Finding the Impedance of a Parallel-Wire Transmission Line

Introduction

A parallel-wire transmission line is composed of two conducting wires in a dielectric such as air. The fields around such a transmission line are not directly confined by the conductors but extend to infinity, although they drop off rapidly away from the wires. This model demonstrates how to compute the fields and impedance of such an unshielded transmission line and compares the results to the analytic solution.

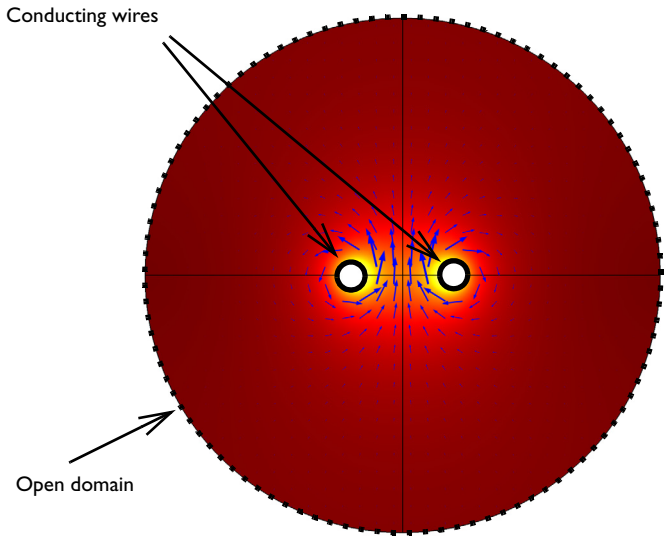


Figure 1: A parallel-wire transmission line. The electric field strength is plotted in color while arrows show the magnetic field.

Model Definition

Because a parallel-wire transmission line operates in TEM mode—with the electric and magnetic fields normal to the direction of propagation along the cable—modeling a 2D cross section suffices to compute the fields and the impedance. For this model,

assume perfect conductors and a lossless air region. The wires, of radius is 1 mm, are separated by a center-to-center distance of 8 mm.

Because the structure is open, the fields extend infinitely far away from the wires. However, they drop off quickly in magnitude. This raises the question about what boundary condition to use on the air domain's outer boundary. The surrounding dielectric medium can be thought of as a perfect insulator, as opposed to the wires, which are modeled as perfect conductors. Thus, the model uses a perfect magnetic conductor (PMC) boundary condition, because this condition is, in a sense, the opposite of the perfect electric conductor (PEC) boundary condition. However, it must be placed some distance away from the wires or else it would artificially confine the fields. In this model, the air domain's radius is chosen to be five times the distance from its center to the center of each wire. Increasing this radius would give a more accurate solution at the cost of a more memory-intensive computation.

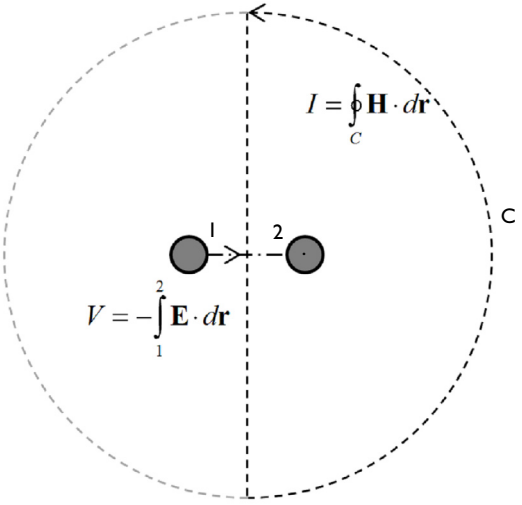


Figure 2: The impedance of a parallel-wire transmission line can be found from the voltage, V, and current, I, which are computed via line integrals as shown.

