Created in COMSOL Multiphysics 6.2

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 yet 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 a circulator is used to isolate microwave components from each other, for example, 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.

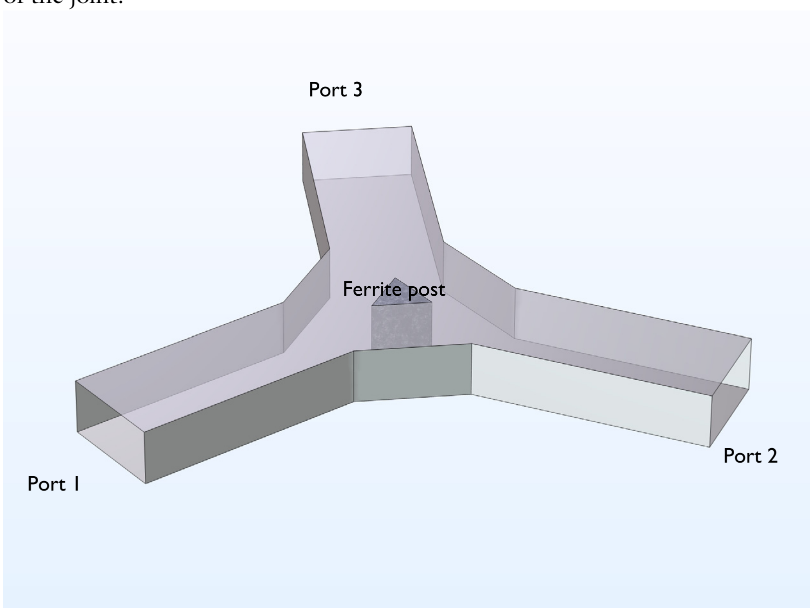


Figure 1: The post is magnetized by a static H_0 bias field along its axis. The bias field is supplied by external permanent magnets which are not explicitly modeled in this example.

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 back to 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 chosen as 3 GHz. The circulator can be expected to perform reasonably well in a narrow frequency band around 3 GHz, and so a frequency range of 2.8–3.2 GHz is studied. It is desired that the device operates in single mode. Thus a rectangular waveguide cross section of 6.67 cm by 3.33 cm is selected to set the cutoff frequency for the fundamental TE_{10} mode to 2.25 GHz. The cutoff 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 is the angular frequency, σ is the conductivity tensor, ε_0 is the permittivity of vacuum, ε_r is the relative permittivity tensor, and k_0 is the free space wave number. In this particular model, 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. The expressions are also included in the next section for reference.

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:

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

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

where

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

$$\omega_{\mathrm{m}} = \mu_0\gamma M_s$$

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

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, 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 ($\alpha = 0$), the permeability becomes infinite at $\omega = \omega_0$. In a lossy ferrite ($\alpha \neq 0$), this resonance becomes finite and is broadened. The loss factor, α, is related to the line width, ΔH, of the susceptibility curve near the resonance as given by the last expression above. The material data,

$$M_{\mathrm{s}} = 5.41\cdot 10^4 \text{ A/m},\ \varepsilon_{\mathrm{r}} = 14.5$$

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 $H_0 = 7.96{\cdot}103$ A/m. The electron gyromagnetic ratio taken from Ref. 2 is $1.759{\cdot}10^{11}$ C/kg.

Results and Discussion

The default multislice plot shows the electric field norm. 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.

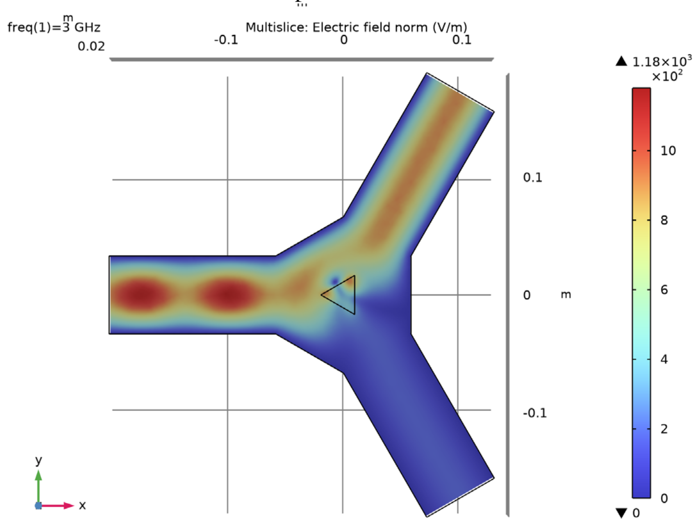


Figure 2: The default electric field norm plot shown on xy-plane.

