Frequency Domain Modeling of a Capacitor

Introduction

A capacitor with an applied sinusoidally time-varying voltage difference is modeled. A wide frequency range is considered and the impedance of the device is computed. Solver accuracy is addressed. The relationship between the frequency domain impedance and the steady-state capacitance and resistance of the device is discussed.

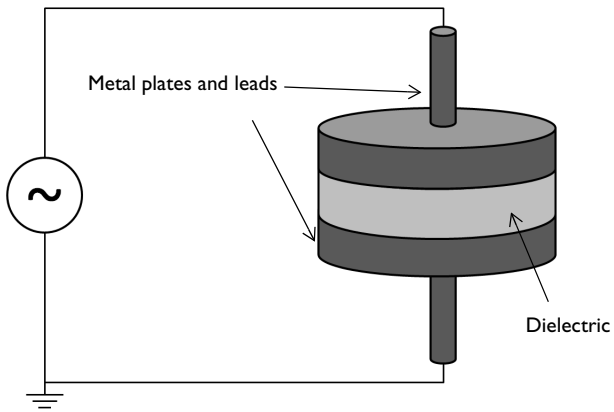


Figure 1: A simple capacitor composed of a disk of dielectric with metal plates on either side, and lead wires. An AC signal is applied.

Model Definition

The capacitor being modeled is shown in Figure 1, two metal disks, with leads, are separated by a disk of quartz glass with relative permittivity $\varepsilon_r = 4.2$ and a small electric conductivity $\sigma = 10^{-14}$ S/m. A region of surrounding air ($\varepsilon_r = 1.0$, $\sigma = 0$ S/m) is also modeled to account for the fringing fields. A sinusoidally time-varying 10 kV source is connected to the leads and the voltage fields and admittance of the device is computed.

It is assumed that the capacitor plates are made of a highly conductive material, that their total effective resistivity is much lower than that of the quartz. It is further assumed that the absolute magnitude of the current flow is small, which implies negligible magnetic fields and magnetically induced currents. Under these assumptions, it is not necessary to solve for the fields inside of the capacitor plates, they can be assumed to be at an equipotential at any instant in time.

A separate electrostatic analysis can be used to compute the capacitance of the device, $C = 43.4 \times 10^{-12}$ F. Similarly, a separate stationary Electric Currents analysis can be used to compute the conductance of the cylinder of quartz: $G = 1.05 \times 10^{-14}$ S. Under the previous assumptions, and for a device with no frequency-dependent material properties, the admittance can be computed from:

$$Y(\omega) = G + j\omega C \tag{1}$$

Where ω is the angular frequency of excitation. It can be observed from this equation that as the frequency increases, the imaginary component of the admittance, the capacitive contribution, becomes many orders of magnitude larger than the real component, the resistive contribution.

In such cases, where the model is dominated by either conductive or capacitive effects, it can become necessary to change the default solver settings. The default solver for 3D electromagnetic problems uses an iterative approach, which means that the solution is found to within a relative tolerance. The default relative tolerance for the iterative solver is 10^{-3}, but this can be tightened up to 10^{-6} if needed. Beyond a relative tolerance of 10^{-6} it may become difficult for the iterative solver to converge. If even higher accuracy is needed, it is also possible to use a direct solver, although this does take more time and memory to solve. Both iterative and direct solvers are used, and compared.

Results and Discussion

The electric potential is plotted in Figure 2. The admittance is computed and the real and imaginary components of the admittance are shown to be related to the steady state capacitance and inductance. When using the iterative solver, and as the capacitive effects begin to dominate the admittance, the resistive component of the admittance becomes less accurate. Once the real component of the admittance is less than six orders of magnitude smaller than the imaginary component, the results are below the tolerance threshold of the iterative solver, and the real component of the admittance is no longer accurate. Using the direct solver can improve this accuracy, if desired.

