away from each other as possible will ensure a good specification of the required mirror plane.

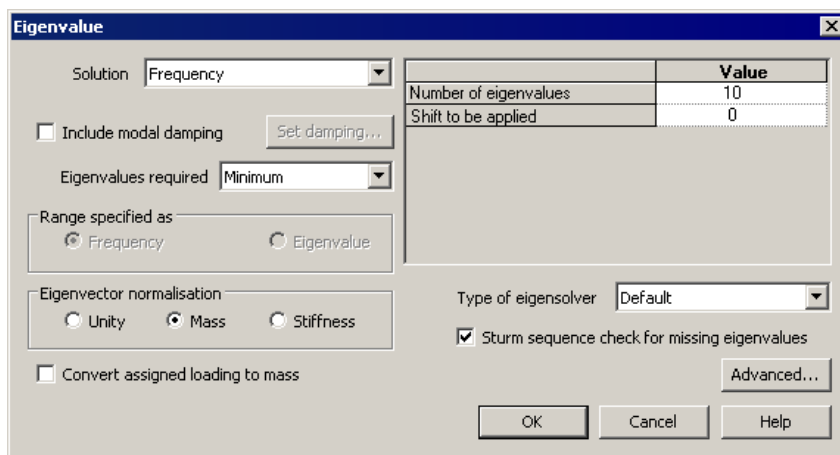
The model is now complete. All attributes assigned to the original half-model features, including the support and mesh assignments, will have been identically reproduced on the duplicated features.

## Eigenvalue Analysis Control

The eigenvalue analysis control parameters are applied as properties 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 a specified number of the minimum eigenvalues.

- Set the **Number of eigenvalues** required to **10**
- Set the **Shift to be applied** to **0**
- Leave the type of eigensolver as Default



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

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

### Saving the model



Save the model file.

File  
Save


## 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...

LUSAS eigenvalue results will be added to  Treeview.

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



- fork.out** this output file contains details of model data, assigned attributes and selected statistics of the analysis.
- fork.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...

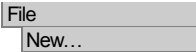
If the analysis fails, information relating to the nature of the error encountered can be written to an output file in addition to the text output window. Any errors listed in the text output window should be corrected in LUSAS Modeller before saving the model and re-running the analysis.

## 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.

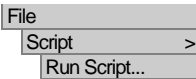


- fork\_modelling.vbs** carries out the modelling of the example.



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 **fork**



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




Rerun the analysis to generate the results

## 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.
- Mode Animation Sequence** Animation of selected mode shapes.
- Printing Eigenvalue Results** Printing results to a text window.
- Modal Dynamics (IMD)** Graphing of Displacement vs. Frequency for a selected node (all frequencies) using a linear scale.

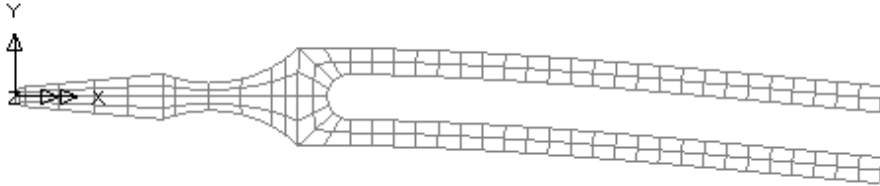
## Selecting a Results Loadcase

Eigenvalue loadcase results are present in the  Treeview, and for an eigenvalue analysis, the first eigenvalue (Eigenvalue 1) is set to be active by default.


## Plotting Mode Shapes

If present, turn off the **Geometry**, **Attributes** and **Mesh** layers in the  Treeview.

- If not already on, turn on the **Deformed mesh** layer. With no features selected click the right-hand mouse button in a blank part of the View window and select **Deformed mesh** to add the deformed mesh layer to the  Treeview.
- Click the **OK** button and the deformed mesh plot for Eigenvalue 1 will be displayed.




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

To view other mode shapes, in the  Treeview right-click on the Eigenvalue required and select the **Set Active** option.

