Created in COMSOL Multiphysics 5.5

Magnetic Damping of Vibrating Conducting Solids

Introduction

When a conductive solid material moves through a static magnetic field, an eddy current density is induced. That induced eddy current density interacts with the static magnetic field and the result is a Lorentz force back on the solid that counteracts the motion. Therefore, a conducting solid that is vibrating in a static magnetic field experiences a structural damping.

This example computes the damping effect when a cantilever beam is harmonically excited across a range of frequencies and placed in a strong magnetic field. The approach presented here assumes that the relative magnitude of the structural displacements are small, that the material has isotropic and linear properties, and that the damping Lorentz force can be computed from the static magnetic field and the motion induced AC eddy current density. Second order effects arising from the AC magnetic field generated by the eddy currents are not included in the computation. The AC magnetic field is also computed and found to be 2-3 orders of magnitude smaller than the DC magnetic field.

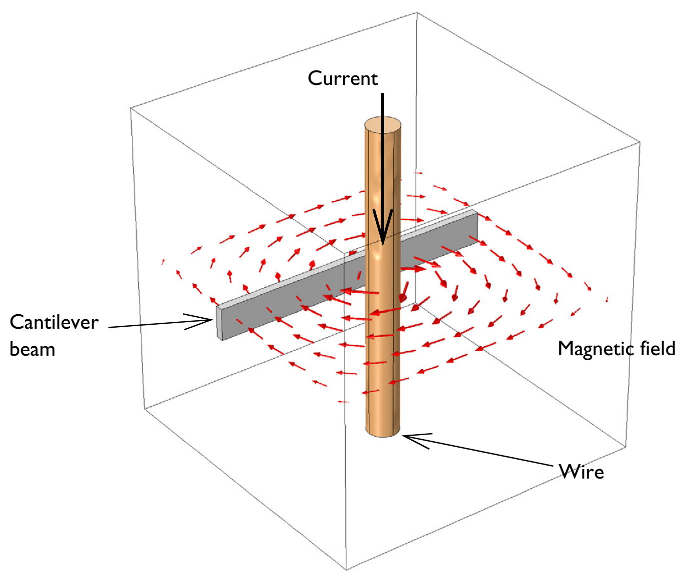


Figure 1: A vibrating beam next to a current carrying wire experiences magnetic damping.

Model Definition

For a solid material experiencing a time-harmonic forced excitation, the displacement field is of the form

$$\mathbf{u}(\mathbf{r}, t) = \hat{\mathbf{u}}(\mathbf{r})\sin(\omega t)$$

which can also be written in the frequency domain as a phasor:

$$\mathbf{u}(\mathbf{r}, t) = Re(\hat{\mathbf{u}}(\mathbf{r})e^{i\omega t})$$

Thus, the velocity field is given by

$$\mathbf{v}(\mathbf{r}, t) = Re(i\omega\hat{\mathbf{u}}(\mathbf{r})e^{i\omega t})$$

Next, consider the effect of a spatially non-uniform but static magnetic flux density, $\mathbf{B}_{DC}(\mathbf{r})$. Under the assumption that the local displacements are small enough so that each moving point in the solid sees only the magnetic flux density in the undeformed state, the velocity induced current density is given by

$$\boldsymbol{J}_i = \sigma\mathbf{v} \times \boldsymbol{B}_{DC}(\boldsymbol{r})$$

where σ is the material conductivity. The resulting total AC current density is however different as the metallic cantilever beam is inductive, that is it exhibits skin effect. Thus a second, frequency domain, magnetodynamic, problem has to be solved in order to compute the AC current density.

The body forces experienced by a current-carrying domain moving through a magnetic field are then given by the cross product between the AC current density and the static magnetic flux density.

$$\boldsymbol{F}_B = \boldsymbol{J}_{AC} \times \boldsymbol{B}_{DC}(\boldsymbol{r})$$

These body forces are then applied to the frequency domain structural mechanics problem and act as a damping force on the system.