The plot of the S-parameter from the parametric sweep of `sc_ferrite` indicates a minimum for a scale factor of 0.518.

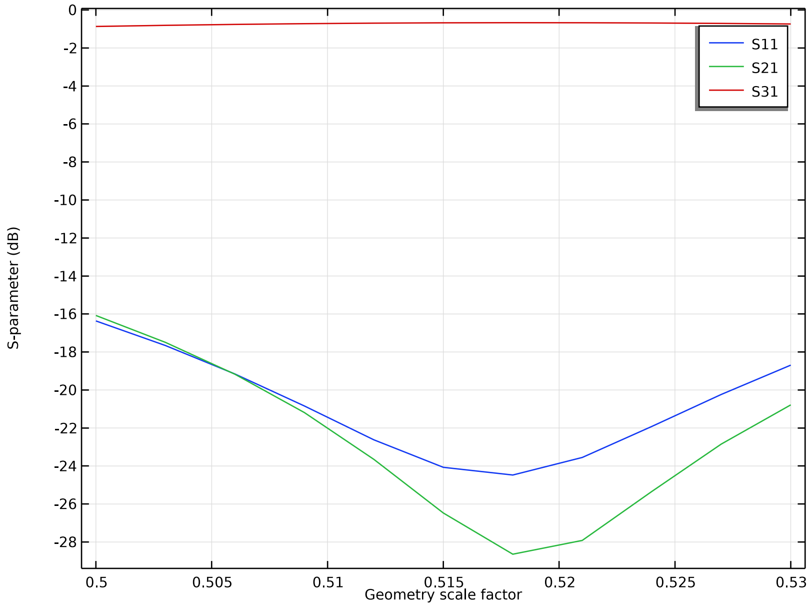


Figure 3: S-parameter as a function of sc_ferrite parameter.

The plot of the S-parameter from the parametric sweep of `sc_chamfer` indicates a minimum for a scale factor of about 3.0.

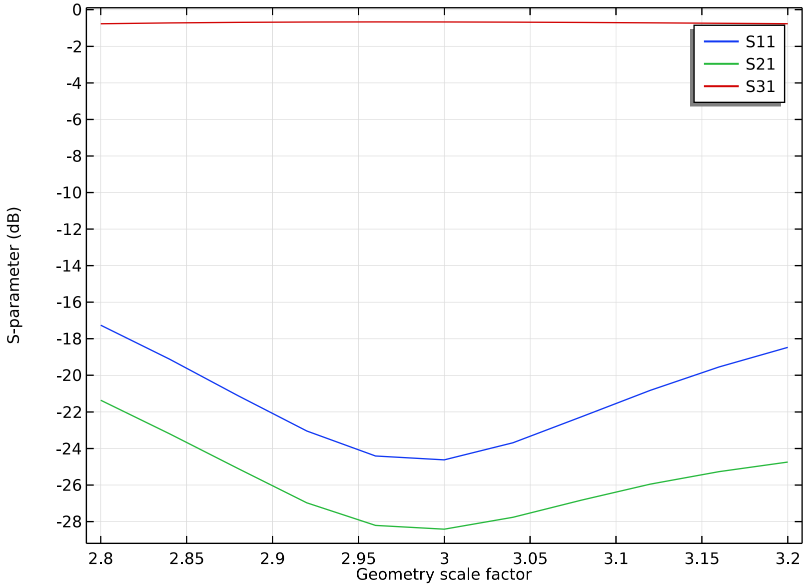


Figure 4: S-parameter as a function of sc_chamfer parameter.

At the center frequency most of the standing waves are gone with the optimized values of sc_ferrite and sc_chamfer.

