Model created in COMSOL Multiphysics 6.4

Piezomagnetic Cell Rover

Introduction

A conventional antenna relies on electromagnetic resonance, which means that it must be of a size comparable to the electromagnetic wavelength. This has been a difficulty for the development of smaller, millimeter-sized antennas. This tutorial is intended to show how to model a miniaturized magnetostrictive antenna developed for operating inside living cells, based on the so-called Cell Rover design presented in Ref. 1.

The Cell Rover antenna is ideal for operating inside living systems because of its small size and its relatively low resonance frequency at around 4.5 MHz. To understand the effect of mass loading and viscous damping, the device is studied in air. By exiting the antenna with an AC magnetic field applied along its length, the stress of the antenna, the magnetic flux density, the current density, and the displacement of the tip of the device are investigated.

Model Definition

The geometry of the Cell Rover is shown in Figure 1. The model is solved in 3D, and consists of a magnetostrictive antenna made of metglas placed inside a cylinder filled with air. The antenna is excited using a uniform magnetic background field. A relatively fine mesh is used for the antenna, while a coarser mesh is used for the surrounding cylinder. The computational mesh can be seen in Figure 2.

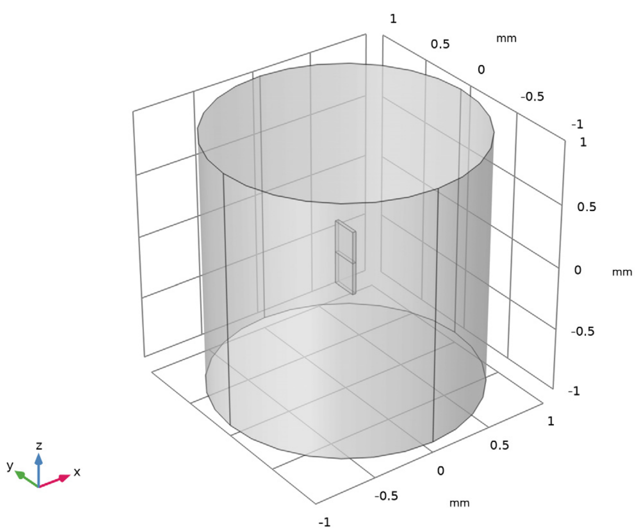


Figure 1: The model geometry.

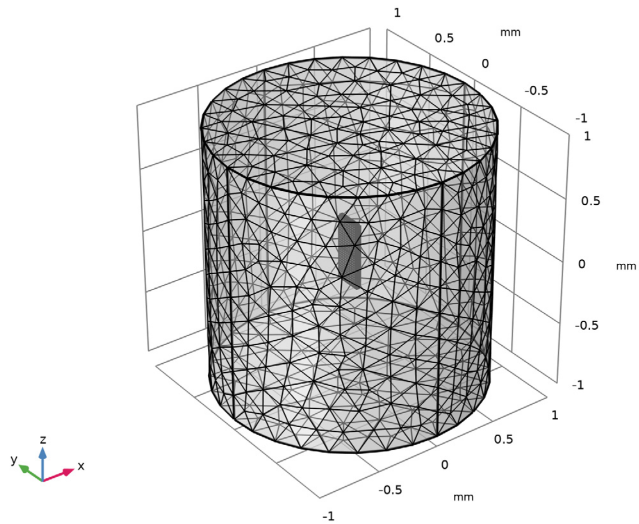


Figure 2: The mesh used in the model.

The implementation makes use of a predefined multiphysics interface available in COMSOL, called Piezomagnetism. Selecting this interface in the Model Wizard adds the Structural Mechanics and Magnetic Fields interfaces together with the corresponding multiphysics coupling feature, Piezomagnetism.

PARAMETERS

The model uses parameters for the excitation, the quality factor, and the metglas material properties, as given in Table 1. The properties of air are taken from the built-in COMSOL material library.

TABLE 1: MODEL PARAMETERS.

