# Magnet Falling Through Copper Tube

## *Introduction*

This model 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.

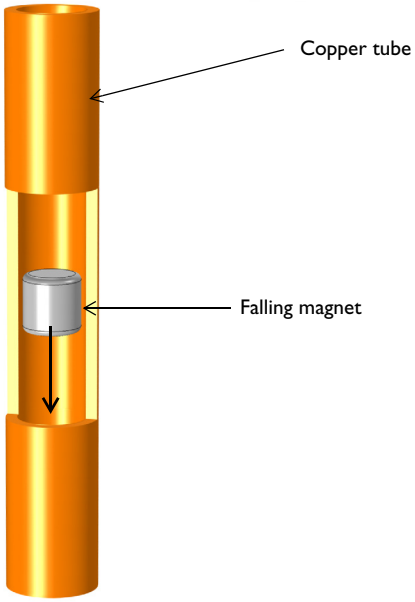



*Figure 1: Model illustration of the magnet falling through the copper tube. The cylindrical magnet inside the copper tube is also shown in this figure.*

## *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, $F_g$ is the force due to gravity on the magnet, $m$ is the magnet mass, and $F_z$ is the magnetic force.

## *Results and Discussion*

In this model, 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 0.26 m/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.

 ©2013 COMSOL

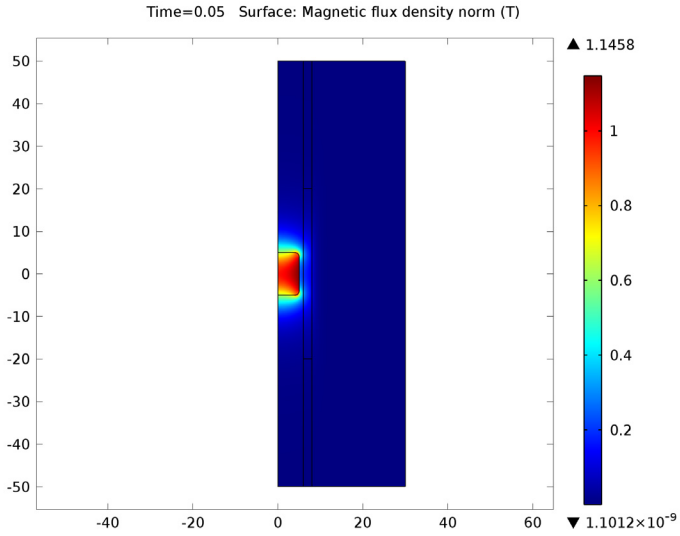



*Figure 2: Magnetic flux density norm at t = 50ms.*

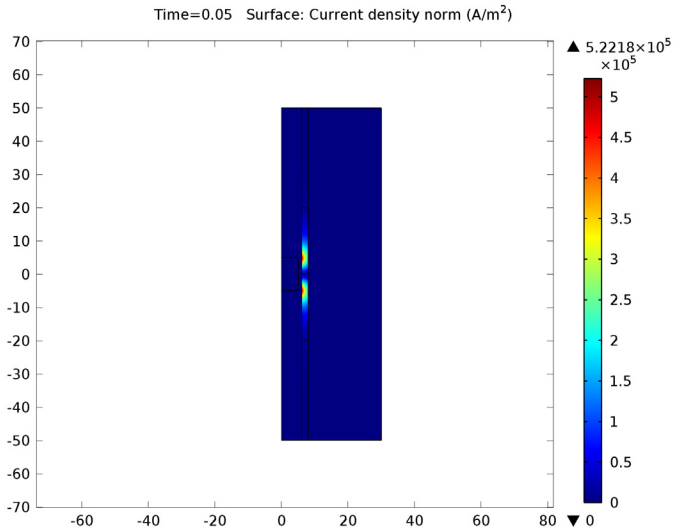



*Figure 3: Current density norm at t = 50ms.*

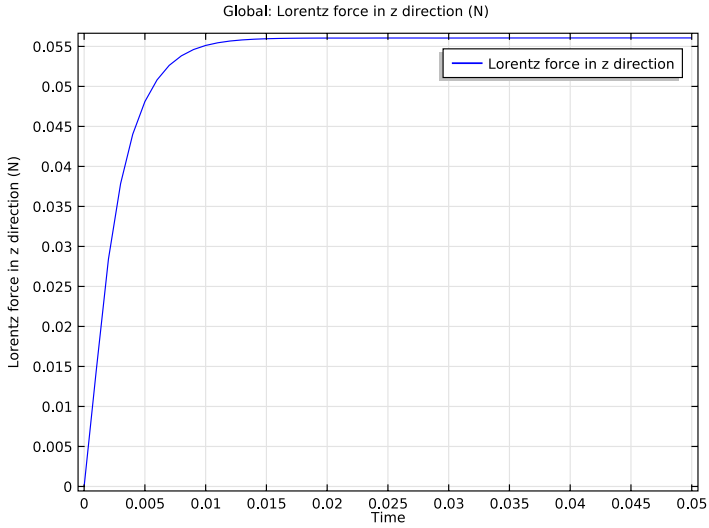



*Figure 4: Total force acting on the magnet versus time. The positive force indicates that the force is acting upward on the magnet.*

