Created in COMSOL Multiphysics 5.5

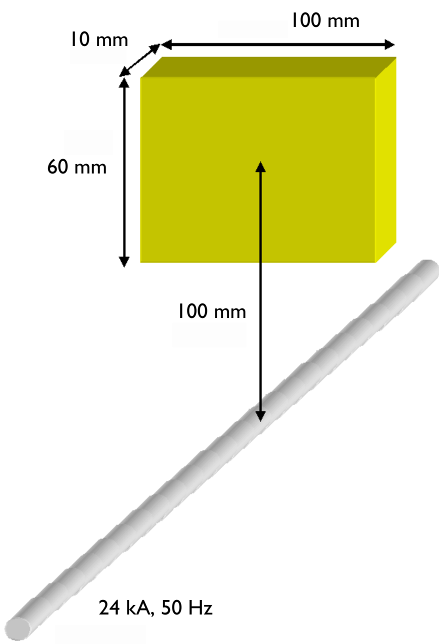
# Eddy Currents

## *Introduction*

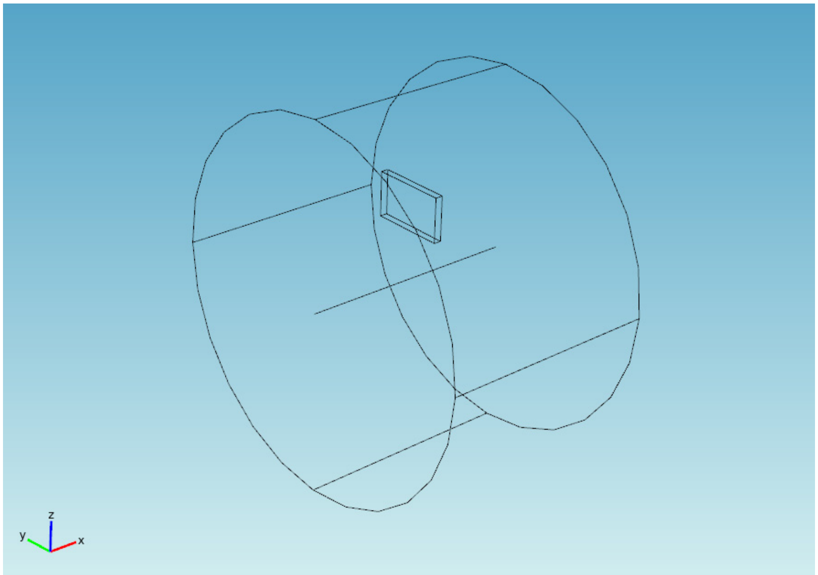
Induced eddy currents and associated thermal loads are of interest in many high power AC applications. This example is of general nature and illustrates some of the involved physics as well as suitable modeling techniques in the AC/DC Module.

## *Model Definition*

A metallic plate is placed near a 50 Hz AC conductor. The resulting eddy current distribution in the plate depends on the conductivity and permeability of the plate. The discussion considers four different materials: copper, aluminum, stainless steel, and magnetic iron. The step-by-step instructions focus on copper and iron. The geometry consists of a single wire and a plate with dimensions as shown below.




Because one cannot afford meshing an infinite volume, it is necessary to specify a finite volume to mesh and solve for. In this case, it makes sense to enclose the wire and the plate in a cylinder with the wire on the axis of this cylinder. This is a good choice when used with the default boundary condition, Magnetic Insulation. This condition forces the field to be tangential to the exterior boundaries. For the cylindrical shape of the domain that is a reasonable approximation to reality.




The conductor is modeled as a line current with 0° phase and an effective (RMS) value of 24 kA.

The magnetic vector potential is calculated from

$$(j\omega\sigma - \omega^2\varepsilon)\mathbf{A} + \nabla \times \left(\frac{1}{\mu}\nabla \times \mathbf{A}\right) = \mathbf{0}$$

where $\sigma$ is the conductivity, $\varepsilon$ the permittivity, and $\mu$ the permeability.

