

# COMSOL MULTIPHYSICS®

## Thermoelastic Damping in a MEMS Resonator

SOLVED WITH COMSOL MULTIPHYSICS 3.5a

# Thermoelastic Damping in a MEMS Resonator

## *Introduction*

---

A high Q value is a key factor of a MEMS resonator. It is essential that the resonator vibrates consistently at the desired frequency and that it requires as little energy as possible to maintain its vibration. These features can be characterized by the resonator's Q value, which is a measure of the sharpness of its spectrum's peak. There are several ways to define the Q value, for example:

$$Q = \frac{2\pi W_0}{\Delta W} = \frac{\omega_0}{2\delta} = \frac{\omega_0}{\Delta\omega} \quad (1)$$

where  $W_0$  is the total stored vibrational energy,  $\Delta W$  is the energy lost per cycle,  $\omega_0$  is the natural angular frequency,  $\delta$  is the damping factor (vibration decays exponentially with  $\delta t$ ), and  $\Delta\omega$  is the half power width of the spectrum.

In order to improve the resonator, the designer needs to consider all aspects that produce damping and noise to the system. For example, resonators are usually run in vacuum to minimize effects of air and squeeze-film damping.

Thermoelastic damping (Ref. 1, Ref. 2, and Ref. 3) is an important factor that the resonator designer needs to address. It is a result of a phenomenon called thermoelastic friction, which takes place when you subject any material to cyclic stress. The stress results in deformation, and the required energy is mostly stored as internal potential energy. However, materials heat under compressive stress and cool under tensile stress. Thus, due to the heat flow from warmer to cooler regions energy is also lost as nonrecoverable thermal energy. The amount of thermoelastic friction and damping depends on the rate of this energy loss. The magnitude of the energy loss depends on the vibrational frequency and on the structure's thermal relaxation time constant, which is the effective time the material requires to relax after an applied constant stress or strain. Therefore, the effect of thermoelastic dissipation, and consequently the damping, is most pronounced when the vibration frequency is close to the thermal relaxation frequency.

For simple structures, researchers have developed analytical expressions to estimate thermoelastic damping. According to Zener (Ref. 1 and Ref. 2), you can calculate the Q value for a resonator with a single thermal mode by:

$$\frac{1}{Q} = \left( \frac{E\alpha^2 T_0}{\rho C_p} \right) \left( \frac{\omega\tau}{1 + (\omega\tau)^2} \right) \quad (2)$$

where  $E$  is the Young's modulus,  $\alpha$  is the thermal expansion coefficient,  $T_0$  is the resonator temperature at rest,  $\rho$  is the density,  $C_p$  is the heat capacity of the material,  $\omega$  is the vibration angular frequency, and  $\tau$  is the thermal relaxation time of the system. Thus it is easy to see that in order to have good Q value, the system needs to be designed so that  $\omega$  is as far from  $1/\tau$  as possible.

The natural frequency of a beam clamped at both ends can be calculated as (Ref. 4)

$$\omega_0 = a_0 \frac{2h}{L^2} \sqrt{\frac{E}{12\rho}} \quad (3)$$

where  $a_0$  equals 4.730;  $h$  and  $L$  are the thickness and length of the beam, respectively; and  $E$  and  $\rho$  are material parameters as above.

The thermal relaxation time of the beam is given by

$$\tau = \frac{\rho C_p h^2}{\pi^2 \kappa} \quad (4)$$

where  $\kappa$  is the thermal conductivity and other parameters are as above.

The problem is that Equations 2 through 4 are valid only for very simple structures. Therefore more advanced methods, such as FEA, are preferable.

This example shows how to model thermoelastic damping with COMSOL Multiphysics. To be able to compare with measurements and analytical expressions, this example illustrates a simple beam resonator in 2D and 3D. The Q value and natural frequency is solved with an eigenfrequency analysis that combines heat transfer and structural mechanics in one equation system. Thus the eigenmodes are thermoelastic.

This example was inspired by the work of Amy Duwel and others (Ref. 1).

## Model Definition

Figure 1 shows the geometry. The resonator is a beam of silicon with length  $400\ \mu\text{m}$ , height  $12\ \mu\text{m}$ , and width  $20\ \mu\text{m}$ . The beam is fixed at both ends, and it vibrates in a flexural mode in the  $z$  direction (that is, along the smallest dimension). The model assume that the vibration takes place in vacuum. Thus there is no transfer of heat from the free boundaries. The model also assumes that the contact boundaries are thermally insulated.

