Created in COMSOL Multiphysics 6.1

Contact Impedance Comparison

Introduction

The contact impedance boundary condition is meant to approximate a thin layer of material that impedes the flow of current normal to the boundary, but does not introduce any additional conduction path tangential to the boundary. This example compares the contact impedance boundary condition to a full-fidelity model and discusses the range of applicability of this boundary condition.

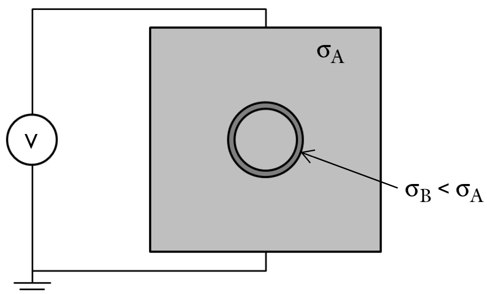


Figure 1: A square two-dimensional domain of conductive material, with a circular inclusion. The wall of the inclusion is made of a material with lower conductivity.

Model Definition

The situation being modeled is shown in Figure 1. A square two-dimensional domain of conductive material has a DC voltage difference applied to it. Within the square domain, there is a circular inclusion. The walls of this inclusion are modeled two ways: using a full fidelity model that includes the thickness of the walls, and using the contact impedance boundary condition. The interior of the inclusion has the same properties as the bulk.

The location of the contact impedance condition is at the centerline, midway between the inner and outer radii of the full fidelity model. Note that, when using the contact impedance boundary condition, the total volume of the surrounding material is slightly larger, since the thickness of the wall is not being explicitly modeled. The conductivity of the wall of the inclusion is varied between a value several orders of magnitude smaller to a value equal to the conductivity of the bulk.

Results and Discussion

The voltage distribution and the electric field strength is plotted in Figure 2 for the case where the electric conductivity is a thousand times smaller in the wall of the inclusion. This case represents a thin walled object that resists current flow through the surface, that is, it presents a high impedance to the voltage source. The field lines can be observed to deform around the object. The solutions agree well for the cases where the conductivity of the contact impedance boundary condition is less than the surrounding medium.

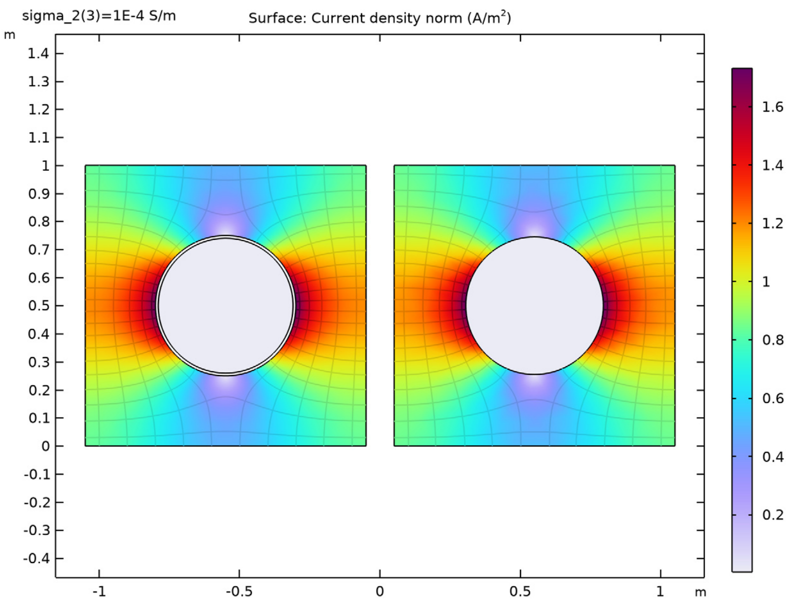


Figure 2: A surface plot of the current density norm, together with isolines of the electric potential and streamlines indicating the current direction, for the case of a thin layer of material that has a high impedance. The full fidelity and contact impedance solutions are almost identical.

The case of equal conductivities is plotted in Figure 3, and shows the limit of the contact impedance boundary condition. As the conductivities becomes equal, there can be no current flow tangential to the boundary.