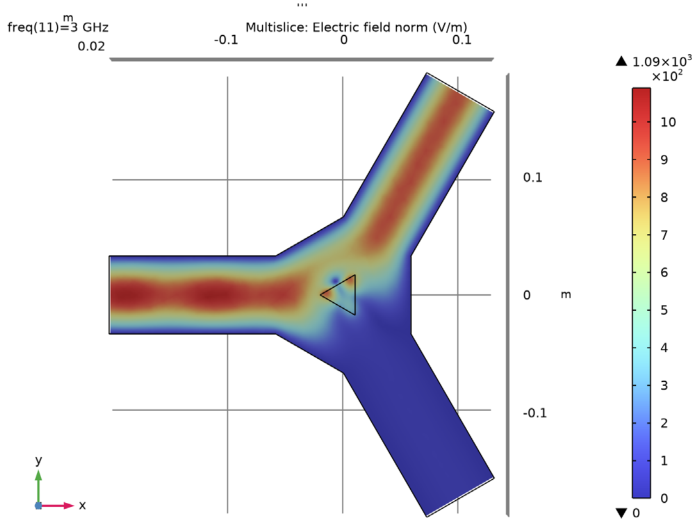


Figure 5: Electric field norm plot with the optimized sc_ferrite and sc_chamfer values.

This is the frequency response of the final design.

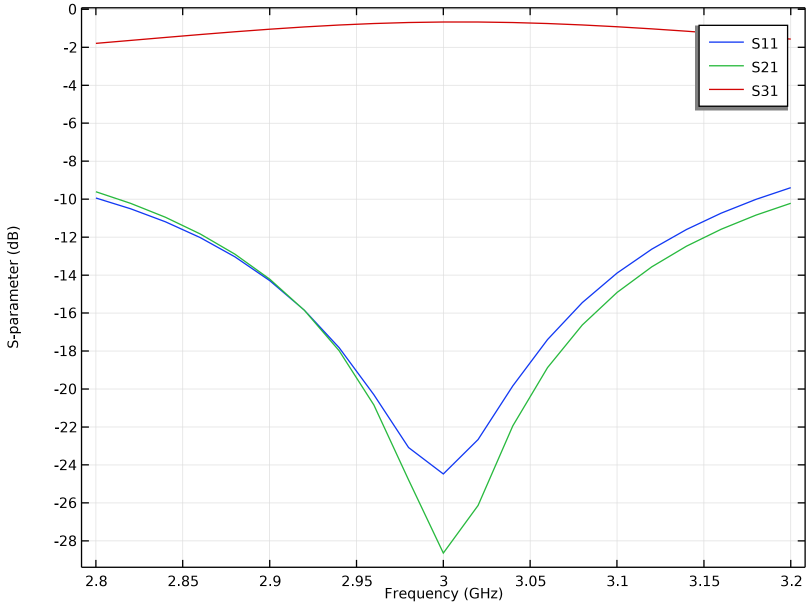


Figure 6: S-parameter as a function of frequency with the optimized sc_ferrite and sc_chamfer values.

From the plot below, it should be possible to identify the model at first glance so it has to display the geometry and some characteristic simulation results.

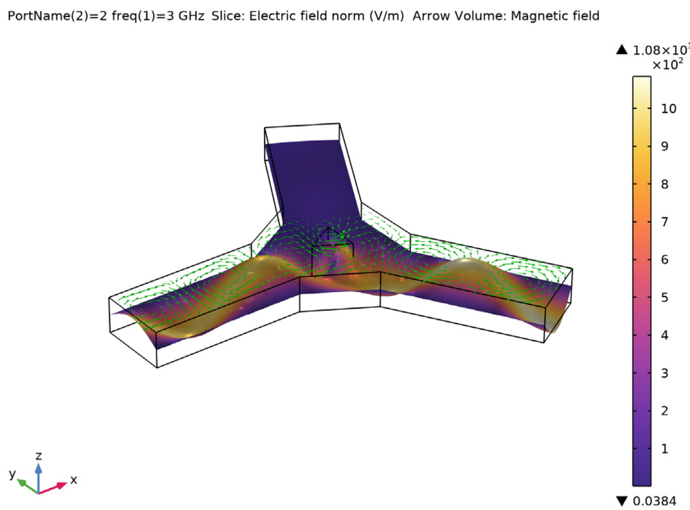


Figure 7: 3D plot used for model thumbnail generation.

Reference

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.

Application Library path: `RF_Module/Ferrimagnetic_Devices/lossy_circulator_3d`

Modeling Instructions

APPLICATION LIBRARIES

1 From the **File** menu, choose **Application Libraries**.

2 In the **Application Libraries** window, select **RF Module>Ferrimagnetic Devices> lossy_circulator_3d_geom** in the tree.