An important parameter in eddy current modeling is the skin depth, $\delta$.

$$\delta = \sqrt{\frac{2}{\omega\mu\sigma}}$$

The following table lists the skin depth for the different materials at a frequency of 50 Hz.

| MATERIAL | REL. PERMEABILITY | CONDUCTIVITY | SKIN DEPTH |
|---|---|---|---|
| Copper | 1 | $5.998 \cdot 10^7$ S/m | 9 mm |
| Aluminum | 1 | $3.774 \cdot 10^7$ S/m | 12 mm |

| MATERIAL | REL. PERMEABILITY | CONDUCTIVITY | SKIN DEPTH |
| --- | --- | --- | --- |
| Stainless steel | 1 | $1.137 \cdot 10^6$ S/m | 67 mm |
| Iron | 4000 | $1.12 \cdot 10^7$ S/m | 0.34 mm |

In order for the model to produce accurate results, the mesh needs to resolve the evanescent fields in the metal. In practice, this means you need to resolve the skin depth with at least a bit more than 1 element, preferably closer to 2 or even more. This application uses a maximum element size of 5 mm for the copper plate.

When the skin depth is small in comparison to the size of the conducting objects, it can be practically impossible to resolve the skin depth. This often happens at high frequencies, in large structures, or with highly conductive and permeable materials. These cases require a different technique: Exclude the interior of the conducting objects from the model. Instead, represent them with an impedance boundary condition. This condition essentially sets the skin depth to zero, making all induced currents flow on the surface of the conductors. Mathematically, the relation between the magnetic and electric field at the boundary reads:

$$\mathbf{n} \times \mathbf{H} + \sqrt{\frac{\varepsilon - j\sigma/\omega}{\mu}}\mathbf{n} \times (\mathbf{E} \times \mathbf{n}) = \mathbf{0}$$

The dissipated power density, $P_\mathrm{d}$ (SI unit: $\mathrm{W/m^2}$) can be calculated from

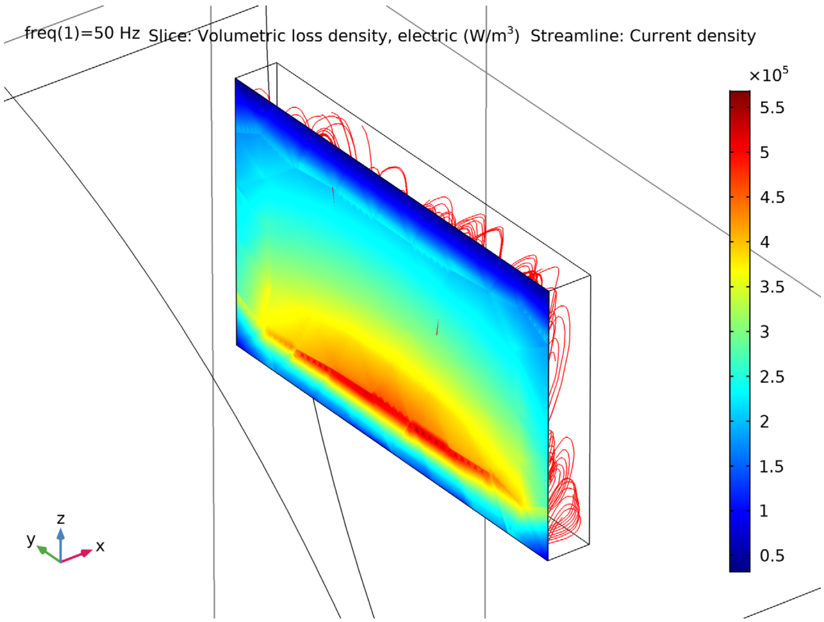
$$P_\mathrm{d} = \frac{1}{2}(\mathbf{J}_S \cdot \mathbf{E}^*)$$

where $\mathbf{J}_S$ is the induced surface current density, and the asterisk (*) denotes the complex conjugate.

This model has the interior of the plate included in the model for copper, and uses the impedance boundary condition for magnetic iron.

