Created in COMSOL Multiphysics 5.3a

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, 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.

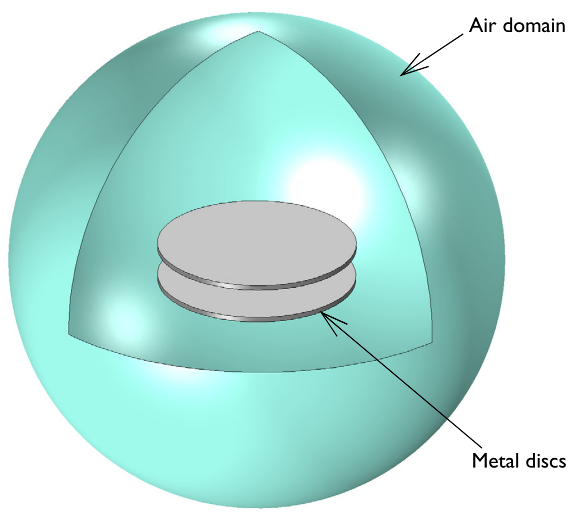


Figure 1: A simple capacitor consisting of two metal discs in an air domain.

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 upon the 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 fields 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.

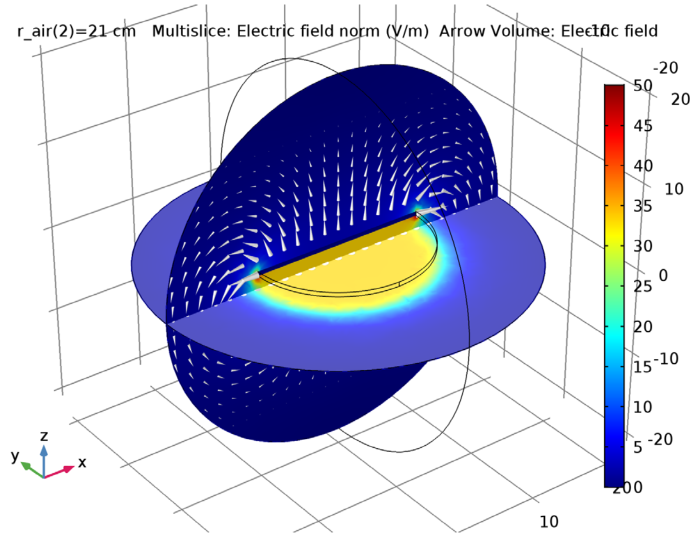


Figure 2: The electric field norm (multislices) and electric field (arrows) for the case of the Zero Charge boundary condition.

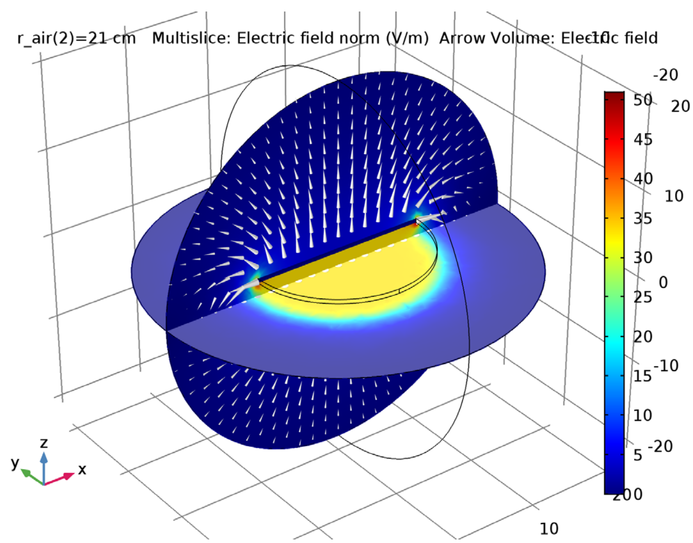


Figure 3: The electric field norm (multislices) and electric field (arrows) for the case of the Floating Potential boundary condition.

