Created in COMSOL Multiphysics 6.3

# Thermo-Structural Effects on a Cavity Filter

## *Introduction*

This example investigates the electrical performance of a cascaded cavity filter operating in the millimeter-wave 5G band, including temperature changes. The thermal variations result in structural deformations of the filter geometry. Thus, the resonant frequencies of the filter elements (cavities) are affected by the thermo-structural phenomena. S-parameters are computed for different thermal expansion configurations.

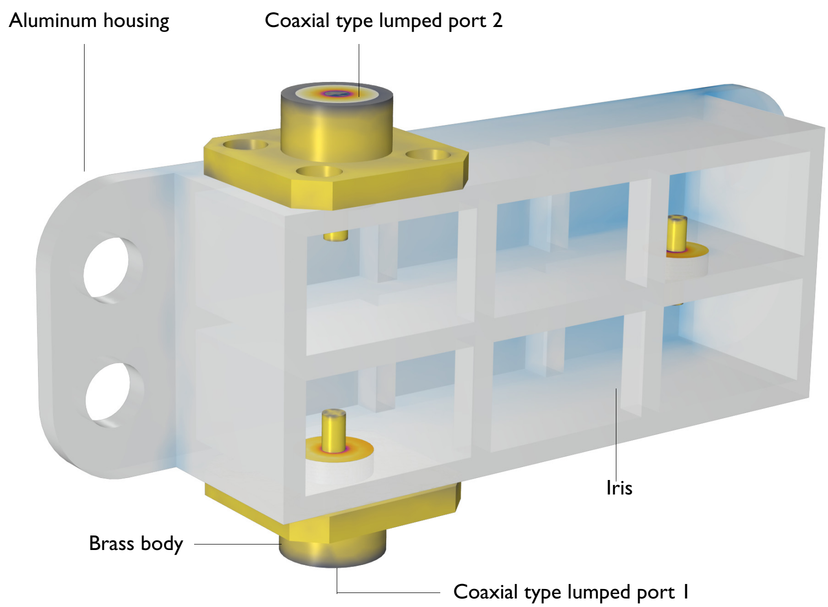



*Figure 1: Cascaded cavity filter with 2.92 mm (K) connectors. The front panel is removed to show the interior parts.*

## *Model Definition*

This example consists of three modeling parts:

- Cascaded cavity filter covering two millimeter-wave 5G bands separately using two different parameter input files. One frequency band is 26.5–29.5 GHz for Japan, Korea, and the United States, whereas the other band, for the European Union and China, is 24.25–27.5 GHz.

- Thermal deformation with the prescribed uniform temperature distribution and its impact on the bandpass filter performance.
- Thermal deformation with the computed nonuniform temperature distribution and its impact on the bandpass filter performance.

In all cases, the properties of the dielectric are assumed to be independent of the temperature. Of course, one could add a temperature dependency as in the RF Heating waveguide example (Ref. 1).

### Cavity Filters for the US and EU Millimeter-Wave 5G Bands

The filter includes six rectangular cavities and two 2.92 mm (K) connectors. The first series of cavities are combined through irises that are considered as inductive due to the polarization excited from the coaxial pin and the orientation of the irises. The housing body and connector volume are quite thick with respect to the simulation frequencies, and no wave penetration is expected. Thus, only the cavity and connector walls are part of the model domain. All metallic boundaries are considered to be lossy with finite conductivity and modeled using an impedance boundary condition. The dielectric part of the coaxial connectors is defined as lossless. The two connectors are excited and terminated using the coaxial type of lumped ports with 50 Ω reference impedance, respectively.

The material labeled as Coated Surfaces is added for the impedance boundary condition feature. The electrical property of the high-conductivity coating of the exposed surfaces is a function of the temperature. See Ref. 2 for details. The frequency domain study is run for the US 5G band (26.5–29.5 GHz) first. The second study is using a different parameter input file, which updates the geometry to be resonant for the EU 5G band (24.25–27.5 GHz).

### Thermal Deformation with the Prescribed Uniform Temperature Distribution

To include thermal deformation effects in the model, material properties are updated. The dielectric material in the coaxial connectors is modified to include the coefficient of thermal expansion (CTE) Young modulus, Poisson ratio, density, thermal conductivity, and specific heat. The values are chosen to be similar to those of FR4 material. The thermal deformation analysis requires to include the entire body of filter and connectors. So, the Aluminum material is added from the library. Brass material is added, with representative properties. Adhesive Layer material is also defined.

The model incorporates the effect of temperature variations on device performance with two cases:

- The part operating at different (but uniform) ambient temperatures.
- What happens when there is a temperature difference across the part. This can occur when a nearby electrical component has a thermal excursion due to overheating, for example.

Several effects can be considered. The solid structure itself will expand and contract in response to temperature increases and decreases. This will lead to a change in the sizes of the air cavities, irises, and dielectric materials within the filter. Both the deformations of the solid and the air domains need to be considered. Next, the material properties, most notably the electric conductivity of the coating, will change with temperature. The relative permittivity as well as the structural and thermal properties, can also be functions of temperature. However, in this model, they are assumed to be constant.

For the structural deformation is concerned, one must make a few assumptions about the attachments of the filter to the surroundings. One can assume that the structure is firmly attached to the flexible surrounding supporting structure, which would require that a structural model of the surroundings also need to be considered. One can also assume that the surroundings are perfectly rigid and that there is a small layer of adhesive attaching the filter to the rigid substrate structure. The latter assumption is simpler, and the one that will be used here. For modeling the connection between the deformed filter to the rigid substrate, the Spring Foundation feature is used. The deformations are initially negligible and the connection will gradually relax into its deformed state.

On two faces at which the lumped port boundary conditions are applied, it is assumed that these faces remain planar and that the cross-section is still an annulus under the thermal variations. These faces are modeled via the two Rigid Connector features, which enforce that the faces do not change shape or size, but can move or rotate due to the deformation of the filter itself.

The filter itself is simply modeled at three different isothermal conditions, -40°C, 20°C, and 120°C (that are -60 K, 0 K, and 100 K deviations in temperature about 20°C). See Parameters 2 node. The effect of the thermal expansion is considered by adding the Thermal Expansion subfeature to the Linear Elastic Material feature within Solid Mechanics interface.

A Moving Mesh feature is added under Definitions node to define the deformation of the air domain. The deformation settings follow the example Microwave Filter on PCB with Stress (Ref. 3).

The study uses a parametric sweep to update the temperature variation. The parametric sweep solves for the deformation of the structure and the air domains for three different operating temperatures. For each temperature, a frequency sweep is performed.

### THERMAL DEFORMATION WITH THE COMPUTED NONUNIFORM TEMPERATURE DISTRIBUTION

This part is meant to address the case of a 5G filter with a nonuniform temperature distribution that needs to be computed, rather than an imposed fixed uniform temperature deviation. The assumption of the analysis here is that the plate upon which the device sits experiences nonuniform heating due to some outside (nonmodeled) source. This nonuniform heating, linearly increasing to the positive $x$ direction, imposes a temperature variation across the bottom face that will nonuniformly heat and distort the filter and affect its performance.

The Heat Transfer in Solids interface is added and is applied to all solid domains. This means that the heat flux inside of the cavities due to conduction and convection through the air is neglected. This a reasonable assumption for such small cavities, as air is a good insulator. Furthermore, radiation within the cavities and to the surrounding is also neglected. This is reasonable since the conductive heat path to the baseplate is the most significant. The thermal effects of the thin high-conductivity coatings, added for improved electromagnetic performance, are neglected, as they are thermally insignificant.

The temperature variation of the baseplate is modeled via the Heat Flux boundary condition. The baseplate is assumed to be a relatively large plate of metal with a known temperature variation across it with an external temperature distribution.

Within the settings for the Heat Transfer in Solids, the Discretization is set to Linear. This is necessary as the discretization of the thermal problem should be one order lower than the Solid Mechanics problem. The latter is Quadratic by default. When starting from the Thermal Stress entry point within the Add Physics wizard, this combination of discretizations is set automatically; otherwise, it needs to be set manually. The Solid Mechanics is identical to the previous model configuration.

In the Impedance Boundary Condition, the Model Input setting has the Temperature set to Temperature (ht), the computed temperature distribution. This physics interface is otherwise identical to the previous model.

The Multiphysics branch has a Thermal Expansion feature that applies the computed temperature distribution as a thermal expansion within the Solid Mechanics interface. This combines two physics interfaces for bi-directional coupling of multiple physical phenomena.

Step 1 first solves the thermo-structural problem, and then the electromagnetic problem is solved on the deformed state.

## *Results and Discussion*

Total four studies are run and some of the default plots are modified. Since the front surfaces are removed from the physics view, the plots of the stress and heat distribution on the aluminum housing do not show the results for the boundaries removed from the view. These can be retrieved by deleting/disabling the Hide for Physics under the View node.

### CAVITY FILTERS WITHOUT THERMAL DEFORMATION

The default multislice plot is adjusted to show the electric field norm inside each series of cavities. The field pattern indicates that the basic $TE_{101}$ cavity mode is excited around the center frequency. The field strength is also sustained from the input port 1 to output port 2; see Figure 2. The volume of the aluminum housing, connector body, and interior part of the coaxial pin are not included in the simulation and results analysis.

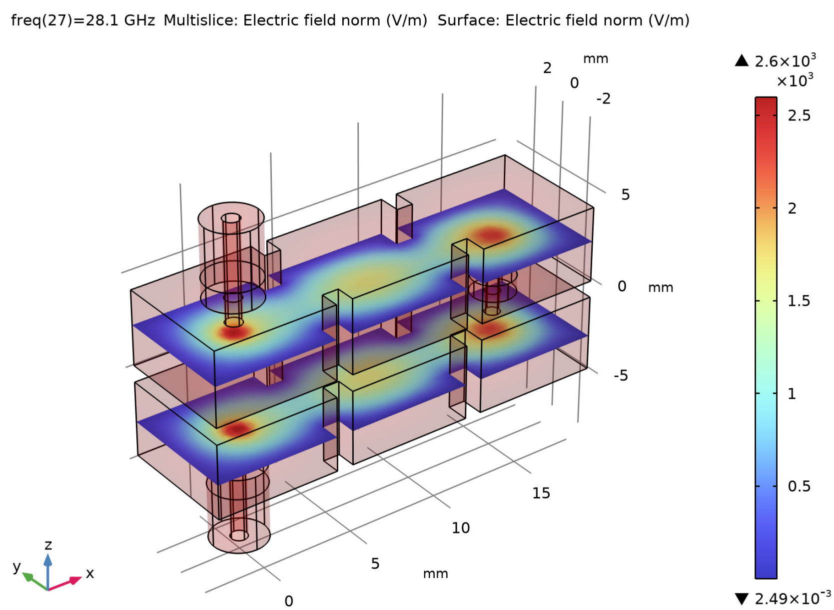



*Figure 2: Electric field norm inside the cavities. The dominant TE resonance at each cavity is observed when the plotting frequency is close to the center of the millimeter-wave 5G band for Japan, Korea, and the US.*

The S-parameter plot shows the impedance matching properties ($S_{11}$) with respect to the reference impedance 50 Ω and insertion loss ($S_{21}$) in dB scale. The first parameter input file shapes the cavities, irises, and coaxial pins to comply with the millimeter-wave 5G band from 26.5 to 29.5 GHz. The insertion loss $S_{21}$ is better than 0.25 dB, and $S_{11}$ is below -17.5 dB in the given bandwidth (Figure 3).

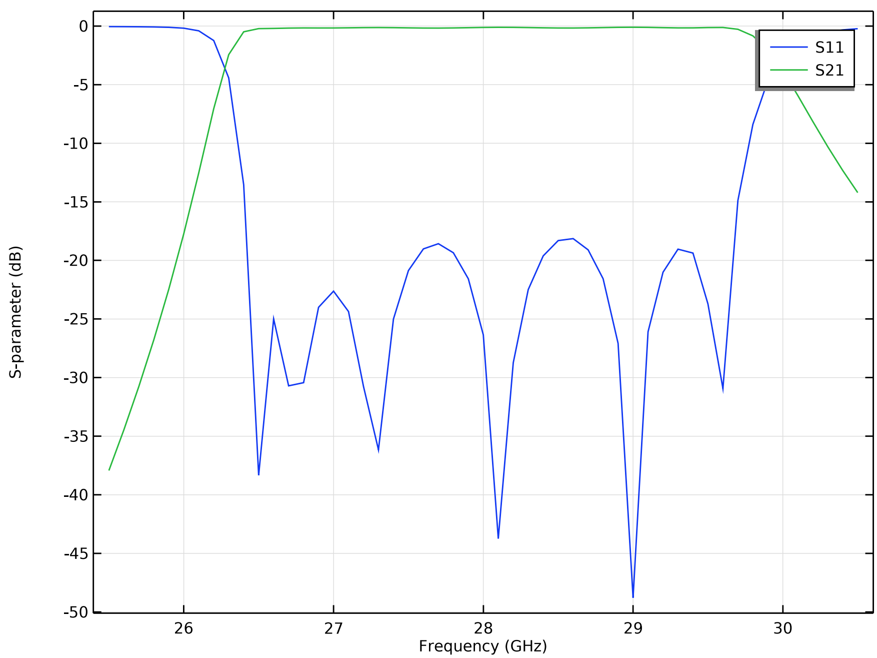



*Figure 3: S-parameter plot for the millimeter-wave 5G band for Japan, Korea, and the US.*

The simulation is repeated with the updated geometry to satisfy the EU millimeter-wave 5G band. The generated results show reasonably acceptable performance similar to that of the geometry for the US band. The factional bandwidth of the EU band (from 24.25 to 27.5 GHz) is wider than the US bandwidth. With the same number of cavity elements, the wider bandwidth results in degraded $S_{11}$, overall below -13 dB, and 0.3 dB of insertion loss in Figure 5.

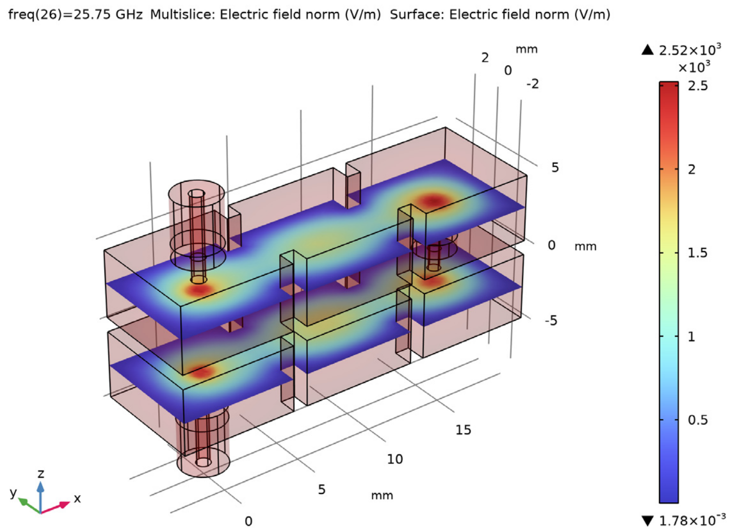



*Figure 4: Electric field norm inside the cavities. The dominant mode TE resonance at each cavity is observed when the plotting frequency is close to the center of the millimeter-wave 5G band for the EU and China.*

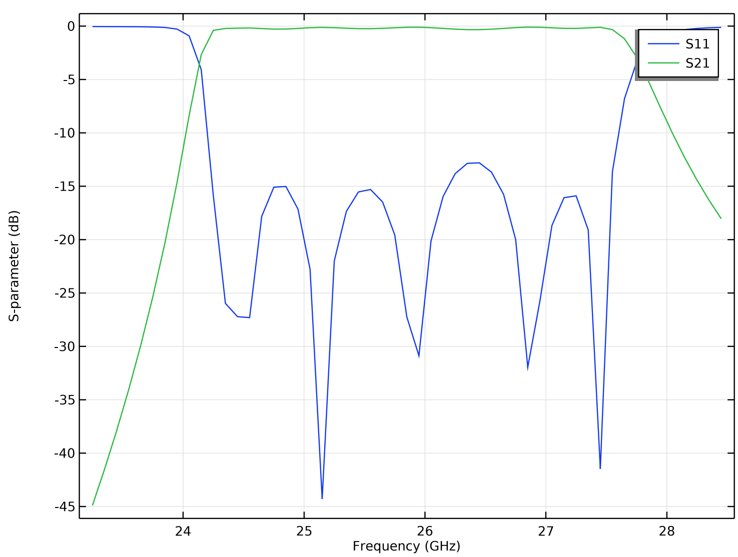



*Figure 5: S-parameter plot for the millimeter-wave 5G band for EU and China.*

## Thermal Deformation

The structural mechanics analysis is added in the study, and the default plot includes a thermal expansion, von Mises stress (Figure 8) plot.

