Transient Modeling of a Capacitor in a Circuit

Introduction

This example solves a transient model of a capacitor in combination with an external electrical circuit. The finite element model of the capacitor is combined with a circuit model of a voltage source and a resistor. A step change in voltage is applied, and the transient current through the capacitor is computed and compared to the analytic result.

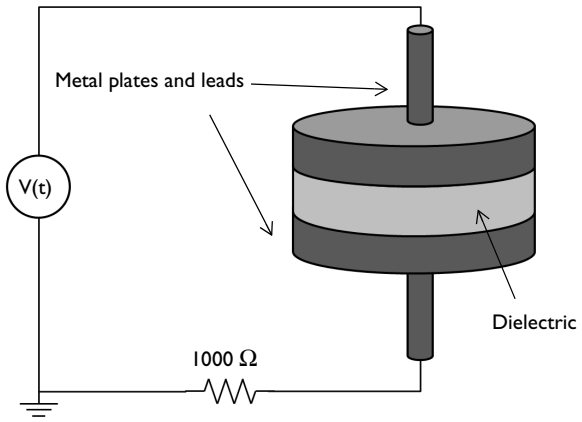


Figure 1: A simple capacitor composed of a disk of dielectric with metal plates on either side and lead wires is connected to an circuit model of a voltage source and a resistor.

Model Definition

The modeled capacitor consists of two metal disks with leads separated by a disk of quartz glass with relative permittivity $\varepsilon_r = 4.2$ and a small electric conductivity $\sigma = 10^{-14}$ S/m (Figure 1). The model also includes a region of surrounding air ($\varepsilon_r = 1.0$, $\sigma = 0$ S/m) to account for the fringing fields. The capacitor is connected to an external circuit composed of a voltage source and a resistor. Initially, the capacitor

is in an equipotential state, with no potential difference between the plates. The voltage across the system is turned on instantaneously, and the potential fields and current through the device is computed.

The air surrounding the capacitor has zero conductivity, which means that only displacement currents can exist. The quartz does have a small electric conductivity, and thus supports both displacement and conduction currents. Because both displacement and conduction currents exist in this model, use the Electric Currents interface.

Assume that the capacitor plates themselves are highly conductive, so that their total effective resistivity is much lower than the external resistor. Under this assumption, it is not necessary to solve for the fields inside of the capacitor plates; you can assume them to be at an equipotential at any instant in time.

A separate electrostatic analysis can be used to compute the capacitance of the device with the result $C = 43.4$ pF. The external resistor has a resistance of $R = 1000\ \Omega$. The analytic solution for the current through a resistor and capacitor in series is

$$I(t) = \frac{V_0}{R}\exp\left(\frac{-t}{RC}\right) \tag{1}$$

where V_0 is the applied voltage.

When solving for a finite element model and a circuit model in combination, it is sometimes necessary to adjust the solver settings. Here, you solve the electric currents problem and the electric circuits problem using a coupled direct solver. This is the most robust solver combination, but also the most memory intensive one.

Results and Discussion

Figure 2 compares the model result for the current through the capacitor as a function of time for a unit change in the applied voltage with the analytic solution. As the figure shows, the agreement is very close.

The displacement and conduction current densities are plotted in Figure 3 and Figure 4 at a point midway between the capacitor plates. While the displacement current density drops off to zero, the induced conduction current density rises in time to a steady-state value. The magnitude of the displacement current density is much higher than the magnitude of the conduction current density, which means that the device has only small leakage currents and losses.

 ©2013 COMSOL

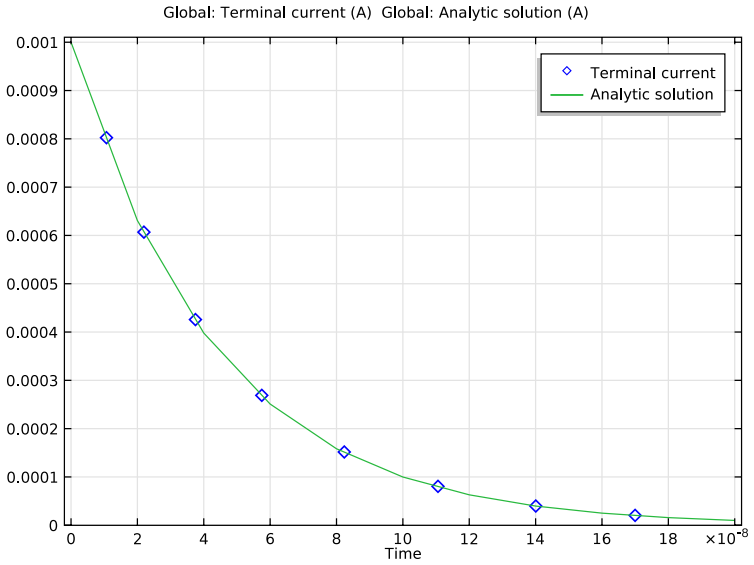


Figure 2: The current through the capacitor after a change in voltage is applied across the system.

