Created in COMSOL Multiphysics 6.2

Thermal Drift in a Microwave Cavity Filter

Introduction

Microwave filters serve to suppress unwanted frequencies in the output of microwave transmitters. Amplifiers are in general nonlinear and produce harmonics that must be suppressed using one or several narrow passband filters on the output. High-frequency stability can be hard to achieve in such filters because microwave systems may be subject to thermal drift caused by high-power loads or harsh environmental conditions like exposure to direct sunlight in the desert. Thus, system engineers need to estimate the drift of the passband frequency that arises due to thermal expansion of a filter.

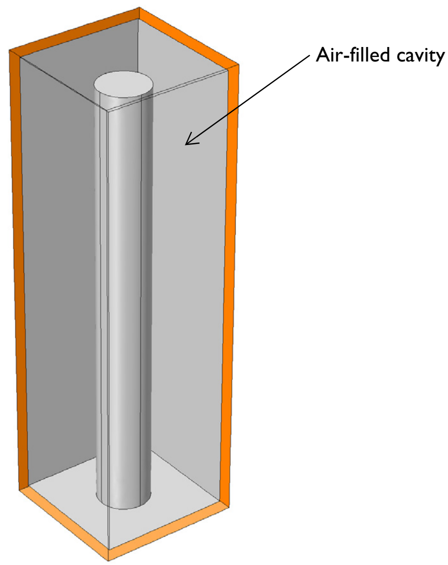


Figure 1: The microwave filter in this example consists of a thin metallic box, made of copper that contains a cylindrical post. This configuration forms a closed air-filled electromagnetic cavity between the box walls and the post.

Note: This example requires the RF Module and the Structural Mechanics Module.

Model Definition

Figure 1 shows the filter geometry. It consists of a box with a cylindrical post centered on one face. It is made of copper covered with a thin layer of silver to minimize losses. The silver layer not modeled is sufficiently thin to have a negligible influence on the device's thermal and mechanical properties. The all-copper design will be compared to a second design using both copper and steel.

Thermal expansion and the associated drift in eigenfrequency are caused by a uniform increase in the temperature of the cavity walls. The thermal expansion is readily computed using the Solid Mechanics interface from the Structural Mechanics Module. The eigenfrequency analysis of the cavity structure is easily performed using the 3D Electromagnetic Waves, Frequency Domain interface in the RF Module.

The model uses the Deformed Geometry (dg) interface to address the distorted shape of the filter geometry due to the thermal expansion. The deformed shape is used for the electromagnetic analysis.

Results and Discussion

The filter's temperature can rise due to power dissipation in the filter itself, in the surrounding electronics, or due to external heating. Figure 2 shows the thermal expansion results for a filter made entirely of copper.

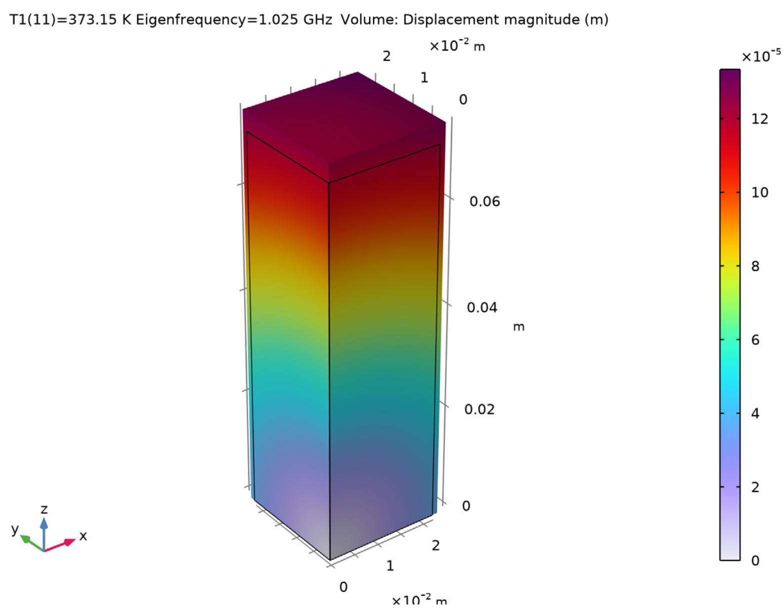


Figure 2: Thermal expansion at 100 °C above the reference temperature.

