Created in COMSOL Multiphysics 6.2

Computing the Resistance of a Wire

Introduction

Applying a voltage difference to a conductor creates a current flow, and the intensity of the current is usually a function of the applied voltage difference. In the simplest (linear) case, the current flow and the voltage difference are proportional, and the proportionality constant is the resistance of the device. This model demonstrates how to compute the resistance of a short section of copper wire. The convergence of the solution with respect to the mesh size is also studied.

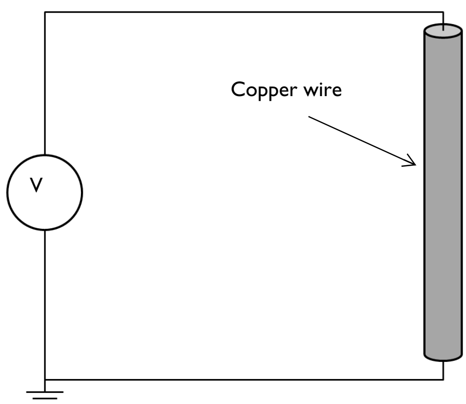


Figure 1: A short section of copper wire. The objective is to compute the equivalent resistance of this wire.

Model Definition

A 10 mm long section of copper wire of 0.5 mm radius, as shown in Figure 1, is studied. A constant current of 1 A is passed through the wire and the voltage drop is measured, from which the resistance of the wire is computed.

The boundary conditions used are meant to represent a connection to a DC source of current. One end of the wire is grounded, representing a current sink, and the other end is connected to a constant current source of 1 A, using the Terminal boundary condition.

Three different meshes are studied, to demonstrate that the results are converged with respect to mesh refinement — any further refinement of the mesh would only marginally improve the precision of the results. The results are compared, and mesh convergence is shown.

Results and Discussion

The voltage distribution is plotted in Figure 2. A linear drop in the voltage along the length of the wire can be observed. The resistance of this 10 mm long wire is computed to be 0.212 mΩ., a value that agrees within 1% for all meshes.

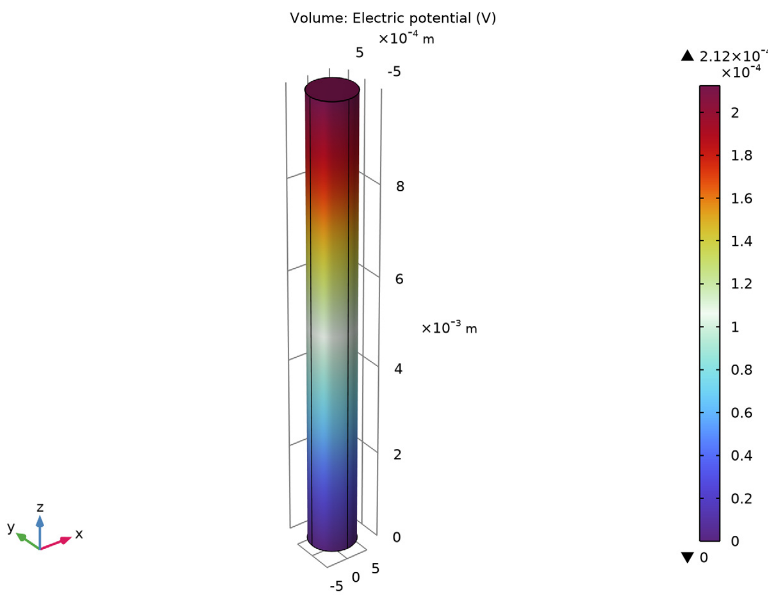


Figure 2: The voltage decreases linearly along the length of the wire.

Application Library path: `ACDC_Module/Introductory_Electric_Currents/simple_resistor`

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>Electric Fields and Currents>Electric Currents (ec)**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GEOMETRY 1

Begin by creating a cylinder for the copper wire.

