Model created in COMSOL Multiphysics 6.4

Eigenmodes of a Viscoelastic Structural Damper

Introduction

The example studies the natural frequencies and corresponding eigenmodes of a typical viscoelastic damper. Damping elements involving layers of viscoelastic materials are often used for the reduction of seismic and wind induced vibrations in buildings and other tall structures. The common feature for such structures is that the frequency of their possible forced vibrations is low. Thus, it is important to use dampers with natural frequencies that are high enough to avoid any failures due to resonances.

Solving for eigenfrequencies in a structure where the deformation to a large extent is controlled by a viscoelastic material (or any other material which has frequency dependent material properties) requires special techniques. A standard eigenfrequency problem without damping can be formulated as

$$[\mathbf{K} - f^2\mathbf{M}]\mathbf{u} = 0 \qquad (1)$$

where $\mathbf{K}$ is the stiffness matrix, $\mathbf{M}$ is the mass matrix, $\mathbf{u}$ is the eigenmode displacement vector, and f is the frequency. In most cases, $\mathbf{K}$ is independent of frequency, but for a viscoelastic material the eigenvalue equation actually is

$$[\mathbf{K}(f) - f^2\mathbf{M}]\mathbf{u} = 0 \qquad (2)$$

In this example, it is shown how you can handle this type of problem.

Model Definition

The geometry of the viscoelastic damper is shown in Figure 1 (from Ref. 1). The damper consists of two layers of viscoelastic material confined between mounting elements made of steel.

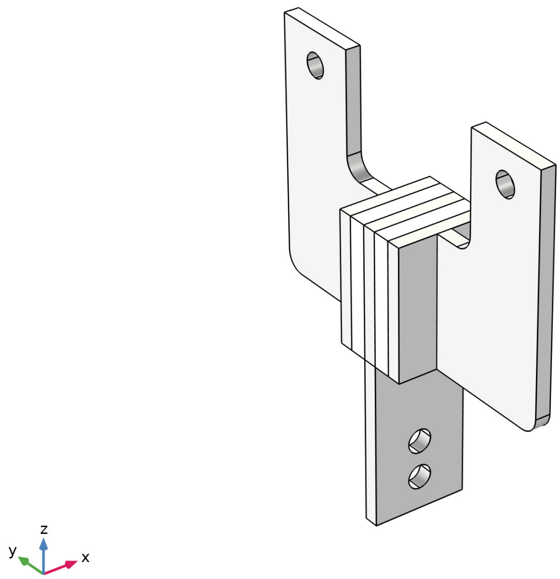


Figure 1: Viscoelastic damping element.

One of the mounting elements is modeled as fixed, and two other elements are partially constrained to represent a typical operating regime of the damper.

The viscoelastic layers are modeled using complex valued material data. For frequency domain and eigenfrequency analyses, the frequency decomposition for the deviatoric stress and strain tensors is in general performed as:

$$S_{\mathrm{d}} = \hat{S}_{\mathrm{d}} e^{2\pi i f t}$$

$$\varepsilon_{\mathrm{d}} = \hat{\varepsilon}_{\mathrm{d}} e^{2\pi i f t}$$

where f is the frequency.

The deviatoric stress is related then to the strain as

$$\hat{S}_{\mathrm{d}} = 2G(f)\hat{\varepsilon}_{\mathrm{d}} \tag{3}$$

where the complex valued shear modulus is written as

$$G = G'(f) + iG''(f)$$

where the G' and G'' are the storage and loss moduli, respectively.

In this example, the moduli are specified by their reference values given at two reference frequencies f_{r1} = 200 Hz and f_{r2} =1000 Hz:

TABLE 1: STORAGE AND LOSS MODULI.

