Created in COMSOL Multiphysics 5.6

Computing Q-Factors and Resonant Frequencies of Cavity Resonators

Introduction

A classic benchmark example in computational electromagnetics is to find the resonant frequency and Q-factor of a cavity with lossy walls. Here, models of rectangular, cylindrical, and spherical cavities are shown to be in agreement with analytic solutions.

Model Definition

This example considers three geometries:

- a rectangular cavity of dimensions 0.9 in-by-0.9 in-by-0.4 in;
- a cylindrical cavity of radius 0.48 in and height 0.4 in; and
- a spherical cavity of radius 1.35 cm.

The cavity walls are assumed to be a good conductor, such as copper, with an electric conductivity of $5.7 \cdot 10^7$ S/m, and relative permeability and permittivity of unity. The interior of the cavity is assumed to be vacuum, with zero electric conductivity, and unit permeability and permittivity. The analytic solutions to these three cases are given in Ref. 1.

The lossy walls of the cavity are represented via the impedance boundary condition. This boundary condition accounts for the frequency dependent losses on the walls of a cavity due to the nonzero electric conductivity, which makes the eigenvalue problem nonlinear. When solving any eigenvalue problem, it is necessary to provide a frequency around which to search for modes. In addition, when solving a nonlinear eigenvalue problem, it is also necessary to provide a frequency at which to initially evaluate the frequency-dependent surface losses. Although the guesses for these frequencies do not need to be very close, solution time is less the closer they are.

It is usually possible to estimate the resonant frequency of interest, and to use this as an initial guess. It is also possible to quickly estimate the resonant frequency by building a second model that uses the perfect electrical conductor (PEC) boundary condition instead of the impedance boundary condition. A model that uses only PEC boundaries results in a linear eigenvalue problem, and is less computationally intensive to solve. Such a model only requires a rough guess at the frequency of the mode, and does not require a frequency at which to evaluate the surface losses. Therefore, it is often convenient to also solve a version of a model without losses.

Q-FACTOR AND RESONANT FREQUENCY IN CAVITY STRUCTURES

Q-factor is one of important parameters characterizing a resonant structure and defined as $Q = \omega$ (average energy stored/dissipated power). The average energy stored can be evaluated as a volume integral of Energy density time average (`emw.Wav`) and the dissipated power can be evaluated as a surface integral of Surface losses (`emw.Qsh`).

Another way to calculate Q-factor at the dominant mode is via equations in Ref. 1. For a rectangular cavity, the dominant mode is TE_{101}, at which the cavity provides the lowest resonant frequency. The Q-factor and resonant frequency at this mode is

$$Q_{TE_{101}} = \frac{1.1107\eta}{R_s\left(1+\frac{a}{2b}\right)},\quad f_{TE_{101}} = \frac{1}{2\pi\sqrt{\mu\varepsilon}}\sqrt{\left(\frac{\pi}{a}\right)^2+\left(\frac{\pi}{c}\right)^2}$$

There are two dominant modes for a cylindrical cavity. One dominant mode of the cylindrical cavity is TE_{111} when the ratio between the height and radius is more than 2.03. The other dominant mode is TM_{010} when the ratio is less than 2.03. For this case, the Q-factor and resonant frequency are given as

$$Q_{TM_{010}} = \frac{1.2025\eta}{R_s\left(1+\frac{a}{h}\right)},\quad f_{TM_{010}} = \frac{1}{2\pi\sqrt{\mu\varepsilon}}\sqrt{\left(\frac{2.40492}{a}\right)^2}$$

For a spherical cavity, TM mode provides the lowest resonant frequency.

$$Q_{TM_{011}} = \frac{1.0041\eta}{R_s},\quad f_{TM_{011}} = \frac{2.744}{2\pi a\sqrt{\mu\varepsilon}}$$

In the above equations, R_s is surface resistance defined as

$$R_s = \sqrt{\frac{\omega_r\mu}{2\sigma}}$$

and η is the characteristic impedance of free space, $\sqrt{\mu_0/\varepsilon_0}$.

These two analytical approaches are compared with the Q-factor obtained from Eigenfrequency analysis.

Results and Discussion

The analytic resonant frequencies and Q-factors for these three cases, and the results of the COMSOL model for various levels of mesh refinement, are shown below. These show that the solutions agree. As the mesh is refined, the polynomial basis functions used by the finite element method better approximate the analytic solutions, which are described by sinusoidal functions for the rectangular cavity and Bessel functions for the cylindrical and spherical cavities. This difference between the numerical results and the analytic solution is discretization error, and is always reduced with mesh refinement.

