

RF MODULE

MODEL LIBRARY

VERSION 3.5a

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@comsol.nl
www.comsol.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

COMSOL Multiphysics GmbH
Berliner Str. 4
D-37073 Göttingen
Phone: +49-551-99721-0
Fax: +49-551-99721-29
info@comsol.de
www.comsol.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 636020
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

Company home page
www.comsol.com

COMSOL user forums
www.comsol.com/support/forums

RF Module Model Library

© COPYRIGHT 1998–2008 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.

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

Version: November 2008 COMSOL 3.5a

Part number: CM021002

C O N T E N T S

Chapter 1: Introduction

Model Library Guide	2
Typographical Conventions	6

Chapter 2: Tutorial Models

Dielectric Scattering PML	8
Model Definition	8
Results and Discussion.	9
Modeling Using the Graphical User Interface	10
 Magnetic Frill	 14
Introduction	14
Model Definition	14
Results and Discussion.	15
Reference	15
Modeling Using the Graphical User Interface	15
 Microstrip on a Printed Circuit Board	 20
Model Definition	20
Results.	20
Modeling Using the Graphical User Interface	21
Solving with Lumped Port Termination.	28
 Waveguide Optimization	 31
Introduction	31
Model Definition	32
Results and Discussion.	33
Reference	34
Modeling Using the Graphical User Interface	35

Calculating the Schumann Resonance Frequencies	42
Introduction	42
Model Definition	42
Modeling in COMSOL Multiphysics	44
Results and Discussion.	45
Modeling Using the Graphical User Interface	46

Chapter 3: RF and Microwave Models

Three-Port Ferrite Circulator	50
Introduction	50
Model Definition	51
Results and Discussion.	53
References	55
Modeling Using the Graphical User Interface	56
Extracting the S-Parameter Matrix	63
Exporting S-Parameters vs. Frequency to Touchstone	64
 Monoconical RF Antenna	 65
Introduction	65
Model Definition	65
Results and Discussion.	66
Modeling Using the Graphical User Interface	70
Antenna Impedance and Radiation Pattern	75
Far-Field Computation.	77
Transient Analysis with External Circuit	78
 Magnetic Dipole Antenna	 80
Introduction	80
Model Definition	80
Results and Discussion.	81
Modeling Using the Graphical User Interface	81
 Optimizing the Shape of a Dipole Antenna	 86
Introduction	86
Model Definition	86

Results and Discussion.	87
Modeling Using the Graphical User Interface	88
Thermal Drift in a Microwave Filter	96
Introduction	96
Model Definition	96
Results and Discussion.	97
Modeling Using the Graphical User Interface	100
H-Bend Waveguide with S-parameters	111
Introduction	111
Model Definition	112
Results and Discussion.	112
Modeling Using the Graphical User Interface	113
Waveguide Adapter	116
Introduction	116
Model Definition	116
Results.	118
Modeling in COMSOL Multiphysics	121
Modeling Using the Graphical User Interface	122
Mode Analysis	122
S-Parameter Analysis	125
Creating the Geometry in MATLAB.	128
Microwave Cancer Therapy	129
Introduction	129
Model Definition	129
Results and Discussion.	133
Reference	136
Modeling in COMSOL Multiphysics	136
Modeling Using the Graphical User Interface	137
Absorbed Radiation (SAR) in the Human Brain	142
Introduction	142
Model Definition	142
Results and Discussion.	144
References	145

Modeling Using the Graphical User Interface	146
Microwave Oven	156
Introduction	156
Model Definition	156
Results.	157
Modeling in COMSOL Multiphysics	160
Modeling Using the Graphical User Interface	160
Electromagnetic Analysis	160
Microwave Filter on PCB	168
Introduction	168
Model Definition	168
Results and Discussion.	170
References	171
Modeling Using the Graphical User Interface	172
Mechanical Stress Analysis	178
Balanced Patch Antenna for 6 GHz	183
Introduction	183
Model Definition	184
Results and Discussion.	186
Modeling in COMSOL Multiphysics	188
Reference	189
Modeling Using the Graphical User Interface	189
Coaxial to Waveguide Coupling	204
Introduction	204
Results and Discussion.	204
Modeling Using the Graphical User Interface	206
Sea Bed Logging	211
Introduction	211
Model Definition	211
Results.	211
Modeling in COMSOL Multiphysics	213
Modeling Using the Graphical User Interface	213

Chapter 4: Optics and Photonics Models

Photonic Microprism	222
Introduction	222
Model Definition	223
Results and Discussion.	224
References	224
Modeling Using the Graphical User Interface	225
Photonic Crystal	230
Introduction	230
Model Definition	231
Results and Discussion.	231
References	232
Modeling Using the Graphical User Interface	232
Bandgap Analysis of a Photonic Crystal	239
Introduction	239
Model Definition	239
Results and Discussion.	242
References	244
Modeling Using the Graphical User Interface	245
Modeling Larger Sweeps With MATLAB	251
Step-Index Fiber	254
Introduction	254
Model Definition	255
Results and Discussion.	255
Reference	256
Modeling Using the Graphical User Interface	256
Stress-Optical Effects in a Silica-on-Silicon Waveguide	260
Introduction	260
The Stress-Optical Effect and Plane Strain	260
Perpendicular Hybrid-Mode Waves	261
Modeling Using the Graphical User Interface—Plane Strain Analysis	262
Modeling Using the Graphical User Interface—Optical Mode Analysis.	270

Convergence Analysis 273

Stress-Optical Effects with Generalized Plane Strain 277

Introduction 277

Generalized Plane Strain 277

Modeling Using the Graphical User Interface—Plane Strain Analysis . . 283

Modeling Using the Graphical User Interface—Optical Mode Analysis. . 291

Second Harmonic Generation of a Gaussian Beam 294

Introduction 294

Model Definition 294

Results and Discussion. 296

Reference 298

Modeling Using the Graphical User Interface 298

Propagation of a 3D Gaussian Beam Laser Pulse 307

Introduction 307

Model Definition 307

Results and Discussion. 308

Modeling Using the Graphical User Interface 309

INDEX 317

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 *RF Module User's Guide*, which introduces you to the basic functionality in the module, reviews new features in the version 3.5a 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 and Quick Reference* manual. An explanation on how to model with a programming language is available in yet another book, the *COMSOL Multiphysics MATLAB Interface Guide*.

The book you are reading, 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.

Note: The full documentation set is available in electronic formats—PDF and HTML—through the COMSOL Help Desk after installation.

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 combined solution time for all solution steps. Additional columns point out the modeling features that a given example

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

TABLE 1-1: RF MODULE MODEL LIBRARY

MODEL	PAGE	APPLICATION MODE	SOLUTION TIME	STATIONARY	TIME-HARMONIC	TIME DEPENDENT	EIGENFREQUENCY/EIGENMODE	SENSITIVITY/OPTIMIZATION	MULTIPHYSICS	PARAMETRIC STUDY
TUTORIAL MODELS										
Periodic Boundary Conditions	37*	3D Electromagnetic Waves	30 s				√			
Far-Field Postprocessing	51*	Axisymmetric TE Waves	1 s		√					
Lossy Circular Waveguide	104*	Perpendicular Hybrid-Mode Waves	2 s				√			
Dielectric Scattering PML	8	In-Plane TE Waves	2 s		√					
Magnetic Frill	14	3D Electromagnetic Waves	31 s		√					
Microstrip on a Printed Circuit Board	20	3D Electromagnetic Waves, Boundary Mode Analysis, Hybrid-Mode Waves	2 min		√		√			
Waveguide Optimization ^α	31	In-Plane TE Waves, Sensitivity, Optimization	3 min		√			√		√
Schumann Resonances	42	Electromagnetic Waves	2 min				√			
RF AND MICROWAVE MODELS										
H-Bend Waveguide	21*	3D Electromagnetic Waves, In-Plane TE Waves	7 s		√					
Three-Port Ferrite Circulator	50	In-Plane TE Waves	60 s		√					

TABLE 1-1: RF MODULE MODEL LIBRARY

MODEL	PAGE	APPLICATION MODE	SOLUTION TIME	STATIONARY	TIME-HARMONIC	TIME DEPENDENT	EIGENFREQUENCY/EIGENMODE	SENSITIVITY/OPTIMIZATION	MULTIPHYSICS	PARAMETRIC STUDY
Monoconical RF Antenna	65	Axisymmetric TM Waves	50 s		√					√
—with External Circuit			30 s			√				
Magnetic Dipole Antenna	80	Axisymmetric TE Waves	1 s		√					
Optimizing the Shape of a Dipole Antenna ²¹	86	Axisymmetric TM Waves, Optimization, Parameterized Geometry	4 min		√			√	√	
Thermal Drift in a Microwave Filter	96	3D Electromagnetic Waves, Moving Mesh, Shell	2 min	√			√		√	√
H-Bend Waveguide with S-parameters	111	In-Plane TE Waves	21 s		√					√
Waveguide Adapter	116	Boundary Mode Analysis, 3D Electromagnetic Waves	4 min		√		√		√	√
Microwave Cancer Therapy	129	Axisymmetric TM Waves, Bioheat Equation	17 s		√				√	
Absorbed Radiation (SAR) in the Human Brain	142	3D Electromagnetic Waves, Bioheat Equation	11 min	√					√	
Microwave Oven	156	3D Electromagnetic Waves, General Heat Transfer	3 min		√	√			√	
Microwave Filter on PCB	168	3D Electromagnetic Waves, Solid, Stress-Strain, Moving Mesh	83 min		√				√	√

TABLE I-I: RF MODULE MODEL LIBRARY

MODEL	PAGE	APPLICATION MODE	SOLUTION TIME	STATIONARY	TIME-HARMONIC	TIME DEPENDENT	EIGENFREQUENCY/EIGENMODE	SENSITIVITY/OPTIMIZATION	MULTIPHYSICS	PARAMETRIC STUDY
Balanced Patch Antenna for 6 GHz [†]	183	3D Electromagnetic Waves	91 min		√					√
Coaxial to Waveguide Coupling	204	3D Electromagnetic Waves	8 min		√					
Sea Bed Logging	211	3D Electromagnetic Waves	2 min		√					
OPTICS AND PHOTONICS MODELS										
Photonic Microprism	222	In-Plane TE Waves	2 s		√					
Photonic Crystal	230	In-Plane TE Waves	4 s		√					
Bandgap Analysis of a Photonic Crystal	239	In-Plane TE Waves	2 min				√			√
Step-Index Fiber	254	Perpendicular Hybrid-Mode Waves	7 s				√			
Stress-Optical Effects in a Silica-on-Silicon Waveguide	260	Plane Strain, Perpendicular Hybrid-Mode Waves	3 s	√		√	√		√	
Stress-Optical Effects with Generalized Plane Strain	277	Plane Strain, Perpendicular Hybrid-Mode Waves, Weak Form, Point	6 s	√			√		√	
Second Harmonic Generation of a Gaussian Beam	294	In-Plane TE Waves, ODE	3 min			√				
Propagation of a 3D Gaussian Beam Laser Pulse	307	3D Electromagnetic Waves	17 s			√				

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

□ This model requires the Optimization Lab.

† 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 code.
- 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 45 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.

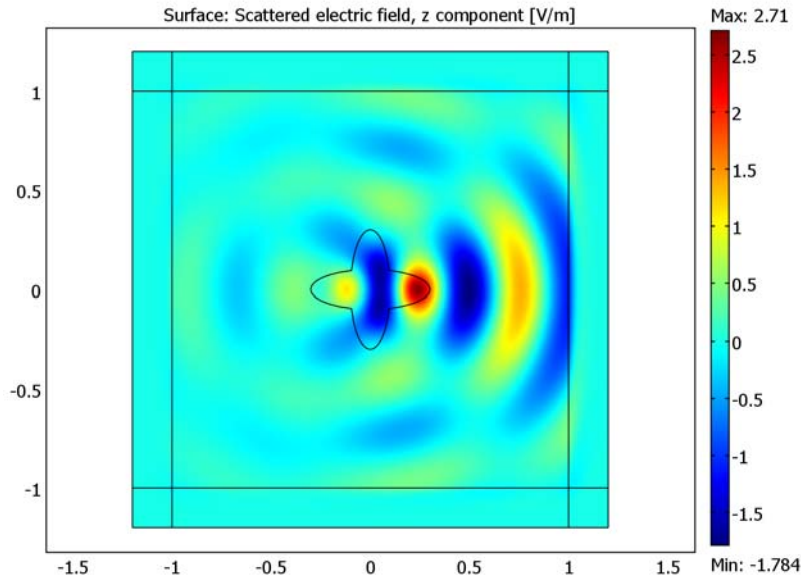


Figure 2-1: The figure shows the scattered field from the dielectric object.

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.

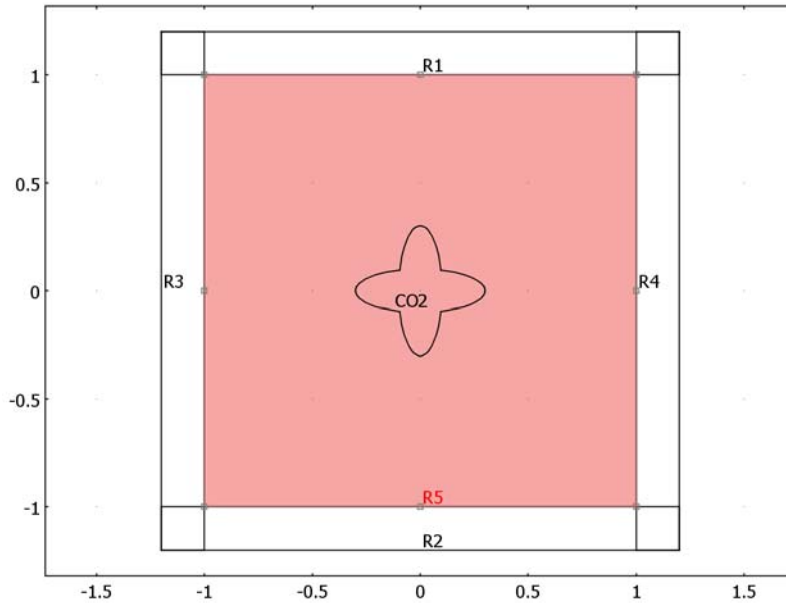
PROPERTY	R1	R2	R3	R4
Width	2.4	2.4	0.2	0.2
Height	0.2	0.2	2.4	2.4
Position corner x	-1.2	-1.2	-1.2	1
Position corner y	1	-1.2	-1.2	-1.2

- 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**.

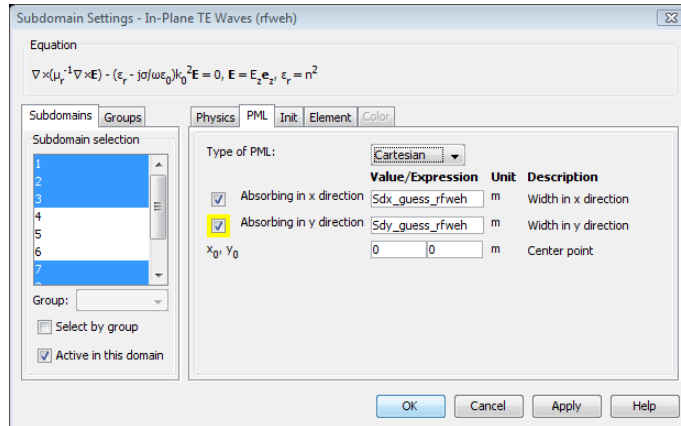
Application Scalar Variables

- 1 From the **Physics** menu, choose **Scalar Variables** to open the **Application Scalar Variables** dialog box.
- 2 Type **0.5** in the edit field corresponding to the variable `lambda0_rfweh`.
- 3 Click **OK**.

Subdomain Settings

- 1 From the **Physics** menu, 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 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 0.05.
- 3 Click **OK**.
- 4 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, which shows the field in the physical domains and the PML domains. 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** page, select **Scattered electric field, z component** from the **Predefined quantities** list.
- 5** Click **OK**.

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 of defining 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 A/m^2 , the magnetic current density has the unit V/m^2 .

A closed loop of magnetic current therefore has the unit 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 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.

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

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, λ .

WAVELENGTH	LENGTH	RADIUS	IMPEDANCE
0.3	0.47λ	0.005λ	

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

Reference

1. C.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. It is sufficient to include only one quarter of the antenna with surroundings.

- 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 Shift-click the **Rectangle/Square** button on the Draw toolbar to open the **Rectangle** dialog box. Specify the rectangle R1 according to the table below, then click **OK**.

NAME	WIDTH	HEIGHT	BASE	X	Y
R1	0.5	0.6	Corner	0	-0.3
R2	0.005*0.3	0.47*0.3	Corner	0	-0.47*0.15

- 4 Repeat the previous step for the rectangle R2.
- 5 Select both rectangles and click the **Difference** button on the Draw toolbar.
- 6 Shift-click the **Point** button on the Draw toolbar. In the **Point** dialog box, specify the point PT1 according to the table below, then click **OK**.

NAME	X	Y
PT1	0.005*0.3	0
PT2	0.01	0

- 7 Repeat the previous step for the point PT2.
- 8 Press Ctrl+A to select all objects, then choose **Revolve** from the **Draw** menu.
- 9 Enter -90 in the $\alpha 2$ edit field, then click **OK**.

This completes the geometry modeling. You created the two loops 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 14 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; when done, click **OK**.

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 From the **Mesh** menu, open the **Free Mesh Parameters** dialog box.
- 2 From the **Predefined mesh sizes** list, select **Coarser**.

- 3 Click the **Custom mesh size** option button and type 0.07 in the **Maximum element size** edit field.
- 4 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.
- 5 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.
- 6 Click the **Remesh** button and then click **OK**.

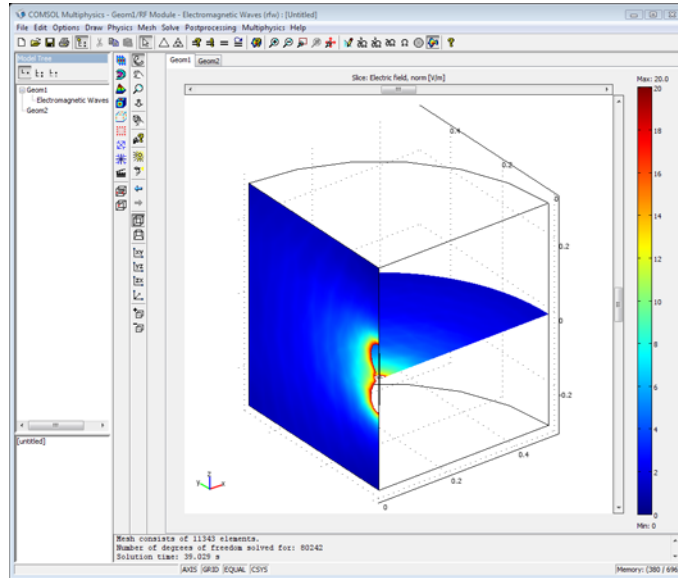
COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar. The solver takes a few minutes to complete the analysis.

POSTPROCESSING AND VISUALIZATION

- 1 From the **Postprocessing** menu, open the **Plot Parameters** dialog box.
- 2 On the **Slice** page, 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