The characteristic impedance, $Z_0 = V/I$, of a transmission line relates the voltage to the current. Although the model does not involve computing the potential field, the voltage of the TEM waveguide can be evaluated as a line integral of the electric field between the conductors:

 ©2013 COMSOL

$$V = V_2 - V_1 = -\int_1^2 \mathbf{E} \cdot d\mathbf{r} \quad (1)$$

Similarly, the current is obtained as a line integral of the magnetic field along the boundary of either conductor, or any closed contour, C, bisecting the space between the conductors:

$$I = \oint_C \mathbf{H} \cdot d\mathbf{r} \quad (2)$$

The voltage and current in the direction out of the plane are positive for integration paths oriented as in Figure 2.

The value of Z_0 obtained in this way, should be compared with the analytic result

$$Z_{0,\,\text{analytic}} = \frac{1}{\pi}\sqrt{\frac{\mu_0}{\varepsilon_0 \varepsilon_r}}\,\text{acosh}\left(\frac{r_d}{r_a}\right) = 247\,\Omega$$

Here r_a is the wire radius and r_d is the center-to-center distance between the wires.

Results and Discussion

Figure 3 is a combined plot of the electric field magnitude and the magnetic field visualized as an arrow plot.

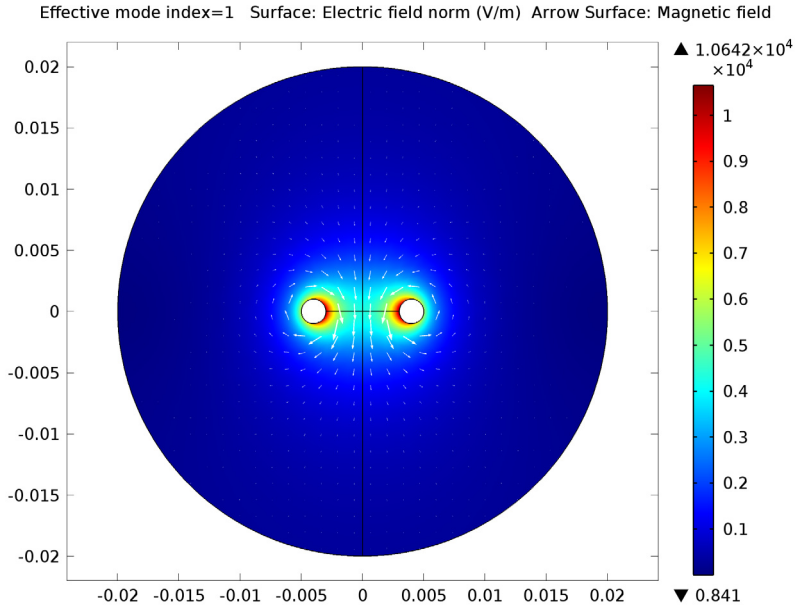


Figure 3: Electric field magnitude (surface) and magnetic field (arrows) around the two parallel wires.

The impedance computed with the default mesh is $Z_0 = 255.8\ \Omega$. As the radius of the dielectric domain is increased, the numerical solution will approach the analytic value of $247.4\ \Omega$.

Notes About the COMSOL Implementation

Solve this model using a Mode Analysis study and the default frequency, $f = 1$ GHz.

Model Library path: `RF_Module/Verification_Models/parallel_wires_impedance`

 ©2013 COMSOL

Modeling Instructions

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Click the **2D** button.

3 Click **Next**.

4 In the **Add physics** tree, select **Radio Frequency>Electromagnetic Waves, Frequency Domain (emw)**.

5 Click **Add Selected**.

6 Click **Next**.

7 Find the **Studies** subsection. In the tree, select **Preset Studies>Mode Analysis**.

8 Click **Finish**.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.

2 In the **Parameters** settings window, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Description
r_a	1[mm]	Wire radius
r_d	4[mm]	Center-to-center distance between wires
r_air	5*r_d	Air-domain radius
Z0_analytic	(Z0_const/pi)*log(r_d/r_a+sqrt((r_d/r_a)^2-1))	Characteristic impedance, analytic

Here, `Z0_const` is a predefined COMSOL Multiphysics constant for the characteristic impedance of vacuum, $Z_0 = \sqrt{(\mu_0/\varepsilon_0)}$. From the Value column you can read off the value $Z_{0,\text{analytic}} = 247\ \Omega$. Note also that the logarithm in the definition for `Z0_analytic` is an equivalent way of writing $\operatorname{acosh}(r_d/r_a)$

GEOMETRY 1

First, create a circle for the air domain.

