



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.

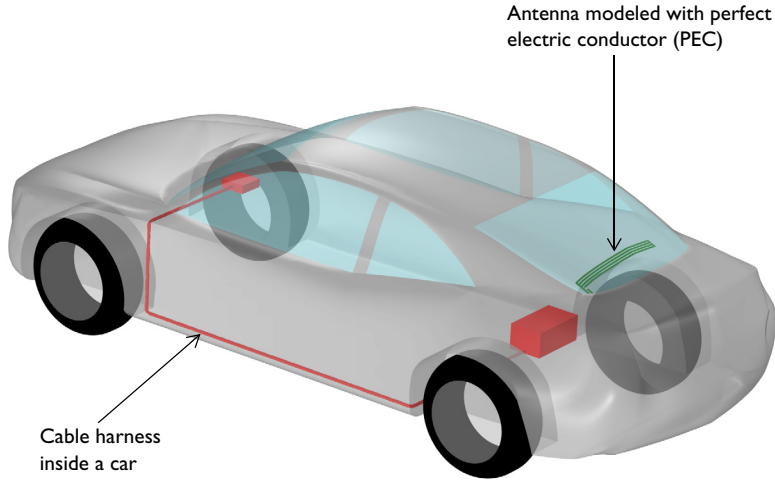


Figure 1: A simplified car model consisting of a metallic body, lossy tires, tire rims, thin dielectric windshields, a printed antenna, and a cable harness connected to electronic component enclosures. The surrounding air domain and ground plane are not included in this figure.

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.

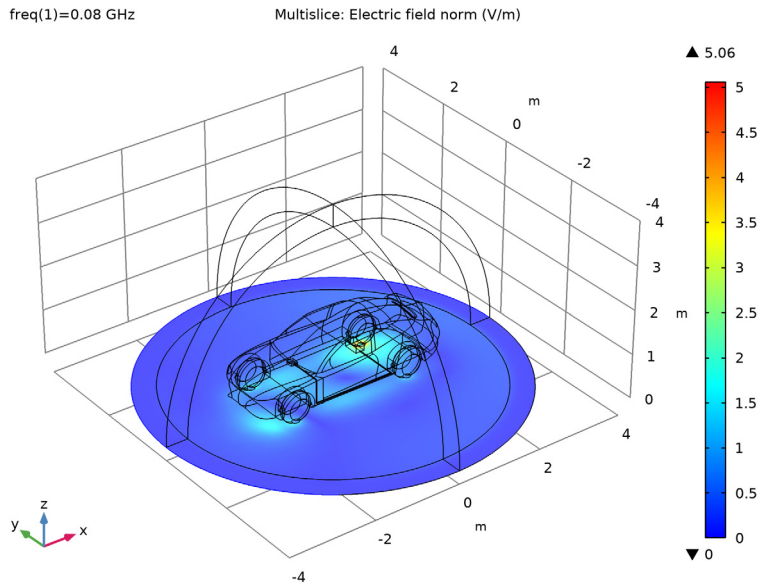


Figure 2: The electric field is nonuniformly illuminated over the ground, which contributes to the distorted radiation pattern of the antenna.

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.

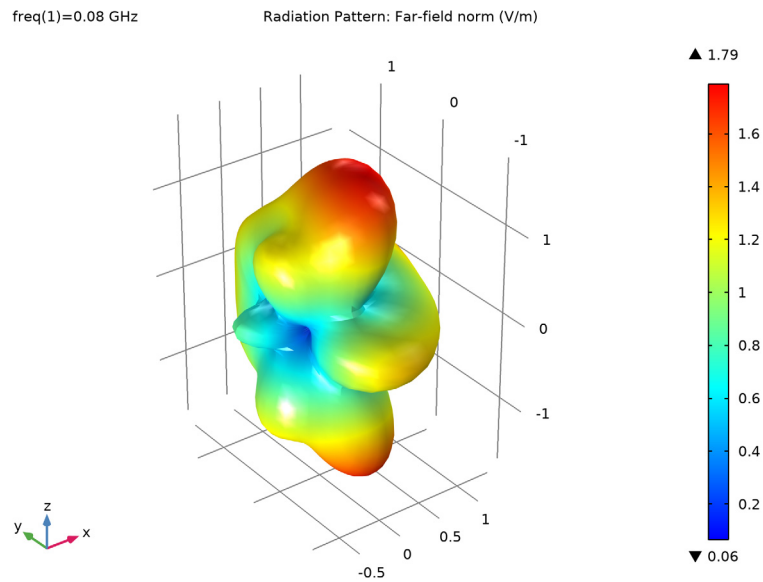


Figure 3: 3D far-field radiation pattern of the printed antenna.

