Created in COMSOL Multiphysics 6.1

Magnet Falling Through Copper Tube

Introduction

This application illustrates the phenomenon of eddy current braking. A cylindrical magnet falling through a copper tube induces eddy currents on the tube walls. The eddy currents, in turn, create a magnetic field that opposes the magnetic field of the magnet and induces a braking force that opposes the motion of the magnet. This opposing force increases with increasing velocity. Thus, there is a terminal velocity at which the magnetic braking force equals the force of gravity. The model computes the velocity of the magnet after it is dropped as it reaches its terminal velocity.

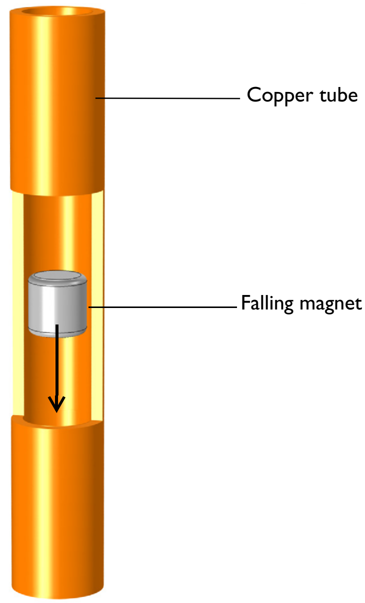


Figure 1: Model illustration of the magnet falling through the copper tube. The cylindrical magnet inside the copper tube is also shown in this figure.

Model Definition

Model the problem in the *rz*-plane, with the *r*-axis lying in the horizontal plane and the *z*-axis representing the vertical axis. Solve the problem in a moving reference frame, where the origin moves with the magnet. Include the velocity of the magnet inside the copper cylinder as a Lorentz term in Ampère's law. The use of the Lorentz term to include the motion is a valid approach in situations when the moving domains do not contain magnetic sources such as currents or magnetization (fixed or induced) that move along with the material, and when the moving domains are invariant in the direction of motion.

A falling magnet in an infinitely long tube is therefore a good example of the correct use of Lorentz terms provided that the model is in a frame where the magnet is fixed and the pipe is moving.

Neglecting the aerodynamic drag force on the magnet and the eddy currents inside the magnet, the equation of motion for the magnet becomes

$$\frac{dv}{dt} - \frac{F_g - F_z}{m} = 0 \tag{1}$$

where v is the magnet velocity, $\mathbf{F}_g$ is the force due to gravity on the magnet, m is the magnet mass, and $\mathbf{F}_z$ is the magnetic force.

Results and Discussion

In this application, a time-domain study is performed to investigate the eddy current effect on the falling magnet through a copper tube. Figure 2 and Figure 3 display a surface plot of the magnetic flux density norm and the current density norm at $t = 50$ ms, respectively.

Figure 4 illustrates the braking force produced by the eddy currents on the copper tube as a function of time. The force is calculated by the volume integration of the Lorentz force in the copper tube. The force is acting upward on the magnet.

Figure 5 shows the velocity of the magnet as a function of time. It shows that the magnet is falling at a constant velocity of about 2.6 cm/s after $t = 20$ ms.

Finally, Figure 6 displays the acceleration of the magnet as a function of time. In this figure, the magnet is initially at the acceleration equal to the acceleration due to gravity 9.81 m/s^2. The acceleration decreases and becomes zero after 20 ms which corresponds to a constant velocity as shown in Figure 5.

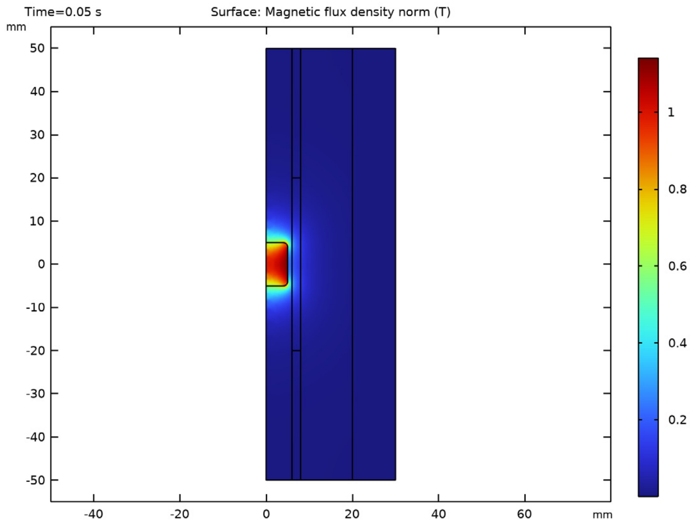


Figure 2: Magnetic flux density norm at t = 50ms.

