Created in COMSOL Multiphysics 6.2

Bracket — Eigenfrequency Analysis

Introduction

In this example you learn how to perform an eigenfrequency analysis for an unloaded structure as well as for a prestressed structure.

When the structure is subjected to a constant external load, the stiffness generated by the stress may affect the natural frequencies of the structure. Tensile stresses tend to increase the natural frequencies, while compressive stresses tend to decrease them.

It is recommended you review the *Introduction to the Structural Mechanics Module*, which includes background information and discusses the `bracket_basic.mph` model relevant to this example.

Model Definition

This tutorial is an extension of the example described in the section "The Fundamentals: A Static Linear Analysis" in the *Introduction to the Structural Mechanics Module*.

The model geometry is represented in Figure 1.

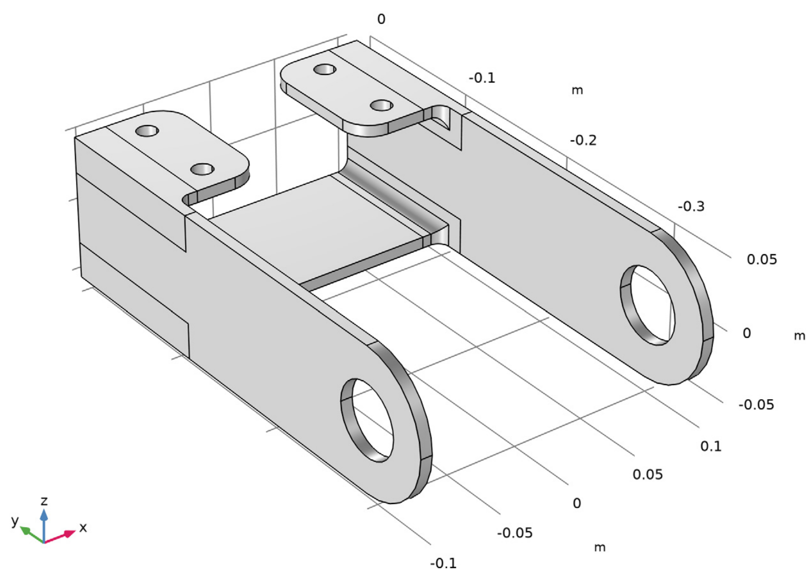

Figure 1: Geometry of the bracket.

In the first case, the natural frequency of the unloaded bracket is studied, while in the second case it is considered how the natural frequencies are affected by a static external load applied at the bracket holes. The left arm is under a pure tensile load while the right arm is under a pure compressive load.

Results and Discussion

Figure 2 and Figure 3 show the first six eigenmodes for both the unloaded and the prestressed case, respectively. The difference in the two first mode shapes between the two load cases is significant.

The two first mode shapes correspond to the bending mode in the *x* direction in the bracket arms. For the unloaded case these are expected to be approximately equal because of the symmetry. For the prestressed case, however, there will be a difference because of stress stiffening (left arm) and stress softening (right arm).

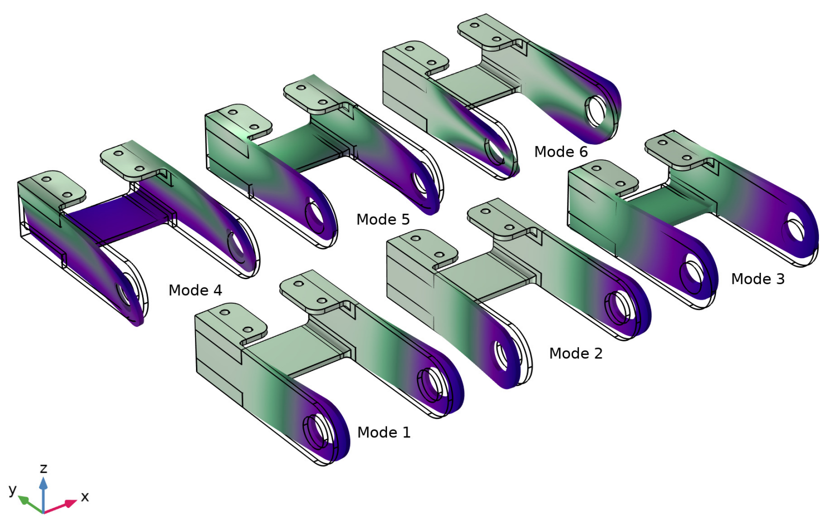

Figure 2: Six first eigenmode shapes for the unloaded case.