The application first computes the static magnetic field due to a current-carrying wire which is next to an aluminum beam. In the second solution step, the beam experiences a forced harmonic vibration and the resulting mechanical beam displacement field and AC current density are computed, yielding also the damping electromagnetic force. The second step is set as a Frequency Domain Perturbation study. The strength of the magnetic field is varied, and the effect of the magnetic damping on the response of the system is observed.

Results and Discussion

Figure 2 shows the magnetic flux density computed for the structure at rest. Figure 3 displays the magnitude of the displacement of the tip of the beam versus excitation frequency for two different magnetic field intensities for the frequency-domain structural dynamics problem. The magnetic field provides significant additional damping. Figure 4 shows a snapshot of the induced eddy current distribution in the beam.

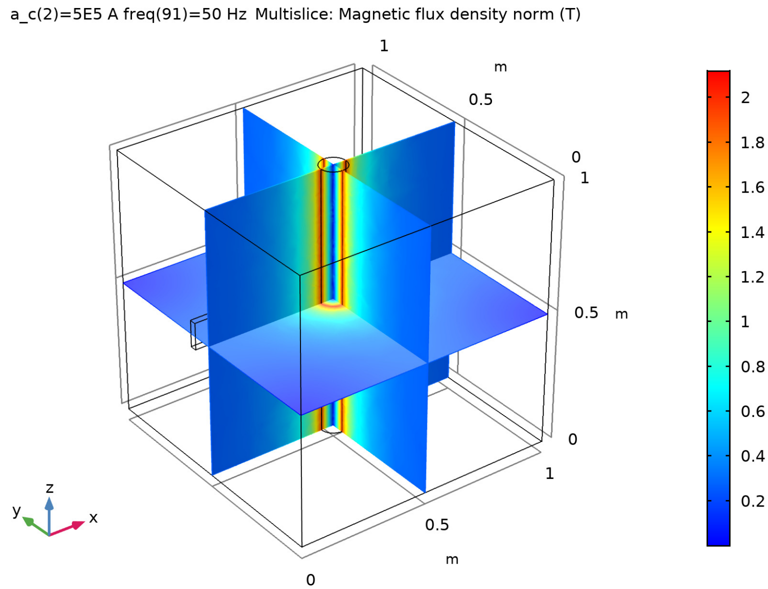


Figure 2: The magnetic field around a current carrying wire.

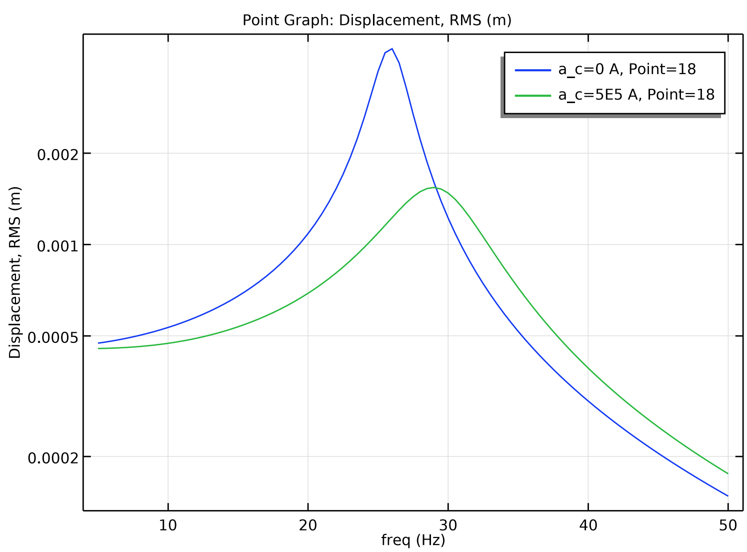


Figure 3: Displacement of the tip of the beam versus excitation frequency for differing magnetic field strengths.