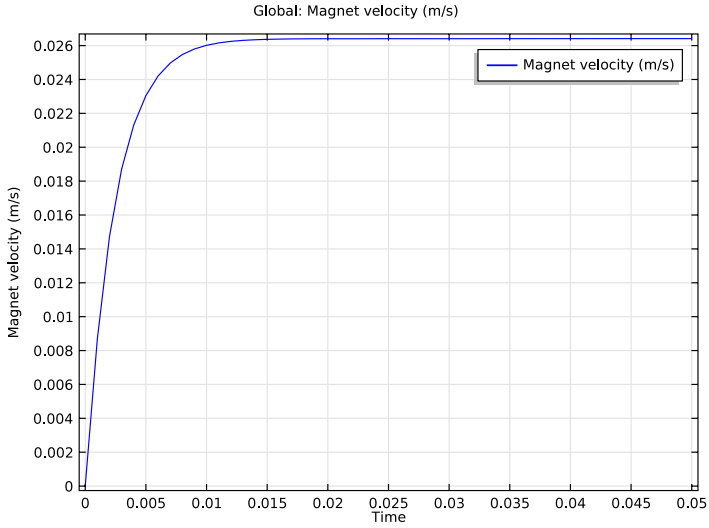



*Figure 5: Terminal velocity of the falling magnet versus time.*


©2013 COMSOL

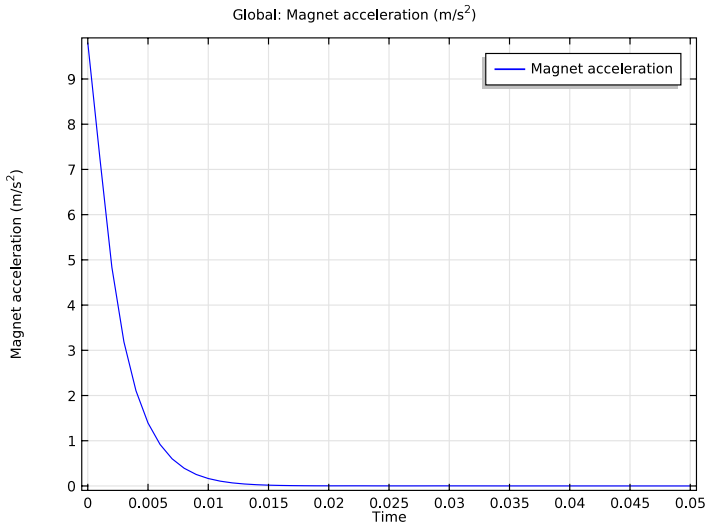



*Figure 6: Acceleration of the falling magnet versus time.*

## *Notes About the COMSOL Implementation*

Use the Magnetic Fields physics 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.

**Model Library path:** `ACDC_Module/Motors_and_Actuators/falling_magnet`

## *Modeling Instructions*

### 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 **Add Selected**.

6 In the **Add physics** tree, select **Mathematics>ODE and DAE Interfaces>Global ODEs and DAEs (ge)**.

7 Click **Add Selected**.

8 Click **Next**.

9 Find the **Studies** subsection. In the tree, select **Preset Studies for Selected Physics>Stationary**.

10 Click **Finish**.

### GLOBAL DEFINITIONS

Define all the required parameters.

*Parameters*

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.