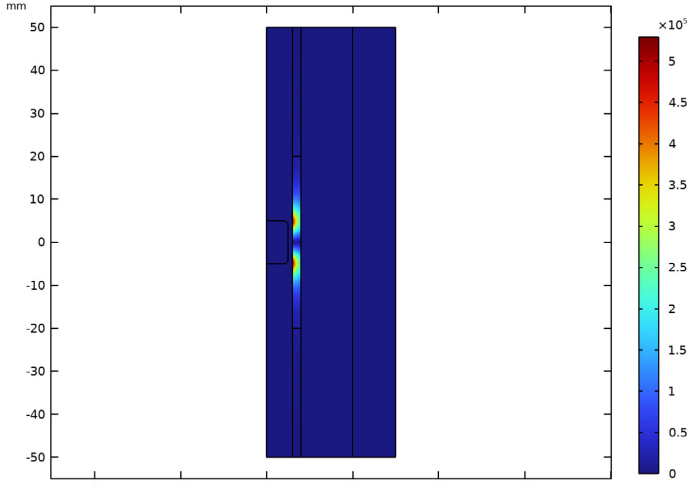


Figure 3: Current density norm at t = 50 ms.

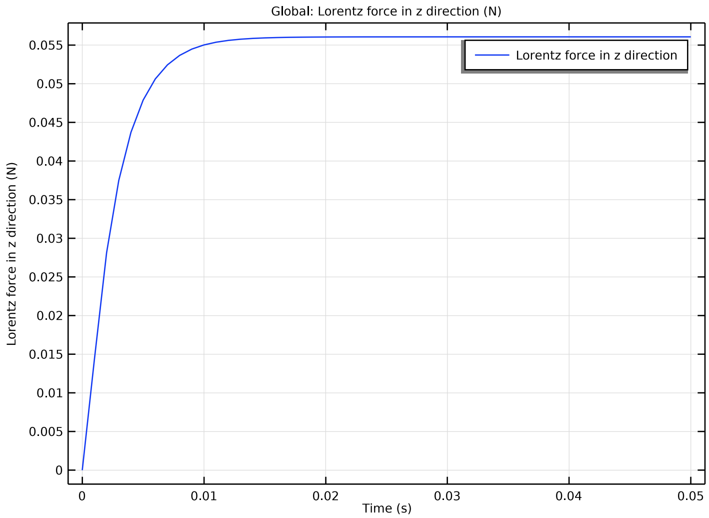


Figure 4: Total force acting on the magnet versus time. The positive force indicates that the force is acting upward on the magnet.

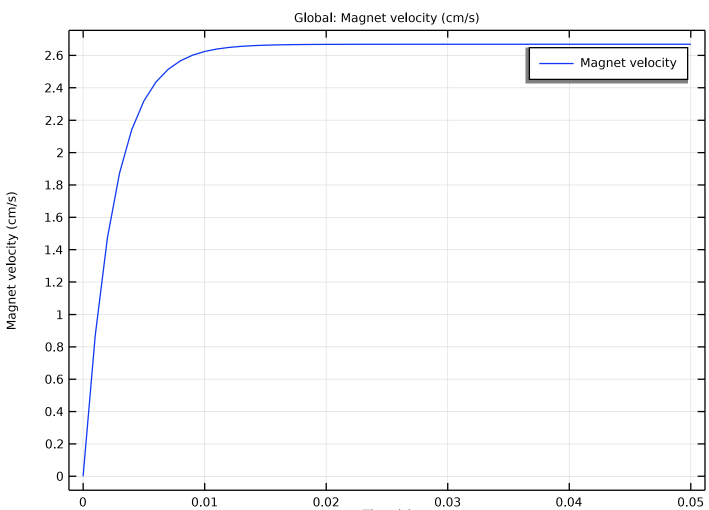


Figure 5: Terminal velocity of the falling magnet versus time.