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.

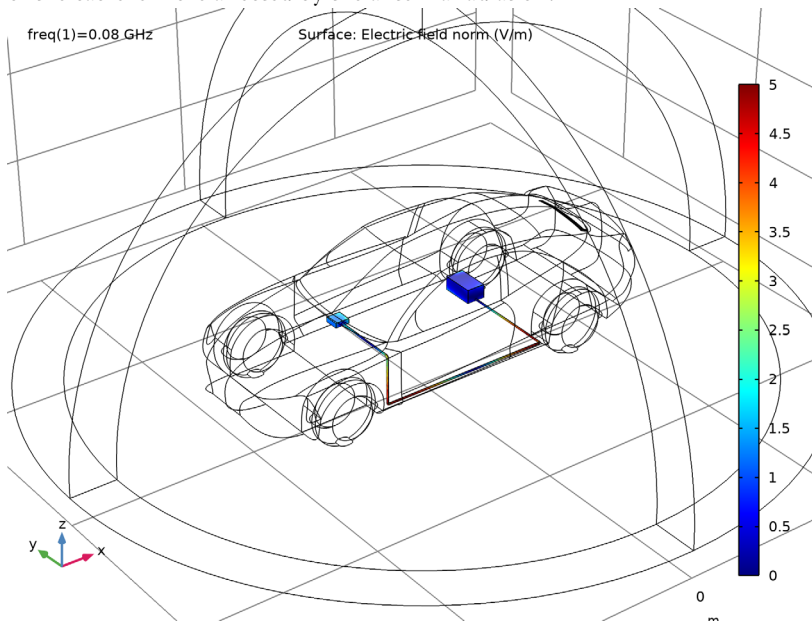



Figure 4: The cable harness that's closer to the right-side tires is more exposed to the antenna radiation.

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 I

Step 1: Frequency Domain


- 1 In the **Model Builder** window, under **Study I** 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 I

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 / (difl)

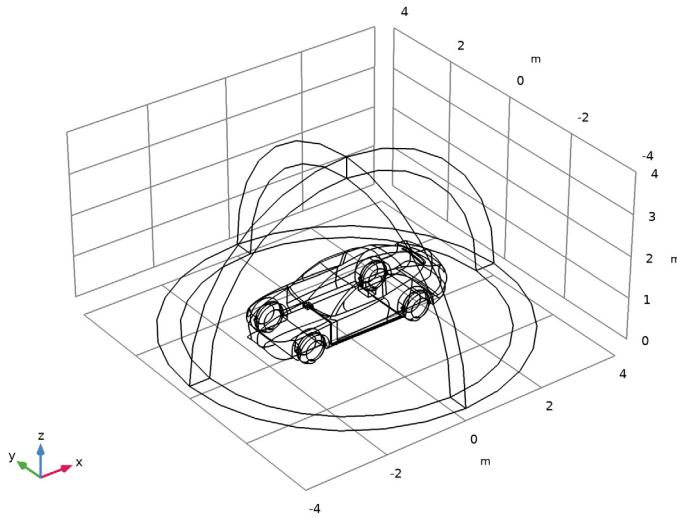
- 1 In the **Geometry** toolbar, click  **Booleans and Partitions** and choose **Difference**.
- 2 Select the object **sphl** only.
- 3 In the **Settings** window for **Difference**, locate the **Difference** section.
- 4 Find the **Objects to subtract** subsection. Select the  **Activate Selection** toggle button.
- 5 Select the object **blk1** only.
- 6 Click  **Build All Objects**.

Ignore Vertices / (igvl)

- 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  **Paste Selection**.
- 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.
- 4 Locate the **Output Entities** section. From the **Output entities** list, choose **Adjacent boundaries**.

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 **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  **Boundaries** and choose **Perfect Electric Conductor**.
- 2 In the **Settings** window for **Perfect Electric Conductor**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **Harness**.
- 4 Click  **Paste Selection**.
- 5 In the **Paste Selection** dialog box, type 79, 142 in the **Selection** text field.
- 6 Click **OK**.