Model Definition

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
```

Results

The analysis tests two terminations: lumped port termination and numerical port termination. Both versions use a lumped port with 50 Ω 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 gives 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 Ω
Numerical port termination	36.44 Ω
Lumped port termination	39.91 Ω

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

Modeling Using the Graphical User Interface

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 ϵ_r , μ_r , and σ** option button, and then enter 3 in the ϵ_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).

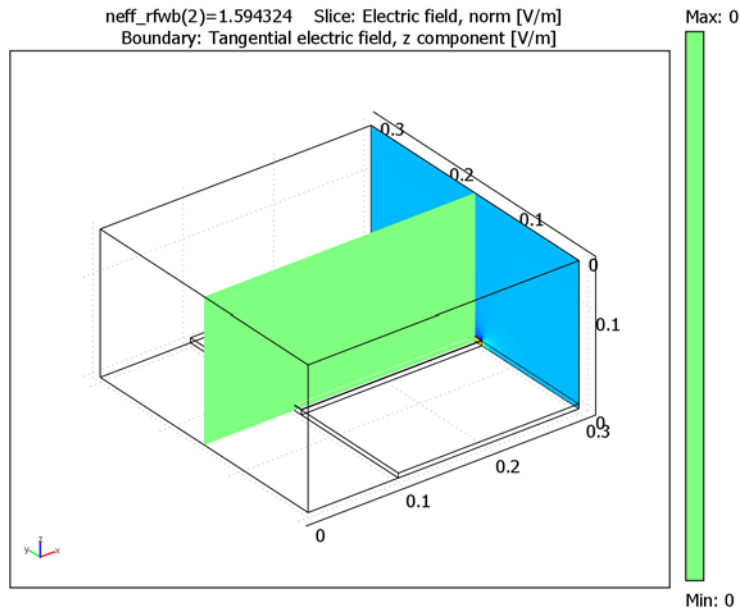
- 1 From the **Solve** menu, open the **Solver Manager** dialog box.
- 2 Click the **Solve For** tab, and select **Boundary Mode Analysis, Hybrid-Mode Waves (rfwb)** from the **Solve for variables** list.
- 3 Click **OK**.

- 4 From the **Solve** menu, open the **Solver Parameters** dialog box.
- 5 From the **Solver** list, choose **Eigenvalue**.
- 6 Enter 1.7 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

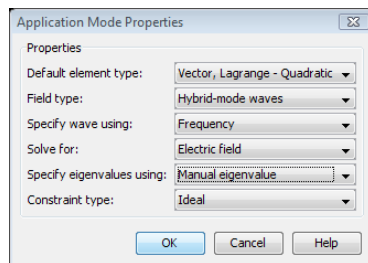
- 1 From the **Postprocessing** menu, open the **Plot Parameters** dialog box.
- 2 Click the **Boundary** tab and select the **Boundary plot** check box. From the **Predefined quantities** list, select **Boundary Mode Analysis, Hybrid-Mode Waves (rfwb)>Tangential electric field, z component**.
- 3 Clear the **Color legend** 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 in the **Solution to use** area, select the second solution from the **Effective mode index** list (with an effective mode index around 1.59).
- 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**.

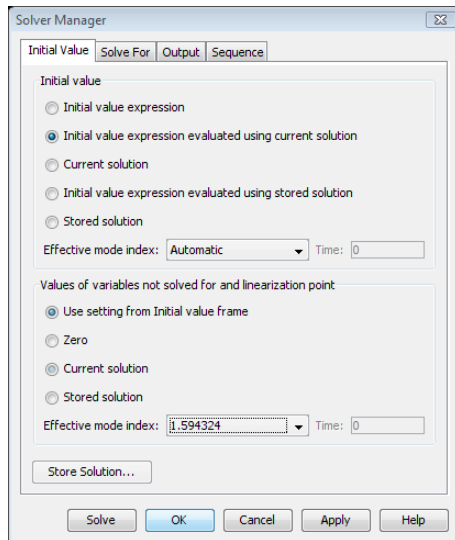


- 3 Click **OK**.

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

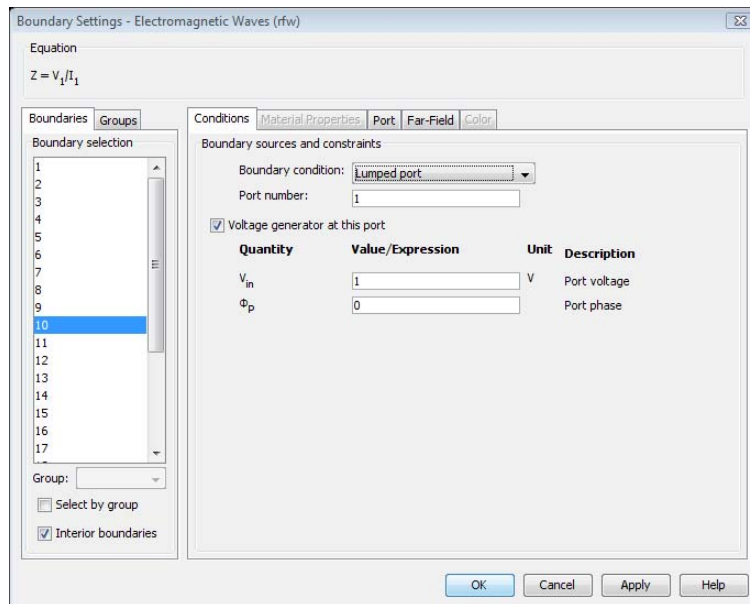
- 1 Open the **Subdomain Settings** dialog box.
- 2 Select Subdomains 2, 3, and 4, then enter 3 in the ϵ_r edit field.

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.

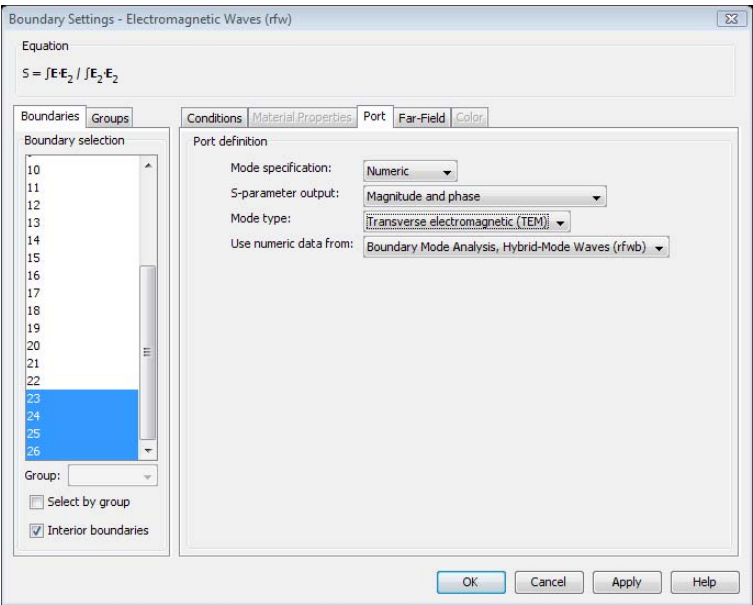
SETTINGS	BOUNDARY 10	BOUNDARIES 23–26	BOUNDARIES 2, 4, 5, 7, 22
Boundary condition	Lumped port	Port	Perfect magnetic conductor
Port number	1	2	
Voltage generator at this port	Selected		
Wave excitation at this port		Cleared	



- 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)

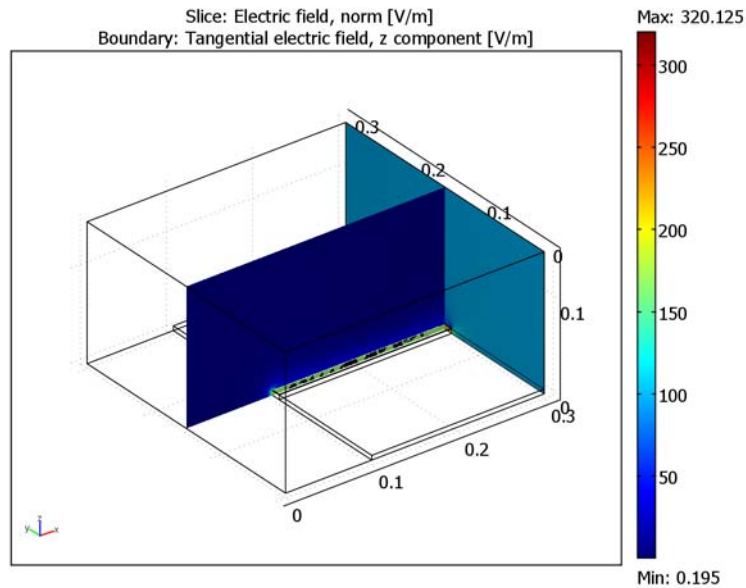


6 Click **OK**.

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

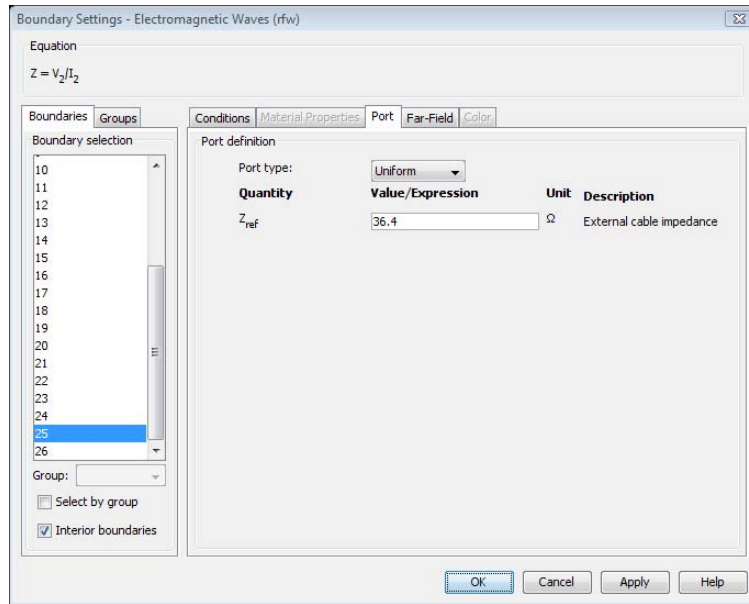
- 1 From the **Postprocessing** menu, choose **Data Display>Global**.
- 2 In the dialog box that appears, choose **Port impedance, port 1** from the **Predefined quantities** list.
- 3 Click **OK**. The result shows up in the message log and is close to 36.44 Ω .

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**.

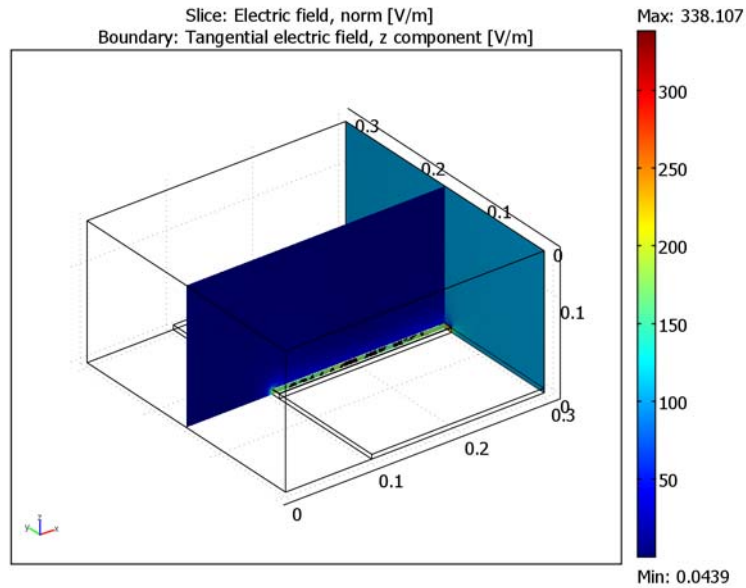
COMPUTING THE SOLUTION

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 $39.91\ \Omega$. 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.



Waveguide Optimization

Introduction

The optimization of components with respect to geometrical design parameters is a common task in microwave engineering. In this model the reflection coefficient of a waveguide structure is minimized, and the sensitivities of the real and imaginary parts of the reflection coefficient are extracted using the parameterized geometry and adjoint sensitivity features of COMSOL Multiphysics. The model was inspired by a conference paper by Li and Nikolova (Ref. 1) and comparisons are made to that paper. The example geometry is a waveguide 90-degree corner bend with the outer corner replaced by a metallic wall section at 45 degrees angle to the connecting straight waveguide sections.

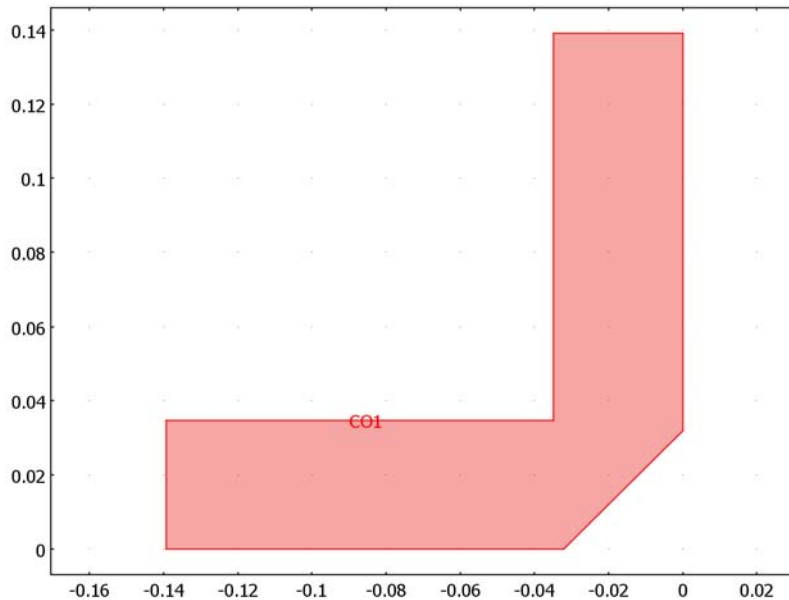


Figure 2-2: 2D waveguide geometry with a 45 degree cut at the outer corner.

For a general introduction to the modeling of rectangular waveguide structures, see the H-Bend Waveguide model on page 111 in the *RF Module User's Guide*. Optimizing the waveguide bend involves calculating how well a TE_{10} wave propagates between ports in the waveguide structure for different positions of the 45 degree wall.

This is done by calculating the scattering parameters, or S-parameters, of the structure as a function of the wall position. The S-parameters are a measure of the transmittance and reflectance of the bend. For a theoretical background on S-parameters, see the section “S-Parameters and Ports” on page 57 in the *RF Module User’s Guide*.

This model only includes the TE₁₀ mode of the waveguide. Thus the model can be made in 2D as the fields of the TE₁₀ mode have no variation in the transverse direction. Figure 2-2 shows the 2D geometry.

Model Definition

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 μ_r denotes the relative permeability, ω the angular frequency, σ the conductivity, ϵ_0 the permittivity of vacuum, ϵ_r the relative permittivity, and k_0 the free space wave number. Losses are neglected so the conductivity is zero everywhere.

BOUNDARY CONDITIONS

The waveguide walls are assumed to be perfect, lossless metallic walls which is the default boundary condition. The two ends of the waveguide are assigned as ports, where the inport is the left-most boundary, and the top boundary is the outport.

Results and Discussion

Figure 2-3 shows the electric field norm for the optimal position of the 45-degree wall. It is flat along the direction of propagation which indicates that there is no reflected wave within the structure. By inspecting S_{11} (-75 dB), this is confirmed.

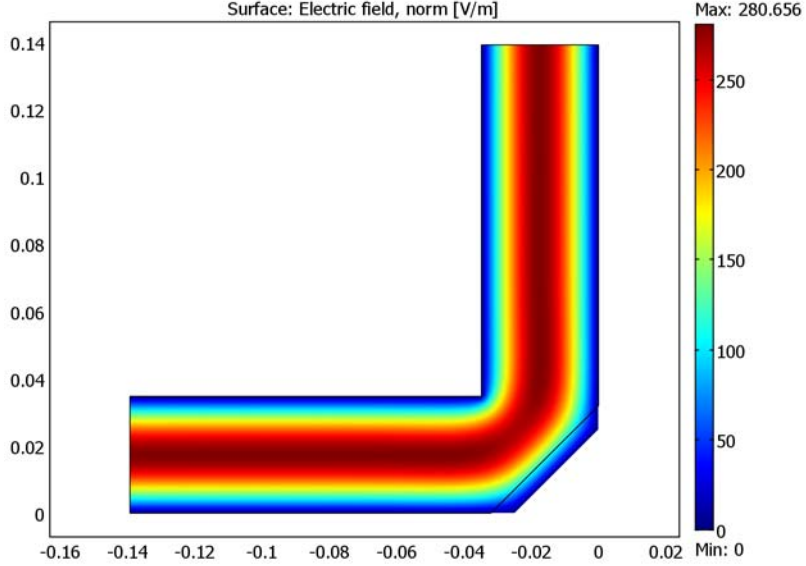


Figure 2-3: Electric field norm plot for the optimized position of the 45-degree wall.

In the Ref. 1, no optimization is performed but the sensitivities for the S-parameters with respect to the position of the 45-degree wall, calculated using the self-adjoint method as well as using a finite difference approximation, are presented. The corresponding sensitivities for the real and imaginary parts of S_{11} calculated by this

model agree very well with those found by Li and Nikolova in Ref. 1. Figure 2-4 shows the computed sensitivity curves for this model.

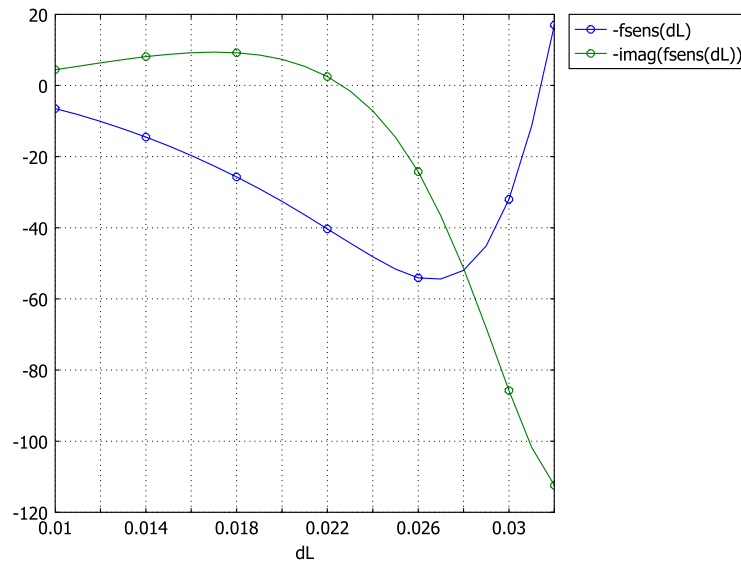


Figure 2-4: Sensitivity curves for real and imaginary parts of S_{11} with respect to the 45-degree wall position.

It should be noted that the paper is not 100% clear about the sign convention for the position parameter (called dL here) so we used the sign that gives agreement. It is trivial to perform the sensitivity analysis also for S_{21} , but in that case the results do not agree with those in the paper. One possible explanation is that a different phase reference may have been used for S_{21} in the paper.

Reference

1. D. Li and N.K. Nikolova, “S-parameter sensitivity analysis of waveguide structures with FEMLAB,” *Proc. COMSOL User’s Conference 2005, Boston*.

Model Library path: RF_Module/Tutorial_Models/waveguide_optimization

MODEL NAVIGATOR

- 1** Start COMSOL Multiphysics.
- 2** In the **Model Navigator**, select **2D** from the **Space dimension** list.
- 3** Click the **Multiphysics** button.
- 4** In the **Model Navigator** open the **COMSOL Multiphysics** folder, then select **Deformed Mesh>Parameterized Geometry**.
- 5** Click the **Add** button.
- 6** In the **Model Navigator** open the **RF Module** folder, then select **In-Plane Waves>TE Waves>Harmonic propagation**.
- 7** Click the **Add** button.
- 8** In the **Model Navigator** open the **COMSOL Multiphysics** folder, then select **Optimization and Sensitivity>Optimization**.
- 9** Click the **Add** button.
- 10** Click **OK**.

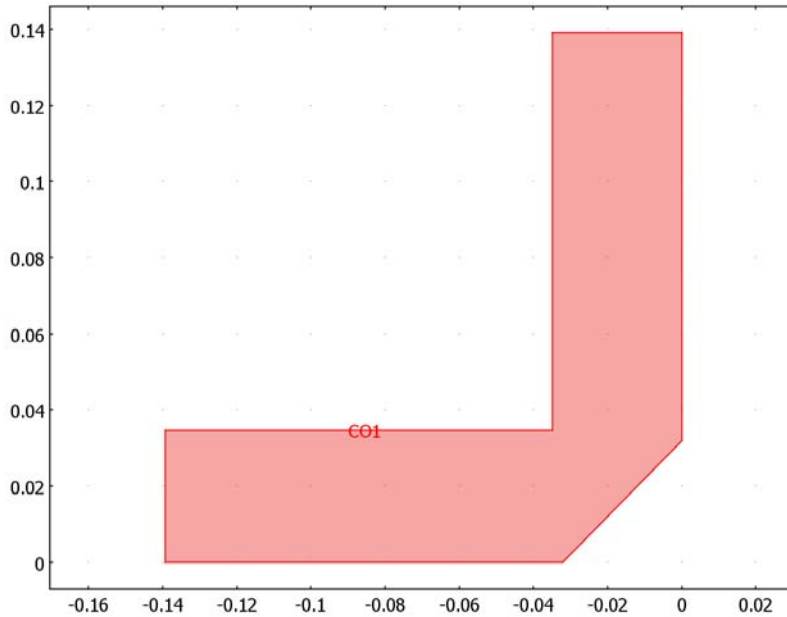
GEOMETRY MODELING

- 1** Go to the **Draw** menu and select **Specify Objects > Line**.
- 2** Under **Style**, select **Closed polyline (solid)**.
- 3** In the **X** and **Y** edit fields, enter the following coordinate values:

PROPERTY	EXPRESSION
X	-0.1392 -0.0348 -0.0348 0 0 -0.032 -0.1392
Y	0.0348 0.0348 0.1392 0.1392 0.032 0 0

- 4** Click **OK**.

- 5 Click the **Zoom Extents** button on the Main toolbar.



PHYSICS SETTINGS—PARAMETERIZED GEOMETRY

Application Mode Properties

- 1 From the **Multiphysics** menu, select the **Parameterized geometry (pg)** application mode.
- 2 Choose **Properties** from the **Physics** menu.
- 3 In the **Application Mode Properties** dialog box, change **Weak constraints** to **On**.
- 4 Click **OK**.

Subdomain Settings

Use the default subdomain settings (**Free displacement**).

Boundary Conditions

Here you describe how the boundaries are going to behave when changing the geometry. The geometry change itself is driven by the point settings specified in the next section.

- 1 From the **Physics** menu, choose **Boundary Settings**.
- 2 Select Boundary 2 and set the **Condition on X displacement** to **Free**.

- 3 Select Boundary 7 and set the **Condition on Y displacement** to **Free**.
- 4 Select Boundary 6 and select **Similarity transformation**.
- 5 Click **OK**.

Point Settings

The point settings are used to specify the position of the 45-degree wall. The parameter dL is the distance from the outer corner, so that $dL = 0$ corresponds to a 90-degree corner.

- 1 From the **Physics** menu, choose **Point Settings**.
- 2 Select Point 5, check **X-displacement**, and enter $-(dL - 0.032)$.
- 3 Select Point 6, check **Y-displacement**, and enter $(dL - 0.032)$.
- 4 Click **OK**.

PHYSICS SETTINGS—IN-PLANE TE WAVES

Scalar Variables

- 1 From the **Multiphysics** menu, select the **In-Plane TE Waves (rfwe)** application mode.
- 2 From the **Physics** menu, choose **Scalar Variables**.
- 3 In the **Application Scalar Variables** dialog box, set the frequency ν_{rfwe} to $5.16e9$, and then click **OK**.

Boundary Conditions

- 1 From the **Physics** menu, choose **Boundary Settings**.
- 2 Select Boundaries 2–4 and 6–7.
- 3 In the **Boundary condition** list, keep the default **Perfect electric conductor** as the boundary condition. These boundaries represent the inside of the walls of the waveguide which is plated with a metal, such as silver, and considered to be a perfect conductor.
- 4 On Boundaries 1 and 5, specify the **Port** boundary condition. On the **Port** page, set the values according to the following table; when done, click **OK**.

SETTINGS	BOUNDARY 1	BOUNDARY 5
Port number	1	2
Wave excitation at this port	Selected	Cleared
Mode specification	Analytic	Analytic
Mode number	1	1

Subdomain Settings

Use the default values for ϵ_r , μ_r , and σ , because the waveguide is filled with air.

PHYSICS SETTINGS—OPTIMIZATION

Scalar Settings

- 1 From the **Multiphysics** menu, select the **Optimization (opt)** application mode.
- 2 Choose **Scalar Settings** from the **Physics** menu.
- 3 In the **Scalar Settings** dialog box, on the **Objective** page, enter S11_rfwe for **Qs**.
- 4 On the **Scalar Constraints** page, enter dL as **Expression** and set upper and lower bounds (**ub** and **lb**) to 0.01 and 0.03 respectively.
- 5 On the **Variables** page, enter dL for **Variable** and 0.014 for **Init**.
- 6 Click **OK**.

MESH GENERATION

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

- 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 3e-3.
- 3 Click **OK**.
- 4 Initialize the mesh by clicking the **Initialize Mesh** button on the Main toolbar.

COMPUTING THE SOLUTION—SENSITIVITY

- 1 In the **Solver Parameters** dialog box, select the **Optimization/Sensitivity** check box.
- 2 Set the **Linear system solver** to **Direct (PARDISO)**.
- 3 In the **Solver Parameters** dialog box, select the **Parametric** solver.
- 4 Enter dL as **Parameter names** and range(0.01,0.001,0.032) as **Parameter values**.
This instructs the solver to calculate the solution at a number of equidistant wall positions in the range 0.01 m to 0.032 m from the outer corner.
- 5 Click the **Optimization/Sensitivity** tab.
- 6 Set the **Analysis** to **Sensitivity**.
- 7 Click the **Advanced** tab.
- 8 Select the **Use complex functions with real input** box.
- 9 Click **OK**.
- 10 Click the **Solve** button on the Main toolbar to compute the solution.

POSTPROCESSING AND VISUALIZATION

Follow these steps to visualize the electric field norm as in Figure 2-5 below.

- 1 Open **Plot Parameters**.
- 2 On the **Surface** page, select **Electric field, norm** from the **Predefined quantities** list.
- 3 Click **OK**.

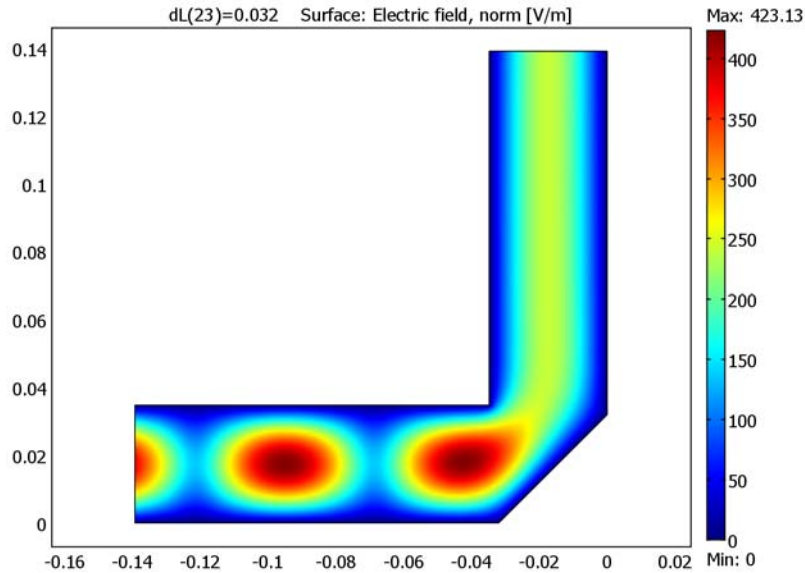
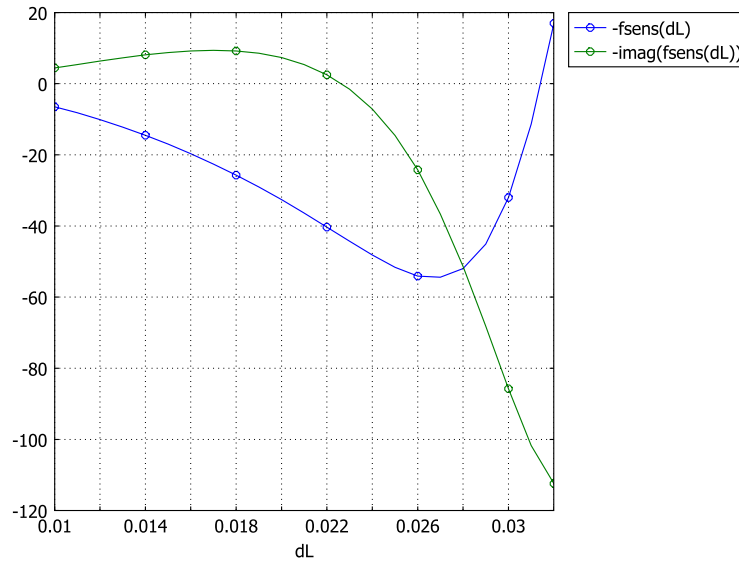


Figure 2-5: Electric field norm plot for the final position of the 45-degree wall.

You can plot the S-parameter sensitivity using a global variables plot.

- 1 From the **Postprocessing** menu, choose **Global Variables Plot**.
- 2 In the **Global Variables Plot** dialog box, select all solutions in the **Solutions to use** list.
- 3 Type `-fsens(dL)` in the **Expression** field, then click the **Add Entered Expression** button (**>**) to add it to the **Quantities to plot** list.
- 4 Repeat the previous step for the expression `-imag(fsens(dL))`.
- 5 Click the **Line Settings** button.
- 6 In the **Line Settings** dialog box, select **Circle** as **Line marker** and select the **Legend** box.
- 7 Click **OK** to close the **Line Settings** dialog box.

8 Click **OK** in the **Global Variables Plot** dialog box.



COMPUTING THE SOLUTION—OPTIMIZATION

Scalar Settings

- 1 From the **Multiphysics** menu, select the **Optimization (opt)** application mode.
- 2 From the **Physics** menu, select **Scalar Settings**.
- 3 On the **Objective** page, set Q_s to `realdot(S11_rfwe,S11_rfwe)`.
- 4 Click **OK**.

Solver Settings

- 1 In the **Solver Parameters** dialog box, select the **Stationary** solver on the **General** page.
- 2 Click the **Optimization/Sensitivity** tab.
- 3 Set the **Analysis** to **Optimization**.
- 4 Click **OK**.
- 5 Click the **Solve** button on the Main toolbar to start the optimization.

The solution takes a few minutes to compute on a standard PC.

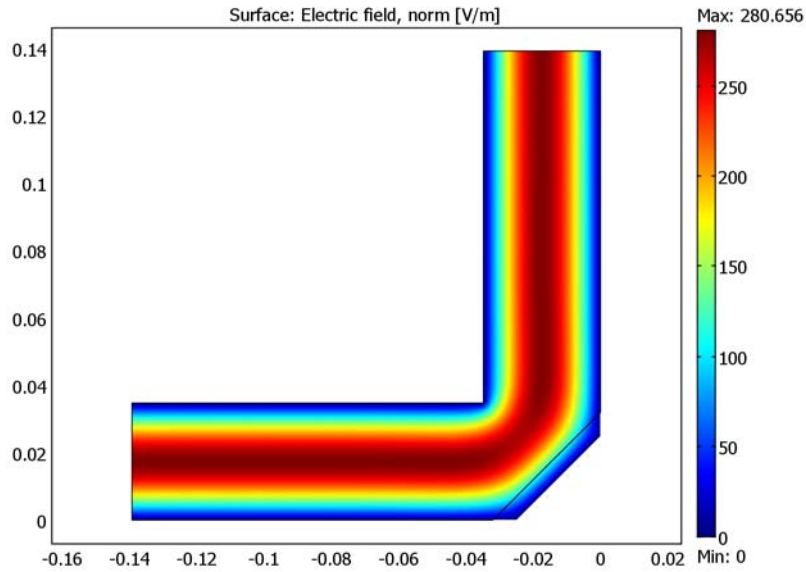


Figure 2-6: Electric field norm plot for the optimal position of the 45-degree wall.

POSTPROCESSING AND VISUALIZATION

Follow these steps to display the optimal value for dL and the corresponding value for S_{11} .

- 1 From the **Postprocessing** menu, select **Data Display>Global**.
- 2 In the **Global Data Display** dialog box, select **Optimization (opt)> dL** in the **Predefined quantities** list.
- 3 Click **Apply** to display the optimal value of dL in the message log at the bottom of the main user interface. It evaluates to about 0.024995 m.
- 4 In the **Predefined quantities** list, select **In-Plane TE Waves (rfwe)>S-parameter dB (S_{11})**.
- 5 Click **OK** to display the optimal value of S_{11} in the log field at the bottom of the main window. It evaluates to about -100 dB.

Calculating the Schumann Resonance Frequencies

Introduction

The Schumann resonance frequencies are a set of eigenmodes in the extremely low frequency (ELF) portion of the electromagnetic field spectrum. These frequencies are only present for planetary-like bodies with a substantial ionosphere acting as a perfect electric conductor, thus forming a resonance cavity with the surface. In this example, the Schumann resonance frequencies for the Earth are calculated.

Model Definition

The Schumann resonance frequencies are extensively studied due to the fact that they provide information about the planet. Intense lightning activity can, for example, be predicted since this is a source for the excitation of electromagnetic waves in the ELF range (3 Hz–30 Hz).

The model geometry is depicted in Figure 2-7, where the brown area in the center, representing the Earth interior, is included only for illustration purposes.

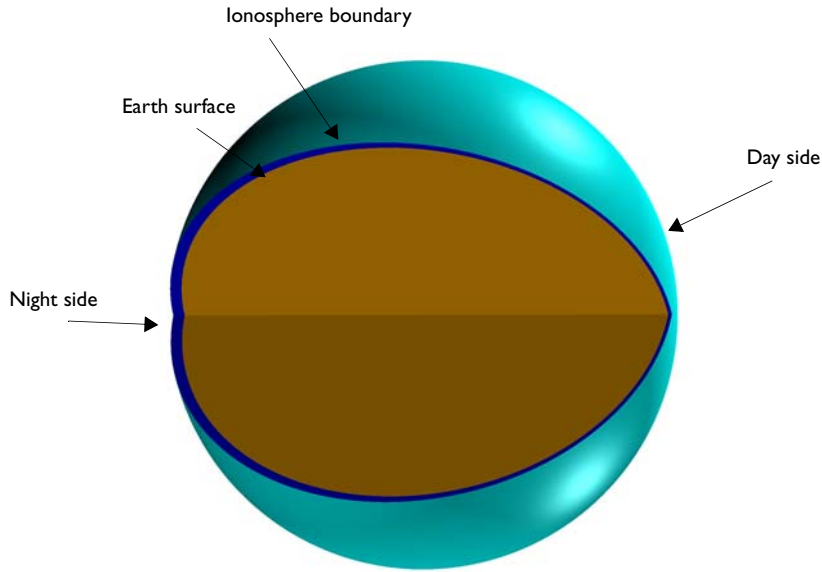


Figure 2-7: The model geometry is in blue. The brown area representing the Earth is included only for illustration. The cavity thickness is 300 km on the night side and 100 km on the day side.

The Earth's ionosphere is composed of different layers that are rather characterized by local maxima in electron density than distinct layers. The most important layers ordered from the inner are referred to as the D, E, F1, and F2 layers. The height of the different layers depends on ionization, which is due to sun radiation, while at night time some of the layers are reduced or even disappear. On the day side of the Earth, the height of the cavity between the inner layer of the ionosphere and the Earth is approximately 100 km, while on the night side, the height is about 300 km. Most of the radiation that ionizes the upper layers is absorbed so that below the D layer, negligible ionization occurs. The conductivity therefore decreases rapidly toward zero.

The conductivity profile in the cavity between the Earth surface and the innermost layer of the ionosphere varies over time with several different parameters. Furthermore the profile is not linear. This model, however, assumes a fixed value at the upper boundary of the cavity that decreases linearly to zero at the Earth's surface.

The conductivity of the Earth's surface is of the magnitude of 0.01 S/m. The skin depth, δ , is approximately:

$$\delta = \sqrt{\frac{2}{\mu_0 \omega \sigma}}$$

For $\omega = 40$ Hz, $\delta = 2$ km, which is 50 times less than the height of the cavity. The Earth surface can therefore be approximated as a perfect electric conductor:

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

Inside the cavity, the electric field \mathbf{E} is described by the following equation:

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

The Schumann resonance frequencies for the Earth are nowadays measured at several stations around the globe. The eigenmodes vary due to several different factors, such as changes in the global climate and variations in the solar wind which affect the height and conductivity profile of the cavity.

Modeling in COMSOL Multiphysics

In this model, the varying height of the cavity depending on day and night sides of the Earth is modeled geometrically with two concentric spheres, the outer sphere having a different center point than the inner. Let \mathbf{x} be a point inside the cavity, r the radial distance to the Earth's center from \mathbf{x} , R_E the Earth radius, σ_{ion} the conductivity at the ionosphere boundary, and d the distance to the ionosphere boundary. The conductivity profile is approximated with the following linear expression:

$$\sigma = \frac{\sigma_{\text{ion}}(r - R_E)}{r + d} \quad (2-1)$$

In COMSOL Multiphysics, the distance, d , to the boundary is available by defining a boundary distance variable.

Table 2-2 presents the data used in the model.

TABLE 2-2: MODEL DATA

PROPERTY	VALUE
Earth radius	6371 km
Ionosphere height	100 km–300 km
Conductivity at inner D layer, σ_{ion}	$1 \cdot 10^{-8}$ S/m

Results and Discussion

The model predicts the following three first eigenfrequencies: 7.5 Hz, 8.13 Hz, and 24.4 Hz, which are in the range of experimental values found in the literature (experimental values often include 14 Hz and 20 Hz as well).

The following plot shows the norm of the electric field at the fundamental tone.

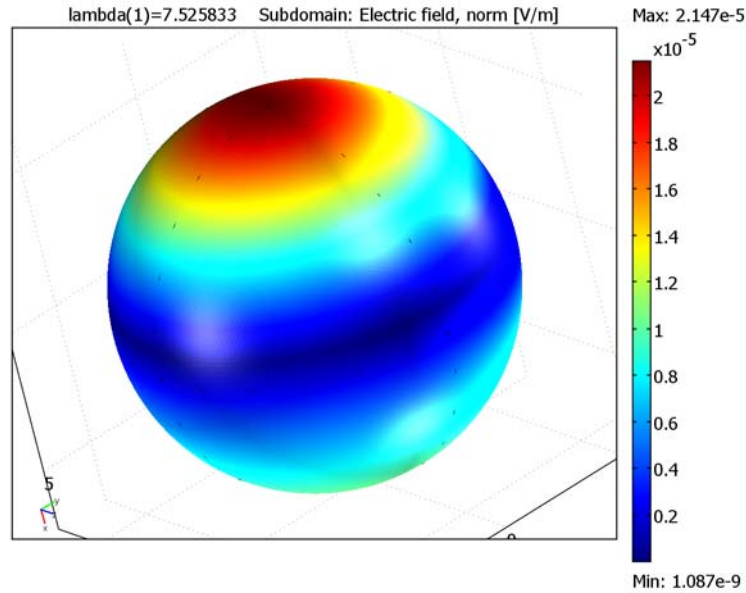


Figure 2-8: The norm of the electric field for the fundamental tone.

Figure 2-9 shows the norm of the electric field at the eigenfrequency of 24 Hz.

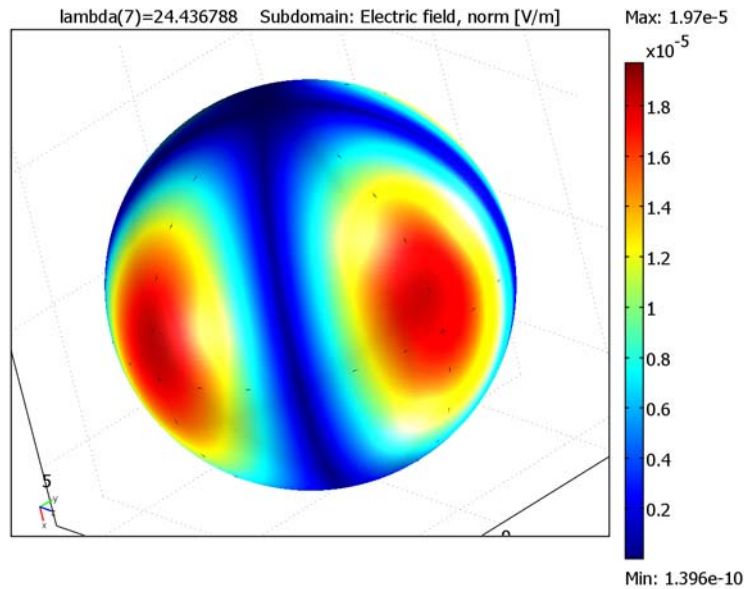


Figure 2-9: Norm of the electric field at the eigenfrequency of 24 Hz.

Model Library path: RF_Module/Tutorial_Models/schumann_resonance

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

- 1 In the **Model Navigator**, begin by selecting **3D** from the **Space dimension** list.
- 2 Select **RF Module>Electromagnetic Waves>Eigenfrequency analysis**, then click **OK**.

GEOMETRY MODELING

- 1 Draw a sphere by clicking on the **Sphere** button on the Draw toolbar.
- 2 Specify the **Radius** as 6371000, then click **OK**.
- 3 Draw another sphere, but this time enter 6371000+2e5 in the **Radius** edit field and 1e5 for the **x-coordinate** in the **Axis base point** area.

4 Select both spheres and click the **Difference** button on the Draw toolbar.

OPTIONS AND SETTINGS

1 Select **Physics>Properties**.

2 From the **Specify eigenvalues using** list, select **Eigenvalue**.

3 Set the **Divergence condition** to **On**.

4 Click **OK**.

5 Select **Options>Constants**.

6 Enter **Constants** according to the following table (the descriptions are optional); when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
sig_ion	1e-8[S/m]	Conductivity at ionosphere boundary
R_earth	6371[km]	Earth radius

7 Select **Options>Boundary Distance Variables**.

8 Click the **New** button.

9 Select Boundaries 1–4 and 13–16.

10 Click **OK**.

11 Select **Options>Expressions>Scalar Expressions**.

12 Enter **Scalar Expressions** according to the following table (the descriptions are optional); when done click **OK**.

NAME	EXPRESSION	DESCRIPTION
r	$\sqrt{x^2+y^2+z^2}$	Distance from Earth center
sigma	$\text{sig_ion} * (r - R_earth) / (r + \text{dist1} - R_earth)$	Conductivity profile

PHYSICS SETTINGS

Subdomain Settings

1 From the **Physics** menu, open the **Subdomain Settings** dialog box.

2 Select Subdomain 1.

3 In the **Electric conductivity** edit field, type sigma.

4 Click **OK**.

The perfect electric conductor boundary condition is the default boundary condition, therefore no changes needs to be made for the this.

MESHING

- 1 From the **Mesh** menu, open the **Free Mesh Parameters** dialog box.
- 2 Click the **Boundary** tab.
- 3 Select all boundaries.
- 4 In the **Maximum element size** edit field, type $5e5$.
- 5 From the **Method** list, select **Triangle (advancing front)**.
- 6 Click the **Remesh** button. When the mesher has finished, click **OK**.

COMPUTING THE SOLUTION

- 1 Open the **Solver Parameters** dialog box.
- 2 In the **Desired number of eigenvalues** edit field, type 20.
- 3 Click **OK**.
- 4 Click the **Solve** button on the Main toolbar to solve the model.

POSTPROCESSING AND VISUALIZATION

- 1 To create the plot in Figure 2-8, begin by opening the **Plot Parameters** dialog box from the **Postprocessing** menu.
- 2 On the **General** page, clear the **Slice** check box and select the **Subdomain** check box.
- 3 Click the **Subdomain** tab.
- 4 From the **Predefined quantities** list, select **Electric field, norm**, then click **Apply**.
- 5 To reproduce the plot in Figure 2-9, click the **General** tab, select **24.426** from the **Eigenvalue** list, and click **OK**.

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° where a ferrite post is inserted at the center of the joint. Figure 3-1 shows the geometry of the circulator.

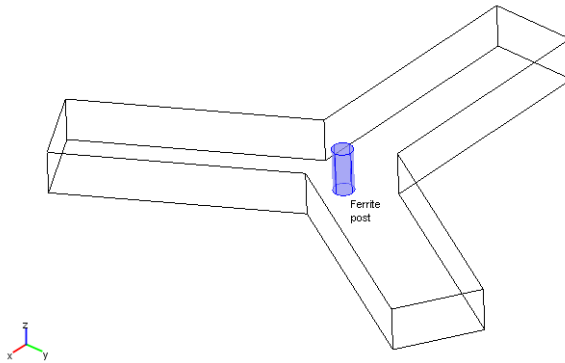


Figure 3-1: Geometry of the three-port microwave circulator.

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 H-Bend Waveguide model on page 21 in the *RF Module User's Guide*. 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 57 in the *RF Module User’s Guide*.

This model only includes the TE₁₀ mode of the waveguide. Thus the model can be made in 2D as the fields of the TE₁₀ mode have no variation in the transverse direction. Figure 3-2 shows the 2D geometry including the dielectric tuning elements.

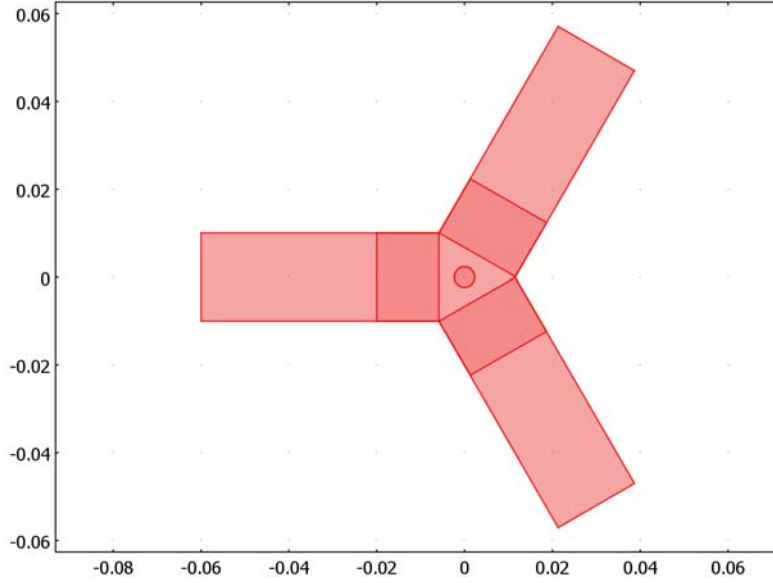


Figure 3-2: 2D geometry with dielectric tuning elements.

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 μ_r denotes the relative permeability, ω the angular frequency, σ the conductivity, ϵ_0 the permittivity of vacuum, ϵ_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 μ_0 denotes the permeability of free space; ω is the angular frequency of the microwave field; ω_0 is the precession frequency or Larmor frequency of a spinning electron in the applied magnetic bias field, H_0 ; ω_m is the electron Larmor frequency at the saturation magnetization of the ferrite, M_s ; and γ 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 TE₁₀ 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, ϵ_{r} , is shown in Figure 3-3. The parameter sweep also used adaptive meshing on the S_{11} parameter, yielding an optimized mesh for the extraction of this parameter. 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} .

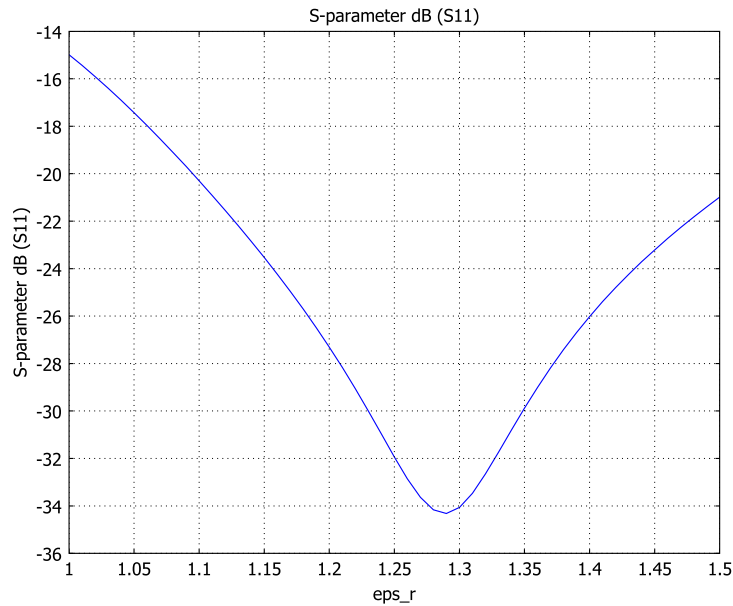
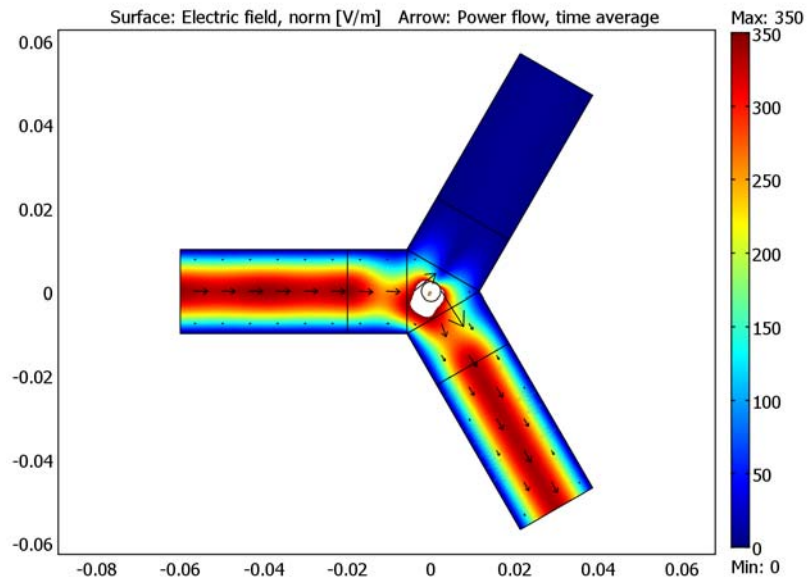


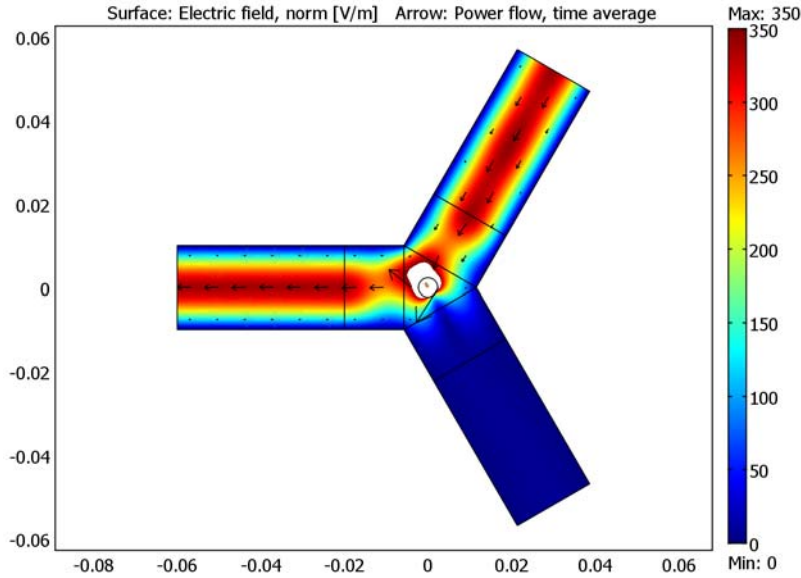
Figure 3-3: S_{11} parameter as a function of the relative permittivity.

By choosing ϵ_{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 ϵ_{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. When done, click **OK**.

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; when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
gamma	1.759e11[C/kg]	Gyromagnetic ratio
H0	omega_rfwe/(gamma*mu0_rfwe+1e4[m/C])	Applied magnetic bias field
w0	mu0_rfwe*gamma*H0	Larmor frequency
Ms	0.3[Wb/m^2]/mu0_rfwe	Saturation magnetization
wm	mu0_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 56. Select the **circulator.mphbin** file, and click **Import**.
- 4 Skip the section “Creating the Geometry from Scratch” and begin at the 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 by pressing Ctrl+C and then press Ctrl+V to paste it without any displacement by clicking **OK** in the **Paste** dialog box.
- 3 Click the **Rotate** button on the Draw toolbar and rotate the pasted rectangle 120° around $(0,0)$.
- 4 Repeat Step 2 to make a new copy of the original rectangle without any displacement.
- 5 Click the **Rotate** button on the Draw toolbar and rotate the pasted rectangle -120° around $(0, 0)$.
- 6 Select all objects and click the **Union** button on the Draw toolbar.
- 7 Click the **Delete Interior Boundaries** button on the Draw 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 Press the Shift key and click the **Ellipse/Circle (centered)** button on the Draw toolbar; then use the **Circle** dialog box to create 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.

- 1 From the **Physics** menu, choose **Boundary Settings**.
- 2 At Boundaries 1, 20, and 21, specify the **Port** boundary condition. On the **Conditions** and **Port** pages set the values according to the following table; when done, click **OK**.

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 From the **Physics** menu, choose **Subdomain Settings**.
- 2 In the **Subdomain Settings** dialog box, click the **Groups** tab and create three groups called air, dielectric, and ferrite, respectively, by clicking the **New** button and replacing **new group** in the **Name** edit field with the name of each group.
- 3 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: $\mu_r -i*k_r i*k_r \mu_r$.
- 4 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.
- 5 Click **OK**.

MESH GENERATION

This model uses the default mesh as initial mesh for the adaptive meshing. It is important to remember to initialize the mesh prior to each new solution step. Otherwise the solver applies the adaptive mesh refinement algorithm on an already adapted mesh.

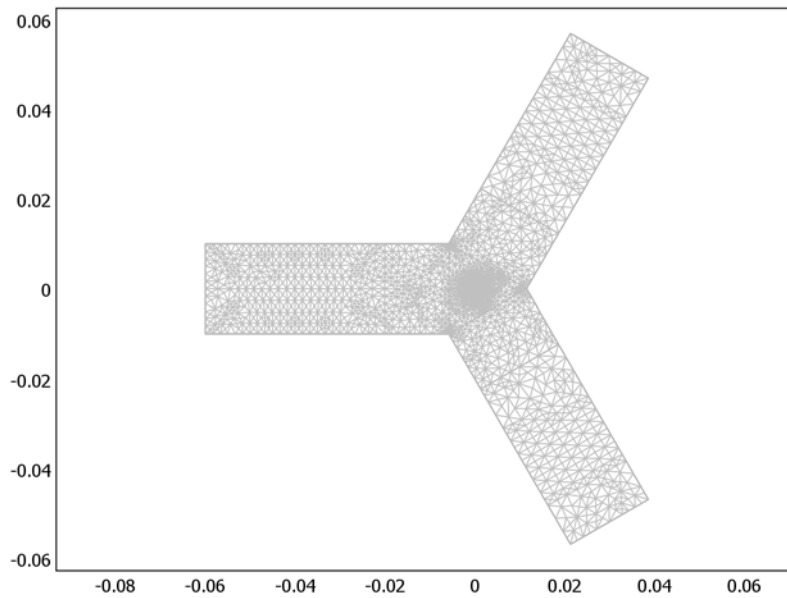
COMPUTING THE SOLUTION

Use the parametric solver to study the S-parameters as a function of the relative permittivity in the matching elements. Also activate adaptive meshing on the S_{11} -parameter.

- 1 Click the **Solver Parameters** button on the Main toolbar.
- 2 From the **Solver** list, select **Parametric**.
- 3 Enter `eps_r` as **Parameter names** and `range(1,0.01,1.5)` as **Parameter values**. The solver will calculate the solution for 51 equidistant permittivity values in the range 1 to 1.5.
- 4 Select the **Adaptive mesh refinement** check box.
- 5 Go to the **Adaptive** page. From the **Error estimate** list, choose **Functional**. Then type `S11_rfwe` in the **Functional** edit field.
- 6 Click **OK**.
- 7 Click the **Solve** button on the Main toolbar.

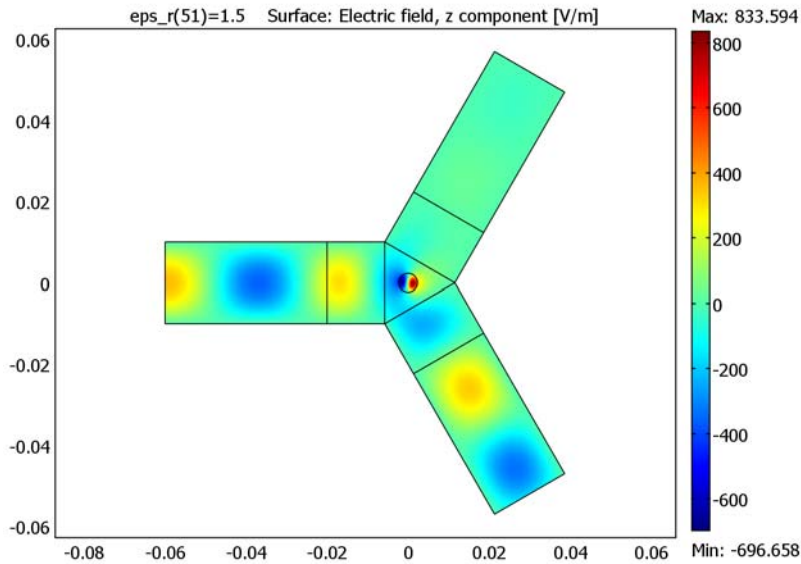
The solver performs a full sweep before each adaptive mesh refinement. It is possible to speed up the parameter sweep by first doing the adaptive meshing for one parameter value, and then turn off the adaptive mesh refinement before running the full sweep.

The following figure shows how the mesh looks after the parameter sweep with adaptive meshing.



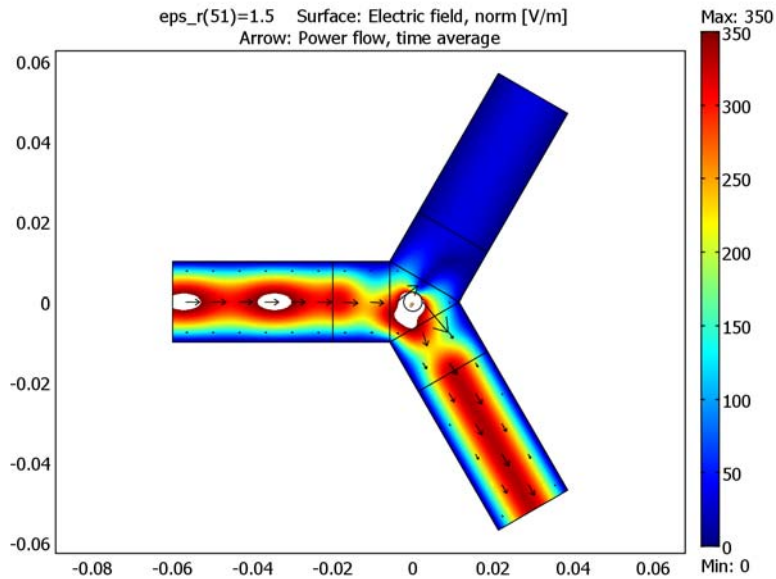
POSTPROCESSING AND VISUALIZATION

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



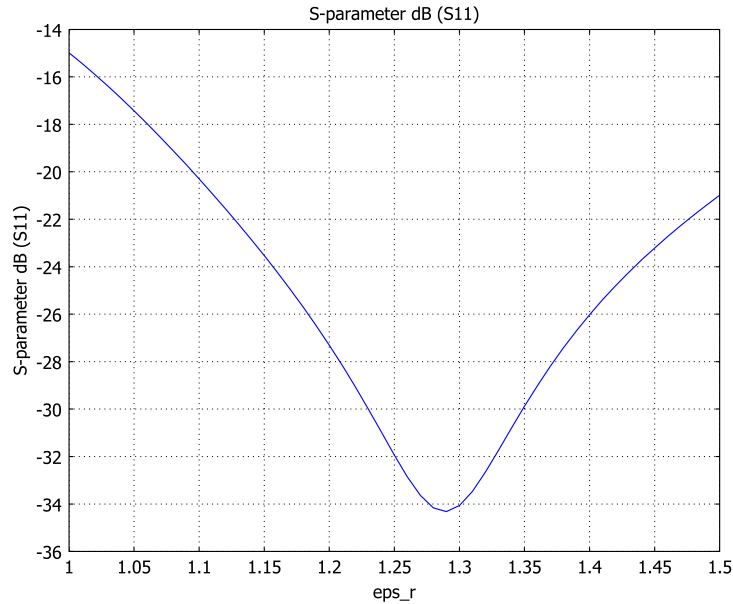
It is more instructive to look at the magnitude of the electric field and to set the color range manually because the strong electric field in the ferrite post otherwise dominates the plot:

- 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 Click the **Color** button and make the arrow color black using the **Arrow Color** dialog box, then click **OK** twice to close the **Arrow Color** and **Plot Parameters** dialog boxes and create the plot.



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 eps_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 eps_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** page.

The plot shows that S_{11} is minimized by setting eps_r to 1.28. Make this change in the **Constants** dialog box, open the **Solver Parameters** dialog, select the **Stationary** solver, initialize the mesh, 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 run COMSOL Multiphysics with MATLAB.

- I From the **File** menu, choose **Client/Server/MATLAB>Connect to MATLAB**.

2 From the **File** menu, choose **Export>FEM Structure as 'fem'**, then switch to the MATLAB window.

3 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 vs. Frequency to Touchstone

This section requires that you run COMSOL Multiphysics with MATLAB.

1 If you continue from the previous section it is not necessary to open a MATLAB connection and export the FEM structure. Otherwise, do Steps 1 and 2 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,'frequit','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 `frequit` 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. As a final step, you conduct a transient analysis of the model and add an external circuit.

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.

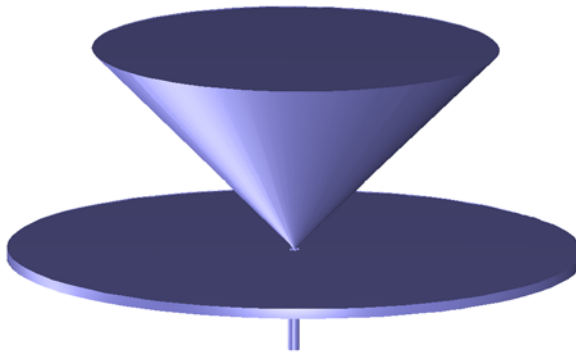


Figure 3-4: The geometry of the antenna. The central conductor of the coaxial cable is connected to the metallic cone, and the cable screen is connected to the finite ground plane.

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 , ϕ , 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 ω . The propagation constant, k , relates to the wavelength in the medium λ 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_ϕ :

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

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 Ω to obtain maximum transmission into free space. This is quite well fulfilled in the high frequency range.

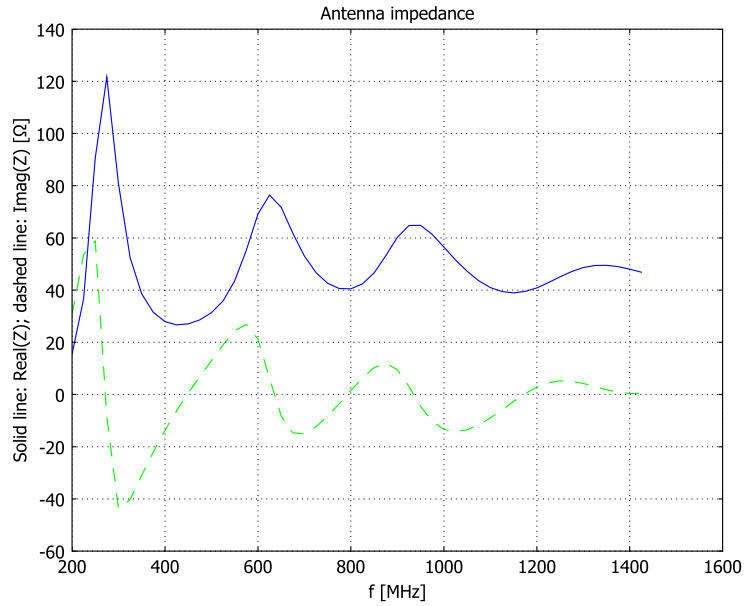


Figure 3-5: The antenna impedance as a function of frequency from 200 MHz to 1.5 GHz. The solid line shows the radiation resistance and the dashed line represents the reactance.

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.

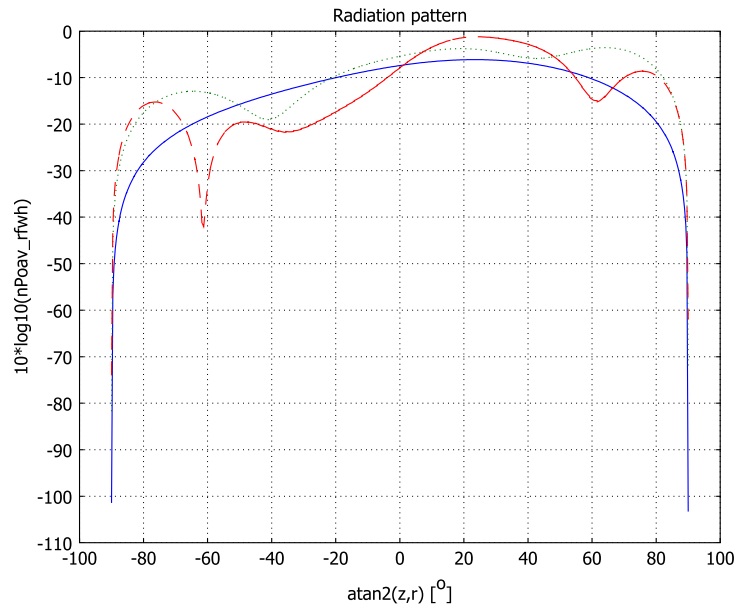


Figure 3-6: The antenna radiation pattern in the near-field as a function of the elevation angle for 200 MHz (solid line), 863 MHz (dotted line) and 1.5 GHz (dashed line).

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.

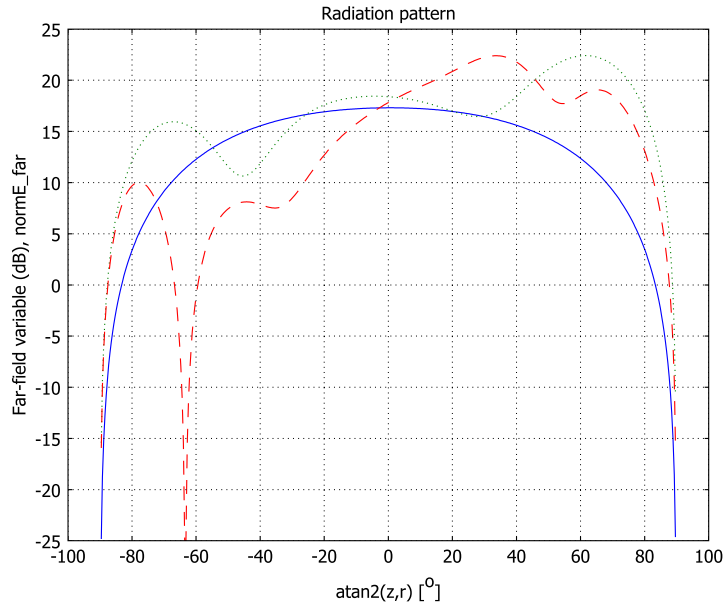


Figure 3-7: The antenna radiation pattern for the far-field as function of elevation angle for 200 MHz (solid line), 863 MHz (dotted line) and 1.5 GHz (dashed line).

As the frequency increases the antenna impedance gets closer to $50\ \Omega$, which means that a voltage generator connected to the input of the antenna should have an output impedance of $50\ \Omega$. Using a transient analysis it is possible to add an external circuit,

and after 3 ns the outgoing wave has reached the outer boundary, see Figure 3-8 below.

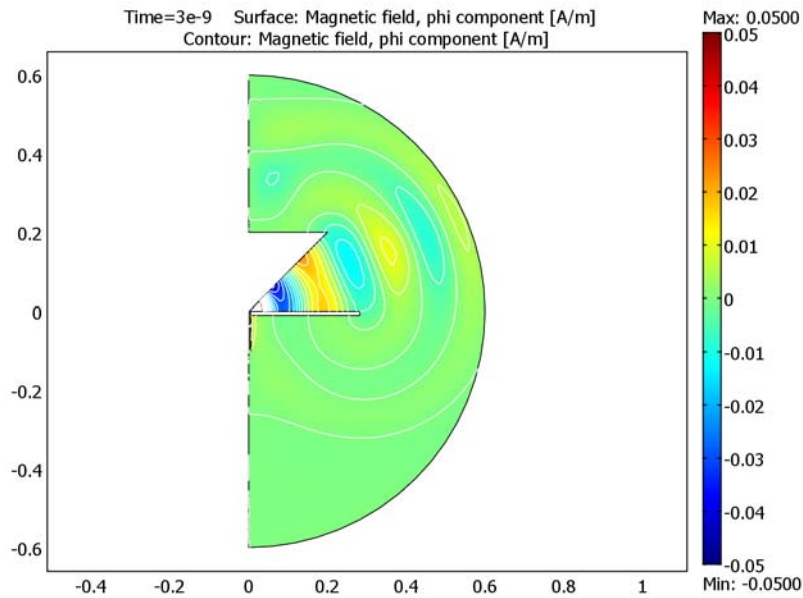


Figure 3-8: The magnetic field after 3 ns. The front of the outgoing wave has just reached the outer boundary.

Model Library path: RF_Module/RF_and_Microwave_Engineering/
conical_antenna

Model Library path: RF_Module/RF_and_Microwave_Engineering/
conical_antenna_circuit

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

- I 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**.

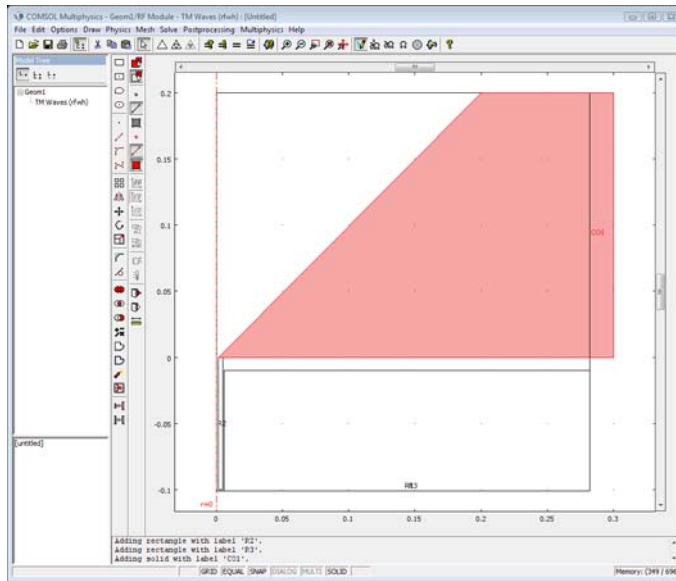
GRID	
r spacing	0.05
Extra r	0.0015 0.004916
z spacing	0.05
Extra z	

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 $R1 - (R2 + R3 + C01)$.
- 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.

-
- File Edit Options Draw Physics Mesh Solve Postprocessing Multiphysics Help
- COMSOL Multiphysics - Geometry (fwh) [Untitled]
- Geometry
- TM Waves (fwh)
- Grid
- Equal
- Snaps
- Plate
- Solid
- Memory: 1326 / 1326

- 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
ϵ_r	1	2.07
μ_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

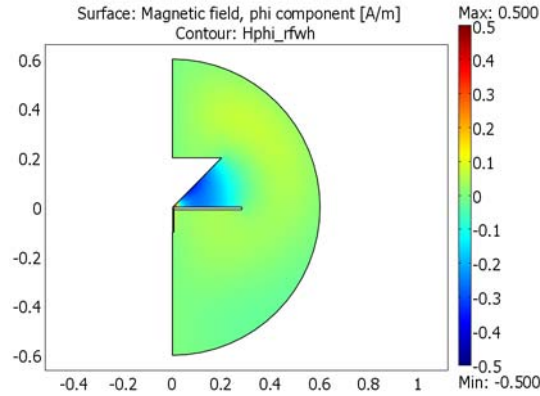
Click the **Solve** button on the Main toolbar.

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 range (-0.4,0.05,0.4) 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 legend** 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

- 1 In the **Constants** dialog box, enter the following variable name and expression:

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

- 2 Click **OK**.

3 In the **Scalar Expressions** dialog box, define the following variable:

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

4 Click **OK**.

SOLVING THE MODEL

1 Open the **Solver Parameters** dialog box and select the **Parametric** solver.

2 Set the **Parameter names** to **f** and the **Parameter values** to **range(200e6, 25e6, 1.5e9)**, 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 option 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 From the **Physics** menu, select **Boundary settings**.
- 2 Select Boundaries 14 and 15.
- 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 From the **Postprocessing** menu, open the **Domain Plot Parameters** dialog box.
- 2 Select a few frequencies in the **Solutions 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.

Transient Analysis with External Circuit

To add an external circuit, it is necessary to first change the analysis type to transient analysis. Then you add the circuit using the SPICE circuit editor.

OPTIONS AND SETTINGS

- 1 From the **Physics** menu, choose **Properties**.
- 2 In the **Application Mode Properties** dialog box, select **Transient propagation** from the **Analysis type** list.
- 3 Click **OK**.

PHYSICS SETTINGS

- 1 From the **Physics** menu, choose **Boundary Settings**.
- 2 Select Boundary 6, and choose the **Circuit port** boundary condition from the **Boundary Condition** list.
- 3 Click **OK**.
- 4 From the **Physics** menu, choose **SPICE Circuit Editor**.
- 5 In the **SPICE Circuit Editor** dialog box, enter the following netlist in the **SPICE netlist** area.


