Introduction to RF Module

VERSION 4.3

COMSOL

Introduction to the RF Module

Version: May 2012 COMSOL 4.3

Contact Information

Visit www.comsol.com/contact for a searchable list of all COMSOL offices and local representatives. From this web page, search the contacts and find a local sales representative, go to other COMSOL websites, request information and pricing, submit technical support queries, subscribe to the monthly eNews email newsletter, and much more.

If you need to contact Technical Support, an online request form is located at www.comsol.com/support/contact.

Other useful links include:

- Technical Support www.comsol.com/support
- Software updates: www.comsol.com/support/updates
- Online community: www.comsol.com/community
- Events, conferences, and training: www.comsol.com/events
- Tutorials: www.comsol.com/products/tutorials
- Knowledge Base: www.comsol.com/support/knowledgebase

Part No. CM021004

Contents

Introduction

The RF Module is used by engineers and scientists to understand, predict, and design electromagnetic wave propagation and resonance effects in high-frequency applications. Simulations of this kind result in more powerful and efficient products and engineering methods. It allows its users to quickly and accurately predict electromagnetic field distributions, transmission, reflection, and power dissipation in a proposed design. Compared to traditional prototyping, it offers the benefits of lower cost and the ability to evaluate and predict entities that are not directly measurable in experiments. It also allows the exploration of operating conditions that would destroy a real prototype or be hazardous.

This module covers electromagnetic fields and waves in two-dimensional and three-dimensional spaces along with traditional circuit-based modeling of passive and active devices. All modeling formulations are based on Maxwell's equations or subsets and special cases of these together with material laws for propagation in various media. The modeling capabilities are accessed via predefined physics interfaces, referred to as RF interfaces, which allow you to set up and solve electromagnetic models. The RF interfaces cover the modeling of electromagnetic fields and waves in frequency domain, time domain, eigenfrequency, and mode analysis.

Under the hood, the RF interfaces formulate and solve the differential form of Maxwell's equations together with the initial and boundary conditions. The equations are solved using the finite element method with numerically stable edge element discretization in combination with state-of-the-art algorithms for preconditioning and solution of the resulting sparse equation systems. The results are presented using predefined plots of electric and magnetic fields, S-parameters, power flow, and dissipation. You can also display your results as plots of expressions of the physical quantities that you define freely, or as tabulated derived values obtained from the simulation.

The work flow is straightforward and can be described by the following steps: define the geometry, select materials, select a suitable RF interface, define boundary and initial conditions, define the finite element mesh, select a solver, and visualize the results. All these steps are accessed from the COMSOL Desktop. The solver step is usually carried out automatically using default settings, which are tuned for each specific RF interface.

The RF Module's Model Library describes the interfaces and their different features through tutorial and benchmark examples for the different formulations. The library includes models from RF and microwave engineering, optics and photonics, tutorial models for education, and benchmark models for verification and validation of the RF interfaces.

This introduction is intended to give you a jump start in your modeling work. It has examples of the typical use of the RF Module, a list of the interfaces with a short description, and a tutorial example that introduces the modeling workflow.

The Use of the RF Module

The RF interfaces are used to model electromagnetic fields and waves in high frequency applications. The latter means that it covers the modeling of devices that are above about 0.1 electromagnetic wavelength in size. Thus, it may be used to model microscale optical devices or for human size devices operating at frequencies above 10 MHz.

RF simulations are frequently used to extract S-parameters characterizing the transmission and reflection of a device. Figure 1 shows the electric field distribution in a dielectric loaded H-bend waveguide component. A rectangular TE_{10} waveguide mode is launched into an inport at the near end of the device and absorbed by the outport at the far end. The bend region is filled by silica glass.

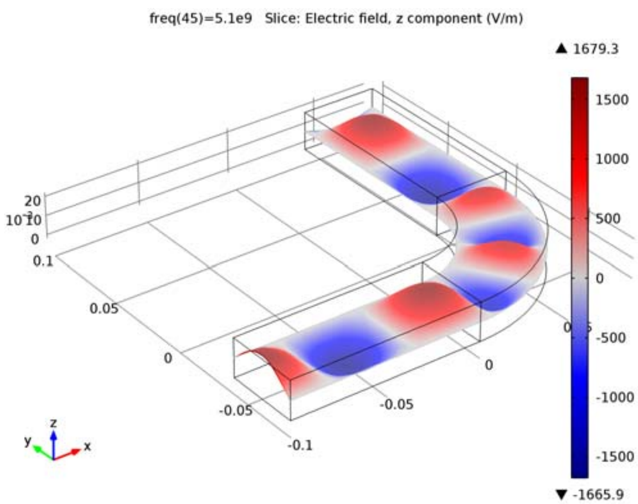


Figure 1: Electric field distribution in a dielectric loaded H-bend waveguide. From the RF Module Model Library model H-Bend Waveguide 3D.

The transmission and reflection of the device is obtained in the form of S-parameters, as in Figure 2.

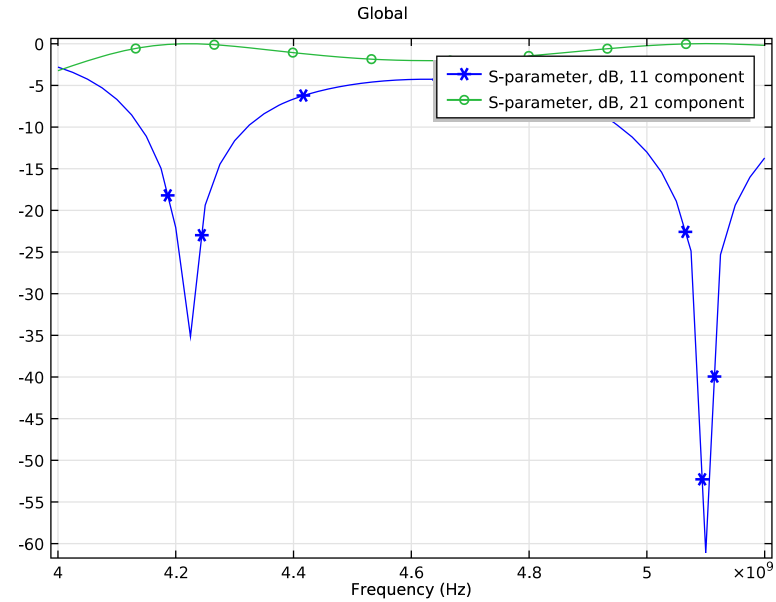


Figure 2: The S-parameters, on a dB scale, as a function of the frequency.

S-parameters can be exported in the Touchstone file format for further use in system simulations and are, by themselves, useful performance measures of a design.

In Figure 3 and Figure 4, a model from the RF Module Model Library shows how a human head absorbs a radiated wave from an antenna held next to the ear. The temperature is increased by the absorbed radiation.

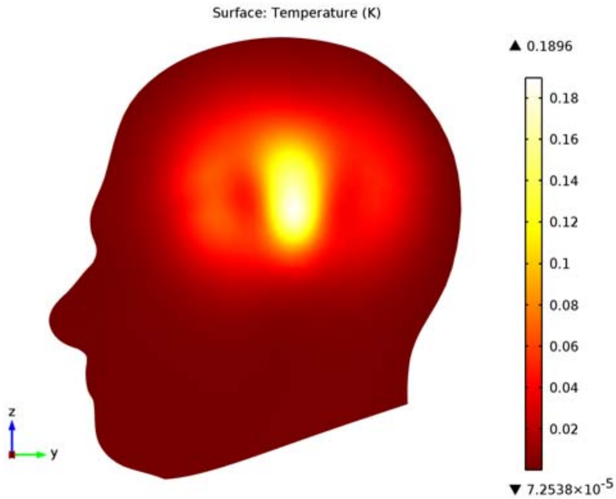


Figure 3: The local increase in temperature in a human head due to absorption of electromagnetic energy from an antenna held next to the ear. From the RF Module Model Library model Absorbed Radiation (SAR) in the Human Brain.

The SAR (specific absorption rate) value is of specific interest to designers of mobile telephones and is readily obtained from the simulation.

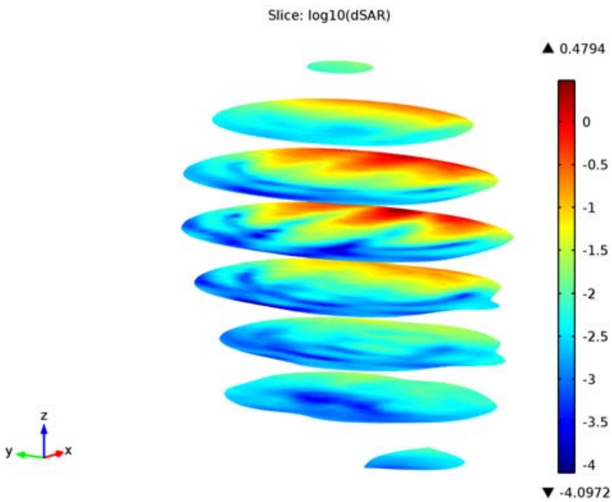


Figure 4: Log-scale slice plot of the local specific absorption rate (SAR) in a human head.

The RF Module also offers a comprehensive set of features for 2D modeling including both source driven wave propagation and mode analysis. Figure 5 shows mode analysis of a step-index profile optical fiber.

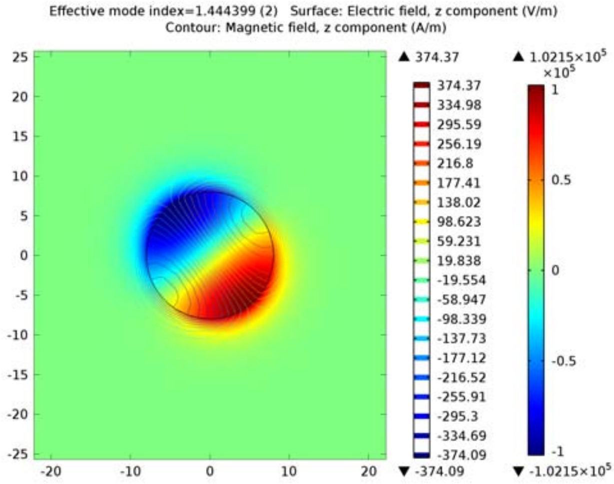


Figure 5: The surface plot visualizes the longitudinal component of the electric field in the fiber core. From the RF Module Model Library model Step Index Fiber.

Both in 2D and 3D, the analysis of periodic structures is popular. Figure 6 is an example of wave propagation in a photonic crystal.

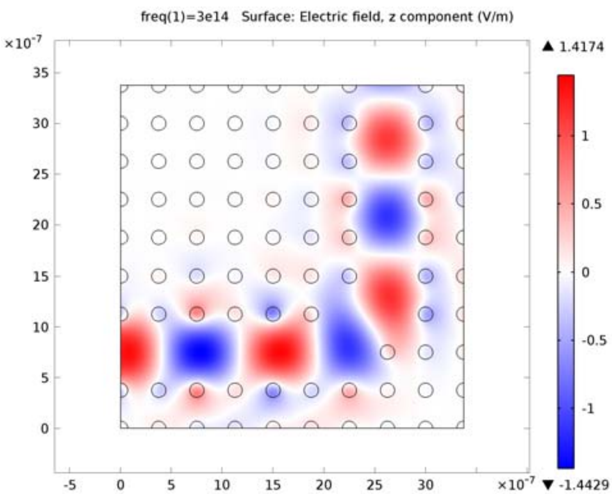


Figure 6: The out-of-plane component of the electric field shows how the wave is confined to propagate along a path defined by removing some pillars in the photonic crystal. From the RF Module Model Library model Photonic Crystal.