3 Click Open.

GEOMETRY 1

Form Union (fin)

Next add material settings to the model. The lossy ferrite does not fit easily into the material settings so it will be taken care of later. Air is the only material to enter here.

ADD MATERIAL

1 In the **Home** toolbar, click **Add Material** to open the **Add Material** window.

2 Go to the **Add Material** window.

3 In the tree, select **Built-in>Air**.

4 Click **Add to Component** in the window toolbar.

5 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Air (mat1)

In the Electromagnetic Waves interface, the ferrite is entered as a separate, user-defined equation model referring to the global variables defined above.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

Wave Equation, Electric 2

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain (emw)** node.

2 Right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Wave Equation, Electric**.

3 Select Domain 2 only.

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

5 From the **Electric displacement field model** list, choose **Dielectric loss**.

6 From the ε' list, choose **User defined**. In the associated text field, type `eps_r_p`.

7 From the ε'' list, choose **User defined**. In the associated text field, type `eps_r_b`.

8 Locate the **Magnetic Field** section. From the μ_r list, choose **User defined**. From the list, choose **Full**.

9 In the μ_r table, enter the following settings:

murxx	murxy	murxz
muryx	muryy	muryz
murzx	murzy	murzz

10 Locate the **Conduction Current** section. From the σ list, choose **User defined**.

One inport for excitation and two outports need to be added next.

Port 1

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

2 Select Boundary 1 only.

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

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

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

Port 2

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

2 Select Boundary 18 only.

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

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

Port 3

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

2 Select Boundary 19 only.

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

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

The mesh needs to resolve the local wavelength and, for lossy domains, the skin depth. The skin depth in the ferrite is large 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 smaller.

MESH 1

Free Tetrahedral 1

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

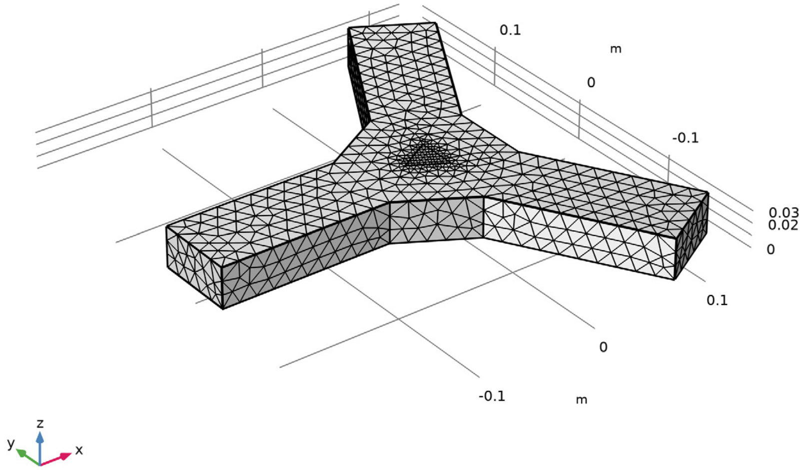
Size 1

1. Right-click **Free Tetrahedral 1** and choose **Size**.
2. In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
3. From the **Geometric entity level** list, choose **Domain**.
4. Select Domain 1 only.
5. Locate the **Element Size** section. Click the **Custom** button.
6. Locate the **Element Size Parameters** section.
7. Select the **Maximum element size** check box. In the associated text field, type `1.5e-2`.

Size 2

1. In the **Model Builder** window, right-click **Free Tetrahedral 1** and choose **Size**.
2. In the **Settings** window for **Size**, locate the **Element Size** section.
3. Click the **Custom** button.
4. Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Domain**.
5. Select Domain 2 only.
6. Locate the **Element Size Parameters** section.
7. Select the **Maximum element size** check box. In the associated text field, type `4.5e-3`.

8 In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.



The mesh should now look as in the above figure.

The final step in the model set up is to solve it for the nominal frequency and inspect the results for possible modeling errors.

STUDY 1

Step 1: Frequency Domain

1 In the **Model Builder** window, expand the **Study 1** node, then click **Step 1: Frequency Domain**.
2 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.
3 In the **Frequencies** text field, type `3[GHz]`.
4 In the **Home** toolbar, click **Compute**.

RESULTS

Electric Field (emw)

The default plot shows a slice plot of the electric field norm. It is best viewed from above.