```
V1 1 0 SIN 0 100 1.5GHz
R1 1 2 50
X1 2 0 conical
.SUBCKT conical 1 COMSOL: *
.ENDS
```
- 6 Click **OK**.

SOLVING THE MODEL

- 1 From the **Solve** menu, open the **Solver Parameters** dialog box.
- 2 Type 0.3×10^{-9} in the **Times** edit field.
- 3 Go to the **Time Stepping** page, choose **Time steps from solver** from the **Times to store in output** list, and **Manual** from the **Time steps taken by solver** list.
- 4 Type $1/15 \times 10^9$ in the **Time step** edit field.
- 5 Click **OK**.

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

POSTPROCESSING AND VISUALIZATION

The level of the field is about one order of magnitude lower for the transient analysis compared so it is necessary to adjust the range.

- 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 -0.05 and 0.05 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 range (-0.04,0.005,0.04) in the associated edit field.
- 5 Click **OK** to see the plot in Figure 3-8 on page 70.

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.

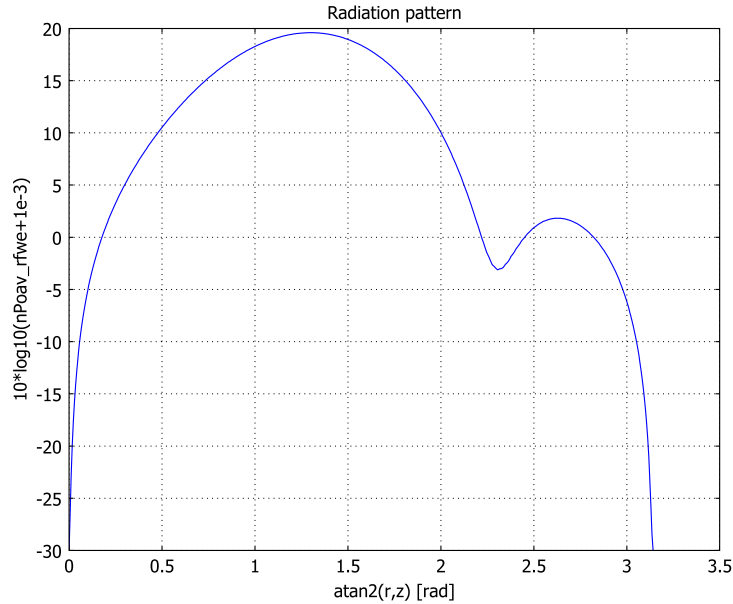
The emitted waves from the dipoles are axisymmetric TE waves. The governing equation for the waves is

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

where μ_r is the relative permeability, ϵ_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_ϕ .

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

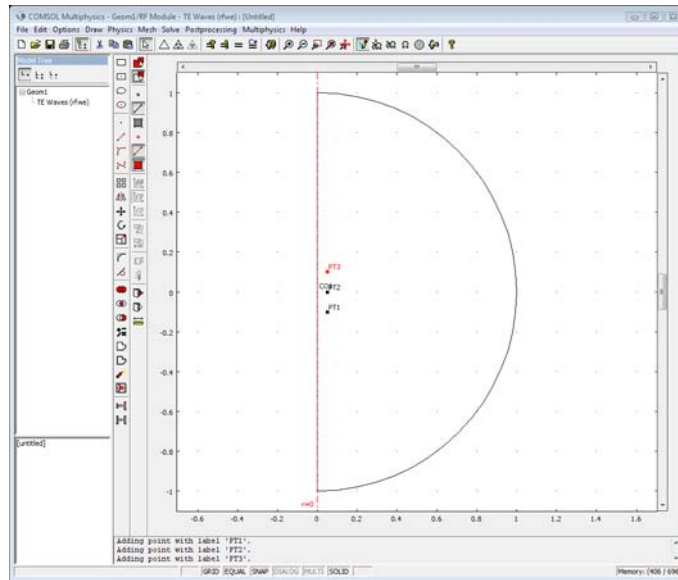
1 In the **Constants** dialog box, enter the following variable names and expressions.

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	I0	$I0 \cdot \exp(j \cdot \theta_1)$	$I0 \cdot \exp(j \cdot \theta_2)$

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

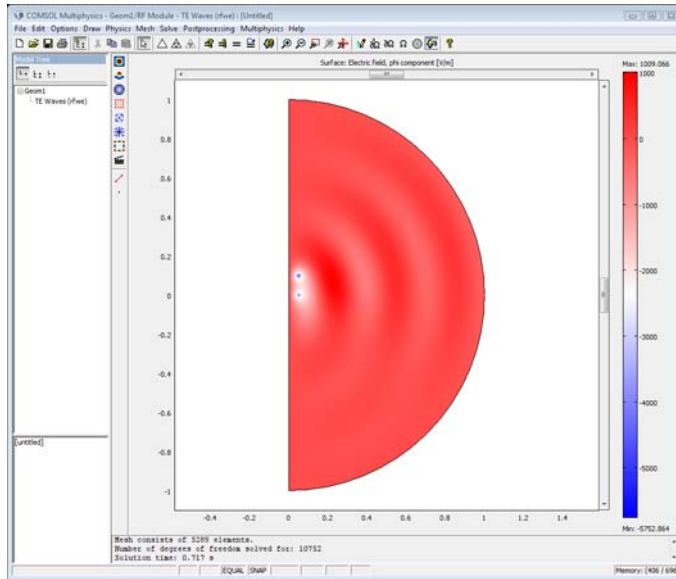
- 1 To obtain a uniform mesh, click the **Custom mesh size** button and set **Maximum element size** to 0.03 in the **Free Mesh Parameters** dialog box.
- 2 Generate the mesh.

COMPUTING THE SOLUTION

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

POSTPROCESSING AND VISUALIZATION

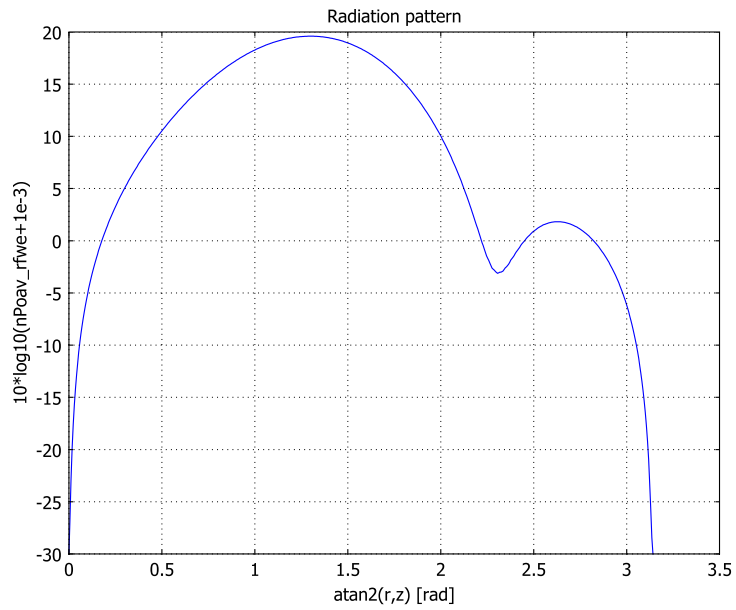
The default plot shows the electric field component of the transmitted wave.



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

- I In the **Domain Plot Parameters** dialog box select **Line** plot. Set the **y-axis data** to $10 \cdot \log_{10}(\text{nPoav_rfwe} + 1e-3)$ 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** option button. Then click the **Expression** button and type $\text{atan2}(r,z)$ in the **Expression** edit field.



Optimizing the Shape of a Dipole Antenna

Introduction

The impedance of an antenna is an important parameter that determines how it behaves in transmitter circuits. Impedance matching and low reactive components are important to get reliable operation, which can be achieved either by proper design or a matching circuit. This model optimizes the length and diameter of a dipole antenna so that its input impedance matches a specified value.

Note: This model requires the Optimization Lab.

Model Definition

A deformed mesh application mode controls the shape of the antenna, by changing the displacement of the antenna's boundaries. The antenna is a long half-wavelength dipole antenna, so the parameters controlling the dimensions are given in terms of the wavelength. Because the dipole is a cylindrical wire, the entire geometry is axisymmetric. The magnetic field from the antenna is polarized out of the plane (ϕ direction), so the TM Waves application mode is appropriate. A magnetic frill generator excites a voltage signal to the antenna. The current comes from a line integral around the antenna, which collapses to a point evaluation in axial symmetry:

$$I = \oint H_{\phi} r d\phi = 2\pi r H_{\phi}$$

If the generated voltage equals unity (that is, 1 V), the impedance is

$$Z = \frac{1 \text{ V}}{I}$$

THE OPTIMIZATION SETUP

The physical model calculates the impedance of the antenna, and the goal is to adjust the antenna's dimensions so that the impedance gets a specified value, denoted Z_0 .

Because the impedance is inversely proportional to the degree of freedom H_ϕ , it is better to use the admittance in the optimization:

$$Y_0 = \frac{I}{1 \text{ V}} \sim H_\phi$$

The objective function for the optimization is then the squared absolute value of the difference multiplied with Z_0 ,

$$f_{\text{obj}}(k, l) = |IZ_0 - 1 \text{ V}|^2 \quad (3-1)$$

where k and l are the design parameters controlling the dipole's diameter and length,

$$\begin{aligned} L &= l\lambda \\ \Phi &= k\lambda \end{aligned}$$

where λ is the wavelength.

The antenna is a long half-wavelength dipole, so the parameters have to fulfill certain constraints

$$\begin{aligned} 0.4 &< l < 0.6 \\ 0.001 &< k < 0.1 \\ k &< \frac{l}{10} \end{aligned}$$

Results and Discussion

The optimization should give a value of the impedance as close as possible to the real value 74Ω . The initial guess gives a value of $94 + i51 \Omega$, which is quite far from the goal. After roughly 120 iterations, the optimization process returns an impedance close to

$$Z = 74 - i5.9 \cdot 10^{-6} \Omega$$

which is very close to the desired value. The optimization parameters for this impedance (there may be small deviations between platforms) are

$$\begin{aligned} k &= k_{\text{opt}} = 0.0322 \\ l &= l_{\text{opt}} = 0.446 \end{aligned}$$

which corresponds to the antenna dimensions

$$\Phi = k_{\text{opt}}\lambda = 1.04 \text{ cm}$$

$$L = l_{\text{opt}}\lambda = 14.5 \text{ cm}$$

The figure below shows the magnetic field for the optimal design.

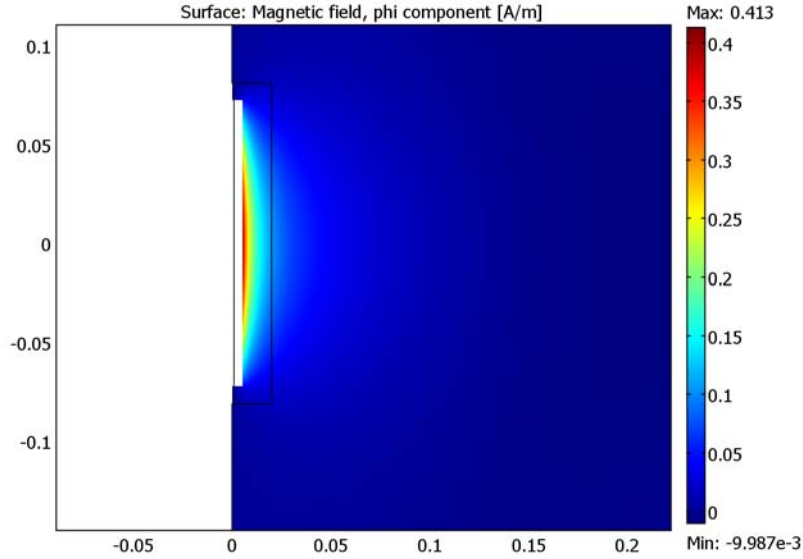


Figure 3-9: The magnetic field close to the antenna.

Model Library path: RF_Module/RF_and_Microwave_Engineering/
shape_optimize_dipole_antenna

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

- 1** From the **Space dimension** list, select **Axial symmetry (2D)**.
- 2** From the **Application Modes** list, select **COMSOL Multiphysics>Optimization and Sensitivity>Optimization**.
- 3** Click the **Multiphysics** button, then click the **Add** button.

- 4 From the **Space dimension** list, select **2D**. The Parameterized Geometry application mode is not present in the axisymmetric view, but the 2D version can be used in this case.
- 5 Select the application mode **COMSOL Multiphysics>Deformed Mesh>Parameterized Geometry**, then click **Add**.
- 6 Select the application mode **RF Module>Electromagnetic Waves>TM Waves**. Click **Add**.
- 7 Click **OK**.

GEOMETRY MODELING

- 1 From the **Draw** menu, use **Specify Object>Circle** to specify the following circles. Note that the circles get their labels automatically.

LABEL	RADIUS	(R, Z)
C1	0.3	(0, 0)
C2	0.5	(0, 0)

- 2 Use **Specify Object>Rectangle** to draw the following rectangles.

LABEL	WIDTH	HEIGHT	BASE	(R, Z)
R1	2e-3	0.162	Center	(0, 0)
R2	0.5	1	Corner	(-0.5, -0.5)
R3	20e-3	0.162	Corner	(0, -0.081)

- 3 Use **Specify Object>Point** to define the following points.

LABEL	R	Z
PT1	1e-3	0
PT2	5e-3	0

- 4 From the **Draw** menu, choose **Create Composite Object**.
- 5 In the **Create Composite Object** dialog box, enter $(C1+C2+R3) - R1 - R2$ in the **Set formula** edit field.
- 6 Click **OK**.

OPTIONS AND SETTINGS

Global Variables

- 1 From the **Options** menu, choose **Expressions>Global Expressions**.

- 2 In the **Global Expressions** dialog box, define the following variables with names, expressions, and (optionally) descriptions:

NAME	EXPRESSION	DESCRIPTION
lambda0	0.324[m]	Wavelength
Z0	74+i*0	Desired input impedance
L	l*lambda0	Dipole length
d_wire	k*lambda0	Wire diameter
Z_frill	1/I_frill	Dipole impedance

The variables k and l are design parameters. You define I_frill next.

- 3 Click **OK**.

Integration Variables

- 1 From the **Options** menu, choose **Integration Coupling Variables>Point Variables**.
- 2 In the **Point Integration Variables** dialog box, select Point 8, and define a variable with the **Name** I_frill by the **Expression** $2\pi r \cdot H_{\phi}$. Make sure that the **Global destination** check box is selected.
- 3 Click **OK**.

PHYSICS SETTINGS—TM WAVES

Properties and Scalar Variables

- 1 From the **Physics** menu, choose **Properties**.
- 2 In the **Application Mode Properties** dialog box, select **Free space wavelength** from the **Specify wave using** list. Click **OK**.
- 3 From the **Physics** menu, choose **Scalar Variables**.
- 4 In the **Expression** column for the variable lambda_rfw, type lambda0.
- 5 Click **OK**.

Subdomain Settings

- 1 From the **Physics** menu, open the **Subdomain Settings** dialog box.
- 2 Click the **PML** tab and select Subdomain 1.
- 3 From the **Type of PML** list, select **Spherical**.
- 4 Click **OK**.

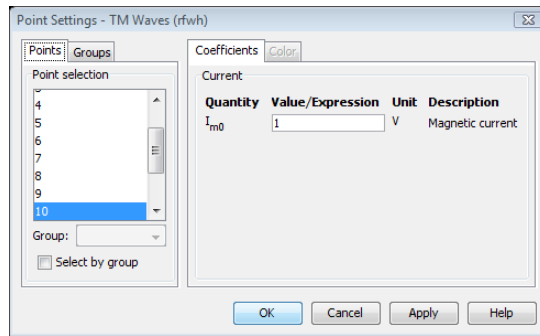
Boundary Conditions

- 1 From the **Physics** menu, open the **Boundary Settings** dialog box.

- 2 Select Boundaries 1, 2, 4, and 6. From the **Boundary condition** list, select **Axial symmetry**.
- 3 Select Boundaries 12 and 15. From the **Boundary condition** list, select **Scattering boundary condition**.
- 4 Click **OK**.

Point Settings

- 1 From the **Physics** menu, open the **Point Settings** dialog box.
- 2 Select Point 10. In the **Magnetic current** edit field, type 1.



- 3 Click **OK**.

PHYSICS SETTINGS—PARAMETERIZED MESH

From the **Multiphysics** menu, select **Parameterized Geometry (pg)**.

Subdomain Settings

- 1 From the **Physics** menu, open the **Subdomain Settings** dialog box.
- 2 Select Subdomain 1, and click the **No displacement** option button.
- 3 Click **OK**.

Boundary Conditions

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

3 Adjust the boundary settings according to the tables below.

PROPERTY	BOUNDARIES 2, 4	BOUNDARY 3	BOUNDARY 5
Condition on R displacement	Prescribed displacement	Free	Free
Value of displacement	0		
Condition on Z displacement	Free	Prescribed displacement	Prescribed displacement
Value of displacement		$-(L - \lambda_0/2)/2$	$(L - \lambda_0/2)/2$

PROPERTY	BOUNDARY 8	BOUNDARY 10	BOUNDARIES 7, 9, 11
Condition on R displacement	Prescribed displacement	Prescribed displacement	Prescribed displacement
Value of displacement	$d_{\text{wire}}/2 - 1\text{e-}3$	$d_{\text{wire}}/2 - 1\text{e-}3$	$d_{\text{wire}}/2 - 1\text{e-}3$
Condition on Z displacement	Prescribed displacement	Prescribed displacement	Free
Value of displacement	$-(L - \lambda_0/2)/2$	$(L - \lambda_0/2)/2$	

4 Click **OK**.

PHYSICS SETTINGS—OPTIMIZATION

From the **Multiphysics** menu, select **Optimization (opt)**.

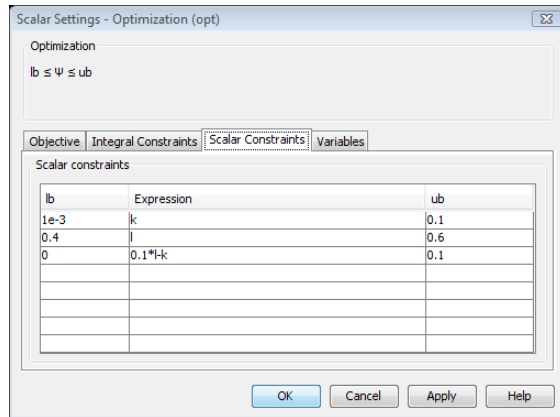
Scalar Settings

- 1 From the **Physics** menu, choose **Scalar Settings**.
- 2 On the **Objective** page, type `realdot(I_frill*Z0-1,I_frill*Z0-1)` in the **q_s** edit field.
- 3 On the **Variables** page, specify the following parameters:

VARIABLE	INIT
l	0.5
k	0.01

- 4 On the **Pointwise Constraints** page, specify the following bounds for the parameters you just defined:

LB	EXPRESSION	UB
1e-3	k	1e-1
0.4	l	0.6
0	0.1*1-k	0.1



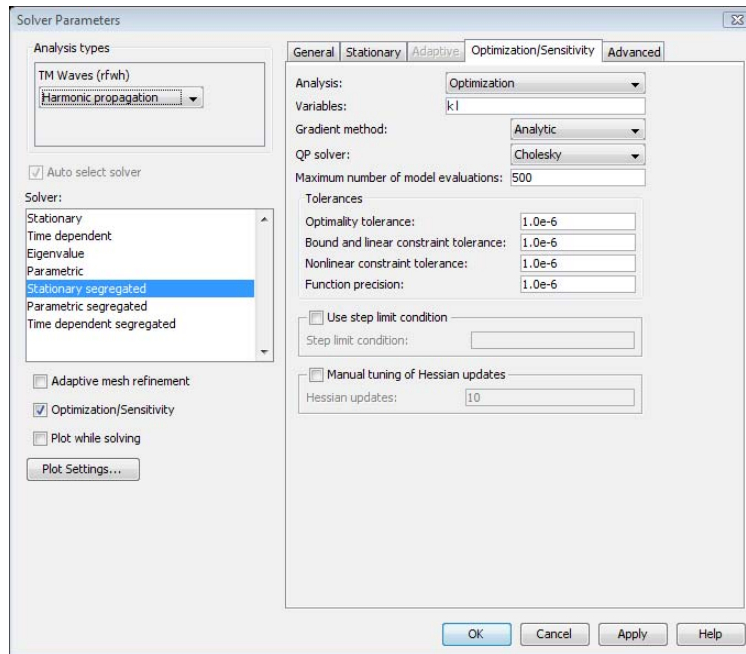
MESH GENERATION

- 1 From the **Mesh** menu, choose **Free Mesh Parameters**.
- 2 Click the **Point** tab.
- 3 Select Points 8 and 10, then set the **Maximum element size** to $5e-4$.
- 4 Click the **Boundary** tab.
- 5 Select Boundaries 3 and 5, then set the **Maximum element size** to $5e-4$.
- 6 Select Boundaries 7 and 9, then set the **Maximum element size** to $2e-3$.
- 7 Click the **Global** tab.
- 8 From the **Predefined mesh sizes** list, select **Fine**.
- 9 Click **OK**.
- 10 Click the **Initialize Mesh** button on the Main toolbar.

COMPUTING THE SOLUTION

- 1 Click the **Solver Parameters** button on the Main toolbar.
- 2 Clear the **Optimization/Sensitivity** check box

- 3 From the **Solver** list, select **Stationary segregated**.
- 4 In the **Segregated groups** frame, type $r_z k_1$ in the **Components** edit field for Group 1.
- 5 On the **General** page, select the **Manual specification of segregated steps** check box.
- 6 For Groups 1 and 2, in turn, set the **Damping** to 1 and the **Number of iterations** to 2.
- 7 Click **Apply**.
- 8 Click the **Solve** button on the Main toolbar.
- 9 Return to the **Solver Parameters** dialog box and select the **Optimization/Sensitivity** check box.
- 10 Click the **Optimization/Sensitivity** tab. In the **Variables** edit field, type k_1 . The gradient method is analytical, which is possible because you used the special function `realdot` instead of `abs` in the goal function. The use of `realdot` circumvents issues involved in computing sensitivities for a nonanalytic objective function, such as the absolute value of a complex number (see the section “Objective Functions for Frequency-Domain Modeling” on page 291 of the *COMSOL Multiphysics Modeling Guide*).



- 11 Click **OK** to close the **Solver Parameters** dialog box.

12 Click the **Solve** button on the Main toolbar to solve the optimization problem.

POSTPROCESSING AND VISUALIZATION

Reproduce the plot in Figure 3-9 on page 88 with the following steps:

- 1** Click the **Plot Parameters** button on the Main toolbar.
- 2** Click the **Surface** tab. From the **Predefined quantities** list on the **Subdomain Data** page, select **TM Waves (rfwh)>Magnetic field, phi component**.
- 3** Click **OK** to close the dialog box and generate the plot you see when you open the model from the Model Library.
- 4** To finish the plot in Figure 3-9, use the **Zoom Window** tool on the Main toolbar to zoom in on the antenna.

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 because microwave systems may be subject to thermal drift caused by high-power loads or harsh environmental conditions like exposure to direct sunlight in the desert. Thus, system engineers need to estimate the drift of the passband frequency that arises due to thermal expansion of a filter.

Model Definition

Figure 3-10 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.

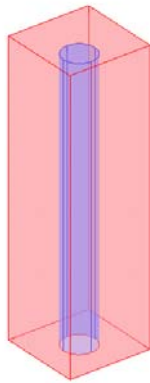


Figure 3-10: A microwave filter consists of a box-shaped thin metallic shell, typically made of brass, that contains a cylindrical post. This configuration forms a closed air-filled electromagnetic cavity between the box walls and the post.

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 model uses the Moving Mesh (ALE) application mode for deformed geometry in COMSOL Multiphysics 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 3-11 shows the thermal expansion results for a filter entirely made of brass.

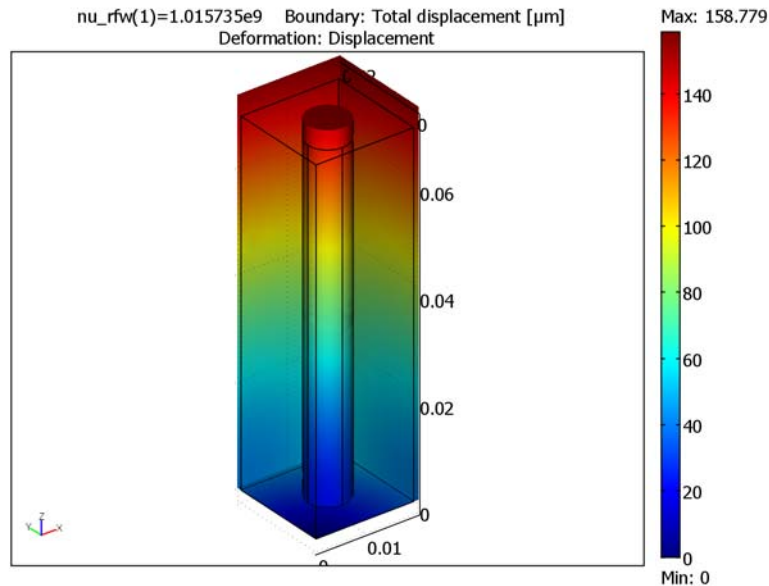


Figure 3-11: Thermal expansion at 100 °C above the reference temperature.

ELECTROMAGNETIC MODE ANALYSIS

An actual filter usually consists of multiple cavities cascaded, but this discussion limits the analysis to one cell. Figure 3-12 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.

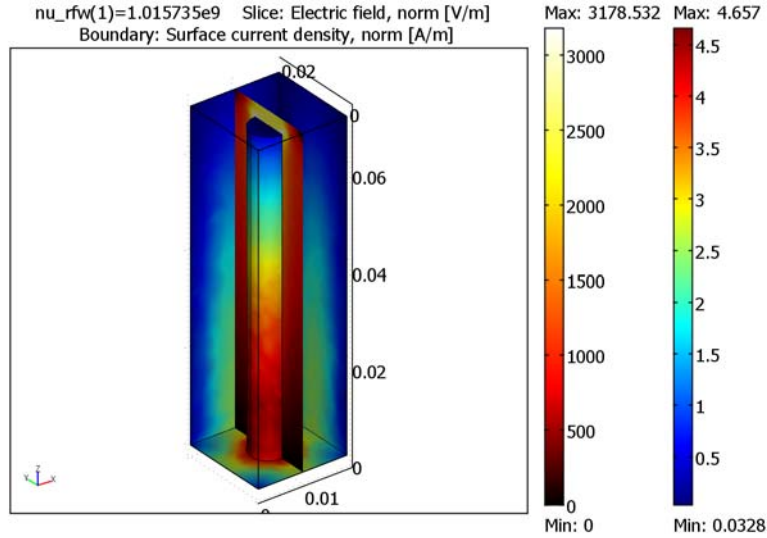


Figure 3-12: Results from the electromagnetic mode analysis. The plot shows the electric and surface current patterns of the fundamental mode.

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 3-13.

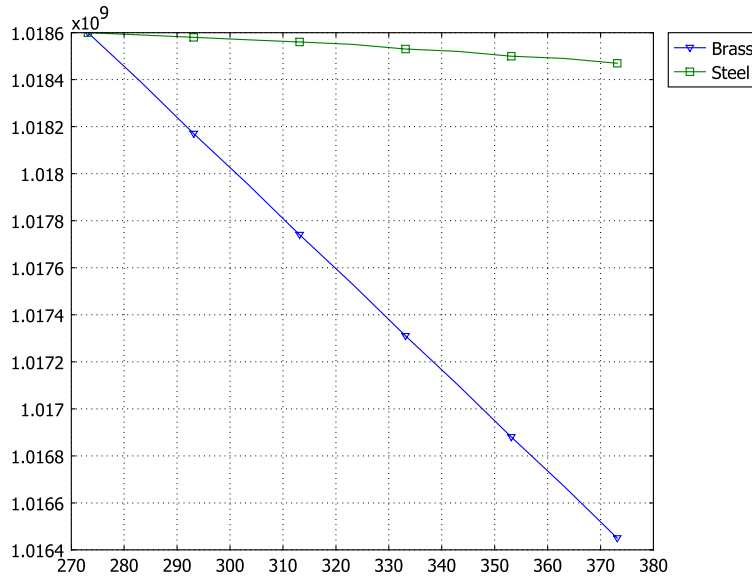


Figure 3-13: Eigenfrequency (Hz) versus temperature (K) for two different designs.

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

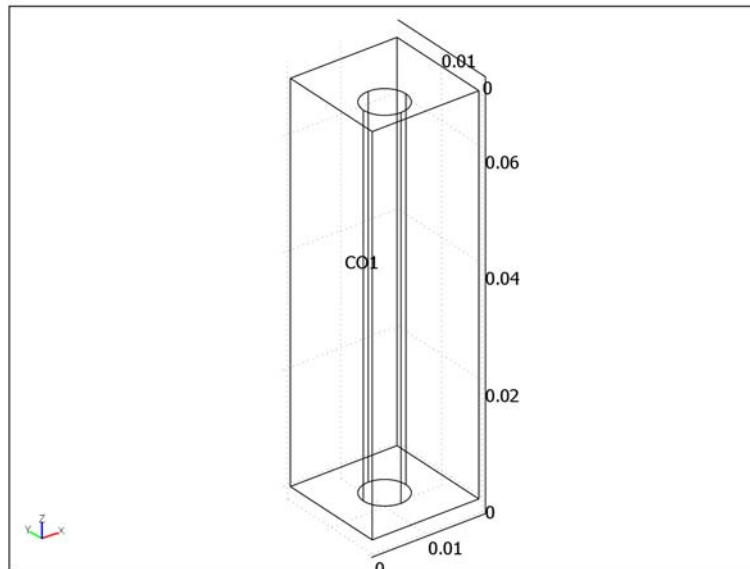
- 1 From the **Options** menu, choose **Constants**.
- 2 In the **Constants** dialog box, define the following constants. The description field is optional. When done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
T0	0[degC]	Reference temperature
T1	100[degC]	Operating temperature

For the parametric analysis, you replace the constant T1 for the operating temperature, T_1 , with a parameter T1 in the **Parametric Sweep** 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 1.05×10^{11} (Pa).
- 5 Change the **Poisson's ratio** to 0.35.
- 6 Change the **Density** to $8400 \text{ (kg/m}^3\text{)}$.
- 7 Change the **Thermal expansion coeff.** to $2.1 \times 10^{-5} \text{ (K}^{-1}\text{)}$.
- 8 Change the **Thickness** to 0.002 (m).
- 9 On the **Load** page, select the **Include thermal expansion** check box.
- 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 2×10^{11} (Pa).
- 14 Change the **Density** to $7800 \text{ (kg/m}^3\text{)}$.
- 15 Change the **Thermal expansion coeff.** to $1.15 \times 10^{-5} \text{ (K}^{-1}\text{)}$.
- 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_x**, **R_y**, and **R_z** to 0.
- 3 Select Point 3 and constrain **R_x** and **R_z** to 0.
- 4 Select Point 13 and constrain **R_y** and **R_z** to 0.
- 5 Click **OK**.

BOUNDARY CONDITIONS—ALE

The default subdomain setting in ALE is “free displacement,” which is right for this model. Therefore 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.

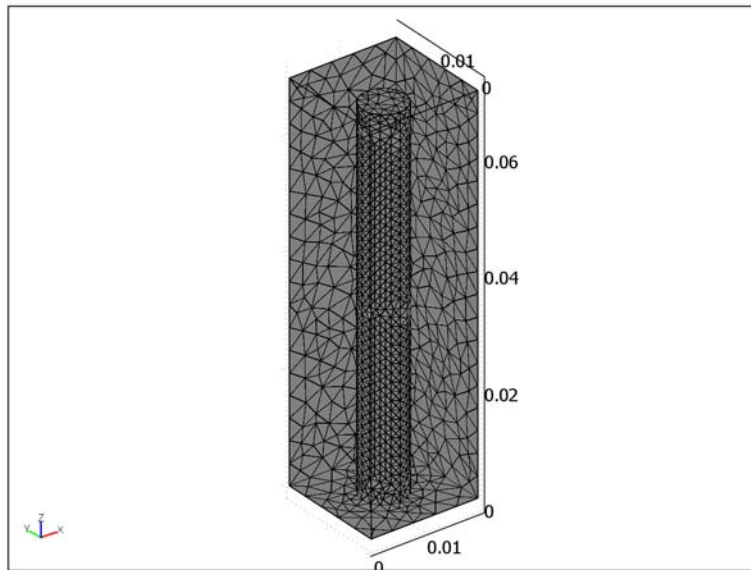
- 3 Set **dx**, **dy**, and **dz** to u, v, and w, respectively.
- 4 Click **OK**.

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 From the **Mesh** menu, open the **Free Mesh Parameters** dialog box. Select **Finer** from the **Predefined mesh sizes** list, then close the dialog box by clicking **OK**.
- 2 Click the **Initialize Mesh** button on the Main toolbar to generate the mesh.
- 3 To see the mesh and later the solution better, choose **Options>Suppress>Suppress Boundaries**. Select Boundaries 1, 2, and 4, then click **OK**. Go to mesh mode by clicking the **Mesh Mode** button on the Main toolbar.



COMPUTING THE SOLUTION

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

application modes and then solve the model for a single operating temperature to verify that the setup is correct.

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.

- 1 Click the **Solver Parameters** button on the Main toolbar.
- 2 From the **Solver** list, select **Stationary**.
- 3 Click **OK** to close the **Solver Parameters** dialog box.
- 4 Click the **Solver Manager** button on the Main toolbar.
- 5 On the **Solve For** page, select only the **Shell (smsh)** variables.
- 6 On the **Sequence** page, click the **Store Current Settings** button.
- 7 In the **Store Solver Settings** dialog box, type smsh in the **Name** edit field. Click **OK**.
- 8 Click the **Initial Value** tab. In the **Values of variables not solved for and linearization point** area, click the **Current** solution button.
- 9 On the **Solve For** page, select only the **Moving Mesh ALE (ale)** variables. (By solving the ALE problem you map the shell deformations smoothly to the volume mesh.)
- 10 On the **Sequence** page, click the **Store Current Settings** button.
- 11 In the **Store Solver Settings** dialog box, type ale in the **Name** edit field. Click **OK**.
- 12 Leaving the **Solver Manager** open, click the **Solver Parameters** button on the Main toolbar.
- 13 In the **Solver Parameters** dialog box, set the **Solver** to **Eigenfrequency**.
- 14 In the **Eigenfrequency** area on the **General** page, set the **Desired number of eigenfrequencies** to 1 and instruct the solver to **Search for eigenfrequencies around 1e9**.
- 15 Click **OK** to close the **Solver Parameters** dialog box.
- 16 Back in the **Solver Manager**, click the **Solve For** tab and select only the **Electromagnetic Waves (rfw)** variables.
- 17 On the **Sequence** page, click the **Store Current Settings** button.
- 18 In the **Store Solver Settings** dialog box, type eig in the **Name** edit field. Click **OK**.
- 19 Select all three entries in the **Stored solver settings** area and then click **>** to copy them to the **Solver sequence** area on the right (preserving the order in which you entered them).

- 20 Select the **Solve using solver sequence** check box, then click **OK** to close the **Solver Manager**.
- 21 Solve by clicking the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

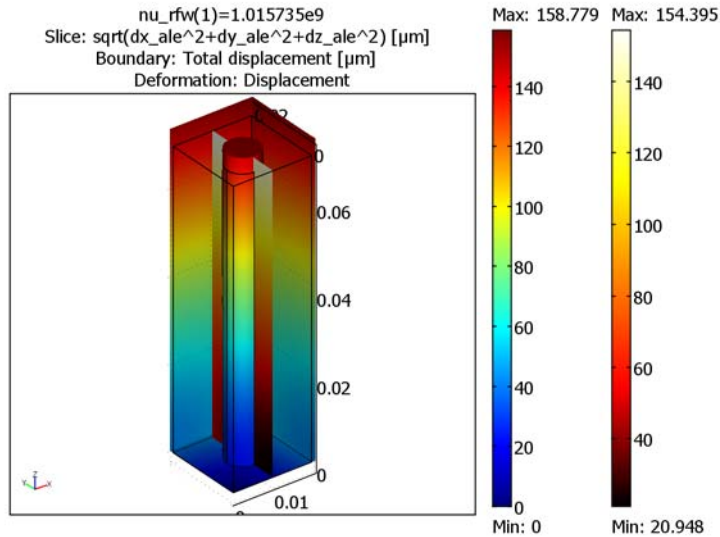
Follow these instructions to reproduce the plot in Figure 3-11 on page 97 visualizing the structural deformation.

- 1 Open the **Plot Parameters** dialog box and clear the **Slice** check box in the **Plot type** area on the **General** page. Instead, select the **Boundary** and **Deformed shape** plots.
- 2 On the **Boundary** page, select **Shell (smsh)>Total displacement** from the **Predefined quantities** list. From the **Unit** list, select μm .
- 3 On the **Deform** page, clear the **Subdomain** check box in the **Domain types to deform** area.
- 4 Click **Apply** to generate the plot, then click the **Headlight** button on the Camera toolbar to finish it.

To visualize the total mesh displacement, add a slice plot according to the following steps.

- 1 In the **Plot Parameters** dialog box, click the **Slice** tab and select the **Slice plot** check box.
- 2 In the **Slice data** area, enter the **Expression** $\sqrt{dx_ale^2+dy_ale^2+dz_ale^2}$. Change the **Unit** to μm .
- 3 In the **Slice positioning** area, change the number of **X levels** to 1.
- 4 In the **Slice color** area, change the **Color table** to **Thermal**.

5 Click **Apply** to generate the plot in the following figure.



Thermal expansion (boundary plot) and the resulting mesh displacement (slice plot) for the brass-only design at an operating temperature of 100 °C.

Next, reproduce the electric field and surface current plot for the filter's lowest eigenmode shown in Figure 3-12 on page 98.

- 1 Return to the **Slice** page of the **Plot Parameters** dialog box and select **Electromagnetic Waves (rfw)>Electric field, norm** from the **Predefined quantities** list in the **Slice data** area.
- 2 Click the **Boundary** tab. In the **Boundary data** area, select **Electromagnetic Waves (rfw)>Surface current density, norm** from the **Predefined quantities** list.
- 3 On the **General** page, clear the **Deformed shape** check box in the **Plot type** area.
- 4 Click **OK** to close the **Plot Parameters** dialog box and generate the plot.

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 THERMAL DRIFT

Having confirmed the correctness of the model setup, proceed to compute the thermal drift over the operating temperature range 0° C through 100° C and compare the brass-only design with one in which the post is made of steel.

Computing the Solution Using a Parametric Sweep—Brass

- 1 From the **Solve** menu, choose **Parametric Sweep**.
- 2 In the **Parameters** area, enter the following settings:

Parameter names	T1
Parameter values	range(273.15, 10, 373.15)

To compute the thermal drift of the filter's lowest eigenfrequency, you want to record the value of `nu_rfw` for each operating temperature in the sweep. However, for this particular case you need not enter this variable in the **Output** area because for output from the eigenvalue solver, COMSOL Multiphysics by default logs the eigenvalue, `lambda`, in the second data column after the parameter value. Thus, leave the **Global variables to evaluate** edit field empty and specify only a log file name.

- 3 Click the **Browse** button, navigate to a directory where you have writing permissions, and then type `mw_filter_freqs_brass.txt` in the **Log file name** edit field.
- 4 Click the **Solve** button to start the parametric sweep.
- 5 When the solver has finished, click **OK** to close the dialog box.

Boundary Conditions—Shell

Change the material in the post from brass to steel.

- 1 From the **Multiphysics** menu, select **Geom1: Shell (smsh)**.
- 2 From the **Physics** menu, open the **Boundary Settings** dialog box.
- 3 Select Boundaries 6–10, then change the **Group** from **Brass** to **Steel**.
- 4 Click **OK**.

Computing the Solution Using a Parametric Sweep—Steel

- 1 From the **Solve** menu, choose **Parametric Sweep**.
- 2 In the **Output** area, change the **Log file name** to `mw_filter_freqs_steel.txt`.
- 3 Click the **Solve** button.
- 4 When the solver has finished, click **OK** to close the dialog box.

Importing the Frequency-Temperature Data

The next step is to import the eigenfrequency versus temperature data to the model. Do this by creating two interpolation functions.

- 1 From the **Options** menu, choose **Functions**.
- 2 In the **Functions** dialog box, click the **New Function** button.

- 3 In the **New Function** dialog box, enter the **Function name** `nu_brass`.
- 4 Click the **Interpolation** button, then select **File** from the **Use data from** list.
- 5 Click the **Browse** button, locate and select the file `mw_filter_freqs_brass.txt` that you just generated, and finally click **Open**. Alternatively, you can use the file with this name located in the models Model Library folder (`models/RF_Module/RF_and_Microwave_Engineering` under the COMSOL installation directory).
- 6 Optionally, select the **Store data in file** check box. This makes the model MPH-file self contained, but it has the potential drawback that changes in the source data file are not taken into account.
- 7 Click **OK** to close the **New Function** dialog box.
- 8 Repeat the Steps 2 through 7 to define the interpolation function `nu_steel` using the data file `mw_filter_freqs_steel.txt`.