An actual filter usually consists of multiple cavities cascaded, but this discussion limits the analysis to one cell. Figure 3 shows the filter's lowest eigenfrequency. The typical quarter-wave resonance of the cylindrical post is clearly visible. A strong capacitive coupling between the top of the post and the nearby face of the box is also obvious.

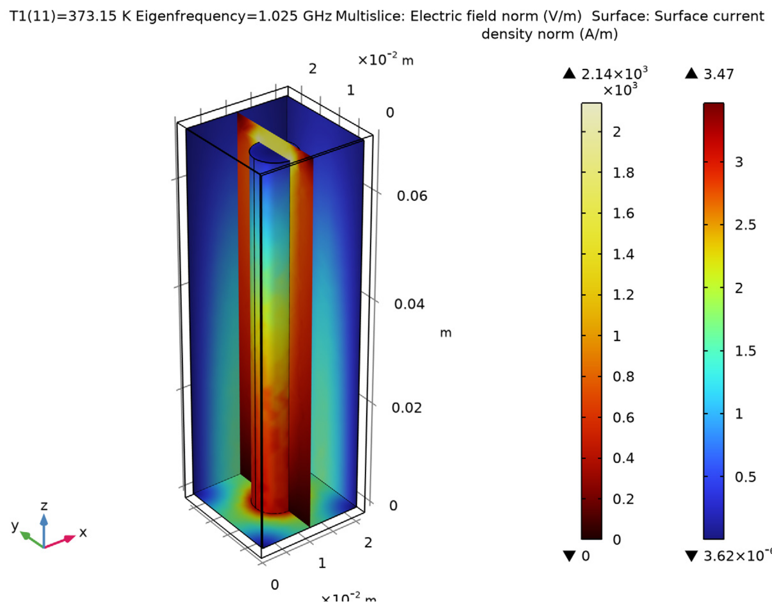


Figure 3: Results from the electromagnetic mode analysis. The plot shows the electric field and surface current patterns of the fundamental mode.

By repeating the structural and electromagnetic analyses for a number of operating temperatures, an eigenfrequency-versus-temperature curve is obtained. The results for two different designs appear in Figure 4.

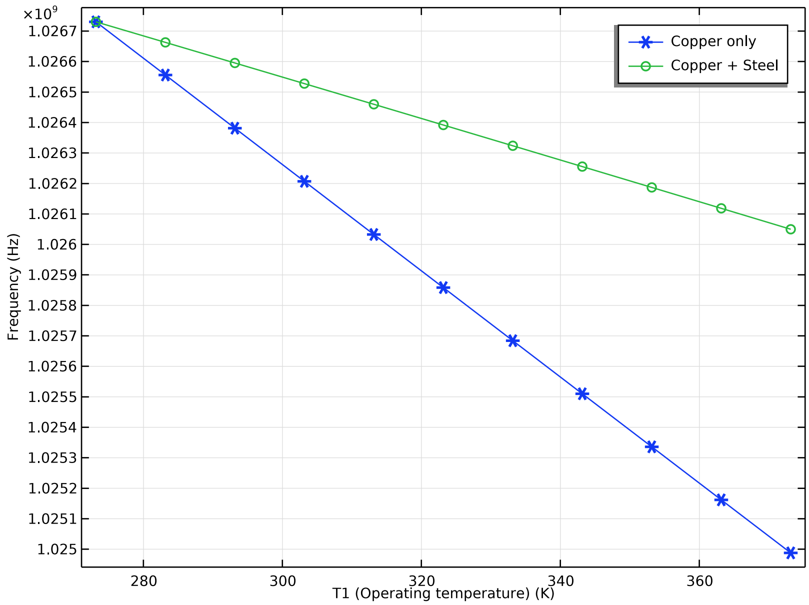


Figure 4: Eigenfrequency (Hz) versus temperature (K) for two different designs.

The first design, where the entire filter is made of copper, has been discussed already. For the second design the post is made of steel. It is obvious that the combination of the steel post and copper box is superior to a design using copper alone. The reason is the reduced capacitive coupling between the top of the post and the nearby face of the box, which results from the different coefficients of thermal expansion for the two materials. This coupling has a strong influence on the resonant frequency and, when reduced, counteracts the effects of an overall increase in cavity size.