It is also possible to perform Body-of-Revolution (BOR) simulations in 2D axisymmetry. Figure 7 illustrates the modeling of a monoconical antenna with coaxial feed.

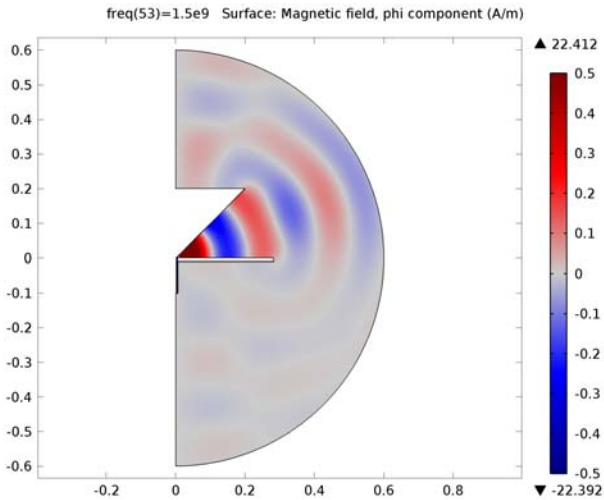


Figure 7: Axisymmetric model of a monoconical antenna with coaxial feed. The azimuthal component of the magnetic field is shown. From the RF Module Model Library model Conical Antenna.

The RF Module has a vast range of tools to evaluate and export the results, for example, evaluation of far-field, feed impedance, and scattering matrices (S-parameters). S-parameters can be exported in the Touchstone file format.

The combination of full-wave electromagnetic field modeling and simplified circuit-based modeling is the ideal basis for design, exploration, and optimization. More complex system models can be exploited using circuit-based modeling while maintaining links to full field models for key devices in the circuit allows for design innovation and optimization on both levels.

The RF Module Interfaces

The RF interfaces are based upon Maxwell's equations or subsets and special cases of these together with material laws. In the module, these laws of physics are translated by the RF interfaces to sets of partial differential equations with corresponding initial and boundary conditions.

The RF physics define a number of features. Each feature represents an operation that describes a term or condition in the underlying Maxwell-based formulation. Such a term or condition may be defined in a geometric entity of the model, such as a domain, boundary, edge (for 3D models), or point.

Figure 8 uses the Coaxial Waveguide Coupling model from the RF Module Model Library to show the Model Builder window and the settings window for the selected Wave Equation, Electric 1 feature node. The Wave Equation, Electric 1 node adds the terms to the model equations to a selected geometrical domain representing the Electromagnetic Waves domain in the model.

Furthermore, the Wave Equation, Electric 1 feature node may link to the Materials feature node to obtain physical properties such as relative permittivity, in this case the relative permittivity of a user-defined dielectric. The properties, defined by the Dielectric material, can be functions of the modeled physical quantities, such as temperature. In the same fashion, the Perfect Electric Conductor 1 boundary condition feature adds the reflecting boundary conditions that limit the Electromagnetic Waves domain.

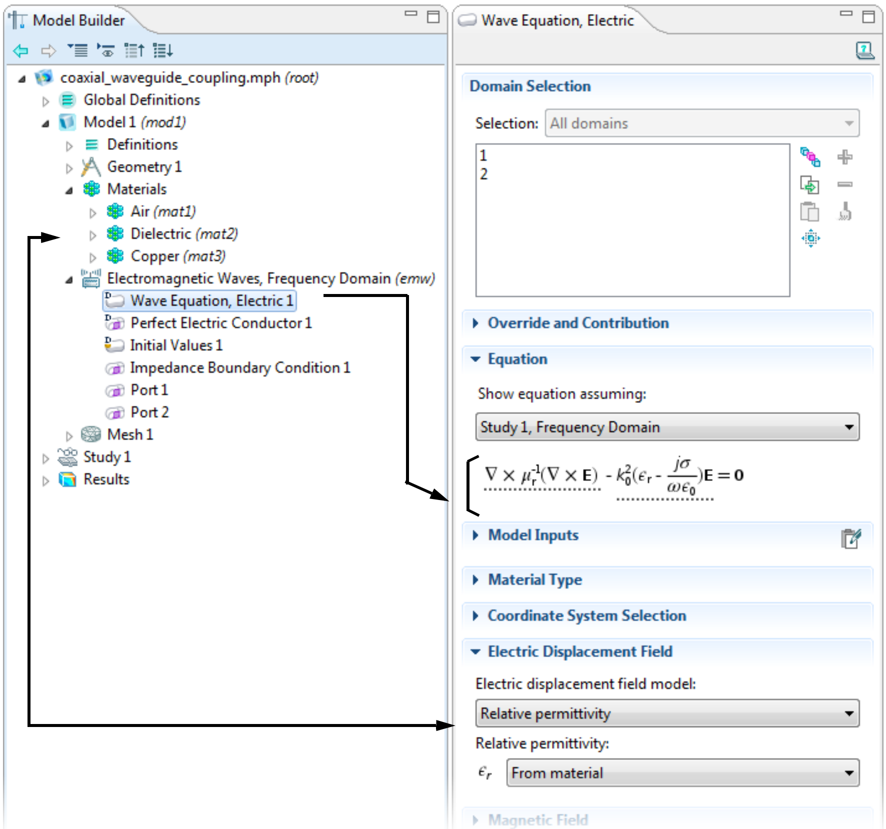
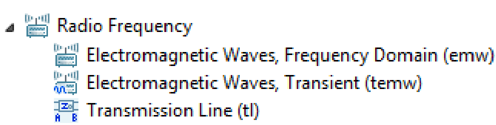


Figure 8: The Model Builder (left), and the Wave Equation, Electric settings window (right). The Equation section shows the model equations and the terms added by the Wave Equation, Electric 1 node to the model equations. The added terms are underlined with a dotted line. The text also explains the link between the Dielectric node and the values for the relative permittivity.

The figure below shows the RF interfaces as displayed in the **Model Wizard** for this module.



This module includes RF interfaces ( ) for frequency-domain modeling and time-domain modeling, respectively. It also has the Microwave Heating interface found under Heat Transfer. Also see “Physics List by Space Dimension and Study Type” on page 14. A brief overview of the RF interfaces follows.

Electromagnetic Waves, Frequency Domain

The Electromagnetic Waves, Frequency Domain interface ( ) solves a frequency-domain wave equation for the electric field given its sources. The sources can be in the form of point dipoles, line currents, or incident fields on boundaries or domains. It is used primarily to model electromagnetic wave propagation in different media and structures. Variants of the formulation solves an eigenvalue problem to find the eigenfrequencies of a structure or, at a prescribed frequency, solves an eigenvalue problem to find the propagating modes in waveguides and transmission lines. Some typical applications that are simulated in the interface are waveguides and transmission lines, filters and resonators, antennas, and RF connectors and couplers.

Electromagnetic Waves, Transient

The Transient Electromagnetic Waves, Transient interface ( ) solves a time-domain wave equation for the electric field given its sources. The sources can be in the form of point dipoles, line currents, or incident fields on boundaries or domains. It is used primarily to model electromagnetic wave propagation in different media and structures when a time-domain solution is required—for example, for non-sinusoidal waveforms or for nonlinear media. Typical applications involve the propagation of electromagnetic pulses and the generation of harmonics in nonlinear optical media.

Transmission Line

The Transmission Line interface ( ), solves the time-harmonic transmission line equation for the electric potential. The interface is used when solving for electromagnetic wave propagation along one-dimensional transmission lines and is available in 1D, 2D and 3D. The Eigenfrequency and Frequency Domain study types are available. The frequency domain study is used for source driven simulations for a single frequency or a sequence of frequencies. Typical applications involve the design of impedance matching elements and networks.

Microwave Heating

The Microwave Heating interface ( ) combines the features of an Electromagnetic Waves, Frequency Domain interface with those of the Heat Transfer interface. The predefined interaction adds the electromagnetic losses from the electromagnetic waves as a heat source. This interface is based on the assumption that the electromagnetic cycle time is short compared to the thermal time scale (adiabatic assumption).

Electrical Circuits

The Electrical Circuit interface (icon) can be connected to an RF interface. The lumped voltage and current variables from the circuits are translated into boundary conditions applied to the distributed field model. Typical applications include the modeling of transmission lines and antenna feeding.

Physics List by Space Dimension and Study Type

The table below list the interfaces available specifically with this module in addition to the COMSOL Multiphysics basic license.

PHYSICS	ICON	TAG	SPACE DIMENSION	PRESET STUDIES
AC/DC				
Electrical Circuit		cir	Not space dependent	stationary; frequency domain; time dependent
Heat Transfer				
Electromagnetic Heating				
Microwave Heating		mh	3D, 2D, 2D axisymmetric	stationary; frequency domain; time dependent; boundary mode analysis; frequency-stationary; frequency transient
Radio Frequency				
Electromagnetic Waves		emw	3D, 2D, 2D axisymmetric	eigenfrequency; frequency domain; frequency-domain modal; boundary mode analysis
Transient Electromagnetic Waves		temw	3D, 2D, 2D axisymmetric	eigenfrequency; time dependent; time dependent modal
Transmission Line		tl	3D, 2D, 1D	eigenfrequency; frequency domain

Opening the Model Library

To open any RF Module Model Library model, select **View > Model Library** from the main menu in COMSOL Multiphysics. In the Model Library window that opens, expand the RF Module folder and browse or search the contents. Click **Open Model**

and PDF to open the model in COMSOL Multiphysics and a PDF to read background theory about the model including the step-by-step instructions to build it.

The MPH-files in the COMSOL model libraries can have two formats—Full MPH-files or Compact MPH-files.