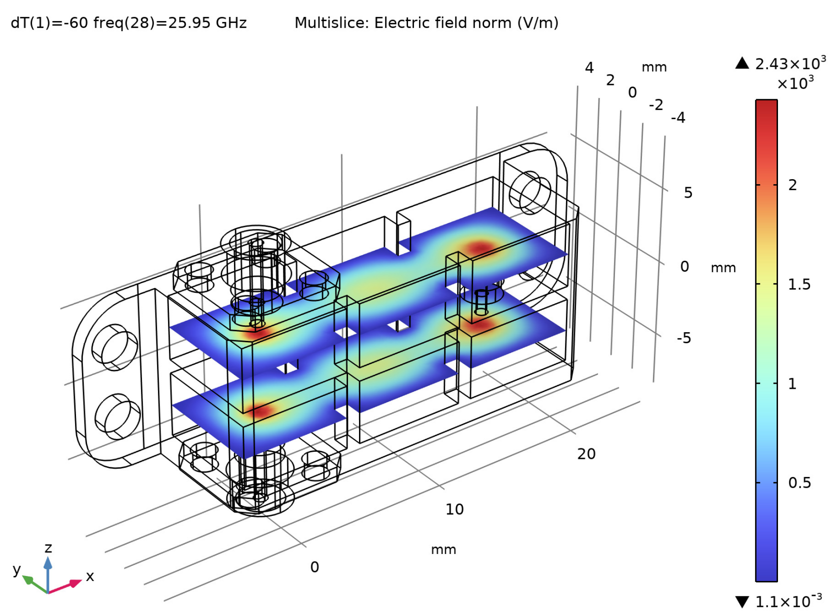



*Figure 6: Electric field norm around the center frequency of the EU millimeter-wave 5G band when the temperature is -40°C. The thermal-structural deformation did not distort the dominant TE resonant modes in the cavities.*

The default plot of the electromagnetics simulation is modified to show the cavity resonances in Figure 6. By looking at the field plot, the impact from the thermal expansion does not distort the dominant TE cavity modes at the center frequency. When the temperature is decreased, the thermal-structural deformation induces smaller cavities, and the S-parameters are shifted toward the higher frequencies. On the other hand, with the increased temperature, frequency responses are downshifted. The overall filter performance in terms of S-parameters (Figure 7) is not severely affected by the thermal expansion of the filter geometry. The frequency shift of -3 dB insertion loss $S_{21}$ or -10 dB $S_{11}$ is less than 100 MHz when the ambient temperature changes from -40 to 120°C.

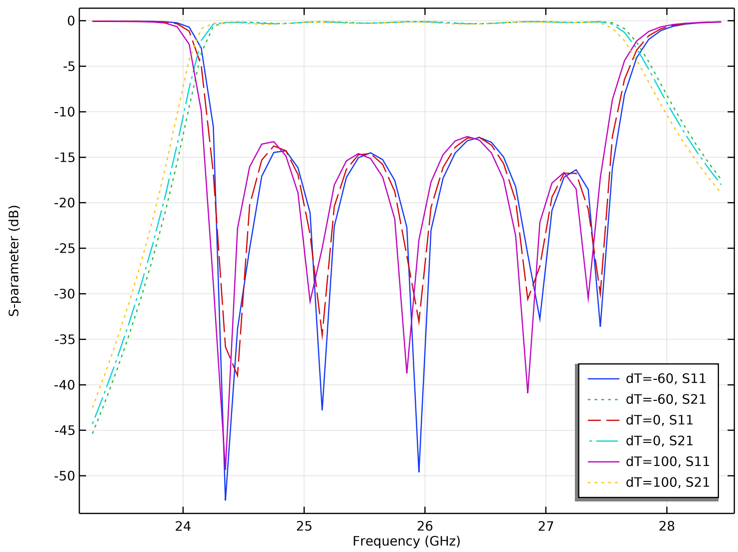



*Figure 7: S-parameters are slightly shifted as a function of ambient temperature.*

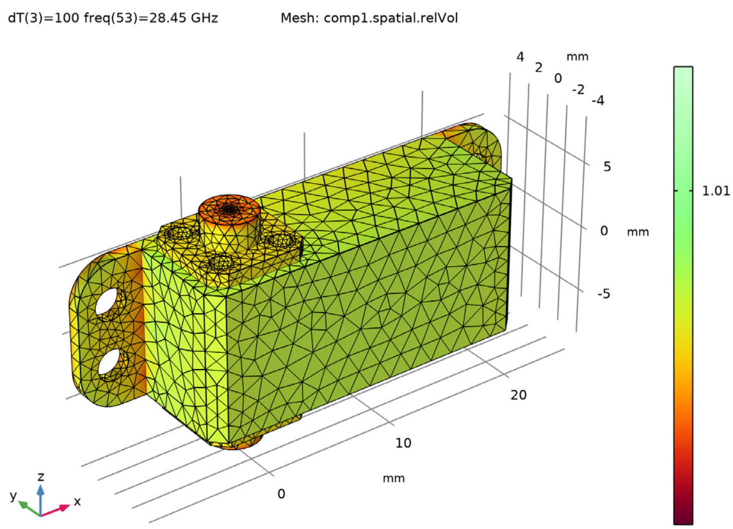



*Figure 8: Thermal expansion at 120°C that is 100K above the reference temperature.*

## THERMAL DEFORMATION WITH COMPUTED TEMPERATURE

The last study is a combination of electromagnetics, structural mechanics, and heat transfer. The default plots include the electric field norm distribution, S-parameters, von Mises stress, and heat distribution. The top multislice plot in the Figure 9 is the electric field norm plot outside the passband. The plotted field strength indicates the input signal at port 1 does not propagate to the output port 2. The bottom multislice plot describes the electric field distribution at the passband.

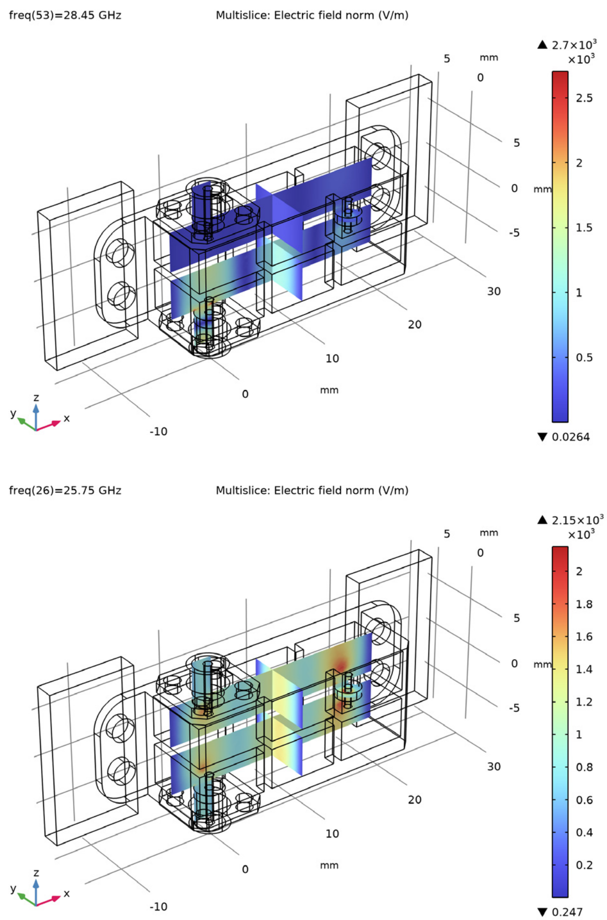



*Figure 9: The electric field norm beyond the passband is presented at the top, with the input signal failing to reach the output port. Conversely, the electric field emanating from the excited port was detected with minimal attenuation at the center of the passband, as illustrated at the bottom.*

The filter is deformed by the uneven heat source on the baseplate. However, as shown in Figure 10, the S-parameters are not significantly distorted. The level of deformation in Figure 11 is less compared to the results of the ambient temperature change in Figure 8. The temperature distribution plots (Figure 12 and Figure 13) give an idea intuitively on which part of the aluminum housing and coaxial structures are hotter and suffer more from the thermo-structural effects.

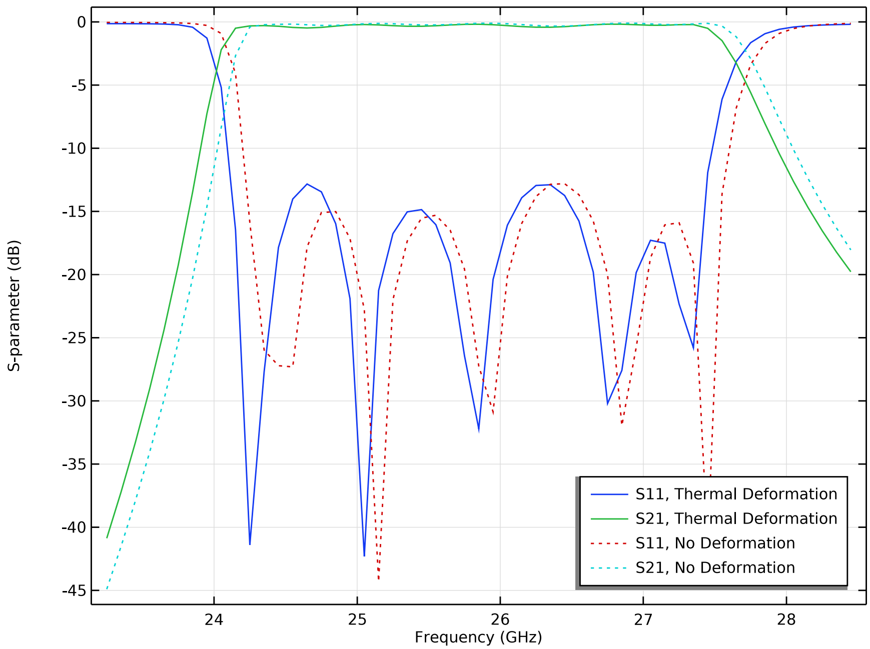



*Figure 10: S-parameters affected by the uneven heat source from the baseplate.*

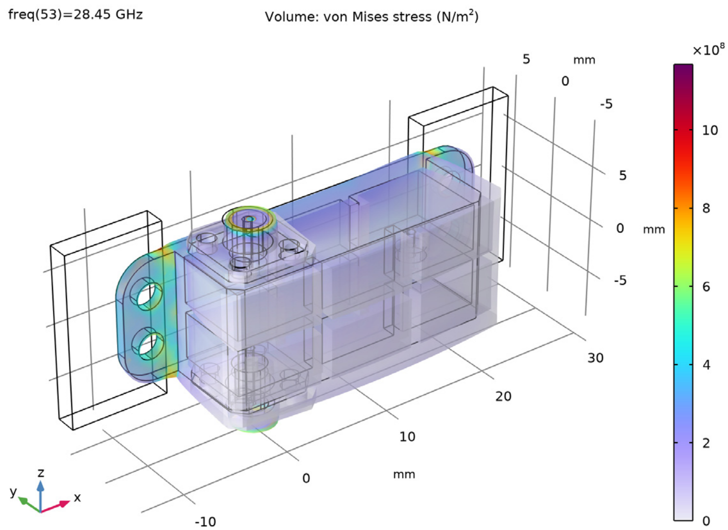



*Figure 11: Deformed aluminum housing due to the heat expansion.*

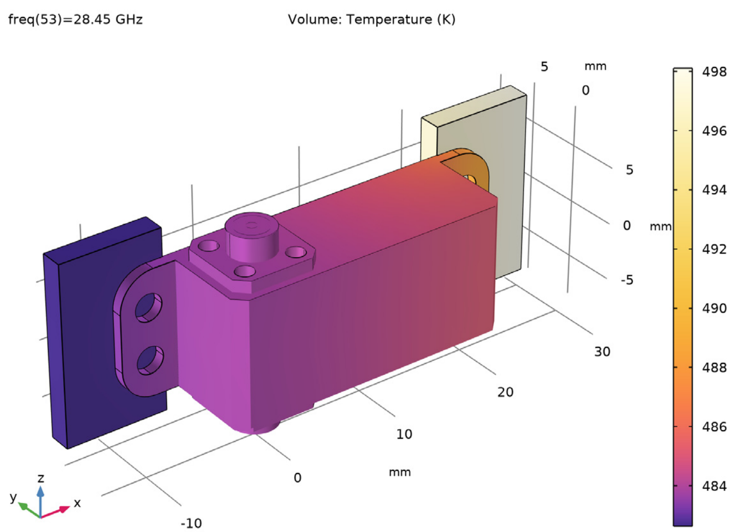



*Figure 12: Surface plot of the temperature field.*

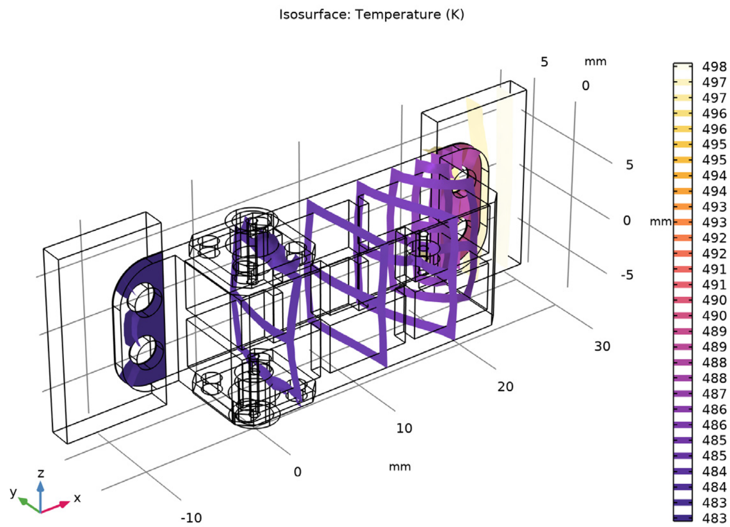



*Figure 13: Isosurface plot of the temperature field.*

## *References*

1. https://www.comsol.com/model/rf-heating-6078

2. https://www.comsol.com/video-training/getting-started/use-functions-define-material-property

3. https://www.comsol.com/model/microwave-filter-on-pcb-with-stress-47501

**Application Library path:** `RF_Module/Filters/cavity_filter_5g`

## *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 **Radio Frequency > Electromagnetic Waves, Frequency Domain (emw)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **General Studies > Frequency Domain**.
6 Click **Done**.

## 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 Click **Load from File**.
4 Browse to the model's Application Libraries folder and double-click the file `cavity_filter_5g_us.txt`.

   The loaded parameters will be used to build the filter geometry providing reasonable performance such as low insertion loss and impedance matching for the millimeter-wave 5G band in Japan, Korea, and the United States.

### *Parameters 2*

1 In the **Home** toolbar, click **Parameters** and choose **Add > Parameters**.
2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
3 In the table, enter the following settings:

| Name | Expression | Value | Description |
|---|---|---|---|
| dT | 0 | 0 | |
| T0 | 20[degC] | 293.15 K | |

## 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 `a0`.

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

5 In the **Height** text field, type `h0`.

6 Locate the **Position** section. In the **x** text field, type `-a0/2`.

7 In the **y** text field, type `-a0/2`.

8 In the **z** text field, type `-h0-w0/2`.

### Array 1 (arr1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Array**.

2 Select the object **blk1** only.

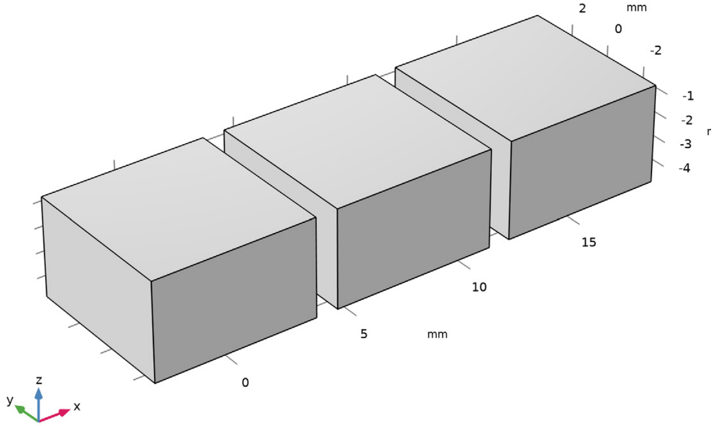
3 In the **Settings** window for **Array**, locate the **Size** section.