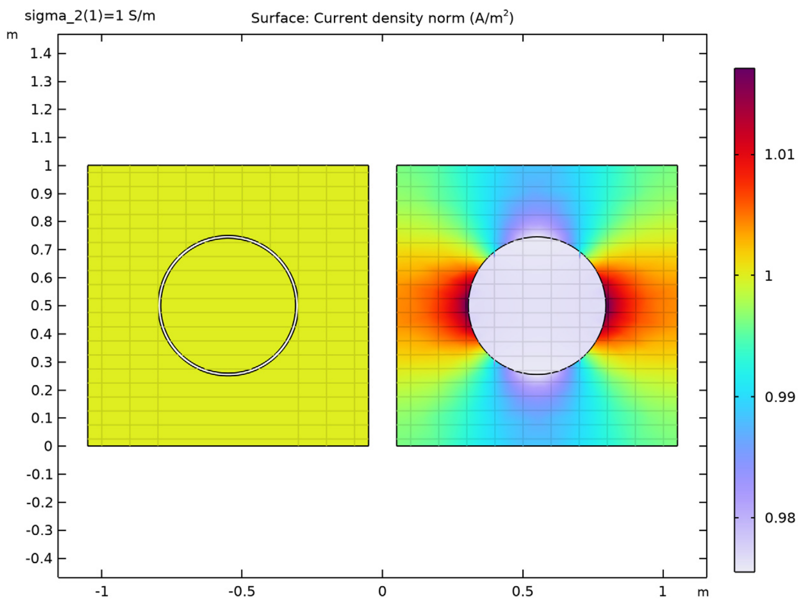


Figure 3: A surface plot of the current density norm, together with isolines of the electric potential and streamlines indicating the current direction. The contact impedance boundary condition (right) prevents any tangential current flow.

The Contact Impedance boundary condition can be used in cases where the thickness of the boundary being approximated is much smaller than the characteristic size of the model domain, and when the conductivity of the layer is smaller than the surrounding medium. When this boundary condition can be used, the resulting number of mesh element is much smaller, saving solution time and memory.

Application Library path: `ACDC_Module/Introductory_Electric_Currents/contact_impedance_comparison`

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 **2D**.

2 In the **Select Physics** tree, select **AC/DC>Electric Fields and Currents>Electric Currents (ec)**.

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
sigma_1	1[S/m]	1 S/m	Conductivity, material 1
sigma_2	1[S/m]	1 S/m	Conductivity, material 2

GEOMETRY 1

Square 1 (sq1)

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

2 In the **Settings** window for **Square**, locate the **Position** section.

3 In the **x** text field, type 0.05.

Square 2 (sq2)

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

2 In the **Settings** window for **Square**, locate the **Position** section.

3 In the **x** text field, type -1.05.

4 Click **Build All Objects**.

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

Circle 1 (c1)

1 In the **Geometry** toolbar, click **Circle**.
2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.
3 In the **Radius** text field, type `0.245`.
4 Locate the **Position** section. In the **x** text field, type `0.55`.
5 In the **y** text field, type `0.5`.

Circle 2 (c2)

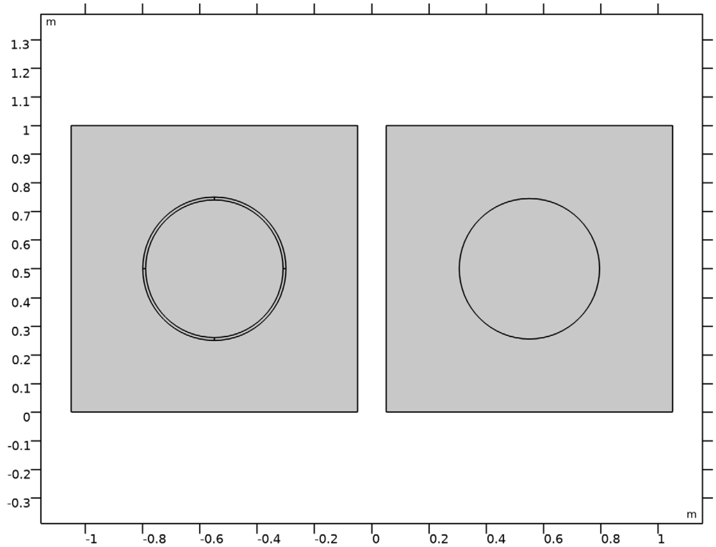
1 In the **Geometry** toolbar, click **Circle**.
2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.
3 In the **Radius** text field, type `0.25`.
4 Locate the **Position** section. In the **x** text field, type `-0.55`.
5 In the **y** text field, type `0.5`.
6 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	0.01

Form Union (fin)

1 In the **Geometry** toolbar, click **Build All**.

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



The geometry on the left side describes the full fidelity model. The geometry on the right side replaces the thin layer with a boundary in order to use the **Contact Impedance** feature.

