Created in COMSOL Multiphysics 6.2

Car Windshield Antenna Effect on a Cable Harness

Introduction

This example simulates an FM antenna printed on the rear windshield of a vehicle. The simulation computes the far-field radiation pattern of the antenna and the electric fields on an interior cable harness.

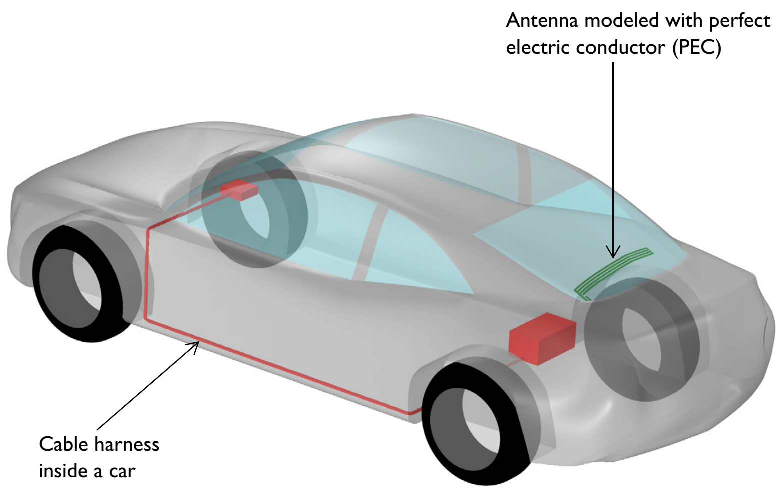


Figure 1: A simplified car model consisting of a metallic body, lossy tires, tire rims, thin dielectric windshields, a printed antenna, and a cable harness connected to electronic component enclosures. The surrounding air domain and ground plane are not included in this figure.

Model Definition

Modeling begins by importing the geometry that describes a car body, cable harness, and a windshield FM antenna (Figure 1). Interior objects inside the car are not included. All metal parts are modeled as perfect electric conductors (PEC), which include the car body, a printed antenna on the rear windshield, tire rims, a cable harness connected to electronic component enclosures, and the ground plane. The tire domains are modeled as a lossy medium, using a loss tangent constitutive relation. Except for the ground plane, the car is surrounded by an air domain, which is enclosed by perfectly matched layers (PML). The 1 cm thick windshield is considered transparent and very thin in the FM frequency range. It is configured using the Transition boundary condition.

To calculate the Far-field radiation pattern over the ground plane (which is simplified as a PEC surface) and create an image of a radiating source, a symmetry condition in the Far-field Calculation Boundary settings is applied.

The antenna is excited by a lumped port with a 50 ohm reference impedance.

Results and Discussion

In Figure 2, the default electric field norm is visualized on the ground plane.

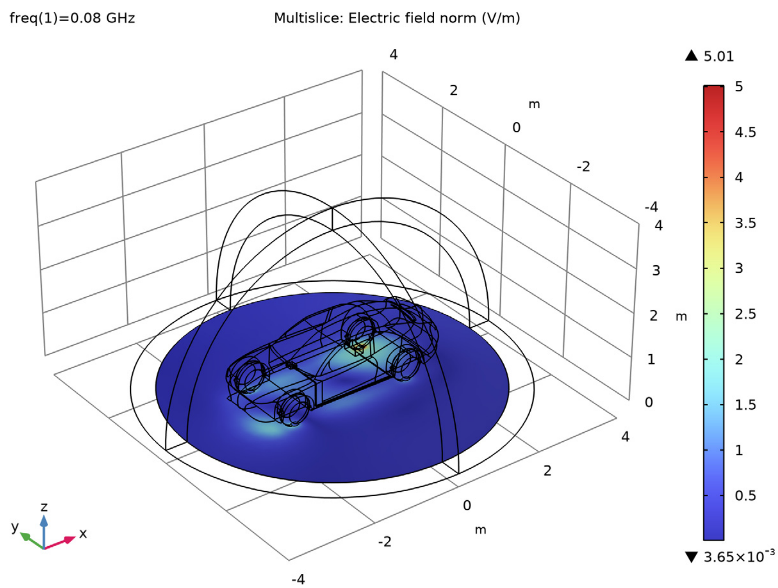


Figure 2: The electric field is nonuniformly illuminated over the ground, which contributes to the distorted radiation pattern of the antenna.