TABLE 1: RESULTS FOR THE TE101 MODE OF A RECTANGULAR CAVITY.

MAXIMUM MESH SIZE	RESONANT FREQUENCY, GHZ (ANALYTIC=9.273)	Q-FACTOR (ANALYTIC=7770)
h_max	9.706	7039
h_max/2	9.283	7687
h_max/4	9.273	7765
h_max/8	9.273	7770

TABLE 2: RESULTS FOR THE TM010 MODE OF A CYLINDRICAL CAVITY.

MAXIMUM MESH SIZE	RESONANT FREQUENCY, GHZ (ANALYTIC=9.412)	Q-FACTOR (ANALYTIC=8065)
h_max	9.458	7891
h_max/2	9.419	8004
h_max/4	9.411	8056
h_max/8	9.411	8065

TABLE 3: RESULTS FOR THE TM011 MODE OF A SPHERICAL CAVITY.

MAXIMUM MESH SIZE	RESONANT FREQUENCY, GHZ (ANALYTIC=9.698)	Q-FACTOR (ANALYTIC=14594)
h_max	9.752	14121
h_max/2	9.723	14430
h_max/4	9.701	14616
h_max/8	9.697	14641

Note that convergence with respect to the mesh is fastest for the rectangular cavity and slowest for the spherical cavity. This is because the isoparametric finite-element mesh represents curved surfaces approximately, via second order polynomials by default. This introduces some small geometric discretization error that is always reduced with mesh refinement. Although it is possible to use different element orders, the default second-order curl element (also known as a vector or Nedelec element) is the best compromise between accuracy and memory requirements. Because memory requirements for three-dimensional models and direct solvers increase close to quadratically with increasing number of unknowns, there is strong motivation to use as coarse a mesh as reasonable. For larger models, one may use an iterative solver that scales more favorably with the number of unknowns but then solution time typically goes up substantially. Figure 1 shows the fields within the cavities, as well as the surface currents and surface losses.

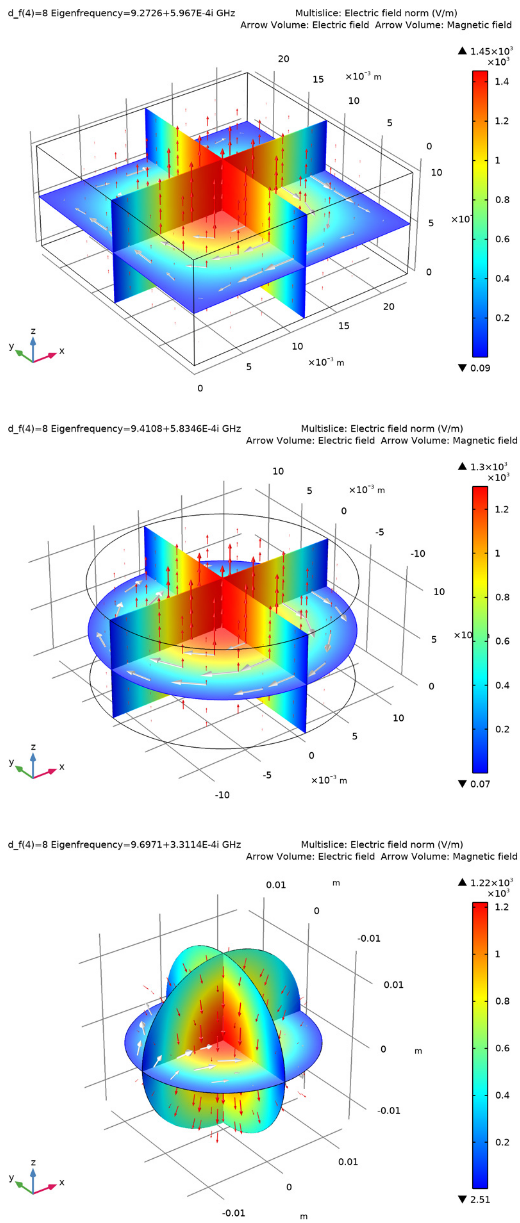


Figure 1: Arrow plots of electric and magnetic fields. Slice plot of electric field.