f	f_{r1}	f_{r2}
G'	3.0848E7 Pa	7.8348E7 Pa
G''	3.6551E7 Pa	8.4935E7 Pa

The frequency dependency is approximated by straight lines in the log-log space using the above data. Thus, the following expressions are used for the moduli:

$$G'(f) = G'(f_{\mathrm{r1}})\left(\frac{f}{f_{\mathrm{r1}}}\right)^n$$

where

$$n = \log\left(\frac{G'(f_{\mathrm{r2}})}{G'(f_{\mathrm{r1}})}\right) \Big/ \log\left(\frac{f_{\mathrm{r2}}}{f_{\mathrm{r1}}}\right)$$

and

$$G''(f) = G''(f_{\mathrm{r1}})\left(\frac{f}{f_{\mathrm{r1}}}\right)^m$$

where

$$m = \log\left(\frac{G''(f_{\mathrm{r2}})}{G''(f_{\mathrm{r1}})}\right) \Big/ \log\left(\frac{f_{\mathrm{r2}}}{f_{\mathrm{r1}}}\right)$$

Substituting the deviatoric stress given by Equation 3 into the equation of motion gives

$$-\rho 4\pi^2 f^2 \hat{\mathbf{u}} = -\nabla\hat{p} + 2(G'(f) + iG''(f))\nabla\cdot\hat{\varepsilon}_{\mathrm{d}}$$

which, together with the boundary conditions, will result in a nonlinear eigenvalue problem for f. The eigenvalue problem will determine the natural frequencies of the system.

COMSOL Multiphysics solves such nonlinear problems by expanding all expressions containing the frequency down to quadratic polynomials using a frequency linearization point f_L which you can specify in the **Eigenvalue Solver** node (100 Hz is used by default).

The eigenvalue problem, which is solved, is then

$$[\mathbf{K}(f_{\mathrm{L}}) - f^2\mathbf{M}]\mathbf{u} = 0$$

Thus, the results become dependent on the choice of the frequency linearization point.

Starting from COMSOL 6.3, there are two options to handle the nonlinearity.

You can use a new eigenvalue solver, ARPACK nonlinear. This solver approximates the nonlinear functions with polynomials using Taylor expansion, and then it will build and solve a number of equivalent linear eigenvalue problems. The use of this approach can be computationally expensive for larger models.

As an alternative, you can compute an approximation of the complex valued shear modulus using the partial fraction fit. The approximation corresponds to an equivalent Generalized viscoelasticity model, and it can be used in both Eigenfrequency and Time Dependent study steps. For the eigenfrequency computations, the eigenvalues problem will be linear. The approximation computation needs to be performed as a preprocessing step, but it is very fast and independent of the model size.

This example demonstrates the use of both approaches.

You model in 3D using the Solid Mechanics interface with a **Linear Elastic Material** and add a **Viscoelasticity** subnode to the domains representing the viscoelastic layers. The approximation computation becomes available directly on the **Viscoelasticity** subnode when the **User defined** viscoelasticity model is selected.

Results and Discussion

Six eigenfrequencies are initially computed using the default linear eigenvalue solver using 100 Hz as the default frequency linearization point. The first computed eigenmode is shown in Figure 2.

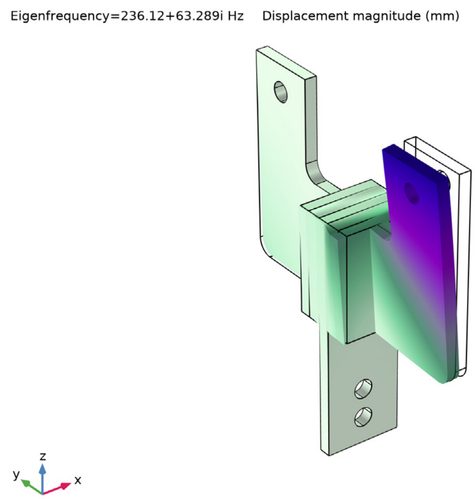


Figure 2: First eigenmode computed using linear solver with default frequency linearization point.

The computed eigenfrequencies can only be expected to be correct by the order of magnitude. This allows you to identify the frequency range 200 – 1000 Hz for further investigations.

The approximation computed for the storage and loss moduli contributions from the Viscoelasticity subnode is shown in Figure 3.