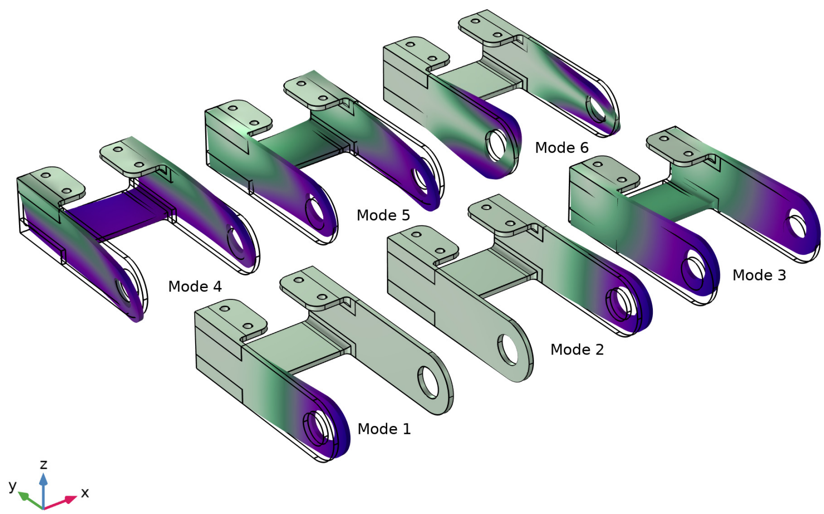

Figure 3: Six first eigenmode shapes for the prestressed case.

When comparing eigenmodes, it should be noted that the modes may be computed with reversed signs. This can even happen when the same study is run twice. Note also that the magnitudes of an eigenmode does not have any physical significance; that is why the default mode shape plots do not have a color legend.

In Figure 4, the stress state from the static preload is shown.

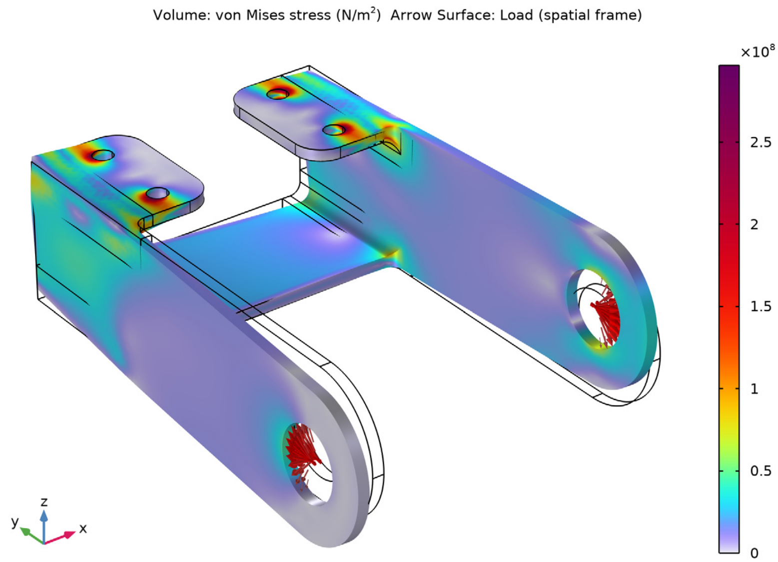

Figure 4: Equivalent stress from the preload. Note the nonlinear color scale.

In Figure 5 below, the frequency shift in the two first eigenmodes is clearly visible.