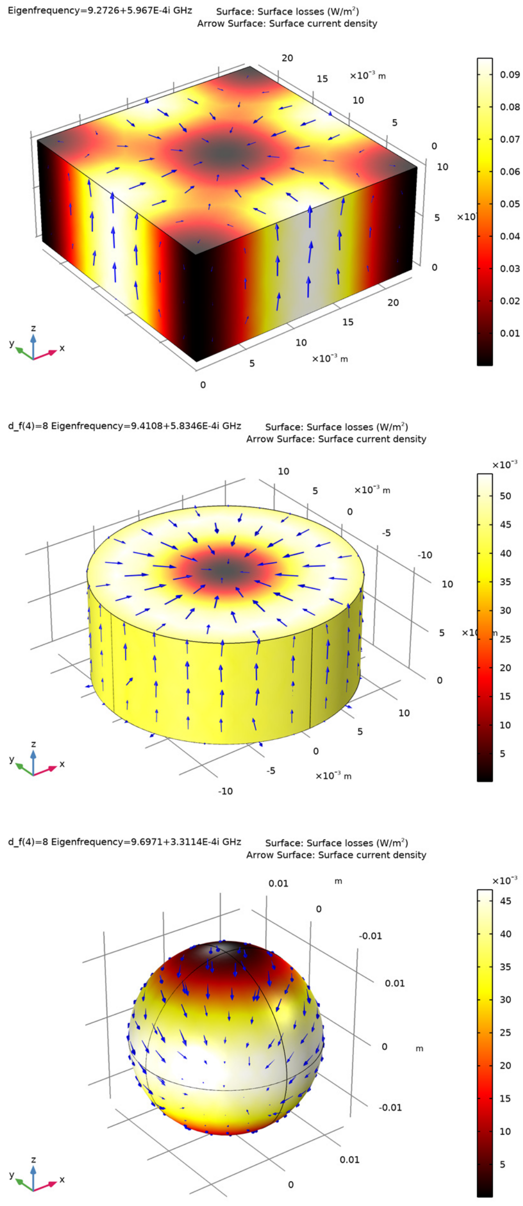


Figure 2: Arrow plots of surface currents. Surface plot of surface losses.

Notes About the COMSOL Implementation

Solve this example using an Eigenfrequency study. Search for a single eigenfrequency around $9 \cdot 10^9$ Hz. Because of the impedance boundary condition with a finite conductivity value, the model becomes a nonlinear eigenvalue problem and it is necessary to provide a frequency at which to initially evaluate the frequency-dependent surface losses. In the Eigenvalue Solver settings window you can see the linearization point is automatically specified to the value in "Search for eigenfrequencies around" in the study settings.

Reference

1. C.A. Balanis, *Advanced Engineering Electromagnetics*, John Wiley & Sons, 1989.

Application Library path: `RF_Module/Verification_Examples/cavity_resonators`

Modeling Instructions

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

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

2 In the **Select Physics** tree, select **Radio Frequency>Electromagnetic Waves, Frequency Domain (emw)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies>Eigenfrequency**.

6 Click **Done**.

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 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `cavity_resonators_parameters.txt`.

   Here, `mu0_const` and `epsilon0_const` in the imported table are predefined COMSOL constants for the permeability and permittivity in free space. From the Value column you can read off the values `f_TE101_analytic_r` = 9.273 GHz, `Q_TE101_analytic_r` = 7770 for the rectangular cavity, `f_TM010_analytic_c` = 9.412 GHz, `Q_TM010_analytic_c` = 8065 for the cylindrical cavity, `f_TM011_analytic_s` = 9.698 GHz, and `Q_TM011_analytic_s` = 14594 for the spherical cavity.

Since air and lossy wall materials will be used on multiple components, add them on the global material node. They will be linked to each individual component later on.

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 Global Materials**.

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

GLOBAL DEFINITIONS

Lossy Wall

1 In the **Model Builder** window, under **Global Definitions** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Lossy Wall` in the **Label** text field.

3 Click to expand the **Material Properties** section. In the **Material properties** tree, select **Basic Properties>Relative Permittivity**.

4 Click **Add to Material**.

5 In the **Material properties** tree, select **Basic Properties>Relative Permeability**.

6 Click **Add to Material**.

7 In the **Material properties** tree, select **Basic Properties>Electrical Conductivity**.

8 Click **Add to Material**.

9 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	1	1	Basic
Relative permeability	mur_iso ; murii = mur_iso, murij = 0	1	1	Basic
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	`sigma_wall`	S/m	Basic

GEOMETRY 1

Create a block for the rectangular cavity.