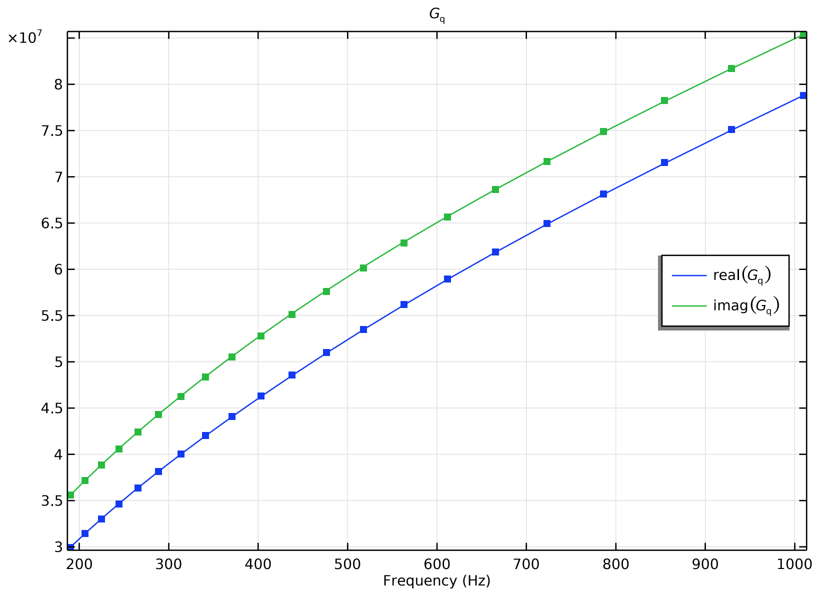


Figure 3: Storage and loss moduli contribution. Solid line is the approximation, the line markers correspond to the user defined expressions.

Figure 4 shows the eigenfrequencies computed using different approaches.

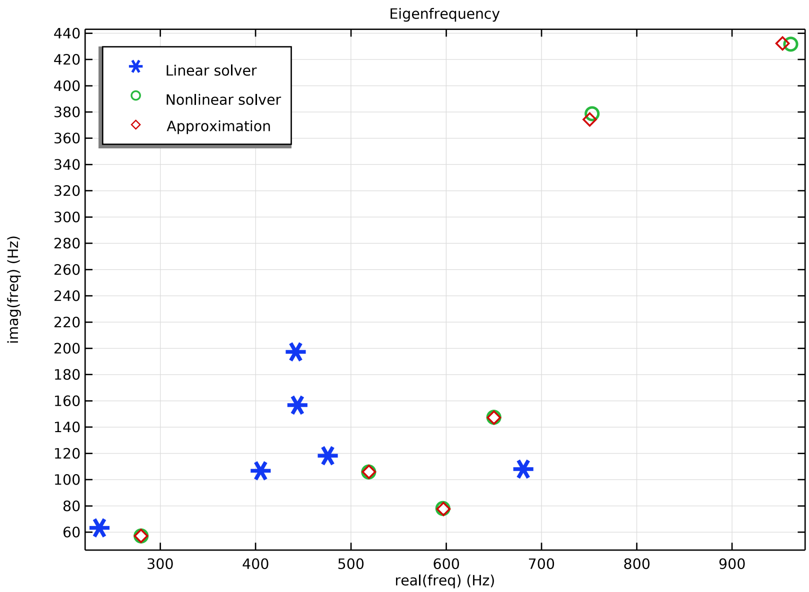


Figure 4: The eigenfrequency distribution in the complex plane.

Thus, the results of using the approximation and nonlinear eigenvalue solver are in good agreement with each other. The first eigenmode computed using the approximation is shown in Figure 5.

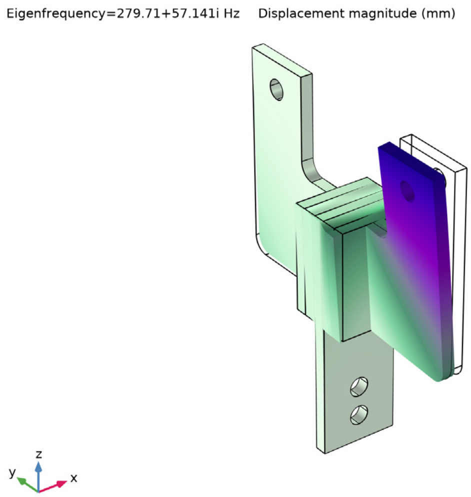


Figure 5: First eigenmode computed a linear eigenvalue solver together with the approximation of viscoelastic data via the partial fraction fit.