4 In the **x size** text field, type `3`.

5 Locate the **Displacement** section. In the **x** text field, type `a0+w0`.

6 Click **Build Selected**.

7 Click the **Zoom Extents** 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 `w0`.

4 In the **Depth** text field, type `i0`.

5 In the **Height** text field, type `h0`.

6 Locate the **Position** section. In the **x** text field, type `a0/2`.

7 In the **y** text field, type `-i0/2`.

8 In the **z** text field, type `-h0-w0/2`.

9 Click **Build Selected**.

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

*Block 3 (blk3)*

1 Right-click **Block 2 (blk2)** and choose **Duplicate**.

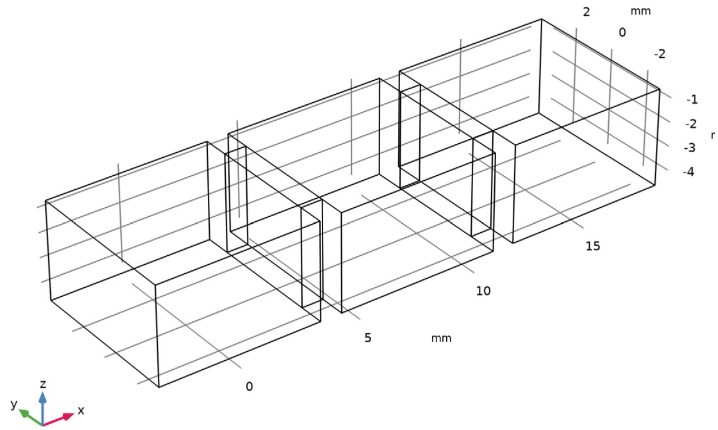
2 In the **Settings** window for **Block**, locate the **Size and Shape** section.

3 In the **Depth** text field, type `i1`.

4 Locate the **Position** section. In the **x** text field, type `a0*1.5+w0`.

5 In the **y** text field, type `-i1/2`.

6 Click **Build Selected**.




*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 `2.92[mm]/2`.

4 In the **Height** text field, type `c0`.

5 Locate the **Position** section. In the **z** text field, type `-8.5[mm]`.

*Cylinder 2 (cyl2)*

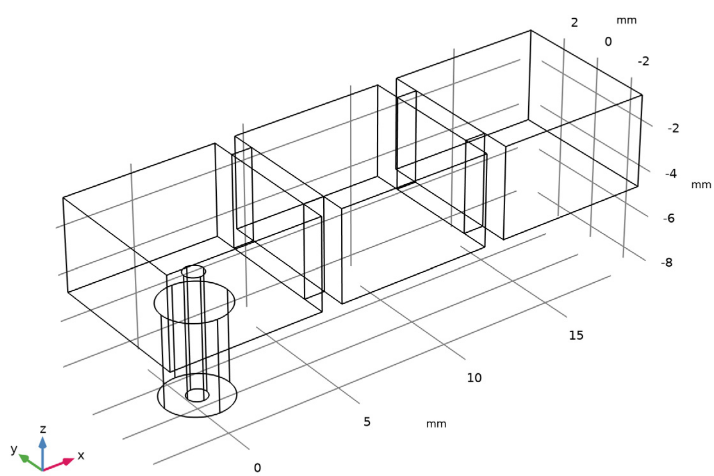
1 Right-click **Cylinder 1 (cyl1)** and choose **Duplicate**.

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

3 In the **Radius** text field, type `0.435[mm]`.

4 In the **Height** text field, type `p0`.

5 Click **Build Selected**.




*Union 1 (uni1)*

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Click the **Select All** button in the **Graphics** toolbar.

*Rotate 1 (rot1)*

1 In the **Geometry** toolbar, click **Transforms** and choose **Rotate**.

2 Select the object **uni1** only.

3 In the **Settings** window for **Rotate**, locate the **Input** section.

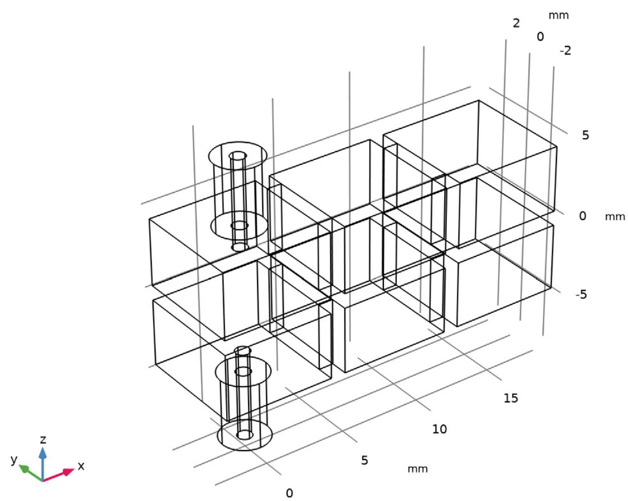
4 Select the **Keep input objects** checkbox.

5 Locate the **Rotation** section. From the **Axis type** list, choose **x-axis**.

**6** In the **Angle** text field, type `180`.

**7** Click **Build Selected**.

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




*Cylinder 3 (cyl3)*

**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 `0.4[mm]`.

**4** In the **Height** text field, type `p1`.

**5** Locate the **Position** section. In the **x** text field, type `2*(w0+a0)`.

**6** In the **z** text field, type `-p1/2`.

*Cylinder 4 (cyl4)*

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

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

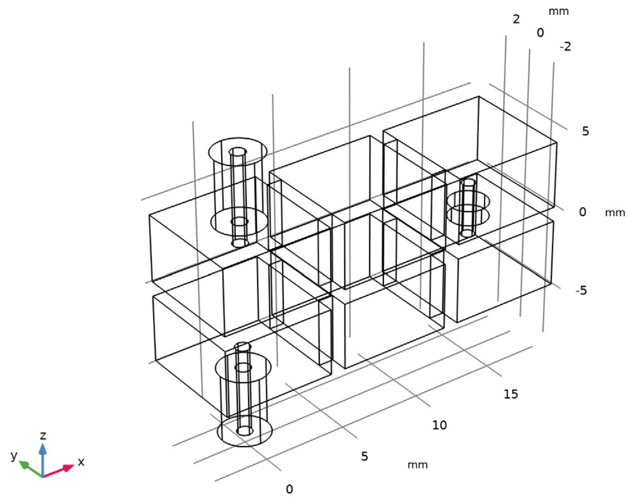
**3** In the **Radius** text field, type `r1`.

**4** In the **Height** text field, type `w0`.

**5** Locate the **Position** section. In the **x** text field, type `2*(w0+a0)`.

**6** In the **z** text field, type `-w0/2`.

7 Click **Build Selected**.




*Import 1 (imp1)*

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

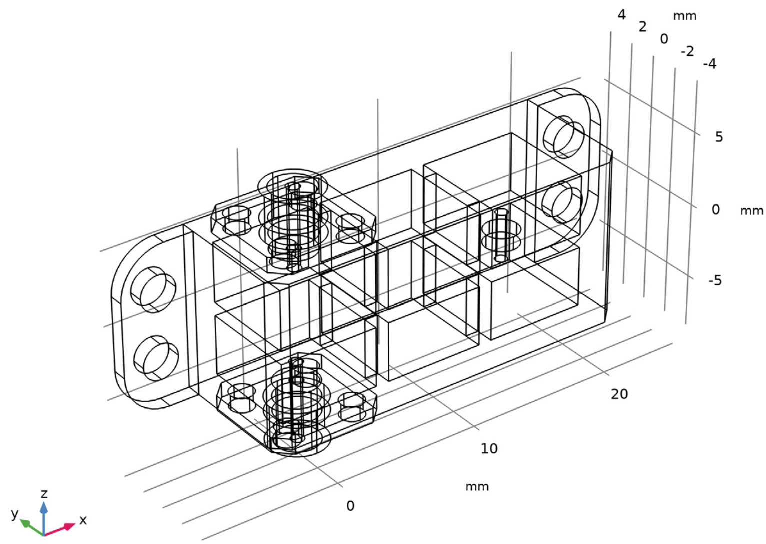
2 In the **Settings** window for **Import**, locate the **Source** section.

3 Click **Browse**.

4 Browse to the model's Application Libraries folder and double-click the file `cavity_filter_5g_case.mphbin`.

5 Click **Import**.

**6** Click **Build All Objects**.




*Work Plane 1 (wp1)*

**1** In the **Geometry** toolbar, click **Work Plane**.

**2** In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

**3** From the **Plane type** list, choose **Face parallel**.

**4** On the object **imp1**, select Boundary 21 only.

*Work Plane 1 (wp1) > Plane Geometry*

In the **Model Builder** window, click **Plane Geometry**.

*Work Plane 1 (wp1) > Rectangle 1 (r1)*

**1** In the **Work Plane** toolbar, click **Rectangle**.

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

**3** In the **Width** text field, type `16`.

**4** In the **Height** text field, type `9`.

**5** Locate the **Position** section. In the **xw** text field, type `-8`.

**6** In the **yw** text field, type `-22`.

### *Work Plane 1 (wp1) > Mirror 1 (mir1)*

1 In the **Work Plane** toolbar, click **Transforms** and choose **Mirror**.

2 Select the object **r1** only.

3 In the **Settings** window for **Mirror**, locate the **Input** section.

4 Select the **Keep input objects** checkbox.

5 Locate the **Normal Vector to Line of Reflection** section. In the **xw** text field, type `0`.

6 In the **yw** text field, type `1`.

7 Click **Build Selected**.

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

### *Extrude 1 (ext1)*

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

2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 In the table, enter the following settings:

| Distances (mm) |
|---|
| 2 |

4 Click **Build All Objects**.

Next, the model utilizes virtual geometry operations and selections. The **Form Composite Domains** feature is used four times to simplify the geometry by reducing the number of different distinct domains within the model. They are labeled to represent their materials.

### *Form Composite Domains (Cavity Air)*

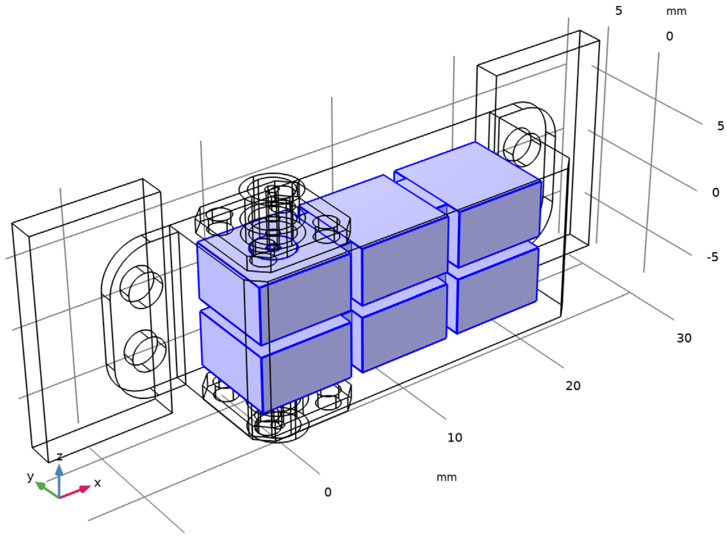
1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Form Composite Domains**.

2 In the **Settings** window for **Form Composite Domains**, type `Form Composite Domains (Cavity Air)` in the **Label** text field.

3 Locate the **Input** section. Click the **Paste Selection** button for **Domains to composite**.

4 In the **Paste Selection** dialog, type `6, 7, 26-33` in the **Selection** text field.

**5** Click **OK**.




*Form Composite Domains (Coax Brass)*

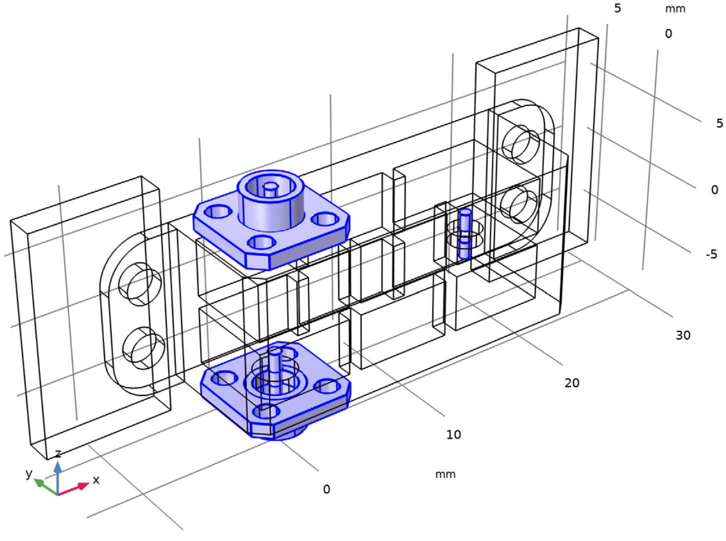
**1** In the **Geometry** toolbar, click **Virtual Operations** and choose **Form Composite Domains**.

**2** In the **Settings** window for **Form Composite Domains**, type `Form Composite Domains (Coax Brass)` in the **Label** text field.

**3** Locate the **Input** section. Click the **Paste Selection** button for **Domains to composite**.

**4** In the **Paste Selection** dialog, type `4, 5, 8-11, 18-25, 27, 29` in the **Selection** text field.

**5** Click **OK**.

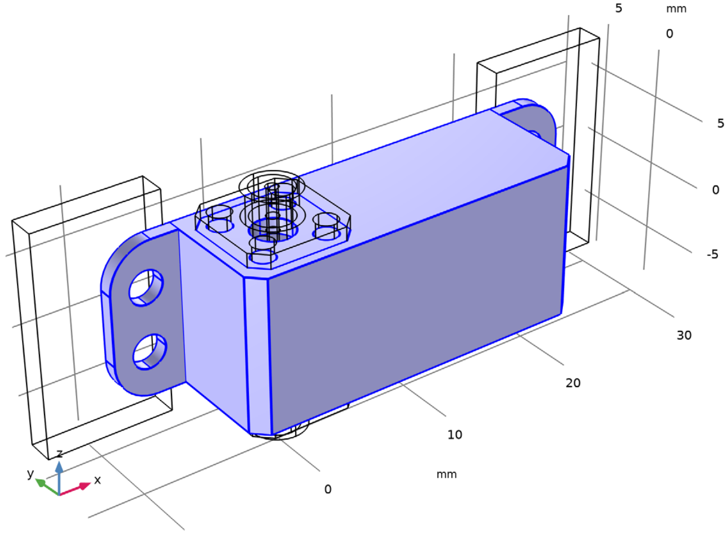



*Form Composite Domains (Aluminum Housing)*

**1** In the **Geometry** toolbar, click **Virtual Operations** and choose **Form Composite Domains**.

**2** In the **Settings** window for **Form Composite Domains**, type `Form Composite Domains (Aluminum Housing)` in the **Label** text field.

**3** On the object **cmd2**, select Domains 2, 3, and 20 only.

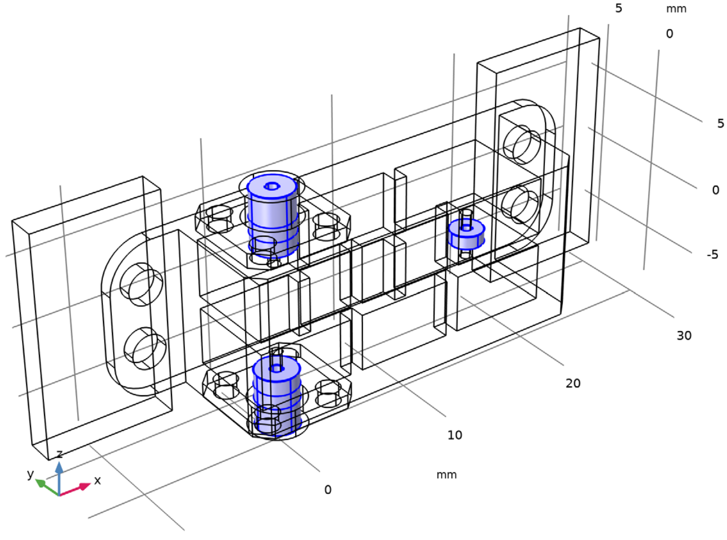



*Form Composite Domains (Coax Dielectric)*

**1** In the **Geometry** toolbar, click **Virtual Operations** and choose **Form Composite Domains**.

**2** In the **Settings** window for **Form Composite Domains**, type `Form Composite Domains (Coax Dielectric)` in the **Label** text field.