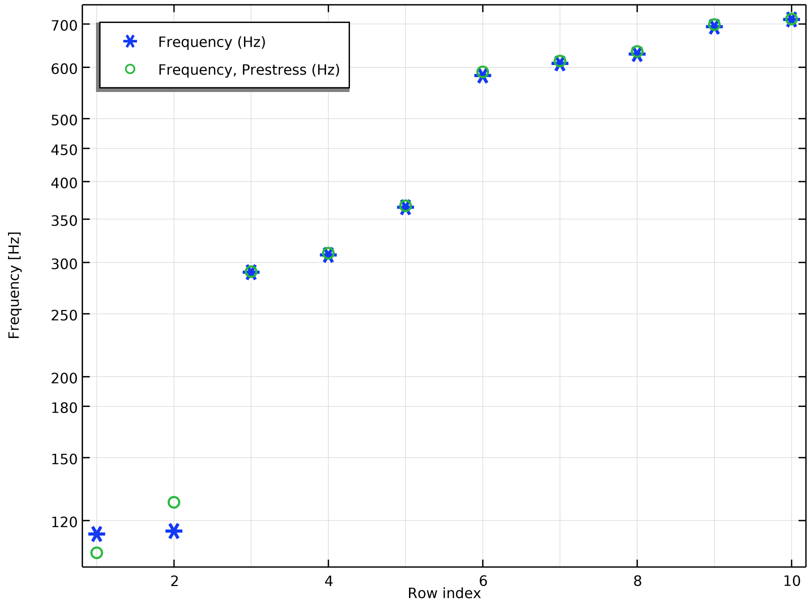

Figure 5: The six first eigenfrequencies for the unloaded case (stars) and the prestressed case (circles).

For the unloaded case, the two first eigenfrequencies are approximately 115 Hz. They correspond to the bending mode in the *x* direction for the two bracket arms. For the prestressed load case, the eigenfrequencies for the bending modes are 107 Hz for one of the arms and 128 Hz for the other. Such a frequency shift is expected since a tensile load causes stress stiffening, while a compressive load causes stress softening. The other mode shapes are not significantly affected by the prestress.

Notes About the COMSOL Implementation

For a structural mechanics physics interface in COMSOL Multiphysics, there are two predefined study types available for eigenfrequency analysis: **Eigenfrequency** and **Eigenfrequency, Prestressed**.

The plain eigenfrequency analysis computes the natural frequencies of the unloaded structure. The contribution of any load boundary condition is disregarded and prescribed displacement constraints are considered as having the value zero.

In the prestressed eigenfrequency analysis, a stationary analysis is first performed to take into account the different loads and nonzero displacement constraints. The resulting stress is then automatically taken into account in the stiffness used in the eigenfrequency calculation.

Application Library path: `Structural_Mechanics_Module/Tutorials/bracket_eigenfrequency`

Modeling Instructions

APPLICATION LIBRARIES

1. From the **File** menu, choose **Application Libraries**.
2. In the **Application Libraries** window, select **Structural Mechanics Module>Tutorials>bracket_basic** in the tree.
3. Click **Open**.

ADD STUDY

1. In the **Home** toolbar, click **Add Study** to open the **Add Study** window.
2. Go to the **Add Study** window.
3. Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Eigenfrequency**.
4. Click **Add Study** in the window toolbar.
5. In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 1

Step 1: Eigenfrequency

In the **Eigenfrequency** study node you have the possibility to select the number of eigenfrequencies to compute, and the frequency around which you would like to search for these frequencies. By default, the eigenvalue solver finds the six lowest frequencies.

1. In the **Settings** window for **Eigenfrequency**, locate the **Study Settings** section.
2. Select the **Desired number of eigenfrequencies** check box. In the associated text field, type `10`.
3. In the **Home** toolbar, click **Compute**.

RESULTS

Mode Shapes, Original

You can access the different eigenfrequency solutions in the 3D Plot Group Settings.

Note that the displacement values are normalized and have no physical significance. The normalization method can be changed in the **Eigenvalue Solver** node, located under the **Solver Configuration** node.

Modify the plot so that the first six eigenmodes are plotted side by side.