Application Library path: `RF_Module/Filters/cavity_filter_thermal_expansion`

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. In the **Select Physics** tree, select **Mathematics>Deformed Mesh>Moving Mesh>Free Deformation**.
5. Click **Add**.
6. In the **Select Physics** tree, select **Radio Frequency>Electromagnetic Waves, Frequency Domain (emw)**.
7. Click **Add**.
8. Click **Done**.

ADD STUDY

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

ADD STUDY

1. Go to the **Add Study** window.
2. Find the **Physics interfaces in study** subsection. In the table, enter the following settings:

Physics	Solve
Solid Mechanics (solid)	√
Electromagnetic Waves, Frequency Domain (emw)	

3. Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Stationary**.
4. Click **Add Study** in the window toolbar.
5. In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 1

Step 2: Eigenfrequency

1. In the **Study** toolbar, click **Study Steps** and choose **Eigenfrequency>Eigenfrequency**.
2. In the **Settings** window for **Eigenfrequency**, locate the **Physics and Variables Selection** section.

3 In the table, enter the following settings:

Physics interface	Solve for	Equation form
Solid Mechanics (solid)		Automatic (Stationary)
Moving mesh (Component 1)		Automatic

4 Locate the **Study Settings** section.

5 Select the **Desired number of eigenfrequencies** check box. In the associated text field, type `1`.

In this model you use a parametric sweep to study thermal expansion as a function of the operating temperature. Begin by setting up a solver sequence for the three physics interfaces.

GLOBAL DEFINITIONS

Parameters 1

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

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
`T0`	`0[degC]`	273.15 K	`Reference temperature`
`T1`	`100[degC]`	373.15 K	`Operating temperature`

GEOMETRY 1

Block 1 (blk1)

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

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

3 In the **Width** text field, type `0.02`.

4 In the **Depth** text field, type `0.02`.

5 In the **Height** text field, type `0.07`.

6 Click **Build Selected**.

7 Click the **Wireframe Rendering** button in the **Graphics** toolbar.

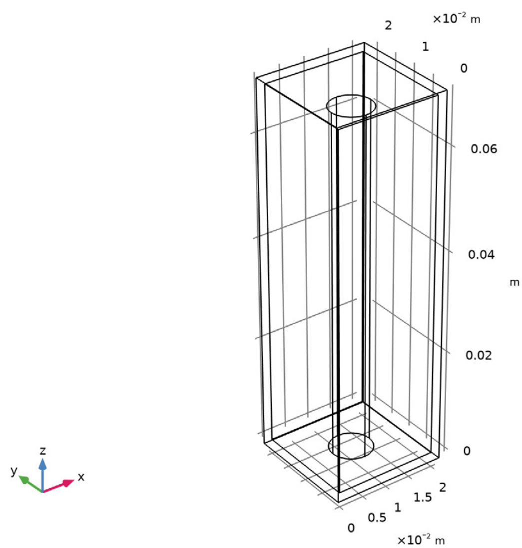
Block 2 (blk2)

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

2. In the **Settings** window for **Block**, locate the **Size and Shape** section.
3. In the **Width** text field, type `0.022`.
4. In the **Depth** text field, type `0.022`.
5. In the **Height** text field, type `0.072`.
6. Locate the **Position** section. In the **x** text field, type `-0.001`.
7. In the **y** text field, type `-0.001`.
8. In the **z** text field, type `-0.001`.

Cylinder 1 (cyl1)

1. In the **Geometry** toolbar, click **Cylinder**.
2. In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.
3. In the **Radius** text field, type `0.004`.
4. In the **Height** text field, type `0.067`.
5. Locate the **Position** section. In the **x** text field, type `0.01`.
6. In the **y** text field, type `0.01`.
7. In the **Geometry** toolbar, click **Build All**.
8. In the **Model Builder** window, click **Geometry 1**.



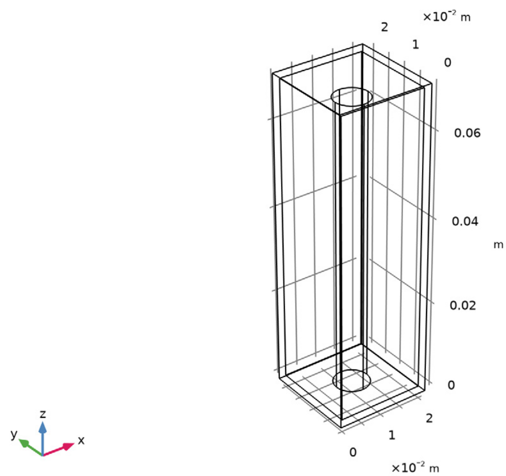
DEFINITIONS

Hide for Geometry 1

1 In the **Model Builder** window, right-click **View 1** and choose **Hide for Geometry**.

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

3 From the **Geometric entity level** list, choose **Boundary**.



4 On the object **fin**, select Boundaries 1, 2, 4, 6, 7, and 9 only.

It might be easier to select the correct boundaries by using the **Selection List** window. To open this window, in the **Home** toolbar click **Windows** and choose **Selection List**. (If you are running the cross-platform desktop, you find **Windows** in the main menu.)