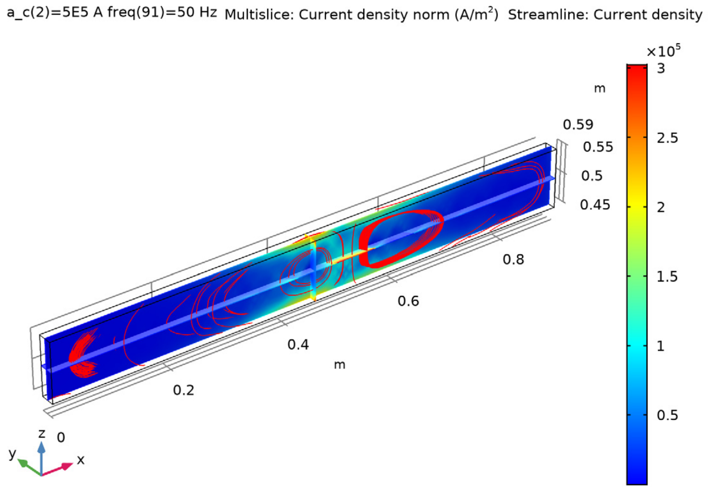


Figure 4: The AC current distribution.

Notes About the COMSOL Implementation

Solve this application with two physics interfaces — the Magnetic Fields and Solid Mechanics interfaces. Use a Stationary study for the first Magnetic Fields interface and a Frequency Domain Perturbation study for the Magnetic Fields and Solid Mechanics interfaces. The coupling between the two interfaces is automatically considered by using the multiphysics coupling feature referred to as Lorentz Coupling.

Application Library path: `ACDC_Module/Motors_and_Actuators/magnetic_damping`

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 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.

5 Click **Add**.

6 Click **Study**.

7 In the **Select Study** tree, select **Empty Study**.

8 Click **Done**.

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
sigma	3.774e7[S/m]	3.774E7 S/m	Material conductivity
a_c	5e5[A]	5E5 A	Applied current on the wire

The Applied current will be used as a sweep parameter.

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.

2 In the **Geometry** toolbar, click **Block** to create a block for the simulation domain. Leave the default block size.

3 In the **Geometry** toolbar, click **Block** again to create a block for the cantilever beam.

Block 2 (blk2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Block 2 (blk2)**.

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

3 In the **Width** text field, type `0.9`.

4 In the **Depth** text field, type `0.025`.

5 In the **Height** text field, type `0.1`.

6 Locate the **Position** section. In the **y** text field, type `0.575`.

7 In the **z** text field, type `0.45`.

Finally, add a cylinder for the wire generating the static magnetic field.

Cylinder 1 (cyl1)

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

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

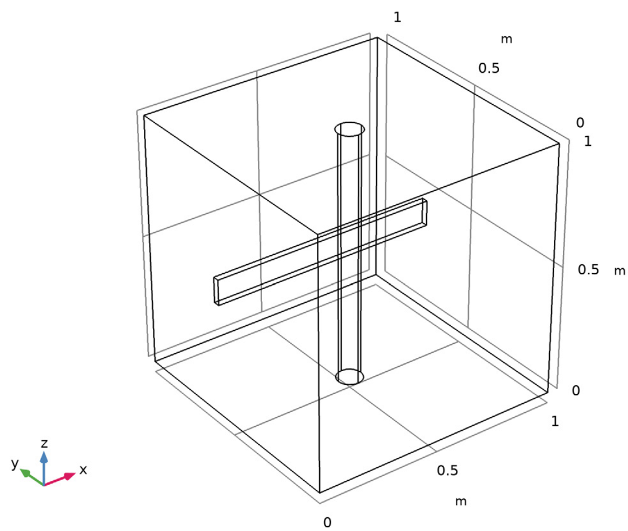
3 In the **Radius** text field, type `0.05`.

