Created in COMSOL Multiphysics 6.1

Heat Generation in a Vibrating Structure

This model is licensed under the COMSOL Software License Agreement 6.1.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

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 represents the heat generation due to mechanical losses. The simulation is based on 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 the 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 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

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

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 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 periodic force, use the thermal insulation condition.
- The boundary between the layers of different materials is an interior boundary.
- At all other boundaries, use the convective cooling condition:

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

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

The stress solution computed in frequency domain is shown in Figure 1. It appears that the maximum stresses are located at the fixed end. As consequence more energy is dissipated at this location.

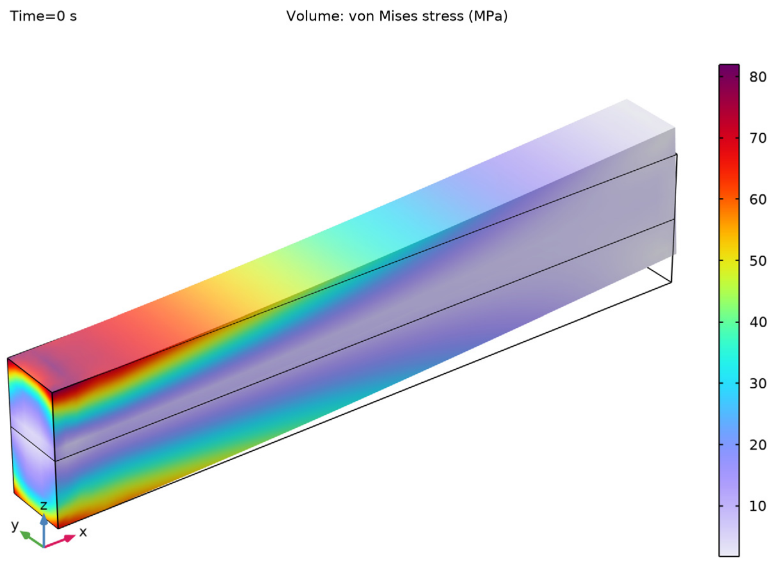

Figure 1: von Mises stress from the frequency domain solution

Figure 2 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 0.18 K.

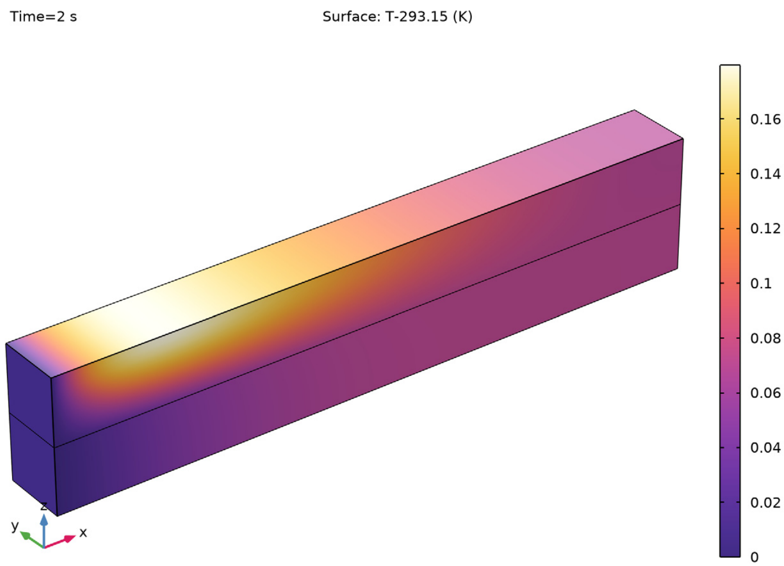

Figure 2: Temperature increase in the beam after 2 seconds of forced vibrations.

The maximum temperature increase is plotted in Figure 3, it shows that the maximum temperature increases in the first time steps, then starts to stabilize around the end time.

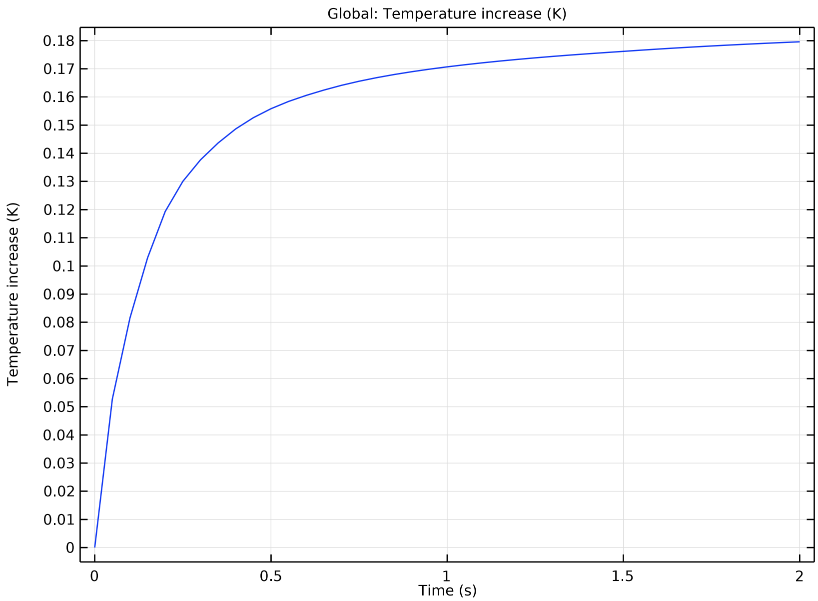

Figure 3: Maximum temperature increase with time

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

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 **Heat Transfer>Heat Transfer in Solids (ht)**.