## *Results and Discussion*

The induced eddy current distribution for a plate made of copper is shown as streamlines, whereas the distribution of the ohmic losses is shown as a slice plot.




A total dissipated power of 6 W was obtained from integration through the plate. If you repeat the simulation for different materials, the application shows that lowering the conductivity decreases the dissipated power. However, for high permeability materials like soft iron, the dissipated power is higher than in copper (27 W) despite a much lower conductivity.

---

**Application Library path:** `ACDC_Module/Inductive_Devices_and_Coils/eddy_currents`

---

## *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>Electromagnetic Fields>Magnetic Fields (mf)**.
3. Click **Add**.
4. Click **Study**.
5. In the **Select Study** tree, select **General Studies>Frequency Domain**.
6. Click **Done**.

## 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 **mm**.

### *Cylinder 1 (cyl1)*

1. In 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 250.
4. In the **Height** text field, type 300.
5. Locate the **Axis** section. From the **Axis type** list, choose **Cartesian**.
6. In the **x** text field, type 1.
7. In the **z** text field, type 0.

### *Line Segment 1 (ls1)*

1. In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.
2. In the **Settings** window for **Line Segment**, locate the **Starting Point** section.
3. From the **Specify** list, choose **Coordinates**.
4. Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.
5. In the **x** text field, type 300.
6. Click **Build All Objects**.

### *Block 1 (blk1)*

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

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

3 In the **Width** text field, type `10`.

4 In the **Depth** text field, type `100`.

5 In the **Height** text field, type `60`.

6 Locate the **Position** section. In the **x** text field, type `145`.

7 In the **y** text field, type `-50`.

8 In the **z** text field, type `70`.

9 Click **Build All Objects**.

10 Click the **Transparency** button in the **Graphics** toolbar.

The model geometry is now complete. Prepare your simulation by defining selection groups.

## DEFINITIONS

*Explicit 1*

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

2 Right-click **Explicit 1** and choose **Rename**.

3 In the **Rename Explicit** dialog box, type `Plate` in the **New label** text field.

4 Click **OK**.

5 Select Domain 2 only.

*Explicit 2*

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

2 In the **Model Builder** window, right-click **Explicit 2** and choose **Rename**.

3 In the **Rename Explicit** dialog box, type `Plate Boundaries` in the **New label** text field.

4 Click **OK**.

5 Select Domain 2 only.

6 In the **Settings** window for **Explicit**, locate the **Output Entities** section.

7 From the **Output entities** list, choose **Adjacent boundaries**.

## ADD MATERIAL

1 In 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** in the window toolbar.

#### MATERIALS

*Air (mat1)*

**1** In the **Settings** window for **Material**, locate the **Material Contents** section.

**2** In the table, enter the following settings:

| Property | Variable | Value | Unit | Property group |
|---|---|---|---|---|
| Electrical conductivity | sigma_iso ; sigmaii = sigma_iso, sigmaij = 0 | `100[S/m]` | S/m | Basic |

Note that a small nonzero value of the air conductivity will not substantially affect the results, but it will help the solver to converge.

Next, override **Air** as the material for the plate domain by **Copper**.

#### ADD MATERIAL

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

**2** In the tree, select **Built-in>Copper**.

**3** Click **Add to Component** in the window toolbar.

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

#### MATERIALS

*Copper (mat2)*

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

**2** From the **Selection** list, choose **Plate**.

#### MAGNETIC FIELDS (MF)

*Edge Current 1*

**1** In the **Model Builder** window, under **Component 1 (comp1)** right-click **Magnetic Fields (mf)** and choose **Edges>Edge Current**.

**2** Select Edge 6 only.

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

**4** In the $I_0$ text field, type `sqrt(2)*24[kA]`, resulting in an RMS current of 24 kA.

### MESH 1

To resolve the skin depth while maintaining a good mesh economy, mesh the copper plate finer than the rest of the geometry. Note that you need to add the finer mesh first in the sequence; otherwise you would get a coarse mesh on the common domain boundaries that would constrain the mesh inside the copper plate domain.