 ©2012 COMSOL

Irrespective of the solver used, and the tolerance specified, the dominant component of the admittance is computed correctly. This device is primarily capacitive, and such capacitive behavior will be captured correctly, even with the default solver settings. The total resistive losses within the dielectric, are also computed correctly, regardless of solution method.

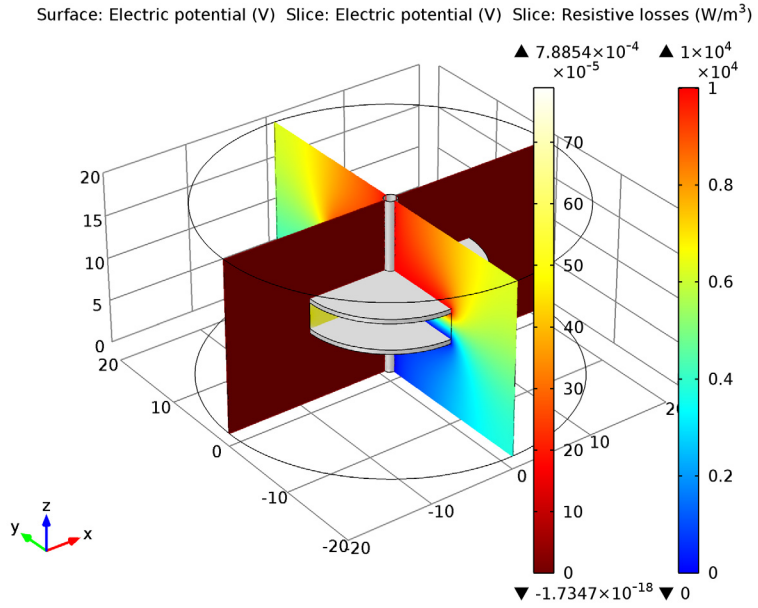


Figure 2: The electric field strength and resistive losses in the dielectric and air domain surrounding the capacitor.

Model Library path: `ACDC_Module/Capacitive_Devices/capacitor_ac`

Modeling Instructions

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Click **Next**.

3 In the **Add physics** tree, select **AC/DC>Electric Currents (ec)**.

4 Click **Add Selected**.

5 Click **Next**.

6 Find the **Studies** subsection. In the tree, select **Preset Studies>Frequency Domain**.

7 Click **Finish**.

GEOMETRY 1

1 In the **Model Builder** window, under **Model 1** click **Geometry 1**.

2 In the **Geometry** settings window, locate the **Units** section.

3 From the **Length unit** list, choose **cm**.

First, create a cylinder for the model domain.

Cylinder 1

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Cylinder**.

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

3 In the **Radius** edit field, type `20`.

4 In the **Height** edit field, type `20`.

5 Click the **Build Selected** button.

Choose wireframe rendering to get a better view of the interior parts.

6 Click the **Wireframe Rendering** button on the Graphics toolbar.

Then, add a cylinder for the disk of dielectric with two metal plates.

Cylinder 2

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Cylinder**.

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

3 In the **Radius** edit field, type `10`.

4 In the **Height** edit field, type `4`.

5 Locate the **Position** section. In the **z** edit field, type `8`.

6 Click to expand the **Layers** section. In the table, enter the following settings:

LAYER NAME	THICKNESS (CM)
`Layer 1`	`5[mm]`

7 Clear the **Layers on side** check box.

8 Select the **Layers on bottom** check box.

 ©2012 COMSOL

9 Select the **Layers on top** check box.

Finish the geometry creation by adding two cylinders for the leads.

Cylinder 3

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Cylinder**.

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

3 In the **Radius** edit field, type `0.75`.

4 In the **Height** edit field, type `8`.

Cylinder 4

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Cylinder**.

