

# RF MODULE

MODEL LIBRARY

**VERSION 3.4**

## How to contact COMSOL:

### Benelux

COMSOL BV  
Röntgenlaan 19  
2719 DX Zoetermeer  
The Netherlands  
Phone: +31 (0) 79 363 4230  
Fax: +31 (0) 79 361 4212  
info@femlab.nl  
www.femlab.nl

### Denmark

COMSOL A/S  
Diplomvej 376  
2800 Kgs. Lyngby  
Phone: +45 88 70 82 00  
Fax: +45 88 70 80 90  
info@comsol.dk  
www.comsol.dk

### Finland

COMSOL OY  
Arabianranta 6  
FIN-00560 Helsinki  
Phone: +358 9 2510 400  
Fax: +358 9 2510 4010  
info@comsol.fi  
www.comsol.fi

### France

COMSOL France  
WTC, 5 pl. Robert Schuman  
F-38000 Grenoble  
Phone: +33 (0)4 76 46 49 01  
Fax: +33 (0)4 76 46 07 42  
info@comsol.fr  
www.comsol.fr

### Germany

FEMLAB GmbH  
Berliner Str. 4  
D-37073 Göttingen  
Phone: +49-551-99721-0  
Fax: +49-551-99721-29  
info@femlab.de  
www.femlab.de

### Italy

COMSOL S.r.l.  
Via Vittorio Emanuele II, 22  
25122 Brescia  
Phone: +39-030-3793800  
Fax: +39-030-3793899  
info.it@comsol.com  
www.it.comsol.com

### Norway

COMSOL AS  
Søndre gate 7  
NO-7485 Trondheim  
Phone: +47 73 84 24 00  
Fax: +47 73 84 24 01  
info@comsol.no  
www.comsol.no

### Sweden

COMSOL AB  
Tegnérsgatan 23  
SE-111 40 Stockholm  
Phone: +46 8 412 95 00  
Fax: +46 8 412 95 10  
info@comsol.se  
www.comsol.se

### Switzerland

FEMLAB GmbH  
Technoparkstrasse 1  
CH-8005 Zürich  
Phone: +41 (0)44 445 2140  
Fax: +41 (0)44 445 2141  
info@femlab.ch  
www.femlab.ch

### United Kingdom

COMSOL Ltd.  
UH Innovation Centre  
College Lane  
Hatfield  
Hertfordshire AL10 9AB  
Phone: +44-(0)-1707 284747  
Fax: +44-(0)-1707 284746  
info.uk@comsol.com  
www.uk.comsol.com

### United States

COMSOL, Inc.  
1 New England Executive Park  
Suite 350  
Burlington, MA 01803  
Phone: +1-781-273-3322  
Fax: +1-781-273-6603

COMSOL, Inc.  
10850 Wilshire Boulevard  
Suite 800  
Los Angeles, CA 90024  
Phone: +1-310-441-4800  
Fax: +1-310-441-0868

COMSOL, Inc.  
744 Cowper Street  
Palo Alto, CA 94301  
Phone: +1-650-324-9935  
Fax: +1-650-324-9936

info@comsol.com  
www.comsol.com

For a complete list of international  
representatives, visit  
[www.comsol.com/contact](http://www.comsol.com/contact)

**Company home page**  
[www.comsol.com](http://www.comsol.com)

**COMSOL user forums**  
[www.comsol.com/support/forums](http://www.comsol.com/support/forums)

## *RF Module Model Library*

© COPYRIGHT 1994–2007 by COMSOL AB. All rights reserved

Patent pending

The software described in this document is furnished under a license agreement. The software may be used or copied only under the terms of the license agreement. No part of this manual may be photocopied or reproduced in any form without prior written consent from COMSOL AB.

COMSOL, COMSOL Multiphysics, COMSOL Reaction Engineering Lab, and FEMLAB are registered trademarks of COMSOL AB. COMSOL Script is a trademark of COMSOL AB.

Other product or brand names are trademarks or registered trademarks of their respective holders.

Version:                      October 2007                      COMSOL 3.4

# C O N T E N T S

## Chapter 1: Introduction

Model Library Guide . . . . .	2
Typographical Conventions . . . . .	5

## Chapter 2: Tutorial Models

<b>Dielectric Scattering PML</b>	<b>8</b>
Model Definition . . . . .	8
Results and Discussion. . . . .	9
Modeling Using the Graphical User Interface . . . . .	10
<b>Magnetic Frill</b>	<b>14</b>
Introduction . . . . .	14
Model Definition . . . . .	14
Results and Discussion. . . . .	14
Reference . . . . .	15
Modeling Using the Graphical User Interface . . . . .	15
<b>Microstrip on a Printed Circuit Board</b>	<b>20</b>
Solving with Lumped Port Termination. . . . .	28

## Chapter 3: RF and Microwave Models

<b>Three-Port Ferrite Circulator</b>	<b>32</b>
Introduction . . . . .	32
Model Definition . . . . .	33
Results and Discussion. . . . .	35
References . . . . .	37
Modeling Using the Graphical User Interface . . . . .	38
Extracting the S-Parameter Matrix . . . . .	43
Exporting S-Parameters versus Frequency to Touchstone . . . . .	44

<b>Monoconical RF Antenna</b>	<b>45</b>
Introduction . . . . .	45
Model Definition . . . . .	45
Results and Discussion. . . . .	46
Modeling Using the Graphical User Interface . . . . .	49
Antenna Impedance and Radiation Pattern . . . . .	54
Far-Field Computation. . . . .	56
<b>Magnetic Dipole Antenna</b>	<b>57</b>
Introduction . . . . .	57
Model Definition . . . . .	57
Results and Discussion. . . . .	58
Modeling Using the Graphical User Interface . . . . .	58
Modeling Using the Programming Language . . . . .	62
<b>Thermal Drift in a Microwave Filter</b>	<b>64</b>
Introduction . . . . .	64
Model Definition . . . . .	64
Results and Discussion. . . . .	65
Modeling Using the Graphical User Interface . . . . .	68
<b>Waveguide H-Bend with S-parameters</b>	<b>80</b>
Introduction . . . . .	80
Model Definition . . . . .	81
Results and Discussion. . . . .	81
Modeling Using the Graphical User Interface . . . . .	82
<b>Waveguide Adapter</b>	<b>85</b>
Introduction . . . . .	85
Model Definition . . . . .	85
Results. . . . .	87
Modeling in COMSOL Multiphysics . . . . .	90
Modeling Using the Graphical User Interface . . . . .	91
Mode Analysis . . . . .	91
S-Parameter Analysis . . . . .	94
Creating the Geometry in COMSOL Script or MATLAB . . . . .	96

<b>Microwave Cancer Therapy</b>	<b>98</b>
Introduction . . . . .	98
Model Definition . . . . .	98
Results and Discussion. . . . .	102
Reference . . . . .	104
Modeling in COMSOL Multiphysics . . . . .	104
Modeling Using the Graphical User Interface . . . . .	105
<b>Absorbed Radiation (SAR) in the Human Brain</b>	<b>110</b>
Introduction . . . . .	110
Model Definition . . . . .	110
Results and Discussion. . . . .	112
References . . . . .	113
Modeling Using the Graphical User Interface . . . . .	114
<b>Microwave Oven</b>	<b>124</b>
Introduction . . . . .	124
Model Definition . . . . .	124
Results. . . . .	125
Modeling in COMSOL Multiphysics . . . . .	128
Modeling Using the Graphical User Interface . . . . .	128
Electromagnetic Analysis . . . . .	129
<b>Balanced Patch Antenna for 6 GHz</b>	<b>136</b>
Introduction . . . . .	136
Model Definition . . . . .	137
Results and Discussion. . . . .	139
Modeling in COMSOL Multiphysics . . . . .	141
Reference . . . . .	142
Modeling Using the Graphical User Interface . . . . .	142
<b>Coaxial to Waveguide Coupling</b>	<b>157</b>
Introduction . . . . .	157
Results and Discussion. . . . .	157
Modeling Using the Graphical User Interface . . . . .	159
<b>Sea Bed Logging</b>	<b>164</b>
Introduction . . . . .	164

Model Definition . . . . .	164
Results. . . . .	164
Modeling in COMSOL Multiphysics . . . . .	166
Modeling Using the Graphical User Interface . . . . .	167

## Chapter 4: Optics and Photonics Models

<b>Photonic Microprism</b>	<b>174</b>
Introduction . . . . .	174
Model Definition . . . . .	175
Results and Discussion. . . . .	176
References . . . . .	176
Modeling Using the Graphical User Interface . . . . .	177
<b>Photonic Crystal</b>	<b>182</b>
Introduction . . . . .	182
Model Definition . . . . .	183
Results and Discussion. . . . .	183
References . . . . .	184
Modeling Using the Graphical User Interface . . . . .	184
Modeling Using the Programming Language . . . . .	190
<b>Bandgap Analysis of a Photonic Crystal</b>	<b>192</b>
Introduction . . . . .	192
Model Definition . . . . .	192
Results and Discussion. . . . .	195
References . . . . .	197
Modeling Using the Graphical User Interface . . . . .	198
Modeling Larger Sweeps With COMSOL Script . . . . .	204
<b>Step-Index Fiber</b>	<b>208</b>
Introduction . . . . .	208
Model Definition . . . . .	209
Results and Discussion. . . . .	209
Reference . . . . .	210
Modeling Using the Graphical User Interface . . . . .	210

Modeling Using the Programming Language . . . . .	213
<b>Stress-Optical Effects in a Silica-on-Silicon Waveguide</b>	<b>215</b>
Introduction . . . . .	215
The Stress-Optical Effect and Plane Strain . . . . .	215
Perpendicular Hybrid-Mode Waves . . . . .	216
Plane Strain Analysis . . . . .	217
Optical Mode Analysis . . . . .	225
Convergence Analysis . . . . .	227
<b>Stress-Optical Effects with Generalized Plane Strain</b>	<b>232</b>
Introduction . . . . .	232
Generalized Plane Strain . . . . .	232
Plane Strain Analysis . . . . .	238
Optical Mode Analysis . . . . .	246
<b>Second Harmonic Generation of a Gaussian Beam</b>	<b>249</b>
Introduction . . . . .	249
Model Definition . . . . .	249
Results and Discussion. . . . .	251
Reference . . . . .	253
Modeling Using the Graphical User Interface . . . . .	253
<b>Propagation of a 3D Gaussian Beam Laser Pulse</b>	<b>261</b>
Introduction . . . . .	261
Model Definition . . . . .	261
Results and Discussion. . . . .	262
Modeling Using the Graphical User Interface . . . . .	263
<b>INDEX</b>	<b>271</b>





# Introduction

The *RF Module Model Library* consists of a set of models from various areas of electromagnetic wave simulations. Their purpose is to assist you in learning, by example, how to model sophisticated electromagnetic components, systems, and effects. Through them, you can tap the expertise of the top researchers in the field, examining how they approach some of the most difficult modeling problems you might encounter. You can thus get a feel for the power that COMSOL Multiphysics offers as a modeling tool. In addition to serving as a reference, the models can also give you a big head start if you are developing a model of a similar nature.

We have divided these models into three groups: optics and photonics, RF and microwave engineering, and tutorial models. The models also illustrate the use of the various electromagnetics-specific application modes from which we built them. These specialized modes are unavailable in the base COMSOL Multiphysics package, and they come with their own graphical-user interfaces that make it quick and easy to access their power. You can even modify them for custom requirements. COMSOL Multiphysics itself is very powerful and, with sufficient expertise in a given field, you certainly could develop these modes by yourself—but why spend the hundreds or thousands of hours that would be necessary when our team of experts has already done the work for you?

Note that the model descriptions in this book do not contain every detail on how to carry out every step in the modeling process. Before tackling these in-depth models, we urge you to first read the second book in the RF Module documentation set. Titled the *RF Module User's Guide*, it introduces you to the basic functionality in the module, reviews new features in the version 3.4 release, covers basic modeling techniques and includes reference material of interest to those working with electromagnetic field simulations. For more information on how to work with the COMSOL Multiphysics graphical user interface, please refer to the *COMSOL Multiphysics User's Guide* or the *COMSOL Multiphysics Quick Start* manual. An explanation on how to model with a programming language is available in yet another book, the *COMSOL Multiphysics Scripting Guide*.

The book in your hands, the *RF Module Model Library*, provides details about a large number of ready-to-run models that illustrate real-world uses of the module. Each entry comes with theoretical background as well as instructions that illustrate how to set it up. They were written by our staff engineers who have years of experience in electromagnetics; they are your peers, using the language and terminology needed to get across the sophisticated concepts in these advanced topics.

Finally note that we supply these models as COMSOL Multiphysics Model MPH-files so you can import them into COMSOL Multiphysics for immediate execution, allowing you to follow along with these examples every step along the way.

### *Model Library Guide*

---

The table below summarizes key information about the entries in the *RF Module Model Library*. A series of columns states the application mode (such as Perpendicular Currents) used to solve the corresponding model. The solution time is the elapsed time measured on a machine running Windows Vista with a 2.6 GHz AMD Athlon X2 Dual Core 500 CPU and 2 GB of RAM. For models with a sequential solution strategy, the Solution Time column shows the elapsed time for the longest solution step. Additional columns point out the solution properties a given example highlights.

The choices here include the type of analysis (such as time dependent) and whether multiphysics or parametric studies are included.

TABLE I-1: RF MODULE MODEL LIBRARY

MODEL	PAGE	APPLICATION MODE	SOLUTION TIME	STATIONARY	TIME-HARMONIC	TIME DEPENDENT	EIGENFREQUENCY/EIGENMODE	NONLINEAR	MULTIPHYSICS	PARAMETRIC STUDY
<b>TUTORIAL MODELS</b>										
Periodic Boundary Conditions	37*	3D Electromagnetic Waves	11 s				√			
Dielectric Scattering PML	48*	In-Plane TE Waves	2 s		√					
Far-Field Postprocessing	50*	Axisymmetric TE Waves	1 s		√					
Microstrip on a Printed Circuit Board	68*	3D Electromagnetic Waves; Boundary Mode Analysis, Hybrid-Mode Waves	2 min		√		√			
Lossy Circular Waveguide	81*	Perpendicular Hybrid-Mode Waves	3 s				√			
Magnetic Frill	14	3D Electromagnetic Waves	34 s		√					
<b>RF AND MICROWAVE MODELS</b>										
Waveguide H-Bend	21*	3D Electromagnetic Waves, In-Plane TE Waves	8 s		√					
		2D In-Plane TE Waves	1 s		√					
Three-Port Ferrite Circulator	32	In-Plane TE Waves	18 s		√					
Monoconical RF Antenna	45	Axisymmetric TM Waves	57 s		√					√
Magnetic Dipole Antenna	57	Axisymmetric TE Waves	1 s		√					

TABLE 1-1: RF MODULE MODEL LIBRARY

MODEL	PAGE	APPLICATION MODE	SOLUTION TIME	STATIONARY	TIME-HARMONIC	TIME DEPENDENT	EIGENFREQUENCY/EIGENMODE	NONLINEAR	MULTIPHYSICS	PARAMETRIC STUDY
Thermal Drift in a Microwave Filter	64	3D Electromagnetic Waves, Moving Mesh, Shell	48 s	√			√		√	√
Waveguide H-Bend with S-parameters	80	In-Plane TE Waves	24 s		√					√
Waveguide Adapter	85	Boundary Mode Analysis, 3D Electromagnetic Waves	4 min		√		√		√	√
Microwave Cancer Therapy	98	Axisymmetric TM Waves, Bioheat Equation	52 s		√				√	
Absorbed Radiation (SAR) in the Human Brain	110	3D Electromagnetic Waves, Bioheat Equation	52 min	√					√	
Microwave Oven	124	3D Electromagnetic Waves, General Heat Transfer	19 min		√	√			√	
Balanced Patch Antenna for 6 GHz	136	3D Electromagnetic Waves	≈7 h		√					√
Coaxial to Waveguide Coupling	157	3D Electromagnetic Waves	9 min		√					
Sea Bed Logging	164	3D Electromagnetic Waves	2 min		√					
<b>OPTICS AND PHOTONICS MODELS</b>										
Photonic Microprism	174	In-Plane TE Waves	2 s		√					
Photonic Crystal	182	In-Plane TE Waves	2 s		√					
Bandgap Analysis of a Photonic Crystal	192	In-Plane TE Waves	3 min				√			√

TABLE 1-1: RF MODULE MODEL LIBRARY

MODEL	PAGE	APPLICATION MODE	SOLUTION TIME	STATIONARY	TIME-HARMONIC	TIME DEPENDENT	EIGENFREQUENCY/EIGENMODE	NONLINEAR	MULTIPHYSICS	PARAMETRIC STUDY
Step-Index Fiber	208	Perpendicular Hybrid-Mode Waves	7 s				√			
Stress-Optical Effects in a Silica-on-Silicon Waveguide	215	Plane Strain, Perpendicular Hybrid-Mode Waves	2 s	√			√		√	
Stress-Optical Effects with Generalized Plane Strain	232	Plane Strain, Perpendicular Hybrid-Mode Waves, Weak Form, Point	7 s	√			√		√	
Second Harmonic Generation of a Gaussian Beam	249	In-Plane TE Waves, ODE	4 min			√		√		
Propagation of a 3D Gaussian Beam Laser Pulse	261	3D Electromagnetic Waves	128 min			√				

\* this page number refers to the *RF Module User's Guide*.

□ this model requires a 64-bit platform.

### *Typographical Conventions*

All COMSOL manuals use a set of consistent typographical conventions that should make it easy for you to follow the discussion, realize what you can expect to see on the screen, and know which data you must enter into various data-entry fields. In particular, you should be aware of these conventions:

- A **boldface** font of the shown size and style indicates that the given word(s) appear exactly that way on the COMSOL graphical user interface (for toolbar buttons in the corresponding tooltip). For instance, we often refer to the **Model Navigator**, which is the window that appears when you start a new modeling session in

COMSOL; the corresponding window on the screen has the title **Model Navigator**. As another example, the instructions might say to click the **Multiphysics** button, and the boldface font indicates that you can expect to see a button with that exact label on the COMSOL user interface.

- The names of other items on the graphical user interface that do not have direct labels contain a leading uppercase letter. For instance, we often refer to the Draw toolbar; this vertical bar containing many icons appears on the left side of the user interface during geometry modeling. However, nowhere on the screen will you see the term “Draw” referring to this toolbar (if it were on the screen, we would print it in this manual as the **Draw** menu).
- The symbol > indicates a menu item or an item in a folder in the **Model Navigator**. For example, **Physics>Equation System>Subdomain Settings** is equivalent to: On the **Physics** menu, point to **Equation System** and then click **Subdomain Settings**. **COMSOL Multiphysics>Heat Transfer>Conduction** means: Open the **COMSOL Multiphysics** folder, open the **Heat Transfer** folder, and select **Conduction**.
- A Code (monospace) font indicates keyboard entries in the user interface. You might see an instruction such as “Type 1.25 in the **Current density** edit field.” The monospace font also indicates COMSOL Script codes.
- An *italic* font indicates the introduction of important terminology. Expect to find an explanation in the same paragraph or in the Glossary. The names of books in the COMSOL documentation set also appear using an italic font.

## Tutorial Models

This chapter contains a selection of models that demonstrate important features and modeling techniques. Therefore, this chapter contains models from many different application areas. For pedagogical reasons, these models are kept as simple as possible with the main focus on the feature that they demonstrate. See also the chapter “RF Modeling” on page 11 in the *RF Module User’s Guide* for additional tutorial models.

# Dielectric Scattering PML

This is a tutorial example to show how to add PMLs and how to use the scattered field formulation. In the RF Module you can define perfectly matched layers by adding additional subdomains outside the boundaries that you would like to be absorbing. Because this model is in 2D, the PML regions are from the 2D cross section of the PML configuration in Figure 2-4 on page 44 in the *RF Module User's Guide*.

## *Model Definition*

---

The model shows a wave propagating from left to right through air. A dielectric object scatters the wave in different directions.

### **DOMAIN EQUATIONS**

The model solves the following equation for the  $z$  component of the scattered  $\mathbf{E}$  field:

$$\nabla \times (\mu_r^{-1} \nabla \times \mathbf{E}_z) - \left( \epsilon_r - j \frac{\sigma}{\omega \epsilon_0} \right) k_0^2 \mathbf{E}_z = 0$$

To define the PMLs you introduce new subdomains around the air domain representing absorbing layers.

- In the corners the PMLs absorb the waves in both the  $x$  and  $y$  directions.
- On the sides only the waves propagating in  $x$  direction are absorbed.
- On the top and bottom the waves propagating in the  $y$  direction are absorbed.

### **BOUNDARY CONDITIONS**

Using an absorbing boundary condition at the outer boundaries of the PML regions improves the efficiency of the PMLs. As a result, use the scattering boundary condition on all outer boundaries.



## Results and Discussion

---

Figure 2-1 shows the scattered field without the plane wave.

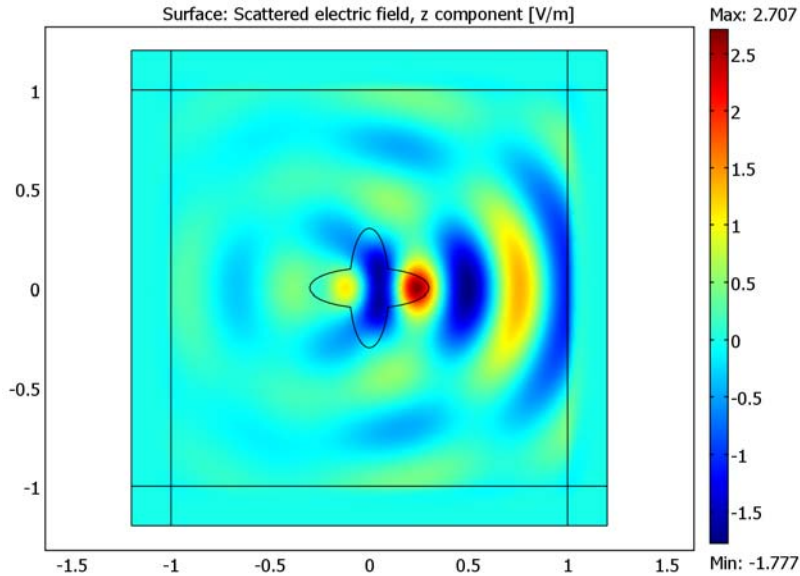


Figure 2-1: The figure shows the scattered field from the dielectric object.

The PMLs absorb the scattered energy with a minimum of reflections. If there was no dielectric object inside the domain, the low-reflecting and PMC boundary conditions would be enough. In that case the waves would propagate only in the  $x$  direction and the low-reflecting boundary condition would absorb the plane wave perfectly. With a scatterer present, the PMLs are necessary to absorb the scattered wave that you can see in Figure 2-1.

---

**Model Library path:** RF\_Module/Tutorial\_Models/  
dielectric\_scattering\_PML

---

**MODEL NAVIGATOR**

- 1 In the **Model Navigator** open the **RF Module** folder, then select **In-Plane Waves>TE Waves>Scattered Harmonic Propagation**.
- 2 Click **OK**.

**GEOMETRY MODELING**

- 1 Draw four rectangles by Shift-clicking the **Rectangle/Square** button on the Draw toolbar. The rectangles have the following dimensions.

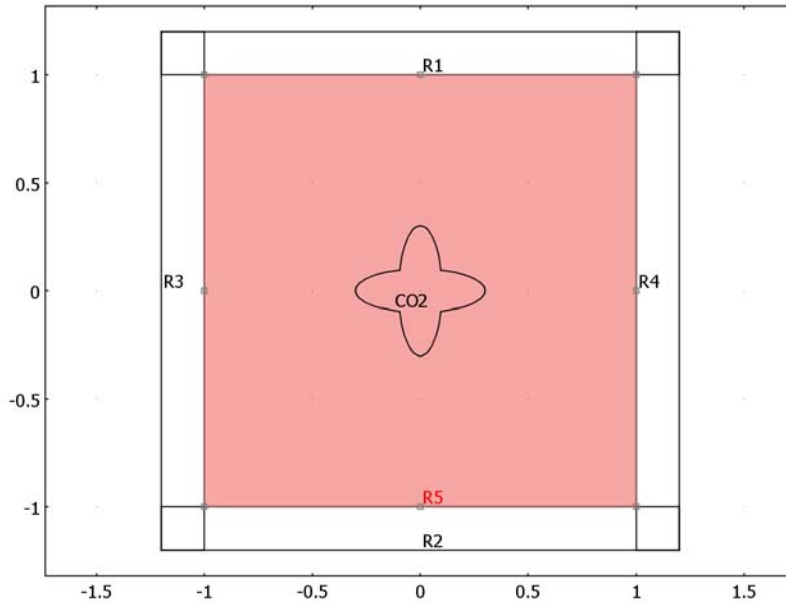
PROPERTY	R1	R2	R3	R4
Width	2.4	2.4	0.2	0.2
Height	0.2	0.2	2.4	2.4
Position corner x	-1.2	-1.2	-1.2	1
Position corner y	1	-1.2	-1.2	-1.2

- 2 Draw two ellipses centered at the origin with the following semi-axis lengths by Shift-clicking the **Ellipse/Circle** button on the Draw toolbar.

PROPERTY	E1	E2
A-semiaxis	0.1	0.3
B-semiaxis	0.3	0.1

- 3 Select both ellipses and click the **Union** button and then the **Delete Interior Boundaries** button on the Draw toolbar.

- 4 Draw a square with corners at  $(-1, -1)$  and  $(1, 1)$ .



## PHYSICS SETTINGS

### *Application Mode Properties*

- 1 Choose **Properties** from the **Physics** menu.
- 2 In the **Application Mode Properties** dialog box, select **Free space wavelength** from the **Specify wave using** list.
- 3 Click **OK**.

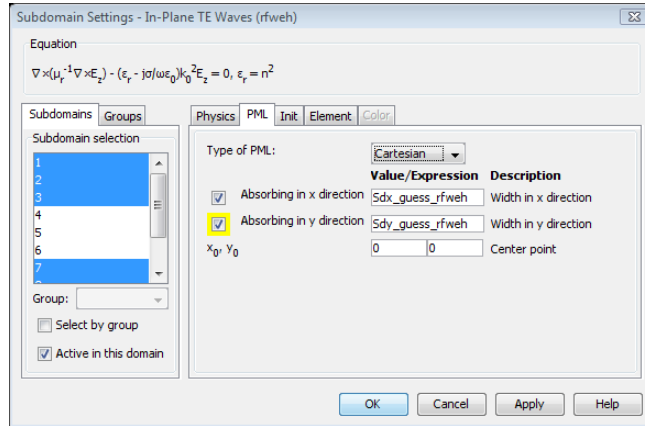
### *Application Scalar Variables*

- 1 Open the **Application Scalar Variables** dialog box.
- 2 Type 0.5 in the edit field corresponding to the variable `lambda0_rfwe`.
- 3 Click **OK**.

### *Subdomain Settings*

- 1 Open the **Subdomain Settings** dialog box.
- 2 Click the **PML** tab, and select Subdomains 1–4 and 6–9.
- 3 Choose **Cartesian** from the **Type of PML** list
- 4 Select Subdomains 1–3 and 7–9.

- 5 Select the **Absorbing in x direction** check box.
- 6 Select Subdomains 1, 3, 4, 6, 7, and 9.
- 7 Select the **Absorbing in y direction** check box.



- 8 Select Subdomain 10, click the **Physics** tab, and set the **Relative permittivity** to 3. Let the **Relative permeability** have the default value 1.
- 9 In Subdomain 5, use the default values.
- 10 Click **OK**.

### Boundary Conditions

- 1 From the **Physics** menu, choose **Boundary Settings**.
- 2 Select Boundary 1 and then select the **Select by group** check box to select all outer boundaries.
- 3 Select **Scattering boundary condition** from the **Boundary condition** list.
- 4 Click **OK**.

### MESH GENERATION

To resolve the waves, you need to use a finer mesh than the default.

- 1 Open the **Free Mesh Parameters** dialog box, click the **Custom mesh size** button, and set the **Maximum element size** to 0.05.
- 2 Click **OK**.
- 3 Initialize the mesh by clicking the **Initialize Mesh** button on the Main toolbar.

## COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar to compute the solution.

## POSTPROCESSING AND VISUALIZATION

Follow these steps to visualize the scattered field without the incident plane wave as in Figure 2-1. To plot the solution in the physical domains only, you suppress the PML subdomains.

- 1 Choose **Suppress>Suppress Subdomains** from the **Options** menu.
- 2 Select Subdomains 1–4 and 6–9. Click **OK**.
- 3 Open **Plot Parameters**.
- 4 On the **Surface** tab, select **Scattered electric field, z component** from the **Predefined quantities** list.
- 5 Click **OK**. This creates the plot in Figure 2-1 on page 9.

# Magnetic Frill

## *Introduction*

---

Feeding antennas with proper signals can be difficult. The signal is often described as a voltage, and voltages are not well defined in electromagnetic wave formulations. There are several tricks to model voltage generators in such situations, and one is the *magnetic frill*. This model shows the basic steps how to define a magnetic frill voltage generator for a dipole antenna, and it also compares the resulting antenna impedance with known results.

## *Model Definition*

---

Magnetic frills can only be defined for the wave formulation using the  $\mathbf{H}$ -field, which is based on the time-harmonic Faraday's law.

$$\nabla \times \mathbf{E} = -j\omega\mathbf{B}$$

Although there are no magnetic charges, it is possible to mathematically define a current of magnetic charges, called a magnetic current. This current enters the right-hand side of Faraday's law in the same manner as the ordinary current enters the right-hand side of Ampère's law. Similar to the ordinary current density that has the unit  $\text{A}/\text{m}^2$ , the magnetic current density has the unit  $\text{V}/\text{m}^2$ .

A closed loop of magnetic current therefore has the unit  $\text{V}$ , and represents a voltage generator for the surface closed by the loop. In this model, the loop is located around a thin straight wire and acts as a voltage source at the center of the wire. This is actually a dipole antenna fed by a voltage signal in the center.

The current through the wire is measured with another loop, along which a line integral of the  $\mathbf{H}$ -field is specified.

$$\int \mathbf{H} \cdot d\mathbf{l} = I$$

Note that this loop and the magnetic current loop must be two different loops.

## *Results and Discussion*

---

The dipole antenna is fed with a voltage signal of 1 V, and from the measured current it is possible to extract the impedance. Taken from Ref. 1, the impedance and

dimensions of a typical dipole antenna are shown in the table below. The dimensions are given in terms of the wavelength,  $\lambda$ .

WAVELENGTH	LENGTH	RADIUS	IMPEDANCE
0.3	$0.47\lambda$	$0.005\lambda$	

The impedance from the COMSOL Multiphysics model is  $78.08+11.88i$ , which agrees well with the results from Ref. 1.

### *Reference*

1. Constantine A. Balanis, *Advanced Engineering Electromagnetics*, John Wiley and Sons, 1989.

---

**Model Library path:** RF\_Module/Tutorial\_Models/magnetic\_frill

---

### *Modeling Using the Graphical User Interface*

#### **MODEL NAVIGATOR**

- 1 Select **3D** from the **Space dimension** list.
- 2 Select the **RF Module>Electromagnetic Waves>Harmonic propagation** application mode.
- 3 From the **Element** list choose **Vector - Quadratic**.
- 4 Click the **Multiphysics** button followed by the **Add** button.
- 5 Click the **Application Mode Properties** button to open the dialog box with the same name.
- 6 In the dialog box, choose **Magnetic Field** from the **Solve for** list, then click **OK**.
- 7 Click **OK**.

#### **OPTIONS AND SETTINGS**

Open the **Global Expressions** dialog box from the **Options** menu, and define the variable.

NAME	EXPRESSION	DESCRIPTION
Z	$1/I$	Impedance of antenna

## GEOMETRY MODELING

You can use symmetries to reduce the size of the model. Only one quarter of the antenna with surroundings is modeled.

- 1 Create a work plane by opening the **Work-Plane Settings** dialog box from the **Draw** menu, click the **yz** button.
- 2 Click **OK** to obtain a work plane in the *yz*-plane.
- 3 Choose **Draw>Specify Objects>Rectangle** and define the following rectangles.

NAME	WIDTH	HEIGHT	BASE	CORNER
R1	0.5	0.6	Corner	(0, -0.3)
R2	0.005*0.3	0.47*0.3	Center	(0, -0.47*0.15)

- 4 Select both rectangles and click the **Difference** toolbar button.
- 5 From the **Draw** menu, use **Specify Objects>Point** to specify two points with the following coordinates.

NAME	X	Y
PT1	0.005*0.3	0
PT2	0.01	0

- 6 Select all objects and choose **Revolve** from the **Draw** menu.
- 7 Enter -90 in the  $\alpha_2$  edit field and click **OK**.

The geometry is now finished, and the two loops were created by revolving the two points in the work plane. If you zoom in closer to the origin, you see two quarter loops.

## PHYSICS SETTINGS

### Variables

- 1 Choose **Options>Integration Coupling Variables>Edge Variables**.
- 2 Select edge I4 and define the line integration for the current in the dipole according to the table below.

NAME	EXPRESSION	INTEGRATION ORDER	GLOBAL DESTINATION
I	$4 * (tHx * t1x + tHy * t1y)$	4	Selected

- 3 Click **OK**.



### Boundary Conditions

From the **Physics** menu, open the **Boundary Settings** dialog box. Then enter the boundary conditions according to the following table.

SETTINGS	BOUNDARIES 1, 2	BOUNDARIES 3, 6, 9	ALL OTHERS
Boundary condition	Perfect magnetic conductor	Scattering boundary condition	Perfect electric conductor
Wave type		Cylindrical wave	
$r_0$		0 0 0.3	

### Subdomain Settings

Leave the subdomain settings at their defaults.

### Edge Settings

- 1 From the **Physics** menu, open the **Edge Settings** dialog box. Select Edge 16 and type 1 in the **Magnetic current in edge segment direction** edit field.
- 2 Click **OK**.

The last step defines the magnetic current in the outer loop. The current flows in the direction of the edge segment, so it is important to know the direction to determine the sign of the magnetic current value.

## MESH GENERATION

The boundary mesh settings below are made to handle the small dimensions of the antenna compared to the rest of the geometry. The global maximum mesh element size is set to resolve the wavelength properly.

- 1 Open the **Free Mesh Parameters** dialog box from the **Mesh** menu, and select **Coarser** from the **Predefined mesh sizes** list.
- 2 Click the **Custom mesh size** option button and type 0.07 in the **Maximum element size** edit field.
- 3 Click the **Boundary** tab, and select Boundaries 4, 5, 7, and 8. Type 0.2 in the **Mesh curvature factor** edit field, and type 0.01 in the **Mesh curvature cutoff** edit field.
- 4 Click the **Advanced** tab and enter 0.75 in the **z-direction scale factor** edit field. This ensures that the small elements around the antenna become stretched a bit along the antenna, reducing the total number of mesh elements efficiently.
- 5 Click the **Remesh** button and then click **OK**.

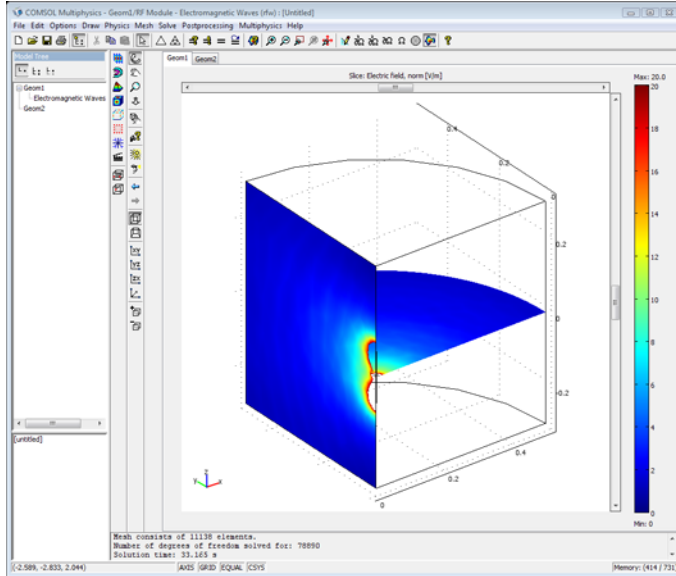
## COMPUTING THE SOLUTION

- 1 Open the **Solver Parameters** dialog box from the **Solve** menu.
- 2 Select **Geometric multigrid** from the **Linear system solver** list.
- 3 Click the **Settings** button. In the dialog box that appears, make sure that **Linear system solver** is selected in the field to the left, and choose **Lower element order first** from the **Hierarchy generation method** list.
- 4 All other settings can be left at their default values. For details on the default settings, see “Solving Large 3D Problems” on page 69 of the *RF Module User’s Guide*.
- 5 Click **OK** to close the **Linear System Solver Settings** dialog.
- 6 Click the **Stationary** tab, and then select **Linear** in the **Linearity** list. This model is linear, but the solver interprets the model as nonlinear, because of the use of integration coupling variables.
- 7 Click **OK**.
- 8 Click the **Solve** button on the Main toolbar.  
The solver takes a few minutes to complete the analysis.

## POSTPROCESSING AND VISUALIZATION

- 1 Open the **Plot Parameters** dialog box from the **Postprocessing** menu.
- 2 On the **Slice** tab, choose **Electric field, norm** from the list of **Predefined quantities**.
- 3 Set the **z levels** to 1 in the **Slice positioning** area. For the **x levels** select **Vector with coordinates** and type 0 in the corresponding edit field.
- 4 Click the **Range** button, clear the **Auto** check box, and type 0 and 20 in the **Min** and **Max** edit fields, respectively. Click **OK**.

5 Click **OK** to close the dialog box and make the following plot.



6 To calculate the impedance, choose **Postprocessing>Data Display>Global**. In the dialog box, enter  $Z$  in the **Expression** edit field. The impedance is close to  $78+12i$ .

# Microstrip on a Printed Circuit Board

This example calculates the impedance of a microstrip line on a printed circuit board. It compares the impedance with that obtained from an analytical formula given the dimensions of the micro strip and permittivity of the circuit board. This formula is available as a script with the following arguments: width, height, height of conductor, and permittivity of circuit board. The following command gives the analytical characteristic impedance of the microstrip:

```
zmustrip(2e-2,5e-3,5e-4,3)
```

```
35.4780
```

The analysis tests two terminations: lumped port termination and numerical port termination. Both versions use a lumped port with  $50\ \Omega$  cable impedance to excite the signal into the microstrip. The numerical port version does a boundary mode analysis on the terminating end, which results in almost no reflection from the termination of the microstrip. The lumped port termination uses the calculated impedance as cable impedance. This matching is less accurate and give reflections, which interfere with the impedance calculation and therefore deviates from the analytical result. Table 2-1 below summarizes the result.

TABLE 2-1: RESULT OF THE IMPEDANCE COMPARISON

ANALYSIS	MICROSTRIP IMPEDANCE
Analytical	35.48 $\Omega$
Numerical port termination	36.39 $\Omega$
Lumped port termination	39.83 $\Omega$

Note that the analytical result does not represent the most accurate value because it assumes infinite extension of the surroundings. This model is terminated by perfect magnetic conductors, which influence the impedance of the microstrip. It is clear that the numerical port has much less reflection compared to the lumped port. This is important to consider when lumped ports are used as terminations.

---

**Model Library path:** RF\_Module/Tutorial\_Models/microstrip\_on\_pcb

---

## MODEL NAVIGATOR

- 1 In the **Model Navigator**, choose **3D** from the **Space dimension** list, and select **RF Module>Electromagnetic Waves**.
- 2 Click **OK**.
- 3 From the **Multiphysics** menu, choose **Model Navigator** to open the same dialog box again.
- 4 Select **RF Module>Boundary Mode Analysis>Hybrid-Mode Waves**, and click the **Add** button.
- 5 Choose **Boundary Mode Analysis, Hybrid-Mode Waves (rfwb)** from the **Ruling application mode** list.
- 6 Click **OK**.

The ruling application mode determines the default settings for the solver and how you can specify the eigenvalues in the **Solver Parameters** dialog box. By reopening the **Model Navigator** dialog box, you can get the default settings for the iterative solvers used by the electromagnetic waves application mode and the more user-friendly eigenvalue specifications of the boundary mode analysis application mode.

## GEOMETRY MODELING

- 1 Draw four blocks by clicking the **Block** button on the Draw toolbar. Specify the following dimensions for the blocks. For each block, click **OK** when done.

PROPERTY	BLK1	BLK2	BLK3	BLK4
X length	0.3	0.2	0.2	0.2
Y length	0.3	0.3	2E-2	2E-2
Z length	0.15	5E-3	5E-3	5E-4
X base	0	0.1	0.1	0.1
Y base	0	0	0.14	0.14
Z base	0	0	0	5E-3

- 2 Select BLK1 and BLK4 and click the **Difference** toolbar button on the Draw toolbar.

## PHYSICS SETTINGS (BOUNDARY MODE ANALYSIS)

### *Boundary Conditions*

- 1 From the **Physics** menu, choose **Boundary Settings**.

- 2 Select Boundaries 23, 25, and 26, select the **Specify material parameters in terms of  $\epsilon_r$ ,  $\mu_r$ , and  $\sigma$**  option button, and then enter 3 in the  $\epsilon_r$  edit field.
- 3 Select Boundaries 1–22, and clear the **Active in this domain** check box.
- 4 Click **OK**.

#### *Edge Settings*

- 1 From the **Physics** menu, choose **Edge Settings**.
- 2 Select Edges 32, 34, 36, 46, and 47, then click the **Perfect magnetic conductor** button.
- 3 Select Edges 33, 38, 39, 40, 41, 43, and 44, and click the **Perfect electric conductor** button.
- 4 Click **OK**.

#### **MESH GENERATION**

- 1 From the **Mesh** menu, open the **Free Mesh Parameters** dialog box.
- 2 From the **Predefined mesh sizes** list, choose **Coarser**.
- 3 Click **OK**.
- 4 Click the **Initialize Mesh** button on the Main toolbar.

#### **COMPUTING THE SOLUTION**

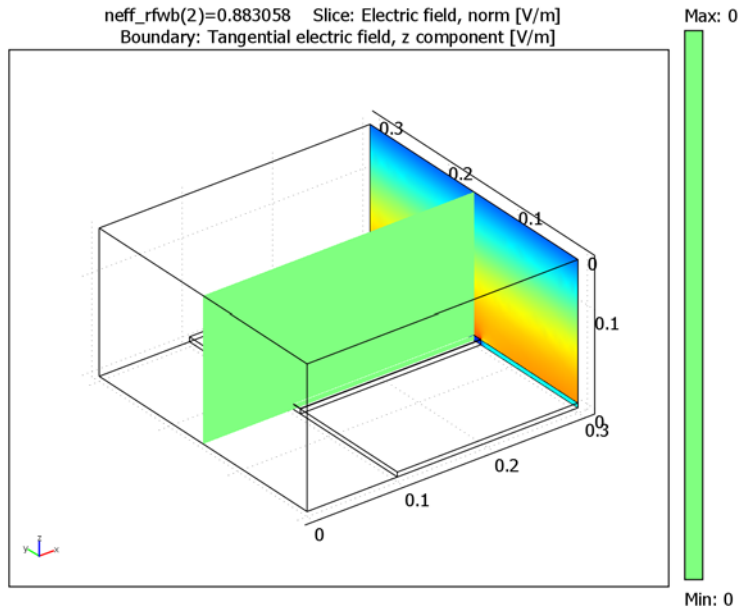
It is important that you solve the boundary mode analysis first because it gives the mode and propagation constant for the numerical port. The boundary mode analysis is an eigenvalue simulation where the propagation constant is an eigenvalue. The second step solves a stationary problem for the wave propagation, so you must switch analysis type between the solver steps. It is also important to store the eigenvalue before you start the second step, especially if the boundary mode analysis gives several possible propagation constants (eigenvalues).

- 1 From the **Solve** menu, open the **Solver Manager** dialog box.
- 2 Click the **Solve For** tab, and select **Boundary Mode Analysis, Hybrid-Mode Waves (rfwb)** from the **Solve for variables** list.
- 3 Click **OK**.
- 4 From the **Solve** menu, open the **Solver Parameters** dialog box.
- 5 From the **Solver** list, choose **Eigenvalue**.
- 6 Enter 0.9 in the **Search for effective mode indices around** edit field.
- 7 Enter 2 in the **Desired number of effective mode indices** edit field.

- 8 From the **Linear system solver** list, choose **Direct (UMFPACK)**, then click **OK**.
- 9 Click the **Solve** button on the Main toolbar.

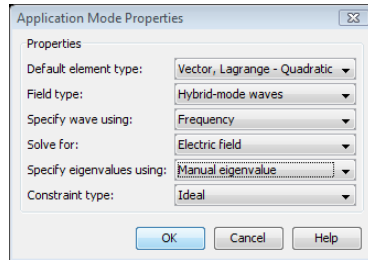
### POSTPROCESSING AND VISUALIZATION

- 1 From the **Postprocessing** menu, open the **Plot Parameters** dialog box.
- 2 Click the **Boundary** tab and select the **Boundary plot** check box. From the **Predefined quantities** list, select **Boundary Mode Analysis, Hybrid-Mode Waves (rfwb)>Tangential electric field, z component**.
- 3 Clear the **Color scale** check box.
- 4 Click the **Slice** tab, then choose **Electromagnetic Waves (rfw)>Electric field, norm** from the **Predefined quantities** list, enter 0 in the **x levels** edit field and 1 in the **y levels** edit field.
- 5 Click the **General** tab, and select the solution with an effective mode index of 0.88 from the **Effective mode index** list.
- 6 Click **OK** to see the following plot. You have not solved for the electric field on the slice yet, so it just shows up as a zero solution. In the subsequent wave propagation simulation you use the plotted boundary mode analysis solution for the numerical port.



## FREEZING THE BOUNDARY MODE ANALYSIS APPLICATION MODE

- 1 In the **Physics** menu, select **Properties** to open the **Application Mode Properties** dialog box.
- 2 From the **Specify eigenvalues using** list, choose **Manual eigenvalue**.



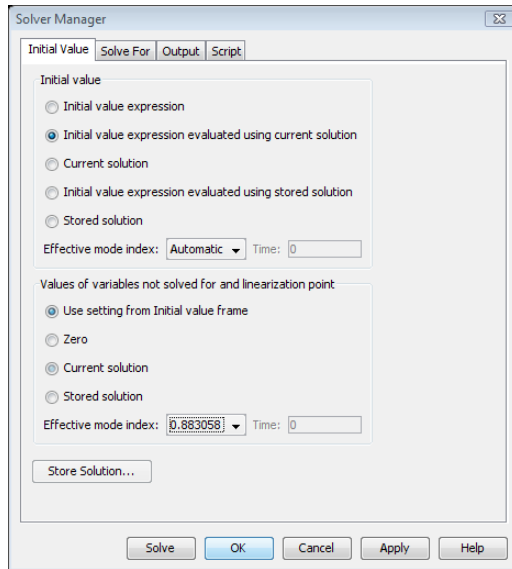
- 3 Click **OK**.

When you perform this operation, you copy the propagation constant for the current plot to an application scalar variable for the propagation constant. All the postprocessing variables now use this propagation constant instead of calculating them from the eigenvalues. The application mode is in a state where it makes no sense to solve for eigenvalues. So before re-solving this application mode again, switch back to the original setting in the **Application Mode Properties** dialog box.

- 4 From the **Solve** menu, open the **Solver Manager** dialog box.
- 5 Click the **Solve For** tab, and choose only the **Electromagnetic Waves (rfw)** application mode.
- 6 Click the **Initial Value** tab, and in the **Values of variables not solved for and linearization point** area, click the **Current solution** option button.



- 7 From the **Effective mode index** list, choose same effective mode index as you plotted, **0.88**. It is important that this mode index matches the propagation constant stored in the scalar variables.



- 8 Click **OK**. You are now ready to continue with the setup of the electromagnetic waves application mode.

## PHYSICS SETTINGS (ELECTROMAGNETIC WAVES)

From the **Multiphysics** menu, choose **Electromagnetic Waves (rfw)**.

### *Subdomain Settings*

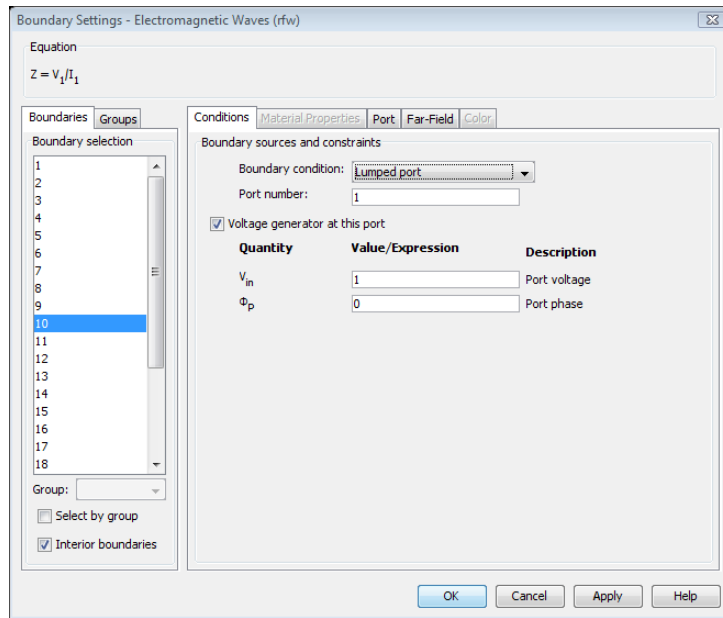
- 1 Open the **Subdomain Settings** dialog box.
- 2 Select Subdomains 2, 3, and 4, then enter 3 in the  $\epsilon_r$  edit field.
- 3 Click **OK**.

### *Boundary Conditions*

- 1 From the **Physics** menu, choose **Boundary Settings**.
- 2 Select the **Interior boundaries** check box.

- 3 On the **Conditions** page, set the values according to the following table. Leave all values not shown at their defaults. Note that the controls do not show up until you choose the boundary condition.

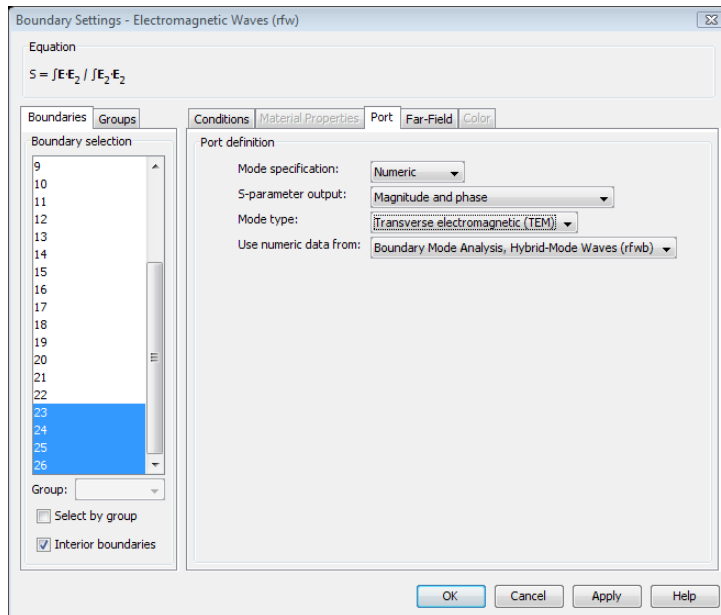
SETTINGS	BOUNDARY 10	BOUNDARIES 23–26	BOUNDARIES 2, 4, 5, 7, 22
Boundary condition	Lumped port	Port	Perfect magnetic conductor
Port number	1	2	
Voltage generator at this port	Selected		
Wave excitation at this port		Cleared	



- 4 Select Boundary 1 and choose **Scattering boundary condition** from the **Boundary condition** list.

5 On the **Port** page, set the values according to the table below:

SETTINGS	BOUNDARIES 23–26
Mode specification	Numeric
Mode type	Transverse electromagnetic (TEM)
Use numeric data from	Boundary Mode Analysis, Hybrid-Mode Waves (rfwb)

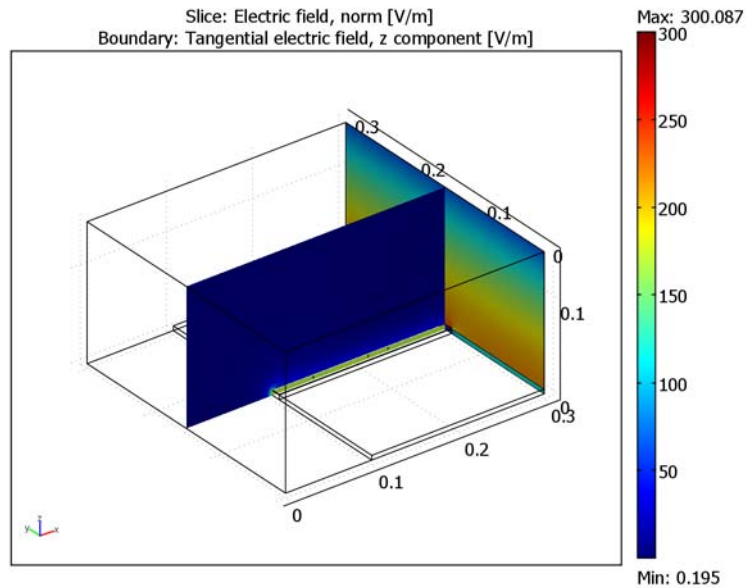


6 Click **OK**.

## COMPUTING THE SOLUTION

- 1 From the **Solve** menu, open the **Solver Parameters** dialog box.
- 2 Switch to the **Stationary** solver and choose **GMRES** from the **Linear system solver** list. All the other settings for this iterative solver are properly setup for an electromagnetic wave calculation because of what you did under the section “Model Navigator” on page 21.
- 3 Click **OK**, and then click the **Solve** button on the Main toolbar to solve the problem.

When the solver finishes, the result looks like the following figure.



#### *Extracting the Microstrip Impedance*

- 1 From the **Postprocessing** menu, choose **Data Display>Global**.
- 2 In the dialog box that appears, choose **Port impedance, port 1** from the **Predefined quantities** list.
- 3 Click **OK**. The result shows up in the message log and is close to  $36.39 \Omega$ .

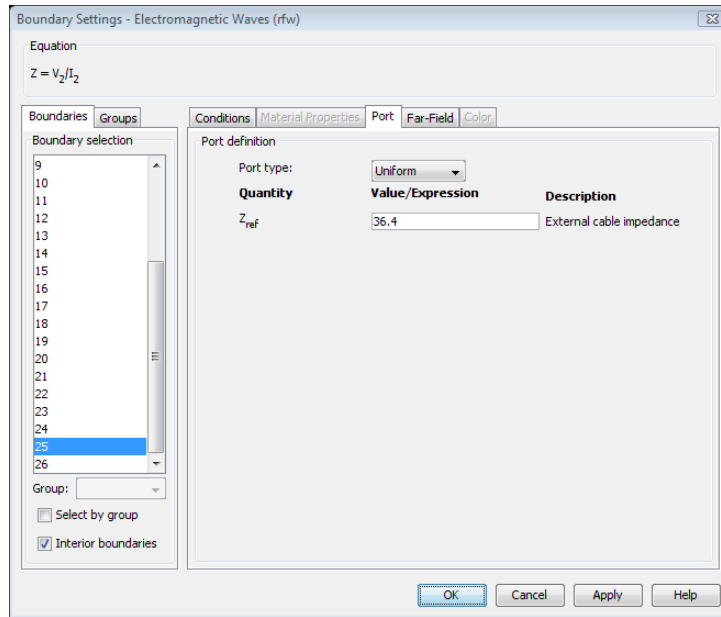
#### *Solving with Lumped Port Termination*

---

##### **PHYSICS SETTINGS (ELECTROMAGNETIC WAVES)**

- 1 From the **Physics** menu, open the **Boundary Settings** dialog box.
- 2 Click the **Conditions** tab, select boundaries 23, 24, and 26, and choose **Perfect magnetic conductor** from the **Boundary condition** list.
- 3 Select Boundary 25, and choose the **Lumped port** boundary condition.
- 4 Enter 2 in the **Port number** edit field.

- 5 On the **Port** page, enter 36.4 in the  $Z_{ref}$  edit field. This is the value you extracted in the previous section.



- 6 Click **OK**.

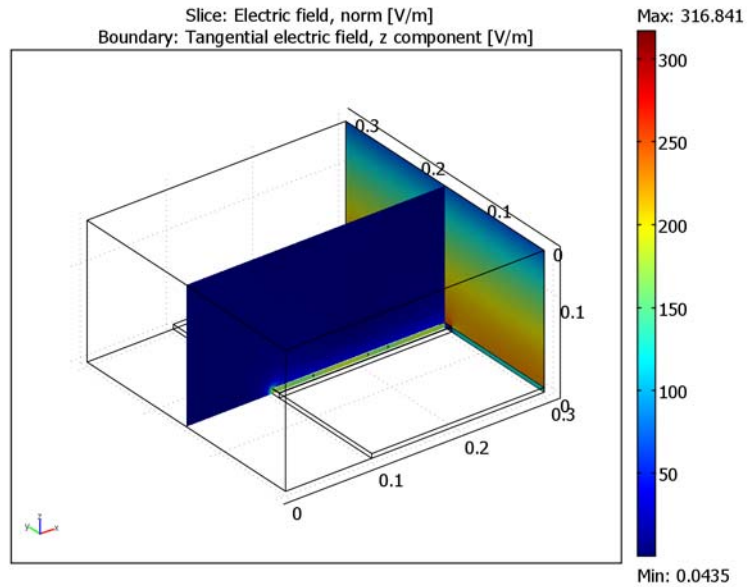
## COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar to solve the model again.

### *Extracting the Microstrip Impedance*

- 1 From the **Postprocessing** menu, choose **Data Display>Global**.
- 2 In the dialog box that appears, pull down the **Predefined quantities** list and choose **Port impedance, port 1**.
- 3 Click **OK**.

The result shows up in the message log and is roughly  $39.83 \Omega$ . This is a little bit different from the previous result, and it is also possible to see the small standing wave shape in the slice plot.



## RF and Microwave Models

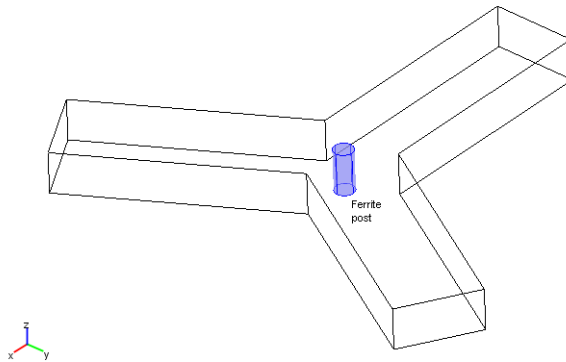
In this chapter, you find models within RF and microwave engineering. From the mathematical viewpoint, the same physics formulations are used in the chapter on optics and photonics. The terminology and the way numerical results are presented differ slightly, but anyone interested in RF and microwave engineering may benefit from also reading that chapter.

# Three-Port Ferrite Circulator

## *Introduction*

---

A microwave circulator is a multiport device that has the property that a wave incident in port 1 is coupled into port 2 only, a wave incident in port 2 is coupled into port 3 only, and so on. Circulators are used to isolate microwave components to couple a transmitter and a receiver to a common antenna, for example. They typically rely on the use of anisotropic materials, most commonly ferrites. In this example, a three-port circulator is constructed from three rectangular waveguide sections joining at  $120^\circ$  where a ferrite post is inserted at the center of the joint. Figure 3-1 shows the geometry of the circulator.



*Figure 3-1: Geometry of the three-port microwave circulator.*

To match the junction, identical dielectric tuning elements are inserted into each branch (not shown above). The ferrite post is magnetized by a static  $H_0$  bias field along the axis. The bias field is usually supplied by external permanent magnets. Here, the focus is on the modeling of the ferrite and how to minimize reflections at the inport by matching the junction by the proper choice of tuning elements. For a general introduction to the modeling of rectangular waveguide structures, see the Waveguide H-bend model on page 21 in the *RF Module User's Guide* and the corresponding 2D



model found on page 80 in this document. Matching the circulator junction involves calculating how well a  $TE_{10}$  wave propagates between ports in the circulator for different materials in the tuning element. This is done by calculating the scattering parameters, or S-parameters, of the structure as a function of the permittivity of the tuning elements for the fundamental  $TE_{10}$  mode. The S-parameters are a measure of the transmittance and reflectance of the circulator. For a theoretical background on S-parameters, see the section “S-Parameters and Ports” on page 56 in the *RF Module User’s Guide*.

This model only includes the  $TE_{10}$  mode of the waveguide. Thus the model can be made in 2D as the fields of the  $TE_{10}$  mode have no variation in the transverse direction. Figure 3-2 shows the 2D geometry including the dielectric tuning elements.

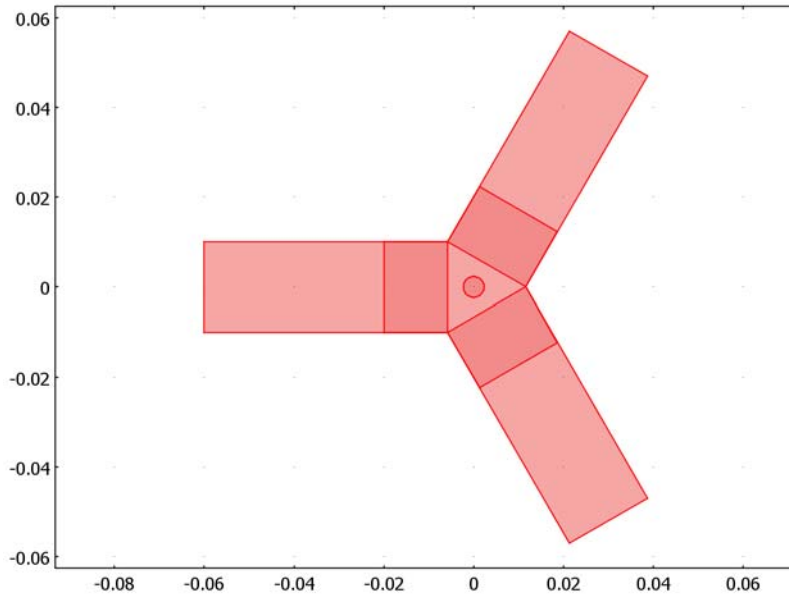


Figure 3-2: 2D geometry with dielectric tuning elements.