*Free Tetrahedral 1*

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Free Tetrahedral**.
2. In the **Settings** window for **Free Tetrahedral**, locate the **Domain Selection** section.
3. From the **Geometric entity level** list, choose **Domain**.
4. From the **Selection** list, choose **Plate**.
5. Click to expand the **Scale Geometry** section. In the **y-direction scale** text field, type `0.4`.
6. In the **z-direction scale** text field, type `0.4`.

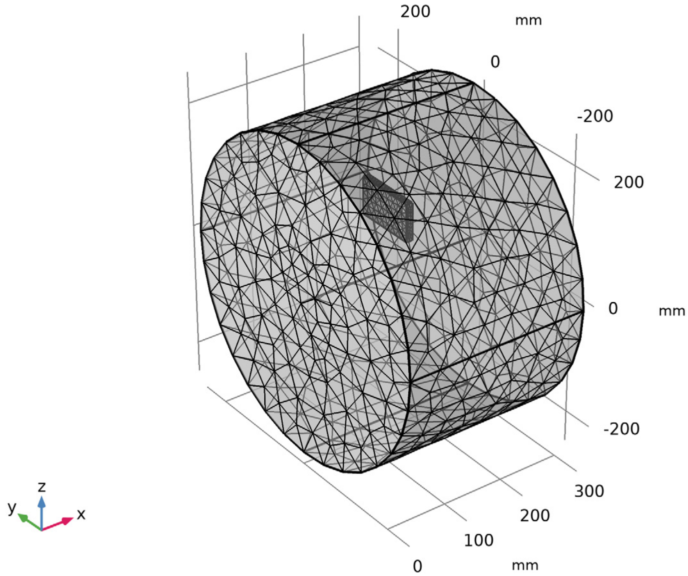
*Size 1*

1. Right-click **Free Tetrahedral 1** and choose **Size**.
2. In the **Settings** window for **Size**, locate the **Element Size** section.
3. Click the **Custom** button.
4. Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
5. In the associated text field, type `5`.

*Free Tetrahedral 2*

1. In the **Model Builder** window, right-click **Mesh 1** and choose **Free Tetrahedral**.

**2** In the **Settings** window for **Mesh**, click **Build All**.




## STUDY 1

*Step 1: Frequency Domain*

**1** In the **Model Builder** window, under **Study 1** click **Step 1: Frequency Domain**.

**2** In the **Settings** window for **Frequency Domain**, locate the **Study Settings** section.

**3** In the **Frequencies** text field, type 50.

**4** In the **Home** toolbar, click **Compute**.

## RESULTS

*Magnetic Flux Density Norm (mf)*

The default plot shows the magnetic flux density norm in three cross sections. The following instructions show you how to visualize the eddy currents and the resistive heating in the copper plate.

*Multislice 1*

**1** In the **Model Builder** window, expand the **Magnetic Flux Density Norm (mf)** node.

**2** Right-click **Multislice 1** and choose **Delete**. Click **Yes** to confirm.

*Slice 1*

1 In the **Model Builder** window, right-click **Magnetic Flux Density Norm (mf)** and choose **Slice**.

2 In the **Settings** window for **Slice**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Magnetic Fields> Heating and losses>mf.Qrh - Volumetric loss density, electric - W/m³**.

3 Locate the **Plane Data** section. From the **Entry method** list, choose **Coordinates**.

4 In the **X-coordinates** text field, type `145.1`.

5 In the **Magnetic Flux Density Norm (mf)** toolbar, click **Plot**.

6 Click the **Transparency** button in the **Graphics** toolbar. This returns the transparency setting to its default state.

*Streamline 1*

1 Right-click **Magnetic Flux Density Norm (mf)** and choose **Streamline**.

2 In the **Settings** window for **Streamline**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Magnetic Fields> Currents and charge>mf.Jx,mf.Jy,mf.Jz - Current density**.