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

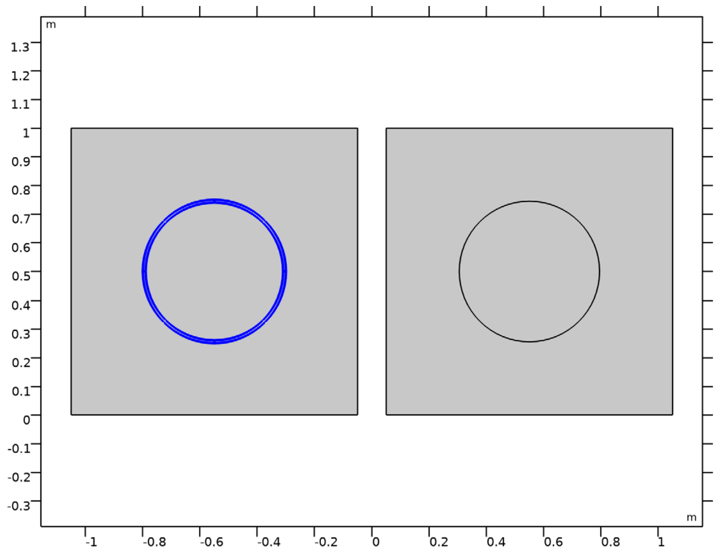
DEFINITIONS

Full Fidelity

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

2 Select Domains 2–5 only.

3 In the **Settings** window for **Explicit**, type `Full Fidelity` in the **Label** text field.



Add a selection for the contact impedance boundaries.

Contact Impedance

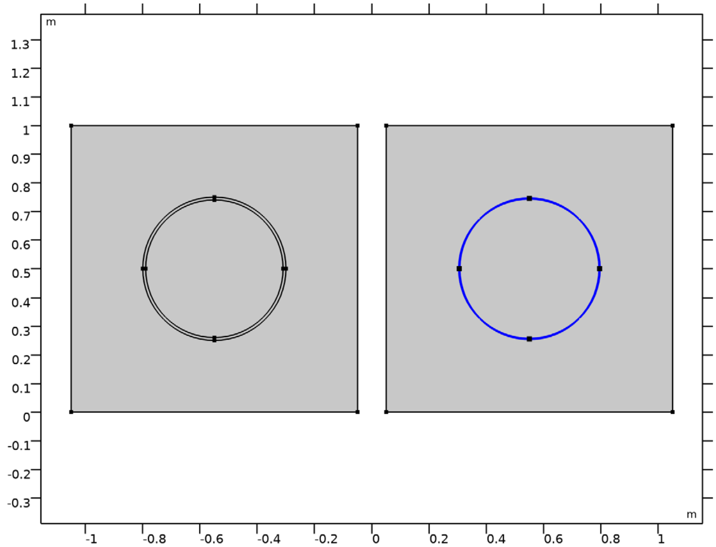
1 In the **Definitions** toolbar, click **Explicit**.

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

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

4 Select Boundaries 21–24 only.

5 In the **Label** text field, type `Contact Impedance`.

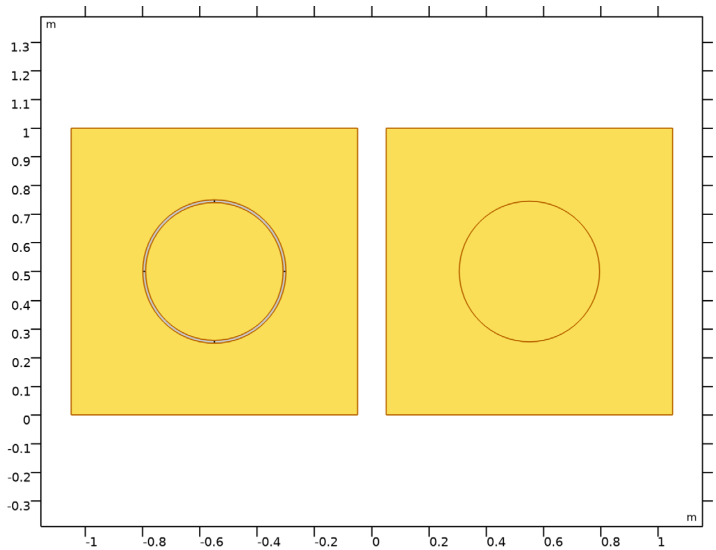


Add a selection for the bulk area. These are the domains complementary to the wall of the inclusion.

