Model created in COMSOL Multiphysics 6.4

Eigenfrequency Shifts Caused by Temperature Changes

Introduction

Some devices require a very high degree of frequency stability with respect to changes in the environment. In some applications, particularly within the MEMS field, it is important to study the sensitivity of a device's eigenfrequencies with respect to a variation in temperature. The same type of phenomena could, for example, also be caused by hygroscopic swelling due to changes in humidity. In very high precision applications, the frequency stability requirements might specify a precision at the ppb (parts-per-billion, 10^{-9}) level. Setting up simulations that accurately capture such small effects can be a challenging task, since several phenomena can interact.

This COMSOL Multiphysics example shows how to perform a sensitivity analysis of the eigenfrequencies of a beam subjected to thermal expansion. Effects like stress softening, geometric changes, and the temperature dependence of material properties are explored.

Model Definition

Consider a rectangular beam with the data listed in Table 1.

TABLE 1: PHYSICAL PROPERTIES.

Property	**Symbol**	**Value**
Length	L	10 mm
Width	a	1 mm
Height	b	0.5 mm
Young's modulus	E	100 GPa
Poisson's ratio	ν	0
Mass Density	ρ	1000 kg/m^3
Coefficient of thermal expansion, x direction	α_x	$1\cdot10^{-5}$ 1/K
Coefficient of thermal expansion, y direction	α_y	$2\cdot10^{-5}$ 1/K
Coefficient of thermal expansion, z direction	α_z	$3\cdot10^{-5}$ 1/K
Temperature shift	ΔT	10 K

The material parameters have values that are of the same order of magnitude as those for many other engineering materials. To better separate the various effects, Poisson's ratio is set to zero, but this assumption does not change the results in any fundamental way. Orthotropic thermal expansion coefficients are used to highlight some properties of the solution.

Results and Discussion

The eigenfrequencies of the beam are computed for two different types of boundary conditions: a doubly clamped beam, and a cantilever beam, where one end is fixed and the other end is free.

The mode shapes of the doubly clamped beam are shown in Figure 1.

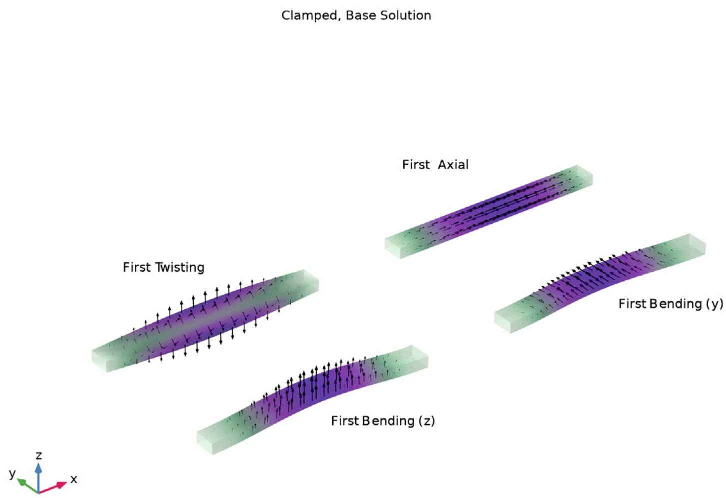

Figure 1: Mode shapes for the doubly clamped beam.

The eigenfrequencies computed for the doubly clamped beam, without any thermal expansion, are listed in Table 2 along with the eigenfrequencies obtained with a temperature rise of 10 K, and the ratio between the two.

TABLE 2: EIGENFREQUENCIES FOR THE DOUBLY CLAMPED BEAM.

Mode type	**Eigenfrequency, Hz**	**Eigenfrequency, Hz $\Delta T=10$ K**	**Ratio**
First bending, z direction	50713.9	50425.1	0.9943
First bending, y direction	97659.6	97526.2	0.9986
First twisting	266902	266917	1.00006
First axial	500000	500025	1.00005

Figure 2 shows the shift in frequency as a function of the temperature variation.

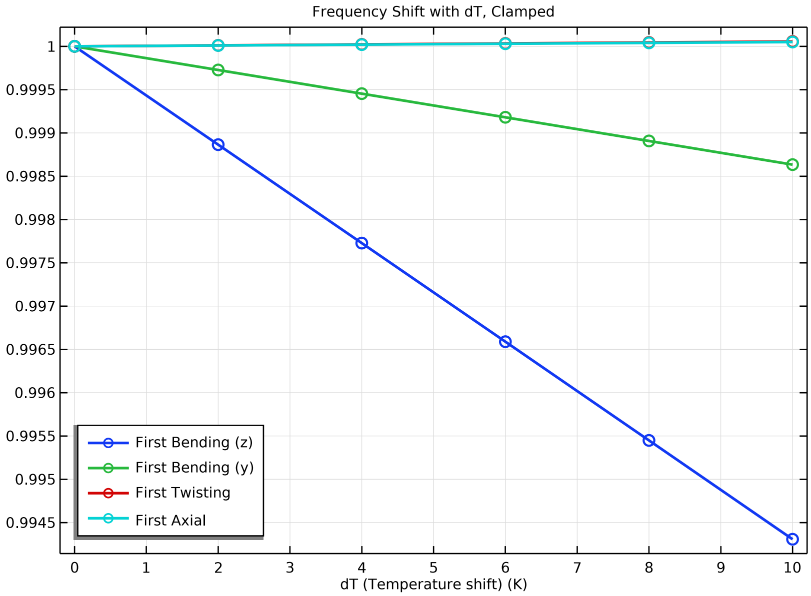

Figure 2: Frequency shift as a function of the temperature change for the doubly clamped beam.

STRESS SOFTENING EFFECT

The first thing to note is that the bending eigenmodes for the doubly clamped beam stand out and have a strong temperature dependence. If a thinner beam is considered, this difference would be even more pronounced. The reason for this behavior is related to the fact that in the case of a doubly clamped beam, the thermal expansion causes a compressive axial stress. With the given data, the stress is −10 MPa (computed as $E\alpha_x\Delta T$). This stress causes a significant reduction in the stiffness of the beam — an effect often called stress stiffening, since it typically occurs in structures with tensile stresses. However, compressive stresses soften the structure.

Another way of looking at this is by performing a linear buckling analysis. You can do so by adding a **Linear Buckling** study to the model and using the thermal expansion caused by $\Delta T = 10$ K as a unit load. You will then find that the critical load factor is 80 (see Figure 3).

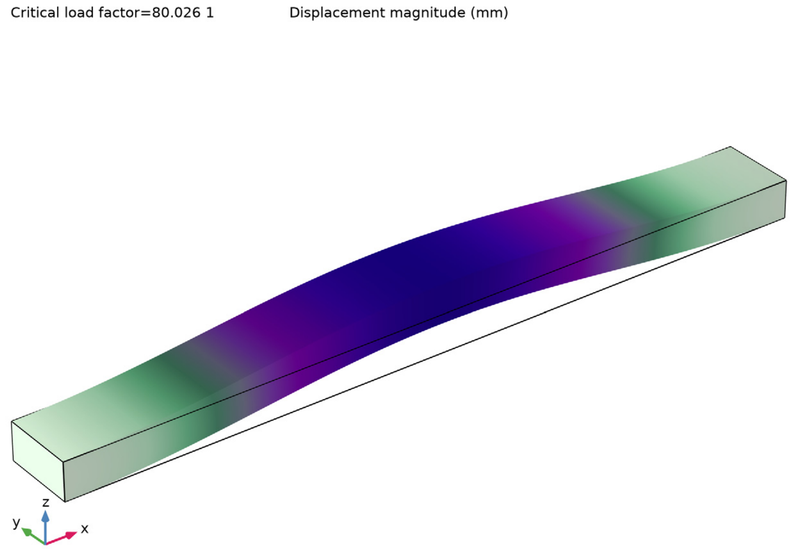

Figure 3: The first buckling mode.

With a linear assumption, the beam becomes unstable at an 800 K temperature increase. At the buckling load, the stiffness has reached 0. Assuming that the stiffness decreases linearly with the compressive stress, the stiffness at $\Delta T = 10$ K should be reduced by a factor of

$$1 - \frac{1}{80} = 0{,}9875$$

Since a natural frequency is proportional to the square root of the stiffness, you can estimate the decrease to

$$\sqrt{0{,}9875} = 0{,}9937$$

which matches the computed value of 0.9943 well.

Stress softening also affects the twisting and axial modes, but the effect is not as obvious as it is in the bending modes.

EFFECT OF GEOMETRY CHANGE

The mode shapes of the cantilever beam are shown in Figure 4.

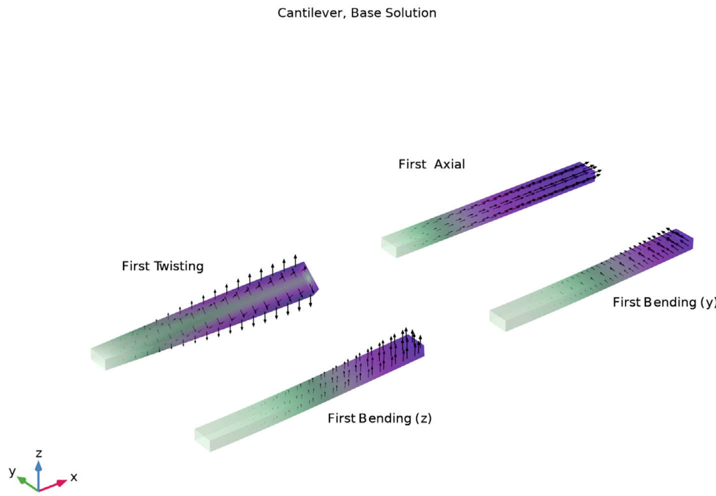

Figure 4: Mode shapes for the cantilever beam.

InTable 3, sthe numerical results for the eigenfrequencies with, and without temperature effects. are shown

TABLE 3: EIGENFREQUENCIES FOR THE CANTILEVER BEAM.

Mode type	**Eigenfrequency, Hz**	**Eigenfrequency, Hz $\Delta T=10$ K**	**Ratio**
First bending, z direction	8063.79	8066.92	1.00039
First bending, y direction	16049.1	16053.7	1.00028
First twisting	132233	132265	1.00025
First axial	250000	250050	1.00020

In the cantilever beam, no stresses develop when it is heated, as it simply expands. In this case, the frequency shift is due solely to the change in geometry — an effect that is much smaller than the stress-softening effect.

The natural frequencies for the bending, torsional, and axial vibration of a beam have the following dependencies on the physical properties

$$f_b \propto \frac{1}{L^2}\sqrt{\frac{EI}{\rho A}}$$

$$f_t \propto \frac{1}{L}\sqrt{\frac{GK}{\rho J}}$$

$$f_a \propto \frac{1}{L}\sqrt{\frac{E}{\rho}}$$

where I stands for the area moment of inertia around the bending axis, G is the shear modulus, K is the torsional modulus, and J is the polar moment of inertia around the beam axis.

It is assumed that the initial dimensions of the beam are L_0-by-a_0-by-b_0, where $a_0 > b_0$. After thermal expansion, the deformed size is L-by-a-by-b.

The expansions (strains) in the three orthogonal directions are called ε_x, ε_y, and ε_z; respectively. In this case, they are linearly related to the thermal expansion by $\varepsilon_x = \alpha_x\Delta T$, $\varepsilon_y = \alpha_y\Delta T$ and $\varepsilon_z = \alpha_z\Delta T$; but in principle, it could be any type of inelastic strain.

