



Model created in COMSOL Multiphysics 6.4

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.

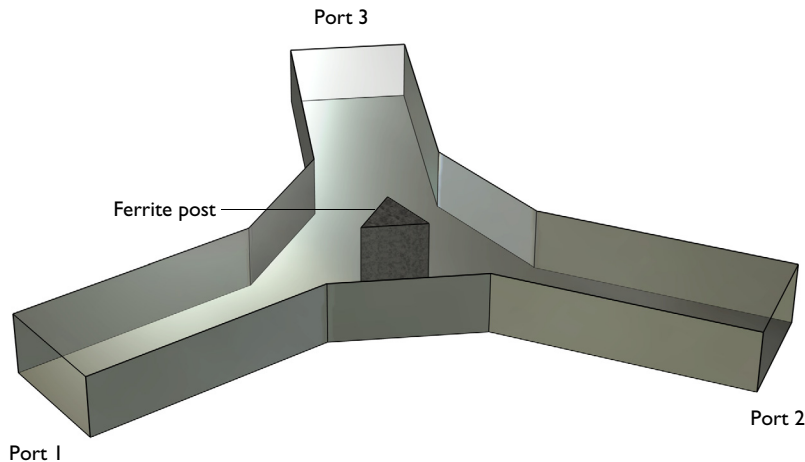


Figure 1: The post is magnetized by a static H_0 bias field along its axis. The bias field is supplied by external permanent magnets which are not explicitly modeled in this example.

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 input. 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 TE₁₀ mode to 2.25 GHz. The cutoff frequencies for the two nearest higher modes, the TE₂₀ and TE₀₁ modes, are both at 4.5 GHz, leaving a reasonable safety margin.

Model Definition

One of the rectangular ports is excited by the fundamental TE₁₀ mode. At the ports, the boundaries are transparent to the TE₁₀ mode. The following equation applies to the electric field vector \mathbf{E} inside the circulator:

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

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 defined using the loss tangent and the complex-valued permeability tensor. 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. The material expressions are discussed 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, \mathbf{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, 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 damping 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,

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

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.

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.

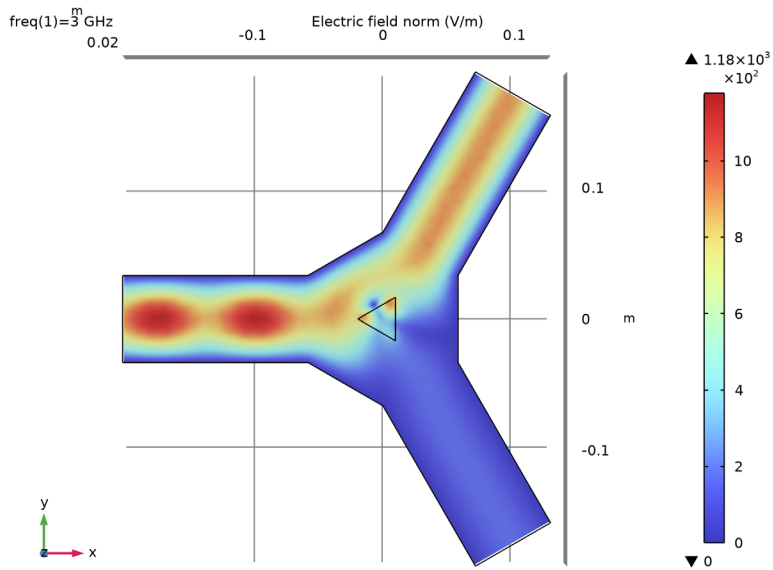


Figure 2: The default electric field norm plot shown on *xy*-plane.

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

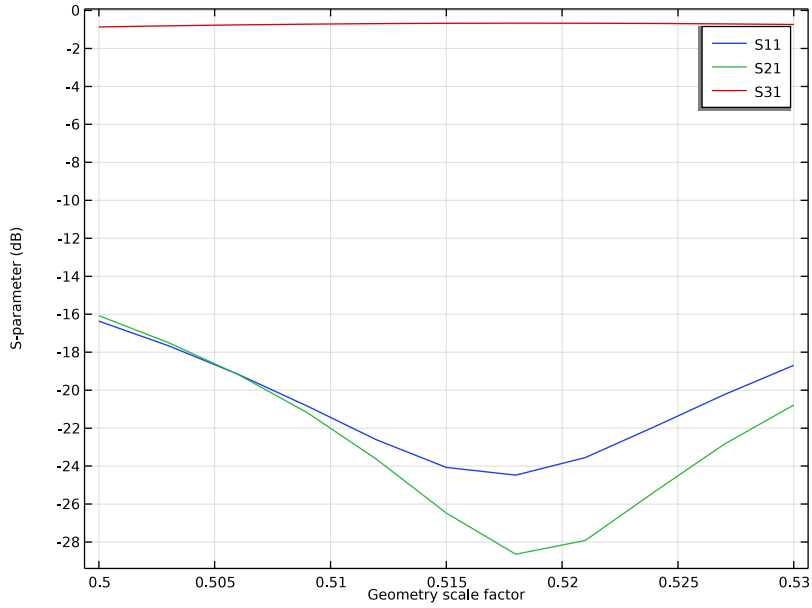


Figure 3: S-parameter as a function of `sc_ferrite` parameter.