**3** On the object **cmd3**, select Domains 7–12 and 15 only.



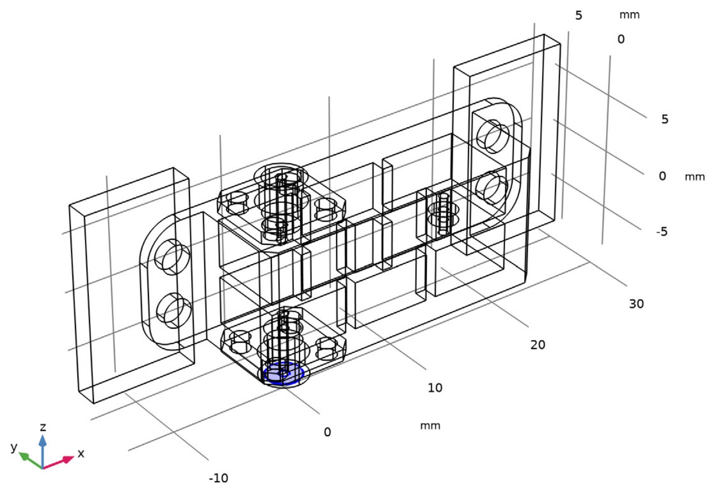

Define selections that are useful for configuring physics settings and material properties later on. Selections are defined for the various materials, as well as the surfaces that need to be coated with a high-conductivity material.

## DEFINITIONS

### *Lumped Port 1*

1. In the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for **Explicit**, type `Lumped Port 1` in the **Label** text field.
3. Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

**4** Select Boundary 80 only.



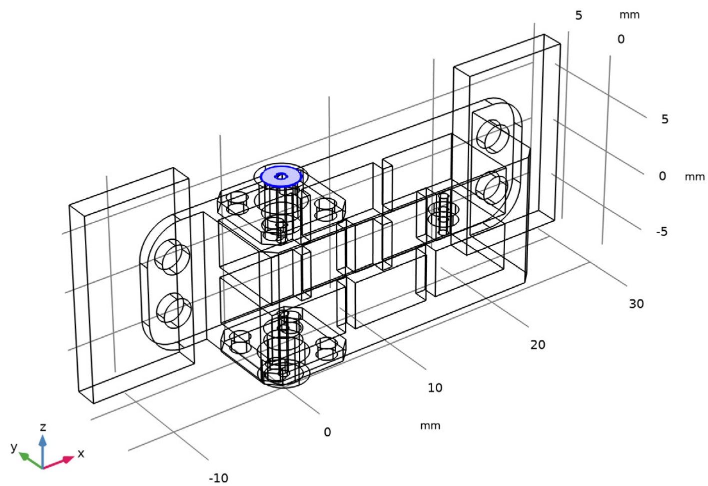

*Lumped Port 2*

**1** In the **Definitions** toolbar, click **Explicit**.

**2** In the **Settings** window for **Explicit**, type `Lumped Port 2` in the **Label** text field.

**3** Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 89 only.

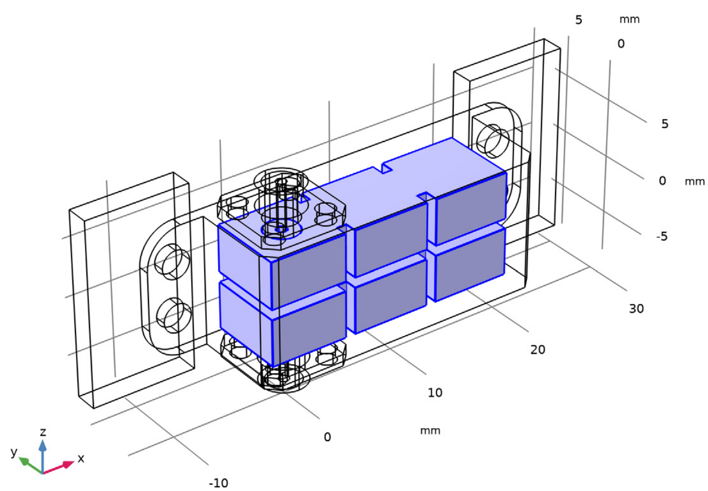



*Air Domains*

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Air Domains` in the **Label** text field.

3 Select Domains 5 and 6 only.

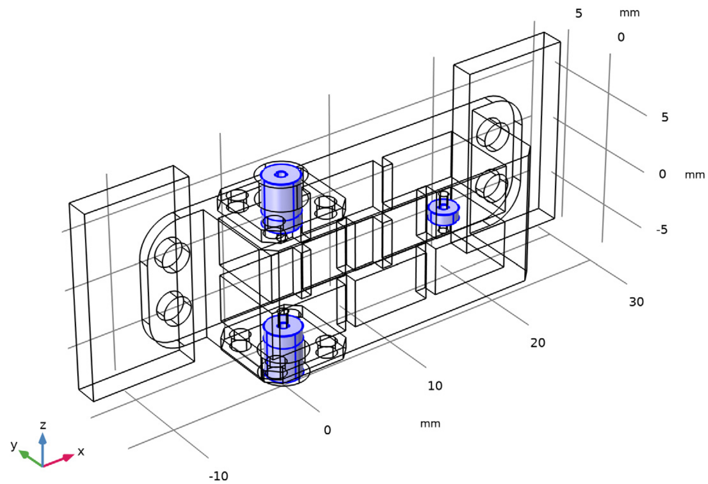


*Dielectric Domains*

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Dielectric Domains` in the **Label** text field.

3 Select Domains 7, 8, and 11 only.




*Surfaces of Electromagnetic Domains*

1 In the **Definitions** toolbar, click **Adjacent**.

2 In the **Settings** window for **Adjacent**, type `Surfaces of Electromagnetic Domains` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Input selections**, click **Add**.

4 In the **Add** dialog, select **Air Domains** in the **Input selections** list.

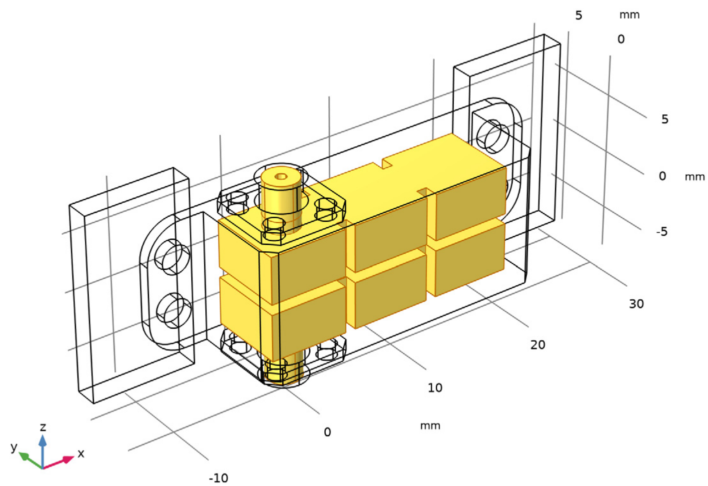
5 Click **OK**.

6 In the **Settings** window for **Adjacent**, locate the **Input Entities** section.

7 Under **Input selections**, click **Add**.

8 In the **Add** dialog, select **Dielectric Domains** in the **Input selections** list.

**9** Click **OK**.




### *Surfaces of Domains for Coating*

**1** In the **Definitions** toolbar, click **Difference**.

**2** In the **Settings** window for **Difference**, type `Surfaces of Domains for Coating` in the **Label** text field.

**3** Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

**4** Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

**5** In the **Add** dialog, select **Surfaces of Electromagnetic Domains** in the **Selections to add** list.

**6** Click **OK**.

**7** In the **Settings** window for **Difference**, locate the **Input Entities** section.

**8** Under **Selections to subtract**, click **Add**.

**9** In the **Add** dialog, select **Lumped Port 1** in the **Selections to subtract** list.

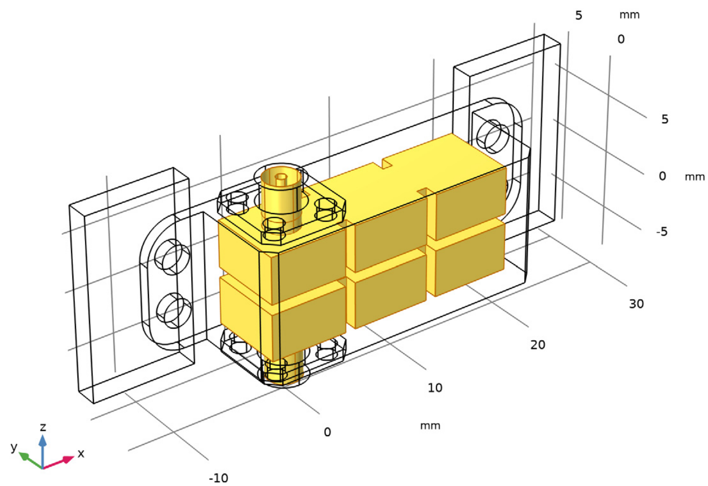
**10** Click **OK**.

**11** In the **Settings** window for **Difference**, locate the **Input Entities** section.

**12** Under **Selections to subtract**, click **Add**.

**13** In the **Add** dialog, select **Lumped Port 2** in the **Selections to subtract** list.

**14** Click **OK**.




*Electromagnetic Domains*

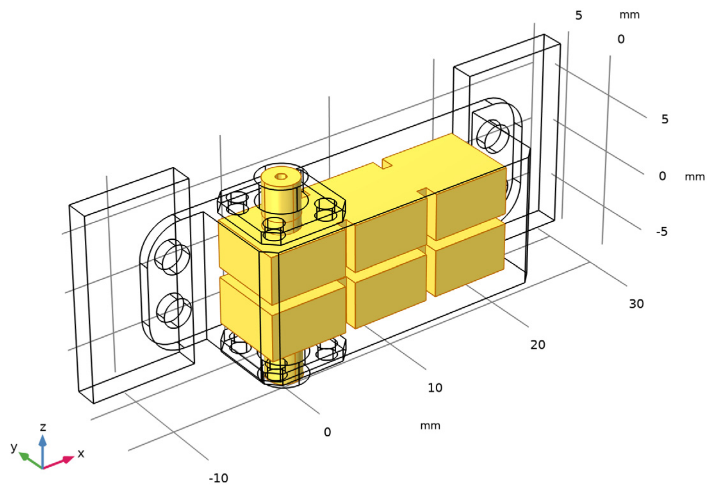
**1** In the **Definitions** toolbar, click **Union**.

**2** In the **Settings** window for **Union**, type `Electromagnetic Domains` in the **Label** text field.

**3** Locate the **Input Entities** section. Under **Selections to add**, click **Add**.

**4** In the **Add** dialog, in the **Selections to add** list, choose **Air Domains** and **Dielectric Domains**.

**5** Click **OK**.




### ADD MATERIAL

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

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

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

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

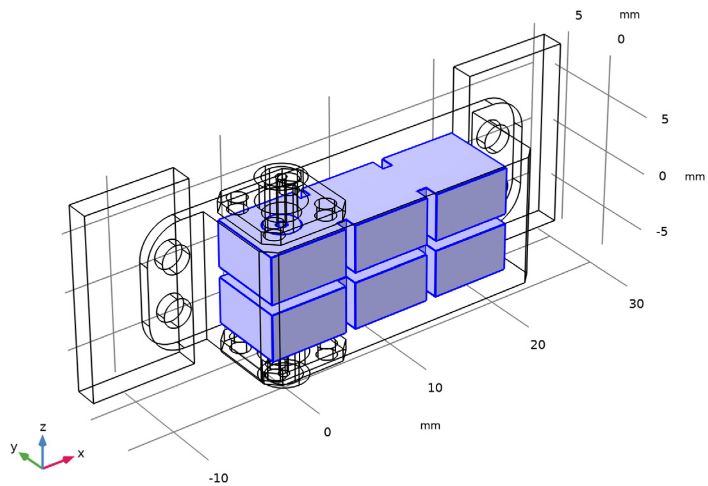
**5** In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

### MATERIALS

*Air (mat1)*

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

2 From the **Selection** list, choose **Air Domains**.

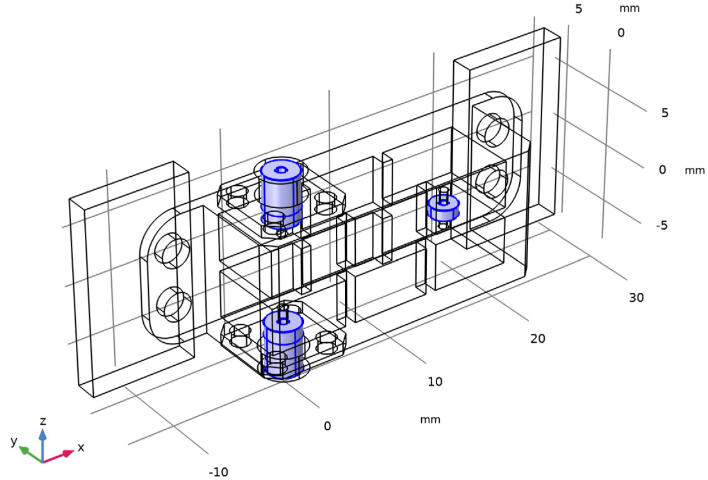



### *Coax Dielectric*

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

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

**3** Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Dielectric Domains**.




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

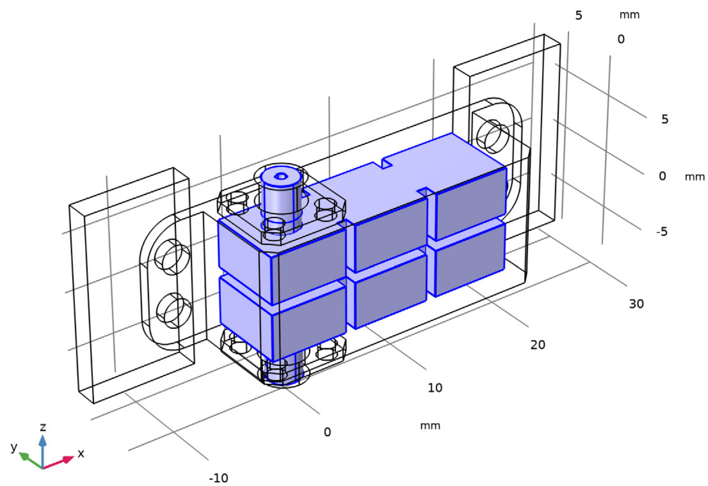
| Property | Variable | Value | Unit | Property group |
|---|---|---|---|---|
| Relative permittivity | epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0 | 2.1 | 1 | Basic |
| Relative permeability | mur_iso ; murii = mur_iso, murij = 0 | 1 | 1 | Basic |
| Electric conductivity | sigma_iso ; sigmaii = sigma_iso, sigmaij = 0 | 0 | S/m | Basic |

### ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

**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 **Electromagnetic Domains**.

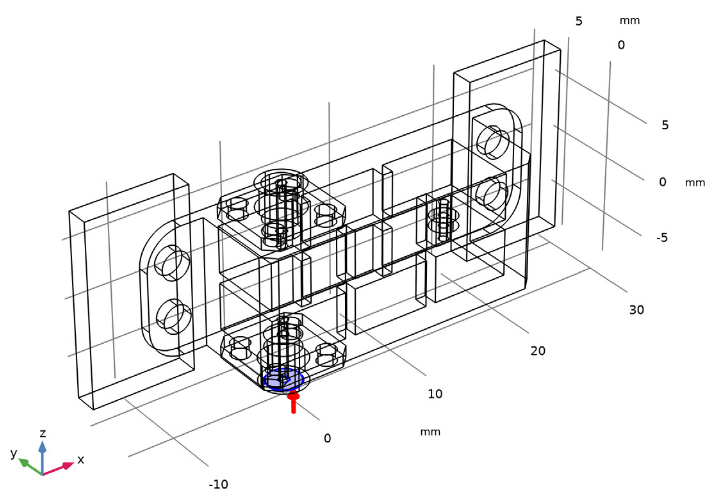



*Lumped Port 1*

1 In the **Physics** toolbar, click **Boundaries** and choose **Lumped Port**.

2 In the **Settings** window for **Lumped Port**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Lumped Port 1**.



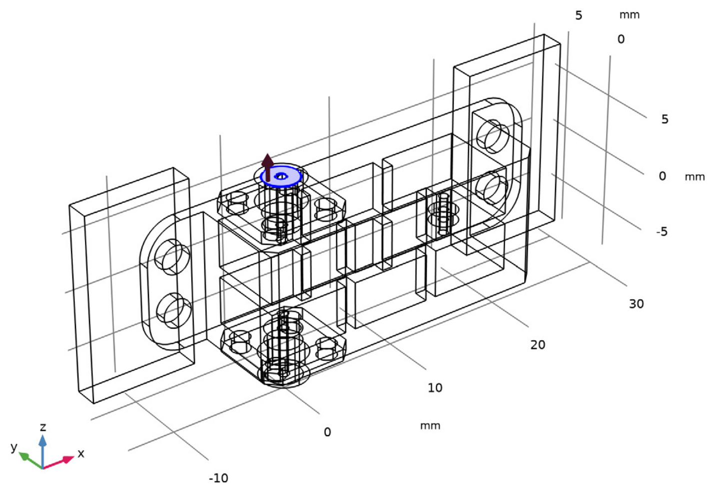
4 Locate the **Lumped Port Properties** section. From the **Type of lumped port** list, choose **Coaxial**.