4 Locate the **Position** section. In the **x** text field, type `0.5`.

5 In the **y** text field, type `0.5`.

6 Click **Build All Objects**.

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



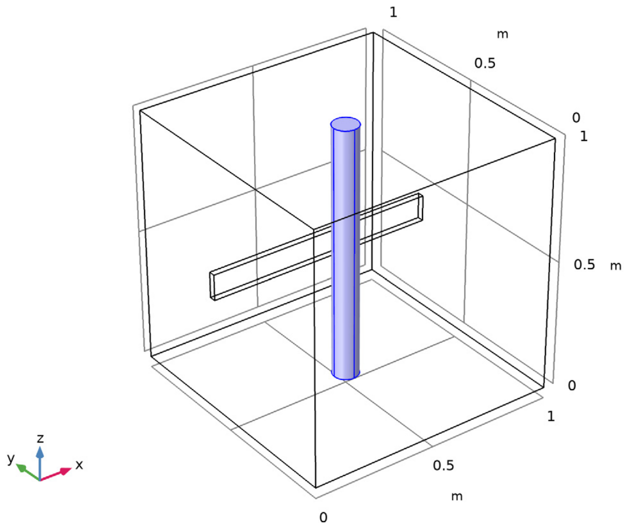
Add variables for the induced current density and body force on the cantilever beam.

MAGNETIC FIELDS (MF)

Coil 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Magnetic Fields (mf)** and choose the domain setting **Coil**.

2 Select Domain 3 only.



3 In the **Settings** window for **Coil**, locate the **Coil** section.

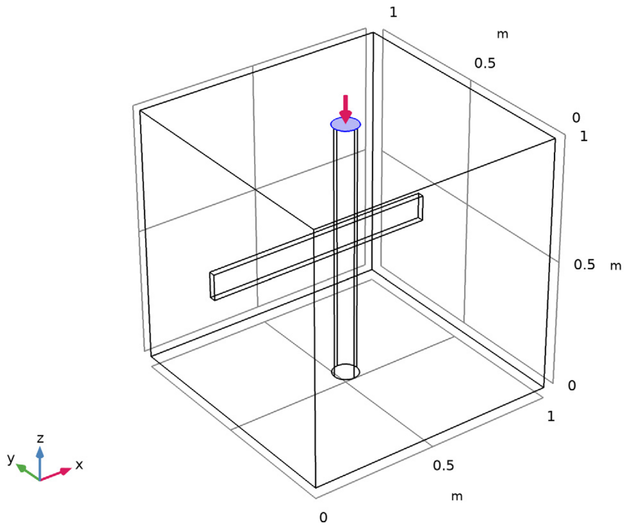
4 In the I_{coil} text field, type `a_c`.

5 In the **Model Builder** window, expand the **Coil 1** node.

Input 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Magnetic Fields (mf)> Coil 1>Geometry Analysis 1** node, then click **Input 1**.

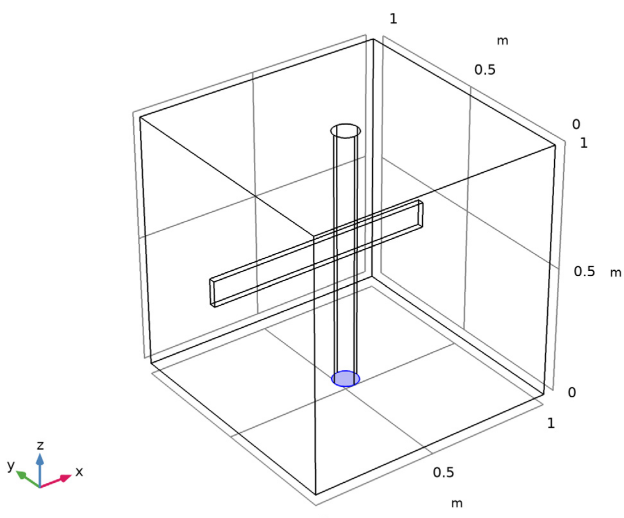
2 Select Boundary 14 only.