Name	Value	Description
E	152e9[Pa]	Young's modulus
nu	0.22	Poisson's ratio
d	1.19e-8[m/A]	Magnetostrictivity
sigma	7.25e5[S/m]	Conductivity
mu	300	Relative permeability
rho	7900[kg/m^3]	Density

TABLE 1: MODEL PARAMETERS.

Name	Value	Description
Hac	3[Oe]	Magnetic field
Q	497	Quality factor

Results and Discussion

Two studies are performed: a Frequency Domain study and an Adaptive Frequency Sweep study. The plots shown in Figures 3-5 show the solution from the Frequency Domain study at the resonance frequency of 4.535 MHz, while Figure 6 compares the solutions from both studies.

Figure 3 shows the magnetic flux density norm and the magnetic flux density streamlines in a slice of the antenna.

Figure 4 shows the volume plot of the stress distribution in the antenna.

Figure 5 displays the volume plot of the current density norm in the antenna.

Figure 6 compares the resulting tip displacements of the first and the second study. The Adaptive Frequency Sweep study has a higher resolution output than the Frequency Domain study. The maximum displacement occurs at the resonance frequency of 4.535 MHz and has a value of 17.1 nm.

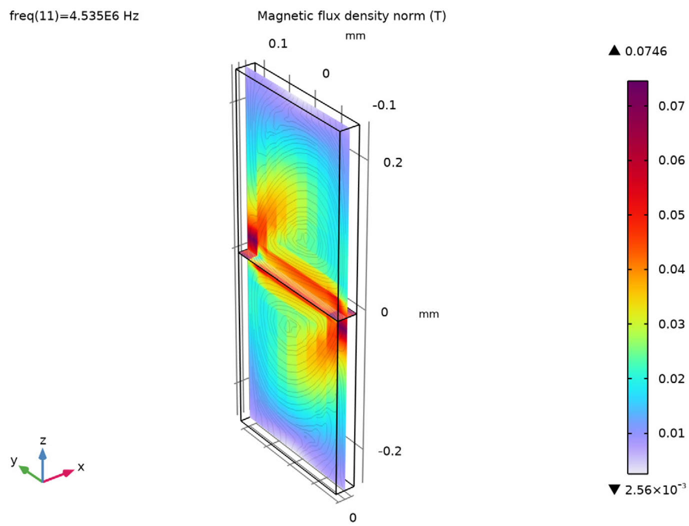


Figure 3: Magnetic flux density norm at f = 4.535 MHz.

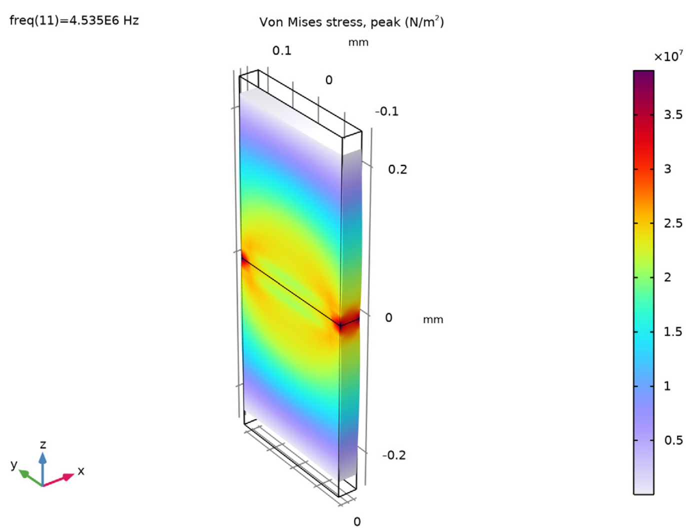


Figure 4: Stress distribution at f = 4.535 MHz.