Bulk

1. In the **Definitions** toolbar, click **Complement**.
2. In the **Settings** window for **Complement**, locate the **Input Entities** section.
3. Under **Selections to invert**, click **Add**.
4. In the **Add** dialog box, select **Full Fidelity** in the **Selections to invert** list.
5. Click **OK**.

6 In the **Settings** window for **Complement**, type `Bulk` in the **Label** text field.

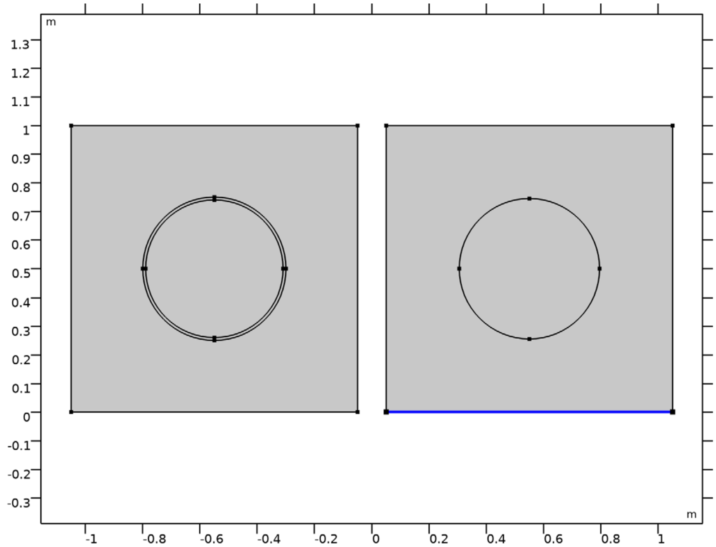


ELECTRIC CURRENTS (EC)

Ground 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Electric Currents (ec)** and choose **Ground**.

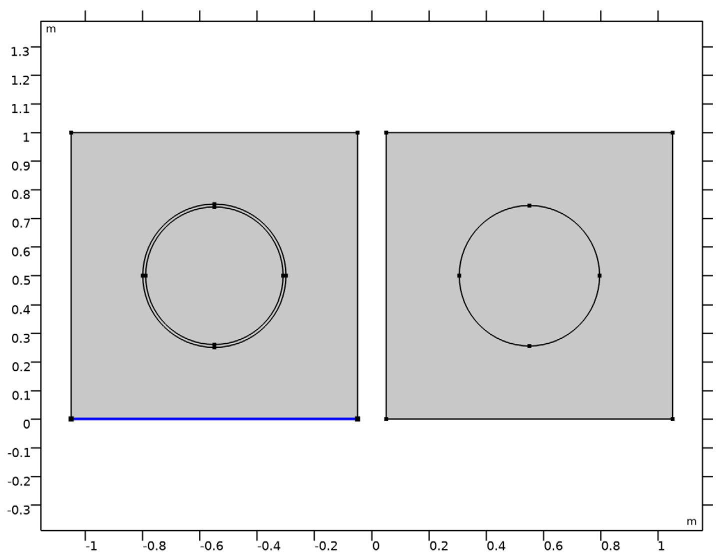
2 Select Boundary 10 only.



Ground 2

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

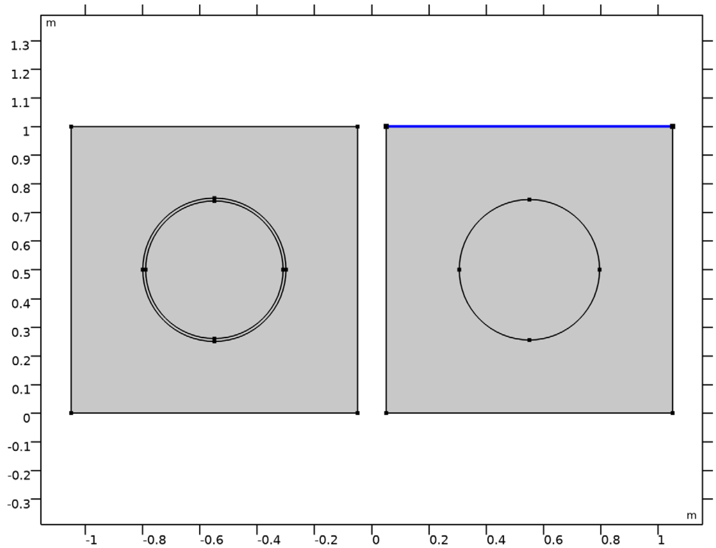
2 Select Boundary 2 only.



