Created in COMSOL Multiphysics 5.6

Viscoelastic Structural Damper — Transient Analysis

Introduction

The model studies the forced response of a typical viscoelastic damper. Damping elements involving layers of viscoelastic materials are often used for reduction of seismic and wind induced vibrations in buildings and other tall structures. The common feature is that the frequency of the forced vibrations is low.

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.

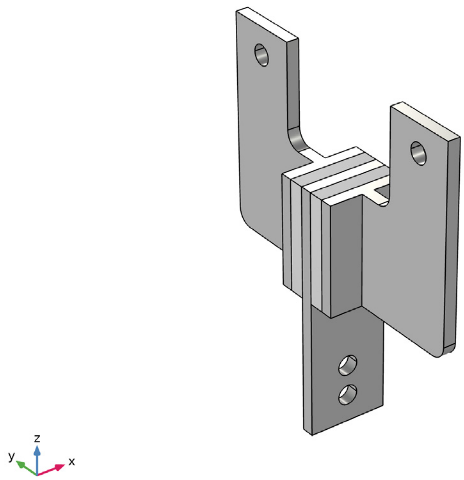
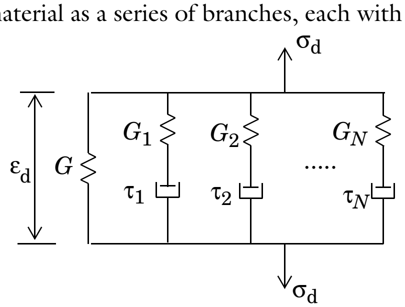


Figure 1: Viscoelastic damping element.

The viscoelastic layers are modeled with the generalized Maxwell model available in COMSOL Multiphysics. The generalized Maxwell model represents the viscoelastic material as a series of branches, each with a spring-dashpot pair.



Eighteen viscoelastic branches guarantee accurate representation of the material behavior for different excitation frequencies, when the damper is subjected to forced vibration. The values of the shear moduli and relaxation times for each branch are available in Ref. 1. They are summarized in the following table:

TABLE 1: MODEL DATA FOR THE VISCOELASTIC DAMPER MODEL.

PROPERTY	VALUE	DESCRIPTION
K	40 GPa	Bulk modulus
G	58.6 kPa	Long time shear modulus
ρ	1.06 g/cm^3	Density
G_1	13.3 MPa	Shear modulus, branch 1
τ_1	10^{-7} s	Relaxation time, branch 1
G_2	286 MPa	Shear modulus, branch 2
τ_2	10^{-6} s	Relaxation time, branch 2
G_3	291 MPa	Shear modulus, branch 3
τ_3	$3.16\cdot10^{-6}$ s	Relaxation time, branch 3
G_4	212 MPa	Shear modulus, branch 4
τ_4	10^{-5} s	Relaxation time, branch 4
G_5	112 MPa	Shear modulus, branch 5
τ_5	$3.16\cdot10^{-5}$ s	Relaxation time, branch 5
G_6	61.6 MPa	Shear modulus, branch 6
τ_6	10^{-4} s	Relaxation time, branch 6
G_7	29.8 MPa	Shear modulus, branch 7
τ_7	$3.16\cdot10^{-4}$ s	Relaxation time, branch 7
G_8	16.1 MPa	Shear modulus, branch 8
τ_8	10^{-3} s	Relaxation time, branch 8
G_9	7.83 MPa	Shear modulus, branch 9
τ_9	$3.16\cdot10^{-3}$ s	Relaxation time, branch 9
G_{10}	4.15 MPa	Shear modulus, branch 10
τ_{10}	10^{-2} s	Relaxation time, branch 10
G_{11}	2.03 MPa	Shear modulus, branch 11
τ_{11}	$3.16\cdot10^{-2}$ s	Relaxation time, branch 11
G_{12}	1.11 MPa	Shear modulus, branch 12
τ_{12}	0.1 s	Relaxation time, branch 12
G_{13}	0.491 MPa	Shear modulus, branch 13

TABLE 1: MODEL DATA FOR THE VISCOELASTIC DAMPER MODEL.