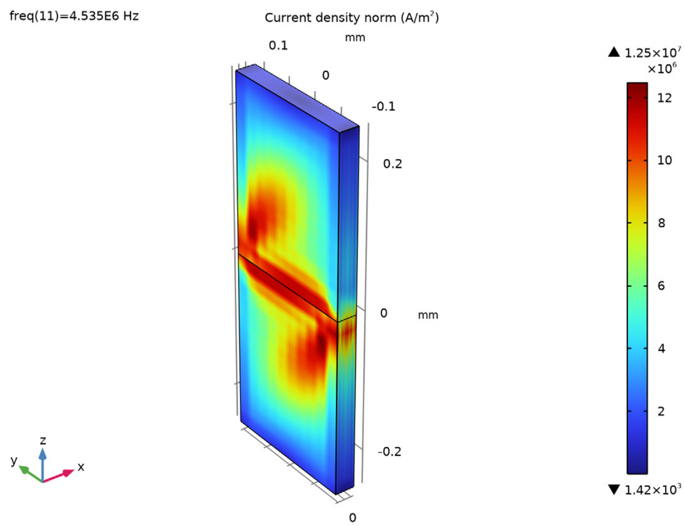


Figure 5: Current density norm at f = 4.535 MHz.

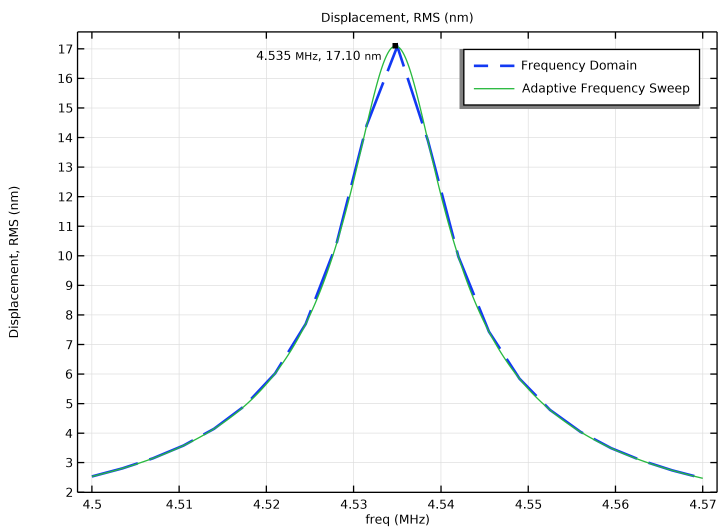


Figure 6: The tip displacement versus frequency for the Frequency Domain study and the Adaptive Frequency Sweep study.

Reference

1. B. Joy, Y. Cai, D.C. Bono, and others, "Cell Rover — a miniaturized magnetostrictive antenna for wireless operation inside living cells," *Nat. Commun.*, vol. 13, p. 5210, 2022.

Article available at: www.nature.com/articles/s41467-022-32862-4.

The *Piezomagnetic Cell Rover* tutorial model uses the device dimensions and material properties of Metglas® 2826 MB as given by the article, and the magnetic field strength and quality factor are as given for the studies done in air. The model reproduces the resonance frequency in air, as obtained from the article.

Application Library path: `ACDC_Module/Electromagnetics_and_Mechanics/piezomagnetic_cell_rover`

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** > **Electromagnetics and Mechanics** > **Magnetostriction** > **Piezomagnetism**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies** > **Frequency Domain**.

6 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 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `piezomagnetic_cell_rover_parameters.txt`.

The imported parameters are used for the excitation, the quality factor, and the metglas material properties.

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**.

Cell Rover

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

2 In the **Settings** window for **Block**, type `Cell Rover` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `0.5`.

4 In the **Depth** text field, type `0.2`.

5 In the **Height** text field, type `0.028`.

6 Locate the **Position** section. From the **Base** list, choose **Center**.

7 Locate the **Axis** section. From the **Axis type** list, choose **x-axis**.

8 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	0.25

9 Find the **Layer position** subsection. Select the **Left** checkbox.

10 Clear the **Bottom** checkbox.

11 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.

12 Click **Build Selected**.

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 **Height** text field, type `2`.

4 Locate the **Position** section. In the **z** text field, type `-1`.

5 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.
6 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**.