Circle 1

1 In the **Model Builder** window, under **Model 1** right-click **Geometry 1** and choose **Circle**.

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

3 In the **Radius** edit field, type `r_air`.

4 Click the **Build Selected** button.

Add a circle for one wire.

Circle 2

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

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

3 In the **Radius** edit field, type `r_a`.

4 Locate the **Position** section. In the **x** edit field, type `r_d`.

5 Click the **Build Selected** button.

Then, generate the other wire by mirroring the above one.

Mirror 1

1 Right-click **Geometry 1** and choose **Transforms>Mirror**.

2 Select the object **c2** only.

3 In the **Mirror** settings window, locate the **Input** section.

4 Select the **Keep input objects** check box.

5 Click the **Build Selected** button.

Difference 1

1 Right-click **Geometry 1** and choose **Boolean Operations>Difference**.

2 Select the object **c1** only.

3 In the **Difference** settings window, locate the **Difference** section.

4 Under **Objects to subtract**, click **Activate Selection**.

5 Select the objects **c2** and **mir1** only.

6 Click the **Build Selected** button.

Create a line for computing the voltage as a line integral of the electric field.

Bézier Polygon 1

1 Right-click **Geometry 1** and choose **Bézier Polygon**.

2 In the **Bézier Polygon** settings window, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click the **Add Linear** button.

 ©2013 COMSOL

4 Find the **Control points** subsection. In row **1**, set **x** to `-r_d+r_a`.

5 In row **2**, set **x** to `r_d-r_a`.

6 Click the **Build Selected** button.

Add a line, a part of the closed contour for computing the current as a line integral of the magnetic field.

Bézier Polygon 2

1 Right-click **Geometry 1** and choose **Bézier Polygon**.

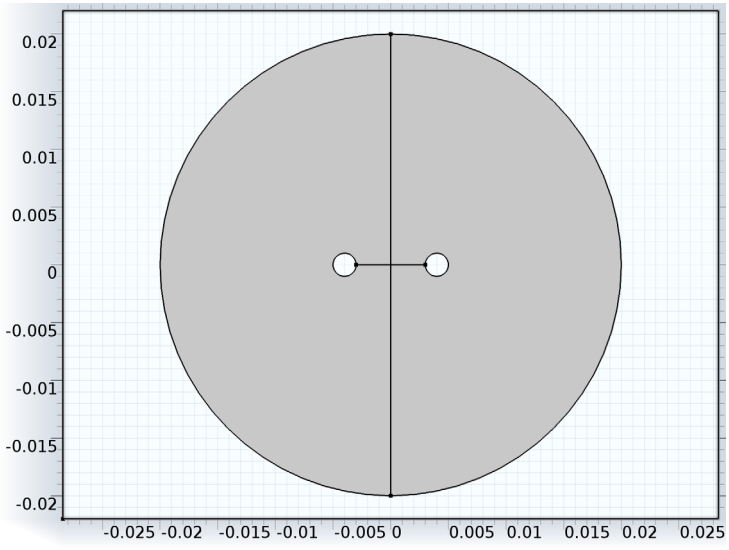
2 In the **Bézier Polygon** settings window, locate the **Polygon Segments** section.

3 Find the **Added segments** subsection. Click the **Add Linear** button.

4 Find the **Control points** subsection. In row **1**, set **y** to `-r_air`.

5 In row **2**, set **y** to `r_air`.

6 Click the **Build Selected** button.



The model layout describes two parallel wires in the air.

