Modal Analysis of a Tuning Fork

For LUSAS version:	19.0
For software product(s):	All (except LT versions)
With product option(s):	None.
Note: The example exceeds the limits of the LUSAS Teaching and Training Version.	

Description

This example demonstrates a natural frequency analysis of a stainless steel tuning fork. The dimensions are those of an A tuning fork which vibrates at 440 Hz. The overall dimensions of the fork are as shown.
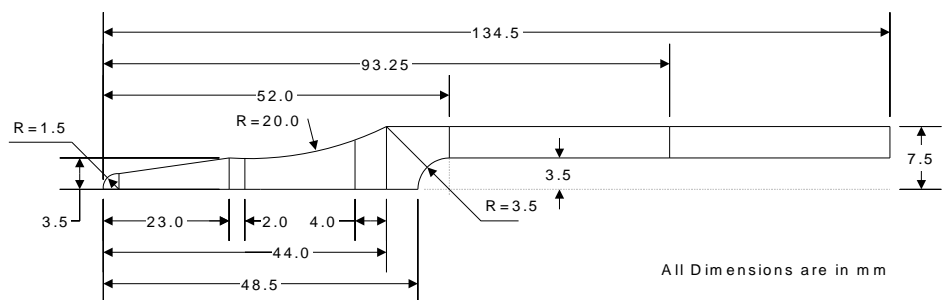
Units of N, mm, t, s, C are used throughout.

❑ For natural frequency analysis consistent units must be adopted.
❑ A model for an eigenvalue analysis is created in an identical way to that required for a static analysis, using features and attributes, but a relatively coarse mesh is used since stress output is not required.
❑ No loading is applied to the structure.
❑ Eigenvalue control data is included to specify details of the analysis required.

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 = ASin(ωt).

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**

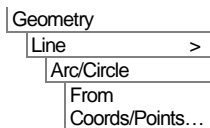- Enter the title as **Tuning Fork - Frequency Analysis** and click the **OK** button.

> 📝 **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

Geometry	
Point	>
By Coords...	
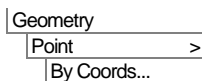
🔘 Define a Point at a coordinate of (**1.5, 0**) and click **OK**

Geometry	
Line	>
Arc/Circle	
From	
Coords/Points…	

- 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**

- Drag a box to select the 2 Points shown.

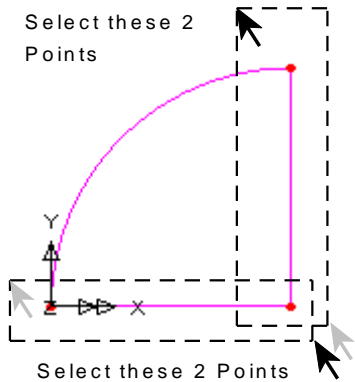S e l e c t t h e s e 2
P o i n t s

/ 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.

S e l e c t t h e s e 2 P o i n t s

▢ 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.

Geometry
  Point   >
    By Coords...

○ 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.

S e l e c t t h e s e 4 L i n e s

Geometry
  Point   >
    By Coords...
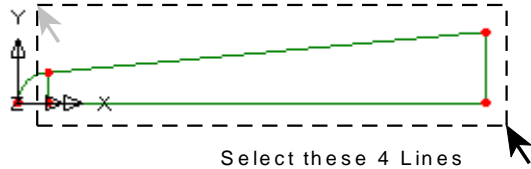
○ Click on a blank part of the view window to clear the selection and then Define Points at coordinates of (**44,0**) and (**44,7.5**) which form the ends of the radius handle of the fork.
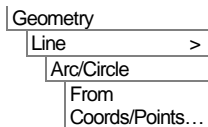

- Select pairs of Points and use the New Line button to create only the straight Lines shown.