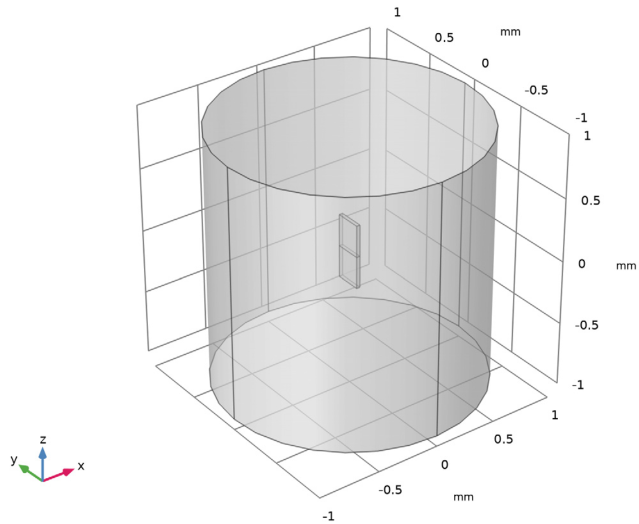
Air

1 In the **Geometry** toolbar, click **Selections** and choose **Complement Selection**.
2 In the **Settings** window for **Complement Selection**, type `Air` in the **Label** text field.
3 Locate the **Input Entities** section. Click **Add**.
4 In the **Add** dialog, select **Cell Rover** in the **Selections to invert** list.
5 Click **OK**.

Exterior Boundaries

1 In the **Geometry** toolbar, click **Selections** and choose **Adjacent Selection**.
2 In the **Settings** window for **Adjacent Selection**, type `Exterior Boundaries` in the **Label** text field.
3 Locate the **Input Entities** section. Click **Add**.
4 In the **Add** dialog, select **Cylinder 1** in the **Input selections** list.
5 Click **OK**.
6 In the **Geometry** toolbar, click **Build All**.
7 Click the **Transparency** button in the **Graphics** toolbar.
8 Click the **Zoom Extents** button in the **Graphics** toolbar.

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



DEFINITIONS

Next, create a selection for a point of interest that can used for the stored solution later.

Explicit 1

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

2 In the **Settings** window for **Explicit**, locate the **Input Entities** section.

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

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type 3 in the **Selection** text field.

6 Click **OK**.

Integration 1 (intop1)

1 In the **Model Builder** window, expand the **Definitions** node.

2 Right-click **Definitions** and choose **Nonlocal Couplings** > **Integration**.

3 In the **Settings** window for **Integration**, locate the **Source Selection** section.

4 From the **Geometric entity level** list, choose **Point**.

5 Select Point 3 only.

ADD MATERIAL

1 In the **Materials** 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 the **Add to Component** button in the window toolbar.

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

MATERIALS

Metglas

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Metglas` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Cell Rover**.

4 Click to expand the **Material Properties** section. In the **Material properties** tree, select **Solid Mechanics** > **Linear Elastic Material** > **Young's Modulus and Poisson's Ratio**.

5 Click **Add to Material**.

6 Locate the **Material Contents** section. In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Young's modulus	E	`root.E`	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	`root.nu`	1	Young's modulus and Poisson's ratio
Electric conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	`sigma`	S/m	Basic
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	1	1	Basic
Compliance matrix, Voigt notation	{sH11, sH12, sH22, sH13, sH23, sH33, sH14, sH24, sH34, sH44, sH15, sH25, sH35, sH45, sH55, sH16, sH26, sH36, sH46, sH56, sH66} ; sHij = sHji	`{1/E, -nu/E, 1/E, -nu/E, -nu/E, 1/E, 0, 0, 0, (1+nu)/E, 0, 0, 0, 0, (1+nu)/E, 0, 0, 0, 0, 0, (1+nu)/E}`	1/Pa	Strain-magnetization form

Property	Variable	Value	Unit	Property group
Piezomagnetic coupling matrix, Voigt notation	{dHT11, dHT21, dHT31, dHT12, dHT22, dHT32, dHT13, dHT23, dHT33, dHT14, dHT24, dHT34, dHT15, dHT25, dHT35, dHT16, dHT26, dHT36}	{d, 0, 0, 0, d, 0, 0, 0, d, 0, 0, 0, 0, 0, 0, 0, 0, 0}	m/A	Strain-magnetization form
Relative permeability	murT_iso ; murTii = murT_iso, murTij = 0	mu	1	Strain-magnetization form
Density	rho	root.rho	kg/m³	Basic

The warning sign shown in the **Material** node will disappear when the physics is set up.

