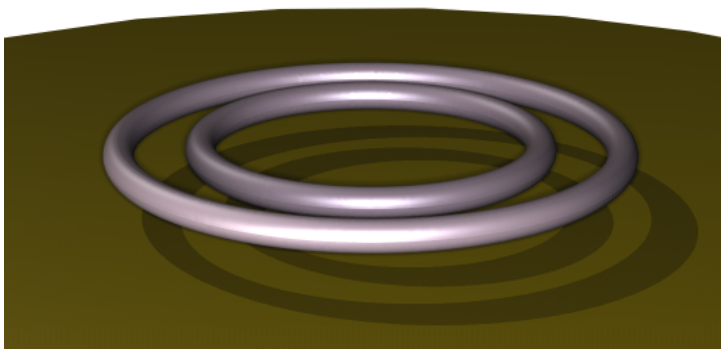
Induction Currents from Circular Coils

Introduction

A time-varying current induces a varying magnetic field. This field induces currents in neighboring conductors. The induced currents are called eddy currents. The following model illustrates this phenomenon by a time-harmonic field simulation as well as a transient analysis, which provides a study of the eddy currents resulting from switching on the source.



Two current-carrying coils are placed above a copper plate. They are surrounded by air, and there is a small air gap between the coils and the metal plate. A potential difference provides the external source. To obtain the total current density in the coils you must take the induced currents into account. The time-harmonic case shows the skin effect, that is, that the current density is high close to the surface and decreases rapidly inside the conductor.

Model Definition

EQUATION

To solve the problem, use a quasi-static equation for the magnetic potential **A**:

$$\sigma\frac{\partial \mathbf{A}}{\partial t} + \nabla \times (\mu_0^{-1}\mu_r^{-1}\nabla \times \mathbf{A}) = \sigma\frac{V_{\mathrm{coil}}}{2\pi r}$$

Here μ_0 is the permeability of vacuum, μ_r the relative permeability, σ the electric conductivity, and V_{coil} the voltage over one turn in the coil. In the time-harmonic case the equation reduces to

$$j\omega\sigma\mathbf{A} + \nabla \times (\mu_0^{-1}\mu_r^{-1}\nabla \times \mathbf{A}) = \sigma\frac{V_{\mathrm{coil}}}{2\pi r}$$

FORCES

The total electromagnetic force acting on region of space Ω can be obtained by integrating Maxwell's stress tensor on the delimiting boundary $\partial\Omega$:

$$\mathbf{F} = \int_{\partial\Omega} T\mathbf{n}\,dS$$

The Force Calculation feature automatically performs the integral along the boundaries of the desired region, considering also the axisymmetric geometry of the problem. The computed force will be available in results processing as a global variable.

Results and Discussion

In the time-harmonic regime, the varying magnetic field induces electrical currents in the metallic plate. The currents, in turn, act as sources of an opposing magnetic field "shielding" the plate from the magnetic field. As a result of this phenomenon, the region in which electrical currents are generated is confined in proximity of the surface and reduces in size with increasing frequency. Figure 1 and Figure 2 show the induced current density at 10 Hz and 300 Hz, respectively.

In this model, a time-domain study is performed to investigate the step response of the system. Figure 3 displays a snapshot of the induced current density and magnetic flux density for the transient solution in a combined surface and arrow plot.

Finally, Figure 4 shows the total axial force between the coils and the plate as a function of time computed by the Force Calculation feature. For the chosen current direction, the force is repulsive (negative).

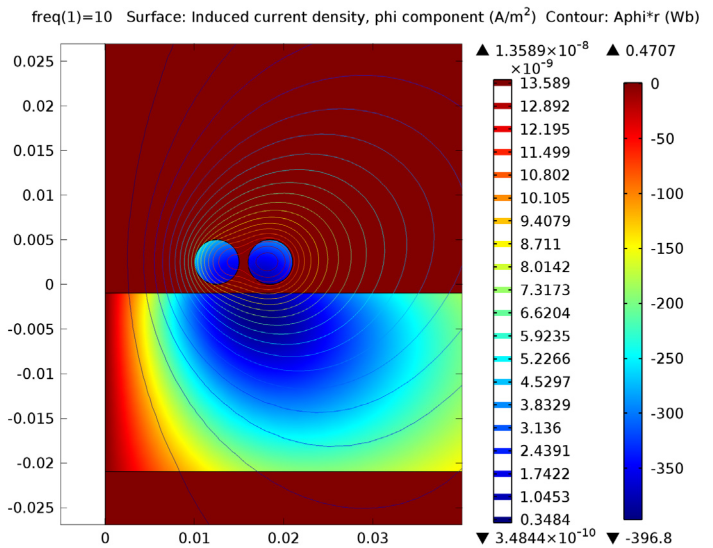


Figure 1: The φ component of the induced current density for the time-harmonic solution plotted together with a contour plot of the magnetic vector potential at a frequency of 10 Hz.