### Creating Animations of Mode Shapes

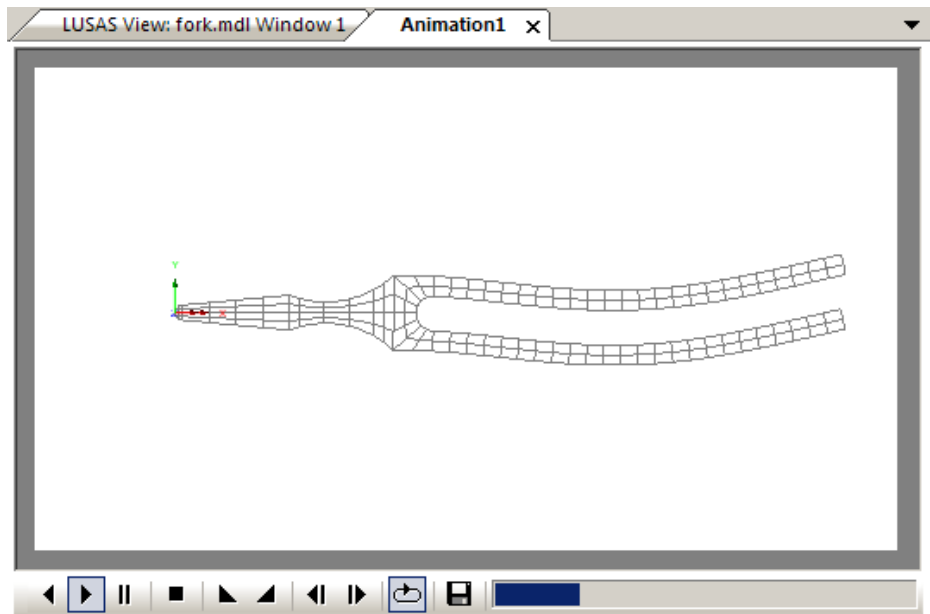
This section will create an animation of the third mode shape.

- In the  Treeview right-click on **Eigenvalue 3** and select the **Set Active** option.
- The deformed shape for Eigenvalue 3 will be displayed.
- Select the **Active loadcase** button and select the **Next** button.
- Use a **Sine** deformation with **8** frames. Set the range to **-1 to 1**. Set the deformation magnitude of **6** mm.
- Click **Finish** and LUSAS will create the animation sequence and display the animation in a new window.

Tools

Animation Wizard...






- The buttons at the bottom of the window may be used to slow-down, speed-up, pause, step through frame by frame, or stop the animation.

## Saving Animations

Animations may be saved for replay in other Windows applications.

- Ensure the animation window is the active window.
- Enter **fork\_mode3** for the animation file name. An **.avi** file extension is automatically appended to the file name when the file is saved. Click **OK**.

 Close the animation window without saving changes.

## Printing Eigenvalue Results

Eigenvalue results for the model can be displayed in a Print Results window.

- Select the Results type **Eigenvalues**
- Select to display results for **All** loadcases.
- Select to display the results type **Eigenvalues (Frequency)** and click the **OK** button to output the eigenvalues to the print results window. Your values should be similar to these:

File  
Save As AVI...

Utilities  
Print Results  
Wizard...

## Modal Analysis of a Tuning Fork

---

MODE	EIGENVALUE	FREQUENCY	PERIOD	ERROR NORM
1	7.25647E6	428.729	2.33248E-3	1.45128E-9
2	7.40305E6	433.037	2.30927E-3	1.35483E-9
3	272.988E6	2.62961E3	0.380284E-3	52.9392E-12
4	283.643E6	2.68044E3	0.373073E-3	48.95E-12
5	2.00119E9	7.11974E3	0.140455E-3	33.5345E-12
6	2.14224E9	7.36638E3	0.135752E-3	7.36666E-12
7	3.75737E9	9.75579E3	0.102503E-3	11.1482E-12
8	5.72457E9	12.0418E3	83.044E-6	0.356697E-9
9	6.68763E9	13.0154E3	76.8323E-6	3.3329E-9
10	8.10989E9	14.3327E3	69.7706E-6	1.28167E-6