References

1. S.W. Park “Analytical Modeling of Viscoelastic Dampers for Structural and Vibration Control,” *Int. J. Solids and Structures*, vol. 38, pp. 694–701, 2001.

2. K.L. Shen and T.T. Soong, “Modeling of Viscoelastic Dampers for Structural Applications,” *J. Eng. Mech.*, vol. 121, pp. 694–701, 1995.

Application Library path: `Structural_Mechanics_Module/Dynamics_and_Vibration/viscoelastic_damper_eigenmodes`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

2 In the **Select Physics** tree, select **Structural Mechanics** > **Solid Mechanics (solid)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies** > **Eigenfrequency**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

Import the viscoelastic material data from a file.

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

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

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `viscoelastic_damper_eigenmodes_parameters.txt`.

The data contains the reference values of the storage and loss moduli given at two reference frequencies. You approximate the data by straight lines in the log-log space. This can be done by using analytic functions as follows.

Analytic 1 (an1)

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

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

3 Locate the **Definition** section. In the **Expression** text field, type `gsr1*(f/fr1)^ns`.

4 In the **Arguments** text field, type `f`.

5 Locate the **Units** section. In the **Function** text field, type `Pa`.

6 In the table, enter the following settings:

Argument	Unit
f	Hz

7 Click to expand the **Advanced** section. Select the **May produce complex output for real arguments** checkbox.

8 Locate the **Plot Parameters** section. In the table, enter the following settings:

Plot	Argument	Lower limit	Upper limit	Fixed value	Unit
√	f	`fr1`	`fr2`	0	Hz

Analytic 2 (gstor2)

1 Right-click **Analytic 1 (gstor)** and choose **Duplicate**.

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

3 Locate the **Definition** section. In the **Expression** text field, type `glr1*(f/fr1)^nl`.

GEOMETRY 1

Import the predefined geometry from a file.

1 In the **Geometry** toolbar, click **Insert Sequence** and choose **Insert Sequence**.

2 Browse to the model's Application Libraries folder and double-click the file `viscoelastic_damper_geom_sequence.mph`.

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

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

The imported geometry should look similar to that shown in Figure 1.

SOLID MECHANICS (SOLID)

Linear Elastic Material 2

1 In the **Physics** toolbar, click **Domains** and choose **Linear Elastic Material**.

2 In the **Settings** window for **Linear Elastic Material**, locate the **Linear Elastic Material** section.

3 From the **Specify** list, choose **Bulk modulus and shear modulus**.

4 From the **Use mixed formulation** list, choose **Pressure formulation**.

5 Select Domains 2 and 5 only.

Viscoelasticity 1

1 In the **Physics** toolbar, click **Attributes** and choose **Viscoelasticity**.

Use the analytic functions to enter the viscoelastic moduli as functions of frequency.

2 In the **Settings** window for **Viscoelasticity**, locate the **Viscoelasticity Model** section.

3 From the **Material model** list, choose **User defined**.

4 In the *G′* text field, type `gstor(solid.freq)`.

5 In the *G″* text field, type `gloss(solid.freq)`.

ADD MATERIAL

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

2 Go to the **Add Material** window.

3 In the tree, select **Built-in > Steel AISI 4340**.

4 Click the **Add to Component** button in the window toolbar.

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

MATERIALS

Viscoelastic

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Viscoelastic` in the **Label** text field.

3 Select Domains 2 and 5 only.

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

Property	Variable	Value	Unit	Property group
Bulk modulus	K	4e8	N/m²	Bulk modulus and shear modulus
Shear modulus	G	5.86e4	N/m²	Bulk modulus and shear modulus
Density	rho	1060	kg/m³	Basic

5 In the **Model Builder** window, under **Component 1 (comp1)** click **Materials**.

6 In the **Settings** window for **Materials**, in the **Graphics** window toolbar, click next to **Colors**, then choose **Show Material Color and Texture**.

SOLID MECHANICS (SOLID)

Fixed Constraint 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.

2 In the **Settings** window for **Fixed Constraint**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Bottom Holes**.

Prescribed Displacement 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Prescribed Displacement**.
2. In the **Settings** window for **Prescribed Displacement**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Right Hole**.
4. Locate the **Prescribed Displacement** section. From the **Displacement in x direction** list, choose **Prescribed**.
5. From the **Displacement in y direction** list, choose **Prescribed**.