1 Click the **Go to XY View** button in the **Graphics** toolbar.

The electric field norm gives a good indication on where the main power is flowing and where there are standing waves due to reflections from the impedance mismatch at the center. See Figure 2.

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).

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.

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

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
sc_ferrite (Geometry scale factor)	`range(0.5,3e-3,0.53)`	

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

RESULTS

S-parameter (emw)

Compare with the plot shown Figure 3. The plot of the S-parameter indicates a minimum for a scale factor of 0.518, so freeze the parameter at this value and add a new study for varying the next scale factor.

GLOBAL DEFINITIONS

Parameters 1

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

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	**Expression**	**Value**	**Description**
`sc_ferrite`	`0.518`	0.518	`Geometry scale factor`

RESULTS

Global I

1 In the **Model Builder** window, expand the **Results>S-parameter (emw)** node, then click **Global 1**.

2 In the **Settings** window for **Global**, locate the **x-Axis Data** section.

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

STUDY I

Parametric Sweep

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

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 In the table, enter the following settings:

Parameter name	**Parameter value list**	**Parameter unit**
sc_chamfer (Geometry scale factor)	`range(2.8,0.04,3.2)`	

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

See Figure 4. The plot of the S-parameter indicates a minimum for a scale factor of about 3.0, so leave the parameter at this value and add a study for the frequency response.

ADD STUDY

1 In the **Study** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies> Frequency Domain**.

4 Click **Add Study** in the window toolbar.

5 In the **Study** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Frequency Domain

1. In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.
2. In the **Frequencies** text field, type `range(2.8[GHz],20[MHz],3.2[GHz])`.
3. In the **Study** toolbar, click **Compute**.

RESULTS

Electric Field (emw) 2

At the final frequency, there are pronounced standing waves. Change to the center frequency.

1. In the **Settings** window for **3D Plot Group**, locate the **Data** section.
2. From the **Parameter value (freq (GHz))** list, choose **3**.
3. In the **Electric Field (emw) 2** toolbar, click **Plot**.

In the reproduced Figure 5 most of the standing waves are gone at the center frequency.

Finally plot all the S-parameters as a function of frequency.

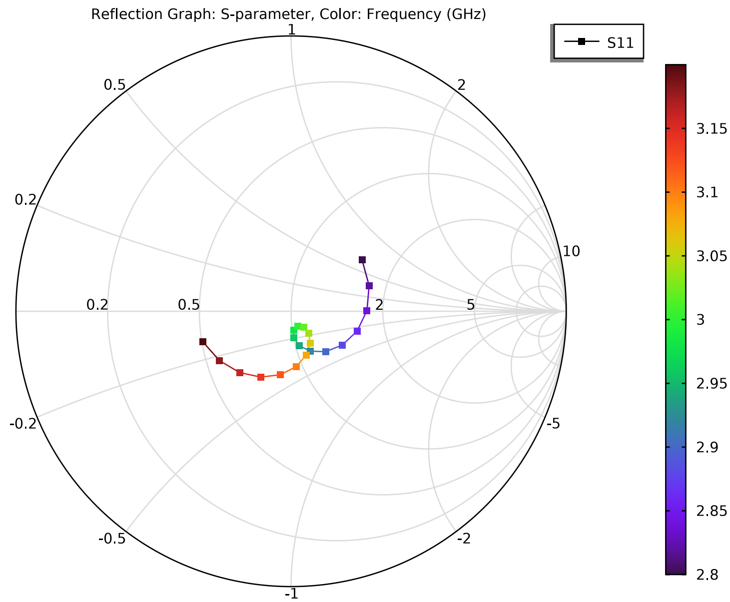
Global 1

Reproduce Figure 6. This is the frequency response of the final design.

Now, let the solver excite one port at a time in order to get the full S-parameter matrix exported to a Touchstone file for potential use in a system simulation tool. The necessary steps are as follows:

Smith Plot (emw) 1

In the **Model Builder** window, expand the **Results>S-parameter (emw) 1** node, then click **Results>Smith Plot (emw) 1**.



GLOBAL DEFINITIONS

Parameters 1

1. In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
2. In the **Settings** window for **Parameters**, locate the **Parameters** section.
3. In the table, enter the following settings:

Name	Expression	Value	Description
PortName	1	1	Port name

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Electromagnetic Waves, Frequency Domain (emw)**.
2. In the **Settings** window for **Electromagnetic Waves, Frequency Domain**, locate the **Port Sweep Settings** section.
3. Select the **Use manual port sweep** check box.