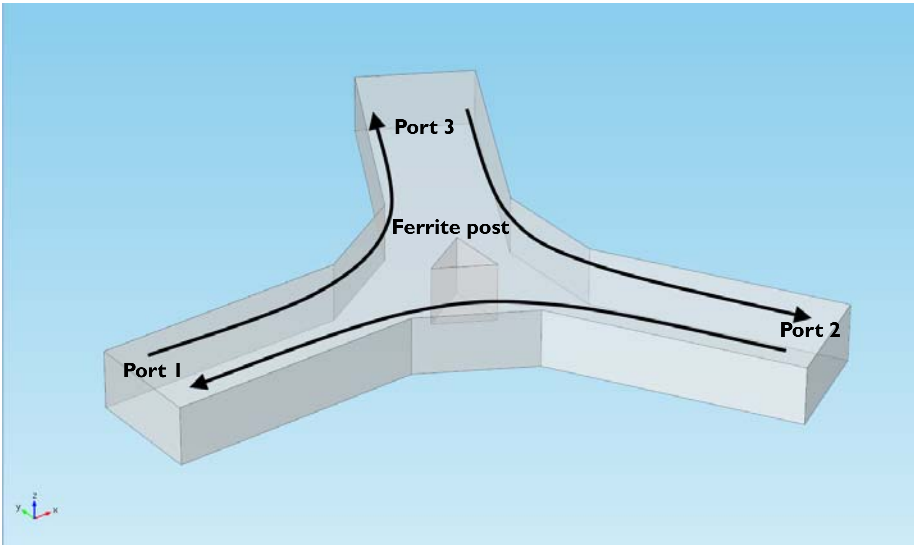
- Full MPH-files, including all meshes and solutions. In the Model Library these models appear with the icon. If the MPH-file's size exceeds 25MB, a tip with the text "Large file" and the file size appears when you position the cursor at the model's node in the Model Library tree.
- Compact MPH-files with all settings for the model but without built meshes and solution data to save space on the DVD (a few MPH-files have no solutions for other reasons). You can open these models to study the settings and to mesh and re-solve the models. It is also possible to download the full versions—with meshes and solutions—of most of these models through Model Library Update. In the Model Library these models appear with the icon. If you position the cursor at a compact model in the Model Library window, a No solutions stored message appears. If a full MPH-file is available for download, the corresponding node's context menu includes a Model Library Update item.

A model from the Model Library is used as a tutorial in this guide. See "Tutorial Example: Impedance Matching of a Lossy Ferrite 3-port Circulator", which starts on the next page.

Tutorial Example: Impedance Matching of a Lossy Ferrite 3-port Circulator

Introduction

A microwave circulator is a nonreciprocal multiport device. It has the property that a wave incident on port 1 is routed into port 3 but a wave incident on port 3 is not routed back into port 1 but is instead routed into port 2, and so on. This property of circulators is used to isolate microwave components from each other. A typical example is when connecting a transmitter and a receiver to a common antenna. By connecting the transmitter, receiver, and antenna to different ports of a circulator, the transmitted power is routed to the antenna whereas any power received by the antenna goes into the receiver. Circulators typically rely on the use of ferrites, a special type of highly permeable and low-loss magnetic material that is anisotropic for a small RF signal when biased by a much larger static magnetic field. In the example, a three-port circulator is constructed from three rectangular waveguide sections joining at 120° and with a ferrite post inserted at the center of the joint.



The post is magnetized by a static H_0 bias field along its axis. The bias field is supplied by external permanent magnets which are not explicitly modeled in this tutorial.

Impedance Matching

An important step in the design of any microwave device is to match its input impedance for a given operating frequency. Impedance matching is equivalent to minimizing the reflections at the inport. The parameters that need to be determined are the size of the ferrite post and the width of the wider waveguide section surrounding the ferrite. In this tutorial, these are varied in order to minimize the reflectance. The scattering parameters (S-parameters) used as measures of the reflectance and transmittance of the circulator are automatically computed.

The nominal frequency for the design of the device is selected to 3 GHz. The circulator can be expected to perform reasonably well in a narrow frequency band around 3 GHz so a frequency range of 2.6 – 3.4 GHz is studied. It is desired that the device operates in single mode. Thus a rectangular waveguides cross section of 6.67 cm by 3.33 cm is selected. This sets the cut-off frequency for the fundamental TE_{10} mode to 2.25 GHz. The cut-off frequencies for the two nearest higher modes, the TE_{20} and TE_{01} modes, are both at 4.5 GHz, leaving a reasonable safety margin.

Model Definition

One of the rectangular ports is excited by the fundamental TE_{10} mode. At the ports, the boundaries are transparent to the TE_{10} mode. The following equation applies to the electric field vector $\mathbf{E}$ inside the circulator:

$$\nabla\times(\mu_r^{-1}\nabla\times\mathbf{E})-k_0^2\left(\varepsilon_r-\frac{j\sigma}{\omega\varepsilon_0}\right)\mathbf{E} = 0$$

where μ_r denotes the relative permeability tensor, ω the angular frequency, σ the conductivity tensor, ε_0 the permittivity of vacuum, ε_r the relative permittivity tensor, and $\mathbf{k}_0$ is the free space wave number. The conductivity is zero everywhere. Losses in the ferrite are introduced as complex-valued permittivity and permeability tensors. The magnetic permeability is of key importance as it is the anisotropy of this parameter that is responsible for the nonreciprocal behavior of the circulator. For simplicity, the rather complicated material expressions are predefined in a text file that is imported into the model. As a reference, the expressions are also included in the next section.

The Lossy Ferrite Material Model

Complete treatises on the theory of magnetic properties of ferrites can be found in 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 expressions are a linearization for a small-signal analysis around this operating point. Under these assumptions, and including 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

$$\kappa = -j\mu_0\chi_{xy}$$

$$\mu = \mu_0(1+\chi_{xx})$$

and the unique elements of the magnetic susceptibility tensor χ are given by:

$$\chi_{xx} = \frac{\omega_0\omega_m(\omega_0^2 - \omega^2) + \omega_0\omega_m\omega^2\alpha^2}{(\omega_0^2 - \omega^2(1+\alpha^2))^2 + 4\omega_0^2\omega^2\alpha^2} - j\frac{\alpha\omega\omega_m(\omega_0^2 + \omega^2(1+\alpha^2))}{(\omega_0^2 - \omega^2(1+\alpha^2))^2 + 4\omega_0^2\omega^2\alpha^2}$$

$$\chi_{xy} = \frac{2\omega_0\omega_m\omega^2\alpha}{(\omega_0^2 - \omega^2(1+\alpha^2))^2 + 4\omega_0^2\omega^2\alpha^2} + j\frac{\omega\omega_m(\omega_0^2 - \omega^2(1+\alpha^2))}{(\omega_0^2 - \omega^2(1+\alpha^2))^2 + 4\omega_0^2\omega^2\alpha^2}$$

where

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

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

$$\alpha = \frac{\mu_0\gamma\Delta H}{2\omega}$$

Here μ_0 denotes the permeability of free space; ω is the angular frequency of the microwave field; ω_0 is the precession resonance frequency (Larmor frequency) of a spinning electron in the applied magnetic bias field, $\mathbf{H}_0$; ω_m is the electron Larmor frequency at the saturation magnetization of the ferrite, $\mathbf{M}_\mathrm{s}$; and γ is the gyromagnetic ratio of the electron. For a lossless ferrite ($\alpha = 0$), the permeability becomes infinite at $\omega = \omega_0$. In the lossy ferrite ($\alpha \neq 0$), this resonance becomes finite and is broadened. The loss factor, α, is related to the line width, $\Delta\mathbf{H}$, of the susceptibility curve near the resonance as given by the last expression above. The material data,

$$M_s = 5.41 \cdot 10^4 \text{ A/m}, \varepsilon_r = 14.5$$

with an effective loss tangent of $2 \cdot 10^{-4}$ and $\Delta H = 3.18 \cdot 10^3$ A/m, are taken for aluminum garnet from Ref. 2. The applied bias field is set to $H_0 = 7.96 \cdot 10^3$ A/m. The electron gyromagnetic ratio taken from Ref. 2 is $1.759 \cdot 10^{11}$ C/kg.

References

1. R.E. Collin, ***Foundations for Microwave Engineering***, 2nd ed., IEEE Press/Wiley-Interscience, 2000.

2. D.M. Pozar, ***Microwave Engineering***, 3rd ed., John Wiley & Sons Inc, 2004.

MODEL WIZARD

These step-by-step instructions guide you through the design and modeling of the lossy three-port circulator in 3D. The first part involves the geometric design and impedance matching at a nominal frequency of 3 GHz. After that, a frequency sweep is performed to see how well it performs in a frequency band of 400 MHz centered at 3 GHz. Finally, computation and a Touchstone file export of the entire S-parameter matrix is performed.

1. Open COMSOL Multiphysics. In the **Model Wizard**, the default **Space Dimension** is **3D**. Click **Next**.
2. In the **Add Physics** tree under **Radio Frequency**, double-click **Electromagnetic Waves, Frequency Domain** to add it to the **Selected physics** list. Click **Next**.
3. Under **Studies>Preset Studies** click **Frequency Domain**.
4. Click **Finish**.

GLOBAL DEFINITIONS - PARAMETERS AND VARIABLES

The geometry is set up using a parameterized approach. This allows you to match the input impedance to that of the connecting waveguide sections by variation of two geometric design parameters. These are dimensionless numbers used to scale selected geometric building blocks.

Note: In this section, two parameters are entered and a set of variables imported from a file to prepare for drawing the circulator geometry, which is described in the section "Geometry Sequence" on page 40. Alternatively, a predefined Model Library file containing the geometry, parameters, and variables can be imported, as described in "Geometry" on page 21. If you import the geometry, you only need to review this section for information.

1 In the **Model Builder**, right-click **Global Definitions** and select **Parameters**.
2 In the **Parameters** settings window under **Parameters**, enter these settings in the **Parameters** table.

Parameters

Name	Expression	Value	Description
sc_chamfer	3	3.0000	Geometry scale factor
sc_ferrite	0.5	0.50000	Geometry scale factor

The lossy ferrite material model is set up by referring to global variables. For convenience the definitions are stored in an external text file that is imported into the model. The external text file also contains comments.

Note: The location of the text files vary based on the installation. For example, if the installation is on your hard drive, the file path might be similar to `C:\Program Files\COMSOL43\models\`.

1 In the **Model Builder**, right-click **Global Definitions** and choose **Variables**.
2 Go to the **Variables** settings window. Under **Variables** click **Load from File**.
3 Browse to the Model Library folder **\RF_Module\RF_and_Microwave_Engineering** and double-click the file `lossy_circulator_3d_parameters.txt`. The variables are imported into the table.

Variables

Name	Expression	Unit	Description
eps0	8.854187817e-12[F/m]	F/m	Permittivity of free space
mu0	4e-7*pi[H/m]	H/m	Permeability of free space
w	2*pi*freq	1/s	Angular frequency
gamma	1.759e11[C/kg]	s·A/kg	Gyromagnetic ratio
H0	(100*1e3/(4*pi))[A/m]	A/m	Applied magnetic bias f...
w0	mu0*gamma*H0	1/s	Larmor frequency
Ms	680e-4[Wb/m^2]/mu0	A/m	Saturation magnetization
wm	mu0*gamma*Ms	1/s	Larmor frequency at sat...
dH	(40*1e3/(4*pi))[A/m]	A/m	Line width
alpha	dH*mu0*gamma/(2*w)		Damping factor

GEOMETRY

In the Global Definitions section, you entered parameters and imported variables in preparation for drawing the geometry. To learn how to draw the circulator, go to "Geometry Sequence" on page 40.

To save time, a predefined model containing the parameters, variables, and geometry can instead be opened from the Model Library.

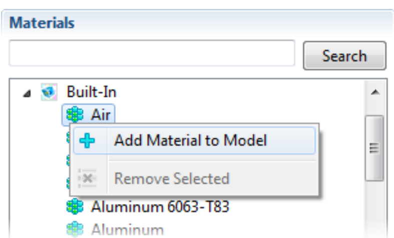
1 From the **View** menu, select **Model Library** .
2 In the **Model Library**, under **RF Module>RF and Microwave Engineering** double-click **lossy_circulator_3d_geom** to open it.