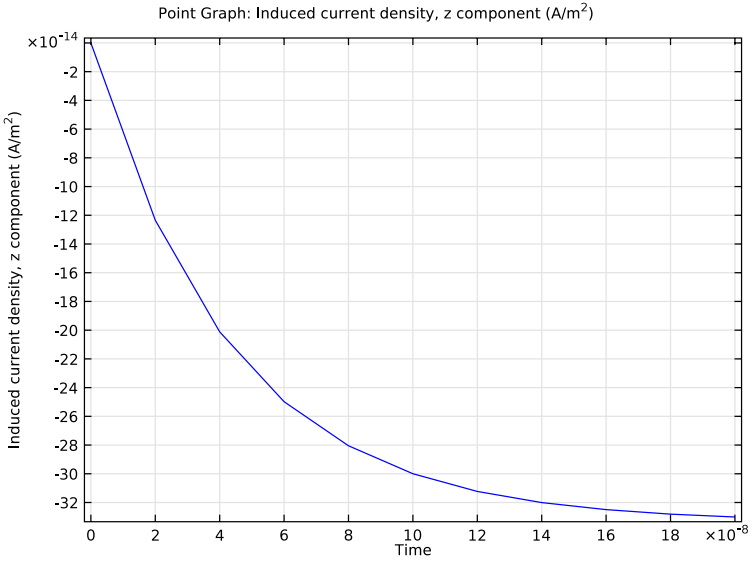


Figure 3: The induced conduction current density in the quartz dielectric.

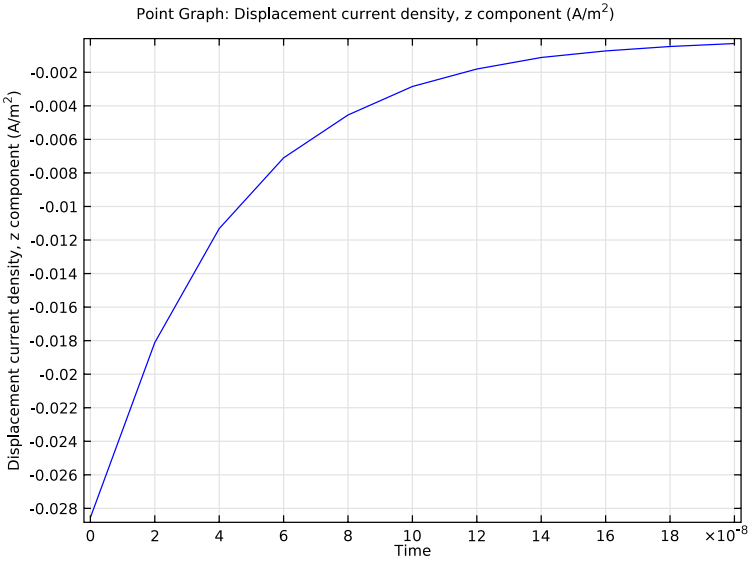


Figure 4: The displacement current density in the quartz dielectric.

Model Library path: `ACDC_Module/Capacitive_Devices/capacitor_transient`

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 In the **Add physics** tree, select **AC/DC>Electrical Circuit (cir)**.

6 Click **Add Selected**.

7 Click **Next**.

 ©2013 COMSOL

8 Find the **Studies** subsection. In the tree, select **Preset Studies for Selected Physics>Time Dependent**.

9 Click **Finish**.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.

2 In the **Parameters** settings window, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Description
R	1000[ohm]	External resistor resistance
C	43.4[pF]	Device capacitance
V0	1[V]	Applied voltage

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.

Next add a cylinder for the disc of dielectric with two metal plates.

Cylinder 2

1 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.

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

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

Cylinder 3

1 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 Right-click **Geometry 1** and choose **Cylinder**.