**Note.** The error norms may vary as they are dependent on the Eigensolver used for the solution.

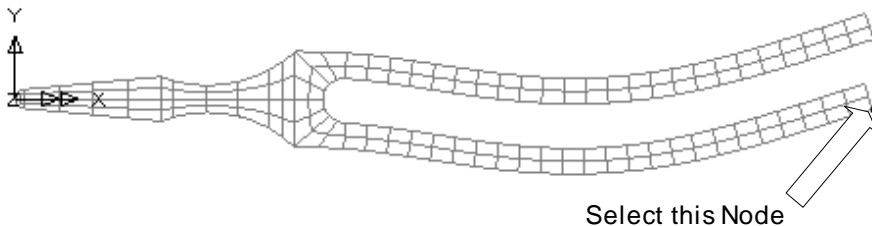


Close the results window.

## Interactive Modal Dynamics

The vertical displacement response of a selected node for a unit vertical force is to be plotted against the sampling frequency over the entire solved frequency range (0-15000 Hz) on a linear scale.

- Select the node at the end of the arm shown.



## Using the Graph Wizard

The graph wizard provides a step-by-step means of selecting results to be plotted on the X and Y axes of a graph. The X axis data is always defined first.

- Select the **Modal Expansion** option and click the **Next** button.
- The **Frequency** response for **All** modes is to be calculated for the X axis data. Ensure the Damping Type is set to **None**

Utilities  
Graph Wizard...

- On the Modal Excitation section of the dialog ensure that **Point** excitation is selected and click the adjacent **Set** button.
- Select the **Node number** previously selected from the drop-down list. Select component **Y** and click the **OK** button to return to the main dialog.
- Click the **Next** button.

The excitation loading has been defined. The response has now to be defined.

- On the Modal Frequency Domain dialog select **Displacement** results of displacement **DY**. Enter **Sampling frequency** values of **Start** as **0**, **End** as **15000** and **Step** as **100**
- Click the **Next** button.

The response has now been defined. Frequency (X) and Amplitude (Y) axis datasets are now generated to graph the displacement frequency response at the selected node.

Additional information for the graph can now be added.

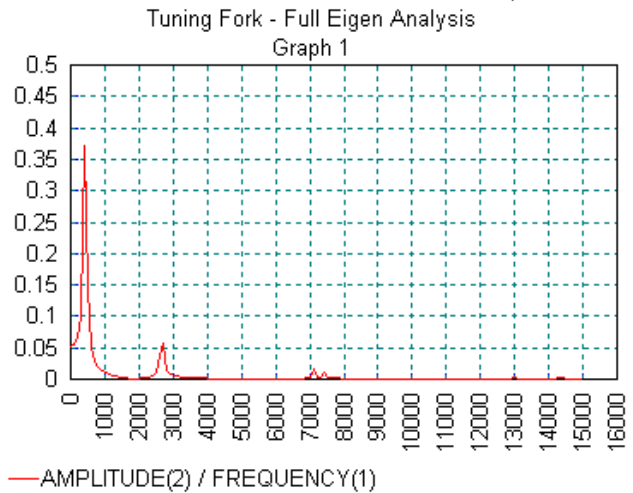
- Leave all graph title information blank.
- Deselect the **Show symbols** button.
- Click the **Finish** button to end.



**Note.** If no graph or axis titles are entered default names will be used. The graph attributes may be edited by right clicking on the graph and selecting **Edit Graph Properties**

LUSAS will create the graph in a new window and display the values used in an adjacent table. To see the graph at the best resolution enlarge the window to a full size view.

- Position the cursor over the first peak on the graph to see the values at the peak of X=400 (Frequency) and Y=0.371 (Amplitude DY)



**Note.** Using a large frequency range with a 100Hz interval means the actual mode 1 frequency (of 428.729Hz) has been missed. When plotting graphs of this nature it is therefore better to use a smaller frequency range and interval in order to isolate the peak results.

### To produce a more detailed graph

- Close the current graph window.
- Ensure the same node is selected on the fork.
- With the **Modal Expansion** option selected click the **Next** button.
- With all options that were set previously selected click the **Next** button.
- Select **Displacement** results of component **DY**. Enter sampling frequency values of **Start** as **0**, **End** as **800** and **Step** as **1**
- Click the **Next** button.
- Leave all graph title information blank.
- Ensure the **Show symbols** option is not selected
- Click the **Finish** button to end.