1. Select this Point 2. Select this Point

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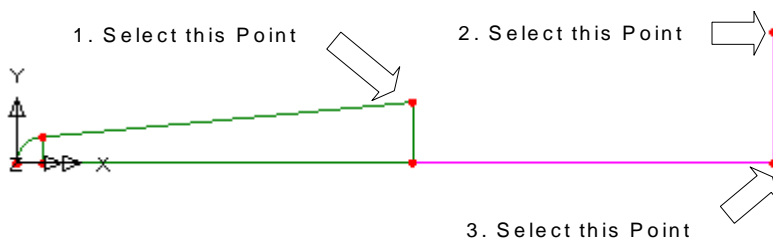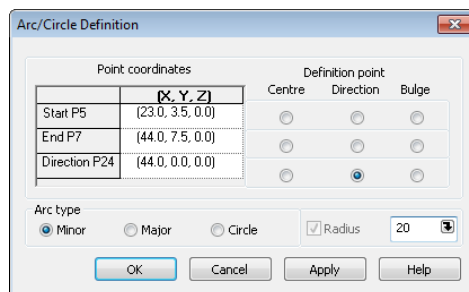
- To define the arc. Select the arc start and end Points, and the Point at the bottom right of the model.

- 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.

2. Select this Line

1. Select this Line
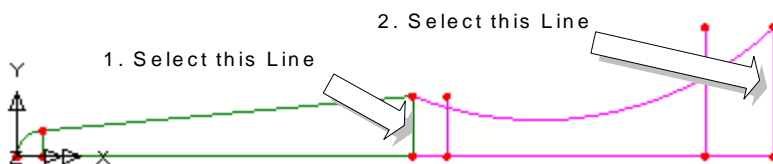
- Select the first Line shown.

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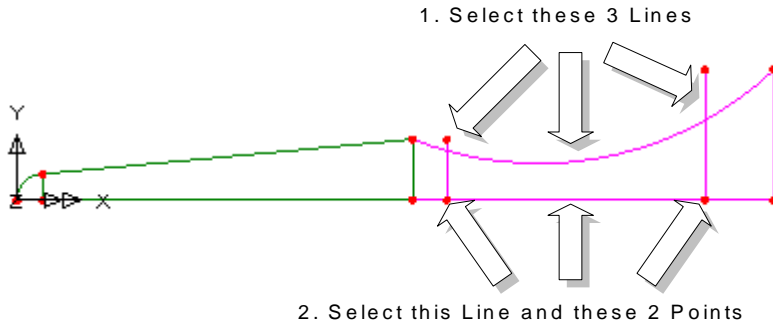
- Select the second Line shown

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**.

1. Select these 3 Lines

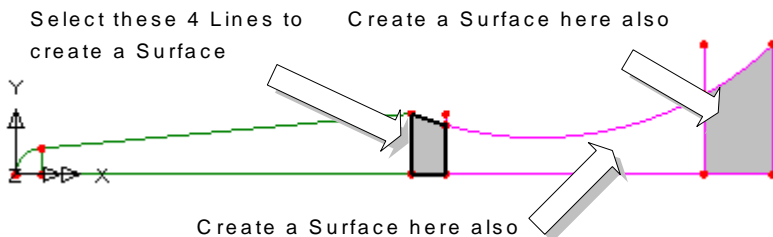2. Select this Line and these 2 Points

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.

Geometry	
Point	>
	By Intersection..

- 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.

Geometry	
Line	>
By Splitting	>
	At a point …

- Ensure that **Delete original geometry after splitting** is selected and click **OK** to create 3 new Lines.

Three Surfaces are now defined using the Lines previously created.

Select these 4 Lines to create a Surface   Create a Surface here also

Create a Surface here also

- 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
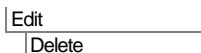　Surface　　　　　>
　　By Lines...
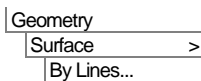
 Use the New Surface button to create a Surface.

- 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.

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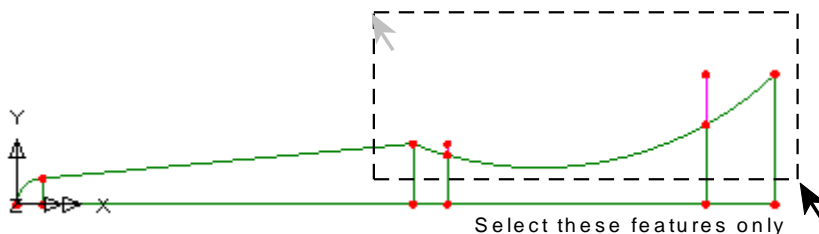
- Drag a box around the features shown, ensuring that no Surfaces are selected.