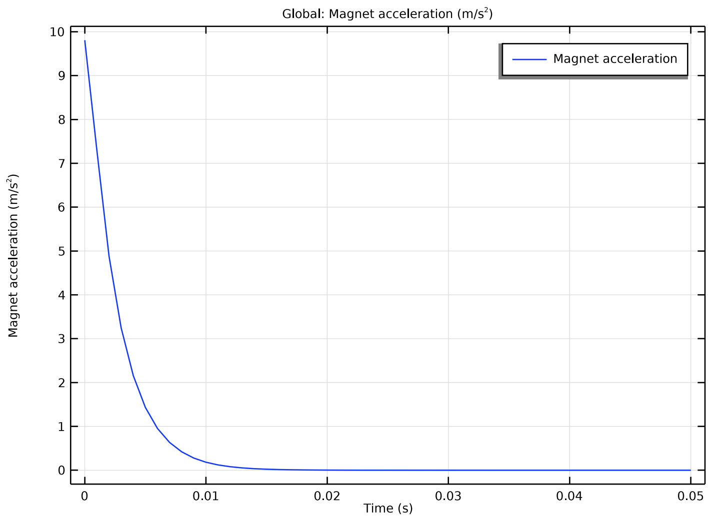


Figure 6: Acceleration of the falling magnet versus time.

Notes About the COMSOL Implementation

Use the Magnetic Fields interface to model the magnetic field, including a Velocity (Lorentz Term) in the copper tube domain. Calculate the Lorentz force as a volume integral over the copper tube. Furthermore, use an Infinite Element Domain feature to model the region of free space surrounding the copper tube, and implement the equation of motion for the falling magnet using a Global ODEs and DAEs interface. Solve the model using two study steps. First, a Stationary study step computes the vector potential field inside and around the stationary permanent magnet. Then, using this stationary solution as an initial condition, a Time Dependent study step determines the terminal velocity and acceleration of the falling magnet.

Application Library path: `ACDC_Module/Devices,_Transducers_and_Actuators/falling_magnet`

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 Axisymmetric**.

2 In the **Select Physics** tree, select **AC/DC>Electromagnetic Fields>Magnetic Fields (mf)**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Mathematics>ODE and DAE Interfaces> Global ODEs and DAEs (ge)**.

5 Click **Add**.

6 Click **Study**.

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

8 Click **Done**.

GLOBAL DEFINITIONS

Define all the required parameters.

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
mr	5[mm]	0.005 m	Magnet radius
mh	10[mm]	0.01 m	Magnet height
r_i	6[mm]	0.006 m	Tube inner radius
r_o	8[mm]	0.008 m	Tube outer radius
dm	7.4[g/(cm)^3]	7400 kg/m³	Density of magnet

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**.

Use the following instructions to construct the model geometry. First, create the magnet.

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 `mr`.

4 In the **Height** text field, type `mh`.

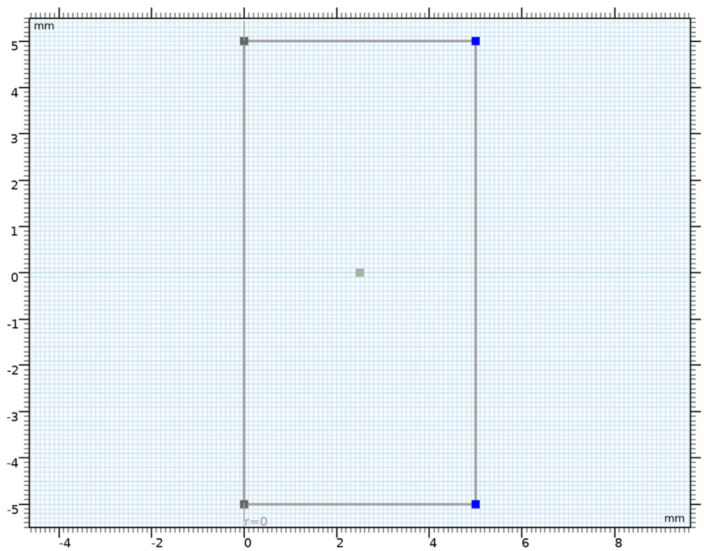
5 Locate the **Position** section. In the **z** text field, type `-mh/2`.