PROPERTY	VALUE	DESCRIPTION
τ_{13}	0.316 s	Relaxation time, branch 13
G_{14}	0.326 MPa	Shear modulus, branch 14
τ_{14}	1 s	Relaxation time, branch 14
G_{15}	$8.25 \cdot 10^{-2}$ MPa	Shear modulus, branch 15
τ_{15}	3.16 s	Relaxation time, branch 15
G_{16}	0.126 MPa	Shear modulus, branch 16
τ_{16}	10 s	Relaxation time, branch 16
G_{17}	$3.73 \cdot 10^{-2}$ MPa	Shear modulus, branch 17
τ_{17}	100 s	Relaxation time, branch 17
G_{18}	$1.18 \cdot 10^{-2}$ MPa	Shear modulus, branch 18
τ_{18}	1000 s	Relaxation time, branch 18

One of the mounting elements is fixed; the other two are loaded with periodic forces with a frequency of 3 Hz.

Results and Discussion

Figure 2 shows the results of the transient computations after two seconds of forced vibrations.

The typical viscoelastic hysteresis loops for a point within one of the mounting elements are shown in Figure 3.

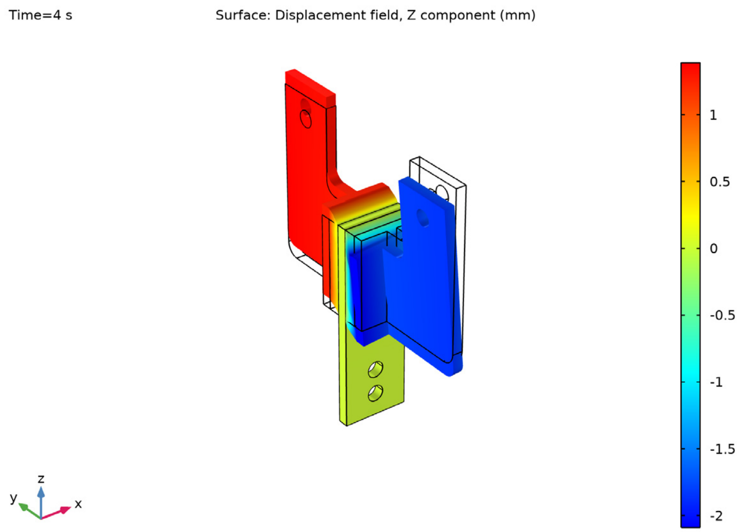


Figure 2: Displacement field.

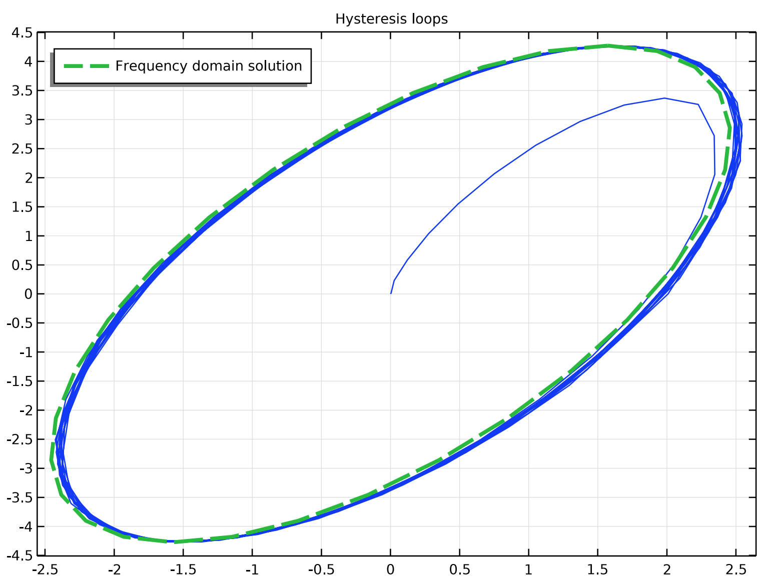


Figure 3: Displacement vs. applied force.

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_transient`

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>Time Dependent**.