2 In the **Parameters** settings window, locate the **Parameters** section.

3 In the table, enter the following settings:

| Name | Expression | Description |
|---|---|---|
| mr | 5[mm] | Magnet radius |
| mh | 10[mm] | Magnet height |
| r_i | 6[mm] | Tube inner radius |
| r_o | 8[mm] | Tube outer radius |
| dm | 7.4[g/(cm)^3] | Density of magnet |
| Br | 1.4[T] | Remanent flux density |

### GEOMETRY 1

1 In the **Model Builder** window, under **Model 1** click **Geometry 1**.

2 In the **Geometry** settings window, locate the **Units** section.

 ©2013 COMSOL

**3** From the **Length unit** list, choose **mm**.

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

*Rectangle 1*

**1** Right-click **Model 1>Geometry 1** and choose **Rectangle**.

**2** In the **Rectangle** settings window, locate the **Size** section.

**3** In the **Width** edit field, type `mr`.

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

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

**6** Click the **Build Selected** button.

*Fillet 1*

**1** In the **Model Builder** window, right-click **Geometry 1** and choose **Fillet**.

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

**3** In the **Fillet** settings window, locate the **Radius** section.

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

**5** Click the **Build Selected** button.

Create the geometry of the copper tube.

*Rectangle 2*

**1** Right-click **Geometry 1** and choose **Rectangle**.

**2** In the **Rectangle** settings window, locate the **Size** section.

**3** In the **Width** edit field, type `r_o-r_i`.

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

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

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

**7** Click the **Build Selected** button.

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

*Rectangle 3*

**1** Right-click **Geometry 1** and choose **Rectangle**.

**2** In the **Rectangle** settings window, locate the **Size** section.

**3** In the **Width** edit field, type `r_o-r_i`.

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

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

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

7 Click the **Build Selected** button.

Finish the geometry by creating the outer boundary.

*Rectangle 4*

1 Right-click **Geometry 1** and choose **Rectangle**.

2 In the **Rectangle** settings window, locate the **Size** section.

3 In the **Width** edit field, type `30`.

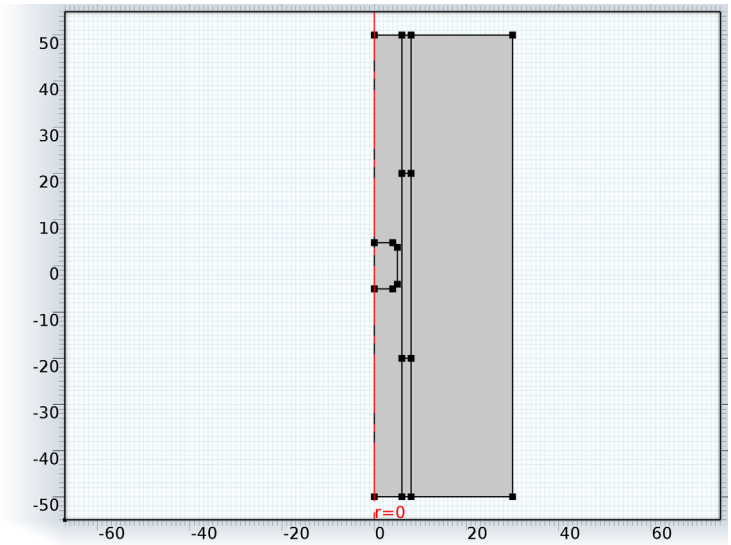
4 In the **Height** edit field, type `100`.

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

6 Click the **Build Selected** button.

*Form Union*

1 In the **Model Builder** window, under **Model 1>Geometry 1** right-click **Form Union** and choose **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.

 ©2013 COMSOL

*Explicit 1*

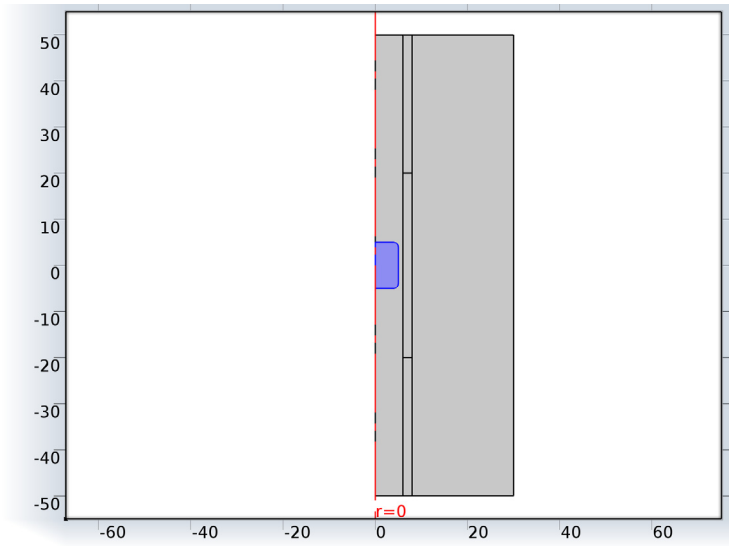
1 In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Selections>Explicit**.

