

# Thermoelectric Leg

## Introduction

---

A thermocouple is made of two different conductors (legs) in contact with each other at one point (junction). When a temperature difference is established between the two legs, then a voltage is established across the junction. Therefore a thermocouple properly calibrated is a temperature sensor and can convert temperature gradients into electric currents. In this validation example, we verify the response of one leg when a current is passed through the device. A cooling effect, known as the Peltier effect, is expected.

## Model Definition

---

The component is 1-by-1-by-6 mm, as shown in [Figure 1](#). The core of the device, the thermoelectric part, is made of bismuth telluride ( $\text{Bi}_2\text{Te}_3$ ). It is capped by two thin copper electrodes, 0.1 mm thick.

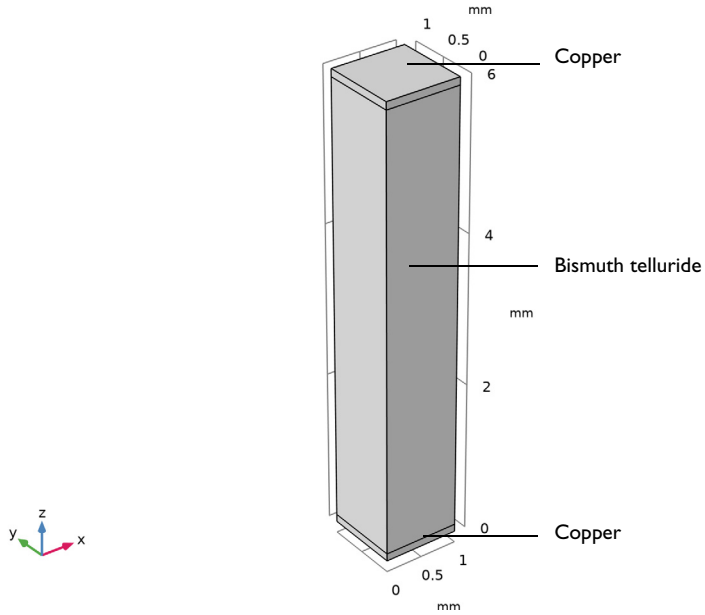


Figure 1: Thermoelectric leg geometry.

The material properties are available in COMSOL Multiphysics Material Library. However, since the properties for bismuth telluride slightly differ from these from the original benchmark, values from [Ref. 1](#) are used in this application.

TABLE 1: MATERIAL PROPERTIES FOR BISMUTH TELLURIDE.

Property	Value
Thermal conductivity	1.6 W/(m·K)
Density	7740 kg/m <sup>3</sup>
Heat capacity at constant pressure	154.4 J/(kg·K)
Electrical conductivity	1.1e5 S/m
Relative permittivity	1
Seebeck coefficient	2e-4 V/K

In addition Seebeck coefficient for copper,  $6.5 \cdot 10^{-6}$  V/K, is also taken from [Ref. 1](#).

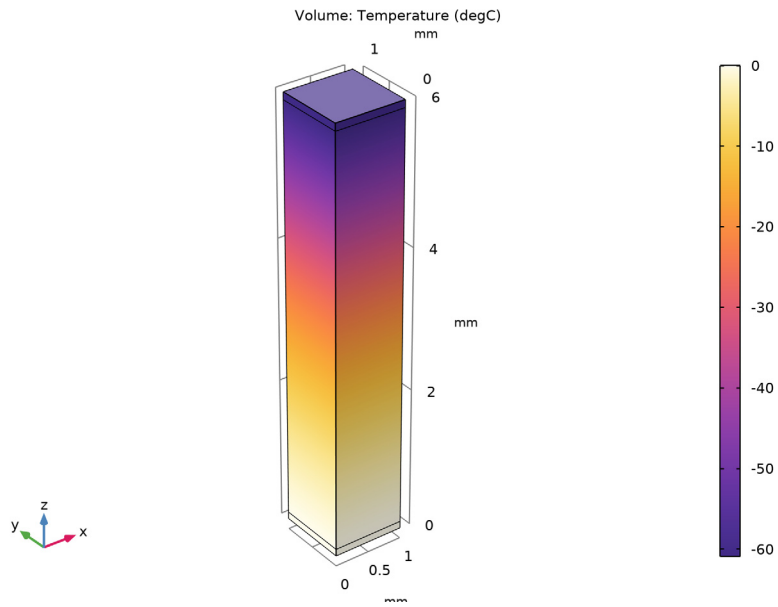
The bottom electrode surface is held at 0°C while the top electrode and the lateral surfaces are thermally insulated.

The bottom electrode is electrically grounded at 0 V. The total inward electric current through the top electrode is 0.7 A. The lateral surfaces are electrically insulated.