6 Click **Build Selected**.

Fillet 1 (fil1)

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

2 On the object **r1**, select Points 2 and 3 only.



3 In the **Settings** window for **Fillet**, locate the **Radius** section.

4 In the **Radius** text field, type `1`.

5 Click **Build Selected**.

Create the geometry of the copper tube.

Rectangle 2 (r2)

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 `r_o-r_i`.

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

5 Locate the **Position** section. In the **r** text field, type `r_i`.

6 In the **z** text field, type `-50`.

7 Click **Build Selected**.

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

Rectangle 3 (r3)

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 `r_o-r_i`.

4 In the **Height** text field, type `40`.

5 Locate the **Position** section. In the **r** text field, type `r_i`.

6 In the **z** text field, type `-20`.

7 Click **Build Selected**.

Finish the geometry by creating the outer boundary.

Rectangle 4 (r4)

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 `30`.

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

5 Locate the **Position** section. In the **z** text field, type `-50`.

6 Click to expand the **Layers** section. Select the **Layers to the right** check box.

7 Clear the **Layers on bottom** check box.

8 In the table, enter the following settings:

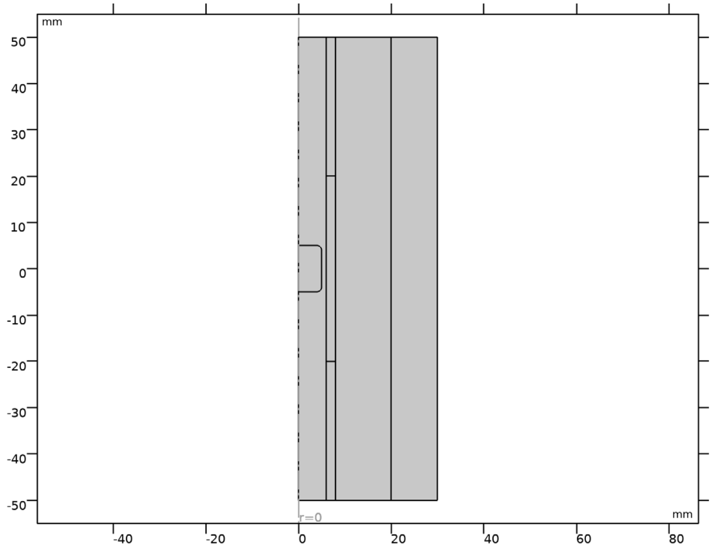
Layer name	Thickness (mm)
Layer 1	10

9 Click **Build Selected**.

Form Union (fin)

1 In the **Model Builder** window, click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, click **Build Selected**.



DEFINITIONS

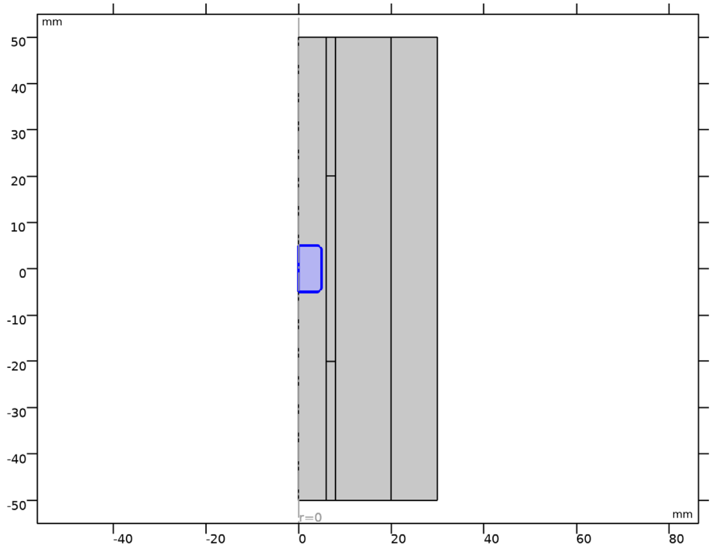
Define domain selections for the magnet and the copper tube before setting up the physics. First, create a selection for the magnet domain.