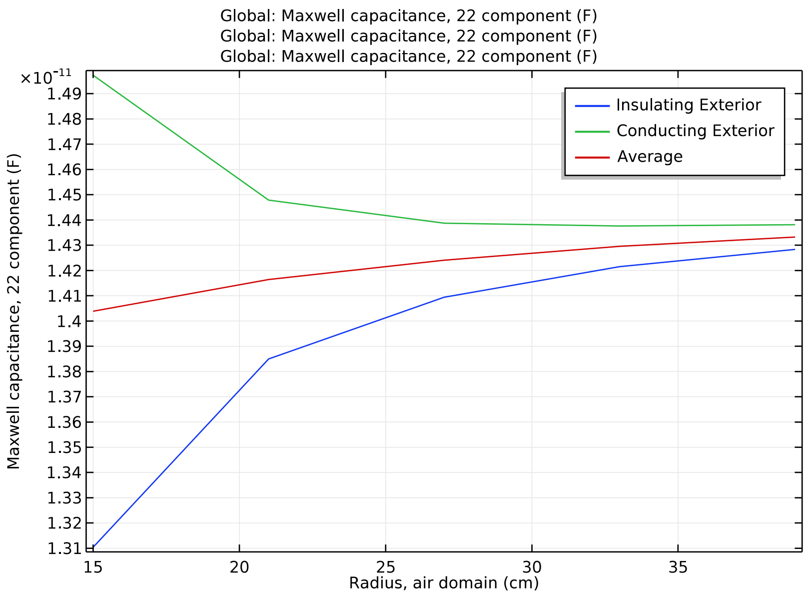


Figure 4: Convergence of the device capacitance as the size of the surrounding air sphere is increased. Electric insulation and fixed potential boundary conditions converge to the same result. The average of the two is also plotted.

Application Library path: `ACDC_Module/Capacitive_Devices/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>Electrostatics (es)**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters

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

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 On 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 On 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** check box.

5 Click **Build Selected**.

Sphere 1 (sph1)

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

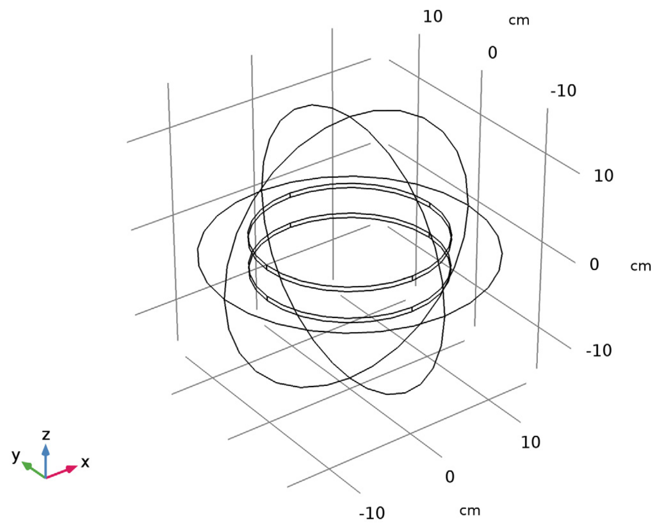
2 In the **Settings** window for **Sphere**, locate the **Size** section.

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

4 Click **Build Selected**.

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

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



The geometry describes two metal discs in an air domain.

Explicit 1

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

On the **Definitions** toolbar, click **Explicit**.

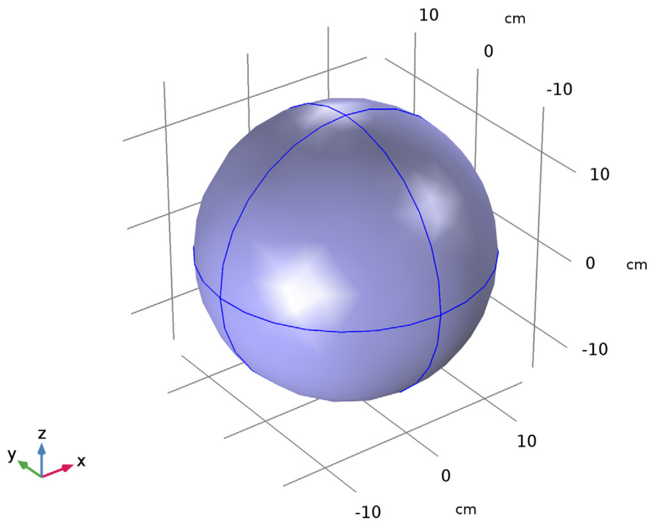
DEFINITIONS

Explicit 1

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

2 Locate the **Input Entities** section. Select the **All domains** check box.

3 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.