Cylinder 1 (cyl1)

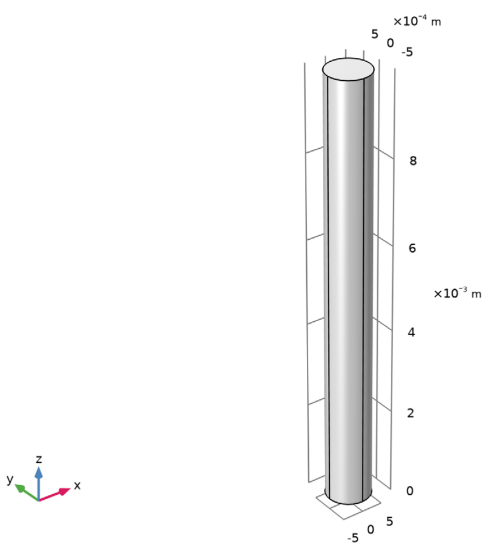
1 In the **Geometry** toolbar, click **Cylinder**.

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

3 In the **Radius** text field, type `0.5[mm]`.

4 In the **Height** text field, type `10[mm]`.

5 Click **Build All Objects**.



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

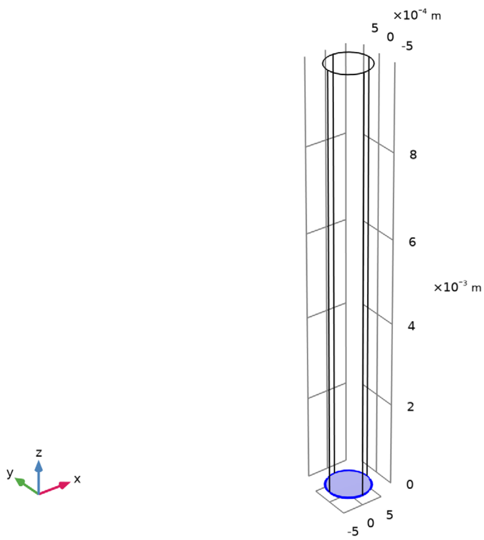
ELECTRIC CURRENTS (EC)

Set up the **Electric Current** physics. Specify the ground and terminal boundaries.

Ground 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Electric Currents (ec)** and choose **Ground**.

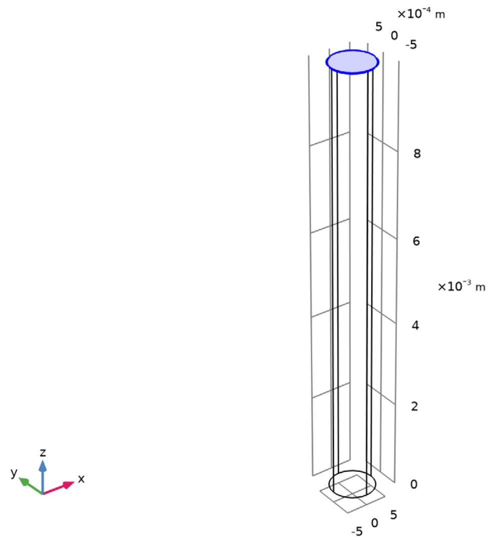
2 Select Boundary 3 only.



Terminal 1

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

2 Select Boundary 4 only.



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

4 In the I_0 text field, type `1`.

MATERIALS

Then, assign material properties. Use copper for all domains.

ADD MATERIAL

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

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

3 In the tree, select **Built-in>Copper**.

4 Click **Add to Component** in the window toolbar.

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

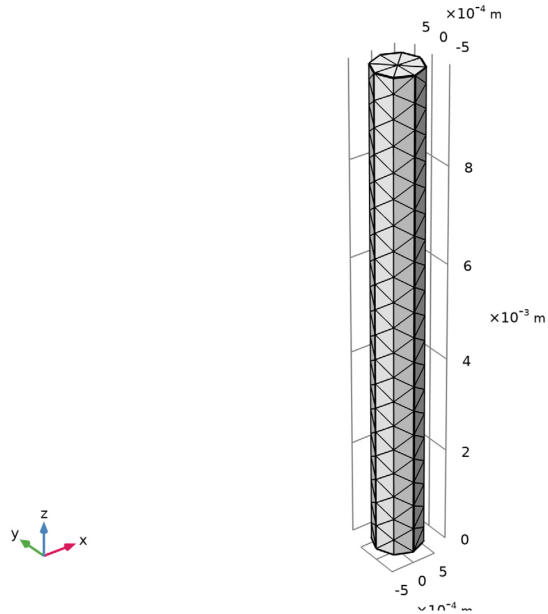
MESH 1

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 **Extra coarse**.