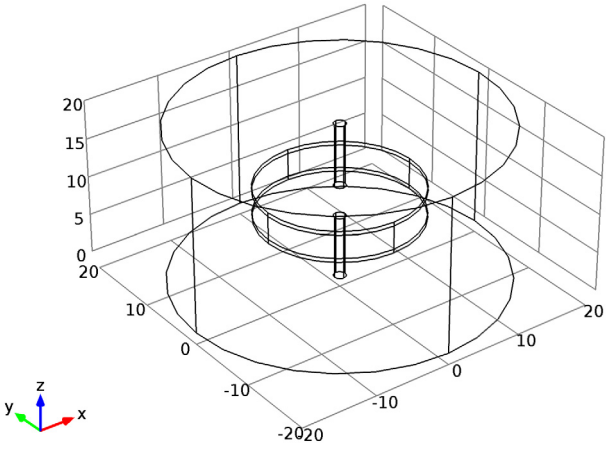
2 In the **Cylinder** settings window, locate the **Size and Shape** section.

3 In the **Radius** edit field, type `0.75`.

4 In the **Height** edit field, type `8`.

5 Locate the **Position** section. In the **z** edit field, type `12`.

6 Click the **Build All** button.



The model is composed of the disk of dielectric with metal plates on either side, and lead wires.

DEFINITIONS

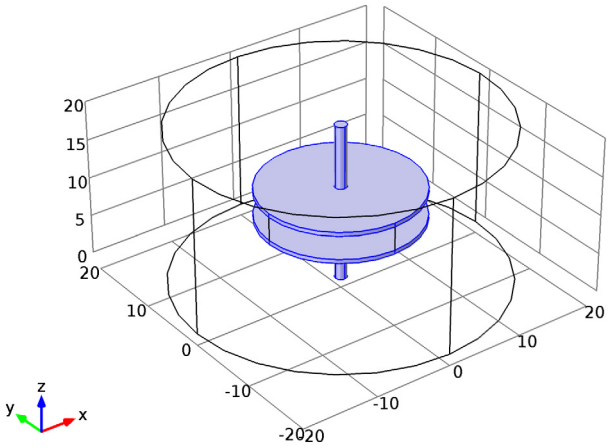
Create a set of selections for use before setting up the physics. First, create a selection for the metal domain.

Explicit 1

1 In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Selections>Explicit**.

2 In the **Model Builder** window, click **Explicit 1**.

3 Select Domains 2 and 4–6 only.



4 Right-click **Explicit 1** and choose **Rename**.

5 Go to the **Rename Explicit** dialog box and type `Metal` in the **New name** edit field.

6 Click **OK**.

Add a selection for the model domain.

Complement 1

1 Right-click **Definitions** and choose **Selections>Complement**.

2 In the **Complement** settings window, locate the **Input Entities** section.

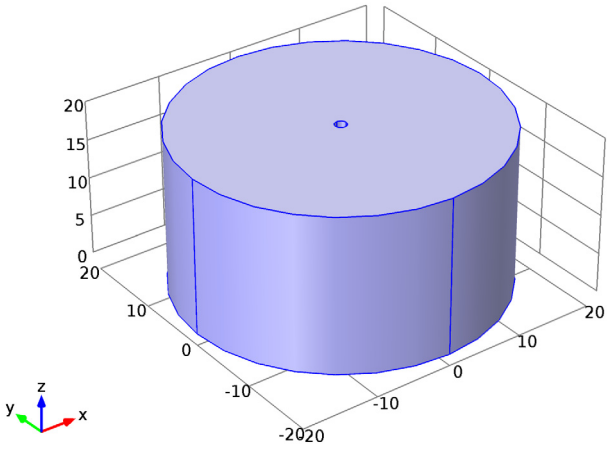
3 Under **Selections to invert**, click **Add**.

4 Go to the **Add** dialog box.

5 In the **Selections to invert** list, select **Metal**.

 ©2012 COMSOL

6 Click the **OK** button.



7 Right-click **Model 1>Definitions>Complement 1** and choose **Rename**.

8 Go to the **Rename Complement** dialog box and type `Model domain` in the **New name** edit field.

9 Click **OK**.

Add a selection for the ground boundaries.