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

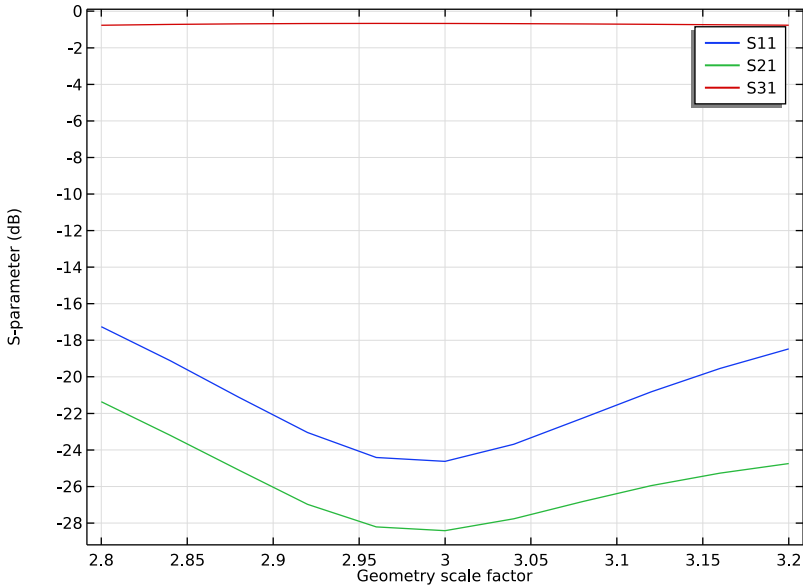


Figure 4: S-parameter as a function of `sc_chamfer` parameter.

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

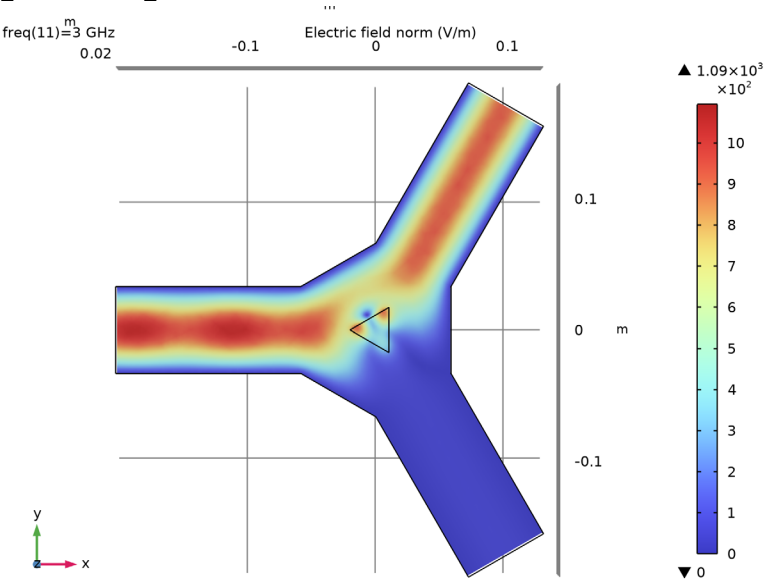


Figure 5: Electric field norm plot with the optimized `sc_ferrite` and `sc_chamfer` values.

This is the frequency response of the final design.