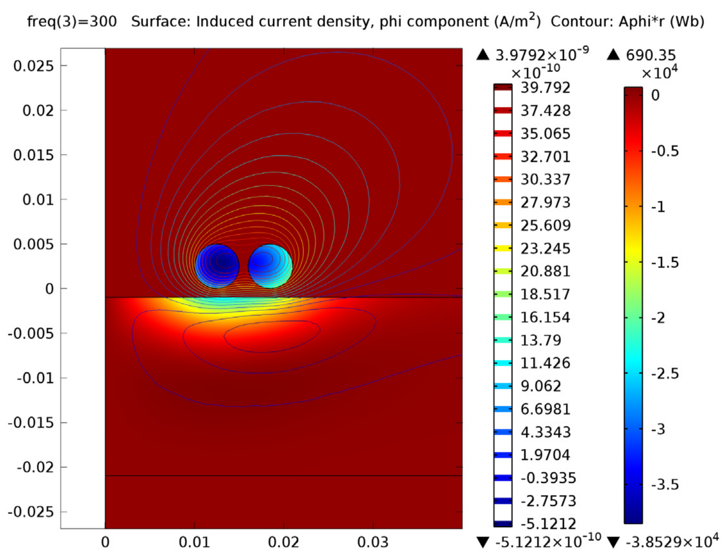


Figure 2: Plot of the same quantities at a frequency of 300 Hz.

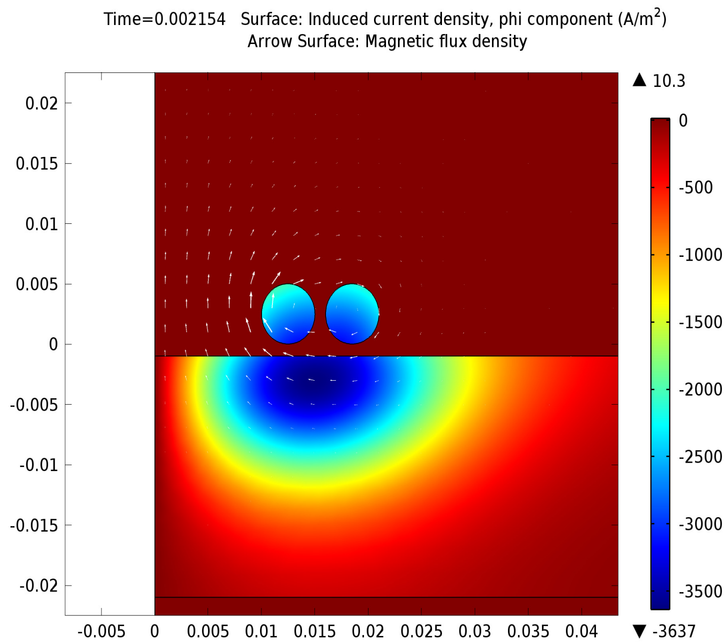


Figure 3: Snapshot of the induced current density (surface plot) and the magnetic flux density (arrow plot) during the transient study.