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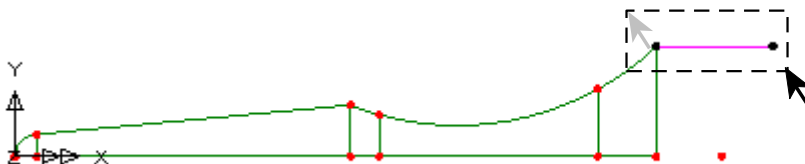
- Click on a blank part of the view window to clear the selection.

Geometry
　Point　　　　　>
　　By Coords...
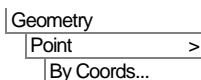
 Enter coordinates of (**48.5,0**) and (**52,7.5**) and click **OK** to define the points.
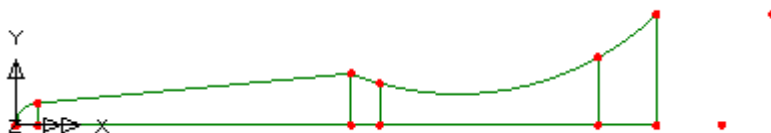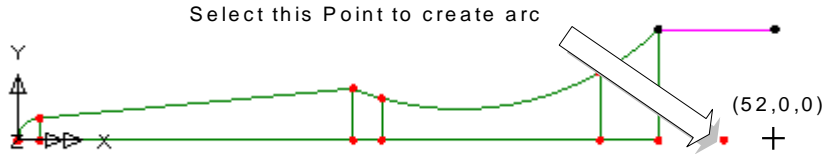

A horizontal Line will now be created.

- 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 this Point to create arc

(52,0,0)

- Select the single Point.

Geometry
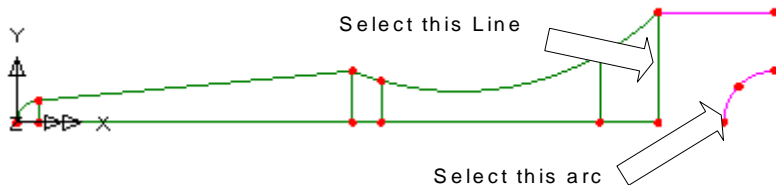| Line | > |
| Arc/Circle | > |
| By Sweeping Points… | |

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.

Geometry
Line	>
By Splitting	>
In equal divisions…	

- 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.

Select this Line
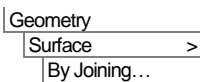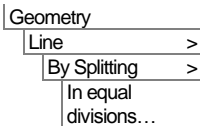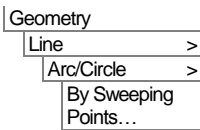
Select this arc

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.

Geometry
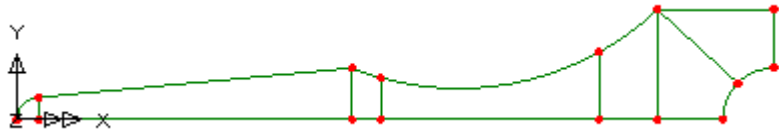| Surface | > |
| By Joining… | |

Create the first Surface.

- Repeat the previous procedure to create the second Surface as shown below.
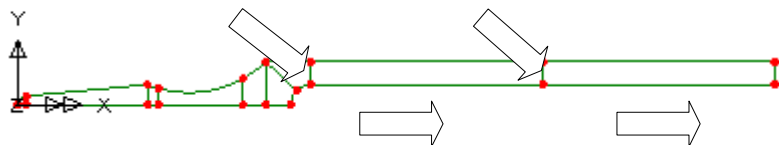
The remainder of the fork will be created by sweeping the vertical Line at the right-hand end of the model through a distance in the X direction.

⊕ 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.

