Model created in COMSOL Multiphysics 6.4

RF Heating

Introduction

This is a model of an RF waveguide bend with a dielectric block inside. There are electromagnetic losses in the block as well as on the waveguide walls which cause the assembly to heat up over time. The material properties of the block are functions of temperature. The transient thermal behavior, as well as the steady-state solution, are computed.

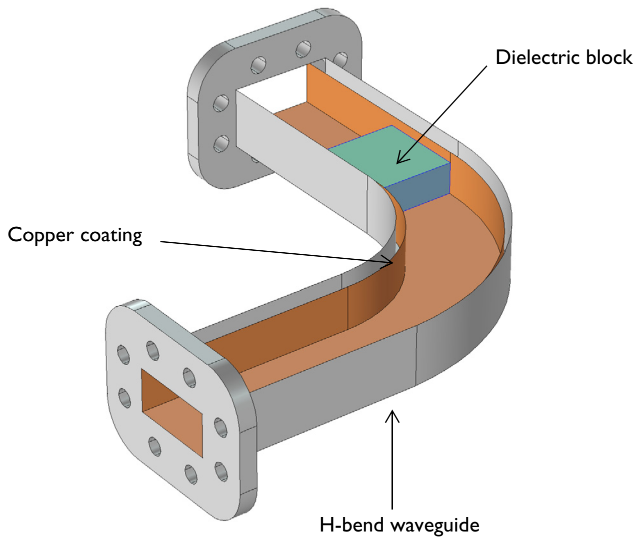


Figure 1: A waveguide bend with a dielectric block inside. Top boundaries of the waveguide are removed only for visualization.

Model Definition

The waveguide bend shown in Figure 1 is connected to a 100 W power source, operating at 10 GHz, via a rectangular waveguide operating in the TE_{10} mode. The other end of the bend is also connected to a rectangular waveguide operating in the TE_{10} mode. The objective of such a bend is primarily to change the direction of propagation of the energy. Here, however, a block of dielectric is introduced as an example of a lossy material interacting with an electromagnetic field.

The waveguide is made of aluminum. To reduce surface losses, the inside walls are coated with copper, a high-conductivity metal. The dielectric block is modeled as having electric conductivity of $\sigma = 0$, relative permeability of $\mu_r = 1$, and a relative permittivity of $\varepsilon_r = 2.1$, with a loss tangent that is a function of temperature, $\delta = 0.001\times(T/300\ \mathrm{K})$. The thermal conductivity of this block is also a function of temperature, $k = 0.3\times(\mathrm{T}/300\ \mathrm{K})$ W/m/K. Furthermore, the density is 2200 kg/m^3 and the specific heat is 1050 J/kg/K. These are generic properties representative of a dielectric material.

At the operating frequency, the skin depth of the copper coating is much smaller than the dimensions of the waveguide, that is, the electromagnetic fields penetrate a negligible distance into the walls. This means that the electromagnetic losses can be localized entirely on the surface, and that there is no need to solve Maxwell's equations inside of the walls themselves. Thus, Maxwell's equations only need to be solved in the air domain inside of the waveguide, as well as inside of the block. The heat transfer equation is solved in the block as well as the waveguide walls.

The objective of the analysis is to observe how the assembly of the dielectric block and waveguide heat up over time, as well as to find the steady-state temperature. The waveguide is initially assumed to be at a constant temperature throughout. After the power source is turned on, the electromagnetic fields interact with the highly conductive interior boundaries of the waveguide, as well as the lossy dielectric block. The losses in the block and on the walls are sources of heat that raises the temperature. The block is assumed to be in perfect thermal contact with the walls of the waveguide, that is, any heat generated in the block is conducted away into the walls. The outside boundaries of the walls are assumed to be facing ambient air, which leads to free convective cooling off of these faces. This example uses an averaged heat transfer coefficient to represent this free convection to ambient air.

The model solves two governing equations: Maxwell's equations, which describe the electromagnetic fields, and the heat transfer equation, which describes the temperature. It is assumed that the operating frequency is much higher than any thermal transients, and thus it is possible to solve the problem either in a frequency-transient or a frequency-stationary sense.

A *frequency-transient* simulation solves Maxwell's equations in the frequency domain. This implicitly assumes that all material properties used to solve Maxwell's equations are constant over a single period of oscillation of the electromagnetic wave. The heat transfer equation is, on the other hand, solved transiently. The electromagnetic fields are only recomputed when the material properties have changed significantly, as determined by a criterion involving the relative tolerance of the time-dependent solver. The objective of the analysis is to determine the change in temperature from given initial conditions and how long these changes take.