The geometric properties scale as:

$$L = L_0(1+\varepsilon_x)$$

$$A = ab = a_0 b_0(1+\varepsilon_y)(1+\varepsilon_z)$$

$$I_y = \frac{ab^3}{12} = \frac{a_0(b_0)^3}{12}(1+\varepsilon_y)(1+\varepsilon_z)^3$$

$$I_z = \frac{a^3 b}{12} = \frac{b_0(a_0)^3}{12}(1+\varepsilon_z)(1+\varepsilon_y)^3$$

$$K = \frac{ab^3}{12}F_1\left(\frac{a}{b}\right) \approx \frac{a_0(b_0)^3}{12}F_1\left(\frac{a_0}{b_0}\right)(1+\varepsilon_y)(1+\varepsilon_z)^3$$

$$J = \frac{ab^3}{12}+\frac{a^3 b}{12} = \frac{a_0(b_0)^3}{12}(1+\varepsilon_y)(1+\varepsilon_z)^3+\frac{b_0(a_0)^3}{12}(1+\varepsilon_z)(1+\varepsilon_y)^3$$

The mass density also changes. Since the same mass is now confined in a larger volume,

$$\rho = \frac{\rho_0}{(1+\varepsilon_x)(1+\varepsilon_y)(1+\varepsilon_z)}$$

By introducing these expressions into the formulas for the natural frequencies, you arrive at the following expected eigenfrequency shifts:

$$\frac{f_{b,z}}{f_{b0,z}} = \sqrt{\frac{(1+\varepsilon_y)(1+\varepsilon_z)^3}{(1+\varepsilon_x)}} \approx 1 - \frac{3\varepsilon_x}{2} + \frac{\varepsilon_y}{2} + \frac{3\varepsilon_z}{2}$$

$$\frac{f_{b,y}}{f_{b0,y}} = \sqrt{\frac{(1+\varepsilon_z)(1+\varepsilon_y)^3}{(1+\varepsilon_x)}} \approx 1 - \frac{3\varepsilon_x}{2} + \frac{3\varepsilon_y}{2} + \frac{\varepsilon_z}{2}$$

$$\frac{f_a}{f_{a0}} = \sqrt{\frac{(1+\varepsilon_z)(1+\varepsilon_y)}{(1+\varepsilon_x)}} \approx 1 - \frac{\varepsilon_x}{2} + \frac{\varepsilon_y}{2} + \frac{\varepsilon_z}{2}$$

Since the thermal expansions are very small, the approximate first-order series expansions can be expected to be accurate.

For the torsional vibrations, the situation is slightly more complicated, since the powers of a and b are mixed in the expression for the polar moment J. But if you make use of the fact that $a = 2b$ for this geometry, then it is possible to derive a similar expression:

$$\frac{f_t}{f_{t0}} = \sqrt{\frac{5(1+\varepsilon_z)(1+\varepsilon_y)^3}{(1+\varepsilon_x)((1+\varepsilon_z)^2 + 4(1+\varepsilon_z)^2)}} \approx 1 - \frac{\varepsilon_x}{2} - \frac{3\varepsilon_y}{10} + \frac{6\varepsilon_z}{5}$$

The computed frequency shifts are shown in Figure 5 along with the analytical predictions for the cantilever beam.

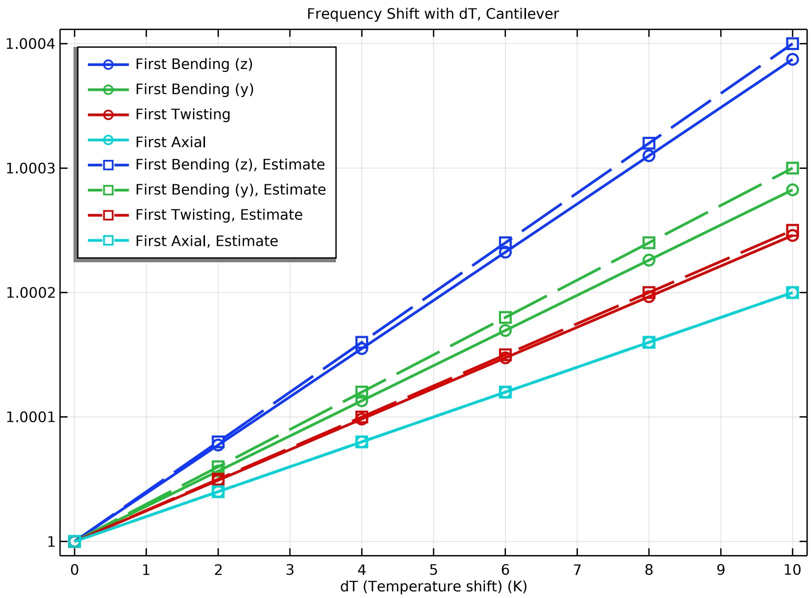

Figure 5: Frequency shift as a function of the temperature variation for a cantilever beam. Numerical results are compared with the estimates.

Table 4 collects and compares the values for $\Delta T = 10$ K.

TABLE 4: COMPUTED VERSUS PREDICTED SHIFT FOR THE CANTILEVER BEAM.

Mode type	Computed shift	Predicted shift
First bending, z direction	1.00039	1.00040
First bending, y direction	1.00028	1.00030
First twisting	1.00025	1.00025
First axial	1.00020	1.00020

EFFECT OF CONSTRAINT MODELING

The fixed constraints at the ends of the beam cause local stress concentrations when the temperature is increased, as the transverse displacement is constrained.

This can have two effects: on one hand, stress stiffening might be induced in a component that is expected to experience only volumetric changes, on the other side, the cross section dimension is no longer constant, due to the restrained transverse displacement (as in the example above).

To determine what effects the constraints should have, you must rely on your engineering judgment. Usually, the component and its surroundings are subject to temperature changes. In this situation you can add a thermal expansion to constraints..

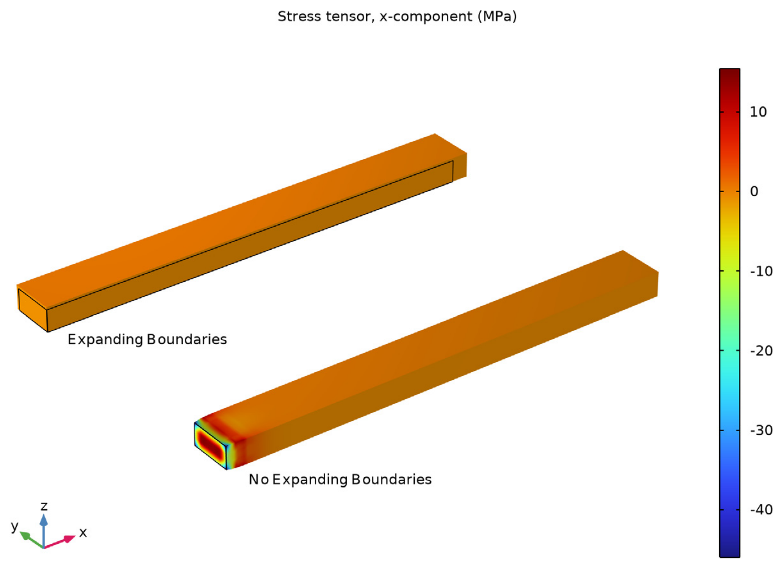

Figure 6: Axial stress distribution for the cantilever beam, comparing the case when thermal expansion in the boundary is considered to when it is not included.

Figure 7 shows the frequency shift dependence on the temperature variation for the cantilever beam when thermal expansion is included in the analysis.

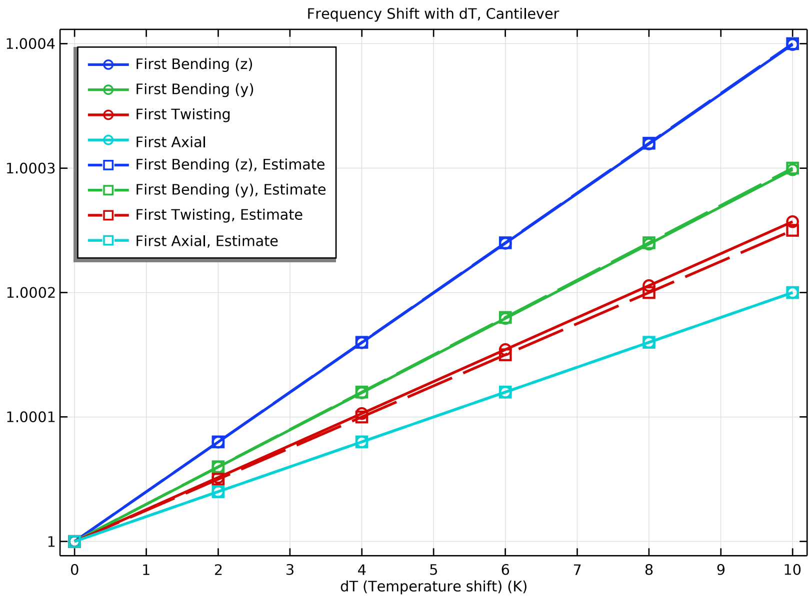

Figure 7: Frequency shift as a function at the temperature variation for a cantilever beam with thermal expansion of boundaries. Numerical results are compared with the estimates.

The results now change so that they match closely the analytical values, as shown both in Figure 7 and Table 5.

TABLE 5: COMPUTED VERSUS PREDICTED SHIFT FOR THE CANTILEVER BEAM WITH EXPANDING BOUNDARIES.

Mode type	**Fixed constraints**	**Stress-free constraints**	**Predicted shift**
First bending, z direction	1.00039	1.00040	1.00040
First bending, y direction	1.00028	1.00030	1.00030
First twisting	1.00025	1.00026	1.00025
First axial	1.00020	1.00020	1.00020

INCLUDING TEMPERATURE DEPENDENCE IN MATERIAL DATA

In the analyses above, it is assumed that the material data does not depend on temperature. When looking at constrained structures (dominated by the stress-softening effect), this might be an acceptable approximation. However, with the small frequency shifts caused by geometric changes, the temperature dependence of the material must also be taken into account.

For many materials, the relative change in stiffness is of the order of 10^{-4} 1/K. This means that for a temperature change of 10 K, you can expect a relative change in material stiffness that is of the order of 0.1%. This effect might actually be larger than the geometric effect computed above.

Note however that, when measuring the temperature dependence of Young's modulus, it is important to know whether or not the geometric change caused by thermal expansion has been taken into account or not. In other words, you must know whether the Young's modulus is measured with respect to the original dimensions or the heated dimensions.

When performing structural mechanics analyses in COMSOL Multiphysics, the equations are formed in the material frame. Thus, the mass density should never be given an explicit temperature dependence, since that would violate mass conservation.

The coefficient of thermal expansion (CTE) usually increases with temperature. The relative sensitivity is often of the order of 10^{-3} 1/K. This may sound large, but it is not usually important when looking at the way the CTE enters the equations.

After including a reduction of Young's modulus by $1 \cdot 10^{-4}$ 1/K, the resulting frequency shift turns out to be negative, rather than the positive shift observed with a constant Young's modulus as it can be seen in Figure 8.

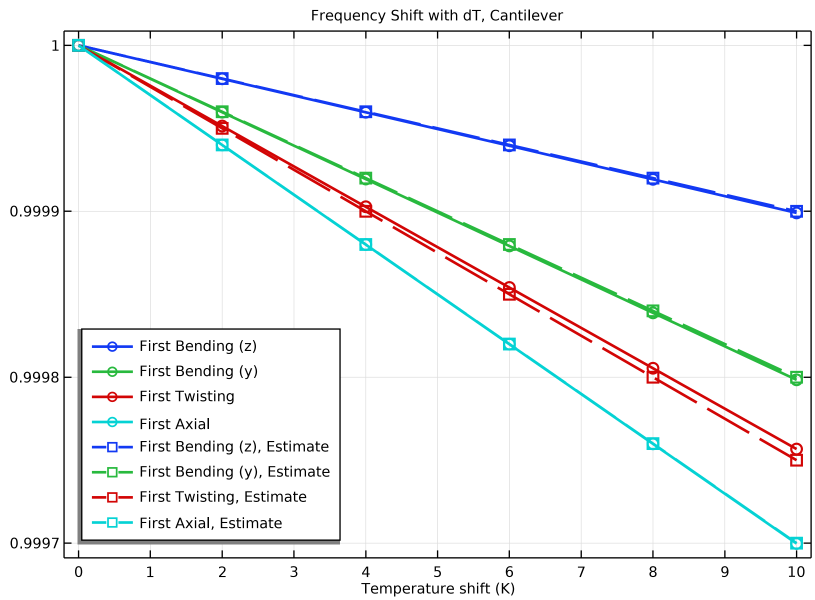

Figure 8: Frequency shift as a function at the temperature variation for a cantilever beam with temperature dependent Young's modulus. Numerical results are compared with the estimates.

The frequency shift at 10 K is reported in Table 6.

TABLE 6: COMPUTED VERSUS PREDICTED SHIFT FOR TEMPERATURE-DEPENDENT YOUNG'S MODULUS.