Geometry	
Surface	>
By Sweeping…	

Geometry	
Surface	>
By Sweeping…	

- 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.
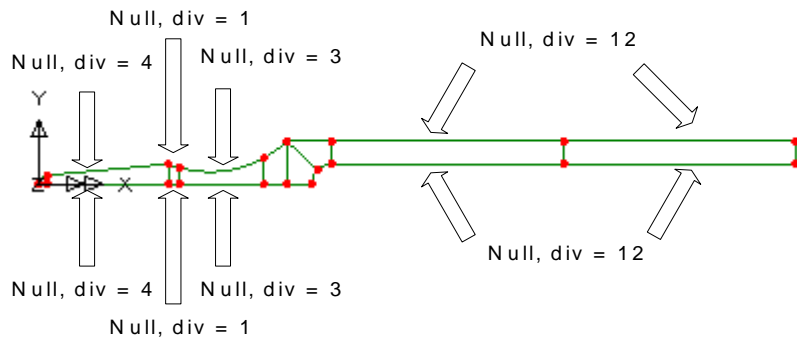
Surface mesh

- 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.
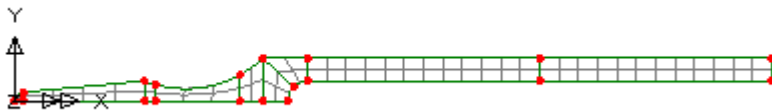

Note. Since all of the Surfaces are 4 (or 3) sided, a regular mesh pattern is created.

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

Attributes	
Geometric	>
Surface…	
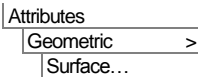
- Specify a thickness of **4** and leave the eccentricity blank.

- 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

Attributes	
Material	>
Material Library…	
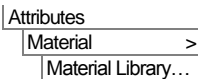
- Select material **Steel – Structural,** Region **None**, and Grade **Stainless steel,** and click **OK**

- With the whole model selected, drag and drop the material dataset **Iso3 (Stainless Steel | Steel – Structural)** from the 🔵 Treeview onto the selected features.

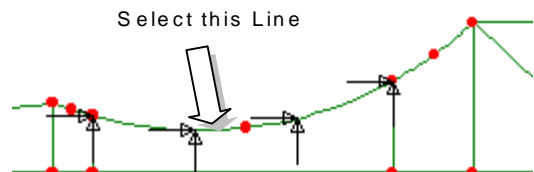- Ensure the **Assign to surfaces** option is selected and click **OK**

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.

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

Select this 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.
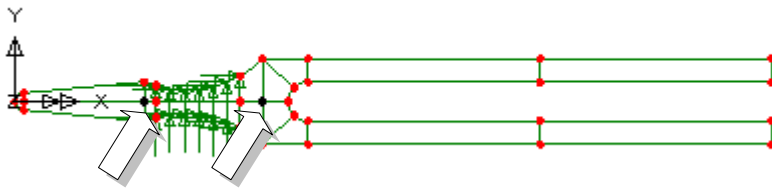
Select these 2 Points to define mirror plane

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

Edit
Selection Memory >
Set

The Points are stored in memory.

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

Geometry
Surface
Copy…

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.

Edit
Selection Memory >
Clear

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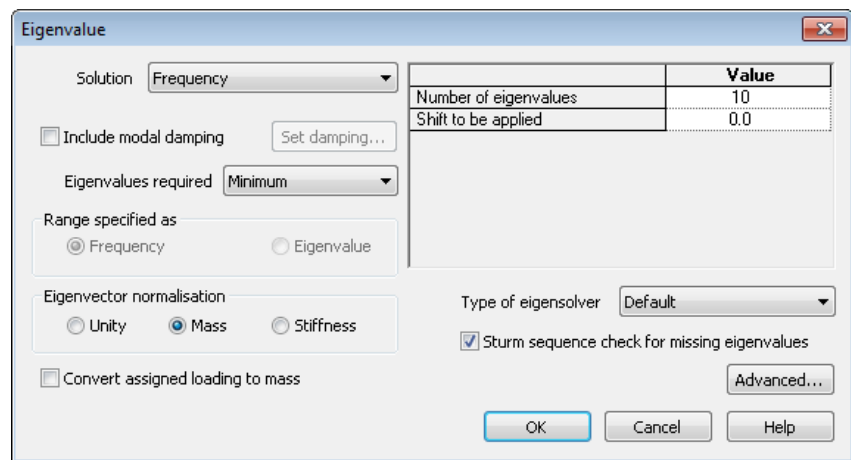
The model is now complete. All attributes assigned to the original half-model features, including the support and mesh assignments, will have been identically reproduced on the duplicated features.