Magnet

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

2 Select Domain 2 only.

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



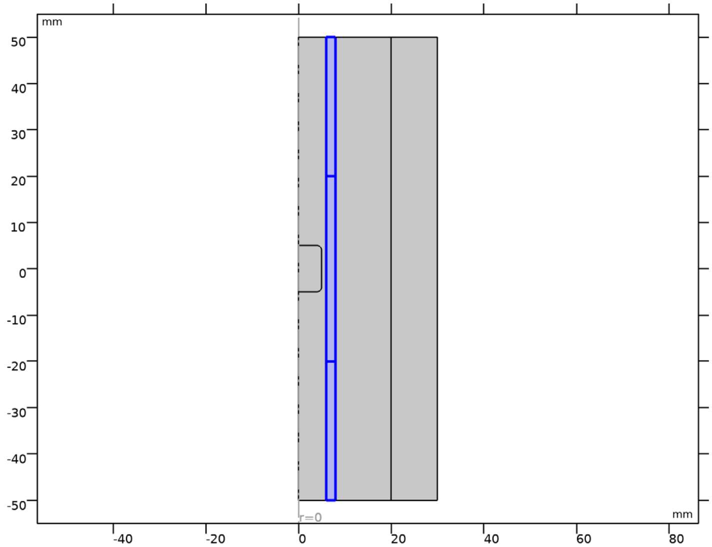
Add a selection for the copper tube domain.

Copper Tube

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

2 Select Domains 3–5 only.

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



Define an integration variable for the magnet domain.

Integration over Magnet

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type `intmag` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Selection** list, choose **Magnet**.

4 Locate the **Advanced** section. Clear the **Compute integral in revolved geometry** check box.

5 In the **Label** text field, type `Integration over Magnet`.

Add variables for the mass and the gravitational force of the magnet.

Variables 1

1 In the **Definitions** toolbar, click **Local Variables**.

2 In the **Settings** window for **Variables**, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
m	intmag(2*pi*r*dm)	kg	Mass of the magnet
Fg	m*g_const	N	Gravitational force on the magnet

Here, `g_const` is a predefined constant for the acceleration of gravity near the surface of the Earth.

Add a nonlocal integration coupling to integrate on the tube domain.

Integration over Tube

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Integration**.

2 In the **Settings** window for **Integration**, type `inttube` in the **Operator name** text field.

3 Locate the **Source Selection** section. From the **Selection** list, choose **Copper Tube**.

4 Locate the **Advanced** section. Clear the **Compute integral in revolved geometry** check box.

5 In the **Label** text field, type `Integration over Tube`.

Define a variable for the Lorentz force and acceleration of the magnet.

Variables 2

1 In the **Definitions** toolbar, click **Local Variables**.

2 In the **Settings** window for **Variables**, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
Fz	inttube(-mf.FLtzz*2*pi*r)	N	Lorentz force in z direction
a	(Fg-Fz)/m	m/s²	Magnet acceleration

Here, the `mf.` prefix identifies variables defined by the **Magnetic Fields** interface.

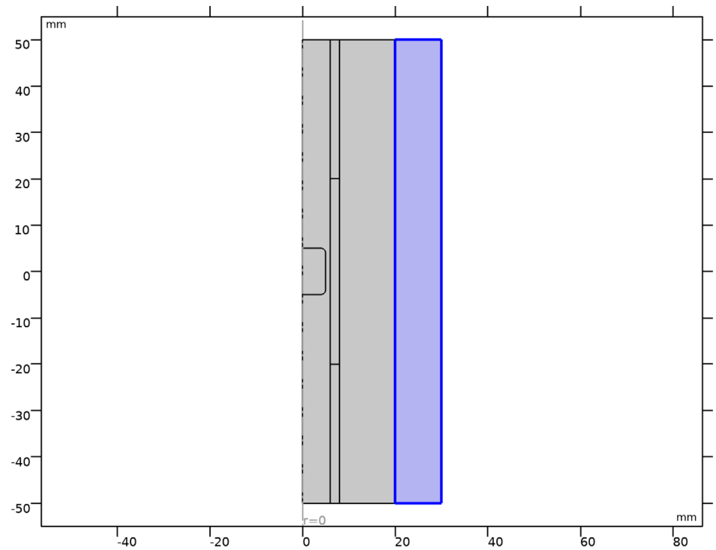
Infinite Element Domain 1 (ie1)

1 In the **Definitions** toolbar, click **Infinite Element Domain**.

2 Select Domain 7 only.

3 In the **Settings** window for **Infinite Element Domain**, locate the **Geometry** section.

4 From the **Type** list, choose **Cylindrical**.



GLOBAL ODES AND DAES (GE)

Global Equations 1

Implement the differential equation for the velocity of the magnet (Equation 1) as a global equation.