Mode type	**Stress-free constraints**	**Stress-free constraints, temperature-dependent E**	**Difference**
First bending, z direction	1.00040	0.99990	-0.00050
First bending, y direction	1.00030	0.99980	-0.00050
First twisting	1.00026	0.99976	-0.00050
First axial	1.00020	0.99970	-0.00050

Young's modulus is reduced by a factor $1\cdot10^{-3}$ and the natural frequencies are proportional to its square root. Actually, you can include the change in Young's modulus in the linearized expressions for the frequency shifts as (see Figure 8):

$$\frac{f_{b,z}}{f_{b0,z}} \approx 1 - \frac{3\varepsilon_x}{2} + \frac{\varepsilon_y}{2} + \frac{3\varepsilon_z}{2} + \frac{\beta\Delta T}{2}$$

$$\frac{f_{b,y}}{f_{b0,y}} \approx 1 - \frac{3\varepsilon_x}{2} + \frac{3\varepsilon_y}{2} + \frac{\varepsilon_z}{2} + \frac{\beta\Delta T}{2}$$

$$\frac{f_a}{f_{a0}} \approx 1 - \frac{\varepsilon_x}{2} + \frac{\varepsilon_y}{2} + \frac{\varepsilon_z}{2} + \frac{\beta\Delta T}{2}$$

$$\frac{f_t}{f_{t0}} \approx 1 - \frac{\varepsilon_x}{2} - \frac{3\varepsilon_y}{10} + \frac{6\varepsilon_z}{5} + \frac{\beta\Delta T}{2}$$

VERIFICATION AGAINST LARGE DEFORMATION

A multiplicative decomposition of deformation gradients is the default in COMSOL Multiphysics. This is one key concept to understand why it is possible to perform this type of analysis with a very high accuracy. Consider now a somewhat artificial case where the temperature increase is $3\cdot10^4$ K and there are no temperature dependencies in the material properties. This means that the stretches are

$$1 + \varepsilon_x = 1{,}3$$

$$1 + \varepsilon_y = 1{,}6$$

$$1 + \varepsilon_z = 1{,}9$$

resulting in the volume changing by a factor of 3.952. You can then compare the results from the prestressed eigenfrequency analysis with a standard eigenfrequency analysis on a larger beam with $L = 13$ mm, $a = 1.6$ mm, and $b = 0.95$ mm (see Figure 9), and a lower density scaled by a volume factor of 3.952, $\rho = 253.036\ \mathrm{kg/m^3}$.

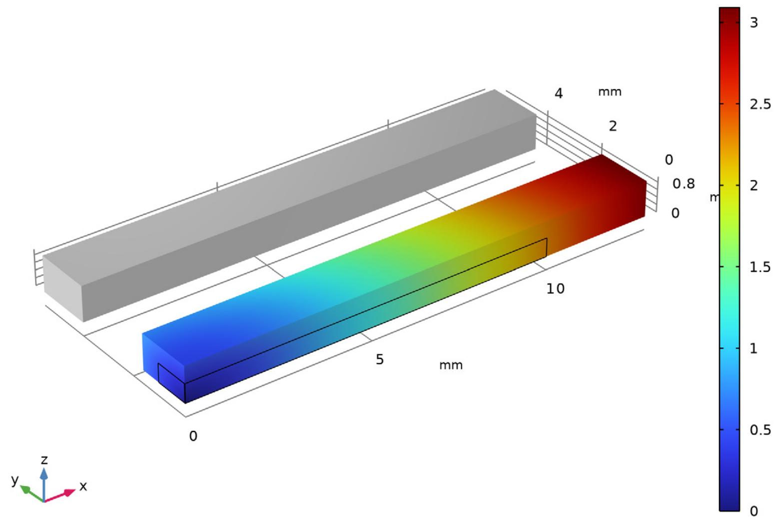

Figure 9: Deformed geometry after a temperature increase of 30,000 K and an undeformed beam with the same geometry.

This of course leads to large increases in the natural frequencies, as the heated object is much larger but with a lower density. The relative changes in frequency for the two approaches are shown in Table 7.

TABLE 7: EIGENFREQUENCY SHIFT.

Mode type	**Thermal expansion and prestressed eigenfrequency**	**Larger geometry and lower density**
First bending, z direction	2.2309	2.2308
First bending, y direction	1.8759	1.8759
First twisting	1.6702	1.6695
First axial	1.5292	1.5292

As can be seen above, the correspondence is in excellent agreement. There is a slight difference in the twisting mode, but that disappears with a refined mesh. Actually, refining the mesh shows that the best prediction is the one from the prestressed eigenfrequency analysis.

Notes About the COMSOL Implementation

- To analyze the effect of thermal expansion, add a **Prestressed Analysis, Eigenfrequency** study. This study consists of two study steps: a **Stationary** study step that computes the displacements and stresses caused by the thermal expansion, and an **Eigenfrequency** study step in which the previously computed solution is used. To compute the reference solution, you either add a separate **Eigenfrequency** study or run the same study sequence, but without thermal expansion.
- Most materials in the Material Library in COMSOL Multiphysics come with temperature-dependent material properties. As an alternative, you can manually add a linear temperature dependence to the Young's modulus as it is shown in this example.
- Computing frequency changes that are at the ppm (parts-per-million) level requires high precision. This means that it is important to avoid spurious rounding errors. There are some actions that you can take to ensure optimal accuracy. In the settings for the **Eigenfrequency** node, set **Search for eigenfrequencies around** to a value of the correct order of magnitude. Then, decrease the **Relative tolerance** in the settings for the **Eigenvalue Solver** node. Change only the parameters necessary for capturing the physics. For example, use the same mesh for all studies.
- If you have reason to believe that the problem is ill-conditioned, as can be the case for a slender structure, select **Iterative refinement** in the settings for the **Direct solver**.

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

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**.

First, set up geometrical parameters and draw the geometry of the beam.

GLOBAL DEFINITIONS

Geometrical Parameters

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

2 In the **Settings** window for **Parameters**, type `Geometrical Parameters` in the **Label** text field.

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

Name	Expression	Value	Description
L0	10[mm]	0.01 m	Reference length (x)
a0	1[mm]	0.001 m	Reference width (y)
b0	0.5[mm]	5E-4 m	Reference height (z)

GEOMETRY 1

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

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

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 `L0`.

4 In the **Depth** text field, type `a0`.

5 In the **Height** text field, type `b0`.

6 Click **Build Selected**.

Add material properties for a linear elastic material.

GLOBAL DEFINITIONS

Material Properties

1 In the **Home** toolbar, click **Pi** **Parameters** and choose **Add** > **Parameters**.

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

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

Name	Expression	Value	Description
`E0`	`100[GPa]`	1E11 Pa	`Reference Young's modulus`
`rho0`	`1000[kg/m^3]`	1000 kg/m³	`Reference mass density`
`nu0`	`0`	0	`Poisson's ratio`

MATERIALS

Material 1 (mat1)

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

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

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	`E0`	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	`nu0`	1	Young's modulus and Poisson's ratio
Density	rho	`rho0`	kg/m³	Basic

SOLID MECHANICS (SOLID)

Add a **Thermal Expansion** subnode to introduce thermal deformation in the material model.

Linear Elastic Material 1

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

Thermal Expansion 1

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

2 In the **Settings** window for **Thermal Expansion**, locate the **Model Input** section.

3 From the T_{ref} list, choose **User defined**. From the T list, choose **User defined**. In the associated text field, type `293.15[K]+dT`.

Specify the required thermal material properties.

GLOBAL DEFINITIONS

Material Properties

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

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
alpha_x	1E-5[1/K]	1E-5 1/K	CTE, x direction
alpha_y	2E-5[1/K]	2E-5 1/K	CTE, y direction
alpha_z	3E-5[1/K]	3E-5 1/K	CTE, z direction
dT	10[K]	10 K	Temperature shift

MATERIALS

Material 1 (mat1)

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

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

3 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Coefficient of thermal expansion	{alpha11, alpha22, alpha33} ; alphaij = 0	{alpha_x, alpha_y, alpha_z}	1/K	Basic

Set boundary conditions.

SOLID MECHANICS (SOLID)

Fixed Constraint, Left End

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

2 In the **Settings** window for **Fixed Constraint**, type `Fixed Constraint, Left End` in the **Label** text field.

3 Select Boundary 1 only.

Fixed Constraint, Right End

1 Right-click **Fixed Constraint, Left End** and choose **Duplicate**.

2 In the **Settings** window for **Fixed Constraint**, type `Fixed Constraint, Right End` in the **Label** text field.

3 Locate the **Boundary Selection** section. Click **Clear Selection**.

4 Select Boundary 6 only.

Use a swept mesh.

MESH 1

Swept 1

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

Size

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

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

3 From the **Predefined** list, choose **Extremely fine**.

4 Click **Build All**.

First solve for the "base case", where no temperature effect is considered.

STUDY 1, CLAMPED, BASE SOLUTION

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

2 In the **Settings** window for **Study**, type `Study 1, Clamped, Base Solution` in the **Label** text field.

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Study 1, Clamped, Base Solution** click **Step 1: Eigenfrequency**.

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

3 Select the **Desired number of eigenfrequencies** checkbox. In the associated text field, type `10`.

4 In the **Search for eigenfrequencies around shift** text field, type `10000[Hz]`.

Disable **Temperature Effects** in the solver.

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 1** > **Thermal Expansion 1**.

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

Modify the default solver sequence.

Solution 1 (sol1)

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

Reduce the relative tolerance to ensure high accuracy.

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

3 In the **Settings** window for **Eigenvalue Solver**, locate the **General** section.

4 In the **Relative tolerance** text field, type `1.0E-30`.

Normalize the eigenmodes to have a maximum of 1.

5 Locate the **Output** section. In the **Maximum absolute value** text field, type `1`.

6 In the **Study** toolbar, click **= Compute**.

RESULTS

Mode Shape, Clamped, Base Solution

1 In the **Settings** window for **3D Plot Group**, type `Mode Shape, Clamped, Base Solution` in the **Label** text field.

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

3 In the **Title** text area, type `Clamped, Base Solution`.

4 Clear the **Parameter indicator** text field.

5 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

6 Click to expand the **Plot Array** section. From the **Array type** list, choose **Square**.

7 In the **Relative row padding** text field, type `0.6`.

8 In the **Relative column padding** text field, type `0.4`.

First Bending, z Direction

1 In the **Model Builder** window, expand the **Mode Shape, Clamped, Base Solution** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, type `First Bending, z Direction` in the **Label** text field.

First Bending, y Direction

1 Right-click **First Bending, z Direction** and choose **Duplicate**.

2 In the **Settings** window for **Surface**, type `First Bending, y Direction` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1, Clamped, Base Solution/ Solution 1 (sol1)**.

4 From the **Eigenfrequency (Hz)** list, choose **97660**.

5 In the **Mode Shape, Clamped, Base Solution** toolbar, click **Plot**.

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

First Twisting

1 Right-click **First Bending, y Direction** and choose **Duplicate**.

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

3 From the **Eigenfrequency (Hz)** list, choose **2.669E5**.

4 In the **Label** text field, type `First Twisting`.

5 In the **Mode Shape, Clamped, Base Solution** toolbar, click **Plot**.

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

First Axial

1 Right-click **First Twisting** and choose **Duplicate**.

2 In the **Settings** window for **Surface**, type `First Axial` in the **Label** text field.

3 Locate the **Data** section. From the **Eigenfrequency (Hz)** list, choose **5E5**.

4 In the **Mode Shape, Clamped, Base Solution** toolbar, click **Plot**.

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

Deformation

1 In the **Model Builder** window, expand the **Results** > **Mode Shape, Clamped, Base Solution** > **First Bending, z Direction** node, then click **Deformation**.

2 In the **Settings** window for **Deformation**, locate the **Scale** section.

3 Select the **Scale factor** checkbox. In the associated text field, type `4E-4`.

First Bending, y Direction

1 In the **Model Builder** window, under **Results** > **Mode Shape, Clamped, Base Solution** click **First Bending, y Direction**.

2 In the **Settings** window for **Surface**, click to expand the **Inherit Style** section.

3 From the **Plot** list, choose **First Bending, z Direction**.

First Twisting

1 In the **Model Builder** window, click **First Twisting**.

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

3 From the **Plot** list, choose **First Bending, z Direction**.

First Axial

1 In the **Model Builder** window, click **First Axial**.

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

3 From the **Plot** list, choose **First Bending, z Direction**.

4 In the **Mode Shape, Clamped, Base Solution** toolbar, click **Plot**.

Arrow Surface, First Bending, z Direction

1 In the **Model Builder** window, right-click **Mode Shape, Clamped, Base Solution** and choose **Arrow Surface**.

2 In the **Settings** window for **Arrow Surface**, type `Arrow Surface, First Bending, z Direction` in the **Label** text field.