Prescribed Displacement 2

1. In the **Physics** toolbar, click **Boundaries** and choose **Prescribed Displacement**.
2. In the **Settings** window for **Prescribed Displacement**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Left Hole**.
4. Locate the **Prescribed Displacement** section. From the **Displacement in y direction** list, choose **Prescribed**.

MESH 1

Free Quad 1

1. In the **Mesh** toolbar, click **More Generators** and choose **Free Quad**.
2. Select Boundary 30 only.

Size 1

1. Right-click **Free Quad 1** and choose **Size**.
2. In the **Settings** window for **Size**, locate the **Element Size** section.
3. From the **Predefined** list, choose **Finer**.

Distribution 1

1. In the **Model Builder** window, right-click **Free Quad 1** and choose **Distribution**.
2. Select Edge 65 only.

Swept 1

1. In the **Mesh** toolbar, click **Swept**.
2. In the **Settings** window for **Swept**, locate the **Domain Selection** section.
3. From the **Geometric entity level** list, choose **Domain**.
4. Select Domain 7 only.

Distribution 1

1 Right-click **Swept 1** and choose **Distribution**.

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

3 In the **Number of elements** text field, type 2.

Free Quad 2

1 In the **Mesh** toolbar, click **More Generators** and choose **Free Quad**.

2 Select Boundaries 2 and 61 only.

Size 1

1 Right-click **Free Quad 2** and choose **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Predefined** list, choose **Fine**.

Swept 2

1 In the **Mesh** toolbar, click **Swept**.

2 In the **Settings** window for **Swept**, locate the **Domain Selection** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 Select Domains 1, 2, and 4 only.

Distribution 1

1 Right-click **Swept 2** and choose **Distribution**.

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

3 In the **Number of elements** text field, type 2.

Copy Domain 1

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Copying Operations** > **Copy Domain**.

2 Select Domains 1, 2, and 7 only.

3 In the **Settings** window for **Copy Domain**, locate the **Destination Domains** section.

4 Click to select the **Activate Selection** toggle button.

5 Select Domains 5, 6, and 8 only.

Free Quad 3

1 In the **Mesh** toolbar, click **More Generators** and choose **Free Quad**.

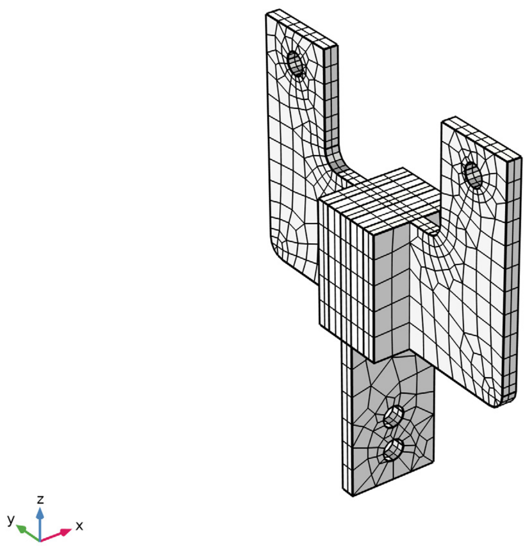
2 Select Boundary 10 only.

Swept 3

1 In the **Mesh** toolbar, click **Swept**.

2 In the **Settings** window for **Swept**, click **Build All**.

The complete mesh should look similar to that shown in the figure below.



Perform the initial eigenfrequency analysis.

STUDY 1

1 In the **Study** toolbar, click **Compute**.

The computed eigenvalues will be automatically evaluated into a table.

RESULTS

Eigenfrequencies (Study 1)

These values are not exact because of their distance from the frequency linearization point (by default, set to 100 Hz). However, they can indicate a region of the real and imaginary parts for more accurate eigenfrequency analysis.

Eigenfrequency

1 In the **Results** toolbar, click **1D Plot Group**.

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

Global 1

1 Right-click **Eigenfrequency** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`imag(freq)`	`Hz`	

4 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

5 In the **Expression** text field, type `real(freq)`.

6 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

7 From the **Width** list, choose **3**.

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

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

Add one more eigenfrequency study and configure it to use a nonlinear eigenvalue solver.