- 9 Click **OK** to close the **Functions** dialog box.

To make the interpolation functions available for postprocessing, create two global expression.

- 1 From the **Options** menu, point to **Expressions** and then choose **Global Expressions**.
- 2 In the **Global Expressions** dialog box, define the following expressions:

NAME	EXPRESSION	DESCRIPTION
<code>nu_brass</code>	<code>nu_brass(T1)</code>	Frequency, brass
<code>nu_steel</code>	<code>nu_steel(T1)</code>	Frequency, steel

- 3 Click **OK**.

Updating the Model

Next, use the parametric solver to generate a set of “dummy” solutions for the same parameter and range as the parametric sweeps you just performed.

- 1 Click the **Solver Parameters** button on the Main toolbar.
- 2 Change the **Solver** to **Parametric**.
- 3 In the **Parameter names** edit field type `T1`, and in the **Parameter values** edit field type `range(273.15, 10, 373.15)`.
- 4 Click **OK** to close the **Solver Parameters** dialog box.
- 5 Click the **Solver Manager** button on the Main toolbar.
- 6 On the **Solve For** page, select only the **Shell (smsh)** variables.
- 7 On the **Sequence** page, clear the **Solve using a solver sequence** check box.

- 8 Click **Solve**.
- 9 When the parametric solver has finished, select the **Solve using a solver sequence** check box again.
- 10 Click **OK** to close the **Solver Manager**.

Postprocessing and Visualization

Follow these steps to reproduce the plot in Figure 3-13 on page 99.

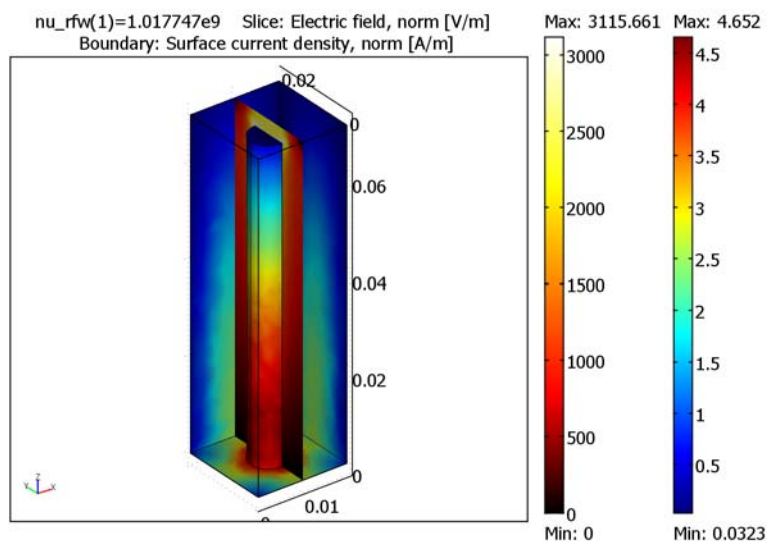
- 1 From the **Postprocessing** menu, choose **Global Variables Plot**.
- 2 From the **Predefined quantities** list, select **Frequency, brass** and **Frequency, steel**.
- 3 Click the **Add Selected Predefined Quantities** button (marked **>**) to add these two expressions to the **Quantities to plot** list on the right.
- 4 Click the **Line Settings** dialog box. Change the selection in the **Line marker** list to **Cycle** and then click **OK** to close the **Line Settings** dialog box.
- 5 Click **OK** to close the **Global Variables Plot** dialog box and generate the plot.
- 6 Click the **Edit Plot** toolbar button in the figure window.
- 7 In the **Edit Plot** dialog box, edit plot title, axis labels, and legends to finish the plot.
- 8 When done, click **OK**.

RESTORING THE MODEL

To save the model's MPH-file with the solution for the combined brass-steel design at an operating temperature of 100° C instead of the dummy solution generated in the previous step, follow these steps.

- 1 Optionally, clear the previous solution history by choosing **Reset Model** from the **File** menu.
- 2 If you chose to reset the model, click the **Initialize Mesh** button on the Main toolbar to re-create a mesh using the settings you specified earlier in the **Free Mesh Parameters** dialog box.
- 3 Click the **Solve** button on the Main toolbar to solve the model using the solver sequence. Note that the solver sequence overrides the current settings in the **Solver Parameters** and **Solver Manager** dialog boxes.

When the solver finishes, the plot in the drawing area should resemble that in the following figure.



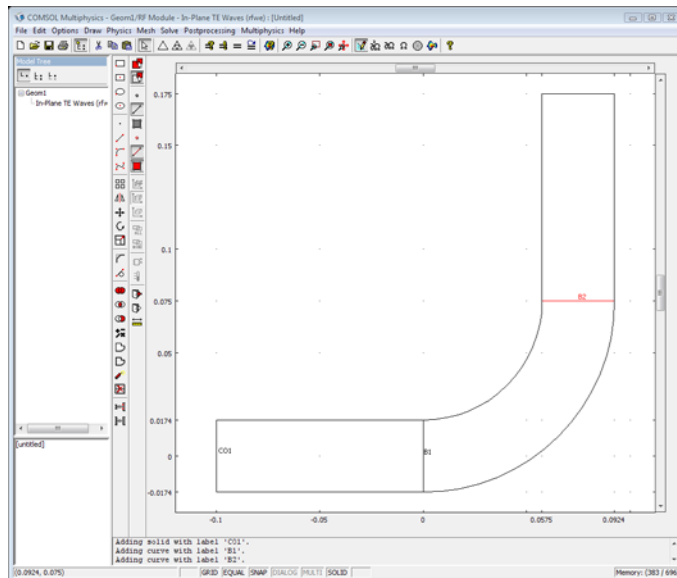
H-Bend Waveguide with S-parameters

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 57 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

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

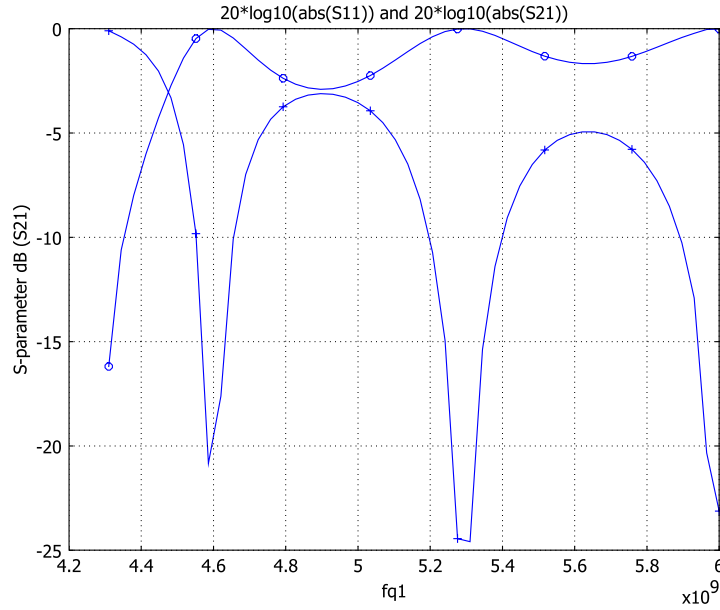
where n is the refractive index, and k_0 is the free space wave number.

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/
waveguide_hbend_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 (0.0576, 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.

- 1 From the **Physics** menu, open the **Boundary Settings** dialog box.
- 2 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); when done, click **OK**.

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

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

- 2 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.
- 3 Click **OK**.

MESH GENERATION

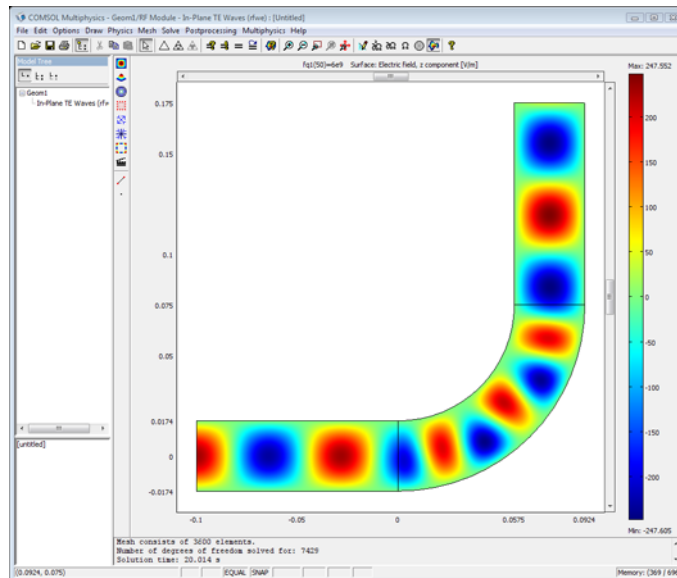
Initialize a mesh and refine it twice using the **Initialize Mesh** and **Refine Mesh** buttons on the Main toolbar.

COMPUTING THE SOLUTION

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

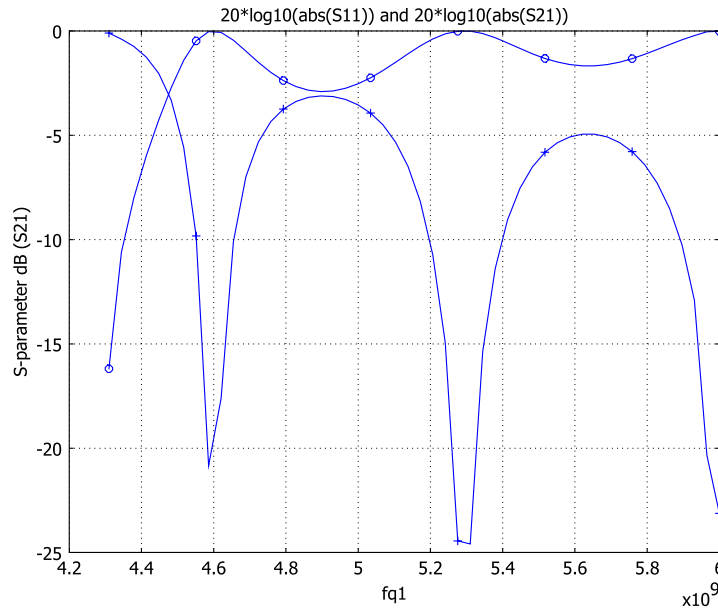
- 1 In the **Solver Parameters** dialog box select the **Parametric** solver.
- 2 Enter **fq1** as **Parameter names**, then click the **Edit** button for the **Parameter values**.
- 3 In the **Edit** dialog box, set the **First value** to 4.31e9, the **Last value** to 6e9, and the **Number of values** to 50, and then finish by clicking **Add**. 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.
- 4 Solve by clicking the **Solve** button.

POSTPROCESSING AND VISUALIZATION



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

- 1 From the **Postprocessing** menu, choose **Domain Plot Parameters**.
- 2 In the **Domain Plot Parameters** dialog box 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**.
- 4 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} .
- 5 On the **General** page, select **Keep current plot**.
- 6 On the **Point** page, select **S-parameter dB(S21)** from the list of **Predefined quantities** and click **OK**.



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, 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** page.

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 3-14.

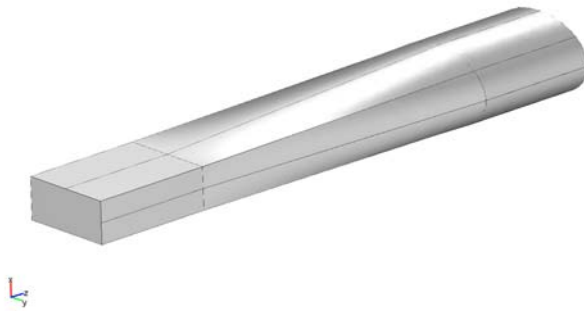


Figure 3-14: The geometry of the waveguide adapter.

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 TE₁₀ 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 TE₁₀ mode, $m = 1$ and $n = 0$. With the dimensions of the rectangular cross section ($a = 2.286$ cm and $b = 1.016$ cm), the TE₁₀ 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, β 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 μ_r denotes the relative permeability, j the imaginary unit, σ the conductivity, ω the angular frequency, ϵ_r the relative permittivity, and ϵ_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

Figure 3-15 displays the first eigenmode of the elliptical port. The effective mode index is $\beta/k_0 = 0.535$.

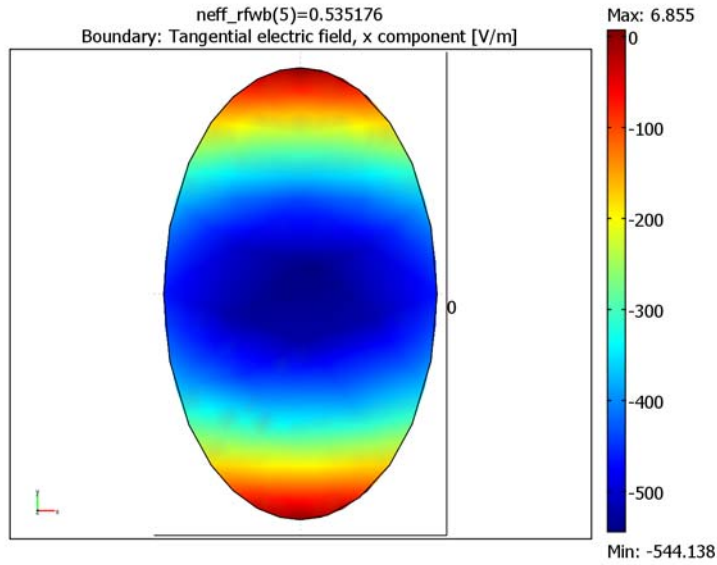


Figure 3-15: The x component of the electric field for the first eigenmode of the elliptical port.

Figure 3-16 shows a single-mode wave propagating through the waveguide.

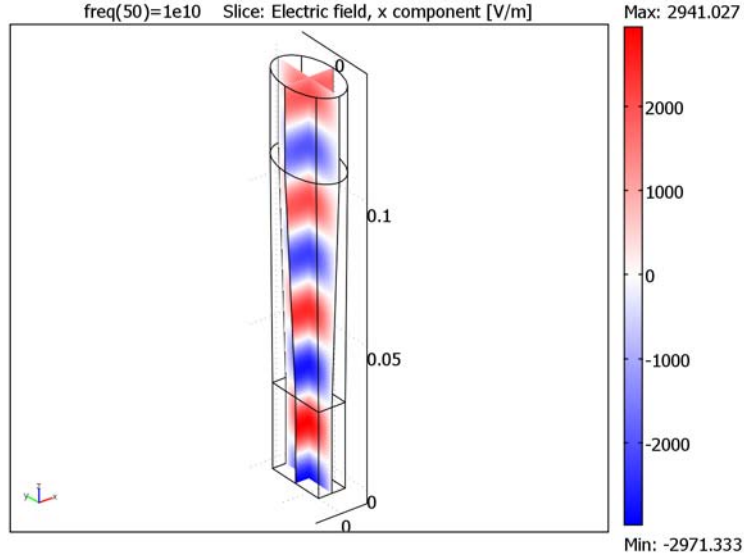


Figure 3-16: The x component of the propagating wave inside the waveguide adapter at the frequency 10 GHz.

Naming the rectangular port Port 1 and the elliptical port 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 3-17 and Figure 3-18 show the S_{11} and S_{21} parameters as functions of the frequency.

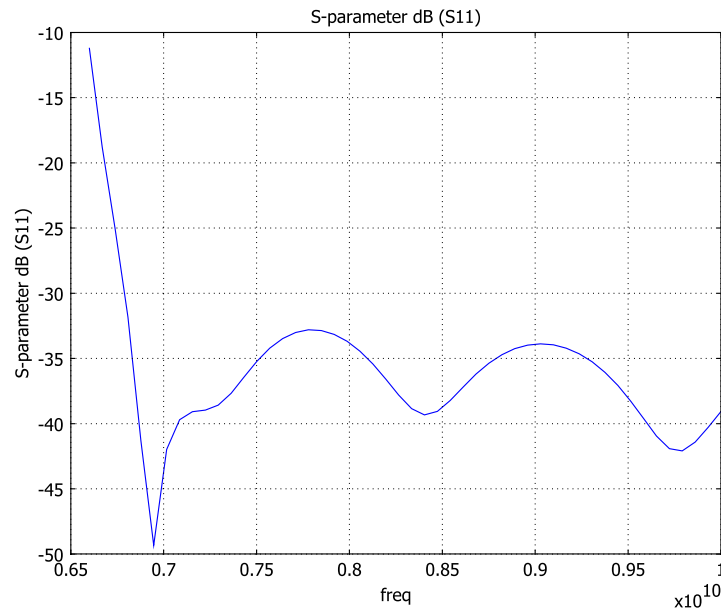


Figure 3-17: The S_{11} parameter (in dB) as a function of the frequency. This parameter describes the reflections when the waveguide adapter is excited at the rectangular port.

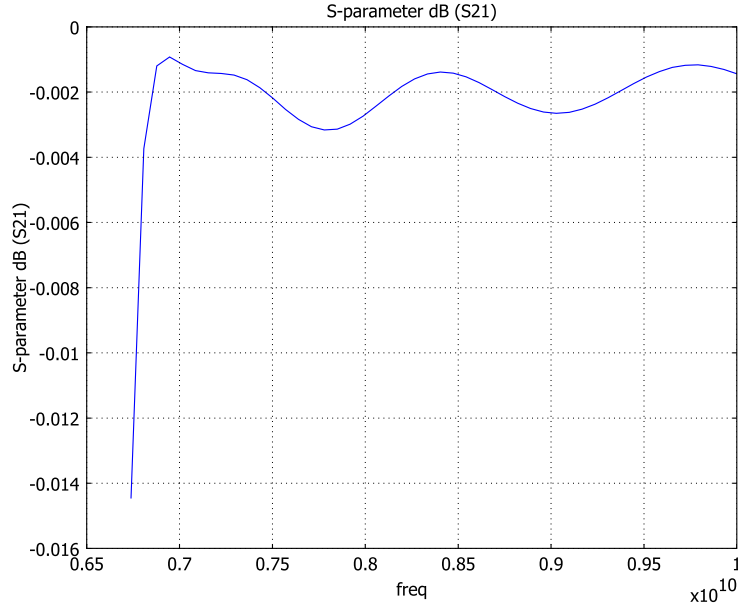


Figure 3-18: The S_{21} parameter (in dB) as a function of the frequency. This parameter is a measure of the part of the wave that is transmitted through the elliptical port when the waveguide adapter is excited at the rectangular port.

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.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.mphbin** file in the model path specified on page 122, 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 (not applicable to interior edges)** 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

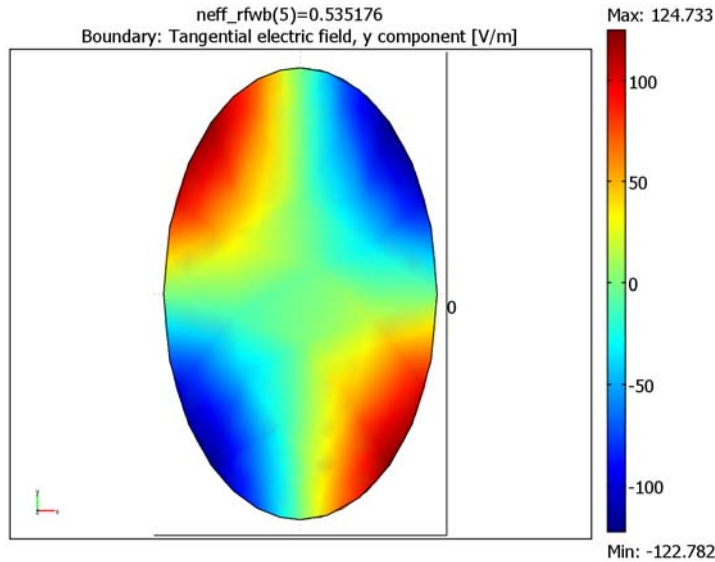
- 1 Open the **Solver Manager** dialog box from the **Solve** menu and select only the **Boundary Mode Analysis, TE Waves (rfwb)** application mode on the **Solve For** tab. Click **OK**.
- 2 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

Visualize the eigenmodes as a boundary plot:

- 1 From the **Postprocessing** menu, choose **Plot Parameters**.
- 2 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.
- 3 Select `0.535` from the **Effective mode index** list.

- 4 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.
- 5 Click **OK** to plot the field and click the **Go to XY View** button to better see the field (see Figure 3-15 on page 118)
- 6 Make the same type of plot of the *y* component of the electric field, `tEy_rfwb`.

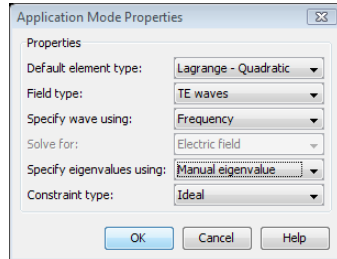


*Figure 3-19: The *y*-component of the electric field for the first eigenmode of the elliptical port.*

- 7 To see the normal component of the magnetic field, start by selecting **Normal magnetic field** from the **Predefined quantities** list.
- 8 On the **General** tab, enter 90 in the **Solution at angle (phase)** edit field. This compensates for the phase shift of 90° between the electric and the magnetic field.
- 9 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 from the **Physics** menu.
- 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

- 4 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.

- 5 Click **OK**.

COMPUTING THE SOLUTION

- 1 Open the **Solver Manager** dialog box. On the **Initial Value** page, 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**.
- 2 In the **Initial value** area, click the **Stored solution** option button.
- 3 On the **Solve For** page, select only the **Electromagnetic Waves (rfw)** application mode.
- 4 Click **OK** to close the **Solver Manager**.
- 5 In the **Solver Parameters** dialog box, select the **Parametric** solver.
- 6 Set the **Parameter names** to **freq**.
- 7 Click the **Edit** button next to the **Parameter values** edit field.
- 8 In the **Edit** dialog box, specify the **First value** as 6.6e9, the **Last value** as 1e10, and set the **Number of values** to 50. Click **Add** to confirm the range of parameter values and close the **Edit** dialog box.
- 9 Click **OK** to close the **Solver Parameters** dialog box.
- 10 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.
- 11 Set the frequency variable **nu_rfw** to **freq**. This is the frequency variable of the 3D Electromagnetic Waves application mode. Click **OK**.

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

POSTPROCESSING AND VISUALIZATION

- 1 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 **WaveLight** from the **Color table** list.
- 7 Click **OK** to see the plot in Figure 3-16 on page 119.

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 From the **Postprocessing** menu, choose **Domain Plot Parameters**.
- 2 On the **Point** tab in **Domain Plot Parameters** dialog box select Point 1 and select **Electromagnetic Waves (rfw)>S-parameter dB (S_{11})**. Click **Apply** to plot S_{11} on a dB scale.
- 3 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 (S_{21})** and click **OK** to make the plot on page 121.

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 MATLAB

To create the geometry, start COMSOL Multiphysics with MATLAB or initiate a MATLAB connection from the **File** menu. 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});
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 3-20 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

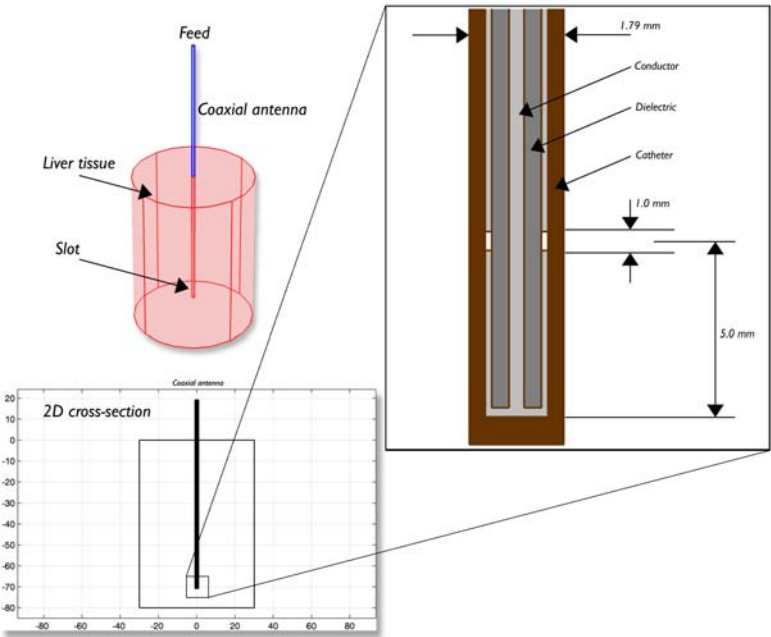


Figure 3-20: Antenna geometry for microwave coagulation therapy. A coaxial cable with a ring-shaped slot cut on the outer conductor is short-circuited at the tip. A plastic catheter surrounds the antenna.

The model takes advantage of the problem's rotational symmetry, which allows modeling in 2D using cylindrical coordinates as indicated in Figure 3-21. 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.

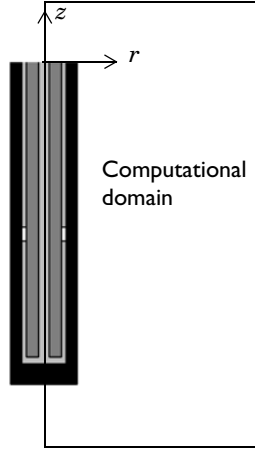


Figure 3-21: The computational domain appears as a rectangle in the rz -plane.

The radial and axial extent of the computational domain is in reality larger than indicated in Figure 3-21. 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

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

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

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

where z is the direction of propagation, and r , ϕ , and z are cylindrical coordinates centered on the axis of the coaxial cable. \mathbf{P}_{av} is the time-averaged power flow in the cable, Z is the wave impedance in the dielectric of the cable, while r_{inner} and r_{outer} are the dielectric's inner and outer radii, respectively. Further, ω denotes the angular frequency. The propagation constant, k , relates to the wavelength in the medium, λ , 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_ϕ :

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

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 $r = 0$.

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 ($\text{W}/(\text{m}\cdot\text{K})$), ρ_b represents the blood density (kg/m^3), C_b is the blood's specific heat capacity ($\text{J}/(\text{kg}\cdot\text{K})$), and ω_b denotes the blood perfusion rate ($1/\text{s}$). Further, Q_{met} is the heat source from metabolism, and Q_{ext} is an external heat source, both measured in W/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 ω_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 3-22 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°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.

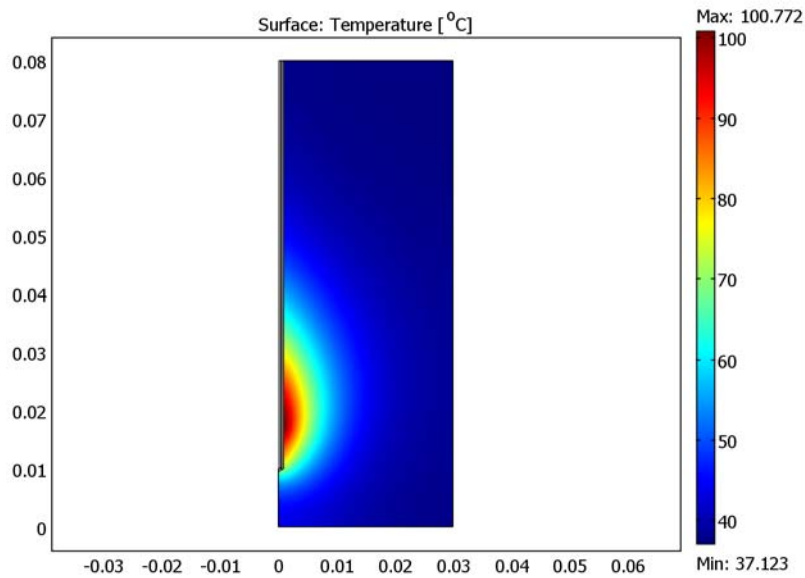


Figure 3-22: Temperature in the liver tissue.

Figure 3-23 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.

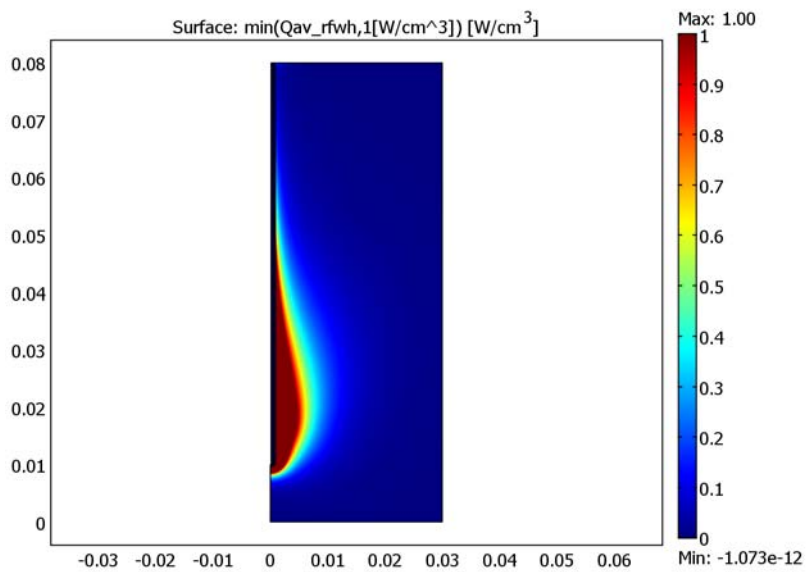


Figure 3-23: The computed microwave heat-source density takes on its highest values near the tip and the slot. The scale is cut off at $1 W/cm^3$.

Figure 3-24 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.

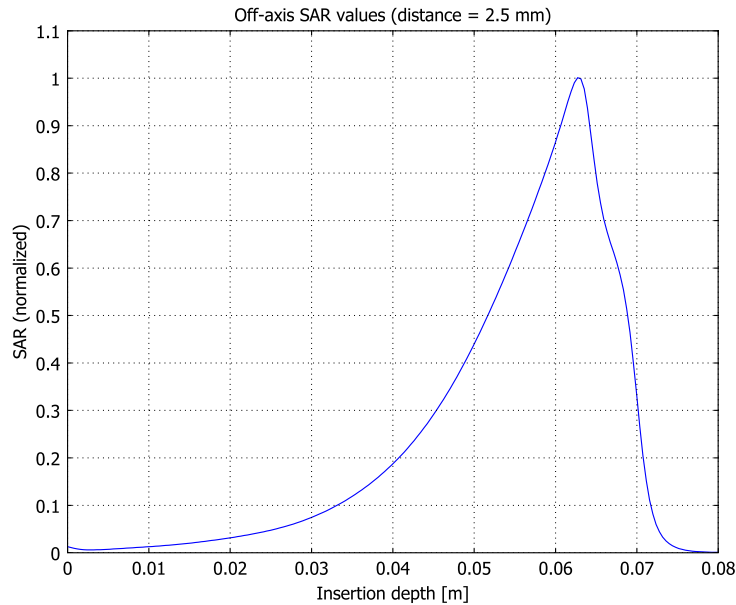


Figure 3-24: Normalized SAR value along a line parallel to the antenna and at a distance 2.5 mm from the antenna axis. The tip of the antenna is located at 70 mm, and the slot is at 65 mm.

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 Click the **Zoom Extents** button on the Main toolbar.
- 3 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**.
- 4 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
0.335e-3	0.0699	0.135e-3	0.0101

- 5 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**.

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}
ρ_b	ρ_{blood}
C_b	C_{blood}
ω_b	ω_{blood}
T_b	T_{blood}
Q_{met}	0
Q_{ext}	$Q_{\text{av_rfwh}}$

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_{met} to 0. The variable $Q_{\text{av_rfwh}}$ 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 Boundary 8 go to the **Port** page); when finished, click **OK**.

SETTINGS	BOUNDARIES 1, 3	BOUNDARIES 2, 14, 18, 20, 21	BOUNDARY 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
ϵ_r (isotropic)	eps_liver	eps_cat	eps_diel	1
σ (isotropic)	sig_liver	0	0	0
μ_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 3-22, 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 $\min(Q_{\text{av_rfwh}}, 1 [\text{W}/\text{cm}^3])$, then click **OK**.

In the resulting plot, which reproduces that in Figure 3-23, the region around the antenna in which the time-averaged resistive heating exceeds $1 \text{ W}/\text{cm}^3$ has a uniform, deep red color. Outside this region, you can read off the heating distribution from the color legend 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 3-24, 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 $Q_{\text{av_rfwh}}/3.01 [\text{W}/\text{cm}^3]$ in the **Expression** edit field. In both the **r0** and **r1** edit fields type $2.5\text{e-}3$; 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 σ is the conductivity of human brain tissue, ρ 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 256×256 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 55×50×50 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 acts 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 44 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 μ_r is the relative permeability, k_0 is the free-space wave vector, and ϵ_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
σ	835 MHz	1.35 S/m	Conductivity
ϵ_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.

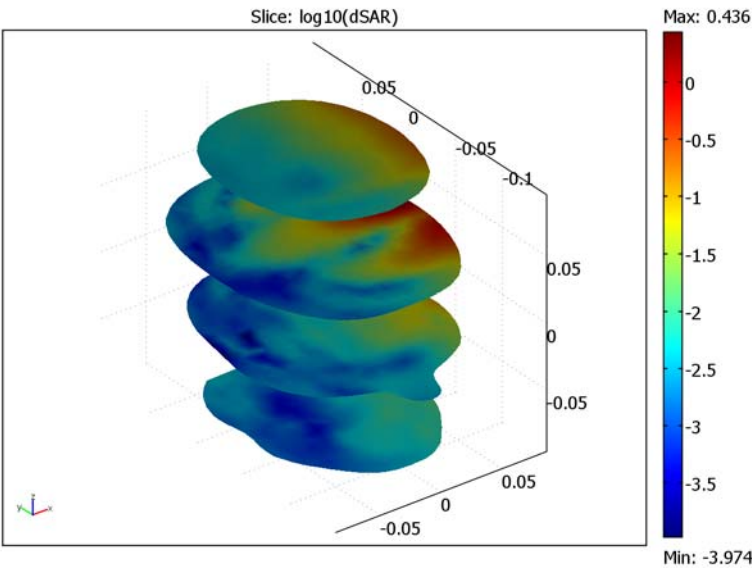


Figure 3-25: Log-scale slice plot of the local SAR value.

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.

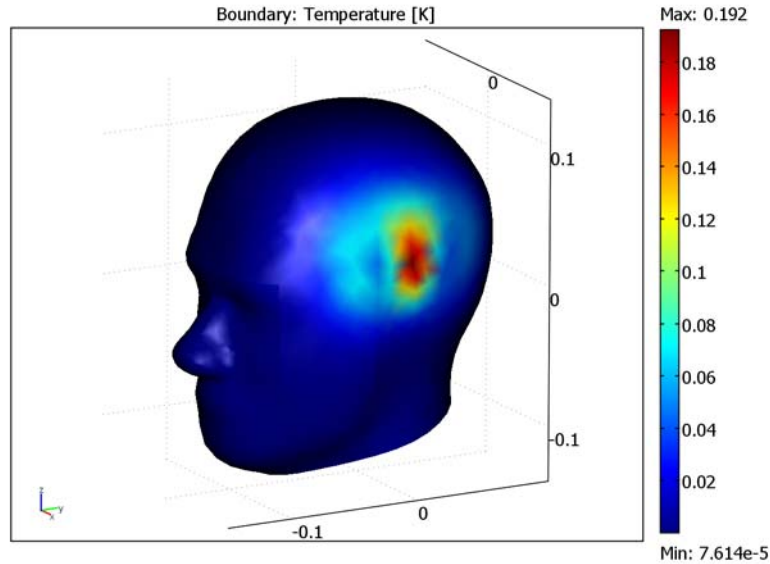


Figure 3-26: The local increase in temperature at the surface has a maximum of 0.2 °C right beneath the antenna.

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

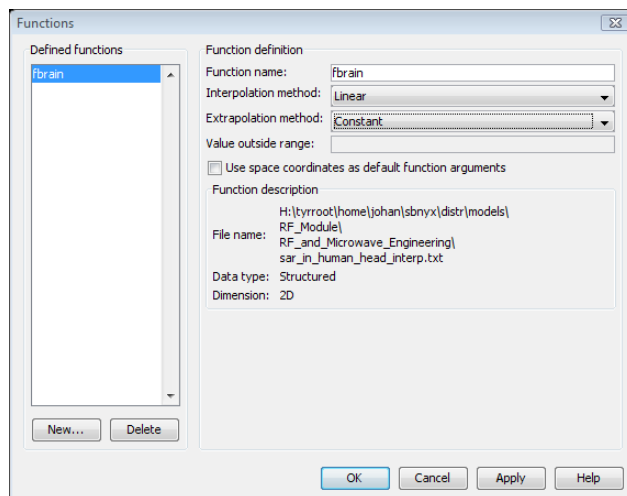
- 1 In the **Model Navigator**, select **3D** in the **Space dimension** list.
- 2 In the **RF Module** folder, select **Electromagnetic Waves>Harmonic propagation**.
- 3 Click the **Multiphysics** button.
- 4 Click the **Add** button.
- 5 In the **Heat Transfer Module** folder, select **Bioheat Equation>Steady-state analysis**.
- 6 In the **Dependent variables** edit field, type **dT**. 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 _{nr_pcb}	5.23	Permittivity for the patch antenna board
epsilon _{nr0_brain}	56	Permittivity for the brain tissue
sigma _{0_brain}	1.35[S/m]	Conductivity for the brain tissue
rho _{brain}	1.03e3[kg/m ³]	Density of brain tissue
sdamping	2e-4	Sampling parameter
edamping	4e-4	Sampling parameter
soffset	-1.0[S/m]	Sampling parameter
offset	-50	Sampling parameter
rho _{blood}	1000[kg/m ³]	Density of blood
c _{blood}	3639[J/(kg*K)]	Heat capacity of blood
odamping	1.08e-6[1/s]	Sampling parameter
offset	-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** option button, and select **File** from the **Use data from** drop-down menu.
- 7 Click the **Browse** button and browse to the file named `sar_in_human_head_interp.txt` in the `models/RF_Module/RF_and_Microwave_Engineering` folder in the COMSOL installation directory. 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` in the `models/RF_Module/RF_and_Microwave_Engineering` folder in the COMSOL installation directory, and 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.

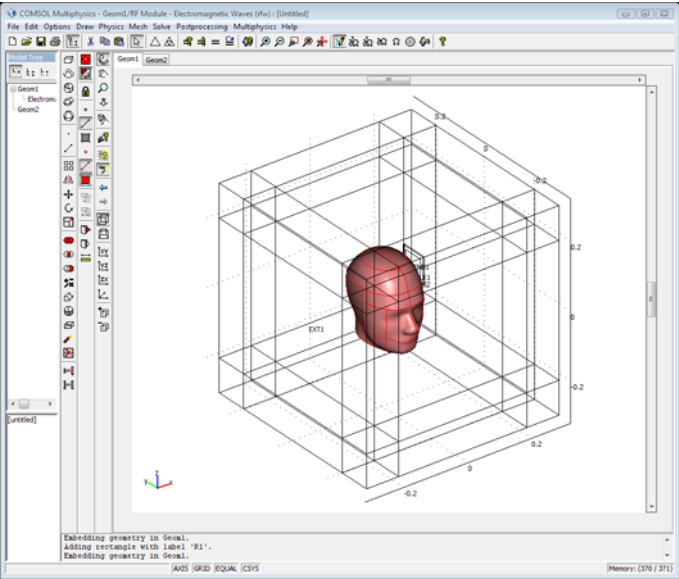
WIDTH	HEIGHT	X	Y
0.06	0.06	0.01	0.01
0.005	0.01	0.04-0.0025	0

- 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 **Scalar Variables**.
- 3 In the **Application 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 _r 0_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_{in}** edit field.
- 8 Click the **Port** tab and type 75 in the **Z_{ref}** 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 Ω.

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
ε _r	epsilon _r _brain	epsilon _r _pcb
σ	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
ρ_b	rho_blood
C_b	c_blood
ω_b	omega_head
T_b	0
Q_{ext}	Qav_rfw

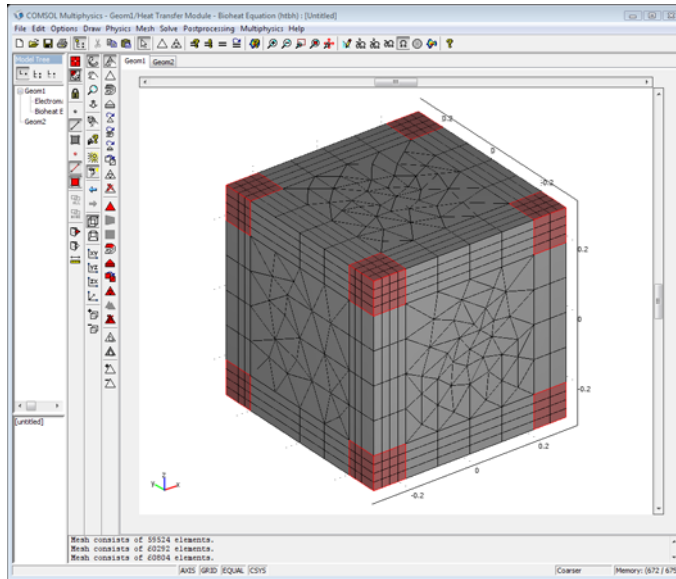
4 Click the **Init** tab and type 0 in the **dt(t₀)** edit field.

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. To do so, follow these steps:

- 1 Click the **Solver Manager** button on the Main toolbar.
- 2 On the **Solve For** page, select **Electromagnetic Waves (rfw)**. Click the **Solve** button.
- 3 When the solution for the electromagnetic wave has finished, select **Bioheat Equation (htbh)**.
- 4 Click the **Initial Value** tab, and then click the **Current solution** option buttons in both the **Initial value** and the **Values of variables not solved for and linearization point** areas.
- 5 Click the **Solve** button.
- 6 Click **OK** when the solution for the temperature has finished.

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 On the **Slice** page, type $\log_{10}(\text{dSAR})$ in the **Expression** edit field.
- 4 Enter 0 in the **x-levels** edit field and 10 in the **z-levels** edit field. Click **Apply**.
You should now see the plot in Figure 3-25 on page 144.
- 5 On the **Slice** page, clear the **Slice plot** check box.
- 6 Click the **Boundary** tab, select the **Boundary plot** check box, and type dT in the **Expression** edit field.
- 7 Click **OK** to see the plot in Figure 3-26 on page 145.

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 the oven, waveguide, potato, and plate. The reduced geometry is shown in Figure 3-27.

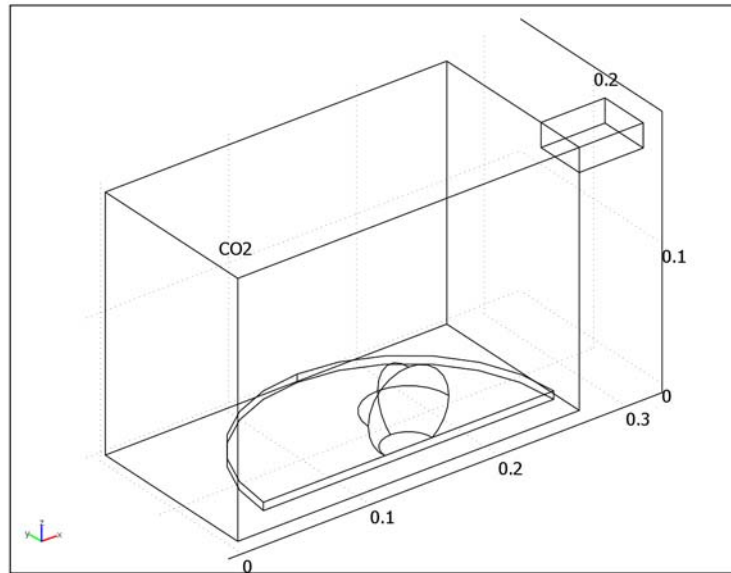


Figure 3-27: Geometry of microwave oven, potato, and waveguide feed.

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

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 TE₁₀ 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 TE₁₀ mode, $m = 1$ and $n = 0$. With the dimensions of the rectangular cross section ($a = 7.8$ cm and $b = 1.8$ cm), the TE₁₀ 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 μ_r denotes the relative permeability, j the imaginary unit, σ the conductivity, ω the angular frequency, ϵ_r the relative permittivity, and ϵ_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.

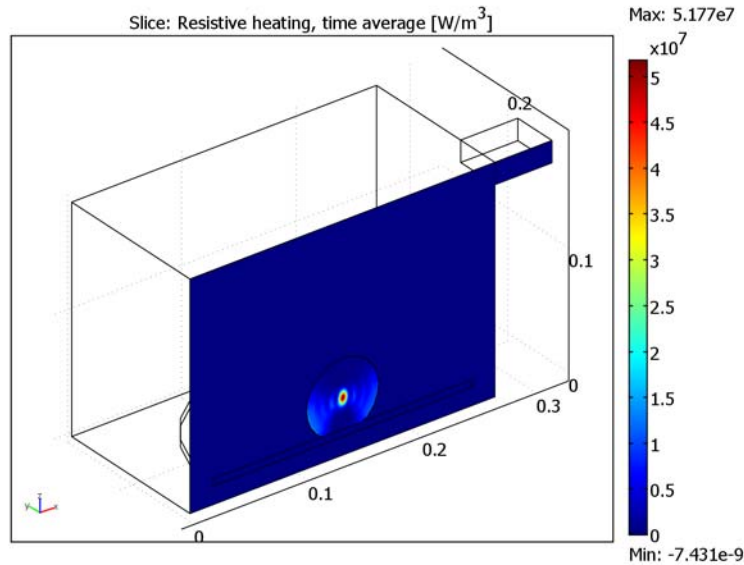


Figure 3-28: Dissipated microwave power distribution.

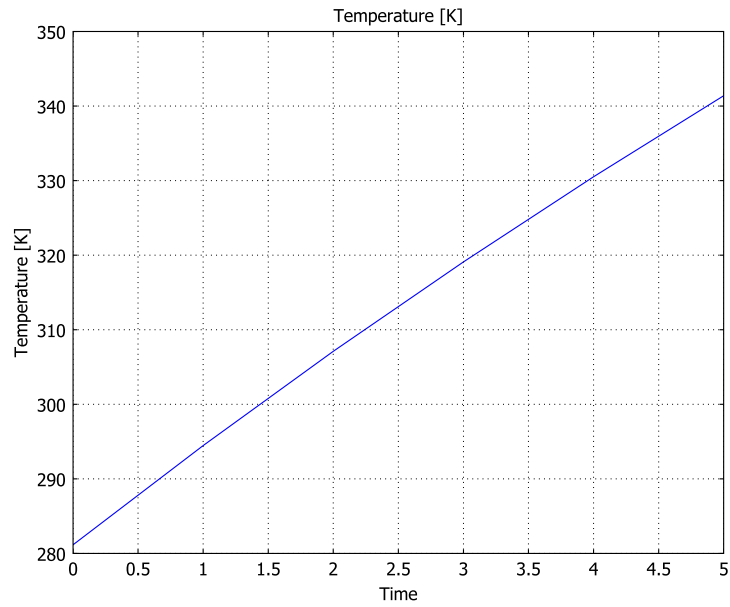


Figure 3-29: Temperature in the center of the potato during the first 5 seconds of heating.

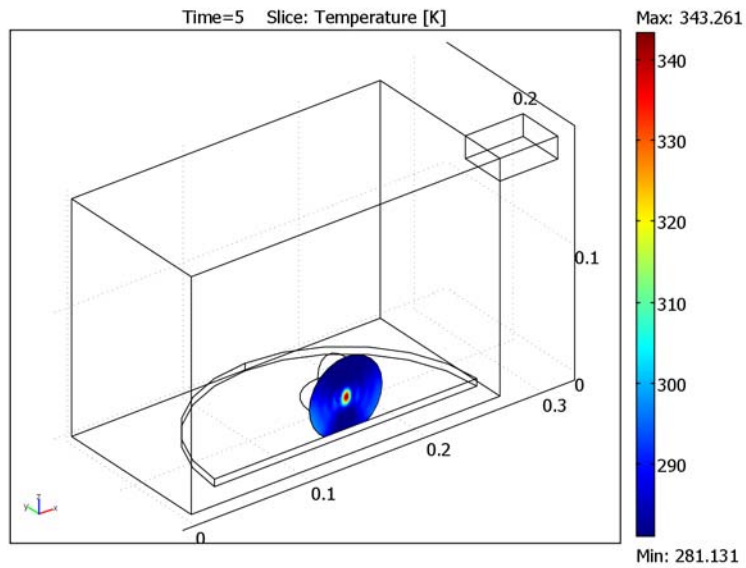


Figure 3-30: Temperature distribution after 5 seconds of heating.

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.

Electromagnetic Analysis

MODEL NAVIGATOR

- 1 Select **3D** from 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.

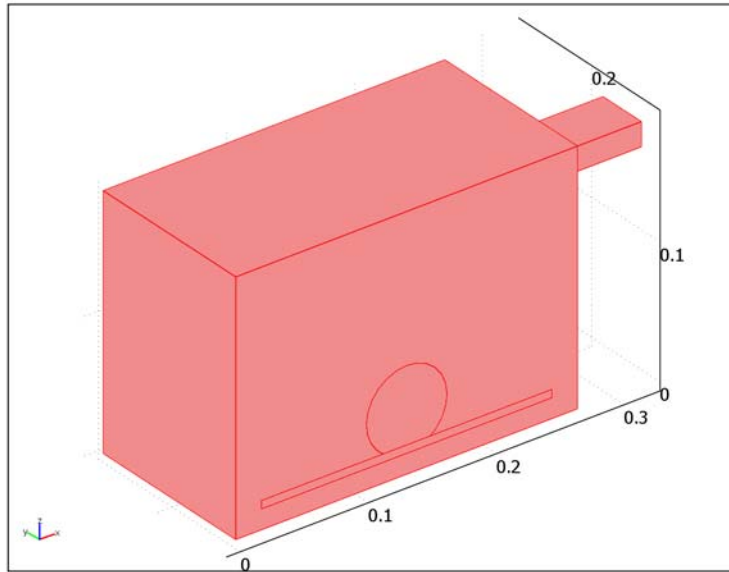
NAME	EXPRESSION	DESCRIPTION
nu_co	$c0_rfw / (2 * 7.8 [cm])$	Cutoff frequency

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 $2.45\text{e}9$.

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	$0 \ 0 \ \cos(\pi * (y - 13.5[cm]) / 7.8[cm])$
Specify cutoff frequency	selected
V_{cutoff}	nu_co

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

MESH GENERATION

- I 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 2** 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.
- II 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 on the Main toolbar.

COMPUTING THE SOLUTION

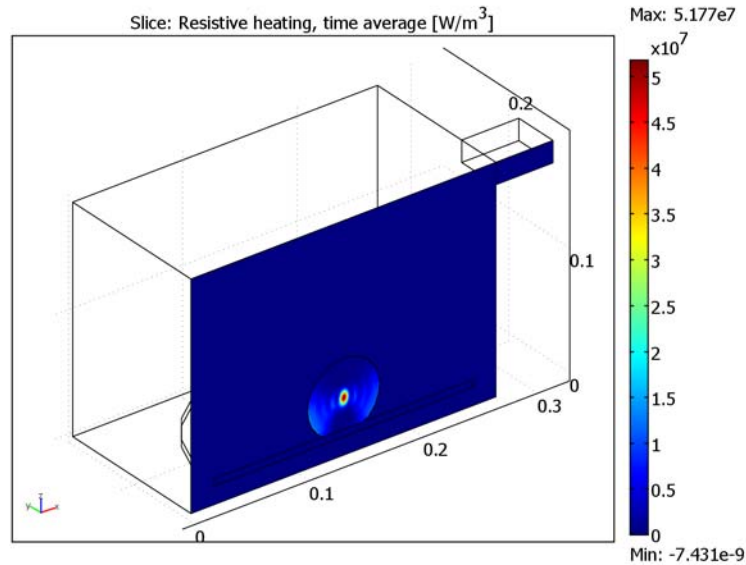
- 1 Open the **Solver Manager** dialog box and on the **Solve For** tab select only the **Electromagnetic Waves** application mode.
- 2 Click **OK** to close the **Solver Manager**.
- 3 In the **Solver Parameters** dialog box, with the **Auto select solver** check box selected, select **Stationary** from the list of analysis types for the **Heat Transfer by Conduction** (or **General Heat Transfer**) application mode in the **Analysis types** area.
- 4 Set **Linear system solver** to **GMRES** and **Preconditioner** to **Geometric Multigrid**.
- 5 Click the **Settings** button.
- 6 Select the **Preconditioner** in the tree on the left of the **Linear System Solver Settings** dialog.
- 7 Set the **Hierarchy generation method** to **Manual**.
- 8 Click **OK** to close the **Linear System Solver Settings** dialog.
- 9 Click **OK** to close the **Solver Parameters** dialog box.
- 10 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, S_{11_rfw} and S_{11dB_rf} which is the S-parameters on a dB scale. To evaluate S_{11dB_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 S_{11dB_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/m³, 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.

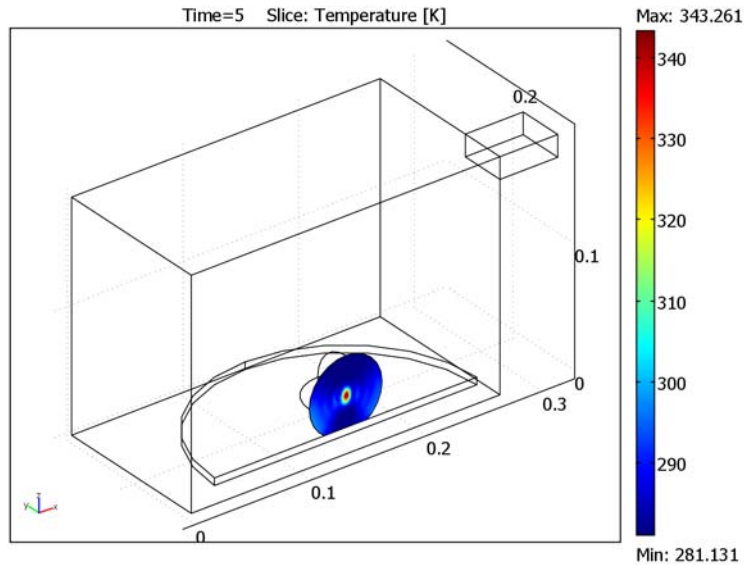
COMPUTING THE SOLUTION

- 1 Open the **Solver Manager** dialog box and on the **Solve For** page select only the **Heat Transfer by Conduction** (or **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 list of analysis types for the **Heat Transfer by Conduction** (or **General Heat Transfer**) application mode in the **Analysis types** area.
- 5 Set **Linear system solver** to **Direct (SPOOLES)**.
- 6 Under **Time stepping** set **Times** to range(0,5) and **Absolute tolerance** to 1.
- 7 Click **OK** to close the **Solver Parameters** dialog box.
- 8 Click the **Solve** button on the Main toolbar.

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 **Heat Transfer by Conduction (ht)>Temperature** (or **General Heat Transfer (htgh)>Temperature**) from 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 **Heat Transfer by Conduction (ht)>Temperature** (or **General Heat Transfer (htgh)>Temperature**) from 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 3-29 on page 159.

Microwave Filter on PCB

This model analyzes the transfer function of a low-pass filter on a printed circuit board.

Introduction

Microstrip filters can be fabricated directly on a printed circuit board (PCB) with a microstrip line going from the input to the output. Along the microstrip line there are a number of stubs of certain lengths and widths. The design of the filter involves choosing the impedance of the microstrip line, the impedance of the stub microstrips, and the length of the stubs. This particular filter is based on a textbook example from Ref. 2. It is also used as example in Ref. 1, which contains results from other simulation tools and methods and is freely available online. The filter has a seven-pole low-pass Chebyshev response with a cutoff frequency of 1 GHz. The entire layout including the dielectric layer is imported from an ODB++(X) file using the ECAD Import feature.