### Model Definition

The dependent variable in this application mode is the  $z$ -component of the electric field  $\mathbf{E}$ . It obeys the following relation:

$$\nabla \times (\mu_r^{-1} \nabla \times E_z) - \left( \epsilon_r - \frac{j\sigma}{\omega\epsilon_0} \right) k_0^2 E_z = 0$$

where  $\mu_r$  denotes the relative permeability,  $\omega$  the angular frequency,  $\sigma$  the conductivity,  $\epsilon_0$  the permittivity of vacuum,  $\epsilon_r$  the relative permittivity, and  $k_0$  is the free space wave number. Losses are neglected so the conductivity is zero everywhere. The magnetic permeability is of key importance in this model as it is the anisotropy of this parameter that is responsible for the nonreciprocal behavior of the circulator. For the theory of the magnetic properties of ferrites, see Ref. 1 and Ref. 2. The model assumes that the static magnetic bias field,  $H_0$ , is much stronger than the alternating magnetic field of the microwaves, so the quoted results are a linearization for a small-signal analysis around this operating point. Further assume that the applied magnetic bias field is strong enough for the ferrite to be in magnetic saturation. Under these assumptions and neglecting losses, the anisotropic permeability of a ferrite magnetized in the positive  $z$  direction is given by:

$$[\mu] = \begin{bmatrix} \mu & j\kappa & 0 \\ -j\kappa & \mu & 0 \\ 0 & 0 & \mu_0 \end{bmatrix}$$

where

$$\kappa = \mu_0 \left( \frac{\omega \omega_m}{\omega_0^2 - \omega^2} \right)$$

$$\mu = \mu_0 \left( 1 + \frac{\omega_0 \omega_m}{\omega_0^2 - \omega^2} \right)$$

and

$$\omega_0 = \mu_0 \gamma H_0$$

$$\omega_m = \mu_0 \gamma M_s$$

Here  $\mu_0$  denotes the permeability of free space;  $\omega$  is the angular frequency of the microwave field;  $\omega_0$  is the precession frequency or Larmor frequency of a spinning electron in the applied magnetic bias field,  $H_0$ ;  $\omega_m$  is the electron Larmor frequency at the saturation magnetization of the ferrite,  $M_s$ ; and  $\gamma$  is the gyromagnetic ratio of the electron. For a lossless ferrite, the permeability clearly becomes unbounded at  $\omega = \omega_0$ . In a real ferrite, this resonance becomes finite and is broadened due to losses. For complete expressions including losses, see Ref. 1 and Ref. 2. In this analysis the operating frequency is chosen sufficiently off from the Larmor frequency to avoid the

singularity. The material data,  $M_s = 2.39 \cdot 10^5$  A/m and  $\epsilon_r = 12.9$ , are taken for magnesium ferrite from Ref. 2. The applied bias field is set to  $H_0 = 2.72 \cdot 10^5$  A/m, which is well above saturation. The electron gyromagnetic ratio is set to  $1.759 \cdot 10^{11}$  C/kg. Finally, the model uses an operating frequency of 10 GHz. This is well above the cutoff for the TE<sub>10</sub> mode, which for a waveguide cross section of 2 cm by 1 cm is at about 7.5 GHz. At the ports, matched port boundary conditions make the boundaries transparent to the wave.

### Results and Discussion

The  $S_{11}$  parameter as a function of the relative permittivity of the matching elements,  $\epsilon_{ps\_r}$ , is shown in Figure 3-3. The  $S_{11}$  parameter corresponds to the reflection coefficient at port 1. Thus matching the junction is equivalent to minimizing the magnitude of  $S_{11}$ .

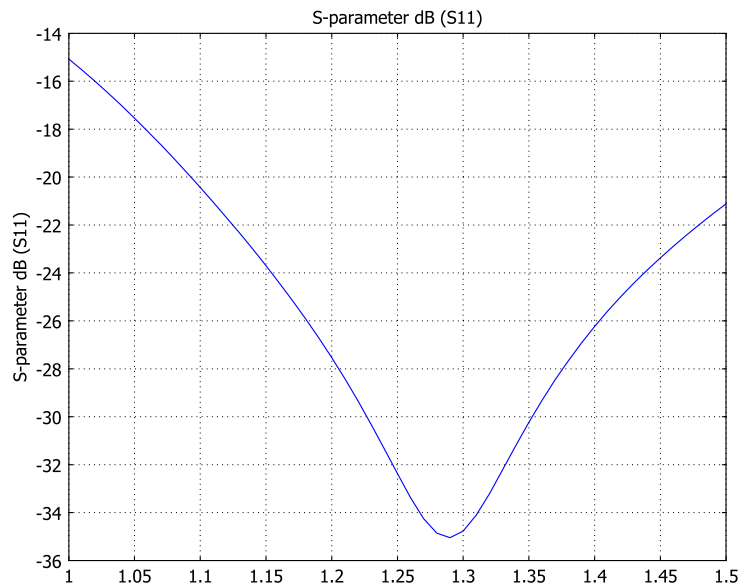
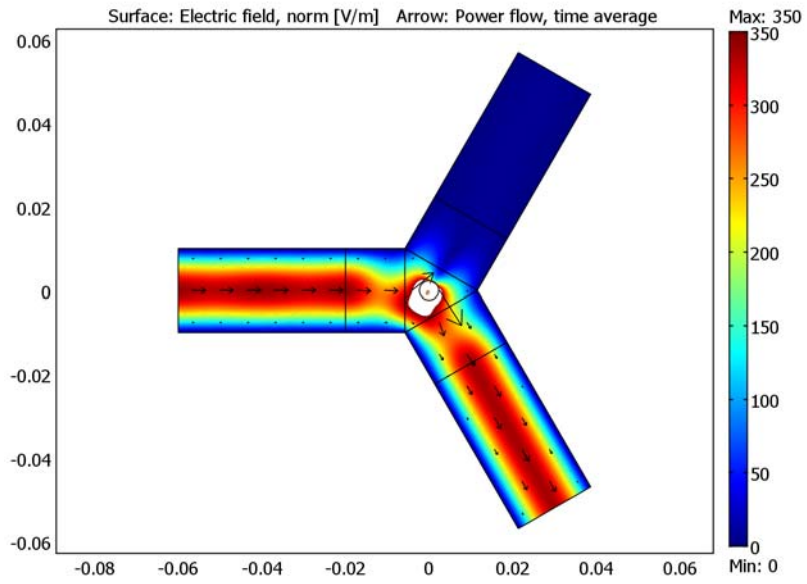


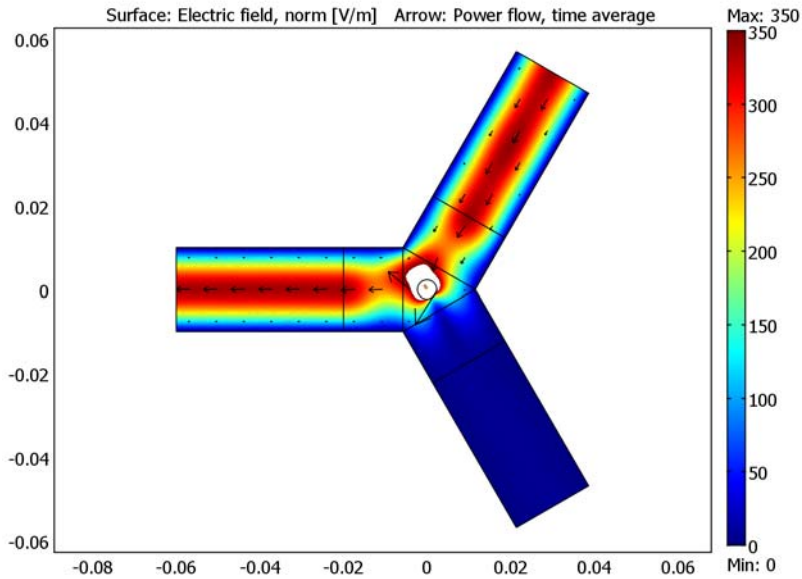
Figure 3-3:  $S_{11}$  parameter as a function of the relative permittivity.

By choosing  $\epsilon_{ps\_r}$  to about 1.28, you obtain a reflection coefficient of about  $-35$  dB, which is a good value for a circulator design. Judging from the absence of standing wave patterns in the magnitude plot of the electric field and by looking at the direction

of the microwave energy flow in the result plot below, it is clear that the circulator behaves as desired.



By feeding the circulator at a different port, the nonreciprocal behavior of the device becomes quite apparent.



As a final study, extract the full S-parameter matrix at the optimum value of  $\epsilon_{ps\_r}$  for a range of frequencies. The script `sparametermatrix.m` automatically uses all ports, one at a time, as inputs and extracts the matrix. The matrix contains all the information about the relation between the ports.

$$\mathbf{S} = \begin{bmatrix} -34.8 & -0.003 & -34.9 \\ -35.0 & -34.8 & -0.003 \\ -0.003 & -34.9 & -34.8 \end{bmatrix}$$

If you feed the signal through, say, Port 2, check column 2, and you see that it is passed on to Port 1 where you find the highest value. The calculated S-parameter data are also exported to a Touchstone file using the command `touchwrite`.

### References

1. R.E. Collin, *Foundations for Microwave Engineering*, 2nd ed., IEEE Press/Wiley-Interscience.
2. D.M. Pozar, *Microwave Engineering*, 3rd ed., John Wiley & Sons Inc.

---

**Model Library path:** RF\_Module/RF\_and\_Microwave\_Engineering/circulator

---

## *Modeling Using the Graphical User Interface*

---

### MODEL NAVIGATOR

- 1 From the **Space dimension** list, select **2D**.
- 2 From the **Application Modes** folder, select  
**RF Module>In-Plane Waves>TE Waves>Harmonic propagation**.
- 3 Click **OK**.

### OPTIONS AND SETTINGS

- 1 Define the following constants in the **Constants** dialog box. The description field is optional and can be omitted.

NAME	EXPRESSION	DESCRIPTION
freq	300[MHz]/0.03	Frequency
eps_r	1	Relative permittivity in matching layer

- 2 Define the following scalar variables in the **Scalar Expressions** dialog box.

NAME	EXPRESSION	DESCRIPTION
gamma	1.759e11[C/kg]	Gyromagnetic ratio
H0	$\omega_{rfwe}/(\gamma * \mu_{0\_rfwe} + 1e4[m/C])$	Applied magnetic bias field
w0	$\mu_{0\_rfwe} * \gamma * H0$	Larmor frequency
Ms	0.3[Wb/m <sup>2</sup> ]/ $\mu_{0\_rfwe}$	Saturation magnetization
wm	$\mu_{0\_rfwe} * \gamma * Ms$	Larmor frequency at saturation limit
mur	$1 + w0 * wm / (w0^2 - \omega_{rfwe}^2)$	Relative permeability tensor element
kr	$\omega_{rfwe} * wm / (w0^2 - \omega_{rfwe}^2)$	Relative permeability tensor element

### GEOMETRY MODELING

The geometry modeling of this model is rather extensive, so it is possible to import the circulator geometry from a binary file. Select the section that you prefer.

### *Importing the Geometry from a Binary File*

- 1 From the **File** menu, select **Import>CAD Data From File**.
- 2 In the **Import CAD Data From File** dialog box, make sure that the **COMSOL Multiphysics file** or **All 2D CAD files** is selected in the **Files of type** list.
- 3 From the `models` directory under the COMSOL Multiphysics installation directory, go to the model library path given on page 38. Select the `circulator.mphbin` file, and click **Import**.
- 4 Skip section “Creating the Geometry from Scratch” and begin at section “Scalar Variables”.

### *Creating the Geometry from Scratch*

Use the buttons on the Draw toolbar and the **Draw** and **Edit** menus.

- 1 Draw a **Rectangle** with the lower left corner in  $(-0.06, -0.01)$  and the width 0.06 and height 0.02.
- 2 **Copy** the rectangle and paste it without any displacement.
- 3 **Rotate** the pasted rectangle  $120^\circ$  around  $(0,0)$ .
- 4 **Paste** a new copy of the original rectangle without any displacement.
- 5 **Rotate** the pasted rectangle  $-120^\circ$  around  $(0, 0)$ .
- 6 Select all objects and click the **Union** button in the toolbar.
- 7 Click the **Delete Interior Boundaries** button in the toolbar.
- 8 Draw another **Rectangle** with the lower left corner in  $(-0.02, -0.01)$  and the width 0.0142264973081037 and height 0.02. You must enter the width to all decimal places, otherwise the two right corners of the rectangle do not connect to the corners in the previously created object. An alternative way to define this rectangle is: Go to the **Draw** menu and select **Draw Objects>Rectangle/Square**. Point the mouse to  $(-0.02, -0.01)$ . Click the left mouse button and keep pressing it down while you move the pointer diagonally up and to the right. When it eventually snaps to the wanted corner in the existing object, release the mouse button.
- 9 Repeat Steps 2–5 above for the latest rectangle.
- 10 Draw a **Circle** with a radius of 0.0023 centered at the origin.

### **SCALAR VARIABLES**

- 1 From the **Physics** menu, select **Scalar Variables**.
- 2 In the **Application Scalar Variables** dialog box, set the **Frequency** to `freq`.
- 3 Click **OK**.

## PHYSICS SETTINGS

### *Boundary Conditions*

The default boundary condition is perfect electric conductor which is fine for all exterior boundaries except at the ports. Interior boundaries are by default not active meaning that continuity is imposed.

At boundaries number 1, 20, and 21, specify the **Port** boundary condition. On the **Port** tab set the values according to the table below:

SETTINGS	BOUNDARY 1	BOUNDARY 20	BOUNDARY 21
Port number	1	3	2
Wave excitation at this port	Selected	Cleared	Cleared
Mode specification	Analytic	Analytic	Analytic
Mode number	1	1	1

### *Subdomain Settings*

- 1 Click the **Groups** tab in the **Subdomain Settings** dialog box and create three groups called air, dielectric, and ferrite, respectively.
- 2 Keep the default subdomain settings for the air group. For the dielectric group, change the **Relative permittivity** to `eps_r`. For the ferrite group, set the **Relative permittivity** to 12.9, and enter the **Relative permeability**, as four space-separated entries in this edit field: `mur -i*kr i*kr mur`.
- 3 On the **Subdomains** page, assign Subdomains 1, 3, 6, and 7 to the **air** group; Subdomains 2, 4, and 5 to the **dielectric** group; and Subdomain 8 to the **ferrite** group.
- 4 Click **OK**.

## MESH GENERATION

- 1 From the **Mesh** menu, open the **Free Mesh Parameters** dialog box.
- 2 On the **Subdomain** page, set the **Maximum element size** to 0.003 for Subdomains 1–7 and to 0.0005 for Subdomain 8. This will ensure that the waves are properly resolved everywhere. Especially in the ferrite, the wavelength will be much shorter than in free space, calling for a much denser mesh here.
- 3 Click **OK** to close **Free Mesh Parameters** dialog box.
- 4 Click the **Initialize Mesh** button on the Main toolbar.



### COMPUTING THE SOLUTION

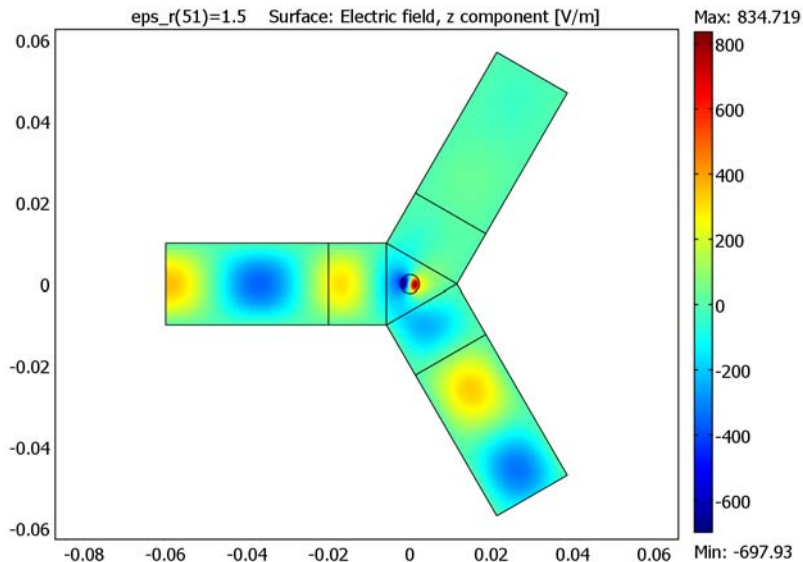
Use the parametric solver in order to study the S-parameters as a function of the relative permittivity in the matching elements.

- 1 Click the **Solver Parameters** button on the Main toolbar.
- 2 From the **Solver** list, select **Parametric**.
- 3 Enter `eps_r` as **Parameter name** and `linspace(1, 1.5, 51)` as **Parameter values**. The solver will calculate the solution using 51 equidistant permittivity values in the range 1 to 1.5. Click **OK**.
- 4 Click the **Solve** button on the Main toolbar.

The solution time is about 30 s on a PC with a 2.2 GHz P4 processor running Windows XP.

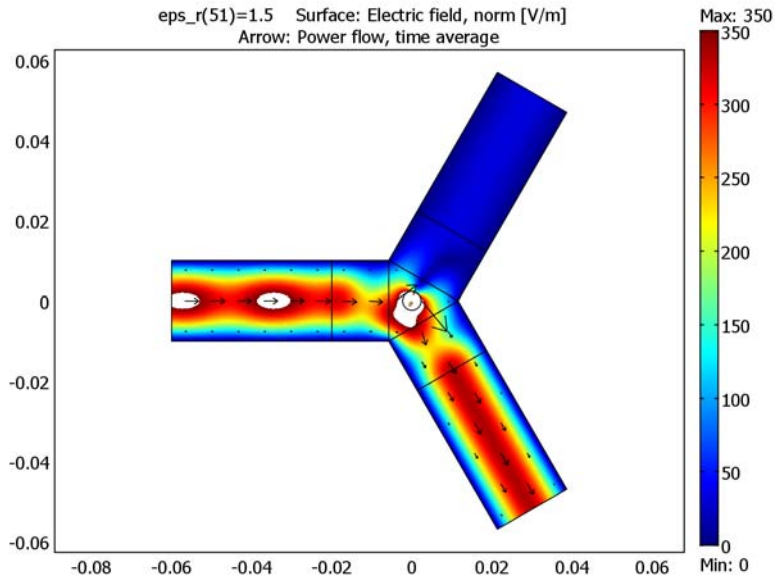
### POSTPROCESSING AND VISUALIZATION

The default plot shows the instantaneous electric field at the final solution step; see the following figure.



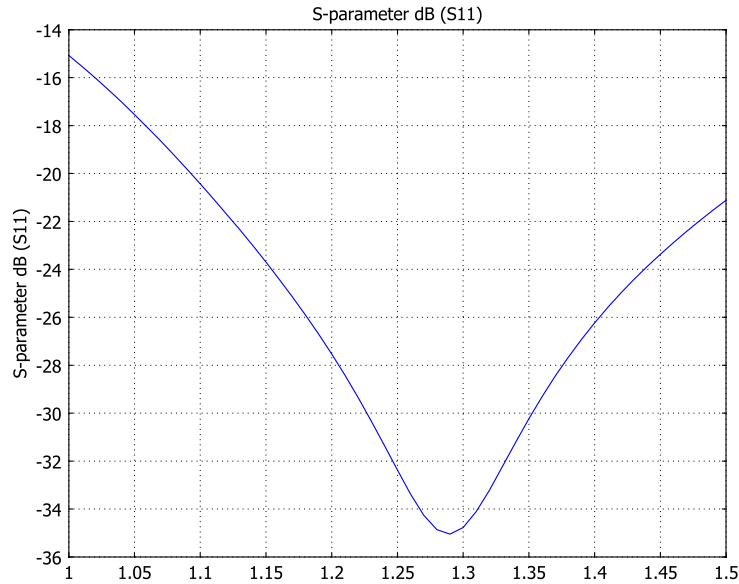
It is more instructive to look at the magnitude of the electric field and to set the color range manually as the plot otherwise will be dominated by the strong electric field in the ferrite post:

- 1 Click the **Plot Parameters** button on the Main toolbar.
- 2 On the **Surface** page, select **Electric field, norm** from the list of **Predefined quantities**.
- 3 Click the **Range** button. Specify a range from 0 to 350, then click **OK**.
- 4 On the **Arrow** page, select the **Arrow plot** check box.
- 5 From the list of **Predefined quantities**, select **Power flow, time average**.
- 6 Set the arrow **Color** to black, then click **OK**.



The presence of standing waves at the input is clearly visible for this value (1.5) of the relative permittivity in the matching elements. The next step is to plot the reflection coefficient  $S_{11}$  as a function of  $\epsilon_{ps\_r}$ . You can do this using a domain plot.

- 1 From the **Postprocessing** menu, select **Domain Plot Parameters**.
- 2 On the **General** page, select all solutions in the **Solutions to use** list.
- 3 On the **Point** page, select **S\_parameter dB(S11)** from the list of **Predefined quantities**.  
 $S_{11}$  is defined everywhere, but you only need to evaluate it at a single point to make a plot as a function of  $\epsilon_{ps\_r}$ .
- 4 Select Point 1, then click **Apply** to plot  $S_{11}$ .




---

**Note:** When displaying S-Parameter variables, the phase of the solution must be set to zero. Otherwise, incorrect values will be displayed. In the various dialogs for displaying and plotting data that are available from the **Postprocessing** menu, the phase of the solution can be set via the **Solution at angle (phase)** field. If the dialog box contains several tabs, it usually appears on the **General** tab.

---

The plot shows that  $S_{11}$  is minimized by setting  $\epsilon_r$  to 1.28. Make this change in the **Constants** dialog box, open the **Solver Parameters** dialog, select the **Stationary** solver, and solve again. Any standing wave pattern should barely be visible now. Change the inport from Port 1 to Port 2 by editing the boundary settings for these boundaries and solve again to see the nonreciprocal behavior of the device.

### *Extracting the S-Parameter Matrix*

---

This section requires that you have access to COMSOL Script or MATLAB.

- I From the **File** menu, choose **Export>FEM Structure as 'fem'**. Switch to the **COMSOL Script** command window.

- 2 Enter the following command at the prompt.

```
S = sparametermatrix(fem, 'scale', 'db')
```

```
S =
```

```
-34.7786    -0.0028    -34.9297  
-34.9500   -34.7802    -0.0028  
-0.0028   -34.9280   -34.7998
```

The command gives the S-parameters in dB scale.

### *Exporting S-Parameters versus Frequency to Touchstone*

---

This section requires that you have access to COMSOL Script or MATLAB.

- 1 If you continue from the previous section it is not necessary to export the `fem` structure. Otherwise, do step 1 from that section.
- 2 Enter the following commands at the prompt:

```
freq = linspace(1e10,1.5e10,25);  
S = sparametermatrix(fem, 'pname', 'freq', 'plist', freq);  
touchwrite('circulator', freq, S, 'frequnit', 'MHz');
```

The first command creates a list of frequencies to export, the second command calculates the S-parameter matrix for all those frequencies, and the final command exports the data to a file named `circulator.s3p`. If you do not provide a path for the file, the `touchwrite` command saves the file in the current directory. It is Touchstone convention to put the number 3 in the filename extension because the model has three ports. The `frequnit` option specifies that the frequency is given in MHz in the exported file. The format for the S-parameters in the file is magnitude and angle, but you can change to another format by using the options specified in Chapter 4, “Function Reference,” in the *RF Module Reference Guide*.

# Monoconical RF Antenna

## *Introduction*

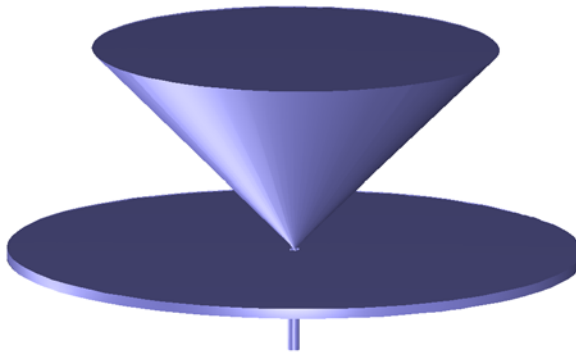
---

Conical antennas are useful for many applications due to their broadband characteristics and relative simplicity. This example includes an analysis of the antenna impedance and the radiation pattern as functions of the frequency for a monoconical antenna with a finite ground plane and a  $50\ \Omega$  coaxial feed. The rotational symmetry makes it possible to model this in 2D using one of the axisymmetric electromagnetic wave propagation application modes. When modeling in 2D, you can use a dense mesh, giving an excellent accuracy for a wide range of frequencies.

## *Model Definition*

---

The antenna geometry consists of a 0.2 m tall metallic cone with a top angle of 90 degrees on a finite ground plane of a 0.282 m radius. The coaxial feed has a central conductor of 1.5 mm radius and an outer conductor (screen) of 4.916 mm radius separated by a teflon dielectric of relative permittivity of 2.07. The central conductor of the coaxial cable is connected to the cone, and the screen is connected to the ground plane.



*Figure 3-4: The geometry of the antenna. The central conductor of the coaxial cable is connected to the metallic cone, and the cable screen is connected to the finite ground plane.*

The model takes advantage of the rotational symmetry of the problem, which allows modeling in 2D using cylindrical coordinates. You can then use a very fine mesh to achieve an excellent accuracy.

## DOMAIN EQUATIONS

An electromagnetic wave propagating in a coaxial cable is characterized by transverse electromagnetic fields (TEM). Assuming time-harmonic fields with complex amplitudes containing the phase information, you have:

$$\mathbf{E} = \mathbf{e}_r \frac{C}{r} e^{j(\omega t - kz)}$$
$$\mathbf{H} = \mathbf{e}_\phi \frac{C}{rZ} e^{j(\omega t - kz)}$$

where  $z$  is the direction of propagation and  $r$ ,  $\phi$ , and  $z$  are cylindrical coordinates centered on axis of the coaxial cable.  $Z$  is the wave impedance in the dielectric of the cable, and  $C$  is an arbitrary constant. The angular frequency is denoted by  $\omega$ . The propagation constant,  $k$ , relates to the wavelength in the medium  $\lambda$  as

$$k = \frac{2\pi}{\lambda}$$

In the air, the electric field also has a finite axial component whereas the magnetic field is purely azimuthal. Thus it is possible to model the antenna using an axisymmetric transverse magnetic (TM) formulation, and the wave equation becomes scalar in  $H_\phi$ :

$$\nabla \times \left( \frac{1}{\epsilon} \nabla \times H_\phi \right) - \mu \omega^2 H_\phi = 0$$

## BOUNDARY CONDITIONS

The boundary conditions for the metallic surfaces are:

$$\mathbf{n} \times \mathbf{E} = 0$$

At the feed point, a matched coaxial port boundary condition is used to make the boundary transparent to the wave. The antenna is radiating into free space, but you can only discretize a finite region. Therefore, truncate the geometry some distance from the antenna using a scattering boundary condition allowing for outgoing spherical waves to pass without being reflected. Finally, apply a symmetry boundary condition for boundaries at  $r = 0$ .

## *Results and Discussion*

---

Figure 3-5 shows the antenna impedance as a function of frequency. Ideally, the antenna impedance should be matched to the characteristic impedance of the feed,

50  $\Omega$ , to obtain maximum transmission into free space. This is quite well fulfilled in the high frequency range.

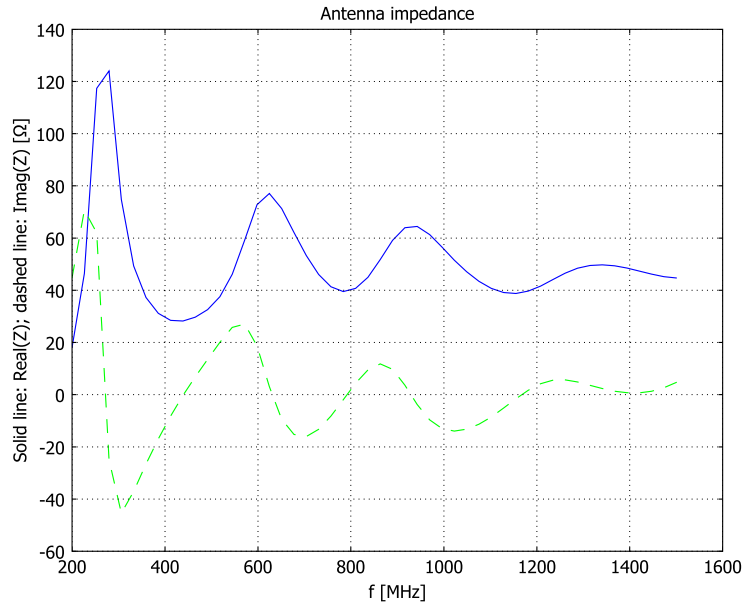


Figure 3-5: The antenna impedance as a function of frequency from 200 MHz to 1.5 GHz. The solid line shows the radiation resistance and the dashed line represents the reactance.

Figure 3-6 shows the antenna radiation pattern in the near-field for three different frequencies. The effect of the finite diameter of the ground plane is to lift the main lobe from the horizontal plane. For an infinite ground plane or in the high frequency limit, the radiation pattern is symmetric around zero elevation. This is easy to understand, as an infinite ground plane can be replaced by a mirror image of the monocone below the plane. Such a biconical antenna is symmetric around zero elevation and has its main

lobe in the horizontal direction. The decreased lobe lifting at higher frequencies is just about visible in Figure 3-6.

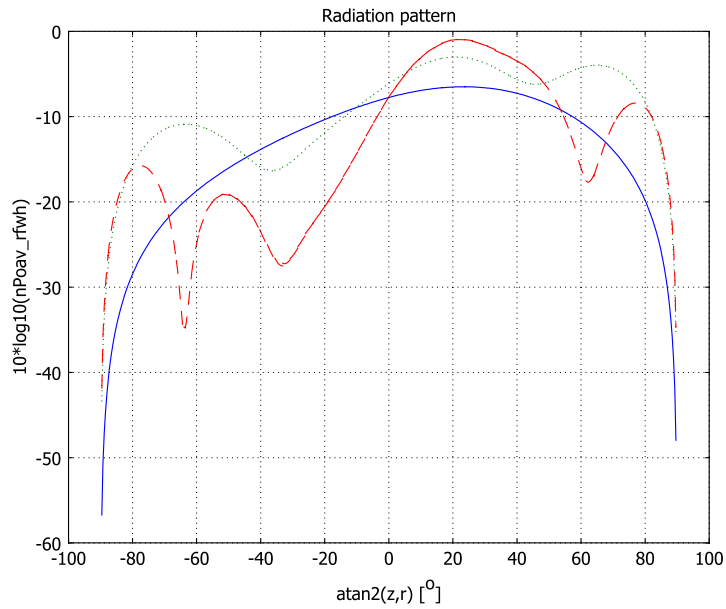


Figure 3-6: The antenna radiation pattern in the near-field as a function of the elevation angle for 200 MHz (solid line), 863 MHz (dotted line) and 1.5 GHz (dashed line).



Figure 3-7 shows the antenna radiation pattern in the far-field for the same frequencies as the radiation pattern at the boundary in Figure 3-6.

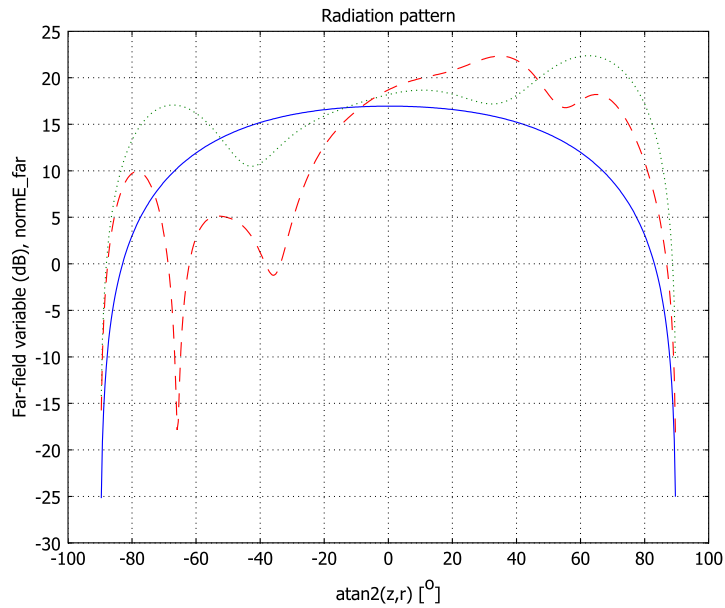


Figure 3-7: The antenna radiation pattern for the far-field as function of elevation angle for 200 MHz (solid line), 863 MHz (dotted line) and 1.5 GHz (dashed line).

---

**Model Library path:** RF\_Module/RF\_and\_Microwave\_Engineering/  
conical\_antenna

---

### *Modeling Using the Graphical User Interface*

---

#### **MODEL NAVIGATOR**

- 1 Select **Axial symmetry (2D)** in the **Space dimension** list.
- 2 Select the **RF Module>Electromagnetic Waves>TM Waves>Harmonic propagation** application mode and then click **OK**.

#### **OPTIONS AND SETTINGS**

- 1 From the **Options** menu, choose **Constants**.

- 2 Enter the following variable name, expression, and (optionally) description; when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
f	500 [MHz]	Frequency

- 3 From the **Options** menu, choose **Axes/Grid Settings**.
- 4 On the **Grid** page, clear the **Auto** check box.
- 5 Enter grid settings according to the following table; when done, click **OK**.

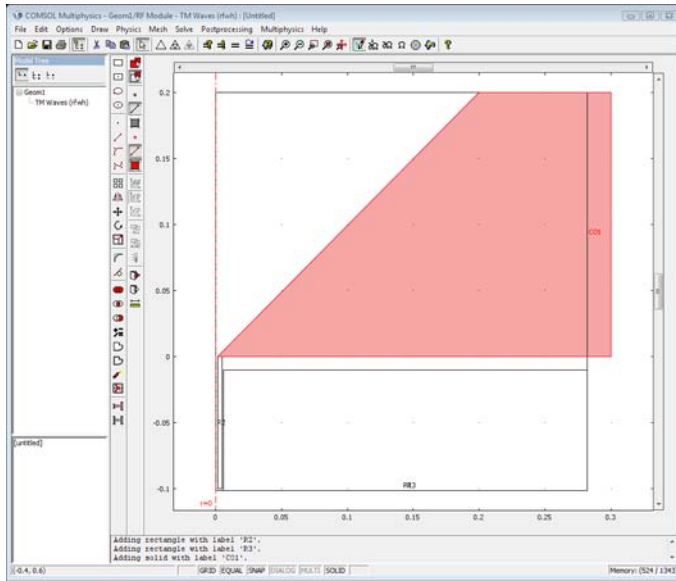
GRID	
r spacing	0.05
Extra r	0.0015 0.004916
z spacing	0.05
Extra z	

### GEOMETRY MODELING

To create the model geometry, draw the right half of the cross section of the antenna and the coaxial feed in a truncated half space.

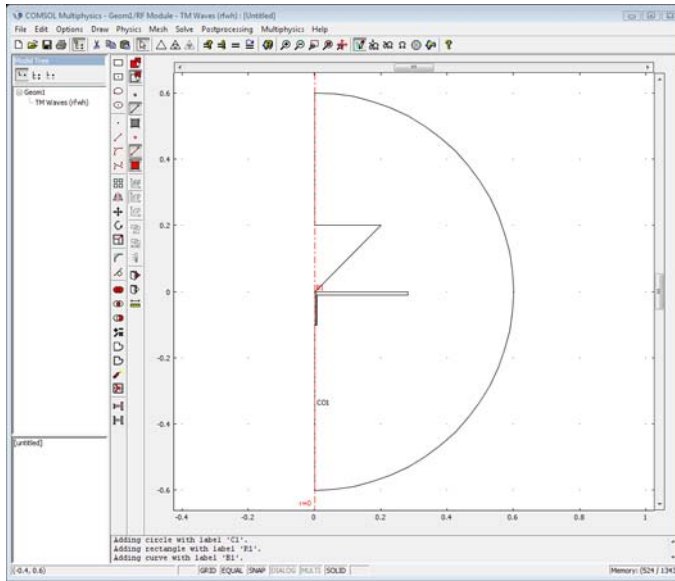
- 1 Open the **Rectangle** dialog box by shift-clicking the **Rectangle/Square** button on the Draw toolbar. Set the **Width** to 0.282, **Height** to 0.301, **Base** to **Corner**, **r** to 0, and **z** to -0.101. Click **OK**.
- 2 Using the same approach, draw a second rectangle and set the **Width** to 0.003416, **Height** to 0.1, **r** to 0.0015, and **z** to -0.1.
- 3 Draw a third rectangle and set the **Width** to 0.276, **Height** to 0.091, **r** to 0.006, and **z** to -0.101.
- 4 Click the **Zoom Extents** button on the Main toolbar.
- 5 Click the **Line** button on the Draw toolbar.

- 6 Draw a polygon with corners at (0.3, 0), (0.3, 0.2), (0.2, 0.2), and (0.0015, 0).  
Finally right-click to create a solid object CO1.



- 7 From the **Draw** menu, open the **Create Composite Object** dialog box.
- 8 In the **Set formula** edit field, type  $R1 - (R2+R3+CO1)$ .
- 9 Click **OK** to create the composite object.
- 10 Zoom out and draw a circle with radius 0.6 centered at (0, 0).
- 11 Draw a rectangle with opposite corners at (-0.6, -0.6) and (0, 0.6).
- 12 Select the three objects R1, C1, and CO2, then click the **Difference** button on the Draw toolbar.

- Zoom in around (0, 0) using the **Zoom Window** tool, and draw a line from (0.0015, 0) to (0.004916, 0).



## PHYSICS SETTINGS

### Scalar Variables

- From the **Physics** menu, choose **Scalar Variables**.
- In the **Application Scalar Variables** dialog box, set the **Frequency** to  $f$  to obtain a 500 MHz wave.
- Click **OK**.

### Boundary Conditions

- From the **Physics** menu, choose **Boundary Settings**.
- Enter boundary conditions according to the following table:

SETTINGS	BOUNDARIES 1, 3	BOUNDARY 6	BOUNDARIES 14, 15	ALL OTHERS
Boundary condition	Axial symmetry	Port	Scattering boundary condition	Perfect electric conductor
Wave type			Spherical wave	

- Select Boundary 6 and click the **Port** tab.

- 4 Select the **Wave excitation at this port** check box.
- 5 Set **Mode specification** to **Coaxial**.
- 6 Click **OK**.

#### *Subdomain Settings*

The inner of the coaxial line is made of teflon. The subdomain parameters need to be specified accordingly.

- 1 From the **Physics** menu, choose **Subdomain Settings**.
- 2 Specify settings according to the following table; when done, click **OK**.

SETTINGS	SUBDOMAIN 1	SUBDOMAIN 2
$\epsilon_r$	1	2.07
$\mu_r$	1	1

#### **MESH GENERATION**

- 1 From the **Mesh** menu, open the **Free Mesh Parameters** dialog box.
- 2 Click the **Custom mesh size** button and set the **Maximum element size** to  $2.5e-2$ .
- 3 To make the mesh finer inside the coaxial line, where the wavelength is shorter due to higher permittivity, click the **Subdomain** tab, select Subdomain 2, and set the **Maximum element size** to  $5e-4$ .
- 4 To make the mesh finer near the antenna, click the **Boundary** tab, select Boundaries 4 and 8, and set the **Maximum element size** to  $2.5e-3$ .
- 5 Click the **Remesh** button. When the mesher has finished, click **OK**.

#### **COMPUTING THE SOLUTION**

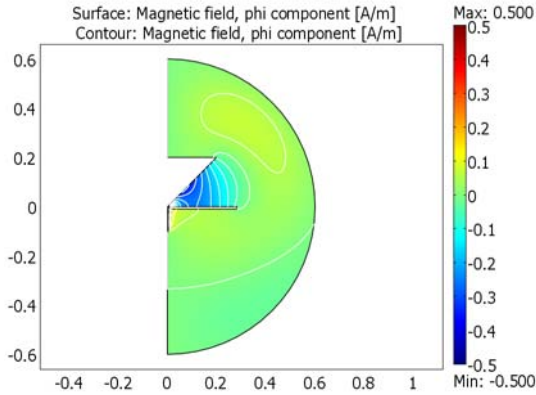
Click the **Solve** button on the Main toolbar.

#### **POSTPROCESSING AND VISUALIZATION**

The default plot shows the azimuthal magnetic field component of the transmitted wave. Due to the strong field in the coaxial line, you need to manually adjust the plot parameters.

- 1 Click the **Plot Parameters** button on the Main toolbar.
- 2 On the **Surface** page, click the **Range** button, clear the **Auto** check box and set **Min** and **Max** to  $-0.5$  and  $0.5$  respectively. Click **OK**.
- 3 On the **Contour** page, select the **Contour plot** check box. On the **Contour Data** page, choose **Magnetic field, phi component** from the **Predefined quantities** list.

- 4 In the **Contour levels** area, click the **Vector with isolevels** option button, then type `-0.5:0.01:0.1` in the associated edit field.
- 5 In the **Contour color** area, click first the **Uniform color** option button and then the **Color** button. Set the color to white, then click **OK**.
- 6 Clear the **Color scale** check box, then click **OK** to generate the following plot:



### *Antenna Impedance and Radiation Pattern*

A frequency sweep for the radiation pattern and the antenna impedance can be made from the graphical user interface or using a script file. The  $S_{11}$  scattering parameter is automatically computed when using the port boundary condition at the feed boundary. From  $S_{11}$ , the antenna impedance is deduced using the relation

$$Z = Z_{t1} \frac{1 + S_{11}}{1 - S_{11}}$$

where  $Z_{t1} = 50 \Omega$  is the characteristic impedance of the coaxial line.

#### **OPTIONS AND SETTINGS**

- I In the **Constants** dialog box, enter the following variable name and expression.

NAME	EXPRESSION	DESCRIPTION
Z_t1	50[ohm]	Characteristic transmission-line impedance

- 2 In the **Scalar Expressions** dialog box, define the following variables.

VARIABLE NAME	EXPRESSION
Z	$Z_{t1} * (1 + S11_{rfwh}) / (1 - S11_{rfwh})$

### SOLVING THE MODEL

- 1 Open the **Solver Parameters** dialog box and select the **Parametric** solver.
- 2 Set the **Parameter name** to  $f$  and the **Parameter values** to `linspace(200e6, 1.5e9, 50)`, then click **OK**.
- 3 Click the **Solve** button on the Main toolbar.

### POSTPROCESSING AND VISUALIZATION

To plot the antenna impedance as a function of frequency, do the following steps:

- 1 Open the **Domain Plot Parameters** dialog box.
- 2 On the **General** page, select all frequencies in the **Solutions to use** list.
- 3 Go to the **Point** page and select Point 1. Then enter `real(Z)` as **Expression** to plot the radiation resistance.
- 4 Click the **Line Settings** button. From the **Line style** list, select **Solid line**. Click **OK**.
- 5 Click **Apply** to generate the resistance plot.
- 6 To generate a second plot in the same figure window, go to the **General** page and select the **Keep current plot** check box.
- 7 Go to the **Point** page and enter `imag(Z)` as **Expression** to plot the reactance.
- 8 Click the **Line Settings** button. From the **Line color** list, select **Color**, then click the **Color** button. Select a green color, then click **OK**. From the **Line style** list, select **Dashed line**, then click **OK** to close the **Line Settings** dialog box.
- 9 On the **General** page, click the **Title/Axis** button. Enter `Antenna impedance` as **Title** and `Solid line: Real(Z); dashed line: Imag(Z) [\OMEGA]` as **Second axis label**. Click **OK**.
- 10 Click **Apply** to reproduce the plot in Figure 3-5.

To plot the radiation pattern as a function of frequency, follow these steps:

- 1 On the **General** page, select a few frequencies in the **Solutions to use** list, for example, **2e8**, **8.632653e8**, and **1.5e9**.
- 2 Clear the **Keep current plot** check box. Click the **Title/Axis** button and enter `Radiation pattern (dB)` as **Title**. Click the **Auto** option button for **Second axis label**, then click **OK**.

- 3 Click the **Line/Extrusion** tab. Select Boundaries 14 and 15, then enter the **Expression**  $10 \cdot \log_{10}(nPoav\_r\text{fwh})$ . Clear the **Smooth** check box.
- 4 In the **x-axis data** area, click first the lower radio button and then the **Expression** button. In the **X-Axis Data** dialog box, enter the **Expression**  $\text{atan2}(z, r)$ , select  $^{\circ}$  (degrees) from the **Unit** list, and then click **OK**.
- 5 Click the **Line Settings** button. Select **cycle** from both the **Line color** list and the **Line style** list, then click **OK**.
- 6 Click **OK** to generate the plot in Figure 3-6.

### *Far-Field Computation*

---

You can easily add far-field calculation to the model, and then plot the radiation pattern in the far field as shown in Figure 3-7.

#### **PHYSICS SETTINGS**

- 1 Select **Boundary settings** from the **Physics** menu.
- 2 Select Boundaries 14 and 15 in the **Boundary selection**.
- 3 Click the **Far-Field** tab.
- 4 Enter a far-field variable with name  $E\_far$  and make sure the tangential electric field and the curl of the electric field are entered automatically in the next two columns.
- 5 Click **OK**.

#### **COMPUTING THE SOLUTION**

To evaluate the far-field variables, select **Update Model** from the **Solver** menu.

#### **POSTPROCESSING AND VISUALIZATION**

To plot the radiation pattern of the far field, do the following steps:

- 1 Open the **Domain Plot Parameters** dialog box from the **Postprocessing** menu.
- 2 Select a few frequencies in the **Solution to use** list, for example **2e8**, **8.632653e8**, and **1.5e9**.
- 3 Click the **Line/Extrusion** tab and select Boundaries 14 and 15.
- 4 In the **Expression** edit field, type  $\text{norm}E\_far\text{dB}$ .
- 5 Click the **Line Settings** button. Make sure that **Line color** and **Line style** are set to **cycle**, then click **OK** to close the **Line Settings** dialog box.
- 6 Click **OK** to generate the plot in Figure 3-7.



# Magnetic Dipole Antenna

## *Introduction*

---

A magnetic dipole consists in its simplest form of a current carrying coil. Three such coils coupled together in an AC power circuit constitute a radiating system. Changing the phases of the coils alters the transmission angle of the waves.

## *Model Definition*

---

An array of three magnetic dipoles with a radius of 5 cm will be modeled. The dipoles are placed 0.1 m apart. The cylindrical symmetry of the problem allows us to set up a 2D axisymmetric model. The dipoles then appear as points in the  $rz$ -modeling plane.

The emitted waves from the dipoles are axisymmetric TE waves. The governing equation for the waves is

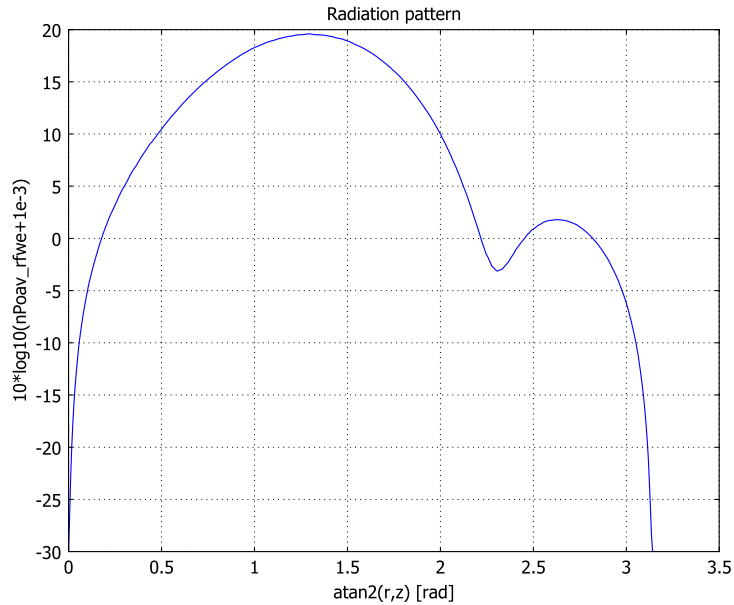
$$\nabla \times (\mu_r^{-1} \nabla \times \mathbf{E}) - k_0^2 \epsilon_r \mathbf{E} = 0$$

where  $\mu_r$  is the relative permeability,  $\epsilon_r$  the relative permittivity, and  $k_0$  the free space wave number. In the axisymmetric case this equation can be reduced to a scalar equation for the azimuthal component of the electric field  $E_\phi$ .

## Results and Discussion

---

The figure below shows the radiation pattern from an array of three coils, each carrying a current of 1 A. The relative phases of the currents are  $\varphi_1 = 0$ ,  $\varphi_2 = 0.7$  rad, and  $\varphi_3 = 1.4$  rad.



---

**Model Library path:** RF\_Module/RF\_and\_Microwave\_Engineering/  
dipole\_array

---

## Modeling Using the Graphical User Interface

---

### MODEL NAVIGATOR

- 1 Select **Axial symmetry (2D)** from the **Space dimension** list.
- 2 Select the **RF Module>Electromagnetic Waves>TE waves, harmonic propagation** application mode.
- 3 Click **OK**.

## OPTIONS AND SETTINGS

1 In the **Constants** dialog box, enter the following variable names and expressions.

NAME	EXPRESSION
I0	1
th1	-0.7
th2	2*th1

2 Use the default frequency of 1 GHz, which you find in the **Application Scalar Variables** dialog box.

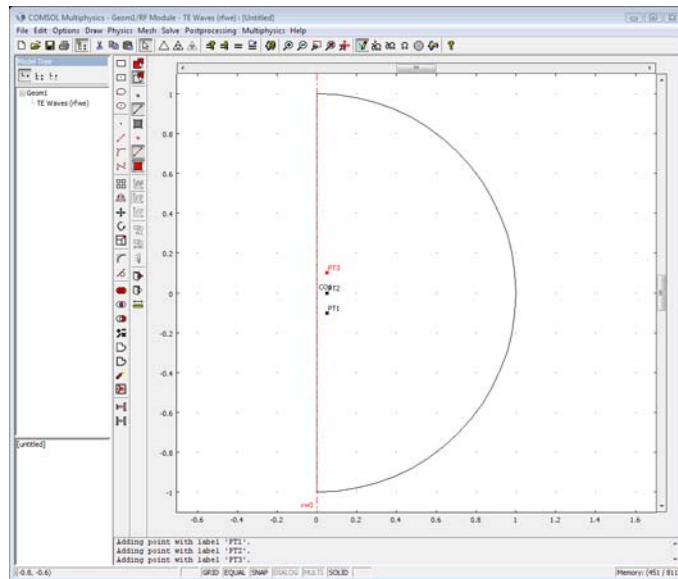
## GEOMETRY MODELING

1 Draw a circle with the radius 1 centered at (0,0).

2 Draw a rectangle with opposite corners at (0,-1) and (1,1).

3 Select both the circle and rectangle and click the **Intersection** button.

4 Draw three points at the coordinates (0.05, -0.1), (0.05, 0), and (0.05, 0.1).



## PHYSICS SETTINGS

### *Point Settings*

Define the three dipoles by entering a current at the points according to the following table:

SETTING	POINT 3	POINT 4	POINT 5
$I_0$	$I_0$	$I_0 \cdot \exp(j \cdot \theta_1)$	$I_0 \cdot \exp(j \cdot \theta_2)$

### *Boundary Conditions*

Enter boundary conditions according to the following table:

SETTING	BOUNDARY 1	BOUNDARIES 2, 3
Boundary condition	Axial symmetry	Scattering boundary condition

### *Subdomain Settings*

Use the default values applicable for air.

## MESH GENERATION

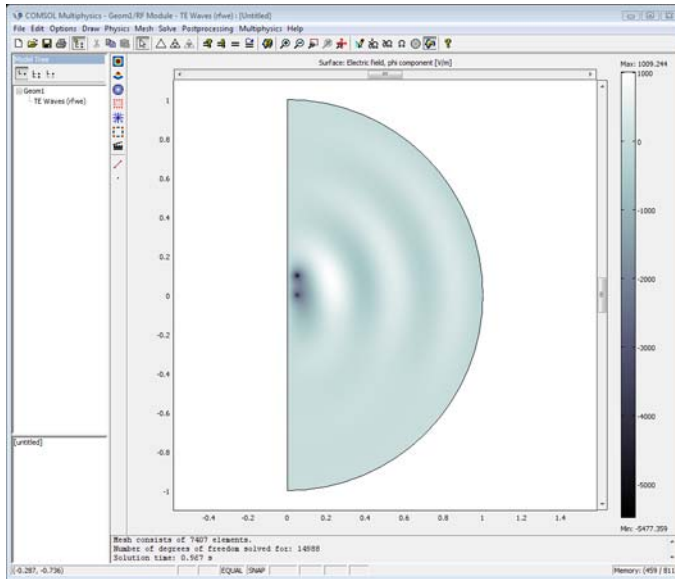
- 1 To obtain a uniform mesh, click the **Custom mesh size** button and set **Maximum element size** to 0.03 in the **Free Mesh Parameters** dialog box.
- 2 Generate the mesh.

## COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar.

## POSTPROCESSING AND VISUALIZATION

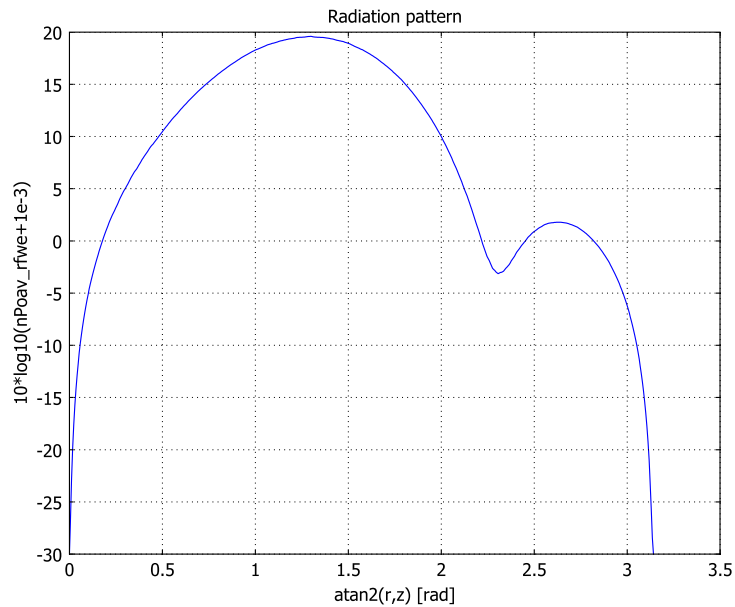
The default plot shows the electric field component of the transmitted wave.



To show the radiation pattern from the antenna array, make a plot along the half-circle boundary.

- In the **Domain Plot Parameters** dialog box select **Line** plot. Set the **y-axis data** to  $10 \cdot \log_{10}(nPoav\_rfwe + 1e-3)$  and select Boundaries 2 and 3. The term  $1e-3$  is to avoid singularities at the end points.

- 2 In the **x-axis** area, select the **Expression** radio button. Then click the **Expression** button and enter  $\text{atan2}(r, z)$  in the **Expression** edit field.



### *Modeling Using the Programming Language*

```
% When programming the same model you start by defining the
% constants. Use a constant named I1 rather than I0 to avoid
% conflict with the point variable I0 generated by the
% application mode.
```

```
clear fem
fem.const={...
    'I1',1,...
    'th1',-0.7,...
    'th2',-1.4};
```

```
% You can now define the coordinates and the geometry.
```

```
fem.sdim={'r' 'z'};
clear draw
draw.s.objs={circ2(0,0,1)*rect2(0,1,-1,1)};
draw.s.name={'C01'};
draw.p.objs={point2(0.05,-0.1),point2(0.05,0),point2(0.05,0.1)};
draw.p.name={'PT1','PT2','PT3'};
fem.draw=draw;
fem.geom=geomcsg(fem);
```

```

% Create the application structure and compute the corresponding
% FEM coefficients.
clear appl
appl.mode.class='AxisymmetricWaves';
appl.mode.type='axi';
appl.bnd.type={'ax' 'SC'};
appl.bnd.ind=[1 2 2];
appl.pnt.I0={'0' 'I1' 'I1*exp(j*th1)' 'I1*exp(j*th2)'};
appl.pnt.ind=[1 1 2 3 4 1];
fem.appl=appl;
fem=multiphysics(fem);

% You can now mesh the problem using the proper parameters, and
% then solve it.
fem.mesh=meshinit(fem,'hmax',...
    {0.03 [] [] []});
fem.xmesh=meshextend(fem);
fem.sol=femlin(fem);

% Make a visualization of the field by the following command.
postplot(fem,'tridata','Ephi')

% The radiation pattern is visualized using a cross-section plot.
postcrossplot(fem,1,[2 3],'lindata','10*log10(nPoav)');

```

# Thermal Drift in a Microwave Filter

## *Introduction*

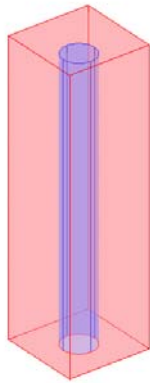
---

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

## *Model Definition*

---

Figure 3-8 shows the filter geometry. It consists of a box with a cylindrical post centered on one face. It is typically made of brass covered with a thin layer of silver to minimize losses. The silver layer is sufficiently thin to have a negligible influence on the device's thermal and mechanical properties.



*Figure 3-8: A microwave filter consists of a box-shaped thin metallic shell, typically made of brass, that contains a cylindrical post. This configuration forms a closed air-filled electromagnetic cavity between the box walls and the post.*

The thermal expansion and the associated drift in eigenfrequency are caused by a uniform increase of the temperature of the cavity walls. The thermal expansion is



readily computed using the Shell application mode from the Structural Mechanics Module. The electromagnetic cavity mode analysis is easily performed using the 3D Electromagnetic Waves application mode in the RF Module. In principle, it is possible to determine the thermal drift in the passband frequency by repeated calculation of the thermal expansion followed by an electromagnetic mode analysis of the deformed geometry. A complication arises, however, because it is not wise to regenerate the mesh—doing so would introduce too much numerical noise.

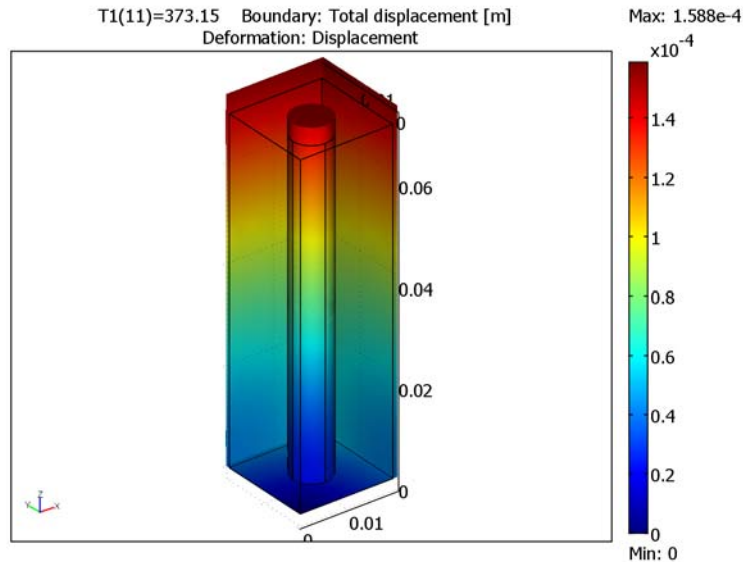
A better approach is to use a special set of equations to smoothly deform the original mesh according to the thermal expansion. Thus, the Moving Mesh (ALE) application mode for deformed geometry in COMSOL Multiphysics is used to map the thermal expansion to the geometry used for the electromagnetic analysis.

## *Results and Discussion*

---

### **MODELING THE THERMAL EXPANSION**

The filter's temperature can rise due to power dissipation in the filter itself, in the surrounding electronics, or due to external heating. Figure 3-9 shows the thermal expansion results for a filter entirely made of brass.



*Figure 3-9: Thermal expansion at 100 °C above the reference temperature.*

## ELECTROMAGNETIC MODE ANALYSIS

An actual filter usually consists of multiple cavities cascaded, but this discussion limits the analysis to one cell. Figure 3-10 shows the filter's lowest eigenmode. The typical quarter-wave resonance of the cylindrical post is clearly visible. A strong capacitive coupling between the top of the post and the nearby face of the box is also obvious.

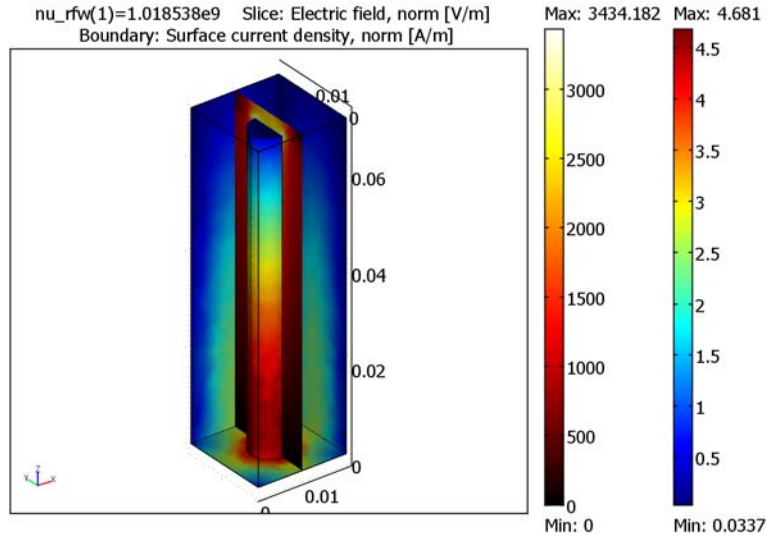


Figure 3-10: Results from the electromagnetic mode analysis. The plot shows the electric and surface current patterns of the fundamental mode.

## EIGENFREQUENCY VS. TEMPERATURE

By repeating the structural and electromagnetic analyses for a number of operating temperatures, an eigenfrequency-versus-temperature curve is obtained. The results for two different designs appear in Figure 3-11.

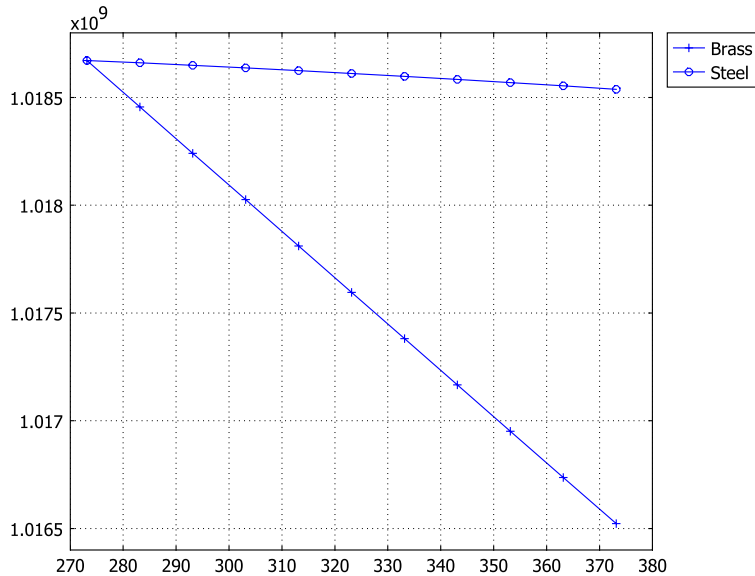


Figure 3-11: Eigenfrequency (Hz) versus temperature (K) for two different designs.