Once the geometry is either drawn or imported, you can then experiment with different dimensions and update the values of `sc_chamfer` and `sc_ferrite` and re-run the geometry sequence.

MATERIALS

The next step is to add material settings to the model. The air that fills most of the volume is available as a built-in material. The lossy ferrite has material assigned to it later, and illustrates how external material data can also be entered directly into the electromagnetic waves model. The walls of the waveguide sections are modeled as perfect conductors and do not require a material.

1 From the main menu, select **View>Material Browser** .
2 Go to the **Material Browser** window. In the **Materials** tree under **Built-In**, right-click **Air** and choose **Add Material to Model** .

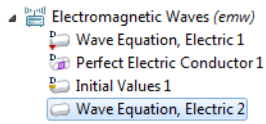


The ferrite enters in the physics interface as a separate, user-defined equation model, referring to the global variables defined on page 19.

Wave Equation, Electric 2

1 In the **Model Builder** right-click **Electromagnetic Waves, Frequency Domain (emw)** node and at the domain level choose **Wave Equation, Electric**.

A **Wave Equation, Electric 2** node is added to the **Model Builder**. The nodes with a 'D' in the upper left corner indicate a default node.

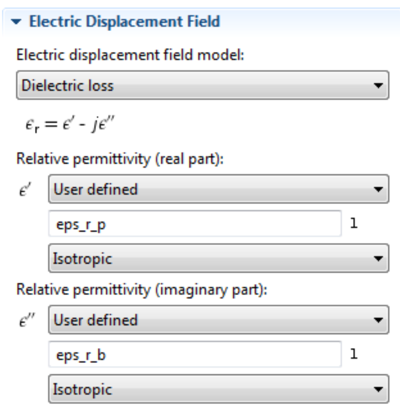


2 Select Domain 2 only.

There are many ways to select geometric entities. When you know the domain to add, such as in this exercise, you can click the **Paste Selection** button and enter the information in the **Selection** field. In this example enter 2 in the **Paste Selection** window. For more information about selecting geometric entities in the **Graphics** window, see the *COMSOL Multiphysics User's Guide*

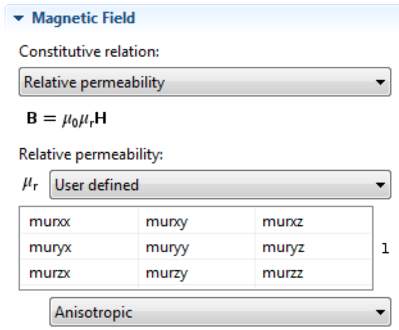
3 Go to the **Wave Equation, Electric** settings window. Under **Electric Displacement Field**:

- From the **Electric displacement field model** list, select **Dielectric loss**.
- From the ε′ list, select **User defined**. In the associated field, enter `eps_r_p`.
- From the ε″ list, select **User defined**. In the associated field, enter `eps_r_b`.

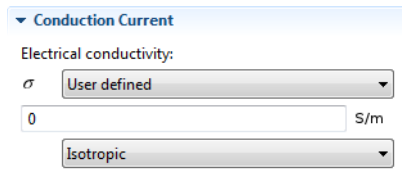


4 Under **Magnetic Field** from the μ_r list, select **User defined** and **Anisotropic**.

5 In the μ_r table, enter the settings as in the figure to the right.



6 Under **Conduction Current** from the σ list, select **User defined** and leave the default value at 0.



Now add ports for excitation and transmission.

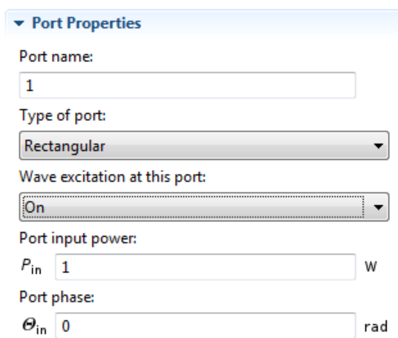
Port 1, Port 2, and Port 3

1 In the **Model Builder**, right-click **Electromagnetic Waves, Frequency Domain (emw)** and at the boundary level choose **Port**.

2 Select Boundary 1 only.

3 Go to the **Port** settings window. Under **Port Properties** from the **Type of port** list, select **Rectangular**.

4 From the **Wave excitation at this port** list, select **On**.

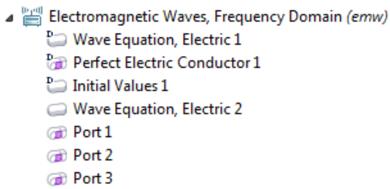


5 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and add another **Port** node. For **Port 2**:

- Select Boundary 18
- Select **Rectangular** as the **Type of port** list

6 Add another **Port** node. For **Port 3**:

- Select Boundary 19
- Select **Rectangular** as the **Type of port** list



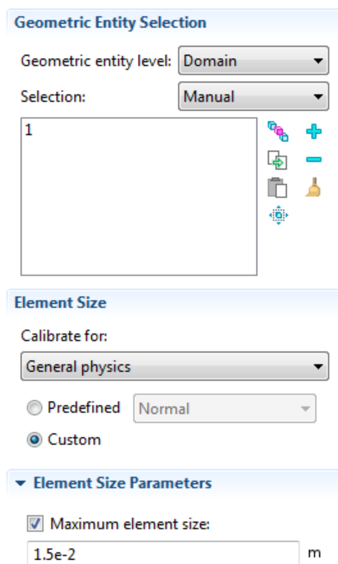
The node sequence in the **Model Builder** should match this figure.

MESH

The mesh respects the geometry but in addition some physics related considerations are needed. The mesh needs to resolve the local wavelength and, for lossy domains, the skin depth. The skin depth in the ferrite is large compared to the size of the domain so the main concern is to resolve the local wavelength. This is done by providing maximum mesh sizes per domain. The rule of thumb is to use a maximum element size that is one fifth of the local wavelength (at the maximum frequency) or less.

Free Tetrahedral 1

1 In the **Model Builder**, right-click **Mesh 1** and choose **Free Tetrahedral**.
2 Right-click **Free Tetrahedral 1** and choose **Size**.
3 Go to the **Size** settings window. Under **Geometric Entity Selection** from the **Geometric entity level** list, select **Domain**.
4 Select Domain 1 only.
5 Under **Element Size** click the **Custom** button.
6 Under **Element Size Parameters** select the **Maximum element size** check box. Enter `1.5e-2` in the field.

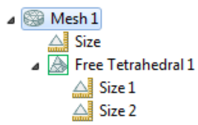


Size 2

1 In the **Model Builder**, right-click **Free Tetrahedral 1** and choose **Size**. A second **Size** node is added to the sequence.
2 Go to the **Size** settings window. Under **Geometric Entity Selection** from the **Geometric entity level** list, select **Domain**.
3 Select Domain 2 only.
4 Under **Element Size** click the **Custom** button.

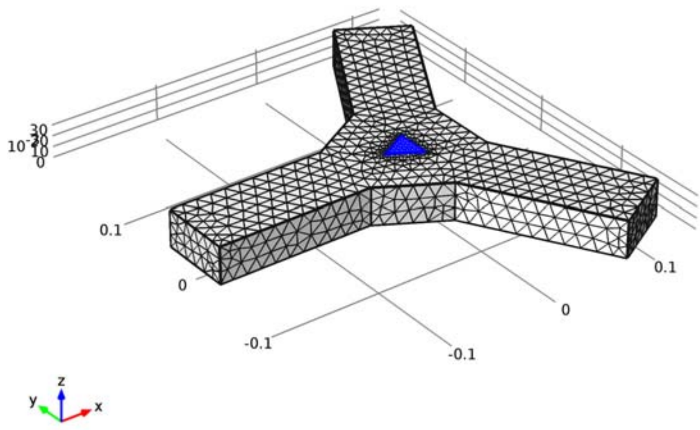
5 Under **Element Size Parameters** select the **Maximum element size** check box. Enter `4.5e-3` in the field.

6 In the **Size** settings window, click **Build All**.



The node sequence in the **Model Builder** should match the figure to the left.

The mesh should match the figure below.



STUDY 1

The final step is to solve for the nominal frequency and inspect the results for possible modeling errors.

1 In the **Model Builder**, expand the **Study 1** node, then click **Step 1: Frequency Domain**.

2 Go to the **Frequency Domain** settings window. Under **Study Settings**, in the **Frequencies** field, enter `3e9`.

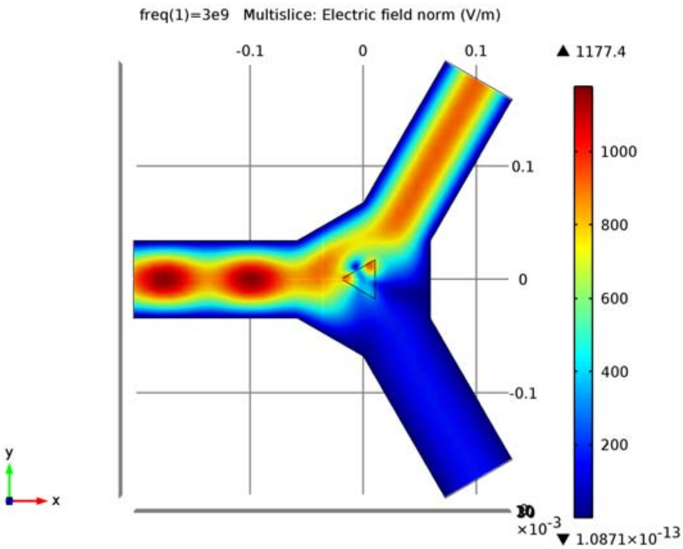
3 In the **Model Builder**, right-click **Study 1** and choose **Compute**.

RESULTS

Electric Field

The default multislice plot shows the electric field norm. It is best viewed from above, so click the **Go to XY View** button on the **Graphics** toolbar.

The electric field norm gives a good indication of where the main power is flowing and where there are standing waves due to reflections from the impedance mismatch at the center.

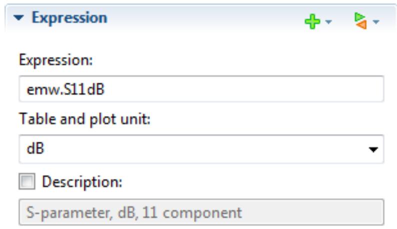


DEFINITIONS - PROBES

The remaining work is to vary the two design parameters in order to minimize reflections at the nominal frequency. To do this, perform parametric sweeps over the design parameters (scale factors). To avoid accumulating a lot of data while solving, throw away the solutions and log only the S-parameter representing reflection in a table. For this purpose, add a Global Variable Probe to the model.

1. In the **Model Builder** under **Model 1**, right-click **Definitions** and choose **Probes>Global Variable Probe**.

2 Go to the **Global Variable Probe** settings window. In the upper-right corner of the **Expression** section, click **Replace Expression**.



3 From the menu, choose **Electromagnetic Waves, Frequency Domain>Ports>S-parameter, dB, 11 component (emw.S11dB)**(or when you know the variable name, enter `emw.S11dB` in the **Expression** field).