3 Locate the **Arrow Positioning** section. In the **Number of arrows** text field, type `100`.

4 Locate the **Coloring and Style** section.

5 Select the **Scale factor** checkbox. In the associated text field, type `6E-4`.

6 Click to expand the **Plot Array** section. Select the **Manual indexing** checkbox.

7 Locate the **Coloring and Style** section. From the **Color** list, choose **Black**.

Deformation 1

1 Right-click **Arrow Surface, First Bending, z Direction** and choose **Deformation**.

2 In the **Settings** window for **Deformation**, locate the **Scale** section.

3 Select the **Scale factor** checkbox. In the associated text field, type `4E-4`.

4 In the **Mode Shape, Clamped, Base Solution** toolbar, click **Plot**.

RESULTS

Arrow Surface, First Bending, z Direction

In the **Model Builder** window, collapse the **Results** > **Mode Shape, Clamped, Base Solution** > **Arrow Surface, First Bending, z Direction** node.

Arrow Surface, First Bending, y Direction

1 In the **Model Builder** window, right-click **Mode Shape, Clamped, Base Solution** and choose **Arrow Surface**.
2 In the **Settings** window for **Arrow Surface**, type `Arrow Surface, First Bending, y Direction` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Study 1, Clamped, Base Solution/ Solution 1 (sol1)**.
4 From the **Eigenfrequency (Hz)** list, choose **97660**.
5 Locate the **Arrow Positioning** section. In the **Number of arrows** text field, type `100`.
6 Locate the **Plot Array** section. Select the **Manual indexing** checkbox.
7 In the **Column index** text field, type `1`.
8 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Arrow Surface, First Bending, z Direction**.

Deformation 1

Right-click **Arrow Surface, First Bending, y Direction** and choose **Deformation**.

Arrow Surface, First Bending, y Direction

In the **Model Builder** window, collapse the **Results** > **Mode Shape, Clamped, Base Solution** > **Arrow Surface, First Bending, y Direction** node.

Arrow Surface, First Twisting

1 Right-click **Arrow Surface, First Bending, y Direction** and choose **Duplicate**.
2 In the **Settings** window for **Arrow Surface**, type `Arrow Surface, First Twisting` in the **Label** text field.
3 Locate the **Data** section. From the **Eigenfrequency (Hz)** list, choose **2.669E5**.
4 Locate the **Plot Array** section. In the **Row index** text field, type `1`.
5 In the **Column index** text field, type `0`.
6 In the **Mode Shape, Clamped, Base Solution** toolbar, click  **Plot**.

Arrow Surface, First Axial

1 Right-click **Arrow Surface, First Twisting** and choose **Duplicate**.
2 In the **Settings** window for **Arrow Surface**, locate the **Data** section.
3 From the **Eigenfrequency (Hz)** list, choose **5E5**.
4 In the **Label** text field, type `Arrow Surface, First Axial`.
5 Locate the **Plot Array** section. In the **Column index** text field, type `1`.

Transparency 1

In the **Model Builder** window, right-click **First Bending, z Direction** and choose **Transparency**.

Transparency 1

In the **Model Builder** window, right-click **First Bending, y Direction** and choose **Transparency**.

Transparency 1

In the **Model Builder** window, right-click **First Twisting** and choose **Transparency**.

Transparency 1

In the **Model Builder** window, right-click **First Axial** and choose **Transparency**.

First Bending, z Direction

In the **Model Builder** window, collapse the **First Bending, z Direction** node.

First Bending, y Direction

In the **Model Builder** window, collapse the **Results** > **Mode Shape, Clamped, Base Solution** > **First Bending, y Direction** node.

First Twisting

In the **Model Builder** window, collapse the **Results** > **Mode Shape, Clamped, Base Solution** > **First Twisting** node.

First Axial

In the **Model Builder** window, collapse the **Results** > **Mode Shape, Clamped, Base Solution** > **First Axial** node.

Mode Shape, Clamped, Base Solution

In the **Model Builder** window, click **Mode Shape, Clamped, Base Solution**.

Table Annotation 1

1 In the **Mode Shape, Clamped, Base Solution** toolbar, click **More Plots** and choose **Table Annotation**.

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

3 From the **Source** list, choose **Local table**.

4 In the table, enter the following settings:

x-coordinate	y-coordinate	z-coordinate	Annotation
`L0/2`	`0`	`0`	`First Bending (z)`
`L0/2+1.4*L0`	`0`	`0`	`First Bending (y)`
`L0/2`	`L0*1.4`	`0`	`First Twisting`
`L0/2+1.4*L0`	`L0*1.4`	`0`	`First  Axial`

5 Locate the **Coloring and Style** section. Clear the **Show point** checkbox.

6 In the **Mode Shape, Clamped, Base Solution** toolbar, click **Plot**.

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

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

Add a view for the plot

DEFINITIONS

View 2, 4 Beams

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **View**.

2 In the **Settings** window for **View**, type `View 2, 4 Beams` in the **Label** text field.

RESULTS

Mode Shape, Clamped, Base Solution

1 In the **Model Builder** window, under **Results** click **Mode Shape, Clamped, Base Solution**.

2 In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.

3 From the **View** list, choose **View 2, 4 Beams**.

Return to the default view to continue the modeling.

SOLID MECHANICS (SOLID)

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

RESULTS

In the **Model Builder** window, collapse the **Results** > **Mode Shape, Clamped, Base Solution** node.

Evaluate eigenfrequencies.

Eigenfrequencies (Study 1, Clamped, Base Solution)

1 In the **Model Builder** window, click **Eigenfrequencies (Study 1, Clamped, Base Solution)**.

2 In the **Eigenfrequencies (Study 1, Clamped, Base Solution)** toolbar, click **Evaluate**.

Participation Factors (Study 1, Clamped, Base Solution)

1 In the **Model Builder** window, click **Participation Factors (Study 1, Clamped, Base Solution)**.

2 In the **Participation Factors (Study 1, Clamped, Base Solution)** toolbar, click **Evaluate**.

Add a study for the cantilever beam case.

ADD STUDY

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

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

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

4 Click the **Add Study** button in the window toolbar.

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

STUDY 2

1 In the **Settings** window for **Study**, locate the **Study Settings** section.

2 Clear the **Generate default plots** checkbox.

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 Select the **Desired number of eigenfrequencies** checkbox. In the associated text field, type `10`.

4 In the **Search for eigenfrequencies around shift** text field, type `10000[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 1** > **Thermal Expansion 1**.

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

Disable the fixed constraint at the right end of the beam to obtain a cantilever.

8 In the tree, select **Component 1 (comp1)** > **Solid Mechanics (solid)** > **Fixed Constraint, Right End**.

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

10 In the **Model Builder** window, click **Study 2**.

11 In the **Settings** window for **Study**, type `Study 2, Cantilever, Base solution` in the **Label** text field.

Solution 2 (sol2)

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

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node, then click **Eigenvalue Solver 1**.
3 In the **Settings** window for **Eigenvalue Solver**, locate the **General** section.
4 In the **Relative tolerance** text field, type `1.0E-30`.
5 Locate the **Output** section. In the **Maximum absolute value** text field, type `1`.
6 In the **Study** toolbar, click **Compute**.

RESULTS

Mode Shape, Cantilever, Base Solution

1 In the **Model Builder** window, right-click **Mode Shape, Clamped, Base Solution** and choose **Duplicate**.
2 In the **Settings** window for **3D Plot Group**, type `Mode Shape, Cantilever, Base Solution` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Study 2, Cantilever, Base solution/ Solution 2 (sol2)**.
4 Locate the **Title** section. In the **Title** text area, type `Cantilever, Base Solution`.

First Bending, y Direction

1 In the **Model Builder** window, expand the **Mode Shape, Cantilever, Base Solution** node, then click **First Bending, y Direction**.
2 In the **Settings** window for **Surface**, locate the **Data** section.
3 From the **Dataset** list, choose **Study 2, Cantilever, Base solution/Solution 2 (sol2)**.

First Twisting

1 In the **Model Builder** window, click **First Twisting**.
2 In the **Settings** window for **Surface**, locate the **Data** section.
3 From the **Dataset** list, choose **Study 2, Cantilever, Base solution/Solution 2 (sol2)**.
4 From the **Eigenfrequency (Hz)** list, choose **1.3223E5**.

First Axial

1 In the **Model Builder** window, click **First Axial**.
2 In the **Settings** window for **Surface**, locate the **Data** section.
3 From the **Dataset** list, choose **Study 2, Cantilever, Base solution/Solution 2 (sol2)**.
4 From the **Eigenfrequency (Hz)** list, choose **2.5E5**.

Arrow Surface, First Bending, y Direction

1 In the **Model Builder** window, click **Arrow Surface, First Bending, y Direction**.
2 In the **Settings** window for **Arrow Surface**, locate the **Data** section.
3 From the **Dataset** list, choose **Study 2, Cantilever, Base solution/Solution 2 (sol2)**.

Arrow Surface, First Twisting

1 In the **Model Builder** window, click **Arrow Surface, First Twisting**.
2 In the **Settings** window for **Arrow Surface**, locate the **Data** section.
3 From the **Dataset** list, choose **Study 2, Cantilever, Base solution/Solution 2 (sol2)**.
4 From the **Eigenfrequency (Hz)** list, choose **1.3223E5**.

Arrow Surface, First Axial

1 In the **Model Builder** window, click **Arrow Surface, First Axial**.
2 In the **Settings** window for **Arrow Surface**, locate the **Data** section.
3 From the **Dataset** list, choose **Study 2, Cantilever, Base solution/Solution 2 (sol2)**.
4 From the **Eigenfrequency (Hz)** list, choose **2.5E5**.
5 In the **Mode Shape, Cantilever, Base Solution** toolbar, click **Plot**.
6 Click the **Show Grid** button in the **Graphics** toolbar.

RESULTS

Mode Shape, Cantilever, Base Solution

In the **Model Builder** window, collapse the **Results** > **Mode Shape, Cantilever, Base Solution** node.

Eigenfrequencies (Study 2, Cantilever, Base Solution)

1 In the **Model Builder** window, right-click **Eigenfrequencies (Study 1, Clamped, Base Solution)** and choose **Duplicate**.
2 In the **Settings** window for **Evaluation Group**, locate the **Data** section.
3 From the **Dataset** list, choose **Study 2, Cantilever, Base solution/Solution 2 (sol2)**.
4 In the **Label** text field, type `Eigenfrequencies (Study 2, Cantilever, Base Solution)`.
5 In the **Eigenfrequencies (Study 2, Cantilever, Base Solution)** toolbar, click **Evaluate**.

Participation Factors (Study 2, Cantilever, Base Solution)

1 In the **Model Builder** window, right-click **Participation Factors (Study 1, Clamped, Base Solution)** and choose **Duplicate**.

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

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

4 In the **Label** text field, type `Participation Factors (Study 2, Cantilever, Base Solution)`.

5 In the **Participation Factors (Study 2, Cantilever, Base Solution)** toolbar, click **Evaluate**.

Eigenfrequencies (Study 1, Clamped, Base Solution), Mode Shape, Clamped, Base Solution, Participation Factors (Study 1, Clamped, Base Solution)

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Mode Shape, Clamped, Base Solution**, **Eigenfrequencies (Study 1, Clamped, Base Solution)**, and **Participation Factors (Study 1, Clamped, Base Solution)**.

2 Right-click and choose **Group**.

Clamped

In the **Settings** window for **Group**, type `Clamped` in the **Label** text field.

Eigenfrequencies (Study 1, Clamped, Base Solution)

In the **Model Builder** window, collapse the **Results** > **Clamped** > **Eigenfrequencies (Study 1, Clamped, Base Solution)** node.

Participation Factors (Study 1, Clamped, Base Solution)

In the **Model Builder** window, collapse the **Results** > **Clamped** > **Participation Factors (Study 1, Clamped, Base Solution)** node.

Mode Shape, Clamped, Base Solution

In the **Model Builder** window, collapse the **Results** > **Clamped** > **Mode Shape, Clamped, Base Solution** node.

Clamped

In the **Model Builder** window, collapse the **Results** > **Clamped** node.

Eigenfrequencies (Study 2, Cantilever, Base Solution), Mode Shape, Cantilever, Base Solution, Participation Factors (Study 2, Cantilever, Base Solution)

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Mode Shape, Cantilever, Base Solution**, **Eigenfrequencies (Study 2, Cantilever, Base Solution)**, and **Participation Factors (Study 2, Cantilever, Base Solution)**.

2 Right-click and choose **Group**.

Cantilever

In the **Settings** window for **Group**, type `Cantilever` in the **Label** text field.

Mode Shape, Cantilever, Base Solution

In the **Model Builder** window, collapse the **Results** > **Cantilever** > **Mode Shape, Cantilever, Base Solution** node.

Eigenfrequencies (Study 2, Cantilever, Base Solution)

In the **Model Builder** window, collapse the **Results** > **Cantilever** > **Eigenfrequencies (Study 2, Cantilever, Base Solution)** node.