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

---

**Model Library path:** RF\_Module/RF\_and\_Microwave\_Engineering/mw\_filter

---

---

**Note:** This model requires the RF Module and the Structural Mechanics Module.

---

## *Modeling Using the Graphical User Interface*

---

### **MODEL NAVIGATOR**

The model uses linear elements to reduce the memory requirements.

- 1 Click the **Multiphysics** button.
- 2 Select **3D** from the **Space dimension** list.
- 3 In the **Structural Mechanics Module** folder, select **Shell>Static analysis**.
- 4 Click **Add** to add a copy of this application mode.
- 5 In the **COMSOL Multiphysics** folder, select **Deformed Mesh>Moving Mesh (ALE)>Static analysis**. Select **Lagrange - Linear** from the **Element** list.
- 6 Click **Add**.
- 7 In the **RF Module** folder, select **Electromagnetic Waves>Eigenfrequency analysis**. Select **Vector - Linear** from the **Element** list.
- 8 Click **Add**.
- 9 The **Electromagnetic Waves>Eigenfrequency analysis** application mode now appears under **Frame (ale)** in the displayed tree structure.
- 10 Change **Ruling application mode** to **Electromagnetic Waves (rfw)**.
- 11 Click **OK** to close the **Model Navigator**.

### **OPTIONS AND SETTINGS**

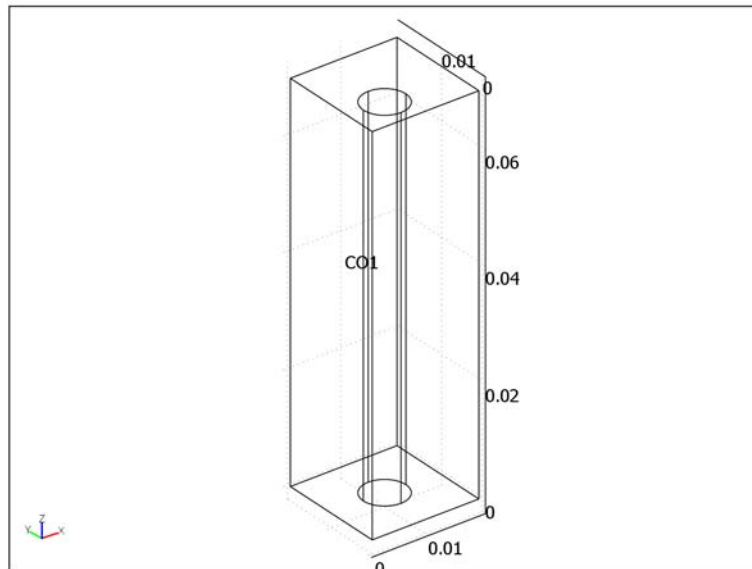
- 1 From the **Options** menu, choose **Constants**.
- 2 In the **Constants** dialog box, define the following constants. The description field is optional. When done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
T0	0[degC]	Reference temperature
T1	0[degC]	Operating temperature

For the parametric analysis, you replace the constant T1 for the operating temperature,  $T_1$ , with a parameter T1 in the **Solver Parameter** dialog box.

## GEOMETRY MODELING

- 1 Go to the **Draw** menu and click the block symbol and define a block with **Length X:** 0.02, **Length Y:** 0.02, **Length Z:** 0.07. Leave the other parameters at their defaults.
- 2 Click **OK**.
- 3 Click the **Zoom Extents** button to get a better view of the geometry.
- 4 Select **Work-Plane Settings** from the **Draw** menu.
- 5 Click **OK** to create a default xy work plane at  $z = 0$ .
- 6 Click the **Zoom Extents** button.
- 7 In the 2D work plane (**Geom 2**), go to **Draw>Specify Objects>Circle** to create a circle C1 with radius 0.004 and center at (0.01, 0.01).
- 8 Go to **Draw>Extrude** and extrude C1 a **Distance** of 0.067.
- 9 Press Ctrl+A to select all objects in the 3D geometry.
- 10 Click the **Difference** button on the Draw toolbar.
- 11 Click the **Zoom Extents** button on the Main toolbar.



## BOUNDARY CONDITIONS (SHELL)

- 1 From the **Multiphysics** menu, select **Geom 1: Shell (smsh)**.
- 2 In the **Boundary settings** dialog box, click the **Groups** tab.

- 3 Select the **unnamed1** group and change the **Name** to Brass.
- 4 Change the **Young's modulus** to 1.05e11 (Pa).
- 5 Change the **Poisson's ratio** to 0.35.
- 6 Change the **Density** to 8400 (kg/m<sup>3</sup>).
- 7 Change the **Thermal expansion coeff.** to 2.1e-5 (K<sup>-1</sup>).
- 8 Change the **Thickness** to 0.002 (m).
- 9 Click the **Load** tab and click **Include thermal expansion**.
- 10 Set **Strain temperature** to T1 and **Strain ref. temperature** to T0.
- 11 Click **New** to create a new group that inherits the settings for “Brass” and name it “Steel.”
- 12 Click the **Material** tab.
- 13 Change the **Young's modulus** to 2e11 (Pa).
- 14 Change the **Density** to 7800 (kg/m<sup>3</sup>).
- 15 Change the **Thermal expansion coeff.** to 1.15e-5 (K<sup>-1</sup>).
- 16 Click the **Boundaries** tab. By now, all boundaries, should belong to the “Brass” group and “Steel” is only there for future use.
- 17 Click **OK**.

#### POINT SETTINGS

Appropriate point settings must be applied to eliminate any translation or rotation of the structure.

- 1 In the **Point settings** dialog box, select Point 1.
- 2 Constrain **R<sub>x</sub>**, **R<sub>y</sub>**, and **R<sub>z</sub>** to 0.
- 3 Select Point 3 and constrain **R<sub>x</sub>** and **R<sub>z</sub>** to 0.
- 4 Select Point 13 and constrain **R<sub>y</sub>** and **R<sub>z</sub>** to 0.
- 5 Click **OK**.

#### BOUNDARY CONDITIONS (ALE)

The default subdomain settings in ALE is “free displacement,” which is right for this model so you only need to set the ALE boundary displacements to the displacements from the shell application mode.

- 1 From the **Multiphysics** menu, select **Geom1: Moving Mesh ALE (ale)**.
- 2 In the **Boundary settings** dialog box, select all boundaries.

3 Set **dx**, **dy**, and **dz** to u, v, and w, respectively.

4 Click **OK**.

### **BOUNDARY CONDITIONS (EM WAVES)**

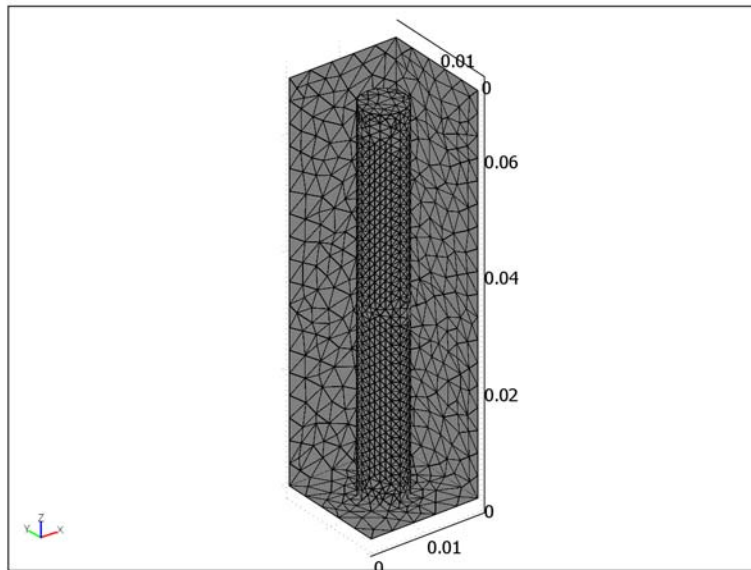
The default boundary and subdomain settings in the Electromagnetic Wave application mode are “perfect electric conductor” and “air,” which is fine for this analysis.

### **MESH GENERATION**

1 Open the **Free Mesh Parameters** dialog and select **Finer** from the **Predefined mesh sizes** list and close the dialog by clicking **OK**.

2 Click the **Initialize Mesh** button in the toolbar to generate the mesh.

3 To see the mesh and later the solution better, go to the **Options** menu and select **Suppress>Suppress Boundaries** and select Boundaries 1, 2, and 4. Click **OK** and go to mesh mode by clicking the **Mesh Mode** button on the Main toolbar.



### **COMPUTING THE SOLUTION (SHELL)**

Use the parametric solver to study thermal expansion as a function of temperature. The solver settings are adapted for the ruling application mode, which is the electromagnetic problem. You need to change these settings to fit also the structural

problem. Also make use of the option to record the solver calls in a log that you can replay later.

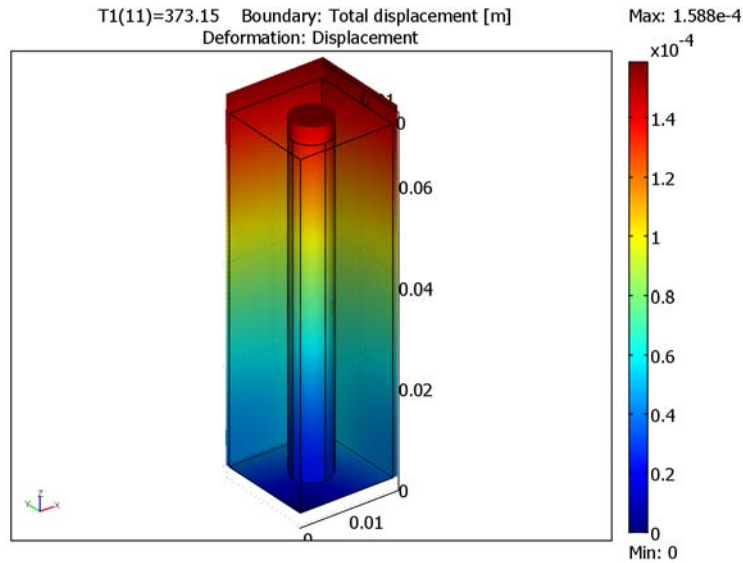
- 1 In the **Solver Parameters** dialog box select the **Parametric** solver.
- 2 Enter T1 as **Parameter name** and `linspace(273.15,373.15,11)` as **Parameter values**. The solver then calculates the solution using 11 equidistant temperature values in the range 273.15 K to 373.15 K.
- 3 Click **OK**.
- 4 Open the **Solver Manager** from the **Solve** menu.
- 5 Click the **Initial value expression** button in the **Initial value** area.
- 6 Click the **Solve For** tab and select only the **Shell (smsh)** variables.
- 7 Click the **Script** tab and select the **Automatically add commands when solving** check box.
- 8 Click **OK**.
- 9 Solve by clicking the **Solve** button on the Main toolbar.

#### **POSTPROCESSING AND VISUALIZATION**

- 1 Open the **Plot Parameters** dialog box and clear the **Slice** check box in the **Plot type** area on the **General** tab. Instead, select the **Boundary** and **Deformed shape** plots.
- 2 Click the **Boundary** tab and select **Shell (smsh)>Total displacement** from the list of **Predefined quantities**.
- 3 Click the **Deform** tab and clear the **Subdomain** check box in the **Domain types to deform** area.



4 Click **OK**.



### COMPUTING THE SOLUTION (ALE)

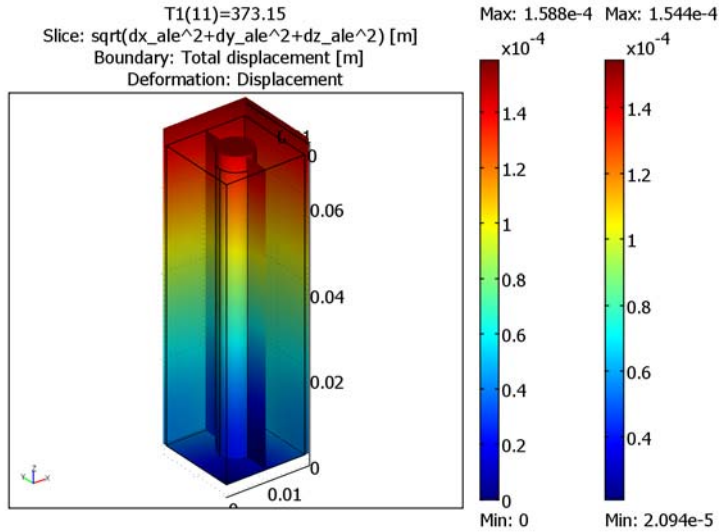
To map the shell deformations smoothly to the volume mesh, you must solve the ALE problem. Use the parametric solver again, now starting from the previous solutions:

- 1 Open the **Solver Manager**.
- 2 Click the **Initial value** tab.
- 3 In the **Values of variables not solved for and linearization point** area, click the **Stored solution** option button.
- 4 Click **Store Solution** and then click **OK** in the dialog that opens (store all solutions).
- 5 Select **All** from the **Parameter value** list.
- 6 Click the **Solve For** tab and select only the **Moving Mesh (ALE) (ale)** variables.
- 7 Click **OK** to close the **Solver Manager**.
- 8 Solve by clicking the **Solve** button on the Main toolbar.

### POSTPROCESSING AND VISUALIZATION

- 1 Open the **Plot Parameters** dialog box and select the **Slice** check box in the **Plot type** area on the **General** tab.

- 2 Click the **Slice** tab and change **Number of levels** to **1 X level** and type  $\text{sqrt}(dx\_ale^2+dy\_ale^2+dz\_ale^2)$  in the **Expression** field.
- 3 Click **OK**.



#### EXPORTING DATA TO COMSOL SCRIPT

- 1 Go to the **File** menu and select **Export>FEM Structure**.
- 2 Type `fem_brass` in the dialog that appears.
- 3 Click **OK** and close the COMSOL Script window that appears.

#### BOUNDARY CONDITIONS (SHELL)

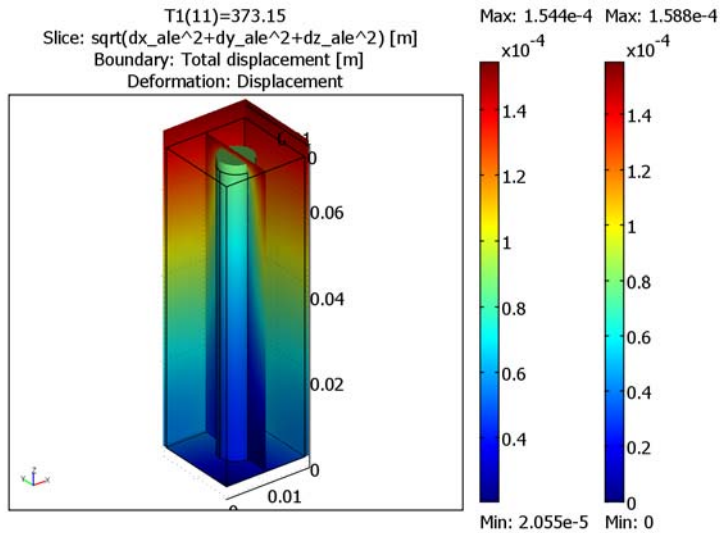
Change the material in the post from brass to steel.

- 1 From the **Multiphysics** menu, select **Geom1: Shell (smsh)**.
- 2 In the **Boundary settings** dialog box, change **Group** from **Brass** to **Steel** for Boundaries 6–10.
- 3 Click **OK**.

#### COMPUTING THE SOLUTION USING A SOLVER SCRIPT

Now, you must repeat the solution steps above. To automate this, use the solver script that has been recorded.

- 1 Open the **Solver Manager**.
- 2 Click the **Script** tab.
- 3 Select the **Solve using a script** check box.
- 4 Click **OK**.
- 5 Solve by clicking the **Solve** button.



#### EXPORTING DATA TO COMSOL SCRIPT

- 1 Go to the **File** menu and select **Export>FEM Structure**.
- 2 Type `fem_steel` in the dialog that appears.
- 3 Click **OK** and close the COMSOL Script window that appears.

#### COMPUTING THE SOLUTION (EM WAVES)

From the graphical user interface, the electromagnetic problem can only be solved for one temperature at a time. Perform this for one temperature. Then go to the COMSOL Script command window for the rest of the analysis.

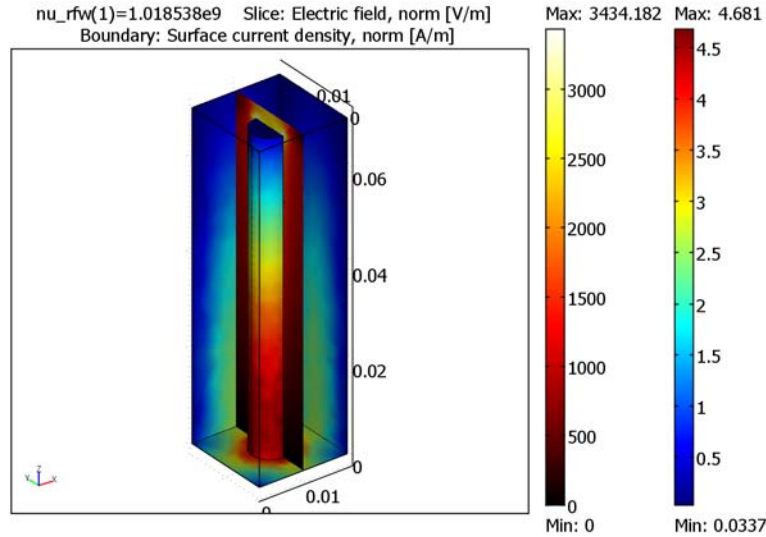
- 1 Open the **Solver Manager**.
- 2 Click the **Script** tab.
- 3 Clear the **Solve using a script** check box.

- 4 Click the **Solve For** tab and select only the **Electromagnetic Waves (rfw)** variables.
- 5 Click the **Initial Value** tab.
- 6 Change **Parameter value** to 373.15.
- 7 Click the **Store Solution** button and then click **OK** in the dialog that opens (store all solutions).
- 8 Click **OK** to close the **Solver Manager**.
- 9 Open the **Solver Parameters** dialog.
- 10 Change **Solver** to **Eigenfrequency**.
- 11 Set **Desired number of eigenfrequencies** to 1 and **Search for eigenfrequencies around to** 1e9.
- 12 Click **OK** to close the **Solver Parameters** dialog box.
- 13 Solve by clicking the **Solve** button on the Main toolbar.

#### **POSTPROCESSING AND VISUALIZATION**

- 1 Open the **Plot Parameters** dialog box and clear the **Deformed shape** check box under **Plot type** on the **General** tab. Keep the **Boundary** and **Slice** plots.
- 2 Click the **Boundary** tab and select **Electromagnetic Waves (rfw)>Surface current density, norm** from the **Predefined quantities** list.
- 3 Click the **Slice** tab and select **Electromagnetic Waves (rfw)>Electric field, norm** from the **Predefined quantities** list.

- 4 Click **OK**. Note that the results might differ slightly from the plot below because the mesh on your computer can be different and the plotted quantities are rather mesh sensitive.



### COMPUTING THE SOLUTION IN COMSOL SCRIPT

Open the COMSOL Script command window from the **File** menu. Then issue the following commands to solve the electromagnetic problem for all temperature values for the two filter designs. The solver calls can essentially be copied from the **Script** page in the **Solver Manager**. COMSOL Script can open and run M-files, so the most convenient procedure is to save the commands in a text file and save it with the file extension `.m`, for example, `eigloop.m`.

```
fem = fem_brass;
fem1 = fem;
T = fem.sol.plist;
freq = zeros(size(T));
for k=1:length(T)
    u = asseminit(fem1,'init',fem.sol,'solnum',k);
    fem1.sol=femeig(fem1,'u',u, ...
        'solcomp',{'tExEyEz10'}, ...
        'outcomp',{'z','thz','w','lm9','lm8', ...
        'v','tExEyEz10','thx','lm7','u','y', ...
        'thy','x'},'neigs',1, ...
        'shift',-6.283185e9*i);
```

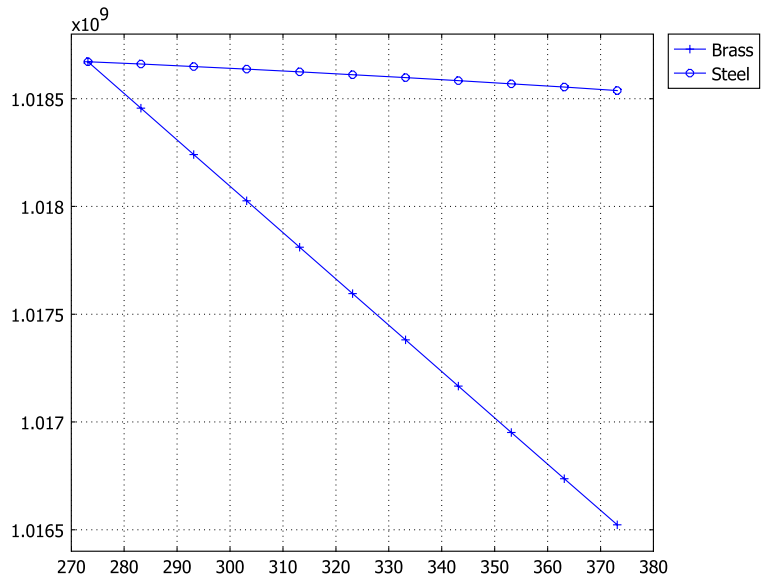
```

    freq(k) =
postinterp(fem1,'nu_rfw',zeros(0,1),'dom',1,'solnum',1);
end;
plot(T,freq,'+-');
grid on;
hold on;

fem = fem_steel;
fem1 = fem;
T = fem.sol.plist;
freq = zeros(size(T));
for k=1:length(T)
    u = assemnit(fem1,'init',fem.sol,'solnum',k);
    fem1.sol=femeig(fem1,'u',u, ...
                    'solcomp',{'tExEyEz10'}, ...
                    'outcomp',{'z','thz','w','lm9','lm8', ...
                               'v','tExEyEz10','thx','lm7','u','y', ...
                               'thy','x'},'neigs',1, ...
                    'shift',-6.283185e9*i);

    freq(k) =
postinterp(fem1,'nu_rfw',zeros(0,1),'dom',1,'solnum',1);
end;
plot(T,freq,'o-');
grid on;
hold on;
legend('Brass','Steel');

```



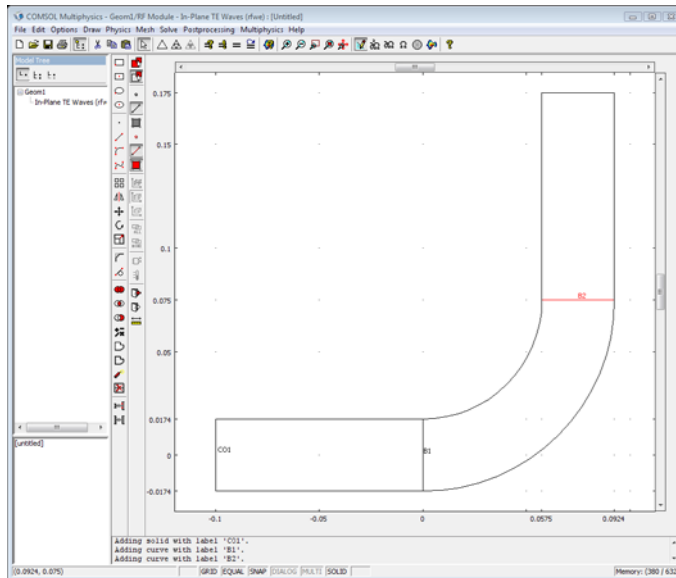
# Waveguide H-Bend with S-parameters

## *Introduction*

This example is an extension of the waveguide H-bend model found on page 21 in the *RF Module User's Guide*. For a general introduction to the model, see the *RF Module User's Guide*.

The purpose of the model is to calculate how well a  $TE_{10}$  wave propagates through a bend in a waveguide filled with a dielectric medium. This is done by calculating the scattering parameters, or S-parameters, of the waveguide bend as a function of the frequency, for the fundamental  $TE_{10}$  mode. The S-parameters are a measure of the transmittance and reflectance of the waveguide bend. For a theoretical background on S-parameters, see the section “S-Parameters and Ports” on page 56 in the *RF Module User's Guide*.

This model only includes the  $TE_{10}$  mode of the waveguide, and you can therefore make the model in 2D, because the fields of the  $TE_{10}$  mode have no variation in the transverse direction. The figure below shows the modeling geometry.



The bend of the waveguide is made of a dielectric medium with the refractive index  $n = 1.45$ .



## Model Definition

---

The governing equation for transverse electric (TE) waves with the transversal component  $E_z$  is given by

$$-\nabla \cdot \nabla E_z - n^2 k_0^2 E_z = 0$$

where  $n$  is the refractive index, and  $k_0$  is the free space wave number.

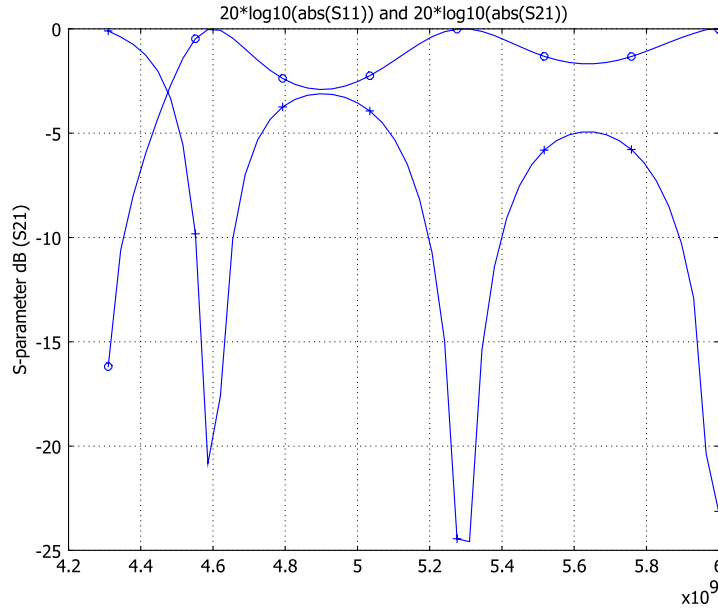
The entrance and exit ports do not represent physical boundaries in the geometry. To avoid reflection at these boundaries, matched boundary conditions make the boundaries transparent to the wave.

## Results and Discussion

---

The following figure shows the S-parameters as a function of frequency. The  $S_{11}$  parameter corresponding to the reflectance is given by the curve with plus sign markers. The  $S_{21}$  parameter corresponding to the transmittance is given by the curve with circle markers.

The resonances at 4.6 GHz, 5.3 GHz, and 6.0 GHz are the cavity resonances of the dielectric region in the bend.



---

**Model Library path:** RF\_Module/RF\_and\_Microwave\_Engineering/  
hband\_S\_parameters

---

### *Modeling Using the Graphical User Interface*

---

Begin by following the steps in the section “2D Modeling Using the Graphical User Interface” on page 33 in the *RF Module User’s Guide*. Then continue as follows.

#### **GEOMETRY MODELING**

- 1 Click the **Line** button, and click at (0, 0.0174) and at (0, -0.0174) with the left mouse button. Then click with the right mouse button to add the line to the geometry.
- 2 Draw another line from (0.0575, 0.075) to (0.0924, 0.075).

#### **SCALAR VARIABLES**

In the **Application Scalar Variables** dialog box, set the frequency nu\_rfwe to fq1.

#### **BOUNDARY CONDITIONS**

The default boundary condition is perfect electric conductor which is fine for all exterior boundaries except at the ports. Interior boundaries are by default not active meaning that the software imposes continuity.

For Boundaries 1 and 7, specify the **Port** boundary condition with the following settings (to set the mode specification and the mode number, go to the **Port** page):

SETTINGS	BOUNDARY 1	BOUNDARY 7
Port number	1	2
Wave excitation at this port	Selected	Cleared
Mode specification	Analytic	Analytic
Mode number	1	1

#### **SUBDOMAIN SETTINGS**

In the **Subdomain Settings** dialog box, select **Specify material parameters in terms of refractive index** in Subdomain 2. Then set the refractive index to 1.45.

## MESH GENERATION

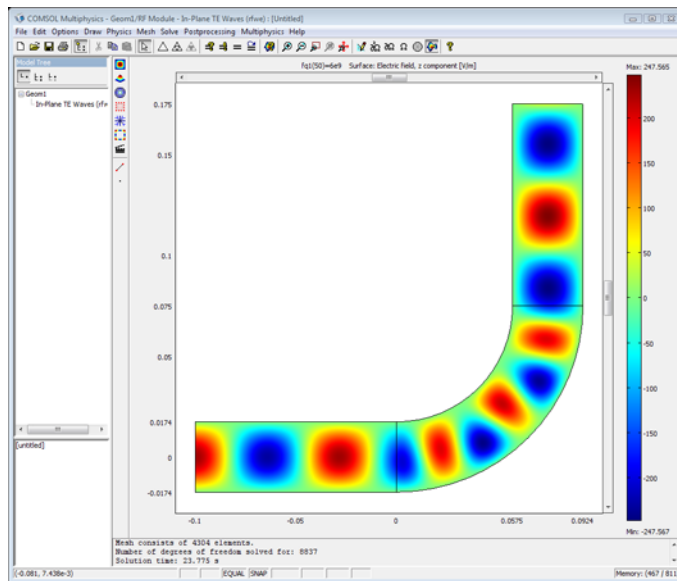
Initialize a mesh and refine it twice.

## COMPUTING THE SOLUTION

Use the parametric solver to study the S-parameters as functions of the frequency.

- 1 In the **Solver Parameters** dialog box select the **Parametric** solver.
- 2 Enter  $f_{q1}$  as **Parameter name** and  $\text{linspace}(4.31e9, 6e9, 50)$  as **Parameter values**.  
The solver will calculate the solution using 50 equidistant frequency values in the range 4.31 GHz to 6 GHz. The frequency 4.31 GHz is just above the cutoff frequency.
- 3 Solve by clicking the **Solve** button.

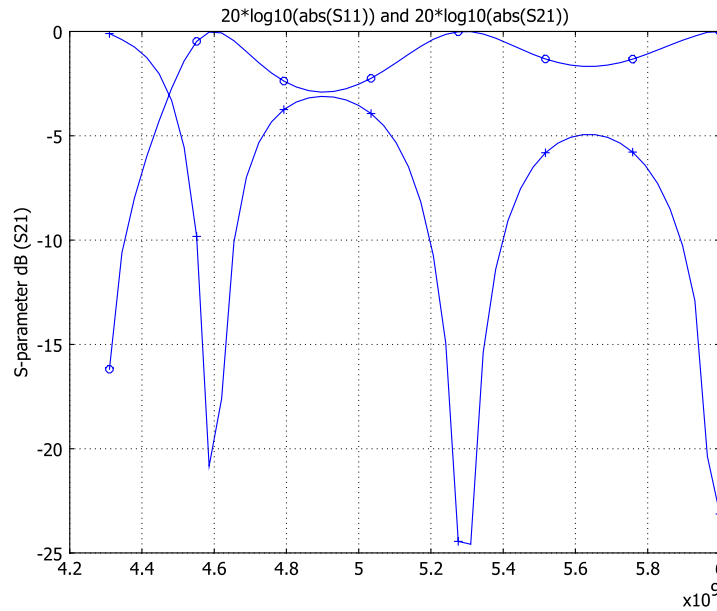
## POSTPROCESSING AND VISUALIZATION



You can plot the S-parameters using a domain plot.

- 1 In the **Domain Plot Parameters** dialog box select all solutions in the **Solutions to use** list.
- 2 Select the **Point** tab and select **S\_parameter dB(S11)** from the list of **Predefined quantities**.

- 3  $S_{11}$  is defined everywhere, but you only need to evaluate it at a single point to make a plot as a function of frequency. Select point 1 and click **Apply** to plot  $S_{11}$ .
- 4 On the **General** tab select **Keep current plot**.
- 5 Select the **Point** tab and select **S\_parameter dB(S21)** from the list of **Predefined quantities** and click **Apply**.




---

**Note:** When displaying S-Parameter variables, the phase of the solution must be set to zero. Otherwise, incorrect values will be displayed. In the various dialogs for displaying and plotting data that are available from the **Postprocessing** menu, the phase of the solution can be set via the **Solution at angle (phase)** field. If the dialog box contains several tabs, it usually appears on the **General** tab.

---

# Waveguide Adapter

## *Introduction*

---

This is a model of an adapter for microwave propagation in the transition between a rectangular and an elliptical waveguide. Such waveguide adapters are designed to keep energy losses due to reflections at a minimum for the operating frequencies. To investigate the characteristics of the adapter, the simulation includes a wave traveling from a rectangular waveguide through the adapter and into an elliptical waveguide. The S-parameters are calculated as functions of the frequency. The involved frequencies are all in the single-mode range of the waveguide, that is, the frequency range where only one mode is propagating in the waveguide.

## *Model Definition*

---

The waveguide adapter consists of a rectangular part smoothly transcending into an elliptical part as seen in Figure 3-12.



*Figure 3-12: The geometry of the waveguide adapter.*

The walls of manufactured waveguides are typically plated with a good conductor such as silver. The model approximates the walls by perfect conductors. This is represented by the boundary condition  $\mathbf{n} \times \mathbf{E} = \mathbf{0}$ .

The rectangular port is excited by a transverse electric (TE) wave, which is a wave that has no electric field component in the direction of propagation. This is what an incoming wave would look like after traveling through a straight rectangular waveguide with the same cross section as the rectangular part of the adapter. The excitation frequencies are selected so that the TE<sub>10</sub> mode is the only propagating mode through the rectangular waveguide. The cutoff frequencies for the different modes are given analytically from the relation

$$(v_c)_{mn} = \frac{c}{2} \sqrt{\left(\frac{m}{a}\right)^2 + \left(\frac{n}{b}\right)^2}$$

where  $m$  and  $n$  are the mode numbers, and  $c$  is the speed of light. For the TE<sub>10</sub> mode,  $m = 1$  and  $n = 0$ . With the dimensions of the rectangular cross section ( $a = 2.286$  cm and  $b = 1.016$  cm), the TE<sub>10</sub> mode is the only propagating mode for frequencies between 6.6 GHz and 14.7 GHz.

At the elliptical port of the waveguide, the solution of an eigenmode problem on the port boundary gives the propagating mode. The elliptical port is passive, but the eigenmode is still used in the boundary condition of the 3D propagating wave simulation. Using the RF Module's Boundary Mode Analysis application mode, the eigenmode equation is

$$\nabla \times (n^{-2} \nabla \times H_n) + (n^{-2} \beta^2 - k_0^2) H_n = 0$$

Here  $H_n$  is the component of the magnetic field perpendicular to the boundary,  $n$  the refractive index,  $\beta$  the propagation constant in the direction perpendicular to the boundary, and  $k_0$  the free space wave number. The eigenvalues are  $\lambda = -j\beta$ . The dimensions of the elliptical end of the waveguide are such that the frequency range for the lowest propagating mode overlaps that of the rectangular port.

With the stipulated excitation at the rectangular port and the numerically established mode shape at the elliptical port as boundary conditions, the following equation is solved for the electric field vector  $\mathbf{E}$  inside the waveguide adapter:

$$\nabla \times (\mu_r^{-1} \nabla \times \mathbf{E}) - k_0^2 \left( \epsilon_r - \frac{j\sigma}{\omega \epsilon_0} \right) \mathbf{E} = 0$$

where  $\mu_r$  denotes the relative permeability,  $j$  the imaginary unit,  $\sigma$  the conductivity,  $\omega$  the angular frequency,  $\epsilon_r$  the relative permittivity, and  $\epsilon_0$  the permittivity of free space. The model uses the following material properties for free space:  $\sigma = 0$  and  $\mu_r = \epsilon_r = 1$ .

## Results

---

The first eigenmode of the elliptical port is displayed in Figure 3-13. The effective mode index is  $\beta/k_0 = 0.535$ .

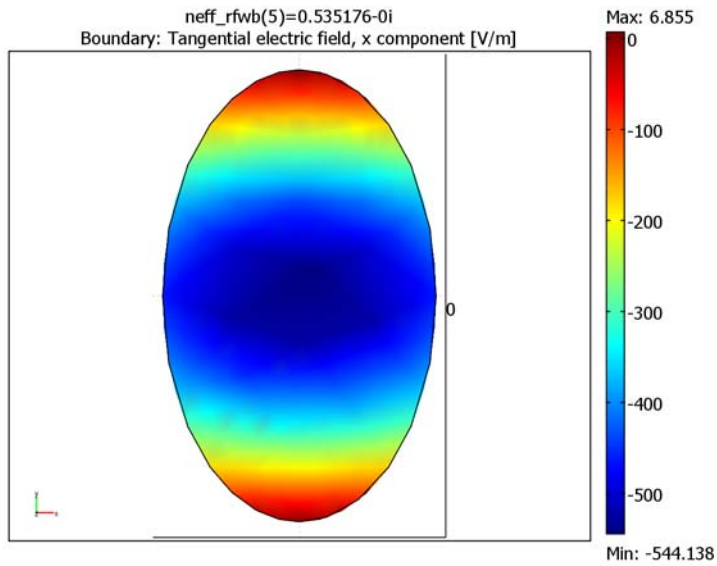


Figure 3-13: The  $x$  component of the electric field for the first eigenmode of the elliptical port.

Figure 3-14 shows a single-mode wave propagating through the waveguide.

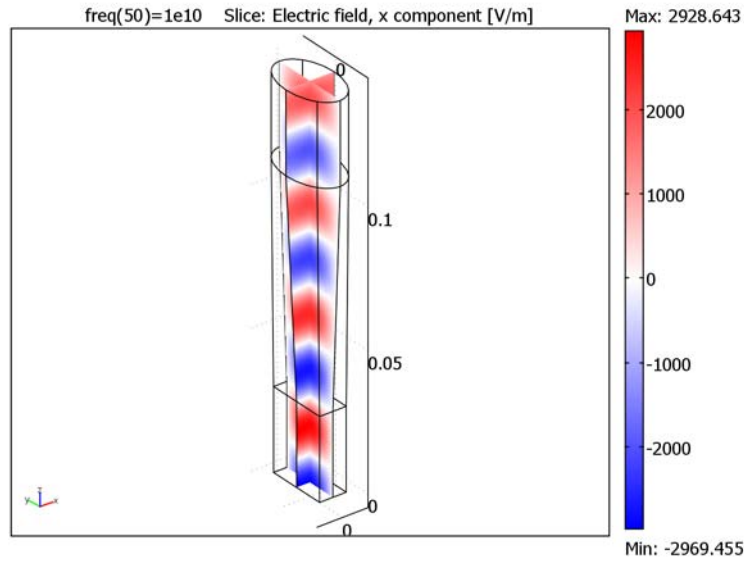


Figure 3-14: The  $x$  component of the propagating wave inside the waveguide adapter at the frequency 10 GHz.

Naming the rectangular Port 1 and the elliptical Port 2, the S-parameters describing the reflection and transmission of the wave are defined as follows:

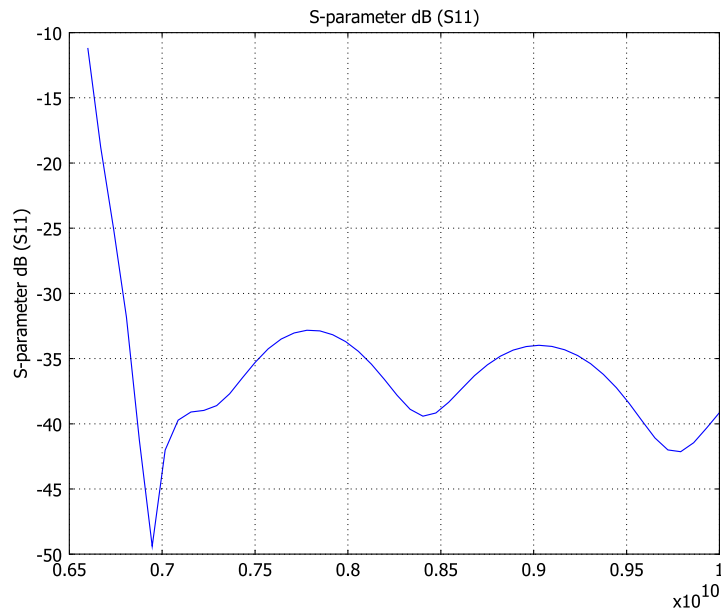
$$S_{11} = \frac{\int_{\text{Port 1}} ((E_c - E_1) \cdot E_1^*) dA_1}{\int_{\text{Port 1}} (E_1 \cdot E_1^*) dA_1}$$

$$S_{21} = \frac{\int_{\text{Port 2}} (E_c \cdot E_2^*) dA_2}{\int_{\text{Port 2}} (E_2 \cdot E_2^*) dA_2}$$

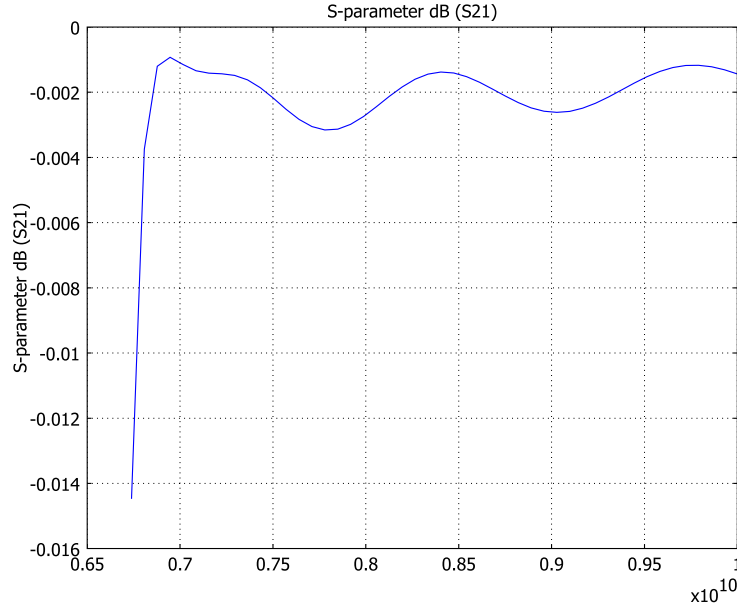
Here  $E_c$  is the calculated total field.  $E_1$  is the analytical field for the port excitation, and  $E_2$  is the eigenmode calculated from the boundary mode analysis and normalized



with respect to the outgoing power flow. Figure 3-15 and Figure 3-16 show the  $S_{11}$  and  $S_{21}$  parameters as functions of the frequency.



*Figure 3-15: The  $S_{11}$  parameter (in dB) as a function of the frequency. This parameter describes the reflections when the waveguide adapter is excited at the rectangular port.*



*Figure 3-16: The  $S_{21}$  parameter (in dB) as a function of the frequency. This parameter is a measure of the part of the wave that is transmitted through the elliptical port when the waveguide adapter is excited at the rectangular port.*

### *Modeling in COMSOL Multiphysics*

---

This model uses the RF Module’s Port boundary condition for the wave propagation problem. With this boundary condition the software computes the S-parameters automatically. The mode of the incoming wave is specified. The mode of the outgoing wave is given from an eigenmode analysis on the cross section of the elliptical part of the waveguide adapter.

To create the geometry of the waveguide adapter from the command line, use the `loft` command, which provides a smooth connection between the box and the ellipse. The resulting geometry is available in a COMSOL binary file, `waveguide_adapter_geometry.mphbin`. The script for creating the geometry is available at the end of the model description.

---

**Model Library path:** RF\_Module/RF\_and\_Microwave\_Engineering/  
waveguide\_adapter

---

### *Modeling Using the Graphical User Interface*

---

To set up the waveguide adapter model, the analysis contains two steps: First perform the boundary mode analysis on the elliptical port of the adapter, then solve the wave propagation problem using the mode shape as a boundary condition.

### *Mode Analysis*

---

#### **MODEL NAVIGATOR**

- 1 Select **3D** in the **Space dimension** list.
- 2 Select the **RF Module>Boundary Mode Analysis>TE Waves** application mode, click the **Multiphysics** button, and click the **Add** button to add the application mode to the model.
- 3 Then select the **RF Module>Electromagnetic Waves>Harmonic propagation** application mode from the tree view.
- 4 From the **Element** list, select **Vector - Quadratic**, and click the **Add** button.
- 5 Click **OK** to close the **Model Navigator**.

#### **GEOMETRY MODELING**

- 1 From the **File** menu, choose **Import>CAD Data From File**.
- 2 In the **Import CAD Data From File** dialog box, make sure that the **COMSOL Multiphysics file** or **All 3D CAD files** is selected in the **Files of type** list.
- 3 From the **models** directory under the COMSOL Multiphysics installation folder, locate the **waveguide\_adapter\_geometry.mphbin** file in the model path specified on page 91, and click **Import**.

First use the Boundary Mode Analysis application mode to compute the eigenmode of the elliptical end. This eigenmode is then part of the boundary condition for the electromagnetic wave propagation simulation.

## PHYSICS SETTINGS

First set up the boundary mode analysis problem. Select **Boundary Mode Analysis** from the **Multiphysics** menu.

### *Scalar Variables*

In the **Application Scalar Variables** dialog box, set the frequency `nu_rfwb` to `7e9`. This is a frequency above the cutoff frequency, which ensures that you find a propagating wave mode when doing the mode analysis.

### *Boundary Settings*

In the **Boundary Settings** dialog box, select all boundaries with `Ctrl+A` and clear the **Active in this domain** check box. Then select Boundary 6 and select the same check box again, so this is the only active boundary. This boundary is the elliptical end of the waveguide, where you perform the mode analysis.

### *Edge Settings*

In the **Edge Settings** dialog box select the **Perfect electric conductor** boundary condition at Edges 5, 6, 10, 14, 29, 39, 52, and 55. These are the exterior edges of the ellipse.

## MESH GENERATION

- 1 Open the **Free Mesh Parameters** dialog box from the **Mesh** menu.
- 2 Click the **Custom mesh size** button, and enter `0.006` in the **Maximum element size** edit field.
- 3 Click **OK**.

## COMPUTING THE SOLUTION

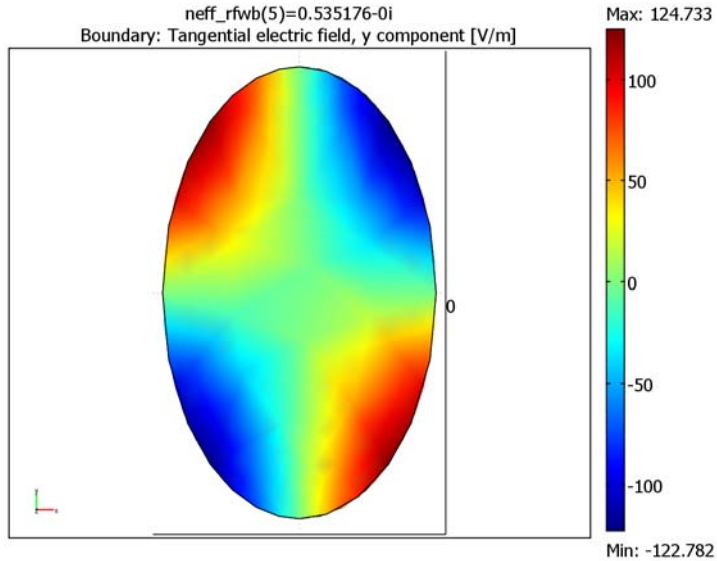
- 1 Open the **Solver Manager** dialog box from the **Solve** menu and select only the **Boundary Mode Analysis** application mode on the **Solve For** tab. Click **OK**.
- 2 Click the **Solve** button on the Main toolbar.

## POSTPROCESSING AND VISUALIZATION

Visualize the eigenmodes as a boundary plot:

- 1 In the **Plot Parameters** dialog box, clear the **Slice** check box and select the **Boundary** check box in the **Plot type** area on the General page.
- 2 Select `0.535` from the **Effective mode index** list.
- 3 On the **Boundary** tab, select the **Tangential electric field, x component**. There are two field variables one for each of the two application modes. Pick the `tEx_rfwb` variable.

- 4 Click **OK** to plot the field and click the **Go to XY View** button to better see the field (see Figure 3-13 on page 87)
- 5 Make the same type of plot of the  $y$  component of the electric field,  $tEy\_rfwb$ .

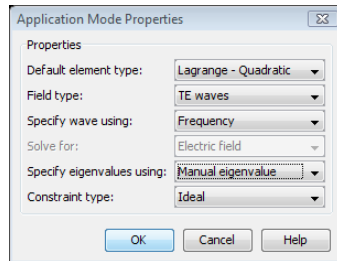


*Figure 3-17: The  $y$ -component of the electric field for the first eigenmode of the elliptical port.*

- 6 To see the normal component of the magnetic field, start by selecting **Normal magnetic field** from the **Predefined quantities** list.
- 7 On the **General** tab, enter 90 in the **Solution at angle (phase)** edit field. This compensates for the phase shift of  $90^\circ$  between the electric and the magnetic field.
- 8 Click **OK** to see the plot.

## FREEZING THE BOUNDARY MODE ANALYSIS APPLICATION MODE

- 1 From the **Physics** menu, choose **Properties** to open the **Application Mode Properties** dialog box. From the **Specify eigenvalues using** list, choose **Manual eigenvalue**.



- 2 Click **OK**.

When you perform this operation, you copy the propagation constant for the current plot to an application mode scalar variable for the propagation constant. All postprocessing variables now use this propagation constant instead of computing it from the eigenvalues. The application mode is in a state where it makes no sense to solve for eigenvalues. So before re-solving this application mode again, switch back to the original setting in the **Application Mode Properties** dialog box.

## *S-Parameter Analysis*

---

Now continue to set up the 3D problem to do the S-parameter analysis. Start by clicking the **Go to Default 3D View** button in order to see the full geometry.

### BOUNDARY SETTINGS

- 1 Choose **Electromagnetic Waves (rfw)** from the **Multiphysics** menu.
- 2 Open the **Boundary Settings** dialog box.
- 3 At the rectangular end, Boundary 13, select the **Port** boundary condition and enter the port number (1). Then click the **Port** tab to specify the mode at this port according to the following table:

DESCRIPTION	VALUE
Port number	1
Wave excitation at this port	selected
Mode specification	Rectangular
Mode type	Transverse electric (TE)
Mode number	10

- At the elliptical end (Boundary 6) also select the **Port** boundary condition and enter the port number (2) on the **Conditions** page. Then click the **Port** tab to specify the mode at this port according to the following table:

DESCRIPTION	VALUE
Port number	2
Wave excitation at this port	cleared
Mode specification	Numeric
Mode type	Automatic
Use numeric data from	Boundary Mode Analysis, TE Waves (rfwb)

- Use the default **Perfect electric conductor** boundary condition at all other boundaries.

### COMPUTING THE SOLUTION

- Open the **Solver Manager** dialog box. On the **Initial Value** tab, click the **Store Solution** button. In the dialog box that appears, select the solution with the effective mode index close to 0.535 and click **OK**.
- In the **Initial value** area, click the **Stored solution** option button.
- On the **Solve For** tab select only the **Electromagnetic Waves** application mode.
- Click **OK** to close the **Solver Manager**.
- In the **Solver Parameters** dialog box, select the **Parametric** solver.
- Set **Parameter name** to `freq` and **Parameter values** to `linspace(6.6e9, 1e10, 50)`.
- Click **OK** to close the **Solver Parameters** dialog box.
- Open the **Application Scalar Variables** dialog box from the **Physics** menu and clear the **Synchronize equivalent variables** check box. This allows you to set different values for the two frequency variables `nu_rfwb` and `nu_rfw`. `nu_rfwb` is the frequency variable for the Boundary Mode Analysis application mode. It is important that it retains its value of  $7 \cdot 10^9$  for the data from the mode analysis solution to be evaluated correctly.
- Set the frequency variable `nu_rfw` to `freq`. This is the frequency variable of the 3D Electromagnetic Waves application mode. Click **OK**.
- Click the **Solve** button on the Main toolbar.

### POSTPROCESSING AND VISUALIZATION

- To visualize the  $x$  component of the electric field in the waveguide, start by opening the **Plot Parameters** dialog box.

- 2 On the **General** tab, make sure that the **Solution at angle (phase)** text field reads 0.
- 3 Clear the **Boundary** check box, and select the **Slice** check box.
- 4 Click the **Slice** tab and select **Electromagnetic Waves (rfw)>Electric field, x component** from the **Predefined quantities** list.
- 5 In the **Slice positioning** area, set the number of **x levels**, **y levels**, and **z levels** to 1, 1, and 0 respectively.
- 6 Choose **wave** from the **Colormap** list.
- 7 Click **OK** to see the plot in Figure 3-14 on page 88.

The Port boundary conditions generate two S-parameter variables,  $S_{11\_rfw}$  and  $S_{21\_rfw}$  as well as two variables  $S_{11dB\_rfw}$  and  $S_{21dB\_rfw}$ , which are the S-parameters on a dB scale. The names of the variables reflect the port numbers that you entered in the **Boundary Settings** dialog box. Make a domain plot in a point to plot the S-parameters as functions of frequency:

- 1 On the **Point** tab in **Domain Plot Parameters** dialog box select Point 1 and select **Electromagnetic Waves (rfw)>S-parameter dB (S11)**. Click **Apply** to plot  $S_{11}$  on a dB scale.
- 2 When plotting  $S_{21}$  first deselect the first two data points on the **General** tab to better see the variation. Select the variable **Electromagnetic Waves (rfw)>S-parameter dB (S21)** and click **OK** to make the plot on page 90.

---

**Note:** When displaying S-parameter variables, you must set the phase of the solution to zero. Otherwise, the result displays show incorrect values. In most dialog boxes that are available on the **Postprocessing** menu, you can set the phase of the solution via the **Solution at angle (phase)** edit field. If the dialog box contains several tabs, it usually appears on the **General** tab.

---

### *Creating the Geometry in COMSOL Script or MATLAB*

To create the geometry, run COMSOL Script from within COMSOL Multiphysics or start COMSOL Multiphysics with MATLAB. Open a new 3D model by selecting **3D** from the **Space dimension** list in the Model Navigator and clicking **OK**. After running the script below from the command line, choose **File>Import>Geometry Objects**. In the dialog box that appears, enter the geometry object `g7`.

```
lx = 1.827/2; ly = 3.015/2; e1 = ellip2(lx,ly);
```



```

y0 = 1.14299999817202; x0 = sqrt((1-y0^2/ly^2)*lx^2);
p1 = point2(x0, y0); p2 = scale(p1,1,-1);
p3 = rotate(p1,pi); p4 = scale(p3,1,-1);
g1 = solid2(geomcsg({e1},{},{p1},{p2},{p3},{p4}));
lx2 = 1.016; ly2 = 2.286;
r1 = solid2(geomcsg({rect2(lx2,ly2,'Base','center')},...
{point2(0,-2.286/2)},{point2(1.016/2,0)},...
{point2(0,2.286/2)},{point2(-1.016/2,0)}));
z0 = 3.0; z1 = 11.0;
wrk0 = geomgetwrkpln('quick',{'xy',z0});
wrk1 = geomgetwrkpln('quick',{'xy',z1});
t = cell(1,8);
t{1} = 3; t{2} = 2; t{3} = 4; t{4} = 5;
t{5} = 6; t{6} = 7; t{7} = 8; t{8} = 1;
g2 = loft({r1 g1},'LoftEdge',t,...
'wrkpln',{wrk0 wrk1},'loftmethod','linear');
g3 = extrude(r1,'distance',3);
g4 = extrude (g1,'distance',3,'wrkpln',wrk1);
g5 = geomcomp({g2,g3,g4},'repairtol',1e-8);
g6 = geomdel(g5);
g7 = scale(g5,0.01,0.01,0.01);
geomplot(g7);

```

# Microwave Cancer Therapy

## *Introduction*

---

Electromagnetic heating appears in a wide range of engineering problems and is ideally suited for modeling in COMSOL Multiphysics because of its multiphysics capabilities. This example comes from the area of hyperthermic oncology and it models the electromagnetic field coupled to the bioheat equation. The modeling issues and techniques are generally applicable to any problem involving electromagnetic heating.

In hyperthermic oncology, cancer is treated by applying localized heating to the tumor tissue, often in combination with chemotherapy or radiotherapy. Some of the challenges associated with the selective heating of deep-seated tumors without damaging surrounding tissue are:

- Control of heating power and spatial distribution
- Design and placement of temperature sensors

Among possible heating techniques, RF and microwave heating have attracted much attention from clinical researchers. Microwave coagulation therapy is one such technique where a thin microwave antenna is inserted into the tumor. The microwaves heat up the tumor, producing a coagulated region where the cancer cells are killed.

This model computes the temperature field, the radiation field, and the specific absorption rate (SAR)—defined as the ratio of absorbed heat power and tissue density—in liver tissue when using a thin coaxial slot antenna for microwave coagulation therapy. It closely follows the analysis found in Ref. 1. It computes the temperature distribution in the tissue using the bioheat equation.

## *Model Definition*

---

Figure 3-18 shows the antenna geometry. It consists of a thin coaxial cable with a ring-shaped slot measuring 1 mm cut on the outer conductor 5 mm from the short-circuited tip. For hygienic purposes, the antenna is enclosed in a sleeve (catheter) made of PTFE (polytetrafluoroethylene). The following tables give the relevant

geometrical dimensions and material data. The antenna operates at 2.45 GHz, a frequency widely used in microwave coagulation therapy.

TABLE 3-1: DIMENSIONS OF THE COAXIAL SLOT ANTENNA.

PROPERTY	VALUE
Diameter of the central conductor	0.29 mm
Inner diameter of the outer conductor	0.94 mm
Outer diameter of the outer conductor	1.19 mm
Diameter of catheter	1.79 mm

TABLE 3-2: MATERIAL PROPERTIES.

PROPERTY	INNER DIELECTRIC OF COAXIAL CABLE	CATHETER	LIVER TISSUE
Relative permittivity	2.03	2.60	43.03
Conductivity			1.69 S/m

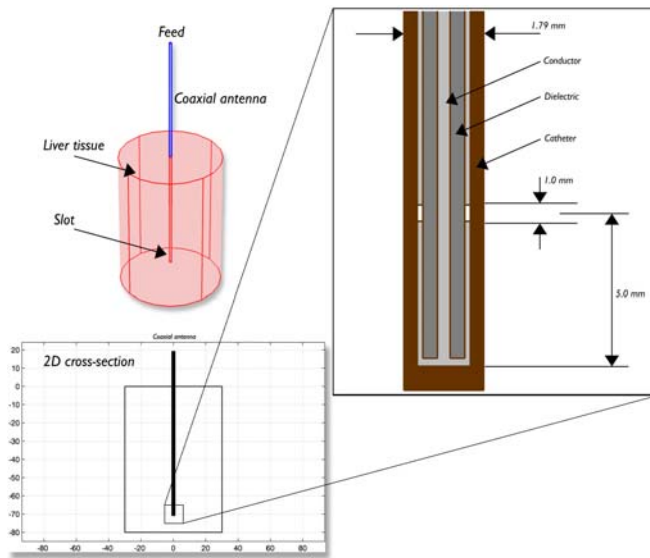


Figure 3-18: Antenna geometry for microwave coagulation therapy. A coaxial cable with a ring-shaped slot cut on the outer conductor is short-circuited at the tip. A plastic catheter surrounds the antenna.

The model takes advantage of the problem's rotational symmetry, which allows modeling in 2D using cylindrical coordinates as indicated in Figure 3-19. When

modeling in 2D, you can select a fine mesh and achieve excellent accuracy. The model uses a frequency-domain problem formulation with the complex-valued azimuthal component of the magnetic field as the unknown.

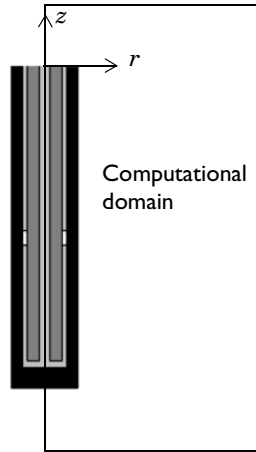


Figure 3-19: The computational domain appears as a rectangle in the  $rz$ -plane.

The radial and axial extent of the computational domain is in reality larger than indicated in Figure 3-19. This problem does not model the interior of the metallic conductors, and it models metallic parts using boundary conditions, setting the tangential component of the electric field to zero.

### DOMAIN AND BOUNDARY EQUATIONS—ELECTROMAGNETICS

An electromagnetic wave propagating in a coaxial cable is characterized by transverse electromagnetic fields (TEM). Assuming time-harmonic fields with complex amplitudes containing the phase information, the appropriate equations are

$$\mathbf{E} = \mathbf{e}_r \frac{C}{r} e^{j(\omega t - kz)}$$

$$\mathbf{H} = \mathbf{e}_\phi \frac{C}{rZ} e^{j(\omega t - kz)}$$

$$\mathbf{P}_{\text{av}} = \int_{r_{\text{inner}}}^{r_{\text{outer}}} \text{Re} \left( \frac{1}{2} \mathbf{E} \times \mathbf{H}^* \right) 2\pi r dr = \mathbf{e}_z \pi \frac{C^2}{Z} \ln \left( \frac{r_{\text{outer}}}{r_{\text{inner}}} \right)$$

where  $z$  is the direction of propagation, and  $r$ ,  $\phi$ , and  $z$  are cylindrical coordinates centered on the axis of the coaxial cable.  $\mathbf{P}_{\text{av}}$  is the time-averaged power flow in the cable,  $Z$  is the wave impedance in the dielectric of the cable, while  $r_{\text{inner}}$  and  $r_{\text{outer}}$  are

the dielectric's inner and outer radii, respectively. Further,  $\omega$  denotes the angular frequency. The propagation constant,  $k$ , relates to the wavelength in the medium,  $\lambda$ , as

$$k = \frac{2\pi}{\lambda}.$$

In the tissue, the electric field also has a finite axial component whereas the magnetic field is purely in the azimuthal direction. Thus, you can model the antenna using an axisymmetric transverse magnetic (TM) formulation. The wave equation then becomes scalar in  $H_\phi$ :

$$\nabla \times \left( \left( \epsilon_r - \frac{j\sigma}{\omega\epsilon_0} \right)^{-1} \nabla \times H_\phi \right) - \mu_r k_0^2 H_\phi = 0.$$

The boundary conditions for the metallic surfaces are

$$\mathbf{n} \times \mathbf{E} = 0.$$

The feed point is modeled using a port boundary condition with a power level set to 10 W. This is essentially a first-order low-reflecting boundary condition with an input field  $H_{\phi 0}$ :

$$\mathbf{n} \times \sqrt{\epsilon} \mathbf{E} - \sqrt{\mu} H_\phi = -2\sqrt{\mu} H_{\phi 0}$$

where

$$H_{\phi 0} = \frac{\sqrt{\frac{\mathbf{P}_{av} Z}{\pi r \ln\left(\frac{r_{outer}}{r_{inner}}\right)}}}{r}$$

for an input power of  $\mathbf{P}_{av}$  deduced from the time-average power flow.