DEFINITIONS

Add a variable for the characteristic impedance computed as the voltage between the wires divided by the current through the wires. Define two integration coupling operators for computing the voltage and the current.

Integration 1

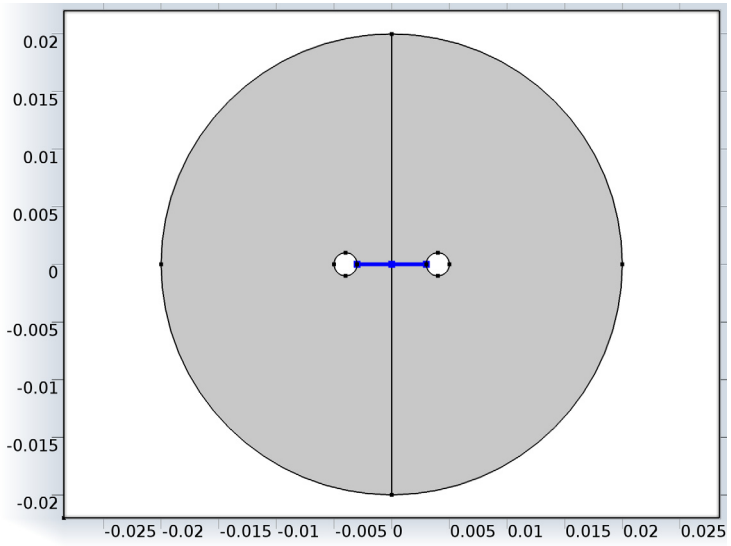
1 In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Model Couplings>Integration**.

2 In the **Integration** settings window, locate the **Operator Name** section.

3 In the **Operator name** edit field, type `int_E`.

4 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

5 Select Boundaries 1 and 4 only.



Integration 2

1 In the **Model Builder** window, right-click **Definitions** and choose **Model Couplings>Integration**.

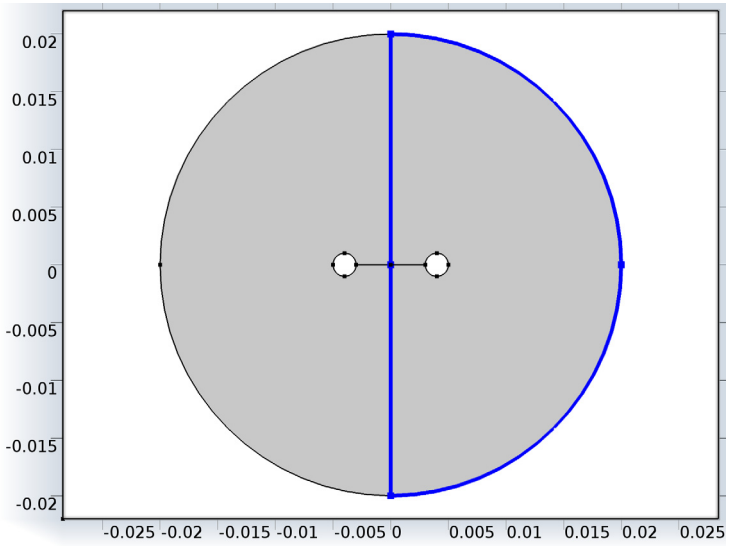
2 In the **Integration** settings window, locate the **Operator Name** section.

3 In the **Operator name** edit field, type `int_H`.

4 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

 ©2013 COMSOL

5 Select Boundaries 2, 3, 11, and 12 only.



Because of the PMC boundary condition, there is no tangential *H*-field on the outermost boundaries. It is therefore possible to omit Boundaries 11 and 12 when computing the current even though the model includes those boundaries.