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 **Background Field** section.

3 From the **Solve for** list, choose **Reduced field**.

4 From the **Background field specification** list, choose **Uniform magnetic flux density**.

5 Specify the $\mathbf{B}_b$ vector as

Hac*mu0_const	z

6 Click to expand the **Discretization** section. From the **Magnetic vector potential** list, choose **Linear**.

Ampère's Law, Piezomagnetic 1

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Magnetic Fields (mf)** click **Ampère's Law, Piezomagnetic 1**.

2 In the **Settings** window for **Ampère's Law, Piezomagnetic**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Cell Rover**.

Gauge Fixing for A-Field 1

In the **Physics** toolbar, click **Domains** and choose **Gauge Fixing for A-Field**.

External Magnetic Vector Potential 1

1 In the **Physics** toolbar, click **Boundaries** and choose **External Magnetic Vector Potential**.

The **External Magnetic Vector Potential** feature enforces the chosen background field on the selected boundaries.

2 In the **Settings** window for **External Magnetic Vector Potential**, locate the **Boundary Selection** section.

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

SOLID MECHANICS (SOLID)

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

2 In the **Settings** window for **Solid Mechanics**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Cell Rover**.

Fixed Constraint 1

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

2 Drag and drop below **Initial Values 1**.

3 In the **Settings** window for **Fixed Constraint**, locate the **Boundary Selection** section.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog, type 10 in the **Selection** text field.

6 Click **OK**.

Piezomagnetic Material 1

1 In the **Model Builder** window, click **Piezomagnetic Material 1**.

2 In the **Settings** window for **Piezomagnetic Material**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Cell Rover**.

Mechanical Damping 1

1 In the **Physics** toolbar, click **Attributes** and choose **Damping**.
2 In the **Settings** window for **Mechanical Damping**, locate the **Damping Settings** section.
3 From the η_s list, choose **User defined**. In the associated text field, type `1/Q`.

   To make sure that the Air material has updated its properties, click on the **Material** node to confirm that the warning has disappeared.

MESH 1

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.
2 In the **Settings** window for **Mesh**, locate the **Sequence Type** section.
3 From the list, choose **User-controlled mesh**.

Swept 1

1 In the **Mesh** toolbar, click **Swept**.
2 Drag and drop below **Size 2**.
3 In the **Settings** window for **Swept**, locate the **Domain Selection** section.
4 From the **Geometric entity level** list, choose **Domain**.
5 From the **Selection** list, choose **Cell Rover**.

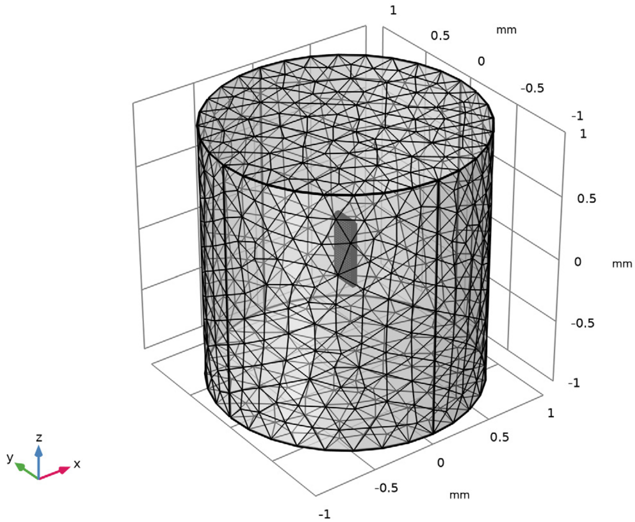
Distribution 1

Right-click **Swept 1** and choose **Distribution**.

Size 1

1 In the **Model Builder** window, right-click **Swept 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.
5 Select the **Maximum element size** checkbox. In the associated text field, type `0.02`.
6 In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.