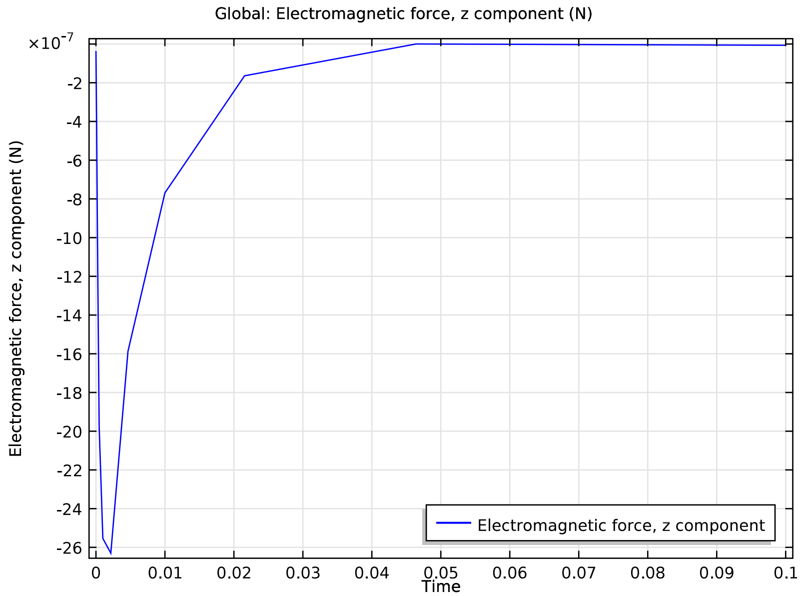


Figure 4: Total force acting on the copper plate plotted as a function of time.

Model Library path: `ACDC_Module/Inductive_Devices_and_Coils/coil_above_plate`

Modeling Instructions—Frequency Domain

MODEL WIZARD

1. Go to the **Model Wizard** window.
2. Click the **2D axisymmetric** button.
3. Click **Next**.
4. In the **Add physics** tree, select **AC/DC>Magnetic Fields (mf)**.
5. Click **Next**.
6. Find the **Studies** subsection. In the tree, select **Preset Studies>Frequency Domain**.
7. Click **Finish**.

GEOMETRY 1

Square 1

1 In the **Model Builder** window, under **Model 1** right-click **Geometry 1** and choose **Square**.
2 In the **Square** settings window, locate the **Size** section.
3 In the **Side length** edit field, type `0.1`.
4 Locate the **Position** section. In the **z** edit field, type `-0.05`.

Rectangle 1

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Rectangle**.
2 In the **Rectangle** settings window, locate the **Size** section.
3 In the **Width** edit field, type `0.08`.
4 In the **Height** edit field, type `0.02`.
5 Locate the **Position** section. In the **z** edit field, type `-0.021`.

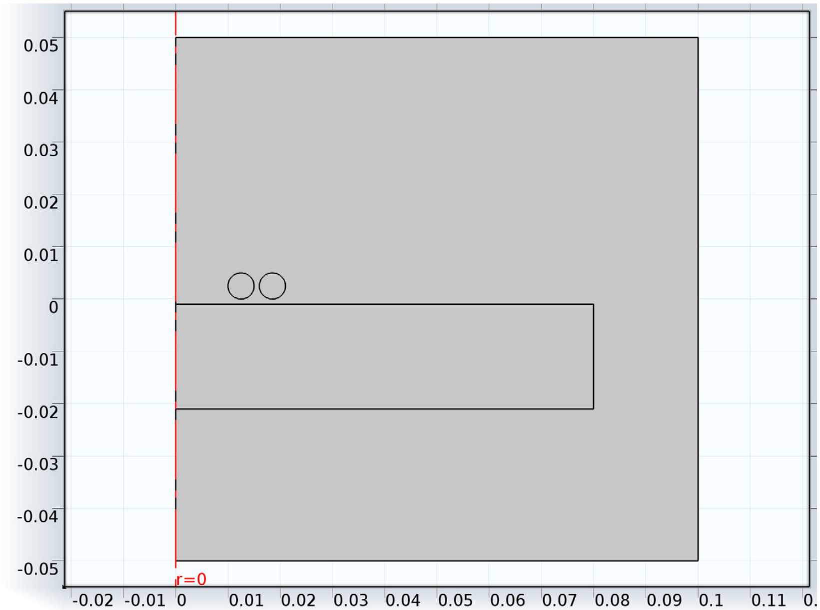
Circle 1

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Circle**.
2 In the **Circle** settings window, locate the **Size and Shape** section.
3 In the **Radius** edit field, type `0.0025`.
4 Locate the **Position** section. In the **r** edit field, type `0.0125`.
5 In the **z** edit field, type `0.0025`.

Circle 2

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Circle**.
2 In the **Circle** settings window, locate the **Size and Shape** section.
3 In the **Radius** edit field, type `0.0025`.
4 Locate the **Position** section. In the **r** edit field, type `0.0185`.
5 In the **z** edit field, type `0.0025`.

6 Click the **Build All** button. The geometry is now complete.



Next, add the materials relevant to the model.

