Model created in COMSOL Multiphysics 6.4

Magnetic Field from an Infinite Conductor

Introduction

This introduction model creates a simple model of the magnetostatics problem with a coaxial cable of infinite length carrying current, which is commonly found in textbooks. Since there is an analytical solution to this problem, the model can be used to compare theory with numerical results from the simulation. The region outside of the coaxial cable is not part of this model.

Model Definition

First, consider the analytical solution to this problem, which can be used for comparison. Let r_i and r_o be the radii of the inner and outer conductors, respectively. In this case, the latter is also equivalent to the radius of the modeling domain. In the air region between the two conductors, — that is, for $r_i < r < r_o$ — the magnetic flux density, **B**, satisfies

$$\mathbf{B} = \frac{\mu_0 I_0}{2\pi r}\hat{\varphi}$$

where μ_0 is the vacuum permeability, I_0 is the current through the conductor, and φ is a unit vector in the azimuthal direction. Inside the center conductor, where $r < r_i$, the solution instead reads

$$\mathbf{B} = \frac{\mu_0 J_0 r}{2}\hat{\varphi}$$

where J_0 is the current density. These expressions can, for example, be derived by using the integral formulation of Ampère's law. From the symmetry of the problem, it follows that the magnetic field is oriented purely in the azimuthal direction and that its strength depends only on the radial distance from the conductor.

The geometry of the conductor will be created in two different ways:

- By using a 2D component, where the Magnetic Fields interface solves for the out-of-plane magnetic vector potential. The 2D component then corresponds to the cross section of the infinite conductor.
- By using a 2D axisymmetric component, where the Magnetic and Electric Fields interface solves for the in-plane magnetic vector potential as well as the electric potential. This component, when revolved around the axis, represents the full conductor.

Even though the outer conductor of the coaxial cable is seemingly not part of the model geometry, which only contains the inner conductor and an air domain, the return currents are still present in the model. This is achieved by using the Magnetic Insulation boundary condition, which is the default boundary condition in the Magnetic and Electric Fields physics interface. This condition applies the constraint $\mathbf{n} \times \mathbf{A} = 0$, where $\mathbf{A}$ is the magnetic vector potential. Current can therefore flow along this boundary, and as shown in Figure 5, the current forms a closed loop through the inner conductor.

Results

The magnetic flux density norm in the 2D component is plotted in Figure 1, while the magnetic flux density norm in the 2D axisymmetric component is shown in Figure 2 (symmetry cross section) and Figure 3 (revolved geometry). Figure 4 shows a plot of the magnetic flux density as a function of the radial coordinate for the two numerical solutions as well as for the analytical solution discussed above. There, it can be seen how well the numerical solutions corresponding to the two different approaches agree with the analytical solution. Finally, Figure 5 shows the return currents along the outside of the air domain in the 2D axisymmetric case.

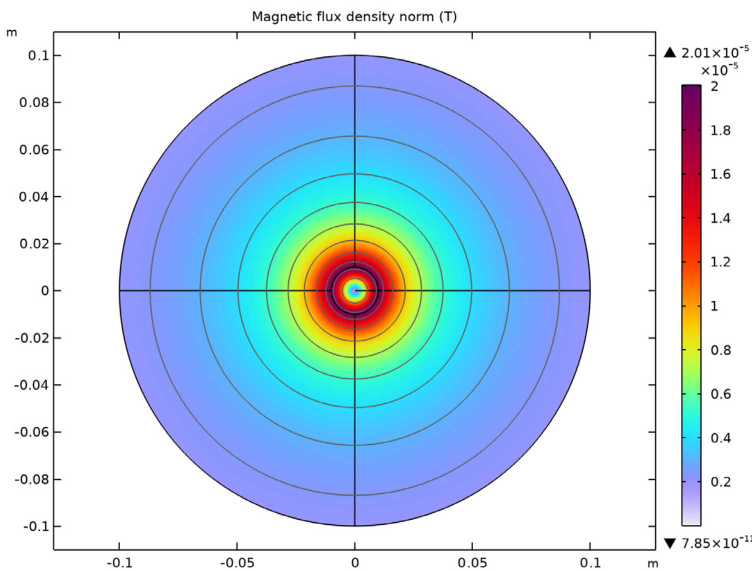

Figure 1: Magnetic flux density norm in the 2D component.