ADD STUDY

1 In the **Home** toolbar, click **Windows** and choose **Add Study**.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies** > **Eigenfrequency**.

4 Right-click and choose **Add Study**.

STUDY 2

Step 1: Eigenfrequency

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

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

3 From the **Eigenfrequency solver** list, choose **ARPACK nonlinear**.

4 From the **Eigenfrequency search method** list, choose **Rectangle**.

5 Find the **Rectangle search region** subsection. In the **Smallest real part (Eigenfrequency)** text field, type `200`.

6 In the **Largest real part (Eigenfrequency)** text field, type `1000`.

7 In the **Largest imaginary part (Eigenfrequency)** text field, type `1000`.

The nonlinear eigenvalue solver will iterate using a Taylor expansion at the center of the specified frequency region.

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

RESULTS

Eigenfrequency

1 In the **Model Builder** window, under **Results** click **Eigenfrequency**.

2 In the **Settings** window for **1D Plot Group**, locate the **Plot Settings** section.

3 Select the **x-axis label** checkbox.

4 Select the **y-axis label** checkbox. In the associated text field, type `imag(freq) (Hz)`.

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

6 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

7 In the **Title** text area, type `Eigenfrequency`.

Global 1

1 In the **Model Builder** window, click **Global 1**.

2 In the **Settings** window for **Global**, click to expand the **Legends** section.

3 From the **Legends** list, choose **Manual**.

4 In the table, enter the following settings:

Legends
`Linear solver`

Global 2

1 Right-click **Results** > **Eigenfrequency** > **Global 1** and choose **Duplicate**.

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

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

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

Legends
`Nonlinear solver`

5 In the **Eigenfrequency** toolbar, click **Plot**.

To use a nonlinear eigenvalue solver can be a computationally expensive solution particularly for large models. Instead, you can compute an approximation for the storage and loss data using a partial fraction fitting algorithm. This you can set up directly on the Viscoelasticity node.

SOLID MECHANICS (SOLID)

Viscoelasticity 2

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Solid Mechanics (solid)** > **Linear Elastic Material 2** right-click **Viscoelasticity 1** and choose **Duplicate**.

2 In the **Settings** window for **Viscoelasticity**, locate the **Viscoelasticity Model** section.

3 Select the **Low frequency limit** checkbox.

4 Locate the **Time Domain and Eigenfrequency** section. From the **Frequency range** list, choose **Minimum and maximum**.

5 In the f_{min} text field, type `fr1-10[Hz]`.

6 In the f_{max} text field, type `fr2+10[Hz]`.

7 Click **Approximation** in the upper-right corner of the **Time Domain and Eigenfrequency** section. From the menu, choose **Compute Approximation**.

8 Click **Section_bar** in the upper-right corner of the **Time Domain and Eigenfrequency** section. From the menu, choose **Preview Approximation**.

9 Click **Preview Approximation** in the upper-right corner of the **Time Domain and Eigenfrequency** section. From the menu, choose **Create Approximation Plot**.

RESULTS

Approximation Plot

Thus computed approximation can be used for solving with a linear eigenvalue solver. It can be also used for modeling the problem in time domain.

Add one more eigenfrequency study and compute the eigenvalues.

ADD STUDY

1 In the **Home** toolbar, click **Windows** and choose **Add Study**.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies** > **Eigenfrequency**.

4 Right-click and choose **Add Study**.

STUDY 3

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Study 3** click **Step 1: Eigenfrequency**.

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

3 From the **Search method around shift** list, choose **Larger real part**.

4 In the **Search for eigenfrequencies around shift** text field, type `200[Hz]`.

5 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** checkbox.

6 In the tree, select **Component 1 (comp1)** > **Solid Mechanics (solid)** > **Linear Elastic Material 2** > **Viscoelasticity 1**.

7 Right-click and choose **Disable**.

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

Plot and compare the results of all three eigenfrequency calculations.

RESULTS

Global 3

1 In the **Model Builder** window, under **Results** > **Eigenfrequency** right-click **Global 2** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study 3/Solution 3 (sol3)**.

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

Legends
`Approximation`

5 In the **Eigenfrequency** toolbar, click **Plot**.