STUDY 1

Modify the study in order to vary the scale factor determining the size of the ferrite post. The study type is still Frequency Domain.

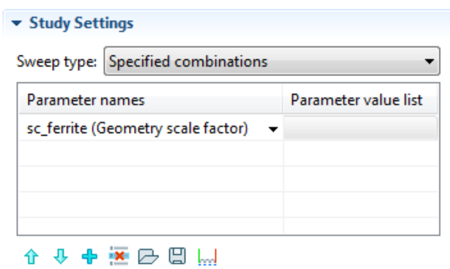
Parametric Sweep

The parametric sweep over the scale factor is added as an extension to the frequency domain study.

1 In the **Model Builder**, right-click **Study 1** and choose **Parametric Sweep**.

2 Go to the **Parametric Sweep** settings window. Under **Study Settings** click **Add** under the **Parameter names** table.

3 In the **Parameter names** list, select **sc_ferrite (geometry scale factor)**.



4 Under **Study Settings** click the **Range** button under the **Parameter names** table.

5 In the **Range** dialog box:

- In the **Start** field, enter `0.5`.
- In the **Step** field, enter `0.003`.
- In the **Stop** field, enter `0.53`.

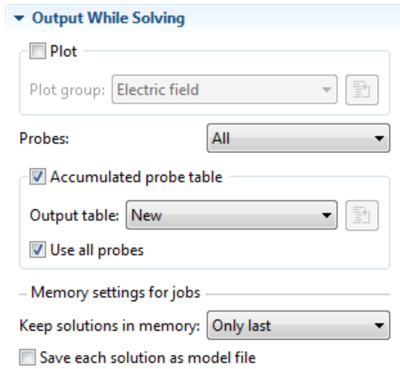
Note: Or enter `range(0.5,0.003,0.53)` in the **Parameter value list**.

6 Click **Replace**.

7 Under **Output While Solving**, select the **Accumulated probe table** check box under **Probes**.

8 Choose **Only last** from the **Keep solutions in memory** list.

9 In the **Model Builder**, right-click **Study 1** and choose **Compute** (or press F8).



RESULTS

Probe 1D Plot Group 2

The probe with the reflection coefficient versus the scale parameter is automatically logged to a table. A dedicated 1D plot group is also created but it plots the S-parameter versus frequency. To plot versus the geometry parameter, proceed as follows.

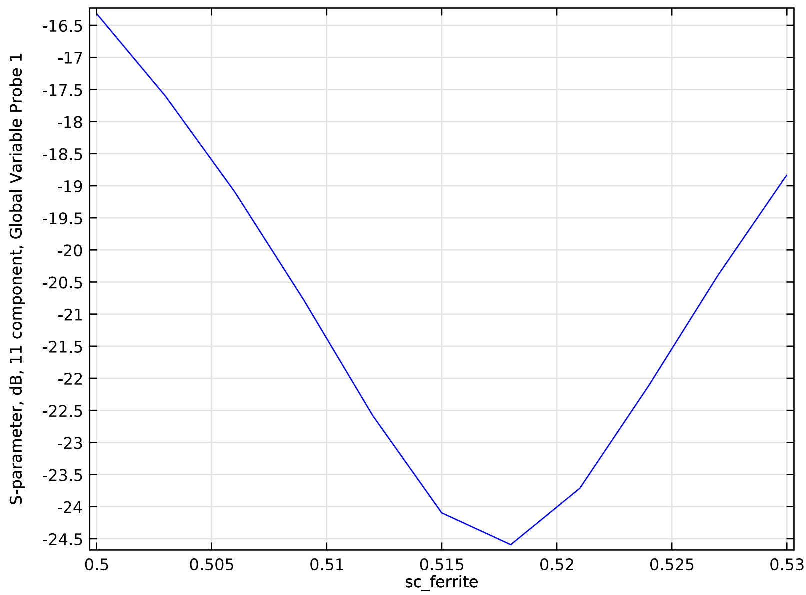
1 In the **Model Builder** under **Results**, expand the **Tables** node.

2 Select **Accumulated Probe Table 1**.

3 In the **Results** window, right-click the **freq** column and select **Delete Column**.

Note: The **Results** window table is by default located in the lower right side of the COMSOL Desktop. Or select **View>Results** from the main menu to open the window.

4 In the **Results** window, click the **Graph Plot**.

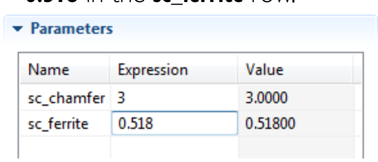


The plot of the S-parameter indicates a minimum for a scale factor of 0.518. Freeze the parameter at this value and add a new study to vary the next scale factor.

GLOBAL DEFINITIONS - PARAMETERS

1 In the **Model Builder** under **Global Definitions**, click **Parameters**.

2 In the **Parameters** settings window under **Parameters**, in the **Expression** column enter **0.518** in the **sc_ferrite** row.


▾ Parameters

Name	Expression	Value
sc_chamfer	3	3.0000
sc_ferrite	0.518	0.51800

STUDY 1

Parametric Sweep

1 In the **Model Builder** under **Study 1**, click **Parametric Sweep**.

2 Go to the **Parametric Sweep** settings window. Under **Study Settings** in the **Parameter names** list, select **sc_chamfer (Geometry scale factor)**.

3 Under **Study Settings** click the **Range** button under the **Parameter names** table.

4 In the **Range** dialog box:

- In the **Start** field, enter `2.8`.
- In the **Step** field, enter `0.04`.
- In the **Stop** field, enter `3.2`.

Note: Or enter `range(2.8,0.04,3.2)` in the **Parameter value list**.

5 Click **Replace**.

6 In the **Model Builder**, right-click **Study 1** and choose **Compute**.

RESULTS

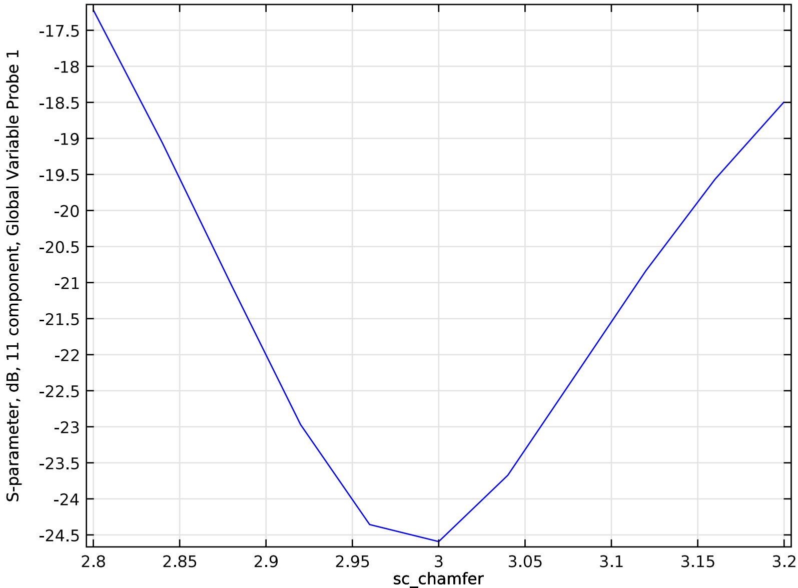
Probe1D Plot Group 2

Again, the probe with the reflection coefficient versus the frequency is automatically logged to a table. To get the desired plot versus the geometry parameter, proceed as follows.

1 In the **Model Builder**, go to **Results** and select **Accumulated Probe Table 1**.

2 In the **Results** window, right-click the **freq** column and select **Delete Column**.

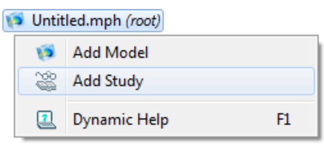
In the **Results** window, click the **Graph Plot**.



The plot of the S-parameter indicates a minimum for a scale factor of about `3.0`. Leave the parameter at this value and add a study for the frequency response.

MODEL WIZARD

1 Open the **Model Wizard** by right-clicking the root node and selecting **Add Study**.
2 Under **Preset Studies** select **Frequency Domain**.
3 Click **Finish**.

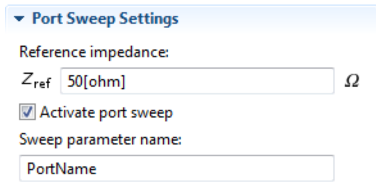


ELECTROMAGNETIC WAVES (EMW)

At this stage, it is convenient to partially invoke some higher-level control of what port is excited. This is done by letting the value of a parameter decide. The parameter can in turn be controlled by the solver when computing the full S-parameter matrix by exciting one port at a time. Another advantage is that when this control is activated, a plot group for the S-parameters is automatically created.

1 In the **Model Builder** click the **Electromagnetic Waves, Frequency Domain (emw)** node.

2 Go to the **Electromagnetic Waves, Frequency Domain** settings window. Under **Port Sweep Settings** select the **Activate port sweep** check box.



Note: The default variable name `PortName` is automatically added to the **Port parameter name** field but must be declared as a global parameter to be available for the parametric sweep.

GLOBAL DEFINITIONS - PARAMETERS

1 In the **Model Builder** under **Global Definitions**, click **Parameters** .

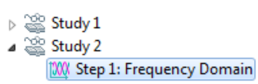
2 In the **Parameters** settings window under **Parameters**:

- In the **Name** column, enter `PortName`
- In the **Expression** column, enter `1`

STUDY 2

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

2 Go to the **Frequency Domain** settings window. Under **Study Settings** click the **Range** button.

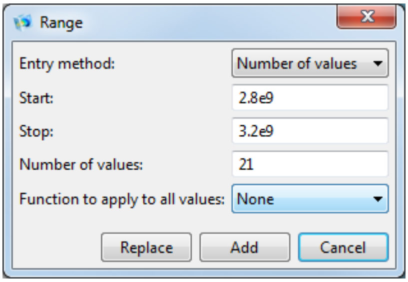


3 From the **Entry method** list, select **Number of values**.

- In the **Start** field, enter `2.8e9`.
- In the **Stop** field, enter `3.2e9`.
- In the **Number of values** field, enter `21`.

4 Click **Replace**.

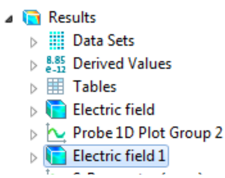
5 In the **Model Builder**, right-click **Study 2** and choose **Compute** (or press F8).



RESULTS

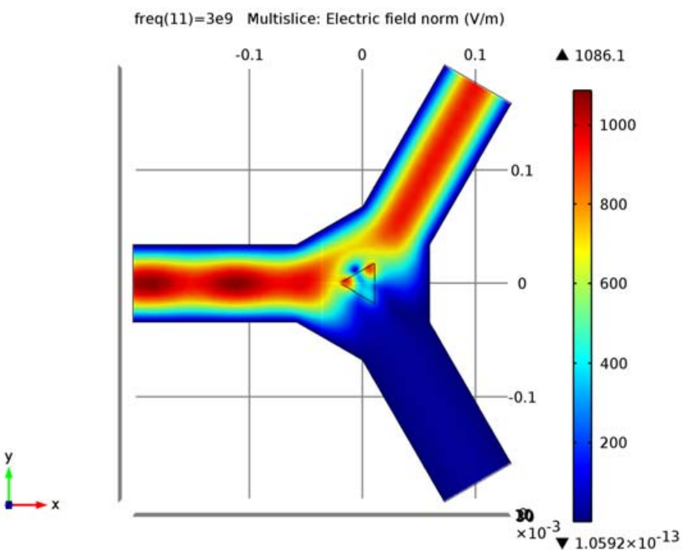
The probe with the reflection coefficient versus the frequency is automatically logged to a table and plotted while solving. At the last frequency, there are pronounced standing waves. Look at the center frequency instead.

