Model created in COMSOL Multiphysics 6.4

Magnetic Frill

Introduction

Feeding antennas with proper signals can be difficult. The signal is often described as a voltage, and voltages are not well defined in electromagnetic wave formulations. There are several tricks to model voltage generators in such situations, and one is the *magnetic frill*. This example shows the basic steps of defining a magnetic frill voltage generator for a dipole antenna, and it also compares the resulting antenna impedance with known results.

Model Definition

Magnetic frills can only be defined using Electromagnetic Waves interface, which is based on the time-harmonic Faraday's law.

$$\nabla \times \mathbf{E} = -j\omega\mathbf{B}$$

Although there are no magnetic charges, it is possible to mathematically define a current of magnetic charges, called a magnetic current. This current enters the right-hand side of Faraday's law in the same manner as the ordinary current enters the right-hand side of Ampère's law. Similar to the ordinary current density that has the unit A/m^2, the magnetic current density has the unit V/m^2.

A closed loop of magnetic current therefore has the unit V and represents a voltage generator for the surface closed by the loop. In this example, the loop is located around a thin straight wire and acts as a voltage source at the center of the wire. This is a dipole antenna fed by a voltage signal in the center.

The current through the wire is measured with another loop, along which a line integral of the **H**-field is specified.

$$\int \mathbf{H} \cdot \mathbf{dl} = I$$

Note that this loop and the magnetic current loop must be two different loops.

The antenna is placed in a spherical air domain surrounded by a perfectly matched layer (PML) serving to absorb the radiation from the antenna with a minimum of reflection.

Results and Discussion

The dipole antenna is fed with a voltage signal of 1 V, and from the measured current it is possible to extract the impedance. Taken from Ref. 1, the impedance and dimensions of a typical dipole antenna are shown in the table below. The dimensions are given in terms of the wavelength, λ.

WAVELENGTH	LENGTH	RADIUS
0.3	0.47λ	0.005λ

The impedance from the COMSOL Multiphysics model is $76.00 + 15.98i$, which agrees well with the results from Ref. 1.

Reference

1. C.A. Balanis, *Advanced Engineering Electromagnetics*, John Wiley & Sons, 1989.

Application Library path: `RF_Module/Antennas/magnetic_frill`

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 **Radio Frequency** > **Electromagnetic Waves, Frequency Domain (emw)**.

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 In the table, enter the following settings:

Name	Expression	Value	Description
`lda0`	`0.3[m]`	0.3 m	`Wavelength`
`l`	`0.47`	0.47	`Scale factor`
`k`	`0.005`	0.005	`Scale factor`
`L`	`l*lda0`	0.141 m	`Dipole length`
`r_wire`	`k*lda0`	0.0015 m	`Wire radius`
`f0`	`1/lda0/sqrt(epsilon0_const* mu0_const)`	9.9931E8 1/s	`Frequency`

STUDY 1

Step 1: Frequency Domain

Define the study frequency ahead of performing any frequency-dependent operation such as building mesh. The physics-controlled mesh uses the specified frequency value.

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 `f0`.

GEOMETRY 1

Work Plane 1 (wp1)

1. In the **Geometry** toolbar, click **Work Plane**.
2. In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.
3. From the **Plane** list, choose **yz-plane**.
4. Click **Go to Plane Geometry**.

Work Plane 1 (wp1) > Rectangle 1 (r1)

1. In the **Work Plane** toolbar, click **Rectangle**.
2. In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3. In the **Width** text field, type `r_wire`.
4. In the **Height** text field, type `L`.
5. Locate the **Position** section. In the **yw** text field, type `-L/2`.

Work Plane 1 (wp1) > Rectangle 2 (r2)

1. In the **Work Plane** toolbar, click **Rectangle**.

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

3 In the **Width** text field, type `0.3`.

4 In the **Height** text field, type `0.6`.

5 Locate the **Position** section. In the **xw** text field, type `-0.3`.

6 In the **yw** text field, type `-0.3`.

Work Plane 1 (wp1) > Circle 1 (c1)

1 In the **Work Plane** toolbar, click **Circle**.

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

3 In the **Radius** text field, type `0.3`.

Work Plane 1 (wp1) > Circle 2 (c2)

1 In the **Work Plane** toolbar, click **Circle**.

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

3 In the **Radius** text field, type `0.2`.

Work Plane 1 (wp1) > Difference 1 (dif1)

1 In the **Work Plane** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the objects **c1** and **c2** only.

Alternatively, you can select all objects and remove `r1` and `r2` from the selection list.

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

4 Click to select the **Activate Selection** toggle button for **Objects to subtract**.

5 Select the objects **r1** and **r2** only.

Work Plane 1 (wp1) > Line Segment 1 (ls1)

1 In the **Work Plane** 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 Locate the **Starting Point** section. In the **xw** text field, type `0.2`.

6 Locate the **Endpoint** section. In the **xw** text field, type `0.3`.

This line segment simplifies meshing.