3 Locate the **Streamline Positioning** section. From the **Positioning** list, choose **Starting-point controlled**.

4 In the **Points** text field, type `50`.

5 Click to expand the **Advanced** section. In the **Maximum number of integration steps** text field, type `200`. This reduces the length of the streamlines.

6 In the **Magnetic Flux Density Norm (mf)** toolbar, click **Plot**.

Due to its nonzero conductivity, the air domain too will contain streamlines. Create a selection to see the streamlines in the copper plate only.

*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 **Plate**.

Zoom in to get a better view of the results.

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

**6** Click the **Zoom to Selection** button in the **Graphics** toolbar.

*Magnetic Flux Density Norm (mf)*

As a final step, integrate the resistive heating in the copper to compute the total heating power.

*Volume Integration 1*

**1** In the **Results** toolbar, click **More Derived Values** and choose **Integration>Volume Integration**.

**2** In the **Settings** window for **Volume Integration**, locate the **Selection** section.

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

**4** Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Magnetic Fields>Heating and losses>mf.Qrh - Volumetric loss density, electric - W/m³**.

**5** Click **Evaluate**.

The result should be close to 6 W.

If you would like to repeat the analysis for aluminum or some other material with a skin depth of the order of 1 cm or greater, just change the material in the plate and run the simulation again. In the remaining part of this tutorial, the **Impedance Boundary Condition** will be used to compute the results for magnetic iron, which has a skin depth far less than the thickness of the plate.

#### MAGNETIC FIELDS (MF)

**1** In the **Model Builder** window, under **Component 1 (comp1)** click **Magnetic Fields (mf)**.

**2** In the **Settings** window for **Magnetic Fields**, locate the **Domain Selection** section.

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

**4** Select Domain 1 only.

Removing the plate means no equations will be solved inside it. Add an **Impedance Boundary Condition** to switch to a surface representation instead.

*Impedance Boundary Condition 1*

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

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

**3** From the **Selection** list, choose **Plate Boundaries**.

### ADD MATERIAL

1 In 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>Iron**.

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

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

### MATERIALS

*Iron (mat3)*

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

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

3 From the **Selection** list, choose **Plate Boundaries**.

### STUDY 1

In order to keep the results for copper, disable **Solver 1** before computing the study. This way, COMSOL Multiphysics will generate a second solver branch.

*Solution 1 (sol1)*

1 In the **Model Builder** window, expand the **Study 1>Solver Configurations** node.

2 Right-click **Solution 1 (sol1)** and choose **Disable**.

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

### RESULTS

*Magnetic Flux Density Norm (mf) 1*

In the **Settings** window for **3D Plot Group**, type `Surface loss density, electric (mf)` in the **Label** text field.

The resistive heating variable is now available on the surface of the plate. You can remove the default multislice plot and add a surface plot for the surface resistive heating.

*Multislice 1*

1 In the **Model Builder** window, expand the **Results>Surface loss density, electric (mf)** node.

2 Right-click **Multislice 1** and choose **Delete**. Click **Yes** to confirm.

*Surface 1*

1 In the **Model Builder** window, right-click **Surface loss density, electric (mf)** 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>Magnetic Fields> Heating and losses>mf.Qsrh - Surface loss density, electric - W/m²**.

3 Locate the **Coloring and Style** section. From the **Color table** list, choose **HeatCamera**.

4 In the **Surface loss density, electric (mf)** toolbar, click **Plot**.

*Study 1/Solution 2 (sol2)*

In the **Model Builder** window, click **Study 1/Solution 2 (sol2)**.

*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 **Boundary**.

4 From the **Selection** list, choose **Plate Boundaries**.

*Surface Integration 2*

1 In the **Results** toolbar, click **More Derived Values** and choose **Integration> Surface Integration**.

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

3 From the **Dataset** list, choose **Study 1/Solution 2 (sol2)**.

4 Locate the **Selection** section. From the **Selection** list, choose **Plate Boundaries**.

5 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1>Magnetic Fields>Heating and losses>mf.Qsrh - Surface loss density, electric - W/m²**.

6 Click **Evaluate**.

The result should be close to 27 W.