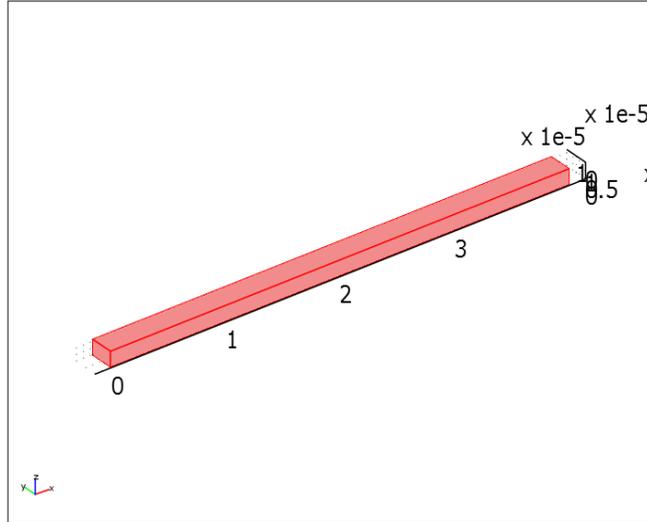


Figure 1: Geometry of the modeled beam. The beam is fixed at both ends.

Table 1 lists the physical properties of the beam material and the surroundings:

TABLE 1: MATERIAL PROPERTIES (POLYSILICON)

PROPERTY	VALUE
Young's modulus, $E$	157 GPa
Density, $\rho$	$2330\ \text{kg/m}^3$
Poisson's ratio, $\nu$	0.3
Thermal expansion coefficient, $\alpha$	$2.6 \cdot 10^{-6}\ \text{K}^{-1}$
Specific heat, $C_p$	$700\ \text{J}/(\text{kg}\cdot\text{K})$
Thermal conductivity, $k$	$90\ \text{W}/(\text{m}\cdot\text{K})$
Ambient and initial beam temperature, $T_{\text{init}}$	300 K

To gain information about the quality of the resonator, it is of interest to know its natural frequency and Q value. To do this, you run an eigenfrequency analysis to find the eigenvalues for this system. For a system with damping, the eigenvalue  $\lambda$  contains information about the natural frequency and Q value according to:

$$\begin{aligned} \omega_0 &= |\text{Im}(\lambda)| \\ Q &= \frac{|\text{Im}(\lambda)|}{2\text{Re}(\lambda)} \end{aligned} \quad (5)$$

The eigenvalues appear as complex conjugates, and  $\omega_0$  and  $Q$  are therefore given as absolute values.

At this point, to avoid any confusion it is good to note that here  $Q$  refers to the resonator's quality, whereas later in this text  $Q_{\text{heat}}$  refers to the heat source term in the heat equation.

To model thermoelastic damping, you must consider both the thermal problem and the structural problem. Furthermore, there is a 2-way coupling between them: the strain rate heats or cools the material locally, which produces thermal strains.

The relation between the material stress  $\sigma$  and strain  $\epsilon$  is given by

$$\sigma = D\epsilon_{\text{el}} = D(\epsilon - \epsilon_{\text{th}}) \quad (6)$$

where  $\epsilon_{\text{el}}$  and  $\epsilon_{\text{th}}$  are the elastic and thermal strains, respectively,  $D$  is the  $6 \times 6$  elasticity matrix, and all stresses and strains are denoted with 6-component vectors consisting of  $x$ ,  $y$ , and  $z$  normal components followed by the  $xy$ ,  $yx$ , and  $xz$  shear components.

It is possible to expand this for an isotropic material:

$$\begin{bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \sigma_{xy} \\ \sigma_{yz} \\ \sigma_{xz} \end{bmatrix} = \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1-\nu & \nu & \nu & 0 & 0 & 0 \\ \nu & 1-\nu & \nu & 0 & 0 & 0 \\ \nu & \nu & 1-\nu & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1-2\nu}{2} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1-2\nu}{2} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1-2\nu}{2} \end{bmatrix} \begin{bmatrix} \epsilon_x - \alpha(T - T_{\text{ref}}) \\ \epsilon_y - \alpha(T - T_{\text{ref}}) \\ \epsilon_z - \alpha(T - T_{\text{ref}}) \\ \epsilon_{xy} \\ \epsilon_{yz} \\ \epsilon_{xz} \end{bmatrix} \quad (7)$$

where  $T$  is the strain temperature,  $T_{\text{ref}}$  is the stress-free reference temperature, and  $\alpha$  is the thermal expansion coefficient.