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

File
Save

💾 Save the model file.

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.
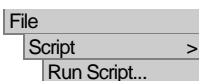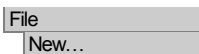
File
New…

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**

File
Script >
Run Script...

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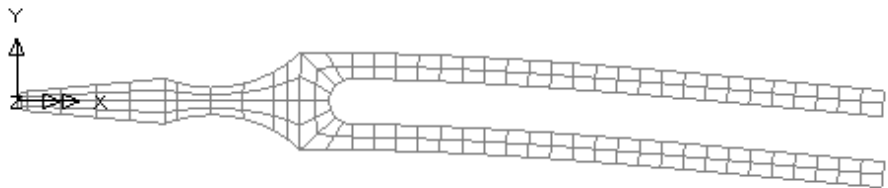
Note. The mode shape may be inverted. This is because the sense is arbitrary since during vibration the deformed shape will appear in both directions.

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.

Tools
   Animation Wizard...

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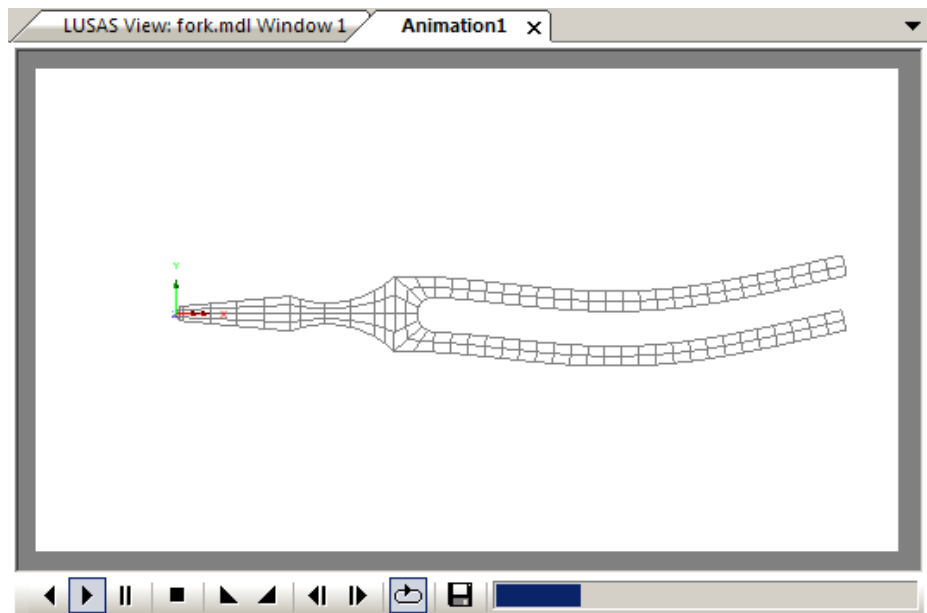
- The buttons at the bottom of the window may be used to slow-down, speed-up, pause, step through frame by frame, or stop the animation.

Saving Animations

Animations may be saved for replay in other Windows applications.

- Ensure the animation window is the active window.

File
   Save As AVI...

- 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**.

$\boxed{\times}$ Close the animation window without saving changes.