1 In the **Model Builder** under **Results**, click **Electric Field 1**.



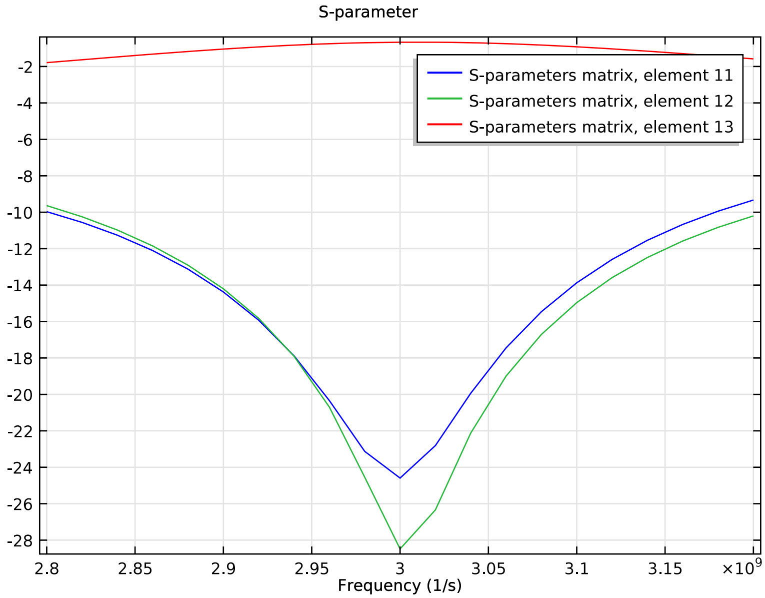
2 Go to the **3D Plot Group** settings window. Under **Data** from the **Parameter value (freq)** list, select **3e9**.

3 Click the **Plot** button. Click the **Go to XY View** button.



At the center frequency most of the standing waves are gone.

Finally look at all the S-parameters plotted versus the frequency. As the sweep was activated a plot group has been created automatically.



This is the frequency response of the final design.

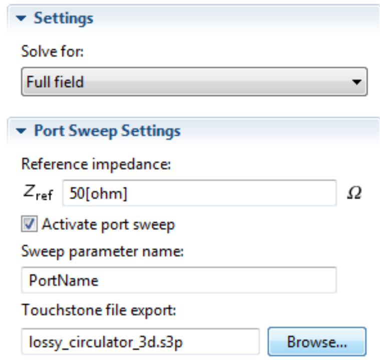
ELECTROMAGNETIC WAVES (EMW)

So far, only the first port has been excited so the full S-parameter matrix remains to be calculated by exciting one port at a time. This is also needed in order to confirm that the circulator behaves as desired. In the RF Module, this procedure is referred to as performing a port sweep. During this stage, the S-parameters can optionally be exported to a Touchstone file for documentation purposes and for use in external system simulation tools. Add the name of the Touchstone file.

- In the **Model Builder** click the **Electromagnetic Waves, Frequency Domain (emw)** node.

2 Go to the **Electromagnetic Waves, Frequency Domain** settings window. Under **Port Sweep Settings** in the **Touchstone file export** field, enter: `lossy_circulator_3d.s3p`.

Note: The Touchstone file is saved in the start directory of COMSOL Multiphysics unless a different file path is specified.



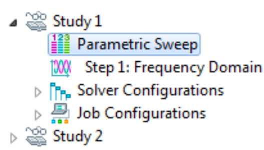
STUDY 1

Reuse the first study for the port sweep. The study is solved for a single frequency to keep simulation time to a minimum although it is possible to solve for a range of frequencies. All solutions are needed in order to display the S-parameter matrix in a table so this setting has to be changed.

Parametric Sweep

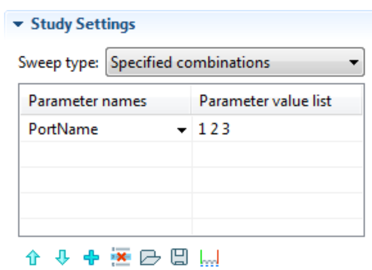
The parametric sweep is used to control which port is excited. It overrides the settings on individual port features and drives one port at a time using 1 W of input power.

1 In the **Model Builder** under **Study 1**, click **Parametric Sweep**.



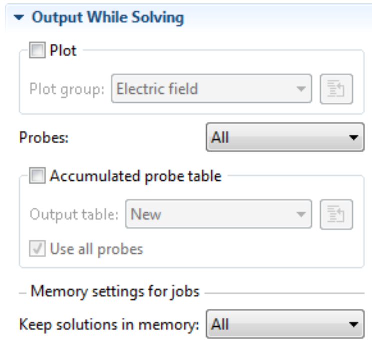
2 Go to the **Parametric Sweep** settings window. Under **Study Settings** in the **Parameter names** list, select **PortName**.

3 In the **Parameter value list**, enter (space separated) `1 2 3`.



4 Under **Output While Solving**, choose **All** from the **Keep solutions in memory** list.

5 In the **Model Builder**, right-click **Study 1** and choose **Compute**.



RESULTS

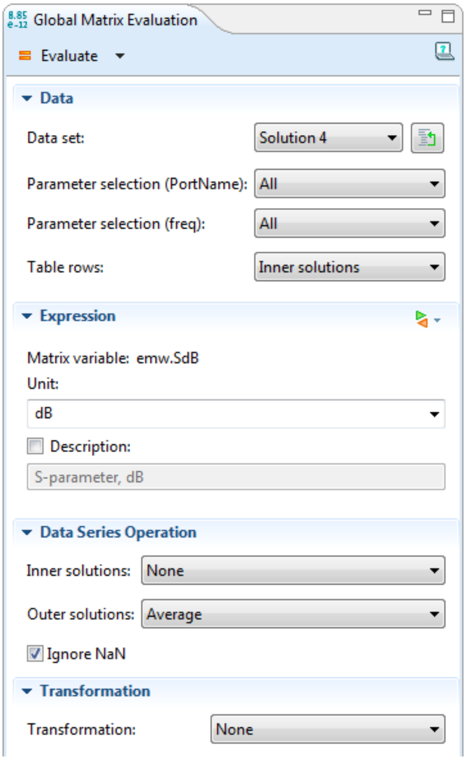
After the computation finishes, the Touchstone file can be inspected in a text editor. The S-parameter matrix can also be displayed in a table.

1 In the **Model Builder** under **Results**, right-click **Derived Values** and choose **Global Matrix Evaluation**.

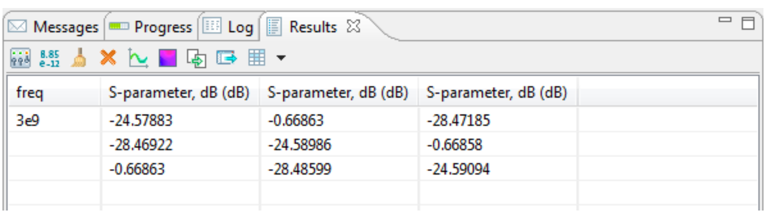
2 Go to the **Global Matrix Evaluation** settings window. Under **Data** from the **Data set** list, choose **Solution 4**.

3 In the upper-right corner of the **Expression** section, click **Replace Expression**.

4 From the menu, choose **Electromagnetic Waves, Frequency Domain>Ports>S-parameter, dB>S-parameter, dB (emw.SdB)**.



5 Click the **Evaluate** button and the S-parameter matrix displays in the **Results** window table located in the lower right side of the COMSOL Desktop. Or select **View>Results** from the main menu to open the **Results** window.



freq	S-parameter, dB (dB)	S-parameter, dB (dB)	S-parameter, dB (dB)
3e9	-24.57883	-0.66863	-28.47185
	-28.46922	-24.58986	-0.66858
	-0.66863	-28.48599	-24.59094

Note: The matrix is non-symmetric, which is typical for a device based on a gyrotropic material. There are three groups of matrix elements that differ in the fourth digit. These should in theory be equal within each group so this gives some indication of the discretization errors.

DEFINITIONS

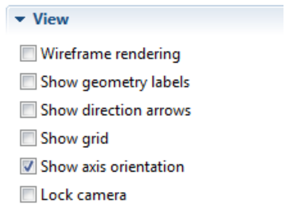
As a final step, create a plot and also use it as a model thumbnail. From this plot, it should be possible to identify the model at first glance so it has to display the geometry and some characteristic simulation results. First change to the default 3D view and switch off the grid.

1 Click the **Go to Default 3D View** button on the **Graphics** toolbar.

2 In the **Model Builder** right-click **Definitions** and choose **View**.

3 Go to the **View** settings window.

4 Under **View** click to clear the **Show grid** check box.

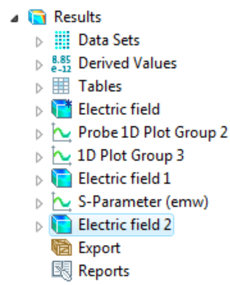


RESULTS

1 In the **Model Builder** under **Results**, click **Electric Field 2**.

2 Go to the **3D Plot Group** settings window. Under **Plot Settings** from the **View** list, choose **View 3**.

3 Click the **Plot** button.



Next, delete the multislice and create a single slice.

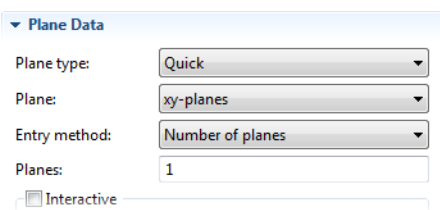
1 In the **Model Builder** under **Electric Field 2**, right-click **Multislice 1** and choose **Delete** (or press Delete on the keyboard).

2 Click **Yes** to confirm.

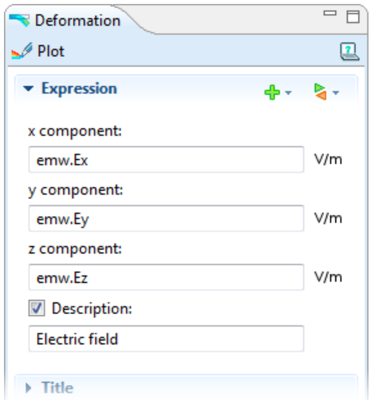
3 Right-click **Electric Field 2** and choose **Slice**.

4 In the **Slice** settings window, under **Plane data** from the **Plane** list, choose **xy-planes**.

5 In the **Planes** field, enter 1 .

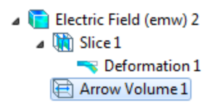


Add a deformation proportional to the electric field to the slice.