Block 1 (blk1)

1 In the **Geometry** toolbar, click **Block**.

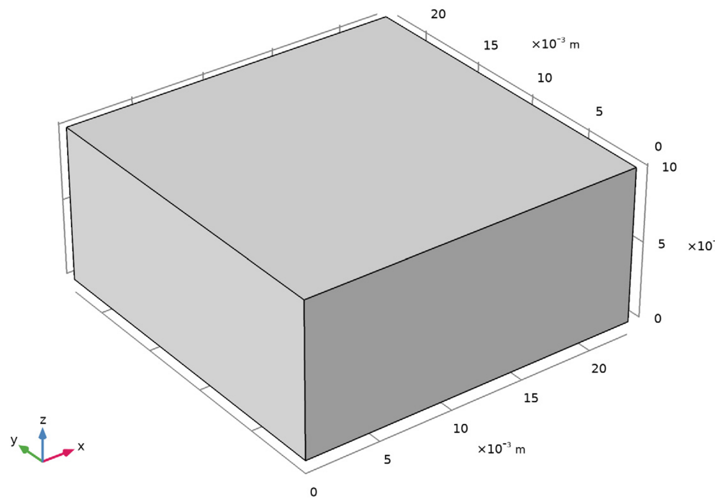
2 In the **Settings** window for **Block**, locate the **Size and Shape** section.

3 In the **Width** text field, type `a_r`.

4 In the **Depth** text field, type `a_r`.

5 In the **Height** text field, type `b_r`.

6 Click **Build All Objects**.



ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

Now set up the physics. Override the default perfect electric conductor condition on the exterior boundaries by an impedance condition.

Impedance Boundary Condition 1

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose **Impedance Boundary Condition**.
2. In the **Settings** window for **Impedance Boundary Condition**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **All boundaries**.

MATERIALS

Assign material properties on the model by linking the global material already created. First, apply air to all domains.

Material Link 1 (matlnk1)

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **More Materials>Material Link**.

Material Link 2 (matlnk2)

1 Right-click **Materials** and choose **More Materials>Material Link**.

Define a lossy conductive material for all exterior boundaries.

2 In the **Settings** window for **Material Link**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **All boundaries**.

5 Locate the **Link Settings** section. From the **Material** list, choose **Lossy Wall (mat2)**.

DEFINITIONS

Add variables for Q-factor calculation and visualization. For this Q-factor calculation, add two nonlocal integration couplings: one for volume and the other for surface integration.

Integration 1 (intop1)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type `int_v` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Selection** list, choose **All domains**.

Integration 2 (intop2)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type `int_s` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **All boundaries**.

Variables 1

1 In the **Definitions** toolbar, click a= **Local Variables**.

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

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `cavity_resonators_model1_variables.txt`.

The `emw.` prefix is for the Electromagnetic Waves, Frequency Domain interface in the first model. `Wav` and `Qsh` are Energy density time average and Surface losses, respectively. `Qfactor` included in this text file shows up in orange indicating an unknown variable. It will be known after solving the model.

MESH 1

The maximum mesh size is one dimension of the cavity scaled inversely by `d_f`, a discretization factor defined in Parameters. The discretization factor is also used as a parametric sweep variable to see the effect of the mesh refinement.

Free Tetrahedral 1

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

Size

1 In the **Model Builder** window, click **Size**.
2 In the **Settings** window for **Size**, locate the **Element Size** section.
3 Click the **Custom** button.
4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `h_max_r/d_f`.
5 In the **Maximum element growth rate** text field, type `2`.
6 In the **Curvature factor** text field, type `1`.
7 In the **Resolution of narrow regions** text field, type `0.1`.
8 Click **Build All**.

Provide the number of modes and a frequency around which to search for modes.

STUDY 1

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Study 1** click **Step 1: Eigenfrequency**.
2 In the **Settings** window for **Eigenfrequency**, locate the **Study Settings** section.
3 Select the **Desired number of eigenfrequencies** check box.
4 In the associated text field, type `1`.
5 In the **Search for eigenfrequencies around** text field, type `9[GHz]`.

Add a Parametric Sweep over the discretization factor, d_f.

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
d_f (Discretization factor)	1 2 4 8	

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

RESULTS

Electric Field (emw)

The default plot shows the distribution of the norm of the electric field. Add arrow plots of the electric and magnetic fields.

Arrow Volume 1

1 Right-click **Electric Field (emw)** 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>Electric>emw.Ex,emw.Ey,emw.Ez - Electric field**.

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

Arrow Volume 2

1 Right-click **Electric Field (emw)** 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 **Arrow Positioning** section. Find the **Z grid points** subsection. In the **Points** text field, type 1.

4 Locate the **Coloring and Style** section. From the **Color** list, choose **White**.

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

6 Click the **Zoom Extents** button in the **Graphics** toolbar.

Compare the resulting plot with that shown in Figure 1, top. The exact numbers that you get may differ slightly.

Add a surface plot of the surface losses and an arrow plot of the surface current (Figure 2, top).

