Model created in COMSOL Multiphysics 6.4

Computing the Effect of Fringing Fields on Capacitance

Introduction

A typical capacitor is composed of two conductive objects with a dielectric in between. Applying a potential difference between these objects results in an electric field. This electric field exists not just directly between the conductive objects, but extends some distance away, which is a phenomenon known as a fringing field. To accurately predict the capacitance of a capacitor, the domain used to model the fringing field must be sufficiently large, and the appropriate boundary conditions must be used. This example models a parallel plate capacitor in air and studies the size of the air domain. The choice of boundary condition is addressed as well.

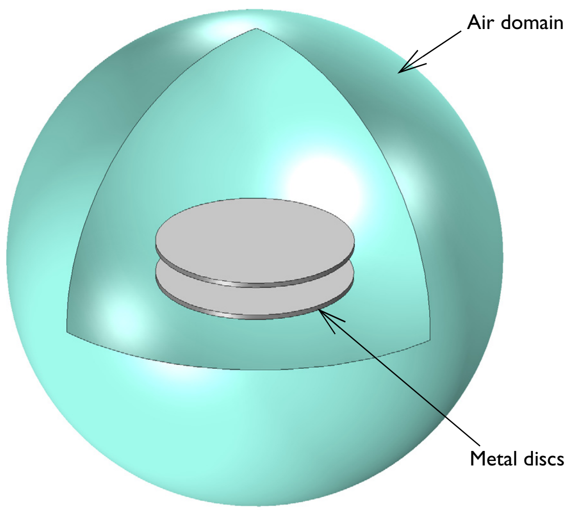


Figure 1: A simple capacitor consisting of two metal discs in an air domain.

Model Definition

Figure 1 shows the capacitor consisting of two metal discs in a spherical volume of air. The size of the sphere truncates the modeling space. This model studies the size of this air domain and its effect on the computed capacitance.

One of the plates is specified as ground, with the electric potential 0 V. The other plate has an applied potential of 1 V.

The air sphere boundary can be thought of as one of two different physical situations: It can be treated as a perfectly insulating surface, across which charge cannot redistribute itself, or as a perfectly conducting surface, over which the potential does not vary.

A perfectly insulating surface is modeled using the Zero Charge boundary condition. This boundary condition implies that the electric field lines are tangential to the boundary.

A perfectly conducting surface can be modeled with the Floating Potential boundary condition. This boundary condition fixes the electric potential on the surface of the sphere to a constant value that is computed during the solution. The boundary condition also implies that the electric field lines are perpendicular to the boundary.

When studying convergence of results with respect to the surrounding domain size, it is important to fix the element size. In this model, the element size is fixed as the domain size is varied.

Results and Discussion

Figure 2 and Figure 3 plot the electric field for the cases where the air sphere boundary is treated as perfectly insulating and perfectly conducting, respectively. The field lines terminate differently on the boundaries of the air sphere.

Figure 4 compares the capacitance values of the device with respect to air sphere radius for the two boundary conditions. The figure also plots the average of the two values. Notice that all three capacitance calculations converge to the same value as the radius grows. In practice, it is often sufficient to model a small air sphere with the electric insulation and floating potential boundary conditions and take the average of the two.

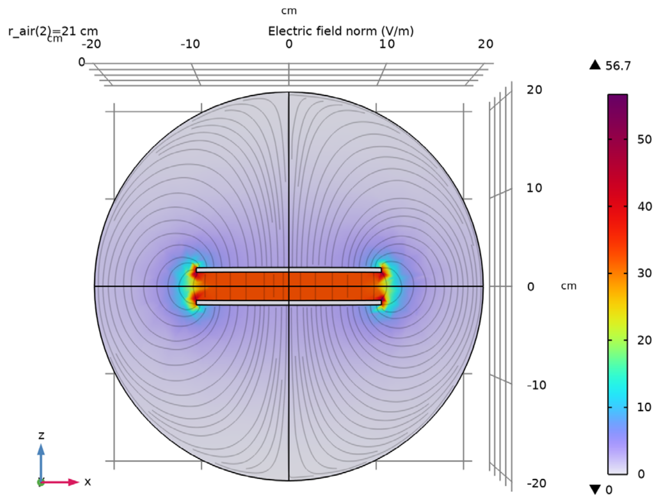


Figure 2: The electric field norm and electric field lines for the case of the Zero Charge boundary condition.