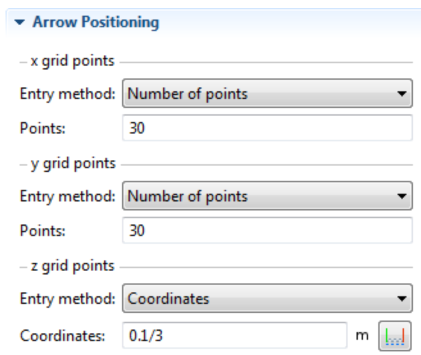


1. Right-click **Slice 1** and choose **Deformation**.
2. Go to the **Deformation** settings window. In the upper-right corner of the **Expression** section, click **Replace Expression**.
3. From the menu, choose **Electromagnetic Waves, Frequency Domain>Electric>Electric field (emw.Ex,emw.Ey,emw.Ez)**.
4. Under **Expression** select the **Description** check box.

Display the magnetic field as arrows. Use logarithmic length scaling to make sure that the arrows are clearly visible everywhere. Place the arrows well above the slice.

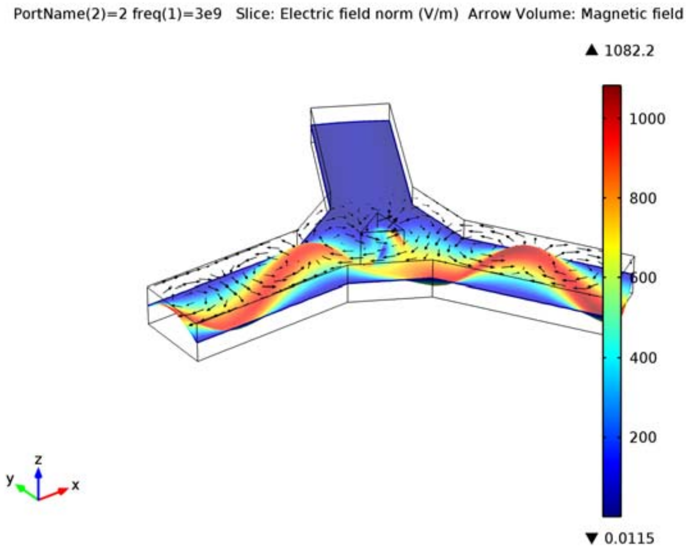


1. In the **Model Builder**, right-click **Electric Field 2** and choose **Arrow Volume**.
2. Go to the **Arrow Volume** settings window. In the upper-right corner of the **Expression** section, click **Replace Expression**.
3. From the menu, choose **Electromagnetic Waves, Frequency Domain>Magnetic>Magnetic field (emw.Hx,emw.Hy,emw.Hz)**.
4. Under **Expression** select the **Description** check box.
5. Under **Arrow Positioning**:
   - In the **Points** field for **x grid points**, enter 30.
   - In the **Points** field for **y grid points**, enter 30
   - For **z grid points** from the **Entry method** list, select **Coordinates**.
   - For **z grid points** in the **Coordinates** field, enter `0.1/3`.
6. Under **Coloring and Style** from the **Arrow length** list, choose **Logarithmic**. From the **Color** list, choose **Black**.



The port excitation can now be selected for the plot group. For the model thumbnail, select the second port.

1 In the **Model Builder**, click **Electric Field 2**.
2 In the **3D Plot Group** settings window under **Data**, choose **2** from the **Parameter value (PortName)** list.
3 Click the **Plot** button.



Select this plot to use as a model thumbnail.

1 In the **Model Builder** under **Results** click **Electric Field 2**.
2 From the **File** menu, choose **Save Model Thumbnail**.

To view the thumbnail image, click the **Root** node and look under the **Model Thumbnail** section.

This concludes the modeling session unless you want to practice drawing a "Geometry Sequence" on page 40.

GEOMETRY SEQUENCE

In "Global Definitions - Parameters and Variables" on page 19 parameters were entered to prepare for drawing the circulator geometry. Once the geometry is created, you can then experiment with different dimensions and update the values of `sc_chamfer` and `sc_ferrite` and rerun the geometry sequence. These step-by-step instructions build the same geometry that is contained in the Model Library file **lossy_circulator_3d_geom** (imported in the section "Geometry" on page 21).

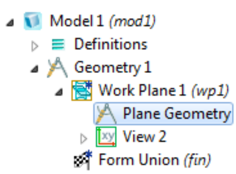
The geometry is built by first defining a 2D cross section of the 3D geometry in a work plane. The 2D geometry is then extruded into 3D.

Note: You need to complete the first two sections "Model Wizard" on page 19 and "Global Definitions - Parameters and Variables" on page 19 before defining the geometry.

Start by defining one arm of the circulator, then copy and rotate it twice to build all three arms.

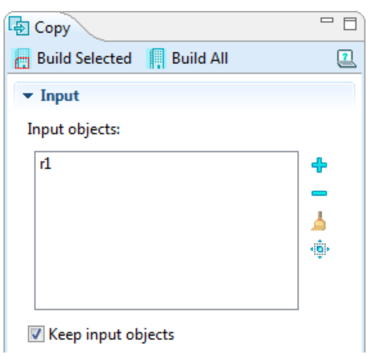
Rectangle 1

1 In the **Model Builder**, right-click **Geometry 1** and choose **Work Plane**.
2 Under **Work Plane 1 (wp1)**, click **Plane Geometry**.
3 Right-click **Plane Geometry** and choose **Rectangle**.
4 Go to the **Rectangle** settings window. Under **Size** in the:
   - **Width** field, enter `0.2-0.1/(3*sqrt(3))`.
   - **Height** field, enter `0.2/3`.
5 Under **Position** in the:
   - **xw** field, enter `-0.2`.
   - **yw** field, enter `-0.1/3`.
6 Click the **Build Selected** button.

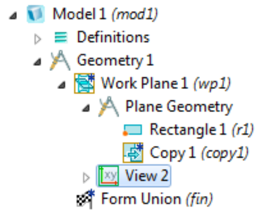


Copy 1

1 In the **Model Builder** under **Work Plane 1 (wp1)**, right-click **Plane Geometry** and choose **Transforms>Copy**.
2 Select the object **r1** by left-clicking (the rectangle is highlighted red) and then right-click the rectangle (it is highlighted blue). The object r1 is added to the **Input objects** list on the **Copy** settings window.
3 Click the **Build Selected** button.

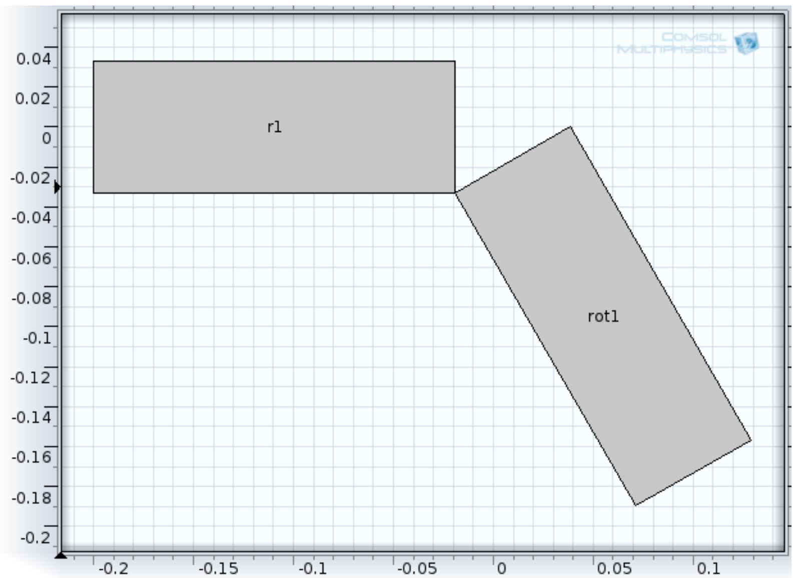


Note: To turn on the geometry labels in the **Graphics** window, in the **Model Builder** under **Geometry 1>WorkPlane 1>Plane Geometry**, click the **View 2** node. Go to the **View** settings window and select the **Show geometry labels** check box.



Rotate 1

1. In the **Model Builder** under **Work Plane 1 (wp1)**, right-click **Plane Geometry** and choose **Transforms>Rotate**.
2. Select only the object **copy1** (left-click twice before right-clicking or you get **r1**).
3. Go to the **Rotate** settings window. Under **Rotation Angle** in the **Rotation** field, enter 120.
4. Click the **Build Selected** button and then click the **Zoom Extents** button on the **Graphics** toolbar. The geometry should match the figure so far.



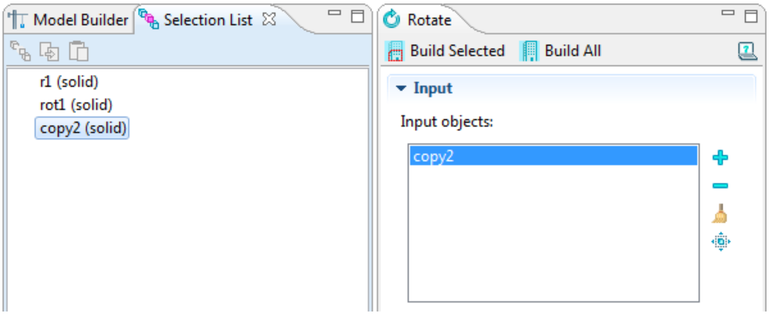
Copy 2

1. Right-click **Plane Geometry** and choose **Transforms>Copy**.
2. Select the object **r1** only and add it to the **Input objects** list in the **Copy** settings window.
3. Click the **Build Selected** button.

Rotate 2

1. Right-click **Plane Geometry** and choose **Transforms>Rotate**.
2. Select the object **copy2** only.

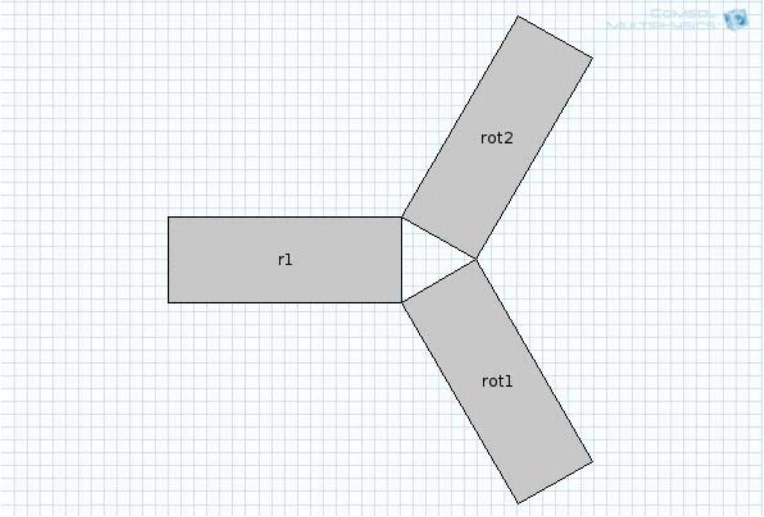
Note: If you cannot locate **copy2** in the **Graphics** window, select **View>Selection List** and click to highlight **copy2** in the list. Then right-click **copy2** in the list to add it to the **Input objects** list.