Utilities

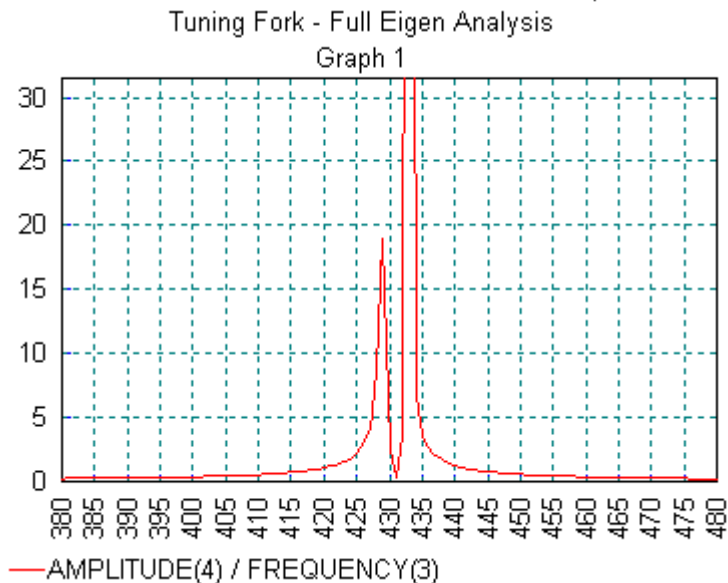
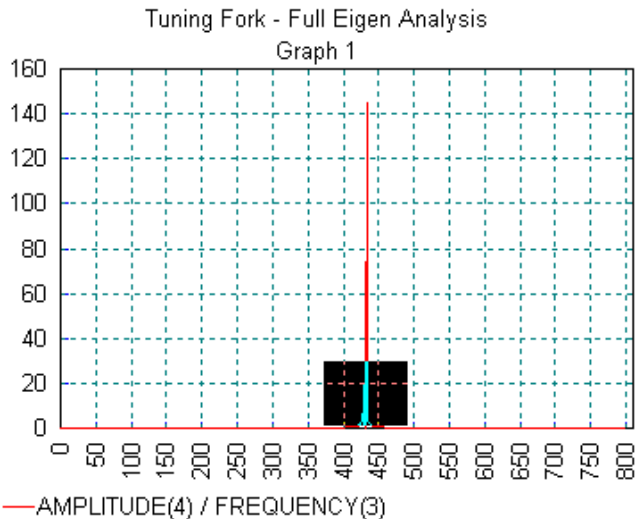
Graph Wizard...

LUSAS will create a graph in a new window and display the values used in an adjacent table. To see the graph at the best resolution enlarge the window to a full size view.

- Position the cursor over the main peak on the graph to see the values at the peak of X=433 (Frequency) and Y=145.153 (Amplitude DY).

This is a theoretical amplitude and is related to no damping being present.

- Click and drag the cursor over the region as shown and a new graph the peak for the first mode shape will be seen



This displays a better representation of the displacement/ frequency response in the vicinity of the first two mode shapes.

## Modal Analysis of a Tuning Fork

---

- Position the cursor over the first peak on the graph to see the values at the peak of X=429 (Frequency) and Y=18.8863 (Amplitude DY)

These values compare more favourably with those obtained from using the Print Results Wizard for mode 1 earlier in the example.

- Right-click and select **Unzoom** to return to the previous graph view.



Close all graph windows to leave the model window active.

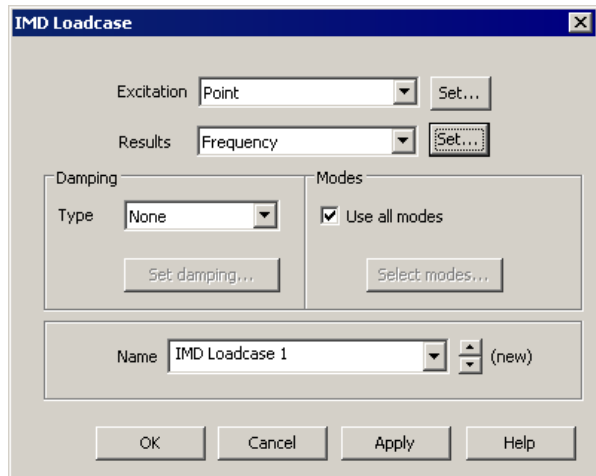
- Ensure the node at the end of the fork is still selected.