A 3D far-field radiation pattern is shown in Figure 3. Due to the shape and placement of the antenna, the overall shape of the radiation pattern is asymmetric.

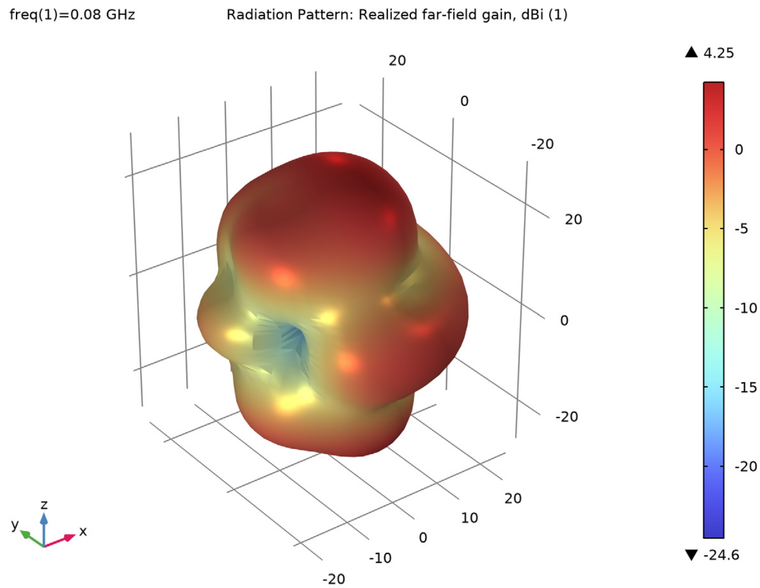


Figure 3: 3D far-field radiation pattern of the printed antenna.

Figure 4 shows the electric field norm over the cable harness surface as well as which part of the cable is more affected by the antenna radiation.

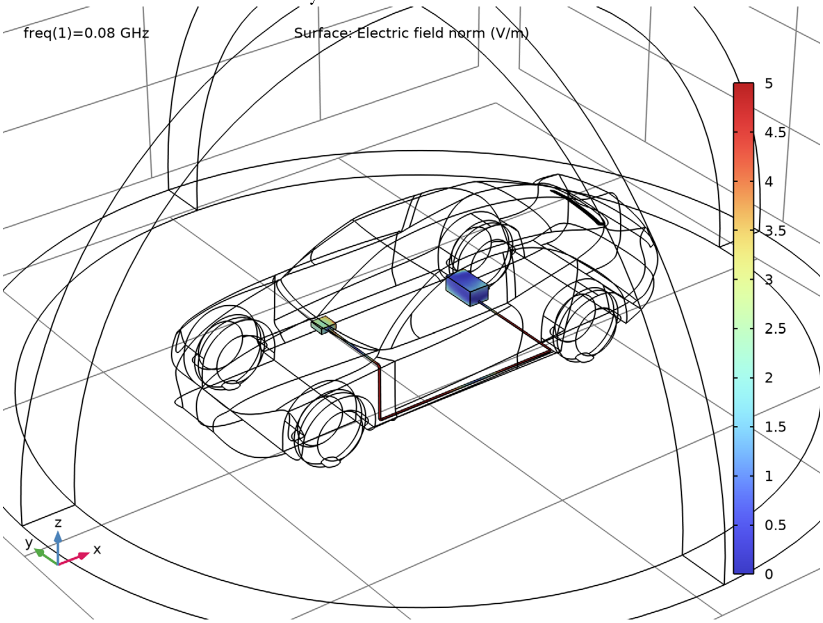


Figure 4: The cable harness that's closer to the right-side tires is more exposed to the antenna radiation.