2 Select Domain 2 only.

3 Right-click **Model 1>Definitions>Explicit 1** and choose **Rename**.

4 Go to the **Rename Explicit** dialog box and type `Magnet` in the **New name** edit field.

5 Click **OK**.




Add a selection for the copper tube domain.

*Explicit 2*

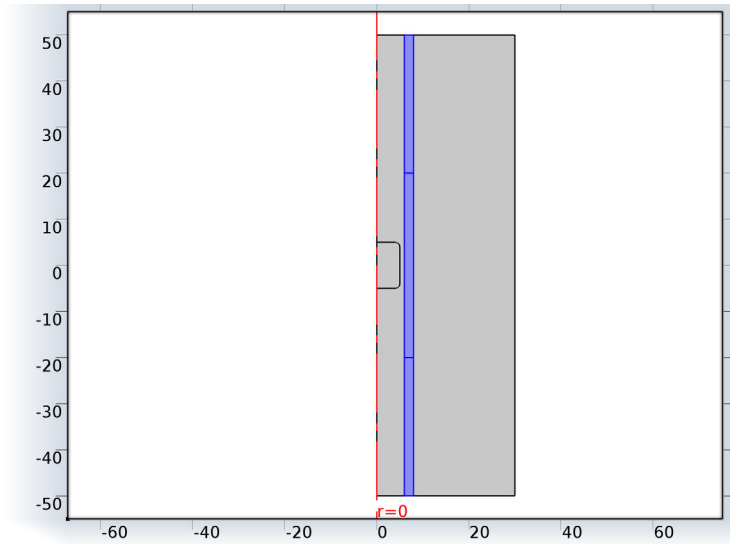
1 Right-click **Definitions** and choose **Selections>Explicit**.

2 Select Domains 3–5 only.

3 Right-click **Model 1>Definitions>Explicit 2** and choose **Rename**.

4 Go to the **Rename Explicit** dialog box and type `Copper Tube` in the **New name** edit field.

**5** Click **OK**.




Define an integration variable for the magnet domain.

*Integration 1*

**1** Right-click **Definitions** and choose **Model Couplings>Integration**.

**2** In the **Integration** settings window, locate the **Operator Name** section.

**3** In the **Operator name** edit field, type `intmag`.

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

**5** Right-click **Model 1>Definitions>Integration 1** and choose **Rename**.

**6** Go to the **Rename Integration** dialog box and type `Integration over Magnet` in the **New name** edit field.

**7** Click **OK**.

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

*Variables 1*

**1** Right-click **Definitions** and choose **Variables**.

**2** In the **Variables** settings window, locate the **Variables** section.

 ©2013 COMSOL

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

| Name | Expression | Description |
|---|---|---|
| m | intmag(2*pi*r*dm) | Magnet mass |
| F_g | m*g_const | Gravitational force on magnet |

Here, g_const is a predefined COMSOL constant for the acceleration due to gravity.

Add an integration variable for the tube domain.

*Integration 2*

**1** Right-click **Definitions** and choose **Model Couplings>Integration**.

**2** In the **Integration** settings window, locate the **Operator Name** section.

**3** In the **Operator name** edit field, type inttube.

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

**5** Right-click **Model 1>Definitions>Integration 2** and choose **Rename**.

**6** Go to the **Rename Integration** dialog box and type Integration over Tube in the **New name** edit field.

**7** Click **OK**.

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

*Variables 2*

**1** Right-click **Definitions** and choose **Variables**.

**2** In the **Variables** settings window, locate the **Variables** section.

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

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

Here, the mf prefix refers to the Magnetic Fields physics interface.