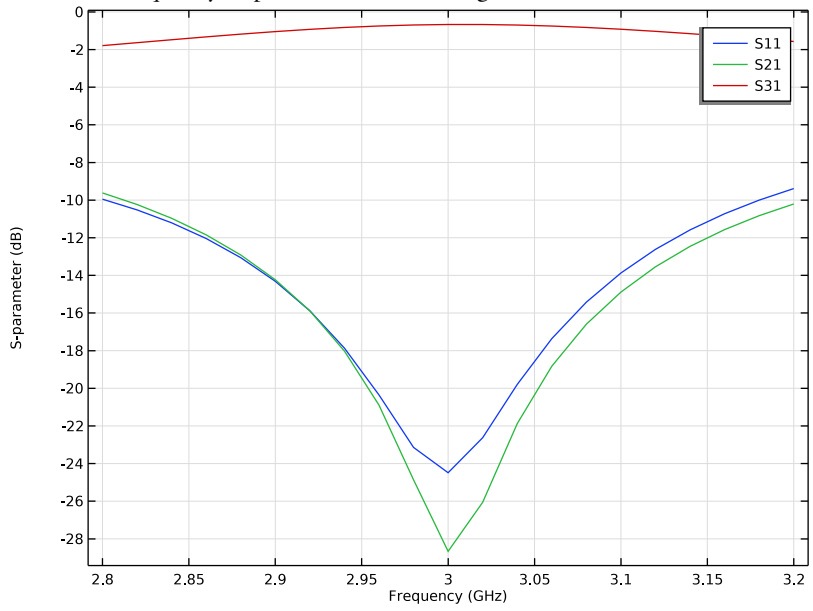


Figure 6: S-parameter as a function of frequency with the optimized $sc_ferrite$ and $sc_chamfer$ values.

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.

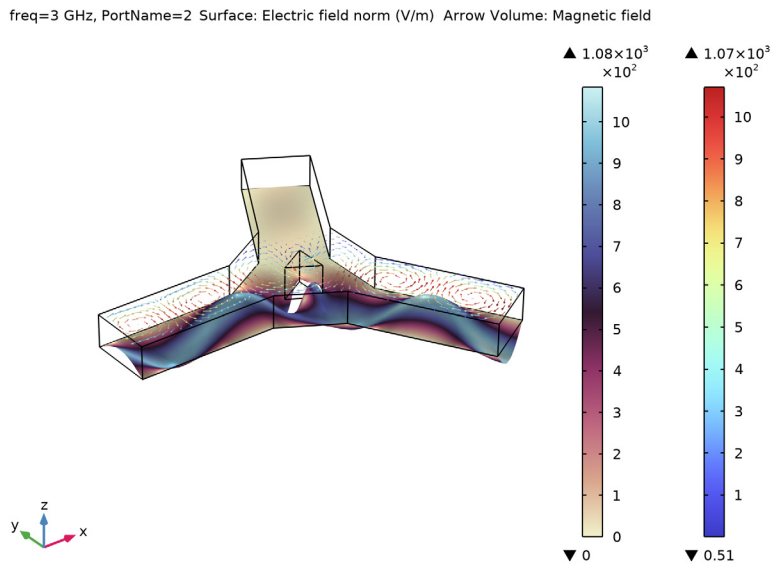


Figure 7: 3D plot used for model thumbnail generation.

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.

Application Library path: RF_Module/Ferrimagnetic_Devices/
lossy_circulator_3d

Modeling Instructions

From the **File** menu, choose **Open**.

Browse to the model's Application Libraries folder and double-click the file `lossy_circulator_3d_geom.mph`.

DEFINITIONS

Variables 1

- 1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variables**.
- 2 In the **Settings** window for **Variables**, locate the **Variables** section.
- 3 In the table, enter the following settings:



Name	Expression	Unit	Description
mur_xx	$1 + ((\omega_0 + j\omega) * \omega_m) / ((\omega_0 + j\omega)^2 - (2\pi \text{freq})^2)$		Complex relative magnetic permeability, xx-component
mur_xy	$(j * 2\pi \text{freq} * \omega_m) / ((\omega_0 + j\omega)^2 - (2\pi \text{freq})^2)$		Complex relative magnetic permeability, xy-component

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 **Materials** 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 the **Add to Component** button in the window toolbar.
- 5 In the **Materials** 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, Ferrite


- 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 In the **Settings** window for **Wave Equation, Electric**, type Wave Equation, Electric 2, Ferrite in the **Label** text field.
- 4 Select Domain 2 only.
- 5 Locate the **Electric Displacement Field** section. From the **Electric displacement field model** list, choose **Loss tangent, dissipation factor**.
- 6 From the ϵ' list, choose **User defined**. In the associated text field, type 14.5.
- 7 From the $\tan\delta$ list, choose **User defined**. In the associated text field, type 0.0002.
- 8 Locate the **Magnetic Field** section. From the μ_r list, choose **User defined**. From the list, choose **Full**.
- 9 Specify the μ_r matrix as


μ_{r_xx}	μ_{r_xy}	0
$-\mu_{r_xy}$	μ_{r_xx}	0
0	0	1

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** checkbox. 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.