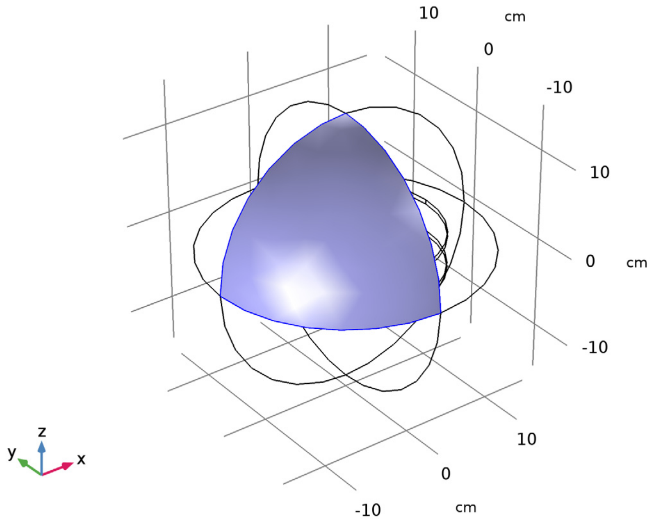
Hide for Physics 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** 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)

Terminal 1

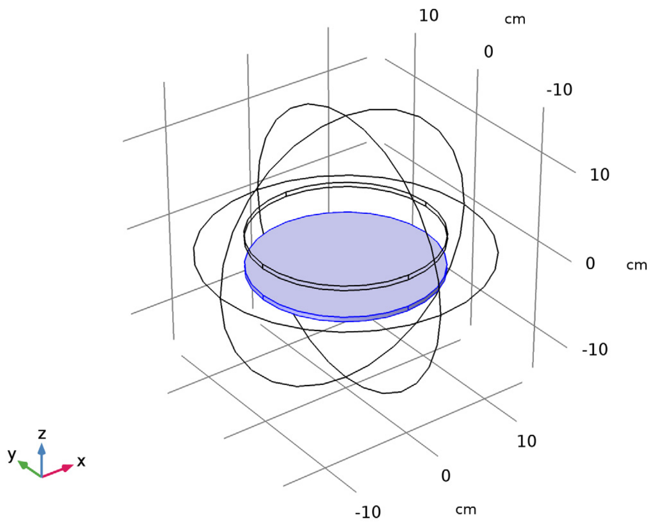
1 On the **Physics** toolbar, click **Domains** and choose **Terminal**.

2 Select Domain 2 only.

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

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

5 In the V_0 text field, type 0.



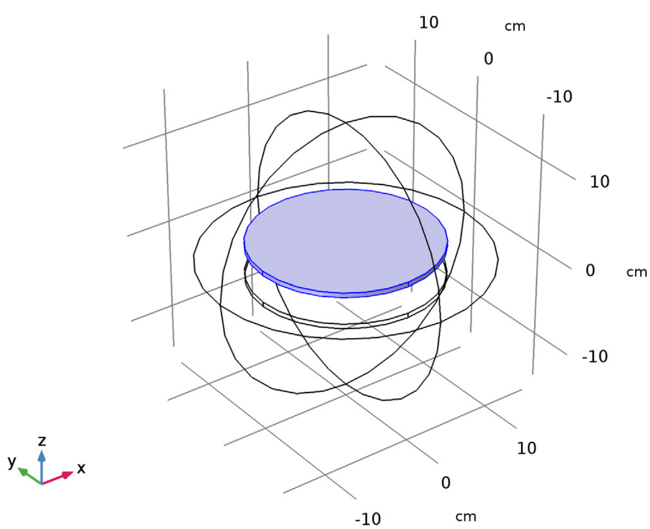
Terminal 2

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

2 Select Domain 3 only.

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

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



Next, assign material properties on the model. Specify **Air** for all domains.