*Lumped Port 2*

1 In the **Physics** toolbar, click **Boundaries** and choose **Lumped Port**.

2 In the **Settings** window for **Lumped Port**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Lumped Port 2**.



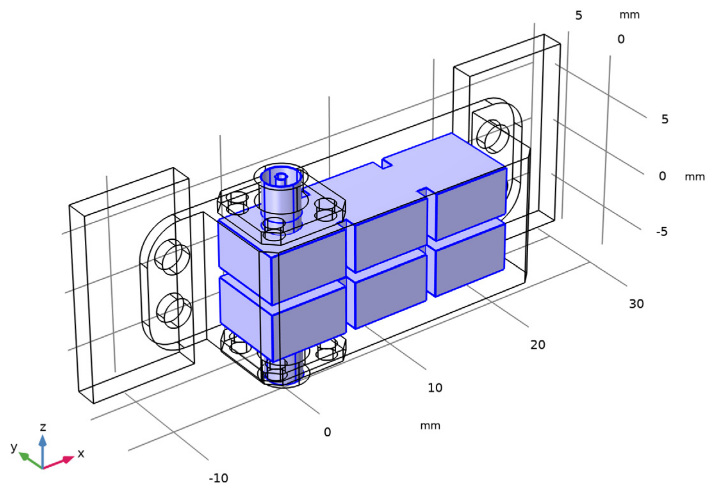

4 Locate the **Lumped Port Properties** section. From the **Type of lumped port** list, choose **Coaxial**.

*Impedance Boundary Condition 1*

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

2 In the **Settings** window for **Impedance Boundary Condition**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Surfaces of Domains for Coating**.




4 Click to expand the **Model Input** section. In the $T$ text field, type `T0+dT`.

In the **Impedance Boundary Condition** feature, the Model Input for Temperature is defined as `T0+dT` where `T0` is the reference ambient temperature and `dT` is the temperature deviation. This is used to evaluate the conductivity of the coating at the specified temperature.

## MATERIALS

### *Coated Surfaces*

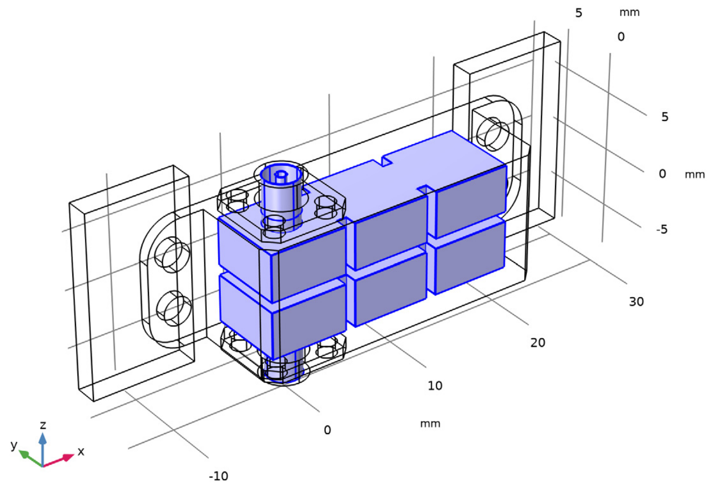
1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

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

3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.

**4** From the **Selection** list, choose **Surfaces of Domains for Coating**.

This is the same boundary selection as you specified the **Impedance Boundary Condition**.




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

| Property | Variable | Value | Unit | Property group |
|---|---|---|---|---|
| Relative permittivity | epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0 | 1 | 1 | Basic |
| Relative permeability | mur_iso ; murii = mur_iso, murij = 0 | 1 | 1 | Basic |
| Electric conductivity | sigma_iso ; sigmaii = sigma_iso, sigmaij = 0 | rho(T) | S/m | Basic |

**6** In the **Model Builder** window, expand the **Component 1 (comp1)** > **Materials** > **Coated Surfaces (mat3)** node, then click **Basic (def)**.

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

**8** Click + **Select Quantity**.

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

**10** Click **OK**.

*Analytic 1 (an1)*

1 Right-click **Component 1 (comp1)** > **Materials** > **Coated Surfaces (mat3)** > **Basic (def)** and choose **Functions** > **Analytic**.

2 In the **Settings** window for **Analytic**, type `rho` in the **Function name** text field.

3 Locate the **Definition** section. In the **Expression** text field, type `1/(2e-8[ohm*m]*(1+0.004[1/K]*(T-293.15[K])))`.

4 In the **Arguments** text field, type `T`.

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

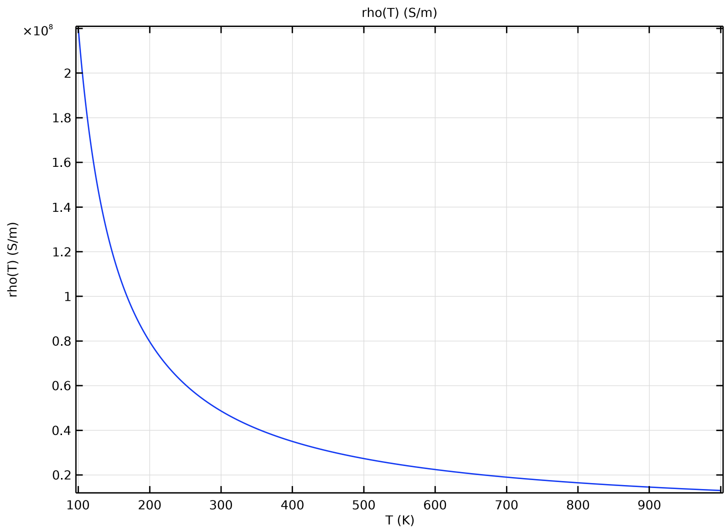
| Argument | Unit |
|---|---|
| T | K |

6 In the **Function** text field, type `S/m`.

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

| Plot | Argument | Lower limit | Upper limit | Fixed value | Unit |
|---|---|---|---|---|---|
| √ | T | 100 | 1000 | 0 | K |

**8** Click Plot.




These settings are for the electrical property used in the **Impedance Boundary Condition** representing the high conductivity coating of the exposed surfaces.

## STUDY 1 - US BAND

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

**2** In the **Settings** window for **Study**, type `Study 1 - US Band` in the **Label** text field.

### *Step 1: Frequency Domain*

Define the study frequency ahead of performing any frequency-dependent operation such as building mesh. The physics-controlled mesh uses the highest frequency value in the specified range.

**1** In the **Model Builder** window, under **Study 1 - US Band** click **Step 1: Frequency Domain**.

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

**3** In the **Frequencies** text field, type `range(25.5,0.1,30.5)`.

## MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Build All**.

#### DEFINITIONS

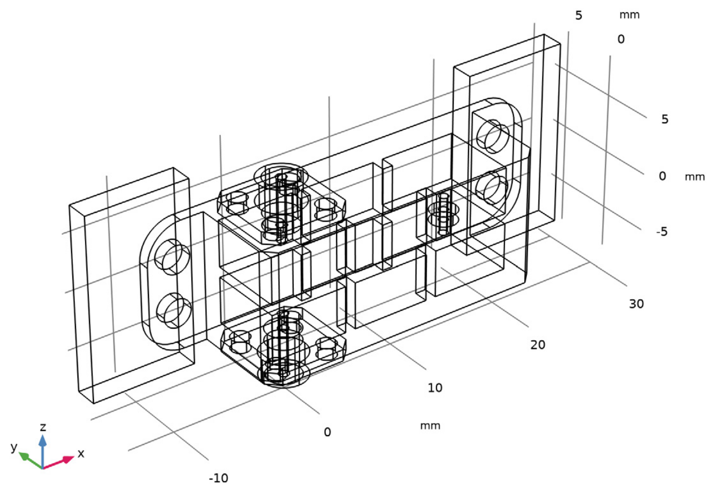
*Hide for Physics 1*

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

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

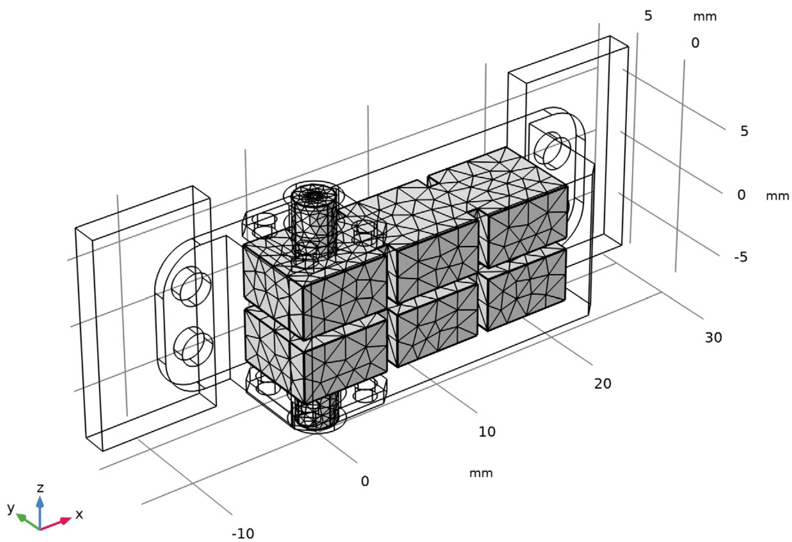
3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 37, 39, 43, 157, 159, 173, and 175 only.



### MESH 1

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




### STUDY 1 - US BAND

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

### RESULTS

*Electric Field (emw)*

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

2 From the **Parameter value (freq (GHz))** list, choose **28.1**.

*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 **X-planes** subsection. In the **Planes** text field, type `0`.

4 Find the **Y-planes** subsection. In the **Planes** text field, type `0`.

5 Find the **Z-planes** subsection. From the **Entry method** list, choose **Coordinates**.

6 In the **Coordinates** text field, type `-2.6 2.6`.

7 In the **Electric Field (emw)** toolbar, click **Plot**.

### *Surface 1*

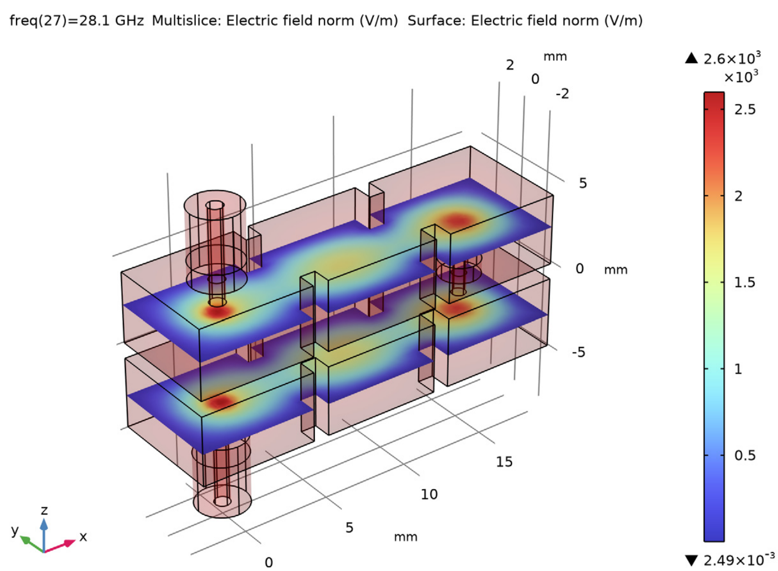
In the **Model Builder** window, right-click **Electric Field (emw)** and choose **Surface**.

### *Transparency 1*

1. In the **Model Builder** window, right-click **Surface 1** and choose **Transparency**.
2. In the **Settings** window for **Transparency**, locate the **Transparency** section.
3. Set the **Transparency** value to **0.85**.

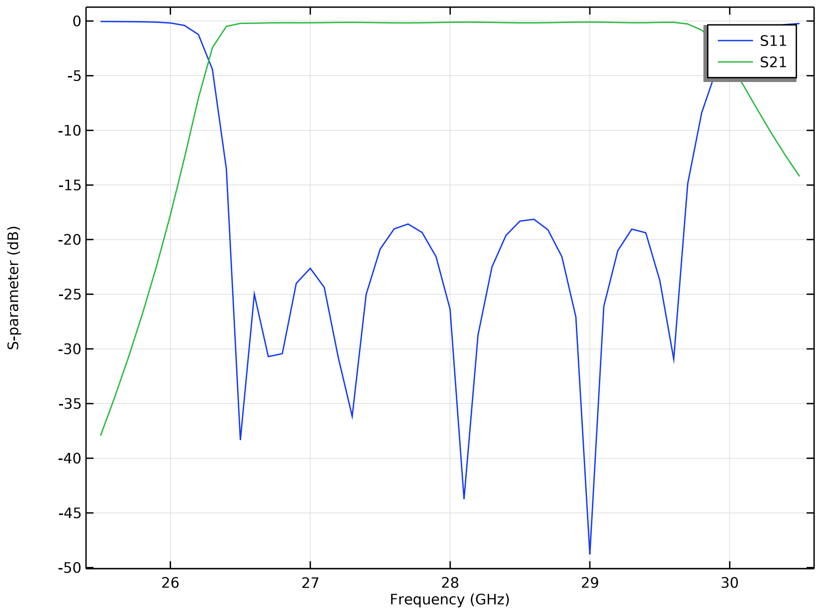
### *Surface 1*

1. In the **Model Builder** window, click **Surface 1**.
2. In the **Settings** window for **Surface**, locate the **Coloring and Style** section.
3. From the **Color table** list, choose **Thermal**.
4. Clear the **Color legend** checkbox.



### *S-parameter (emw)*

In the **Model Builder** window, under **Results** click **S-parameter (emw)**.




The following instruction shows how to use the **Graph Marker** subfeature to analyze 1D plots. When plotting transmittivity properties of a bandpass filter, the half-power bandwidth of the passband can be computed through a graph marker.

### *Passband with Graph Marker*

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

**2** In the **Settings** window for **1D Plot Group**, type `Passband with Graph Marker` in the **Label** text field.

### *Global 1*

**1** Right-click **Passband with Graph Marker** and choose **Global**.

**2** In the **Settings** window for **Global**, click **Add Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)** > **Electromagnetic Waves, Frequency Domain** > **Ports** > **S-parameter, dB** > **emw.S21dB - S21**.