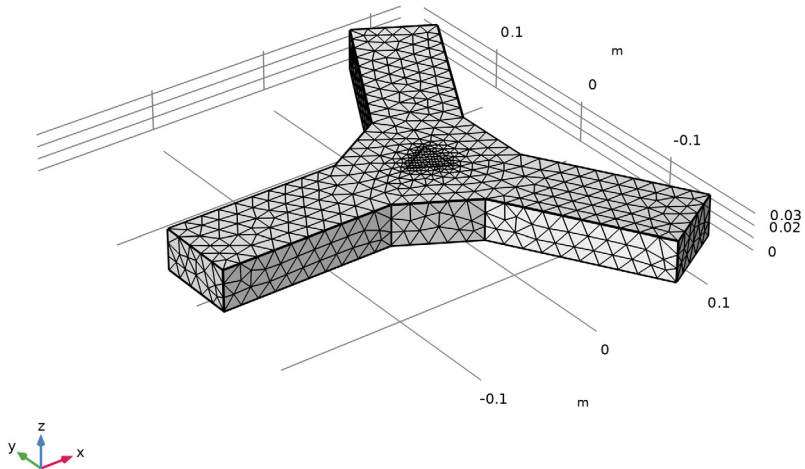
4 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Domain**.

5 Select Domain 2 only.

6 Locate the **Element Size Parameters** section.

7 Select the **Maximum element size** checkbox. 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 **Study** 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

Electric Field (emw) 1

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 1


- 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 1

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 the **Add Study** button 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.

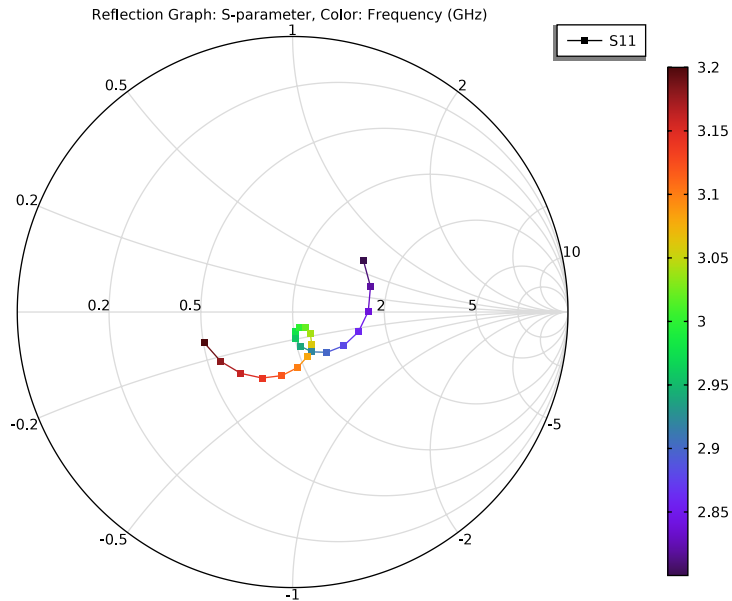
Global 1

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**.




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 **Preset Studies for Selected Physics Interfaces > Frequency Domain Source Sweep**.
- 4 Click the **Add Study** button in the window toolbar.
- 5 In the **Home** toolbar, click  **Add Study** to close the **Add Study** window.

STUDY 3

Step 1: Frequency Domain Source Sweep

- 1 In the **Settings** window for **Frequency Domain Source Sweep**, locate the **Study Settings** section.
- 2 In the **Frequencies** text field, type 3[GHz].
- 3 In the **Study** toolbar, click  **Compute**.

Check the S-parameter matrix in a table.

RESULTS

Electric Field, Logarithmic (emw) 2

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 5

- 1 In the **Model Builder** window, expand the **Results > Derived Values** node.
- 2 Right-click **Component 1 (comp1) > Definitions** and choose **View**.
- 3 In the **Settings** window for **View**, locate the **View** section.
- 4 Clear the **Show grid** checkbox.

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 5**.
- 4 Click the  **Go to Default View** button in the **Graphics** toolbar.
- 5 In the **Electric Field (emw) 3** toolbar, click  **Plot**.

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

Multislice 1

- 1 In the **Model Builder** window, expand the **Electric Field (emw) 3** node.
- 2 Right-click **Multislice 1** and choose **Delete**.

Surface 1

- 1 Right-click **Electric Field (emw) 3** and choose **Surface**.
- 2 In the **Settings** window for **Surface**, locate the **Expression** section.
- 3 In the **Expression** text field, type `emw.normE`.
- 4 Locate the **Coloring and Style** section. From the **Color table** list, choose **Avicularia**.

Selection 1

- 1 Right-click **Surface 1** and choose **Selection**.
- 2 Select Boundary 3 only.

Add deformation proportional to the electric field.

Deformation 1

- 1 In the **Model Builder** window, right-click **Surface 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**.

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** checkbox.
- 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**.

Color Expression 1


- 1 Right-click **Arrow Volume 1** and choose **Color Expression**.
- 2 In the **Settings** window for **Color Expression**, locate the **Expression** section.
- 3 In the **Expression** text field, type `emw.normE`.

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

While the arrow represents the direction of the magnetic field, the color indicates the strength of the electric field norm.

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, under **Results** 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.