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



The mesh should look like the figure above.

8 In the **Mesh** toolbar, click **Plot**.

RESULTS

Mesh 1

Click the **Show Grid** button in the **Graphics** toolbar.

Filter 1

1 Right-click **Mesh 1** and choose **Filter**.

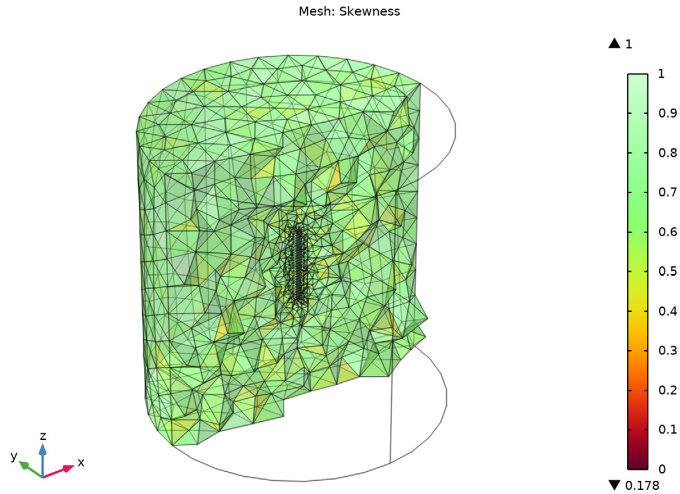
2 In the **Settings** window for **Filter**, locate the **Element Selection** section.

3 In the **Logical expression for inclusion** text field, type `y>0`.

Mesh Plot 1

1 In the **Model Builder** window, under **Results** click **Mesh Plot 1**.

2 In the **Mesh Plot 1** toolbar, click **Plot**.



The mesh plot should look like the figure above.

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 `range(4.5[MHz],(4.57[MHz]-(4.5[MHz]))/20, 4.57[MHz])`.

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

5 In the **Settings** window for **Study**, type `Study 1 Frequency Domain` in the **Label** text field.

Solution 1 (sol1)

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

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

3 In the **Model Builder** window, expand the **Study 1 Frequency Domain** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 1** node.

4 Right-click **Study 1 Frequency Domain** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 1** and choose **Fully Coupled**.

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

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

7 In the **Study** toolbar, click **= Compute**.

RESULTS

Study 1 Frequency Domain/Solution 1 (sol1)

In the **Model Builder** window, expand the **Results > Datasets** node, then click **Study 1 Frequency Domain/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 **Cell Rover**.

Magnetic Flux Density Norm (mf)

1 In the **Model Builder** window, under **Results** click **Magnetic Flux Density (mf)**.

2 In the **Settings** window for **3D Plot Group**, type `Magnetic Flux Density Norm (mf)` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter value (freq (Hz))** list, choose **4.535E6**.

Multislice 1

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

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

3 Find the **y-planes** subsection. From the **Entry method** list, choose **Number of planes**.

4 In the **Planes** text field, type `0`.

Streamline Multislice 1

1 In the **Model Builder** window, click **Streamline Multislice 1**.

2 In the **Settings** window for **Streamline Multislice**, locate the **Multiplane Data** section.

3 Find the **y-planes** subsection. From the **Entry method** list, choose **Number of planes**.

4 In the **Planes** text field, type `0`.

Magnetic Flux Density Norm (mf)

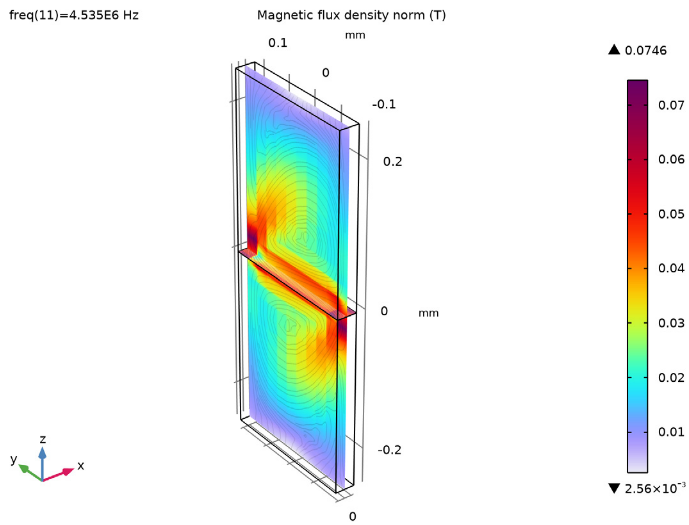
1 Click the **Zoom Extents** button in the **Graphics** toolbar.