Participation Factors (Study 2, Cantilever, Base Solution)

In the **Model Builder** window, collapse the **Results** > **Cantilever** > **Participation Factors (Study 2, Cantilever, Base Solution)** node.

Cantilever

In the **Model Builder** window, collapse the **Results** > **Cantilever** node.

Now, analyze the effect of a temperature shift.

ADD STUDY

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

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces** > **Eigenfrequency, Prestressed**.

4 Click the **Add Study** button in the window toolbar.

STUDY 3, CLAMPED, DT

1 In the **Settings** window for **Study**, type `Study 3, Clamped, dT` in the **Label** text field.

2 Locate the **Study Settings** section. Clear the **Generate default plots** checkbox.

In this case, it would be possible to use a geometrically linear study.

Step 1: Stationary

1 In the **Model Builder** window, under **Study 3, Clamped, dT** click **Step 1: Stationary**.

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

3 Clear the **Include geometric nonlinearity** checkbox.

Step 2: Eigenfrequency

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

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

3 Select the **Desired number of eigenfrequencies** checkbox. In the associated text field, type `10`.

4 In the **Search for eigenfrequencies around shift** text field, type 10000[Hz].

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
dT (Temperature shift)		K

5 Click **Range**.

6 In the **Range** dialog, type 0 in the **Start** text field.

7 In the **Step** text field, type 2.

8 In the **Stop** text field, type 10.

9 Click **Add**.

Solution 3 (sol3)

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

Solution 3 (sol3)

1 In the **Model Builder** window, expand the **Study 3, Clamped, dT** > **Solver Configurations** > **Solution 3 (sol3)** node, then click **Eigenvalue Solver 1**.

2 In the **Settings** window for **Eigenvalue Solver**, locate the **General** section.

3 In the **Relative tolerance** text field, type 1.0E-30.

4 Locate the **Output** section. From the **Scaling of eigenvectors** list, choose **RMS**.

STUDY 3, CLAMPED, DT

1 In the **Model Builder** window, collapse the **Study 3, Clamped, dT** > **Solver Configurations** > **Solution 3 (sol3)** node.

2 In the **Study** toolbar, click **Add Study** to close the **Add Study** window.

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

Plot the frequency shift as a function of the temperature variation.

RESULTS

Frequency Shift with Temperature, Clamped

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

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

3 Locate the **Data** section. From the **Dataset** list, choose **Study 3, Clamped, dT/ Parametric Solutions 1 (sol5)**.

4 From the **Eigenfrequency selection** list, choose **First**.

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

6 In the **Title** text area, type `Frequency Shift with dT, Clamped`.

First Bending Mode, z Direction

1 Right-click **Frequency Shift with Temperature, Clamped** and choose **Global**.

2 In the **Settings** window for **Global**, type `First Bending Mode, z Direction` in the **Label** text field.

3 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
solid.freq/withsol('sol1',freq,setind(lambda, 1))	1	

4 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **Outer solutions**.

5 Click to expand the **Coloring and Style** section. From the **Width** list, choose **2**.

6 Find the **Line markers** subsection. From the **Marker** list, choose **Circle**.

7 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

8 In the table, enter the following settings:

Legends
First Bending (z)

9 In the **Frequency Shift with Temperature, Clamped** toolbar, click **Plot**.

First Bending Mode, y Direction

1 Right-click **First Bending Mode, z Direction** and choose **Duplicate**.

2 In the **Settings** window for **Global**, type `First Bending Mode, y Direction` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 3, Clamped, dT/ Parametric Solutions 1 (sol5)**.

4 From the **Eigenfrequency selection** list, choose **Manual**.

5 In the **Eigenfrequency indices (1-10)** text field, type `2`.

6 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`solid.freq/withsol('sol1',freq,setind(lambda, 2))`	1	

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

Legends
`First Bending (y)`

8 In the **Frequency Shift with Temperature, Clamped** toolbar, click **Plot**.

First Twisting Mode

1 Right-click **First Bending Mode, y Direction** and choose **Duplicate**.

2 In the **Settings** window for **Global**, type `First Twisting Mode` in the **Label** text field.

3 Locate the **Data** section. In the **Eigenfrequency indices (1-10)** text field, type `6`.

4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`solid.freq/withsol('sol1',freq,setind(lambda, 6))`	1	

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

Legends
`First Twisting`

6 In the **Frequency Shift with Temperature, Clamped** toolbar, click **Plot**.

First Axial Mode

1 Right-click **First Twisting Mode** and choose **Duplicate**.

2 In the **Settings** window for **Global**, type `First Axial Mode` in the **Label** text field.

3 Locate the **Data** section. In the **Eigenfrequency indices (1-10)** text field, type `9`.

4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`solid.freq/withsol('sol1',freq,setind(lambda, 9))`	1	

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

Legends
First Axial

6 In the **Frequency Shift with Temperature, Clamped** toolbar, click **Plot**.

Frequency Shift with Temperature, Clamped

1 In the **Model Builder** window, click **Frequency Shift with Temperature, Clamped**.

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

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

4 In the **Frequency Shift with Temperature, Clamped** toolbar, click **Plot**.

Repeat for the cantilever.

ADD STUDY

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

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces** > **Eigenfrequency, Prestressed**.

4 Click the **Add Study** button in the window toolbar.

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

STUDY 4, CANTILEVER, DT

1 In the **Settings** window for **Study**, type `Study 4, Cantilever, dT` in the **Label** text field.

2 Locate the **Study Settings** section. Clear the **Generate default plots** checkbox.

Step 1: Stationary

1 In the **Model Builder** window, under **Study 4, Cantilever, dT** click **Step 1: Stationary**.

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

3 Clear the **Include geometric nonlinearity** checkbox.

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

5 In the tree, select **Component 1 (comp1)** > **Solid Mechanics (solid)** > **Fixed Constraint, Right End**.

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

Step 2: Eigenfrequency

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

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

3 Select the **Desired number of eigenfrequencies** checkbox. In the associated text field, type 10.

4 In the **Search for eigenfrequencies around shift** text field, type 10000[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), Controls spatial frame** > **Fixed Constraint, Right End**.

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

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
dT (Temperature shift)		K

5 Click **Range**.

6 In the **Range** dialog, type 0 in the **Start** text field.

7 In the **Step** text field, type 2.

8 In the **Stop** text field, type 10.

9 Click **Add**.

Solution 12 (sol12)

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

2 In the **Model Builder** window, expand the **Solution 12 (sol12)** node, then click **Eigenvalue Solver 1**.

3 In the **Settings** window for **Eigenvalue Solver**, locate the **General** section.

4 In the **Relative tolerance** text field, type 1.0E-30.

5 Locate the **Output** section. From the **Scaling of eigenvectors** list, choose **RMS**.

STUDY 4, CANTILEVER, DT

1 In the **Model Builder** window, collapse the **Study 4, Cantilever, dT** > **Solver Configurations** > **Solution 12 (sol12)** node.
2 In the **Study** toolbar, click **Compute**.

RESULTS

Frequency Shift with Temperature, Cantilever

1 In the **Model Builder** window, right-click **Frequency Shift with Temperature, Clamped** and choose **Duplicate**.
2 In the **Settings** window for **1D Plot Group**, type `Frequency Shift with Temperature, Cantilever` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Study 4, Cantilever, dT/ Parametric Solutions 2 (sol14)**.
4 Locate the **Title** section. In the **Title** text area, type `Frequency Shift with dT, Cantilever`.

First Bending Mode, z Direction

1 In the **Model Builder** window, expand the **Frequency Shift with Temperature, Cantilever** node, then click **First Bending Mode, z Direction**.
2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.
3 In the table, enter the following settings:

Expression	Unit	Description
`solid.freq/withsol('sol2',freq,setind(lambda, 1))`	1	

First Bending Mode, y Direction

1 In the **Model Builder** window, click **First Bending Mode, y Direction**.
2 In the **Settings** window for **Global**, locate the **Data** section.
3 From the **Dataset** list, choose **Study 4, Cantilever, dT/Parametric Solutions 2 (sol14)**.
4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`solid.freq/withsol('sol2',freq,setind(lambda, 2))`	1	

First Twisting Mode

1 In the **Model Builder** window, click **First Twisting Mode**.

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

3 From the **Dataset** list, choose **Study 4, Cantilever, dT/Parametric Solutions 2 (sol14)**.

4 In the **Eigenfrequency indices (1-10)** text field, type 5.

5 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
solid.freq/withsol('sol2',freq,setind(lambda, 5))	1	

First Axial Mode

1 In the **Model Builder** window, click **First Axial Mode**.

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

3 From the **Dataset** list, choose **Study 4, Cantilever, dT/Parametric Solutions 2 (sol14)**.

4 In the **Eigenfrequency indices (1-10)** text field, type 7.

5 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
solid.freq/withsol('sol2',freq,setind(lambda, 7))	1	

6 In the **Frequency Shift with Temperature, Cantilever** toolbar, click **Plot**.

Frequency Shift with Temperature, Cantilever

1 In the **Model Builder** window, click **Frequency Shift with Temperature, Cantilever**.

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

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

4 In the **Frequency Shift with Temperature, Cantilever** toolbar, click **Plot**.

5 In the **Model Builder** window, collapse the **Frequency Shift with Temperature, Cantilever** node.

Frequency Shift with Temperature, Clamped

1 In the **Model Builder** window, expand the **Results** > **Clamped** node, then click **Frequency Shift with Temperature, Clamped**.

2 Drag and drop below **Clamped** > **Mode Shape, Clamped, Base Solution**.

Frequency Shift with Temperature, Cantilever

1 In the **Model Builder** window, expand the **Results** > **Cantilever** node, then click **Frequency Shift with Temperature, Cantilever**.

2 Drag and drop below **Cantilever** > **Mode Shape, Cantilever, Base Solution**.

Frequency Shift with Temperature, Clamped

1 In the **Results** toolbar, click **Evaluation Group**.

2 In the **Settings** window for **Evaluation Group**, type `Frequency Shift with Temperature, Clamped` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 3, Clamped, dT/Parametric Solutions 1 (sol5)**.

4 Click to expand the **Format** section. From the **Concatenation** list, choose **Vertical**.

First Bending (z)

1 Right-click **Frequency Shift with Temperature, Clamped** and choose **Global Evaluation**.

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

3 From the **Dataset** list, choose **Study 3, Clamped, dT/Parametric Solutions 1 (sol5)**.

4 From the **Eigenfrequency selection** list, choose **First**.

5 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`freq/withsol('sol1',freq,setind(lambda,1))`	1	

6 Locate the **Data** section. From the **Parameter selection (dT)** list, choose **Last**.

7 In the **Label** text field, type `First Bending (z)`.

First Bending (y)

1 Right-click **First Bending (z)** and choose **Duplicate**.

2 In the **Settings** window for **Global Evaluation**, type `First Bending (y)` in the **Label** text field.

3 Locate the **Data** section. From the **Eigenfrequency selection** list, choose **Manual**.

4 In the **Eigenfrequency indices (1-10)** text field, type `2`.

5 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`freq/withsol('sol1',freq,setind(lambda,2))`	1	

First Twisting

1 Right-click **First Bending (y)** and choose **Duplicate**.

2 In the **Settings** window for **Global Evaluation**, type `First Twisting` in the **Label** text field.

3 Locate the **Data** section. In the **Eigenfrequency indices (1-10)** text field, type 6.

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

Expression	Unit	Description
freq/withsol('sol1',freq,setind(lambda,6))	1	

First Axial

1 Right-click **First Twisting** and choose **Duplicate**.

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

3 In the **Eigenfrequency indices (1-10)** text field, type 9.

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

Expression	Unit	Description
freq/withsol('sol1',freq,setind(lambda,9))	1	

5 In the **Label** text field, type First Axial.

6 In the **Frequency Shift with Temperature, Clamped** toolbar, click **Evaluate**.

Frequency Shift with Temperature, Clamped

In the **Model Builder** window, right-click **Frequency Shift with Temperature, Clamped** and choose **Copy**.

Frequency Shift with Temperature, Cantilever

1 In the **Model Builder** window, right-click **Cantilever** and choose **Paste Evaluation Group**.

2 In the **Settings** window for **Evaluation Group**, type Frequency Shift with Temperature, Cantilever in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 4, Cantilever, dT/Parametric Solutions 2 (sol14)**.

First Bending (z)

1 In the **Model Builder** window, expand the **Frequency Shift with Temperature, Cantilever** node, then click **First Bending (z)**.