Model Definition

The model uses the Electromagnetic Waves application mode that solves the vector Helmholtz wave equation. The PCB layout of the Chebyshev filter is imported using the ECAD Import feature, which creates the entire 3D geometry from the layout and stackup information of an ODB++ file. It also creates useful groups in the dialog boxes for subdomain and boundary settings.

The cutoff frequency of the filter is 1 GHz by design, and the dielectric layer of the PCB has a relative permittivity of 10.8. The metal layers are modeled as perfect electric conductors with zero thickness, thereby avoiding a dense meshing of thin conductive layers. The ECAD import has an option that causes the import engine to ignore the thickness of metal layers and insert them as faces on the dielectric layer. The width of the microstrip line is 0.1 mm and the width of the stubs is 5 mm.

The characteristics of the filter are sensitive to the placement and length of the stubs; therefore this model also analyzes the change in filter characteristics as a function of mechanical deformation. This is done by adding a Solid, Stress-Strain application mode together with a Moving Mesh application mode. The Moving Mesh application mode is necessary to enable the Electromagnetic Waves application mode to account for the deformation of the PCB. The deformation comes from a uniform load across the board with fixed input and output faces.

Because the filter cutoff should be close to 1 GHz, and its distortion caused by PCB bending is small, it is necessary to sweep the frequency near the cutoff, from 750 MHz up to 1.5 GHz. The first solution step performs this sweep for the Electromagnetics application mode without any mechanical deformation. Then the same sweep is performed with a mechanical deformation from a uniform load of 40 N. The Solid, Stress-Strain application mode calculates the deformation, and the Moving Mesh application mode applies this deformation to the coordinate system that the Electromagnetic Waves application mode use. Due to the one-way coupling between the three application modes, the Solid, Stress-Strain and Moving Mesh application modes can be solved more efficiently using the segregated solver. After this step, the frequency sweep is performed again for the Electromagnetic Waves application mode using the parametric solver. It is possible to solve for all three application modes simultaneously with the segregated solver. Doing so is unnecessary because the Solid, Stress-Strain and Moving Mesh application modes are frequency independent.

Results and Discussion

The purpose of this simulation is to analyze how the S-parameter curve changes when a force of 40 N is applied on the circuit board. This force significantly bends the PCB as you can see in Figure 3-31.

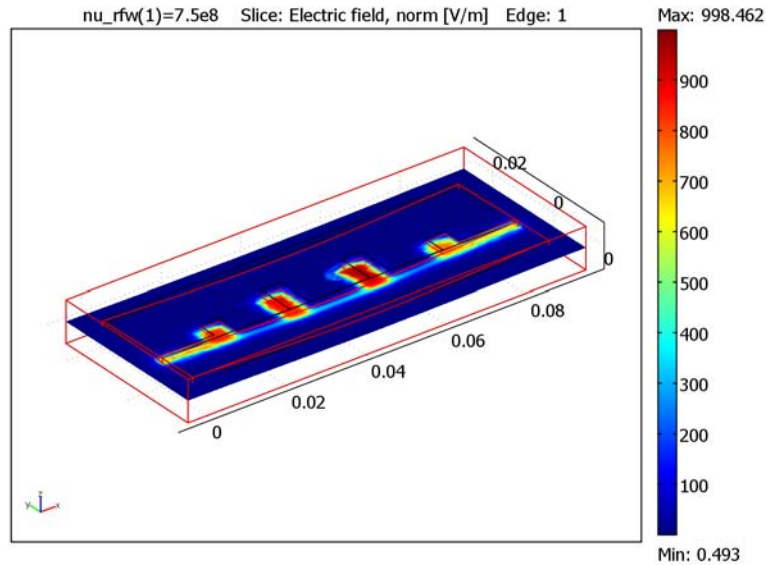


Figure 3-31: The black and red outlines of the geometry show the shape before and after the force has been applied. The figure also shows the norm of the electric field inside the PCB at 1.5 GHz.

Although the PCB deformation is fairly large the S-parameter curve does not change that much. The cutoff frequency is shifted about 10 MHz to a higher frequency when

the force is applied. Figure 3-32 displays the difference between the S-parameter curves with and without an applied force.

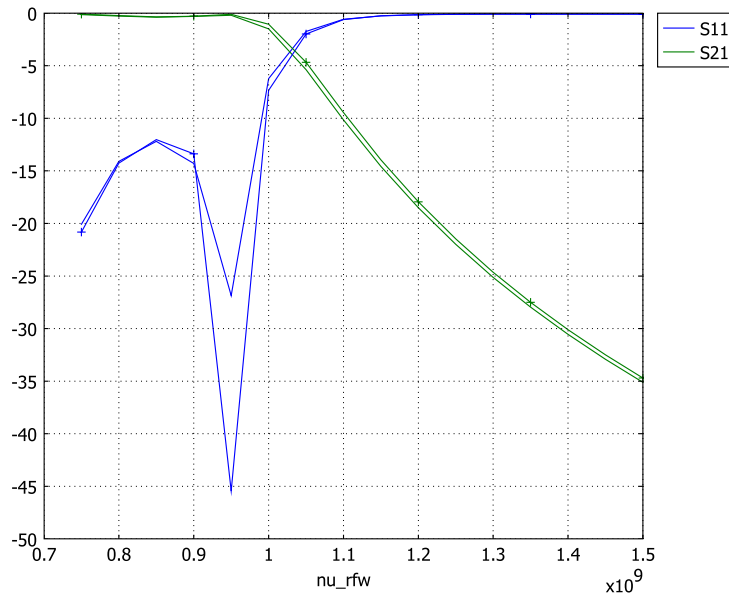


Figure 3-32: A comparison between the S11 and S21 parameters before and after a force of 40 N has been applied on the PCB. The S-parameter curves for the filter with an applied force are marked with a plus sign.

References

1. D.V. Tosic and M. Potrebic, "Software Tools for Research and Education," *Microwave Review*, vol. 12, no. 2, p. 45, 2006.
2. J.-S.G. Hong and M.J. Lancaster, *Microstrip Filters for RF/Microwave Applications*, John Wiley, 2001.

Model Library path: RF_Module/RF_and_Microwave_Engineering/
pcb_microwave_filter_with_stress

MODEL NAVIGATOR

- 1 From the **Space dimension** list, select **3D**.
- 2 Select the application mode **RF Module>Electromagnetic Waves**.
- 3 Click **OK** to close the **Model Navigator**.

GEOMETRY MODELING

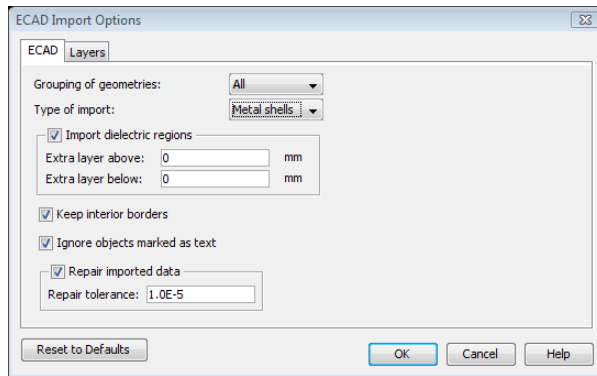
The major part of the geometry is imported using the ECAD Import tool, which automatically reads a PCB layout and extrude the layers to a 3D geometry.

- 1 Click the **Block** button on the Draw toolbar.
- 2 In the **Block** dialog box specify the dimensions according to the following table:

PROPERTY		VALUE
Length	(X, Y, Z)	(0.1, 0.04, 0.01)
Axis base point	(x, y, z)	(-0.005, -0.015, -0.005)

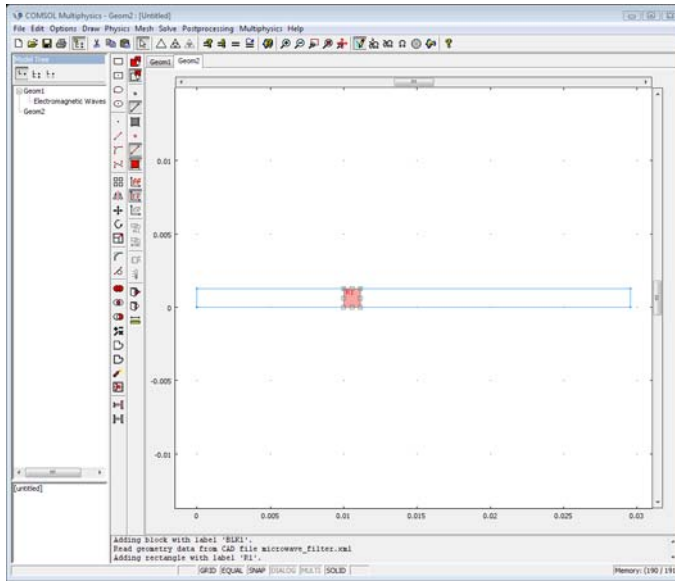
- 3 From the **File** menu, choose **Import>ECAD Import**.
- 4 In the dialog box that appears, browse to the file called `microwave_filter.xml`.
It is located in the same folder as specified in the Model Library Path on page 171.
This path originates from the folder `models` under the COMSOL Multiphysics installation folder.
- 5 Click **Import** when you have selected the file.

- 6 Then the **ECAD Import Options** dialog box appears. From the **Import type** list, choose **Metal shells**. This will import all metal layers as faces, which drastically reduce the problem size in this model without compromising the accuracy.



- 7 Click **OK** to start importing the geometry. After the geometry objects have been imported, the program automatically switch to Subdomain mode. It does this to add named subdomain and boundary groups, which helps you select the subdomains or boundaries that are electrically connected.
- 8 The PCB is now placed within the drawn block. The next step is to add boundary faces for the input and output ports. Choose **Work-Plane Settings** from the **Draw** menu.
- 9 In the **Work-Plane Settings** dialog box, go to the **Face Parallel** page. Browse to the vertical face of the PCB board that is closest to you (**CO1>2**).
- 10 Click **OK**. You now get a new 2D work plane where you can see the cross section of the 3D geometry in blue. Click the **Zoom Extents** button on the Main toolbar if the view is too large.
- 11 Make sure that point snapping is activated. It is if the word **SNAP** shows up black in the status bar at the bottom of the main window. If not, double-click on the text to activate it.
- 12 With point snapping activated, draw a rectangle with its upper corners coincident with the two points in the middle of the PCB cross section. These two points are part of the microstrip. Place the bottom side on the lower boundary of the PCB. When drawing, start with the upper-left corner and extend the rectangle so it only gets the correct height.

- 13 When the rectangle is drawn, select and drag the upper-right corner so it gets aligned with the right point on the microstrip. The rectangle should look like the figure below.



- 14 From the **Draw** menu, choose **Embed**.
- 15 Make sure that the rectangle is selected and click **OK**.
- 16 After embedding, you are back at the 3D geometry again. Open the **Work-Plane Settings** dialog box again.
- 17 Select the vertical face of the PCB board that is farthest from you (**COI>5**) and click **OK**.
- 18 You can re-use the rectangle you created earlier, so just choose **Embed** from the **Draw** menu and click **OK**.

OPTIONS AND SETTINGS

Constants

- 1 From the **Options** menu, choose **Constants**.
- 2 In the **Constants** dialog box, add one constant with the **Name** **Fload** defined by the **Expression** **40[N]**.
- 3 Click **OK**.

PHYSICS SETTINGS

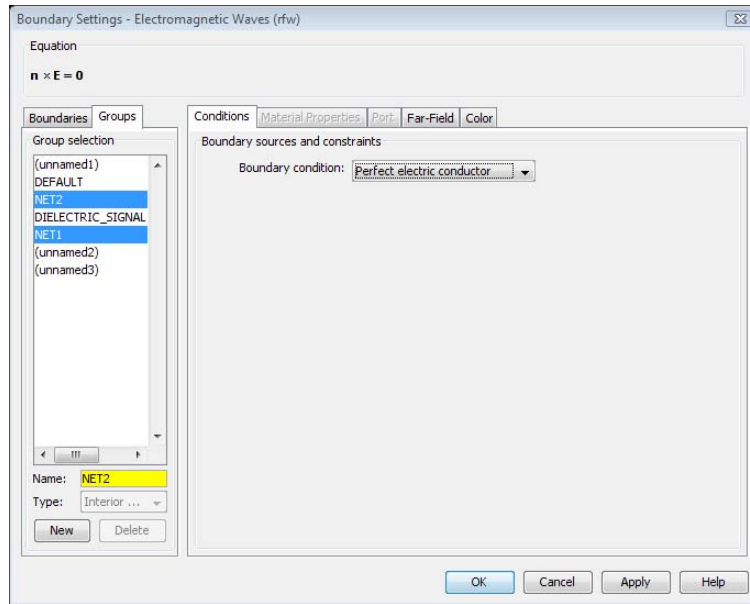
Subdomain Settings

- 1 Go to the **Physics** menu and choose **Subdomain Settings**.
- 2 In the **Subdomain Settings** dialog box, select Subdomain 2 and type 10.8 in the **Relative permittivity** edit field. Leave the other subdomain at its default value, which represents air.
- 3 Click **OK**.

Boundary Conditions

- 1 From the **Physics** menu, choose **Boundary Settings**.
- 2 In the **Boundary Settings** dialog box, select the **Select by group** check box and click on Boundary 1; this selects all outer boundaries.
- 3 From the **Boundary condition** list, choose the **Scattering boundary condition**.
- 4 Clear the **Select by group** check box and select the **Interior boundaries** check box.
- 5 Select Boundary 10 and choose the **Lumped port** boundary condition from the **Boundary condition** list.
- 6 Select the **Voltage generator at this port** check box to make this port act as input for the signal.
- 7 Select Boundary 16 and set the boundary condition to **Lumped port**.
- 8 Change the port number by typing 2 in the **Port number** edit field.
- 9 Go to the **Groups** page in the left part of the dialog box.
- 10 Select the groups NET1 and NET2. These groups are automatically generated by the ECAD Import and represent the two electrical nets present in the file. Set the

boundary condition **Perfect electric conductor** for these two groups. They represent the microstrip and the ground plane on the PCB.



II Click **OK**.

MESH PARAMETERS

- I From the **Mesh** menu choose **Free Mesh Parameters**.
- 2 Select **Coarse** from the **Predefined mesh sizes** list.
- 3 Go to the **Boundary** page and select boundaries 10 and 16. Type **1e-3** in the **Maximum element size** edit field to increase the accuracy of the S-parameter calculations.
- 4 Click **OK**. It is not necessary to create the mesh because the solver does this automatically prior to the first solution step.

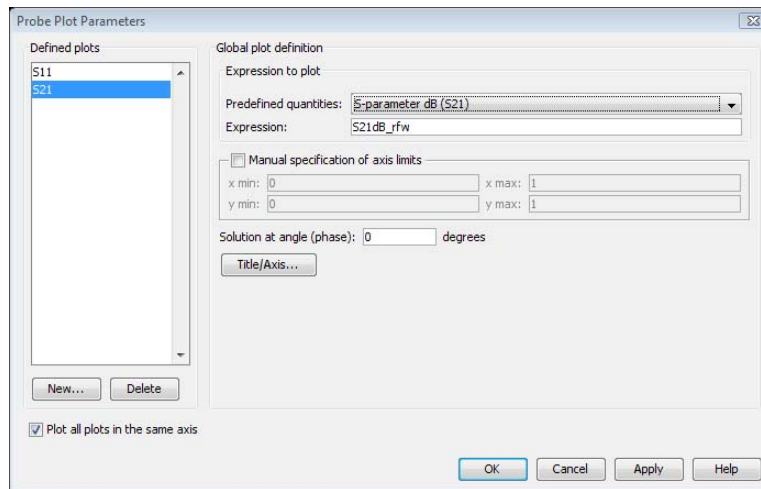
COMPUTING THE SOLUTION

Probe Plot Parameters

Probe plotting is a convenient technique to plot while solving, which is very useful for parameter sweeps. It is possible to discover problems before the solution step has finished and then stop the sweep to save time. It is also useful in situations when the

solver takes more steps than it stores in the output. The probe plot contains all steps that the solver takes.

- 1 From the **Postprocessing** menu, choose **Probe Plot Parameters**.
- 2 In the **Probe Plot Parameters** dialog box, click the **New** button.
- 3 In the **New Probe Plot** dialog box, choose **Global** from the **Plot type** list. Type S11 in the **Plot name** edit field and click **OK**.
- 4 From the **Predefined quantities** list, choose **Electromagnetic Waves (rfw)>S-parameter dB (S11)**.
- 5 Repeat the same procedure to add a variable with name S21 and plotting the quantity **S-parameter dB (S21)**.
- 6 Select the **Plot all plots in the same axis** check box and then click **OK**.



Solver Parameters

- 1 Go to the **Solve** menu and select **Solver Parameters**.
- 2 In the **Solver Parameters** dialog box, select **Parametric** in the **Solver** area.
- 3 Type nu_rfw in the **Parameter names** edit field and range (750e6, 50e6, 1.5e9) in the **Parameter values** edit field. This will tell the solver to perform a sweep from 750 Mhz up to 1.5 GHz with a step size of 50 MHz. The variable nu_rfw is the application mode variable for frequency that you manually can change from the **Application Scalar Variables** dialog box. Using this variable as parameter name will override the entered value in that dialog box.
- 4 Click **OK**.

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

Mechanical Stress Analysis

MODEL NAVIGATOR

- 1 From the **Multiphysics** menu choose **Model Navigator**.
- 2 In the **Model Navigator** dialog box, select the application mode **COMSOL Multiphysics>Structural Mechanics>Solid, Stress-Strain**.
- 3 Click the **Add** button.
- 4 Then select **COMSOL Multiphysics>Deformed Mesh>Moving Mesh (ALE)** and click the **Add** button.
- 5 Click **OK** to close the dialog box.

Subdomain Settings (Moving Mesh (ALE), ale)

- 1 Choose the **Multiphysics** menu and make sure that the **Moving Mesh (ALE) (ale)** application mode is active.
- 2 Go to the **Physics** menu and choose **Subdomain Settings**.
- 3 In the **Subdomain Settings** dialog box, select subdomain 2, click the **Physics induced displacement** button, and type u , v , and w in the three edit fields labeled **dX**, **dY**, **dZ**. Subdomain 1 keeps its default setting, that is **Free displacement**.
- 4 Click **OK**.

Boundary Settings (Moving Mesh (ALE), ale)

- 1 Open the **Boundary Settings** dialog box from the **Physics** menu.
- 2 Select Boundaries 1–5 and 18, then select the check boxes **dx**, **dy**, and **dz**. Leave the default value of zero in the corresponding edit fields.
- 3 Select the **Select by group** check box and click on boundary 6 to select all boundaries adjacent to the PCB.
- 4 Select the check boxes **dx**, **dy**, and **dz**. Type u , v , and w in the edit fields next to the check boxes.
- 5 Click **OK**.

Subdomain Settings (Solid, Stress-Strain, sld)

- 1 Choose the **Multiphysics** menu and make the **Solid, Stress-Strain (sld)** application mode active.
- 2 Open the **Subdomain Settings** dialog box.

- 3 Select Subdomain 1 and clear the **Active in this domain** check box.
- 4 Select Subdomain 2 and click the **Load** button to open the **Materials/Coefficients Library** dialog box.
- 5 Expand the **Basic Material Properties** library and choose the **FR4 (Circuit Board)** material. The PCB material of the microwave filter is not FR4 but its mechanical properties are used here as an approximation.
- 6 Click **OK** to select the material for Subdomain 2 and to close the dialog box.
- 7 Go to the **Load** page and type $-Fload/3.357289e-6$ in the **Fz** edit field.
The denominator in this expression is the volume of the PCB board. You can calculate this by clicking the **Geometric Properties** toolbar button when Subdomain 2 is selected, which is the PCB. The volume appears in the message log in the bottom of the main window, and then you can simply copy and paste it into the expression.
- 8 Click **OK** to close the **Subdomain Settings** dialog box.

Boundary Settings (Solid, Stress-Strain, sld)

The port boundaries are mechanically attached to a surrounding structure and cannot move. You must therefore specify zero displacements for these boundaries.

- 1 Open the **Boundary Settings** dialog box.
- 2 Select Boundaries 10 and 16 (the port boundaries).
- 3 Select the **R_x**, **R_y**, and **R_z** check boxes to specify zero displacements in all directions.
- 4 Click **OK**.

COMPUTING THE SOLUTION

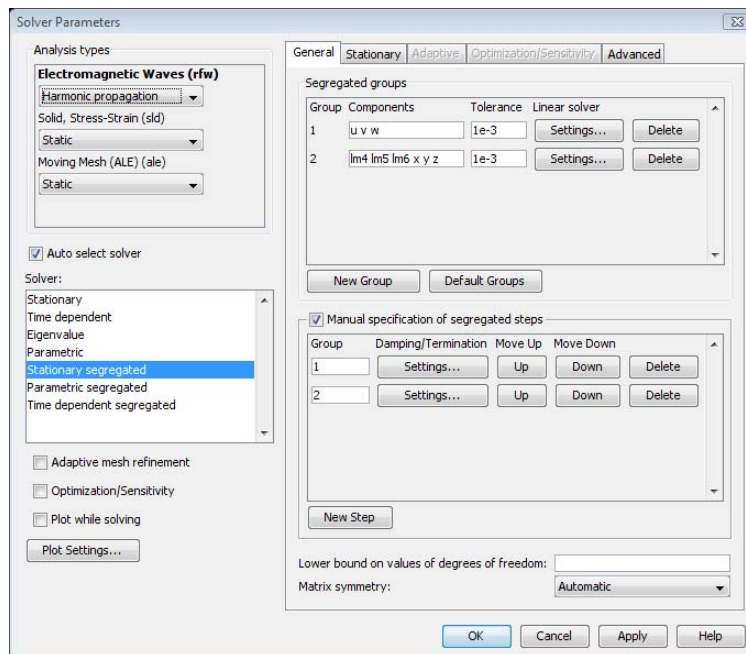
Solver Manager

- 1 From the **Solve** menu, choose **Solver Manager**.
- 2 In the **Solver Manager** dialog box, locate the frame labeled **Values of variables not solved for and linearization point**. Then click the **Current solution** option button in that frame. This tells the solver to keep the solution for all variables that it does not solve for.
- 3 Go to the **Solve For** page and select **Solid, Stress-Strain (sld)** and **Moving Mesh (ALE) (ale)**. Make sure that **Electromagnetic Waves (rfw)** is not selected.
- 4 Click **OK**.

Solver Parameters

- 1 From the **Solve** menu, choose **Solver Parameters**.

- 2 In the **Solver** area, select **Stationary segregated**.
- 3 In the **Segregated groups** area, locate the group with the dependent variables from the Electromagnetic Waves application mode (begins with **tExEyEz10**) and click its **Delete** button.
- 4 Then click the **Settings** button for the first group in the list.
- 5 In the dialog that appears, choose **Direct (PARDISO)** from the **Linear system solver** list.
- 6 Click **OK**.
- 7 Repeat Steps 4 through 6 for the second group.
- 8 Select the **Manual specification of segregated steps** check box. Then click the **Settings** button for group 1 beneath this check box.
- 9 In the dialog box that appears, type 1 in the **Damping constant** edit field.
- 10 Click **OK**.
- 11 Do the same thing for group 2. This ensures that the full step is taken for each segregated solve steps, which is most efficient for one-way coupled problems like this one.



- 12 Click **OK** to close the **Solver Parameters** dialog box.

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

Application Mode Properties

Before solving for the electromagnetic waves, you must define the Electromagnetics Waves application mode in the moving frame in order to see the displacements from the previous calculation.

- 1 From the **Multiphysics** menu, choose **Electromagnetic Waves (rfw)**.
- 2 From the **Physics** menu, choose **Properties** to open the **Application Mode Properties** dialog box.
- 3 Select **Frame (ale)** from the **Frame** list.
- 4 Click **OK**.

Solver Manager

- 1 From the **Solve** menu, choose **Solver Manager**.
- 2 In the **Solve For** page, select **Electromagnetic Waves (rfw)** and make sure that the other two application modes are not selected.
- 3 Click **OK**.

Solver Parameters

- 1 From the **Solve** menu, choose **Solver Parameters**.
- 2 In the **Solver** area, select **Parametric** then click **OK**.
- 3 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

- 1 From the **Postprocessing** menu, choose **Plot Parameters**.
- 2 In the **Plot Parameters** dialog box, go to the **Slice** page. Choose **Electric field, norm** from the **Predefined quantities** list.
- 3 Type 0 in the edit field next to the **X-levels** button, and then click the **Vector with coordinates** option button for **Z levels**. Type $1e-4$ in the edit field next to this button.
- 4 Go to the **Edge** page, select the **Edge plot** check box, and type 1 in the **Expression** edit field.
- 5 Click the **Uniform color** button.
- 6 Go to the **General** page, and select the first frequency (750 MHz) from the **Parameter value** list.
- 7 Clear the **Element refinement** button and type 3 in the edit field next to it.
- 8 Click **OK** to see the plot in Figure 3-31 on page 170.

The probe plotting leaves an open figure window when the final step is done, so you should have one from the first sweep showing the S-parameters without any applied force. To compare this S-parameter curve with the ones with stress you just plot the S-parameters in the old figure and keeping the old plot. The steps below assume that the S-parameter plot without force is in the plot labeled **Figure 1**.

- 9** From the **Postprocessing** menu, open the **Global Variables Plot** dialog box.
- 10** Choose the variables **S-parameter dB (S11)** and **S-parameter dB (S21)** from the **Predefined quantities** list. Then add them to the **Quantities to plot** list by clicking the **>** button.
- 11** Select the **Keep current plot** check box, and choose **Figure 1** from the **Plot in** list.
- 12** Click the **Line Settings** button.
- 13** In the dialog box that appears, choose **Plus sign** from the **Line marker** list. Click **OK** to close the dialog box.
- 14** Click **OK** to see the comparison shown in Figure 3-32 on page 171.

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 3-33 shows the antenna that this model simulates.

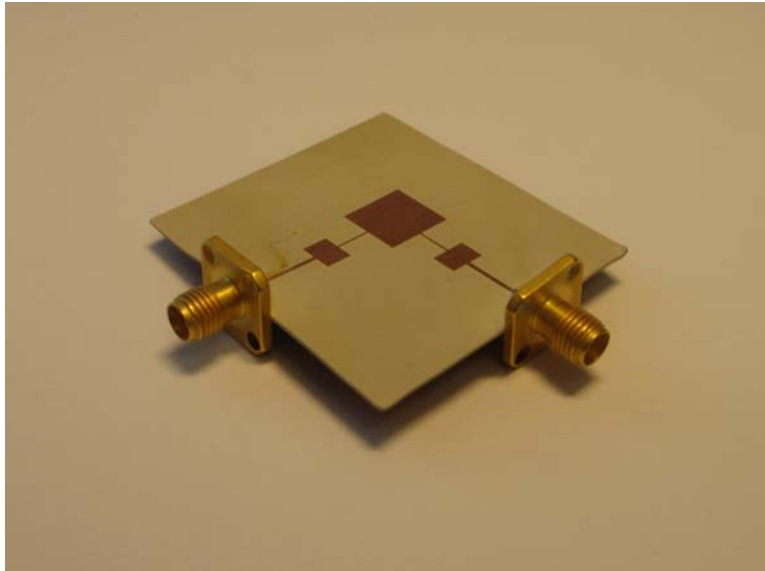


Figure 3-33: A photo of the real antenna that the model extracts the properties for.

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 3-34 below.

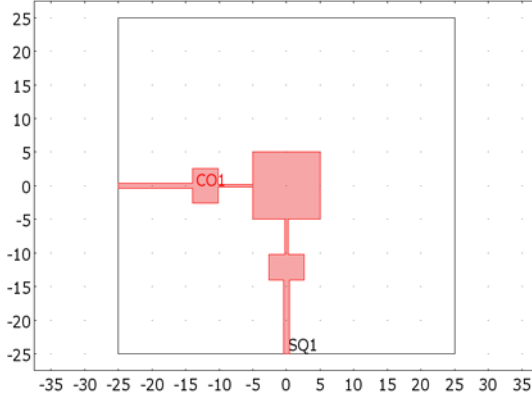


Figure 3-34: The printed pattern of the patch antenna shown as a top view. The large square is 10 mm by 10 mm, the smaller rectangles are 5.2 mm by 3.8 mm, the thicker lines are 0.6 mm wide, and the thinner lines are 0.2 mm wide and 5.2 mm long. The entire pcb is 50 mm by 50 mm and 0.7 mm thick.

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 Ω . 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 44 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_{PML} , and the length of the PML layer, d_{PML} , using the following formula:

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

For frequencies around 6 GHz the damping parameter α 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.

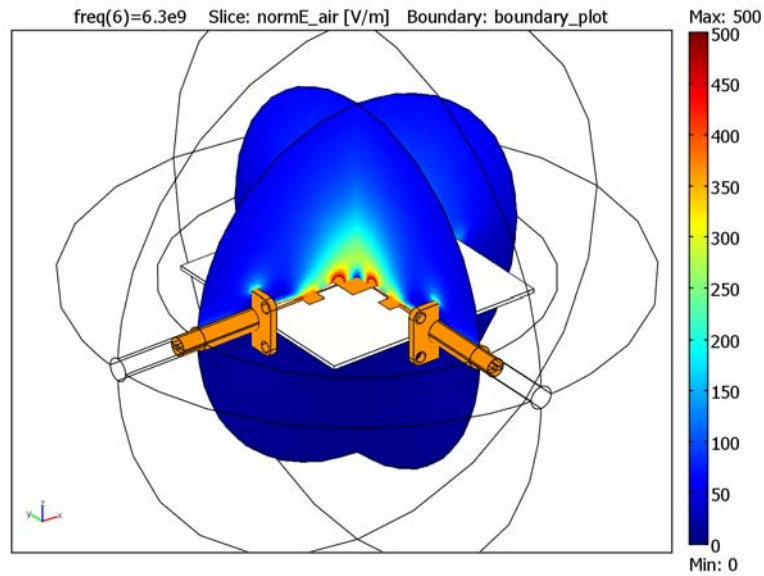


Figure 3-35: The final geometry of the patch antenna plotted together with a slice plot of the electric field in the surrounding air. The spherical PML is located outside the air domain and outside all cable regions.

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 3-36 this efficiency is plotted against the frequency, showing that the optimum frequency is located at 6.3 GHz.

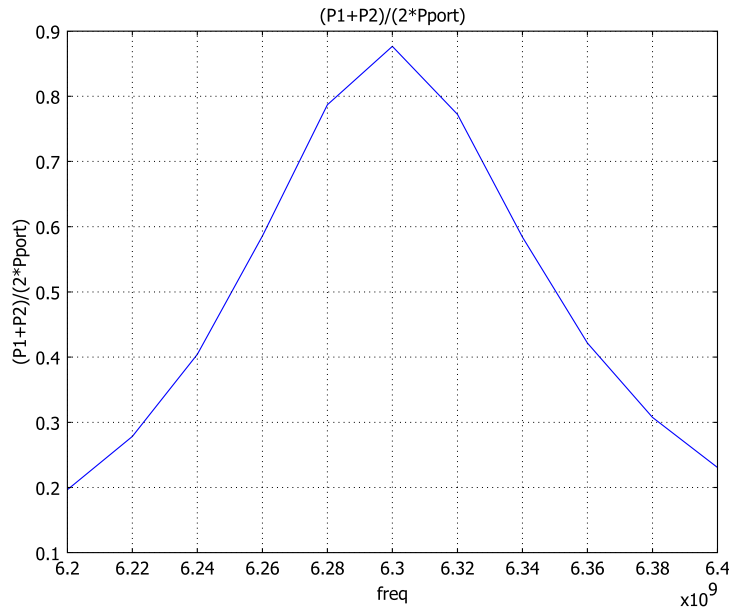


Figure 3-36: A resolved sweep in the range around 6 GHz, where the region between 6.2 GHz and 6.4 GHz was resolved in detail to locate the peak.

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 3-37 for the optimum frequency, 6.3 GHz.

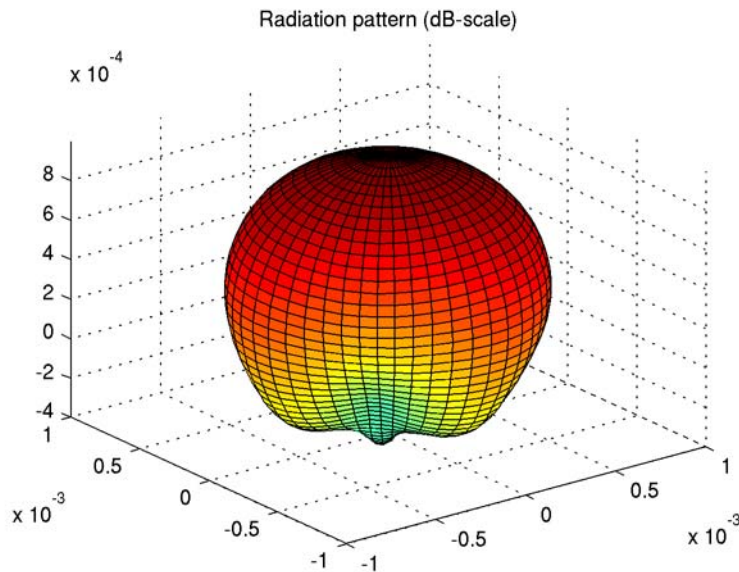


Figure 3-37: The radiation pattern plotted as a lobe pattern, using the deformed plot feature. It is clear that the wave mainly propagates in two lobes, with minimal radiation out of the backside of the PCB.

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>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, expressions, and (optionally) descriptions; when done, click **OK**.

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_r_coax	2.07	Rel. permittivity of the insulator in the cable
epsilon_r_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

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 188. 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 194.

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, then 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, type -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 CO1, 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** option 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)
C1	1.25	Center	(12.5/2-1.75, 12.5/2-1.75)

- 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.

4 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

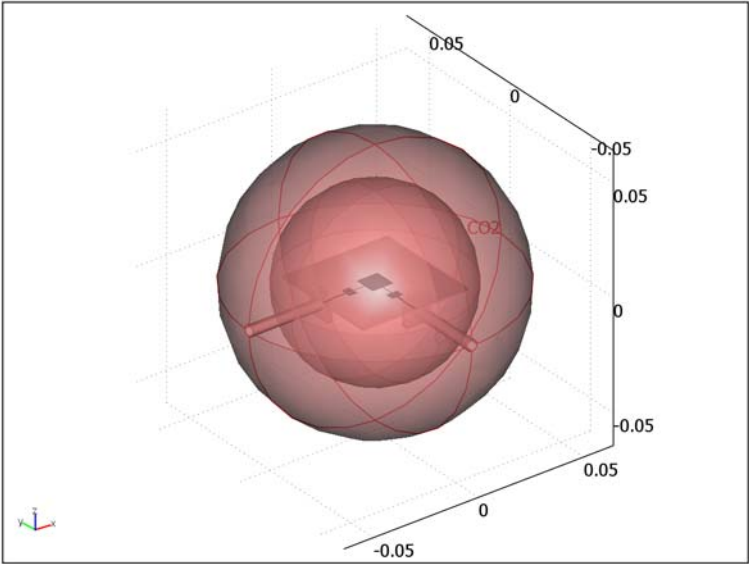
The entire geometry is now completed in the `patch_antenna` geometry. To help the software coerce the geometry to its minimal regions (subdomains) you can do step-by-step union and difference operations until there is only two solid objects (and one face object). Do all the operations below in the specified 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	$\sqrt{1 - P1 / P_{port}}$	S11 parameter for first input
Z_coax	$\sqrt{\mu_0 \cdot r_{fw} / (\epsilon_0 \cdot r_{fw} \cdot \epsilon_{r_coax})} / (2 \cdot \pi) \cdot \log(b_coax / a_coax)$	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 an 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	-nP0av_rfw		
P2		-nP0av_rfw	

10 Click **OK**.

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

12 In the **Application 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 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
ϵ_r	epsilon _r _coax	epsilon _r _pcb	1
μ_r	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 that one wavelength fits into the width of the PML. Using the free-space wavelength, λ_0 , instead makes the program assume that

the width of the PML is equal to λ_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.

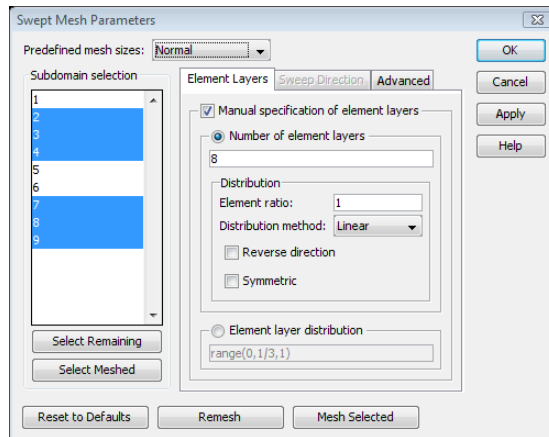
MESH GENERATION

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**.

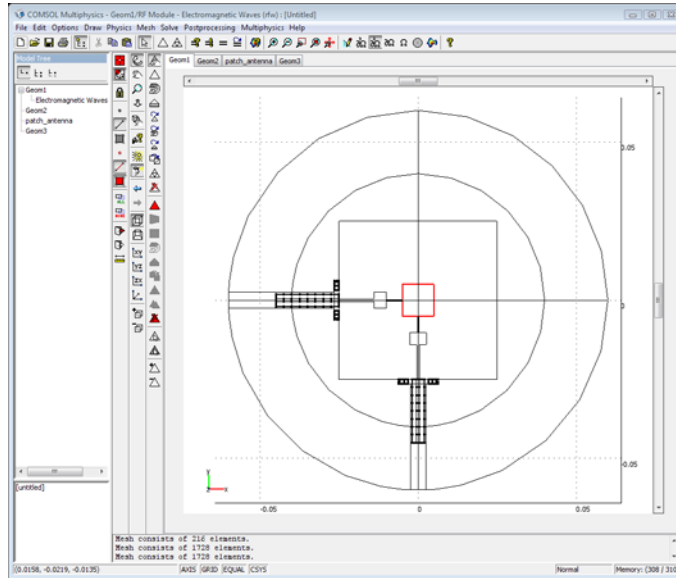
- 8 In the Swept Mesh Parameter dialog box, select Subdomains 2–4 and 7–9. On the **Element layers** page, select the **Manual selection of element layers** check box and set the **Number of element layers** to 8.



- 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.

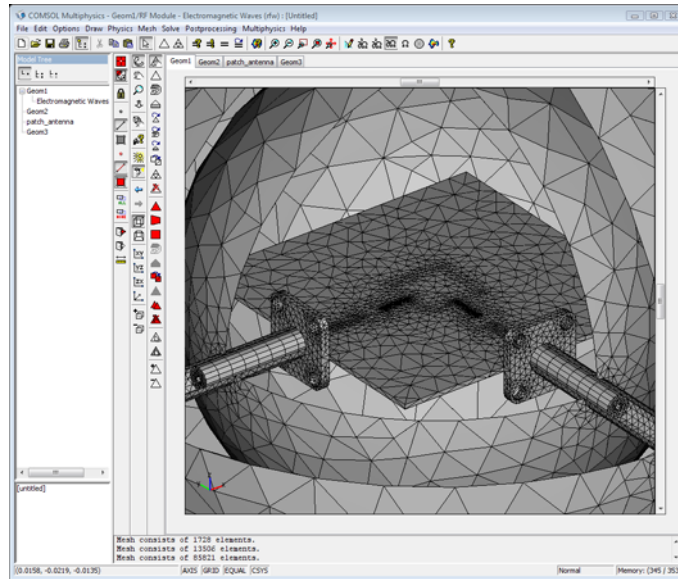


- 19 Click the **Boundary** tab, select Boundary 70 and click the **Mesh Selected** button.
- 20 Click the **Global** tab and type 0.5 in the **Resolution of narrow regions** edit field.
- 21 Click the **Advanced** tab and type 2 in the **z-direction scale factor** edit field.
- 22 Click the **Subdomain** 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.

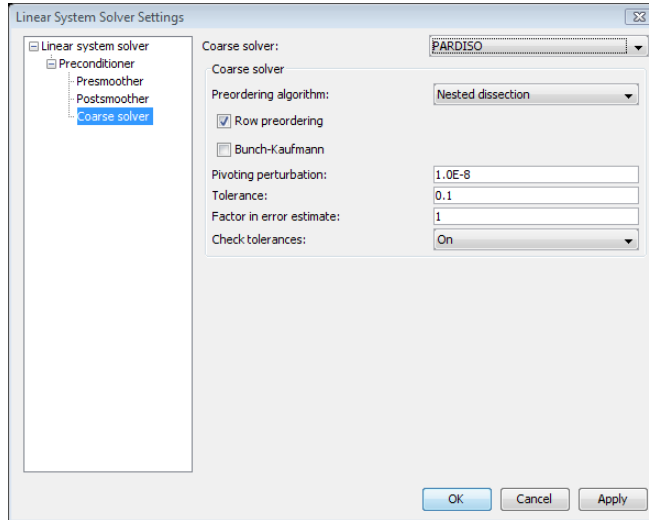


COMPUTING THE SOLUTION

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 names** edit field type `freq`, and in the **Parameter values** edit field type range `(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. From the **Color table** list, select **Thermal**.

- 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 3-35 on page 186.

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 3-36 on page 187.

RADIATION PATTERN PLOT IN MATLAB

The final step is to plot the radiation pattern shown in Figure 3-37 on page 188. These steps assume that you run COMSOL Multiphysics with MATLAB.

- 1 Press CTRL+F to export the entire model to the MATLAB workspace.
- 2 Switch to the MATLAB 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 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,'normEfardB',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 option to make a nicer plot. The plot should look like Figure 3-37 on page 188.