3 Go to the **Rotate** settings window. Under **Rotation Angle** in the **Rotation** field, enter -120.

4 Click the **Build Selected** button and then click the **Zoom Extents** button on the **Graphics** toolbar.

The geometry should match this figure.



Next, unite the three arms to one object.

Union 1

1 Under **Work Plane 1 (wp1)**, right-click **Plane Geometry** and choose **Boolean Operations>Union**.

2 Select the objects **r1**, **rot1**, and **rot2** only and add these to the **Input objects** list in the **Union** settings window.

3 Click the **Build Selected** button. There is one object created called **uni1**.

Now, build the central connecting region and add the ferrite domain. During these stages, the geometric design parameters are used. Start by creating a triangle connecting the arms and then by subtracting a copy of what has already been drawn from a circle of proper radius.

Circle 1

1 Under **Work Plane 1 (wp1)**, right-click **Plane Geometry** and choose **Circle**.

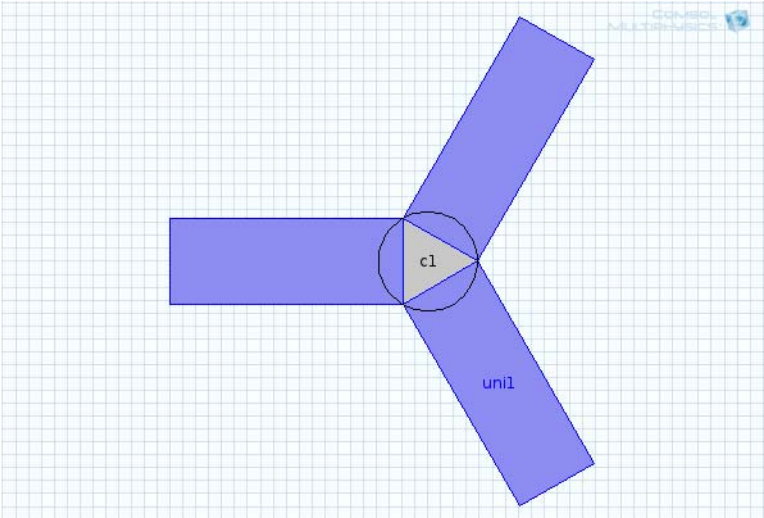
2 In the **Circle** settings window under **Size and Shape**, enter `0.2/(3*sqrt(3))` in the **Radius** field.

3 Click the **Build Selected** button.

Copy 3

1 Under **Work Plane 1 (wp1)**, right-click **Plane Geometry** and choose **Transforms>Copy**. A **Copy 3** node is added to the sequence.

2 Select the object **uni1** only.



Difference 1

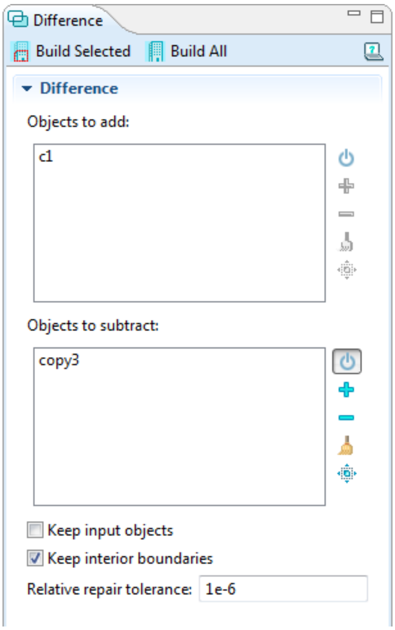
1 Right-click **Plane Geometry** and choose **Boolean Operations>Difference**.

2 Select the object **c1** only to add it to the **Objects to add** list in the **Difference** settings window.

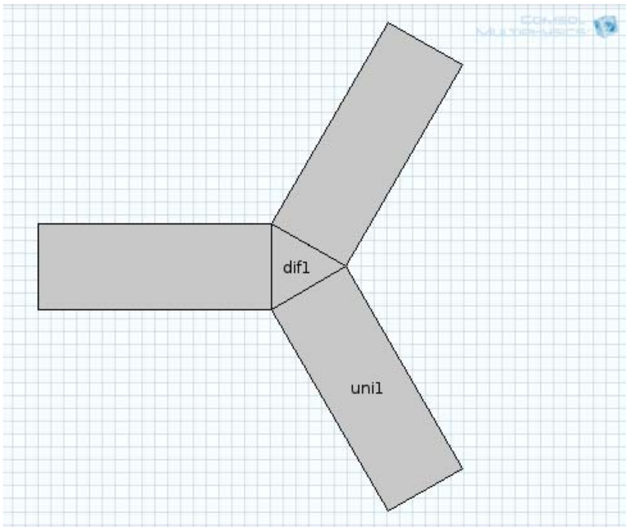
3 Go to the **Difference** settings window. To the right of the **Objects to subtract** section, click the **Activate Selection** button.

4 Select **View>Selection List** and click to highlight **copy3** in the list. Then right-click **copy3** in the list to add it to the **Objects to subtract** list.

5 Click the **Build Selected** button.



The geometry should match this figure so far.



Now, rotate the newly created triangle 180 degrees and use one scaled copy of it to create linear fillets for impedance matching. Use another scaled copy to define the ferrite.

Rotate 3

1 In the **Model Builder** under **Work Plane 1 (wp1)**, right-click **Plane Geometry** and choose **Transforms>Rotate**.
2 Select the object **dif1** only.
3 Go to the **Rotate** settings window. Under **Rotation Angle** in the **Rotation** field, enter 180.
4 Click the **Build Selected** button.

Copy 4

1 Right-click **Plane Geometry** and choose **Transforms>Copy**.
2 Select the object **rot3** only.

It is now time to apply the first scaling for the impedance matching.

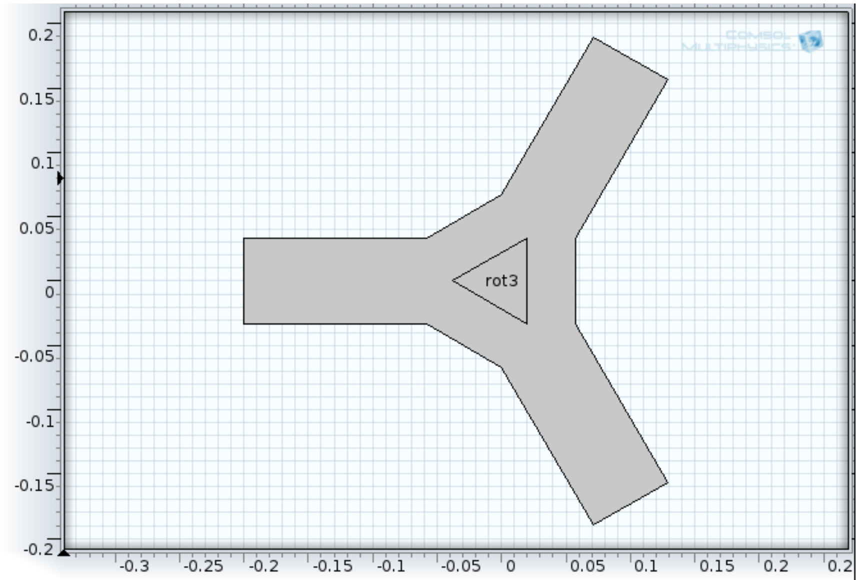
Scale 1

1 Right-click **Plane Geometry** and choose **Transforms>Scale**.
2 Go to the **Scale** settings window. Under **Scale Factor** in the **Factor** field, enter `sc_chamfer` (one of the parameters entered in the step "Global Definitions - Parameters and Variables" on page 19).
3 Select the object **copy4** only.
4 Click the **Build Selected** button.

Union 2

1 Right-click **Plane Geometry** and choose **Boolean Operations>Union**.
2 Select the objects **uni1** and **sca1** only. Use the **Selection List** to select the objects if required.
3 Go to the **Union** settings window. Under **Union** click to clear the **Keep interior boundaries** check box.
4 Click the **Build Selected** button.

The geometry should match this figure.



Next, apply the scaling for the ferrite region.

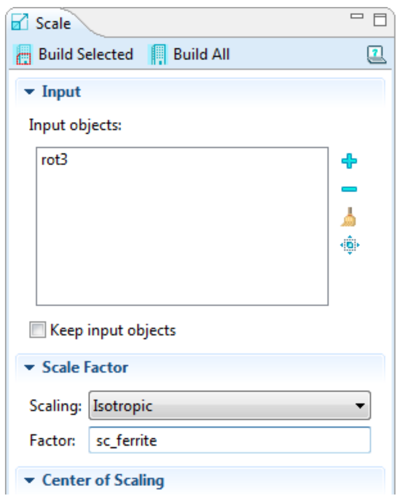
Scale 2

1 In the **Model Builder** right-click **Plane Geometry** and choose **Transforms>Scale**.

2 Select the object **rot3** only.

3 Go to the **Scale** settings window. Under **Scale Factor** in the **Factor** field, enter `sc_ferrite`.

4 Click the **Build Selected** button.



Extruding the 2D cross-section into a 3D solid geometry finalizes the geometry.

Extrude 1

1 Right-click **Work Plane 1 (wp1)** and choose **Extrude**.

2 Go to the **Extrude** settings window. Under **Distances from Plane** in the associated table, enter `0.1/3` in the **Distances (m)** column.

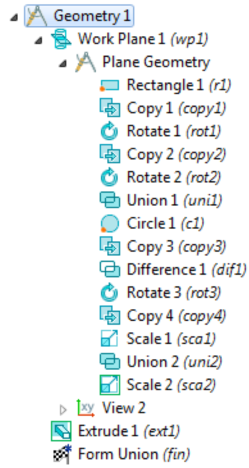
3 Click the **Build Selected** button and then click the **Zoom Extents** button on the **Graphics** toolbar.

Form Union

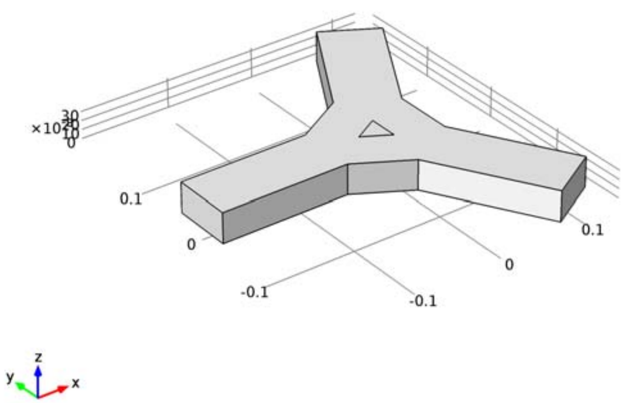
1 In the **Model Builder**, click **Form Union (fin)**.

2 In the **Form Union** settings window, click **Build All**.

The final sequence of **Geometry** nodes in the **Model Builder** should match the figure.



The last step finalizes the geometry and turns it into a form suitable for the simulation by removing duplicate faces, for example. It is performed automatically when material is added or when physics are defined, but it is good practice to perform it manually as any error messages from this step may be confusing when appearing at a later stage. The geometry should match this figure.



Note: If you skipped to this section to learn how to create the geometry, you can now return to the next tutorial step: "Materials" on page 21.