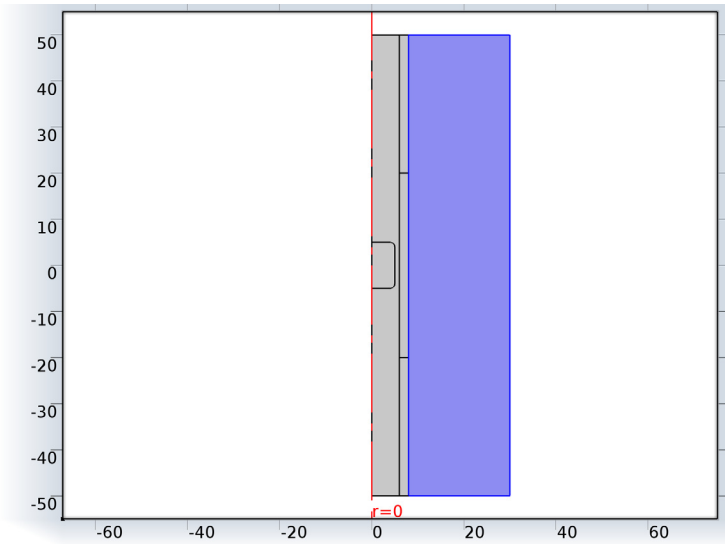
*Infinite Element Domain 1*

**1** Right-click **Definitions** and choose **Infinite Element Domain**.

**2** Select Domain 6 only.

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

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




## GLOBAL ODES AND DAES

### *Global Equations 1*

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

**1** In the **Model Builder** window, under **Model 1>Global ODEs and DAEs** click **Global Equations 1**.

**2** In the **Global Equations** settings window, locate the **Global Equations** section.

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

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

## MAGNETIC FIELDS

Now set up the physics for the magnetic field. Apply Ampere's law in the magnet and the air domain.

 ©2013 COMSOL

*Ampère's Law 2*

1 In the **Model Builder** window, under **Model 1** right-click **Magnetic Fields** and choose **Ampère's Law**.

2 In the **Ampère's Law** settings window, locate the **Domain Selection** section.

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

4 Locate the **Magnetic Field** section. From the **Constitutive relation** list, choose **Remanent flux density**.

5 In the $\mathbf{B}_r$ table, enter the following settings:

| | |
|---|---|
| 0 | r |
| 0 | phi |
| Br | z |

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

*Velocity (Lorentz Term) 1*

1 In the **Model Builder** window, right-click **Magnetic Fields** and choose **Velocity (Lorentz Term)**.

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

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

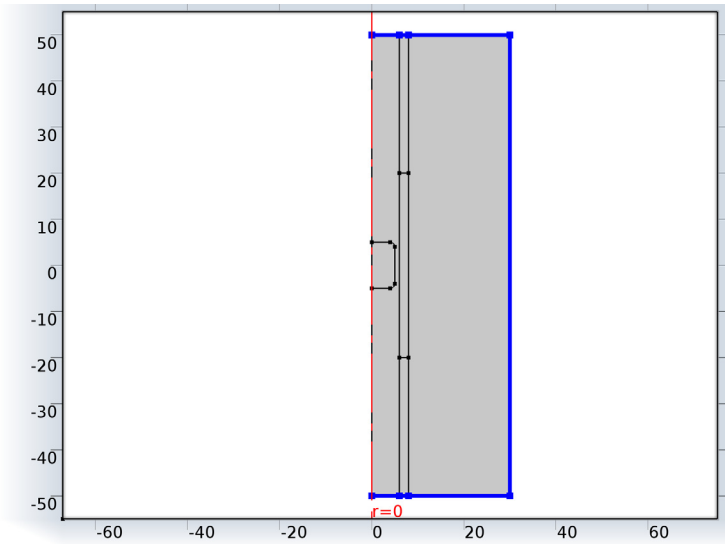
4 Locate the **Velocity (Lorentz Term)** section. In the $\mathbf{v}$ table, enter the following settings:

| | |
|---|---|
| 0 | r |
| v | z |

*Perfect Magnetic Conductor 1*

1 Right-click **Magnetic Fields** and choose **Perfect Magnetic Conductor**.

**2** Select Boundaries 2, 7, 10, 15, 17, 20, and 21 only.




## MATERIALS

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

### *Material Browser*

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

**2** In the **Material Browser** settings window, In the tree, select **Built-In>Air**.

**3** In the **Material_browser** window, click **Add Material to Model**.

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

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

**5** In the **Material Browser** settings window, In the tree, select **AC/DC>Copper**.

**6** In the **Material_browser** window, click **Add Material to Model**.

### *Copper*

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

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

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

## MESH 1

In the **Model Builder** window, under **Model 1** right-click **Mesh 1** and choose **Size**.