1 In the **Model Builder** window, under **Component 1 (comp1)>Global ODEs and DAEs (ge)** click **Global Equations 1**.

2 In the **Settings** window for **Global Equations**, locate the **Global Equations** section.

3 In the table, enter the following settings:

Name	f(u,ut,utt,t) (1)	Initial value (u_0) (1)	Initial value (u_t0) (1/s)	Description
`v`	`d(v,t)-(Fg-Fz)/m`	`0`	`0`	`Magnet velocity`

4 Locate the **Units** section. Click **Select Dependent Variable Quantity**.

5 In the **Physical Quantity** dialog box, type `velocity` in the text field.

6 Click **Filter**.

7 In the tree, select **General>Velocity (m/s)**.

8 Click **OK**.

9 In the **Settings** window for **Global Equations**, locate the **Units** section.

10 Click **Select Source Term Quantity**.

11 In the **Physical Quantity** dialog box, type `acceleration` in the text field.

12 Click **Filter**.

13 In the tree, select **General>Acceleration (m/s^2)**.

14 Click **OK**.

MAGNETIC FIELDS (MF)

Now set up the physics for the magnetic field. Apply **Ampère's Law** in the magnet and the air domain.

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

Ampère's Law - Magnet

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

2 In the **Settings** window for **Ampère's Law**, type `Ampère's Law - Magnet` in the **Label** text field.

3 Locate the **Domain Selection** section. From the **Selection** list, choose **Magnet**.

4 Locate the **Constitutive Relation B-H** section. From the **Magnetization model** list, choose **Remanent flux density**.

5 Specify the **e** vector as

0	r
0	phi
1	z

Specify the velocity for the copper tube domain using a Lorentz term.

Velocity (Lorentz Term) 1

1 In the **Physics** toolbar, click **Domains** and choose **Velocity (Lorentz Term)**.

2 In the **Settings** window for **Velocity (Lorentz Term)**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Copper Tube**.

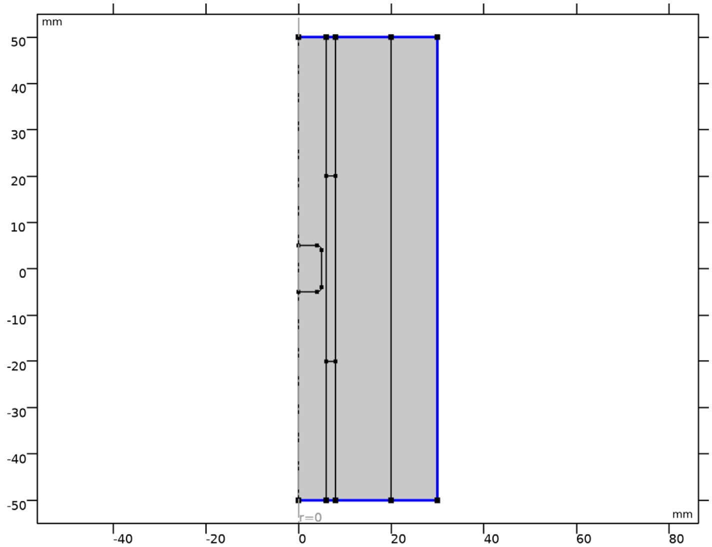
4 Locate the **Velocity (Lorentz Term)** section. Specify the **v** vector as

0	r
v	z

Perfect Magnetic Conductor 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Perfect Magnetic Conductor**.

2 Select Boundaries 2, 7, 10, 15, 17, 20, and 22–24 only.



MATERIALS

Assign materials for the model. Begin by specifying air for all domains.

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)

Next, assign the material to the copper tube and to the magnet. This will automatically override the air.