3D Plot Group 2

In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

Surface 1

1 Right-click **3D Plot Group 2** and choose **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Electromagnetic Waves, Frequency Domain>Heating and losses>emw.Qsh - Surface losses - W/m²**.

3 Locate the **Coloring and Style** section. From the **Color table** list, choose **ThermalEquidistant**.

Surface Losses (emw)

1 In the **Model Builder** window, under **Results** click **3D Plot Group 2**.

2 In the **Settings** window for **3D Plot Group**, type `Surface Losses (emw)` in the **Label** text field.

Arrow Surface 1

1 Right-click **Surface Losses (emw)** and choose **Arrow Surface**.

2 In the **Settings** window for **Arrow Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Electromagnetic Waves, Frequency Domain>Currents and charge>emw.Jsx,...,emw.Jsz - Surface current density**.

3 Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.

4 In the **Surface Losses (emw)** toolbar, click **Plot**.

ROOT

Next, set up a model for the cylindrical cavity.

ADD COMPONENT

In the **Model Builder** window, right-click the root node and choose **Add Component>3D**.

ADD PHYSICS

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

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

3 In the tree, select **Radio Frequency>Electromagnetic Waves, Frequency Domain (emw)**.

4 Click **Add to Component 2** in the window toolbar.

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

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 **Empty Study**.

You will copy the settings from the existing study later on.

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

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

GEOMETRY 2

Cylinder 1 (cyl1)

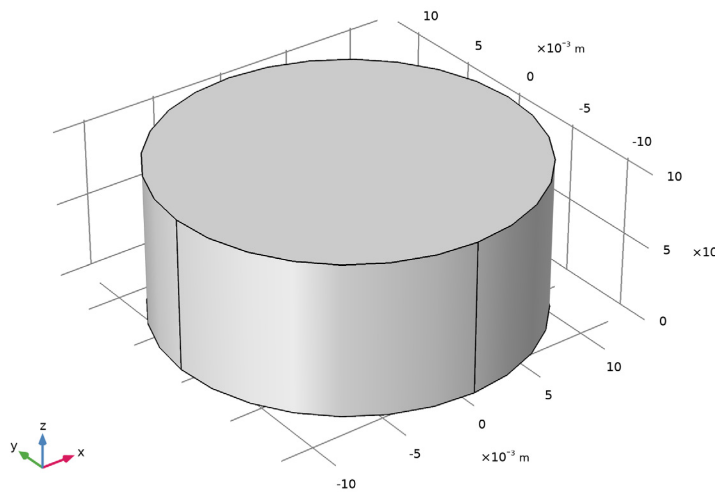
1 In the **Geometry** toolbar, click **Cylinder**.

2 In the **Settings** window for **Cylinder**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `a_c`.

4 In the **Height** text field, type `height_c`.

5 Click **Build All Objects**.



ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN 2 (EMW2)

Set up the second physics interface. The steps are same as for the first model.

1 In the **Model Builder** window, under **Component 2 (comp2)** click **Electromagnetic Waves, Frequency Domain 2 (emw2)**.

Impedance Boundary Condition 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Impedance Boundary Condition**.

2 In the **Settings** window for **Impedance Boundary Condition**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **All boundaries**.

MATERIALS

Assign material properties on the second model. Apply air to all domains.

Material Link 3 (matlnk3)

In the **Model Builder** window, under **Component 2 (comp2)** right-click **Materials** and choose **More Materials>Material Link**.

Material Link 4 (matlnk4)

1 Right-click **Materials** and choose **More Materials>Material Link**.

Define a lossy conductive material for all exterior boundaries.

2 In the **Settings** window for **Material Link**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **All boundaries**.

5 Locate the **Link Settings** section. From the **Material** list, choose **Lossy Wall (mat2)**.

DEFINITIONS (COMP2)

Add variables and two nonlocal integration couplings. The purpose of these are the same as in the first model.

Integration 3 (intop3)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type `int_v` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Selection** list, choose **All domains**.

Integration 4 (intop4)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type `int_s` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **All boundaries**.

Variables 2

1 In the **Definitions** toolbar, click **Local Variables**.

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

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `cavity_resonators_model2_variables.txt`.

The `emw2.` prefix refers to the Electromagnetic Waves, Frequency Domain interface for the second model.

MESH 2

Apply the same logic in the mesh set up as you have done in the first model.

Free Tetrahedral 1

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

Size

1 In the **Model Builder** window, click **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `h_max_c/d_f`.

5 In the **Maximum element growth rate** text field, type `2`.

6 In the **Curvature factor** text field, type `1`.

7 In the **Resolution of narrow regions** text field, type `0.1`.

8 Click **Build All**.

STUDY 1

Parametric Sweep, Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Study 1**, Ctrl-click to select **Parametric Sweep** and **Step 1: Eigenfrequency**.