ADD MATERIAL

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

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

3 In the tree, select **Built-in>Copper**.

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

5 In the tree, select **Built-in>Air**.

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

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

MATERIALS

Copper (mat1)

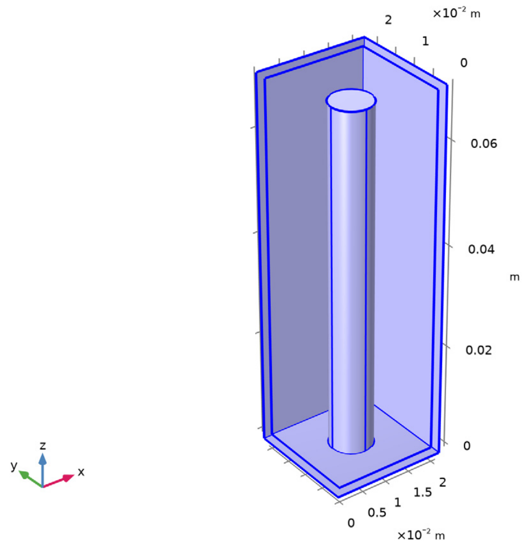
1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Copper (mat1)**.

2 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

3 In the list, select **2**.

4 Click **Remove from Selection**.

5 Select Domains 1 and 3 only.



6 Click **Create Selection**.

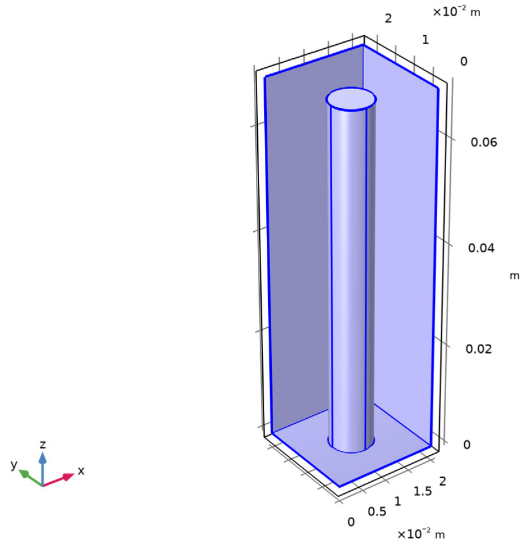
7 In the **Create Selection** dialog box, type `Metal` in the **Selection name** text field.

8 Click **OK**.

Air (mat2)

1 In the **Model Builder** window, click **Air (mat2)**.

2 Select Domain 2 only.



3 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

4 Click **Create Selection**.

5 In the **Create Selection** dialog box, type `Air` in the **Selection name** text field.

6 Click **OK**.

MOVING MESH

Deforming Domain 1

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

2 In the **Settings** window for **Deforming Domain**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Air**.

SOLID MECHANICS (SOLID)

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

2 In the **Settings** window for **Solid Mechanics**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Metal**.

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 **Geometric Nonlinearity** section.
3. From the **Formulation** list, choose **Geometrically linear**.

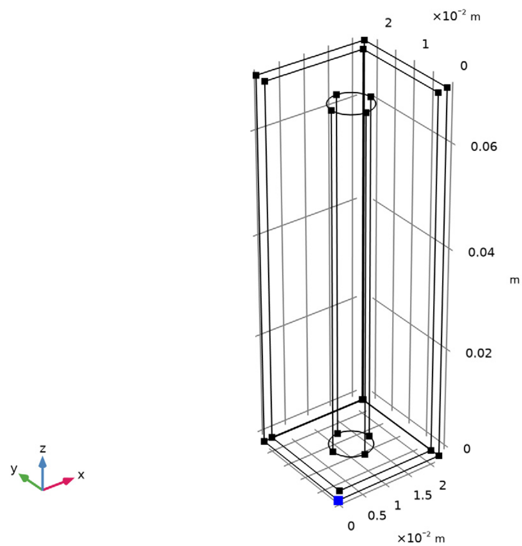
Thermal Expansion 1

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**. In the associated text field, type `T0`.
4. From the T list, choose **User defined**. In the associated text field, type `T1`.