```

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.

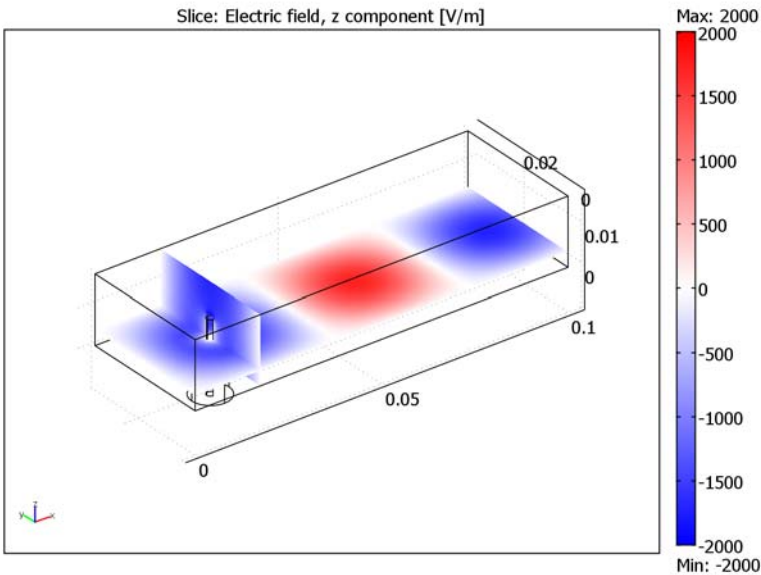
Results and Discussion

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
S1I	-8.6 dB	Reflected wave into the coaxial cable
S2I	-0.65 dB	Wave exiting the rectangular waveguide
S3I	-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_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 $4 \times Sdz_guess_rfw$ 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. Set the **Number of element layers** to 5.
- 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** option button, type 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.

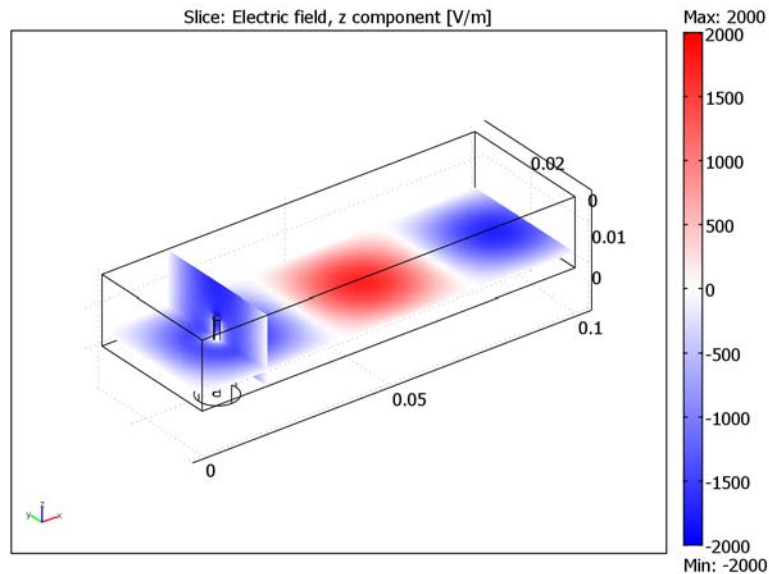
COMPUTING THE SOLUTION

- 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, type -2000 and 2000 in the **Min** and **Max** edit fields, respectively.
- 5 Click **OK**.
- 6 Choose the **WaveLight** color table 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 = -8.59dB
S21 = -.64dB
S31 = -22.23dB

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 μ_r denotes the relative permeability, j the imaginary unit, σ the conductivity, ω the angular frequency, ϵ_r the relative permittivity, and ϵ_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.

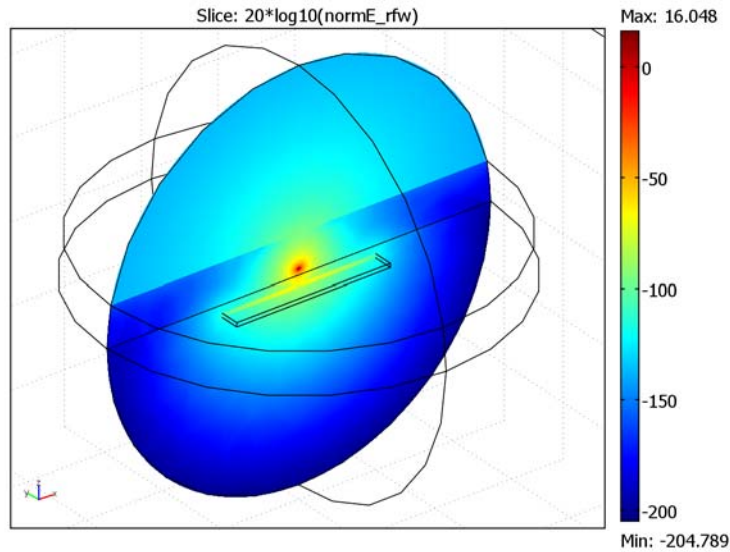


Figure 3-38: The electric field magnitude is plotted in a slice containing the antenna (red spot). The guiding effect of the hydrocarbon layer is clearly visible.

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.

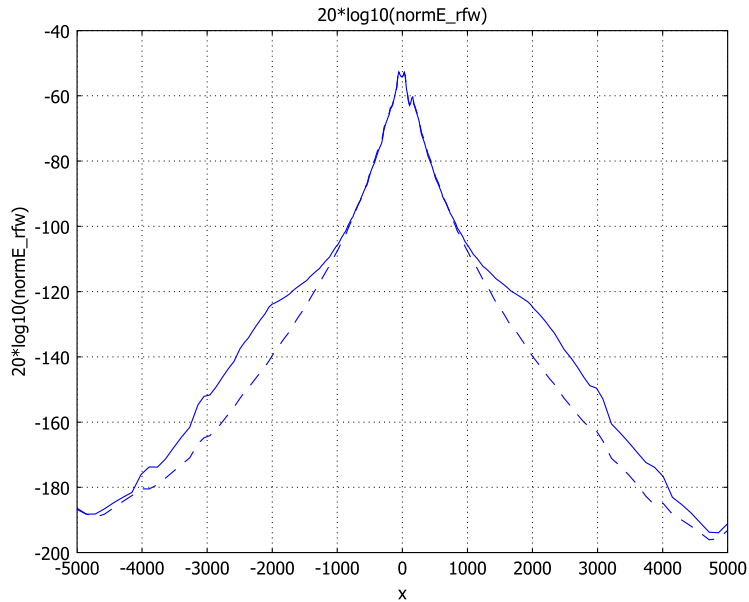


Figure 3-39: Electric field magnitude on the sea floor as a function of distance. The dashed line represents the case without a hydrocarbon layer.

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

Modeling Using the Graphical User Interface

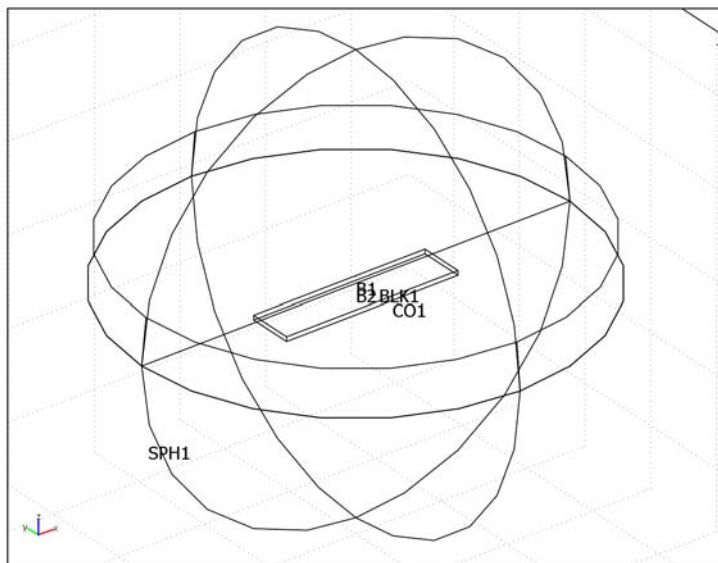
MODEL NAVIGATOR

- I 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

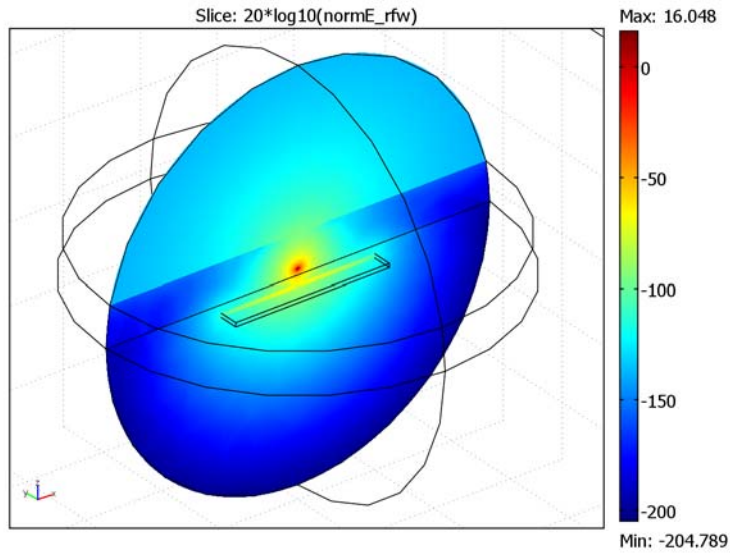
Click the **Solve** button on the Main toolbar.

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.

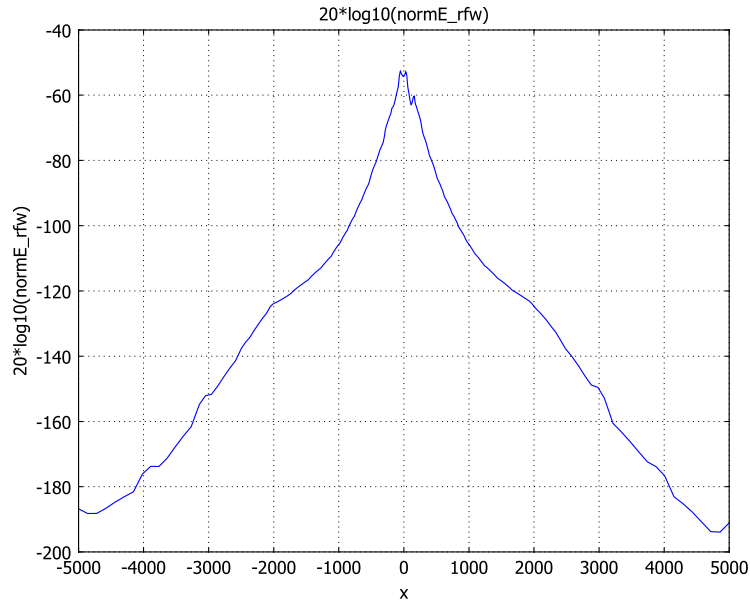
- 1 Open the **Plot Parameters** dialog box.
- 2 Click the **Slice** tab and type $20 \cdot \log_{10}(\text{normE_rfw})$ in the **Expression** edit field.
- 3 In the **Slice positioning** area, set the number of **x levels** and **z levels** to 0 and number of **y levels** to 1.

- 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**.

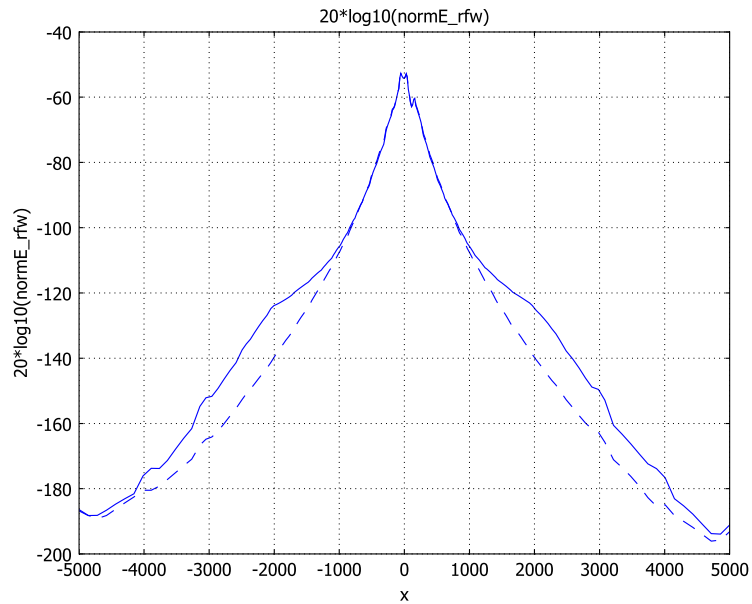
COMPUTING THE SOLUTION

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.



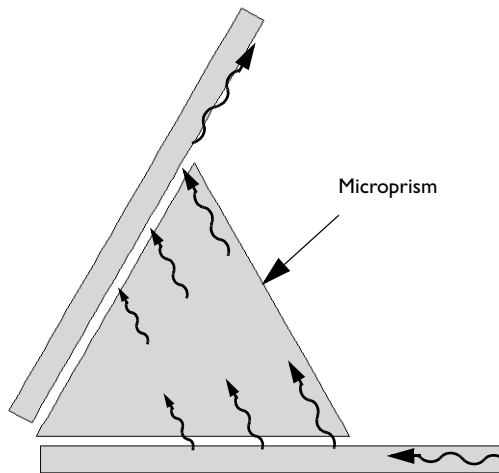
Optics and Photonics Models

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 44 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 μ_r denotes the relative permeability, ω the angular frequency, σ the conductivity, ϵ_0 the permittivity of vacuum, ϵ_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.

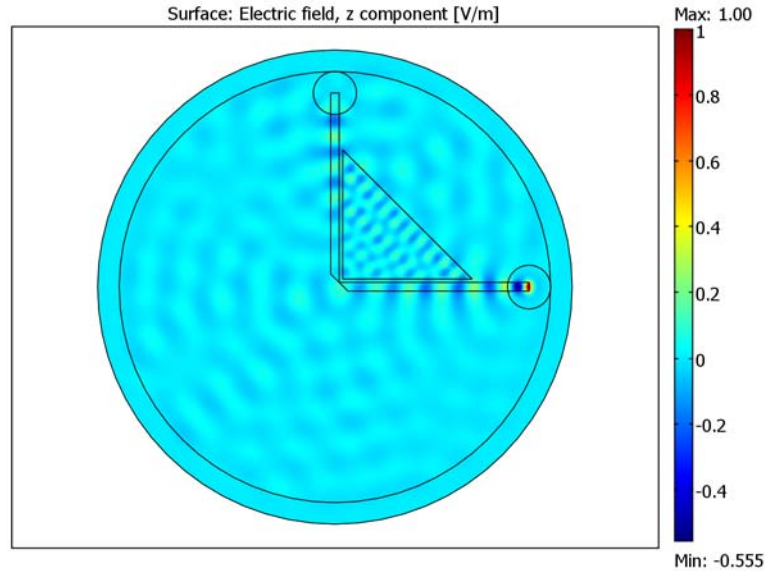


Figure 4-1: The z component of the electric field.

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 μm .

- 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.

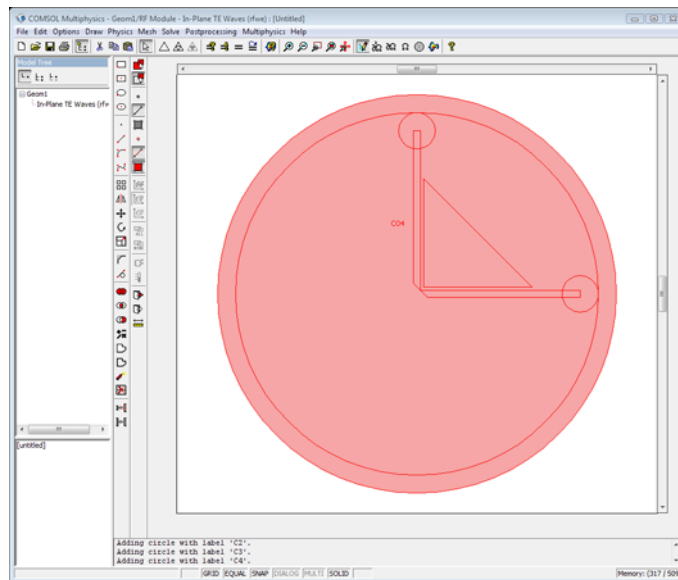


Figure 4-2: Model geometry.

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 OTHERS
Boundary condition	Electric field	Scattering boundary condition	Continuity
E_{0z}	$\cos(y*\pi/0.4[\mu\text{m}])$	0	
Wave type		Cylindrical wave	

Subdomain Settings

- 1 Subdomains 1 and 5 represent air or vacuum. This is the default setting.
- 2 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.
- 3 Subdomains 2, 4, and 6 are the free part of the waveguide. Use n_{Guide} as the refractive index here.
- 4 Subdomains 3, 8, and 9 are used for damping the wave. Choose to represent the material properties in terms of ϵ_r , μ_r , and σ . Enter $n_{\text{Guide}}^2 * I_{zz}$ for the **Relative permittivity** and $I_{xx} \ 0 \ 0 \ I_{yy}$ for the **Relative permeability**, which is anisotropic.
- 5 Subdomain 10 also damps the wave to reduce the effect of the exterior boundary. Enter I_{zz} for the **Relative permittivity** and $I_{xx} \ 0 \ 0 \ I_{yy}$ for the **Relative permeability**.

Expression Variables

- 1 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
I_{xx}	$s_y * s_z / s_x$
I_{yy}	$s_x * s_z / s_y$
I_{zz}	$s_x * s_y / s_z$

- 2 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.

SETTINGS	SUBDOMAINS 3, 4, 8	SUBDOMAINS 9, 10	ALL OTHERS
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** page, click the **Custom mesh size** button and set **Maximum element size** to $250\text{e-}9$. On the **Subdomain** page, select Subdomains 2, 4, and 6–8, and set **Maximum element size** to $100\text{e-}9$.
- 2 Initialize the mesh.

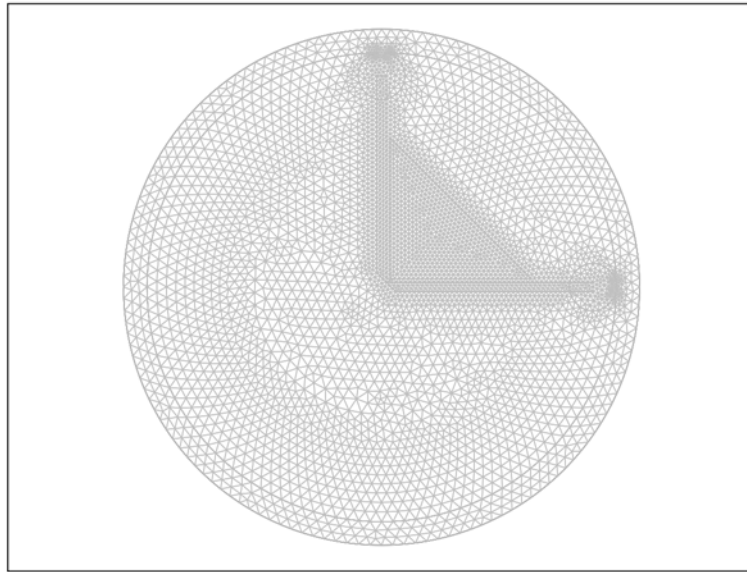


Figure 4-3: Mesh

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** page of the **Plot Parameters** dialog box, and use the **WaveLight** color table. 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.

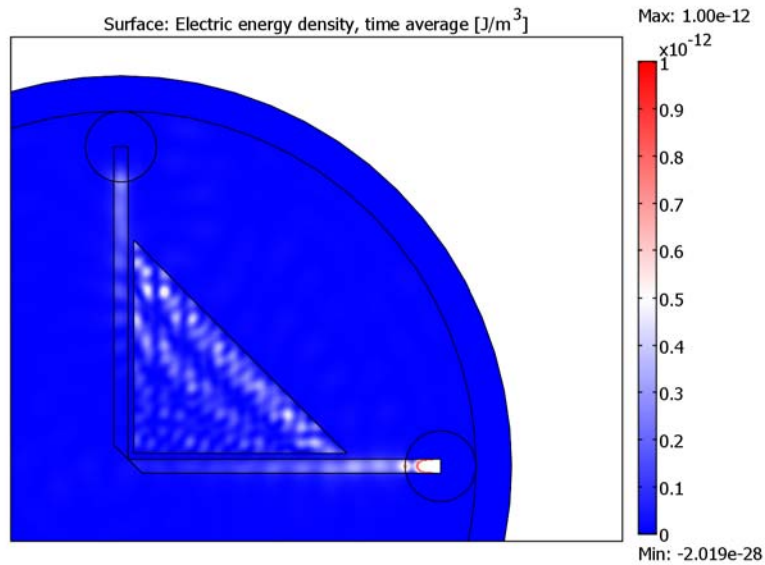


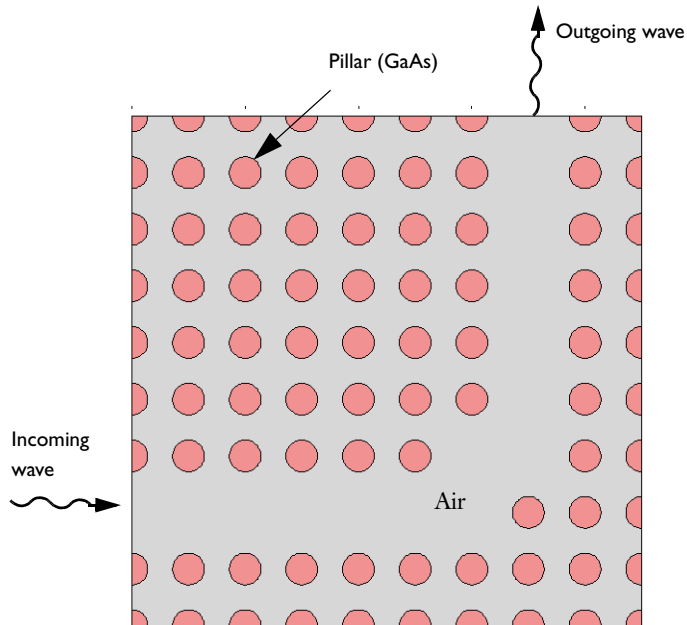
Figure 4-4: Electric energy density.

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.

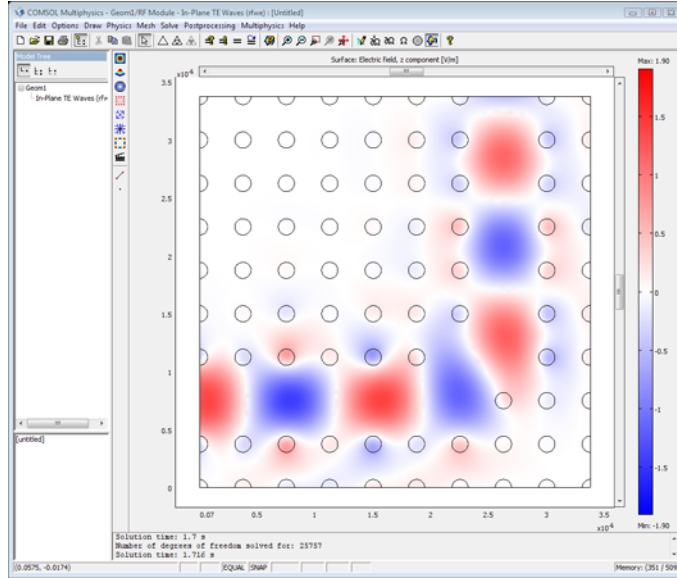


Figure 4-5: The z component of the electric field showing how the wave propagates along the path defined by the pillars.

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.

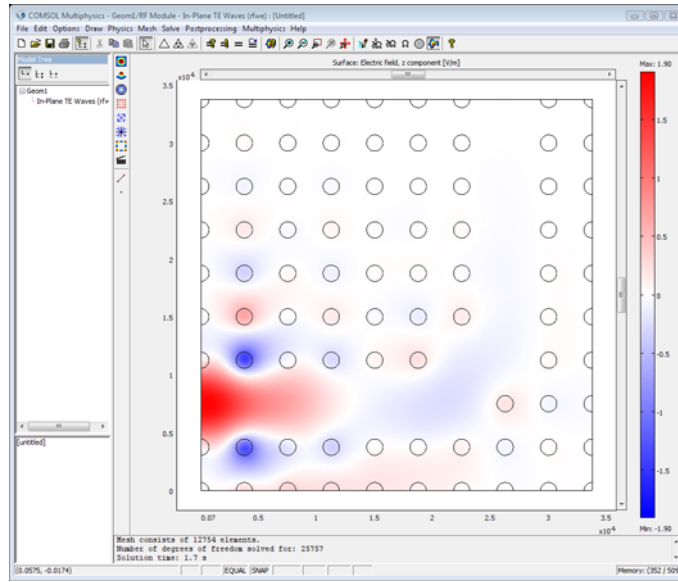


Figure 4-6: A longer wavelength will not propagate through the guide.

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	

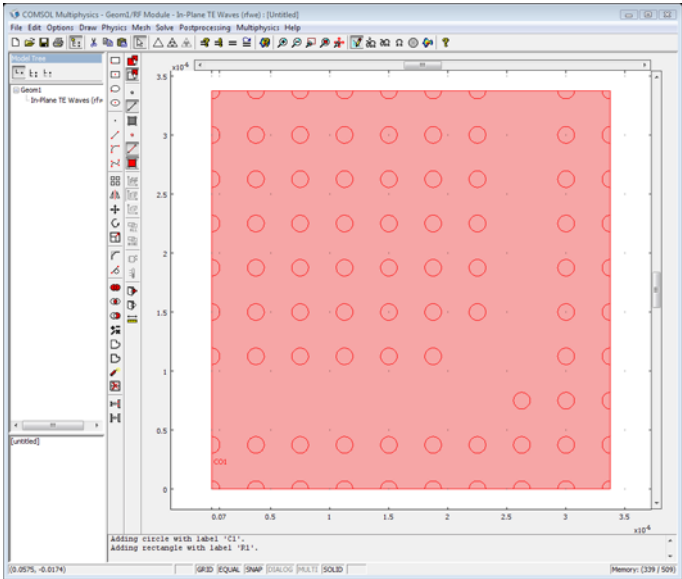
- 3 Click **OK**.

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° 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*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 \star 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 $1\text{e-}6$.

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	SUBDOMAIN 2
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 μm and 1.2339 μm according to Ref. 2. In the **Options>Expressions>Scalar Expressions** dialog box, define the variables `n_Air` and `n_GaAs`. The descriptions are optional. When done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
n_Air	1	Refractive index, air
n_GaAs	-3.3285e5*lambda0_rfwe[1/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.

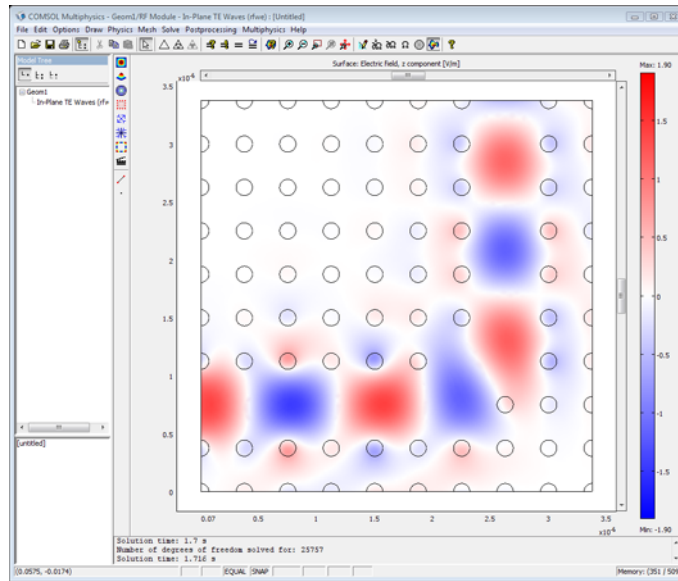
COMPUTING THE SOLUTION

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. On the **Surface** page in the **Plot Parameters**

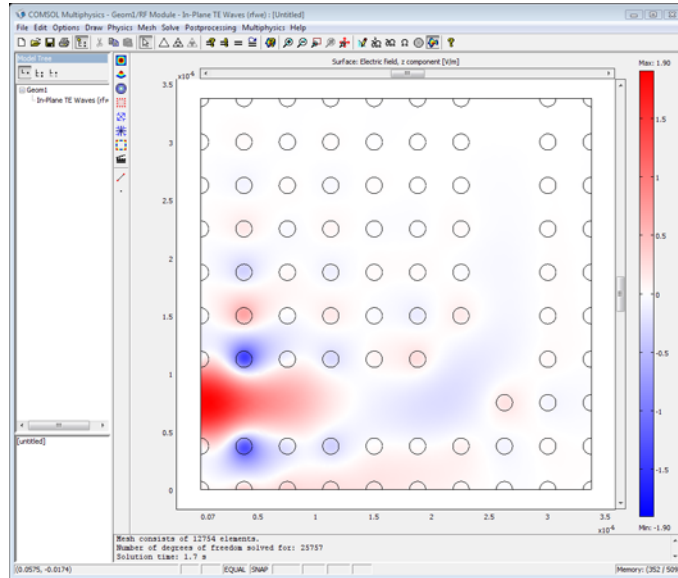
dialog box, change the **Color table** to **WaveLight**. This makes it easier to identify the wave.