2 Right-click and choose **Copy**.

STUDY 2

Parametric Sweep

In the **Model Builder** window, right-click **Study 2** and choose **Paste Multiple Items**.

STUDY 2

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Study 2** click **Step 1: Eigenfrequency**.
2 In the **Settings** window for **Eigenfrequency**, locate the **Physics and Variables Selection** section.
3 In the table, enter the following settings:

Physics interface	Solve for	Discretization
Electromagnetic Waves, Frequency Domain (emw)		Physics settings
Electromagnetic Waves, Frequency Domain 2 (emw2)	√	Physics settings

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

RESULTS

Arrow Volume 1

1 Right-click **Electric Field (emw2)** 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 2 (comp2)>Electromagnetic Waves, Frequency Domain 2>Electric>emw2.Ex,emw2.Ey,emw2.Ez - Electric field**.
3 In the **Electric Field (emw2)** toolbar, click **Plot**.

Arrow Volume 2

1 Right-click **Electric Field (emw2)** 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 2 (comp2)>Electromagnetic Waves, Frequency Domain 2>Magnetic>emw2.Hx,emw2.Hy,emw2.Hz - Magnetic field**.
3 Locate the **Arrow Positioning** section. Find the **Z grid points** subsection. In the **Points** text field, type 1.

4 Locate the **Coloring and Style** section. From the **Color** list, choose **White**.

5 In the **Electric Field (emw2)** toolbar, click **Plot**.

6 Click the **Zoom Extents** button in the **Graphics** toolbar.

The plot should now look like that in Figure 1, middle.

Again, add a surface plot of the surface losses and an arrow plot of the surface current (Figure 2, middle).

3D Plot Group 4

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

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

3 From the **Dataset** list, choose **Study 2/Parametric Solutions 2 (6) (sol8)**.

Surface 1

1 Right-click **3D Plot Group 4** and choose **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 2 (comp2)>Electromagnetic Waves, Frequency Domain 2>Heating and losses>emw2.Qsh - Surface losses - W/m²**.

3 Locate the **Coloring and Style** section. From the **Color table** list, choose **ThermalEquidistant**.

Surface Losses (emw2)

1 In the **Model Builder** window, under **Results** click **3D Plot Group 4**.

2 In the **Settings** window for **3D Plot Group**, type `Surface Losses (emw2)` in the **Label** text field.

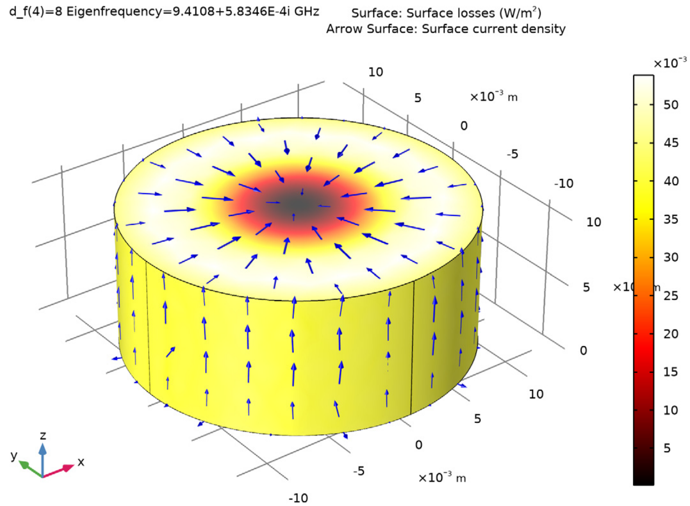
Arrow Surface 1

1 Right-click **Surface Losses (emw2)** and choose **Arrow Surface**.

2 In the **Settings** window for **Arrow Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 2 (comp2)>Electromagnetic Waves, Frequency Domain 2>Currents and charge>emw2.Jsx,...,emw2.Jsz - Surface current density**.

3 Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.

4 In the **Surface Losses (emw2)** toolbar, click **Plot**.



ROOT

Now add a model for the spherical cavity.

ADD COMPONENT

In the **Model Builder** window, right-click the root node and choose **Add Component>3D**.

ADD PHYSICS

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

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

3 In the tree, select **Radio Frequency>Electromagnetic Waves, Frequency Domain (emw)**.

4 Click **Add to Component 3** in the window toolbar.

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

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 **Empty Study**.