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

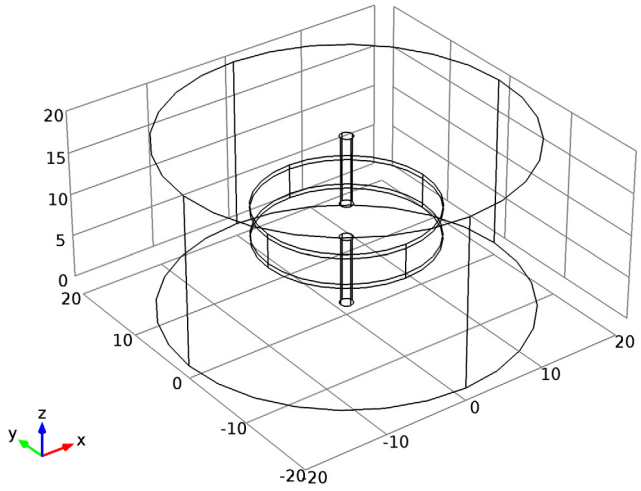
3 In the **Radius** edit field, type `0.75`.

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

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

 ©2013 COMSOL

6 Click the **Build All** button.



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

DEFINITIONS

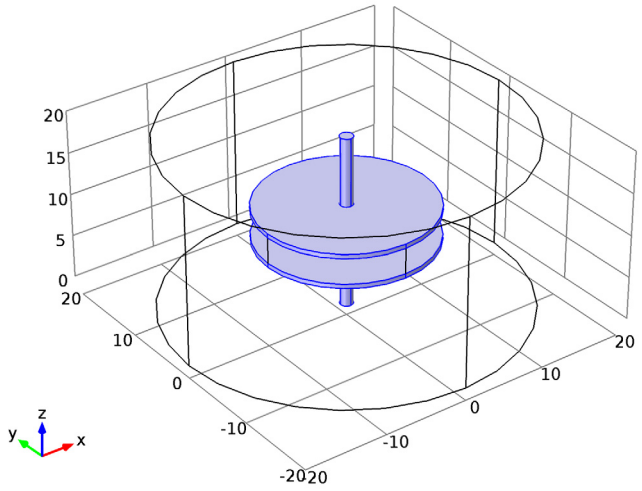
Create a set of selections to use when setting up the physics. First, create a selection for the metal domain.

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.

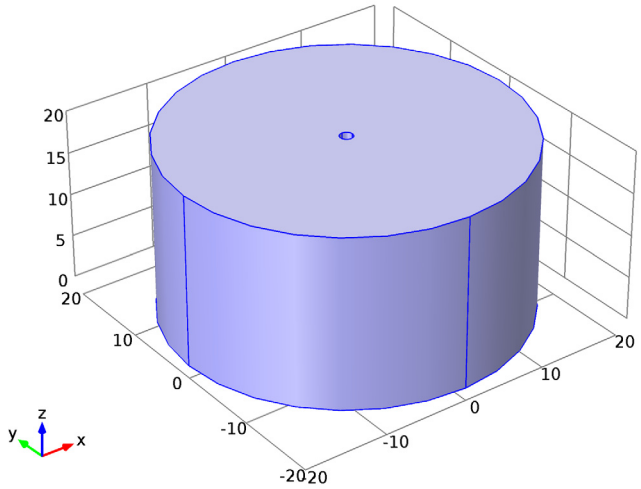
3 Under **Selections to invert**, click **Add**.

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

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

 ©2013 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.

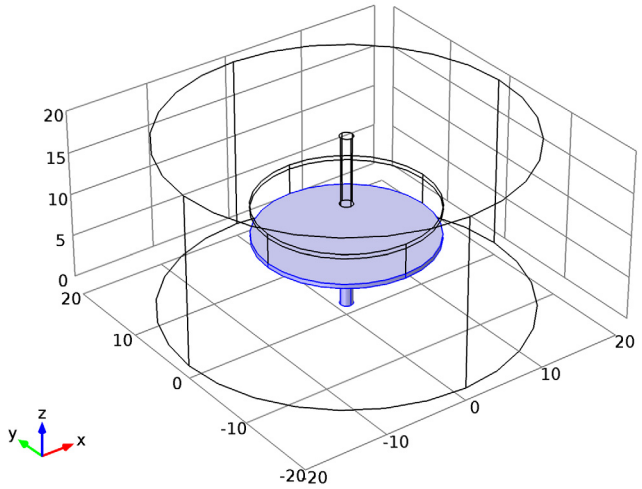
Explicit 2

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

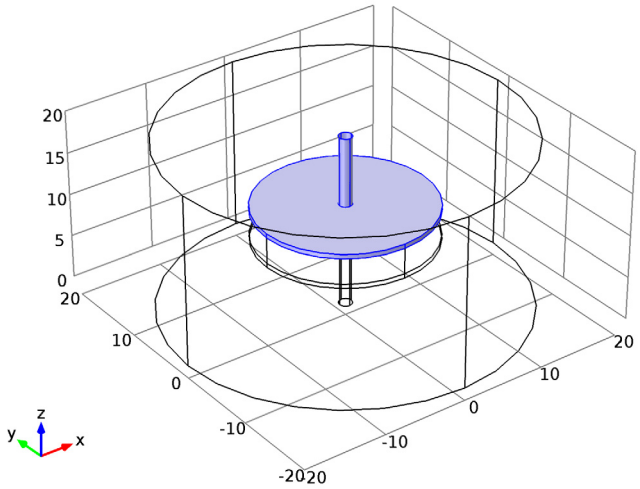
1 Right-click **Definitions** and choose **Selections>Explicit**.