The heat balance equation is

$$T \frac{\partial S}{\partial t} - \nabla \cdot (k \nabla T) = 0 \quad (8)$$

where  $k$  is the thermal conductivity. For a linear thermoelastic solid, the entropy per unit volume is:

$$S = \rho C_P \log(T/T_0) + S_{\text{elast}} \quad (9)$$

where  $T_0$  is the reference temperature, the volumetric heat capacity  $\rho C_P$  is assumed independent of the temperature, and

$$S_{\text{elast}} = \alpha_{\text{vec}} \cdot \sigma \quad (10)$$

where  $\sigma$  is the stress vector, and  $\alpha_{\text{vec}}$  is the thermal expansion vector. For an isotropic material, Equation 10 simplifies into

$$S_{\text{elast}} = \alpha(\sigma_x + \sigma_y + \sigma_z) \quad (11)$$

For small deformations and small variation of the temperature, Equation 8 is linearized as:

$$\rho C_P \frac{\partial T}{\partial t} + T_0 \frac{\partial}{\partial t} S_{\text{elast}} - \nabla \cdot (k \nabla T) = 0 \quad (12)$$

Finally, the frequency decomposition for the temperature is performed

$$T = \tilde{T}(x_i) \exp(j\omega t) \quad (13)$$

where  $j\omega = -\lambda$  is the complex angular frequency. Equation 12 gives

$$-\nabla \cdot (k \nabla \tilde{T}) = Q_{\text{heat}} \quad (14)$$

where the heat source term is

$$Q_{\text{heat}} = -j\omega(\rho C_P \tilde{T} + T_0 S_{\text{elast}}) \quad (15)$$

You model the problem with the damped eigenfrequency analysis for the beam, which is coupled with the thermal problem Equation 14.

In the eigenfrequency analysis, you use the temperature computer from Equation 14 as the strain temperature and set the strain reference temperature to zero,  $T_{\text{ref}}=0$ . This is because the solution physically corresponds to thermoelastic oscillations of small amplitude—it is initialized to zero. However,  $T_0$  in Equation 12 and Equation 15 is the actual temperature of the beam at rest.

## *Results and Discussion*

Figure 2 and Figure 3 show the eigenmodes and temperature distribution corresponding to the found eigenvalue. Solved natural frequencies and Q values are given in Table 2. Reference data, one calculated with Equation 2 to Equation 4 and the others from measurements (see Ref. 1), are also given.

TABLE 2: COMPARISON OF NATURAL FREQUENCY AND Q VALUE.

SOURCE	F0 (MHZ)	Q
3D model	0.63 MHz	9151
2D model	0.63 MHz	10,170
Equation	0.63 MHz	10,260
Measurements (Ref. 1)	0.57 MHz	10,281

The Q value given by the 3D model appear to be roughly 10% smaller than the other estimates. One reason for this difference comes from the simplifying assumption that concerns both the Zener's equation (Equation 2) and the plane stress method. For example, the plane stress method assumes that the structure is very thin and that there are no stresses perpendicular to the plane. However, looking at the 3D model in more detail, you find that stresses and their spatial derivatives have components of equal magnitude in all three dimensions. The model also assumes that the beam was perfectly fixed at its ends. Physically, this cannot happen, and allowing more loose contact lowers the natural frequency and improves the Q value.

One factor that also affects the simulated Q values is the boundary condition for the thermal equation. This example uses thermal insulation on all boundaries, but you can assume that there is a flux of heat at least from both ends of the beam. If you simply assign a constant temperature ( $T = 0$ ) to the beam ends, the Q value improves considerably. The physically correct result should be somewhere between these two estimates.

A look at Figure 2 and Figure 3 shows a temperature distribution that appears to agree with the theory: the temperature is higher near the compressive strain and lower near the tensile strain.

Note, however, that you should not take the displayed temperature range ( $-4$  K to  $4$  K) literally because the software normalizes the solution from the eigenfrequency solver.