Terminal 1

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

2 Select Boundary 11 only.



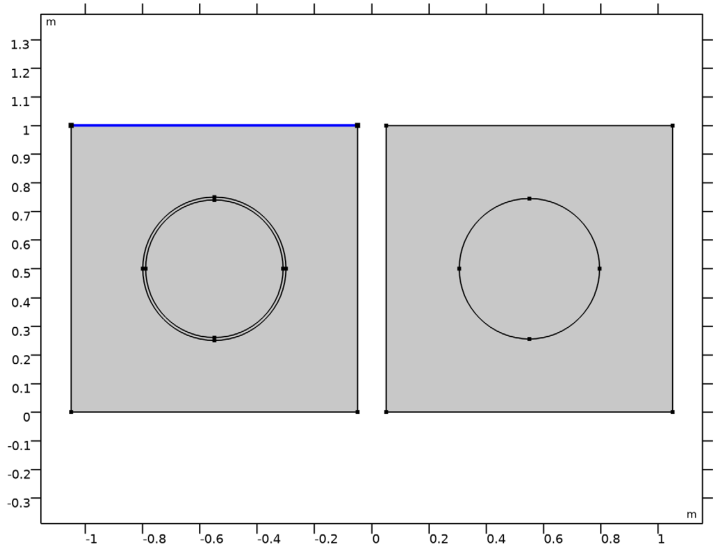
3 In the **Settings** window for **Terminal**, locate the **Terminal** section.

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

Terminal 2

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

2 Select Boundary 3 only.



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

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

Contact Impedance 1

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

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

3 From the **Selection** list, choose **Contact Impedance**.

4 Locate the **Contact Impedance** section. In the d_s text field, type `1[cm]`.

MATERIALS

Material 1 (mat1)

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

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

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

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

Property	Variable	Value	Unit	Property group
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	`sigma_1`	S/m	Basic
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	1	1	Basic

Material 2 (mat2)

1 Right-click **Materials** and choose **Blank Material**.

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

3 From the **Selection** list, choose **Full Fidelity**.

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

Property	Variable	Value	Unit	Property group
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	`sigma_2`	S/m	Basic
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	1	1	Basic

Material 3 (mat3)

1 Right-click **Materials** and choose **Blank Material**.

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

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

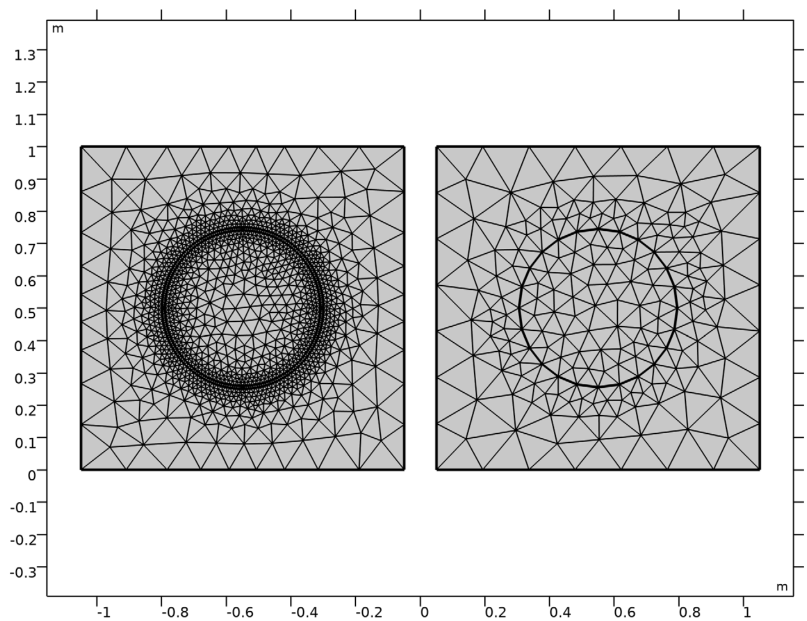
4 From the **Selection** list, choose **Contact Impedance**.