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

3 From the **Dataset** list, choose **Study 4, Cantilever, dT/Parametric Solutions 2 (sol14)**.

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

Expression	Unit	Description
freq/withsol('sol2',freq,setind(lambda,1))	1	

First Bending (y)

1 In the **Model Builder** window, click **First Bending (y)**.

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

3 From the **Dataset** list, choose **Study 4, Cantilever, dT/Parametric Solutions 2 (sol14)**.

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

Expression	Unit	Description
`freq/withsol('sol2',freq,setind(lambda,2))`	1	

First Twisting

1 In the **Model Builder** window, click **First Twisting**.

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

3 From the **Dataset** list, choose **Study 4, Cantilever, dT/Parametric Solutions 2 (sol14)**.

4 In the **Eigenfrequency indices (1-10)** text field, type `5`.

5 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`freq/withsol('sol2',freq,setind(lambda,5))`	1	

First Axial

1 In the **Model Builder** window, click **First Axial**.

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

3 From the **Dataset** list, choose **Study 4, Cantilever, dT/Parametric Solutions 2 (sol14)**.

4 In the **Eigenfrequency indices (1-10)** text field, type `7`.

5 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`freq/withsol('sol2',freq,setind(lambda,7))`	1	

6 In the **Frequency Shift with Temperature, Cantilever** toolbar, click = **Evaluate**.

Frequency Shift with Temperature, Cantilever

In the **Model Builder** window, collapse the **Results** > **Cantilever** > **Frequency Shift with Temperature, Cantilever** node.

Frequency Shift with Temperature, Clamped

In the **Model Builder** window, collapse the **Results** > **Clamped** > **Frequency Shift with Temperature, Clamped** node.

Compute the buckling critical load for the clamped case.

ADD STUDY

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

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces > Linear Buckling**.

4 Click the **Add Study** button in the window toolbar.

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

STUDY 5, BUCKLING

In the **Settings** window for **Study**, type `Study 5, Buckling` in the **Label** text field.

Step 2: Linear Buckling

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

RESULTS

Mode Shape, Buckling

1 In the **Settings** window for **3D Plot Group**, type `Mode Shape, Buckling` in the **Label** text field.

2 Drag and drop below **Clamped > Frequency Shift with Temperature, Clamped**.

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

Add a view for the buckling plot.

DEFINITIONS

View 3: Buckling

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **View**.

2 In the **Settings** window for **View**, type `View 3: Buckling` in the **Label** text field.

RESULTS

Mode Shape, Buckling

1 In the **Model Builder** window, under **Results > Clamped** click **Mode Shape, Buckling**.

2 In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.

3 From the **View** list, choose **View 3: Buckling**.

Return to the default view to continue the modeling.

DEFINITIONS

View 1

Compare the numerical results obtained for the cantilever with the estimated expected shifts.

Variables 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
eps_x	alpha_x*dT		Thermal strain, x direction
eps_y	alpha_y*dT		Thermal strain, y direction
eps_z	alpha_z*dT		Thermal strain, z direction
fbz_fb0z	1-3*eps_x/2+eps_y/2+3*eps_z/2		Estimate of frequency shift, cantilever, bending z direction
fby_fb0y	1-3*eps_x/2+3*eps_y/2+eps_z/2		Estimate of frequency shift, cantilever, bending y direction
fa_fa0	1-eps_x/2+eps_y/2+eps_z/2		Estimate of frequency shift, cantilever, axial
ft_ft0	1-eps_x/2-3*eps_y/10+6*eps_z/5		Estimate of frequency shift, cantilever, torsional

STUDY 4, CANTILEVER, DT

In the **Study** toolbar, click **Update Solution**.

RESULTS

First Bending (z)

1 In the **Model Builder** window, under **Results** > **Cantilever** > **Frequency Shift with Temperature, Cantilever** click **First Bending (z)**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
fbz_fb0z	1	Estimate of frequency shift, cantilever, bending z direction

First Bending (y)

1 In the **Model Builder** window, click **First Bending (y)**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
fby_fb0y	1	Estimate of frequency shift, cantilever, bending y direction

First Twisting

1 In the **Model Builder** window, click **First Twisting**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
ft_ft0	1	Estimate of frequency shift, cantilever, torsional

First Axial

1 In the **Model Builder** window, click **First Axial**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
fa_fa0	1	Estimate of frequency shift, cantilever, axial

4 In the **Frequency Shift with Temperature, Cantilever** toolbar, click **=** **Evaluate**.

FREQUENCY SHIFT WITH TEMPERATURE, CANTILEVER

1 Go to the **Frequency Shift with Temperature, Cantilever** window.

2 Click the **Full Precision** button in the window toolbar to better compare numerical results with estimates.

3 Click the **Full Precision** button in the window toolbar to revert to the default precision.

Add plots of the estimates of the frequency shift versus temperature to get a visual comparison with the numerical results.

RESULTS

Frequency Shift with Temperature, Cantilever

In the **Model Builder** window, expand the **Frequency Shift with Temperature, Cantilever** node.

First Axial Mode, First Bending Mode, y Direction, First Bending Mode, z Direction, First Twisting Mode

1 In the **Model Builder** window, under **Results** > **Cantilever** > **Frequency Shift with Temperature, Cantilever**, Ctrl-click to select **First Bending Mode, z Direction**, **First Bending Mode, y Direction**, **First Twisting Mode**, and **First Axial Mode**.

2 Right-click and choose **Duplicate**.

First Bending Mode, z Direction (Estimate)

1 In the **Settings** window for **Global**, type `First Bending Mode, z Direction (Estimate)` in the **Label** text field.

2 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
fbz_fb0z	1	Estimate of frequency shift, cantilever, bending z direction

3 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

4 Find the **Line markers** subsection. From the **Marker** list, choose **Square**.

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

Legends
First Bending (z), Estimate

First Bending Mode, y Direction (Estimate)

1 In the **Model Builder** window, under **Results** > **Cantilever** > **Frequency Shift with Temperature, Cantilever** click **First Bending Mode, y Direction 1**.

2 In the **Settings** window for **Global**, type `First Bending Mode, y Direction (Estimate)` in the **Label** text field.

3 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
fby_fb0y	1	Estimate of frequency shift, cantilever, bending y direction

4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

5 Find the **Line markers** subsection. From the **Marker** list, choose **Square**.

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

Legends
First Bending (y), Estimate

First Twisting Mode (Estimate)

1 In the **Model Builder** window, under **Results** > **Cantilever** > **Frequency Shift with Temperature, Cantilever** click **First Twisting Mode 1**.

2 In the **Settings** window for **Global**, type `First Twisting Mode (Estimate)` in the **Label** text field.

3 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
ft_ft0	1	Estimate of frequency shift, cantilever, torsional

4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

5 Find the **Line markers** subsection. From the **Marker** list, choose **Square**.

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

Legends
First Twisting, Estimate

First Axial Mode (Estimate)

1 In the **Model Builder** window, under **Results** > **Cantilever** > **Frequency Shift with Temperature, Cantilever** click **First Axial Mode 1**.

2 In the **Settings** window for **Global**, type `First Axial Mode (Estimate)` in the **Label** text field.

3 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
fa_fa0	1	Estimate of frequency shift, cantilever, axial

4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

5 Find the **Line markers** subsection. From the **Marker** list, choose **Square**.

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

Legends
First Axial, Estimate

First Bending Mode, z Direction (Estimate)

1 In the **Model Builder** window, click **First Bending Mode, z Direction (Estimate)**.

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

3 From the **Color** list, choose **Cycle (reset)**.

Frequency Shift with Temperature, Cantilever

In the **Model Builder** window, collapse the **Results** > **Cantilever** > **Frequency Shift with Temperature, Cantilever** node.

Inspect stress concentration at the fixed boundary.

Stress at Clamp

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Stress at Clamp` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 4, Cantilever, dT/ Solution Store 2 (sol13)**.

Volume 1

1 Right-click **Stress at Clamp** and choose **Volume**.

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

3 In the **Expression** text field, type `solid.sx`.

4 In the **Stress at Clamp** toolbar, click **Plot**.

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

6 Locate the **Coloring and Style** section. From the **Color table** list, choose **Rainbow**.

Introduce thermal expansion of constraints.

SOLID MECHANICS (SOLID)

Fixed Constraint, Left End

In the **Model Builder** window, under **Component 1 (comp1)** > **Solid Mechanics (solid)** click **Fixed Constraint, Left End**.

Thermal Expansion 1

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

2 In the **Settings** window for **Thermal Expansion**, locate the **Thermal Expansion Properties** section.

3 Select the **Inherit from domain** checkbox.

Modify the previous studies for future reruns to not include the constraint thermal expansion.

STUDY 1, CLAMPED, BASE SOLUTION

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Study 1, Clamped, Base Solution** click **Step 1: Eigenfrequency**.

2 In the **Settings** window for **Eigenfrequency**, locate the **Physics and Variables Selection** section.

3 In the tree, select **Component 1 (comp1)** > **Solid Mechanics (solid)** > **Fixed Constraint, Left End** > **Thermal Expansion 1**.

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

STUDY 2, CANTILEVER, BASE SOLUTION

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

2 In the **Settings** window for **Eigenfrequency**, locate the **Physics and Variables Selection** section.

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

STUDY 3, CLAMPED, DT

Step 1: Stationary

1 In the **Model Builder** window, under **Study 3, Clamped, dT** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** checkbox.

4 In the tree, select **Component 1 (comp1)** > **Solid Mechanics (solid)** > **Fixed Constraint, Left End** > **Thermal Expansion 1**.

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

Step 2: Eigenfrequency

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

2 In the **Settings** window for **Eigenfrequency**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** checkbox.

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

STUDY 4, CANTILEVER, DT

Step 1: Stationary

1 In the **Model Builder** window, under **Study 4, Cantilever, dT** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

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

Step 2: Eigenfrequency

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

2 In the **Settings** window for **Eigenfrequency**, locate the **Physics and Variables Selection** section.

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

STUDY 5, BUCKLING

Step 1: Stationary

1 In the **Model Builder** window, under **Study 5, Buckling** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** checkbox.

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

Step 2: Linear Buckling

1 In the **Model Builder** window, click **Step 2: Linear Buckling**.

2 In the **Settings** window for **Linear Buckling**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** checkbox.

4 In the tree, select **Component 1 (comp1)** > **Solid Mechanics (solid)** > **Fixed Constraint, Left End** > **Thermal Expansion 1**.

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

STUDY 1, CLAMPED, BASE SOLUTION

In the **Model Builder** window, collapse the **Study 1, Clamped, Base Solution** node.

STUDY 2, CANTILEVER, BASE SOLUTION

In the **Model Builder** window, collapse the **Study 2, Cantilever, Base solution** node.

STUDY 3, CLAMPED, DT

In the **Model Builder** window, collapse the **Study 3, Clamped, dT** node.

STUDY 4, CANTILEVER, DT

In the **Model Builder** window, collapse the **Study 4, Cantilever, dT** node.

STUDY 5, BUCKLING

In the **Model Builder** window, collapse the **Study 5, Buckling** node.

ADD STUDY

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

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces** > **Eigenfrequency, Prestressed**.

4 Click the **Add Study** button in the window toolbar.

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

STUDY 6, CANTILEVER WITH EXPANDING CONSTRAINT

1 In the **Settings** window for **Study**, type `Study 6, Cantilever with Expanding Constraint` in the **Label** text field.

2 Locate the **Study Settings** section. Clear the **Generate default plots** checkbox.

Step 1: Stationary

1 In the **Model Builder** window, under **Study 6, Cantilever with Expanding Constraint** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** checkbox.

4 In the tree, select **Component 1 (comp1)** > **Solid Mechanics (solid), Controls spatial frame** > **Fixed Constraint, Right End**.

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

Step 2: Eigenfrequency

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

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

3 Select the **Desired number of eigenfrequencies** checkbox. In the associated text field, type `10`.

4 In the **Search for eigenfrequencies around shift** text field, type `10000[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), Controls spatial frame** > **Fixed Constraint, Right End**.

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

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
dT (Temperature shift)	`range(0,2,10)`	K

Solution 23 (sol23)

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

2 In the **Model Builder** window, expand the **Solution 23 (sol23)** node, then click **Eigenvalue Solver 1**.

3 In the **Settings** window for **Eigenvalue Solver**, locate the **General** section.

4 In the **Relative tolerance** text field, type `1.0E-30`.

5 Locate the **Output** section. From the **Scaling of eigenvectors** list, choose **RMS**.

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

RESULTS

Frequency Shift with Temperature, Cantilever, Expanding Constraint

1 In the **Model Builder** window, right-click **Frequency Shift with Temperature, Cantilever** and choose **Duplicate**.
2 In the **Settings** window for **1D Plot Group**, type `Frequency Shift with Temperature, Cantilever, Expanding Constraint` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Study 6, Cantilever with Expanding Constraint/Parametric Solutions 3 (sol25)**.