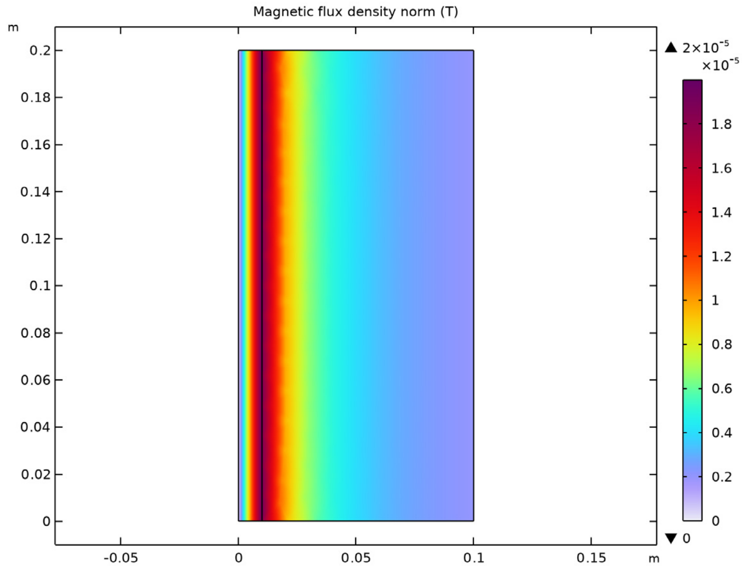

Figure 2: Magnetic flux density norm in the 2D axisymmetric component.

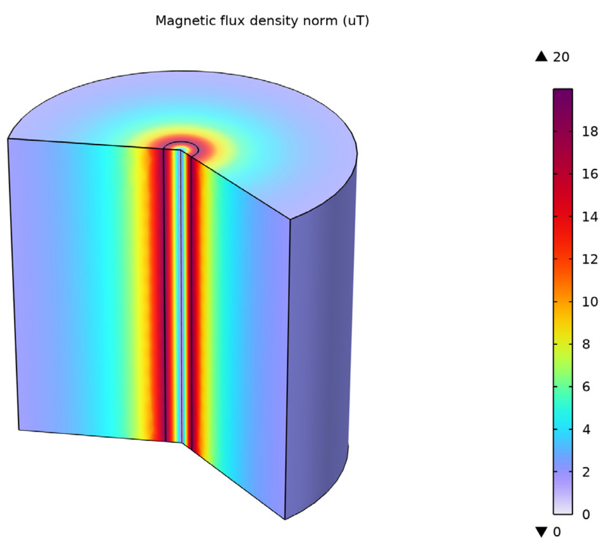

Figure 3: Magnetic flux density norm in the 2D axisymmetric component, shown in the full revolved geometry.

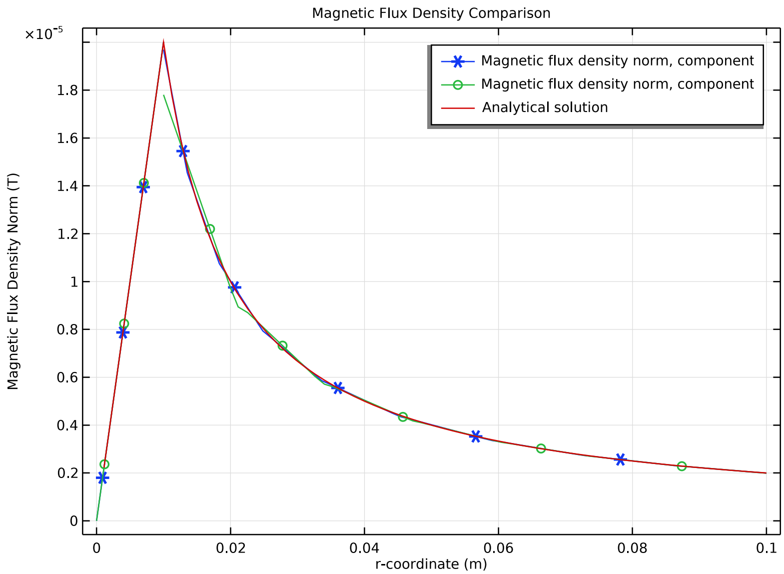

Figure 4: The magnetic flux density norm plotted as a function of the radial coordinate. Here, the numerical solutions from the two different components are shown, along with the analytical solution.