Work Plane 1 (wp1) > Point 1 (pt1)

1 In the **Work Plane** toolbar, click **Point**.

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

3 In the **xw** text field, type `r_wire+0.001`.

Work Plane 1 (wp1) > Point 2 (pt2)

1 In the **Work Plane** toolbar, click **Point**.

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

3 In the **xw** text field, type `0.01`.

4 In the **Work Plane** toolbar, click **Build All**.

Revolve 1 (rev1)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** right-click **Work Plane 1 (wp1)** and choose **Revolve**.

2 In the **Settings** window for **Revolve**, locate the **Revolution Angles** section.

3 Click the **Angles** button.

4 In the **End angle** text field, type `-90`.

5 Click **Build All Objects**.

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

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**.

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

DEFINITIONS

Integration 1 (intop1)

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

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

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

4 Select Edge 20 only.

Variables 1

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

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

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

4 Select Edge 20 only.

5 Locate the **Variables** section. In the table, enter the following settings:

Name	Expression	Unit	Description
intcpl_source_I	4*(emw.Hx*t1x+emw.Hy*t1y)	A/m	

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
I	intop1(intcpl_source_I)	A	Current
Z	1/I	1/A	Impedance

Perfectly Matched Layer 1 (pml1)

1 In the **Definitions** toolbar, click **Perfectly Matched Layer**.

2 Select Domains 1 and 3 only.

3 In the **Settings** window for **Perfectly Matched Layer**, locate the **Geometry** section.

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

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.

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

Perfect Magnetic Conductor 1

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

2 Select Boundaries 1, 2, 4, 5, 9, and 10 only.

Magnetic Current 1

1 In the **Physics** toolbar, click **Edges** and choose **Magnetic Current**.

2 Select Edge 21 only.

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

4 In the I_m text field, type 1.

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 **Coarse**.

4 Click **Build All**.

STUDY 1

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

RESULTS

Electric Field (emw)

The default plot shows the electric field on slices through the geometry.

Visualize the electric field around the antenna by modifying this plot as follows:

Delete the Multislice plot.

Multislice 1

1 In the **Model Builder** window, expand the **Electric Field (emw)** node.

2 Right-click **Multislice 1** and choose **Delete**.

Electric Field (emw)

Add two Slice plots.

Slice 1

1 Right-click **Electric Field (emw)** and choose **Slice**.

2 In the **Settings** window for **Slice**, locate the **Plane Data** section.

3 From the **Plane** list, choose **XY-planes**.

4 From the **Entry method** list, choose **Coordinates**.

5 Click to expand the **Range** section. Select the **Manual color range** checkbox.

6 In the **Maximum** text field, type 20.

7 Select the **Manual data range** checkbox.

8 In the **Maximum** text field, type 20.

Slice 2

1 Right-click **Electric Field (emw)** and choose **Slice**.

2 In the **Settings** window for **Slice**, locate the **Plane Data** section.

3 From the **Entry method** list, choose **Coordinates**.

4 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Slice 1**.

Transparency 1

Right-click **Slice 2** and choose **Transparency**.

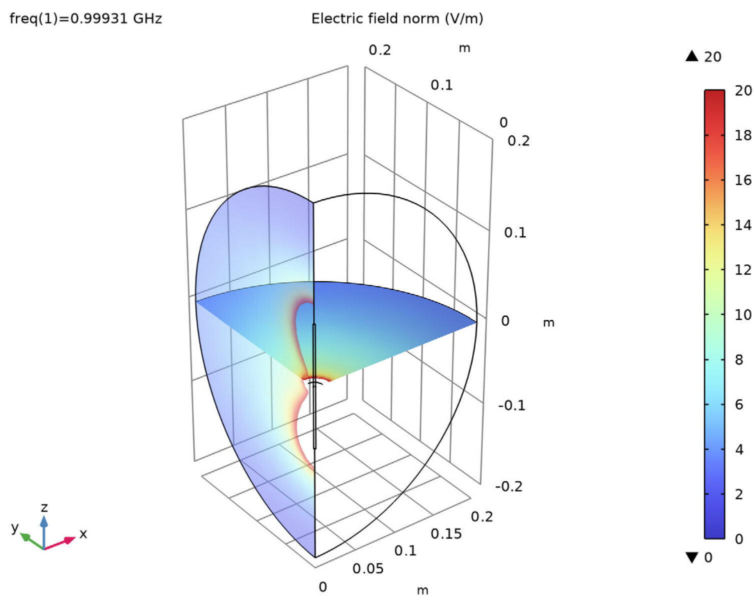
Electric Field (emw)

1 In the **Settings** window for **3D Plot Group**, click to expand the **Title** section.

2 From the **Title type** list, choose **Manual**.

3 In the **Title** text area, type `Electric field norm (V/m)`.

4 In the **Electric Field (emw)** toolbar, click **Plot**.



Global Evaluation 1

1 In the **Results** toolbar, click **Global Evaluation**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`Z`	`1/A`	`Impedance`

4 Click = **Evaluate**.