Heat Generation in a Vibrating Structure

Introduction

When a structure is subjected to vibrations of high frequency, a significant amount of heat can be generated within the structure because of mechanical losses in the material such as, for example, viscoelastic effects.

In this example, you model the slow rise of the temperature in a vibrating beam-like structure. You use a transient heat-transfer problem with source term which represent the heat generation due to mechanical losses, and which is computed from a structural analysis performed in the frequency domain.

Model Definition

The beam consists of two layers made of aluminum and titanium, respectively, with the corresponding loss factors 0.001 and 0.005. One end of the beam is fixed, and the other one is subjected to periodic loading in z-direction, which is represented in the frequency domain as $F_z \exp(j\omega t)$, where j is the imaginary unit, and the angular frequency is

$$\omega = 2\pi f$$

The excitation frequency $f = 7767$ Hz and the load magnitude $F_z = 1.7$ MPa are used in this model example.

The temperature rise is given by the heat-transfer equation

$$\rho C_p \frac{\partial T}{\partial t} - \nabla \cdot (k \nabla T) = Q_h$$

where k is the thermal conductivity, and the volumetric heat capacity ρC_p is independent of the temperature in accordance with the Dulong-Petit law.

Note that T represents the temperature averaged over the time period $2\pi/\omega$. The heat source

$$Q_h = \frac{1}{2}\omega\eta \text{Real}[\varepsilon : \text{Conj}(C : \varepsilon)]$$

presents the internal work of the nonelastic (for example, viscous) forces over the period. In the above expression, η is the loss factor, ε is the strain tensor, and $\mathbf{C}$ is the elasticity tensor. The term is computed from a structural analysis performed in the frequency domain.

The initial state at time $t = 0$ is stress-free, and the initial temperature is 293.15 K over the entire beam.

Use the following boundary conditions:

- At the fixed end, use the temperature condition $T = 293.15$ K.
- At the end subjected to forcing, use the thermal insulation condition.
- The boundary between the layers of different material is an internal boundary.
- At all other boundaries, use the convective cooling condition:

$$\mathbf{n} \cdot (-k\nabla T) = h(T - T_{\text{ext}}) \tag{1}$$

where $h = 5$ W/(m^2·K) is the heat transfer coefficient and $T_{\text{ext}} = 293.15$ K is the external temperature.

For the simulation, apply a periodic loading in the z direction of magnitude 1.7 MPa and frequency 7767 Hz at the free end of the beam for 2 seconds, keeping the fixed end and the structure environment at a constant temperature of 300 K during the process.

Results and Discussion

Figure 1 displays the temperature distribution at the end of the simulated 2-second forced vibrations. As the figure shows, the maximum temperature rise in the beam is about 0.2 K.

 ©2012 COMSOL

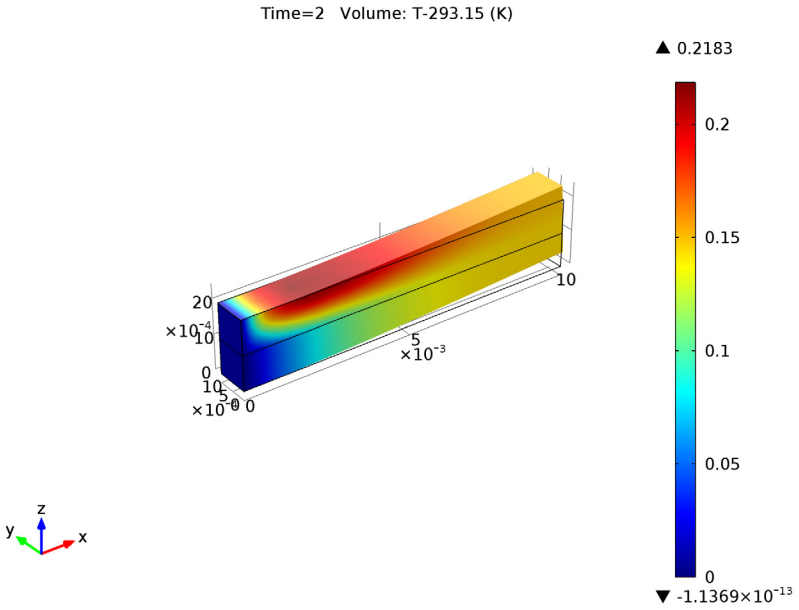

Figure 1: Temperature increase in the beam after 2 seconds of forced vibrations.

Model Library path: `Structural_Mechanics_Module/Thermal-Structure_Interaction/vibrating_beam`

Modeling Instructions

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Click **Next**.

3 In the **Add physics** tree, select **Heat Transfer>Heat Transfer in Solids (ht)**.

4 Click **Add Selected**.

5 In the **Add physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.

6 Click **Add Selected**.

7 Click **Next**.

8 Find the **Studies** subsection. In the tree, select **Preset Studies for Selected Physics>Time Dependent**.

9 Click **Finish**.

GEOMETRY 1

Block 1

1 In the **Model Builder** window, under **Model 1** right-click **Geometry 1** and choose **Block**.

2 In the **Block** settings window, locate the **Size and Shape** section.

3 In the **Depth** edit field, type `0.0010`.

4 In the **Height** edit field, type `0.0010`.

5 In the **Width** edit field, type `0.010`.

Block 2

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Block**.