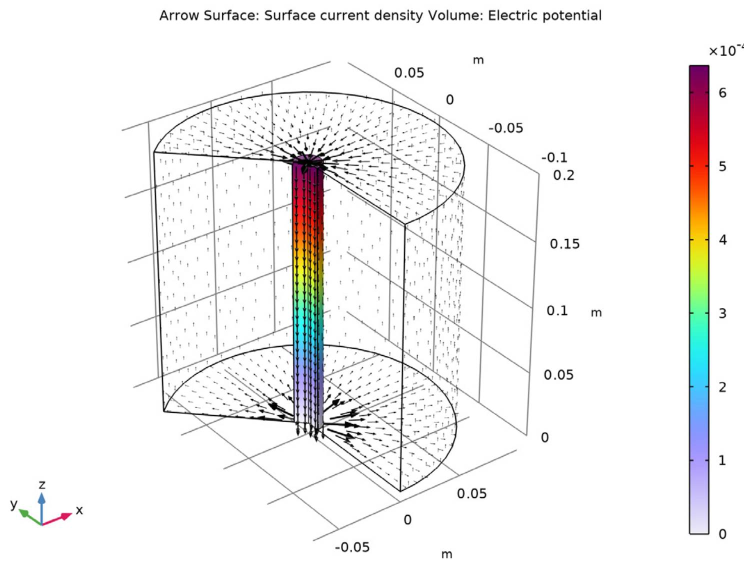

Figure 5: The surface currents plotted on the outside of the modeling domain, along with the currents and the electric potential in the inner conductor.

Application Library path: `ACDC_Module/Introductory_Magnetostatics/magnetic_field_infinite_conductor`

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** > **Electromagnetic Fields** > **Magnetic Fields (mf)**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

First, define some parameters that will be used when building the model.

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
`ri`	`1[cm]`	0.01 m	`Radius of the conductor`
`ro`	`10[cm]`	0.1 m	`Radius of the computation domain`
`I0`	`1[A]`	1 A	`Conducting current`
`J0`	`I0/(pi*ri^2)`	3183.1 A/m²	`Current density in the conductor`

GEOMETRY 1

One way of constructing the geometry is to use a 2D component. Since the Magnetic Fields interface by default only solves for the out-of-plane component of the magnetic vector potential, this is equivalent to having a conductor of infinite length, where the 2D geometry corresponds to the cross section.

Circle 1 (c1)

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Geometry 1** node.

2 Right-click **Geometry 1** and choose **Circle**.

3 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

4 In the **Radius** text field, type `ro`.

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

Layer name	Thickness (m)
`Layer 1`	`ro-ri`

Point 1 (pt1)

In the **Geometry** toolbar, click **Point**.

Line Segment 1 (ls1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 On the object **pt1**, select Point 1 only.

3 In the **Settings** window for **Line Segment**, locate the **Endpoint** section.

4 Click to select the **Activate Selection** toggle button for **End vertex**.

5 On the object **c1**, select Point 7 only.

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

The only modification needed for the physics is to add a current density.

MAGNETIC FIELDS (MF)

External Current Density 1

1 In the **Physics** toolbar, click **Domains** and choose **External Current Density**.

2 Select Domain 4 only.

3 In the **Settings** window for **External Current Density**, locate the **External Current Density** section.

4 Specify the $\mathbf{J}_e$ vector as

0	x
0	y
J0	z

STUDY 1

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

Now, add a second, 2D axisymmetric, component that will be used to illustrate a different approach to modeling the same problem. It will contain an equivalent geometry.

ADD COMPONENT

In the **Model Builder** window, right-click the root node and choose **Add Component** > **2D Axisymmetric**.

GEOMETRY 2

Rectangle 1 (r1)

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

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `ro`.

