

# RF Heating

This model is licensed under the COMSOL Software License Agreement 6.1. All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

# 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 electrical

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,  $\delta = 0.001 \times (T/300 \text{ K})$ . The thermal conductivity of this block is also a function of temperature,  $k = 0.3 \times (T/300 \text{ K}) \text{ W/m/K}$ . Furthermore, the density is 2200 kg/m<sup>3</sup> 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, is 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 fields inside of the waveguide, as well as 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.



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

Surface: Temperature (degC) Arrow Volume: Electric field Arrow Volume: Magnetic field Arrow Volume: Power flow, time average



Figure 3: The electric fields (red arrows) magnetic fields (green arrows) and power flow (blue arrows) are shown inside of the waveguide. 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.

# 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

- I In the Model Wizard window, click 间 3D.
- 2 In the Select Physics tree, select Heat Transfer>Electromagnetic Heating> Microwave Heating.
- 3 Click Add.
- 4 Click  $\bigcirc$  Study.
- 5 In the Select Study tree, select Preset Studies for Selected Multiphysics>Frequency-Transient.
- 6 Click 🗹 Done.

#### **GLOBAL DEFINITIONS**

Parameters 1

- I In the Model Builder window, under Global Definitions click Parameters I.
- 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]	IEI0 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 I

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

Import I (imp1)

- I In the **Home** toolbar, click া Import.
- 2 In the Settings window for Import, locate the Import 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

- I In the **Definitions** toolbar, click **here 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

- I 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

- I 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

- I 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

- I In the **Definitions** toolbar, click **herefore 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.



To get a better view, suppress some of the boundaries. Furthermore, by assigning the resulting settings to a View node, you can easily return to the same view later by clicking the **Go to View 2** button in the **Graphics** toolbar.

#### View 2

- I In the Model Builder window, right-click Definitions and choose View.
- **2** Click the 🔁 Wireframe Rendering button in the Graphics toolbar.

#### Hide for Physics 1

- I In the Model Builder window, right-click View 2 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 18 and 50 only.



#### ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

- I In the Model Builder window, under Component I (compl) 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, select **I**.
- **4** Click  **Remove from Selection**.
- **5** Select Domains 2 and 3 only.

#### Wave Equation, Electric 1

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

I In the Model Builder window, under Component I (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, select 2.
- 4 Click Remove from Selection.
- **5** Select Domains 1 and 3 only.

#### Heat Flux 1

- I 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 I (compl) click Electromagnetic Waves, Frequency Domain (emw).

#### Wave Equation, Electric 2

- I 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

- I 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 I

I 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  $P_{\rm in}$  text field, type 100.

# Port 2

I 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

- I In the Home toolbar, click 🙀 Add Material to open the Add Material window.
- 2 Go to the Add Material window.
- 3 In the tree, select Built-in>Aluminum.
- 4 Click Add to Component in the window toolbar.
- 5 In the tree, select Built-in>Air.
- 6 Click Add to Component in the window toolbar.

# MATERIALS

# Aluminum (mat1)

I In the Model Builder window, under Component I (compl)>Materials click Aluminum (matl).

- 2 In the Settings window for Material, locate the Geometric Entity Selection section.
- **3** From the **Selection** list, choose **Waveguide**.

#### Air (mat2)

- I 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

- I 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_i so; epsilonPrimii = epsilonPrim_i so, 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	Ср	1050	J/(kg·K)	Basic

# ADD MATERIAL

- I Go to the **Add Material** window.
- 2 In the tree, select **Built-in>Copper**.
- **3** Click **Add to Component** in the window toolbar.

4 In the Home toolbar, click 🙀 Add Material to close the Add Material window.

## MATERIALS

Copper (mat4)

- I 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 I

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

- I In the Model Builder window, under Component I (comp1) right-click Mesh I 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 check box. In the associated text field, type h\_max.

Size 2

- I In the Model Builder window, right-click Mesh I 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 check box. In the associated text field, type h\_max/ sqrt(2.1).

Free Tetrahedral I

I In the Mesh toolbar, click \land Free Tetrahedral.

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



# STUDY I

Step 1: Frequency-Transient

- I In the Model Builder window, under Study I click Step I: 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 I.
- 8 In the Settings window for Study, locate the Study Settings section.
- 9 Clear the Generate default plots check box.
- **IO** In the **Home** toolbar, click **= Compute**.

## RESULTS

Plot the transient response of the peak temperature.

#### Maximum I

- I In the Model Builder window, expand the Results node.
- 2 Right-click Results>Datasets and choose More Datasets>Maximum.

#### ID Plot Group 1

- I In the Results toolbar, click  $\sim$  ID Plot Group.
- 2 In the Settings window for ID Plot Group, locate the Data section.
- 3 From the Dataset list, choose Maximum I.

#### Point Graph 1

- I Right-click ID Plot Group I 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 I (compl)> Heat Transfer in Solids>Temperature>T Temperature K.
- 3 Locate the y-Axis Data section. From the Unit list, choose degC.
- 4 In the ID Plot Group I 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 enough long time so the peak temperature is saturated.

## ADD STUDY

- I In the Home toolbar, click  $\sim_1^{\circ}$  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 Add Study in the window toolbar.
- 5 In the Home toolbar, click  $\stackrel{\text{res}}{\longrightarrow}$  Add Study to close the Add Study window.

### STUDY 2

#### Frequency-Stationary

- I In the Study toolbar, click **T** 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 and then, add arrow plots of the electric fields, magnetic fields, and power flow.

# Surface

- I In the Model Builder window, expand the Temperature (ht) node, then click Surface.
- 2 In the Settings window for Surface, locate the Expression section.
- 3 From the Unit list, choose degC.

# Arrow Volume 1

- I In the Model Builder window, right-click Temperature (ht) 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 I (compl)> 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 Temperature (ht) toolbar, click 💽 Plot.

## Arrow Volume 2

- I Right-click Arrow Volume I 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 I (compl)> 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

- I 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 I (compl)>
  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.

Compare the resulting plot with that shown in Figure 3.

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

To create the plot, reuse the plot group named Isothermal Contours (ht).

Loss Tangent (emw)

- I In the Model Builder window, under Results click Isothermal Contours (ht).
- 2 In the Settings window for 3D Plot Group, type Loss Tangent (emw) in the Label text field.

Isosurface

- I In the Model Builder window, expand the Loss Tangent (emw) node.
- 2 Right-click Results>Loss Tangent (emw)>lsosurface and choose Delete.

Surface 1

- I In the Model Builder window, right-click Loss Tangent (emw) and choose Surface.
- 2 In the Settings window for Surface, click Replace Expression in the upper-right corner of the Expression section. From the menu, choose Component I (compl)>
  Electromagnetic Waves, Frequency Domain>Material properties>emw.delta Loss tangent, loss angle rad.

3 In the Loss Tangent (emw) toolbar, click 💽 Plot.

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