Application Library path: `RF_Module/EMI_EMC_Applications/car_emiemc`

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**.

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 `80[MHz]`.

GEOMETRY 1

Import 1 (imp1)

1 In the **Home** toolbar, click **Import**.

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

3 Click **Browse**.

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

5 Click **Import**.

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

Sphere 1 (sph1)

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

2 In the **Settings** window for **Sphere**, locate the **Size** section.

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

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

Layer name	Thickness (m)
`Layer 1`	`0.5`

Block 1 (blk1)

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

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

3 In the **Width** text field, type 8.

4 In the **Depth** text field, type 8.

5 In the **Height** text field, type 4.

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

7 In the **z** text field, type -2.

Difference 1 (dif1)

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

2 Select the object **sph1** only.

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 object **blk1** only.

6 Click **Build All Objects**.

Ignore Vertices 1 (igv1)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Ignore Vertices**.

2 In the **Settings** window for **Ignore Vertices**, locate the **Input** section.

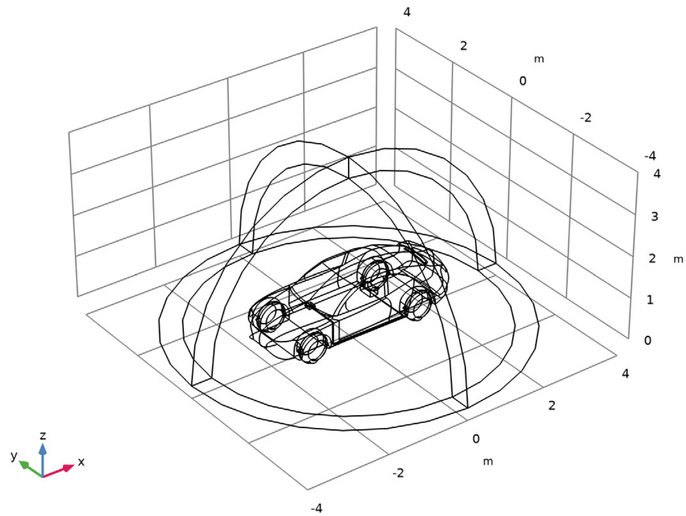
3 Click the **Paste Selection** button for **Vertices to ignore**.

4 In the **Paste Selection** dialog box, type 110 111 117 118 190 191 in the **Selection** text field.

5 Click **OK**.

This removes some vertices generating unnecessary finer mesh elements.

6 In the **Geometry** toolbar, click **Build All**.



DEFINITIONS

Create a set of selections before setting up the physics.

Windshield

1. In the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for **Explicit**, type `Windshield` in the **Label** text field.
3. Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4. Click **Paste Selection**.
5. In the **Paste Selection** dialog box, type `66-67, 104-107, 127-128, 169-170, 191-196, 202` in the **Selection** text field.
6. Click **OK**.

Tire

1. In the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for **Explicit**, type `Tire` in the **Label** text field.
3. Select Domains 5, 6, 18, and 19 only.

Harness

1. In the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for **Explicit**, type `Harness` in the **Label** text field.

3 Select Domains 7–12 and 15–17 only.

Perfectly Matched Layer 1 (pml1)

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

2 Select Domains 1, 2, 13, and 14 only.

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

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

ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN (EMW)

Perfect Electric Conductor 2

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Electromagnetic Waves, Frequency Domain (emw)** and choose the boundary condition **Perfect Electric Conductor**.

2 In the **Settings** window for **Perfect Electric Conductor**, locate the **Boundary Selection** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `9-31, 45-46, 54-60, 88-89, 95-96, 98-103, 108-111, 121-126, 145-155, 172-173, 177-178, 181-182, 185-188, 198-200, 203-206, 208-219` in the **Selection** text field.