STUDY 1

Step 1: Mode Analysis

1 In the **Model Builder** window, under **Study 1** click **Step 1: Mode Analysis**.

2 In the **Mode Analysis** settings window, locate the **Study Settings** section.

3 In the **Desired number of modes** edit field, type 1.

4 In the **Search for modes around** edit field, type 1.

DEFINITIONS

Variables 1

1 In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Variables**.

2 In the **Variables** settings window, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Description
V	`int_E(-emw.Ex*t1x-emw.Ey*t1y)`	Voltage
I	`-int_H(emw.Hx*t1x+emw.Hy*t1y)`	Current
Z_model	V/I	Characteristic impedance

Here, `t1x` and `t1y` are the tangential vector components along the integration boundaries (`'1'` refers to the boundary dimension). The `emw.` prefix gives the correct physics-interface scope for the electric and magnetic field vector components.

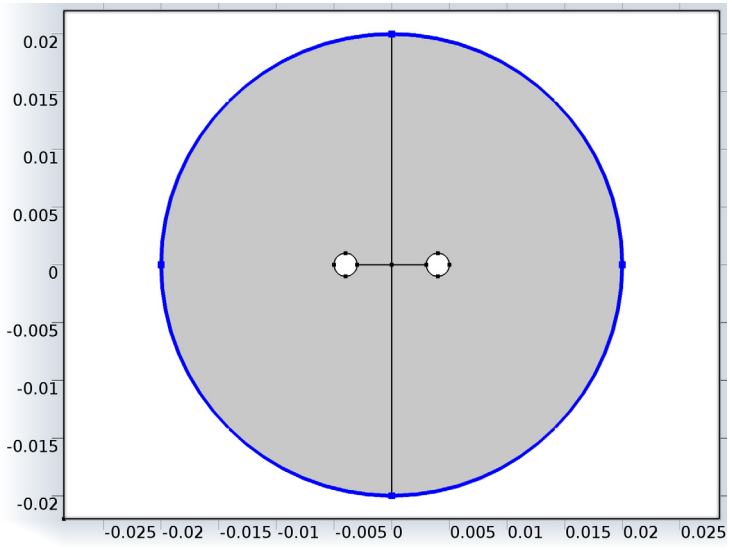
ELECTROMAGNETIC WAVES, FREQUENCY DOMAIN

Now set up the physics. The default boundary condition is perfect electric conductor. Override the outermost boundaries with a perfect magnetic conductor condition to create a virtually infinite modeling space.

Perfect Magnetic Conductor 1

1 In the **Model Builder** window, under **Model 1** right-click **Electromagnetic Waves, Frequency Domain** and choose **Perfect Magnetic Conductor**.

2 Select Boundaries 5, 6, 11, and 12 only.



 ©2013 COMSOL

MATERIALS

Next, assign a material to the modeling domain.

Material Browser

1 In the **Model Builder** window, under **Model 1** right-click **Materials** and choose **Open Material Browser**.

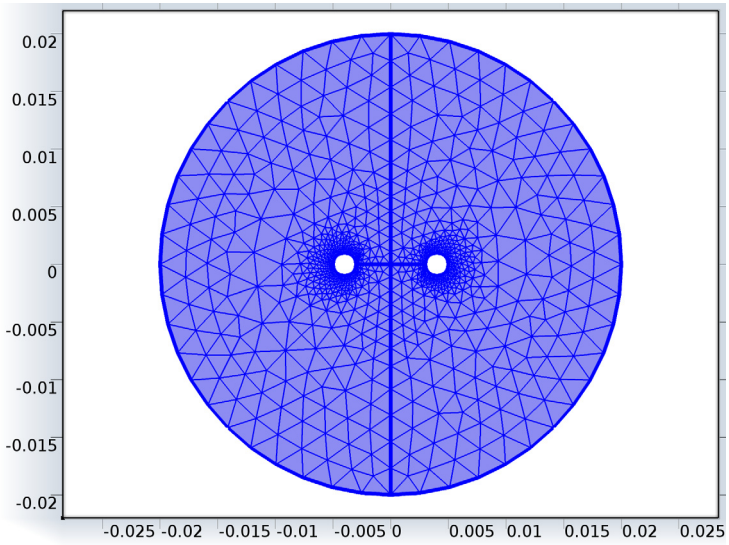
2 In the **Material Browser** settings window, In the tree, select **Built-In>Air**.