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 WaveLight color table 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.

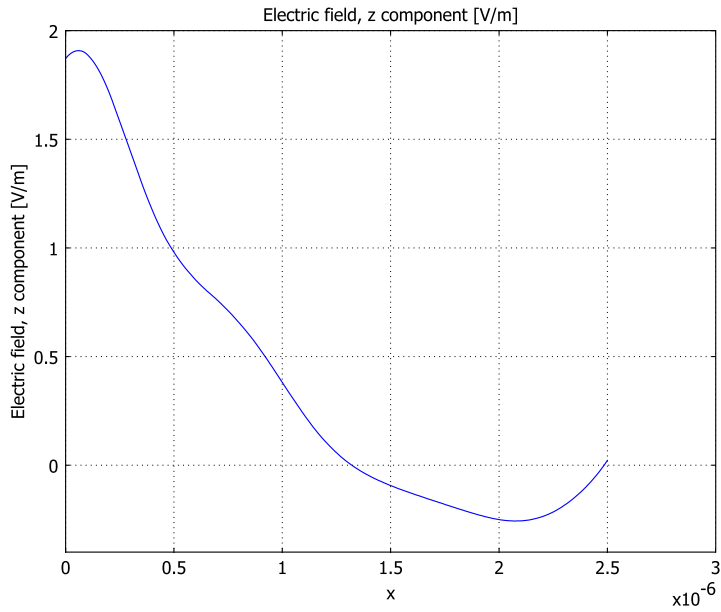
5 Click **OK** twice to see the following plot.



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 2.5×10^{-6} , **y0** to 0.75×10^{-6} , and **y1** to 0.75×10^{-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.

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 230.

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. 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 Λ to avoid confusion with the free-space wavelength, which is denoted with λ_0 . The relation between Λ and λ_0 is

$$\lambda_0 = \frac{2\pi}{\sqrt{\Lambda}} .$$

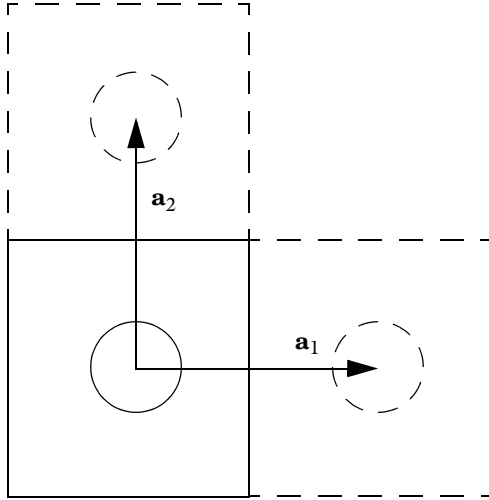
The wave vector for the propagating wave, k , enters the simulation as Floquet periodicity boundary conditions (Ref. 1),

$$E_z(2) = E_z(1)e^{-i\beta} ,$$

where β 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 MATLAB. 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.

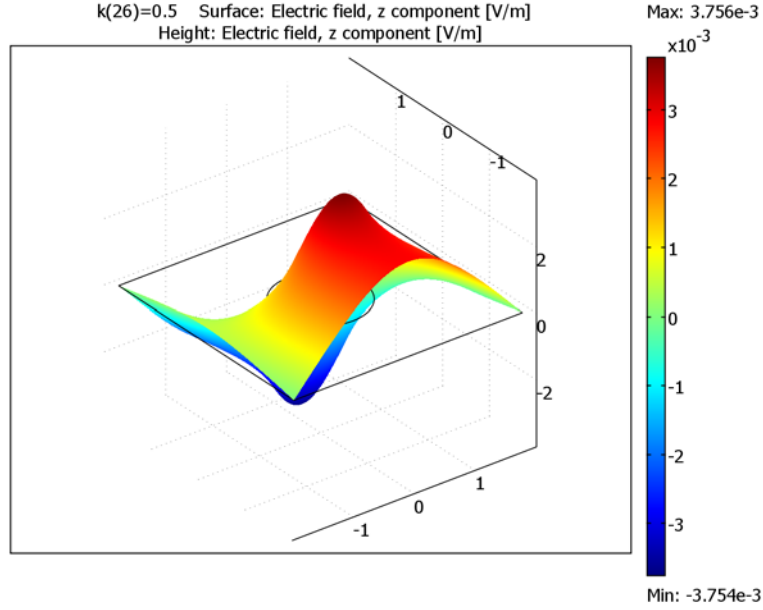


Figure 4-7: Field profile, E_z , at $k = 0.5$ for the first band in the $(1, 1)$ direction.

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.

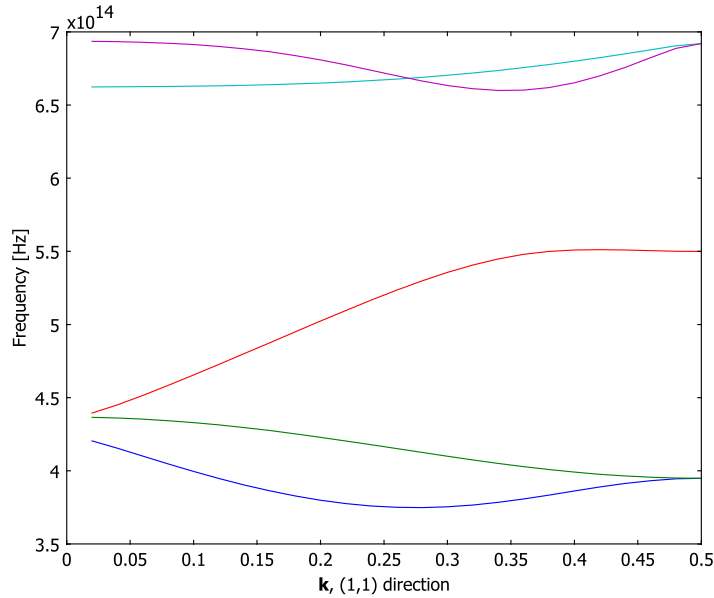


Figure 4-8: Band diagram of the bands in the (1, 1) direction.

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 244 shows the five lowest band surfaces obtained using the script created under the section “Modeling Larger Sweeps With MATLAB” on page 251. There is a large bandgap between the

third and fourth band surface, and no waves in this frequency range can propagate in the crystal.

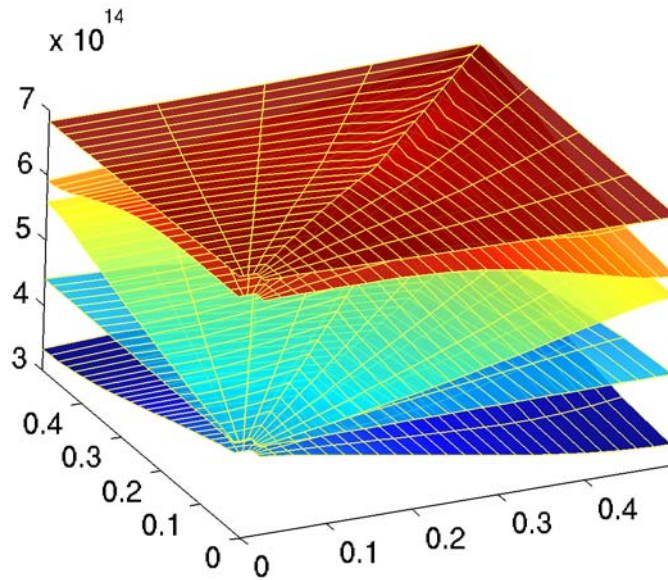


Figure 4-9: Band surfaces for the five lowest bands for one quadrant in k -space.

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	$k \cdot (k1 \cdot b1x + k2 \cdot b2x)$	1st k-component for periodic condition
ky	$k \cdot (k1 \cdot b1y + k2 \cdot b2y)$	2nd k-component for periodic condition
b1x	$2 \cdot \pi \cdot a2y / (a1x \cdot a2y - a1y \cdot a2x)$	x-component of 1st reciprocal lattice vector
b1y	$-2 \cdot \pi \cdot a2x / (a1x \cdot a2y - a1y \cdot a2x)$	y-component of 1st reciprocal lattice vector
b2x	$-2 \cdot \pi \cdot a1y / (a1x \cdot a2y - a1y \cdot a2x)$	x-component of 2nd reciprocal lattice vector

NAME	EXPRESSION	DESCRIPTION
b2y	$2\pi a_1 x / (a_1 x a_2 y - a_1 y a_2 x)$	y-component of 2nd reciprocal lattice vector
n_Air	1	Refractive index of air
n_GaAs	$3.3285e5 [1/m] \cdot \lambda_{0_rfwe} + 3.5031$	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 **\mathbf{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** option button.

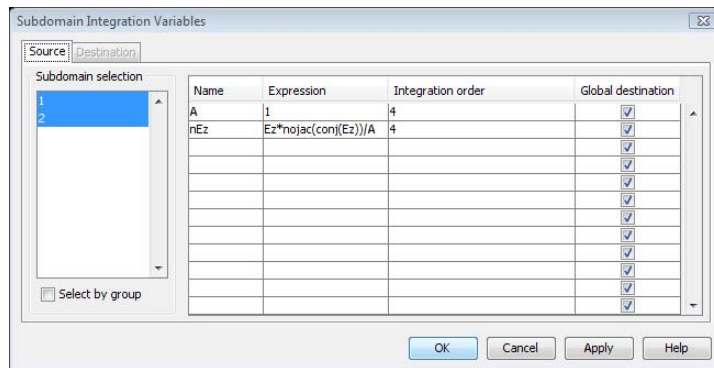
PROPERTY	SUBDOMAIN 1	SUBDOMAIN 2
n	n_{Air}	n_{GaAs}

- 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 **freq** 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 $4\text{e}14$ in the **Search for eigenfrequencies around** edit field.
- 3 Click the **Eigenfrequency** tab, and type $-i*2*\pi*4\text{e}14$ in the **Eigenvalue linearization point** edit field. This is the eigenvalue that corresponds to the eigenfrequency $4\cdot 10^{14}$ 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

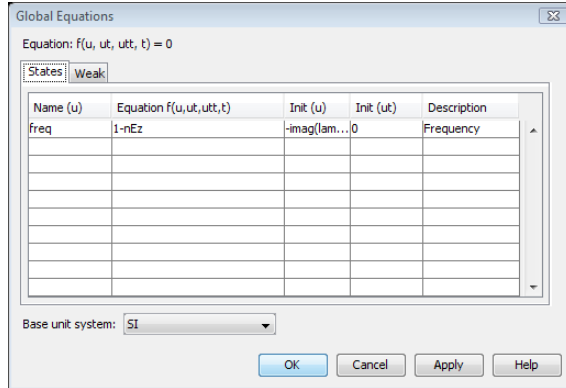
Now you use the eigenvalue solution as initial guess for the nonlinear problem. The use of the nonlinear solver enables sweeping of variables with the parametric solver. Here you ramp the magnitude of the k -vector from 0 to 0.5, which is in units of the reciprocal lattice vectors, \mathbf{b}_1 , \mathbf{b}_2 , \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 range (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**.

Global Equations

- 1 From the **Physics** menu, choose **Global Equations**.
- 2 In the **Global 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 From the **Physics** menu, choose **Scalar Variables**.
- 2 In the dialog box that appears, enter the value **freq** for the **nu_rfw** 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**.

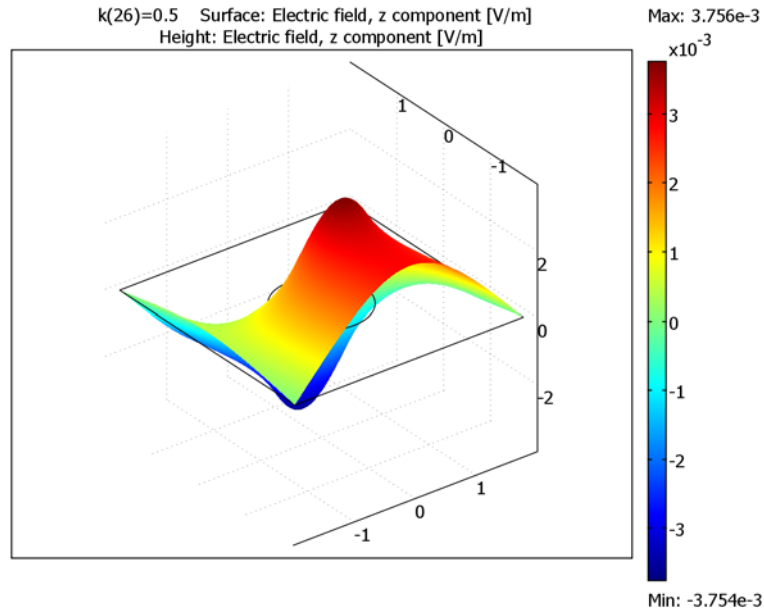
During the sweep, the solver uses the latest solution for both **freq** and **Ez**, tracing the band starting at $k = 0$ when k increases. You need to choose the step size of k carefully, because a too large step may cause the solution to “jump” to a different band if the bands lie close and have similar 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.

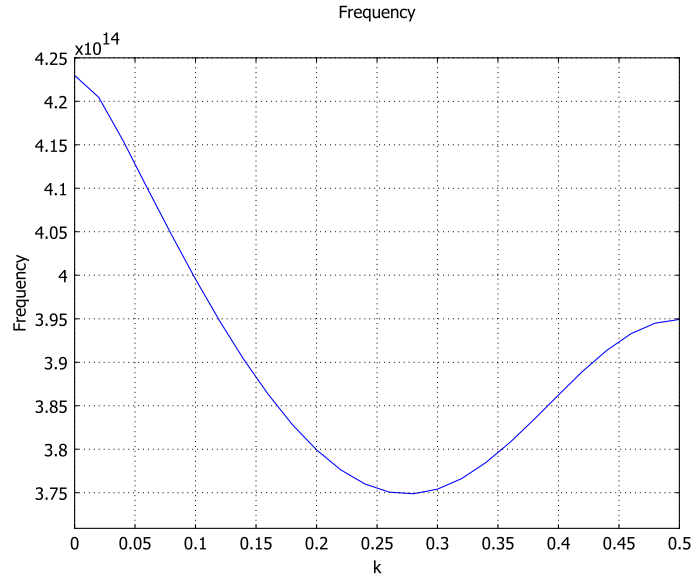
Click the **3D Surface Plot** toolbar button to get a height scale on the default surface plot.



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 248. 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 MATLAB

Performing more than three sweeps is best done using a script or a function. The following script takes the FEM structure, k magnitudes, directions, and eigenfrequency shifts, and 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.

Note: The instructions below assume that you are running COMSOL Multiphysics with MATLAB.

- 1 In COMSOL Multiphysics, press Ctrl+F to export the current model to the MATLAB workspace. The model is now available at the command prompt through the structure named `fem`.
- 2 Type `band_diagram_sweep` to run the following script (available in your path if you have MATLAB), which controls a sweep over five bands and one quadrant in k -space.

Note: The script takes a long time to run because it calculates a large number of data points.

```
% 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 = assemnit(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 = asseminit(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 3rd 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', ...
        'FaceAlpha',0.6);
end
% Band diagram in (1,1) direction
figure(3);
plot(k,permute(real(nu(5,:,:)),[3 2 1]));
xlabel('k, (1,1) direction');
ylabel('Frequency [Hz]');

```

You can find more information on the script commands in the *COMSOL Multiphysics Reference Guide* and the MATLAB documentation.

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 (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 μm (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 μm . 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_{eff} is close to the upper limit in this interval.

Model Definition

The mode analysis is made on a cross-section in the xy -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 ω is the angular frequency and β 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.

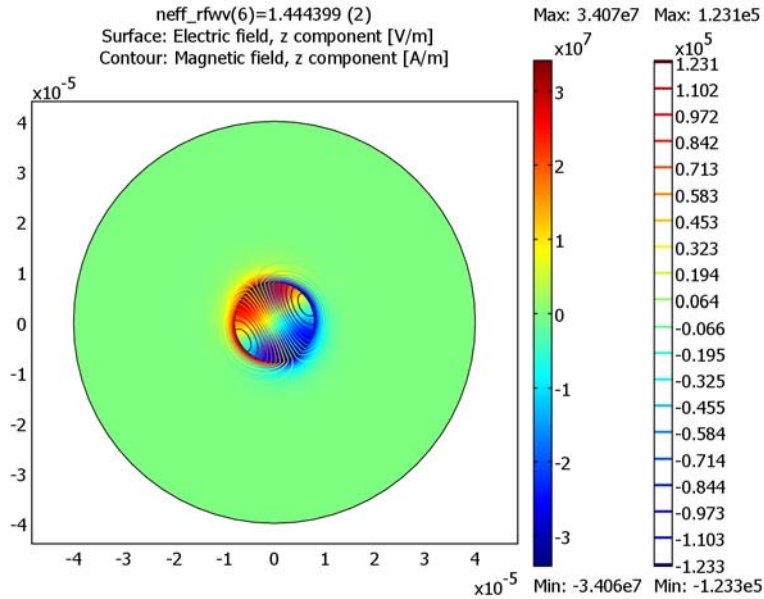


Figure 4-10: The surface plot visualizes the z component of the electric field, and the contour lines show the z component of the magnetic field. This plot is for the effective mode index 1.4444.

Reference

1. A. 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 λ_{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,

$$1.4378 < n_{\text{eff}} < 1.4457$$

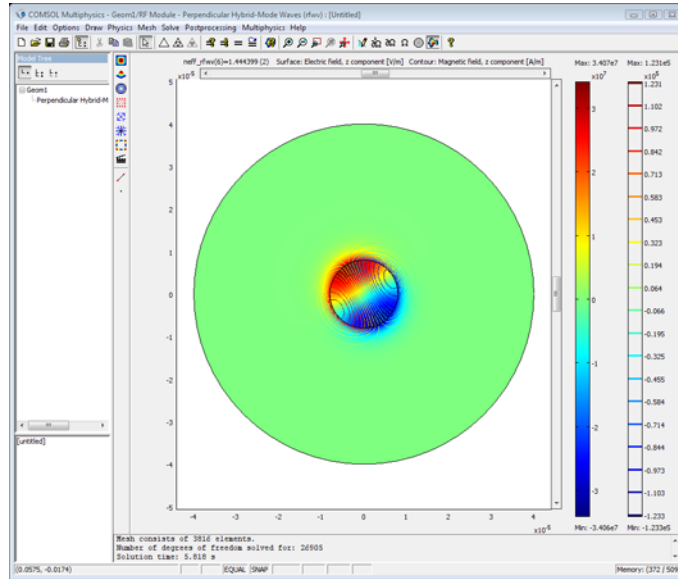
- 1 In the **Solver Parameters** dialog box set the parameter **Search for effective mode indices around** to 1.446. This guarantees that the solver will find the fundamental mode, which has the largest effective mode index.
- 2 Solve the problem.

POSTPROCESSING AND VISUALIZATION

The default plot shows the power flow in the z direction for the fundamental mode. This is the 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** page, set the **Surface data** to **Electric field, z component**.

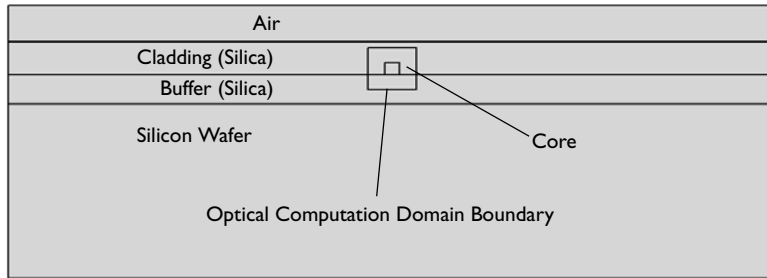
4 On the **Contour** page, give **Magnetic field, z component** as **Contour data**.



Stress-Optical Effects in a Silica-on-Silicon Waveguide

Introduction

Planar photonic waveguides in silica (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°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°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 σ_{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 σ_{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 σ_{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 directions 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 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 277 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 ν , 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 μm . 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_Photonics/stress_optical

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

Modeling Using the Graphical User Interface—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 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.
- 6 Choose **Multiphysics>Model Navigator**. Set the **Ruling application mode** to the **Perpendicular Hybrid-Mode Waves (rfwv)** 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 ₂)
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 From the **Options** menu, choose **Axes/Grid Settings**.
- 3 Give axis and grid settings according to the following table (to enter grid settings, first clear the **Auto** check box on the **Grid** page); when done, click **OK**.

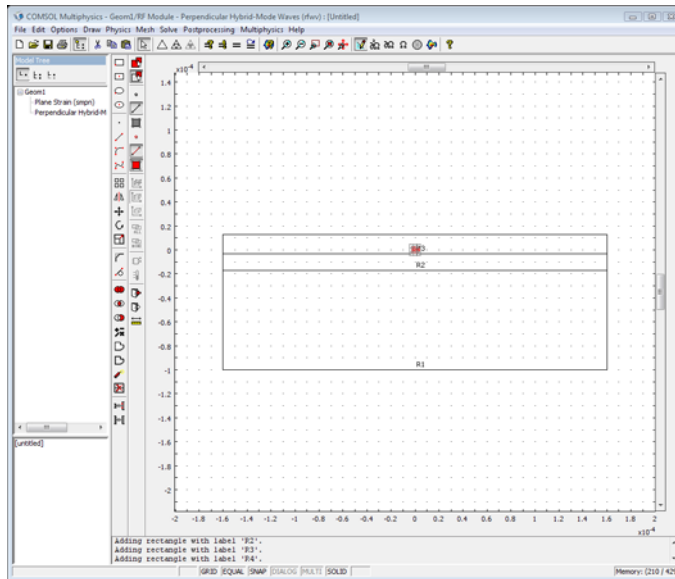
AXIS		GRID	
x min	-0.2e-3	x spacing	1e-5
x max	0.2e-3	Extra x	

AXIS		GRID	
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 (smpn)** application mode from the **Multiphysics** menu.
- 2 From the **Physics** menu, choose **Point Settings**.
- 3 In the **Point Settings** dialog box, choose Point 1 and, on the **Constraint** page, select the **R_x** and **R_y** check boxes to constrain the displacements in the x and y directions to zero in the lower-left corner. This keeps the domain from moving (translational move).
- 4 Select Point 9 and then select the **R_y** check box to constrain the y direction only in the lower-right corner, keeping the domain from rotating but allowing it to slide in the x direction.
- 5 Click **OK**.

Subdomain Settings

The air domain need not be part of the plane strain model.

- 1 From the **Physics** menu, choose **Subdomain Settings**.
- 2 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		
	ν	nuSi	Temp	T1		ν	nuSi02	Temp	T1
	α	alphaSi	Tempref	T0		α	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 ρ , because it does not enter the equation for static problems.

- 3 When done, click **OK**.

EXPRESSION VARIABLES

- 1 From the **Options** menu, choose **Expressions>Subdomain Expressions**.

- 2 Define the following variables in the **Subdomain Expressions** dialog box; when done, click **OK**.

NAME	EXPRESSION			
	SUBDOMAIN 1	SUBDOMAIN 2	SUBDOMAIN 3	SUBDOMAIN 4
N	nSi	nBuf	nClad	nCore

NAME	EXPRESSION	
	SUBDOMAIN 1	SUBDOMAINS 2-4
Nx	N	$N - B1 * sx_smpn - B2 * (sy_smpn + sz_smpn)$
Ny	N	$N - B1 * sy_smpn - B2 * (sx_smpn + sz_smpn)$
Nz	N	$N - B1 * sz_smpn - B2 * (sx_smpn + sy_smpn)$

MESH GENERATION

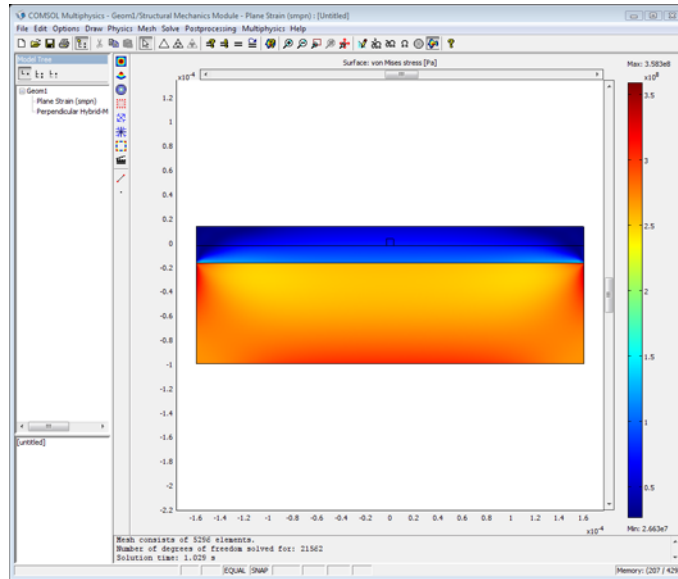
Initialize a mesh and refine it twice, using the **Initialize Mesh** and **Refine Mesh** buttons on the Main toolbar.

COMPUTING THE SOLUTION

- 1 From the **Solve** menu, choose **Solver Parameters**.
- 2 In the **Solver Parameters** dialog box select the **Stationary** solver. Click **OK**.
- 3 From the **Solve** menu, choose **Solver Manager**.
- 4 On the **Solve For** tab in the **Solver Manager** dialog box, select only the **Plane Strain (smpn)** application mode. This makes sure that COMSOL Multiphysics only solves the Plane Strain application mode equations in the first run.
- 5 Click the **Solve** button.
- 6 Click **OK**.

POSTPROCESSING AND VISUALIZATION

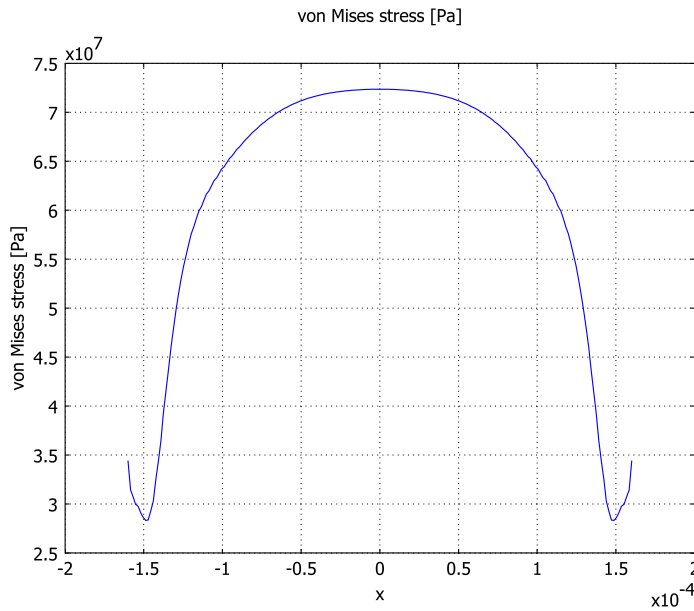
The default plot shows the von Mises effective stress as a colored surface plot.



To view the von Mises stress along a horizontal line through the entire structure, use a cross-section plot.

- 1 From the **Postprocessing** menu, choose **Cross-Section Plot Parameters**.
- 2 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 $-16\text{e-}5$, **y0** to 0, **x1** to $16\text{e-}5$, and **y1** to 0. Use **x** as **x-axis data**.

3 Click **OK**.

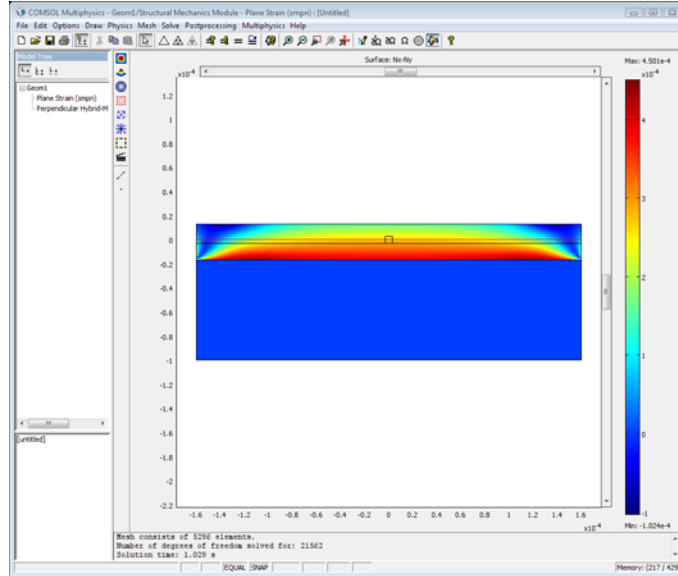


Notice that the effective stress is varying slowly on the horizontal cut. This means that the significant influence on the stress-induced changes in refractive index will come from the stress variations in the vertical direction. This is expected because the extension of the domains in the x direction is chosen to minimize effects of the edges.

Next plot the birefringence:

- 4 From the **Postprocessing** menu, choose **Plot Parameters**.
- 5 In the **Plot Parameters** dialog box, click the **Surface** tab.
- 6 To view the stress-induced birefringence $n_x - n_y$, type $Nx - Ny$ in the **Expression** field on the **Surface Data** page.

7 Click **OK**.

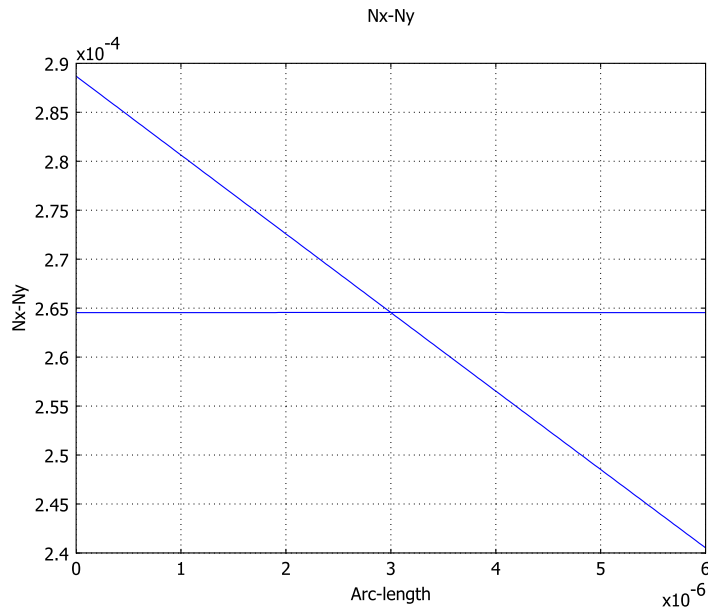


8 From the **Postprocessing** menu, choose **Cross-Section Plot Parameters**.

9 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**. Click **Apply**. 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.

10 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.

II Click **OK**.



The birefringence is constant on the horizontal line, thus the influence of the edges is indeed reduced to a minimum.

Modeling Using the Graphical User Interface—Optical Mode Analysis

Select the **Perpendicular Hybrid-Mode Waves (rfwv)** 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.

Click the **Draw Mode** button on the Main toolbar, and then 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 by clearing the **Active in this domain** check box. This makes sure the problem is only solved in the newly drawn rectangle.
- 3 For all active domains, select the **Specify material properties in terms of refractive index** button. Enter the values for N_x , N_y , and N_z , respectively, as anisotropic refractive indices by selecting **Diagonal** from the **Anisotropy** list in the extended **n** edit field and 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 From the **Mesh** menu, choose **Free Mesh Parameters**.
- 2 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.
- 3 Click the **Remesh** button to initialize the mesh, then click **OK**.

COMPUTING THE SOLUTION

- 1 From the **Solve** menu, choose **Solver Parameters**.
- 2 In the **Solver Parameters** dialog box, select the **Eigenvalue** solver.

- 3 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

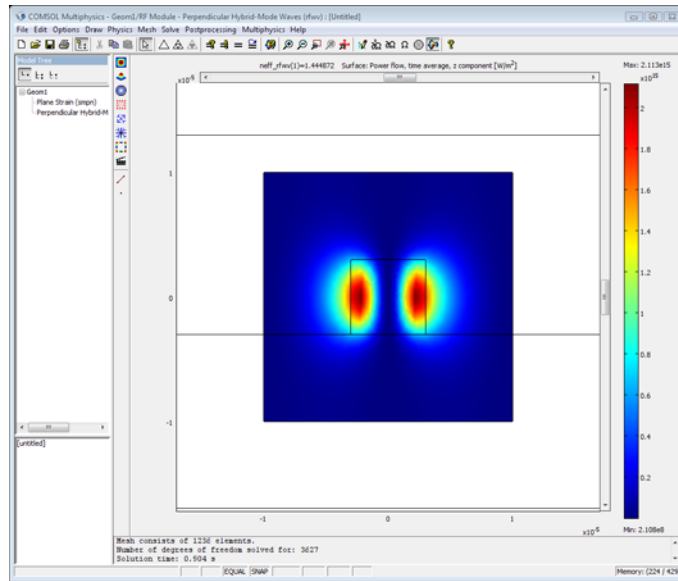
$$n_{\text{eff}} < n_{\text{core}} = 1.456$$

- 4 Click **OK**.
- 5 From the **Solve** menu, choose **Solver Manager**.
- 6 In the **Solver Manager** dialog box, select only the **Perpendicular Hybrid-Mode Waves (rfwv)** application mode on the **Solve For** tab. This ensures that the Plane Strain application mode is not part of the eigenmode computation.
- 7 On the **Initial Value** tab, set the initial value to **Current solution** in the **Initial value** area. This will take the plane strain solution and use it to evaluate the refractive indices for the mode analysis.
- 8 Click the **Solve** button, then click **OK**.

POSTPROCESSING AND VISUALIZATION

- 1 From the **Postprocessing** menu, choose **Plot Parameters**.
- 2 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).

- 3 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.

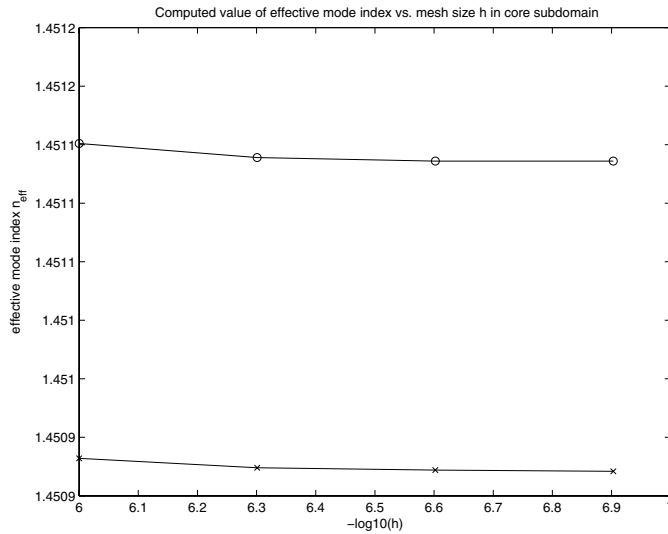
- 1 In the **Free Mesh Parameters** dialog box, lower the value of the maximum element size for the core subdomain, Subdomain 6, to $5\text{e-}7$, remesh, and solve again.
- 2 Repeat this for the maximum element sizes $2.5\text{e-}7$ and $1.25\text{e-}7$.

The solver finds the following effective mode:

ELEMENT SIZE, CORE SUBDOMAIN	1E-6	5E-7	2.5E-7	1.25E-7
EFFECTIVE MODE INDEX				
$n_{\text{eff}1}$	1.444872	1.444873	1.444873	1.444873
$n_{\text{eff}2}$	1.445134	1.445134	1.445134	1.445134
$n_{\text{eff}3}$	1.445138	1.445138	1.445138	1.445138
$n_{\text{eff}4}$	1.445399	1.445399	1.445399	1.445399

ELEMENT SIZE, CORE SUBDOMAIN	1E-6	5E-7	2.5E-7	1.25E-7
EFFECTIVE MODE INDEX				
$n_{\text{eff}5}$	1.450871	1.450871	1.450871	1.450871
$n_{\text{eff}6}$	1.451135	1.451135	1.451135	1.451135

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 **Isotropic** from the **Anisotropy** list in the extended **n** edit field for the refractive index in the **Subdomain Settings** dialog box, and typing N as the refractive index.
- 2 Solve the problem for a maximum element size of $2.5\text{e-}7$ for Subdomain 6.

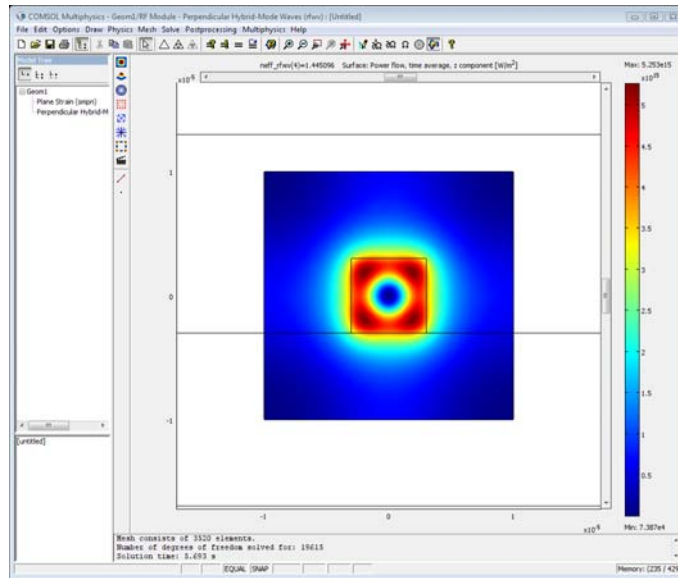
Compare these ideal values of the propagation constants with the stress-optical case.

ELEMENT SIZE, CORE SUBDOMAIN: EFFECTIVE MODE INDEX	2.5E-7 (STRESS)	2.5E-7 (NO STRESS)	DIFFERENCE
			*1e-4
$n_{\text{eff}1}$	1.444873	1.444816	.57
$n_{\text{eff}2}$	1.445134	1.444828	3.06
$n_{\text{eff}3}$	1.445138	1.44507	.63
$n_{\text{eff}4}$	1.445399	1.445096	3.03
$n_{\text{eff}5}$	1.450871	1.45082	.51
$n_{\text{eff}6}$	1.451135	1.45082	3.16

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 277. 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 260, 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

$$\epsilon_z = e_0 + e_1x + e_2y$$

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 ϵ_x and ϵ_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 ϵ_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 ϵ_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,

$$\begin{bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{xz} \end{bmatrix} = D \begin{bmatrix} \epsilon_x \\ \epsilon_y \\ \epsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{xz} \end{bmatrix} - \alpha \begin{bmatrix} T - T_{\text{ref}} \\ T - T_{\text{ref}} \\ T - T_{\text{ref}} \\ 0 \\ 0 \\ 0 \end{bmatrix}$$

where σ_x , σ_y , σ_z , τ_{xy} , τ_{yz} , and τ_{xz} are the components of the stress tensor, ϵ_x , ϵ_y , ϵ_z , γ_{xy} , γ_{yz} , and γ_{xz} are the components of the strain tensor, α is the thermal expansion coefficient, T is the operating temperature, and T_{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, ν . 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

$$\begin{bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \end{bmatrix} = D \begin{bmatrix} \epsilon_x \\ \epsilon_y \\ \epsilon_z \\ \gamma_{xy} \end{bmatrix} - \alpha \begin{bmatrix} T - T_{\text{ref}} \\ T - T_{\text{ref}} \\ T - T_{\text{ref}} \\ 0 \end{bmatrix}$$

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_{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 347 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 $\mathbf{K} = \mathbf{0}$,

$$0 = \int_{\partial\Omega} \begin{bmatrix} u_{\text{test}} & v_{\text{test}} & w_{\text{test}} \end{bmatrix} \begin{bmatrix} \sigma_x & \tau_{xy} & 0 \\ \tau_{yx} & \sigma_y & 0 \\ 0 & 0 & \sigma_z \end{bmatrix} \begin{bmatrix} n_x \\ n_y \\ n_z \end{bmatrix} d\Gamma$$

$$- \int_{\Omega} \begin{bmatrix} u_{x,\text{test}} & u_{y,\text{test}} & u_{z,\text{test}} \\ v_{x,\text{test}} & v_{y,\text{test}} & v_{z,\text{test}} \\ w_{x,\text{test}} & w_{y,\text{test}} & w_{z,\text{test}} \end{bmatrix} \cdot \begin{bmatrix} \sigma_x & \tau_{xy} & 0 \\ \tau_{yx} & \sigma_y & 0 \\ 0 & 0 & \sigma_z \end{bmatrix} d\Omega$$

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

$$\int_{\Omega} \begin{bmatrix} u_{x,\text{test}} & u_{y,\text{test}} & u_{z,\text{test}} \\ v_{x,\text{test}} & v_{y,\text{test}} & v_{z,\text{test}} \\ w_{x,\text{test}} & w_{y,\text{test}} & w_{z,\text{test}} \end{bmatrix} \cdot \begin{bmatrix} \sigma_x & \tau_{xy} & 0 \\ \tau_{yx} & \sigma_y & 0 \\ 0 & 0 & \sigma_z \end{bmatrix} d\Omega$$

$$= \int_{\Omega} \left(\begin{bmatrix} u_{x,\text{test}} & u_{y,\text{test}} & u_{z,\text{test}} \end{bmatrix} \cdot \begin{bmatrix} \sigma_x & \tau_{xy} & 0 \end{bmatrix} \right.$$

$$+ \begin{bmatrix} v_{x,\text{test}} & v_{y,\text{test}} & v_{z,\text{test}} \end{bmatrix} \cdot \begin{bmatrix} \tau_{yx} & \sigma_y & 0 \end{bmatrix}$$

$$+ \left. \begin{bmatrix} w_{x,\text{test}} & w_{y,\text{test}} & w_{z,\text{test}} \end{bmatrix} \cdot \begin{bmatrix} 0 & 0 & \sigma_z \end{bmatrix} \right) d\Omega$$

where

$$\sigma_x = \frac{E}{(1+\nu)(1-2\nu)} ((1-\nu)\epsilon_x + \nu\epsilon_y + \nu\epsilon_z - (1+\nu)\alpha(T - T_{\text{ref}}))$$

$$\sigma_y = \frac{E}{(1+\nu)(1-2\nu)} (\nu\epsilon_x + (1-\nu)\epsilon_y + \nu\epsilon_z - (1+\nu)\alpha(T - T_{\text{ref}}))$$

$$\sigma_z = \frac{E}{(1+\nu)(1-2\nu)} (\nu\epsilon_x + \nu\epsilon_y + (1-\nu)\epsilon_z - (1+\nu)\alpha(T - T_{\text{ref}}))$$

$$\tau_{xy} = \frac{E}{(1+\nu)(1-2\nu)} \left(\frac{1-2\nu}{2} \gamma_{xy} \right)$$

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_{test} and v_{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 $\mathbf{K} = \mathbf{0}$)

$$0 = \int_{\partial\Omega} \mathbf{v} \mathbf{\sigma} \mathbf{n} d\Gamma - \int_{\Omega} (\varepsilon_{x, \text{ test}} \sigma_x + \varepsilon_{y, \text{ test}} \sigma_y + \varepsilon_{z, \text{ test}} \sigma_z + \gamma_{xy, \text{ test}} \tau_{xy}) d\Omega$$

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 σ_z can be separated out according to

$$\begin{bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{bmatrix} = D \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{bmatrix} - \begin{bmatrix} (1+\nu)\alpha(T-T_{\text{ref}}) \\ (1+\nu)\alpha(T-T_{\text{ref}}) \\ 0 \end{bmatrix} + \begin{bmatrix} \nu\varepsilon_z \\ \nu\varepsilon_z \\ 0 \end{bmatrix}$$

$$\sigma_z = \frac{E}{(1+\nu)(1-2\nu)}(\nu\varepsilon_x + \nu\varepsilon_y + (1-\nu)\varepsilon_z - (1+\nu)\alpha(T-T_{\text{ref}}))$$

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 ε_z need to the stress components σ_x and σ_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, σ_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 σ_z was presented earlier.