### *Graph Marker 1*

**1** Right-click **Global 1** and choose **Graph Marker**.

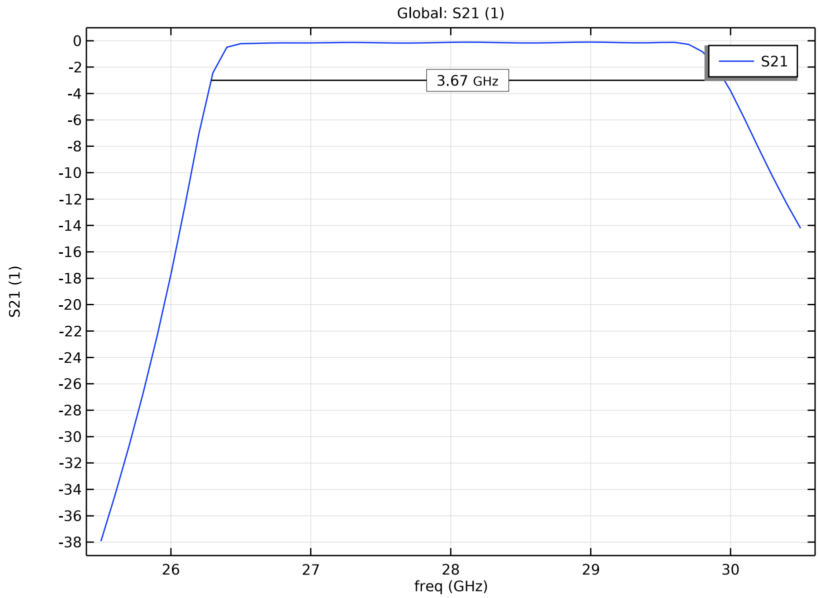
**2** In the **Settings** window for **Graph Marker**, locate the **Display** section.

**3** From the **Display mode** list, choose **Bandwidth**.

**4** Locate the **Text Format** section. In the **Precision** text field, type 3.

**5** Select the **Include unit** checkbox.

**6** Click to expand the **Coloring and Style** section. Select the **Show frame** checkbox.




## 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** Click **Clear Table**.

**4** Click **Load from File**.

**5** Browse to the model's Application Libraries folder and double-click the file `cavity_filter_5g_eu.txt`.

This parameters will update the geometry for the millimeter-wave 5G band in the European Union and China.

## 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 **General Studies** > **Frequency Domain**.

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 2 - EU BAND

In the **Settings** window for **Study**, type `Study 2 - EU Band` in the **Label** text field.

### *Step 1: Frequency Domain*

1 In the **Model Builder** window, under **Study 2 - EU Band** click **Step 1: Frequency Domain**.

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

3 In the **Frequencies** text field, type `range(23.25,0.1,28.5)`.

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

## RESULTS

### *Electric Field (emw) 1*

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

2 From the **Parameter value (freq (GHz))** list, choose **25.75**.

### *Multislice*

1 In the **Model Builder** window, expand the **Electric Field (emw) 1** node, then click **Multislice**.

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

3 Find the **X-planes** subsection. In the **Planes** text field, type `0`.

4 Find the **Y-planes** subsection. In the **Planes** text field, type `0`.

5 Find the **Z-planes** subsection. From the **Entry method** list, choose **Coordinates**.

6 In the **Coordinates** text field, type `-2.6 2.6`.

7 In the **Electric Field (emw) 1** toolbar, click **Plot**.

### *Surface 1*

In the **Model Builder** window, right-click **Electric Field (emw) 1** and choose **Surface**.

### *Transparency 1*

1 In the **Model Builder** window, right-click **Surface 1** and choose **Transparency**.

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

3 Set the **Transparency** value to **0.85**.

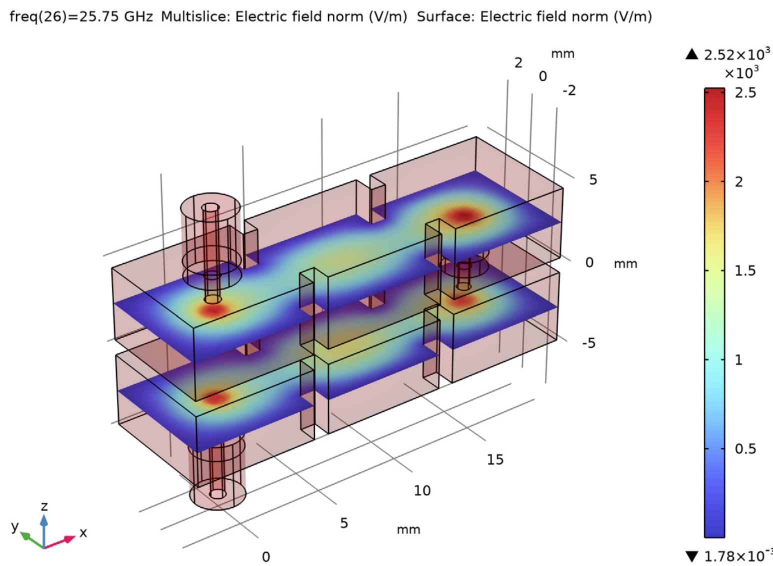
### *Surface 1*

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

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

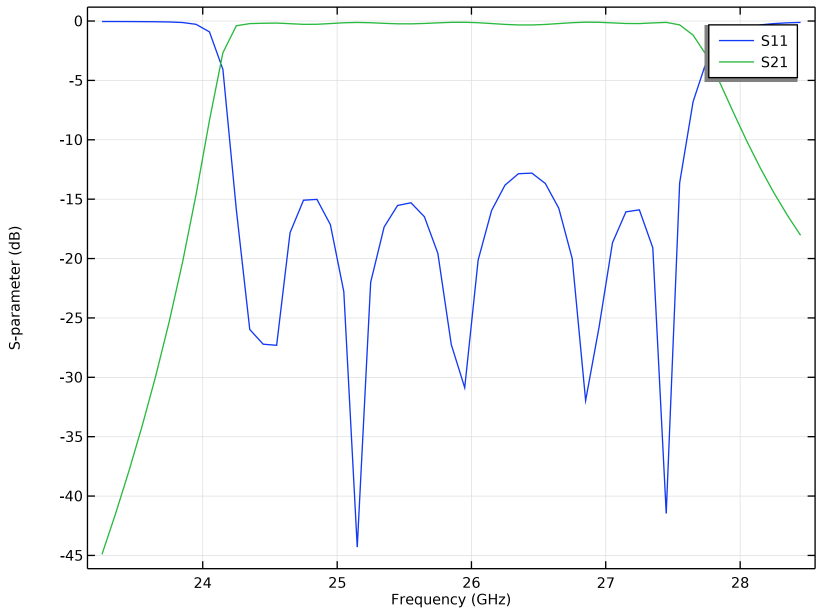
3 From the **Color table** list, choose **Thermal**.

4 Clear the **Color legend** checkbox.



### *S-parameter (emw) 1*

In the **Model Builder** window, under **Results** click **S-parameter (emw) 1**.




## *Thermal Deformation with the Prescribed Uniform Temperature Distribution*

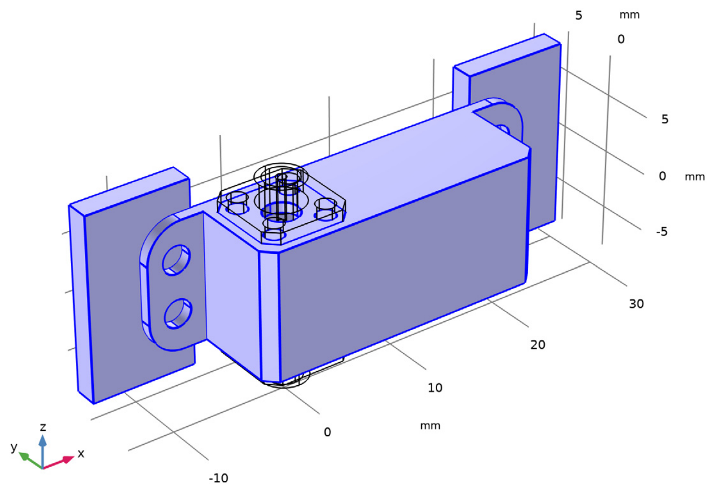
### DEFINITIONS

#### *Aluminum Domain*

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Aluminum Domain` in the **Label** text field.

**3** Select Domains 1, 2, and 15 only.

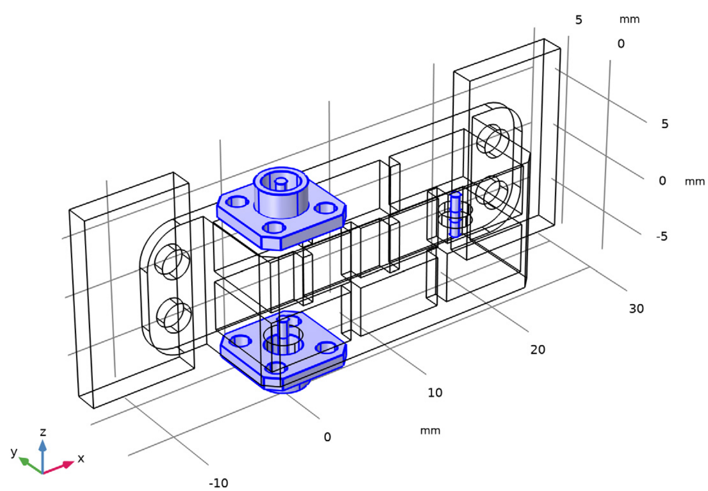



*Brass Domains*

**1** In the **Definitions** toolbar, click **Explicit**.

**2** In the **Settings** window for **Explicit**, type `Brass Domains` in the **Label** text field.

**3** Select Domains 3, 4, 9, 10, and 12–14 only.

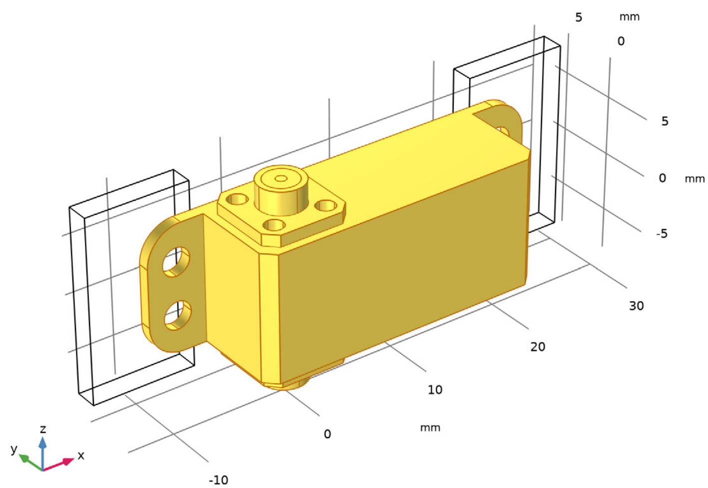


*Enclosure Domains*

1 In the **Definitions** toolbar, click **Explicit**.
2 In the **Settings** window for **Explicit**, type `Enclosure Domains` in the **Label** text field.
3 Select Domain 2 only.

*Solid Domains*

1 In the **Definitions** toolbar, click **Union**.
2 In the **Settings** window for **Union**, type `Solid Domains` in the **Label** text field.
3 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.
4 In the **Add** dialog, in the **Selections to add** list, choose **Dielectric Domains**, **Brass Domains**, and **Enclosure Domains**.
5 Click **OK**.




*Nonuniform Heat Domains*

1 In the **Definitions** toolbar, click **Union**.
2 In the **Settings** window for **Union**, type `Nonuniform Heat Domains` in the **Label** text field.
3 Locate the **Input Entities** section. Under **Selections to add**, click **Add**.
4 In the **Add** dialog, in the **Selections to add** list, choose **Dielectric Domains**, **Aluminum Domain**, **Brass Domains**, and **Enclosure Domains**.
5 Click **OK**.

*Surfaces of Air Domains*

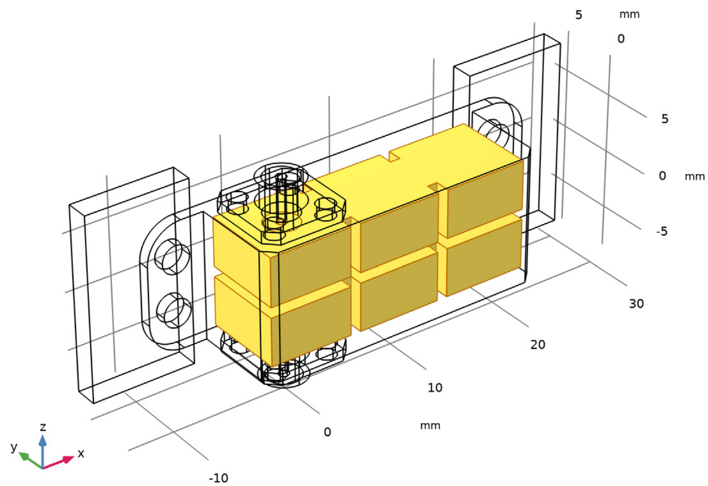
1 In the **Definitions** toolbar, click **Adjacent**.

2 In the **Settings** window for **Adjacent**, type `Surfaces of Air Domains` in the **Label** text field.

3 Locate the **Input Entities** section. Under **Input selections**, click **Add**.

4 In the **Add** dialog, select **Air Domains** in the **Input selections** list.

5 Click **OK**.



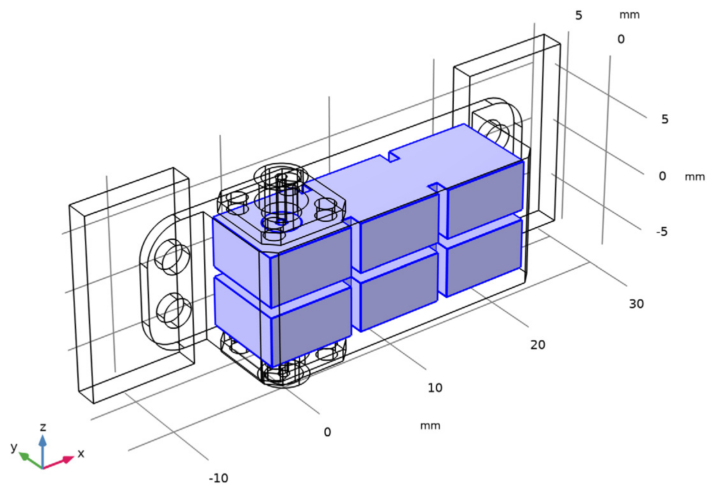

#### COMPONENT 1 (COMP1)

*Deforming Domain 1*

1 In the **Physics** toolbar, click **Moving Mesh** and choose **Free Deformation**.

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

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




The air domain is the deforming domain.

#### ADD PHYSICS

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

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

3 In the tree, select **Structural Mechanics** > **Solid Mechanics (solid)**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** checkboxes for **Study 1 - US Band** and **Study 2 - EU Band**.

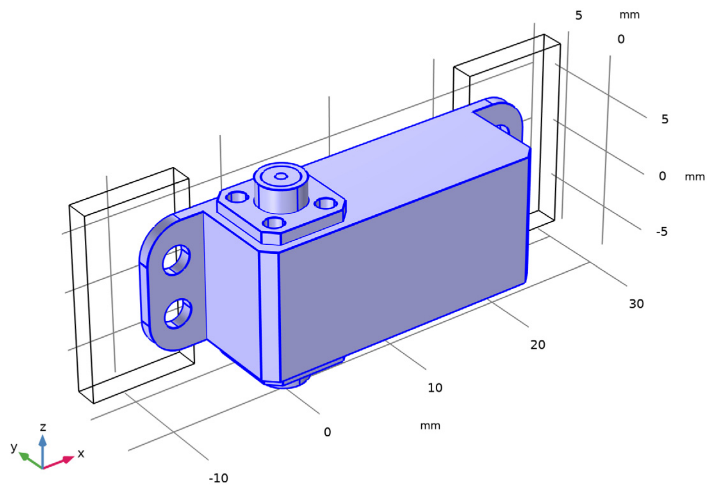
5 Click the **Add to Component 1** button in the window toolbar.

6 In the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

#### SOLID MECHANICS (SOLID)

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

**2** From the **Selection** list, choose **Solid Domains**.




*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.

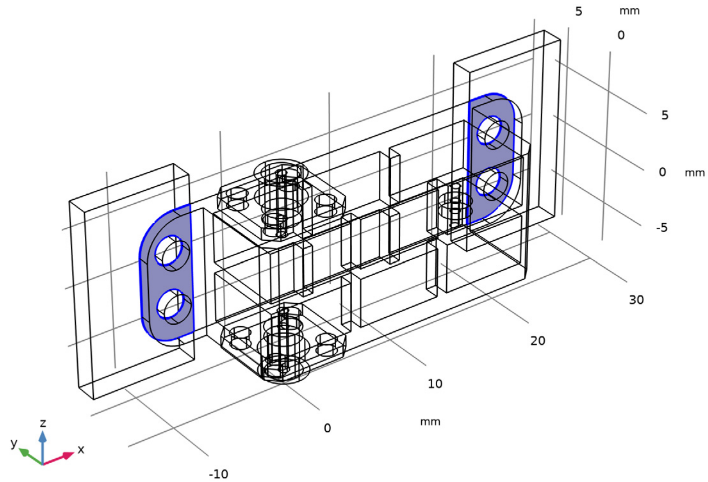
**3** From the $T_{\mathrm{ref}}$ list, choose **User defined**. In the associated text field, type `T0`.