1 In the **Settings** window for **3D Plot Group**, type `Mode Shapes, Original` in the **Label** text field.

2 Click to expand the **Title** section. From the **Title type** list, choose **None**.

3 Click to expand the **Plot Array** section. Select the **Enable** check box.

4 From the **Array shape** list, choose **Square**.

5 In the **Relative row padding** text field, type `0.4`.

6 In the **Relative column padding** text field, type `0.4`.

Surface 1

1 In the **Model Builder** window, expand the **Mode Shapes, Original** node.

2 Right-click **Surface 1** and choose **Duplicate**.

Surface 2

1 In the **Model Builder** window, click **Surface 2**.

2 In the **Settings** window for **Surface**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 1/Solution 1 (sol1)**.

4 From the **Eigenfrequency (Hz)** list, choose **115.57**.

5 Right-click **Surface 2** and choose **Duplicate**.

Surface 3

1 In the **Model Builder** window, click **Surface 3**.

2 In the **Settings** window for **Surface**, locate the **Data** section.

3 From the **Eigenfrequency (Hz)** list, choose **289.82**.

4 Right-click **Surface 3** and choose **Duplicate**.

Surface 4

1 In the **Model Builder** window, click **Surface 4**.

2 In the **Settings** window for **Surface**, locate the **Data** section.

3 From the **Eigenfrequency (Hz)** list, choose **308.19**.

4 Right-click **Surface 4** and choose **Duplicate**.

Surface 5

1 In the **Model Builder** window, click **Surface 5**.

2 In the **Settings** window for **Surface**, locate the **Data** section.

3 From the **Eigenfrequency (Hz)** list, choose **365.06**.

4 Right-click **Surface 5** and choose **Duplicate**.

Surface 6

1 In the **Model Builder** window, click **Surface 6**.

2 In the **Settings** window for **Surface**, locate the **Data** section.

3 From the **Eigenfrequency (Hz)** list, choose **582.86**.

4 Click the **Zoom Extents** button in the **Graphics** toolbar.

5 Click the **Show Grid** button in the **Graphics** toolbar.

Parameters

1 In the **Results** toolbar, click **Parameters**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	**Expression**	**Value**	**Description**
dx	-0.05	-0.05	Annotation offset, X
dy	-0.3	-0.3	Annotation offset, Y