5 Click **OK**.

Perfect Electric Conductor 3

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

2 In the **Settings** window for **Perfect Electric Conductor**, locate the **Boundary Selection** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `44, 47, 171, 174` in the **Selection** text field.

5 Click **OK**.

Perfect Electric Conductor 4

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

2 In the **Settings** window for **Perfect Electric Conductor**, locate the **Boundary Selection** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type `189-190, 197, 207` in the **Selection** text field.

5 Click **OK**.

Perfect Electric Conductor 5

1 In the **Physics** toolbar, click **Domains** and choose **Perfect Electric Conductor**.

2 In the **Settings** window for **Perfect Electric Conductor**, locate the **Domain Selection** section.

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

Transition Boundary Condition 1

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

2 In the **Settings** window for **Transition Boundary Condition**, locate the **Boundary Selection** section.

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

Wave Equation, Electric 2

1 In the **Physics** toolbar, click **Domains** and choose **Wave Equation, Electric**.

2 In the **Settings** window for **Wave Equation, Electric**, locate the **Domain Selection** section.

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

4 Locate the **Electric Displacement Field** section. From the **Electric displacement field model** list, choose **Loss tangent, loss angle**.

Lumped Port 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Lumped Port**.

2 Select Boundary 201 only.

For the first port, wave excitation is **on** by default.

Far-Field Domain 1

In the **Physics** toolbar, click **Domains** and choose **Far-Field Domain**.

Far-Field Calculation 1

1 In the **Model Builder** window, expand the **Far-Field Domain 1** node, then click **Far-Field Calculation 1**.

2 In the **Settings** window for **Far-Field Calculation**, locate the **Far-Field Calculation** section.

3 Select the **Symmetry in the z=0 plane** check box.

4 From the **Symmetry type** list, choose **Symmetry in H (PEC)**.

MATERIALS

Material 1 (mat1)

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.
2. In the **Settings** window for **Material**, locate the **Material Contents** section.
3. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	1	1	Basic
Relative permeability	mur_iso ; murii = mur_iso, murij = 0	1	1	Basic
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	0	S/m	Basic

Material 2 (mat2)

1. Right-click **Materials** and choose **Blank Material**.
2. In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
3. From the **Selection** list, choose **Tire**.
4. Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Relative permittivity (real part)	epsilonPrim_iso ; epsilonPrimii = epsilonPrim_iso, epsilonPrimij = 0	2	1	Loss tangent, loss angle
Loss tangent, loss angle	delta	0.00005	rad	Loss tangent, loss angle
Relative permeability	mur_iso ; murii = mur_iso, murij = 0	1	1	Basic

Material 3 (mat3)

1 Right-click **Materials** and choose **Blank Material**.
2 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
3 From the **Geometric entity level** list, choose **Boundary**.
4 From the **Selection** list, choose **Windshield**.
5 Locate the **Material Contents** section. In the table, enter the following settings:

Property	**Variable**	**Value**	**Unit**	**Property group**
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	4	1	Basic
Relative permeability	mur_iso ; murii = mur_iso, murij = 0	1	1	Basic
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	0	S/m	Basic