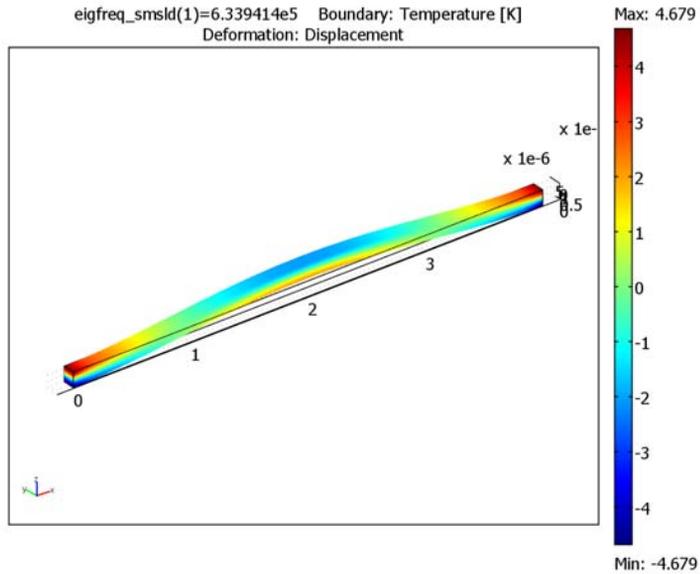


Figure 2: First eigenmode and temperature distribution of the 3D model.

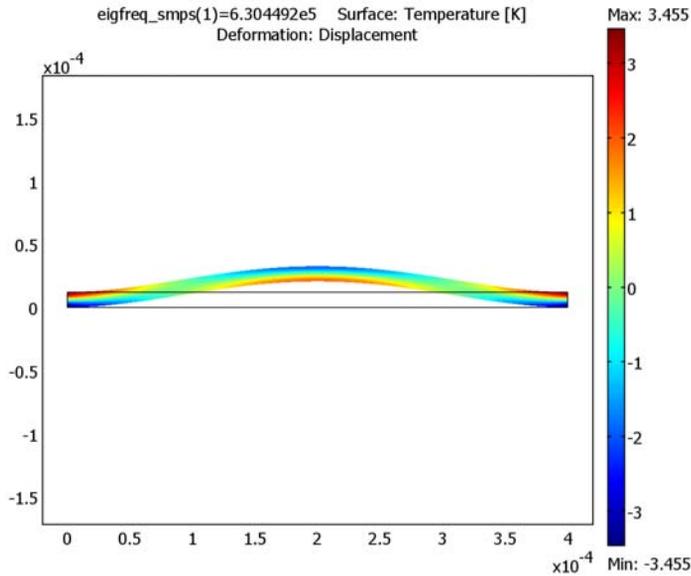


Figure 3: First eigenmode and temperature distribution of the 2D model.

### Modeling in COMSOL Multiphysics

To create a model of thermoelastic damping with COMSOL Multiphysics requires two application modes. For the 3D model, use the Solid, Stress-Strain application mode from the MEMS Module and the steady-state Conduction application mode from the base package. In 2D, use the Plane Stress application mode from the MEMS Module with the same Conduction application mode as in 3D.

In this example you guide the eigenfrequency solver to find the eigenfrequency near the expected natural frequency (Equation 3). Alternatively, you can first solve the undamped eigenfrequency  $f_0$  (select **Eigenfrequency analysis** in the **Solver Parameters** dialog box and solve only for the structural application mode). This approach is also valid for more complex geometries where you do not have an analytical expression for the estimated natural frequency.

This model, which consists of 2nd-order (structural) and 1st-order (thermal) eigenvalue problems, is numerically quite challenging. You can improve the results' accuracy by manually scaling the dependent variables. This is an approach that you can use more generally. Once you know the magnitudes of the dependent variables, go to

the **Advanced** page in the **Solver Parameters** dialog box and define manual scaling of the dependent variables.

For the 3D problem you can utilize its symmetry. The beam is truncated to 10  $\mu\text{m}$  in the width direction. Thus you get a smaller mesh size, but it also makes the problem easier to solve because the symmetry condition also prevents any flexural vibrations in this direction.