A *frequency-stationary* simulation solves Maxwell's equations in the frequency domain, but it solves the stationary heat transfer equation under the assumption that all initial transient variations have died out. Although no transient information is obtainable, this computation is significantly faster than a frequency-transient analysis and gives the steady-state temperature distribution.

Results and Discussion

Figure 2 plots the peak temperature within the dielectric block over time, showing that it takes several minutes for the block to reach thermal equilibrium.

Figure 3 plots the temperature of the assembly, for the steady-state temperature solution after all thermal transients have died out. The dielectric block shows a significant temperature variation, which affects the thermal conductivity and loss tangent, plotted in Figure 4, that also includes the electromagnetic fields inside of the waveguide.

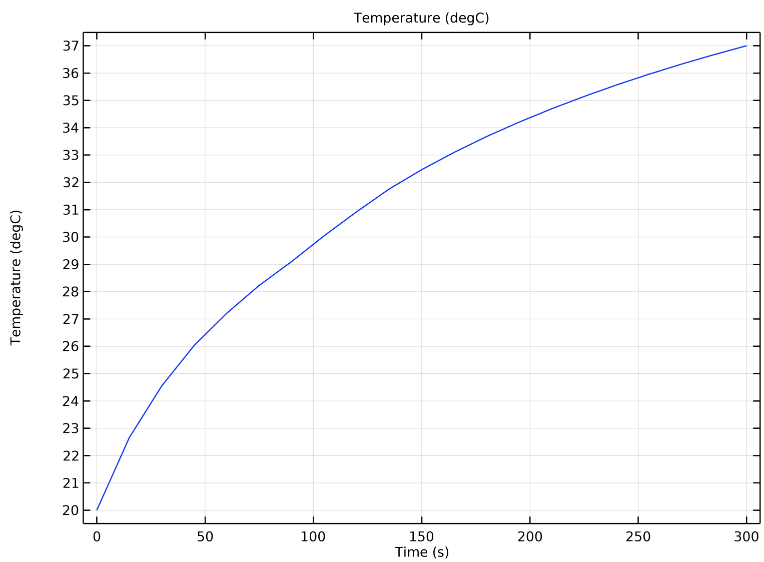


Figure 2: The maximum temperature, evaluated over the volume of the block, is plotted as a function of temperature.

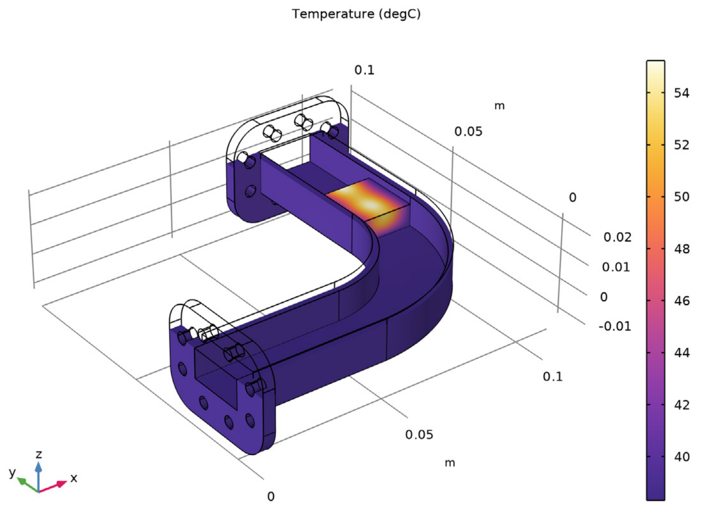


Figure 3: The steady-state temperature is plotted on the block and waveguide walls.

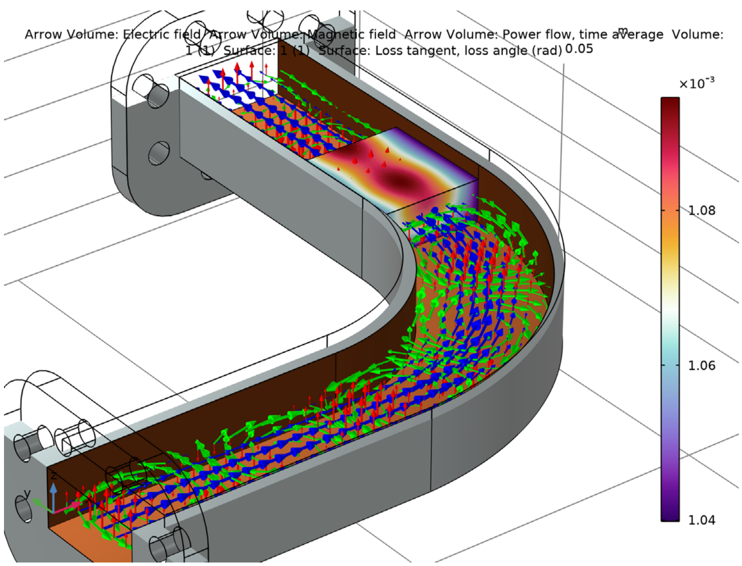


Figure 4: The loss tangent within the dielectric block for the steady-state solution shows that the variation in temperature affects the material properties. The electric fields (red arrows) magnetic fields (green arrows), and power flow (blue arrows) are also shown inside of the waveguide.