MATERIALS

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

2 In the **Material Browser** window, locate the **Materials** section.

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

4 Right-click and choose **Add Material to Model** from the menu.

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

6 In the **Material Browser** window, locate the **Materials** section.

7 In the tree, select **Built-In>Copper**.

8 Right-click and choose **Add Material to Model** from the menu.

Copper

1 In the **Model Builder** window, under **Model 1>Materials** click **Copper**.

2 Select Domains 2–4 only.

MAGNETIC FIELDS

In the **Model Builder** window, expand the **Model 1>Magnetic Fields** node.

Single-Turn Coil Domain 1

1 Right-click **Magnetic Fields** and choose the domain setting **Single-Turn Coil Domain**.
2 Select Domains 3 and 4 only.
3 In the **Single-Turn Coil Domain** settings window, locate the **Single-Turn Coil Domain** section.
4 From the **Coil excitation** list, choose **Voltage**.
5 In the V_{coil} edit field, type `1e-4`. With this setting, the **Single-Turn Coil Domain** feature applies a loop voltage of 1 mV to each of the coil loops.

Now, add a Force Calculation feature that computes the total force acting on the plate.

Force Calculation 1

1 In the **Model Builder** window, right-click **Magnetic Fields** and choose the domain setting **Force Calculation**.
2 Select Domain 2 only.
3 In the **Force Calculation** settings window, locate the **Force Calculation** section.
4 In the **Force name** edit field, type `plate`.

STUDY 1

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 1** click **Step 1: Frequency Domain**.
2 In the **Frequency Domain** settings window, locate the **Study Settings** section.
3 In the **Frequencies** edit field, type `10[Hz],100[Hz],300[Hz]`.

Disable the automatic plot generation.

4 In the **Model Builder** window, click **Study 1**.
5 In the **Study** settings window, locate the **Study Settings** section.
6 Clear the **Generate default plots** check box.
7 Click the **Compute** button.

When the solution process is completed, create plot groups to visualize the results.

RESULTS

2D Plot Group 1

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

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

3 In the **Surface** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Magnetic Fields>Currents and charge>Induced current density>Induced current density, phi component (mf.Jiphi)**.

Add a contour plot to show the field lines of the magnetic flux density. In axial symmetry, those lines can be obtained by plotting the isolines of the magnetic vector potential multiplied by the radial coordinate, r.

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

5 In the **Contour** settings window, locate the **Expression** section.

6 In the **Expression** edit field, type `Aphi*r`.

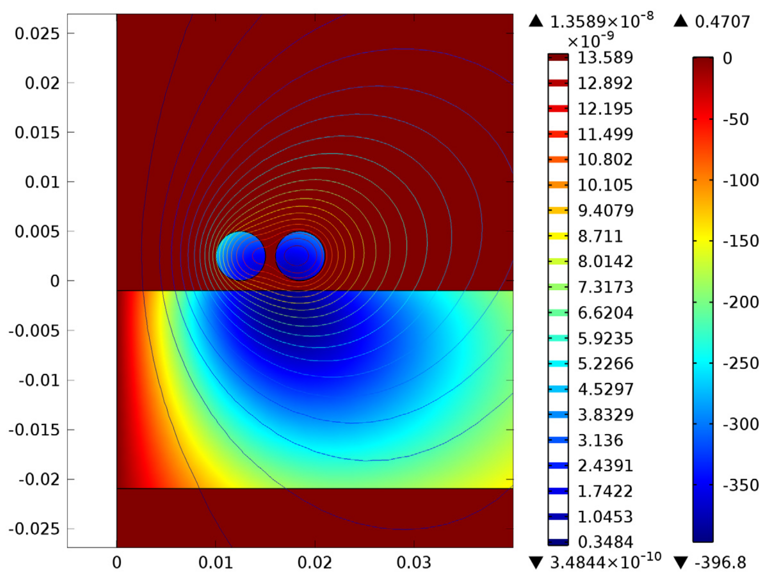
7 In the **Model Builder** window, click **2D Plot Group 1**.