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

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

GEOMETRY 3

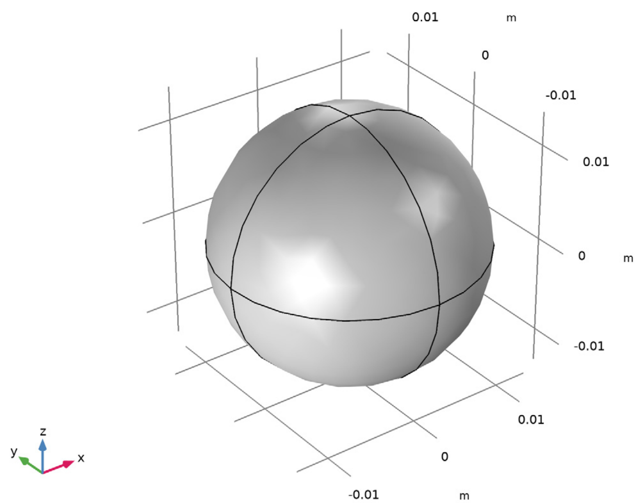
Sphere 1 (sph1)

1 In the **Geometry** toolbar, click **Sphere**.

2 In the **Settings** window for **Sphere**, locate the **Size** section.

3 In the **Radius** text field, type a_s.

4 Click **Build All Objects**.



ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN 3 (EMW3)

Set up the third physics interface.

1 In the **Model Builder** window, under **Component 3 (comp3)** click **Electromagnetic Waves, Frequency Domain 3 (emw3)**.

Impedance Boundary Condition 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Impedance Boundary Condition**.

2 In the **Settings** window for **Impedance Boundary Condition**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **All boundaries**.

MATERIALS

Assign material properties on the third model. Apply air to all domains.

Material Link 5 (matlnk5)

In the **Model Builder** window, under **Component 3 (comp3)** right-click **Materials** and choose **More Materials>Material Link**.

Material Link 6 (matlnk6)

1 Right-click **Materials** and choose **More Materials>Material Link**.

Define a lossy conductive material for all exterior boundaries.

2 In the **Settings** window for **Material Link**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **All boundaries**.

5 Locate the **Link Settings** section. From the **Material** list, choose **Lossy Wall (mat2)**.

DEFINITIONS (COMP3)

Add variables and two nonlocal integration couplings.

Integration 5 (intop5)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type `int_v` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Selection** list, choose **All domains**.

Integration 6 (intop6)

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type `int_s` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **All boundaries**.

Variables 3

1 In the **Definitions** toolbar, click a= **Local Variables**.

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

3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `cavity_resonators_model3_variables.txt`.

The `emw3.` prefix in the imported table is for the physics interface, **Electromagnetic Waves, Frequency Domain**, in the third model.

MESH 3

Free Tetrahedral 1

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

Size

1 In the **Model Builder** window, click **Size**.

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `h_max_s/d_f`.

5 In the **Maximum element growth rate** text field, type `2`.

6 In the **Curvature factor** text field, type `1`.

7 In the **Resolution of narrow regions** text field, type `0.1`.

8 Click **Build All**.

STUDY 2

Parametric Sweep, Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Study 2**, Ctrl-click to select **Parametric Sweep** and **Step 1: Eigenfrequency**.

2 Right-click and choose **Copy**.

STUDY 3

Parametric Sweep

In the **Model Builder** window, right-click **Study 3** and choose **Paste Multiple Items**.

STUDY 3

Step 1: Eigenfrequency

1 In the **Model Builder** window, under **Study 3** click **Step 1: Eigenfrequency**.

2 In the **Settings** window for **Eigenfrequency**, locate the **Physics and Variables Selection** section.

3 In the table, enter the following settings:

Physics interface	Solve for	Discretization
Electromagnetic Waves, Frequency Domain (emw)		Physics settings
Electromagnetic Waves, Frequency Domain 2 (emw2)		Physics settings
Electromagnetic Waves, Frequency Domain 3 (emw3)	√	Physics settings

Solution 13 (sol13)

1 In the **Study** toolbar, click **Show Default Solver**.

2 Click **Compute**.

RESULTS

Arrow Volume 1

1 Right-click **Electric Field (emw3)** 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 3 (comp3)>Electromagnetic Waves, Frequency Domain 3>Electric>emw3.Ex,emw3.Ey,emw3.Ez - Electric field**.

Arrow Volume 2

1 Right-click **Electric Field (emw3)** 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 3 (comp3)>Electromagnetic Waves, Frequency Domain 3>Magnetic>emw3.Hx,emw3.Hy,emw3.Hz - Magnetic field**.