Output 1

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

2 Select Boundary 13 only.

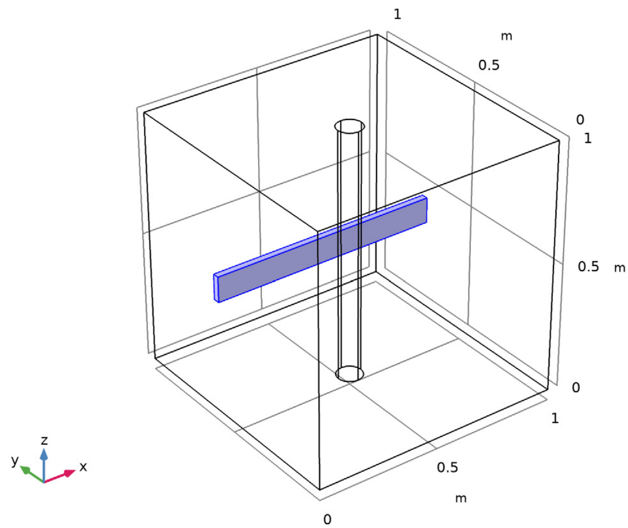


SOLID MECHANICS (SOLID)

The **Solid Mechanics** interface is active only on the cantilever beam.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Solid Mechanics (solid)**.

2 Select Domain 2 only.



Linear Elastic Material 1

Add a damping factor on **Linear Elastic Material Model 1**.

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Linear Elastic Material 1**.

Damping 1

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

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

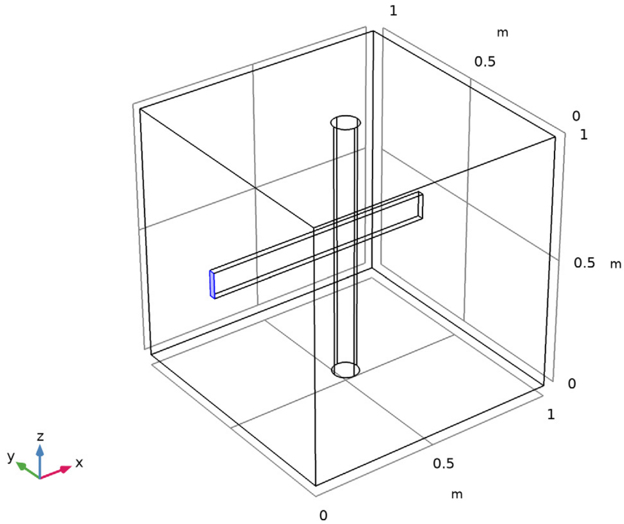
3 From the **Damping type** list, choose **Isotropic loss factor**.

4 From the η_s list, choose **User defined**. In the associated text field, type `0.1`.

Fixed Constraint 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.

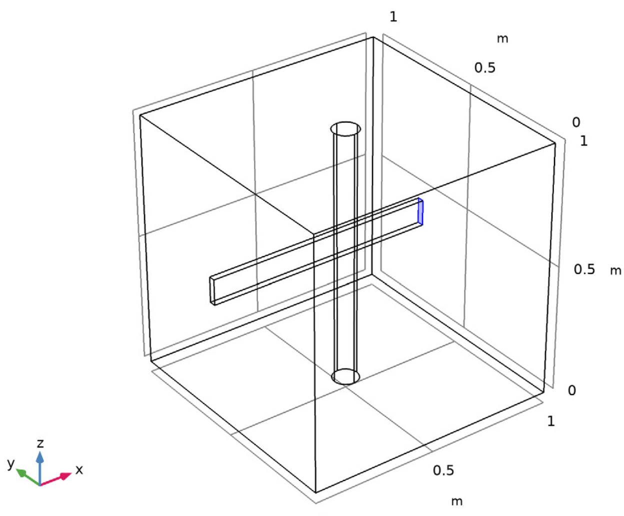
2 Select Boundary 5 only.