8 In the **2D Plot Group** settings window, locate the **Data** section.

9 From the **Parameter value (freq)** list, choose **10**.

10 Click the **Plot** button.

freq(1)=10 Surface: Induced current density, phi component (A/m^2) Contour: Aphi*r (Wb)

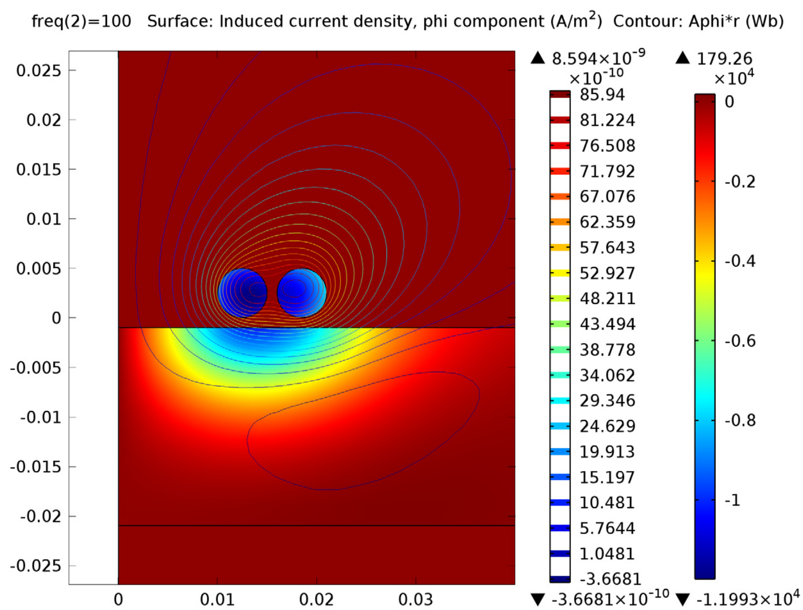


The plot shows the induced current density in the plate. Plotting the other solutions shows how the region in which the currents are induced decreases with increasing frequency.

11 In the **Model Builder** window, click **2D Plot Group 1**.

12 In the **2D Plot Group** settings window, locate the **Data** section.

13 From the **Parameter value (freq)** list, choose **100**.



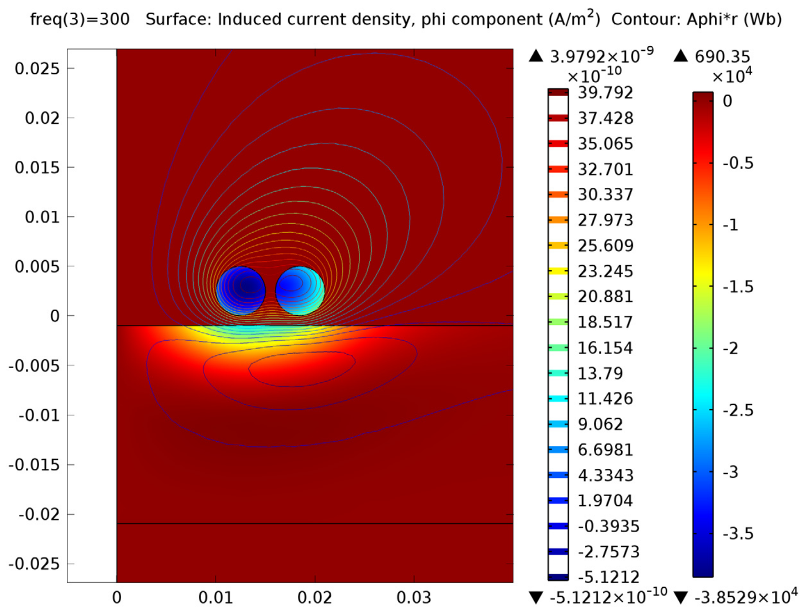
14 Click the **Plot** button.

15 In the **Model Builder** window, click **2D Plot Group 1**.

16 In the **2D Plot Group** settings window, locate the **Data** section.

17 From the **Parameter value (freq)** list, choose **300**.

18 Click the **Plot** button.



Modeling Instructions —Transient Analysis

To set up a time-dependent study to investigate the step response of the system requires only a few additional steps. The **Initial Values** feature automatically included in the **Magnetic Fields** interface specifies the initial value for the magnetic vector potential, defaulted to zero. At the beginning of the transient simulation ($t = 0$), a 0.1 mV voltage is applied to the coil. This corresponds to exciting from an unexcited state the system with a step function.

ROOT

In the **Model Builder** window, right-click the root node and choose **Add Study**.

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Find the **Studies** subsection. In the tree, select **Preset Studies>Time Dependent**.

3 Click **Finish**.

STUDY 2

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 2** click **Step 1: Time Dependent**.