4 In the **Height** text field, type `2*ro`.

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

Layer name	Thickness (m)
`Layer 1`	`ri`

6 Select the **Layers to the left** checkbox.

7 Clear the **Layers on bottom** checkbox.

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

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

ADD PHYSICS

1 In the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.

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

3 In the tree, select **AC/DC** > **Electromagnetic Fields** > **Vector Formulations** > **Magnetic and Electric Fields (mef)**.

4 Click the **Add to Component 2** button in the window toolbar.

5 In the **Home** toolbar, click **Add Physics** to close the **Add Physics** window.

MAGNETIC AND ELECTRIC FIELDS (MEF)

In the Magnetic and Electric Fields interface, solve for the in-plane magnetic vector potential, and use the Terminal feature to excite the conductor.

1 In the **Settings** window for **Magnetic and Electric Fields**, locate the **Components** section.

2 From the **Field components solved for** list, choose **In-plane vector potential**.

MAGNETIC AND ELECTRIC FIELDS (MEF)

In the **Model Builder** window, expand the **Component 1 (comp1)** > **Magnetic Fields (mf)** > **Magnetic Insulation 1** node, then click **Component 2 (comp2)** > **Magnetic and Electric Fields (mef)** > **Magnetic Insulation 1**.

Boundary Terminal 1

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

2 Select Boundary 3 only.

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

4 In the I_0 text field, type `I0`.

Ampère's Law and Current Conservation in Solids 1

1. In the **Physics** toolbar, click **Domains** and choose **Ampère's Law and Current Conservation in Solids**.
2. Select Domain 1 only.

Finally, add an Ampère's law feature to the air domain, since it is not necessary to solve for the electric potential there. Also, add a Gauge Fixing for A-field feature. This was not needed in the component where only the out-of-plane vector potential component is solved for, since that setting acts as an inherent gauge fix.

Ampère's Law 1

1. In the **Physics** toolbar, click **Domains** and choose **Ampère's Law**.
2. Select Domain 2 only.

Gauge Fixing for A-Field 1

In the **Physics** toolbar, click **Domains** and choose **Gauge Fixing for A-Field**.

Now, add another stationary study for the second component, and make sure that the two studies only solve for their respective physics interfaces.

STUDY 1

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. In the **Solve for** column of the table, under **Component 2 (comp2)**, clear the checkbox for **Magnetic and Electric Fields (mef)**.

ADD MATERIAL

1. In the **Materials** 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 the **Add to Component** button in the window toolbar.
5. In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Conductor

1. In the **Model Builder** window, under **Component 2 (comp2)** > **Materials** right-click **Air (mat1)** and choose **Duplicate**.

2 In the **Settings** window for **Material**, type `Conductor` in the **Label** text field.

3 Select Domain 1 only.

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

Property	**Variable**	**Value**	**Unit**	**Property group**
Electric conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	1e6	S/m	Basic

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

Step 1: Stationary

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

2 In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkbox for **Magnetic Fields (mf)**.

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

Now, it is time to compare the results with the known analytical solutions. Add plots of the magnetic flux density computed from each component, as well as a plot containing the known analytical solution to the problem. Note that the solution is different in the regions inside and outside of the conductor.

RESULTS

Magnetic Flux Density Comparison

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

2 In the **Settings** window for **1D Plot Group**, type `Magnetic Flux Density Comparison` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **Label**.

4 Locate the **Data** section. From the **Dataset** list, choose **None**.

Line Graph 1

1 Right-click **Magnetic Flux Density Comparison** and choose **Line Graph**.

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

3 From the **Dataset** list, choose **Study 1/Solution 1 (sol1)**.

4 Select Boundaries 3 and 5 only.

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type `x`.