The antenna radiates into the tissue where a damped wave propagates. Because you can discretize only a finite region, you must truncate the geometry some distance from the antenna using a similar absorbing boundary condition without excitation. Apply this boundary condition to all exterior boundaries. Finally, apply a symmetry boundary condition for boundaries at  $r = 0$ :

$$E_r = 0,$$

$$\frac{\partial E_z}{\partial r} = 0.$$

## DOMAIN AND BOUNDARY EQUATIONS—HEAT TRANSFER

The bioheat equation describes the stationary heat transfer problem as

$$\nabla \cdot (-k \nabla T) = \rho_b C_b \omega_b (T_b - T) + Q_{\text{met}} + Q_{\text{ext}}$$

where  $k$  is the liver's thermal conductivity ( $\text{W}/(\text{m}\cdot\text{K})$ ),  $\rho_b$  represents the blood density ( $\text{kg}/\text{m}^3$ ),  $C_b$  is the blood's specific heat capacity ( $\text{J}/(\text{kg}\cdot\text{K})$ ), and  $\omega_b$  denotes the blood perfusion rate ( $1/\text{s}$ ). Further,  $Q_{\text{met}}$  is the heat source from metabolism, and  $Q_{\text{ext}}$  is an external heat source, both measured in  $\text{W}/\text{m}^3$ .

This model neglects the heat source from metabolism. The external heat source is equal to the resistive heat generated by the electromagnetic field:

$$Q_{\text{ext}} = \frac{1}{2} \text{Re}[(\sigma - j\omega\epsilon) \mathbf{E} \cdot \mathbf{E}^*].$$

The model assumes that the blood perfusion rate is  $\omega_b = 0.0036 \text{ s}^{-1}$ , and that the blood enters the liver at the body temperature  $T_b = 37^\circ\text{C}$  and is heated to a temperature,  $T$ . The blood's specific heat capacity is  $C_b = 3639 \text{ J}/(\text{kg}\cdot\text{K})$ .

For a more realistic model, you might consider letting  $\omega_b$  be a function of the temperature. At least for external body parts such as hands and feet, it is evident that a temperature increase results in an increased blood flow.

This example models the heat-transfer problem only in the liver domain. Where this domain is truncated, it uses insulation, that is

$$\mathbf{n} \cdot \nabla T = 0.$$

### *Results and Discussion*

---

Figure 3-20 shows the resulting steady-state temperature distribution in the liver tissue for an input microwave power of 10 W. The temperature is highest near the antenna. It then decreases with distance from the antenna and reaches  $37^\circ\text{C}$  closer to the outer boundaries of the computational domain. The perfusion of relatively cold blood seems to limit the extent of the area that is heated.

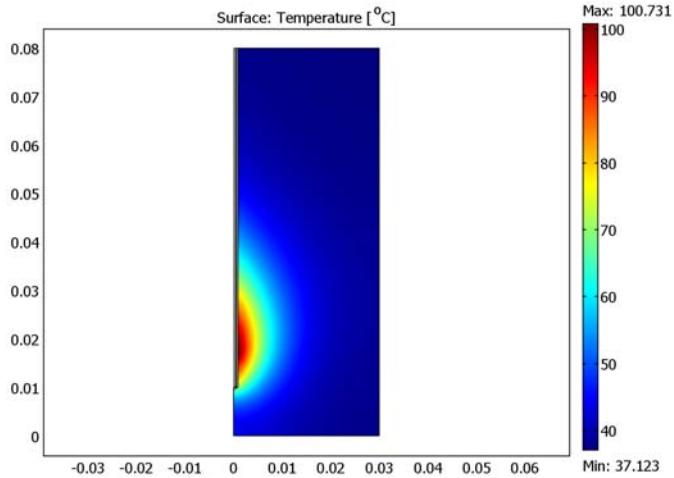


Figure 3-20: Temperature in the liver tissue.

Figure 3-21 shows the distribution of the microwave heat source. Clearly the temperature field follows the heat-source distribution quite well. That is, near the antenna the heat source is strong, which leads to high temperatures, while far from the antenna, the heat source is weaker and the blood manages to keep the tissue at normal body temperature.

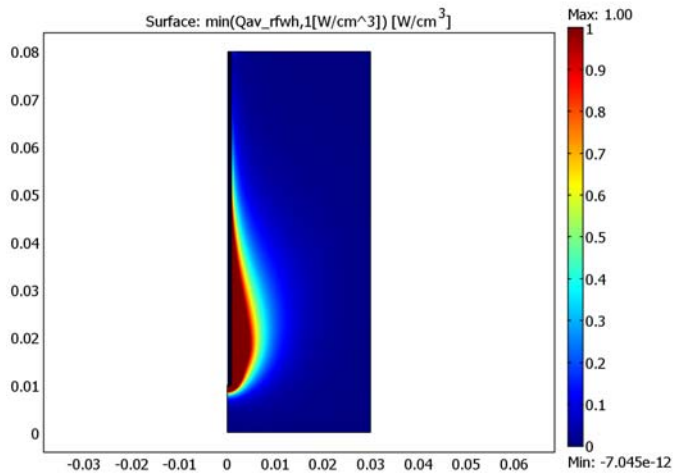


Figure 3-21: The computed microwave heat-source density takes on its highest values near the tip and the slot. The scale is cut off at  $1 \text{ W/cm}^3$ .

Figure 3-22 plots the specific absorption rate (SAR) along a line parallel to the antenna and at a distance of 2.5 mm from the antenna axis normalized by its maximal value along the line. The results are in good agreement with those found in Ref. 1.

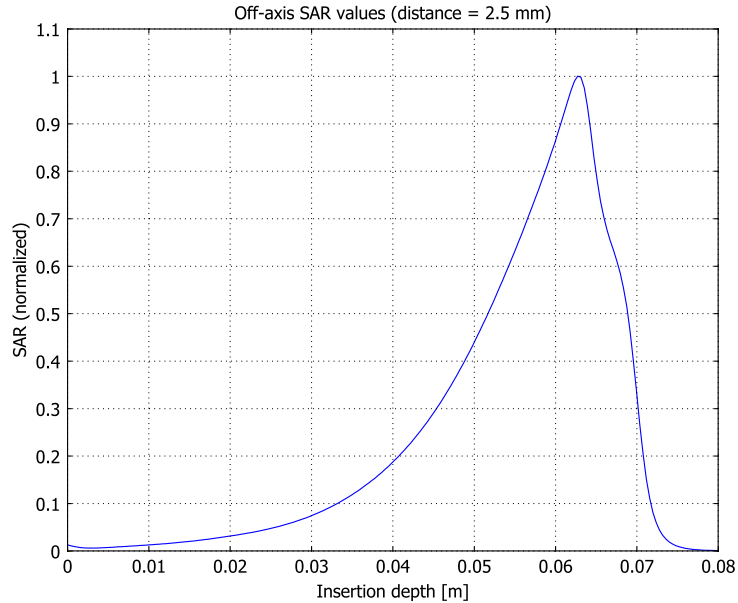


Figure 3-22: Normalized SAR value along a line parallel to the antenna and at a distance 2.5 mm from the antenna axis. The tip of the antenna is located at 70 mm, and the slot is at 65 mm.

### Reference

1. K. Saito, T. Taniguchi, H. Yoshimura, and K. Ito, “Estimation of SAR Distribution of a Tip-Split Array Applicator for Microwave Coagulation Therapy Using the Finite Element Method,” *IEICE Trans. Electronics*, vol. E84-C, 7, pp. 948–954, July 2001.

### Modeling in COMSOL Multiphysics

The COMSOL Multiphysics implementation is straightforward. Drawing the geometry is best done creating rectangles and setting their dimensions directly from the **Draw** menu. The scale differences together with the strong radial dependence of the electromagnetic fields make some manual adjustment of the mesh parameters necessary. In addition, 4th-order elements for the electromagnetic problem and a dense mesh in the dielectric result in well-resolved fields. The model computes the



solutions for both the electromagnetic problem and the heat transfer problem in parallel. This takes into account the coupling of the resistive heating from the electromagnetic solution into the bioheat equation. In principle, however, you could solve the two problems in sequence because there is only a 1-way coupling from the electromagnetic problem to the bioheat problem.

---

**Model Library path:** RF\_Module/RF\_and\_Microwave\_Engineering/  
microwave\_cancer\_therapy

---



---

**Note:** This model requires the RF Module and the Heat Transfer Module.

---

### *Modeling Using the Graphical User Interface*

---

#### **MODEL NAVIGATOR**

- 1 Open the **Model Navigator**. In the **Space dimension** list select **Axial symmetry 2D**.
- 2 In the list of application modes select **Heat Transfer Module>Bioheat Equation>Steady-state analysis**.
- 3 Click the **Multiphysics** button, then click the **Add** button.
- 4 In the list of application modes select **RF Module>Electromagnetic Waves>TM Waves>Harmonic propagation**.
- 5 In the **Element** list select **Lagrange - Quartic**.
- 6 Click **Add**, then click **OK**.

#### **OPTIONS AND SETTINGS**

From the **Options** menu select **Constants**. Enter the following names and expressions; when done, click **OK**.

NAME	EXPRESSION	EXPRESSION
k_liver	0.56[W/(kg*K)]	Thermal conductivity, liver
rho_blood	1000[kg/m^3]	Density, blood
C_blood	3639[J/(kg*K)]	Specific heat, blood
omega_blood	3.6e-3[1/s]	Blood perfusion rate

NAME	EXPRESSION	EXPRESSION
T_blood	37[degC]	Blood temperature
P_in	10[W]	Input microwave power
nu	2.45[GHz]	Microwave frequency
eps_diel	2.03	Relative permittivity, dielectric
eps_cat	2.6	Relative permittivity, catheter
eps_liver	43.03	Relative permittivity, liver
sig_liver	1.69[S/m]	Electric conductivity

### GEOMETRY MODELING

- 1 Create two rectangles. Select the menu item **Draw>Specify Objects>Rectangle**, then enter the following settings; when done with each one, click **OK**.

WIDTH	HEIGHT	BASE CORNER R	BASE CORNER Z
0.595e-3	0.01	0	0
29.405e-3	0.08	0.595e-3	0

- 2 From the Draw menu open the **Create Composite Object** dialog box. Clear the **Keep interior boundaries** check box. In the **Object selection** box select both rectangles, then click the **Union** button. Click **OK**.
- 3 Following the procedure in Step 1, specify two more rectangles with the following properties:

WIDTH	HEIGHT	BASE CORNER R	BASE CORNER Z
0.125e-3	1e-3	0.47e-3	0.0155
3.35e-4	0.0699	0.135e-3	0.0101

- 4 Add a line to the geometry. Select the menu item **Draw>Specify Objects>Line**. In the **r** edit field enter the coordinates 0 8.95e-4 8.95e-4, and in the **z** edit field enter the coordinates 9.5e-3 0.01 0.08. Click **OK**.
- 5 Finally specify another rectangle with these parameters:

WIDTH	HEIGHT	BASE CORNER R	BASE CORNER Z
1.25e-4	1e-3	4.7e-4	0.0155

### PHYSICS SETTINGS

#### *Subdomain Settings—Bioheat Equation*

- 1 From the **Multiphysics** menu select **1 Bioheat Equation (htbh)**.

- 2 From the **Physics** menu select **Subdomain Settings**.
- 3 Select Subdomains 2, 3, and 4, then clear the **Active in this domain** check box.
- 4 Select Subdomain 1, then enter the following settings; when done, click **OK**.

PROPERTY	VALUE
k (isotropic)	k_liver
$\rho_b$	rho_blood
$C_b$	C_blood
$\omega_b$	omega_blood
$T_b$	T_blood
$Q_{\text{met}}$	0
$Q_{\text{ext}}$	Qav_rfwh

*Boundary Conditions—Bioheat Equation*

- 1 From the **Physics** menu select **Boundary Settings**.
- 2 Select all the exterior boundaries (get them by pressing Ctrl+A, and note that the following step ignores the interior boundaries).
- 3 In the **Boundary condition** list select **Thermal insulation**, then click **OK**.

---

**Note:** Because the model neglects metabolic heat generation you set  $Q_{\text{met}}$  to 0. The variable Qav\_rfwh is a subdomain expression for the resistive heating provided by the TM Waves application mode.

---

*Scalar Variables—TM Waves*

- 1 From the **Multiphysics** menu select **2 TM Waves (rfwh)**.
- 2 From the **Physics** menu select **Scalar Variables** to open the **Application Scalar Variables** dialog box.
- 3 Find the variable nu\_rfwh and set its value to nu, then click **OK**.

*Boundary Conditions—TM Waves*

- 1 From the **Physics** menu select **Boundary Settings**.

- Specify boundary settings according to the following table (to enter the port settings for Subdomain 8 go to the **Port** page); when finished, click **OK**.

SETTINGS	SUBDOMAINS 1, 3	SUBDOMAINS 2, 14, 18, 20, 21	SUBDOMAIN 8
Boundary condition	Axial symmetry	Scattering boundary condition	Port
Wave excitation at this port			selected
$P_{in}$			$P_{in}$
Mode specification			Coaxial
Wave type		Spherical wave	

For the (exterior) boundaries not mentioned in the table, the default condition (perfect electric conductor) applies.

#### Subdomain Settings—TM Waves

- From the **Physics** menu select **Subdomain Settings**.
- Enter the following settings; when finished, click **OK**.

SETTINGS	SUBDOMAIN 1	SUBDOMAIN 2	SUBDOMAIN 3	SUBDOMAIN 4
$\epsilon_r$ (isotropic)	eps_liver	eps_cat	eps_diel	1
$\sigma$ (isotropic)	sig_liver	0	0	0
$\mu_r$	1	1	1	1

#### MESH GENERATION

- From the **Mesh** menu open the **Free Mesh Parameters** dialog box.
- Go to the **Global** page, click the **Custom mesh size** button and in the **Maximum element size** edit field type  $3e-3$ .
- Go to the **Subdomain** page and select Subdomain 3. In the **Maximum element size** edit field type  $1.5e-4$ .
- Click **Remesh**, then click **OK**.

#### COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar.

#### POSTPROCESSING AND VISUALIZATION

The default plot shows the temperature field. To change the unit to degrees Celsius, reproducing the plot in Figure 3-20, do as follows:

- 1 Click the **Plot Parameters** button on the Main toolbar.
- 2 Click the **Surface** tab. From the **Unit** list select **°C**, then click **Apply**.

The following steps describe how to visualize the resistive heating of the tissue:

- 1 In the **Predefined quantities** list select **TM Waves (rfwh)>Resistive heating, time average**. In the **Unit** edit field type  $W/cm^3$ , then click **Apply**.

Heating decreases rapidly in the liver tissue, resulting in an almost uniformly blue plot. To get a better feeling for the heating at a distance from the antenna, do as follows:

- 2 In the **Expression** edit field type  $\min(Q_{av\_rfwh}, 1 [W/cm^3])$ , then click **OK**.

In the resulting plot, which reproduces that in Figure 3-21, the region around the antenna in which the time-averaged resistive heating exceeds  $1 W/cm^3$  has a uniform, deep red color. Outside this region, you can read off the heating distribution from the color scale on the right.

To compute the total heating power deposited in the liver, follow these steps:

- 1 From the **Postprocessing** menu open the **Subdomain Integration** dialog box.
- 2 Select Subdomain 1. From the **Predefined quantities** list select **TM Waves (rfwh)>Resistive heating, time average**.
- 3 Select the **Compute volume integral (for axisymmetric modes)** check box. Click **OK**.

The result appears in the message log at the bottom of the user interface. The value of approximately 9.37 W indicates that the tissue absorbs most of the 10 W input power at stationary conditions.

These steps reproduce the plot in Figure 3-22, displaying the normalized SAR value:

- 1 From the **Postprocessing** menu open the **Cross-Section Plot Parameters** dialog box.
- 2 On the **Line/Extrusion** page, type  $Q_{av\_rfwh}/3.01 [W/cm^3]$  in the **Expression** edit field. In both the **r0** and **r1** edit fields type  $2.5e-3$ ; in the **z0** edit field type 0.08; and in the **z1** edit field type 0.
- 3 Click the **General** tab, then click the **Title/Axis** button. In the **Title/Axis Settings** dialog box, select the option button next to the **Title** edit field, then enter the title **Off-axis SAR values (distance = 2.5 mm)**.
- 4 In a similar way, enter the first axis label **Insertion depth [m]** and the second axis label **SAR (normalized)**, then click **OK** to close the **Title/Axis Settings** dialog box.
- 5 Click **OK** to generate the plot.

# Absorbed Radiation (SAR) in the Human Brain

Scientists use the SAR (specific absorption rate) to determine the amount of radiation that human tissue absorbs. This measurement is especially important for mobile telephones, which radiate close to the brain. The model studies how a human head absorbs a radiated wave from an antenna, and the temperature increase that the absorbed radiation causes.

## *Introduction*

---

The increasing use of wireless equipment has also increased the amount of radiation energy to which human bodies are exposed, and it is particularly important to avoid radiation into the brain. Experts continue to debate how dangerous this radiation might be. Almost everyone agrees, however, that it is important to minimize exposure to radiation. A common property that measures absorbed energy is the SAR value, calculated as

$$E_{\text{SAR}} = \sigma \frac{|\mathbf{E}|^2}{\rho}$$

where  $\sigma$  is the conductivity of human brain tissue,  $\rho$  is the density, and  $|\mathbf{E}|$  is the norm of the electric field. The SAR value is an average over a region of either 10 g or 1 g of brain tissue, depending on national rules. This model does not calculate the average value and so it refers to the local SAR value. The maximum local SAR value is always higher than the maximum SAR value.

## *Model Definition*

---

The human head geometry is the same geometry (SAM Phantom) provided by IEEE, IEC and CENELEC from their standard specification of SAR value measurements. The original geometry was imported into COMSOL Multiphysics after minor adjustments of the original geometry.

In addition, the model samples some material parameters with a volumetric interpolation function that estimates the variation of tissue type inside the head. The source data for this function comes directly from a file named

sar\_in\_human\_head\_interp.txt. That data file was created from a magnetic-resonance image (MRI) of a human head; these images contain 109 slices, each with  $256 \times 256$  voxels (Ref. 2). The use of the variation of the data in this file on the tissue material parameters has no scientific background, and this model simply implements it to illustrate a variation in conductivity, permittivity and perfusion rate as a function of the position inside the head. The model reduces the resolution of the volumetric data to  $55 \times 50 \times 50$  interpolation points, which matches the mesh-element density inside the head. Prior to generating the data file, the modeler in this case scaled, translated and rotated the 3D MRI data to match the form of the imported head geometry in COMSOL Multiphysics.

### WAVE PROPAGATION

The radiation comes from a patch antenna placed on the left side of the head. A line current on an edge act as an equivalent current source feeding the two patches of the antenna. To avoid reflections, the model makes use of PMLs; see “Perfectly Matched Layers (PMLs)” on page 43 in the *RF Module User’s Guide*. The model solves the Vector-Helmholtz equation everywhere in the domain for a certain frequency

$$\nabla \times \frac{1}{\mu_r} \nabla \times \mathbf{E} - k_0^2 \epsilon_r \mathbf{E} = \mathbf{0}$$

where  $\mu_r$  is the relative permeability,  $k_0$  is the free-space wave vector, and  $\epsilon_r$  is the permittivity for a vacuum.

For wave-propagation problems such as this one, you must limit the mesh size according to the problem’s minimum wavelength. Typically you need about five elements per wavelength to properly resolve the wave.

This example takes material properties for the human brain from a presentation by G. Schmid (Ref. 1). The following table reviews some important frequency-dependent properties in this publication. The interpolation function samples these values to create a realistic variation.

PARAMETER	FREQUENCY	VALUE	DESCRIPTION
$\sigma$	835 MHz	1.35 S/m	Conductivity
$\epsilon_r$	835 MHz	56	Relative permeability

### HEATING OF THE HEAD

The bioheat equation models the heating of the head with a heating loss due to the blood flow. This heat loss depends on the heat capacity and density of the blood, and

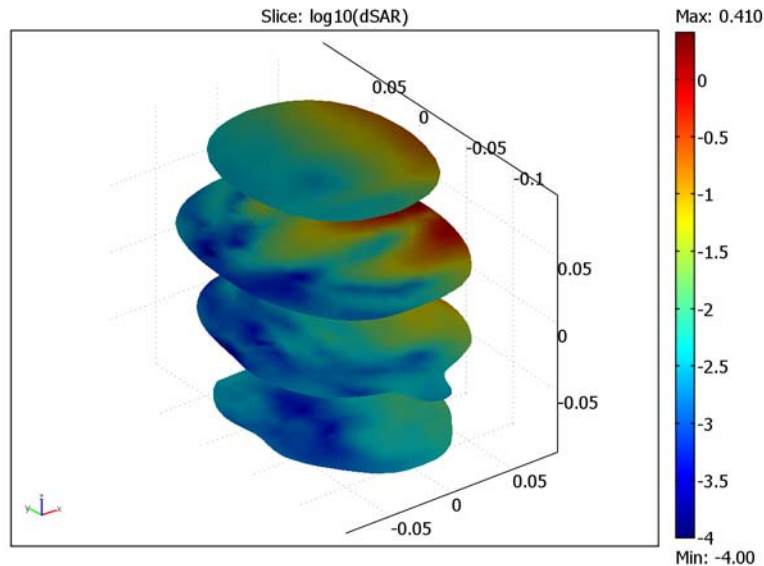
on the blood perfusion rate. The perfusion rate varies significantly in different parts of the human body, and the table below presents the values used here.

PART	PERFUSION RATE
Brain	$2 \cdot 10^{-3}$ (ml/s)/ml
Bone	$3 \cdot 10^{-4}$ (ml/s)/ml
Skin	$3 \cdot 10^{-4}$ (ml/s)/ml

The same interpolation function used for the electric parameters also models the difference in perfusion rate between the brain tissue inside the head and the outer parts of skin and bone. Note again that the use of the interpolation function does not have any physical relevance, it is just to show a realistic effect of a varying material parameter.

### *Results and Discussion*

The model studies the local SAR value in the head using the formula described earlier for the frequency 835 MHz. The SAR value is highest close to the surface of the head facing the incident wave. The differences in electrical properties become visible if you plot the local SAR value on a log scale.



*Figure 3-23: Log-scale slice plot of the local SAR value.*



The bioheat equation produces a similar plot for the heating of the head, which is highest closest to the antenna. The maximum temperature increase (from 37 °C) is less than 0.2 °C, and drops rapidly inside the head.

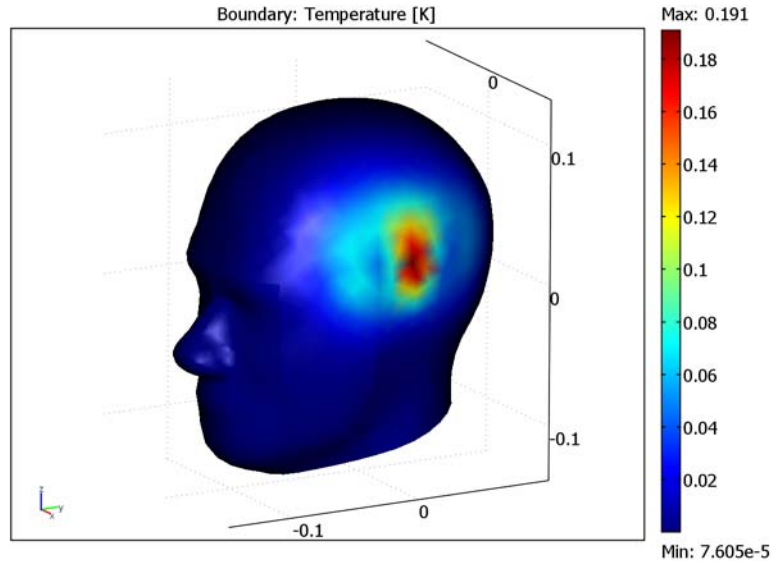


Figure 3-24: The local increase in temperature at the surface has a maximum of 0.2 °C right beneath the antenna.

### References

1. G. Schmid, G. Neubauer, P.R. Mazal, and F. Alesch, *Dielectric properties of brain tissue: measurements on humans*, <http://www.cost281.org/download.php?fid=531>.
2. M. Levoy, MRI data originally from Univ. of North Carolina (downloaded from the Stanford volume data archive at <http://graphics.stanford.edu/data/voldata/>).

---

**Model Library path:** RF\_Module/RF\_and\_Microwave\_Engineering/  
sar\_in\_human\_head

---

**Note:** This model requires both the RF Module and the Heat Transfer Module.

---

#### MODEL NAVIGATOR

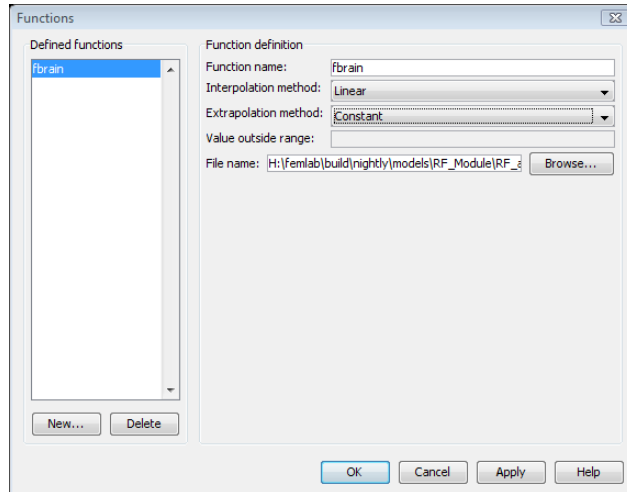
- 1 In the **Model Navigator**, select **3D** in the **Space dimension** list.
- 2 In the **RF Module** folder, select **Electromagnetic Waves>Harmonic propagation**.
- 3 Click the **Multiphysics** button.
- 4 Click the **Add** button.
- 5 In the **Heat Transfer Module** folder, select **Bioheat Equation>Steady-state analysis**.
- 6 In the **Dependent variables** edit field, type  $\Delta T$ . Click **Add**.
- 7 Click **OK**.

#### OPTIONS AND SETTINGS

- 1 From the **Options** menu, choose **Constants**.
- 2 In the **Constants** dialog box, define the following constants with names, expressions, and descriptions (optional):

NAME	EXPRESSION	DESCRIPTION
epsilon <sub>nr_pcb</sub>	5.23	Permittivity for the patch antenna board
epsilon <sub>r0_brain</sub>	56	Permittivity for the brain tissue
sigma <sub>0_brain</sub>	1.35[S/m]	Conductivity for the brain tissue
rho <sub>brain</sub>	1.03e3[kg/m <sup>3</sup> ]	Density of brain tissue
sdamping	2e-4	Sampling parameter
edamping	4e-4	Sampling parameter
soffset	-1.0[S/m]	Sampling parameter
eoffset	-50	Sampling parameter
rho <sub>blood</sub>	1000[kg/m <sup>3</sup> ]	Density of blood
c <sub>blood</sub>	3639[J/(kg*K)]	Heat capacity of blood
odamping	1.08e-6[1/s]	Sampling parameter
ooffset	-7.8e-4[1/s]	Sampling parameter

- 3 Click **OK**.
- 4 From the **Options** menu, choose **Functions**.
- 5 In the **Functions** dialog box, click the **New** button.
- 6 In the **New Function** dialog box, type `fbrain` in the **Function name** edit field, select the **Interpolation** radio button, and select **File** from the **Use data from** drop-down menu.
- 7 Click the **Browse** button and locate the file named `sar_in_human_head_interp.txt`. Select the file and click **Open** to use it. Click **OK** to create the interpolation function.
- 8 From the **Extrapolation method** list, select **Constant**.
- 9 Click **OK**.



## GEOMETRY MODELING

The head geometry has been created outside COMSOL Multiphysics, so you import it from an `mphbin` file (COMSOL's native geometry format). Then create the surrounding domains of PMLs, air, and antenna manually.

- 1 From the **File** menu, choose **Import>CAD Data From File**.
- 2 In the **Import CAD Data From File** dialog box, make sure that **COMSOL Multiphysics file** or **All 3D CAD files** is selected in the **Files of type** list.
- 3 Locate the file `sar_in_human_head.mphbin`, then click **Import**.
- 4 From the **Draw** menu, choose **Work-Plane Settings**.

- 5 In the **Work-Plane Settings** dialog box, type -0.3 in the **z** edit field, and click **OK**.
- 6 From the **Draw** menu, choose **Specify Objects>Rectangle**.
- 7 In the **Rectangle** dialog box, specify a rectangle with the following settings and click **OK** when done.

WIDTH	HEIGHT	X	Y
0.4	0.4	-0.2	-0.2-0.03

- 8 Specify two more rectangles with the following settings:

WIDTH	HEIGHT	X	Y
0.1	0.1	-0.3	-0.3-0.03
0.6	0.6	-0.3	-0.3-0.03

- 9 Select the smaller rectangle with label R2.
- 10 Click the **Array** button on the Draw toolbar. In the dialog box that appears, fill in the following fields. Click **OK** when done.

X (DISPLACEMENT)	Y (DISPLACEMENT)	X (ARRAY SIZE)	Y (ARRAY SIZE)
0.5	0.5	2	2

- 11 Select all objects and click the **Union** button on the Draw toolbar.
- 12 From the **Draw** menu, choose **Extrude**.
- 13 Enter 0.1 0.5 0.6 in the **Distance** edit field and click **OK**.

You have now created the surrounding air and the PML regions. Continue with the patch antenna close to the head.

- 14 Click the **Block** toolbar button.
- 15 In the **Block** dialog box, click the **Center** option button and fill in the following fields. Click **OK** when done.

X (LENGTH)	Y (LENGTH)	Z (LENGTH)	AXIS BASE POINT (X, Y, Z)
0.004	0.08	0.08	(0.1, 0, 0.05)

- 16 From the **Draw** menu, choose **Work-Plane Settings**.
- 17 In the **Work-Plane Settings** dialog box, click the **Face Parallel** tab. Expand the **BLK1** object, and choose face number 2.
- 18 Click **OK**. You now see a work plane parallel to the face of the block facing the head.

19 Specify two rectangles with the following settings.

WIDTH	HEIGHT	X	Y
0.06	0.06	0.01	0.01
0.005	0.01	0.04-0.0025	0

20 Select both rectangles and click the **Union** button on the Draw toolbar.

21 Click the **Delete Interior Boundaries** toolbar button to delete the internal boundary between the rectangles.

22 From the **Draw** menu, choose **Embed**, and click **OK** in the dialog box that appears. This places the drawn object onto the face of the block in the 3D geometry.

23 From the **Draw** menu, choose **Work-Plane Settings**.

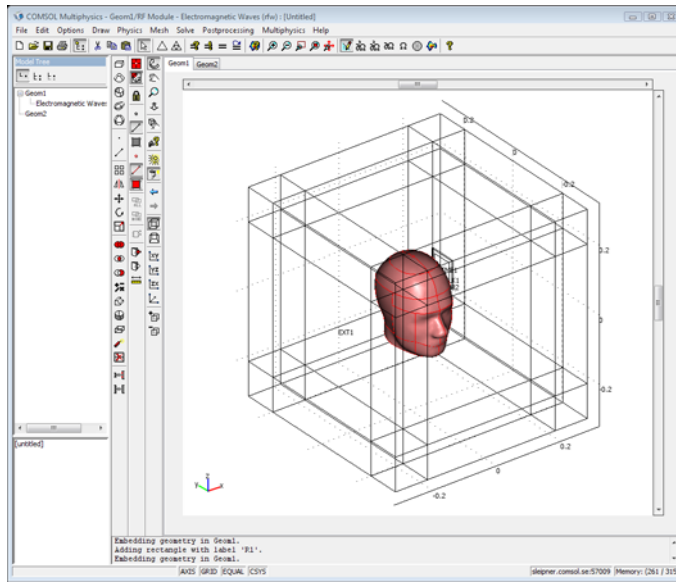
24 In the **Work-Plane Settings** dialog box, click the **Face Parallel** tab. Expand the **BLK1** object, and choose face 1. Click **OK**.

25 Specify one rectangle with the following settings.

WIDTH	HEIGHT	X	Y
0.004	0.005	0	0.04-0.0025

26 From the **Draw** menu, choose **Embed**. Click **OK**.

- 27 Click the **Zoom Extents** button on the Main toolbar and the **Headlight** toolbar button on the Camera toolbar. Select the head geometry objects to see the geometry plotted below.



## PHYSICS SETTINGS (ELECTROMAGNETIC WAVES)

- 1 From the **Multiphysics** menu, choose **Electromagnetic Waves (rfw)**.
- 2 From the **Physics** menu, choose **Application Scalar Variables**.
- 3 In the **Scalar Variables** dialog box, change the value of the frequency, nu\_rfw, to 835 [MHz].
- 4 Click **OK**.
- 5 From the **Options** menu, select **Expressions>Subdomain Expressions**.
- 6 In the **Subdomain Expressions** dialog box, define variables according to the following table:

SETTINGS	SUBDOMAIN 19
dSAR	Qav_rfw/rho_brain
sigma_brain	sigma0_brain*(1+fbrain(x[1/m],y[1/m],z[1/m])*sdamping)+soffset

SETTINGS	SUBDOMAIN 19
epsilon_r_brain	epsilon_r0_brain*(1+fbrain(x[1/m],y[1/m],z[1/m])*edamping)+eoffset
omega_head	fbrain(x[1/m],y[1/m],z[1/m])*odamping+ooffset

7 Click **OK**.

#### *Boundary Conditions*

- 1 From the **Physics** menu, open the **Boundary Settings** dialog box.
- 2 Select all exterior boundaries (use the **Select by group** check box) and choose the **Scattering boundary condition** from the **Boundary condition** list.
- 3 Select the **Interior boundaries** check box to allow settings on interior boundaries. Make sure that the **Select by group** check box is cleared.
- 4 Select Boundaries 92 and 96 (patches on the antenna) and choose **Perfect electric conductor** from the **Boundary condition** list.
- 5 Select Boundary 93 (boundary between patches) and choose **Lumped port** from the **Boundary condition** list.
- 6 Select the **Voltage generator at this port** check box.
- 7 Enter 45.5 in the **V<sub>in</sub>** edit field.
- 8 Click the **Port** tab and type 75 in the **Z<sub>ref</sub>** edit field.
- 9 Click **OK**. The voltage applied to the lumped port is quite high due to the impedance mismatch between the antenna and the external cable impedance, which is set to 75  $\Omega$ .

#### *Subdomain Settings*

- 1 From the **Physics** menu, open the **Subdomain Settings** dialog box.
- 2 Define the subdomain settings according to the following table:

SETTINGS	SUBDOMAIN 19	SUBDOMAIN 20
$\epsilon_r$	epsilon_r_brain	epsilon_r_pcb
$\sigma$	sigma_brain	0

- 3 Click the **PML** tab and define the PML regions according to the table below. Note that the subdomain selections are overlapping. Make sure that all selections, except those specified as “As is,” are white and not yellow. A yellow color means that the

property has different values among the selected subdomains. You fix this by reselecting the property to the specified value.

SETTINGS	SUBDOMAINS 1–9, 21–29	SUBDOMAINS 1–3, 7–12, 16–18, 21–23, 27–29	SUBDOMAINS 1, 3, 4, 6, 7, 9, 10, 12, 13, 15, 16, 18, 21, 23, 24, 26, 27, 29
Type of PML	Cartesian	Cartesian	Cartesian
Absorbing in x direction	Selected	As is	As is
Width in x direction	2*Sdx_guess_rfw	As is	As is
Absorbing in y direction	As is	Selected	As is
Width in y direction	As is	2*Sdy_guess_rfw	As is
Absorbing in z direction	As is	As is	Selected
Width in z direction	As is	As is	2*Sdz_guess_rfw

4 Click **OK**.

#### PHYSICS SETTINGS (BIOHEAT EQUATION)

From the **Multiphysics** menu, choose **Bioheat Equation (htbh)**.

##### *Boundary Conditions*

The default boundary condition, thermal insulation, applies everywhere.

##### *Subdomain Settings*

- 1 From the **Physics** menu, open the **Subdomain Settings** dialog box.
- 2 Select all subdomains except Subdomain 19, and clear the **Active in this domain** check box.
- 3 Then select Subdomain 19, and enter the following expressions.

PARAMETER	EXPRESSION
$\rho_b$	rho_blood
$C_b$	c_blood
$\omega_b$	omega_head
$T_b$	0
$Q_{ext}$	Qav_rfw



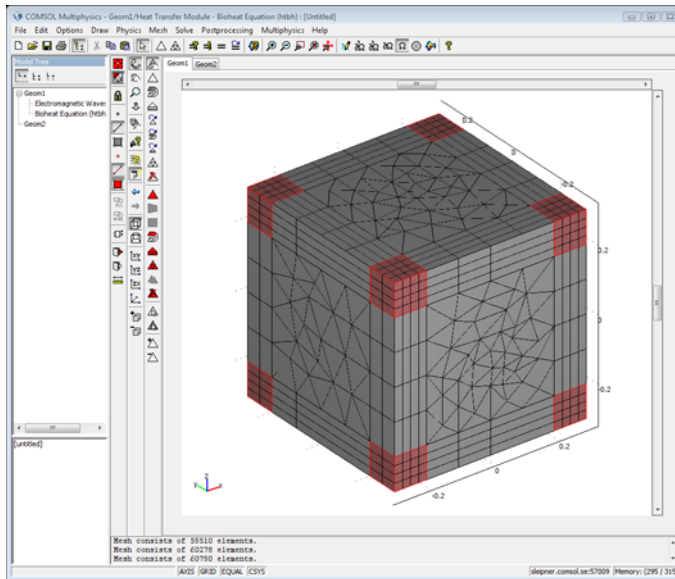
- 4 Click the **Init** tab and type 0 in the **dt(t<sub>0</sub>)** edit field.
- 5 Click **OK**.

### **MESH GENERATION**

The steps to generate the mesh involve interactive meshing, where you use the standard free meshing for the head, patch, and surrounding air. For the PML regions, use swept meshing. This gives more control of the mesh resolution in the absorbing direction, which is crucial to get convergence with iterative solvers.

- 1 From the **Mesh** menu, choose **Free Mesh Parameters**.
- 2 Select **Coarser** from the **Predefined mesh sizes** list.
- 3 Click the **Edge** tab, select all edges of the patch facing the head (Edges 132–135, 137–138, 140–142, and 149). Enter  $2e-3$  in the **Maximum element size** edit field.
- 4 Click the **Boundary** tab, select Boundaries 73 and 78, enter 0.4 in the **Mesh curvature factor** edit field, and enter 0.01 in the **Mesh curvature cutoff** edit field.
- 5 Click the **Subdomain** tab, select Subdomain 19 and enter 0.015 in the **Maximum element size** edit field.
- 6 Select Subdomains 14, 19, and 20, and click the **Mesh Selected** button.
- 7 Click **OK** to close the **Free Mesh Parameters** dialog box.
- 8 Open the **Swept Mesh Parameters** dialog box from the **Mesh** menu.
- 9 Select the Subdomains 5, 11, 13, 15, 17, and 25. These are the PML regions with absorption in a single direction.
- 10 Select the **Manual specification of element layers** check box, and enter 4 in the **Number of element layers** edit field.
- 11 Click the **Mesh Selected** button.
- 12 Select the Subdomains 2, 4, 6, 8, 10, 12, 16, 18, 22, 24, 26, and 28. These are the PML regions with absorption in two directions.
- 13 Click the **Mesh Selected** button. This creates hex elements in the selected regions.
- 14 Click the **Select Remaining** button and then click the **Mesh Selected** button. This creates hex elements in the remaining PML regions.

15 Click **OK** to close the dialog box. The final mesh is a mix between tetrahedron elements and prism elements.



## COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar.

The model solves with the default solver settings, which is the GMRES solver with the GMG preconditioner. It solves the nonlinear problem including both electromagnetic waves and the bioheat equation, which requires a little bit more than 2 GB of memory to solve. It is possible to save some memory and solution time by first solving for the electromagnetic waves and then for the temperature because the system is only one-way coupled. Specify the variables to solve for from the **Solver Manager** dialog box.

## POSTPROCESSING AND VISUALIZATION

- 1 Select **Plot Parameters** from the **Postprocessing** menu.
- 2 On the **General** page, make sure that the **Slice** check box is selected, and that the **Geometry edges** check box is cleared.
- 3 Click the **Slice** tab, and type  $\log_{10}(dSAR)$  in the **Expression** edit field.
- 4 Enter 0 in the **x-levels** edit field, and 10 in the **z-levels** edit field.
- 5 Click **Apply**. You should now see the plot in Figure 3-23 on page 112.
- 6 Under the **Slice** tab, clear the **Slice plot** check box.

- 7 Click the **Boundary** tab, and enter  $dT$  in the **Expression** edit field.
- 8 Click **OK** to see the plot in Figure 3-24 on page 113.

# Microwave Oven

## *Introduction*

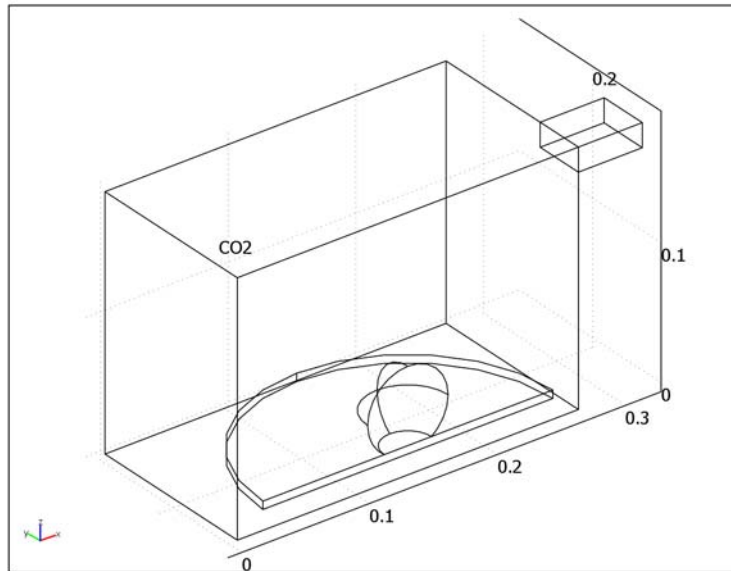
---

This is a model of the heating process in a microwave oven. The distributed heat source is computed in a stationary, frequency domain electromagnetic analysis followed by a transient heat transfer simulation showing how the heat redistributes in the food.

## *Model Definition*

---

The microwave oven is a metallic box connected to a 1 kW, 2.45 GHz microwave source via a rectangular waveguide operating in the  $TE_{10}$  mode. Near the bottom of the oven there is a cylindrical glass plate with a spherical potato placed on top of it. A part of the potato is cut away for mechanical stability which also facilitates the creation of a finite element mesh in the region where it is in contact with the plate. Symmetry is utilized by simulating only half of the problem. The symmetry cut is applied vertically through oven, waveguide, potato and plate. The reduced geometry is shown in Figure 3-12.



*Figure 3-25: Geometry of microwave oven, potato, and waveguide feed.*

The walls of the oven and the waveguide are good conductors. The model approximates these walls as perfect conductors, represented by the boundary condition  $\mathbf{n} \times \mathbf{E} = \mathbf{0}$ . The symmetry cut has mirror symmetry for the electric field and is represented by the boundary condition  $\mathbf{n} \times \mathbf{H} = \mathbf{0}$ .

The rectangular port is excited by a transverse electric (TE) wave, which is a wave that has no electric field component in the direction of propagation. At an excitation frequency of 2.45 GHz, the TE<sub>10</sub> mode is the only propagating mode through the rectangular waveguide. The cutoff frequencies for the different modes are given analytically from the relation

$$(v_c)_{mn} = \frac{c}{2} \sqrt{\left(\frac{m}{a}\right)^2 + \left(\frac{n}{b}\right)^2}$$

where  $m$  and  $n$  are the mode numbers and  $c$  denotes the speed of light. For the TE<sub>10</sub> mode,  $m = 1$  and  $n = 0$ . With the dimensions of the rectangular cross section ( $a = 7.8$  cm and  $b = 1.8$  cm), the TE<sub>10</sub> mode is the only propagating mode for frequencies between 1.92 GHz and 3.84 GHz.

With the stipulated excitation at the rectangular port, the following equation is solved for the electric field vector  $\mathbf{E}$  inside the waveguide and oven:

$$\nabla \times (\mu_r^{-1} \nabla \times \mathbf{E}) - k_0^2 \left( \epsilon_r - \frac{j\sigma}{\omega \epsilon_0} \right) \mathbf{E} = 0$$

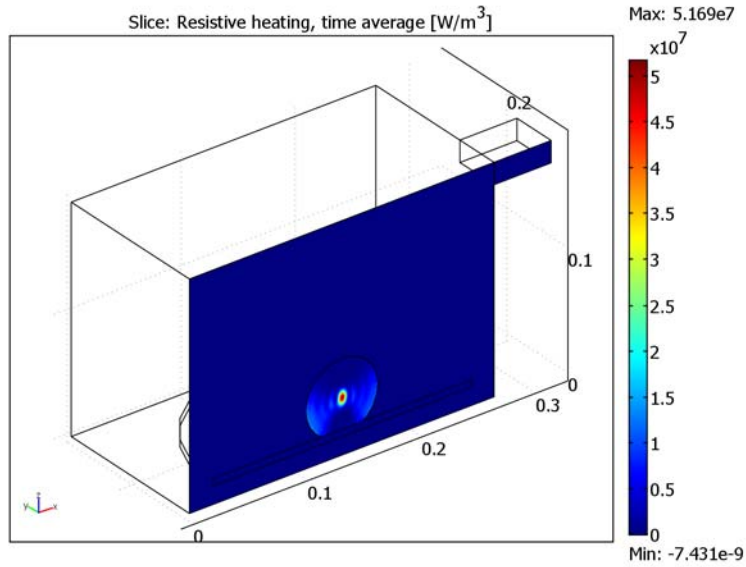
where  $\mu_r$  denotes the relative permeability,  $j$  the imaginary unit,  $\sigma$  the conductivity,  $\omega$  the angular frequency,  $\epsilon_r$  the relative permittivity, and  $\epsilon_0$  the permittivity of free space. The model uses material parameters for air:  $\sigma = 0$  and  $\mu_r = \epsilon_r = 1$ . In the potato the same parameters are used except for the permittivity which is set to  $\epsilon_r = 65 - 20j$  where the imaginary part accounts for dielectric losses. The glass plate has  $\sigma = 0$ ,  $\mu_r = 1$  and  $\epsilon_r = 2.55$ .

## Results

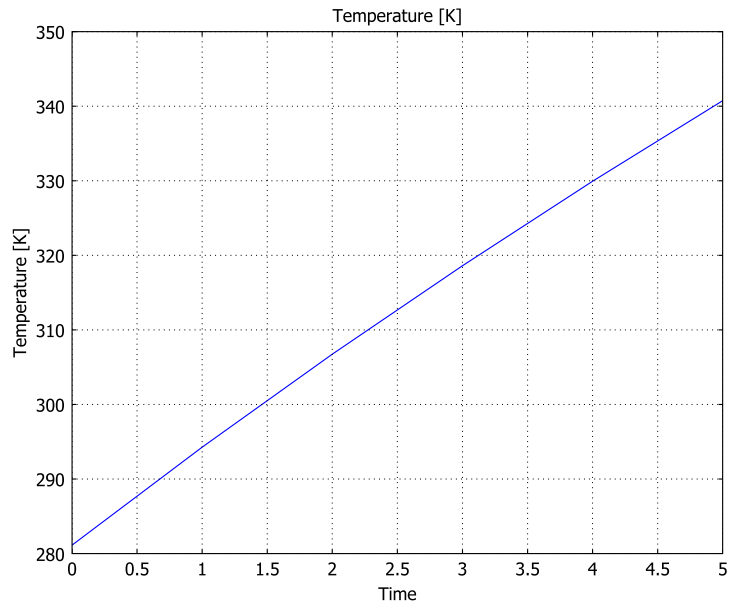
---

The first figure below shows the distributed microwave heat source as a slice plot through the center of the potato. From the rather complicated oscillating pattern that has a strong peak in the center, it is obvious that the potato acts as a resonant cavity for the microwave field. The voltage reflection coefficient at the waveguide port,  $S_{11}$ , evaluates to about  $-4$ dB, meaning that the potato absorbs about 60% of the input microwave power. The second figure shows the temperature in the center of the potato as a function of time for the first 5 seconds. Due to the low thermal conductivity of the

potato, the heat distributes rather slowly and the temperature profile after 5 seconds has a strong peak in the center, see the third figure. When heating the potato further, the temperature in the center eventually reaches 100 °C and the water contents start boiling, drying out the center and transporting heat as steam to outer layers. This also affects the electromagnetic properties of the potato. The simple microwave absorption and heat conduction model used here does not capture these nonlinear effects. However, the model can serve as a starting point for a more advanced analysis.



*Figure 3-26: Dissipated microwave power distribution.*



*Figure 3-27: Temperature in the center of the potato during the first 5 seconds of heating.*

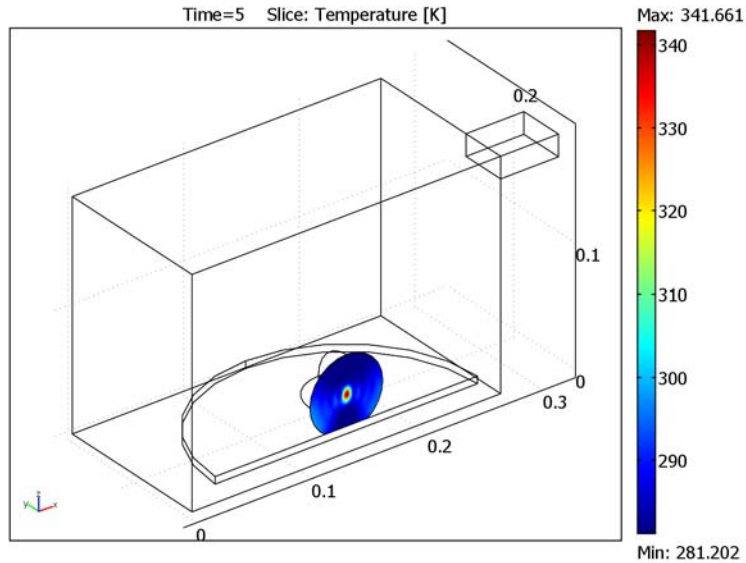


Figure 3-28: Temperature distribution after 5 seconds of heating.

### *Modeling in COMSOL Multiphysics*

This model uses the RF Module's Port boundary condition for the wave propagation problem. With this boundary condition, the S-parameters are calculated automatically. The mode of the incoming wave is specified.

**Model Library path:** RF\_Module/RF\_and\_Microwave\_Engineering/  
microwave\_oven

### *Modeling Using the Graphical User Interface*

The model is set up in two steps: first the wave propagation problem is solved, then the heat transfer problem is solved.



### MODEL NAVIGATOR

- 1 Select **3D** in the **Space dimension** list.
- 2 Select the **RF Module>Electro-Thermal Interaction>Microwave Heating>Transient analysis** application mode from the tree view.
- 3 Click **OK** to close the **Model Navigator** dialog box.

### OPTIONS AND SETTINGS

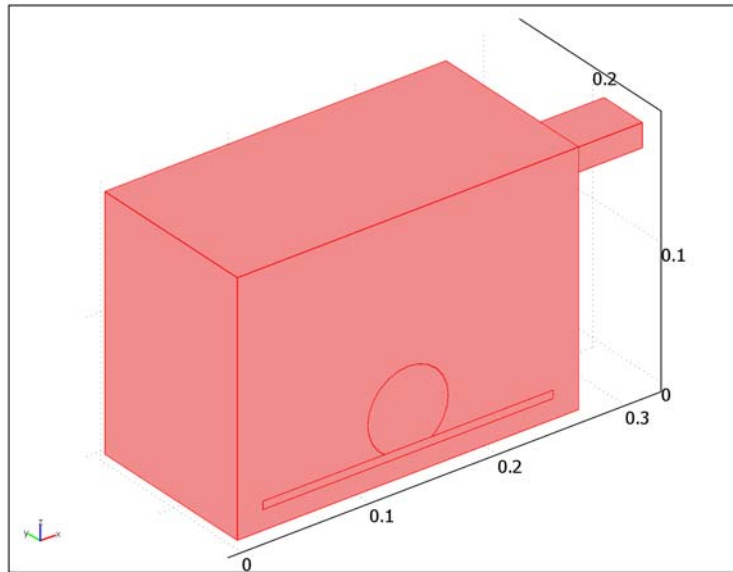
Define the following expression in the **Scalar Expressions** dialog box. The description field is optional and can be omitted.

NAME	EXPRESSION	DESCRIPTION
nu_co	$c0\_rfw / (2 * 7.8 [cm])$	Cutoff frequency

### GEOMETRY MODELING

- 1 Go to the **Draw** menu and click the block symbol and define a block with **Length x:** 0.267, **Length y:** 0.270, **Length z:** 0.188. Leave the other parameters at their defaults.
- 2 Click **OK**.
- 3 Click the block symbol again and define a block with **Length X:** 0.050, **Length Y:** 0.078, **Length Z:** 0.018, **Axis base point x:** 0.267, **Axis base point y:** (0.270-0.078) / 2, **Axis base point z:** 0.188-0.018. Leave the other parameters at their defaults.
- 4 Click **OK**.
- 5 Click the **Zoom Extents** button to get a better view of the geometry.
- 6 In the **Draw** menu choose **Cylinder**, then define a cylinder with **Radius:** 0.227/2, **Height:** 0.006, **Axis base point x:** 0.267/2, **Axis base point y:** 0.270/2, **Axis base point z:** 0.021-0.006. Leave the other parameters at their default values.
- 7 Click **OK**.
- 8 While the cylinder is selected, copy it by pressing Ctrl+C and paste the copy, which you will need later, by pressing Ctrl+V. Use the default displacements of (0, 0, 0).
- 9 In the **Draw** menu choose **Sphere**, then define a sphere with **Radius:** 0.063/2, **Axis base point x:** 0.267/2, **Axis base point y:** 0.270/2, **Axis base point z:** 0.021-0.006+0.063/2. Leave the other parameters at their default values.
- 10 Click **OK**.

- 11** In the **Draw** menu choose **Create Composite Object**. In the **Set formula** edit field type SPH1 - CYL2.
- 12** Click **OK**.
- 13** In the **Draw** menu choose **Block**, then define a block with **Length X:**  $0.267+0.050$ , **Length Y:**  $0.270/2$ , **Length Z:**  $0.188$ . Leave the other parameters at their default values.
- 14** Click **OK**.
- 15** In the **Draw** menu choose **Create Composite Object**. In the **Set formula** edit field, type  $(BLK1+BLK2+CYL1+C01) - BLK3$ .
- 16** Click **OK**.
- 17** Click the **Zoom Extents** button on the Main toolbar.



### PHYSICS SETTINGS

First set up the electromagnetic problem. Select **Electromagnetic Waves (rfw)** from the **Multiphysics** menu.

#### *Scalar Variables*

In the **Scalar Variables** dialog box, set the frequency  $\nu_{rfw}$  to  $2.45e9$ .

### Subdomain Settings

- 1 Open the **Subdomain Settings** dialog box.
- 2 Keep all the subdomains at the default settings (air) except for Subdomain 2 (glass plate) which has a relative permittivity of 2.55 and Subdomain 3 (potato) which has a relative permittivity of  $65 - i*20$ .
- 3 Click **OK**.

### Boundary Conditions

- 1 Open the **Boundary Settings** dialog box.
- 2 At the waveguide end, Boundary 23, select the **Port** boundary condition. On the **Port** tab set the values according to the table below:

DESCRIPTION	VALUE
Port number	1
Wave excitation at this port	selected
$P_{in}$	500
Mode specification	User defined
Mode type	Transverse electric (TE)
$E_0$	$0 \ 0 \ \cos(\pi*(y-13.5[\text{cm}])/7.8[\text{cm}])$
Specify cutoff frequency	selected
$\nu_{cutoff}$	nu_co

- 3 Use the default **Perfect electric conductor** boundary condition at all other boundaries except at the symmetry boundaries 2, 7, 10, and 19 where the **Perfect magnetic conductor** boundary condition is applied.
- 4 Click **OK**.

### MESH GENERATION

- 1 Select **Mesh Cases** from the **Mesh** menu and create two new mesh cases by clicking the **New** button twice in the dialog box that opens.
- 2 Select **Mesh case 1** in the list and select **Use the same mesh as mesh case 0**. Repeat the procedure for mesh case 2.
- 3 Click **OK**.
- 4 Select **Mesh case 0** from the **Mesh** menu.
- 5 Open the **Subdomain Settings** dialog box.

- 6 Click the **Element** tab and select **Vector - Cubic** from the list of **Predefined elements**. Do this for all subdomains.
- 7 Click **OK**.
- 8 Select **Mesh case 1** from the **Mesh** menu.
- 9 Open the **Subdomain Settings** dialog box.
- 10 Click the **Element** tab and select **Vector - Quadratic** from the list of **Predefined elements**. Do this for all subdomains.
- 11 Click **OK**.
- 12 Open the **Free Mesh Parameters** dialog box and click the **Subdomain** tab and set the **Maximum element size** to 0.049 for Subdomains 1 and 4 and to  $0.049/\sqrt{2.55}$  for Subdomain 2 (plate) and to  $0.049/\sqrt{65}$  for Subdomain 3 (potato). This will make sure that the waves are properly resolved everywhere. Especially in the potato, the wavelength will be much shorter than in free space, calling for a much denser mesh here.
- 13 Click **OK**.
- 14 Click the **Initialize Mesh** button in the toolbar.

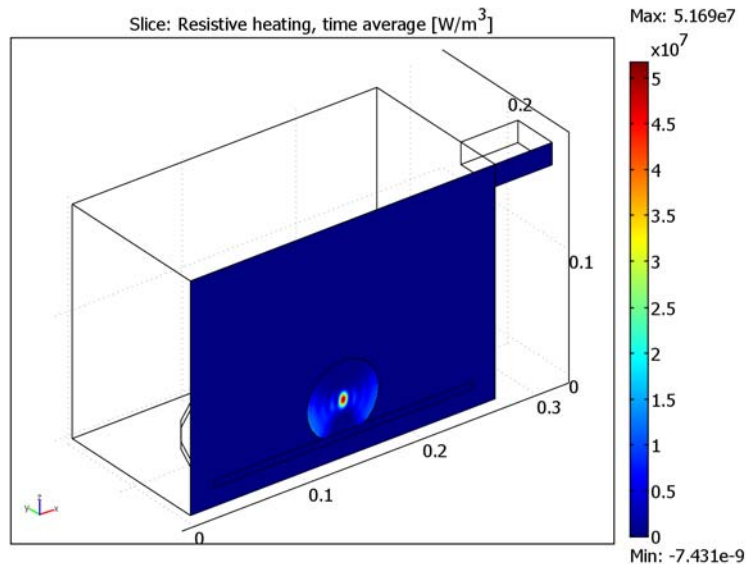
#### COMPUTING THE SOLUTION

- 1 Open the **Solver Manager** dialog box and on the **Solve For** tab select only the **Electromagnetic Waves** application mode.
- 2 Click **OK** to close the **Solver Manager**.
- 3 In the **Solver Parameters** dialog box, with the **Auto select solver** check box selected, set the **Analysis** type to **Stationary**.
- 4 Set **Matrix symmetry** to **Symmetric**.
- 5 Set **Linear system solver** to **GMRES** and **Preconditioner** to **Geometric Multigrid**.
- 6 Click the **Settings** button.
- 7 Select the **Preconditioner** in the tree on the left of the **Linear System Solver Settings** dialog.
- 8 Set the **Hierarchy generation method** to **Manual**.
- 9 Click **OK** to close the **Linear System Solver Settings** dialog.
- 10 Click the **Stationary** tab and set **Linearity** to **Linear**.
- 11 Click the **Advanced** tab and set the **Assembly block size** to 2000.
- 12 Click **OK** to close the **Solver Parameters** dialog box.
- 13 Click the **Solve** button.

## POSTPROCESSING AND VISUALIZATION

The default plot shows the temperature which so far is zero as we have not solved for it yet.

- 1 To visualize the dissipated heat in the potato, start by opening the **Plot Parameters** dialog box.
- 2 Click the **Slice** tab and select **Electromagnetic Waves>Resistive heating, time average** in the **Predefined quantities** list.
- 3 Under **Slice Positioning**, set the number of **x levels** to 0 and for **y levels** check **Vector with coordinates** and enter  $0.135+1e-5$  in that field.
- 4 Click **OK** to generate the plot below.



The port boundary condition generates two S-parameter variables, S11\_rfw and S11dB\_rf which is the S-parameters on a dB scale. To evaluate S11dB\_rf, proceed as follows.

- 1 Open the **Point Evaluation** dialog box from the **Postprocessing** menu.
- 2 Select Point 1 from the **Point selection** list and **S Parameter dB (S11)** from the list of **Predefined quantities**.
- 3 Click **OK** to display the value of S11dB\_rf in log bar at the bottom of the graphical user interface. The value should be about  $-4$  dB.

## PHYSICS SETTINGS

Now it is time to set up and solve the transient heat transfer problem. Select **General Heat Transfer (htgh)** or **Heat Transfer (ht)** from the **Multiphysics** menu. What you select here depends if you have the Heat-Transfer Module or not. The multiphysics group Microwave heating uses the module version (htgh) if you have access to it. However, this model does not require that you have it.

### *Subdomain Settings*

- 1 Open the **Subdomain Settings** dialog box.
- 2 Select Subdomains 1, 2, and 4 and clear the **Active in this domain** check box.
- 3 Select Subdomain 3 and set the **Thermal conductivity** to 0.55 W/(m·K), the **Density** to 1050 kg/m<sup>3</sup>, the **Heat capacity** to 3.64e3 J/(kg·K), and the **Heat source** to Qav\_rfw.
- 4 Click the **Init** tab and set the Initial value to 273.15+8 K.
- 5 Click **OK**.

### *Boundary Conditions*

The default boundary condition, **Thermal insulation**, is used here so no changes are necessary.

## MESH

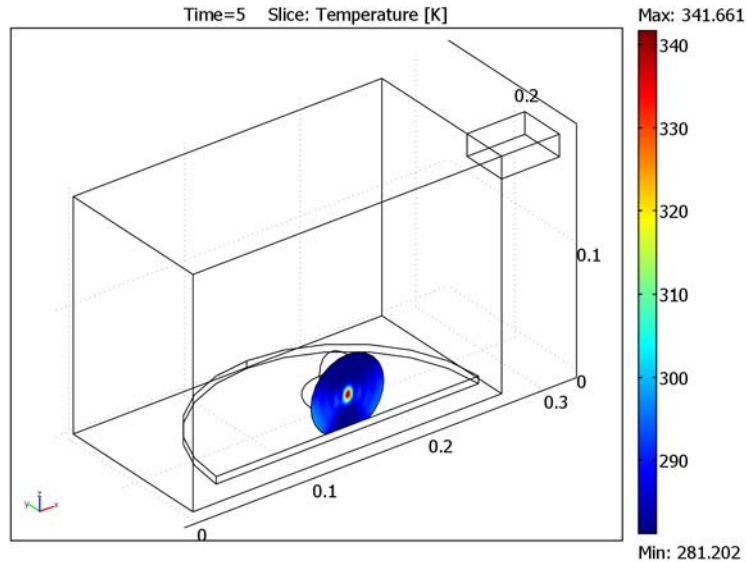
Select **Mesh case 0** from the **Mesh** menu.