Printing Eigenvalue Results

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

Utilities
Print Results
Wizard…

- 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:

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.
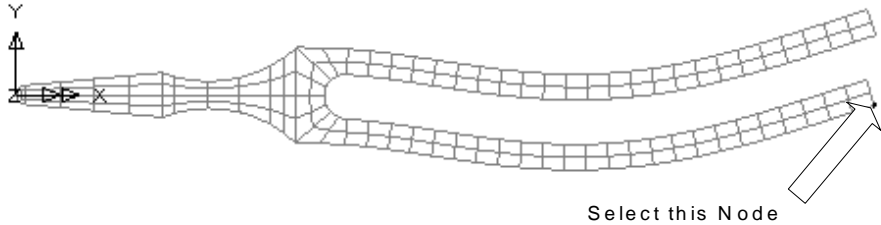
$\boxed{\times}$ Close the results window.

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.

Select this Node

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.

Utilities
  Graph Wizard...

- 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**

- 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.

- <u>Deselect</u> 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)

Tuning Fork - Full Eigen Analysis
Graph 1

—AMPLITUDE(2) / FREQUENCY(1)
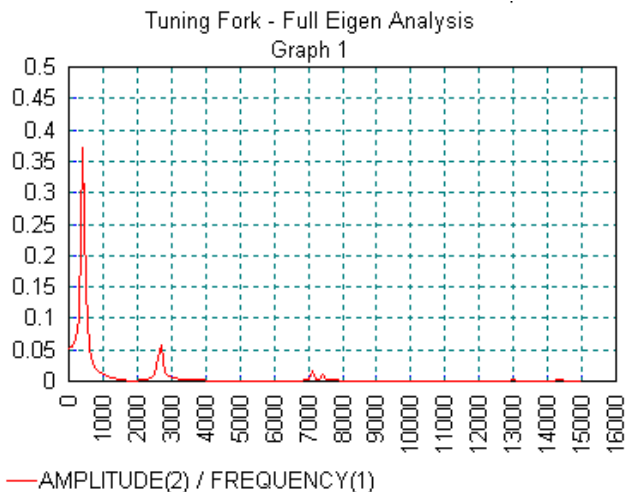
Note. Using a large frequency range with a 100Hz interval means the actual mode 1 frequency (of 428.729Hz) has been missed. When plotting graphs of this nature it is therefore better to use a smaller frequency range and interval in order to isolate the peak results.

To produce a more detailed graph

- Close the current graph window.

- Ensure the same node is selected on the fork.

Utilities
    Graph Wizard...

- 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.

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.

Tuning Fork - Full Eigen Analysis
Graph 1

— AMPLITUDE(4) / FREQUENCY(3)