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

Property	**Variable**	**Value**	**Unit**	**Property group**
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	`sigma_2`	S/m	Basic
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	1	1	Basic

MESH 1

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Build All**.



STUDY 1

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
sigma_2 (Conductivity, material 2)	`1 0.01 0.0001`	`S/m`

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

Begin the result analysis by excluding the interior of the wall of the inclusion which is not of interest.

RESULTS

Study 1/Solution 1 (sol1)

In the **Model Builder** window, expand the **Results>Datasets** node, then click **Study 1/ Solution 1 (sol1)**.

Selection

1 In the **Results** toolbar, click **Attributes** and choose **Selection**.

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

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

4 From the **Selection** list, choose **Bulk**.

Create a custom plot to show the direction and norm of the current density.

Current Density (ec)

1 In the **Results** toolbar, click **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Current Density (ec)` in the **Label** text field.

Surface 1

1 Right-click **Current Density (ec)** 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)>Electric Currents> Currents and charge>ec.normJ - Current density norm - A/m²**.

3 Locate the **Coloring and Style** section. Click **Change Color Table**.

4 In the **Color Table** dialog box, select **Rainbow>Prism** in the tree.

5 Click **OK**.

Next, add a contour plot showing the electric potential.

Contour 1

1. In the **Model Builder** window, right-click **Current Density (ec)** and choose **Contour**.
2. In the **Settings** window for **Contour**, click to expand the **Title** section.
3. From the **Title type** list, choose **None**.
4. Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.
5. Clear the **Color** check box.

Color Expression 1

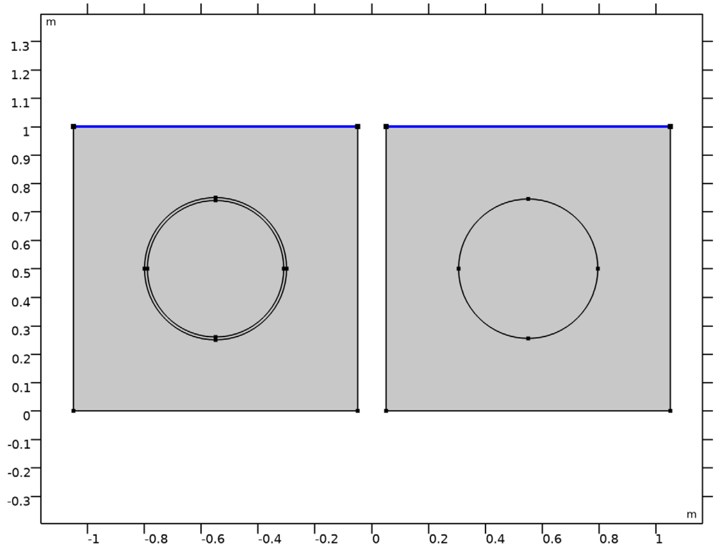
1. Right-click **Contour 1** and choose **Color Expression**.
2. In the **Settings** window for **Color Expression**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **ec.normJ - Current density norm - A/m^2**.
3. Locate the **Coloring and Style** section. Click **Change Color Table**.
4. In the **Color Table** dialog box, select **Rainbow>PrismDark** in the tree.
5. Click **OK**.
6. In the **Settings** window for **Color Expression**, locate the **Coloring and Style** section.
7. Clear the **Color legend** check box.

Then, add a streamline plot showing the current density.

Streamline 1

1. In the **Model Builder** window, right-click **Current Density (ec)** and choose **Streamline**.

2 Select Boundaries 3 and 11 only.



3 In the **Settings** window for **Streamline**, click to expand the **Title** section.

4 From the **Title type** list, choose **None**.

5 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

6 Clear the **Color** check box.

Color Expression 1

In the **Model Builder** window, under **Results>Current Density (ec)>Contour 1** right-click **Color Expression 1** and choose **Copy**.

Color Expression 1

1 In the **Model Builder** window, right-click **Streamline 1** and choose **Paste Color Expression**.

2 In the **Current Density (ec)** toolbar, click **Plot**.

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

Compare the plot with Figure 2.

Current Density (ec)

1 In the **Model Builder** window, under **Results** click **Current Density (ec)**.

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

3 From the **Parameter value (sigma_2 (S/m))** list, choose **1**.

4 In the **Current Density (ec)** toolbar, click **Plot**.

This should look like Figure 3. Note that due to the selection defined in **Solution 1**, the streamlines in the full fidelity model are not completely visualized.