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

*Spring Foundation 1*

**1** In the **Physics** toolbar, click **Boundaries** and choose **Spring Foundation**.

2 Select Boundaries 10 and 210 only.




3 In the **Settings** window for **Spring Foundation**, locate the **Spring** section.

4 From the **Spring type** list, choose **Use material data**.

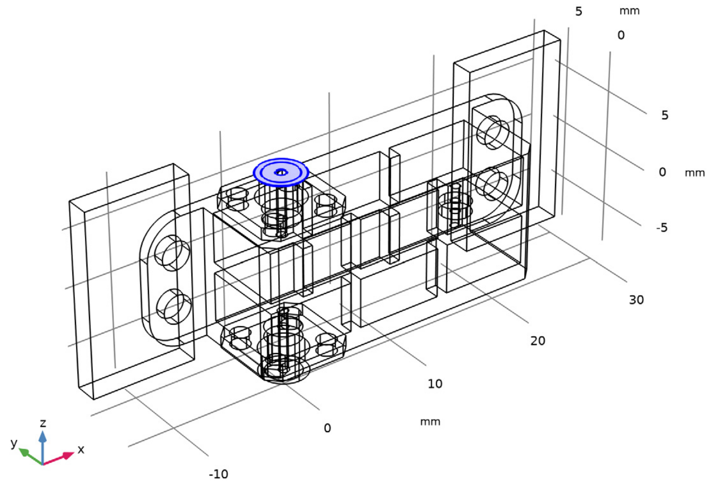
5 In the $d_s$ text field, type `0.1[mm]`.

   This represents a thin, `0.1[mm]`, adhesive layer connecting the filter body to a rigid substrate.

*Rigid Connector 1*

1 In the **Physics** toolbar, click **Boundaries** and choose **Rigid Connector**.

**2** Select Boundaries 77 and 89 only.

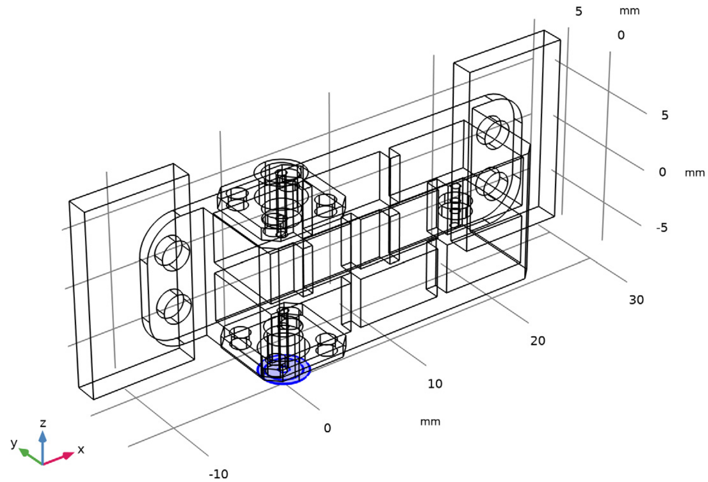



This feature enforces the deformation effects from the thermal variations only within the filter itself excluding the connector port boundaries. So, the lumped port boundaries remain planar during the simulation.

*Rigid Connector 2*

**1** In the **Physics** toolbar, click **Boundaries** and choose **Rigid Connector**.

2 Select Boundaries 74 and 80 only.




**MATERIALS**

*Coax Dielectric (mat2)*

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Materials** click **Coax Dielectric (mat2)**.

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 | `25[GPa]` | Pa | Young's modulus and Poisson's ratio |
| Poisson's ratio | nu | `0.2` | 1 | Young's modulus and Poisson's ratio |
| Density | rho | `2000[kg/m^3]` | $kg/m^3$ | Basic |
| Coefficient of thermal expansion | alpha_iso ; alphaii = alpha_iso, alphaij = 0 | `18e-6[1/K]` | 1/K | Basic |

The material properties are modified to include the thermal deformation analysis. These properties may not represent low-loss material for millimeter-wave applications, but they are used to show the basic modeling workflow.

### ADD MATERIAL

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

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

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

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

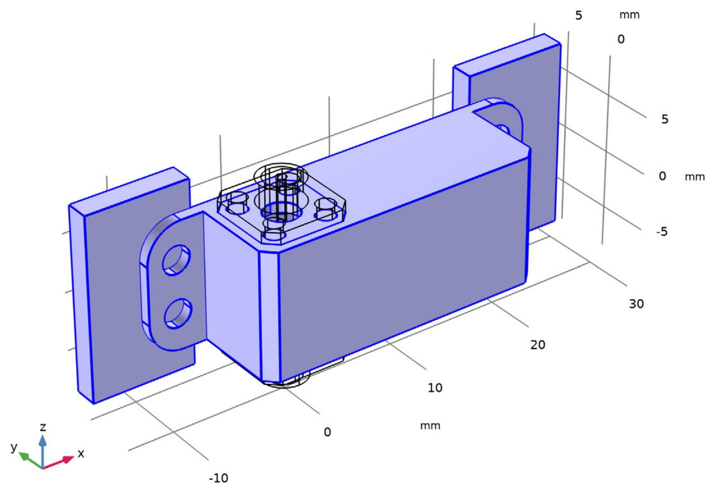
5 In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

### MATERIALS

*Aluminum (mat4)*

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

**2** From the **Selection** list, choose **Aluminum Domain**.

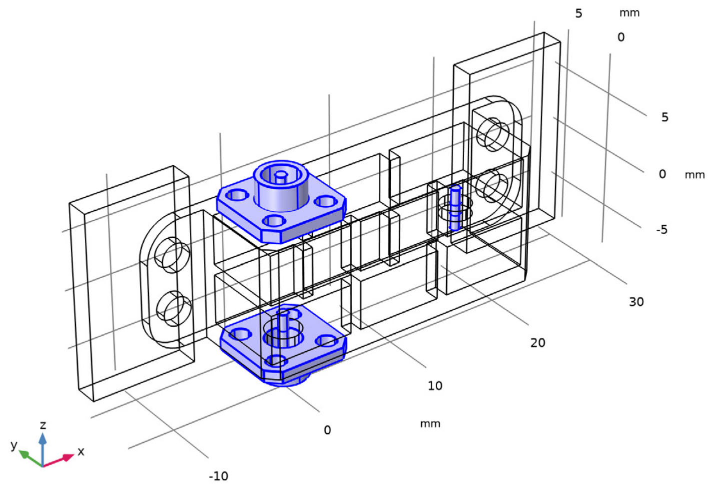



*Brass*

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

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

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Brass Domains**.




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

| **Property** | **Variable** | **Value** | **Unit** | **Property group** |
|---|---|---|---|---|
| Young's modulus | E | `110[GPa]` | Pa | Young's modulus and Poisson's ratio |
| Poisson's ratio | nu | `0.33` | 1 | Young's modulus and Poisson's ratio |
| Density | rho | `8700[kg/m^3]` | kg/m³ | Basic |
| Coefficient of thermal expansion | alpha_iso ; alphaii = alpha_iso, alphaij = 0 | `19e-6[1/K]` | 1/K | Basic |

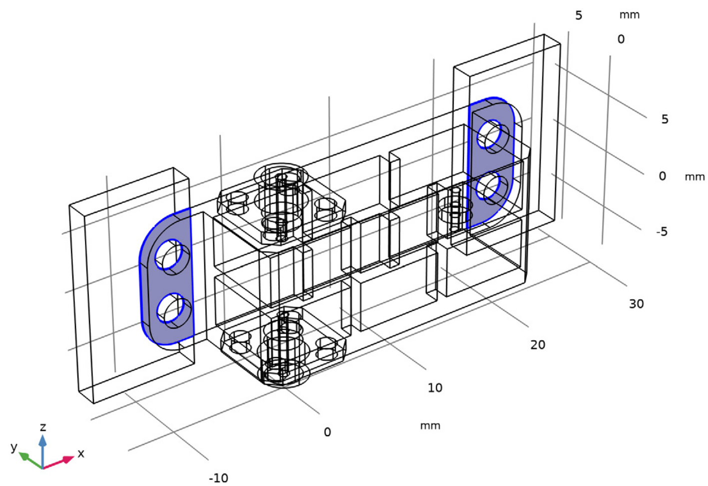
### *Adhesive Layer*

1 Right-click **Materials** and choose **Blank Material**.

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

**3** Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.

**4** Select Boundaries 10 and 210 only.




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

| Property | Variable | Value | Unit | Property group |
|---|---|---|---|---|
| Young's modulus | E | `70[GPa]` | Pa | Young's modulus and Poisson's ratio |
| Poisson's ratio | nu | `0.35` | 1 | Young's modulus and Poisson's ratio |

The following study uses a parametric sweep to wrap around the stationary study solving for the deformation of the structure with the three different temperatures.

### 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 **Physics interfaces in study** subsection. In the table, clear the **Solve** checkbox for **Electromagnetic Waves, Frequency Domain (emw)**.

**4** Find the **Studies** subsection. In the **Select Study** tree, select **General Studies** > **Stationary**.

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

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

## STUDY 3

### *Step 2: Frequency Domain*

**1** In the **Study** toolbar, click **Study Steps** and choose **Frequency Domain** > **Frequency Domain**.

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

**3** In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkboxes for **Solid Mechanics (solid)** and **Moving Mesh**.

**4** Locate the **Study Settings** section. From the **Frequency unit** list, choose **GHz**.

**5** In the **Frequencies** text field, type `range(23.25,0.1,28.5)`.

### *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 | -60 0 100 | |

Set the temperature deviation -60K, 0K, and 100K from the reference temperature 20°C.

**5** In the **Model Builder** window, click **Study 3**.

**6** In the **Settings** window for **Study**, type `Study 3 - Thermal Deformation, Uniform Temperature Distribution` in the **Label** text field.

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

## RESULTS

### *Multislice*

**1** In the **Model Builder** window, expand the **Results** > **Electric Field (emw) 2** node, then click **Multislice**.

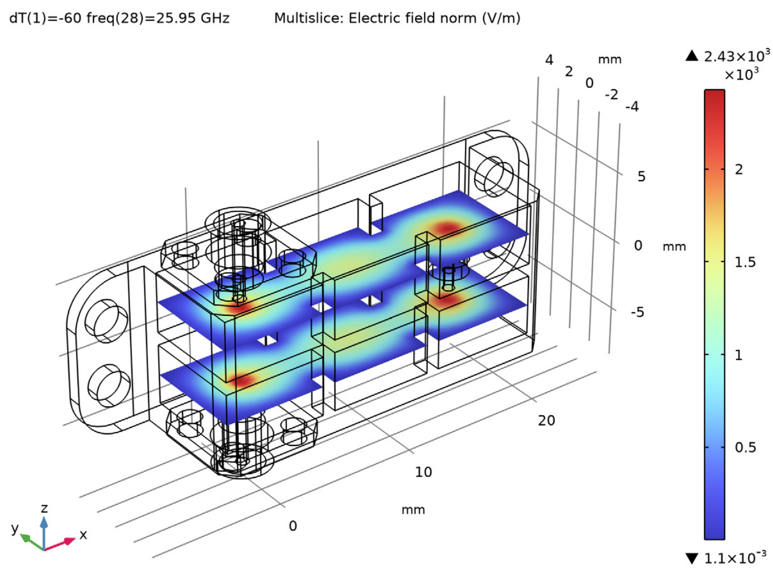
**2** In the **Settings** window for **Multislice**, locate the **Multiplane Data** section.

**3** Find the **x-planes** subsection. In the **Planes** text field, type `0`.

4 Find the **y-planes** subsection. In the **Planes** text field, type `0`.
5 Find the **z-planes** subsection. From the **Entry method** list, choose **Coordinates**.
6 In the **Coordinates** text field, type `-2.6 2.6`.

*Electric Field (emw) 2*

1 In the **Model Builder** window, click **Electric Field (emw) 2**.
2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.
3 From the **Parameter value (freq (GHz))** list, choose **25.95**.
4 From the **Parameter value (dT)** list, choose **-60**.
5 In the **Electric Field (emw) 2** toolbar, click **Plot**.




*S-parameter (emw) 2*

1 In the **Model Builder** window, click **S-parameter (emw) 2**.
2 In the **Settings** window for **1D Plot Group**, locate the **Legend** section.
3 From the **Position** list, choose **Lower right**.

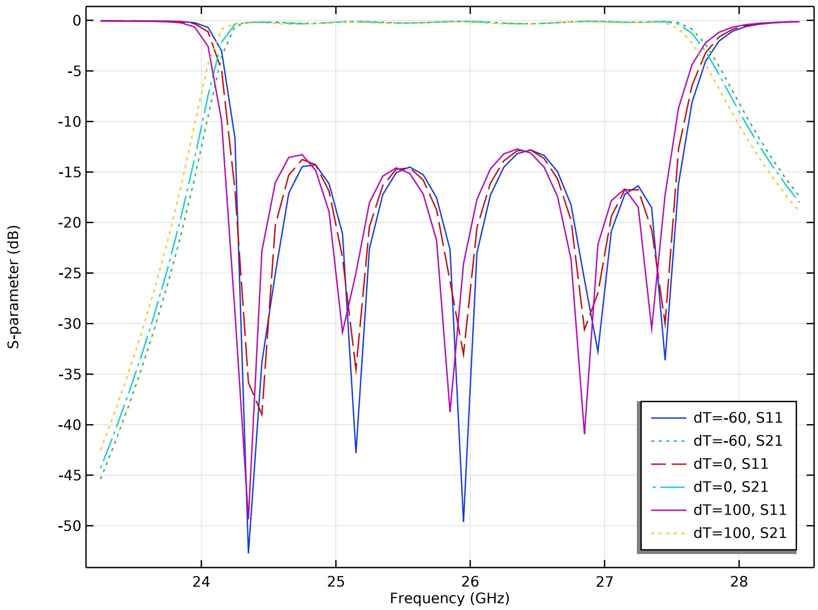
*Global 1*

1 In the **Model Builder** window, expand the **S-parameter (emw) 2** node, then click **Global 1**.
2 In the **Settings** window for **Global**, locate the **x-Axis Data** section.

**3** From the **Axis source data** list, choose **Inner solutions**.

**4** Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Cycle**.

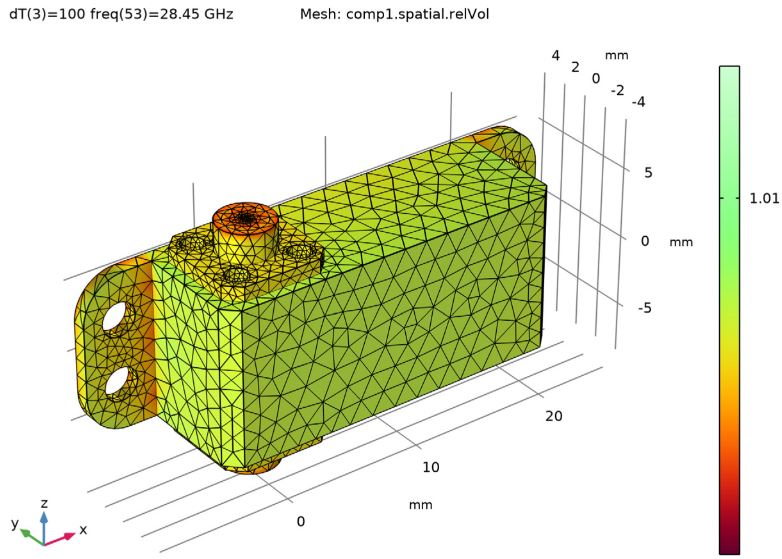
**5** In the **S-parameter (emw) 2** toolbar, click **Plot**.