Boundary Load 1

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

2 Select Boundary 17 only.



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

4 Specify the $\mathbf{F}_A$ vector as

0	x
1e4	y
0	z

5 Right-click **Boundary Load 1** and choose **Harmonic Perturbation**.

ADD MATERIAL

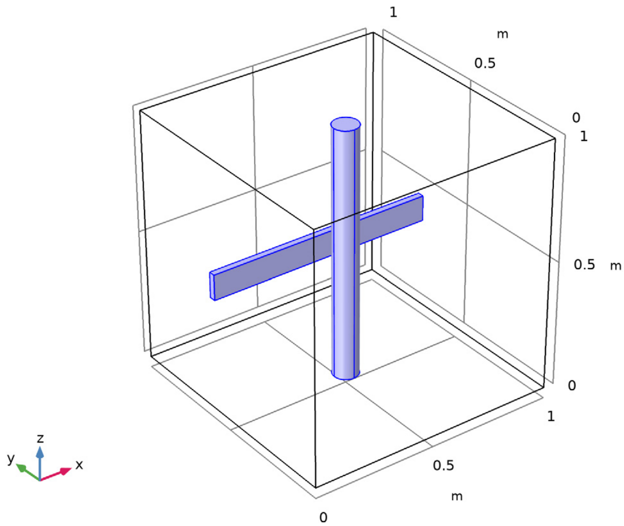
1. In the **Home** toolbar, click **Add Material** to open the **Add Material** window.
2. Go to the **Add Material** window.
3. In the tree, select **Built-in>Aluminum**.
4. Click **Add to Component** in the window toolbar.
5. In the tree, select **Built-in>Air**.
6. Click **Add to Component** in the window toolbar.
7. In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Aluminum (mat1)

1. In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Aluminum (mat1)**.

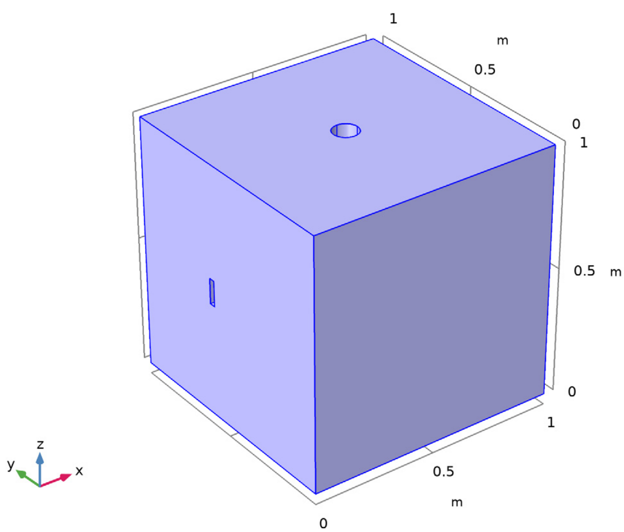
2 Select Domains 2 and 3 only.



Air (mat2)

1 In the **Model Builder** window, click **Air (mat2)**.

2 Select Domain 1 only.



Some artificial conductivity is needed for numerical stability of the AC magnetic simulation.

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

4 In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	10[S/m]	S/m	Basic

MULTIPHYSICS

Lorentz Coupling 1 (ltzc1)

In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Domain>Lorentz Coupling**.

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 From the **Element size** list, choose **Coarser**.

STUDY 1

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
a_c (Applied current on the wire)	0[A] 500000[A]	A

Add a **Coil Geometry Analysis** study step as the first step to compute the direction of the current in the wire.

Coil Geometry Analysis

In the **Study** toolbar, click **Study Steps** and choose **Other>Coil Geometry Analysis**.

Stationary

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

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

3 In the table, under **Solve for**, select **Magnetic Fields** only.

Frequency Domain Perturbation

1 In the **Study** toolbar, click **Study Steps** and choose **Frequency Domain>Frequency Domain Perturbation**.

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

3 In the **Frequencies** text field, type `range(5,0.5,50)`.

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.

Some adjustments to the default solver settings will improve the performance.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 3** node, then click **Fully Coupled 1**.

4 In the **Settings** window for **Fully Coupled**, locate the **General** section.

5 From the **Linear solver** list, choose **Direct**.

6 In the **Model Builder** window, click **Direct**.

7 In the **Settings** window for **Direct**, locate the **General** section.

8 From the **Solver** list, choose **PARDISO**.

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

RESULTS

In the **Model Builder** window, expand the **Results** node.