6 Click **Done**.

GEOMETRY 1

Import the predefined geometry from a file.

Import 1 (imp1)

1 In the **Home** toolbar, click **Import**.

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

3 Click **Browse**.

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

5 Click **Import**.

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

SOLID MECHANICS (SOLID)

Linear Elastic Material 2

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** and choose **Material Models>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 4 only.

Viscoelasticity 1

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

Since there are 18 branches in this material model, the data has been collected in a text file which you can load.

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

3 Click to select row number 1 in the table.

4 Click **Delete**.

5 Click **Load from File**.

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

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 **Add to Component** in the window toolbar.

MATERIALS

Steel AISI 4340 (mat1)

Select Domains 1, 3, and 5 only.

Viscoelastic

1 In the **Model Builder** window, 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 4 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 **Home** toolbar, click **Add Material** to close the **Add Material** window.
6 Click the **Show Material Color and Texture** button in the **Graphics** toolbar.

DEFINITIONS

Ramp 1 (rm1)

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

Create a ramp function to apply the loads smoothly.

2 In the **Settings** window for **Ramp**, locate the **Parameters** section.
3 In the **Location** text field, type `2e-2`.
4 In the **Slope** text field, type `10`.
5 Click to expand the **Smoothing** section. Select the **Size of transition zone at start** check box.
6 In the associated text field, type `0.01`.
7 Locate the **Parameters** section. In the **Location** text field, type `0.02`.
8 Select the **Cutoff** check box.
9 Locate the **Smoothing** section. Select the **Size of transition zone at cutoff** check box.
10 In the associated text field, type `0.01`.

SOLID MECHANICS (SOLID)

Fixed Constraint 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.
2 Select Boundaries 24–27 only.

Prescribed Displacement 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Prescribed Displacement**.

2 Select Boundaries 40 and 41 only.

3 In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.

4 Select the **Prescribed in x direction** check box.

5 Select the **Prescribed in y direction** check box.

Boundary Load 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.

2 Select Boundaries 40 and 41 only.

3 In the **Settings** window for **Boundary Load**, locate the **Force** section.

4 Specify the $\mathbf{F}_A$ vector as

0	x
0	y
8.5[MPa]*sin(pi/2+2*pi*t*3[Hz])*rm1(t[1/s])	z

Prescribed Displacement 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Prescribed Displacement**.

2 Select Boundaries 32 and 33 only.

3 In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.

4 Select the **Prescribed in y direction** check box.

Boundary Load 2

1 In the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.

2 Select Boundaries 32 and 33 only.

3 In the **Settings** window for **Boundary Load**, locate the **Force** section.

4 Specify the $\mathbf{F}_A$ vector as

0.5[MPa]*sin(2*pi*t*3[Hz])*rm1(t[1/s])	x
0	y
8.5[MPa]*sin(2*pi*t*3[Hz])*rm1(t[1/s])	z

DEFINITIONS

Set up a nonlocal integration coupling to compute the total force applied to one of the mounting holes. You configure the integration to be performed in the material frame, because this is the frame used within the Solid Mechanics interface.

Integration 1 (intop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.
2 In the **Settings** window for **Integration**, locate the **Advanced** section.
3 From the **Frame** list, choose **Material (X, Y, Z)**.
4 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.
5 Select Boundaries 32 and 33 only.

Variables 1

1 In the **Definitions** toolbar, click **Local Variables**.
2 In the **Settings** window for **Variables**, locate the **Variables** section.
3 In the table, enter the following settings:

Name	Expression	Unit	Description
Fz1	intop1(solid.FperAreaz)	N	Total force, Z component

MESH 1

Mesh the side surfaces of the viscoelastic layers and then sweep the resulting mesh into the layers.

Mapped 1

1 In the **Mesh** toolbar, click **Boundary** and choose **Mapped**.
2 Select Boundaries 6 and 20 only.
3 In the **Settings** window for **Mapped**, click **Build Selected**.

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 Domains 2 and 4 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 3.