First Bending Mode, y Direction

1 In the **Model Builder** window, expand the **Frequency Shift with Temperature, Cantilever, Expanding Constraint** node, then click **First Bending Mode, y Direction**.
2 In the **Settings** window for **Global**, locate the **Data** section.
3 From the **Dataset** list, choose **Study 6, Cantilever with Expanding Constraint/ Parametric Solutions 3 (sol25)**.

First Twisting Mode

1 In the **Model Builder** window, click **First Twisting Mode**.
2 In the **Settings** window for **Global**, locate the **Data** section.
3 From the **Dataset** list, choose **Study 6, Cantilever with Expanding Constraint/ Parametric Solutions 3 (sol25)**.

First Axial Mode

1 In the **Model Builder** window, click **First Axial Mode**.
2 In the **Settings** window for **Global**, locate the **Data** section.
3 From the **Dataset** list, choose **Study 6, Cantilever with Expanding Constraint/ Parametric Solutions 3 (sol25)**.
4 In the **Frequency Shift with Temperature, Cantilever, Expanding Constraint** toolbar, click **Plot**.

First Bending (z)

1 In the **Model Builder** window, under **Results** > **Cantilever** > **Frequency Shift with Temperature, Cantilever** click **First Bending (z)**.
2 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.

3 In the table, enter the following settings:

Expression	Unit	Description
`freq/withsol('sol2',freq, setind(lambda,1))`	1	`Frequency Shift`
`withsol('sol25',freq, setind(lambda,1))/ withsol('sol2',freq, setind(lambda,1))`	1	`Frequency Shift, Expanding Boundaries`
`fbz_fb0z`	1	`Estimate of frequency shift, cantilever, bending z direction`

First Bending (y)

1 In the **Model Builder** window, click **First Bending (y)**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`withsol('sol25',freq, setind(lambda,2))/ withsol('sol2',freq, setind(lambda,2))`	1	
`fby_fb0y`	1	`Estimate of frequency shift, cantilever, bending y direction`

First Twisting

1 In the **Model Builder** window, click **First Twisting**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`withsol('sol25',freq, setind(lambda,5))/ withsol('sol2',freq, setind(lambda,5))`	1	
`ft_ft0`	1	`Estimate of frequency shift, cantilever, torsional`

First Axial

1 In the **Model Builder** window, click **First Axial**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`withsol('sol25',freq, setind(lambda,7))/ withsol('sol2',freq, setind(lambda,7))`	1	
`fa_fa0`	1	`Estimate of frequency shift, cantilever, axial`

4 In the **Frequency Shift with Temperature, Cantilever** toolbar, click **Evaluate**.

Compare the stress at the fixed boundary with that obtained without thermal expansion in the clamp.

Volume 2

1 In the **Model Builder** window, under **Results** > **Cantilever** > **Stress at Clamp** right-click **Volume 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study 6, Cantilever with Expanding Constraint/ Solution Store 4 (sol24)**.

4 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Volume 1**.

Stress at Clamp

1 In the **Model Builder** window, click **Stress at Clamp**.

2 In the **Settings** window for **3D Plot Group**, locate the **Plot Array** section.

3 From the **Array type** list, choose **Linear**.

4 From the **Array axis** list, choose **y**.

5 In the **Relative padding** text field, type `5`.

6 In the **Stress at Clamp** toolbar, click **Plot**.

7 Locate the **Title** section. From the **Title type** list, choose **Manual**.

8 In the **Title** text area, type `Stress tensor, x-component (MPa)`.

9 In the **Stress at Clamp** toolbar, click **Plot**.

No Expanding Boundaries

1 In the **Model Builder** window, under **Results** > **Cantilever** > **Stress at Clamp** click **Volume 1**.

2 In the **Settings** window for **Volume**, type `No Expanding Boundaries` in the **Label** text field.

Expanding Boundaries

1 In the **Model Builder** window, under **Results** > **Cantilever** > **Stress at Clamp** click **Volume 2**.

2 In the **Settings** window for **Volume**, type `Expanding Boundaries` in the **Label** text field.

Deformation 1

In the **Model Builder** window, right-click **No Expanding Boundaries** and choose **Deformation**.

Deformation 1

In the **Model Builder** window, right-click **Expanding Boundaries** and choose **Deformation**.

Table Annotation 1

1 In the **Stress at Clamp** toolbar, click **More Plots** and choose **Table Annotation**.

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

3 From the **Source** list, choose **Local table**.

4 In the table, enter the following settings:

x-coordinate	y-coordinate	z-coordinate	Annotation
`-0.25`	`-0.75`	`0`	`No Expanding Boundaries`
`-0.25`	`5.25`	`0`	`Expanding Boundaries`

5 Locate the **Coloring and Style** section. Clear the **Show point** checkbox.

6 From the **Anchor point** list, choose **Lower left**.

7 In the **Stress at Clamp** toolbar, click **Plot**.

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

DEFINITIONS

View 4: 2 Beams

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **View**.

2 In the **Settings** window for **View**, type `View 4: 2 Beams` in the **Label** text field.

RESULTS

Stress at Clamp

1 In the **Model Builder** window, under **Results** > **Cantilever** click **Stress at Clamp**.

2 In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.

3 From the **View** list, choose **View 4: 2 Beams**.

RESULTS

Expanding Boundaries

In the **Model Builder** window, collapse the **Results** > **Cantilever** > **Stress at Clamp** > **Expanding Boundaries** node.

No Expanding Boundaries

In the **Model Builder** window, collapse the **Results** > **Cantilever** > **Stress at Clamp** > **No Expanding Boundaries** node.

Frequency Shift with Temperature, Cantilever

In the **Model Builder** window, collapse the **Results** > **Cantilever** > **Frequency Shift with Temperature, Cantilever** node.

Stress at Clamp

In the **Model Builder** window, collapse the **Results** > **Cantilever** > **Stress at Clamp** node.

Frequency Shift with Temperature, Cantilever, Expanding Constraint

In the **Model Builder** window, collapse the **Results** > **Cantilever** > **Frequency Shift with Temperature, Cantilever, Expanding Constraint** node.

Clamped

In the **Model Builder** window, collapse the **Results** > **Clamped** node.

STUDY 1, CLAMPED, BASE SOLUTION

Solver Configurations

In the **Model Builder** window, collapse the **Study 1, Clamped, Base Solution** > **Solver Configurations** node.

Now, analyze the effect of a temperature dependent Young's modulus.

GLOBAL DEFINITIONS

Material Properties

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

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
beta	-1e-4[1/K]*0	0 1/K	Relative temperature sensitivity for Young's modulus

MATERIALS

Material 1 (mat1)

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Materials** > **Material 1 (mat1)** node, then click **Basic (def)**.

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

3 Click **Select Quantity**.

4 In the **Physical Quantity** dialog, type `Temperature` in the text field.

5 In the tree, select **General** > **Temperature (K)**.

6 Click **OK**.

7 In the **Model Builder** window, under **Component 1 (comp1)** > **Materials** > **Material 1 (mat1)** click **Young's modulus and Poisson's ratio (Enu)**.

8 In the **Settings** window for **Young's Modulus and Poisson's Ratio**, locate the **Model Inputs** section.

9 Click **Select Quantity**.

10 In the **Physical Quantity** dialog, select **General** > **Temperature (K)** in the tree.

11 Click **OK**.

12 In the **Model Builder** window, click **Material 1 (mat1)**.

13 In the **Settings** window for **Material**, locate the **Material Contents** section.

14 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	`E0*(1+beta*(T-293.15))`	Pa	Young's modulus and Poisson's ratio

SOLID MECHANICS (SOLID)

Linear Elastic Material 1

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

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

3 From the T_{ref} list, choose **User defined**. From the T list, choose **User defined**. In the associated text field, type `293.15[K]+dT`.

ADD STUDY

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

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces** > **Eigenfrequency, Prestressed**.

4 Click the **Add Study** button in the window toolbar.

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

STUDY 7, CANTILEVER WITH TEMPERATURE DEPENDENT E

1 In the **Settings** window for **Study**, type `Study 7, Cantilever with Temperature Dependent E` in the **Label** text field.

2 Locate the **Study Settings** section. Clear the **Generate default plots** checkbox.

Step 1: Stationary

1 In the **Model Builder** window, under **Study 7, Cantilever with Temperature Dependent E** click **Step 1: Stationary**.

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

3 Clear the **Include geometric nonlinearity** checkbox.

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

5 In the tree, select **Component 1 (comp1)** > **Solid Mechanics (solid)** > **Fixed Constraint, Right End**.

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

Step 2: Eigenfrequency

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

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

3 Select the **Desired number of eigenfrequencies** checkbox. In the associated text field, type `10`.

4 In the **Search for eigenfrequencies around shift** text field, type `10000[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), Controls spatial frame** > **Fixed Constraint, Right End**.

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

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
beta (Relative temperature sensitivity for Young's modulus)	-1E-4	1/K

5 Click **Add**.

6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
dT (Temperature shift)	range(0,2,10)	K

7 From the **Sweep type** list, choose **All combinations**.

Solution 32 (sol32)

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

2 In the **Model Builder** window, expand the **Solution 32 (sol32)** node, then click **Eigenvalue Solver 1**.

3 In the **Settings** window for **Eigenvalue Solver**, locate the **General** section.

4 In the **Relative tolerance** text field, type `1.0E-30`.

5 Locate the **Output** section. From the **Scaling of eigenvectors** list, choose **RMS**.

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

RESULTS

Frequency Shift with Temperature, Cantilever, Temperature Dependent E

1 In the **Model Builder** window, right-click **Frequency Shift with Temperature, Cantilever, Expanding Constraint** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Frequency Shift with Temperature, Cantilever, Temperature Dependent E` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 7, Cantilever with Temperature Dependent E/Parametric Solutions 4 (sol34)**.

4 From the **Parameter selection (beta)** list, choose **First**.

First Bending Mode, z Direction

1 In the **Model Builder** window, expand the **Frequency Shift with Temperature, Cantilever, Temperature Dependent E** node, then click **First Bending Mode, z Direction**.
2 In the **Settings** window for **Global**, locate the **Data** section.
3 From the **Dataset** list, choose **Study 7, Cantilever with Temperature Dependent E/ Parametric Solutions 4 (sol34)**.
4 From the **Parameter selection (beta)** list, choose **First**.
5 From the **Eigenfrequency selection** list, choose **First**.
6 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
7 In the **Expression** text field, type `dT`.

First Bending Mode, y Direction

1 In the **Model Builder** window, click **First Bending Mode, y Direction**.
2 In the **Settings** window for **Global**, locate the **Data** section.
3 From the **Dataset** list, choose **Study 7, Cantilever with Temperature Dependent E/ Parametric Solutions 4 (sol34)**.
4 From the **Parameter selection (beta)** list, choose **First**.
5 From the **Eigenfrequency selection** list, choose **Manual**.
6 In the **Eigenfrequency indices (1-10)** text field, type `2`.
7 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
8 In the **Expression** text field, type `dT`.

First Twisting Mode

1 In the **Model Builder** window, click **First Twisting Mode**.
2 In the **Settings** window for **Global**, locate the **Data** section.
3 From the **Dataset** list, choose **Study 7, Cantilever with Temperature Dependent E/ Parametric Solutions 4 (sol34)**.
4 From the **Parameter selection (beta)** list, choose **First**.
5 From the **Eigenfrequency selection** list, choose **Manual**.
6 In the **Eigenfrequency indices (1-10)** text field, type `5`.
7 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.
8 In the **Expression** text field, type `dT`.

First Axial Mode

1 In the **Model Builder** window, click **First Axial Mode**.

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

3 From the **Dataset** list, choose **Study 7, Cantilever with Temperature Dependent E/ Parametric Solutions 4 (sol34)**.

4 From the **Parameter selection (beta)** list, choose **First**.

5 From the **Eigenfrequency selection** list, choose **Manual**.

6 In the **Eigenfrequency indices (1-10)** text field, type 7.

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

8 In the **Expression** text field, type dT.

First Bending Mode, z Direction (Estimate)

1 In the **Model Builder** window, click **First Bending Mode, z Direction (Estimate)**.

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

3 From the **Dataset** list, choose **Study 7, Cantilever with Temperature Dependent E/ Parametric Solutions 4 (sol34)**.