Prescribed Displacement 1

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

   Appropriate prescribed displacement on points must be applied to eliminate any translation or rotation of the structure.
2. Select Point 1 only.

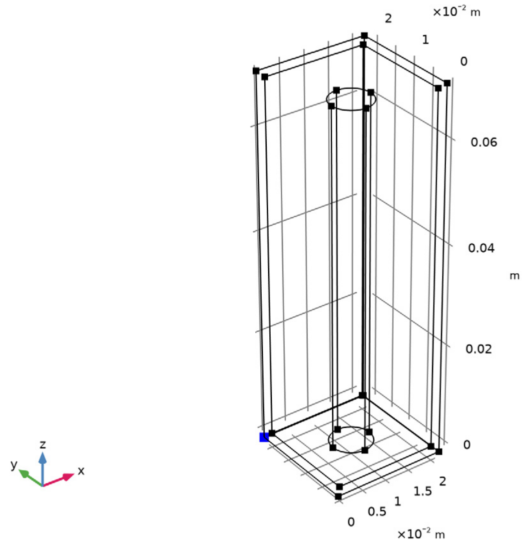


3. In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.
4. From the **Displacement in x direction** list, choose **Prescribed**.
5. From the **Displacement in y direction** list, choose **Prescribed**.
6. From the **Displacement in z direction** list, choose **Prescribed**.

Prescribed Displacement 2

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

2 Select Point 3 only.



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

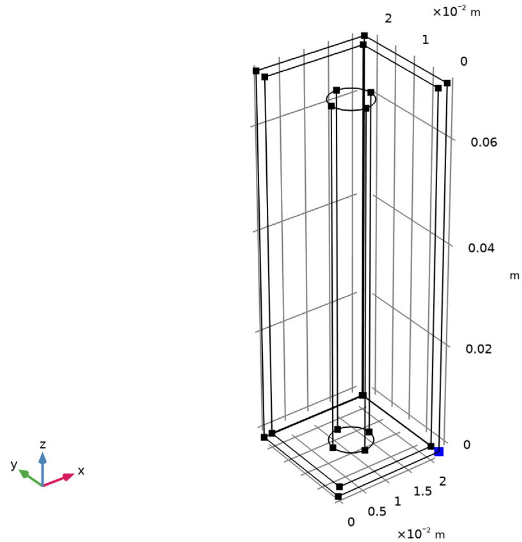
4 From the **Displacement in x direction** list, choose **Prescribed**.

5 From the **Displacement in z direction** list, choose **Prescribed**.

Prescribed Displacement 3

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

2 Select Point 21 only.



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

4 From the **Displacement in y direction** list, choose **Prescribed**.

5 From the **Displacement in z direction** list, choose **Prescribed**.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

The losses on metallic surfaces are negligible since the analysis frequency is low, so the boundaries are modeled as perfect electric conductors (PEC) by default.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electromagnetic Waves, Frequency Domain (emw)**.

2 In the **Settings** window for **Electromagnetic Waves, Frequency Domain**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Air**.

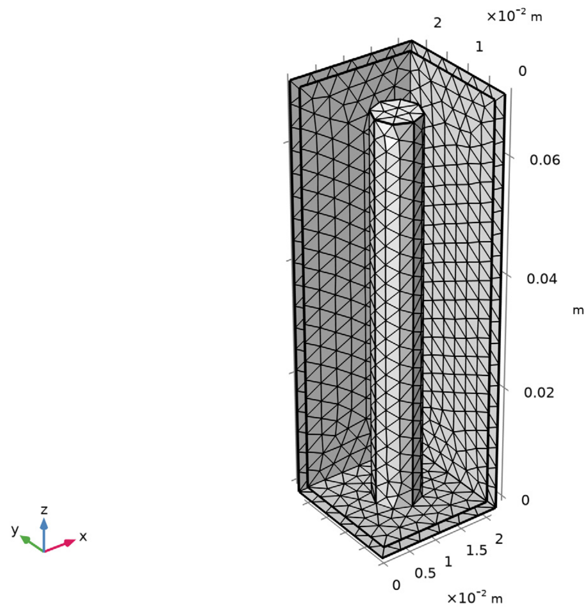
MESH 1

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

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

3 From the **Element size** list, choose **Coarser**.

4 Click **Build All**.



STUDY 1

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 From the list in the **Parameter name** column, choose **T1 (Operating temperature)**.