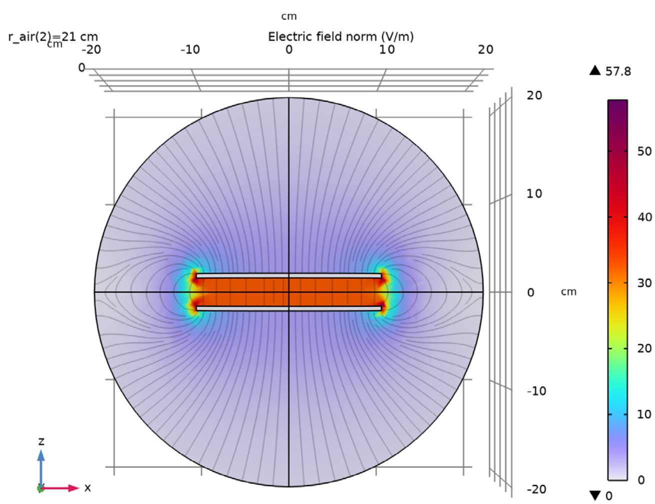


Figure 3: The electric field norm and electric field lines for the case of the Floating Potential boundary condition.

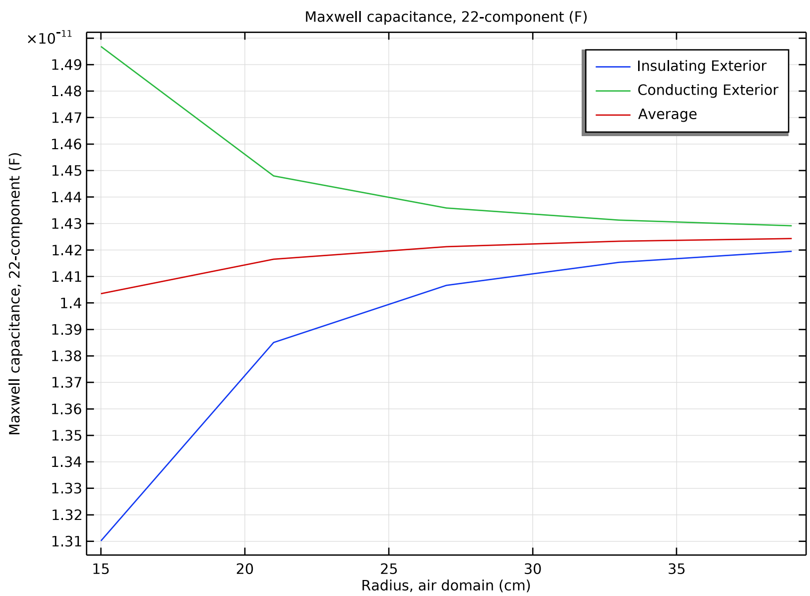


Figure 4: Convergence of the device capacitance as the size of the surrounding air sphere is increased. Electric insulation and fixed potential boundary conditions converge to the same result. The average of the two is also plotted.

Application Library path: `ACDC_Module/Introductory_Electrostatics/capacitor_fringing_fields`

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 **AC/DC** > **Electric Fields and Currents** > **Electrostatics (es)**.

3 Click **Add**.

4 Click **Study**.

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

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 In the table, enter the following settings:

Name	Expression	Value	Description
r_air	15[cm]	0.15 m	Radius, air domain

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

2 In the **Settings** window for **Geometry**, locate the **Units** section.

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

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 `10`.

4 In the **Height** text field, type `0.5`.

5 Locate the **Position** section. In the **z** text field, type `-2`.

6 Click **Build Selected**.

Mirror 1 (mir1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Mirror**.