4 From the **Parameter selection (beta)** list, choose **First**.

5 From the **Eigenfrequency selection** list, choose **First**.

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

7 In the **Expression** text field, type dT.

8 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
fbz_fb0z+beta*dT/2	1	

First Bending Mode, y Direction (Estimate)

1 In the **Model Builder** window, click **First Bending Mode, y Direction (Estimate)**.

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

3 From the **Dataset** list, choose **Study 7, Cantilever with Temperature Dependent E/ Parametric Solutions 4 (sol34)**.

4 From the **Parameter selection (beta)** list, choose **First**.

5 From the **Eigenfrequency selection** list, choose **First**.

6 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
fby_fb0y+beta*dT/2	1	

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

8 In the **Expression** text field, type dT.

First Twisting Mode (Estimate)

1 In the **Model Builder** window, click **First Twisting Mode (Estimate)**.

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

3 From the **Dataset** list, choose **Study 7, Cantilever with Temperature Dependent E/ Parametric Solutions 4 (sol34)**.

4 From the **Parameter selection (beta)** list, choose **First**.

5 From the **Eigenfrequency selection** list, choose **First**.

6 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
ft_ft0+beta*dT/2	1	

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

8 In the **Expression** text field, type dT.

First Axial Mode (Estimate)

1 In the **Model Builder** window, click **First Axial Mode (Estimate)**.

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

3 From the **Dataset** list, choose **Study 7, Cantilever with Temperature Dependent E/ Parametric Solutions 4 (sol34)**.

4 From the **Parameter selection (beta)** list, choose **First**.

5 From the **Eigenfrequency selection** list, choose **First**.

6 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
fa_fa0+beta*dT/2	1	

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

8 In the **Expression** text field, type dT.

9 In the **Frequency Shift with Temperature, Cantilever, Temperature Dependent E** toolbar, click **Plot**.

RESULTS

Frequency Shift with Temperature, Cantilever, Temperature Dependent E

1. In the **Model Builder** window, collapse the **Results** > **Cantilever** > **Frequency Shift with Temperature, Cantilever, Temperature Dependent E** node.
2. In the **Model Builder** window, click **Frequency Shift with Temperature, Cantilever, Temperature Dependent E**.
3. In the **Settings** window for **1D Plot Group**, locate the **Legend** section.
4. From the **Position** list, choose **Lower left**.
5. In the **Frequency Shift with Temperature, Cantilever, Temperature Dependent E** toolbar, click **Plot**.

First Bending (z)

1. In the **Model Builder** window, expand the **Frequency Shift with Temperature, Cantilever** node, then click **First Bending (z)**.
2. In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.
3. In the table, enter the following settings:

Expression	Unit	Description
`withsol('sol34',freq,setind(lambda,1))/withsol('sol2',freq,setind(lambda,1))`	1	Frequency Shift, E(T)
`fbz_fb0z+withsol('sol34',beta)*dT/2`	1	Estimate of frequency shift, E(T)

First Bending (y)

1. In the **Model Builder** window, click **First Bending (y)**.
2. In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.
3. In the table, enter the following settings:

Expression	Unit	Description
`withsol('sol34',freq,setind(lambda,2))/withsol('sol2',freq,setind(lambda,2))`	1	
`fby_fb0y+withsol('sol34',beta)*dT/2`	1	

First Twisting

1. In the **Model Builder** window, click **First Twisting**.
2. In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.

3 In the table, enter the following settings:

Expression	Unit	Description
withsol('sol34',freq,setind(lambda,5))/ withsol('sol2',freq,setind(lambda,5))	1	
ft_ft0+withsol('sol34',beta)*dT/2	1	

First Axial

1 In the **Model Builder** window, click **First Axial**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
withsol('sol34',freq,setind(lambda,7))/ withsol('sol2',freq,setind(lambda,7))	1	
fa_fa0+withsol('sol34',beta)*dT/2	1	

4 In the **Frequency Shift with Temperature, Cantilever** toolbar, click **Evaluate**.

Compute the eigenfrequencies for `dT=30000[K]`.

ADD STUDY

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

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Selected Physics Interfaces** > **Eigenfrequency, Prestressed**.

4 Click the **Add Study** button in the window toolbar.

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

STUDY 8, CANTILEVER, DT=30000

1 In the **Settings** window for **Study**, type `Study 8, Cantilever, dT=30000` in the **Label** text field.

2 Locate the **Study Settings** section. Clear the **Generate default plots** checkbox.

Step 1: Stationary

1 In the **Model Builder** window, under **Study 8, Cantilever, dT=30000** click **Step 1: Stationary**.

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

3 Clear the **Include geometric nonlinearity** checkbox.

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

5 In the tree, select **Component 1 (comp1)** > **Solid Mechanics (solid)** > **Fixed Constraint, Right End**.

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

Step 2: Eigenfrequency

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

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

3 Select the **Desired number of eigenfrequencies** checkbox. In the associated text field, type `10`.

4 In the **Search for eigenfrequencies around shift** text field, type `10000[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), Controls spatial frame** > **Fixed Constraint, Right End**.

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

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
dT (Temperature shift)	30000	K

Solution 41 (sol41)

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

Solution 41 (sol41)

1 In the **Model Builder** window, expand the **Study 8, Cantilever, dT=30000** > **Solver Configurations** > **Solution 41 (sol41)** node, then click **Eigenvalue Solver 1**.

2 In the **Settings** window for **Eigenvalue Solver**, locate the **General** section.

3 In the **Relative tolerance** text field, type `1.0E-30`.

4 Locate the **Output** section. From the **Scaling of eigenvectors** list, choose **RMS**.

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

6 In the **Model Builder** window, collapse the **Study 8, Cantilever, dT=30000** node.

Analyze the natural frequencies of a beam with dimensions equal to those of the beam under analysis when subjected to the thermal deformation.

ADD STUDY

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

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

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

4 Click the **Add Study** button in the window toolbar.

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

STUDY 9

Step 1: Eigenfrequency

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

2 Select the **Desired number of eigenfrequencies** checkbox. In the associated text field, type `10`.

3 In the **Search for eigenfrequencies around shift** text field, type `10000[Hz]`.

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

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

6 In the **Model Builder** window, click **Study 9**.

7 In the **Settings** window for **Study**, type `Study 9, Cantilever, Modified Geometry And Density` in the **Label** text field.

8 Locate the **Study Settings** section. Clear the **Generate default plots** checkbox.

Solution 45 (sol45)

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

2 In the **Model Builder** window, expand the **Solution 45 (sol45)** node.

3 In the **Model Builder** window, under **Study 9, Cantilever, Modified Geometry And Density** > **Solver Configurations** > **Solution 45 (sol45)** click **Eigenvalue Solver 1**.

4 In the **Settings** window for **Eigenvalue Solver**, locate the **General** section.

5 In the **Relative tolerance** text field, type `1.0E-30`.

6 Locate the **Output** section. From the **Scaling of eigenvectors** list, choose **RMS**.

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
L0 (Reference length (x))	13	m

5 Click **Add**.

6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
L0 (Reference length (x))	13	mm
a0 (Reference width (y))	1.6	mm

7 Click **Add**.

8 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
b0 (Reference height (z))	0.95	mm

9 Click **Add**.

10 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
rho0 (Reference mass density)	1000/(1.3*1.6*1.9)	kg/m^3

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

Compare the deformed shape at `dT=30000[K]` with the undeformed modified geometry.

RESULTS

3D Plot Group 9

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 8, Cantilever, dT=30000/Solution Store 6 (sol42)**.

Volume 1

Right-click **3D Plot Group 9** and choose **Volume**.

Deformation 1

1 In the **Model Builder** window, right-click **Volume 1** and choose **Deformation**.

2 In the **Settings** window for **Deformation**, locate the **Scale** section.

3 Select the **Scale factor** checkbox. In the associated text field, type `1`.

4 In the **3D Plot Group 9** toolbar, click **Plot**.

3D Plot Group 9

1 In the **Model Builder** window, under **Results** click **3D Plot Group 9**.

2 In the **Settings** window for **3D Plot Group**, locate the **Title** section.

3 From the **Title type** list, choose **None**.

4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

5 Locate the **Plot Array** section. From the **Array type** list, choose **Linear**.

6 From the **Array axis** list, choose **y**.

7 In the **Relative padding** text field, type `1.5`.

Volume 2

1 In the **Model Builder** window, under **Results > 3D Plot Group 9** right-click **Volume 1** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study 9, Cantilever, Modified Geometry And Density/ Solution 45 (sol45)**.

Deformation 1

1 In the **Model Builder** window, expand the **Volume 2** node.

2 Right-click **Deformation 1** and choose **Delete**.

Volume 2

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

2 In the **Expression** text field, type `1`.

3 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

4 From the **Color** list, choose **Gray**.

Line 1

1 In the **Model Builder** window, right-click **3D Plot Group 9** and choose **Line**.

2 In the **Settings** window for **Line**, click to expand the **Plot Array** section.

3 Select the **Manual indexing** checkbox.

4 Locate the **Expression** section. In the **Expression** text field, type `1`.

5 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

6 From the **Color** list, choose **Black**.

7 In the **3D Plot Group 9** toolbar, click **Plot**.

Thermal Expansion vs. Modified Geometry

1 In the **Model Builder** window, click **3D Plot Group 9**.

2 Drag and drop below **Cantilever** > **Frequency Shift with Temperature, Cantilever, Expanding Constraint**.

3 In the **Model Builder** window, click **3D Plot Group 9**.

4 In the **Settings** window for **3D Plot Group**, type `Thermal Expansion vs. Modified Geometry` in the **Label** text field.

Thermal Expansion

1 In the **Model Builder** window, under **Results** > **Cantilever** > **Thermal Expansion vs. Modified Geometry** click **Volume 1**.

2 In the **Settings** window for **Volume**, type `Thermal Expansion` in the **Label** text field.

Modified Geometry

1 In the **Model Builder** window, under **Results** > **Cantilever** > **Thermal Expansion vs. Modified Geometry** click **Volume 2**.

2 In the **Settings** window for **Volume**, type `Modified Geometry` in the **Label** text field.

Thermal Expansion, Undeformed

1 In the **Model Builder** window, under **Results** > **Cantilever** > **Thermal Expansion vs. Modified Geometry** click **Line 1**.

2 In the **Settings** window for **Line**, type `Thermal Expansion, Undeformed` in the **Label** text field.

Thermal Expansion vs. Modified Geometry

1 In the **Model Builder** window, collapse the **Results** > **Cantilever** > **Thermal Expansion vs. Modified Geometry** node.

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

3 In the **Model Builder** window, click **Thermal Expansion vs. Modified Geometry**.

4 In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.

5 From the **View** list, choose **View 4: 2 Beams**.

dT=30000[K] vs. Modified Geometry

1 In the **Results** toolbar, click **Evaluation Group**.

2 In the **Settings** window for **Evaluation Group**, type `dT=30000[K] vs. Modified Geometry` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 8, Cantilever, dT=30000/ Parametric Solutions 5 (sol43)**.

4 Locate the **Format** section. From the **Concatenation** list, choose **Vertical**.

First Bending (z)

1 Right-click **dT=30000[K] vs. Modified Geometry** and choose **Global Evaluation**.

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

3 From the **Dataset** list, choose **Study 8, Cantilever, dT=30000/Solution 41 (sol41)**.

4 From the **Eigenfrequency selection** list, choose **First**.

5 In the **Label** text field, type `First Bending (z)`.

6 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`freq/withsol('sol45',freq,setind(lambda,1))`	1	

First Bending (y)

1 Right-click **First Bending (z)** and choose **Duplicate**.

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

3 From the **Eigenfrequency selection** list, choose **Manual**.

4 In the **Eigenfrequency indices (1-10)** text field, type `2`.

5 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`freq/withsol('sol45',freq,setind(lambda,2))`	1	

6 In the **Label** text field, type `First Bending (y)`.

First Twisting

1 Right-click **First Bending (y)** and choose **Duplicate**.

2 In the **Settings** window for **Global Evaluation**, type `First Twisting` in the **Label** text field.

3 Locate the **Data** section. In the **Eigenfrequency indices (1-10)** text field, type `5`.

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

Expression	Unit	Description
`freq/withsol('sol45',freq,setind(lambda,5))`	1	

First Axial

1 Right-click **First Twisting** and choose **Duplicate**.

2 In the **Settings** window for **Global Evaluation**, type `First Axial` in the **Label** text field.

3 Locate the **Data** section. In the **Eigenfrequency indices (1-10)** text field, type `7`.

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

Expression	Unit	Description
`freq/withsol('sol45',freq,setind(lambda,7))`	1	

dT=30000[K] vs. Modified Geometry

1 In the **Model Builder** window, click **dT=30000[K] vs. Modified Geometry**.

2 In the **dT=30000[K] vs. Modified Geometry** toolbar, click = **Evaluate**.