## COMPUTING THE SOLUTION

- 1 Open the **Solver Manager** dialog box and on the **Solve For** page select only the **General Heat Transfer** application mode.
- 2 On the **Initial Value** page set the **Variables not solved for and linearization point** to **Current solution**.
- 3 Click **OK** to close the **Solver Manager**.
- 4 In the **Solver Parameters** dialog box, with the **Auto select solver** check box selected, select **Transient** from the **Analysis** list.
- 5 Set **Linear system solver** to **Direct (SPOLES)**.
- 6 Under **Time stepping** set **Times** to `linspace(0,5,6)` and **Absolute tolerance** to 1.
- 7 Click **OK** to close the **Solver Parameters** dialog box.
- 8 Click the **Solve** button.

## POSTPROCESSING AND VISUALIZATION

- 1 To visualize the temperature in the potato, start by opening the **Plot Parameters** dialog box.
- 2 Click the **Slice** tab and select **General Heat Transfer>Temperature** in the **Predefined quantities** list.
- 3 Click **OK** to generate the plot below.



- 1 To visualize the temperature as a function of time in the center of the potato, start by opening the **Cross-section Plot Parameters** dialog box from the **Postprocessing** menu.
- 2 Click the **Point** tab and select **General Heat Transfer>Temperature** in the **Predefined quantities** list.
- 3 Set the **Coordinates** to  $(0.267/2, 0.270/2, 0.015+0.063/2)$ .
- 4 Click **OK** to generate the plot in Figure 3-27 on page 127.

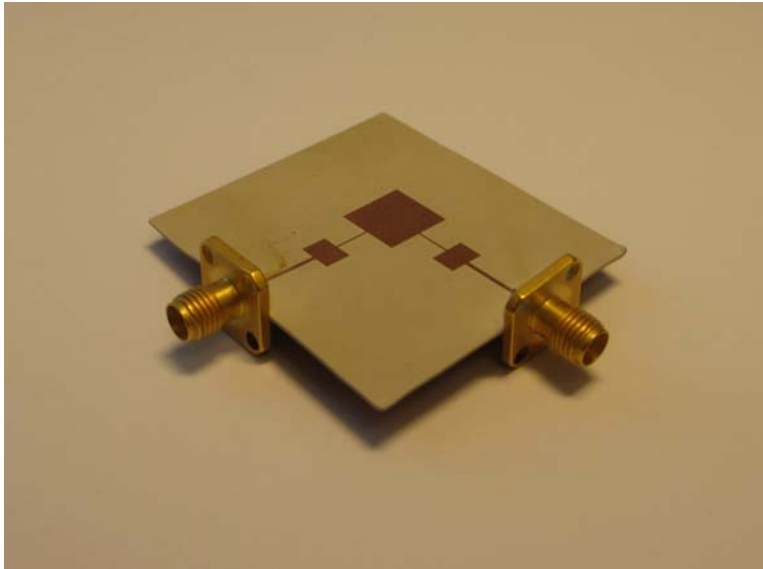
# Balanced Patch Antenna for 6 GHz

## *Introduction*

---

Patch antennas are becoming more common in wireless equipment, like wireless LAN access points, cellular phones, and GPS handheld devices. The antennas are small in size and can be manufactured with simple and cost-effective techniques. Due to the complicated relationship between the geometry of the antenna and the electromagnetic fields, it is difficult to estimate the properties of a certain antenna shape. At the early stages of antenna design the engineer can benefit a lot from using computer simulations. The changes in the shape of the patch are directly related to the changes in radiation pattern, antenna efficiency, and antenna impedance.

Balanced antennas are fed using two inputs, resulting in less disturbances on the total system through the ground. Balanced systems also provide a degree of freedom to alter antenna properties, by adjusting the phase and magnitude of the two input signals. Figure 3-29 shows the antenna that this model simulates.

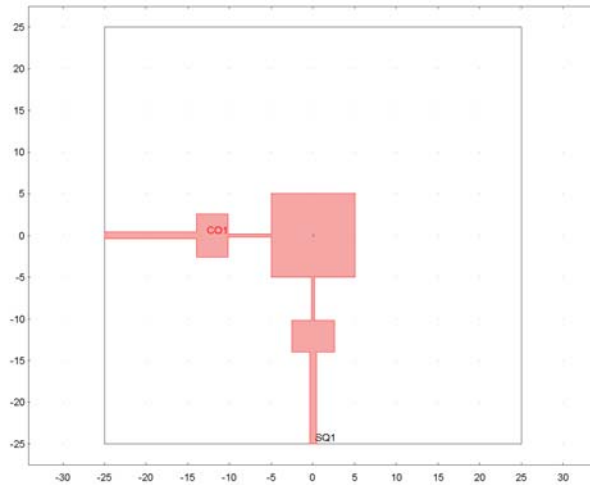


*Figure 3-29: A photo of the real antenna that the model extracts the properties for.*



## Model Definition

The patch antenna is fabricated on a printed circuit board (PCB) with a relative dielectric constant of 5.23 (Ref. 1). The entire backside is covered with copper, and the front side has the pattern of the drawing shown in Figure 3-30 below.



*Figure 3-30: The printed pattern of the patch antenna shown as a top view. The large square is 10×10 mm, the smaller rectangles are 5.2×3.8 mm, the thicker lines are 0.6 mm wide, and the thinner lines are 0.2 mm wide and 5.2 mm long. The entire pcb is 50×50 mm and 0.7 mm thick.*

The coaxial cables have an outer conductor with an inner diameter of 4 mm and a center conductor with a diameter of 1 mm. The gap between the conductors is filled with a material with a dielectric constant of 2.07 (teflon). This gives an impedance of the cables close to 58  $\Omega$ . There are two coaxial cables feeding the patch antenna from two sides, resulting in a balanced feed. The shape of the field at the outer boundary of the cables are known TEM modes, which can be specified with analytical formulas expressed in the local cylindrical coordinate system of each cable. With the port boundary condition you can automatically calculate the coaxial TEM mode, and specify a power level and a phase shift for each defined port. In this model, the signals in the cables have the same magnitude but are shifted 180 degrees in phase.

The entire antenna is modeled in 3D, using the electric field as solution variable. The time-harmonic nature of the signals makes it possible to solve the Vector-Helmholtz equation for the electric field everywhere in the geometry,

$$\nabla \times (\mu^{-1} \nabla \times \mathbf{E}) - k_0^2 \epsilon_r \mathbf{E} = 0$$

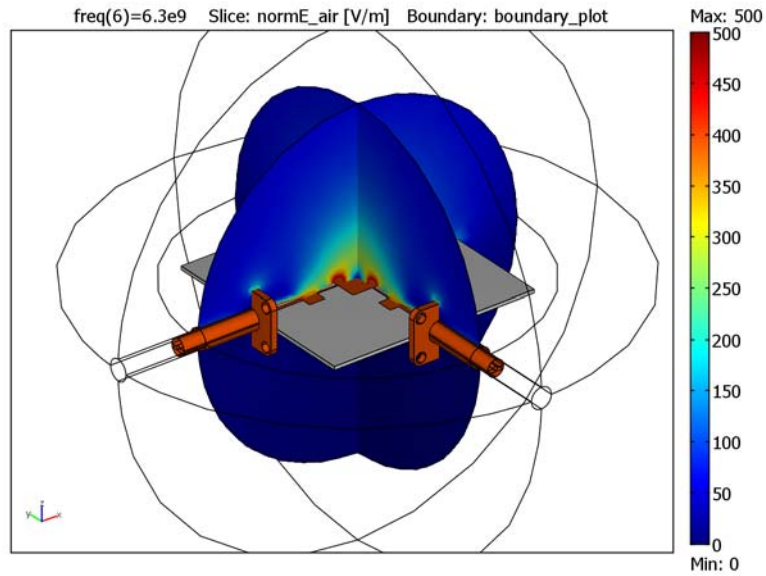
where  $k_0$  is the wave number for free space and is defined as

$$k_0 = \omega \sqrt{\epsilon_0 \mu_0}$$

All metallic objects are defined as perfect electric conductors (PEC), and the input to the coaxial cables uses the low-reflecting scattering boundary condition. The latter is a first-order approximation of an absorbing boundary, which lets all scattered waves pass the boundary without reflections. In addition, you can supply an incident wave, which in this case is the TEM mode of a coaxial cable described earlier in the text. The scattering boundary condition usually works well for waves propagating in cables, but for radiating waves from an antenna they are not very accurate and create reflections. Perfectly matched layers (PMLs) are more suitable in such cases, and for this antenna the model includes a spherical PML. You can find detailed information about PMLs in “Perfectly Matched Layers (PMLs)” on page 43 of the *RF Module User’s Guide*. This description covers the damping of the PML, because it is important when designing the length of the PML. A wave entering the PML is damped without reflections as it travels through the layer. When the wave reaches the outer boundary, the entire wave can be reflected back in the worst-case scenario. During the propagation back to the simulation volume the wave continues to get damped, so it always has to travel twice the distance of the PML layer. The damping is determined by the wave number,  $k_0$ , the PML constant,  $b_{\text{PML}}$ , and the length of the PML layer,  $d_{\text{PML}}$ , using the following formula:

$$\alpha = \exp(-2b_{\text{PML}}d_{\text{PML}}k_0)$$

For frequencies around 6 GHz the damping parameter  $\alpha$  is 0.007, so less than 1% of the radiated signal is reflected back. In addition, a scattering boundary condition is added outside the PML to further reduce the reflection.



*Figure 3-31: The final geometry of the patch antenna plotted together with a slice plot of the electric field in the surrounding air. The spherical PML is located outside the air domain and outside all cable regions.*

Typical parameters important for an antenna are optimum radiation frequency, antenna impedance, and radiation pattern. First, use the parametric solver to find the optimal radiation frequency by sweeping over frequency. It is most efficient to first do a coarse sweep and then continue with a successively more resolved sweep in order to find the peak in the antenna efficiency. The antenna efficiency is the radiated power

divided by the input power. In Figure 3-32 this efficiency is plotted against the frequency, showing that the optimum frequency is located at 6.3 GHz.

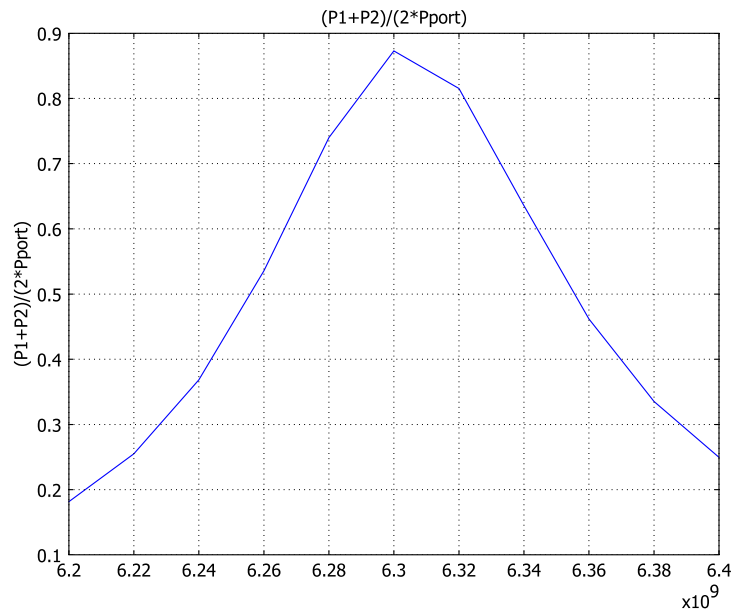
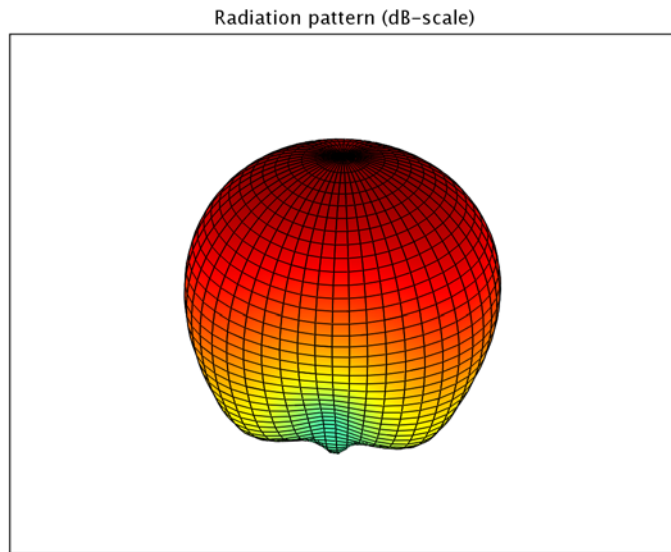


Figure 3-32: A resolved sweep in the range around 6 GHz, where the region between 6.2 GHz and 6.4 GHz was resolved in detail to locate the peak.

The radiation pattern is usually plotted in a lobe pattern, which is a surface representing the directional strength in all directions of the antenna. This plot is interpreted as follows: The vector between the origin and a point on the surface has a magnitude that represents the radiated energy in the direction parallel to the vector. The energy is given in dB, and the 0 dB level is the maximum radiated energy (a point on the unit sphere). The lobe pattern can be created with a deformation plot, where 0 dB is no deformation and  $-40$  dB is the maximum deformation to origin. The

radiation pattern for the modeled patch antenna is shown in Figure 3-33 for the optimum frequency, 6.3 GHz.



*Figure 3-33: The radiation pattern plotted as a lobe pattern, using the deformed plot feature. It is clear that the wave mainly propagates in two lobes, with minimal radiation out of the backside of the PCB.*

---

**Model Library path:** RF\_Module/RF\_and\_Microwave\_Engineering/  
patch\_antenna\_balanced

---

### *Modeling in COMSOL Multiphysics*

---

Antenna simulations in 3D often get very large, so this model should be solved on a 64-bit platform. The final mesh results in about 550,000 complex degrees of freedom, and it is seldom possible to solve problems of such large size without iterative solvers. This model solves with an efficient iterative multigrid solver, consuming around 2.5 GB of memory.

## Reference

---

1. E. Recht and S. Shiran, “A Simple Model for Characteristic Impedance of Wide Microstrip Lines for Flexible PCB,” *Proceedings of IEEE EMC Symposium 2000*, pp. 1010–1014.

## Modeling Using the Graphical User Interface

---

### MODEL NAVIGATOR

- 1 In the **Model Navigator**, select **3D** in the **Space dimension** list.
- 2 In the **RF Module** folder, select **Electromagnetic Waves** and **Harmonic propagation**.
- 3 Click **OK**.

### OPTIONS AND SETTINGS

- 1 From the **Options** menu, choose **Constants**.
- 2 In the **Constants** dialog box, define the following constants with names and expressions:

NAME	VALUE	DESCRIPTION
a_coax	0.5 [mm]	Radius for the inner conductor of the coaxial cable
b_coax	2 [mm]	Inner radius for the outer conductor
epsilon_nr_coax	2.07	Rel. permittivity of the insulator in the cable
epsilon_nr_pcb	5.23	Rel. permittivity of the printed circuit board for the patch antenna
Pport	16.636 [mW]	Input power for the ports
freq	6.3 [GHz]	Default frequency

- 3 Click **OK**.

### GEOMETRY MODELING

The geometry modeling of this model is rather extensive, so it is possible to import the patch antenna geometry from a binary file. Select the section that you prefer.

#### *Importing the Geometry from a Binary File*

- 1 From the **File** menu, select **Import>CAD Data From File**.
- 2 In the **Import CAD Data From File** dialog box, make sure that the **COMSOL Multiphysics file** or **All 3D CAD files** is selected in the **Files of type** list.

- 3 From the `models` directory under the COMSOL Multiphysics installation directory, go to the model library path given on page 141. Select the file `patch_antenna_balanced.mphbin`, then click **Import**.
- 4 Skip section “Creating the Geometry from Scratch” and begin at “Physics Settings” on page 147.

#### *Creating the Geometry from Scratch*

The geometry will be drawn using the unit mm in a separate geometry. As a final step the entire drawn geometry will be converted to meters by copying it to the first geometry followed by a scale operation. All the dialog boxes for specifying the primitive objects are accessed from the **Draw** menu and **Specify Object**. The software generates the Name column in the tables below, so you do not have to enter them. Just check that you get the correct name for the objects that you create.

Begin by creating a work plane for the patch antenna cross section:

- 1 From the **Draw** menu, select **Work-Plane Settings**.
- 2 In the **Work-Plane Settings** dialog box, select the **x-y** plane at z equal to 0. Click **OK**.
- 3 Draw squares with the properties according to the table below.

NAME	WIDTH	BASE	(X, Y)	DESCRIPTION
SQ1	50	Center	(0, 0)	PCB
SQ2	10	Center	(0, 0)	Patch

- 4 Draw rectangles with the properties according to the following table.

NAME	WIDTH	HEIGHT	BASE	(X, Y)	DESCRIPTION
R1	0.4	5.2	Corner	(-0.2, -5-5.2)	Conductor
R2	5.2	3.8	Center	(0, -5-5.2-3.8/2)	Conductor
R3	0.8	25-5-5.2-3.8	Corner	(-0.4, -25)	Conductor

- 5 Click the **Zoom Extents** toolbar button.
- 6 Select all the rectangles, R1, R2, and R3, and click the **Union** toolbar button.
- 7 Click the **Delete Interior Boundaries** toolbar button.
- 8 Copy the selected object by pressing Ctrl+C, and then paste it by pressing Ctrl+V. In the dialog box that appears, just click **OK** to create the copy in the same location as the original.
- 9 Click the **Rotate** toolbar button. In the **Rotate** dialog box, type -90 in the **Rotation angle** edit field.

**10** Select the objects SQ2, CO1, and CO2. Click first the **Union** button and then the **Delete Interior Boundaries** button, both located on the Draw toolbar.

Now it is time to draw the patch antenna in a new 3D geometry called patch\_antenna.

**11** Open the **Model Navigator** from the **Multiphysics** menu.

**12** Click the **Add Geometry** button. In the **Add Geometry** dialog box, enter patch\_antenna in the **Geometry Name** edit field, and select **3D** from the **Space dimension** list. Click **OK**.

**13** Click **OK** to close the **Model Navigator** window.

**14** Click the **Geom2** tab in the main window.

**15** Select the large square, SQ1, and select **Extrude** from the **Draw** menu.

**16** In the **Extrude** dialog box, enter -0.8 in the **Distance** edit field, and select **patch\_antenna** from the **Extrude to geometry** list. Click **OK**.

**17** Go back to **Geom2** to do an embedding.

**18** Select the patch object C01, and select **Embed** from the **Draw** menu. Select **patch\_antenna** in the **Embed to geom** list. Click **OK**.

The following steps create the coaxial cables that feed the antenna. The connectors are also drawn to resemble the real device as closely as possible:

**19** From the **Draw** menu, select **Work-Plane Settings**.

**20** In the **Work-Plane Settings** dialog box, click **Add** to create a new work plane geometry.

**21** Select the **z-x** plane and set **y** equal to -25. Click **OK**.

**22** Draw circles with the properties according to the table below.

NAME	RADIUS	BASE	(X, Y)	DESCRIPTION
C1	0.5	Center	(0, 0)	Inner conductor
C2	2	Center	(0, 0)	Outer conductor inner border
C3	2.5	Center	(0, 0)	Outer conductor outer border

**23** Select the circles C1 and C2, then click the **Difference** button on the Draw toolbar.

**24** Press Ctrl+C to copy the resulting object. This saves the object for later use.



25 Create a rectangle with the following properties:

NAME	WIDTH	HEIGHT	BASE	(X, Y)
R1	0.8	10	Corner	(-0.8, -5)

26 Select the objects R1 and CO1, and click the **Intersection** toolbar button.

27 Paste the copied object with Ctrl+V. Leave all the displacements at zero and click **OK**.

28 Select the objects CO1 and CO2, and click the **Union** button.

29 Extrude the resulting object, CO3, to the patch\_antenna geometry. Enter the expression  $-45+25$  in the **Distance** edit field and click **OK**.

30 Go back to the work plane geometry by clicking the **Geom3** tab.

31 Specify a rectangle with the following properties:

NAME	WIDTH	HEIGHT	BASE	(X, Y)	DESCRIPTION
R1	12.5	12.5	Center	(0, 0)	Connector

32 From the **Draw** menu, select **Fillet/Chamfer**. In the **Fillet/Chamfer** dialog box, select all the points of object R1, make sure that the **Fillet** radio button is selected, and enter 1.75 in the **Radius** edit field. Click **OK**.

33 Create a circle with the following properties:

NAME	RADIUS	BASE	(X, Y)	DESCRIPTION
C1	1.25	Center	$(12.5/2 - 1.75, 12.5/2 - 1.75)$	One of the holes of the connector

34 Select the circle C1 and click the **Array** toolbar button. In the **Array** dialog box, enter the expression  $-12.5+3.5$  in both the **Displacement x** and **y** edit fields, and enter 2 in both the **x** and **y Array size** edit fields. Click **OK**.

35 Select the circles C1, C2, C4, and C5. Click the **Union** button.

36 Select the objects CO1 and CO2. Click the **Difference** button.

37 Extrude the resulting object to the patch\_antenna geometry, using -1.5 in the **Distance** edit field.

38 Open the **Work-Plane Settings** from the **Draw** menu. Enter  $-25 - 1.5$  in the **y** edit field. This changes the y-location of the work plane in the 3D geometry.

39 Extrude the circle C3 to the patch\_antenna geometry, using a distance of -50.

40 Select the extruded objects, EXT2, EXT3, and EXT4. Copy them by pressing Ctrl+C.

- 41 Paste them with Ctrl+V and leave the displacements at zero.
- 42 Click the **Rotate** toolbar button. In the **Rotate** dialog box, enter -90 in the **Rotation angle** edit field. Click **OK**.

The final step is to create the surrounding air volume, and the PML region. This is done with two spheres.

- 43 Specify two spheres with the following properties:

NAME	RADIUS	BASE	CORNER	DESCRIPTION
SPH1	60	Center	(0, 0)	PML+Air
SPH2	40	Center	(0, 0)	Air

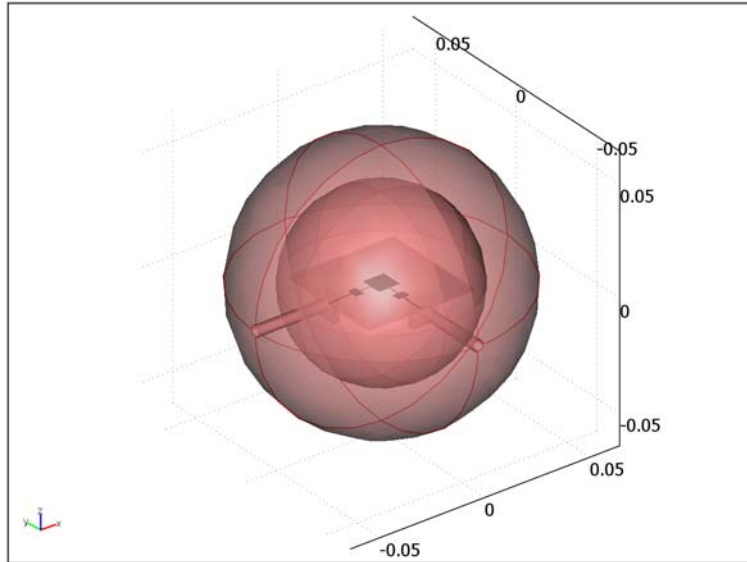
The entire geometry is now completed in the `patch_antenna` geometry. To help the software coerce the geometry to its minimal regions (subdomains) you can do step-by-step union and difference operations until there is only two solid objects (and one face object). Do all the operations below in the specified order, otherwise the names of the geometry objects do not match.

- 1 Select all objects in the `patch_antenna` geometry by pressing Ctrl+A.
- 2 Press Ctrl+C to copy, then go to the first geometry by clicking the **Geom1** tab, and paste the objects with Ctrl+V (and zero displacements).
- 3 Do all the geometry operations according to the table below, starting from the first row:

STEP	OBJECTS	OPERATION
1	SPH1, SPH2	Union
2	CO1, EXT4	Difference
3	CO2, EXT7	Difference
4	CO1, EXT3	Difference
5	CO2, EXT6	Difference
6	EXT1, CO1	Union
7	EXT2, EXT5	Union

- 4 Press Ctrl+A to select all objects.
- 5 Click the **Scale** button on the Draw toolbar. Type  $1e-3$  in all the **Scale factor** edit fields, then click **OK**.

- Click the **Zoom Extents** button on the Main toolbar. Using the tools on the Camera toolbar, turn on the headlight and increase the transparency slightly to get the following image of the final geometry.



## PHYSICS SETTINGS

- Go to the first geometry by clicking the **Geom1** tab.
- From the **Options** menu, choose **Expressions>Scalar Expressions**.
- In the **Scalar Expressions** dialog box, define the following variables with names and expressions (the descriptions are optional):

NAME	EXPRESSION	DESCRIPTION
S11	$\sqrt{1-P1/Pport}$	S11 parameter for first input
Z_coax	$\sqrt{\mu_{0\_rfw}/(\epsilon_{0\_rfw} \cdot \epsilon_{nr\_coax})} / (2 \cdot \pi) \cdot \log(b\_coax/a\_coax)$	Impedance for the coaxial cables
Z0	$Z\_coax \cdot (1+S11)/(1-S11)$	Input impedance for the system on the first cable

The variable P1 is the power flow into Port 1, which is defined as a integration coupling variable later. The impedance variable Z0 is not correct to use when both

inputs are feeding the antenna, like it is done below. To perform a calculation for the antenna impedance, you must turn off one input. The variable is declared here to prepare for such an analysis.

4 Click **OK**.

5 From the **Options** menu, select **Expressions>Subdomain Expressions**.

6 In the **Subdomain Expressions** dialog box, define the following variables:

SETTING	SUBDOMAIN 5	ALL OTHERS
normE_air	normE_rfw	

7 Click **OK**.

8 From the **Options** menu, choose **Integration Coupling Variables>Boundary Variables**.

9 In the **Boundary Integration Variables** dialog box, define the following variables with names and expressions. Use the default **Global destination** and **Integration order 4** for all variables.

VARIABLE	BOUNDARIES 9, 11, 14	BOUNDARIES 99, 100, 105	ALL OTHERS
P1	-nPoav_rfw		
P2		-nPoav_rfw	

10 Click **OK**.

11 From the **Physics** menu, select **Scalar Variables**.

12 In the **Scalar Variables** dialog box, type `freq` in the expression column for variable `nu_rfw`. Click **OK**.

### *Boundary Conditions*

1 From the **Physics** menu, open the **Boundary Settings** dialog box, select the **Interior boundaries** check box, and enter the settings according to the following two tables (leave all fields not specified at their default values):

SETTINGS	BOUNDARIES 9, 11, 14	BOUNDARIES 99, 100, 105	BOUNDARIES 2, 4, 6, 8, 112, 114, 123, 126
Boundary condition	Port	Port	Scattering boundary
Wave type	Plane wave	Plane wave	Spherical wave
Port number	1	2	
Wave excitation at this port	Selected	Selected	
Port power	Pport	Pport	

SETTINGS	BOUNDARIES 9, 11, 14	BOUNDARIES 99, 100, 105	BOUNDARIES 2, 4, 6, 8, 112, 114, 123, 126
Port phase	0	pi	
Mode specification	Coaxial	Coaxial	0

SETTINGS	BOUNDARIES 60, 70
Boundary condition	Perfect electric conductor

- 2 Select Boundaries 26, 28, 30, 32, 120, 122, 124, and 125. On the **Far-Field** page type  $E_{far}$  in the first row of the **Name** column and let the other columns have the default values.
- 3 Click **Apply** and leave the **Boundary Settings** dialog box open. A special variable is needed later to plot all the metal objects in the geometry. Use the boundary settings to select all boundaries with the **Perfect electric conductor** condition. Mark boundary 60 and then select the **Select by group** check box. All metal boundaries should now be selected. Click **OK**.
- 4 Go to the **Options** menu and select **Expressions>Boundary Expressions**.
- 5 In the **Boundary Expressions** dialog box, add the variable `boundary_plot` and type 1 in the **Expression** column.
- 6 Select Boundaries 58, 59, 61, 65, 71, 73, 74, 101, 141, and 142. Type 2 in the **Expression** column for the `boundary_plot` variable.
- 7 Select Boundaries 1, 3, 5, 7, 92, 93, 111, and 113. Clear the **Expression** column for the `boundary_plot` variable.
- 8 Click **OK**.

#### Subdomain Settings

- 1 From the **Physics** menu, open the **Subdomain Settings** dialog box and enter the following settings:

SETTINGS	SUBDOMAINS 2-4, 7-9	SUBDOMAIN 6	ALL OTHERS
$\epsilon_r$ (isotropic)	<code>epsilon0r_coax</code>	<code>epsilon0r_pcb</code>	1
$\mu_r$ (isotropic)	1	1	1

- 2 Select Subdomain 1, click the **PML** tab, and select **Spherical** from the **PML type** list.
- 3 Enter `lambda0_rfw` in the **Width in r direction** edit field.

The default value that you removed is equal to the actual width of the PML. Using that value adjusts the PML design so one wavelength fits into the width of the PML. Using the free-space wavelength,  $\lambda_0$ , instead makes the program assume that the

width of the PML is equal to  $\lambda_0$ , even though it is not. This means that the width of the PML is shorter than one wavelength, and thereby reduces the required number of mesh elements here. The drawback is that the reflection increases. However, the combination of the PML and the scattering boundary condition for spherical waves is efficient, and one-wavelength PMLs are often not necessary.

- 4 Click **OK** to close the **Subdomain Settings** dialog box.

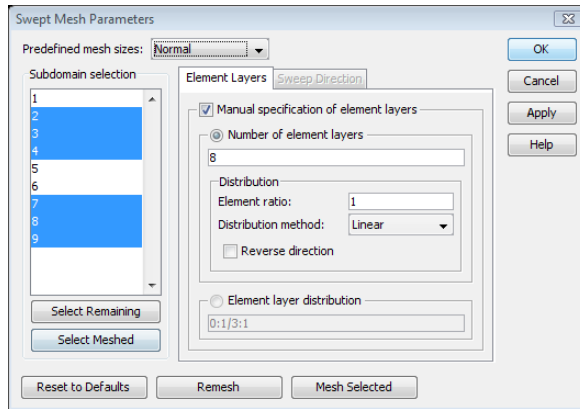
#### **MESH GENERATION**

The structure of the patch antenna benefits significantly by optimizing the mesh with interactive meshing. This feature makes it possible to use individual mesh parameters for subdomains and boundaries.

- 1 Click the **Geom1** tab.
- 2 From the **Mesh** menu, choose **Free Mesh Parameters**.
- 3 Select **Finer** from the **Predefined mesh sizes** list.
- 4 Click the **Boundary** tab.
- 5 Select boundaries representing the faces of the coaxial cable facing the patch antenna. Either use the technique described for the edges below or select them by number. The boundary numbers are: 9, 11, 14, 99, 100, and 105.
- 6 Click the **Mesh Selected** button. Leave this dialog box open.

Due to the structure of the coaxial cable subdomains it is more efficient to mesh them with prism elements. With mesh sweeping, it is possible to create such meshes in extruded-like geometries from a triangular mesh on one of the faces.
- 7 From the **Mesh** menu, choose **Swept Mesh Parameters**.

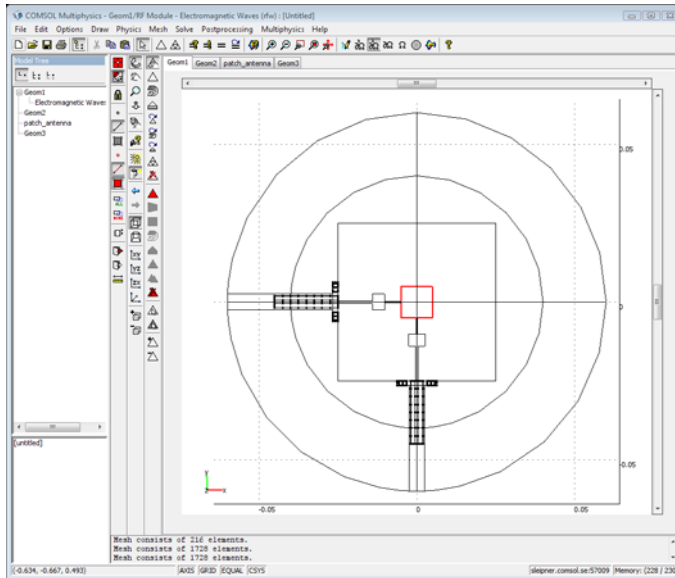
- 8 In the Swept Mesh Parameter dialog box, select Subdomains 2, 3, 4, 7, 8, and 9. Under the **Element layers** tab, select the **Manual selection of element layers** check box, and type 8 in the **Number of element layers** edit field.



- 9 Click the **Mesh Selected** button.
- 10 Click **OK** to close the **Swept Mesh Parameters** dialog box.
- 11 In the **Free Mesh Parameters** dialog box, click the **Global** tab, select **Normal** from the **Predefined mesh sizes** list, and click **Apply**.
- 12 Click the **Custom mesh size** option button, then type 0.01 in the **Mesh curvature cutoff** edit field.
- 13 Click the **Boundary** tab.
- 14 Select boundaries representing the drill holes and fillets of the connectors. Either use the technique described for the edges below or select them by number. The boundary numbers are: 35–40, 42–45, 47–56, 82–85, 87–90, and 130–139.
- 15 Click the **Mesh Selected** button.
- 16 Click the **Global** tab, then type 2 in the **Resolution of narrow regions** edit field.
- 17 Click the **Edge** tab.
- 18 Select Edges 190–193, 284, and 330, then type  $8e-4$  in the **Maximum element size** edit field.

The edges listed above are all edges of the square patch in the middle of the substrate, and there is a technique to select all of them with the mouse. Click the **Go to XY View** button on the Camera toolbar. You now see a top view of the patch antenna. Then click the **Orbit/Pan/Zoom** button, also on the Camera toolbar. This is a toggle button that shifts between rotating the view (button pressed down) and

selecting objects. The button must now be in the selecting objects state (button not pressed). With the mouse, select all edges belonging to the rectangular patch in the middle (mouse click and drag around these edges). Right-click on one of the selected edges to lock the selection (colored blue). The figure below shows what the selection should look like. This is often the most efficient technique to select multiple edges and boundaries in a 3D geometry. It is also possible to change view and select a new set of domains, adding them to the previous selection.



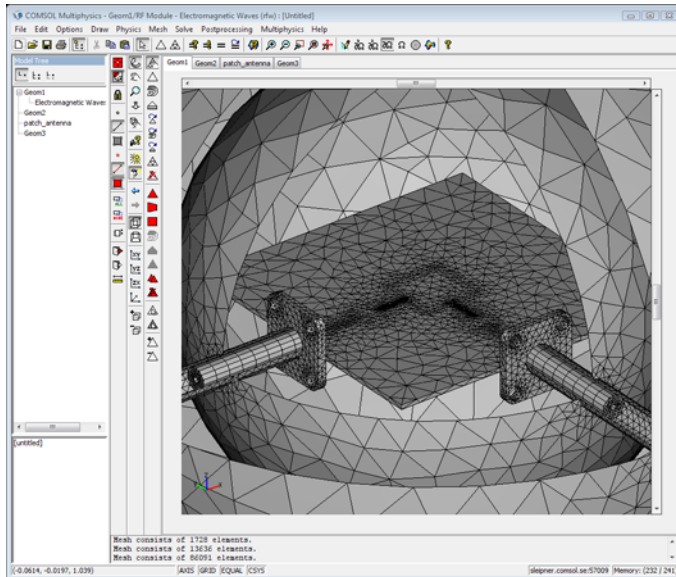
- 19 Click the **Boundary** tab, select Boundary 70 and click the **Mesh Selected** button.
- 20 Click the **Global** tab and type 0.5 in the **Resolution of narrow regions** edit field.
- 21 Click the **Advanced** tab and type 2 in the **z-direction scale factor** edit field.
- 22 Click the **Subdomain** tab, select Subdomain 6, and click the **Mesh Selected** button.
- 23 Click the **Advanced** tab again and restore the value in the **z-direction scale factor** edit field to 1.
- 24 Click the **Subdomain** tab, select Subdomains 1 and 5, and click the **Mesh selected** button.
- 25 Click **OK** to close the **Free Mesh Parameters** dialog box.
- 26 Click **OK** to close the dialog box.



### Hiding Boundaries

To better see the mesh inside the geometry it is necessary to hide (suppress) some boundaries.

- 1 Choose **Suppress>Boundaries** from the **Options** menu.
- 2 Select Boundaries 3, 4, 27, 28, 93, and 95. Click **OK**.
- 3 Zoom in to get a better view of the mesh inside. You should see something like the figure below. Notice the prism elements inside the coaxial cable.

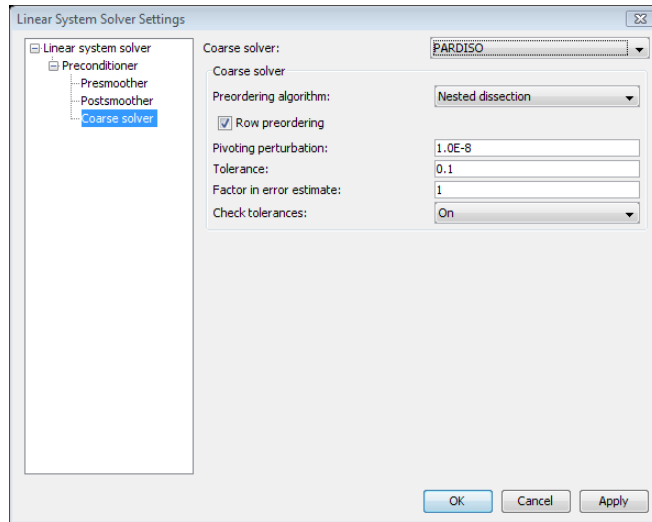


### COMPUTING THE SOLUTION

The program can use the linear order element combination for a coarse solution and solve for the quadratic elements using the Geometric multigrid preconditioner. The linear solution is then used in the preconditioning step. COMSOL Multiphysics does this automatically with the setting you select in Step 6.

- 1 Open the **Solver Parameters** dialog box from the **Solve** menu.
- 2 Select **Parametric** from the **Solver** list.
- 3 In the **Parameter name** edit field type **freq**, and in the **Parameter values** edit field type **6.2e9:0.02e9:6.4e9**.
- 4 Click the **Settings** button.

- 5 Select **Coarse solver** from the tree view. Choose **PARDISO** from the **Coarse solver** list.



All other settings can be left at their default values. For details on the default settings, see “Solving Large 3D Problems” in the *RF Module User’s Guide*.

- 6 Click **OK** to close the **Linear System Solver Settings** dialog box.

- 7 Click **OK**.

- 8 Click the **Solve** button on the Main toolbar.

The solving process takes some time, because this is a large problem, and the analysis includes a parameter sweep of eleven steps.

## POSTPROCESSING AND VISUALIZATION

- 1 Select **Plot Parameters** from the **Postprocessing** menu.
- 2 Make sure that the **Boundary**, **Slice**, and **Geometry edges** check boxes are selected on the **General** tab.
- 3 Choose the solution **6.3e9** from the **Parameter value** list.
- 4 Click the **Slice** tab, and enter `normE_air` in the **Expression** edit field.
- 5 Click the **Range** button, clear the **Auto** check box, and type 500 in the **Max** edit field and 0 in the **Min** edit field. Click **OK**.
- 6 Click the **Boundary** tab, and enter `boundary_plot` in the **Expression** edit field. Select the **Hot** colormap from the **Colormap** list.

- 7 Click the **Range** button, clear the **Auto** check box, and type 2 in the **Max** edit field and 0 in the **Min** edit field. Click **OK**.
- 8 Click **OK**. You should now see the plot in Figure 3-31 on page 139.

To plot the antenna efficiency, perform the following steps:

- 1 From the **Postprocessing** menu, select **Domain Plot Parameters**.
- 2 Choose the **Point** plot type and check that all parameters are selected in the **Parameter value** list.
- 3 On the **Point** tab, type  $(P1+P2)/(2*Pport)$  in the **Expression** edit field.
- 4 Select Point 1, then click **OK**. In a separate window you should see the plot in Figure 3-32 on page 140.

### RADIATION PATTERN PLOT IN COMSOL SCRIPT

The final step is to plot the radiation pattern shown in Figure 3-33 on page 141. These steps require COMSOL Script or MATLAB.

- 1 Press CTRL+F to export the entire model to the COMSOL Script workspace.
- 2 Go to the COMSOL Script window and enter the following commands to define some useful variables.

```
res = 50;
theta = linspace(0,pi,res);
phi = linspace(0,2*pi,res);
[THETA,PHI] = meshgrid(theta,phi);
```

The variable `res` sets the resolution of the radiation plot, the `theta` and `phi` variables defines the ranges of the angular direction to span all directions, and finally the variables `THETA` and `PHI` holds the grid points used for plotting the final surface.

- 3 Then convert the angular direction to the corresponding vector components. Scale the length of the vector so it does not point to a coordinate outside the model geometry.

```
X = 1e-3*sin(THETA).*cos(PHI);
Y = 1e-3*sin(THETA).*sin(PHI);
Z = 1e-3*cos(THETA);
```

- 4 Now use the `postinterp` function to extract the norm of the far field for all directions. It is first necessary to reorganize the coordinates to a 3-by-N matrix.

```
xx = [X(:) Y(:) Z(:)]';
EdB = postinterp(fem, 'normEfar dB', xx, 'solnum', 6);
```

- 5 You need the maximum and minimum values before you calculate the coordinates for the radiation pattern surface.

```
maxE = max(EdB);  
minE = min(EdB);  
EdB = reshape(EdB,size(THETA));  
Xr = X.*(EdB-minE)./(maxE-minE);  
Yr = Y.*(EdB-minE)./(maxE-minE);  
Zr = Z.*(EdB-minE)./(maxE-minE);
```

- 6 Use the `surf` function to plot the radiation pattern, and also specify a few options to make a nicer plot. The plot should look like Figure 3-33 on page 141.

```
h = surf(Xr,Yr,Zr,EdB);  
set(h,'edgecolor','black');  
title('Radiation pattern (dB-scale)');
```

It may take a minute or so before the plot appears, because of the far-field evaluation in all directions.

# Coaxial to Waveguide Coupling

This model shows how to use assembly pairs and the Port boundary conditions to feed a rectangular waveguide.

## *Introduction*

---

Feeding a waveguide from a coaxial cable is a straightforward way to get electromagnetic waves inside the waveguide. The coaxial feed must be modeled with enough length, so reflected waves either are damped out in the cable (evanescent modes) or turn into propagating modes. Propagating modes are efficiently terminated with a matched absorbing boundary condition, called the Port boundary condition in the RF Module. Due to its small size and circular shape, the cable contributes significantly to the overall size of the problem. It is therefore necessary to keep the cable as short as possible. One approach is to add the cable part as a separate assembly, connecting it to the waveguide using the port condition for pairs. This port condition can excite a wave into the master side of the pair while only the reflected waves exiting the master side enters the slave side. The port condition can be placed directly at the surface where the cable enters the waveguide, and only a short cable part, modeled as a perfectly matched layer (PML), is necessary.

In 3D, the electromagnetic waves application mode uses vector elements, and these elements are very sensitive to incompatible meshes. The master and slave side of an assembly pair usually get incompatible meshes in 3D, so the identity coupling must be stabilized with the divergence condition. The divergence condition adds an extra equation that complicates the use of iterative solvers, so if it is possible to create compatible (or identical) meshes it is definitely worth the effort. It is possible to copy a mesh from one boundary in an identity pair to the other side using interactive meshing and script programming. This approach is used in this model to avoid having to turn on the divergence condition.

## *Results and Discussion*

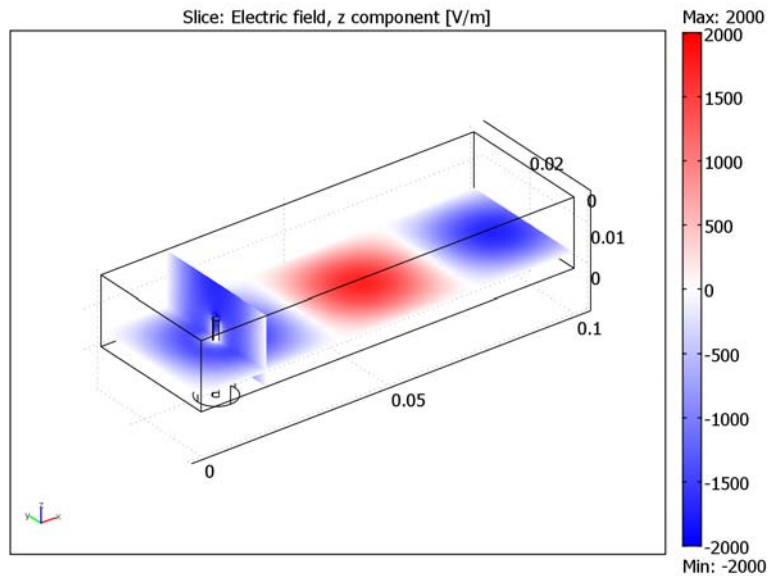
---

For cable feeds it is interesting to compare the reflected wave going back into the coaxial cable with the wave propagating in the rectangular waveguide. You can make such a comparison by evaluating the S-parameter on the coaxial port and compare it with the S-parameter on the rectangular port. There is also a coaxial port on the

exterior of the PML that shows a very low value. The following table shows the result for a frequency of 6 GHz in dB scale:

PARAMETER	VALUE	DESCRIPTION
S11	-8.6 dB	Reflected wave into the coaxial cable
S21	-0.65 dB	Wave exiting the rectangular waveguide
S31	-19 dB	Wave exiting the PML

The following plot shows the  $z$ -component of the electric field. It shows how the rectangular waveguide mode is created directly after the cable feed.



The  $z$ -component is the only component present for the propagating mode in the rectangular waveguide, and it is completely absent from the propagating mode of the coaxial cable.

---

**Model Library path:** RF\_Module/RF\_and\_Microwave\_Engineering/  
coaxial\_to\_waveguide\_coupling

---

**MODEL NAVIGATOR**

- 1 Select **3D** from the **Space dimension** list.
- 2 Select the **RF Module>Electromagnetic Waves>Harmonic propagation** application mode.
- 3 Click **OK**.

**GEOMETRY MODELING**

Use the toolbar buttons and the **Draw** and **Edit** menus.

- 1 Click the **Block** toolbar button. In the **Block** dialog box enter the following properties.

PROPERTY	VALUE
x	0.1
y	0.0175*2
z	0.0175

- 2 Click **OK**.
- 3 From the **Draw** menu, choose **Work-Plane Settings**.
- 4 In the **Work-Plane Settings** dialog box, click the **Face parallel** tab, and select face number 1 of the object BLK1. Click **OK** to create the work plane.
- 5 In the new work plane, draw two circles according to the table below.

LABEL	RADIUS	CENTER (X, Y)
C1	0.005	(0.0175, 0.0175)
C2	0.001	(0.0175, 0.0175)

- 6 Select the circle labeled C2 and press Ctrl+C to copy it.
- 7 Select all objects by pressing Ctrl+A, and then click the **Difference** toolbar button.
- 8 Press Ctrl+V to paste the circle back to the work plane. In the dialog that appears, just click **OK**.
- 9 Select the pasted circle and choose **Extrude** from the **Draw** menu.
- 10 In the **Distance** edit field, type 0.0175/2. Click **OK**.
- 11 In the 3D geometry, select all objects and click the **Difference** toolbar button.
- 12 Go back to the work plane again and select the ring labeled CO1.

**13** Choose **Extrude** from the **Draw** menu, and type -0.01 in the **Distance** edit field.

**14** Select all objects and click the **Create Pairs and Imprints** toolbar button.

### SCALAR VARIABLES

In the **Application Scalar Variables** dialog box (available by choosing **Scalar Variables** from the **Physics** menu), set the **Frequency** to  $6e9$ .

### BOUNDARY CONDITIONS

The default boundary condition is a perfect electric conductor, which is fine for all exterior boundaries except at the ports. The pair condition between the waveguide and the coaxial cable is also a port with wave excitation. The excitation enters the waveguide side because it is the master side of the pair. Only waves reflected back to the cable enter the slave side.

**1** Open the **Boundary Settings** dialog box from the **Physics** menu.

**2** Click the **Pair** tab and select the pair in the list.

**3** From the **Boundary condition** list, select the **Port** boundary condition.

**4** Click the **Port** tab, then select the **Wave excitation at this port** check box.

**5** From the **Mode specification** list, select **Coaxial**.

**6** Click the **Boundaries** tab, and select Boundary 15. Select the **Port** boundary condition.

**7** Click the **Port** tab and type 3 in the **Port number** edit field. Then select the coaxial mode specification also for this port.

**8** Click the **Conditions** tab, select Boundary 12, choose the port boundary condition again, and then click the **Port** tab.

**9** Type 2 in the **Port number** edit field, and choose **Rectangular** from the **Mode specification** list.

**10** Click **OK** to close the **Boundary Settings** dialog box.

### SUBDOMAIN SETTINGS

All subdomains keep the defaults value that represents air. The subdomain for the coaxial cable damps all reflected waves using perfectly matched layers.

**1** Open the **Subdomain Settings** dialog box from the **Physics** menu.

**2** Select Subdomain 2 and click the **PML** tab.

**3** Choose **Cartesian** from the **Type of PML** list.



- 4 Select the **Absorbing in z direction** check box, and enter  $4*Sdz\_guess\_rfw$  in the **Width in z direction** edit field. This setting ensures that a quarter wavelength fits inside the PML. The damping is not as good as the default setting but should be enough because a Port boundary condition terminates the PML.
- 5 Click **OK** to close the dialog box.

### MESH GENERATION

The mesh generation for this model needs interactive meshing, because it is necessary to have identical meshes on the master and slave side of the assembly pair (Boundaries 6 and 16). If the meshes are incompatible (not identical) it is necessary to use the divergence condition, which increases the solution time and memory requirement drastically.

- 1 Open the **Free Mesh Parameters** dialog box from the **Mesh** menu.
- 2 Click the **Boundary** tab, select Boundary 6, and set the **Maximum element size** to 0.002, the **Mesh curvature factor** to 0.4, and the **Mesh curvature cutoff** to 0.01.
- 3 Click the **Mesh Selected** button. The mesh now only contains the boundary mesh of the master side of the pair.
- 4 Select both boundaries 6 and 16, and click the **Copy Mesh** toolbar button. The same mesh now exists on boundary 16.
- 5 Click **OK** to close the **Free Mesh Parameters** dialog box.
- 6 Choose Swept Mesh Parameters from the Mesh menu.
- 7 Select subdomain 2 and select the **Manual specification of element layers** check box. Enter 5 in the **Number of element layers** edit field.
- 8 Click the **Mesh Selected** button.
- 9 Click **OK**.
- 10 Continue with the rest of the mesh. Open the **Free Mesh Parameters** dialog box again. Click the **Custom mesh size** radio button, enter 0.008 in the **Maximum element size** edit field, 0.5 in the **Mesh curvature factor** edit field, and 0.012 in the **Mesh curvature cutoff** edit field.
- 11 Click **OK**.
- 12 Click the **Mesh Mode** button on the Main toolbar, then click the **Mesh Remaining (Free)** button on the Mesh toolbar.

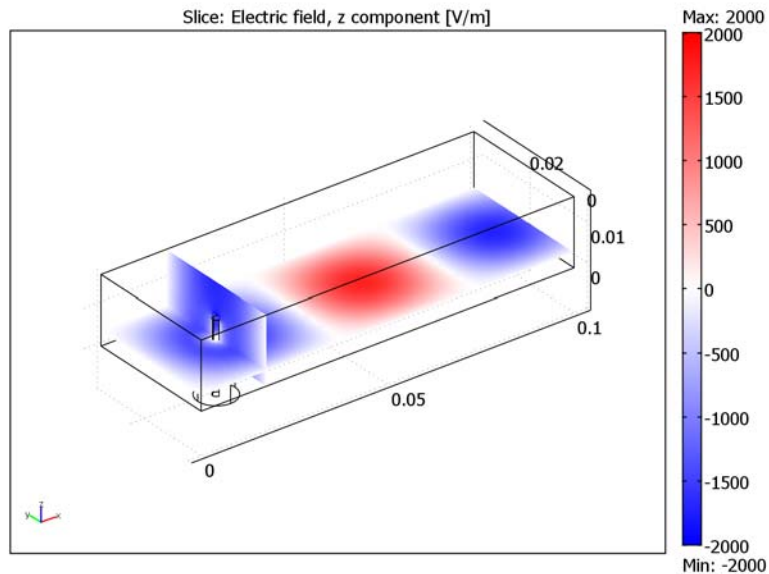
### COMPUTING THE SOLUTION

- 1 Open the **Solver Parameters** dialog box from the **Solve** menu.

- 2 Click the **Settings** button.
- 3 In the dialog box that appears, type 40 in the **Factor in error estimate** edit field and 100 in the **Number of iterations before restart** edit field.
- 4 Click **OK** twice to close both dialog boxes.
- 5 Click the **Solve** button on the Main toolbar to solve the problem. The convergence is rather slow so the solution process might take some time.

#### POSTPROCESSING AND VISUALIZATION

- 1 Open the **Plot Parameters** dialog box from the **Postprocessing** menu.
- 2 Under the **Slice** tab, choose **Electric field, z component** from the **Predefined quantities** list.
- 3 Click the option button **Vector with coordinates** for the **x levels**, and type 0.0175 in the corresponding edit field. Also type 1 in the **Number of levels** edit field for the **z levels**.
- 4 Click the **Range** button, and in the dialog box that appears, enter -2000 and 2000 in the **Min** and **Max** edit fields, respectively.
- 5 Click **OK**.
- 6 Choose the **wave** colormap and click **OK** to get the plot below.



7 The S-parameters S11 and S21 are easiest to get from the menu option **Postprocessing>Data Display>Global**. In the **Global Data Display** dialog box, choose **Electromagnetic Waves (rfw)>S-parameter dB (S11)** and **Electromagnetic Waves (rfw)>S-parameter dB (S21)** from the **Predefined quantities** list and then click **Apply** to see their values in the message log at the bottom of the COMSOL Multiphysics user interface. The S-parameters have the following approximate values:

S11 = -8.59 dB  
S21 = -.64 dB  
S31 = -22.23 dB

# Sea Bed Logging

## *Introduction*

---

The Sea Bed Logging method for oil prospecting has emerged as a promising technique during recent years. This model demonstrates one variant of it. It uses a mobile horizontal 1 Hz electric dipole antenna that is towed 150 m above the sea floor. An array of sea floor receivers measure the electric field at various distances away from the antenna. When measuring at sufficiently large distance, some of the transmitted energy is reflected/guided by the resistive reservoir and results in a higher received signal than if no reservoir were present.

## *Model Definition*

---

The computational domain is a sphere of 5 km radius. The top region of this sphere represents air. At such a low frequency as 1 Hz, some numerical stabilization is required in this domain so an artificial conductivity of 0.001 S/m is specified. A 100 m deep ocean water domain with a conductivity of 3 S/m and a relative permittivity of 80 is specified above the mid plane of the sphere. Below the midplane, a conductivity of 1.5 S/m and a relative permittivity of 30 is specified for the rock. Embedded in the rock at an average depth of 250 m, there is a block-shaped, 100 m deep and 4 km-by-1 km wide hydrocarbon reservoir. The conductivity of the hydrocarbon layer is 0.01 S/m and the permittivity is 4. The transmitter is modeled as a short 10 kA amplitude AC line current segment 150 m above the midplane. At the external spherical boundaries, a scattering type boundary condition absorbs outgoing spherical waves. The following equation is solved for the electric field vector  $\mathbf{E}$  inside the computational domain:

$$\nabla \times (\mu_r^{-1} \nabla \times \mathbf{E}) - k_0^2 \left( \epsilon_r - \frac{j\sigma}{\omega \epsilon_0} \right) \mathbf{E} = 0$$

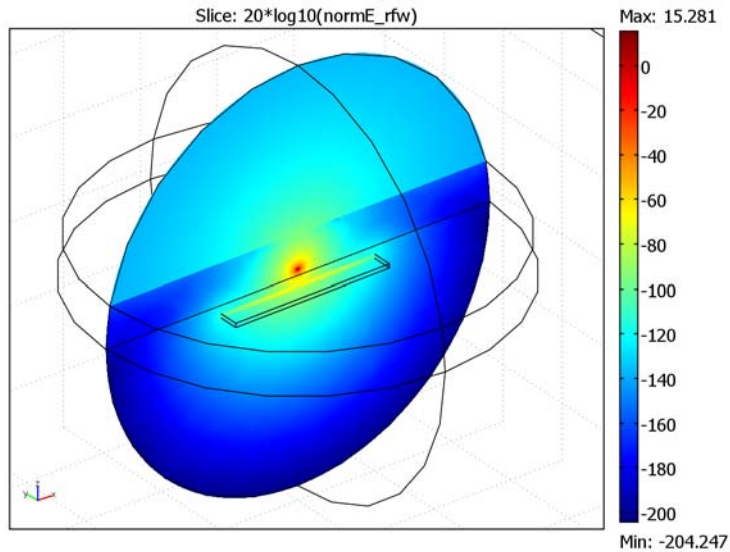
where  $\mu_r$  denotes the relative permeability,  $j$  the imaginary unit,  $\sigma$  the conductivity,  $\omega$  the angular frequency,  $\epsilon_r$  the relative permittivity, and  $\epsilon_0$  the permittivity of free space.

## *Results*

---

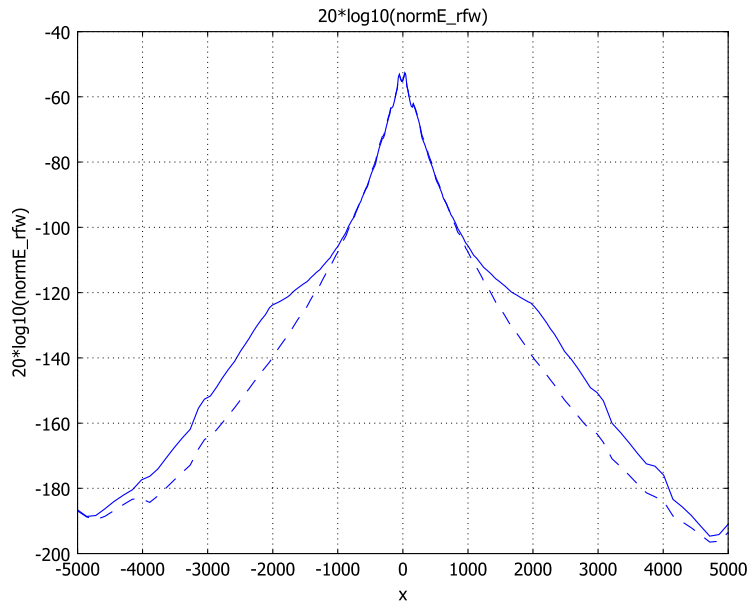
The first figure below shows a slice plot of the electric field magnitude on a dB scale

where the guiding effect of the hydrocarbon layer is clearly visible.



*Figure 3-34: The electric field magnitude is plotted in a slice containing the antenna (red spot). The guiding effect of the hydrocarbon layer is clearly visible.*

The effect is shown quantitatively in the next plot, where a comparison of the electric field magnitude on the sea floor is plotted as a function of distance for simulations with and without a hydrocarbon layer. The maximum is obtained right under the antenna where there is little difference between the two cases. When you move away from the antenna, there is a notably higher signal strength when the hydrocarbon layer is present.



*Figure 3-35: Electric field magnitude on the sea floor as a function of distance. The dashed line represents the case without a hydrocarbon layer.*

### *Modeling in COMSOL Multiphysics*

This model uses the 3D Electromagnetic Waves formulation available in the RF Module although the wavelength is much larger than the computational domain. In principle, you could also create this model using a quasi-static formulation in the AC/DC Module.

---

**Model Library path:** RF\_Module/RF\_and\_Microwave\_Engineering/  
sea\_bed\_logging

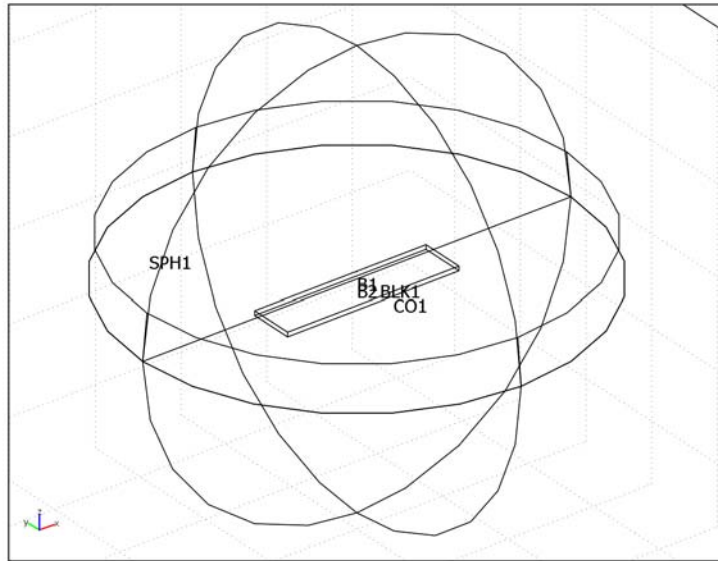
---

#### **MODEL NAVIGATOR**

- 1 Select **3D** from the **Space dimension** list.
- 2 Select the **RF Module>Electromagnetic Waves>Harmonic propagation** application mode from the list of application modes.
- 3 Click **OK** to close the **Model Navigator** dialog box.