2 In the **Model Builder** window, click **Magnetic Flux Density Norm (mf)**.

3 In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.

4 From the **View** list, choose **New view**.

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



The plot shows the magnitude of the magnetic flux density norm in the antenna.

Stress (solid)

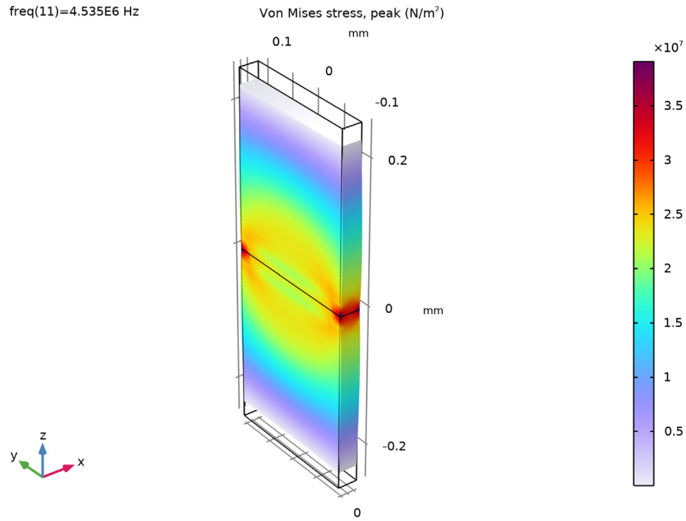
1 In the **Model Builder** window, click **Stress (solid)**.

2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Parameter value (freq (Hz))** list, choose **4.535E6**.

4 Locate the **Plot Settings** section. From the **View** list, choose **View 3D 2**.

5 In the **Stress (solid)** toolbar, click **Plot**.



This plot shows the peak von Mises stress in the antenna.

Current Density Norm

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

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

3 Locate the **Data** section. From the **Parameter value (freq (Hz))** list, choose **4.535E6**.

4 Locate the **Plot Settings** section. From the **View** list, choose **View 3D 2**.

5 From the **Frame** list, choose **Spatial (x, y, z)**.

6 Locate the **Color Legend** section. Select the **Show maximum and minimum values** checkbox.

Volume 1

1 Right-click **Current Density Norm** and choose **Volume**.

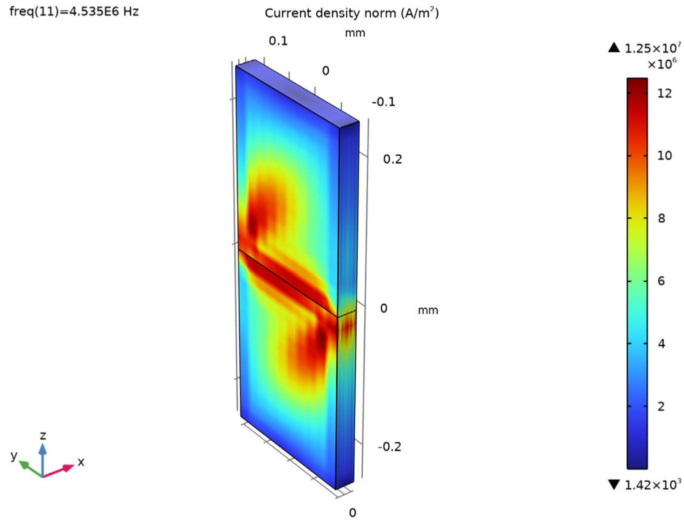
2 In the **Settings** window for **Volume**, locate the **Expression** section.