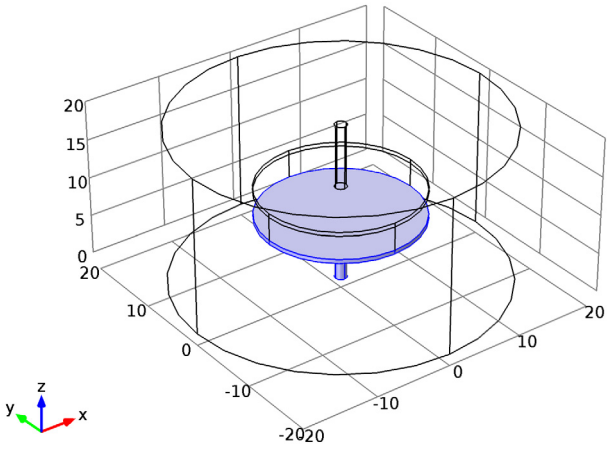
Explicit 2

1 Right-click **Definitions** and choose **Selections>Explicit**.

2 In the **Explicit** settings window, locate the **Input Entities** section.

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

4 Select Boundaries 5–7, 10, 15, 16, 24, 27, 29, and 31 only.



5 Right-click **Model 1>Definitions>Explicit 2** and choose **Rename**.

6 Go to the **Rename Explicit** dialog box and type `Ground boundaries` in the **New name** edit field.

7 Click **OK**.

Add a selection for the terminal boundaries.

Explicit 3

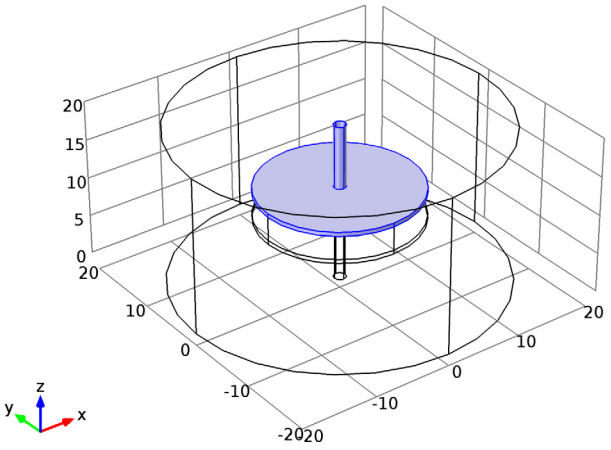
1 Right-click **Definitions** and choose **Selections>Explicit**.

2 In the **Explicit** settings window, locate the **Input Entities** section.

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

 ©2012 COMSOL

4 Select Boundaries 11–14, 19, 20, 26, 28, 30, and 33 only.



5 Right-click **Model 1>Definitions>Explicit 3** and choose **Rename**.

6 Go to the **Rename Explicit** dialog box and type `Terminal boundaries` in the **New name** edit field.

7 Click **OK**.

View 1

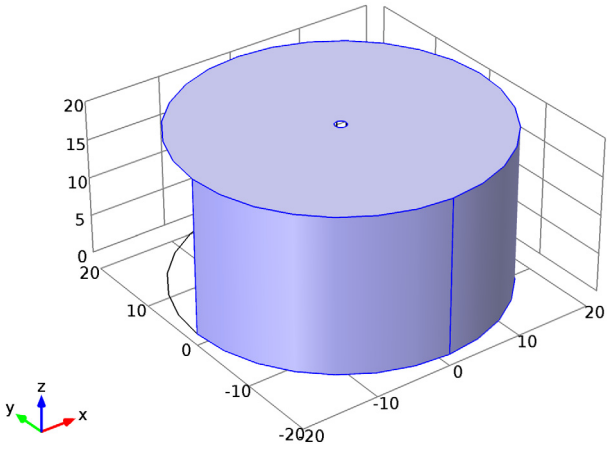
To get a better view, suppress some of the boundaries.

1 In the **Model Builder** window, under **Model 1>Definitions** right-click **View 1** and choose **Hide Geometric Entities**.