7 Click to expand the **Legends** section. Select the **Show legends** checkbox.

8 Find the **Include** subsection. Clear the **Solution** checkbox.

9 Select the **Description** checkbox.

10 Find the **Prefix and suffix** subsection. In the **Suffix** text field, type `, component 1`.

11 Click to expand the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

12 From the **Positioning** list, choose **Interpolated**.

Magnetic Flux Density Comparison

1 In the **Model Builder** window, click **Magnetic Flux Density Comparison**.

2 In the **Settings** window for **1D Plot Group**, locate the **Plot Settings** section.

3 Select the **x-axis label** checkbox. In the associated text field, type `r-coordinate (m)`.

4 Select the **y-axis label** checkbox. In the associated text field, type `Magnetic Flux Density Norm (T)`.

Line Graph 2

1 Right-click **Magnetic Flux Density Comparison** and choose **Line Graph**.

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

3 From the **Dataset** list, choose **Study 2/Solution 2 (3) (sol2)**.

4 Select Boundaries 2 and 5 only.

5 Locate the **y-Axis Data** section. In the **Expression** text field, type `mef.normB`.

6 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

7 In the **Expression** text field, type `r`.

8 Locate the **Legends** section. Select the **Show legends** checkbox.

9 Find the **Include** subsection. Clear the **Solution** checkbox.

10 Select the **Description** checkbox.

11 Find the **Prefix and suffix** subsection. In the **Suffix** text field, type `, component 2`.

12 Locate the **Coloring and Style** section. Find the **Line markers** subsection. From the **Marker** list, choose **Cycle**.

13 From the **Positioning** list, choose **Interpolated**.

Line Graph 3

1 Right-click **Magnetic Flux Density Comparison** and choose **Line Graph**.

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

3 From the **Dataset** list, choose **Study 2/Solution 2 (3) (sol2)**.

4 Select Boundaries 2 and 5 only.

5 Locate the **y-Axis Data** section. In the **Expression** text field, type `mu0_const*J0/2*r*(r<=ri)+mu0_const*I0/(2*pi*r)*(r>ri)`.

6 Locate the **Legends** section. Select the **Show legends** checkbox.

7 Find the **Include** subsection. Clear the **Solution** checkbox.

8 Select the **Expression** checkbox.

9 From the **Legends** list, choose **Manual**.

10 In the table, enter the following settings:

Legends
`Analytical solution`

11 In the **Magnetic Flux Density Comparison** toolbar, click **Plot**.

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

As the last step, create a plot that illustrates the return currents on the outside of the air domain.

Air Domain Return Current (mef)

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Air Domain Return Current (mef)` in the **Label** text field.

Arrow Surface 1

1 Right-click **Air Domain Return Current (mef)** and choose **Arrow Surface**.

2 In the **Settings** window for **Arrow Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 2 (comp2)** >

Magnetic and Electric Fields > Currents and charge > mef.Jsr,...,mef.Jsz - Surface current density.

3 Locate the **Arrow Positioning** section. In the **Number of arrows** text field, type `1500`.

4 Locate the **Coloring and Style** section.

5 Select the **Scale factor** checkbox. In the associated text field, type `0.002`.

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

7 In the **Air Domain Return Current (mef)** toolbar, click **Plot**.

Arrow Surface 2

1 In the **Model Builder** window, right-click **Air Domain Return Current (mef)** and choose **Arrow Surface**.

2 In the **Settings** window for **Arrow Surface**, locate the **Expression** section.

3 In the **r-component** text field, type `mef.Jr`.

4 In the **phi-component** text field, type `mef.Jphi`.

5 In the **z-component** text field, type `mef.Jz`.

6 Locate the **Arrow Positioning** section. In the **Number of arrows** text field, type `300`.

7 Locate the **Coloring and Style** section.

8 Select the **Scale factor** checkbox. In the associated text field, type `4e-6`.

9 From the **Color** list, choose **Black**.

10 Click to expand the **Title** section. From the **Title type** list, choose **None**.

Selection 1

1 Right-click **Arrow Surface 2** and choose **Selection**.

2 Select Domain 1 only.

3 In the **Air Domain Return Current (mef)** toolbar, click **Plot**.

Volume 1

1 In the **Model Builder** window, right-click **Air Domain Return Current (mef)** and choose **Volume**.

2 In the **Settings** window for **Volume**, click to expand the **Title** section.

3 From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Volume: Electric potential`.

5 Locate the **Coloring and Style** section. From the **Color table** list, choose **Prism**.

6 In the **Air Domain Return Current (mef)** toolbar, click **Plot**.