ADD MATERIAL

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

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

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

4 Click **Add to Component 1**.

MATERIALS

Air (mat1)

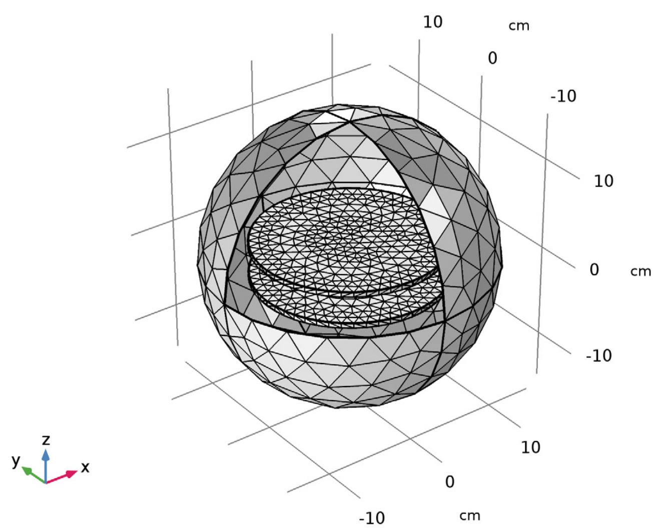
On the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MESH 1

1 In the **Settings** window for **Mesh**, locate the **Mesh Settings** section.

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

3 Click **Build All**.



STUDY 1

Parametric Sweep

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

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

3 Click **Add**.

4 Click to select row number 1 in the table.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
r_air	range(15,6,39)	cm

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

RESULTS

Electric Potential (es)

Modify the default plot to show the electric field norm. Add an arrow plot for the electric field to observe the field direction.

1 In the **Model Builder** window, under **Results** click **Electric Potential (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 **Results>Insulating Exterior** node, then click **Multislice 1**.

2 In the **Settings** window for **Multislice**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Electrostatics>Electric>es.normE - Electric field norm**.

3 Locate the **Multiplane Data** section. Find the **x-planes** subsection. In the **Planes** text field, type `0`.

Arrow Volume 1

1 In the **Model Builder** window, under **Results** right-click **Insulating Exterior** and choose **Arrow Volume**.

2 In the **Settings** window for **Arrow Volume**, locate the **Arrow Positioning** section.

3 Find the **x grid points** subsection. In the **Points** text field, type `25`.

4 Find the **y grid points** subsection. In the **Points** text field, type `1`.

5 Find the **z grid points** subsection. In the **Points** text field, type `25`.

6 Locate the **Coloring and Style** section. From the **Arrow type** list, choose **Cone**.

7 From the **Arrow length** list, choose **Logarithmic**.

8 From the **Color** list, choose **White**.

9 On the **Insulating Exterior** toolbar, click **Plot**.

Compare the resulting plot with Figure 2.

ELECTROSTATICS (ES)

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

Floating Potential 1

1 On 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 from the previous one.

ADD STUDY

1 On 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**.

4 In the **Select Study** tree, select **Preset Studies>Stationary**.

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

6 On the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Parametric Sweep

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

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

3 Click **Add**.

4 Click to select row number 1 in the table.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
r_air	range(15,6,39)	cm

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

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

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

9 On the **Study** toolbar, click **Compute**.

RESULTS

Insulating Exterior 1

1 In the **Model Builder** window, under **Results** 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 **Data set** list, choose **Study 2/ Parametric Solutions 2 (sol9)**.

4 On the **Conducting Exterior** toolbar, click **Plot**.

The resulting plot should look like Figure 3.

Join 1

1 On the **Results** toolbar, click **More Data Sets** 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`.

Global 1

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

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

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

4 From the **Data set** list, choose **Study 1/Parametric Solutions 1 (sol2)**.

5 Click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Model>Component 1>Electrostatics>Terminals>Maxwell capacitance> es.C22 - Maxwell capacitance, 22 component**.

6 Click **Replace Expression** in the upper-right corner of the **x-axis data** section. From the menu, choose **Model>Global Definitions>Parameters>r_air - Radius, air domain**.

7 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

8 In the table, enter the following settings:

Legends
Insulating Exterior

9 On the **1D Plot Group 3** toolbar, click **Plot**.

Global 2

1 Right-click **Results>1D Plot Group 3>Global 1** and choose **Duplicate**.

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

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

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

Legends
Conducting Exterior

5 On the **1D Plot Group 3** toolbar, click **Plot**.

Global 3

1 Right-click **Results>1D Plot Group 3>Global 2** and choose **Duplicate**.

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

3 From the **Data set** list, choose **Join 1**.

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

Legends
Average

5 On the **1D Plot Group 3** 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** check box.

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

5 Click **Disable**.