3 Click **Add Material to Model**.

MESH 1

Use the default mesh.

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



STUDY 1

In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

RESULTS

Electric Field (emw)

The default plot shows the distribution of the norm of the electric field. Add an arrow plot of the magnetic field.

1 In the **Model Builder** window, under **Results** right-click **Electric Field (emw)** and choose **Arrow Surface**.

2 In the **Arrow Surface** settings window, locate the **Arrow Positioning** section.

3 Find the **x grid points** subsection. In the **Points** edit field, type 30.

4 Find the **y grid points** subsection. In the **Points** edit field, type 30.

5 Click the **Plot** button.

6 Locate the **Coloring and Style** section. From the **Color** list, choose **White**.

7 Use the slider to adjust the arrow-length scale factor.

Compare the resulting plot with that shown in Figure 3.

Add an arrow plot along the boundaries to see the orientation of tangent vector field.

8 In the **Model Builder** window, right-click **Electric Field (emw)** and choose **Arrow Line**.

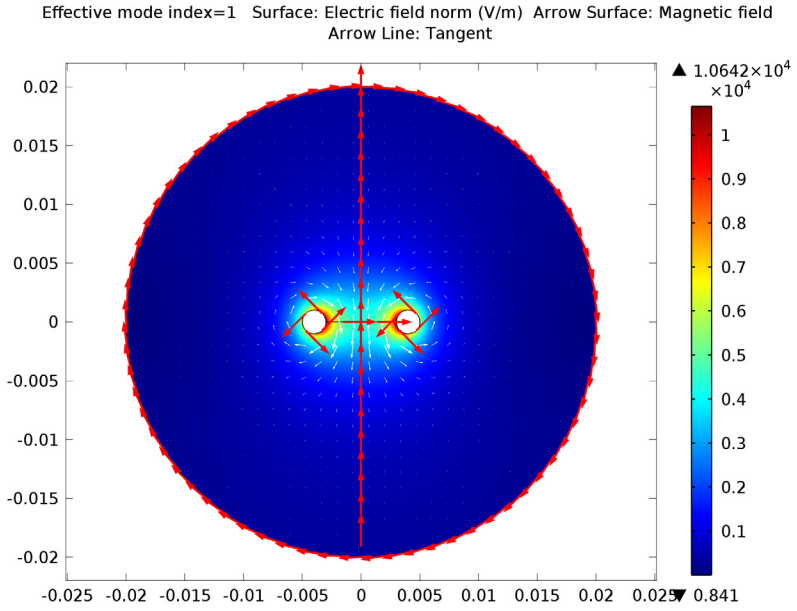
9 In the **Arrow Line** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Geometry>Tangent (tx,ty)**.

10 Click the **Plot** button.

11 Locate the **Coloring and Style** section. In the **Number of arrows** edit field, type 100.

 ©2013 COMSOL

12 Click the **Plot** button.



A comparison with Equation 1 reveals that the line integral for the voltage computes the potential difference $V_2 - V_1$. When computing the line integral for the current, the clockwise orientation of the integration contour would mean that a positive current is directed in the negative z direction, that is, into the modeling plane. The minus sign added in the definition of I reverses this direction.

Derived Values

Finish by computing the characteristic impedance.

1. In the **Model Builder** window, under **Results** right-click **Derived Values** and choose **Global Evaluation**.
2. In the **Global Evaluation** settings window, locate the **Expression** section.
3. In the **Expression** edit field, type `Z_model`.
4. Click the **Evaluate** button.

The value shown in the Results window should be close to 255.8 Ω. You can get closer to the analytic value 247.4 Ω by increasing the air-domain radius. For example, changing the definition of `r_air` to `10*r_d` under **Global Definitions>Parameters** and re-solving the model gives the value 248.9 Ω.

 ©2013 COMSOL