#### **GEOMETRY MODELING**

- 1 Use the **Sphere** tool to create a sphere with a **Radius** of  $5e3$  and other parameters at their defaults.
- 2 Click the **Zoom Extents** button on the Main toolbar.
- 3 Use the **Cylinder** tool to create a cylinder with **Radius**  $6e3$  and **Height**  $1e3$ . Leave all other parameters at their default values.
- 4 Select all object by pressing **Ctrl+A**.
- 5 Click the **Intersection** button on the Draw toolbar.
- 6 Use the **Sphere** tool to create a new sphere with a **Radius** of  $5e3$  and other parameters at their defaults.
- 7 Go to the **Draw** menu and click the block symbol and define a block with **Base: Center**, **Length x:**  $4e3$ , **Length y:**  $1e3$  **Length z:**  $100$ , and **Axis base point z:**  $-250$ . Leave the other parameters at their default values.
- 8 Go to the **Draw** menu and click the line symbol and define a **Polyline** with **Coordinates x:**  $-10$   $10$ , **Coordinates y:**  $0$   $0$  and **Coordinates z:**  $150$   $150$ .
- 9 Go to the **Draw** menu and click the line symbol and define a **Polyline** with **Coordinates x:**  $-5e3$   $5e3$ , **Coordinates y:**  $0$   $0$  and **Coordinates z:**  $0$   $0$ .



## PHYSICS SETTINGS

### Scalar Variables

- 1 From the **Physics** menu, choose **Scalar Variables**.
- 2 In the **Application Scalar Variables** dialog box, set the frequency `nu_rfw` to 1.
- 3 Click **OK**.

### Subdomain Settings

- 1 Open the **Subdomain Settings** dialog box.
- 2 Specify subdomain settings according to the following table.

SETTINGS	SUBDOMAIN 1	SUBDOMAIN 2	SUBDOMAIN 3	SUBDOMAIN 4
Relative permittivity	80	30	1	4
Electric conductivity	3	1.5	0.001	0.01

- 3 Click **OK**.

### Boundary Conditions

- 1 Open the **Boundary Settings** dialog box.



- 2 Select all exterior boundaries, that is, the boundaries that are not grayed out in the **Boundary selection** list. Select the **Scattering boundary condition** and the **Spherical wave** type.
- 3 Click **OK**.

#### *Edge Settings*

- 1 Open the **Edge Settings** dialog box.
- 2 Select Edge 17. In the **Current in edge segment direction** edit field, type  $1e4$ .
- 3 Click **OK**.

#### **MESH GENERATION**

- 1 Open the **Free Mesh Parameters** dialog box.
- 2 Click the **Custom mesh size** button and set the **Maximum element size** to  $1e3$ .
- 3 Click **OK**.
- 4 Click the **Initialize Mesh** button on the Main toolbar.

#### **COMPUTING THE SOLUTION**

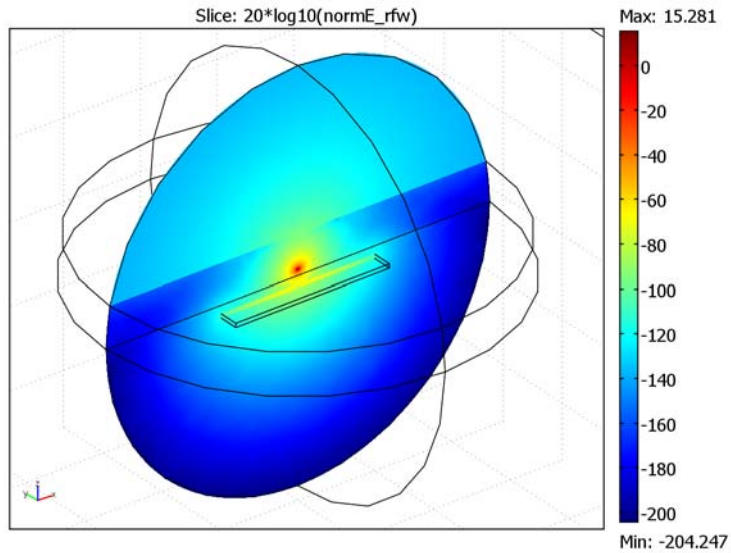
Click the **Solve** button on the Main toolbar.

#### **POSTPROCESSING AND VISUALIZATION**

The default plot shows the total energy density on a linear scale. Follow the instructions below to visualize the electric field magnitude on a dB scale.

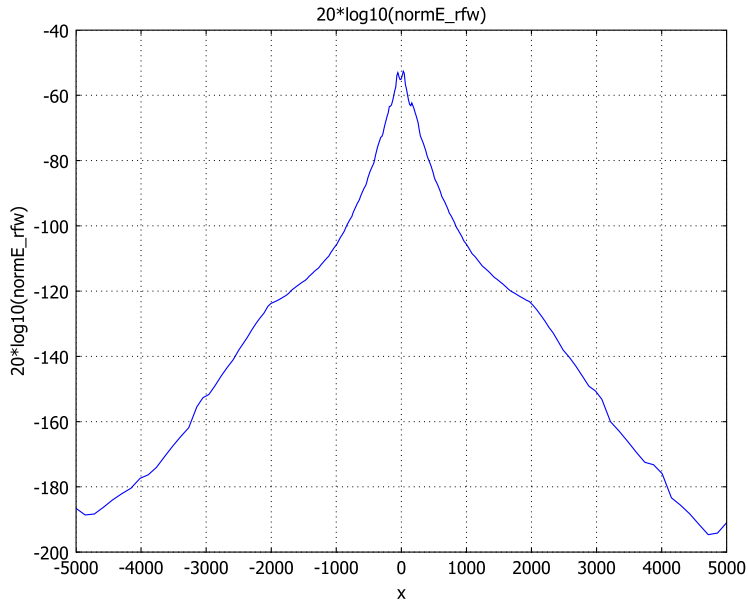
- 1 Open the **Plot Parameters** dialog box.
- 2 Click the **Slice** tab and type  $20 \cdot \log_{10}(\text{normE}_r\text{fw})$  in the **Expression** edit field.
- 3 In the **Slice positioning** area, set the number of **x levels** and **z levels** to 0 and number of **y levels** to 1.

4 Click **OK** to generate the plot below.



- 5 Open the **Domain Plot Parameters** dialog box from the **Postprocessing** menu.
- 6 Click the **Line/Extrusion plot** tab and type  $20 \cdot \log_{10}(\text{normE\_rfw})$  in the **Expression** edit field.
- 7 Set the **x-axis data** to x.
- 8 Select Edge 5 in the **Edge selection** list.

9 Click **OK** to generate the following plot.



Keep this figure window open in the background for later use.

### PHYSICS SETTINGS

Now it is time to simulate the model without the hydrocarbon reservoir.

#### *Subdomain Settings*

- 1 Open the **Subdomain Settings** dialog box.
- 2 Select Subdomain 4 and type in the subdomain settings from Subdomain 2.
- 3 Click **OK**.

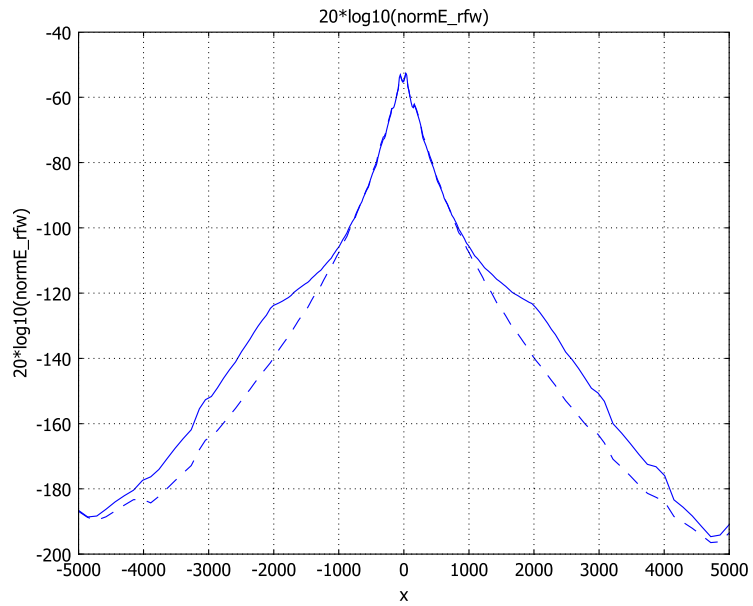
### COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar.

### POSTPROCESSING AND VISUALIZATION

- 1 Open the **Domain Plot Parameters** dialog box from the **Postprocessing** menu.
- 2 On the **Line/Extrusion** page, click the **Line Settings** button.
- 3 In the dialog box that opens, select **Dashed line** from the **Line type** list.
- 4 Click **OK** to close the **Line Settings** dialog.

- 5 Click the **General** tab. From the **Plot in** list, select **Figure 1**. Select the **Keep current plot** check box.
- 6 Select Edge 5 from the **Edge selection** list.
- 7 Click **OK** to generate the plot below.



## Optics and Photonics Models

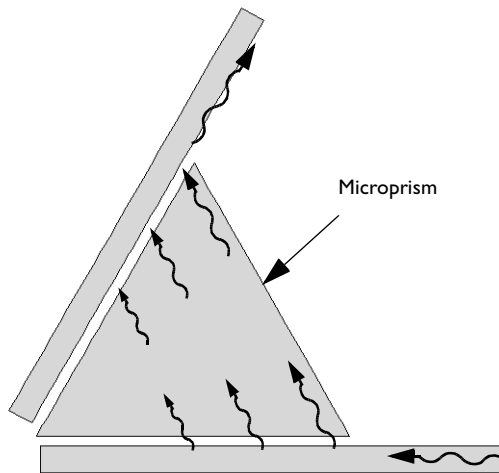
In this chapter you find models within Optics and Photonics. From the mathematical viewpoint, these models use the same physics formulations as in the previous chapter on RF and Microwave Engineering. The terminology and the way numerical results are presented differ slightly though. Anyone interested in RF and Microwave Engineering may benefit from also reading this chapter.

# Photonic Microprism

## *Introduction*

---

A microprism is used for reducing radiative losses in photonic waveguide bends. If you place a microprism between two waveguides forming a sharp bend, light will be guided between the waveguides through the prism.



For a certain refractive index of the prism, the light propagating through the prism couples to the respective mode under just the appropriate resonance angle (see Ref. 1). If the initial field distribution does not diffract while propagating through the prism, the coupling from the prism to the guide is the inverse to the light transfer from the guide to the prism. Therefore, the efficiency of the process is very high.

The interface between the guide and the prism must be sufficiently long to allow almost all the power to exit from the guide into the prism and vice versa. On the other hand, to avoid diffraction, the size of the prism should be kept as small as possible.

This model also makes use of *perfectly matched layers*, PMLs. These are domains with artificial absorption that reduce nonphysical reflections. For more information on PMLs, see Ref. 2 and the section “Perfectly Matched Layers (PMLs)” on page 43 in the *RF Module User’s Guide*.

## *Model Definition*

---

The model is built using the 2D In-Plane TE Waves application mode. The modeling takes place in the  $xy$ -plane.

### **DOMAIN EQUATIONS**

The dependent variable in this application mode is the  $z$  component of the electric field  $\mathbf{E}$ . It obeys the following relation:

$$\nabla \times (\mu_r^{-1} \nabla \times \mathbf{E}_z) - \left( \epsilon_r - \frac{j\sigma}{\omega\epsilon_0} \right) k_0^2 \mathbf{E}_z = 0$$

where  $\mu_r$  denotes the relative permeability,  $\omega$  the angular frequency,  $\sigma$  the conductivity,  $\epsilon_0$  the permittivity of vacuum,  $\epsilon_r$  the relative permittivity, and  $k_0$  the wave number. Different refractive indices are used for the prism and the guides. The wave is dampened by PMLs where the wave enters and exits the setup. The whole geometry is surrounded by another PML, which decreases reflections from the nonphysical exterior boundary. The solution is calculated for IR light with a wavelength in vacuum of 870 nm.

### **BOUNDARY CONDITIONS**

The exterior boundaries in this model use a scattering boundary condition to terminate the PML. Inside the geometry, continuity is applied everywhere except for at the boundary where the wave is entering the structure. This boundary is excited with a cosine function fitted to match the width of the waveguide.

## Results and Discussion

---

Figure 4-1 shows the geometry and the solution of the model. The wave enters the horizontal guide from the right and exits at the top of the vertical guide. The circles surrounding the entry and the exit are PMLs.

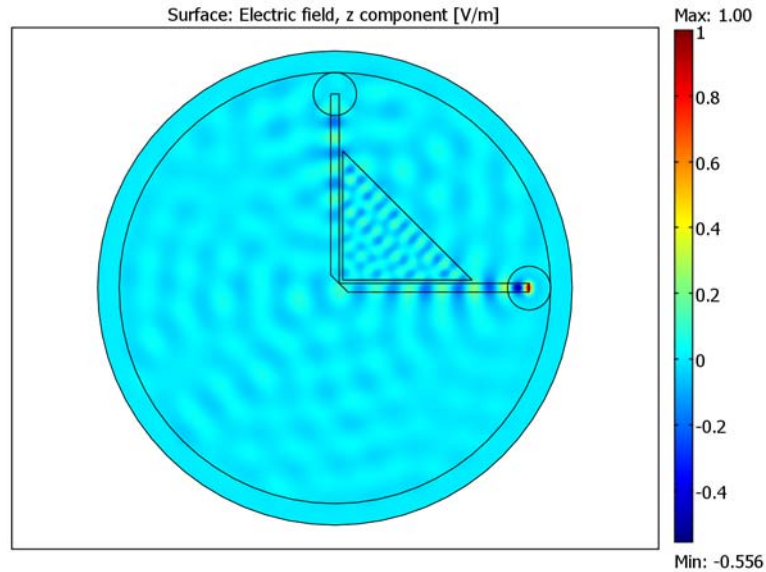


Figure 4-1: The z component of the electric field.

Several variables govern the transmission of the wave through the bend: for example, the relation between the refractive indices of the guide and the prism, the size of the prism, and the gap between the guides and the prism. The optimal values of the parameters also depend on the angle of the bend.

## References

---

1. CLEO 2001, *Conference on Lasers and Electro-Optics*, OSA-Optical Society of America, pp. 129–130.
2. Jianming Jin: *The Finite Element Method in Electromagnetics*, Second Edition, Wiley-Interscience.



---

**Model Library path:** RF\_Module/Optics\_and\_Photonics/  
photonic\_micro\_prism

---

## *Modeling Using the Graphical User Interface*

---

### MODEL NAVIGATOR

- 1 Select **2D** from the **Space dimension** list.
- 2 In the list of application modes, select **RF Module>In-Plane Waves>TE Waves>Harmonic propagation**.
- 3 Click **OK**.

### OPTIONS AND SETTINGS

- 1 In the **Axes/Grid Settings** dialog box, set axis and grid settings according to the following table; when done, click **OK**.

AXIS		GRID	
x min	-6	x spacing	0.5
x max	6	Extra x	-0.1 0.1 0.18 0.3 3.18
y min	-6	y spacing	0.5
y max	6	Extra y	-0.1 0.1 0.18 0.3 3.18

- 2 In the **Constants** dialog box, enter the following variable names, expressions, and descriptions (optional); when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
n_Guide	1.5	Refractive index, waveguide
n_Prism	2.5	Refractive index, microprism

### GEOMETRY MODELING

Start by defining the solid objects for the two guides and the prism.

- 1 Select **Draw Line** and click at (0.3, -0.1), (4.5, -0.1), (4.5, 0.1), and (0.1, 0.1). Create a solid object CO1 by clicking the right mouse button.
- 2 Select **Draw Line** and click at (0.1, 0.1), (0.1, 4.5), (-0.1, 4.5), and (-0.1, 0.3). Create a solid object CO2 by clicking the right mouse button.

- 3 Select **Draw Line** and click at (0.18, 3.18), (0.18, 0.18), and (3.18, 0.18). Create a solid object CO3 by clicking the right mouse button.

Then, define circular domains for the absorbing layers.

- 1 Draw a circle C1 with radius 5 and a circle C2 with radius 5.5, both centered at (0,0).
- 2 Draw a circle C3 centered at (4.5, 0) and a circle C4 centered at (0, 4.5), both with radius 0.5.
- 3 Unite all objects into one solid object by selecting all objects and clicking the **Union** toolbar button.

The model geometry is now ready, except for the size. The dimensions should be given in  $\mu\text{m}$ .

- 1 Click the **Scale** button on the Draw toolbar, and give the value  $1 \text{e-}6$  for both scaling factors.
- 2 Click the **Zoom Extents** button on the Main toolbar to see the resulting object.

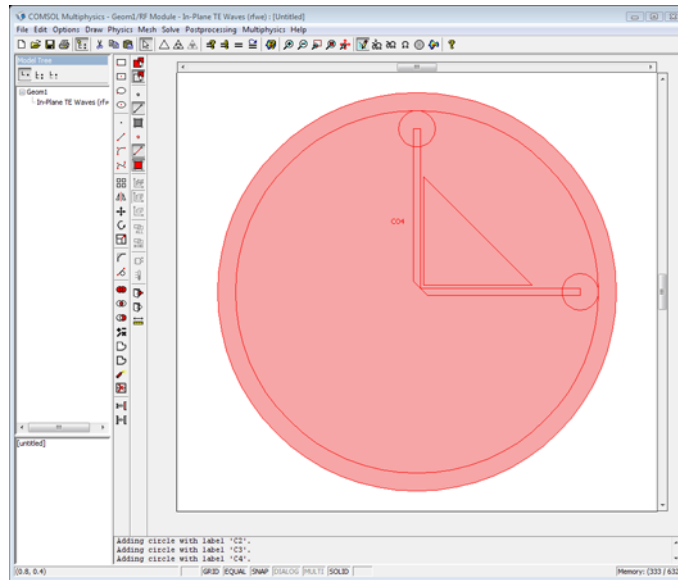


Figure 4-2: Model geometry.

## PHYSICS SETTINGS

### Scalar Variables

Set the frequency to  $3\text{e}8/870\text{e-}9$  in the **Scalar Variables** dialog box.

### Boundary Conditions

Enter the boundary conditions according to the following table. To apply boundary conditions to interior boundaries, it is necessary to select the **Interior boundaries** check box.

SETTINGS	BOUNDARY 15	BOUNDARIES 16, 17, 23, 28	ALL OTHER
Boundary condition	Electric field	Scattering boundary condition	Continuity
$E_{0z}$	$\cos(y*\pi/0.4[\text{um}])$	0	
Wave type		Cylindrical wave	

### Subdomain Settings

- Subdomains 1 and 5 represent air or vacuum. This is the default setting.
- Subdomain 7 is the prism. Choose to represent the material properties in terms of the refractive index, and enter `n_Prism` in the text field.
- Subdomains 2, 4, and 6 are the free part of the waveguide. Use `n_Guide` as the refractive index here.
- Subdomains 3, 8, and 9 are used for damping the wave. Choose to represent the material properties in terms of  $\epsilon_r$ ,  $\mu_r$ , and  $\sigma$ . Enter `n_Guide^2*Izz` for the **Relative permittivity** and `Ixx 0 0 Iyy` for the **Relative permeability**, which is anisotropic.
- Subdomain 10 also damps the wave to reduce the effect of the exterior boundary. Enter `Izz` for the **Relative permittivity** and `Ixx 0 0 Iyy` for the **Relative permeability**.

### Expression Variables

- Open the **Subdomain Expressions** dialog box from the **Options** menu. Mark all subdomains and specify the expressions for diagonal indices of the PML tensor.

NAME	EXPRESSION
Ixx	<code>sy*sz/sx</code>
Iyy	<code>sx*sz/sy</code>
Izz	<code>sx*sy/sz</code>

- For best results, the wave is damped along the expected direction of propagation, except for in the beginning of the waveguide, where it is damped in the perpendicular direction. To achieve this effect, use the following expressions.

SUBDOMAIN	3 4 8	9 10	ALL OTHER
sx	1	1 - i	1
sy	1 - i	1 - i	1
sz	1	1	1

## MESH GENERATION

- 1 Change the default mesh parameters to get an applicable mesh. Open the **Free Mesh Parameters** dialog box. On the **Global** tab, click the **Custom mesh size** button and set **Maximum element size** to  $250e-9$ . On the **Subdomain** tab, select the subdomains 2, 4, 6, 7, and 8, and set **Maximum element size** to  $120e-9$ .
- 2 Initialize the mesh.

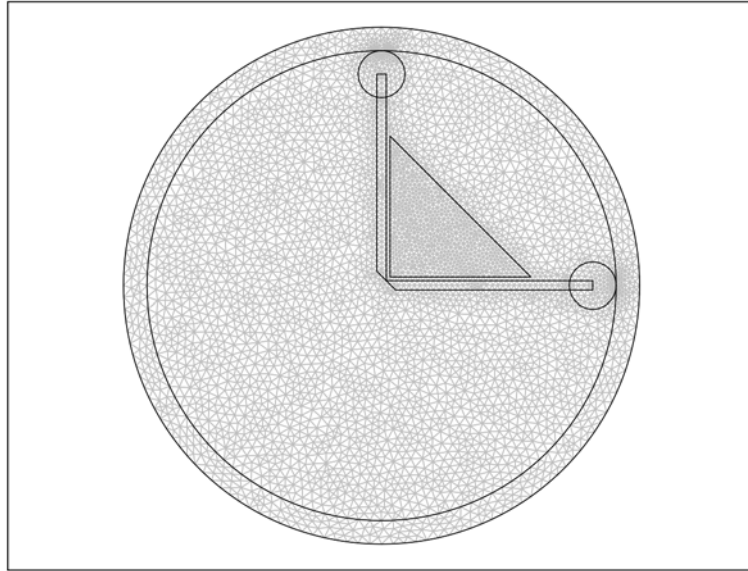


Figure 4-3: Mesh

## COMPUTING THE SOLUTION

- 1 Click the **Solve** button on the Main toolbar to compute the solution.

## POSTPROCESSING AND VISUALIZATION

By default, the  $z$  component of the electric field is visualized. It shows clearly that the wave propagates between the two guides, through the prism; see Figure 4-1. Another interesting entity for visualization is the electric energy density. Select **Electric energy density, time average** on the **Surface** tab of the **Plot Parameters** dialog box, and use the **cool** colormap. The resulting plot (Figure 4-4) shows that the energy density is mainly

localized to the waveguide and the prism, and that there are signs of resonances in the prism.

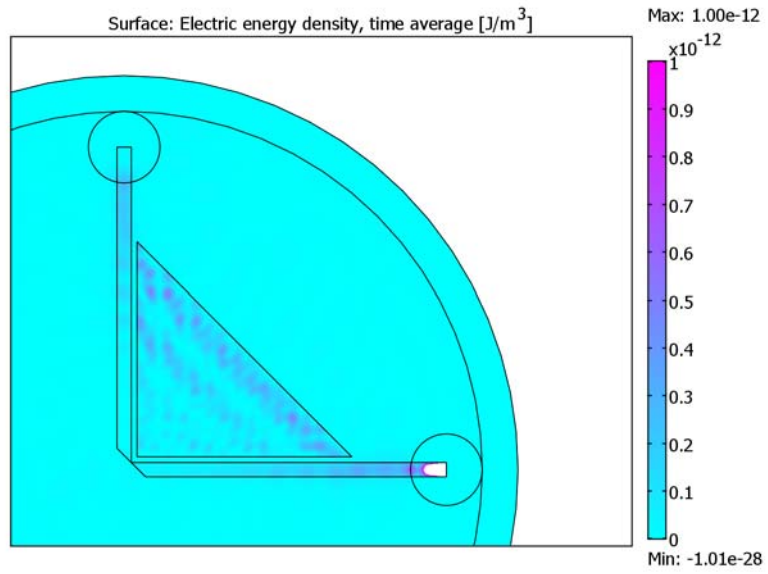


Figure 4-4: Electric energy density.

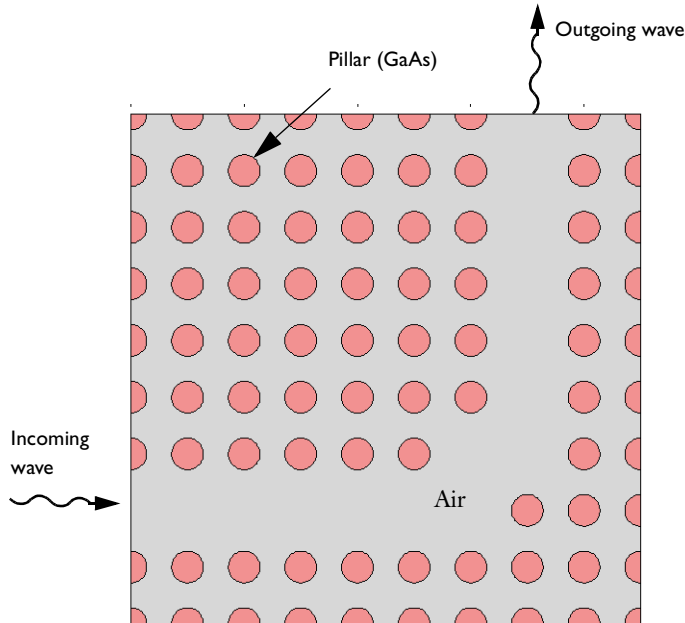
# Photonic Crystal

Photonic crystal devices are periodic structures of alternating layers of materials with different refractive indices. Waveguides that are confined inside of a photonic crystal can have very sharp low-loss bends, which may enable an increase in integration density of several orders of magnitude.

## *Introduction*

---

This model describes the wave propagation in a photonic crystal that consists of GaAs pillars placed equidistant from each other. The distance between the pillars prevents light of certain wavelengths to propagate into the crystal structure. Depending on the distance between the pillars, waves within a specific frequency range are reflected instead of propagating through the crystal. This frequency range is called the photonic bandgap (Ref. 1). By removing some of the GaAs pillars in the crystal structure you can create a guide for the frequencies within the bandgap. Light can then propagate along the outlined guide geometry.



## Model Definition

The geometry is a square of air with an array of circular pillars of GaAs as described above. Some pillars are removed to make a waveguide with a 90° bend.

The objective of the model is to study TE waves propagating through the crystal. To model these, use a scalar equation for the transverse electric field component  $E_z$ ,

$$-\nabla \cdot \nabla E_z - n^2 k_0^2 E_z = 0$$

where  $n$  is the refractive index and  $k_0$  is the free-space wave number.

Because there are no physical boundaries, you can use the scattering boundary condition at all boundaries. Set the amplitude  $E_z$  to 1 on the boundary of the incoming wave.

## Results and Discussion

Figure 4-5 contains a plot of the  $z$  component of the electric field. It clearly shows the propagation of the wave through the guide.

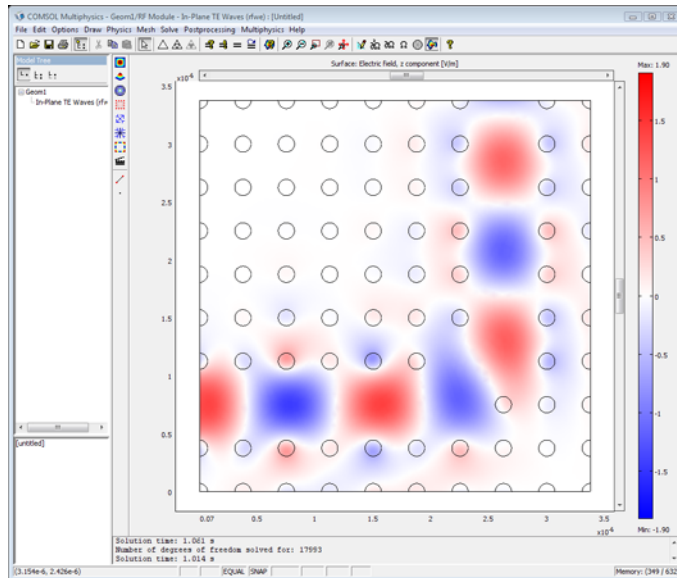


Figure 4-5: The  $z$  component of the electric field showing how the wave propagates along the path defined by the pillars.

If the angular frequency of the incoming wave is less than the cutoff frequency of the waveguide, the wave does not propagate through the outlined guide geometry. In Figure 4-6 the wavelength has been increased by a factor of 1.17.

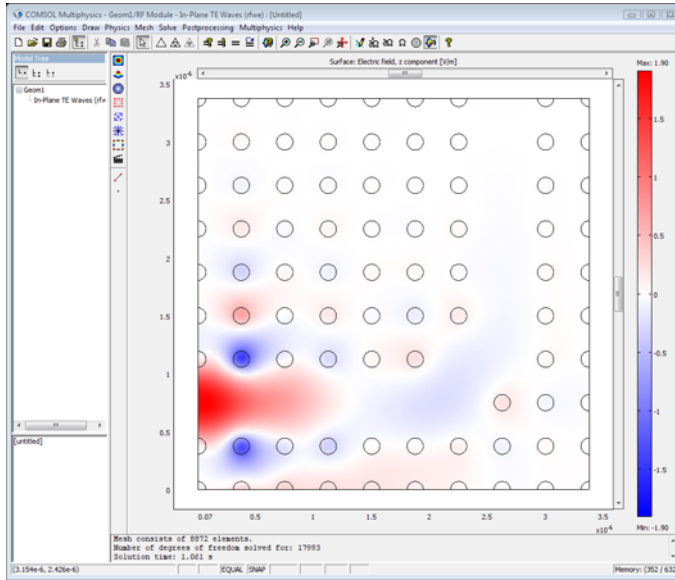


Figure 4-6: A longer wavelength will not propagate through the guide.

## References

1. J. D. Joannopoulos, R. D. Meade, J. N. Winn, *Photonic Crystals (Modeling the Flow of Light)*, Princeton University Press, 1995.
2. Chuang Shun Lien, *Physics of Optoelectronic Devices*, Wiley series in pure and applied optics, 1995.

---

**Model Library path:** RF\_Module/Optics\_and\_Photonics/photonic\_crystal

---

## Modeling Using the Graphical User Interface

---

### MODEL NAVIGATOR

- 1 Select **2D** from the **Space dimension** list.



- 2 In the list of application modes, select **RF Module>In-Plane Waves>TE Waves>Harmonic propagation**.
- 3 Click **OK**.

#### APPLICATION MODE PROPERTIES

For convenience, specify that the wavelength rather than the frequency should be used as input. In the **Application Mode Properties** dialog box set the property **Specify wave using** to **Free space wavelength**.

#### OPTIONS AND SETTINGS

- 1 From the **Options** menu, choose **Axes/Grid Settings**.
- 2 Set axis and grid settings according to the following table (clear the **Auto** check box to enter the grid settings).

AXIS		GRID	
x min	0	x spacing	5e-7
x max	5e-6	Extra x	7e-8
y min	-1e-6	y spacing	5e-7
y max	4e-6	Extra y	

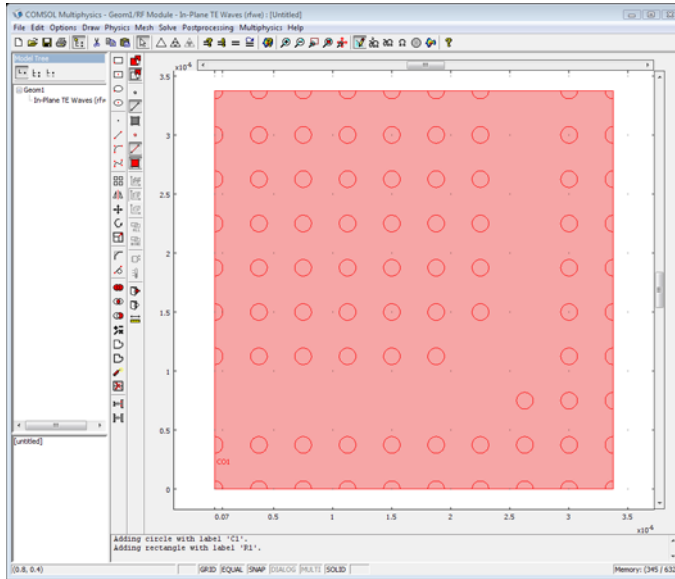
- 3 Click **OK**.

#### GEOMETRY MODELING

The easiest way to create the crystal geometry is using a array of geometry objects.

- 1 Start by drawing a circle with the radius  $7e-8$  and the center at  $x = 0, y = 0$ .
- 2 Select the circle and click the **Array** button. In the **Displacement** edit fields, type  $3.75e-7$  for the displacements in both directions and in the **Array size** edit fields, type 10 in both the  $x$  and  $y$  directions
- 3 Create the guide as a  $90^\circ$  bend by removing some pillars from the array. Remove the circles C3, C13, C23, C33, C43, C53, C63, C64, and C74 to C80.
- 4 Use the **Rectangle** dialog box to create a rectangle intersecting with all pillars. Set both the **Width** and **Height** to  $9 \times 3.75e-7$ .
- 5 Select all objects and open the **Create Composite Object** dialog box. Type  $R1 * (C1 + C2 + \dots + C99 + C100 + R1)$  in the **Set formula** edit field and click **OK** to create an object with the regions outside the rectangle removed. The sum within

parentheses in the set formula creates a union of all objects. The \* operator then takes the intersection of this union and the rectangle.



## PHYSICS SETTINGS

### Scalar Variables

In the **Application Scalar Variables** dialog box, set the wavelength to  $1e-6$ .

### Boundary Conditions

Use low-reflecting boundary conditions on all exterior boundaries. Apply a source at the input port. These settings are summarized in the following table.

SETTINGS	BOUNDARY 5	ALL OTHERS
Boundary condition	Scattering boundary condition	Scattering boundary condition
$E_{0z}$	1	0
Wave type	Plane wave	Plane wave

### Subdomain Settings

Enter subdomain settings according to the following table.

SETTINGS	SUBDOMAINS 1, 3–86	SUBDOMAIN2
$n$	n_GaAs	n_Air

Because the refractive index of GaAs is frequency dependent, define the refractive indices in the model using expression variables. The expression defines a linearized frequency dependency of the refractive index of GaAs between the refractive index values corresponding to the wavelengths 1.0332  $\mu\text{m}$  and 1.2339  $\mu\text{m}$  according to Ref. 2. In the **Options>Expressions>Scalar Expressions** dialog box, define the variables  $n_{\text{Air}}$  and  $n_{\text{GaAs}}$ . The descriptions are optional. When done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
$n_{\text{Air}}$	1	Refractive index, air
$n_{\text{GaAs}}$	$-3.3285e5*\lambda_{\text{rfwe}}[1/\text{m}]+3.5031$	Refractive index, GaAs

### MESH GENERATION

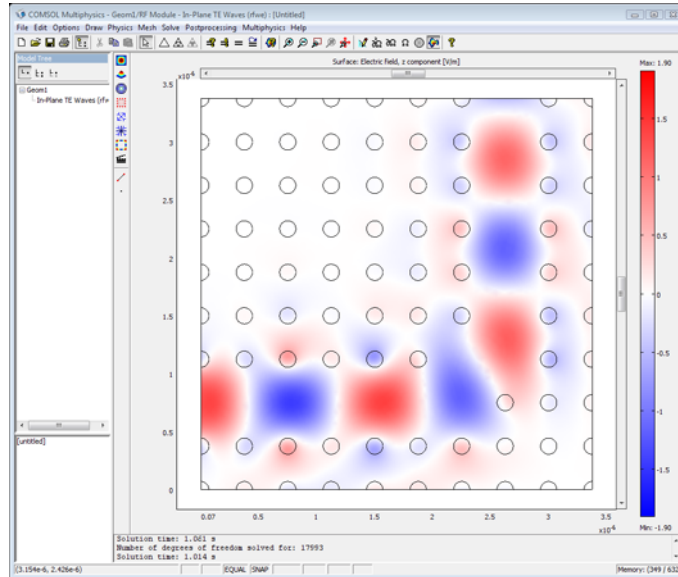
- 1 Change the default mesh parameters to get an applicable mesh. Open the **Free Mesh Parameters** dialog box, click the **Custom mesh size** button, and set **Element growth rate** to 1.55 and **Mesh curvature factor** to 0.65.
- 2 Initialize the mesh.

### COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar to solve the problem with the default solver.

## POSTPROCESSING AND VISUALIZATION

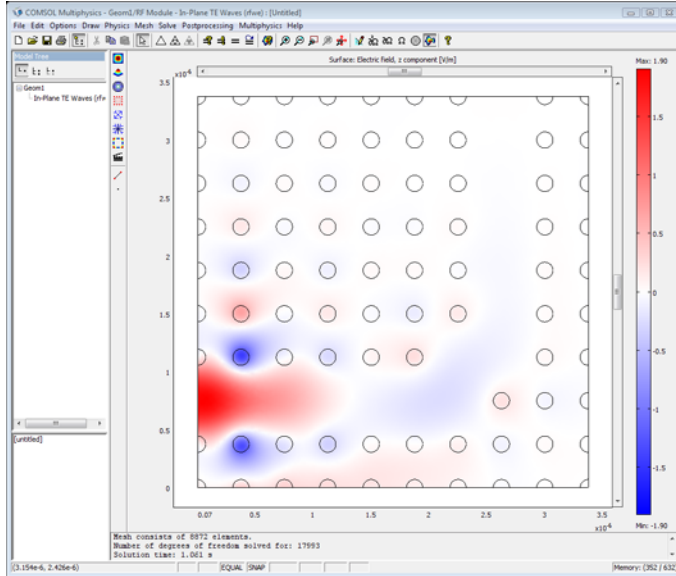
By default, the  $z$  component of the electric field is visualized. This clearly shows the propagation of the wave through the guide. Change the colormap to **wave** under the **Surface** tab in the **Postprocessing** dialog box. This makes it easier to identify the wave.



If the angular frequency of the incoming wave is less than the cutoff frequency of the waveguide, the wave will not propagate through the outlined guide geometry.

- 1 Open the **Application Scalar Variables** dialog box and multiply the wavelength by 1.17.
- 2 Click the **Solve** button.
- 3 Open the **Plot Parameters** dialog box to change the range of the plot. The wave colormap looks best if the range is symmetric around zero.
- 4 Click the **Range** button, then enter -1.9 and 1.9 in the **Min** and **Max** edit fields.

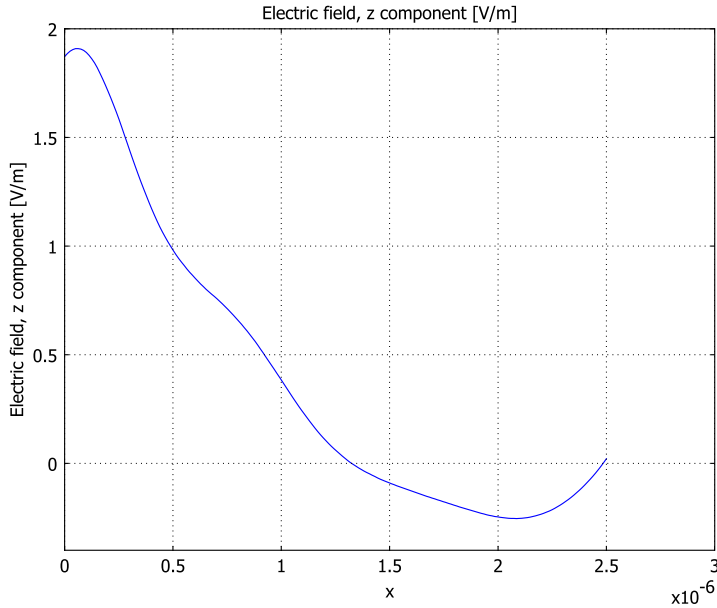
5 Click **OK** twice to see the following plot.



Use a cross-section line plot to visualize the evanescent electric field. Position the plot along the outlined guide between the inlet and the bend.

1 Open the **Cross-Section Plot Parameters** dialog box and select **Line plot**.

2 As **Cross-section line data** set **x0** to 0, **x1** to  $2.5e-6$ , **y0** to  $0.75e-6$ , and **y1** to  $0.75e-6$ . Set the **x-axis data** to **x**.



The resulting plot shows that the *z* component of the electric field declines exponentially along the plot line.

### *Modeling Using the Programming Language*

```
% Start by defining the variables.
clear fem appl
fem.expr.n_GaAs = '-3.3285e5*lambda0+3.5031';
fem.expr.n_Air = '1';

% Define the geometry.
latt_const = 3.75e-7;
oPillar = circ2(0,0,7e-8);
N = 10;
cPillars = geomarrayr(oPillar,latt_const,latt_const,N,N);
aRm_Pillars = [3 13 23 33 43 53 63 64 74:80];
bKeep_Pillars = logical(ones(1,N^2));
bKeep_Pillars(aRm_Pillars) = logical(0);
oPillars = geomcomp(cPillars(bKeep_Pillars));
oBox = rect2(0,(N-1)*latt_const,0,(N-1)*latt_const);
fem.geom = (oBox+oPillars)*oBox;
```

```

% Create the application structure and compute the corresponding
% FEM coefficients.
appl.mode = 'InPlaneWaves';
appl.prop.inputvar = 'lambda';
appl.var.lambda0 = 1e-6;
appl.bnd.type = {'cont' 'SC' 'SC'};
appl.bnd.ind = ones(1,340);
appl.bnd.ind(1:72) = 2;
appl.bnd.ind(5) = 3;
appl.bnd.E0 = {'0' '0' '0'} {'0' '0' '0'} {'0' '0' '1'};
appl.equ.matparams = 'n';
appl.equ.n = {'n_GaAs', 'n_Air'};
appl.equ.ind = ones(1,86);
appl.equ.ind(2) = 2;
fem.appl = appl;
fem = multiphysics(fem);

% Mesh and solve the problem using the proper parameters.
fem.mesh = meshinit(fem, 'hgrad', 1.55, 'hcurve', 0.65);
fem.xmesh = meshextend(fem);
fem.sol = femlin(fem);

% Visualize the resulting electric field.
postplot(fem, 'tridata', 'Ez', 'geom', 'on', 'tribar', 'on');

% Modify the angular frequency and re-solve the problem.
fem.appl.var.lambda0 = 1e-6*1.17;
fem = multiphysics(fem);
fem.xmesh = meshextend(fem);
fem.sol = femlin(fem);

% Visualize the new solution.
postplot(fem, 'tridata', 'Ez', 'geom', 'on', 'tribar', 'on');

% Make the cross section line plot.
postcrossplot(fem, 1, [0 2.5e-6;...
    0.75e-6 0.75e-6], 'lindata', 'Ez', 'title', '', ...
    'axislabel', {'width [m]' 'electric field [V/m]' ''});

```

# Bandgap Analysis of a Photonic Crystal

This model performs a bandgap analysis of a photonic crystal similar to the one used in the model “Photonic Crystal” on page 182.

## *Introduction*

---

The model investigates the wave propagation in a photonic crystal that consists of GaAs pillars placed equidistant from each other. The distance between the pillars determines a relationship between the wave number and the frequency of the light that prevents light of certain wavelengths to propagate inside the crystal structure. This frequency range is called the photonic bandgap (Ref. 2). There are several bandgaps for a certain structure, and this model extracts the bandgaps for the lowest bands of the crystal.

## *Model Definition*

---

This model is similar to the Photonic Crystal waveguide model. The difference is that in this model the crystal itself is analyzed instead of a waveguide. Because it has a repeated pattern it is possible to use periodic boundary conditions. As a result, only one pillar is needed for this simulation. The model contains a small asymmetry, which is not present in the photonic crystal waveguide model, to remove difficulties of eigenfunctions with identical eigenvalues.

There are two main complications with this bandgap analysis. Firstly, the refractive index of GaAs is frequency dependent. Secondly, the wave vector must be ramped for the band diagram. Although you can solve each of these complications with the eigenvalue solver separately, the two combined make it difficult without a script. However, it is possible to solve a nonlinear problem with the stationary solver, using the eigenvalue as an unknown (see “Example: Eigenmode Analysis of an L-Shaped Membrane with Rounded Corner” on page 425 in the *COMSOL Multiphysics Modeling Guide* for more details on such approaches). The equation for the eigenvalue is a normalization of the electric field, so the average field is unity over the domain. The nonlinear solver finds the correct eigenvalue with an updated refractive index to the found eigenvalue. Furthermore, the parametric solver can sweep the wave vector,  $k$ . The eigenvalue is equal to the squared wave vector in free space,



$$k_0^2 = \Lambda.$$

The eigenvalue is denoted  $\Lambda$  to avoid confusion with the free-space wavelength, which is denoted with  $\lambda_0$ . The relation between  $\Lambda$  and  $\lambda_0$  is

$$\lambda_0 = \frac{2\pi}{\sqrt{\Lambda}}.$$

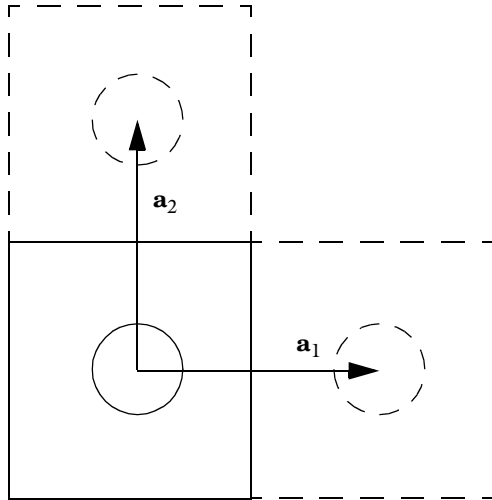
The wave vector for the propagating wave,  $k$ , enters the simulation as Floquet periodicity boundary conditions (Ref. 1),

$$E_z(2) = E_z(1)e^{-i\beta},$$

where  $\beta$  is a phase factor determined by the wave vector and the distance,  $d$ , between the periodic boundaries:

$$\beta = kd.$$

The range for the swept  $k$  is determined by the reciprocal lattice vectors of the photonic crystal, and these are determined from the primitive lattice vectors. For a 2D crystal there are two lattice vectors,  $\mathbf{a}_1$  and  $\mathbf{a}_2$ , defined in the following figure.



The reciprocal lattice vectors are calculated from  $\mathbf{a}_1$  and  $\mathbf{a}_2$  using the relations

$$\mathbf{b}_1 = 2\pi \frac{\mathbf{a}_2 \times \mathbf{a}_3}{\mathbf{a}_1 \cdot (\mathbf{a}_2 \times \mathbf{a}_3)},$$

$$\mathbf{b}_2 = 2\pi \frac{\mathbf{a}_3 \times \mathbf{a}_1}{\mathbf{a}_1 \cdot (\mathbf{a}_2 \times \mathbf{a}_3)},$$

where  $\mathbf{a}_3$  is assumed to be the unit vector  $\mathbf{e}_z$ . When  $\mathbf{a}_1$  and  $\mathbf{a}_2$  are perpendicular to each other and to  $\mathbf{a}_3$ ,  $\mathbf{b}_1$  and  $\mathbf{b}_2$  become

$$\mathbf{b}_1 = \frac{2\pi}{|\mathbf{a}_1|} \frac{\mathbf{a}_1}{|\mathbf{a}_1|},$$

$$\mathbf{b}_2 = \frac{2\pi}{|\mathbf{a}_2|} \frac{\mathbf{a}_2}{|\mathbf{a}_2|}.$$

The solution process is rather complicated, and you have to find proper initial conditions for the nonlinear parameter ramp. This is crucial because the system has several solutions, one for each eigenvalue. In COMSOL Multiphysics you can first use the eigenvalue solver to locate an approximate solution at any  $k$ -vector. The solution is not exact due to the frequency-dependent refractive index of GaAs. It is possible to repeat the eigenvalue calculation to get closer to the final solution. You can often switch directly to the time-harmonic analysis type after the first iteration of the eigenvalue problem, using that solution as initial guess for the nonlinear solver. With the nonlinear solver you can now perform a nonlinear ramp from  $k = 0$  to  $k = 0.5$ , which represents half the reciprocal vector given by a linear combination of  $\mathbf{b}_1$  and  $\mathbf{b}_2$  in some predefined direction, for example (1, 1).

The solution steps described above can be automated with a script in COMSOL Script or MATLAB. Note that it is not possible to automate it with solver scripting because you have to change the analysis type before switching to the nonlinear solver. The automation is recommended if you are interested in doing several sweeps for different bands and for different  $k$  directions.

## Results and Discussion

Figure 4-7 contains the field profile for  $k = 0.5$  in the  $(1, 1)$  direction. The frequency for this band and  $k$ -vector is 395 THz, and the  $k = 0$  eigenvalue for this band corresponds to a frequency of 423 THz.

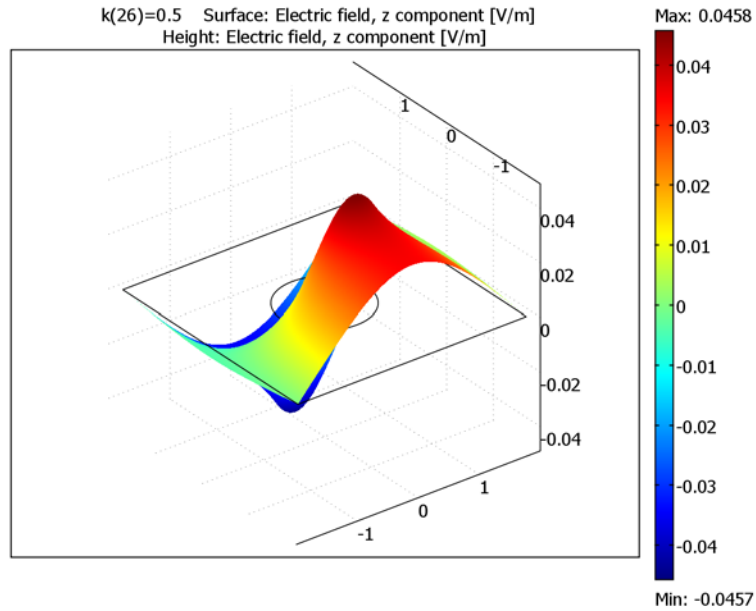
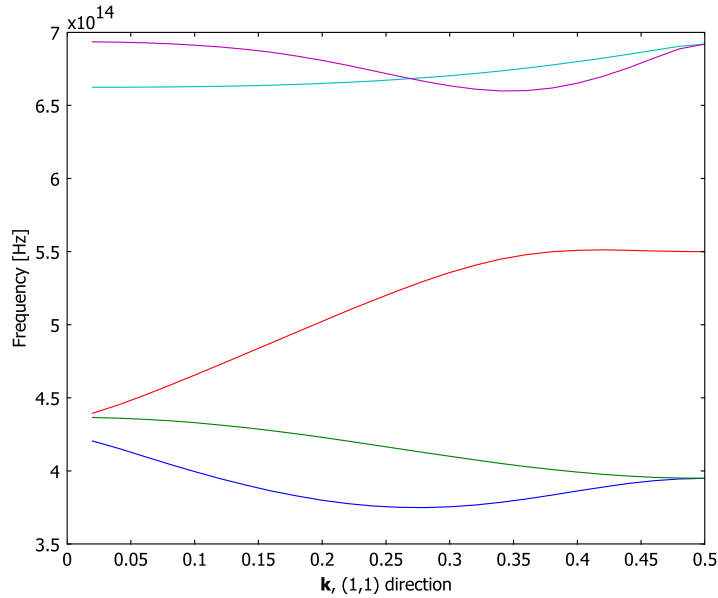


Figure 4-7: Field profile,  $E_z$ , at  $k = 0.5$  for the first band in the  $(1, 1)$  direction.

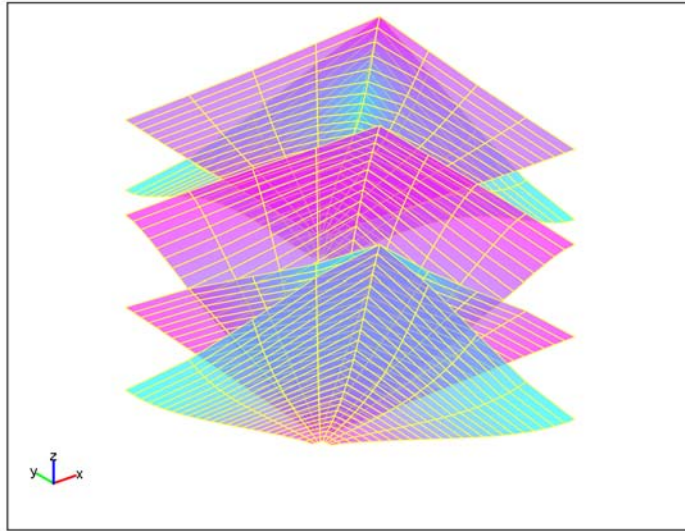
The five lowest bands for the (1, 1) direction appear in Figure 4-8 below. The three lowest bands lie close and do not really show any true bandgap. A bandgap appears between the third and fourth band.



*Figure 4-8: Band diagram of the bands in the (1, 1) direction.*

Using a script, it is also possible to perform a sweep in different directions and for several bands, together with the sweep of the  $k$ -vector magnitude. Such an analysis results in several band surfaces in  $k$ -space. Figure 4-9 on page 197 shows the five lowest band surfaces obtained using the script created under the section “Modeling Larger Sweeps With COMSOL Script” on page 204. There is a large bandgap between

the third and fourth band surface, and no waves in this frequency range can propagate in the crystal.



*Figure 4-9: Band surfaces for the five lowest bands for one quadrant in  $k$ -space.*

### *References*

- 
1. C. Kittel, *Introduction to Solid State Physics*, 7th ed., John Wiley & Sons, New York, 1996.
  2. J. D. Joannopoulos, R. D. Meade, J. N. Winn, *Photonic Crystals (Modeling the Flow of Light)*, Princeton university press, 1995.
- 

**Model Library path:** RF\_Module/Optics\_and\_Photonics/  
bandgap\_photonic\_crystal

---

**MODEL NAVIGATOR**

- 1 Select **2D** from the **Space dimension** list.
- 2 In the list of application modes, select **RF Module>In-Plane Waves>TE Waves>Eigenfrequency analysis**.
- 3 Click **OK**.

**OPTIONS AND SETTINGS**

- 1 From the **Options** menu, choose **Constants**.
- 2 In the **Constants** dialog box, define the following constants with names, expressions, and (optional) descriptions:

NAME	VALUE	DESCRIPTION
k	0	Fraction of wave vector magnitude
k1	1	1st component of wave direction vector
k2	1	2nd component of wave direction vector
a1x	375[nm]	x-component of 1st lattice vector
a1y	0[nm]	y-component of 1st lattice vector
a2x	0[nm]	x-component of 2nd lattice vector
a2y	375[nm]	y-component of 2nd lattice vector

*Scalar Expressions*

- 1 From the **Options** menu, choose **Expressions>Scalar Expressions**.
- 2 In the **Scalar Expressions** dialog box, define the following variables with names, expressions, and (optional) descriptions:

NAME	EXPRESSION	DESCRIPTION
kx	$k*(k1*b1x+k2*b2x)$	1st k-component for periodic condition
ky	$k*(k1*b1y+k2*b2y)$	2nd k-component for periodic condition
b1x	$2*pi*a2y/(a1x*a2y-a1y*a2x)$	x-component of 1st reciprocal lattice vector
b1y	$-2*pi*a2x/(a1x*a2y-a1y*a2x)$	y-component of 1st reciprocal lattice vector
b2x	$-2*pi*a1y/(a1x*a2y-a1y*a2x)$	x-component of 2nd reciprocal lattice vector

NAME	EXPRESSION	DESCRIPTION
b2y	$2\pi a_1x / (a_1x a_2y - a_1y a_2x)$	y-component of 2nd reciprocal lattice vector
n_Air	1	Refractive index of air
n_GaAs	$3.3285e5 [1/m] \lambda + 3.5031$	Dispersive refractive index of GaAs

### GEOMETRY MODELING

- 1 From the **Draw** menu, choose **Specify Objects>Rectangle**.
- 2 In the dialog box that appears, define the rectangle properties according to the table below.

WIDTH	HEIGHT	BASE	(X, Y)
$3.75e-7$	$3.75e-7$	Center	(0,0)

- 3 From the **Draw** menu, choose **Specify Objects>Circle**.
- 4 In the dialog box that appears, define the circle properties according to the table below.

RADIUS	BASE	(X, Y)
$7e-8$	Center	(0,0)

- 5 Finally, click the **Zoom Extents** button on the Main toolbar to simplify the boundary and subdomain selection.

### PHYSICS SETTINGS

#### *Boundary Conditions*

- 1 Open the **Boundary Settings** dialog box from the **Physics** menu.
- 2 Select all boundaries and select the **Periodic condition** from the **Boundary condition** list.
- 3 From the **Type of periodicity** list, choose **Floquet periodicity**.
- 4 Enter  $k_x$  and  $k_y$  in the edit fields for the **k**-vector.
- 5 The selected boundaries represents two groups of periodic conditions, one between Boundaries 1 and 4, and the other between Boundaries 2 and 3. You must specify a different periodic pair index to separate them. Select Boundaries 2 and 3 and enter 2 in the **Periodic pair index** edit field.
- 6 Click **OK**.

### Subdomain Settings

- 1 Open the **Subdomain Settings** dialog box from the **Physics** menu and specify the subdomain settings according to the following table. Prior to specifying the refractive index,  $n$ , it is necessary to click the **Specify material properties in terms of refractive index** radio button.

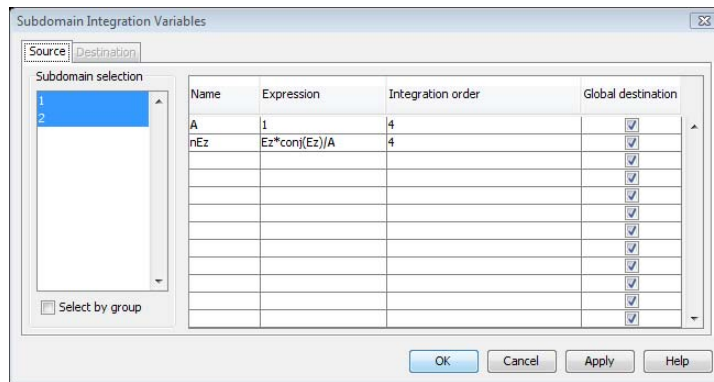
PROPERTY	SUBDOMAIN 1	SUBDOMAIN 2
$n$	$n_{\text{Air}}$	$n_{\text{GaAs}}$

- 2 Click the **Init** tab and enter  $E_z$  in the  **$E_z(t_0)$**  edit field for both subdomains.
- 3 Click **OK**.

### Integration Coupling Variables

When the nonlinear solver calculates the eigenvalue it is necessary to provide a second equation to the system. This model uses a normalization taking the average value of the squared magnitude of the electric field over the entire domain. The average value is calculated using an integration coupling variable, and an ODE forces that variable equal to 1.

- 1 From the **Options** menu choose **Integration Coupling Variables>Subdomain Variables**.
- 2 In the dialog box that appears, define two variables according to the table below. Make sure that the **Global destination** check box is selected for both variables.



NAME	SUBDOMAIN	EXPRESSION
A	1,2	1
nEz	1,2	$E_z \cdot \text{conj}(E_z) / A$

- 3 Click **OK**.



## MESH GENERATION

- 1 Open the **Free Mesh Parameters** dialog box from the **Mesh** menu.
- 2 Select the **Finer** mesh size from the **Predefined mesh sizes** list.
- 3 Click **Remesh** and then click **OK**.

## COMPUTING THE EIGENVALUE SOLUTION

As a first step, calculate a good initial guess for the  $f_{\text{req}}$  ODE variable and the electric field, using the eigenvalue analysis of the photonic crystal. Because the refractive index depends on the wavelength, it is necessary to provide a linearization point for the eigenvalue.

- 1 Open the **Solver Parameters** dialog box from the **Solve** menu. Make sure that **Eigenfrequency** is chosen in the **Analysis** list.
- 2 Type 1 in the **Desired number of eigenfrequencies** edit field, and type  $4e14$  in the **Search for eigenfrequencies around** edit field.
- 3 Click the **Eigenfrequency** tab, and enter  $-i*2*\pi*4e14$  in the **Eigenvalue linearization point** edit field. This is the eigenvalue that corresponds to the eigenfrequency  $4e14$  Hz.
- 4 Click the **Advanced** tab. Select the **Use Hermitian transpose of constraint matrix and in symmetry detection** check box. Click **OK**.
- 5 Click the **Solve** button on the Main toolbar. The calculated solution has not reached the correct eigenvalue, because the frequency  $4 \cdot 10^{14}$  Hz was used for the refractive index calculation of GaAs and the returned eigenvalue is close to  $4.2 \cdot 10^{14}$  Hz. It is possible to update the linearization point and solve again, but for the parametric sweep defined later it is more efficient to solve a nonlinear problem. The current eigenvalue solution is good enough as an initial guess for that nonlinear parametric sweep.

## COMPUTING THE NONLINEAR SOLUTION

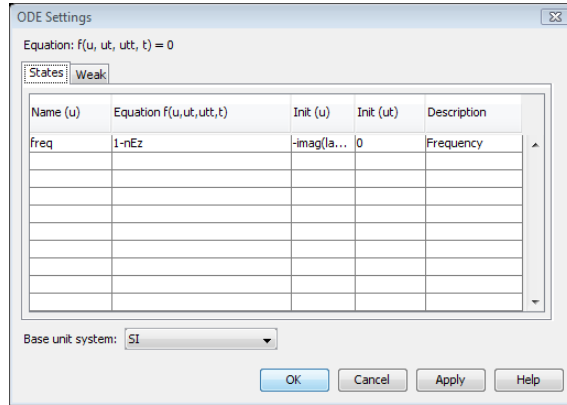
Now you use the eigenvalue solution as initial guess for the nonlinear problem. The use of the nonlinear solver enables sweeping of variables with the parametric solver. Here you ramp the magnitude of the  $k$ -vector from 0 to 0.5, which is in units of the reciprocal lattice vectors,  $\mathbf{b}_1$ ,  $\mathbf{b}_2$ ,  $\mathbf{b}_3$ .

- 1 Open the **Solver Parameters** dialog box from the **Solve** menu. Choose **Harmonic propagation** from the **Analysis** list.
- 2 Select the **Parametric** solver from the **Solver** list. Type  $k$  in the **Parameter name** edit field and  $0:0.02:0.5$  in the **Parameter values** edit field.

- 3 Click the **Parametric** tab. Select the **Manual tuning of parameter step size** check box, and type 1 in the **Minimum step size** and **Maximum step size** edit fields. Click **OK**.

### Space-Independent Equations

- 1 From the **Physics** menu, choose **Space-Independent Equations**.
- 2 In the **Space-Independent Equations** dialog box, define the following variable and equation:



NAME (U)	EQUATION	INIT (U)	DESCRIPTION
freq	1 - nEz	- imag(lambda) / (2 * pi)	Frequency

- 3 Click **OK**.