2 In the **Hide Geometric Entities** settings window, locate the **Geometric Entity Selection** section.

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

4 Select Boundaries 1, 4, and 23 only.



ELECTRIC CURRENTS

1 In the **Model Builder** window, under **Model 1** click **Electric Currents**.

2 In the **Electric Currents** settings window, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Model domain**.

4 Right-click **Model 1>Electric Currents** and choose the boundary condition **Ground**.

Ground 1

1 In the **Ground** settings window, locate the **Boundary Selection** section.

2 From the **Selection** list, choose **Ground boundaries**.

Terminal 1

1 In the **Model Builder** window, right-click **Electric Currents** and choose the boundary condition **Terminal**.

2 In the **Terminal** settings window, locate the **Boundary Selection** section.

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

4 Locate the **Terminal** section. From the **Terminal type** list, choose **Voltage**.

5 In the V_0 edit field, type `10[kV]`.

 ©2012 COMSOL

MATERIALS

Next, assign material properties on the model. Begin by specifying air for all domains.

1 In the **Model Builder** window, under **Model 1** right-click **Materials** and choose **Open Material Browser**.

2 In the **Material Browser** window, locate the **Materials** section.

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

4 Right-click and choose **Add Material to Model** from the menu.

Override the dielectric disk with Glass(quartz).

5 In the **Model Builder** window, right-click **Materials** and choose **Open Material Browser**.

6 In the **Material Browser** window, locate the **Materials** section.

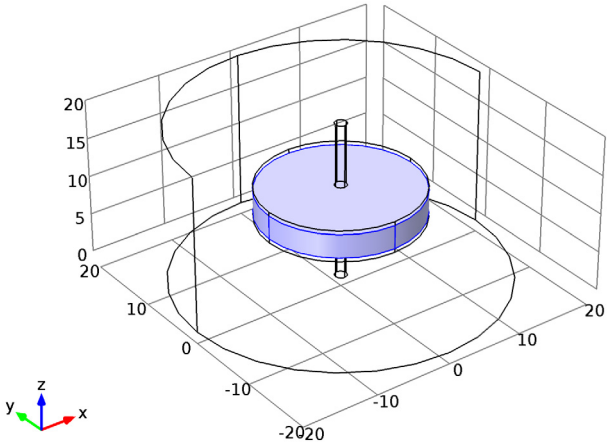
7 In the tree, select **Built-In>Glass (quartz)**.

8 Right-click and choose **Add Material to Model** from the menu.

Glass (quartz)

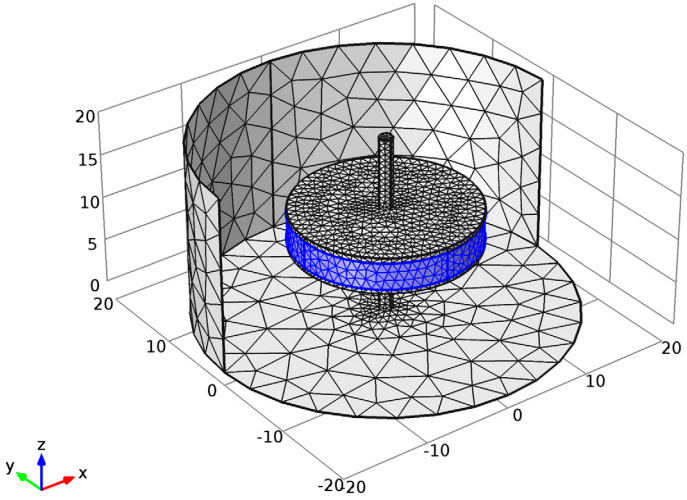
1 In the **Model Builder** window, under **Model 1>Materials** click **Glass (quartz)**.