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	epsilon _{nr_iso} ; epsilon _{nr_ii} = epsilon _{nr_iso} , epsilon _{nr_ij} = 0	1		Basic
Relative permeability	mu _{r_iso} ; mu _{r_ii} = mu _{r_iso} , mu _{r_ij} = 0	1		Basic
Electrical conductivity	sigma _{iso} ; sigma _{ii} = sigma _{iso} , sigma _{ij} = 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)	epsilon _{Prim_iso} ; epsilon _{Prim_ii} = epsilon _{Prim_iso} , epsilon _{Prim_ij} = 0	2		Loss tangent, loss angle
Loss tangent, loss angle	delta	0.00005	rad	Loss tangent, loss angle
Relative permeability	mu _{r_iso} ; mu _{r_ii} = mu _{r_iso} , mu _{r_ij} = 0	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	epsilon _{nr_} iso ; epsilon _{nrii} = epsilon _{nr_} iso, epsilon _{nrij} = 0	4	I	Basic
Relative permeability	mu _{r_} iso ; mu _{rii} = mu _{r_} iso, mu _{rij} = 0	1	I	Basic
Electrical conductivity	sigma _{iso} ; sigma _{ii} = sigma _{iso} , sigma _{ij} = 0	0	S/m	Basic

MESH I

Warning I

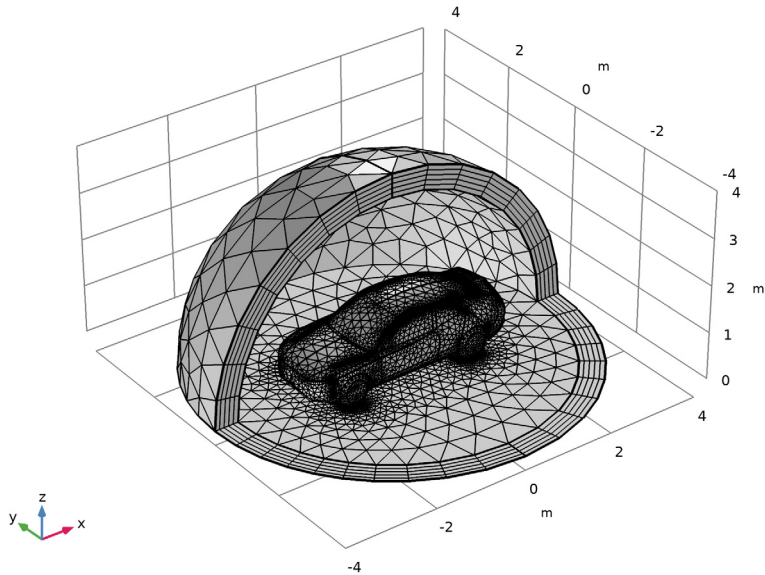
In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Build All**.

DEFINITIONS

Hide for Physics I


- 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




STUDY 1

Step 1: Frequency Domain

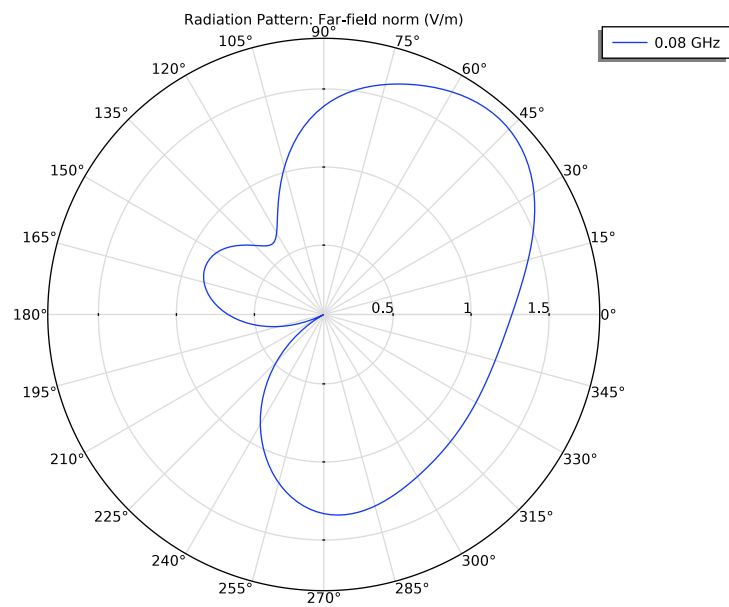
In the **Home** toolbar, click  **Compute**.

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)



Radiation Pattern I

- 1 In the **Model Builder** window, expand the **3D Far Field (emw)** node, then click **Radiation Pattern I**.
- 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 (emw)** toolbar, click  **Plot**.

TABLE

Go to the **Table** window.

RESULTS

3D Plot Group 4

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

Surface I

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

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

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

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