Application Library path: `RF_Module/Microwave_Heating/rf_heating`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

2 In the **Select Physics** tree, select **Heat Transfer** > **Electromagnetic Heating** > **Microwave Heating**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **Preset Studies for Selected Multiphysics** > **Frequency–Transient**.

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 In the table, enter the following settings:

Name	Expression	Value	Description
f0	10[GHz]	1E10 Hz	Current frequency
lda0	c_const/f0	0.029979 m	Wavelength, air
h_max	0.2*lda0	0.0059958 m	Maximum mesh element size, air

Here, `c_const` is a predefined COMSOL constant for the speed of light in vacuum.

GEOMETRY 1

First, import the geometry of the waveguide including a dielectric block inside the waveguide.

Import 1 (imp1)

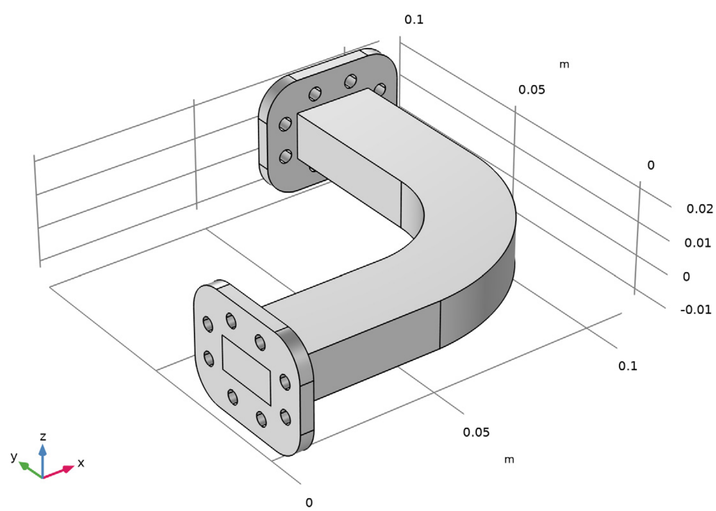
1 In the **Geometry** toolbar, click **Import**.

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 `rf_heating.mphbin`.

5 Click **Import**.



Use the wireframe rendering to see the inner parts of the waveguide.

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

DEFINITIONS

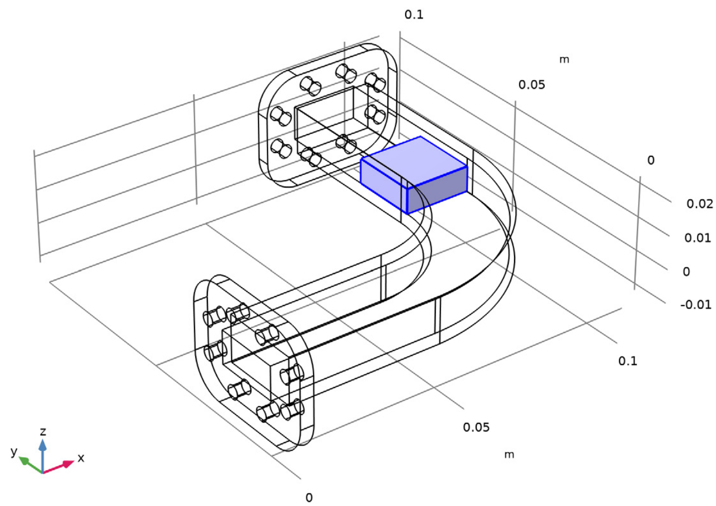
Create a set of selections for use before setting up the physics. First, create a selection for the dielectric block.

Dielectric

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

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

3 Select Domain 3 only.



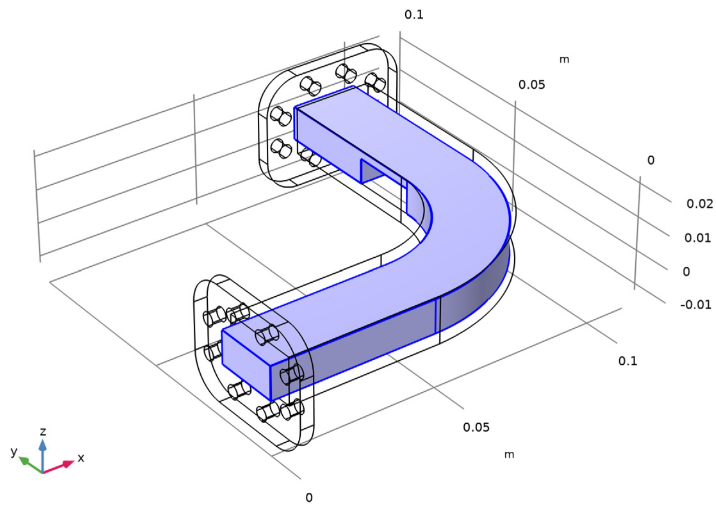
Add a selection for the air-filled region inside the waveguide.