MESH 1

Information 1

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

DEFINITIONS

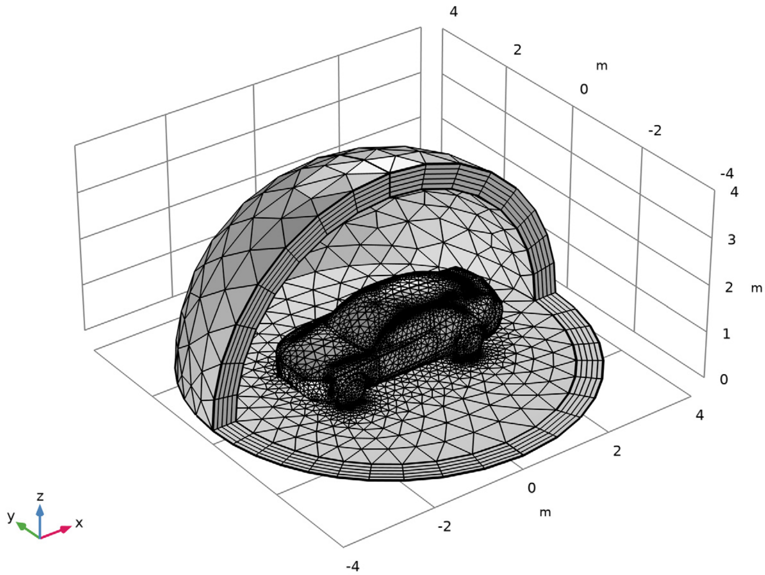
Hide for Physics 1

1 In the **Model Builder** window, right-click **View 1** and choose **Hide for Physics**.

   Suppress some boundaries to get a better view when reviewing the meshed results.
2 In the **Settings** window for **Hide for Physics**, locate the **Geometric Entity Selection** section.
3 From the **Geometric entity level** list, choose **Boundary**.
4 Select Boundaries 4, 6, 112, 114, and 115 only.

MESH 1

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



STUDY 1

Step 1: Frequency Domain

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

RESULTS

Multislice

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

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

3 Find the **X-planes** subsection. In the **Planes** text field, type 0.

4 Find the **Y-planes** subsection. In the **Planes** text field, type 0.

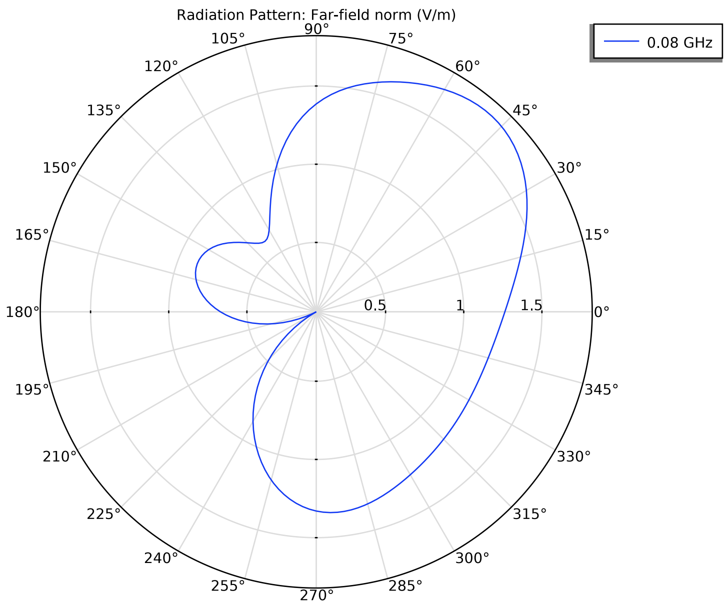
5 Find the **Z-planes** subsection. From the **Entry method** list, choose **Coordinates**.

6 In the **Coordinates** text field, type 0.

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

2D Far Field (emw)

In the **Model Builder** window, under **Results** click **2D Far Field (emw)**.



Radiation Pattern 1

1 In the **Model Builder** window, expand the **3D Far Field, Gain (emw)** node, then click **Radiation Pattern 1**.
2 In the **Settings** window for **Radiation Pattern**, locate the **Evaluation** section.
3 Find the **Angles** subsection. In the **Number of azimuth angles** text field, type 40.
4 In the **3D Far Field, Gain (emw)** toolbar, click **Plot**.

3D Plot Group 4

In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

Surface 1

1 Right-click **3D Plot Group 4** and choose **Surface**.
2 In the **Settings** window for **Surface**, click to expand the **Range** section.
3 Select the **Manual color range** check box.
4 In the **Maximum** text field, type 5.

Selection 1

1 Right-click **Surface 1** and choose **Selection**.

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

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

4 From the **Selection** list, choose **Harness**.

5 In the **3D Plot Group 4** toolbar, click **Plot**.