Annotation 1

1 In the **Model Builder** window, right-click **Mode Shapes, Original** and choose **Annotation**.

2 In the **Settings** window for **Annotation**, locate the **Annotation** section.

3 In the **Text** text field, type `Mode 1`.

4 Locate the **Position** section. In the **X** text field, type `dx`.

5 In the **Y** text field, type `dy`.

6 Locate the **Coloring and Style** section. Clear the **Show point** check box.

7 Click to expand the **Plot Array** section. Select the **Manual indexing** check box.

8 Right-click **Annotation 1** and choose **Duplicate**.

Annotation 2

1 In the **Model Builder** window, click **Annotation 2**.

2 In the **Settings** window for **Annotation**, locate the **Annotation** section.

3 In the **Text** text field, type `Mode 2`.

4 Locate the **Plot Array** section. In the **Column index** text field, type `1`.

5 Right-click **Annotation 2** and choose **Duplicate**.

Annotation 3

1 In the **Model Builder** window, click **Annotation 3**.

2 In the **Settings** window for **Annotation**, locate the **Annotation** section.

3 In the **Text** text field, type `Mode 3`.

4 Locate the **Plot Array** section. In the **Column index** text field, type `2`.

5 Right-click **Annotation 3** and choose **Duplicate**.

Annotation 4

1 In the **Model Builder** window, click **Annotation 4**.

2 In the **Settings** window for **Annotation**, locate the **Annotation** section.

3 In the **Text** text field, type `Mode 4`.

4 Locate the **Plot Array** section. In the **Row index** text field, type `1`.

5 In the **Column index** text field, type `0`.

6 Right-click **Annotation 4** and choose **Duplicate**.

Annotation 5

1 In the **Model Builder** window, click **Annotation 5**.

2 In the **Settings** window for **Annotation**, locate the **Annotation** section.

3 In the **Text** text field, type `Mode 5`.

4 Locate the **Plot Array** section. In the **Column index** text field, type `1`.

5 Right-click **Annotation 5** and choose **Duplicate**.

Annotation 6

1 In the **Model Builder** window, click **Annotation 6**.

2 In the **Settings** window for **Annotation**, locate the **Annotation** section.

3 In the **Text** text field, type `Mode 6`.

4 Locate the **Plot Array** section. In the **Column index** text field, type `2`.

5 In the **Mode Shapes, Original** toolbar, click **Plot**.

Global Evaluation 1

1 In the **Results** toolbar, click **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.

3 In the table, enter the following settings:

Expression	Unit	Description
freq	Hz	Frequency

4 Click **Evaluate**.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
P0	30[MPa]	3E7 Pa	Peak load intensity

DEFINITIONS

Analytic 1 (an1)

1 In the **Home** toolbar, click f(X) **Functions** and choose **Local>Analytic**.

2 In the **Settings** window for **Analytic**, type `load` in the **Function name** text field.

3 Locate the **Definition** section. In the **Expression** text field, type `F*cos(atan2(py, abs(px)))`.

4 In the **Arguments** text field, type `F, py, px`.

5 Locate the **Units** section. In the table, enter the following settings:

Argument	Unit
F	Pa
py	m
px	m

6 In the **Function** text field, type `Pa`.

Boundary System 1 (sys1)

The default boundary coordinate system will have axes that change orientation with the deformation. As an effect, the applied load would be interpreted as a follower load. In this case, the load is intended to have fixed directions.

1 In the **Model Builder** window, click **Boundary System 1 (sys1)**.
2 In the **Settings** window for **Boundary System**, locate the **Settings** section.
3 From the **Frame** list, choose **Reference configuration**.

SOLID MECHANICS (SOLID)

Boundary Load 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** and choose **Boundary Load**.

   Apply a boundary load to the bracket holes.

2 In the **Settings** window for **Boundary Load**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Pin Holes**.
4 Locate the **Coordinate System Selection** section. From the **Coordinate system** list, choose **Boundary System 1 (sys1)**.
5 Locate the **Force** section. Specify the $\mathbf{F}_A$ vector as

`0`	t1
`0`	t2
`load(-P0,Z,Y-PinHoleY)*(sign(X)*(Y-PinHoleY)<0)`	n

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.
2 Go to the **Add Study** window.

   The prestressed eigenfrequency analysis is available as a predefined study.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces>Eigenfrequency, Prestressed**.
4 Click **Add Study** in the window toolbar.
5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Stationary

Note that the newly generated study combines one stationary analysis and one eigenfrequency analysis.

Step 2: Eigenfrequency

1 In the **Model Builder** window, click **Step 2: Eigenfrequency**.

2 In the **Settings** window for **Eigenfrequency**, locate the **Study Settings** section.

3 Select the **Desired number of eigenfrequencies** check box. In the associated text field, type `10`.

4 In the **Search for eigenfrequencies around shift** text field, type `100`.

5 In the **Model Builder** window, click **Study 2**.

6 In the **Settings** window for **Study**, locate the **Study Settings** section.

7 Clear the **Generate default plots** check box.

8 In the **Home** toolbar, click **Compute**.

RESULTS

Mode Shapes, Original

In the **Model Builder** window, under **Results** right-click **Mode Shapes, Original** and choose **Duplicate**.

Mode Shapes, Prestressed

1 In the **Model Builder** window, expand the **Results>Mode Shapes, Original 1** node, then click **Mode Shapes, Original 1**.

2 In the **Settings** window for **3D Plot Group**, type `Mode Shapes, Prestressed` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

Surface 2

1 In the **Model Builder** window, click **Surface 2**.

2 In the **Settings** window for **Surface**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

Surface 3

1 In the **Model Builder** window, click **Surface 3**.

2 In the **Settings** window for **Surface**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

Surface 4

1 In the **Model Builder** window, click **Surface 4**.

2 In the **Settings** window for **Surface**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

Surface 5

1 In the **Model Builder** window, click **Surface 5**.

2 In the **Settings** window for **Surface**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

Surface 6

1 In the **Model Builder** window, click **Surface 6**.

2 In the **Settings** window for **Surface**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

4 In the **Mode Shapes, Prestressed** toolbar, click **Plot**.

In the settings for the second plot group you can see the list of the new eigenfrequencies. You can also add them to the previous table for easy comparison with the corresponding values without preload.