Air

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

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

3 Select Domain 2 only.



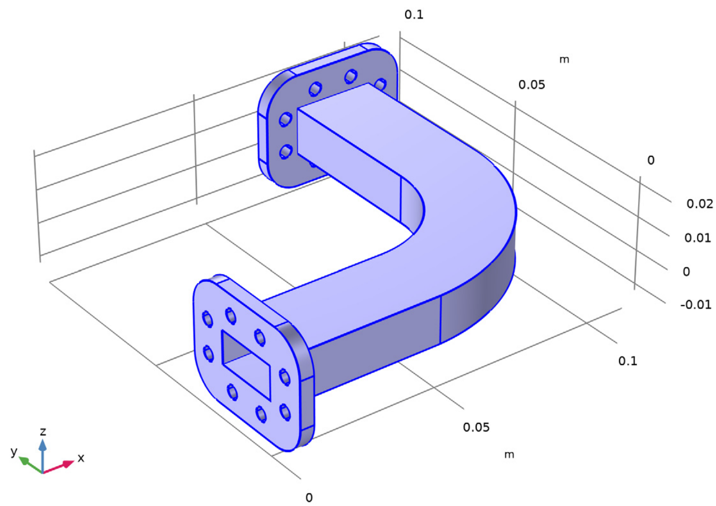
Add a selection for the waveguide structure.

Waveguide

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

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

3 Select Domain 1 only.



Add a selection for the inner surface of the waveguide.

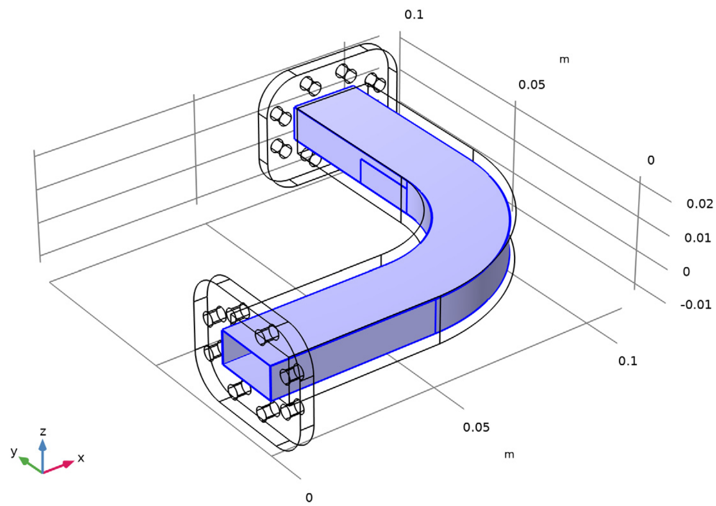
Waveguide inside surfaces

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

2 In the **Settings** window for **Explicit**, type `Waveguide inside surfaces` in the **Label** text field.

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

4 Select Boundaries 16–18, 35, 53, 54, 72, 74, 75, 78, 96, and 97 only.



Add a selection for the outer surface of the waveguide.

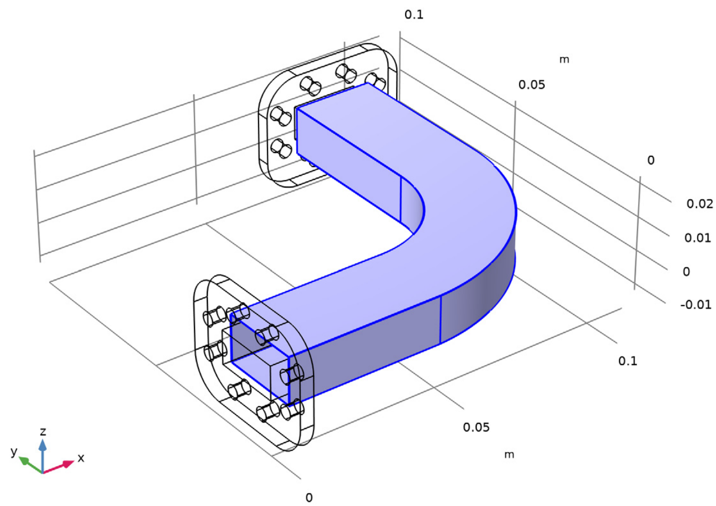
Waveguide outside surfaces

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

2 In the **Settings** window for **Explicit**, type `Waveguide outside surfaces` in the **Label** text field.

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

4 Select Boundaries 48–52, 55, 69, and 98 only.



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 In the list box, select **1**.