2 Select the object **cyl1** only.

3 In the **Settings** window for **Mirror**, locate the **Input** section.

4 Select the **Keep input objects** checkbox.

5 Click **Build Selected**.

Sphere 1 (sph1)

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

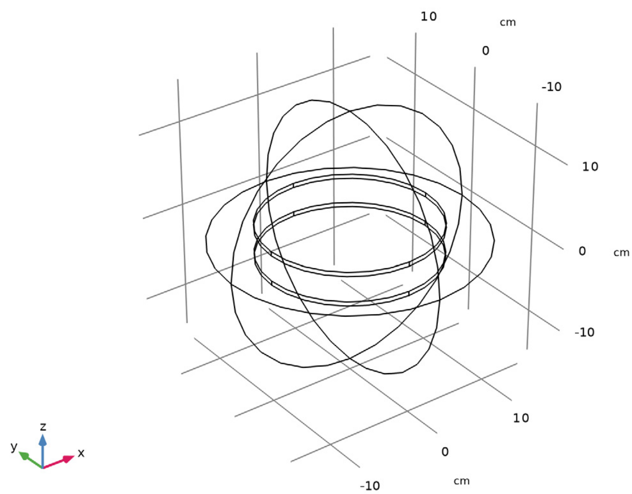
2 In the **Settings** window for **Sphere**, locate the **Size** section.

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

4 Click **Build Selected**.

5 Click the **Wireframe Rendering** button in the **Graphics** toolbar.

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



The geometry describes two metal discs in an air domain.

DEFINITIONS

Create a selection for the exterior boundaries. Later, this will be used for the **Floating Potential** boundary condition.

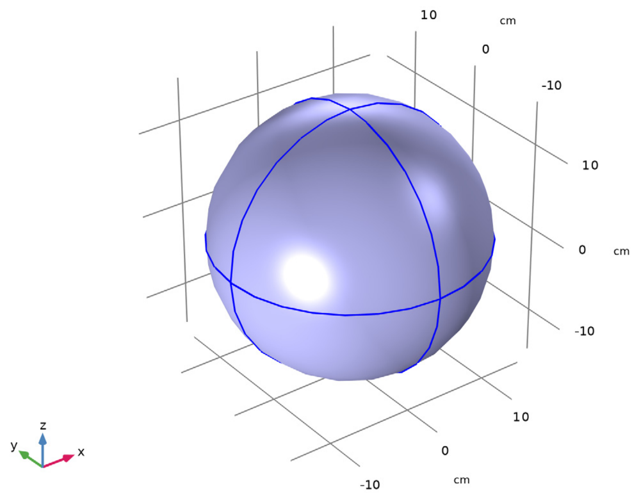
Exterior

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Exterior` in the **Label** text field.

3 Locate the **Input Entities** section. Select the **All domains** checkbox.

4 Locate the **Output Entities** section. From the **Output entities** list, choose **Adjacent boundaries**.



Hide one boundary to get a better view of the interior parts when setting up the physics and reviewing the mesh. Begin by selecting the **Electrostatics** interface, then add a **Hide** node.

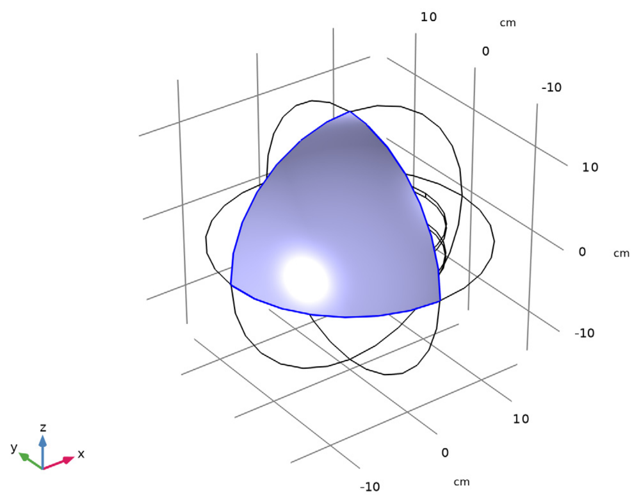
Hide for Physics 1

1 In the **Model Builder** window, right-click **View 1** and choose **Hide for Physics**.

2 In the **Settings** window for **Hide for Physics**, locate the **Geometric Entity Selection** section.

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

4 Select Boundary 2 only.



The default boundary condition is **Zero Charge**, which is applied to all exterior boundaries. Add two **Terminal** features for the electrodes. One connected to the source, and one connected to ground.