Global Evaluation 1

1 In the **Model Builder** window, under **Results>Derived Values** click **Global Evaluation 1**.

2 In the **Settings** window for **Global Evaluation**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 2/Solution 2 (sol2)**.

4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	**Unit**	**Description**
`freq`	`Hz`	`Frequency, Prestress`

5 Click **Evaluate**.

Add a plot with stress and loads for the prestressed state.

ADD PREDEFINED PLOT

1 In the **Home** toolbar, click **Add Predefined Plot** to open the **Add Predefined Plot** window.

2 Go to the **Add Predefined Plot** window.

3 In the tree, select **Study 2/Solution Store 1 (sol3)>Solid Mechanics>Stress (solid)** and **Study 2/Solution Store 1 (sol3)>Solid Mechanics>Applied Loads (solid)> Boundary Loads (solid)**.

4 Click **Add Plot** in the window toolbar.

5 In the **Home** toolbar, click **Add Predefined Plot** to close the **Add Predefined Plot** window.

RESULTS

Boundary Load 1

1 In the **Model Builder** window, expand the **Boundary Loads (solid)** node.

2 Right-click **Boundary Load 1** and choose **Copy**.

Stress (solid)

In the **Model Builder** window, under **Results** right-click **Stress (solid)** and choose **Paste Arrow Surface**.

Boundary Load 1

1 In the **Model Builder** window, click **Boundary Load 1**.

2 In the **Settings** window for **Arrow Surface**, click to expand the **Inherit Style** section.

3 From the **Plot** list, choose **Volume 1**.

4 Clear the **Color** check box.

5 Clear the **Color and data range** check box.

6 Locate the **Coloring and Style** section. From the **Arrow base** list, choose **Head**.

Color Expression

1 In the **Model Builder** window, expand the **Boundary Load 1** node, then click **Color Expression**.

2 In the **Settings** window for **Color Expression**, locate the **Coloring and Style** section.

3 Clear the **Color legend** check box.

Boundary Loads (solid)

In the **Model Builder** window, under **Results** right-click **Boundary Loads (solid)** and choose **Delete**.

Volume 1

1 In the **Model Builder** window, under **Results>Stress (solid)** click **Volume 1**.

2 In the **Settings** window for **Volume**, locate the **Coloring and Style** section.

3 From the **Color table transformation** list, choose **Nonlinear**.

4 Set the **Color calibration parameter** value to **-1.4**.

Stress From Static Load

1 In the **Model Builder** window, under **Results** click **Stress (solid)**.

2 In the **Settings** window for **3D Plot Group**, type `Stress From Static Load` in the **Label** text field.

3 Click the **Zoom Extents** button in the **Graphics** toolbar.

4 In the **Stress From Static Load** toolbar, click **Plot**.

Eigenfrequency Comparison

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, type `Eigenfrequency Comparison` in the **Label** text field.

3 Locate the **Plot Settings** section.

4 Select the **y-axis label** check box. In the associated text field, type `Frequency [Hz]`.

5 Locate the **Grid** section. Select the **Manual spacing** check box.

6 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Table Graph 1

1 Right-click **Eigenfrequency Comparison** and choose **Table Graph**.

2 In the **Settings** window for **Table Graph**, locate the **Data** section.

3 From the **x-axis data** list, choose **Row index**.

4 From the **Plot columns** list, choose **Manual**.

5 In the **Columns** list, choose **Frequency (Hz)** and **Frequency, Prestress (Hz)**.

6 Click to expand the **Legends** section. Select the **Show legends** check box.

7 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

8 Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

9 In the **Eigenfrequency Comparison** toolbar, click **Plot**.

10 Click the **y-Axis Log Scale** button in the **Graphics** toolbar.