4 Click **Remove from Selection**.

5 Select Domains 2 and 3 only.

Wave Equation, Electric 1

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

2 In the **Settings** window for **Wave Equation, Electric**, locate the **Electric Displacement Field** section.

3 From the **Electric displacement field model** list, choose **Loss tangent, loss angle**.

HEAT TRANSFER IN SOLIDS (HT)

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

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

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

4 Click **Remove from Selection**.

5 Select Domains 1 and 3 only.

Heat Flux 1

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

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

3 From the **Selection** list, choose **Waveguide outside surfaces**.

4 Locate the **Heat Flux** section. From the **Flux type** list, choose **Convective heat flux**.

5 In the h text field, type 5.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

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

Wave Equation, Electric 2

1 In the **Physics** toolbar, click **Domains** and choose **Wave Equation, Electric**.

2 In the **Settings** window for **Wave Equation, Electric**, locate the **Domain Selection** section.

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

Impedance Boundary Condition 1

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

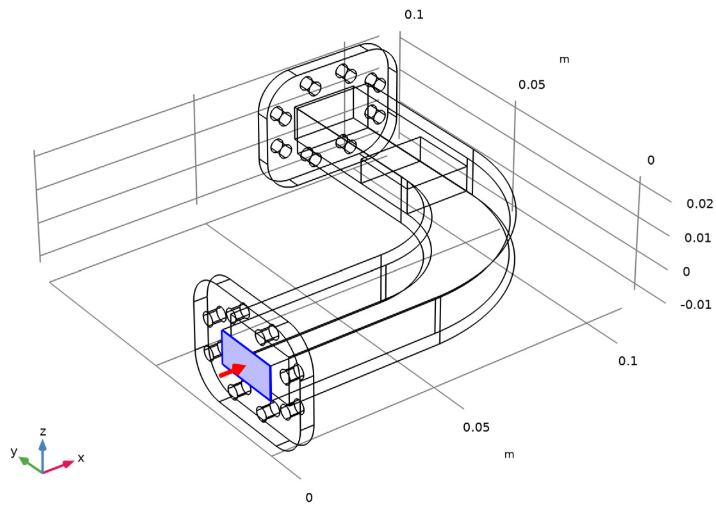
2 In the **Settings** window for **Impedance Boundary Condition**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Waveguide inside surfaces**.

Port 1

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

2 Select Boundary 15 only.



3 In the **Settings** window for **Port**, locate the **Port Properties** section.

4 From the **Type of port** list, choose **Rectangular**.

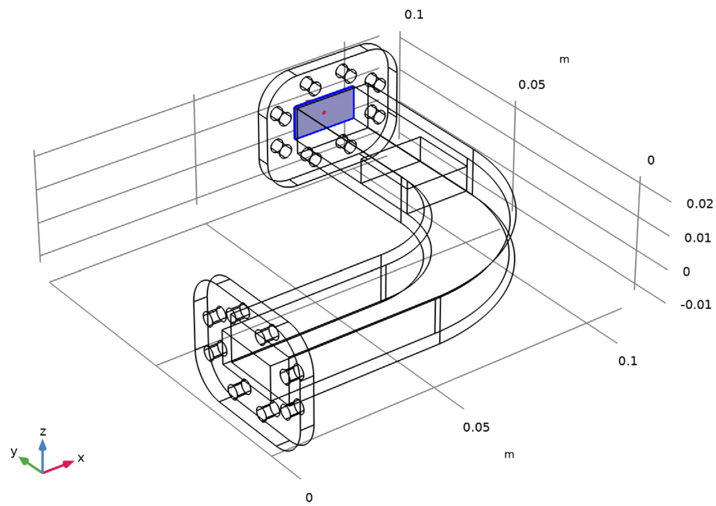
For the first port, wave excitation is **on** by default.

5 In the P_{in} text field, type `100`.

Port 2

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

2 Select Boundary 79 only.



3 In the **Settings** window for **Port**, locate the **Port Properties** section.

4 From the **Type of port** list, choose **Rectangular**.

MATERIALS

Next, assign material properties on the model. Begin by specifying Aluminum for the waveguide structure.

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 tree, select **Built-in > Air**.

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

MATERIALS

Aluminum (mat1)

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

2 From the **Selection** list, choose **Waveguide**.

Air (mat2)

1. In the **Model Builder** window, click **Air (mat2)**.
2. In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
3. From the **Selection** list, choose **Air**.