2 In the **Block** settings window, locate the **Size and Shape** section.

3 In the **Depth** edit field, type `0.0010`.

4 In the **Height** edit field, type `0.0010`.

5 In the **Width** edit field, type `0.010`.

6 Locate the **Position** section. In the **z** edit field, type `0.0010`.

7 In the **Model Builder** window, right-click **Geometry 1** and choose **Build All**.

MATERIALS

1 In the **Model Builder** window, under **Model 1** right-click **Materials** and choose **Open Material Browser**.

2 In the **Material Browser** window, locate the **Materials** section.

3 In the tree, select **Built-In>Aluminum**.

4 Right-click and choose **Add Material to Model** from the menu.

Aluminum

1 In the **Model Builder** window, under **Model 1>Materials** click **Aluminum**.

2 Select Domain 1 only.

3 In the **Material Browser** window, locate the **Materials** section.

4 In the tree, select **Built-In>Titanium beta-21S**.

5 Right-click and choose **Add Material to Model** from the menu.

 ©2012 COMSOL

Titanium beta-21S

1 In the **Model Builder** window, under **Model 1>Materials** click **Titanium beta-21S**.

2 Select Domain 2 only.

SOLID MECHANICS

You need to set up the solid mechanics equation to the frequency-domain form because the study type will be set to time dependent, which must be applied to the heat transfer only.

1 In the **Model Builder** window, under **Model 1** click **Solid Mechanics**.

2 In the **Solid Mechanics** settings window, click to expand the **Equation** section.

3 From the **Equation form** list, choose **Frequency domain**.

4 From the **Frequency** list, choose **User defined**. In the *f* edit field, type `7767`.

5 Right-click **Model 1>Solid Mechanics** and choose the boundary condition **Fixed Constraint**.

Fixed Constraint 1

Select Boundaries 1 and 4 only.

Boundary Load 1

1 In the **Model Builder** window, right-click **Solid Mechanics** and choose the boundary condition **Boundary Load**.

2 Select Boundaries 10 and 11 only.

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

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

`0`	x
`0`	y
`1.7[MPa]`	z

Damping 1

1 In the **Model Builder** window, right-click **Model 1>Solid Mechanics>Linear Elastic Material 1** and choose the domain setting **Damping**.

2 Select Domain 1 only.

3 In the **Damping** settings window, locate the **Damping Settings** section.

4 From the **Damping type** list, choose **Isotropic loss factor**.

5 From the η_s list, choose **User defined**. In the associated edit field, type `0.001`.

Damping 2

1 In the **Model Builder** window, right-click **Model 1>Solid Mechanics>Linear Elastic Material 1** and choose the domain setting **Damping**.

2 Select Domain 2 only.

3 In the **Damping** settings window, locate the **Damping Settings** section.

4 From the **Damping type** list, choose **Isotropic loss factor**.

5 From the η_s list, choose **User defined**. In the associated edit field, type `0.005`.

HEAT TRANSFER IN SOLIDS

Temperature 1

1 In the **Model Builder** window, right-click **Model 1>Heat Transfer in Solids** and choose the boundary condition **Temperature**.

2 Select Boundaries 1 and 4 only.

Heat Flux 1

1 In the **Model Builder** window, right-click **Heat Transfer in Solids** and choose the boundary condition **Heat Flux**.

2 In the **Heat Flux** settings window, locate the **Heat Flux** section.

3 Click the **Inward heat flux** button.

4 In the h edit field, type `5`.

5 Select Boundaries 2, 3, 5, and 7–9 only.

Heat Source 1

1 In the **Model Builder** window, right-click **Heat Transfer in Solids** and choose the domain setting **Heat Source**.

2 Select Domains 1 and 2 only.

3 In the **Heat Source** settings window, locate the **Heat Source** section.

4 From the Q list, choose **Total power dissipation density (solid/lemm1)**.

This choice models the heat generated by the vibrations in the structure.

MESH 1

In the **Model Builder** window, under **Model 1** right-click **Mesh 1** and choose **Build All**.

 ©2012 COMSOL

STUDY 1

Step 1: Time Dependent

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Time Dependent**.

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

3 In the **Times** edit field, type `range(0,0.05,2)`.

4 In the **Model Builder** window, right-click **Study 1** and choose **Show Default Solver**.

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

1 In the **Model Builder** window, right-click **Results** and choose **3D Plot Group**.

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

3 In the **Volume** settings window, locate the **Expression** section.

4 In the **Expression** edit field, type `T-293.15`.

5 Right-click **Results>3D Plot Group 1>Volume 1** and choose **Deformation**.

STUDY 1

Step 1: Time Dependent

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

2 In the **Time Dependent** settings window, click to expand the **Results While Solving** section.

3 Select the **Plot** check box.

Solver 1

You need to enable complex values because they are used in the solid mechanics equations, which you manually reconfigured for the frequency-domain analysis.

1 In the **Model Builder** window, under **Study 1>Solver Configurations>Solver 1** click **Time-Dependent Solver 1**.

2 In the **Time-Dependent Solver** settings window, click to expand the **Advanced** section.

3 Select the **Allow complex numbers** check box.

4 In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

©2012 COMSOL

RESULTS

3D Plot Group 1

1 Click the **Zoom Out** button on the Graphics toolbar.

The computed solution should closely resemble that shown in Figure 1.

 ©2012 COMSOL