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

3 In the **Times** edit field, type `0,10^(range(-4,1/3,-1))`.

4 Select the **Relative tolerance** check box.

5 In the associated edit field, type `0.001`.

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

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

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

9 Click the **Compute** button.

RESULTS

2D Plot Group 2

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

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

3 From the **Data set** list, choose **Solution 2**.

4 From the **Time** list, choose **0.002154**.

5 Right-click **Results>2D Plot Group 2** and choose **Surface**.

6 In the **Surface** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Magnetic Fields>Currents and charge>Induced current density>Induced current density, phi component (mf.Jiphi)**.

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

8 In the **Arrow Surface** settings window, locate the **Arrow Positioning** section.

9 Find the **r grid points** subsection. In the **Points** edit field, type `50`.

10 Find the **z grid points** subsection. In the **Points** edit field, type `50`.

11 Locate the **Coloring and Style** section. From the **Color** list, choose **White**.

12 Click the **Plot** button.

The Force Calculation feature automatically computed the total force acting on the plate and created a global variable that can be plotted as a function of time.

1D Plot Group 3

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

2 In the **1D Plot Group** settings window, locate the **Data** section.

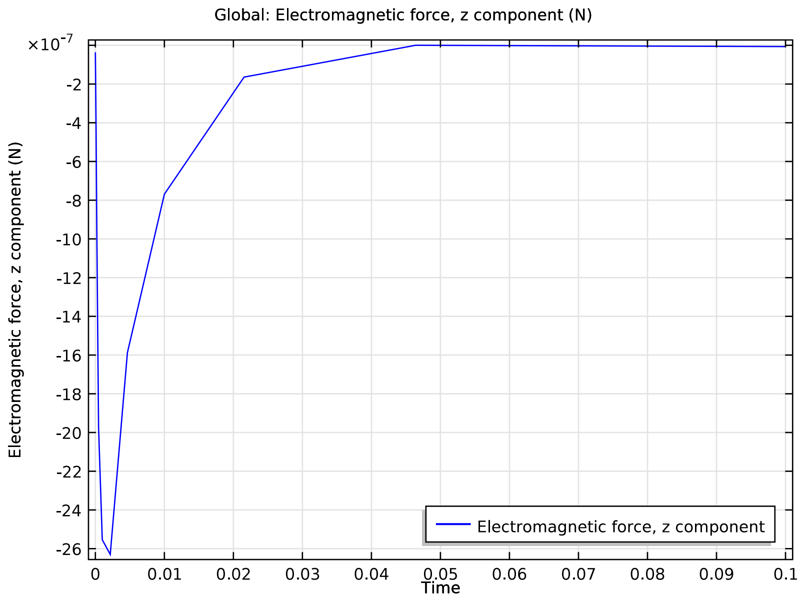
3 From the **Data set** list, choose **Solution 2**.

4 Click to expand the **Legend** section. From the **Position** list, choose **Lower right**.

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

6 In the **Global** settings window, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Magnetic Fields>Mechanical>Electromagnetic force>Electromagnetic force, z component (mf.Forcez_plate)**.

7 Click the **Plot** button. The plot shows that a repulsive force acts on the plate during the transient.



The following instructions explain how to use a **Revolution 2D** data set to obtain a 3D plot from the 2D axisymmetric model.

Data Sets

1 In the **Model Builder** window, expand the **Results>Data Sets** node.

2 Right-click **Data Sets** and choose **Solution**.

3 Right-click **Results>Data Sets>Solution 3** and choose **Add Selection**.

4 In the **Selection** settings window, locate the **Geometric Entity Selection** section.

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

6 Select Domains 2–4 only.

7 In the **Model Builder** window, right-click **Data Sets** and choose **Revolution 2D**.

8 In the **Revolution 2D** settings window, locate the **Data** section.

9 From the **Data set** list, choose **Solution 3**.

10 Click to expand the **Revolution Layers** section. In the **Start angle** edit field, type -90.

11 In the **Revolution angle** edit field, type 255.

3D Plot Group 4

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

2 Right-click **3D Plot Group 4** and choose **Surface**.

3 In the **Surface** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Magnetic Fields>Currents and charge>Induced current density>Induced current density, phi component (mf.Jiphi)**.

4 In the **Model Builder** window, click **3D Plot Group 4**.

5 In the **3D Plot Group** settings window, locate the **Data** section.

6 From the **Parameter value (freq)** list, choose **10**.

7 Click the **Plot** button.

8 Click the **Zoom In** button on the Graphics toolbar.