Dielectric

1. In the **Model Builder** window, right-click **Materials** and choose **Blank Material**.
2. In the **Settings** window for **Material**, type `Dielectric` in the **Label** text field.
3. Select Domain 3 only.
4. Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Relative permittivity (real part)	epsilonPrim_iso ; epsilonPrimii = epsilonPrim_iso, epsilonPrimij = 0	`2.1`	1	Loss tangent, loss angle
Loss tangent, loss angle	delta	`0.001*(T/300[K])`	rad	Loss tangent, loss angle
Relative permeability	mur_iso ; murii = mur_iso, murij = 0	1	1	Basic
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	`0.3[W/m/K]*(T/300[K])`	W/(m·K)	Basic
Density	rho	2200	kg/m³	Basic
Heat capacity at constant pressure	Cp	1050	J/(kg·K)	Basic

ADD MATERIAL

1. Go to the **Add Material** window.
2. In the tree, select **Built-in > Copper**.
3. Click the **Add to Component** button in the window toolbar.
4. In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Copper (mat4)

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

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

3 From the **Selection** list, choose **Waveguide inside surfaces**.

MESH 1

Choose the maximum mesh size in the air domain smaller than 0.2 wavelengths using the parameter h_max that you defined earlier. For the dielectric materials, scale the mesh size by the inverse of the square root of the relative dielectric constant.

Size 1

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

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

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

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

5 Locate the **Element Size** section. Click the **Custom** button.

6 Locate the **Element Size Parameters** section.

7 Select the **Maximum element size** checkbox. In the associated text field, type `h_max`.

Size 2

1 In the **Model Builder** window, right-click **Mesh 1** and choose **Size**.

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

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

4 From the **Selection** list, choose **Dielectric**.

5 Locate the **Element Size** section. Click the **Custom** button.

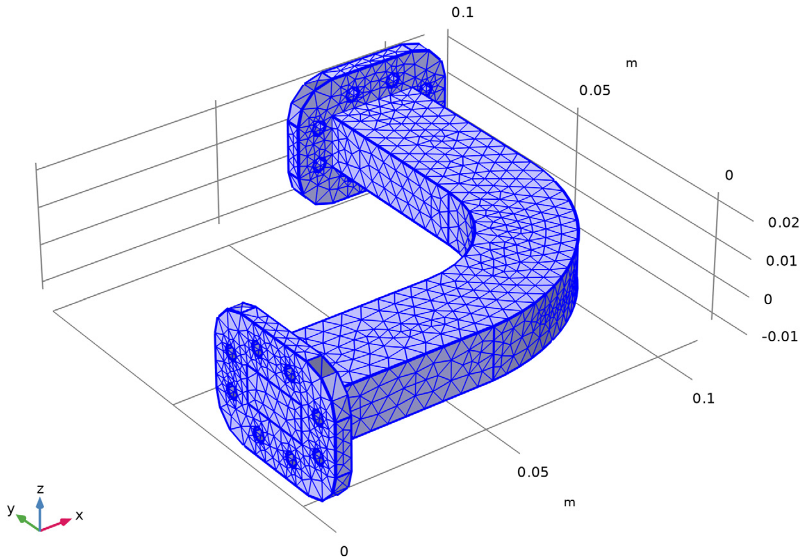
6 Locate the **Element Size Parameters** section.

7 Select the **Maximum element size** checkbox. In the associated text field, type `h_max/sqrt(2.1)`.

Free Tetrahedral 1

1 In the **Mesh** toolbar, click **Free Tetrahedral**.

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



STUDY 1

Step 1: Frequency–Transient

1. In the **Model Builder** window, under **Study 1** click **Step 1: Frequency–Transient**.
2. In the **Settings** window for **Frequency–Transient**, locate the **Study Settings** section.
3. In the **Output times** text field, type `range(0,15,300)`.
4. From the **Tolerance** list, choose **User controlled**.
5. In the **Relative tolerance** text field, type `0.001`.
6. In the **Frequency** text field, type `f0`.
7. In the **Model Builder** window, click **Study 1**.
8. In the **Settings** window for **Study**, locate the **Study Settings** section.
9. Clear the **Generate default plots** checkbox.
10. In the **Study** toolbar, click **Compute**.

RESULTS

Plot the transient response of the peak temperature.