 ©2013 COMSOL

*Size*

1 In the **Model Builder** window, under **Model 1>Mesh 1** click **Size**.

2 In the **Size** settings window, locate the **Element Size** section.

3 From the **Predefined** list, choose **Fine**.

*Size 1*

1 In the **Model Builder** window, under **Model 1>Mesh 1** click **Size 1**.

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

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

4 Select Domain 4 only.

5 Locate the **Element Size** section. Click the **Custom** button.

6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

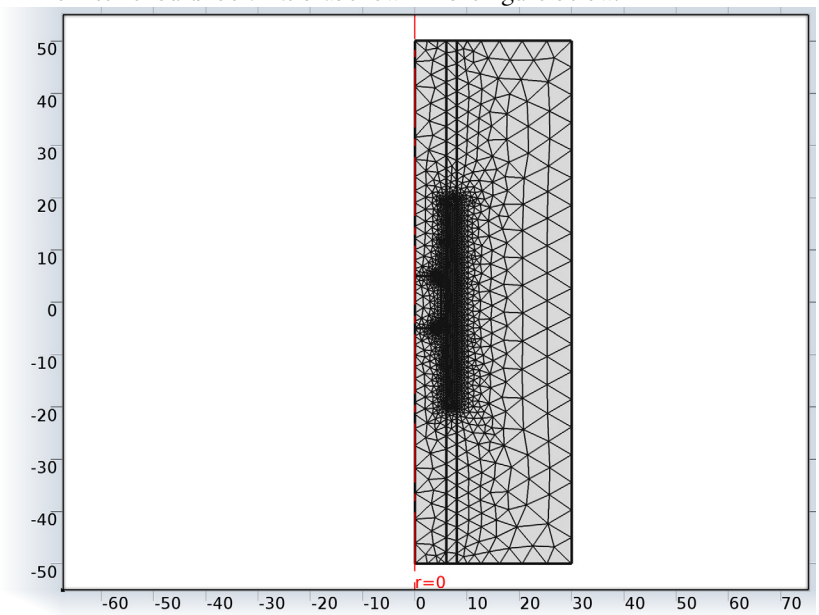
7 In the associated edit field, type `0.5`.

*Free Triangular 1*

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

2 Right-click **Mesh 1** and choose **Build All**.

The mesh should look like that shown in the figure below.



## 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 **Study** settings window, 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 **Stationary** settings window, locate the **Physics and Variables Selection** section.

3 In the table, enter the following settings:

| Physics | Solve for |
|---|---|
| Global ODEs and DAEs | × |

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.

### *Step 2: Time Dependent*

1 In the **Model Builder** window, right-click **Study 1** and choose **Study Steps>Time Dependent**.

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

3 In the **Times** edit field, type `range(0,0.001,0.05)`.

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

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

6 Right-click **Study 1** and choose **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*

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

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

 ©2013 COMSOL

*2D Plot Group 2*

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

2 In the **Model Builder** window, under **Results** right-click **2D Plot Group 2** 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>Current density norm (mf.normJ)**.

4 Click the **Plot** button.

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

*1D Plot Group 3*

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

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

3 In the **Global** settings window, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Definitions>Lorentz force in z direction (Fz)**.

4 Click the **Plot** button.

5 Compare the above plot with Figure 4.

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

7 Go to the **Rename Global** dialog box and type `Lorentz Force, Fz` in the **New name** edit field.

8 Click **OK**.

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

*1D Plot Group 4*

1 Right-click **Results** and choose **1D Plot Group**.

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

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

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

5 Go to the **Rename Global** dialog box and type `Terminal Velocity` in the **New name** edit field.

6 Click **OK**.

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

*1D Plot Group 5*

1 Right-click **Results** and choose **1D Plot Group**.

2 In the **Model Builder** window, under **Results** right-click **1D Plot Group 5** and choose **Global**.

3 In the **Global** settings window, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Definitions>Magnet acceleration (a)**.

4 Click the **Plot** button.

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

6 Go to the **Rename Global** dialog box and type `Magnet acceleration` in the **New name** edit field.

7 Click **OK**.

 ©2013 COMSOL