This example uses both rectangular and brick meshes. In 2D, first draw the geometry and use a mapped mesh to mesh it. In 3D, you utilize work planes by drawing a 2D view of the beam into the work plane. Then mesh it using the mapped mesh, and finally, a mesh extrusion creates the true 3D geometry.

In the damped eigenfrequency analysis, the resonator's quality factor  $Q$  is available as a predefined variable  $Q_{\text{factor}}$ . In the 3D analysis, you can use a predefined variable  $Ent$  for the elastic part of the entropy  $S_{\text{elast}}$ . Use the explicit expression given in Equation 15 for the 2D problem.

### References

---

1. A. Duwel, R.N. Candler, T.W. Kenny, and M. Varghese, *Journal of Microelectromechanical Systems*, vol. 15, no. 6, pp. 1437–1445, 2006.
2. S. Gupta, *Estimation of Thermo-Elastic Dissipation in MEMS*, MSc. Thesis, Dept. Mechanical Engineering, Indian Institute of Science, Bangalore, July 2004.
3. T.V. Roszhart, “The Effect of Thermoelastic Internal Friction on the  $Q$  of Micromachined Silicon Resonators,” *Tech. Dig. Solid-State Sens. Actuator Workshop*, Hilton Head, SC, 1990, pp. 13–16.
4. R. Lifshitz and K.L. Roukes, “Thermoelastic Damping in Micro- and Nanomechanical Systems,” *Physical Review B*, vol. 61, no. 8, 15. Feb. 2000-II.

---

**Model Library path:** MEMS\_Module/Actuator\_Models/  
thermoelastic\_damping\_2d

---

---

**Model Library path:** MEMS\_Module/Actuator\_Models/  
thermoelastic\_damping\_3d

---

### MODEL NAVIGATOR

- 1 In the Model Navigator select **3D** in the **Space dimension** list.
- 2 Click **Multiphysics**.
- 3 From the list of application modes, select **MEMS Module>Structural Mechanics>Solid, Stress-Strain>Damped eigenfrequency analysis**.
- 4 Click **Add**.
- 5 From the list of application modes select **COMSOL Multiphysics>Heat Transfer>Conduction>Steady-state analysis**.
- 6 Click **Add**.
- 7 Click **OK**.

### OPTIONS AND SETTINGS

Select **Options>Constants** and enter constants from the following table; when finished, click **OK**.

NAME	EXPRESSION	DESCRIPTION
E	157[GPa]	Young's modulus
rho	2330[kg/m <sup>3</sup> ]	Density
nu	0.3	Poisson's ratio
alpha	2.6e-6[1/K]	Thermal expansion coefficient
Cp	700[J/(kg*K)]	Specific heat capacity
k	90[W/(m*K)]	Thermal conductivity
T0	300[K]	Ambient and initial beam temperature
f0	$4.730^2 * \sqrt{E / (\rho * 12)} * 12[\mu\text{m}] / (2 * \pi * (400[\mu\text{m}])^2)$	Theoretical estimate for the natural frequency

The constant f0 serves as an initial guess for the eigenfrequency solver.

### GEOMETRY MODELING

- 1 Select **Draw>Work-Plane Settings**.
- 2 On the **Quick** page select the **z-x** check box.
- 3 Click **OK**. The 2D work plane becomes active.
- 4 Shift-click the **Rectangle/Square** button on the Draw toolbar.

- 5 In the **Width** edit field type  $12\text{e-}6$ , and in the **Height** edit field type  $400\text{e-}6$ .
- 6 Click **OK**, then click the **Zoom Extents** button on the Main toolbar.

### MESH GENERATION

- 1 In the work plane select **Mesh>Mapped Mesh Parameters**.
- 2 In the **Subdomain** page select the first subdomain and click the **Boundary** tab.
- 3 Select Boundary 1.
- 4 Select the **Constrained edge element distribution** check box, then enter 25 in the **Number of edge elements** edit field.
- 5 Select Boundary 2.
- 6 Select the **Constrained edge element distribution** check box, then enter 4 in the **Number of edge elements** edit field.
- 7 Click **Remesh**, then click **OK**.
- 8 Select **Mesh>Extrude Mesh**.
- 9 In the **Distance** edit field type  $10\text{e-}6$ .
- 10 Change to the **Mesh** page
- 11 In the **Number of element layers** edit field type 2.
- 12 Click **OK**.

The 3D geometry should open automatically.