ELECTROSTATICS (ES)

Domain Terminal 1

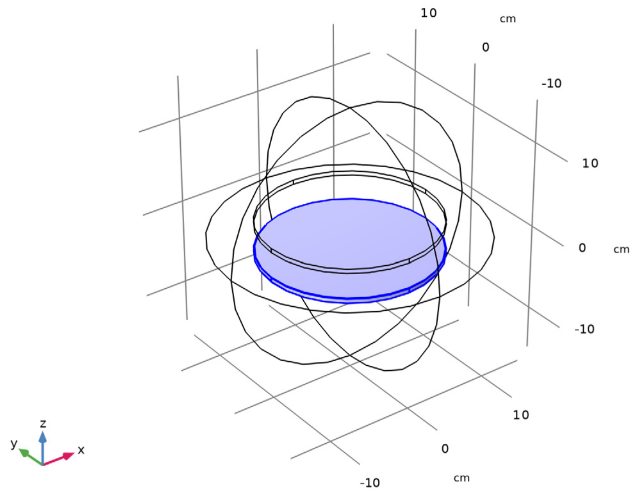
1 In the **Physics** toolbar, click **Domains** and choose **Domain Terminal**.

2 Select Domain 2 only.

3 In the **Settings** window for **Domain Terminal**, locate the **Terminal** section.

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

5 In the V_0 text field, type 0.



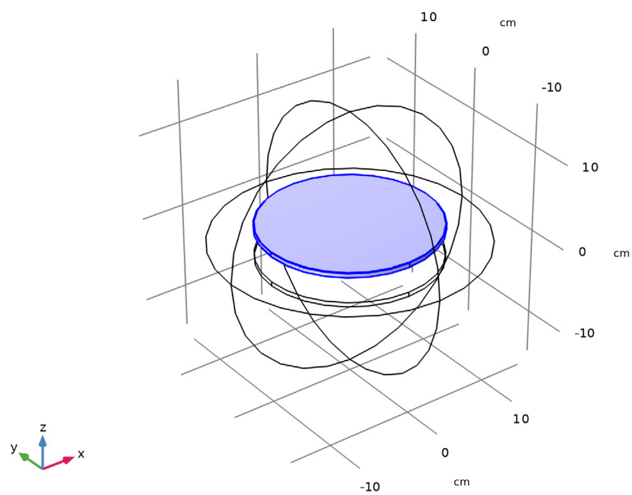
Domain Terminal 2

1 In the **Physics** toolbar, click **Domains** and choose **Domain Terminal**.

2 Select Domain 3 only.

3 In the **Settings** window for **Domain Terminal**, locate the **Terminal** section.

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



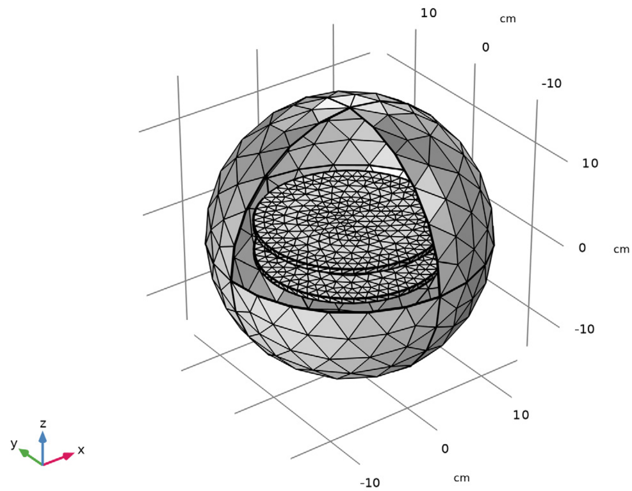
MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 From the **Element size** list, choose **Coarse**.

4 Click **Build All**.



STUDY 1

The stationary study should be solved for different values of the radius of the air domain. Set the range of possible values by using a parametric sweep.

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
r_air (Radius, air domain)	`range(15,6,39)`	`cm`

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

RESULTS

Insulating Exterior

The second default plot shows the electric field. Modify it to get a better view of the field shape in the x,z-plane.

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

2 In the **Settings** window for **3D Plot Group**, type `Insulating Exterior` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter value (r_air (cm))** list, choose **21**.