4 Click **Build All**.



STUDY 1

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

RESULTS

Electric Potential (ec)

The default plot shows the electric potential in the copper wire. See Figure 2.

Evaluate the resistance of the wire with the extra coarse mesh size.

Global Evaluation 1

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

2 In the **Settings** window for **Global Evaluation**, click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)> Electric Currents>Terminals>ec.R11 - Resistance -** Ω.

3 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
ec.R11	mΩ	Resistance

4 Click **Evaluate**.

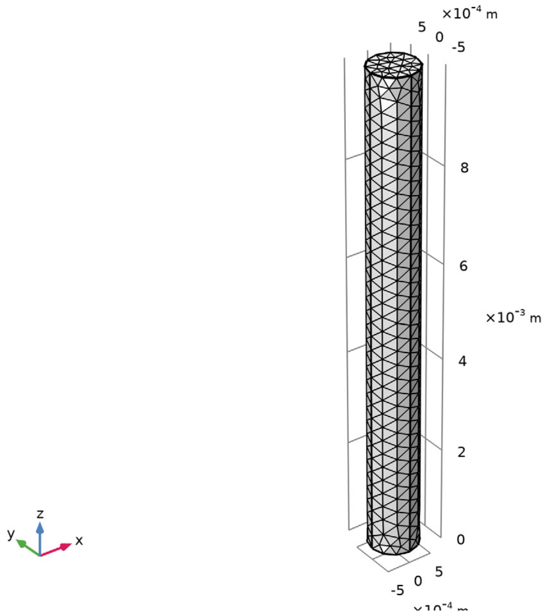
MESH 1

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 **Normal**.

4 Click **Build All**.



STUDY 1

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

RESULTS

Evaluate the resistance of the wire with the normal mesh size.

1 In the **Model Builder** window, under **Results>Derived Values** click **Global Evaluation 1**.

2 In the **Settings** window for **Global Evaluation**, click **Evaluate**.

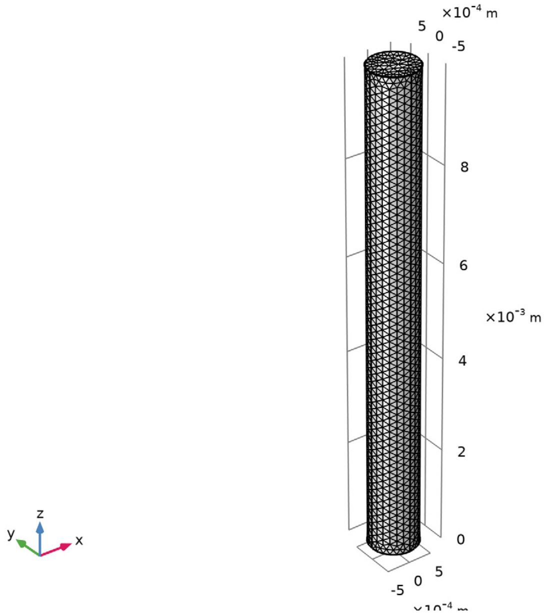
MESH 1

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 **Extra fine**.

4 Click **Build All**.



STUDY 1

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

RESULTS

Finish the result analysis by evaluating the resistance of the wire with the extra fine mesh size.

1 In the **Model Builder** window, under **Results>Derived Values** click **Global Evaluation 1**.

2 In the **Settings** window for **Global Evaluation**, click **Evaluate**.

TABLE 1

1 Go to the **Table 1** window.

The evaluated wire resistance for the three different meshes should agree within 1%.