### Scalar Variables

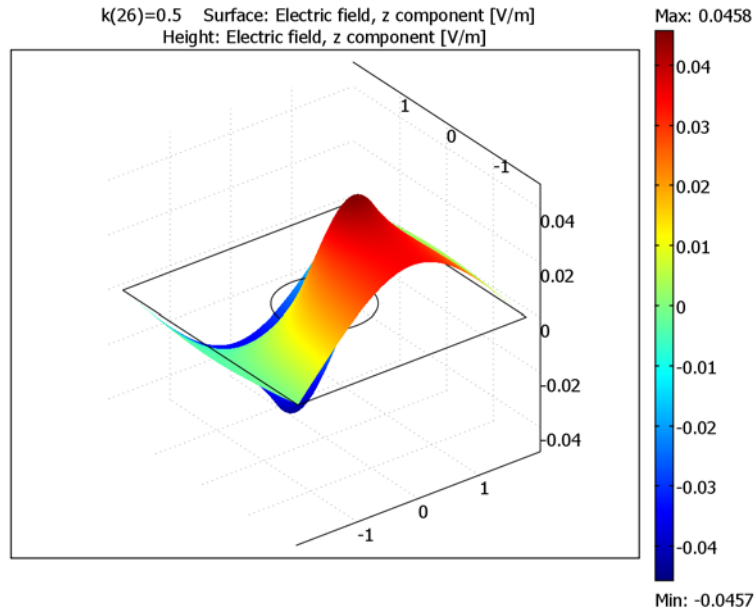
- 1 Choose **Scalar Variables** from the **Physics** menu.
- 2 In the dialog box that appears, enter the value `freq` for the `nu_rfwe` variable. Click **OK**.
- 3 Click the **Solve** button on the Main toolbar. COMSOL Multiphysics uses the latest eigenvalue to calculate the initial guess for the ODE variable `freq`.

During the sweep, the solver uses the latest solution for both `freq` and `Ez`, tracing the band starting at  $k = 0$  when  $k$  increases. You need to choose the step size of  $k$  carefully, because a too large step may cause the solution to “jump” to a different band if the bands lie close and have similar field profiles. This is hard to avoid in some situations, because the bands can meet in some points.

## POSTPROCESSING AND VISUALIZATION

By default you see the field profile for the current solution in the main window. This profile can be used to check that the  $k$ -vector sweep stays on the same band. The field profile changes only slightly when the magnitude of the  $k$ -vector increases, and a sudden change in field profile between two  $k$ -vector steps can indicate that the solution has jumped to a different band.

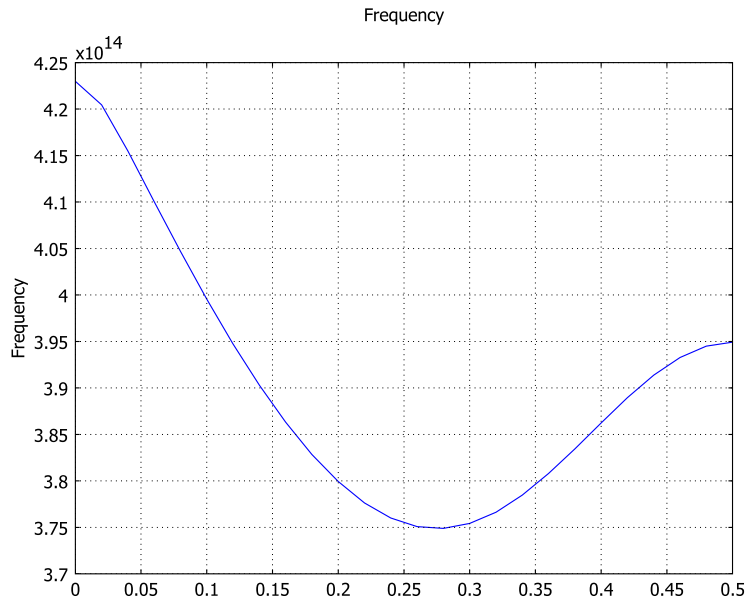
Click the **3D Surface Plot** toolbar button to get a height scale on the default surface plot.



The most interesting plot is the band diagram of the photonic crystal. Follow the steps below to generate the band diagram for one band.

- 1 Choose **Global Variables Plot** from the **Postprocessing** menu.
- 2 In the **Global Variables Plot** dialog box, select the variable **Frequency** from the **Predefined quantities** list and press the **>** button. This is the ODE variable  $f_{req}$ .

3 Click **OK** to see the following plot.



If you plan to do another sweep for a different band, just repeat the steps starting from “Computing the Eigenvalue Solution” on page 201. You have to change the eigenvalue shift and linearization point to match the new band for  $k = 0$ . If you want to plot the new band in the same figure window, make sure that the **Keep current plot** check box is selected in the **Global Variables Plot** dialog box.

### *Modeling Larger Sweeps With COMSOL Script*

---

More than three sweeps are best performed using a script or function. The following script takes the `fem` structure,  $k$  magnitudes, directions, and eigenfrequency shifts, and it then performs several sweeps over all the possible variations. If the direction spans several directions in  $k$ -space, you can plot the results as a surface for each band.

- 1 In COMSOL Multiphysics, press Ctrl+F. This exports the current model to the COMSOL Script command line. The model is now available at the command prompt through the structure named `fem`.
- 2 Open the COMSOL Script text editor by typing `edit` at the command prompt. It is also possible to use a different text editor; just make sure to save the script in a the

current path used in COMSOL Script or in its search path. You can view the current path with the command `pwd`, and the search path with the command `path`.

- 3 In the text editor, enter the following sequence of commands, which controls a sweep over five bands and one quadrant in  $k$ -space. Save the file with an appropriate name.

```
% k-vector factor to sweep
k = 0.02:0.02:0.5;
% Directions to create band surfaces
dir0 = [ones(1,5) 0.75:-0.25:0 ; 0:0.25:1 ones(1,4)];
% Number of bands
nbands = 5;
% Frequency linearization point
freq0 = 4e14;

% Update magnitude
fem.const{2*strmatch('k',fem.const(1:2:end),'exact')} = k(1);

% Clear frequency variable
nu = [];
% Loop over all directions
for dir=dir0
    % Update direction
    fem.const{2*strmatch('k1',fem.const(1:2:end), ...
                        'exact')} = dir(1);
    fem.const{2*strmatch('k2',fem.const(1:2:end), ...
                        'exact')} = dir(2);
    % Switch to eigenvalue analysis
    fem.appl{1}.prop.analysis = 'eigen';
    fem = multiphysics(fem);
    fem.xmesh = meshextend(fem);
    % Solve eigenvalue problem
    init = asseminit(fem,'init','0');
    fem.sol=femeig(fem, ...
                  'init',init, ...
                  'conjugate','on', ...
                  'solcomp',{'Ez'}, ...
                  'neigs',nbands, ...
                  'shift',-i*2*pi*freq0, ...
                  'eigref',sprintf('-i*2*pi*%g',freq0));
    % Store solution guess for all bands
    sole = fem.sol;
    % Switch to harmonic analysis
    fem.appl{1}.prop.analysis = 'harmonic';
    fem = multiphysics(fem);
    fem.xmesh = meshextend(fem);
    % Loop over all bands
    for ind=1:length(sole.lambda)
        % Print some progress information
```

```

disp(sprintf(['Direction = (%g,%g), ' ...
            'Band number %i'],dir,ind));
init = assemnit(fem,'u',sole,'solnum',ind);
% Sweep in direction using eigenvalue solution guess
fem.sol=femnlin(fem, ...
               'init',init, ...
               'conjugate','on', ...
               'pname','k', ...
               'plist',k, ...
               'porder',0, ...
               'pminstep',0.01, ...
               'pmaxstep',0.01);
nu = cat(1,nu,postint(fem,'nu_rfwe','edim',0,'dl',1,...
                    'solnum','all'));
end
end
% Reshape the nu data to have bands in the 3:rd dimension
nu = reshape(nu,[nbands size(dir0,2) length(k)]);
nu = permute(nu,[2 3 1]);
% Save data for later use
save band_diagram_data k dir0 nu

% Create a grid for plotting
[K,DIRx] = meshgrid(k,dir0(1,:));
[K,DIRy] = meshgrid(k,dir0(2,:));
X = DIRx.*K;
Y = DIRy.*K;
% Plot one surface for each band
figure(2);
hold on;
for ind=1:size(nu,3)
    h = mesh(X,Y,real(nu(:,:,ind)));
    % Set transparency and colors
    set(h,'EdgeColor',[1 1 0.3],'FaceColor','interp',...
        'ColorMap','cool(1024)','FaceAlpha',0.6);
end
% Band diagram in (1,1) direction
figure(3);
plot(k,permute(real(nu(5,:,:)),[3 2 1]));
xlabel('<b>k</b>, (1,1) direction');
ylabel('Frequency [Hz]');

```

You can find more information on the script commands in the *COMSOL Script User's Guide* and the *COMSOL Multiphysics Reference Guide*.

- 4 Once you saved the script, you can execute it from the command prompt by typing its name without the extension. If you, for example, named the file `band_diagram_sweep.m`, type the following at the command line:

`band_diagram_sweep`

The script takes a long time to solve because it calculates so many data points.

# Step-Index Fiber

## *Introduction*

---

The transmission speed of optical waveguides is superior to microwave waveguides because optical devices have a much higher operating frequency than microwaves, enabling a far higher bandwidth.

Today the silica glass ( $\text{SiO}_2$ ) fiber is forming the backbone of modern communication systems. Before 1970, optical fibers suffered from large transmission losses, making optical communication technology merely an academic issue. In 1970, researchers showed, for the first time, that low-loss optical fibers really could be manufactured. Earlier losses of 2000 dB/km now went down to 20 dB/km. Today's fibers have losses near the theoretical limit of 0.16 dB/km at 1.55  $\mu\text{m}$  (infrared light).

One of the winning devices has been the single-mode fiber, having a step-index profile with a higher refractive index in the centre core and a lower index in the outer cladding. Numerical software plays an important role in the design of single-mode waveguides and fibers. For a fiber cross section, even the most simple shape is difficult and cumbersome to deal with analytically. A circular step-index waveguide is a basic shape where benchmark results are available (see Ref. 1).

This example is a model of a single step-index waveguide made of silica glass. The inner core is made of pure silica glass with refractive index  $n_1 = 1.4457$  and the cladding is doped, with a refractive index of  $n_2 = 1.4378$ . These values are valid for free-space wavelengths of 1.55  $\mu\text{m}$ . The radius of the cladding is chosen to be large enough so that the field of confined modes is zero at the exterior boundaries.

For a confined mode there is no energy flow in the radial direction, thus the wave must be evanescent in the radial direction in the cladding. This is true only if

$$n_{\text{eff}} > n_2$$

On the other hand, the wave cannot be radially evanescent in the core region. Thus

$$n_2 < n_{\text{eff}} < n_1$$

The waves are more confined when  $n_{\text{eff}}$  is close to the upper limit in this interval.



### *Model Definition*

---

The mode analysis is made on a cross-section in the  $x$ - $y$  plane of the fiber. The wave propagates in the  $z$  direction and has the form

$$\mathbf{H}(x, y, z, t) = \mathbf{H}(x, y)e^{j(\omega t - \beta z)}$$

where  $\omega$  is the angular frequency and  $\beta$  the propagation constant. An eigenvalue equation for the magnetic field  $\mathbf{H}$  is derived from Helmholtz equation

$$\nabla \times (n^{-2} \nabla \times \mathbf{H}) - k_0^2 \mathbf{H} = \mathbf{0}$$

which is solved for the eigenvalue  $\lambda = -j\beta$ .

As boundary condition along the outside of the cladding the magnetic field is set to zero. Because the amplitude of the field decays rapidly as a function of the radius of the cladding this is a valid boundary condition.

### *Results and Discussion*

---

When studying the characteristics of optical waveguides, the effective mode index of a confined mode,

$$n_{\text{eff}} = \frac{\beta}{k_0}$$

as a function of the frequency is an important characteristic. A common notion is the normalized frequency for a fiber. This is defined as

$$V = \frac{2\pi a}{\lambda_0} \sqrt{n_1^2 - n_2^2} = k_0 a \sqrt{n_1^2 - n_2^2}$$

where  $a$  is the radius of the core of the fiber. For this simulation, the effective mode index for the fundamental mode, 1.4444 corresponds to a normalized frequency of 4.895. The electric and magnetic fields for this mode is shown in Figure 4-10 below.

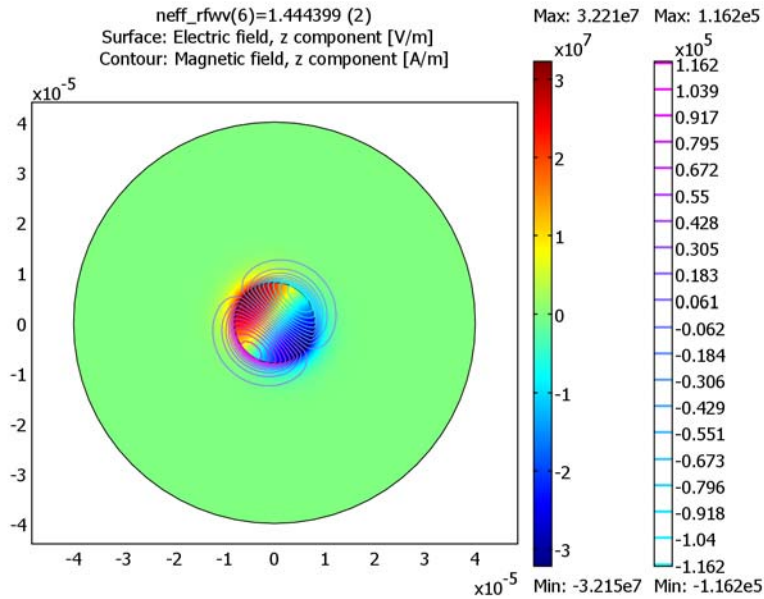


Figure 4-10: The color scale is the z component of the electric field and the contour lines is the z component of the magnetic field. This plot is for the effective mode index 1.4444.

### Reference

1. Amnon Yariv, *Optical Electronics in Modern Communications*, 5th edition, Oxford University Press, 1997.

**Model Library path:** RF\_Module/Optics\_and\_Photonics/step\_index\_fiber

### Modeling Using the Graphical User Interface

#### MODEL NAVIGATOR

I In the **Model Navigator**, select **2D** in the **Space dimension** list.

- 2 Select the **RF Module>Perpendicular Waves>Hybrid-Mode Waves>Mode analysis** application mode.
- 3 Click **OK**.

#### APPLICATION MODE PROPERTIES

- 1 Select **Properties** in the **Physics** menu to open the **Application Mode Properties** dialog box.
- 2 For convenience set the property **Specify wave using** to **Free space wavelength**. This makes the wavelength available in the **Application Scalar Variables** dialog box instead of the frequency.

#### OPTIONS AND SETTINGS

- 1 In the **Constants** dialog box, enter the following names and expressions for the refractive indices.

NAME	EXPRESSION
nSilicaGlass	1.4457
nSilicaGlassDoped	1.4378

#### GEOMETRY MODELING

- 1 Draw a circle C1 centered at (0,0) with radius 8e-6.
- 2 Draw a second circle C2 centered at (0,0). This time with the radius 40e-6.
- 3 Click the **Zoom Extents** button for a full visualization of the two circles.

#### PHYSICS SETTINGS

##### *Scalar Variables*

In the **Application Scalar Variables** dialog box, set the free space wavelength  $\lambda_{0\_rfwv}$  to 1.55e-6.

##### *Boundary Conditions*

Use the default boundary conditions on all exterior boundaries.

##### *Subdomain Settings*

In the **Subdomain Settings** dialog box enter the refractive index in the two domains according to the following table.

SETTING	SUBDOMAIN 1	SUBDOMAIN 2
n	nSilicaGlassDoped	nSilicaGlass

## MESH GENERATION

- 1 Initialize the mesh.
- 2 Refine the mesh once.

## COMPUTING THE SOLUTION

The modes of interest have an effective mode index somewhere between the refractive indices of the two materials, that is,

$$1.4378 < n_{\text{eff}} < 1.4457$$

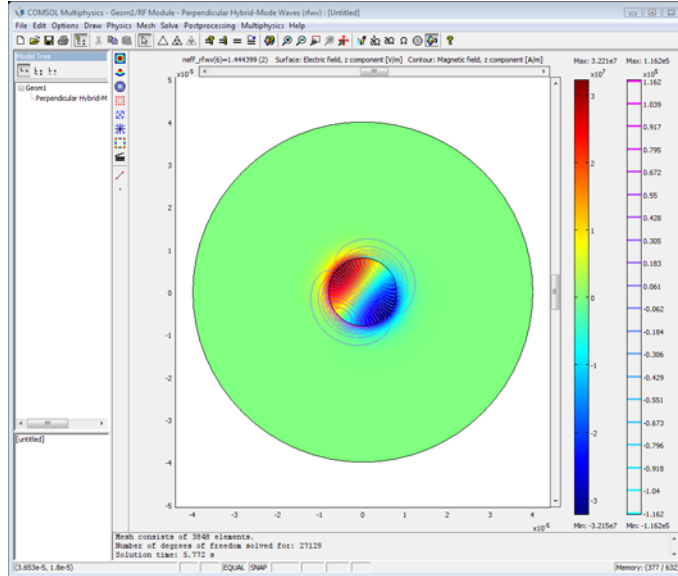
- 1 In the **Solver Parameters** dialog box set the parameter **Search for effective mode indices around** to 1.446. This guarantees that the solver will find the fundamental mode, which has the largest effective mode index.
- 2 Solve the problem.

## POSTPROCESSING AND VISUALIZATION

The default plot shows the power flow in the  $z$  direction for the fundamental mode. This is the  $\text{HE}_{11}$  mode, which is verified by visualizing the transversal components of both the magnetic and the electric field.

- 1 In the **Plot Parameters** dialog box, select surface and contour plots.
- 2 Set **Solution at angle (phase)** to 45.
- 3 On the **Surface** tab, set the **Surface data** to **Electric field, z component**.

#### 4 On the **Contour** tab, give **Magnetic field, z component** as **Contour data**.



### *Modeling Using the Programming Language*

The following command lines performs the calculation in COMSOL Script. The code is also available in the script file `step_index_fiber_cmd.m` in the model library path.

```
% The model can also be simulated by programming. First set up the
% structure with the geometry and the refractive indices as
% constants.
clear fem
fem.sdim={'x' 'y'};
fem.geom=circ2(0,0,8e-6)+circ2(0,0,40e-6);
fem.const={...
    'nSilicaGlassDoped',1.4378,...
    'nSilicaGlass',1.4457};

% Then define the application mode structure and generate the FEM
% structure from it.
clear appl
appl.mode='PerpendicularWaves';
appl.prop.inputvar='lambda';
appl.var.lambda0=1.55e-6;
appl.equ.n={'nSilicaGlassDoped' 'nSilicaGlass'};
fem.appl=appl;
fem=multiphysics(fem);
```

```
% Now generate the mesh, and the extended mesh structure.
fem.mesh=meshinit(fem);
fem.mesh=meshrefine(fem);
fem.xmesh=meshextend(fem);

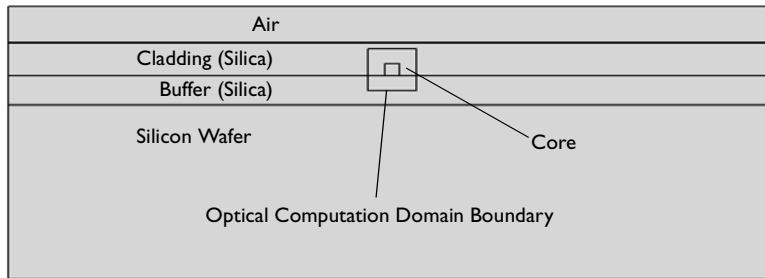
% Solve the problem and visualize the solution.
% Note that the 'shift' parameter to the solver
% must be expressed as the eigenvalue  $\lambda = -j\beta$ .
fem.sol=femeig(fem,'shift',-5.861604e6*i);
postplot(fem,'tridata','Ez','contdata','Hz','phase',45);
```

# Stress-Optical Effects in a Silica-on-Silicon Waveguide

## *Introduction*

---

Planar photonic waveguides in silica ( $\text{SiO}_2$ ) have great potential for use in wavelength routing applications. The major problem with these kinds of waveguides is birefringence. Anisotropic refractive indices result in fundamental mode splitting and pulse broadening. The goal is to minimize birefringence effects by adapting materials and manufacturing processes. One source of birefringence is the use of a silicon (Si) wafer on which the waveguide structure is deposited. After annealing at high temperature (approximately  $1000\text{ }^\circ\text{C}$ ), mismatch in thermal expansivity between the silica and silicon layers results in thermally induced stresses in the structure at the operating temperature (typically room temperature,  $20\text{ }^\circ\text{C}$ ).



## *The Stress-Optical Effect and Plane Strain*

---

The general linear stress-optical relation can be written, using tensor notation, as

$$\Delta n_{ij} = -B_{ijkl}\sigma_{kl}$$

where  $\Delta n_{ij} = n_{ij} - n_0 I_{ij}$ ,  $n_{ij}$  is the refractive index tensor,  $n_0$  is the refractive index for a stress-free material,  $I_{ij}$  is the identity tensor,  $B_{ijkl}$  is the stress-optical tensor, and  $\sigma_{kl}$  is the stress tensor. Due to symmetry the number of independent parameters in the stress-optical tensor that characterizes this constitutive relation can be reduced. Because  $n_{ij}$  and  $\sigma_{kl}$  are both symmetric,  $B_{ijkl} = B_{jikl}$  and  $B_{ijkl} = B_{ijlk}$ . In many cases it

is possible to further reduce the number of independent parameters, and this model includes only two independent parameters,  $B_1$  and  $B_2$ . The stress-optical relation simplifies to

$$\begin{bmatrix} \Delta n_x \\ \Delta n_y \\ \Delta n_z \end{bmatrix} = - \begin{bmatrix} B_2 & B_1 & B_1 \\ B_1 & B_2 & B_1 \\ B_1 & B_1 & B_2 \end{bmatrix} \begin{bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \end{bmatrix}$$

where  $n_x = n_{11}$ ,  $n_y = n_{22}$ ,  $n_z = n_{33}$ ,  $\sigma_x = \sigma_{11}$ ,  $\sigma_y = \sigma_{22}$ , and  $\sigma_z = \sigma_{33}$ .

This translates to

$$\begin{aligned} n_x &= n_0 - B_2 \sigma_x - B_1 (\sigma_y + \sigma_z) \\ n_y &= n_0 - B_2 \sigma_y - B_1 (\sigma_z + \sigma_x) \\ n_z &= n_0 - B_2 \sigma_z - B_1 (\sigma_x + \sigma_y) \end{aligned}$$

Using the two parameters  $B_1$  and  $B_2$ , the model assumes that the nondiagonal parts of  $n_{ij}$  and  $\sigma_{kl}$  are negligible. This means that the shear stress corresponding to  $\sigma_{12} = \tau_{xy}$  is neglected. In addition, the shear stresses corresponding to  $\sigma_{13} = \tau_{xz}$  and  $\sigma_{23} = \tau_{yz}$  are neglected by using the plane strain approximation. The plane strain approximation holds in a situation where the structure is free in the  $x$  and  $y$  direction but where the  $z$  strain is assumed to be zero. Note that this deformation state is not correct if the structure is free also in the  $z$  direction. In such a case a modified deformation state equation, which handles the the  $x$  and  $z$  directions equivalently, is needed.

The first part of this model utilizes the Plane Strain application mode of the Structural Mechanics Module. The resulting birefringent refractive index is computed using expression variables and can be considered a postprocessing step of the plane strain model. The refractive index tensor is used as material data for the second part of the model, the mode analysis.

The model “Stress-Optical Effects with Generalized Plane Strain” on page 232 demonstrates a computation where the structure is free also in the  $z$  direction, using a formulation called generalized plane strain.

### *Perpendicular Hybrid-Mode Waves*

---

For a given frequency  $\nu$ , or equivalently, free-space wavelength  $\lambda_0 = c_0/\nu$ , the RF Module’s Perpendicular Hybrid-Mode Waves application mode can be used for the



mode analysis. In this model the free-space wavelength is 1.55  $\mu\text{m}$ . The simulation is set up with the normalized magnetic field components  $\mathbf{H}=(H_x, H_y, H_z)$  as dependent variables, and the effective mode index  $n_{\text{eff}} = \beta/k_0$  is obtained from the eigenvalues.

Using this application mode, the wave is assumed to have the form

$$\mathbf{H} = \mathbf{H}(x, y)e^{j(\omega t - \beta z)} = (H_x(x, y), H_y(x, y), H_z(x, y))e^{j(\omega t - \beta z)}$$

The computations show a shift in effective mode index due to the stress-induced change in refractive index. The birefringence causes the otherwise two-fold degenerate fundamental mode to split.

---

**Model Library path:** RF\_Module/  
Optics\_and\_PhotonicsStructural\_Mechanics\_Module/  
Stress-Optical\_Effects/stress\_optical

---

---

**Note:** This model requires the RF Module and the Structural Mechanics Module.

---

### *Plane Strain Analysis*

---

- 1 In the **Model Navigator**, select **2D**.
- 2 Select the **Structural Mechanics Module>Plane Strain>Static analysis** application mode.
- 3 Click **Multiphysics** then **Add** to add this application mode to the model.
- 4 Next select the **RF Module>Perpendicular Waves>Hybrid-Mode Waves>Mode analysis** application mode. Click **Add** to add this application mode to the model; then click **OK**.
- 5 Choose **Physics>Properties** to open the **Application Mode Properties** dialog box and set **Field components** to **In-plane components**. This selects the two-components equation formulation for hybrid-mode waves. The default three-component equation formulation can also be used for this problem. See “Selecting Equation Formulation” on page 146 in the *RF Module User’s Guide* for general information about the different options.

- 6 For convenience set the property **Specify wave using** to **Free space wavelength**. This makes the wavelength available in the **Application Scalar Variables** dialog box instead of the frequency. Click **OK** to close the **Application Mode Properties** dialog box.
- 7 Choose **Multiphysics>Model Navigator**. Set the **Ruling application mode** to the **Perpendicular Hybrid-Mode Waves** application mode. This makes this mode specify the interpretation of the parameters to the eigenvalue solver given in the **Solver Parameters** dialog box. Click **OK**.

### OPTIONS AND SETTINGS

- 1 Open the **Constants** dialog box from the **Options** menu and enter the following names, expressions, and descriptions (optional); when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
nSi	3.5	Refractive index, silicon (Si)
nBuf	1.445	Refractive index, silica (SiO <sub>2</sub> )
nClad	1.445	Refractive index, silica
deltan	0.0075	Relative index difference: $\Delta = \frac{(n_{\text{core}}^2 - n_{\text{cladding}}^2)}{2n_{\text{core}}^2}$
nCore	nClad/sqrt(1-2*deltan)	Refractive index, core
alphaSi	2.5e-6[1/K]	Thermal expansion coefficient, silicon
alphaSi02	0.35e-6[1/K]	Thermal expansion coefficient, silica
ESi	110[GPa]	Young's modulus, silicon
ESi02	78[GPa]	Young's modulus, silica
nuSi	0.19	Poisson's ratio, silicon
nuSi02	0.42	Poisson's ratio, silica
B1	4.2e-12[m^2/N]	First stress optical coefficient
B2	0.65e-12[m^2/N]	Second stress optical coefficient
T1	20[degC]	Operating temperature
T0	1000[degC]	Reference temperature

Notice that the temperatures are given in degrees Celsius. This works well because this is a linear model. (Nonlinear models need to have temperatures in kelvin to get the correct thermodynamics.)

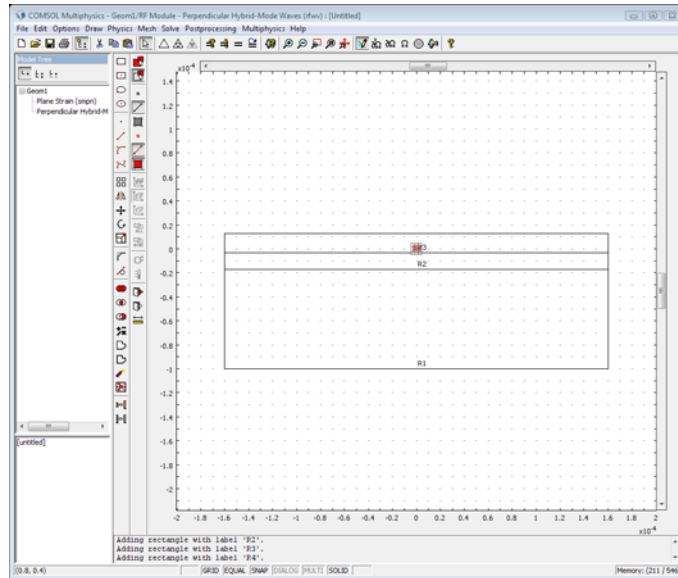
2 Give axis and grid settings according to the following table.

AXIS		GRID	
x min	-0.2e-3	x spacing	1e-5
x max	0.2e-3	Extra x	
y min	-0.1e-3	y spacing	1e-5
y max	0.03e-3	Extra y	

### GEOMETRY MODELING

Draw rectangles corresponding to the different regions.

REGION	LOWER LEFT CORNER	UPPER RIGHT CORNER
Silicon Wafer	(-16e-5, -1e-4)	(16e-5, -1.7e-5)
Buffer	(-16e-5, -1.7e-5)	(16e-5, -3e-6)
Cladding	(-16e-5, -3e-6)	(16e-5, 1.3e-5)
Core	(-3e-6, -3e-6)	(3e-6, 3e-6)



## PHYSICS SETTINGS

### *Point Settings and Boundary Conditions*

All regions have free boundaries, which also is the default boundary condition. However, these conditions will not suffice in creating a unique solution because the computational domain is allowed to move and rotate freely; the problem is ill-posed. The problem becomes well-posed by adding constraints at points to keep the domain fixed.

- 1 Select the **Plane Strain** application mode from the **Multiphysics** menu.
- 2 Open the **Point Settings** dialog box, and check the constraints  $R_x$  and  $R_y$  at Point 1, the lower left corner, for both the  $x$  and  $y$  direction. This will keep the domain from moving (translational move).
- 3 Check the constraint  $R_y$  at Point 9, the lower right corner, for the  $y$  direction only, keeping the domain from rotating but allowing it to slide in the  $x$  direction.

### *Subdomain Settings*

The air domain need not be part of the plane strain model.

- 1 Specify the subdomain settings for the Plane Strain application mode according to the following table:

	SUBDOMAIN 1				SUBDOMAINS 2-4			
Page	Material		Load		Material		Load	
	$E$	ESi			$E$	ESi02		
	$\nu$	nuSi	Temp	T1	$\nu$	nuSi02	Temp	T1
	$\alpha$	alphaSi	Tempref	T0	$\alpha$	alphaSi02	Tempref	T0

Before entering the temperatures on the **Load** page, you need to check **Include thermal expansion**. It is not necessary to specify the density  $\rho$ , because it does not enter the equation for static problems.

### EXPRESSION VARIABLES

In the **Subdomain Expressions** dialog box in the **Options** menu, define the following variables; when done, click **OK**.

VARIABLE NAME	SUBDOMAIN	EXPRESSION
N	1	nSi
	2	nBuf
	3	nClad
	4	nCore
Nx	1	N
	2,3,4	$N - B1 * sx\_smpn - B2 * (sy\_smpn + sz\_smpn)$
Ny	1	N
	2,3,4	$N - B1 * sy\_smpn - B2 * (sx\_smpn + sz\_smpn)$
Nz	1	N
	2,3,4	$N - B1 * sz\_smpn - B2 * (sx\_smpn + sy\_smpn)$

### MESH GENERATION

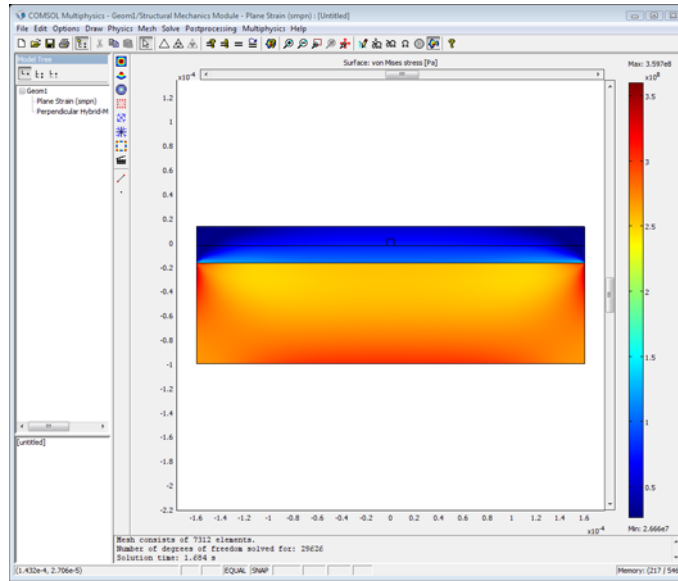
Initialize a mesh and refine it twice.

### COMPUTING THE SOLUTION

- 1 In the **Solver Parameters** dialog box select the **Stationary** solver.
- 2 On the **Solve for** tab in the **Solver Manager** dialog box, select only the **Plane Strain** application mode. This will make sure that only the Plane Strain application mode equations are solved in the first run.
- 3 Click the **Solve** button.

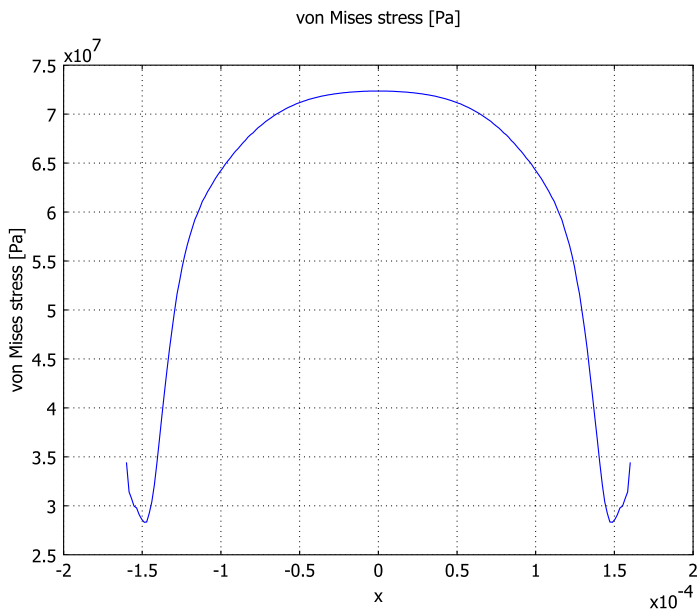
## POSTPROCESSING AND VISUALIZATION

The default plot shows the von Mises effective stress as a colored surface plot.



- I To view the von Mises stress along a horizontal line through the entire structure, use a cross-section plot. In the **Cross-Section Plot Parameters** dialog box, select **von**

**Mises stress** as **y-axis data** on the **Line/Extrusion** tab. As **Cross-section line data** set **x0** to  $-16e-5$ , **y0** to 0, **x1** to  $16e-5$ , and **y1** to 0. Use **x** as **x-axis data**.

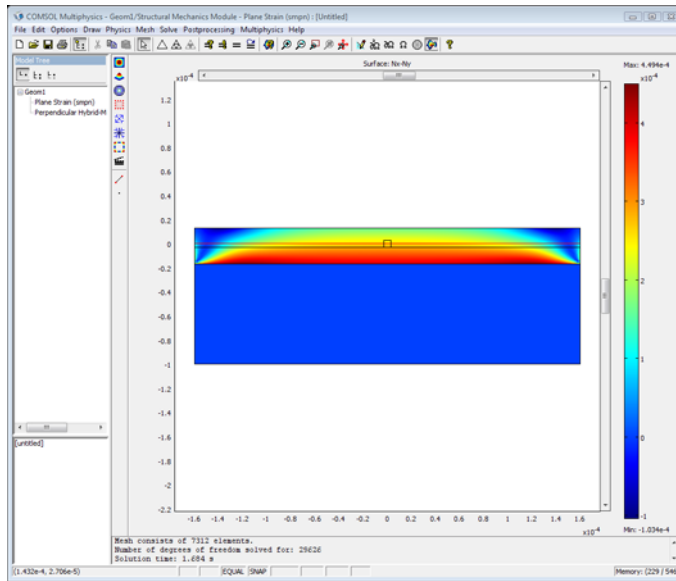


Notice that the effective stress is varying slowly on the horizontal cut. This means that the significant influence on the stress induced changes in refractive index will come from the stress variations in the vertical direction. This is expected since the extension of the domains in the  $x$  direction is chosen to minimize effects of the edges.

Next plot the birefringence:

- 2 In the **Plot Parameters** dialog box, click the **Surface** tab.

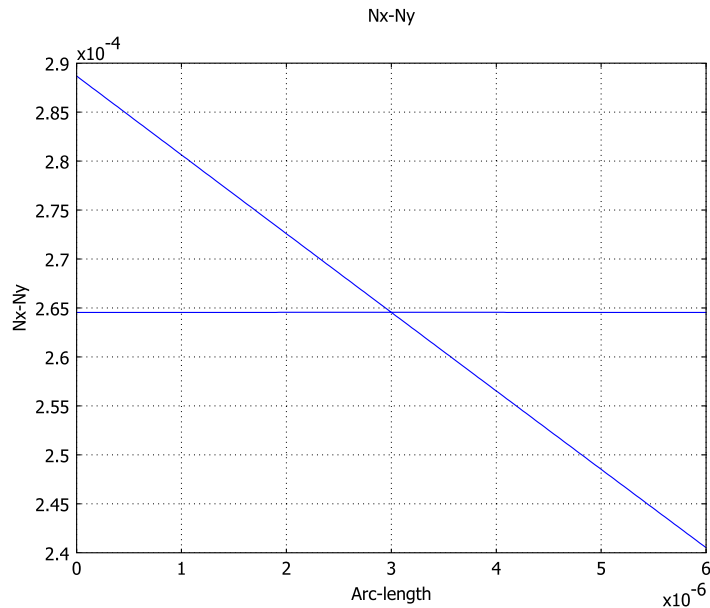
- To view the stress-induced birefringence  $n_x - n_y$ , type  $N_x - N_y$  in the **Expression** field on the **Surface Data** page.



- Now create a cross-section plot of the birefringence  $n_x - n_y$  on the vertical line from  $(0, -3 \cdot 10^{-6})$  to  $(0, 3 \cdot 10^{-6})$ . Do this by entering  $N_x - N_y$  as **y-axis data**, and setting **x0** to 0, **y0** to  $-3e-6$ , **x1** to 0, and **y1** to  $3e-6$  in the **Cross-Section Plot Parameters** dialog box. Set the **x-axis data** to **Arc-length**. The birefringence varies linearly from about  $2.9 \cdot 10^{-4}$  at the bottom of the core to  $2.4 \cdot 10^{-4}$  at the top. Do not close the figure.
- Now create another cross-section plot of the birefringence on the horizontal line from  $(-3 \cdot 10^{-6}, 0)$  to  $(3 \cdot 10^{-6}, 0)$ . On the **General** tab select the **Keep current plot**



check box to make the plot in the same window as the previous one. On the **Line/Extrusion** page, set **x0** to  $-3e-6$ , **y0** to 0, **x1** to  $3e-6$ , and **y1** to 0.



The birefringence is constant on the horizontal line, thus the influence of the edges is indeed reduced to a minimum.

### *Optical Mode Analysis*

Select the **Perpendicular Hybrid-Mode Waves** application mode from the **Multiphysics** menu.

#### **GEOMETRY MODELING**

The computational domain can be reduced significantly for the optical mode analysis, because the energy of the fundamental modes is concentrated in the core region, and the energy density decays rapidly in the cladding and buffer regions.

Draw a rectangle with corners at  $(-1e-5, -1e-5)$  and  $(1e-5, 1e-5)$ .

The rectangular region containing the active domains can be enlarged later on for validating the results. The rectangular region should be chosen large enough so that the computed propagation constants do not change significantly if the region is enlarged.

## PHYSICS SETTINGS

### *Scalar Variables*

- 1 From the **Physics** menu, choose **Scalar Variables**.
- 2 In the **Application Scalar Variables** dialog box, set the free-space wavelength to  $1.55\text{e}-6$ ; when done, click **OK**.

### *Subdomain Settings*

- 1 From the **Physics** menu, choose **Subdomain Settings**.
- 2 Deactivate the Perpendicular Hybrid-Mode Waves application mode in all subdomains except 4, 5, and 6. This makes sure the problem is only solved in the newly drawn rectangle.
- 3 For all active domains, select **n (anisotropic)** and set  $N_x$ ,  $N_y$ , and  $N_z$ , respectively, as anisotropic refractive indices by changing the entries on the diagonal in the refractive index matrix.
- 4 When done, click **OK**.

### *Boundary Conditions*

The only available perfect magnetic conductor boundary condition will suffice. Both the **E** and the **H** fields are assumed to be of negligible size at the boundaries, hence any one of them can be set equal to zero at the boundary.

## MESH GENERATION

- 1 On the **Subdomain** tab in the **Free Mesh Parameters** dialog box, set the **Maximum element size** to  $2\text{e}-6$  for Subdomains 4 and 5, and to  $1\text{e}-6$  for Subdomain 6. This will make the mesh dense in the activated regions and coarse in the deactivated regions. The mesh in the deactivated regions only affects the interpolated values of the expression variables and not the mode computation results.
- 2 Initialize the mesh.

## COMPUTING THE SOLUTION

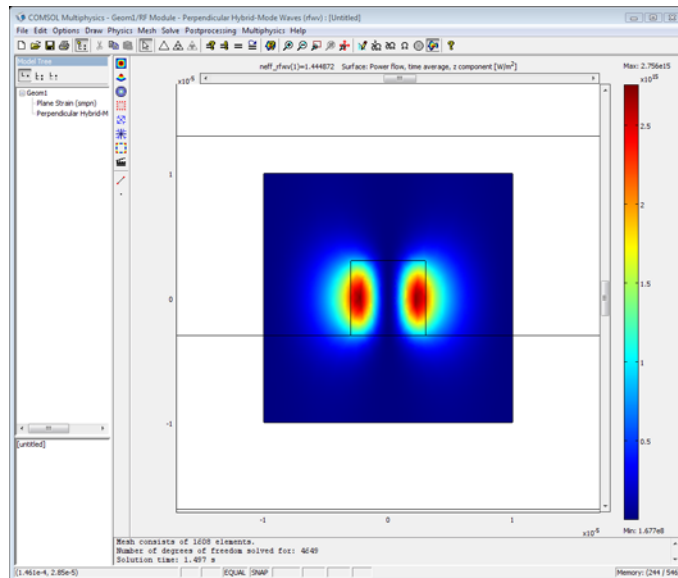
- 1 In the **Solver Parameters** dialog box, select the **Eigenvalue** solver.
- 2 Enter  $1.46$  in the **Search for effective mode indices around** edit field. The default **Desired number of effective mode indices** is set to  $6$ , which is fine. These settings will make the eigenmode solver search for the  $6$  eigenmodes with effective mode indices closest to the value  $1.46$ . This value is an estimate of the effective mode index for the fundamental mode. For propagating modes it must hold that

$$n_{\text{eff}} < n_{\text{core}} = 1.456$$

- 3 In the **Solver Manager** dialog box, select only the **Perpendicular Hybrid-Mode Waves** application mode on the **Solve For** tab. This ensures that the Plane Strain application mode is not part of the eigenmode computation.
- 4 On the **Initial Value** tab, set the initial value to **Current solution**. This will take the plane strain solution and use it to evaluate the refractive indices for the mode analysis.
- 5 Click the **Solve** button.

## POSTPROCESSING AND VISUALIZATION

- 1 On the **Surface** tab in the **Plot Parameters** dialog box, select **Perpendicular Hybrid-Mode Waves (rfwv)>Power flow, time average, z component** from the **Predefined quantities** list on the **Surface Data** tab. Then click **Apply**. This creates a visualization of the power flow, also called optical intensity or the Poynting vector, in the  $z$  direction (out-of-plane direction).
- 2 The default eigenmode is the one with the highest effective mode index corresponding to one of the fundamental modes.



## Convergence Analysis

To check the accuracy of the eigenvalues you need to examine the sensitivity of these values to changes in the mesh density.

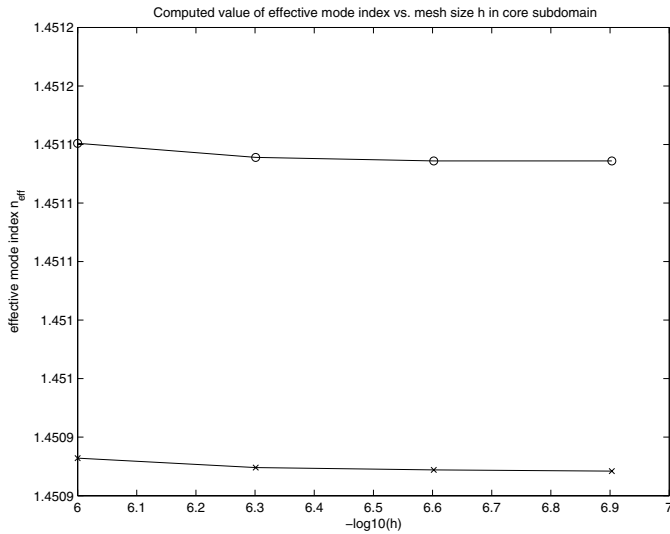
- 1 In the **Free Mesh Parameters** dialog box, lower the value of the maximum element size for the core subdomain, number 6, to  $5e-7$ , remesh, and solve again.
- 2 Repeat this for the maximum element sizes  $2.5e-7$  and  $1.25e-7$ .

The solver finds the following effective mode:

ELEMENT SIZE, CORE SUBDOMAIN	1E-6	5E-7	2.5E-7	1.25E-7
<b>EFFECTIVE MODE INDEX</b>				
$n_{\text{eff}1}$	1.451149	1.451139	1.451136	1.451136
$n_{\text{eff}2}$	1.450883	1.450875	1.450872	1.450871
$n_{\text{eff}3}$	1.445409	1.445407	1.445405	1.445405
$n_{\text{eff}4}$	1.445144	1.445141	1.445139	1.445139
$n_{\text{eff}5}$	1.445139	1.445133	1.445132	1.445131
$n_{\text{eff}6}$	1.444873	1.444867	1.444865	1.444865

It is evident from the table that the fundamental modes, corresponding to the two lowest eigenmodes with the highest effective mode indices, are nearly degenerate. It is also clear that the split of the degenerate fundamental modes that is expected due to the stress-induced anisotropic refractive index has been properly resolved with these

mesh sizes. The convergence of the two fundamental modes is shown in the figure below.



Now, run the model without the stress-induced birefringence.

- 1 Change the refractive index of Subdomains 4, 5, and 6 to an isotropic index by selecting the **n (isotropic)** radio button in the **Subdomain Settings** dialog box, and typing N as refractive index.
- 2 Solve the problem for a maximum element size of  $2.5e-7$  for Subdomain 6.

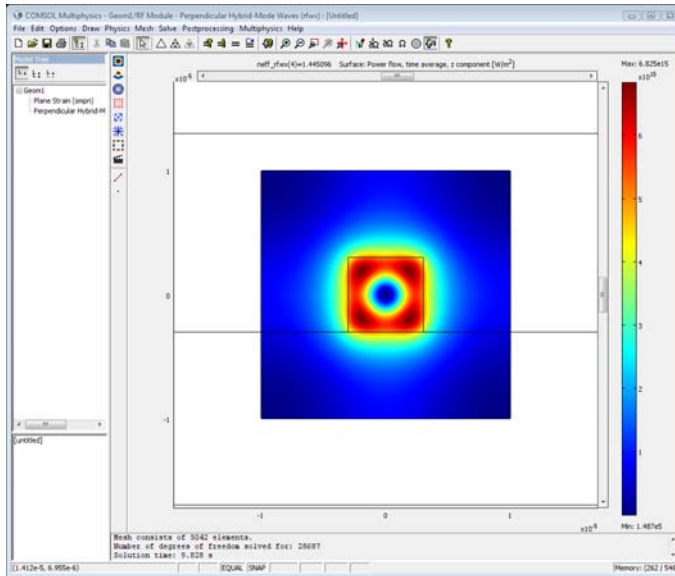
Compare these ideal values of the propagation constants with the stress-optical case.

ELEMENT SIZE, CORE SUBDOMAIN:	2.5E-7 (STRESS)	2.5E-7 (NO STRESS)	DIFFERENCE
EFFECTIVE MODE INDEX			*1e-4
$n_{eff1}$	1.451136	1.450821	3.15
$n_{eff2}$	1.450872	1.45082	.51
$n_{eff3}$	1.445405	1.445093	57.79
$n_{eff4}$	1.445139	1.445083	.56
$n_{eff5}$	1.445132	1.444832	3.00
$n_{eff6}$	1.444865	1.444799	.66

The difference is significant, which shows that the shift in the effective mode indices due to the stress-optical effect is indeed resolved.

Visual inspection of the 4 higher eigenmodes indicates that these are probably degenerate in the ideal case of no thermally induced stresses. Moreover, the higher eigenmodes have a larger portion of energy leaking into the cladding and buffer, and are thus more affected than the fundamental modes of the distance to the air and silicon layers. Because of this leakage, the boundary condition will affect the higher eigenmodes more than the fundamental mode. Thus a refined analysis of the higher modes would make it necessary to enlarge the computation domain.

The figure below shows the  $z$  component of the Poynting vector for the fourth eigenmode.



## MODELING ERRORS AND SENSITIVITY ANALYSIS

Although the fundamental modes have converged to 5 decimal places, the known modeling errors makes the exactness of the numbers uncertain. One major modeling error is due to the fact that the model contains a plane strain assumption in a case where the real-world model does not necessarily conform to this deformation state. This modeling error is reduced in the refined model “Stress-Optical Effects with Generalized Plane Strain” on page 232. Moreover, the material properties are only known to a few decimal places and the computed magnitudes of the effective mode

indices will correspondingly be uncertain. A standard way of examining the effects of uncertainty in material parameters is to perform a sensitivity analysis. That is, you perturb one of the material parameters slightly and then examine the resulting perturbation in the computed parameter. Another source of uncertainty is whether the stress from the thermal expansion is large with respect to the other sources of stress in the material originating from the manufacturing process.

# Stress-Optical Effects with Generalized Plane Strain

## *Introduction*

---

The assumptions made for plane strain in the previous analysis of the waveguide structure, in the model “Stress-Optical Effects in a Silica-on-Silicon Waveguide” on page 215, do not hold in a situation where the silicon-silica laminate is free to expand in the  $z$  direction. Instead it is necessary to use a *generalized plane strain* model that allows for free expansion in the  $z$  direction. The boundary conditions in the  $xy$ -plane already allow the structure to expand freely in all directions in the plane. When the different materials in a laminate expand with different expansion coefficients, the laminate bends. In this model, the silica-silicon laminate bends in both the  $x$  and  $z$  directions. The Plane Strain application mode does not cover the bending in the  $z$  direction, so you must make modifications to the plane strain equations at the equation-system level.

---

**Note:** This model requires the RF Module and the Structural Mechanics Module.

---

## *Generalized Plane Strain*

---

One possible extension of the plane strain formulation is to assume that the strain in the  $z$  direction has the form

$$\varepsilon_z = e_0 + e_1x + e_2y$$

That is, the strain is linearly varying throughout the cross section. This approximation is expected to be good when the bending curvature is small with respect to the extents of the structure in the  $xy$ -plane and corresponds to a small rotation that is representative of each cross section of the structure along the  $z$  axis. (A more general model would include second-degree terms in  $x$  and  $y$ .)

One way of validating the result of this simulation is to plot the  $\varepsilon_x$  and  $\varepsilon_z$  strains on a cross section on the symmetry line at  $x = 0$ . Because you can expect an identical bending in the  $x$  and  $z$  directions, these strains should be identical and vary linearly.



## EXTENSION OF THE PLANE STRAIN EQUATIONS

The objective is to extend the current Plane Strain application mode with additional PDE terms that represent the generalized plane strain deformation state. In COMSOL Multiphysics, the coefficients  $e_0$ ,  $e_1$ , and  $e_2$  in the expression for the  $\varepsilon_z$  strain is modeled with coupling variables associated with a point geometry object (zero dimensional object). This is a convenient way to include degrees of freedom that are constant throughout a subdomain. In order to expand the equations for the Plane Strain application mode, the new  $\varepsilon_z$  strain contributions need to be added as weak terms to the ordinary plane strain equations. The reason for this is that for COMSOL Multiphysics to be able to solve the equations as a linear problem, the assembler requires that all contributions from coupling variables are entered as weak terms.

Start from the 3D stress-strain relation for linear isotropic conditions including thermal effects,

$$\begin{bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{xz} \end{bmatrix} = D \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{xz} \end{bmatrix} - \alpha \begin{bmatrix} T - T_{\text{ref}} \\ T - T_{\text{ref}} \\ T - T_{\text{ref}} \\ 0 \\ 0 \\ 0 \end{bmatrix}$$

where  $\sigma_x$ ,  $\sigma_y$ ,  $\sigma_z$ ,  $\tau_{xy}$ ,  $\tau_{yz}$ , and  $\tau_{xz}$  are the components of the stress tensor,  $\varepsilon_x$ ,  $\varepsilon_y$ ,  $\varepsilon_z$ ,  $\gamma_{xy}$ ,  $\gamma_{yz}$ , and  $\gamma_{xz}$  are the components of the strain tensor,  $\alpha$  is the thermal expansion coefficient,  $T$  is the operating temperature, and  $T_{\text{ref}}$  is the reference temperature corresponding to the manufacturing temperature. The 6-by-6 matrix  $D$  (tensor) has entries that contain expressions in Young's modulus,  $E$ , and Poisson's ratio,  $\nu$ . The matrix  $D$  is presented in the *Structural Mechanics Module User's Guide*.

For generalized plane strain (and ordinary plane strain), assume that  $\gamma_{yz} = \gamma_{xz} = 0$  and two of the equations vanish

$$\begin{bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \end{bmatrix} = D \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \end{bmatrix} - \alpha \begin{bmatrix} T - T_{\text{ref}} \\ T - T_{\text{ref}} \\ T - T_{\text{ref}} \\ 0 \end{bmatrix}$$

The matrix  $D$  shrinks accordingly.

The PDE solved is Navier's equation

$$-\nabla \cdot \boldsymbol{\sigma} = \mathbf{K}$$

where

$$\boldsymbol{\sigma} = \begin{bmatrix} \sigma_x & \tau_{xy} & 0 \\ \tau_{yx} & \sigma_y & 0 \\ 0 & 0 & \sigma_z \end{bmatrix} = c \nabla \mathbf{u} - \boldsymbol{\gamma}$$

in the case of generalized plane strain and  $\mathbf{u} = (u, v, w)$  are the unknown displacement variables solved for.

The coefficient  $c$  above is a tensor containing the elements of  $D$  in a pattern that is listed in the *Structural Mechanics Module User's Guide* and

$$\boldsymbol{\gamma} = \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} (1+\nu)\alpha(T-T_{\text{ref}}) & 0 & 0 \\ 0 & (1+\nu)\alpha(T-T_{\text{ref}}) & 0 \\ 0 & 0 & (1+\nu)\alpha(T-T_{\text{ref}}) \end{bmatrix}$$

contains the contributions from the thermal strains.

In the case of ordinary plane strain the upper left 2-by-2 submatrix of  $\boldsymbol{\sigma}$ ,  $\boldsymbol{\gamma}$ , and  $D$  is used.

It is assumed here that the variables  $T$  and  $T_{\text{ref}}$  are constants. For information on how the temperature variables enter Navier's equation as unknowns in a heat transfer problem, see the *Structural Mechanics Module User's Guide*.

To expand the plane strain equation to a generalized plane strain formulation one needs to consider the weak form of Navier's equation. The weak form of a system of equations is a scalar equation involving integrals. See "Weak Form Modeling" on page 293 in the *COMSOL Multiphysics Modeling Guide* for more information on the appearance of the weak form for systems of equations.

To get the weak form of Navier's equation, multiply by a test function  $\mathbf{v} = [u_{\text{test}}, v_{\text{test}}, w_{\text{test}}]$  and integrate

$$\int_{\Omega} \mathbf{v}(-\nabla \cdot \boldsymbol{\sigma}) d\Omega = \int_{\Omega} \mathbf{v} \cdot \mathbf{K} d\Omega$$

Integration by parts gives

$$0 = \int_{\partial\Omega} \mathbf{v} \mathbf{n} d\Gamma - \int_{\Omega} \nabla \mathbf{v} \cdot \boldsymbol{\sigma} d\Omega + \int_{\Omega} \mathbf{v} \cdot \mathbf{K} d\Omega$$

or in more detail, with  $\mathbf{K} = \mathbf{0}$

$$0 = \int_{\partial\Omega} \begin{bmatrix} u_{\text{test}} & v_{\text{test}} & w_{\text{test}} \end{bmatrix} \begin{bmatrix} \sigma_x & \tau_{xy} & 0 \\ \tau_{yx} & \sigma_y & 0 \\ 0 & 0 & \sigma_z \end{bmatrix} \begin{bmatrix} n_x \\ n_y \\ n_z \end{bmatrix} d\Gamma$$

$$- \int_{\Omega} \begin{pmatrix} \begin{bmatrix} u_{x, \text{test}} & u_{y, \text{test}} & u_{z, \text{test}} \\ v_{x, \text{test}} & v_{y, \text{test}} & v_{z, \text{test}} \\ w_{x, \text{test}} & w_{y, \text{test}} & w_{z, \text{test}} \end{bmatrix} \cdot \begin{bmatrix} \sigma_x & \tau_{xy} & 0 \\ \tau_{yx} & \sigma_y & 0 \\ 0 & 0 & \sigma_z \end{bmatrix} \end{pmatrix} d\Omega$$

where  $u_{y, \text{test}} = \partial u_{\text{test}} / \partial y$  and similarly for the other test function derivatives. The dot product in the domain integral is interpreted as a dot product operating on the rows of the matrices so that

$$\int_{\Omega} \left( \begin{bmatrix} u_{x, \text{test}} & u_{y, \text{test}} & u_{z, \text{test}} \\ v_{x, \text{test}} & v_{y, \text{test}} & v_{z, \text{test}} \\ w_{x, \text{test}} & w_{y, \text{test}} & w_{z, \text{test}} \end{bmatrix} \cdot \begin{bmatrix} \sigma_x & \tau_{xy} & 0 \\ \tau_{yx} & \sigma_y & 0 \\ 0 & 0 & \sigma_z \end{bmatrix} \right) d\Omega$$

$$= \int_{\Omega} \left( \begin{bmatrix} u_{x, \text{test}} & u_{y, \text{test}} & u_{z, \text{test}} \end{bmatrix} \cdot \begin{bmatrix} \sigma_x & \tau_{xy} & 0 \end{bmatrix} + \right.$$

$$\left. \begin{bmatrix} v_{x, \text{test}} & v_{y, \text{test}} & v_{z, \text{test}} \end{bmatrix} \cdot \begin{bmatrix} \tau_{yx} & \sigma_y & 0 \end{bmatrix} + \begin{bmatrix} w_{x, \text{test}} & w_{y, \text{test}} & w_{z, \text{test}} \end{bmatrix} \cdot \begin{bmatrix} 0 & 0 & \sigma_z \end{bmatrix} \right) d\Omega$$

where

$$\begin{aligned}\sigma_x &= \frac{E}{(1+\nu)(1-2\nu)}((1-\nu)\varepsilon_x + \nu\varepsilon_y + \nu\varepsilon_z - (1+\nu)\alpha(T - T_{\text{ref}})) \\ \sigma_y &= \frac{E}{(1+\nu)(1-2\nu)}(\nu\varepsilon_x + (1-\nu)\varepsilon_y + \nu\varepsilon_z - (1+\nu)\alpha(T - T_{\text{ref}})) \\ \sigma_z &= \frac{E}{(1+\nu)(1-2\nu)}(\nu\varepsilon_x + \nu\varepsilon_y + (1-\nu)\varepsilon_z - (1+\nu)\alpha(T - T_{\text{ref}})) \\ \tau_{xy} &= \frac{E}{(1+\nu)(1-2\nu)}\left(\frac{1-2\nu}{2}\gamma_{xy}\right)\end{aligned}$$

and

$$\begin{aligned}\varepsilon_x &= \frac{\partial u}{\partial x} \\ \varepsilon_y &= \frac{\partial v}{\partial y} \\ \varepsilon_z &= \frac{\partial w}{\partial z} \\ \gamma_{xy} &= \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\end{aligned}$$

Note that the displacement variable  $w$  in the  $z$  direction is not available in the Plane Strain application mode. However, it is not needed since the generalized formulation assumes a certain fix shape of the deformation in the  $z$  direction.

The same relations hold for the test functions,

$$\begin{aligned}\varepsilon_{x, \text{test}} &= u_{x, \text{test}} \\ \varepsilon_{y, \text{test}} &= v_{y, \text{test}} \\ \varepsilon_{z, \text{test}} &= w_{z, \text{test}} \\ \gamma_{xy, \text{test}} &= u_{y, \text{test}} + v_{x, \text{test}}\end{aligned}$$

In COMSOL Multiphysics, the test functions  $u_{\text{test}}$  and  $v_{\text{test}}$  are denoted `u_test` and `v_test`, respectively. Their derivatives  $u_{x,\text{test}}$ ,  $u_{y,\text{test}}$ ,  $v_{x,\text{test}}$ , and  $v_{y,\text{test}}$  are denoted `ux_test`, `uy_test`, `vx_test`, and `vy_test`, respectively. Note that the  $z$  components of the test functions are not available in the Plane Strain application mode.

Using this notation, the weak form of Navier's equation becomes (with  $\mathbf{K} = \mathbf{0}$ )

$$0 = \int_{\partial\Omega} \mathbf{v} \mathbf{n} d\Gamma - \int_{\Omega} (\varepsilon_{x, \text{test}} \sigma_x + \varepsilon_{y, \text{test}} \sigma_y + \varepsilon_{z, \text{test}} \sigma_z + \gamma_{xy, \text{test}} \tau_{xy}) d\Omega$$

This expression for the weak equation shows which terms you must add as weak terms to generalize the plane strain formulation.

To see the difference compared to the ordinary plane strain formulation, the equation for  $\sigma_z$  can be separated out according to

$$\begin{bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{bmatrix} = D \left( \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{bmatrix} - \begin{bmatrix} (1+\nu)\alpha(T-T_{\text{ref}}) \\ (1+\nu)\alpha(T-T_{\text{ref}}) \\ 0 \end{bmatrix} + \begin{bmatrix} \nu\varepsilon_z \\ \nu\varepsilon_z \\ 0 \end{bmatrix} \right)$$

$$\sigma_z = \frac{E}{(1+\nu)(1-2\nu)} (\nu\varepsilon_x + \nu\varepsilon_y + (1-\nu)\varepsilon_z - (1+\nu)\alpha(T-T_{\text{ref}}))$$

where  $D$  has been reduced to

$$D = \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1-\nu & \nu & 0 \\ \nu & 1-\nu & 0 \\ 0 & 0 & \frac{1-2\nu}{2} \end{bmatrix}$$

The ordinary plane strain equation is obtained if  $\varepsilon_z = 0$ .

To expand the Plane Strain application mode, you must add weak contributions from  $\varepsilon_z$  need to the stress components  $\sigma_x$  and  $\sigma_y$  according to

$$-\frac{E}{(1+\nu)(1-2\nu)} (\varepsilon_{x, \text{test}} \nu \varepsilon_z + \varepsilon_{y, \text{test}} \nu \varepsilon_z)$$

In addition,  $\sigma_z$  needs to be included in the weak equation from being merely a postprocessing quantity in the original Plane Strain application mode,

$$-\varepsilon_{z, \text{test}} \sigma_z$$

where

$$\varepsilon_{z, \text{test}} = e_{0, \text{test}} + e_{1, \text{test}} x + e_{2, \text{test}} y$$

and  $\sigma_z$  was presented earlier.