## Plotting Dynamic Response for a Particular Frequency

In some analyses, dynamic responses are required at a specified frequency. In these cases, an Interactive Modal Dynamics (IMD) load case is defined to allow the frequency and type of excitation to be specified. In this example, deformed shapes and peak displacements are to be plotted for excitation frequencies of 750Hz and 1500Hz.

On the IMD Loadcase properties dialog:

- Select **Point** excitation and click the adjacent **Set** button.
- On the Point Excitation dialog, for a Force, select the **Node number** previously selected from the drop-down list. Select Direction component **Y** and click the **OK** button to return to the IMD Loadcase main dialog.
- Select **Frequency** results and click the adjacent **Set** button.



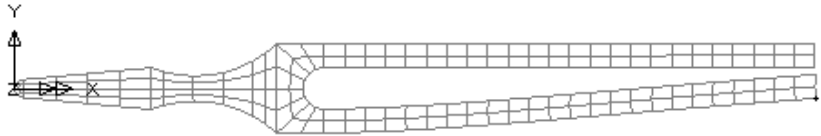
- On the Frequency Results dialog, for Type Amplitude, enter a **Frequency** of **750**
- Click the **OK** button to return to the main IMD Loadcase dialog.
- Ensure the name is set to be **IMD Loadcase 1**
- Click the **OK** button to finish defining the IMD loadcase.

Analyses

IMD Loadcase...


## Selecting the IMD Results Loadcase

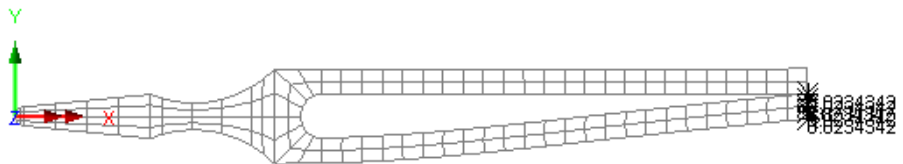
- In the  Treeview right-click on **IMD Loadcase 1** and select the **Set Active** option.




The deformed mesh plot is updated to show the deformed mesh at the specified frequency.

## Marking Peak Values

- With no features selected click the right-hand mouse button in a blank part of the View window and select the **Values** option to add the **Values** layer to the  Treeview.
- The values properties will be displayed.
- Select **Displacement** results of displacement in the Y direction **DY**. Select the **Values Display** tab and select the top **0%** of **Maxima** values.
- Click the **OK** button to display the top value of displacement for this IMD results loadcase frequency.



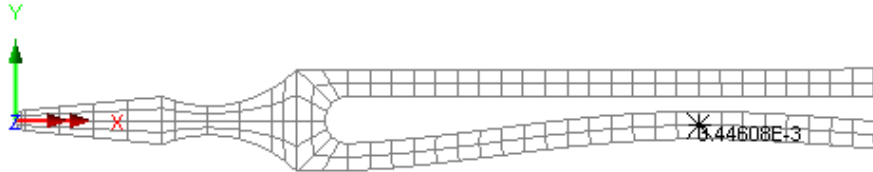
## Changing the results frequency

- In the  Treeview double-click the **IMD 1** dataset name.
- Click the **Set** button adjacent to the **Frequency** option.
- Change the frequency to **1500** and click the **OK** button to return to the main dialog.
- Click the **OK** button to finish defining the IMD loadcase.

## Modal Analysis of a Tuning Fork

---

The deformed mesh plot will be updated to show the revised mode shape and corresponding values for the specified frequency.



**Note.** Since the eigenvalue is independent of sign the deformed shape may appear inverted from that shown.

This completes the example.