### PHYSICS SETTINGS

#### *Subdomain Settings*

- 1 From the **Multiphysics** menu select **I Geom I: Solid, Stress-Strain (smsld)**.
- 2 Select **Physics>Subdomain Settings**.
- 3 Select Subdomain 1, and in the **Material** page enter settings according to the following table:

PROPERTY	VALUE
E	E
$\nu$	nu
$\alpha$	alpha
$\rho$	rho

- 4 Click the **Load** tab.

- 5 Select **Include thermal expansion** and type T for **Temp** and 0 for **Tempref**.
- 6 Change to the **Damping** page.
- 7 From the **Damping model** list select **No damping**.
- 8 Click **OK**.
- 9 From the **Multiphysics** menu select **2 Geom I: Heat Transfer by Conduction (ht)**.
- 10 Select **Physics>Subdomain Settings**.
- 11 See that Subdomain 1 is selected and in the **Material** page enter settings from the following table:

PROPERTY	VALUE
k (isotropic)	k
Q	- j omega_smsld*(rho*Cp*T+T0*Ent_smsld)

- 12 Click **OK**.

#### *Boundary Conditions*

- 1 From the **Multiphysics** menu, select **1 Geom I: Solid, Stress-Strain (smsld)**.
- 2 Select **Physics>Boundary Settings**.
- 3 Select Boundaries 1 and 6.
- 4 On the **Constraint** page, set **Constraint condition** to **Fixed**.
- 5 Select Boundary 2.
- 6 Set **Constraint condition** list to **Symmetry plane**.
- 7 Click **OK**.
- 8 From the **Multiphysics** menu, select **2 Geom I: Heat Transfer by Conduction (ht)**.
- 9 Select **Physics>Boundary Settings**.
- 10 Verify that **Thermal insulation** is the default value.
- 11 Click **OK**.

#### **COMPUTING THE SOLUTION**

- 1 Choose **Solve>Solver Parameters**.
- 2 Go to the **General** page.
- 3 Type 1 for **Desired number of eigenfrequencies**.
- 4 Type f0 for **Search for eigenfrequencies around**.
- 5 Go to the **Advanced** page.
- 6 Locate the **Scaling of variables** area and choose **Manual** for **Type of scaling**.

**7** In the **Manual scaling** edit field, type  $u\ 1e-4\ v\ 1e-4\ w\ 1e-4\ T\ 1$ . Click **OK**.

**8** Click the **Solve** button on the Main toolbar.

The solver finds a solution with an eigenfrequency of approximately 63,394 Hz.

#### **POSTPROCESSING AND VISUALIZATION**

To generate the plot in Figure 2 follow these steps:

**1** Click the **Plot Parameters** button on the Main toolbar.

**2** On the **General Page** verify that the **Boundary**, **Deformed shape**, and **Geometry edges** check boxes are selected and that all others are cleared.

**3** Go to the **Boundary** page and select **Heat Transfer by Conduction (ht)>Temperature** from the **Predefined quantities** list.

**4** Click **OK**.

You can see the eigenfrequency in the postprocessing plot. Use the **Global Data Display** dialog box to calculate the Q value:

**1** Choose **Postprocessing>Data Display>Global**.

**2** In the **Expression** edit field, type `Qfactor_sms1d`.

**3** Click **Apply**. You find the Q value in the message log.

### *Modeling Using the Graphical User Interface—2D Example*

---

Start the model in the Model Navigator.

**1** In the Model Navigator select **2D**.

**2** Click **Multiphysics**.

**3** From the list of application modes select **MEMS Module>Structural Mechanics>Plane Stress>Damped eigenfrequency analysis**.

**4** Click **Add**.

**5** From the list of application modes select **COMSOL Multiphysics>Heat Transfer>Conduction>Steady-state analysis**.

**6** Click **Add**, then click **OK**.

## OPTIONS AND SETTINGS

Choose **Options>Constants** and enter constants from the following table:

NAME	EXPRESSION	DESCRIPTION
E	157[GPa]	Young's modulus
rho	2330[kg/m <sup>3</sup> ]	Density
nu	0.3	Poisson's ratio
alpha	2.6e-6[1/K]	Thermal expansion coefficient
Cp	700[J/(kg*K)]	Specific heat capacity
k	90[W/(m*K)]	Thermal conductivity
T0	300[K]	Ambient and initial beam temperature
f0	$4.730^2 * \sqrt{E / (\rho * l^2)} * l / (2 * \pi * (400[\mu\text{m}])^2)$	Theoretical estimate for the natural frequency

The constant f0 serves as an initial guess for the eigenfrequency solver.