Study 1/Parametric Solutions 1 (5) (sol4)

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

2 Right-click **Results>Datasets>Study 1/Parametric Solutions 1 (sol4)** and choose **Duplicate**.

Selection

1 In the **Model Builder** window, right-click **Study 1/Parametric Solutions 1 (5) (sol4)** 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 Select Domain 2 only.

Magnetic Flux Density Norm (mf)

The first default plot group shows the magnetic field around a current carrying wire; compare with Figure 2. Give it a more descriptive name.

1 In the **Model Builder** window, under **Results** click **Magnetic Flux Density Norm (mf)**.
2 In the **Settings** window for **3D Plot Group**, type `DC Magnetic Flux Density Norm` in the **Label** text field.

Multislice 1

1 In the **Model Builder** window, expand the **Results>DC Magnetic Flux Density Norm** node, then click **Multislice 1**.
2 In the **Settings** window for **Multislice**, locate the **Expression** section.
3 From the **Expression evaluated for** list, choose **Static solution**.
4 In the **DC Magnetic Flux Density Norm** toolbar, click **Plot**.
5 Click the **Go to Default View** button in the **Graphics** toolbar.

DC Magnetic Flux Density Norm 1

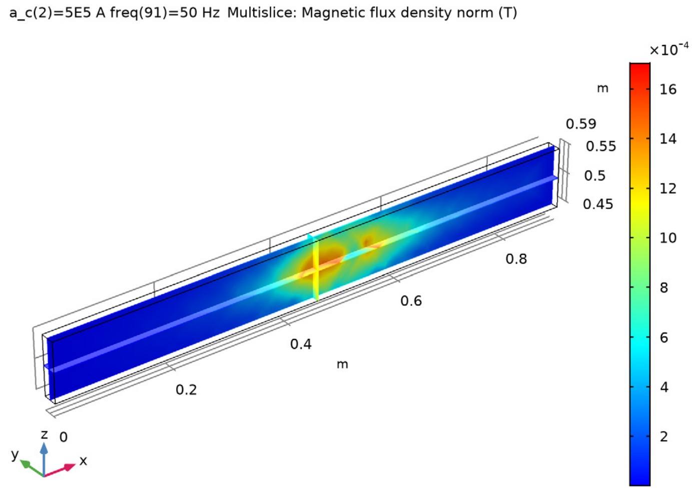
1 In the **Model Builder** window, right-click **DC Magnetic Flux Density Norm** and choose **Duplicate**.
2 In the **Settings** window for **3D Plot Group**, type `AC Magnetic Flux Density Norm` in the **Label** text field.
3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Parametric Solutions 1 (5) (sol4)**.

The second plot group shows the AC magnetic flux density. Improve it by plotting the data in the cantilever beam only.

Multislice 1

1 In the **Model Builder** window, expand the **Results>AC Magnetic Flux Density Norm** node, then click **Multislice 1**.
2 In the **Settings** window for **Multislice**, locate the **Expression** section.
3 From the **Expression evaluated for** list, choose **Harmonic perturbation**.
4 In the **AC Magnetic Flux Density Norm** toolbar, click **Plot**.

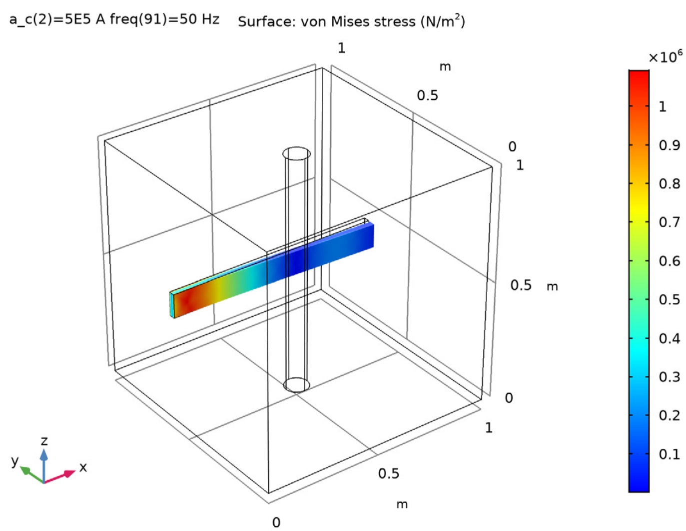
5 Click the **Zoom Extents** button in the **Graphics** toolbar.