3 In the **Expression** text field, type `mf.normJ`.

Current Density Norm

1 In the **Model Builder** window, click **Current Density Norm**.

2 In the **Current Density Norm** toolbar, click **Plot**.



Finish by plotting the displacement of the tip of the antenna as a function of frequency.

Tip Displacement

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

2 In the **Settings** window for **1D Plot Group**, type `Tip Displacement` in the **Label** text field.

Point Graph 1

1 Right-click **Tip Displacement** and choose **Point Graph**.

2 In the **Settings** window for **Point Graph**, locate the **Selection** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog, type `3` in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

7 In the **Expression** text field, type `solid.disp_rms`.

8 From the **Unit** list, choose **nm**.

9 Locate the **x-Axis Data** section. From the **Unit** list, choose **MHz**.

10 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

11 From the **Width** list, choose **2**.

12 Click to expand the **Legends** section. Select the **Show legends** checkbox.

13 From the **Legends** list, choose **Manual**.

14 In the table, enter the following settings:

Legends
`Frequency Domain`

Tip Displacement

1 In the **Model Builder** window, click **Tip Displacement**.

2 In the **Tip Displacement** toolbar, click **Plot**.

Now add an Adaptive Frequency Sweep study to the model for comparison.

ADD STUDY

1 In the **Study** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Empty Study**.

4 Click the **Add Study** button in the window toolbar.

STUDY 2

Step 1: Adaptive Frequency Sweep

1 In the **Study** toolbar, click **More Study Steps** and choose **Frequency Domain** > **Adaptive Frequency Sweep**.

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

3 In the **Frequencies** text field, type `range(4.5[MHz],(4.57[MHz]-(4.5[MHz]))/2000,4.57[MHz])`.

4 From the **AWE expression type** list, choose **User controlled**.

5 In the table, enter the following settings:

Asymptotic waveform evaluation (AWE) expressions
`comp1.intop1(solid.disp_rms)`

Next, set the output selections to a point of interest to reduce the size of the stored solution.

6 Click to expand the **Store in Output** section. In the table, enter the following settings:

Interface	Output	Selection
Magnetic Fields (mf)	None	
Solid Mechanics (solid)	Selection	

7 Under **Selections**, click **Add**.

8 In the **Add** dialog, select **Explicit 1** in the **Selections** list.

9 Click **OK**.

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

11 In the **Settings** window for **Study**, type `Study 2 Adaptive Frequency Sweep` in the **Label** text field.

12 Locate the **Study Settings** section. Clear the **Generate default plots** checkbox.

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

RESULTS

Point Graph 2

1 In the **Model Builder** window, right-click **Tip Displacement** and choose **Point Graph**.

2 In the **Settings** window for **Point Graph**, locate the **Data** section.

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

4 Locate the **Selection** section. Click **Paste Selection**.

5 In the **Paste Selection** dialog, type 3 in the **Selection** text field.

6 Click **OK**.

7 In the **Settings** window for **Point Graph**, locate the **y-Axis Data** section.

8 In the **Expression** text field, type `solid.disp_rms`.

9 From the **Unit** list, choose **nm**.

10 Click to expand the **Title** section. From the **Title type** list, choose **None**.

11 Locate the **x-Axis Data** section. From the **Unit** list, choose **MHz**.

12 Locate the **Legends** section. Select the **Show legends** checkbox.

13 From the **Legends** list, choose **Manual**.

14 In the table, enter the following settings:

Legends
`Adaptive Frequency Sweep`

Graph Marker 1

1. Right-click **Point Graph 2** and choose **Graph Marker**.
2. In the **Settings** window for **Graph Marker**, locate the **Display** section.
3. From the **Display** list, choose **Max**.
4. Locate the **Text Format** section. Select the **Show x-coordinate** checkbox.
5. Select the **Include unit** checkbox.
6. In the **Precision** text field, type 4.
7. Click to expand the **Coloring and Style** section. From the **Anchor point** list, choose **Upper right**.
8. In the **Tip Displacement** toolbar, click **Plot**.