*Moving Mesh*

**1** In the **Model Builder** window, under **Results** click **Moving Mesh**.

2 In the **Moving Mesh** toolbar, click **Plot**.




## *Thermal Deformation with the Computed Nonuniform Temperature Distribution*

### ADD PHYSICS

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

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

3 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** checkboxes for **Study 1 - US Band**, **Study 2 - EU Band**, and **Study 3 - Thermal Deformation, Uniform Temperature Distribution**.

4 In the tree, select **Heat Transfer** > **Heat Transfer in Solids (ht)**.

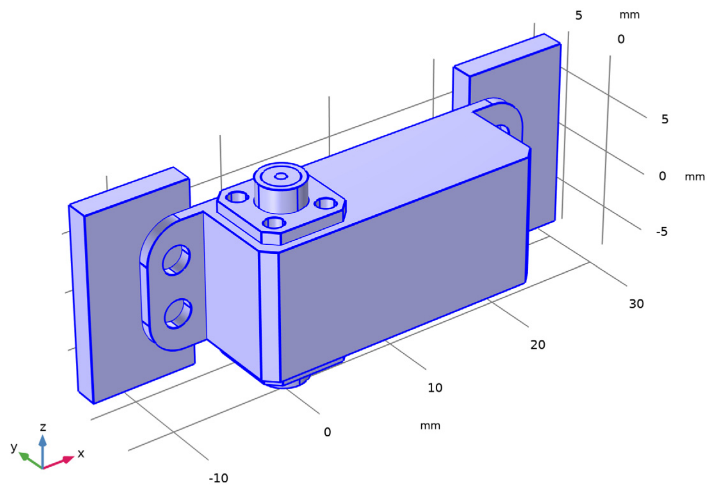
5 Click the **Add to Component 1** button in the window toolbar.

6 In the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

### HEAT TRANSFER IN SOLIDS (HT)

1 In the **Settings** window for **Heat Transfer in Solids**, locate the **Domain Selection** section.

2 From the **Selection** list, choose **Nonuniform Heat Domains**.




3 Click to expand the **Discretization** section. From the **Temperature** list, choose **Linear**.

For the Heat Transfer in Solids, the **Discretization** is set to **Linear**, one order lower than that in the Solid Mechanics problem.

*Heat Flux 1*

1 In the **Physics** toolbar, click **Boundaries** and choose **Heat Flux**.

2 In the **Settings** window for **Heat Flux**, locate the **Boundary Selection** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type `2, 6-9, 21-29, 31, 32, 34-36, 47, 60-63, 72, 73, 75, 76, 102, 103, 120, 121, 142-145, 154, 155, 202, 205-208, 219, 220, 225` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.

7 From the **Flux type** list, choose **Convective heat flux**.

8 In the $h$ text field, type `5[W/m/m/K]`.

9 In the $T_{ext}$ text field, type `25[degC]`.

*Temperature 1*

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

2 Select Boundary 226 only.

3 In the **Settings** window for **Temperature**, locate the **Temperature** section.

4 In the $T_0$ text field, type `225[degC]`.

### *Thin Layer 1*

1 In the **Physics** toolbar, click **Boundaries** and choose **Thin Layer**.

2 Select Boundaries 10 and 210 only.

3 In the **Settings** window for **Thin Layer**, locate the **Heat Conduction** section.

4 From the $k$ list, choose **User defined**. In the associated text field, type `0.42[W/m/K]`.

5 Locate the **Thermodynamics** section. From the ρ list, choose **User defined**. In the associated text field, type `1000[kg/m^3]`.

6 From the $C_p$ list, choose **User defined**. In the associated text field, type `1500[J/(kg*K)]`.

## MATERIALS

### *Aluminum (mat4)*

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Materials** click **Aluminum (mat4)**.

2 Select Domains 1, 2, and 15 only.

### *Adhesive Layer (mat6)*

1 In the **Model Builder** window, click **Adhesive Layer (mat6)**.

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 |
|---|---|---|---|---|
| Thickness | lth | `0.1[mm]` | m | Shell |

## ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

### *Impedance Boundary Condition 1*

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

2 In the **Settings** window for **Impedance Boundary Condition**, locate the **Model Input** section.

**3** From the $T$ list, choose **Temperature (ht)**.

Except for the above update in the **Impedance Boundary Condition**, the settings in this physics interface for simulating electromagnetics are identical to the previous model configurations.

### MULTIPHYSICS

*Thermal Expansion 1 (te1)*

**1** In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Domain** > **Thermal Expansion**.

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

**3** From the **Selection** list, choose **Solid Domains**.

### MATERIALS

*Coax Dielectric (mat2)*

**1** In the **Model Builder** window, under **Component 1 (comp1)** > **Materials** click **Coax Dielectric (mat2)**.

**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** |
|---|---|---|---|---|
| Thermal conductivity | k_iso ; kii = k_iso, kij = 0 | 0.25[W/(m*K)] | W/(m·K) | Basic |
| Heat capacity at constant pressure | Cp | 1350[J/kg/K] | J/(kg·K) | Basic |

*Brass (mat5)*

**1** In the **Model Builder** window, click **Brass (mat5)**.

**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** |
|---|---|---|---|---|
| Thermal conductivity | k_iso ; kii = k_iso, kij = 0 | 105[W/(m*K)] | W/(m·K) | Basic |
| Heat capacity at constant pressure | Cp | 380[J/(kg*K)] | J/(kg·K) | Basic |

Next, add a stationary study step for the thermo-structural analysis. A frequency domain study step is also required to solve the electromagnetics problem on the deformed state.

### 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 **Physics interfaces in study** subsection. In the table, clear the **Solve** checkbox for **Electromagnetic Waves, Frequency Domain (emw)**.

4 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies** > **Stationary**.

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

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

### STUDY 4

*Step 1: Stationary*

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

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

3 In the tree, select **Component 1 (comp1)** > **Electromagnetic Waves, Frequency Domain (emw)**.

4 Click **Disable in Model**.

5 In the tree, select **Component 1 (comp1)** > **Solid Mechanics (solid), Controls spatial frame** > **Linear Elastic Material 1** > **Thermal Expansion 1**.

6 Click **Disable**.

*Step 2: Frequency Domain*

1 In the **Study** toolbar, click **Study Steps** and choose **Frequency Domain** > **Frequency Domain**.

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

3 From the **Frequency unit** list, choose **GHz**.

4 In the **Frequencies** text field, type `range(23.25,0.1,28.5)`.

5 Locate the **Physics and Variables Selection** section. In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkboxes for **Solid Mechanics (solid)** and **Moving Mesh**.

6 In the **Solve for** column of the table, under **Component 1 (comp1)** > **Multiphysics**, clear the checkbox for **Thermal Expansion 1 (te1)**.

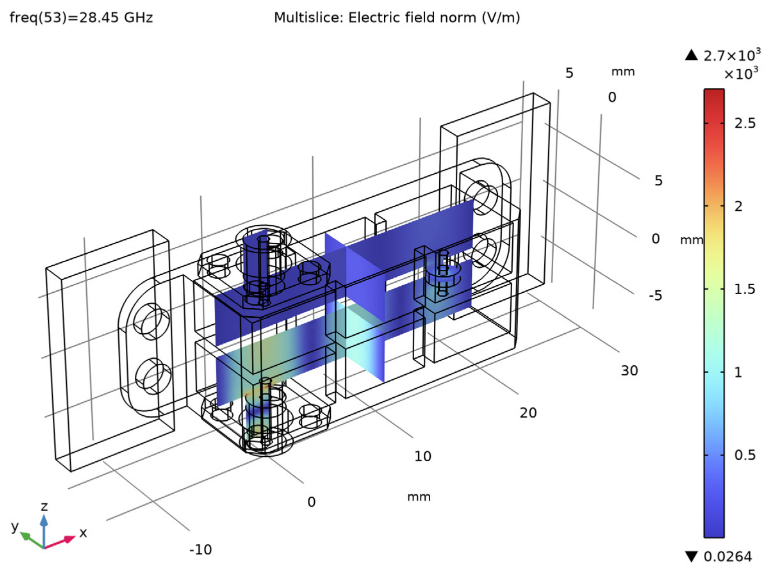
7 In the **Model Builder** window, click **Study 4**.

8 In the **Settings** window for **Study**, type `Study 4 - Thermal Deformation, Nonuniform Computed Temperature` in the **Label** text field.

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

## RESULTS

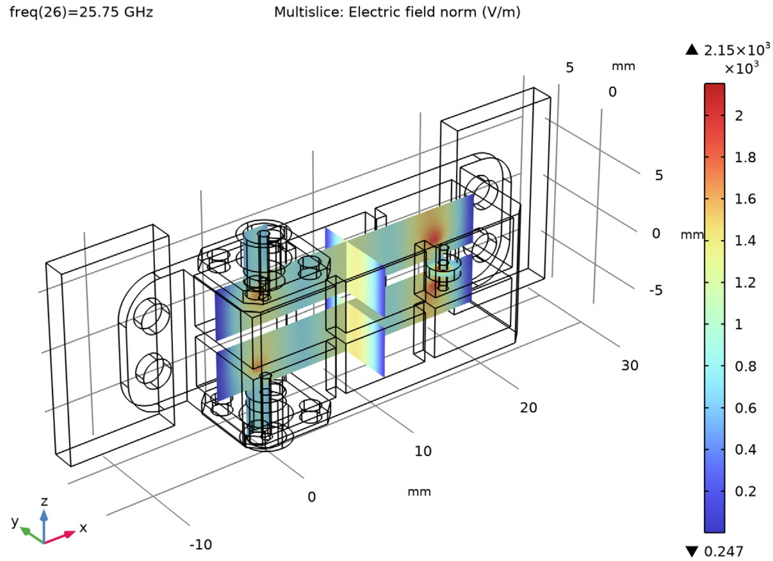
### *Electric Field (emw) 3*




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

2 From the **Parameter value (freq (GHz))** list, choose **25.75**.

3 In the **Electric Field (emw) 3** toolbar, click **Plot**.




*S-parameter (emw) 3*

1 In the **Model Builder** window, click **S-parameter (emw) 3**.

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

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

*Global 2*

1 In the **Model Builder** window, expand the **S-parameter (emw) 3** node.

2 Right-click **Results > S-parameter (emw) 3 > Global 1** and choose **Duplicate**.

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

4 From the **Dataset** list, choose **Study 2 - EU Band/Solution 2 (sol2)**.

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

*Global 1*

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

2 In the **Settings** window for **Global**, click to expand the **Legends** section.

3 From the **Legends** list, choose **Manual**.

**4** In the table, enter the following settings:

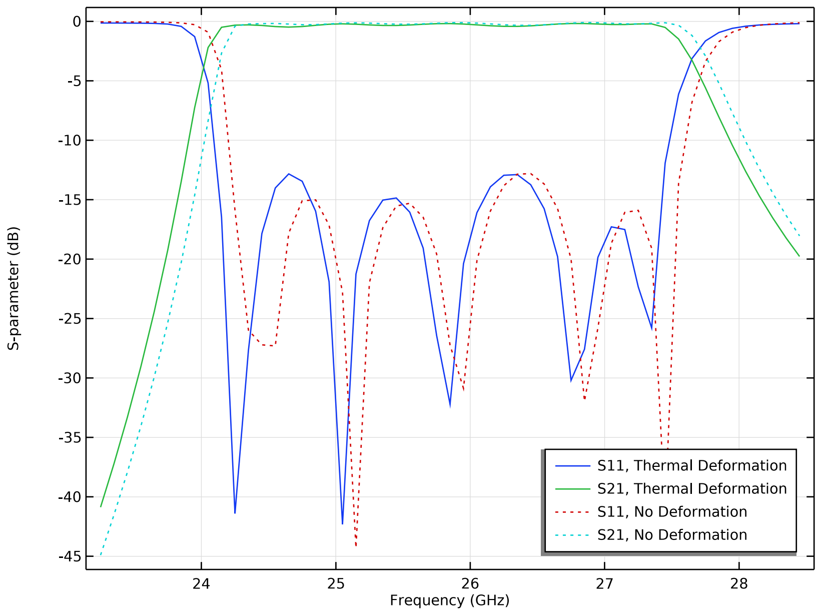
| Legends |
| --- |
| S11, Thermal Deformation |
| S21, Thermal Deformation |

*Global 2*

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

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

**3** From the **Legends** list, choose **Manual**.



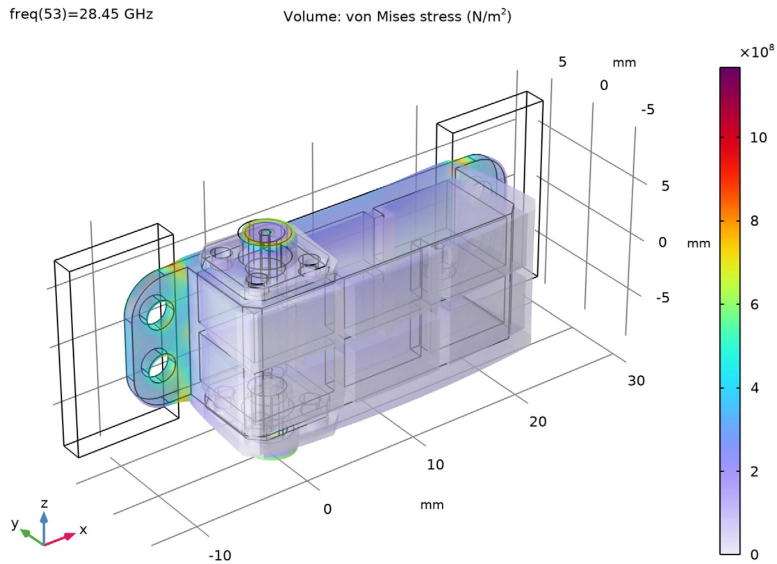

**4** In the table, enter the following settings:

| Legends |
| --- |
| S11, No Deformation |
| S21, No Deformation |

*Transparency 1*

**1** In the **Model Builder** window, expand the **Results** > **Stress (solid) 1** node.

2 Right-click **Volume 1** and choose **Transparency**.




### *Domain*

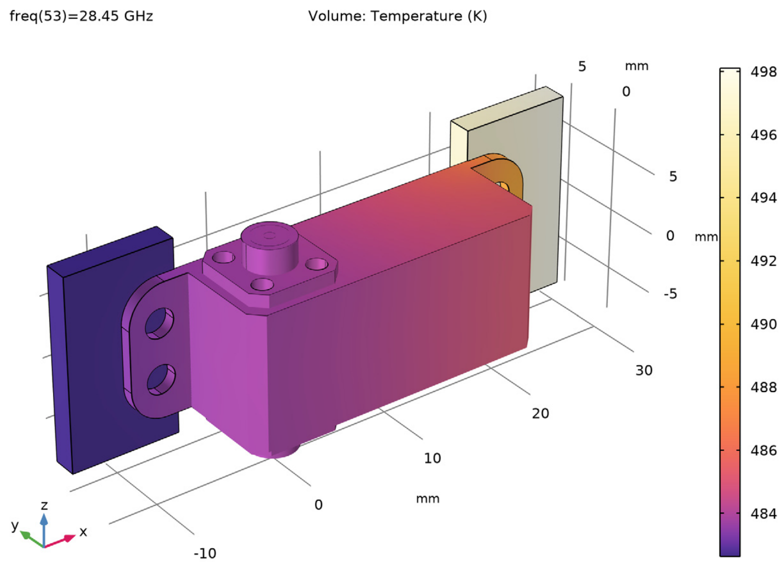
1 In the **Model Builder** window, expand the **Results** > **Temperature (ht)** node, then click **Domain**.

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

3 From the **Color table transformation** list, choose **Nonlinear**.

4 Set the **Color calibration parameter** value to **-1.4**.

**5** In the **Color calibration parameter** text field, type `-1.5`.




#### RESULT TEMPLATES

**1** In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.

**2** Go to the **Result Templates** window.

**3** In the tree, select **Study 4 - Thermal Deformation, Nonuniform Computed Temperature/ Solution Store 2 (sol10)** > **Heat Transfer in Solids** > **Isothermal Contours (ht)**.

**4** Click the **Add Result Template** button in the window toolbar.

**5** In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

#### RESULTS

*Domain*

**1** In the **Model Builder** window, expand the **Isothermal Contours (ht)** node, then click **Domain**.

**2** In the **Settings** window for **Isosurface**, locate the **Levels** section.

**3** In the **Total levels** text field, type 30.

**4** In the **Isothermal Contours (ht)** toolbar, click **Plot**.