- 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

Tuning Fork - Full Eigen Analysis
Graph 1

— AMPLITUDE(4) / FREQUENCY(3)
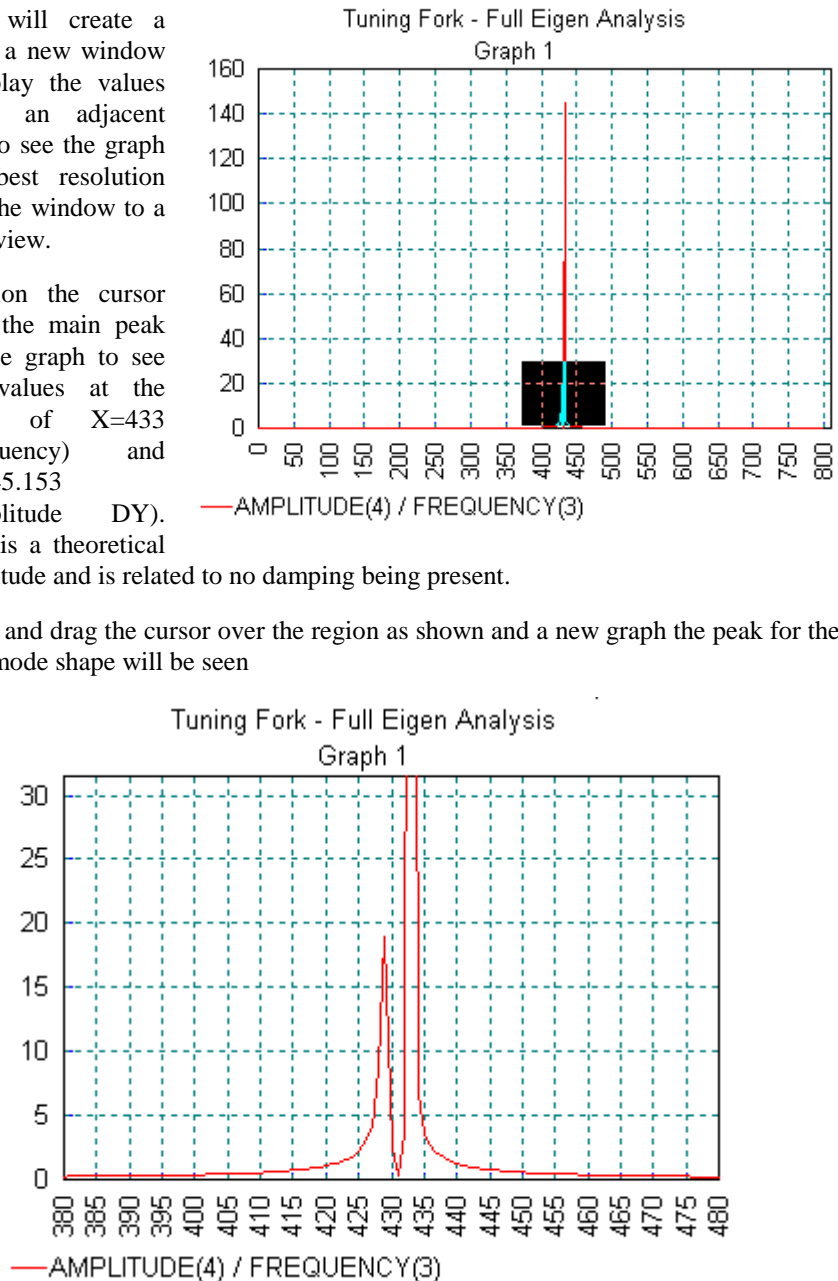
This displays a better representation of the displacement/ frequency response in the vicinity of the first two mode shapes.

- 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.

[x] 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.

Analyses
    IMD Loadcase...

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.
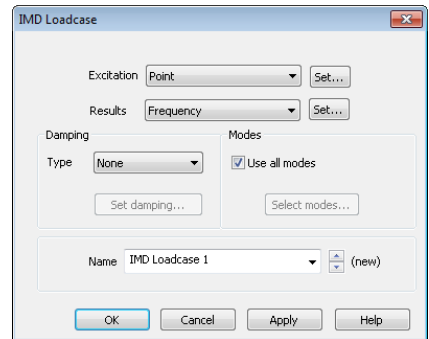
Selecting the IMD Results Loadcase

- In the [icon] 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.
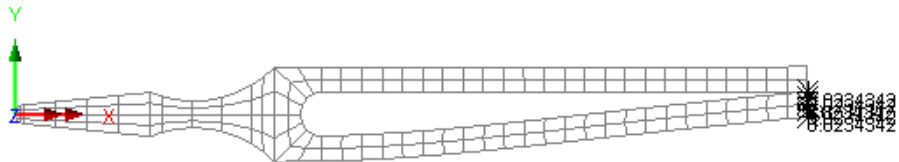
Changing the results frequency

- 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.

The deformed mesh plot will be updated to show the revised mode shape and corresponding values for the specified frequency.
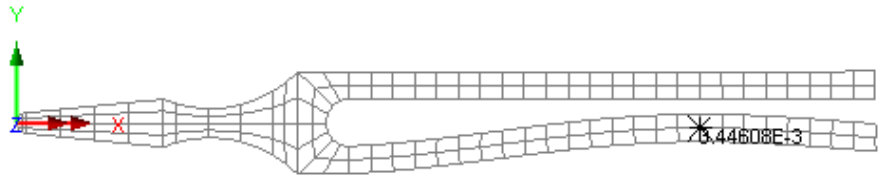
Note. Since the eigenvalue is independent of sign the deformed shape may appear inverted from that shown.

This completes the example.