## GEOMETRY MODELING

- 1 Shift-click the **Rectangle/Square** button on the Draw toolbar.
- 2 In the **Width** edit field type 400e-6, and in the **Height** edit field type 12e-6.
- 3 Click **OK**, then click the **Zoom Extents** button on the Main toolbar.

## MESH GENERATION

- 1 In the work plane select **Mesh>Mapped Mesh Parameters**.
- 2 On the **Subdomain** page, select Subdomain 1.
- 3 Click the **Boundary** tab. Select Boundary 1.
- 4 Select the **Constrained edge element distribution** check box, then type 5 in the **Number of edge elements** edit field.
- 5 Select Boundary 2.
- 6 Select the **Constrained edge element distribution** check box, then type 25 in the **Number of edge elements** edit field.
- 7 Click **Remesh**, then click **OK**.

## PHYSICS SETTINGS

*Subdomain Settings—Plane Stress*

- 1 From the **Multiphysics** menu, select **I Plane Stress (smps)**.
- 2 Choose **Physics>Subdomain Settings**.

3 Select Subdomain 1. On the **Material** page, enter the following settings:

PROPERTY	VALUE
E	E
$\nu$	nu
$\alpha$	alpha
$\rho$	rho
thickness	20e-6

4 Click the **Load** tab.

5 Select the **Include thermal expansion** check box. Enter T for **Temp** and 0 for **Tempref**.

6 Change to the **Damping** page.

7 From the **Damping model** list, select **No damping**, then click **OK**.

*Subdomain Settings—Heat Transfer by Conduction*

1 From the **Multiphysics** menu, select **2 Heat Transfer by Conduction (ht)**.

2 Choose **Physics>Subdomain Settings**.

3 Verify that Subdomain 1 is selected, then on the **Physics** page enter settings according to the following table; when finished, click **OK**.

PROPERTY	VALUE
k (isotropic)	k
Q	-j*omega_smps*(rho*Cp*T+T0*Ent_smps)

*Boundary Conditions—Plane Stress*

1 From the **Multiphysics** menu, select **1 Plane Stress (smps)**.

2 Choose **Physics>Boundary Settings**. Select Boundaries 1 and 4.

3 On the **Constraint** page, set **Constraint condition** to **Fixed**, then click **OK**.

*Boundary Conditions—Heat Transfer by Conduction*

1 From the **Multiphysics** menu, select **2 Heat Transfer by Conduction (ht)**.

2 Choose **Physics>Boundary Settings**.

3 Verify that **Thermal insulation** is the default value, then click **OK**.

**COMPUTING THE SOLUTION**

1 Choose **Solve>Solver Parameters**.

2 On the **General** page, enter 1 for **Desired number of eigenfrequencies**.

- 3 Enter  $f_0$  for **Search for eigenfrequencies around**.
- 4 Go to the **Advanced** page.
- 5 Locate the **Scaling of variables** area and choose **Manual** for **Type of scaling**.
- 6 In the **Manual scaling** edit field, type  $u \ 1e-4 \ v \ 1e-4 \ T \ 1$ . Click **OK**.
- 7 Click the **Solve** button on the Main toolbar.

The solver finds a solution with an eigenfrequency of approximately 63,045 Hz.

#### **POSTPROCESSING AND VISUALIZATION**

To generate the plot in Figure 3 follow the steps below:

- 1 Click the **Plot Parameters** button on the Main toolbar.
- 2 On the **General** page, verify that the **Boundary**, **Deformed shape**, and **Geometry edges** check boxes are selected, and that all others are cleared.
- 3 On the **Surface** page, select **Heat Transfer by Conduction (ht)>Temperature** from the **Predefined quantities** list. Click **OK**.

You can see the eigenfrequency in the postprocessing plot. Use the **Global Data Display** dialog box to calculate the Q value:

- 1 Select **Postprocessing>Data Display>Global**.
- 2 In the **Expression** edit field type **Qfactor\_smgs**.
- 3 Click **Apply**. The Q value appears in the message log below the drawing area.