In COMSOL Multiphysics you model the coefficients  $e_0$ ,  $e_1$ , and  $e_2$  in the expression for the  $\varepsilon_z$  strain with coupling variables as weak contributions to the ordinary plane strain equations. These coefficients represent three additional unknown variables that prevail throughout the modeled domain. In practice, these variables need to be

associated with an arbitrary point of the geometry. Using coupling variables in this way specifies a map from this point onto the entire domain on which the generalized plane strain equations hold. COMSOL Multiphysics creates the test functions corresponding to  $e_0$ ,  $e_1$ , and  $e_2$  automatically.

---

**Model Library path:** RF\_Module/  
Optics\_and\_PhotonicsStructural\_Mechanics\_Module/  
Stress-Optical\_Effects/stress\_optical\_generalized

---

### *Plane Strain Analysis*

---

Many of the modeling steps are similar to the model “Stress-Optical Effects in a Silica-on-Silicon Waveguide” on page 215, but some steps are made in a different order, and there are obviously differences when generalizing the plane strain problem.

#### **MULTIPHYSICS SETTINGS**

- 1 In the **Model Navigator**, select **2D**.
- 2 Select the **Structural Mechanics Module>Plane Strain>Static analysis** application mode.
- 3 Click **Multiphysics** then **Add** to add this application mode to the model.
- 4 Next select the **RF Module>Perpendicular Waves>Hybrid-Mode Waves> Mode analysis** application mode. Add this mode to the model.
- 5 Open the **Application Mode Properties** dialog box from the **Physics** menu (**Properties**) and select **In-plane components** from the **Field components** list. This selects the two-components equation formulation for hybrid-mode waves. It is also possible to use the default three-component equation formulation for this problem. See “Selecting Equation Formulation” on page 146 in the *RF Module User’s Guide* for general information about the different options.
- 6 For convenience select **Free space wavelength** from the **Specify wave using** list. This makes the wavelength available in the **Application Scalar Variables** dialog box instead of the frequency.

The unknown parameters  $e_0$ ,  $e_1$ , and  $e_2$  should be constant within the whole modeling domain. You can achieve this by adding a new geometry with only a single point to the model and define  $e_0$ ,  $e_1$ , and  $e_2$  as the dependent variables at this point. Later on you define coupling variables that map these dependent variables from the point to the whole model.

- 1 From the **Multiphysics** menu, select the **Model Navigator**. Click the **Add Geometry** button to add a 2D geometry (2D is the default space dimension).
- 2 Select the **COMSOL Multiphysics>PDE Modes>Weak Form,Point** application mode from the application mode tree.
- 3 Type  $e_0$   $e_1$   $e_2$  in the **Dependent variables** text field.
- 4 Click **Add** to add the Weak Form, Point application mode to the second geometry. This application mode contains the three degrees of freedom for the parameters  $e_0$ ,  $e_1$ , and  $e_2$  that describe the  $z$  strain.
- 5 Set the **Ruling application mode** to the **Perpendicular Hybrid-Mode Waves** application mode. This makes this application mode specify the interpretation of the parameters to the eigenvalue solver given in the **Solver Parameters** dialog box.
- 6 Click **OK** to close the **Model Navigator**.

The Weak Form, Point application mode becomes the active application mode when you close the Model Navigator.

#### GEOMETRY MODELING

The three degrees of freedom  $e_0$ ,  $e_1$ , and  $e_2$  must be associated with a point. The location of this point is insignificant. This means that the added geometry could equally well have been a 1D or a 3D geometry. Using the programming language, there is also an option of using a 0D geometry.

Add a point at any position, say at the origin (0, 0).

#### OPTIONS AND SETTINGS

In the **Constants** dialog box enter the following names, expressions, and descriptions (optional); when done, click **OK**. They are identical to the constants defined on page 218.

NAME	EXPRESSION	DESCRIPTION
nSi	3.5	Refractive index, silicon (Si)
nBuf	1.445	Refractive index, silica (SiO <sub>2</sub> )
nClad	1.445	Refractive index, silica
deltan	0.0075	Relative index difference: $\Delta = \frac{(n_{\text{core}}^2 - n_{\text{cladding}}^2)}{2n_{\text{core}}^2}$
nCore	nClad/sqrt(1-2*deltan)	Refractive index, core

NAME	EXPRESSION	DESCRIPTION
alphaSi	2.5e-6[1/K]	Thermal expansion coefficient, silicon
alphaSiO2	0.35e-6[1/K]	Thermal expansion coefficient, silica
ESi	110[GPa]	Young's modulus, silicon
ESiO2	78[GPa]	Young's modulus, silica
nuSi	0.19	Poisson's ratio, silicon
nuSiO2	0.42	Poisson's ratio, silica
B1	4.2e-12[m^2/N]	First stress optical coefficient
B2	0.65e-12[m^2/N]	Second stress optical coefficient
T1	20[degC]	Operating temperature
T0	1000[degC]	Reference temperature

### GEOMETRY MODELING

- 1 Switch to **Geom1** by clicking its tab.
- 2 Draw rectangles corresponding to the different regions.

REGION	LOWER LEFT CORNER	UPPER RIGHT CORNER
Silicon Wafer	(-16e-5, -1e-4)	(16e-5, -1.7e-5)
Buffer	(-16e-5, -1.7e-5)	(16e-5, -3e-6)
Cladding	(-16e-5, -3e-6)	(16e-5, 1.3e-5)
Core	(-3e-6, -3e-6)	(3e-6, 3e-6)

### PHYSICS SETTINGS

#### *Point Settings and Boundary Conditions*

All regions have free boundaries, which also is the default boundary condition. However, these conditions do not suffice in creating a unique solution because the computational domain can move and rotate freely; the problem is ill-posed. The problem becomes well-posed by adding constraints at points to keep the domain fixed.

- 1 Select the **Plane Strain** application mode from the **Multiphysics** menu.
- 2 Open the **Point Settings** dialog box, and select the check boxes for the constraints  $R_x$  and  $R_y$  at point 1, the lower left corner, for both the  $x$  and  $y$  directions. Doing so keeps the domain from moving (translational move).
- 3 Select the check box for the constraint  $R_y$  at point 9, the lower-right corner, for the  $y$  direction only, keeping the domain from rotating but allowing it to slide in the  $x$  direction. When done, click **OK**.



### Subdomain Settings

The air domain need not be part of the plane strain model.

- I Specify the subdomain settings for the Plane Strain application mode according to the following table.

	SUBDOMAIN I				SUBDOMAINS 2-4			
Page	Material		Loads		Material		Loads	
	$E$	ESi			$E$	ESi02		
	$\nu$	nuSi	Temp	T1	$\nu$	nuSi02	Temp	T1
	$\alpha$	alphaSi	Tempref	T0	$\alpha$	alphaSi02	Tempref	T0

Before entering the temperatures on the **Load** page, you must select the **Include thermal expansion** check box. It is not necessary to specify the density  $\rho$ , because it does not enter the equation for static problems.

## EXPRESSION VARIABLES

In the **Subdomain Expressions** dialog box in the **Options** menu, define the following variables.

VARIABLE NAME	SUBDOMAIN	EXPRESSION
E	1	ESi
	2,3,4	ESi02
nu	1	nuSi
	2,3,4	nuSi02
alpha	1	alphaSi
	2,3,4	alphaSi02
ex	1,2,3,4	ux
ey	1,2,3,4	vy
ez	1,2,3,4	e00+e11*x+e22*y
sx	1,2,3,4	sx_smpn+E/((1+nu)*(1-2*nu))*nu*ez
sy	1,2,3,4	sy_smpn+E/((1+nu)*(1-2*nu))*nu*ez
sz	1,2,3,4	E/((1+nu)*(1-2*nu))*(nu*ex+nu*ey+(1-nu)*ez-(1+nu)*alpha*(T1-T0))
N	1	nSi
	2	nBuf
	3	nClad
	4	nCore
Nx	1	N
	2,3,4	N-B1*sx-B2*(sy+sz)
Ny	1	N
	2,3,4	N-B1*sy-B2*(sx+sz)
Nz	1	N
	2,3,4	N-B1*sz-B2*(sx+sy)

The variables  $ex$ ,  $ey$ , and  $ez$  are the strains  $\epsilon_x$ ,  $\epsilon_y$ , and  $\epsilon_z$ , and the variables  $sx$ ,  $sy$ , and  $sz$  are the stresses  $\sigma_x$ ,  $\sigma_y$ , and  $\sigma_z$ . The definitions of the refractive index variables  $Nx$ ,  $Ny$ , and  $Nz$  differ slightly compared to the previous model. In this case they are expressed in the generalized stress variables  $sx$ ,  $sy$ , and  $sz$ . Because of the use of integration coupling variables ( $e00$ ,  $e11$ , and  $e22$ ), COMSOL Multiphysics cannot determine the unit for other variables that directly or indirectly depend on these

integration coupling variables. This causes the warnings for inconsistent units here and in the specification of the refractive index. You can disregard these warnings.

### COUPLING VARIABLES

The unknown parameters  $e_0$ ,  $e_1$ , and  $e_2$  should be constant within the whole modeling domain. You can achieve this by defining coupling variables that map these dependent variables from the point to the entire model:

- 1 Switch to **Geom2** by clicking its tab.
- 2 Open the **Point Integration Variables** dialog box by selecting **Integration Coupling Variables>Point Variables** from the **Options** menu.
- 3 Enter the following names and expressions, each on a separate row in the table on the **Source** tab.

NAME	EXPRESSION
e00	e0
e11	e1
e22	e2

The variables e00, e11, and e22 take the values of e0, e1, and e2, respectively, in the entire model.

### SUBDOMAIN SETTINGS

Finally, enter the additional terms in the generalized plane strain problem.

- 1 Switch to **Geom1** by clicking its tab.
- 2 Select **Equation System>Subdomain Settings** from the **Physics** menu.
- 3 On the **Weak** tab in the **Subdomain Settings - Equation System** dialog box, add  $-E/((1+\nu)*(1-2\nu))*(ex\_test*\nu*ez+ey\_test*\nu*ez)-ez\_test*sz$  in the second row of the **weak** term for all subdomains. The two first rows correspond to the plane strain problem. You can enter this expression in any of these two fields, because the software adds them together when assembling the problem.

### MESH GENERATION

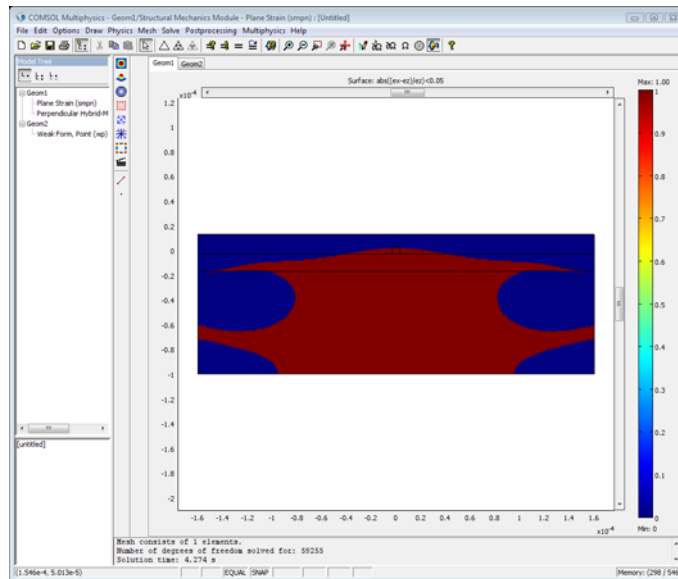
- 1 Initialize the mesh.
- 2 Refine it twice.
- 3 Switch to **Geom2** by clicking its tab.
- 4 Initialize the mesh in this geometry too.

## COMPUTING THE SOLUTION

- 1 In the **Solver Parameters** dialog box select the **Stationary** solver. When done, click **OK**.
- 2 On the **Solve for** tab in the **Solver Manager** dialog box, select only the **Plane Strain** and the **Weak Form, Point** application modes. This excludes the hybrid-mode waves application mode from the equations to solve. When done, click **OK**.
- 3 Click the **Solve** button on the Main toolbar.

## POSTPROCESSING AND VISUALIZATION

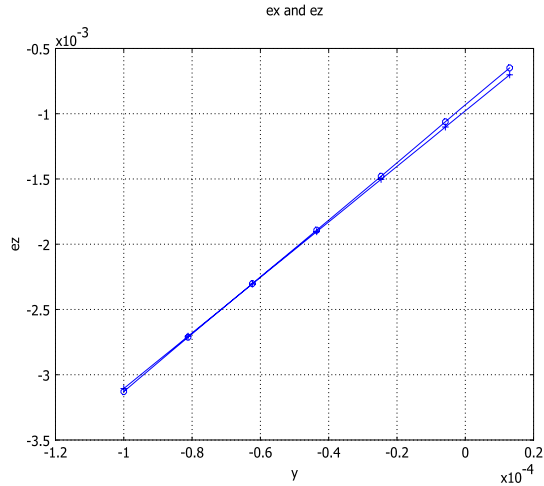
- 1 Switch to **Geom1** by clicking its tab.
- 2 Click the **Surface** tab on the **Plot Parameters** dialog box. Change the string in the **Expression** edit field on the **Surface Data** page to  $\text{abs}((\epsilon_x - \epsilon_z) / \epsilon_z) < 0.05$ . Clear the **Smooth** check box. Click **OK** to visualize the area where the relative difference between the  $x$  and  $z$  strain is within 5%. The model is most accurate in the regions close to the core, far from the boundaries on the far left and right.



- 3 For symmetry reasons the strains  $\epsilon_x$  and  $\epsilon_z$  should be equal. To see how well this model has achieved this, make a cross-section plot of them. Open the **Cross-Section Plot Parameters** dialog box. Enter the **Title**  $\epsilon_x$  and  $\epsilon_z$  by using the **Title/Axis** dialog box on the **General** page. On the **Line/Extrusion** tab, set **x0** to 0, **x1** to 0, **y0** to  $-1 \times 10^{-4}$ , and **y1** to  $1.3 \times 10^{-5}$  to make a vertical line across the structure. Enter the **y-axis data**

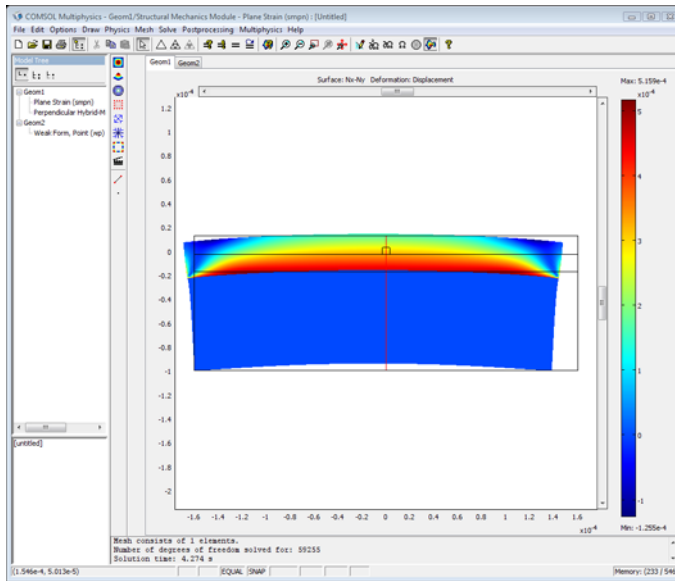
expression  $e_x$ , and set the **Line resolution** to 25. Set the **x-axis data** to  $y$ . Click the **Line Settings** button and select **Circle** as **Line marker**. Click **Apply** to plot  $e_x$ .

- 4 On the **General** tab check **Keep current plot** to be able to plot  $e_z$  in the same figure window.
- 5 Then change the **Expression** to  $e_z$  in **y-axis data**. Use **Plus sign** as **Line marker** for this plot. Click **OK** to make the plot.



- 6 To view the stress-induced birefringence  $n_x - n_y$  together with the deformation, open the **Plot Parameters** dialog box and select **Surface** and **Deformation** plot. On the **Surface** page, enter the  $N_x - N_y$  in the **Expression** edit field on the **Surface Data** page, and select the **Smooth** check box again. As **Deformation data** use **Displacement (smpn)**

from the **Predefined quantities** list on the **Subdomain Data** page, which is the default selected on the **Deform** tab. Click **OK**.



---

**Note:** This model includes an extension of the plane strain PDE system. This means that some of the postprocessing variables in the Plane Strain application mode are no longer valid and give wrong results if used. We recommend that you use only the variables entered in the **Subdomain Expressions** dialog box for postprocessing.

---

### *Optical Mode Analysis*

Now continue with the mode analysis in the same way as was done in the previous model on page 225.

#### **GEOMETRY MODELING**

You can reduce the computational domain significantly for the optical mode analysis, because the energy of the fundamental modes is concentrated in the core region, and the energy density decays exponentially in the cladding and buffer regions.

Draw a rectangle with corners at  $(-1e-5, -1e-5)$  and  $(1e-5, 1e-5)$ .

## PHYSICS SETTINGS

Next, set up the parameters for the hybrid-mode wave problem. Select the **Perpendicular Hybrid-Mode Waves** application mode in the **Multiphysics** menu.

### *Scalar Variables*

- 1 From the **Physics** menu, choose **Scalar Variables**.
- 2 In the **Application Scalar Variables** dialog box, set the free space wavelength to  $1.55\text{e-}6$ . Click **OK**.

### *Subdomain Settings*

- 1 From the **Physics** menu, choose **Subdomain Settings**.
- 2 Deactivate the Perpendicular Hybrid-Mode Waves application mode in all subdomains except 4, 5, and 6 (clear the **Active in this domain** check box for these subdomains). This makes sure that the solution for the electromagnetic waves only includes the newly drawn rectangle.
- 3 For all active domains, select **n (anisotropic)** and set  $N_x$ ,  $N_y$ , and  $N_z$ , respectively, as anisotropic refractive indices by changing the entries on the diagonal in the refractive index matrix. When done, click **OK**.

## EXPRESSION VARIABLES

Occasionally the associative geometry algorithm does not update all variables and settings exactly as expected when modifying the geometry. In this case the expression for  $N$  might have got the wrong value in domain 6. In the **Subdomain Expressions** dialog box change the value of  $N$  in Subdomain 6 to  $n_{\text{Core}}$ .

## MESH GENERATION

- 1 On the **Subdomain** tab in the **Free Mesh Parameters** dialog box, set the **Maximum element size** to  $2\text{e-}6$  for Subdomains 4 and 5, and to  $2.5\text{e-}7$  for Subdomain 6.
- 2 Initialize the mesh.

## COMPUTING THE SOLUTION

- 1 In the **Solver Parameters** dialog box, select the **Eigenvalue** solver.
- 2 Enter 1.46 in the text field **Search for effective mode indices around**. The default **Desired number of effective mode indices** is set to 6, which is fine. These settings make the eigenmode solver search for the 6 eigenmodes with effective mode indices closest to the value 1.46. This value is an estimate of the effective mode index for the fundamental mode. For propagating modes it must hold that

$$n_{\text{eff}} < n_{\text{core}} = 1.456$$

- 3 In the **Solver Manager** dialog box, select only the **Perpendicular Hybrid-Mode Waves** application mode on the **Solve For** page. This ensures that only this application mode is part of the eigenmode computation.
- 4 On the **Initial Value** page, set the initial value to **Current solution**. This takes the plane strain solution and use it to evaluate the refractive indices for the mode analysis.
- 5 Click the **Solve** button.

The following table contains the effective mode indices together with the effective mode indices obtained from the analysis in the previous model without the generalization of the plane strain equations.

<b>EFFECTIVE MODE INDEX</b>	<b>GENERALIZED PLANE STRAIN</b>	<b>PLANE STRAIN</b>	<b>DIFFERENCE</b>
$n_{\text{eff}1}$	1.450403	1.451136	0.73
$n_{\text{eff}2}$	1.450115	1.450872	0.76
$n_{\text{eff}3}$	1.444671	1.445405	0.73
$n_{\text{eff}4}$	1.444398	1.445139	0.74
$n_{\text{eff}5}$	1.444383	1.445132	0.75
$n_{\text{eff}6}$	1.444109	1.444865	0.76

There is a systematic shift in the propagation constants when the strain in the  $z$  direction is taken into account.



# Second Harmonic Generation of a Gaussian Beam

## *Introduction*

---

Laser systems are an important application area in modern electronics. There are several ways to generate a laser beam, but they all have one thing in common: The wavelength is determined by the stimulated emission, which depends on material parameters. It is especially difficult to find lasers that generate short wavelengths (for example, ultraviolet light). With nonlinear materials it is possible to generate harmonics that are an even multiple of the frequency of the laser light. With such materials it is straightforward to generate a laser beam with half the wavelength of the original beam. This model shows how to set up a second harmonic generation as a transient wave simulation using nonlinear material properties. A YAG ( $\lambda=1.06 \mu\text{m}$ ) laser beam is focused on a nonlinear crystal, so that the waist of the beam is inside the crystal.

## *Model Definition*

---

When a laser beam propagates it travels as an approximate plane wave with a cross-section intensity of Gaussian shape. The waist of the beam is defined as the Gaussian function's characteristic length. A focused laser beam has a minimum waist,  $w_0$ , at a certain point along the direction of propagation. The solution of the time-dependent Maxwell's equations gives the following electric field ( $x$ -component):

$$\mathbf{E}(x, y, z) = E_{0x} \frac{w_0}{w(z)} e^{-r^2/\omega(z)^2} \cos\left(\omega t - kz + \eta(z) - r^2 \frac{k}{2R(z)}\right) \mathbf{e}_x ,$$

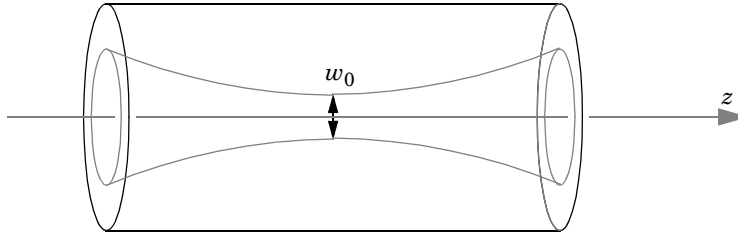
where

$$w(z) = w_0 \sqrt{1 + \left(\frac{z}{z_0}\right)^2}$$

$$\eta(z) = \text{atan}\left(\frac{z}{z_0}\right) .$$

$$R(z) = z \left(1 + \left(\frac{z_0}{z}\right)^2\right)$$

In these expressions,  $w_0$  is the minimum waist,  $\omega$  is the angular frequency,  $r$  is the radial cylindrical coordinate, and  $k$  is the wave number. The wave fronts of the beam is not exactly planar, it propagates like a spherical wave with radius  $R(z)$ . Close to the point of the minimum waist the wave is almost plane. These expressions are used to excite a plane wave at one end of a cylinder.



In this example the problem is solved as a 2D cross section that resembles the axial symmetry of the problem. However, due to the  $x$ -polarization of the  $\mathbf{E}$ -field, true axial symmetry cannot be used (available as an application mode in COMSOL Multiphysics). Here the  $\varphi$  direction is approximates with a standard 2D cross section, neglecting the revolution around the  $z$ -axis. So this actually models an infinitely long Gaussian beam plane. The second harmonics generation is probably not affected significantly by this approximation. In the model “Propagation of a 3D Gaussian Beam Laser Pulse” on page 261, propagation of a true 3D Gaussian beam is calculated without any nonlinear effects.

The nonlinear properties for 2<sup>nd</sup> harmonic generation in a material can be defined with the following matrix,

$$\mathbf{P} = \begin{bmatrix} d_{11} & d_{12} & d_{13} & d_{14} & d_{15} & d_{16} \\ d_{21} & d_{22} & d_{23} & d_{24} & d_{25} & d_{26} \\ d_{31} & d_{32} & d_{33} & d_{34} & d_{35} & d_{36} \end{bmatrix} \cdot \begin{bmatrix} E_x^2 \\ E_y^2 \\ E_z^2 \\ 2E_z E_y \\ 2E_z E_x \\ 2E_x E_y \end{bmatrix},$$

where  $\mathbf{P}$  is the polarization. The model only uses the  $d_{11}$  parameter for simplicity. To keep the problem size small the nonlinear parameter is magnified by some orders of magnitude. The crystal here has a value of  $10^{-17}$  F/V, when the values for most

materials usually are in the  $10^{-22}$  F/V range. Without this magnification, a detectable 2nd harmonics requires the  $z$  direction of the geometry to be much longer, resulting in a large problem.

The Gaussian beam is also excited as a pulse in time, using a Gaussian envelope function. This produces a wave package with a Gaussian frequency spectrum. The frequency spectrum from a series of time samples can be calculated using the fast Fourier transform (FFT), which you can do from the COMSOL Script command line using the function `fft`.

### Results and Discussion

---

The main purpose of this simulation is to calculate the second harmonic generation when the pulse travels along the 20  $\mu\text{m}$  geometry. So you have to solve for the time it takes for the pulse to enter, pass, and disappear from the volume. The pulse has a characteristic time of 10 fs, and below you can see the pulse after it has traveled 61 fs.

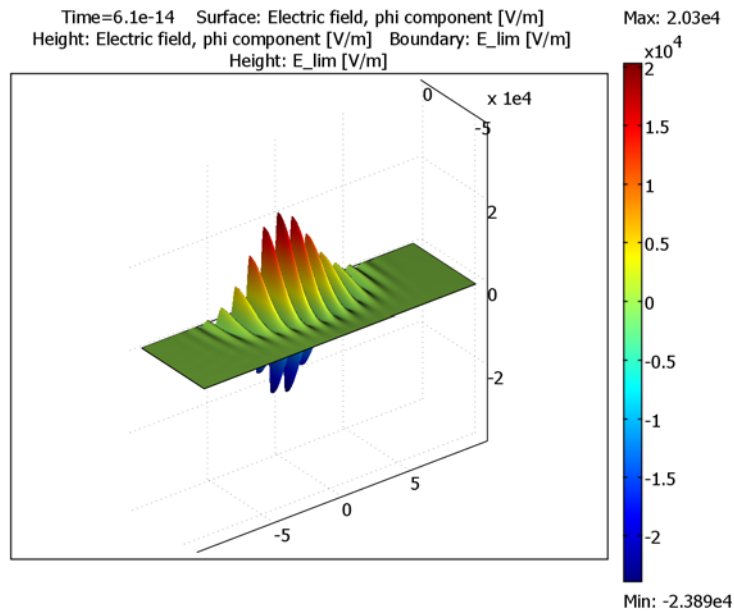


Figure 4-11: The pulse after 61 fs.

After 90 fs the pulse has reached the output boundary (see Figure 4-12). The simulation has to continue for another 30 fs so the pulse completely disappears.

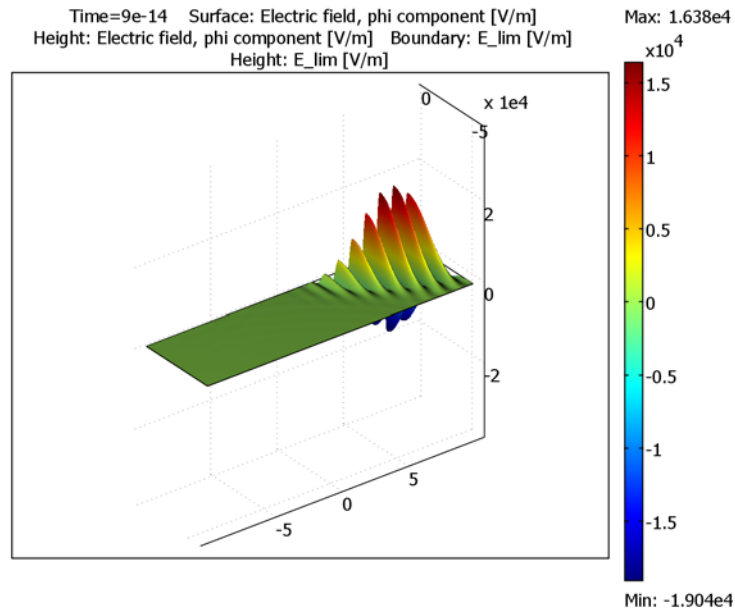
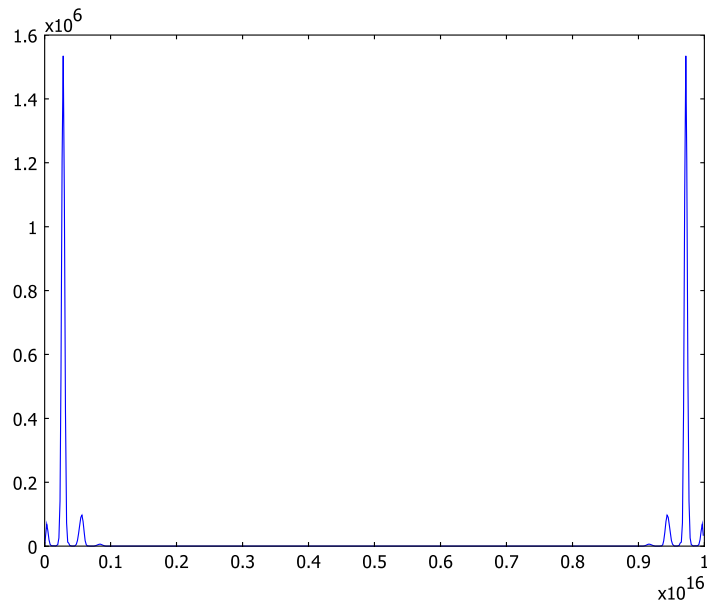


Figure 4-12: The pulse after 90 fs. It has now reached the output boundary.

The simulation stores the times between 60 fs and 120 fs, which is when the pulse passes the output boundary. The electric field at this boundary has a second harmonic component that is extracted using FFT. The extraction and transformation is performed at the COMSOL Script command line. The result appears in Figure 4-13 on page 253. There are two groups of peaks in the figure, where the peaks on the right are mirrors of the ones to the left. This mirror effect is due to the sampling of the harmonic signals, which always creates a mirror spectrum around the sampling frequency.



*Figure 4-13: The result from the FFT on the beam at the output boundary. The two small peaks between the large peaks are the second harmonic generation.*

---

### *Reference*

1. A. Yariv, *Quantum Electronics*, 3rd Edition, John Wiley & Sons, 1988.

---

**Model Library path:** RF\_Module/Optics\_and\_Photonics/  
second\_harmonic\_generation

---

### *Modeling Using the Graphical User Interface*

---

#### **MODEL NAVIGATOR**

- 1** In the **Model Navigator**, select **2D** in the **Space dimension** list.
- 2** Click the **Multiphysics** button, and then Click the **Add Geometry** button.

- 3 In the **Add Geometry** dialog box, type  $z_r\ \phi$  in the **Independent variables** edit field. Click **OK**.
- 4 Select the **RF Module>In-Plane Waves>TE Waves>Transient propagation** application mode.
- 5 Type  $A\phi$  in the **Dependent variables** edit field, and click **Add**.
- 6 Click **OK**.

#### OPTIONS AND SETTINGS

- 1 From the **Options** menu, choose **Constants**.
- 2 In the **Constants** dialog box, define the following constants with names, expressions, and descriptions (optional):

NAME	EXPRESSION	DESCRIPTION
w0	2[um]	Minimum waist of laser beam
lambda0	1.06[um]	Wavelength of laser beam
E0	3e4[V/m]	Peak electric field
z0	$\pi \cdot w0^2 / \lambda0$	Peak electric field z position
k0	$2 \cdot \pi / \lambda0$	Wave number
t0	25[fs]	Pulse time delay
dt	10[fs]	Pulse width
d11	1e-17[F/V]	Matrix element for 2nd generation

- 3 Click **OK**.

#### GEOMETRY MODELING

All the dialog boxes for specifying the primitive objects are accessed from the **Draw** menu and **Specify Objects**. The first column in the table below contains the labels of the geometric objects. These are automatically generated by COMSOL Multiphysics, and you do not have to enter them. Just check that you get the correct label for the objects that you create.

- 1 Draw a rectangle with the properties according to the table below.

LABEL	WIDTH	HEIGHT	BASE	CORNER
R1	20e-6	6e-6	Corner	(-1e-5, -6e-6)

- 2 Click the **Zoom Extents** toolbar button.

## PHYSICS SETTINGS

- 1 From the **Options** menu, choose **Expressions>Scalar Expressions**.
- 2 In the **Scalar Expressions** dialog box, define the following variables with names, expressions, and descriptions (optional):

NAME	VALUE	DESCRIPTION	UNIT
w	$w_0 \sqrt{1 + (z/z_0)^2}$	Analytical waist function along z	m
eta	$\text{atan}(z/z_0)$	Analytical angle along z	m
R	$z \cdot (1 + (z_0/z)^2)$	Analytical radius along z	m
c0	$1/\sqrt{\epsilon_0 \mu_0}$	Speed of light	m/s
omega0	$2\pi c_0/\lambda_0$	Angular frequency	rad/s

- 3 Click **OK**.
- 4 From the **Options** menu, choose **Expressions>Boundary Expressions**.
- 5 In the **Boundary Expressions** dialog box, define the following variables with names and expressions:

SETTINGS	BOUNDARY 1	BOUNDARY 2	BOUNDARY 3
E_bnd	$w_0/w \cdot \exp(-r^2/w_{\text{bnd}}^2) \cdot \cos(\omega_0 t - k_0 z + \eta_{\text{bnd}} - r^2 k_0 / (2R_{\text{bnd}}))$		
E_pulse	$\exp(-(t-t_0)^2/dt^2)$		
E_lim		$E_0 \cdot 1.2$	$-E_0 \cdot 1.2$

Because of the use of an integration coupling variable ( $w_{\text{bnd}}$ ), COMSOL Multiphysics cannot determine the unit for  $E_{\text{bnd}}$  and warns for an inconsistent unit here and in the specification of the scattering boundary condition. You can disregard these warnings.

- 6 Click **OK**.
- 7 From the **Options** menu, point to **Integration Coupling Variable** and then click **Point Variables**.
- 8 In the **Point Integration Variables** dialog box, define the following variables with names and expressions. Use **Global destination** for all variables.

NAME	POINT 2	ALL OTHERS
w_bnd	w	
eta_bnd	eta	
R_bnd	R	

### Boundary Conditions

- 1 From the **Physics** menu, open the **Boundary Settings** dialog box and enter the settings according to the following two tables (leave all fields not specified at their default values):

SETTINGS	BOUNDARY 1	BOUNDARY 4
Boundary condition	Scattering boundary	Scattering boundary
$E_{0,\varphi}$	$E_0 * E\_pulse * E\_bnd$	0

SETTINGS	BOUNDARY 2	BOUNDARY 3
Boundary condition	Perfect electric conductor	Perfect magnetic conductor

- 2 Click **OK**.

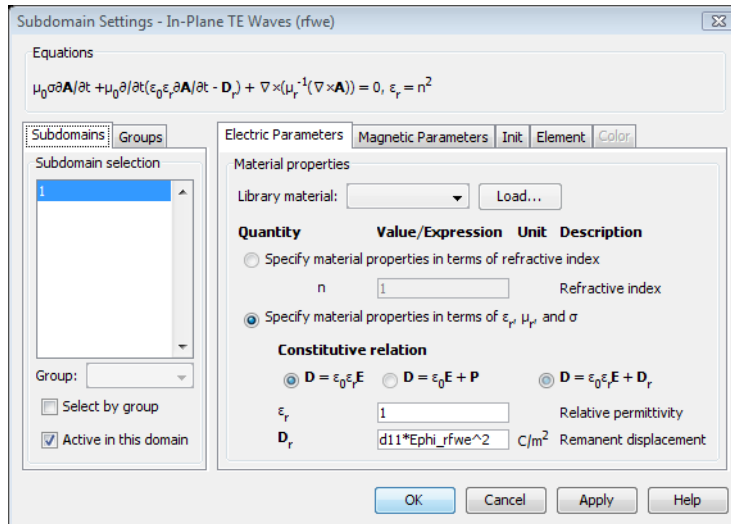
### Subdomain Settings

- 1 From the **Physics** menu, open the **Subdomain Settings** dialog box, select Subdomain 1, and click the  $\mathbf{D} = \epsilon_0 \epsilon_r \mathbf{E} + \mathbf{D}_r$  button for the constitutive relation.
- 2 Define the subdomain settings according to the following table:

SETTINGS	SUBDOMAIN 1
$\epsilon_r$	1
$D_r$	$d11 * E_{phi\_r} fwe^2$

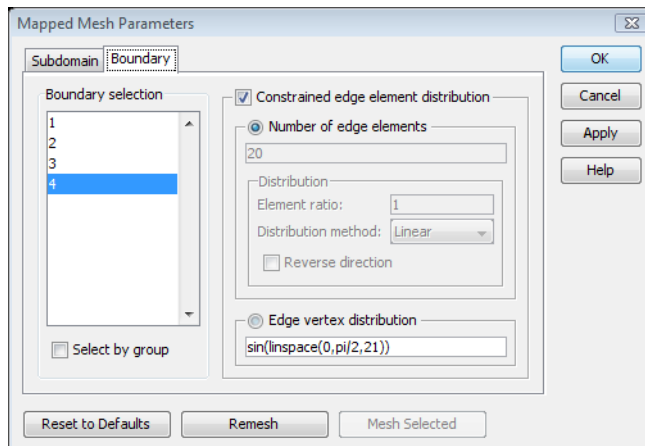
- 3 Click **OK**.



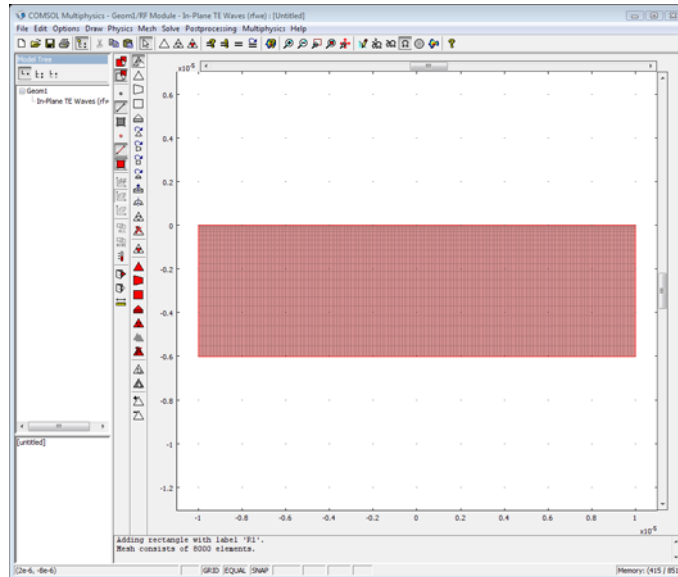


## MESH GENERATION

- 1 From the **Mesh** menu, choose **Mapped Mesh Parameters**.
- 2 Click on the **Boundary** tab, and select Boundary 1.
- 3 Select the **Constrained edge element distribution** check box.
- 4 Click the **Edge vertex distribution** button. In the edit field below, type `sin(linspace(0, pi/2, 21))`. This creates a denser mesh closer to the upper boundary.



- 5 Select Boundary 4 and do the same operations there.
- 6 Select Boundary 2, click the **Edge vertex distribution** button, and type  $0:5e-8:20e-6$  in the edit field.
- 7 Click the **Remesh** button and then click **OK**.



### PROBE PLOT FOR FFT ANALYSIS

The amount of time steps that has to be saved for the FFT analysis is very large, and to store all solutions of the **A**-field results in a huge model file. For the FFT, it is only interesting to look at the field at the output boundary. If you make a probe plot of this quantity, the resulting plot contains all steps taken by the solver. You can later use the data points stored in the figure window to make the FFT analysis. As a result, you can reduce the number of output times to include only those values intended for field plots.

- 1 From the **Postprocessing** menu, choose **Probe Plot Parameters**.
- 2 In the **Probe Plot Parameters** dialog box, click the **New** button.
- 3 In the dialog box that appears, choose **Point probe** from the **Plot type** list, and enter Eout in the **Plot name** edit field. Click **OK**.
- 4 Select boundary 4, and choose **Electric field, phi component** from the **Predefined quantities** list.

- 5 Click the **New** button again and repeat the steps above to define a point probe plot named  $E_{in}$  for boundary 2.
- 6 Select the **Plot all plots in the same axis** check box. Click **OK**.

#### COMPUTING THE SOLUTION

- 1 From the **Solve** menu, choose **Solver Parameters**.
- 2 In the **Times** edit field type 0 6.1e-14 9e-14 1.2e-13.
- 3 Click on the **Time Stepping** tab, and select the **Manual tuning of step size** check box. Enter 1e-16 in the **Initial time step** and **Maximum time step** edit fields.
- 4 Click **OK**.
- 5 Click the **Solve** toolbar button.

#### POSTPROCESSING AND VISUALIZATION

- 1 Select **Plot Parameters** from the **Postprocessing** menu.
- 2 Make sure that the **Surface**, **Boundary**, and **Geometry edges** check boxes are selected under the **General** tab.
- 3 Select time **6.1e-14** from the **Solution at time** list.
- 4 Click the **Surface** tab, and enter  $E_{phi\_rfwe}$  in the **Expression** edit field.
- 5 Click the **Height Data** tab, select the **Height data** check box, and enter  $E_{phi\_rfwe}$  in the **Expression** edit field.
- 6 Click the **Boundary** tab, type  $E_{lim}$  in the **Expression** edit field.
- 7 Click the **Height Data** tab, select the **Height data** check box, and enter  $E_{lim}$  in the **Expression** edit field.
- 8 Click the **Uniform color** option button.
- 9 Click the **Color** button and select white from the palette in the dialog box that appears. Click **OK** to close the dialog box.
- 10 Click **Apply** to get the plot in Figure 4-11.
- 11 Click on the **General** tab, and select time **9e-14**. Click **OK**. You should now see the plot in Figure 4-12.

#### FFT ANALYSIS USING COMSOL SCRIPT

To get the Fourier transform of the output, run some commands in COMSOL Script (or MATLAB):

- 1 From the **File** menu, choose **COMSOL Script**. The **COMSOL Script** window opens automatically.

- 2** In the **COMSOL Script** window, enter the following commands:

```
c = get(gca, 'children');  
xdata = get(c(1), 'xdata');  
ydata = get(c(1), 'ydata');  
ydata = ydata(diff([xdata inf])>0);  
xdata = xdata(diff([xdata inf])>0);
```

These statements first fetch the output field data from the current figure. This data contains a lot of duplicate points that you remove with the last two statements.

- 3** Now you can continue to preform the FFT analysis on a selected time interval of the output field.

```
T = xdata(601:end);  
Eout = ydata(601:end);  
freq = ((1:length(T))-1)/(length(T)*mean(diff(T)));  
xEout = fft(Eout);  
plot(freq,abs(xEout));
```

This produces the plot in Figure 4-13 on page 253.

# Propagation of a 3D Gaussian Beam Laser Pulse

## *Introduction*

---

This is a 3D version of the model “Second Harmonic Generation of a Gaussian Beam” on page 249. The differences are that the nonlinear material parameters have been removed, the geometry is smaller, and the laser pulse is shorter. This is necessary to reduce the execution time and the model file size shipped with the product.

## *Model Definition*

---

All the details about Gaussian laser beams are covered in the model “Second Harmonic Generation of a Gaussian Beam” on page 249. This model uses the same YAG laser pulse, but with a characteristic time of 40 fs. Because the Gaussian pulse has spherical wave fronts when it approaches and leaves the minimum waist, the input and output boundaries are made spherical.

The model also use the symmetry of the cylindrical shape so that it is only necessary to simulate one quarter of the full volume. The perfect electric and perfect magnetic boundary conditions are used as symmetry conditions one these boundaries. It is not straightforward to use axial symmetry, because the polarization of the beam is in the  $x$  direction.

## Results and Discussion

Notice the spherical fronts in the following figures. These figures show the pulse that propagates at two different times: at the entry and just after passing the minimum waist.

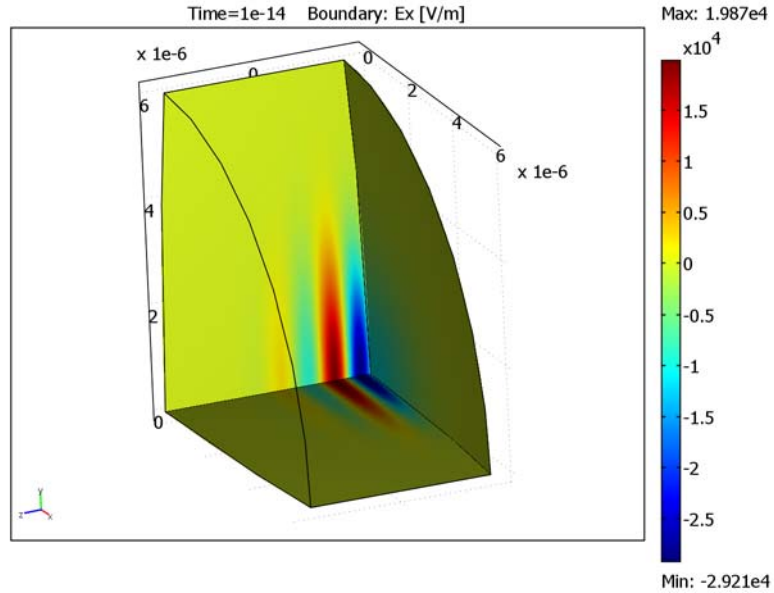
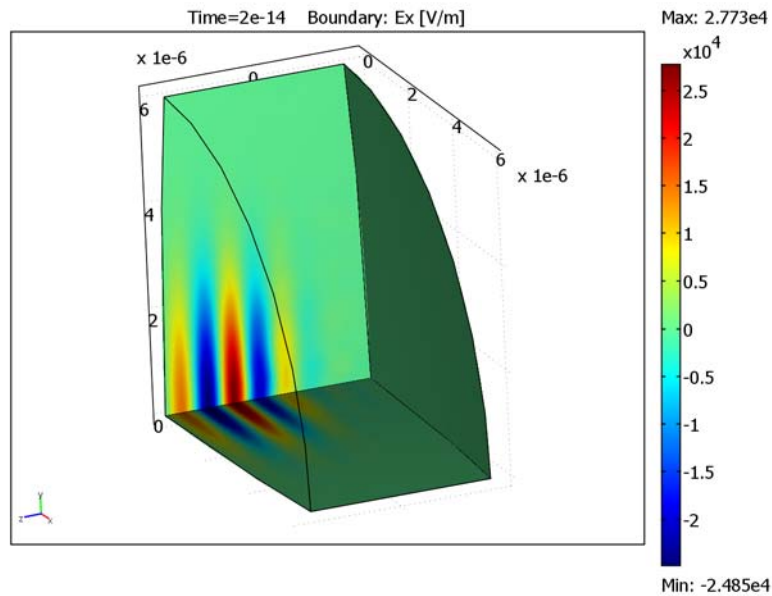


Figure 4-14: The pulse entering the domain at  $t=10$  fs. The spot size is shown on the input boundary to the left.



*Figure 4-15: After 20 fs the pulse has passed the minimum waist and soon reaches the output boundary.*

---

**Model Library path:** RF\_Module/Optics\_and\_Photonics/gaussian\_beam\_3d

---

### *Modeling Using the Graphical User Interface*

---

#### **MODEL NAVIGATOR**

- 1 In the **Model Navigator**, select **3D** in the **Space dimension** list.
- 2 Select the **RF Module>Electromagnetic Waves>Transient propagation** application mode. Click **OK**.

#### **OPTIONS AND SETTINGS**

- 1 From the **Options** menu, choose **Constants**.

- 2 In the **Constants** dialog box, define the following constants with names, expressions, and descriptions (optional):

NAME	EXPRESSION	DESCRIPTION
w0	2[um]	Minimum waist of laser beam
lambda0	1.06[um]	Wavelength of laser beam
E0	30[kV/m]	Peak electric field
z0	$\pi \cdot w0^2 / \lambda0$	Peak electric field z position
k0	$2 \cdot \pi / \lambda0$	Wave number
t0	10[fs]	Pulse time delay
dt	40[fs]	Pulse width

- 3 Click **OK**.

### GEOMETRY MODELING

All the dialog boxes for specifying the primitive objects are accessed from the **Draw** menu and **Specify Objects**. The first column in the tables below contains the labels of the geometric objects. These are automatically generated by COMSOL Multiphysics, and you do not have to enter them. Just check that you get the correct label for the objects that you create.

Begin by drawing the cross section of the simulation domain followed by an extrusion to 3D. The input and output boundaries are then adjusted so they get a slight spherical shape, consistent with the spherical wave fronts of the Gaussian pulse.

- 1 From the **Draw** menu, select **Work-Plane Settings**.

- 2 In the **Work-Plane Settings** dialog box, select the **x-y** plane and enter  $-2e-6$  in the **z** edit field. Click **OK**.

Draw a circle with the properties according to the table below.

NAME	RADIUS	BASE	(X, Y)
C1	6e-6	Center	(0, 0)

- 3 Draw rectangles with the properties according to the following table.

NAME	WIDTH	HEIGHT	BASE	CORNER
R1	6e-6	1.2e-5	Corner	(-6e-6, -6e-6)
R2	6e-6	6e-6	Corner	(0, -6e-6)

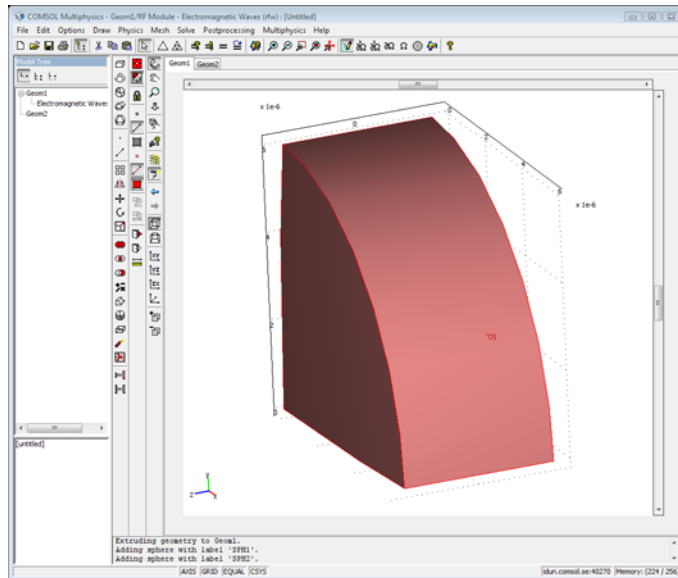
- 4 Click the **Zoom Extents** toolbar button.



- 5 Select all the rectangles, R1 and R2, and click the **Union** toolbar button.
- 6 Click the **Delete Interior Boundaries** toolbar button.
- 7 Select all objects by pressing Ctrl+A, and click the **Create Composite Object** toolbar button.
- 8 In the dialog box that appears, enter C1-C01 in the **Set formula** edit field. Click **OK**.
- 9 Select **Extrude** from the **Draw** menu.
- 10 In the **Extrude** dialog box, enter  $4e-6$  in the **Distance** edit field, and click **OK**.
- 11 It is desired to have the spherical wave fronts of the beam coincide with the input and output boundaries. Do this with two spheres with a radius from the value of the R expression evaluated at one of the boundaries. Specify the two spheres using the following properties:

NAME	RADIUS	BASE	CORNER
SPH1	$7.22713e-5$	Center	$(7.22713e-5-2e-6, 0)$
SPH2	$7.22713e-5$	Center	$(-7.22713e-5+2e-6, 0)$

- 12 Select all objects by pressing Ctrl+A, and click the **Intersection** toolbar button.
- 13 Click the **Headlight** toolbar button. You should now see the geometry shown in the figure below.



## PHYSICS SETTINGS

- 1 Go to geometry **Geom1** by clicking the **Geom1** tab.
- 2 From the **Options** menu, choose **Expressions>Scalar Expressions**.
- 3 In the **Scalar Expressions** dialog box, define the following variables with names, expressions, and descriptions (optional):

NAME	VALUE	DESCRIPTION
w	$w0*\sqrt{1+(z/z0)^2}$	Analytical waist function along z
eta	$\text{atan}(z/z0)$	Analytical angle along z
R	$z*(1+(z0/z)^2)$	Analytical radius along z
r	$\sqrt{x^2+y^2}$	Radial coordinate
c0	$1/\sqrt{\text{epsilon}0_{\text{r}fw}*\mu0_{\text{r}fw}}$	Speed of light
omega0	$2*\pi*c0/\text{lambda}0$	Angular frequency

- 4 Click **OK**.
- 5 From the **Options** menu, choose **Expressions>Boundary Expressions**.
- 6 In the **Boundary Expressions** dialog box, define the following variables with names and expressions:

SETTING	BOUNDARY 3	ALL OTHER
E_bnd	$w0/w*\exp(-r^2/w\_bnd^2)*\cos(\text{omega}0*t-k0*z+\text{eta\_bnd}-r^2*k0/(2*R\_bnd))$	
E_pulse	$\exp(-(t-t0)^2/dt^2)$	

Because of the use of an integration coupling variable ( $w\_bnd$ ), COMSOL Multiphysics cannot determine the unit for  $E\_bnd$  and warns for an inconsistent unit here and in the specification of the scattering boundary condition. You can disregard these warnings.

- 7 Click **OK**.
- 8 From the **Options** menu, select **Integration Coupling Variable>Point Variables**.
- 9 In the **Point Integration Variables** dialog box, define the following variables with names and expressions. Use **Global destination** for all variables.

NAME	POINT 1	ALL OTHERS
w_bnd	w	
eta_bnd	eta	
R_bnd	R	

### Boundary Conditions

- 1 From the **Physics** menu, open the **Boundary Settings** dialog box and enter the settings according the following two tables (leave all fields not specified at their default values):

SETTINGS	BOUNDARY 3	BOUNDARY 4
Boundary condition	Scattering boundary	Scattering boundary
$E_{0,x}$	$E_0 * E_{pulse} * E_{bnd}$	0
$E_{0,y}$	0	0
$E_{0,z}$	0	0

SETTING	BOUNDARY 2	BOUNDARIES 1, 5
Boundary condition	Perfect magnetic conductor	Perfect electric conductor

- 2 Click **OK**.

### Subdomain Settings

Use the default values for the subdomain settings.

### MESH GENERATION

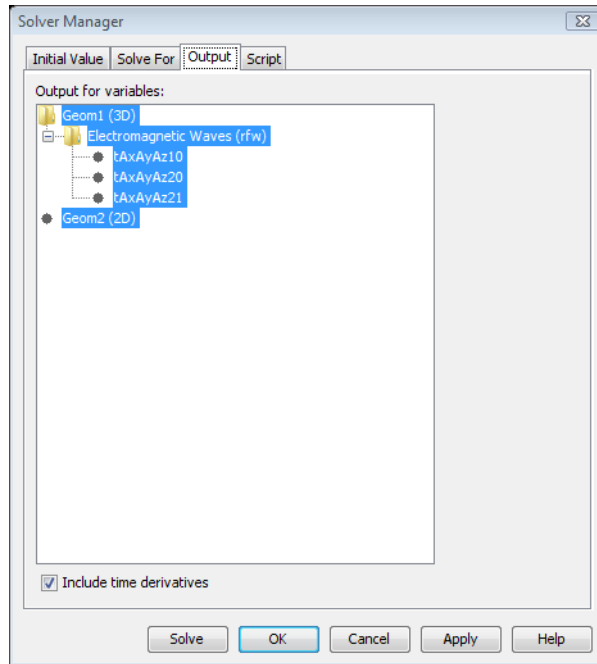
- 1 From the **Mesh** menu, choose **Free Mesh Parameters**.
- 2 Click the **Custom mesh size** button and enter  $1e-7$  in the **Maximum element size** edit field.
- 3 Click the **Advanced** tab.
- 4 Enter  $0.12$  in the **x-direction scale factor** and **y-direction scale factor** edit fields. This keeps the small mesh size in the  $z$  direction but stretches the elements in the  $x$  and  $y$  directions.
- 5 Click the **Remesh** button, then click **OK**.

### COMPUTING THE SOLUTION

- 1 From the **Solve** menu, choose **Solver Parameters**.
- 2 In the **Times** edit field type  $0:1e-14:3e-14$ .
- 3 Select the **GMRES** from the **Linear system solver** list.
- 4 Select **Algebraic multigrid** from the **Preconditioner** list.
- 5 Make sure that the **Symmetric** option is selected from the **Matrix symmetry** list.
- 6 Click the **Time Stepping** tab and select the **Manual tuning of step size** check box. Enter  $1e-16$  in the **Initial time step** and **Maximum time step** edit fields.

7 Click **OK**.

8 From the **Solve** menu, choose **Solver Manager**. Click the **Output** tab and select the **Include time derivatives** check box. This ensures that the time derivatives of the **A**-field, which is the **E**-field, is calculated more accurately.



9 Click **OK**.

10 Click the **Solve** button on the Main toolbar.

The solving process takes some time because this is a large problem that takes about 300 time steps.

## POSTPROCESSING AND VISUALIZATION

Plot the  $x$ -component of the **E**-field on the boundaries. Some boundaries are suppressed so it is possible to get a view into the simulation volume.

- 1 Select **Plot Parameters** from the **Postprocessing** menu.
- 2 Make sure that the **Boundary** and **Geometry edges** check boxes are selected under the **General** tab.
- 3 Click the **Boundary** tab and enter  $E_x$  in the **Expression** edit field.

- 4 Click **OK**.
- 5 From the **Options** menu, select **Suppress>Suppress Boundaries**. In the **Suppress Boundaries** dialog box, select boundaries 4 and 5. Click **OK**.
- 6 Compare the results at times  $1e-14$  and  $2e-14$  to see how the pulse propagates. These results are plotted in Figure 4-14 on page 262 and in Figure 4-15 on page 263.



# I N D E X

- 3D electromagnetic waves model 85, 124
- 3D waveguide 85
- A**
  - absorbing layers 174
  - anisotropic permeability 34
  - antenna impedance 46, 54
  - antennas 136
    - conical 45
    - magnetic dipole 57
    - RF 45
  - application mode
    - Axisymmetric TE waves 58
    - Axisymmetric TM waves 49
    - Bioheat equation 105
    - Boundary mode analysis 91
    - Electromagnetic waves 91
    - In-plane TE waves 177, 185, 198
    - Perpendicular Hybrid-Mode Waves 216
    - Perpendicular hybrid-mode waves 211
    - Plane Strain 216
    - TM waves 105
  - axisymmetric TE waves model 57
  - axisymmetric TM waves model 45
  - azimuthal magnetic field 46
- B**
  - balanced antennas 136
  - bandgap analysis 192
  - bioheat equation model 98
  - birefringence 215
    - stress-induced 224
  - boundary mode analysis 91
  - boundary mode analysis model 85, 124
- C**
  - capacitive coupling 67
  - cavity resonances 81
  - characteristic impedance 54
  - circulator 32
    - cladding 208
    - coaxial cable, feeding a waveguide 157
    - COMSOL Script 77
    - confined mode 208
    - conical antennas 45
    - convergence analysis 227
    - coupling variable
      - scalar 54
    - cross-section plot 155
    - cutoff frequency 86, 125, 184, 188
- D**
  - dipole, magnetic 57
  - drift, in eigenfrequency 64
- E**
  - effective mode index 209
  - eigenfrequency drift 64
  - electromagnetic wave model 3D 85, 124
  - errors in modeling 230
  - expansion coefficient 232
- F**
  - far-field
    - computation 56
    - radiation patterns 49
  - ferrite circulator 32
  - FFT 251, 258
  - fiber
    - optical 208
    - silica glass 208
    - single-mode 208
  - Floquet periodicity boundary conditions 193
  - frequency
    - normalized 209
- G**
  - Gaussian beam 249, 261
  - generalized plane strain 232
  - gyromagnetic ratio 34
- H**
  - H bend 32, 80

- harmonic generation 250
- I** impedance
  - of antenna 46, 54
  - impedance, of microstrip line 20
  - in-plane TE waves model 32, 80, 174, 182
  - intensity, optical 227
- L** Larmor frequency 34
  - lobe pattern 141
  - low-reflecting boundary conditions 138
- M** magnetic dipole 57
  - magnetic frill 14
  - matched boundary conditions 46, 81
  - matched port boundary conditions 35
  - microprism 174
  - microstrip line, impedance of 20
  - microwave circulator 32
  - microwave filters 64
  - microwave waveguide 85
  - mode analysis 225
  - model
    - stress-optical effects 215
    - stress-optical effects, generalized 232
  - modeling errors 230
  - models, overview of 2
  - monoconical antennas 45
  - moving mesh 65
- N** Navier's equation 234
  - weak form 234
  - near-field radiation patterns 47
  - normalized frequency 209
- O** optical fiber 208
  - optical intensity 227
  - optical waveguide 174, 208
  - optics 208
  - optimum radiation frequency 139
  - optoelectronics 208
- P** parametric solver 55
  - patch antennas 136
  - perfect electric conductors 138
  - perfectly matched layers 138, 174
  - Perpendicular Hybrid-Mode Waves 216
  - perpendicular hybrid-mode waves model
    - 208
  - perturbation
    - of material parameters 231
  - photonic bandgap 182, 192
  - photonic crystal 192
  - photonic microprism 174
  - photonic waveguide 215
  - Plane Strain 232
    - application mode 216
    - generalized 232
  - plane strain, generalized 232
  - PML constant 138
  - PML layer 138
  - PMLs. see perfectly matched layers
  - Point Settings dialog box 220, 240
  - Port boundary condition 157
  - power flow 227
  - Poynting vector 227
- Q** quarter-wave resonance 66
- R** radiation patterns 54, 140
  - far-field 49
  - near-field 47
  - reactance 55
  - rectangular waveguides 157
  - refractive index 174, 208
  - resistance 55
  - resonance 66, 81
- S** SAR 110
  - scattering boundary conditions 46
  - scattering parameter 33, 54, 80
  - sea bed logging 164



- second harmonic generation 250
- sensitivity analysis 231
- silica glass fiber 208
- single-mode fiber 208
- single-mode range 85
- solver
  - parametric 55
- S-parameter 33, 54, 80, 85
- S-parameters 163
- specific absorption rate 110
- Specific absorption rate (SAR) 98
- spherical waves 46
- stress-induced birefringence 224
- stress-optical effect 215
- stress-optical effects 215
- stress-optical effects, generalized 232
- stress-optical tensor 215
- symmetry boundary conditions 46
- T** thermal expansion 64
- TM waves model 98
- Touchstone file, exporting to 44
- transverse electric waves 81, 86, 125
- transverse electromagnetic fields 46
- typographical conventions 5
- W** waveguide
  - 3D 85
  - H bend 32, 80
  - microwave 85
  - optical 174, 208
  - photonic 215
  - weak form 234
- Y** YAG laser 249, 261