Maximum 1

1 In the **Model Builder** window, expand the **Results** node.

2 Right-click **Results** > **Datasets** and choose **More Datasets** > **Maximum**.

1D Plot Group 2

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

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

3 From the **Dataset** list, choose **Maximum 1**.

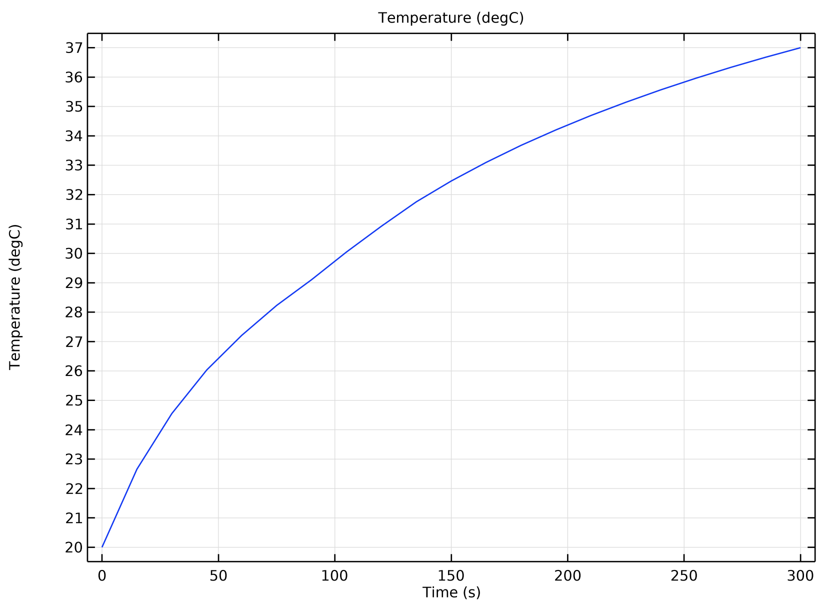
Point Graph 1

1 Right-click **1D Plot Group 2** and choose **Point Graph**.

2 In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)** > **Heat Transfer in Solids** > **Temperature** > **T - Temperature - K**.

3 Locate the **y-Axis Data** section. In the **Unit** field, type `degC`.

4 In the **1D Plot Group 2** toolbar, click **Plot**.



Compare the resulting plot with that shown in Figure 2.

Next, add a Frequency-Stationary study to evaluate the peak temperature which can be observed with the **Frequency-Transient** study after applying a time sufficiently long that the peak temperature is saturated.

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 **Empty Study**.

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

Step 1: Frequency–Stationary

1 In the **Study** toolbar, click **More Study Steps** and choose **Stationary** > **Frequency–Stationary**.

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

3 In the **Frequency** text field, type `f0`.

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

RESULTS

Temperature (ht)

The default plots show the distribution of the electric field norm and the temperature. For the temperature plot, first change the unit to the degree Celsius.

Volume 1

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

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

3 In the **Unit** field, type `degC`.

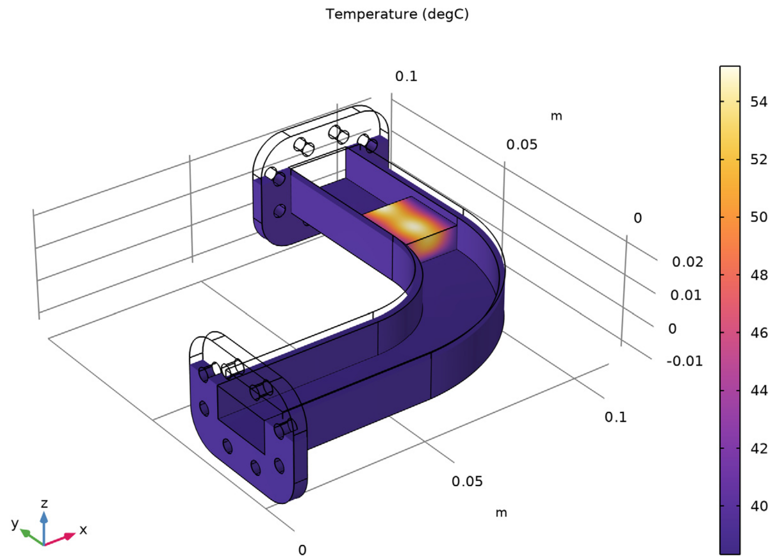
Filter 1

1 Right-click **Volume 1** and choose **Filter**.

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

3 In the **Logical expression for inclusion** text field, type `z<0.01`.

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



Then, add arrow plots of the electric fields, magnetic fields, and power flow.

Electric Field, Magnetic Field, and Power Flow

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

2 In the **Settings** window for **3D Plot Group**, type `Electric Field, Magnetic Field, and Power Flow` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Probe Solution 2 (sol2)**.

Arrow Volume 1

1 Right-click **Electric Field, Magnetic Field, and Power Flow** and choose **Arrow Volume**.

2 In the **Settings** window for **Arrow Volume**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Electromagnetic Waves, Frequency Domain** > **Electric** > **emw.Ex,emw.Ey,emw.Ez - Electric field**.