The plot now shows the magnitude of the AC magnetic flux density in the beam only.

Stress (solid)

The third plot shows the von Mises stress.



DC Magnetic Flux Density Norm 1

1 In the **Model Builder** window, right-click **DC Magnetic Flux Density Norm** and choose **Duplicate**.

   Add a plot of the AC currents in the beam.

2 In the **Settings** window for **3D Plot Group**, type `AC Electric Current Density` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Parametric Solutions 1 (5) (sol4)**.

Multislice 1

1 In the **Model Builder** window, expand the **DC Magnetic Flux Density Norm 1** node, then click **Results>AC Electric Current Density>Multislice 1**.

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

3 From the **Expression evaluated for** list, choose **Harmonic perturbation**.

4 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Magnetic Fields>Currents and charge>mf.normJ - Current density norm - A/m²**.

Streamline 1

1 In the **Model Builder** window, right-click **AC Electric Current Density** and choose **Streamline**.

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

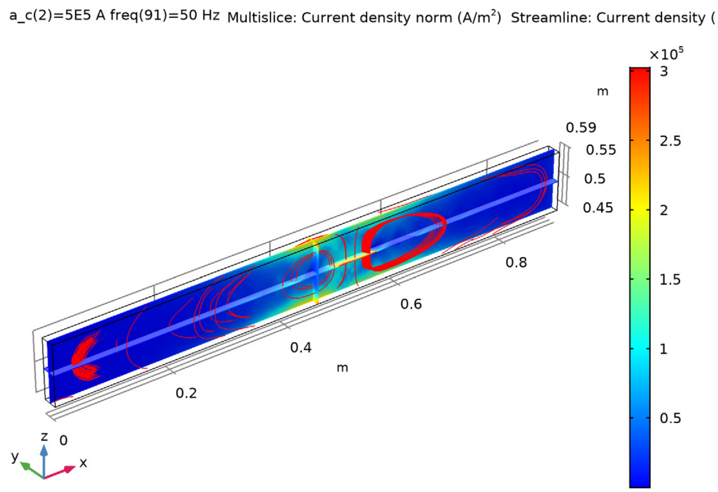
3 From the **Expression evaluated for** list, choose **Harmonic perturbation**.

4 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.Jz - Current density (spatial frame)**.

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

6 In the **AC Electric Current Density** toolbar, click **Plot**.

7 Click the **Go to Default View** button in the **Graphics** toolbar.



The AC eddy currents circulate within the beam.

Finish by plotting the RMS displacement of the tip of the beam as a function of frequency (Figure 3).

1D Plot Group 6

1. In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2. In the **Settings** window for **1D Plot Group**, type `RMS Displacement vs Frequency` in the **Label** text field.
3. Locate the **Data** section. From the **Dataset** list, choose **Study 1/ Parametric Solutions 1 (4) (sol4)**.

Point Graph 1

1. Right-click **RMS Displacement vs Frequency** and choose **Point Graph**.
2. Select Point 18 only.
3. In the **Settings** window for **Point Graph**, click **Replace Expression** in the upper-right corner of the **y-axis data** section. From the menu, choose **Component 1>Solid Mechanics> Displacement>solid.disp_rms - Displacement, RMS - m**.
4. Click to expand the **Legends** section. Select the **Show legends** check box.
5. In the **RMS Displacement vs Frequency** toolbar, click **Plot**.

6 Click the **y-Axis Log Scale** button in the **Graphics** toolbar.

Compare the resulting plot with that shown in Figure 3.