Multislice 1

1 In the **Model Builder** window, expand the **Insulating Exterior** node, then click **Multislice 1**.

2 In the **Settings** window for **Multislice**, locate the **Multiplane Data** section.

3 Find the **x-planes** subsection. From the **Entry method** list, choose **Number of planes**.

4 In the **Planes** text field, type `0`.

Streamline Multislice 1

1 In the **Model Builder** window, click **Streamline Multislice 1**.

2 In the **Settings** window for **Streamline Multislice**, locate the **Multiplane Data** section.

3 Find the **x-planes** subsection. From the **Entry method** list, choose **Number of planes**.

4 In the **Planes** text field, type `0`.

Transparency 1

1 In the **Model Builder** window, expand the **Streamline Multislice 1** node, then click **Transparency 1**.

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

3 Find the **Transparency** subsection. In the **Transparency** text field, type `0.8`.

4 Click the **Go to XZ View** button in the **Graphics** toolbar.

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

6 In the **Insulating Exterior** toolbar, click **Plot**.

Compare the resulting plot with Figure 2.

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

ELECTROSTATICS (ES)

Next, apply a **Floating Potential** boundary condition to the exterior. This condition overrides the default **Zero Charge** condition.

Floating Potential 1

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

2 In the **Settings** window for **Floating Potential**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Exterior**.

Add a new study to keep the result separate from the previous one.

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** > **Stationary**.

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 2

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
r_air (Radius, air domain)	`range(15,6,39)`	`cm`

5 In the **Model Builder** window, click **Study 2**.

6 In the **Settings** window for **Study**, locate the **Study Settings** section.

7 Clear the **Generate default plots** checkbox.

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

RESULTS

Conducting Exterior

1 In the **Model Builder** window, right-click **Insulating Exterior** and choose **Duplicate**.

2 In the **Settings** window for **3D Plot Group**, type `Conducting Exterior` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/ Parametric Solutions 2 (sol9)**.

4 Click the **Go to XZ View** button in the **Graphics** toolbar.

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

6 In the **Conducting Exterior** toolbar, click **Plot**.

The resulting plot should look like Figure 3.

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

Join 1

1 In the **Results** toolbar, click **More Datasets** and choose **Join**.
2 In the **Settings** window for **Join**, locate the **Data 1** section.
3 From the **Data** list, choose **Study 1/Parametric Solutions 1 (sol2)**.
4 Locate the **Data 2** section. From the **Data** list, choose **Study 2/ Parametric Solutions 2 (sol9)**.
5 Locate the **Combination** section. From the **Method** list, choose **General**.
6 In the **Expression** text field, type `(data1+data2)/2`.

1D Plot Group 4

In the **Results** toolbar, click **1D Plot Group**.

Global 1

1 Right-click **1D Plot Group 4** and choose **Global**.
2 In the **Settings** window for **Global**, 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 **y-Axis Data** section. From the menu, choose **Component 1 (comp1)** > **Electrostatics** > **Terminals** > **Maxwell capacitance - F** > **es.C22 - Maxwell capacitance, 22-component**.
5 Click **Replace Expression** in the upper-right corner of the **x-Axis Data** section. From the menu, choose **Global definitions** > **Parameters** > **r_air - Radius, air domain - m**.
6 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.
7 In the table, enter the following settings:

Legends
`Insulating Exterior`

8 In the **1D Plot Group 4** toolbar, click **Plot**.

Global 2

1 Right-click **Global 1** and choose **Duplicate**.
2 In the **Settings** window for **Global**, locate the **Data** section.
3 From the **Dataset** list, choose **Study 2/Parametric Solutions 2 (sol9)**.
4 Click to expand the **Title** section. From the **Title type** list, choose **None**.

5 Locate the **Legends** section. In the table, enter the following settings:

Legends
Conducting Exterior

6 In the **1D Plot Group 4** toolbar, click **Plot**.

Global 3

1 Right-click **Global 2** and choose **Duplicate**.

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

3 From the **Dataset** list, choose **Join 1**.

4 Locate the **Legends** section. In the table, enter the following settings:

Legends
Average

5 In the **1D Plot Group 4** toolbar, click **Plot**.

This should reproduce Figure 4.

STUDY 1

Optionally, to allow recomputing **Study 1**, you can disable the **Floating Potential** boundary condition for that particular study as follows.

Step 1: Stationary

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

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

3 Select the **Modify model configuration for study step** checkbox.

4 In the tree, select **Component 1 (comp1)** > **Electrostatics (es)** > **Floating Potential 1**.

5 Click **Disable**.