Swept 1

1 In the **Model Builder** window, right-click **Swept 1** and choose **Build Selected**.

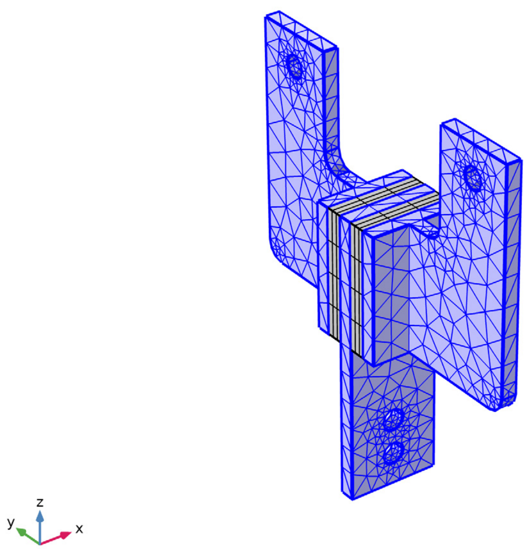
Mesh the rest of the geometry using a free tetrahedral mesh.

Free Tetrahedral 1

1 In the **Mesh** toolbar, click **Free Tetrahedral**.

2 In the **Settings** window for **Free Tetrahedral**, click **Build Selected**.

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



STUDY 1

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 1** click **Step 1: Time Dependent**.

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

3 In the **Output times** text field, type `range(0,0.01,4)`.

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Dependent Variables 1** node, then click **Auxiliary pressure (comp1.solid.pw)**.

4 In the **Settings** window for **Field**, locate the **Scaling** section.

5 In the **Scale** text field, type `1e6`.

Excluding algebraic error estimates allows the solver to take larger time steps.

6 In the **Model Builder** window, click **Time-Dependent Solver 1**.

7 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.

8 Find the **Algebraic variable settings** subsection. From the **Error estimation** list, choose **Exclude algebraic**.

RESULTS

Before computing the solution, set up a displacement plot that will be displayed and updated after every time step of the transient analysis.

3D Plot Group 1

In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

Surface 1

1 Right-click **3D Plot Group 1** and choose **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Displacement>Displacement field - m>w - Displacement field, Z component**.

3 Locate the **Expression** section. From the **Unit** list, choose **mm**.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **RainbowLight**.

Deformation 1

Right-click **Surface 1** and choose **Deformation**.

STUDY 1

Step 1: Time Dependent

1 In the **Settings** window for **Time Dependent**, click to expand the **Results While Solving** section.

2 Select the **Plot** check box.

Solution 1 (sol1)

In the **Model Builder** window, under **Study 1>Solver Configurations** right-click **Solution 1 (sol1)** and choose **Compute**.

RESULTS

The computed solution should closely resemble that shown in Figure 2.

To plot the displacement vs. applied force, follow these steps:

Table 1

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

Import the frequency domain solution.

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

3 Click **Import**.

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

1D Plot Group 2

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

2 In the **Settings** window for **1D Plot Group**, click to expand the **Title** section.

3 From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Hysteresis loops` .

Point Graph 1

1 Right-click **1D Plot Group 2** and choose **Point Graph**.

2 Select Point 23 only.

3 In the **Settings** window for **Point Graph**, locate the **x-Axis Data** section.

4 From the **Parameter** list, choose **Expression**.

5 Click **Replace Expression** in the upper-right corner of the **x-Axis Data** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Displacement>Displacement field - m>w - Displacement field, Z component**.

6 Locate the **x-Axis Data** section. From the **Unit** list, choose **mm**.

7 Locate the **y-Axis Data** section. In the **Expression** text field, type `Fz1`.

8 From the **Unit** list, choose **kN**.

Table Graph 1

1 In the **Model Builder** window, right-click **1D Plot Group 2** and choose **Table Graph**.

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

3 Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

4 Set the **Width** value to **3**.

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

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

7 In the table, enter the following settings:

Legends
`Frequency domain solution`

1D Plot Group 2

1 In the **Model Builder** window, click **1D Plot Group 2**.

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

3 From the **Position** list, choose **Upper left**.

4 In the **1D Plot Group 2** toolbar, click **Plot**.