3 Click **Add**.

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

5 Click **Add**.

6 Click **Study**.

7 In the **Select Study** tree, select **General Studies>Time Dependent**.

8 Click **Done**.

GEOMETRY 1

Block 1 (blk1)

1 In the **Geometry** toolbar, click **Block**.

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

3 In the **Width** text field, type `0.01`.

4 In the **Depth** text field, type `0.001`.

5 In the **Height** text field, type `0.001`.

Block 2 (blk2)

1 In the **Geometry** toolbar, click **Block**.

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

3 In the **Width** text field, type `0.01`.

4 In the **Depth** text field, type `0.001`.

5 In the **Height** text field, type `0.001`.

6 Locate the **Position** section. In the **z** text field, type `0.001`.

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

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>Aluminum**.

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

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

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

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

MATERIALS

Aluminum (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Aluminum (mat1)**.

2 Select Domain 1 only.

Titanium beta-21S (mat2)

1 In the **Model Builder** window, click **Titanium beta-21S (mat2)**.

2 Select Domain 2 only.

SOLID MECHANICS (SOLID)

You need to set up the Solid Mechanics equation form to frequency-domain, since the study type will be set to time dependent. The time dependent equations should be applied to the heat transfer physics only.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

2 In the **Settings** window for **Solid Mechanics**, 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 text field, type `7767`.

Fixed Constraint 1

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

2 Select Boundaries 1 and 4 only.

Boundary Load 1

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

2 Select Boundaries 10 and 11 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
1.7[MPa]	z

Linear Elastic Material 1

In the **Model Builder** window, click **Linear Elastic Material 1**.

Damping 1

1 In the **Physics** toolbar, click **Attributes** and choose **Damping**.
2 Select Domain 1 only.
3 In the **Settings** window for **Damping**, 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 text field, type `0.001`.

Linear Elastic Material 1

In the **Model Builder** window, click **Linear Elastic Material 1**.

Damping 2

1 In the **Physics** toolbar, click **Attributes** and choose **Damping**.
2 Select Domain 2 only.
3 In the **Settings** window for **Damping**, 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 text field, type `0.005`.

HEAT TRANSFER IN SOLIDS (HT)

In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Solids (ht)**.

Temperature 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Temperature**.
2 Select Boundaries 1 and 4 only.

Heat Flux 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.
2 In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.
3 From the **Flux type** list, choose **Convective heat flux**.
4 In the h text field, type `5`.
5 Select Boundaries 2, 3, 5, and 7–9 only.

Heat Source 1

1 In the **Physics** toolbar, click **Domains** and choose **Heat Source**.
2 In the **Settings** window for **Heat Source**, locate the **Domain Selection** section.
3 From the **Selection** list, choose **All domains**.

4 Locate the **Heat Source** section. From the Q_0 list, choose **Total power dissipation density (solid)**.

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

DEFINITIONS

Maximum 1 (maxop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Maximum**.

Add a maximum operator to enable the calculation of maximum temperature after computation.

2 In the **Settings** window for **Maximum**, locate the **Source Selection** section.

3 From the **Selection** list, choose **All domains**.

MESH 1

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

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 From the **Element size** list, choose **Extra fine**.

Swept 1

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

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

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(0,0.05,2)`.

Before computing the solution, generate the default plots.

4 In the **Model Builder** window, right-click **Study 1** and choose **Get Initial Value for Step**.

RESULTS

Surface

1 In the **Model Builder** window, expand the **Temperature (ht)** node, then click **Surface**.

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

3 In the **Expression** text field, type `T-293.15`.

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**, click to expand the **Results While Solving** section.

3 Select the **Plot** check box.

Solver Configurations

In the **Model Builder** window, expand the **Study 1>Solver Configurations** node.

Solution 1 (sol1)

1 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)** node, then click **Time-Dependent Solver 1**.

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

3 From the **Steps taken by solver** list, choose **Intermediate**.

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

4 Click to expand the **Advanced** section. Select the **Allow complex numbers** check box.

5 In the **Home** toolbar, click **=** **Compute**.

RESULTS

Temperature (ht)

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

Volume 1

1 In the **Model Builder** window, expand the **Stress (solid)** node, then click **Volume 1**.

2 In the **Settings** window for **Volume**, locate the **Expression** section.

3 From the **Unit** list, choose **MPa**.

4 In the **Stress (solid)** toolbar, click **Plot**.

Temperature Increase

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

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

Global 1

1 In the **Temperature Increase** toolbar, click **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
maxop1(T-293.15[K])	K	Temperature increase

4 Click to expand the **Legends** section. Clear the **Show legends** check box.