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

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

 ©2013 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

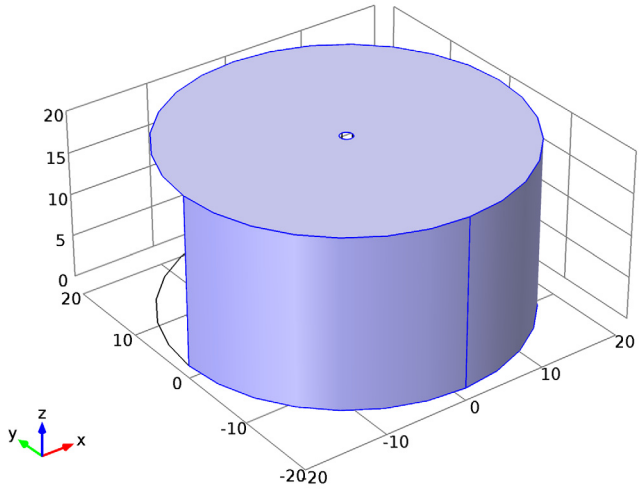
To get a better view, suppress some of the boundaries.

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**.

Ground 1

1 Right-click **Model 1>Electric Currents** and choose **Ground**.

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

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

Terminal 1

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

 ©2013 COMSOL

ELECTRICAL CIRCUIT

Resistor 1

In the **Model Builder** window, under **Model 1** right-click **Electrical Circuit** and choose **Resistor**.

Voltage Source 1

1 In the **Model Builder** window, right-click **Electrical Circuit** and choose **Voltage Source**.

2 In the **Voltage Source** settings window, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	2

External I Vs. U 1

1 Right-click **Electrical Circuit** and choose **External I Vs. U**.

2 In the **External I Vs. U** settings window, locate the **Node Connections** section.

3 In the table, enter the following settings:

Label	Node names
p	2
n	1

4 Locate the **External Device** section. From the *V* list, choose **Terminal voltage (ec/term1)**.

MATERIALS

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

Material Browser

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

2 In the **Material Browser** settings window, In the tree, select **Built-In>Air**.

3 Click **Add Material to Model**.

Override the dielectric disc with glass (quartz).

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

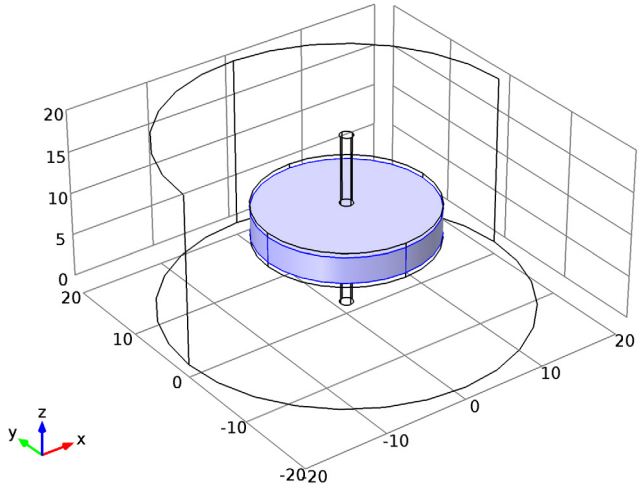
5 In the **Material Browser** settings window, In the tree, select **Built-In>Glass (quartz)**.