2 Select Domain 3 only.



MESH 1

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



STUDY 1

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

2 In the **Study** settings window, locate the **Study Settings** section.

3 Clear the **Generate default plots** check box.

Step 1: Frequency Domain

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

2 In the **Frequency Domain** settings window, locate the **Study Settings** section.

3 In the **Frequencies** edit field, type `1 10 100 1000 10000 100000`.

4 In the **Model Builder** window, right-click **Study 1** and choose **Show Default Solver**.

5 Expand the **Study 1>Solver Configurations** node.

Solver 1

1 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solver 1** node, then click **Stationary Solver 1**.

 ©2012 COMSOL

2 In the **Stationary Solver** settings window, locate the **General** section.

3 In the **Relative tolerance** edit field, type `1e-6`.

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

RESULTS

Data Sets

Add a solution and set a selection for the metal part.

1 In the **Model Builder** window, under **Results** right-click **Data Sets** and choose **Solution**.

2 Right-click **Results>Data Sets>Solution 2** and choose **Add Selection**.

3 In the **Selection** settings window, locate the **Geometric Entity Selection** section.

4 From the **Geometric entity level** list, choose **Domain**.

5 From the **Selection** list, choose **Metal**.

6 Select the **Propagate to lower dimensions** check box.

3D Plot Group 1

1 In the **Model Builder** window, right-click **Results** and choose **3D Plot Group**.

The metal parts, not part of the model domain, can be visualized by choosing Uniform for the coloring type though there is no solution. Create a surface plot for it.

2 In the **Model Builder** window, under **Results** right-click **3D Plot Group 1** and choose **Surface**.

3 In the **Surface** settings window, locate the **Data** section.

4 From the **Data set** list, choose **Solution 2**.

5 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

6 From the **Color** list, choose **Gray**.

Add slice plots for the norm of the electric potential and resistive losses.

7 In the **Model Builder** window, right-click **3D Plot Group 1** and choose **Slice**.

8 In the **Slice** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. Locate the **Plane Data** section. In the **Planes** edit field, type `1`.

9 Locate the **Coloring and Style** section. From the **Color table** list, choose **RainbowLight**.

10 In the **Model Builder** window, right-click **3D Plot Group 1** and choose **Slice**.

11 In the **Slice** settings window, locate the **Plane Data** section.

12 From the **Plane** list, choose **xy-planes**.

©2012 COMSOL

13 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Electric Currents>Heating and losses>Resistive losses (ec.Qrh)**.

14 Locate the **Plane Data** section. From the **Plane** list, choose **zx-planes**.

15 In the **Planes** edit field, type `1`.

16 Locate the **Coloring and Style** section. From the **Color table** list, choose **ThermalLight**.

17 Click the **Plot** button.

Compare the resulting plot with Figure 2.

Derived Values

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

2 In the **Global Evaluation** settings window, locate the **Expression** section.

3 In the **Expression** edit field, type `ec.Y11`.

4 Click the **Evaluate** button.

5 In the **Model Builder** window, right-click **Derived Values** and choose **Global Evaluation**.

6 In the **Global Evaluation** settings window, locate the **Expression** section.

7 In the **Expression** edit field, type `imag(ec.Y11)/ec.omega`.

8 Click the **Evaluate** button.

9 In the **Model Builder** window, right-click **Derived Values** and choose **Integration>Volume Integration**.

10 Select Domain 3 only.

11 In the **Volume Integration** settings window, locate the **Expression** section.

12 In the **Expression** edit field, type `ec.Qrh`.

13 Click the **Evaluate** button.

Next, use the direct solver to improve the accuracy of the resistive component of the admittance.

STUDY 1

Solver 1

1 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solver 1>Stationary Solver 1** node.

2 Right-click **Direct** and choose **Enable**.

3 Right-click **Study 1** and choose **Compute**.

 ©2012 COMSOL

RESULTS

Derived Values

1 In the **Settings** window, click **Evaluate**.

2 In the **Model Builder** window, under **Results>Derived Values** click **Global Evaluation 2**.

3 In the **Settings** window, click **Evaluate**.

4 In the **Model Builder** window, under **Results>Derived Values** click **Volume Integration 1**.

5 In the **Settings** window, click **Evaluate**.

 ©2012 COMSOL