4 Select the **Export Touchstone file** check box.

5 In the **Touchstone file export** text field, type `lossy_circulator_3d.s3p`.

Add a new study for the port sweep. The study is solved for a single frequency to keep down simulation time though it is possible to solve for a range of frequencies.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies> Frequency Domain**.

4 Click **Add Study** in the window toolbar.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 3

Step 1: Frequency Domain

1 In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

2 In the **Frequencies** text field, type `3[GHz]`.

Parametric Sweep

1 In the **Study** toolbar, click **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.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
PortName (Port name)	1 2 3	

5 In the table, click to select the cell at row number 1 and column number 3.

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

Display the S-parameter matrix in a table.

RESULTS

Global Matrix Evaluation 1

1 In the **Results** toolbar, click **More Derived Values** and choose **Other> Global Matrix Evaluation**.

2 In the **Settings** window for **Global Matrix Evaluation**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 3/Parametric Solutions 2 (sol16)**.

4 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Electromagnetic Waves, Frequency Domain>Ports> S-parameter, dB>emw.SdB - S-parameter, dB - 1**.

5 Click **Evaluate**.

Electric Field (emw) 3

As a final step, create a nice plot to use as a thumbnail. First change to the default 3D view and switch off grid.

1 Click the **Go to Default View** button in the **Graphics** toolbar.

DEFINITIONS

View 3

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

2 In the **Settings** window for **View**, locate the **View** section.

3 Clear the **Show grid** check box.

RESULTS

Electric Field (emw) 3

1 In the **Model Builder** window, under **Results** click **Electric Field (emw) 3**.

2 In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.

3 From the **View** list, choose **View 3**.

4 In the **Electric Field (emw) 3** toolbar, click **Plot**.

Next, delete the Multislice plot and add a single slice.

Multislice

1 In the **Model Builder** window, expand the **Electric Field (emw) 3** node.

2 Right-click **Multislice** and choose **Delete**.

Slice 1

1 In the **Model Builder** window, right-click **Electric Field (emw) 3** and choose **Slice**.

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

2 In the **Settings** window for **Slice**, locate the **Plane Data** section.

3 From the **Plane** list, choose **XY-planes**.

4 In the **Planes** text field, type 1.

5 Locate the **Coloring and Style** section. Click **Change Color Table**.

6 In the **Color Table** dialog box, select **Thermal>HeatCameraLight** in the tree.

7 Click **OK**.

Deformation 1

1 Right-click **Slice 1** and choose **Deformation**.

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

3 Locate the **Expression** section. Select the **Description** check box.

Electric Field (emw) 3

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.

Arrow Volume 1

1 In the **Model Builder** window, right-click **Electric Field (emw) 3** and choose **Arrow Volume**.

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

3 Locate the **Expression** section. Select the **Description** check box.

4 Locate the **Arrow Positioning** section. Find the **X grid points** subsection. In the **Points** text field, type 45.

5 Find the **Y grid points** subsection. In the **Points** text field, type 45.

6 Find the **Z grid points** subsection. From the **Entry method** list, choose **Coordinates**.

7 In the **Coordinates** text field, type 0.1/3.

8 Locate the **Coloring and Style** section. From the **Arrow length** list, choose **Logarithmic**.

9 From the **Color** list, choose **Green**.

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

Electric Field (emw) 3

1 In the **Model Builder** window, click **Electric Field (emw) 3**.
2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.
3 From the **Parameter value (PortName)** list, choose **2**.
4 In the **Electric Field (emw) 3** toolbar, click **Plot**.

By plotting Figure 7, conclude this modeling session.

S-parameter (emw) 2

1 In the **Model Builder** window, click **S-parameter (emw) 2**.
2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.
3 From the **Parameter selection (PortName)** list, choose **From list**.
4 In the **Parameter values (PortName)** list, select **1**.

Global 1

1 In the **Model Builder** window, expand the **S-parameter (emw) 2** node, then click **Global 1**.
2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.
3 Ctrl-click to select table rows 2, 3, 5, 6, 8, and 9.
4 Click **Delete**.

The table should now only contain `emw.S11dB`, `emw.S21dB` and `emw.S31dB`.

5 Click to expand the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.
6 In the **S-parameter (emw) 2** toolbar, click **Plot**.