In COMSOL Multiphysics you model the coefficients e_0 , e_1 , and e_2 in the expression for the ε_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_Photonics/
stress_optical_generalized

Modeling Using the Graphical User Interface—Plane Strain Analysis

Many of the modeling steps are similar to the model “Stress-Optical Effects in a Silica-on-Silicon Waveguide” on page 260, 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 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.
- 6 Click **OK**.

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 (rfwv)** 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, for example, the origin (0, 0), on the **Geom2** page in the drawing area.

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 263.

NAME	EXPRESSION	DESCRIPTION
nSi	3.5	Refractive index, silicon (Si)
nBuf	1.445	Refractive index, silica (SiO ₂)
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

NAME	EXPRESSION	DESCRIPTION
B2	$0.65 \times 10^{-12} [\text{m}^2/\text{N}]$	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	(-16×10^{-5} , -1×10^{-4})	(16×10^{-5} , -1.7×10^{-5})
Buffer	(-16×10^{-5} , -1.7×10^{-5})	(16×10^{-5} , -3×10^{-6})
Cladding	(-16×10^{-5} , -3×10^{-6})	(16×10^{-5} , 1.3×10^{-5})
Core	(-3×10^{-6} , -3×10^{-6})	(3×10^{-6} , 3×10^{-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 (smpn)** application mode from the **Multiphysics** menu.
- 2 From the **Physics** menu, choose **Point Settings**.
- 3 In the **Point Settings** dialog box, choose Point 1 and, on the **Constraint** page, select the **R_x** and **R_y** check boxes to constrain the displacements in the x - and y -direction to zero in the lower-left corner. This keeps the domain from moving (translational move).
- 4 Select Point 9 and then select the **R_y** check box to constrain the y direction only in the lower-right corner, keeping the domain from rotating but allowing it to slide in the x direction.
- 5 When done, click **OK**.

Subdomain Settings

The air domain need not be part of the plane strain model.

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

2 Specify the subdomain settings for the Plane Strain application mode according to the following table.

	SUBDOMAIN 1				SUBDOMAINS 2-4			
Page	Material		Loads		Material		Loads	
	E	ESi			E	ESi02		
	ν	ν Si	Temp	T1	ν	ν Si02	Temp	T1
	α	α Si	Tempref	T0	α	α Si02	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 ρ , because it does not enter the equation for static problems.

3 Click **OK**.

EXPRESSION VARIABLES

I From the **Options** menu, choose **Expressions>Subdomain Expressions**.

2 In the **Subdomain Expressions** dialog box in the **Options** menu, define the following variables; when done, click **OK**.

NAME	SUBDOMAIN SELECTION	EXPRESSION
E	I	ESi
	2–4	ESi02
nu	I	nuSi
	2–4	nuSi02
alpha	I	alphaSi
	2–4	alphaSi02
ex	I–4	ux
ey	I–4	vy
ez	I–4	e00+e11*x+e22*y
sx	I–4	sx_smpn+E/((1+nu)*(1-2*nu))*nu*ez
sy	I–4	sy_smpn+E/((1+nu)*(1-2*nu))*nu*ez
sz	I–4	E/((1+nu)*(1-2*nu))*(nu*ex+nu*ey+(1-nu)*ez-(1+nu)*alpha*(T1-T0))
N	I	nSi
	2	nBuf
	3	nCore
	4	nClad
Nx	I	N
	2–4	N-B1*sx-B2*(sy+sz)
Ny	I	N
	2–4	N-B1*sy-B2*(sx+sz)
Nz	I	N
	2–4	N-B1*sz-B2*(sx+sy)

The variables ex, ey, and ez are the strains ϵ_x , ϵ_y , and ϵ_z , and the variables sx, sy, and sz are the stresses σ_x , σ_y , and σ_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** page.

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 $-E/((1+\nu)*(1-2*\nu))*(\text{ex_test}*\nu*\text{ez}+\text{ey_test}*\nu*\text{ez})-\text{ez_test}*sz$ 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.
- 4 Click **OK**.

MESH GENERATION

- 1 Initialize the mesh by clicking the **Initialize Mesh** button on the Main toolbar.
- 2 Refine it twice by clicking the **Refine Mesh** button on the Main toolbar twice.
- 3 Switch to **Geom2** by clicking its tab.

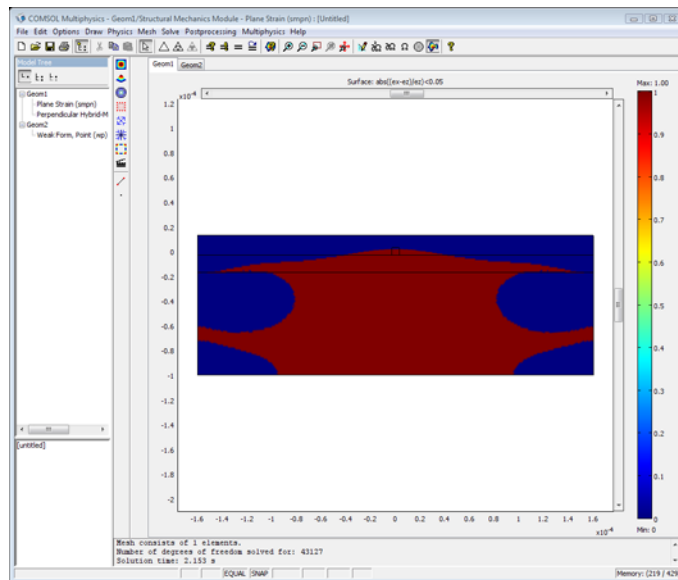
- 4 Initialize the mesh in this geometry too.

COMPUTING THE SOLUTION

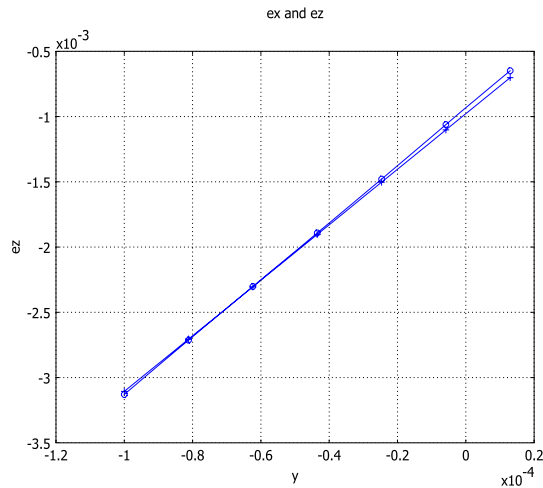
- 1 From the **Solve** menu, choose **Solver Parameters**.
- 2 In the **Solver Parameters** dialog box select the **Stationary** solver. When done, click **OK**.
- 3 From the **Solve** menu, choose **Solver Manager**.
- 4 On the **Solve For** tab in the **Solver Manager** dialog box, select only the **Plane Strain** and the **Weak Form, Point (wp)** application modes. This excludes the hybrid-mode waves application mode from the equations to solve. When done, click **OK**.
- 5 Click the **Solve** button, then click **OK**.

POSTPROCESSING AND VISUALIZATION

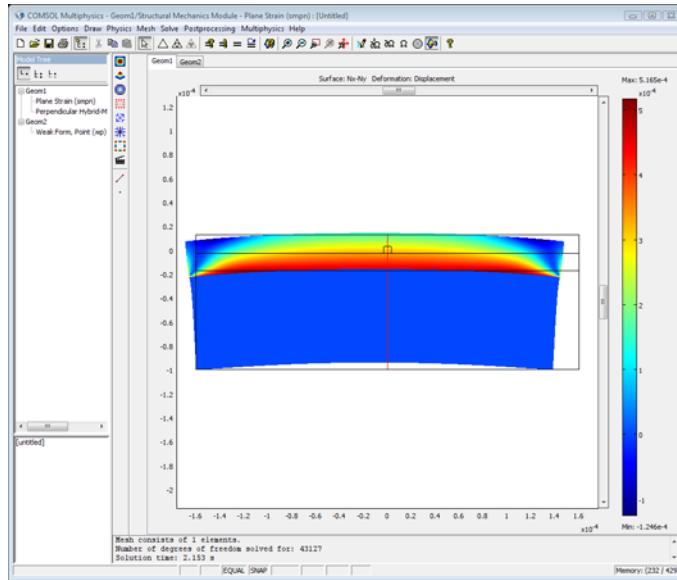
- 1 Switch to **Geom1** by clicking its tab.
- 2 From the **Postprocessing** menu, choose **Plot Parameters**.
- 3 Click the **Surface** tab on the **Plot Parameters** dialog box. Change the string in the **Expression** edit field on the **Surface Data** page to $\text{abs}((e_x - e_z)/e_z) < 0.05$. 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.



- 4 For symmetry reasons the strains ϵ_x and ϵ_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 from the **Postprocessing** menu. Enter the **Title** ϵ_x and ϵ_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 $-1\text{e-}4$, and **y1** to $1.3\text{e-}5$ to make a vertical line across the structure. Enter the **y-axis data expression** ϵ_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**; then click **OK**. Click **Apply** to plot ϵ_x .
- 5 On the **General** tab check **Keep current plot** to be able to plot ϵ_z in the same figure window.
- 6 Then change the **Expression** to ϵ_z in **y-axis data**. Use **Plus sign** as **Line marker** for this plot. Click **OK** to make the plot.



- 7 To view the stress-induced birefringence $n_x - n_y$ together with the deformation, open the **Plot Parameters** dialog box and select the **Surface** and **Deformed shape** check boxes in the **Plot type** area. 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. In the **Deformation data** area, choose **Plane Strain (smpn) > Displacement** 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.

Modeling Using the Graphical User Interface—Optical Mode Analysis

Continue with the mode analysis in the same way as in the previous model on page 270.

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 $(1e-5, 1e-5)$ on the **Geom1** page.

PHYSICS SETTINGS

Next, set up the parameters for the hybrid-mode wave problem. Select the **Perpendicular Hybrid-Mode Waves (rfwv)** 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 those subdomains). This makes sure that the solution for the electromagnetic waves only includes the newly drawn rectangle.
- 3 For all active domains, select the **Specify material properties in terms of refractive index** button. Enter the values for N_x , N_y , and N_z , respectively, as anisotropic refractive indices by selecting **Diagonal** from the **Anisotropy** list in the extended **n** edit field and changing the entries on the diagonal in the refractive index matrix
- 4 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_{Core} .

MESH GENERATION

- 1 From the **Mesh** menu, choose **Free Mesh Parameters**.
- 2 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.
- 3 Click the **Remesh** button to initialize the mesh, then click **OK**.

COMPUTING THE SOLUTION

- 1 From the **Solve** menu, choose **Solver Parameters**.
- 2 In the **Solver Parameters** dialog box, select the **Eigenvalue** solver.

- 3 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. Click **OK**.

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

$$n_{\text{eff}} < n_{\text{core}} = 1.456$$

- 4 From the **Solve** menu, choose **Solver Manager**.
- 5 In the **Solver Manager** dialog box, select only the **Perpendicular Hybrid-Mode Waves (rfwv)** application mode on the **Solve For** page. This ensures that only this application mode is part of the eigenmode computation.
- 6 On the **Initial Value** page, set the initial value to **Current solution** in the **Initial value** area. This takes the plane strain solution and uses it to evaluate the refractive indices for the mode analysis.
- 7 Click the **Solve** button, then click **OK**.

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.

EFFECTIVE MODE INDEX	GENERALIZED PLANE STRAIN	PLANE STRAIN	DIFFERENCE *1e-3
$n_{\text{eff}1}$	1.444923	1.444873	- .05
$n_{\text{eff}2}$	1.445195	1.445134	- .06
$n_{\text{eff}3}$	1.445213	1.445138	- .07
$n_{\text{eff}4}$	1.445487	1.445399	- .09
$n_{\text{eff}5}$	1.450928	1.450871	- .06
$n_{\text{eff}6}$	1.451217	1.451135	- .08

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):

$$\mathbf{E}(x, y, z) = E_{0x} \frac{w_0}{w(z)} e^{-(r/\omega(z))^2} \cos\left(\omega t - kz + \eta(z) - r^2 \frac{k}{2R(z)}\right) \mathbf{e}_x ,$$

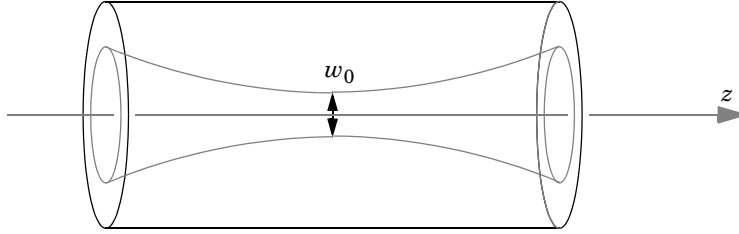
where

$$w(z) = w_0 \sqrt{1 + \left(\frac{z}{z_0}\right)^2}$$

$$\eta(z) = \text{atan}\left(\frac{z}{z_0}\right) .$$

$$R(z) = z \left(1 + \left(\frac{z_0}{z}\right)^2\right)$$

In these expressions, w_0 is the minimum waist, ω is the angular frequency, r is the radial cylindrical coordinate, and k is the wave number. The wave front 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 ϕ direction is approximated 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 307, propagation of a true 3D Gaussian beam is calculated without any nonlinear effects.

The nonlinear properties for 2nd 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. If you have MATLAB, you can calculate the frequency spectrum from a series of time samples with the fast Fourier transform (FFT) 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 $20\text{ }\mu\text{m}$ 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.

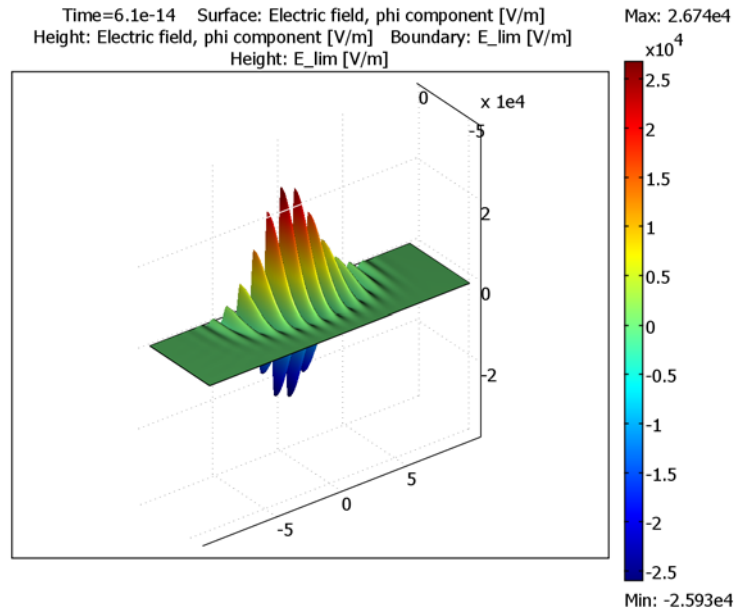


Figure 4-11: The pulse after 61 fs.

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.

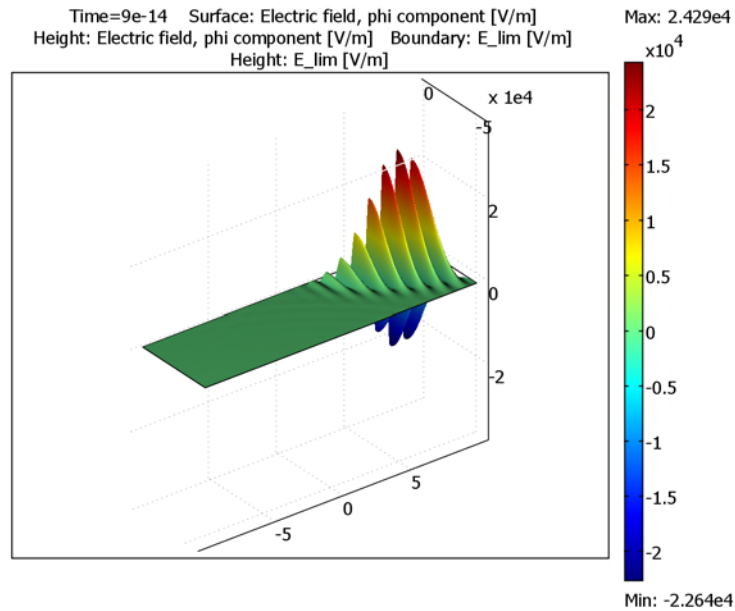


Figure 4-12: The pulse after 90 fs. It has now reached the output boundary.

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 can be extracted using FFT. Perform the extraction and transformation at the MATLAB command line. The result appears in Figure 4-13 on page 298. 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.

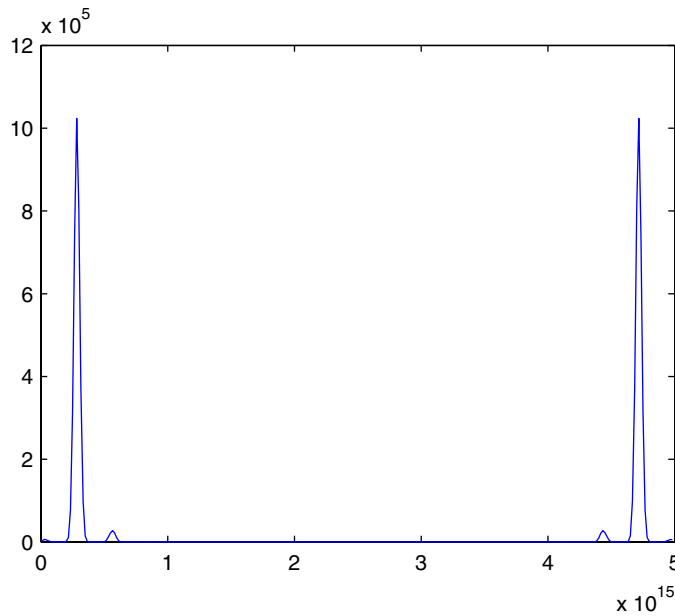


Figure 4-13: The result from the FFT on the beam at the output boundary. The two small peaks between the large peaks are the second harmonic generation.

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 by choosing **Specify Objects** from the **Draw** menu. 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
w	$w0 \cdot \sqrt{1 + (z/z0)^2}$	Analytical waist function along z
eta	$\text{atan}(z/z0)$	Analytical angle along z
R	$z \cdot (1 + (z0/z)^2)$	Analytical radius along z
c0	$1/\sqrt{\epsilon_0 \cdot \mu_0}$	Speed of light
omega0	$2\pi \cdot c0 / \lambda_0$	Angular frequency

- 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	$w0/w \cdot \exp(-r^2/w_bnd^2) \cdot \cos(\omega_0 \cdot t - k_0 \cdot z + \eta_bnd - r^2 \cdot k_0 / (2 \cdot R_bnd))$		
E_pulse	$\exp(-(t-t0)^2/dt^2)$		
E_lim		$E0 \cdot 1.2$	$-E0 \cdot 1.2$

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.

- 6 Click **OK**.

- 7 From the **Options** menu, point to **Integration Coupling Variables** 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. When done, click **OK**.

NAME	POINT 2	POINT 4	ALL OTHERS
w_bnd	w		
eta_bnd	eta		
R_bnd	R		
Eout		Ephi_rfwe	

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 1	BOUNDARY 4
Boundary condition	Scattering boundary	Scattering boundary
$E_{0,\varphi}$	$E0 * E_pulse * E_bnd$	0

SETTINGS	BOUNDARY 2	BOUNDARY 3
Boundary condition	Perfect electric conductor	Perfect magnetic conductor

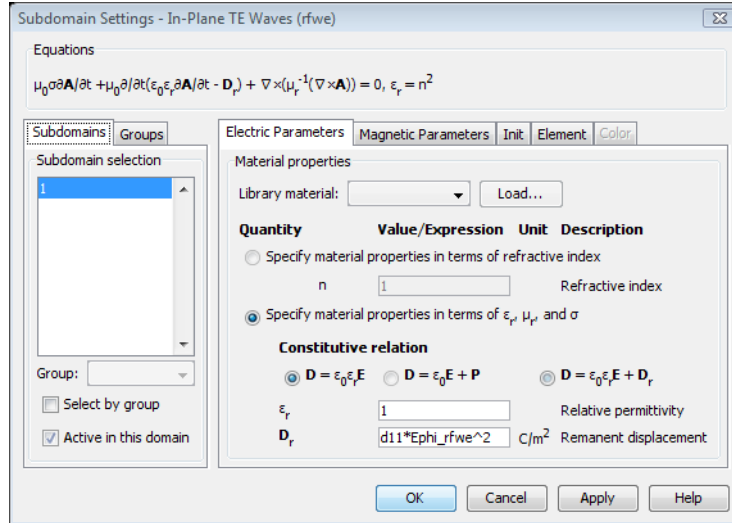
- 2 Click **OK**.

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:

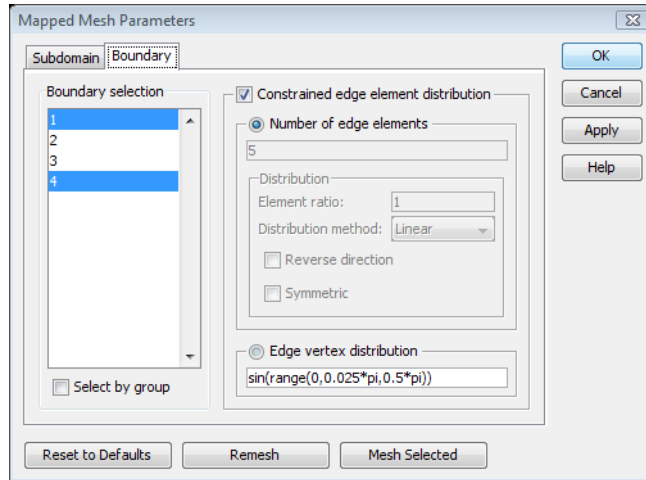
SETTINGS	SUBDOMAIN 1
ϵ_r	1
\mathbf{D}_r	$d11 * E_{phi_rfwe}^2$

- 3 Click **OK**.



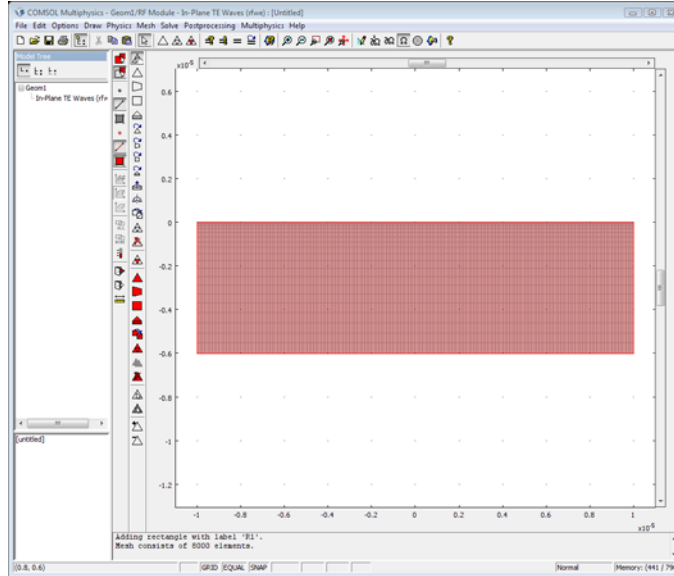
MESH GENERATION

- 1 From the **Mesh** menu, choose **Mapped Mesh Parameters**.
- 2 On the **Boundary** page, select Boundaries 1 and 4 from the **Boundary selection** list.
- 3 Select the **Constrained edge element distribution** check box.
- 4 Click the **Edge vertex distribution** button. In the edit field below, type $\sin(\text{range}(0, 0.025\pi, 0.5\pi))$. This creates a denser mesh closer to the upper boundary.



- 5 Select Boundary 2, click the **Edge vertex distribution** button, and type $\text{range}(0, 5e-8, 20e-6)$ in the edit field.

6 Click the **Remesh** button; when the mesher has finished, click **OK**.



COMPUTING THE SOLUTION

To perform an FFT analysis in MATLAB, the number of time steps that have to be saved is very large, and to store all solutions of the **A**-field results in a huge model file. However, for the FFT, it is only interesting to look at the field at the output boundary. You can take advantage of this fact by first solving the model with a large number of output time steps using a single-step parametric sweep to store the quantity of interest in a global variables log file. Once the solver has finished, you reset the model, change the output time steps, and solve the model again (this time without using a parametric sweep). The following instructions describe the relevant modeling procedure to take whether you want to skip or perform the FFT analysis.

To be able to view the time-evolution of the electric field at the input and output boundaries while the solver computes the solution, first set up a probe plot.

- 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 Point 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 Point 2.
- 6 Select the **Plot all plots in the same axis** check box. Click **OK**.

Proceed to set up the solver.

- 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 On the **Time Stepping** page, select **Manual** from the **Time steps taken by solver** list. Set the **Time step** to 2e-16.
- 4 If you do not intend to perform an FFT analysis or want to use a presaved data file for this purpose, skip directly to Step 17; otherwise continue with the next step.
- 5 Change the **Times to store in output** to **Time steps from solver**.
- 6 Click **Apply**.

Next, set up and solve a single-step parametric sweep to generate a log file.

- 7 From the **Solve** menu, choose **Parametric Sweep**.
- 8 In the **Parameter names** edit field, enter a dummy name, for example, **dummyparam**.
- 9 In the **Parameter values** edit field, enter a single value, for example, 0.
- 10 In the **Global variables to evaluate** edit field, type **E_{out}**.
- 11 In the **Log file name** edit field, type **second_harmonic_generation_outdata.txt** to save the log in the current directory or click **Browse** and use the **Log File** dialog box to save the file in a suitable directory, preferably one on your MATLAB path.
- 12 Click the **Solve** button in the **Parametric Sweep** dialog box.
- 13 When the solver has finished, click **OK** to close the **Parametric Sweep** dialog box.
- 14 From the **File** menu, select **Reset Model**. In the dialog box that appears, click **OK** to confirm.
- 15 Return to the **Solver Parameters** dialog box.
- 16 On the **Time Stepping** page, set the **Times to store in output** to **Specified times**.
- 17 Click **OK** to close the **Solver Parameters** dialog box.
- 18 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

- 1 Select **Plot Parameters** from the **Postprocessing** menu.

- 2 On the **General** page, make sure that the **Surface**, **Boundary**, and **Geometry edges** check boxes are selected.
- 3 From the **Solution at time** list, select **6.1e-14**.
- 4 On the **Surface** page, type `Ephi_rfwe` in the **Expression** edit field.
- 5 Click the **Height Data** tab, select the **Height data** check box, and type `Ephi_rfwe` in the **Expression** edit field.
- 6 On the **Boundary** page, type `E_lim` in the **Expression** edit field.
- 7 Click the **Height Data** tab, select the **Height data** check box, and type `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 On the **General** page, select **9e-14** from the **Solution at time** list. Click **OK**. You should now see the plot in Figure 4-12.

FFT ANALYSIS USING MATLAB

If you have MATLAB, you can go on to compute and plot the Fourier transform of the electric field at the output boundary that you stored in the parametric sweep log file. This data is also provided in the text file `second_harmonic_generation_outdata.txt`, located in the model's Model Library folder.

Now switch to the MATLAB workspace.

- 1 If you stored the data file in a directory that is not on your MATLAB path, use the command `cd` to navigate to that directory. Open the file in a text editor and remove the initial header before saving it again.
- 2 At the MATLAB prompt, enter the following commands to fetch the output field data from the file:

```
S = load('second_harmonic_generation_data.txt', '-ascii');
xdata = S(:,2);
ydata = S(:,3);
```

(The first column in the data text file contains the values for the dummy parameter used in the parametric sweep and can thus be discarded.)

- 3** Now you can proceed to perform the FFT analysis. Because the output boundary field is effectively zero for the first half of the time steps in the file, you can restrict the time interval for the analysis by discarding the first 300 time steps.

```
T = xdata(301:end);  
Eout = ydata(301: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 298.

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 294. The differences are that the nonlinear material parameters have been removed, the geometry is smaller, and the laser pulse is shorter. This is done to reduce the execution time and the size of the model file 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 294. This model uses the same YAG laser pulse, but with a characteristic time of 4 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 on 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.

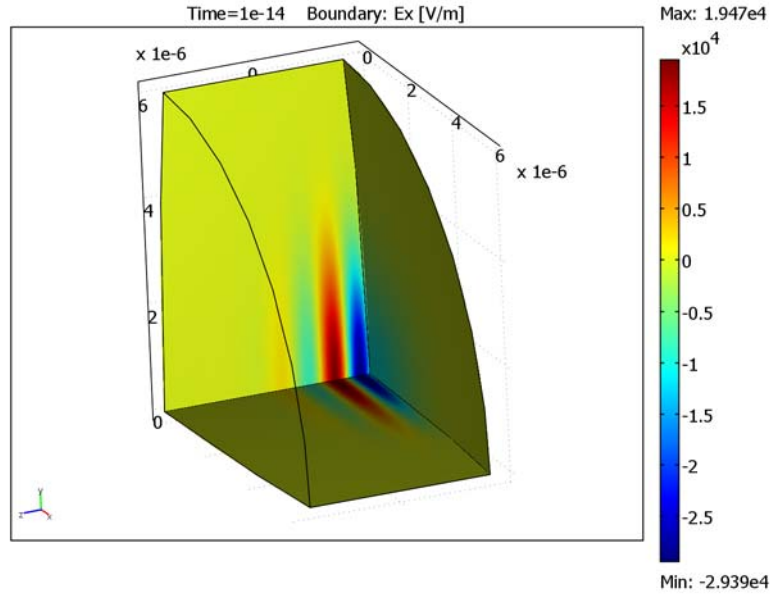


Figure 4-14: The pulse entering the domain at $t=10$ fs. The spot size is shown on the input boundary to the left.

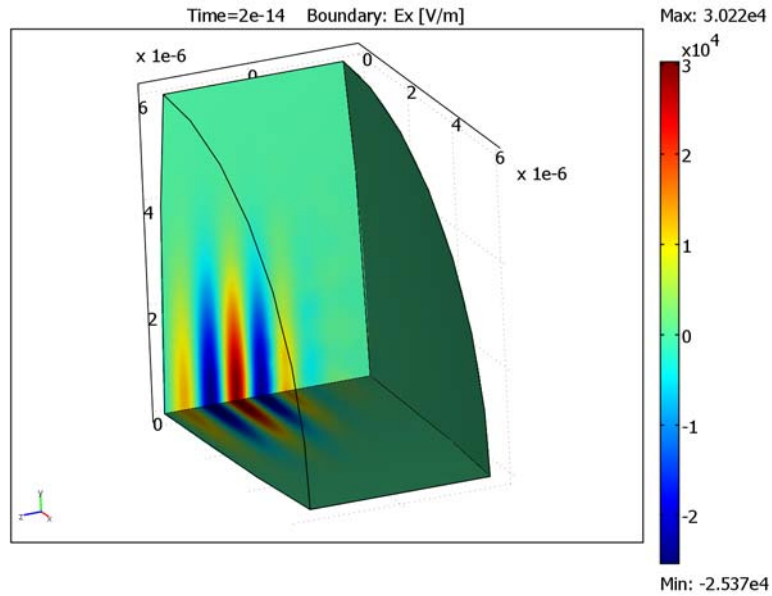


Figure 4-15: After 20 fs the pulse has passed the minimum waist and soon reaches the output boundary.

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** from 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); when done, click **OK**.

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	4[fs]	Pulse width

GEOMETRY MODELING

All the dialog boxes for specifying the primitive objects are available from the **Draw>Specify Objects** menu. 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**.
- 3 Draw a circle with the properties according to the table below.

NAME	RADIUS	BASE	(X, Y)
C1	$6e-6$	Center	(0, 0)

- 4 Draw rectangles with the properties given in the following table:

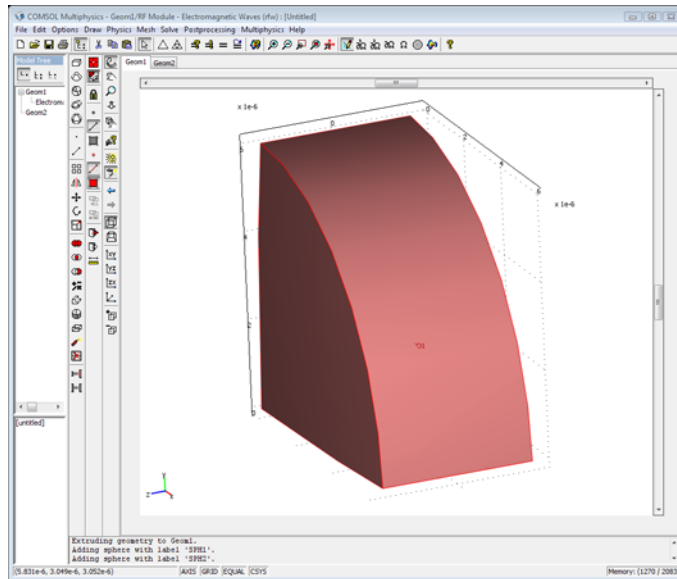
NAME	WIDTH	HEIGHT	BASE	CORNER
R1	$6e-6$	$1.2e-5$	Corner	($-6e-6$, $-6e-6$)
R2	$6e-6$	$6e-6$	Corner	(0, $-6e-6$)

- 5 Click the **Zoom Extents** button on the Main toolbar.
- 6 Select both rectangles, R1 and R2, then click the **Union** toolbar button.

- 7 Click the **Delete Interior Boundaries** toolbar button.
- 8 Press Ctrl+A to select all objects, then click the **Create Composite Object** button on the Draw toolbar. In the dialog box that appears, enter C1-C01 in the **Set formula** edit field, then click **OK**.
- 9 From the **Draw** menu, select **Extrude**. In the **Extrude** dialog box, specify the **Distance** as $4\text{e-}6$, then click **OK** to create the object EXT1.
- 10 It is desired to have the spherical wave fronts of the beam coincide with the input and output boundaries. To achieve this, for each of these two boundaries create a sphere with radius equal to the value of the expression R—the “analytical radius” along the z-axis that you will define shortly—evaluated at the boundary, place it on the z-axis so that it just touches the boundary, and then take the intersection with EXT1. Thus, create two spheres with the properties in the following table (the axis base point is the centerpoint):

NAME	RADIUS	AXIS BASE POINT
SPH1	$7.22713\text{e-}5$	$(0, 0, 7.22713\text{e-}5-2\text{e-}6)$
SPH2	$7.22713\text{e-}5$	$(0, 0, -7.22713\text{e-}5+2\text{e-}6)$

- 11 Press Ctrl+A to select all objects, then click the **Intersection** button.
- 12 Click the **Headlight** toolbar button. You should now see the geometry shown in the figure below. Note that the input and output boundary surfaces are spherical now.



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	$1/\sqrt{\text{epsilon}0_{\text{rfw}}*\mu0_{\text{rfw}}}$	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

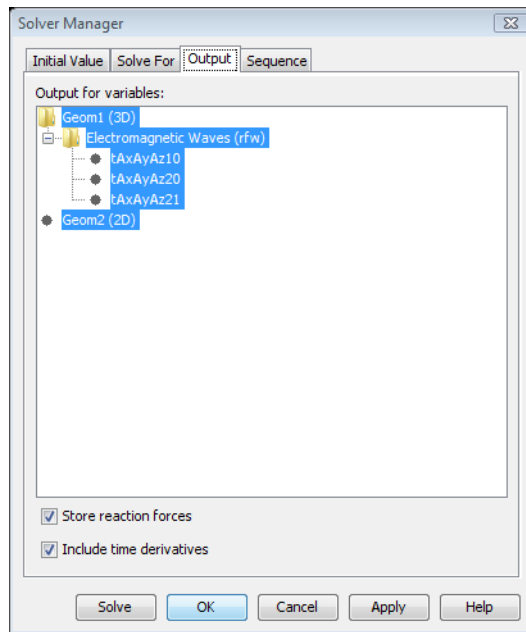
- 1 From the **Mesh** menu, choose **Free Mesh Parameters**.
- 2 Click the **Custom mesh size** button and enter $1e-7$ in the **Maximum element size** edit field.
- 3 Click the **Advanced** tab.
- 4 Enter 0.12 in the **x-direction scale factor** and **y-direction scale factor** edit fields. This keeps 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 range (0, 5e-15, 2e-14).
- 3 Click the **Settings** button.
- 4 In the dialog box that appears, select the **Coarse solver** item in the tree view.
- 5 Choose **Off** from the **Check tolerances** list and click **OK**.
- 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 (hidden) to make it 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 $1\text{e-}14$ and $2\text{e-}14$ to see how the pulse propagates. These results are plotted in Figure 4-14 on page 308 and in Figure 4-15 on page 309.

I N D E X

- 3D electromagnetic waves model 116, 156
- 3D waveguide 116
- A**
 - absorbing layers 222
 - anisotropic permeability 52
 - antenna impedance 66, 75
 - antennas 183
 - conical 65
 - magnetic dipole 80
 - RF 65
 - application mode
 - Axisymmetric TE waves 81
 - Axisymmetric TM waves 71
 - Bioheat equation 137
 - Boundary mode analysis 122
 - Electromagnetic waves 122
 - In-plane TE waves 225, 233, 245
 - Perpendicular currents 35
 - Perpendicular Hybrid-Mode Waves 261
 - Perpendicular hybrid-mode waves 257
 - Plane Strain 261
 - TM waves 137
 - axisymmetric TE waves model 80
 - axisymmetric TM waves model 65
 - azimuthal magnetic field 66
- B**
 - balanced antennas 183
 - bandgap analysis 239
 - bioheat equation model 129
 - birefringence 260
 - stress-induced 268
 - boundary mode analysis 122
 - boundary mode analysis model 116, 156
- C**
 - capacitive coupling 99
 - cavity resonances 112
 - characteristic impedance 75
 - circuit port
 - SPICE Import boundary condition 78
 - circulator 50
 - cladding 254
 - coaxial cable, feeding a waveguide 204
 - confined mode 254
 - conical antennas 65
 - convergence analysis 273
 - coupling variable
 - scalar 75
 - cross-section plot 202
 - cutoff frequency 117, 157, 232, 236
- D**
 - dipole, magnetic 80
 - drift, in eigenfrequency 96
- E**
 - effective mode index 255
 - eigenfrequency drift 96
 - electromagnetic wave model 3D 116, 156
 - errors in modeling 276
 - expansion coefficient 277
- F**
 - far-field
 - computation 77
 - radiation patterns 69
 - ferrite circulator 50
 - FFT 296, 303
 - fiber
 - optical 254
 - silica glass 254
 - single-mode 254
 - Floquet periodicity boundary conditions 240
 - frequency
 - normalized 255
- G**
 - Gaussian beam 294, 307
 - generalized plane strain 277

- gyromagnetic ratio 52
- H**
 - H bend 111
 - harmonic generation 295
 - H-bend waveguide 31, 50
- I**
 - impedance
 - of antenna 66, 75
 - impedance, of microstrip line 20
 - in-plane TE waves model 50, 111, 222, 230
 - intensity
 - optical 272
 - ionosphere, of the Earth 43
- L**
 - Larmor frequency 52
 - lobe pattern 188
 - low-reflecting boundary conditions 185
- M**
 - magnetic dipole 80
 - magnetic frill 14
 - matched boundary conditions 66, 112
 - matched port boundary conditions 53
 - microprism 222
 - microstrip line, impedance of 20
 - microwave circulator 50
 - microwave filters 96
 - microwave waveguide 116
 - mode analysis 270
 - model
 - stress-optical effects 260
 - stress-optical effects, generalized 277
 - modeling errors 276
 - models, overview of 2
 - monoconical antennas 65
 - moving mesh 97
- N**
 - Navier's equation 279
 - weak form 279
 - near-field radiation patterns 67
 - normalized frequency 255
- O**
 - optical fiber 254
 - optical intensity 272
 - optical mode analysis 270
 - optical waveguide 222, 254
 - optics 254
 - optimum radiation frequency 186
 - optoelectronics 254
- P**
 - parametric solver 76
 - patch antennas 183
 - perfect electric conductors 185
 - perfectly matched layers 185, 222
 - Perpendicular Hybrid-Mode Waves 261
 - perpendicular hybrid-mode waves model 254
 - perturbation
 - of material parameters 276
 - photonic bandgap 230, 239
 - photonic crystal 239
 - photonic microprism 222
 - photonic waveguide 260
 - Plane Strain 277
 - application mode 261
 - generalized 277
 - plane strain
 - generalized 277
 - PML constant 185
 - PML layer 185
 - PMLs. *see* perfectly matched layers
 - Port boundary condition 204
 - power flow 272
 - Poynting vector 272
- Q**
 - quarter-wave resonance 98
- R**
 - radiation patterns 75, 187
 - far-field 69
 - near-field 67
 - reactance 76
 - rectangular waveguides 204

- refractive index 222, 254
- resistance 76
- resonance 98, 112
- S** SAR 142
 - scattering boundary conditions 66
 - scattering parameter 32, 50, 75, 111
 - Schumann resonance frequencies 42
 - sea bed logging 211
 - second harmonic generation 295
 - sensitivity analysis 276
 - silica glass fiber 254
 - single-mode fiber 254
 - single-mode range 116
 - skin depth 44
 - solver
 - parametric 76
 - S-parameter 32, 50, 75, 111, 116
 - S-parameters 210
 - specific absorption rate 129, 142
 - spherical waves 66
 - SPICE Import
 - model using 78
 - stress-induced birefringence 268
 - stress-optical effects 260
 - generalized 277
 - stress-optical tensor 260
 - symmetry boundary conditions 66
- T** thermal expansion 96
 - TM waves model 129
 - Touchstone file, exporting to 64
 - transverse electric waves 112, 117, 157
 - transverse electromagnetic fields 66
 - typographical conventions 6
- W** waveguide
 - 3D 116
 - H bend 111
 - H-bend 31, 50
 - microwave 116
 - optical 222, 254
 - photonic 260
 - weak form 279
- Y** YAG laser 294, 307