3 Locate the **Arrow Positioning** section. Find the **X grid points** subsection. In the **Points** text field, type `40`.

4 Find the **Y grid points** subsection. In the **Points** text field, type `40`.

5 Find the **Z grid points** subsection. In the **Points** text field, type `1`.

6 In the **Electric Field, Magnetic Field, and Power Flow** toolbar, click **Plot**.

Arrow Volume 2

1 Right-click **Arrow Volume 1** and choose **Duplicate**.

2 In the **Settings** window for **Arrow Volume**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Electromagnetic Waves, Frequency Domain** > **Magnetic** > **emw.Hx,emw.Hy,emw.Hz - Magnetic field**.

3 Locate the **Coloring and Style** section. From the **Color** list, choose **Green**.

Arrow Volume 3

1 Right-click **Arrow Volume 2** and choose **Duplicate**.

2 In the **Settings** window for **Arrow Volume**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Electromagnetic Waves, Frequency Domain** > **Energy and power** > **emw.Poavx,..., emw.Poavz - Power flow, time average**.

3 Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.

Volume 1

1 In the **Model Builder** window, right-click **Electric Field, Magnetic Field, and Power Flow** and choose **Volume**.

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

3 In the **Expression** text field, type `1`.

Material Appearance 1

1 Right-click **Volume 1** and choose **Material Appearance**.

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

3 From the **Appearance** list, choose **Custom**.

4 From the **Material type** list, choose **Chrome**.

Selection 1

1 In the **Model Builder** window, right-click **Volume 1** and choose **Selection**.

2 Select Domain 1 only.

Filter 1

1 Right-click **Volume 1** and choose **Filter**.

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

3 In the **Logical expression for inclusion** text field, type `z<0.01`.

4 In the **Electric Field, Magnetic Field, and Power Flow** toolbar, click **Plot**.

Surface 1

1 In the **Model Builder** window, right-click **Electric Field, Magnetic Field, and Power Flow** and choose **Surface**.

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

3 In the **Expression** text field, type 1.

Selection 1

1 Right-click **Surface 1** and choose **Selection**.

2 Select Boundaries 16, 17, 35, 53, 54, 72, 74, 75, 78, 96, and 97 only.

Material Appearance 1

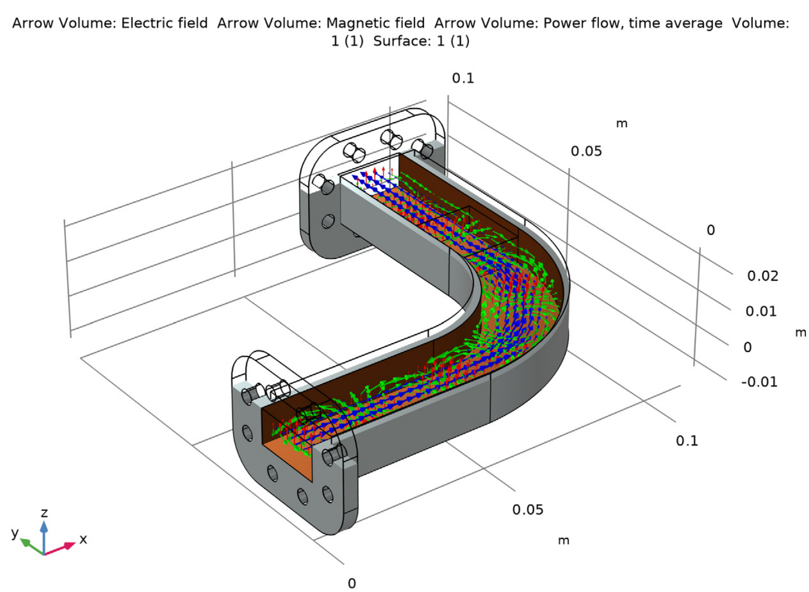
1 In the **Model Builder** window, right-click **Surface 1** and choose **Material Appearance**.

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

3 From the **Appearance** list, choose **Custom**.

4 From the **Material type** list, choose **Copper**.

5 In the **Electric Field, Magnetic Field, and Power Flow** toolbar, click **Plot**.



Finally, reproduce the plot of the loss tangent on the dielectric block shown in Figure 4.

Surface 2

1 In the **Model Builder** window, right-click **Electric Field, Magnetic Field, and Power Flow** 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** > **Material properties** > **emw.delta - Loss tangent, loss angle - rad**.

3 Locate the **Coloring and Style** section. From the **Color table** list, choose **Ctenophora**.

4 In the **Electric Field, Magnetic Field, and Power Flow** toolbar, click **Plot**.

5 Click the **Zoom Box** button in the **Graphics** toolbar and then use the mouse to zoom in.