3 Locate the **Arrow Positioning** section. Find the **Z grid points** subsection. In the **Points** text field, type 1.

4 Locate the **Coloring and Style** section. From the **Color** list, choose **White**.

5 In the **Electric Field (emw3)** toolbar, click **Plot**.

6 Click the **Zoom Extents** button in the **Graphics** toolbar.

Compare the resulting plot with that shown in Figure 1, bottom.

Again, add a surface plot of the surface losses and an arrow plot of the surface current (Figure 2, bottom).

3D Plot Group 6

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.
2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.
3 From the **Dataset** list, choose **Study 3/Parametric Solutions 3 (12) (sol14)**.

Surface 1

1 Right-click **3D Plot Group 6** and choose **Surface**.
2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 3 (comp3)>Electromagnetic Waves, Frequency Domain 3>Heating and losses>emw3.Qsh - Surface losses - W/m**^{2}.
3 Locate the **Coloring and Style** section. From the **Color table** list, choose **ThermalEquidistant**.

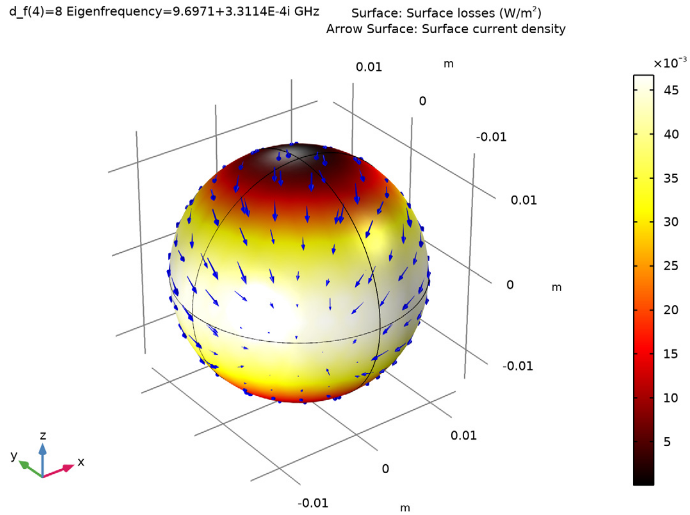
Surface Losses (emw3)

1 In the **Model Builder** window, under **Results** click **3D Plot Group 6**.
2 In the **Settings** window for **3D Plot Group**, type `Surface Losses (emw3)` in the **Label** text field.

Arrow Surface 1

1 Right-click **Surface Losses (emw3)** and choose **Arrow Surface**.
2 In the **Settings** window for **Arrow Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 3 (comp3)>Electromagnetic Waves, Frequency Domain 3>Currents and charge>emw3.Jsx,...,emw3.Jsz - Surface current density**.
3 Locate the **Coloring and Style** section. From the **Color** list, choose **Blue**.

4 In the **Surface Losses (emw3)** toolbar, click **Plot**.



Finish by evaluating the Q-factor and resonant frequency. Compare them with those values in Table 1, Table 2 and Table 3.

Global Evaluation 4

1 In the **Results** toolbar, click **Global Evaluation**.

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

3 From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

4 From the **Eigenfrequency selection** list, choose **First**.

5 From the **Table columns** list, choose **Inner solutions**.

6 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Definitions>Variables>Q_computed - Q-factor, computed from eigenvalue**.

7 Click **Evaluate**.

8 Locate the **Data** section. From the **Dataset** list, choose **Study 2/ Parametric Solutions 2 (6) (sol8)**.

9 Click **Evaluate**.

10 From the **Dataset** list, choose **Study 3/Parametric Solutions 3 (12) (sol14)**.

11 Click **Evaluate**.

Global Evaluation 5

1 Right-click **Global Evaluation 4** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Definitions>Variables>Q_definition - Q-factor, definition**.

5 Click ▾ next to **Evaluate**, then choose **New Table**.

6 Locate the **Data** section. From the **Dataset** list, choose **Study 2/ Parametric Solutions 2 (6) (sol8)**.

7 Click **Evaluate**.

8 From the **Dataset** list, choose **Study 3/Parametric Solutions 3 (12) (sol14)**.

9 Click **Evaluate**.

Global Evaluation 6

1 In the **Model Builder** window, under **Results>Derived Values** right-click **Global Evaluation 4** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Definitions>Variables>frequency - Frequency, simulated - Hz**.

5 Click ▾ next to **Evaluate**, then choose **New Table**.

6 Locate the **Data** section. From the **Dataset** list, choose **Study 2/ Parametric Solutions 2 (6) (sol8)**.

7 Click **Evaluate**.

8 From the **Dataset** list, choose **Study 3/Parametric Solutions 3 (12) (sol14)**.

9 Click **Evaluate**.