5 Click **Range**.

6 In the **Range** dialog box, choose **Number of values** from the **Entry method** list.

7 In the **Start** text field, type `T0`.

8 In the **Stop** text field, type `T1`.

9 In the **Number of values** text field, type `11`.

10 Click **Replace**.

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

RESULTS

Stress (solid)

Follow these instructions to reproduce the plot in Figure 2 that shows the structural deformation.

Volume 1

1. In the **Model Builder** window, expand the **Stress (solid)** node, then click **Volume 1**.
2. In the **Settings** window for **Volume**, locate the **Expression** section.
3. In the **Expression** text field, type `solid.disp`.
4. In the **Stress (solid)** toolbar, click **Plot**.

Next, reproduce the electric field and surface current plot for the filter's lowest eigenmode, shown in Figure 3.

Multislice

1. In the **Model Builder** window, expand the **Electric Field (emw)** node, then click **Multislice**.
2. In the **Settings** window for **Multislice**, locate the **Multiplane Data** section.
3. Find the **y-planes** subsection. In the **Planes** text field, type `0`.
4. Find the **z-planes** subsection. In the **Planes** text field, type `0`.
5. Locate the **Coloring and Style** section. Click **Change Color Table**.
6. In the **Color Table** dialog box, select **Thermal>ThermalDark** in the tree.
7. Click **OK**.

Surface 1

1. In the **Model Builder** window, right-click **Electric Field (emw)** and choose **Surface**.
2. In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Electromagnetic Waves, Frequency Domain>Currents and charge>emw.normJs - Surface current density norm - A/m**.
3. In the **Electric Field (emw)** toolbar, click **Plot**.

Having confirmed the correctness of the model setup, compute the thermal drift over the operating temperature range of 0° C to 100° C.

1D Plot Group 4

1. In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, locate the **Data** section.
3. From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (sol3)**.

4 Click to expand the **Title** section. From the **Title type** list, choose **None**.

Global 1

1 Right-click **1D Plot Group 4** and choose **Global**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`emw.freq`	`Hz`	`Frequency`

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

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

7 In the table, enter the following settings:

Legends
`Copper only`

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

Compare the copper-only design with one in which the post is made of steel.

ADD MATERIAL

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

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

3 In the tree, select **Built-in>Steel AISI 4340**.

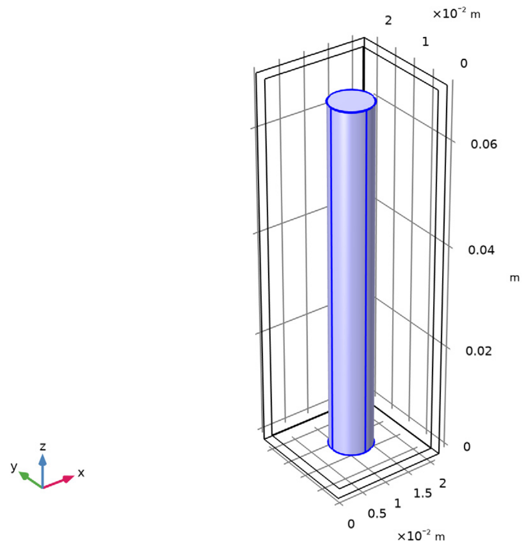
4 Click **Add to Component** in the window toolbar.

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

MATERIALS

Steel AISI 4340 (mat3)

Select Domain 3 only.



STUDY 1

Solver Configurations

In the **Study** toolbar, click **Create Solution Copy**.

RESULTS

Global 1

1 In the **Model Builder** window, under **Results>1D Plot Group 4** click **Global 1**.

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

3 From the **Dataset** list, choose **Study 1/Parametric Solutions 1 - Copy 1 (sol15)**.

Stress (solid)

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

Global 1

Right-click **Global 1** and choose **Duplicate**.

Global 2

1 In the **Model Builder** window, click **Global 2**.

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

3 From the **Dataset** list, choose **From parent**.

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

Legends
Copper + Steel

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

This reproduces the plot in Figure 4.