ADD MATERIAL

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

2 In the tree, select **AC/DC>Copper**.

3 Right-click and choose **Add to Component 1 (comp1)**.

4 In the tree, select **AC/DC>Hard Magnetic Materials> Sintered NdFeB Grades (Chinese Standard)>N50 (Sintered NdFeB)**.

5 Right-click and choose **Add to Component 1 (comp1)**.

6 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 **Copper Tube**.

N50 (Sintered NdFeB) (mat3)

1 In the **Model Builder** window, click **N50 (Sintered NdFeB) (mat3)**.

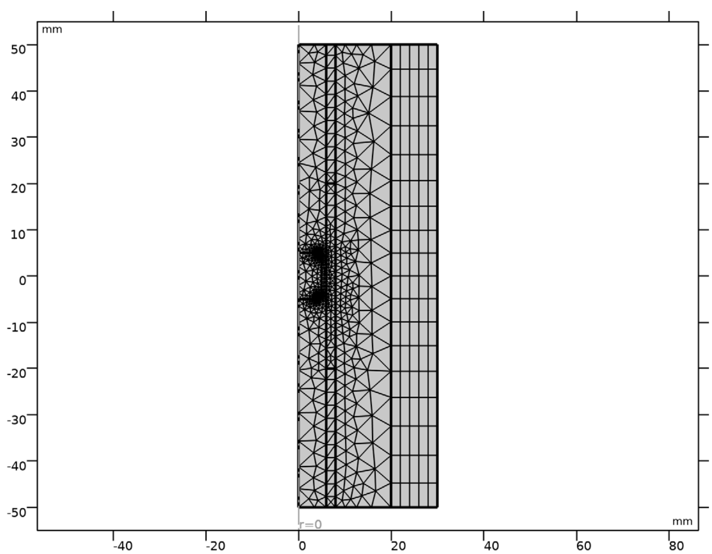
2 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

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

Let **Physics-Controlled Mesh** generate a proper mesh which should look like the figure below.

MESH 1

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



STUDY 1

First, set up the **Stationary** step that computes the vector potential field before the permanent magnet is dropped.

1 In the **Model Builder** window, click **Study 1**.

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

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

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 table, clear the **Solve for** check box for **Global ODEs and DAEs (ge)**.

Now, add a **Time Dependent** study step and solve the problem in time domain from 0 to 50 milliseconds. The **Time Dependent** study will automatically use the stationary solution as the initial condition for the vector potential.

Time Dependent

1 In the **Study** toolbar, click **Study Steps** and choose **Time Dependent>Time Dependent**.

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

3 In the **Output times** text field, type `range(0,0.001,0.05)`.

4 From the **Tolerance** list, choose **User controlled**.

5 In the **Relative tolerance** text field, type `0.001`.

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

RESULTS

Use the following steps to generate a plot of the magnetic flux density norm as shown in Figure 2.

2D Plot Group 1

In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

Surface 1

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

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

Follow the steps below to reproduce the current density norm plot shown in Figure 3.

2D Plot Group 2

In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

Surface 1

1 Right-click **2D Plot Group 2** 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)>Magnetic Fields>Currents and charge>mf.normJ - Current density norm - A/m²**.

3 In the **2D Plot Group 2** toolbar, click **Plot**.

Next, plot the z-component of the Lorentz force on the magnet.

1D Plot Group 3

In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

Lorentz Force, Fz

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

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Definitions>Variables>Fz - Lorentz force in z direction - N**.

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

Compare the resulting plot with Figure 4.

4 In the **Label** text field, type `Lorentz Force, Fz`.

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

Plot the terminal velocity of the magnet using the following instructions. The plot is as shown in Figure 5.

1D Plot Group 4

In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

Terminal Velocity

1 Right-click **1D Plot Group 4** and choose **Global**.

2 In the **Settings** window for **Global**, type `Terminal Velocity` in the **Label** text field.

3 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
v	cm/s	Magnet velocity

4 In the **1D Plot Group 4** toolbar, click **Plot**.

Finally, plot the acceleration of the magnet. The plot is as shown in Figure 6.

1D Plot Group 5

In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

Magnet acceleration

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

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Definitions> Variables>a - Magnet acceleration - m/s²**.

3 In the **Label** text field, type `Magnet acceleration`.

4 In the **1D Plot Group 5** toolbar, click **Plot**.