6 Click **Add Material to Model**.

Glass (quartz)

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

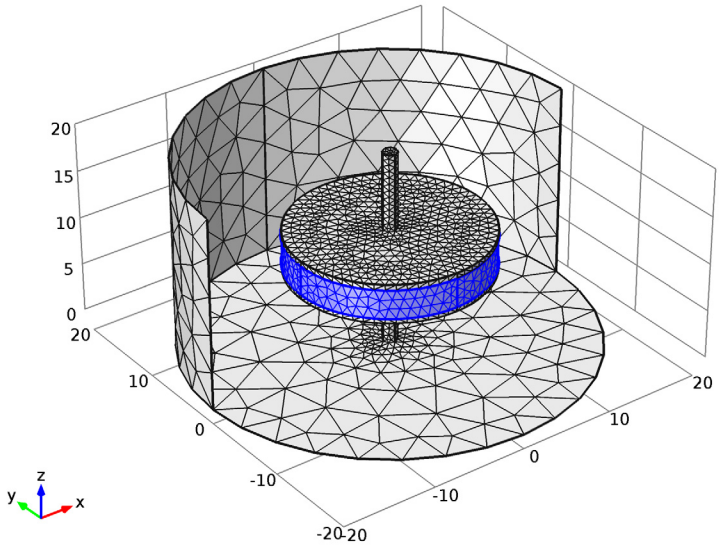
2 Select Domain 3 only.



 ©2013 COMSOL

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: Time Dependent

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

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

3 In the **Times** edit field, type `range(0,2.0e-8,2.0e-7)`.

Solver 1

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

2 Expand the **Solver 1** node.

3 Right-click **Time-Dependent Solver 1** and choose **Fully Coupled**.

4 Right-click **Study 1>Solver Configurations>Solver 1>Time-Dependent Solver 1>Direct** and choose **Enable**.

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

RESULTS

Data Sets

1 In the **Model Builder** window, under **Results** right-click **Data Sets** and choose **Cut Point 3D**.

2 In the **Cut Point 3D** settings window, locate the **Point Data** section.

3 In the **x** edit field, type `0`.

4 In the **y** edit field, type `0`.

5 In the **z** edit field, type `10`.

3D Plot Group 1

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

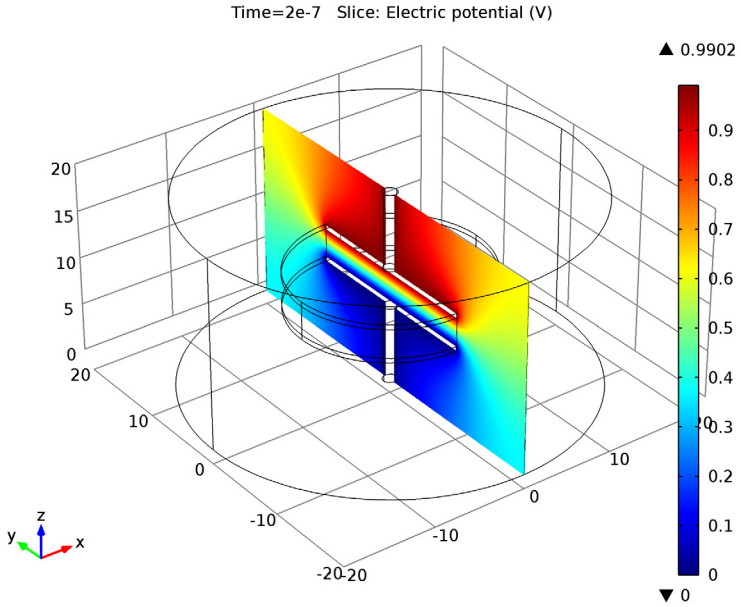
2 Right-click **3D Plot Group 1** and choose **Slice**.

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

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

 ©2013 COMSOL

5 Click the **Plot** button.



1D Plot Group 2

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

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

3 In the **Global** settings window, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
`ec.IO_1`	A	`Terminal current`

4 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **None**.

5 Set the **Width** value to **2**.

6 Find the **Line markers** subsection. From the **Marker** list, choose **Diamond**.

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

8 In the **Global** settings window, locate the **y-Axis Data** section.

9 In the table, enter the following settings:

Expression	Unit	Description
`(V0/R)*exp(-t/(R*C))`	A	Analytic solution

10 Click the **Plot** button.

Compare the resulting plot with Figure 2.

1D Plot Group 3

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

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

3 In the **Point Graph** settings window, locate the **Data** section.

4 From the **Data set** list, choose **Cut Point 3D 1**.

5 Click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Electric Currents>Currents and charge>Induced current density>Induced current density, z component (ec.Jiz)**.

6 Click the **Plot** button.

The reproduced plot should look like Figure 3. Finish the result analysis by reproducing Figure 4.

1D Plot Group 4

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

2 In the **Model Builder** window, expand the **1D Plot Group 4** node, then click **Point Graph 1**.

3 In the **Point Graph** settings window, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Electric Currents>Currents and charge>Displacement current density>Displacement current density, z component (ec.Jdz)**.

4 Click the **Plot** button.

 ©2013 COMSOL