### *Results and Discussion*

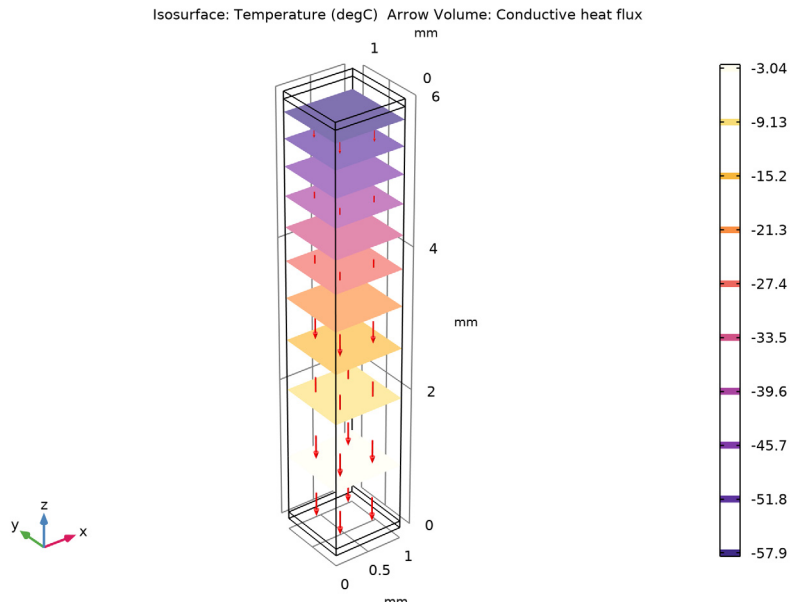
---

The current circulating in the thermoelectric device is responsible for the cooling effect shown in [Figure 2](#). The temperature field is in complete agreement with the results from [Ref. 1](#).



*Figure 2: Temperature field on the thermoelectric leg surface.*

Figure 3 shows the isothermal surfaces and the heat flux which is in the same direction as the electric current (from the top to the bottom).



*Figure 3: Isothermal surfaces and heat flux in the thermoelectric leg.*

The top level electrode reaches an electric potential of around 49.1 mV due to the inward current density set on this boundary. This corresponds to the value presented in [Ref. 1](#).

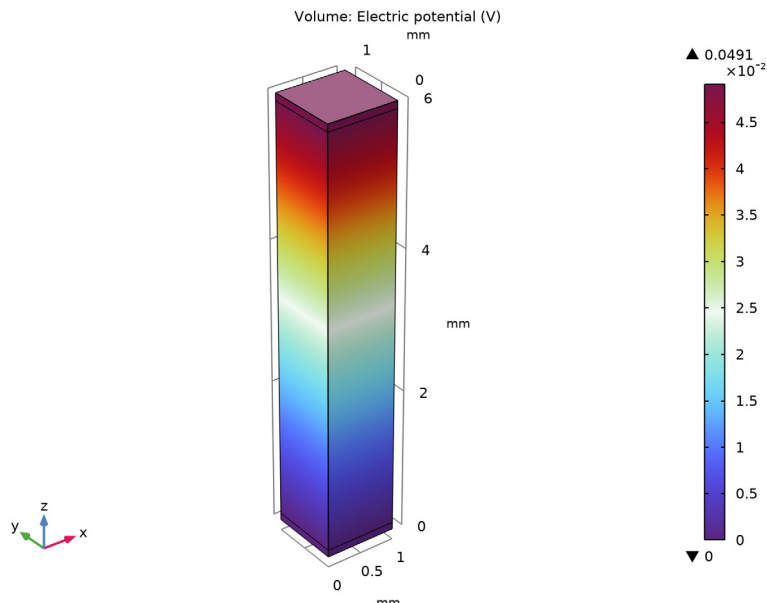


Figure 4: Electric potential in the thermoelectric leg.

## Reference


1. M. Jaegle, Multiphysics Simulation of Thermoelectric Systems, “Modeling of Peltier-Cooling and Thermoelectric Generation,” *Proc. COMSOL Conf. 2008 Hannover*, 2008.

**Application Library path:** Heat\_Transfer\_Module/Verification\_Examples/thermoelectric\_leg




## 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 **Heat Transfer>Thermoelectric Effect**.
- 3 Click **Add**.
- 4 Click  **Study**.
- 5 In the **Select Study** tree, select **General Studies>Stationary**.
- 6 Click  **Done**.



## GEOMETRY I

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.
- 2 In the **Settings** window for **Geometry**, locate the **Units** section.
- 3 From the **Length unit** list, choose **mm**.

### *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 **Height** text field, type 6.
- 4 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	0.1

- 5 Find the **Layer position** subsection. Select the **Top** check box.
- 6 Click  **Build All Objects**.
- 7 Click the  **Zoom Extents** button in the **Graphics** toolbar.

Now define the parameters that will be used for the model. The inward current density,  $J_0$ , corresponds to a total current of 0.7 A through a 1x1 mm square.

## 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
T0	0[degC]	273.15 K	Temperature reference
J0	$0.7[A] / (1[mm])^2$	7E5 A/m <sup>2</sup>	Inward current density



## MATERIALS

*Bismuth Telluride - Bi<sub>2</sub>Te<sub>3</sub>*

- 1 In the **Materials** toolbar, click  **Blank Material**.
- 2 In the **Settings** window for **Material**, type Bismuth Telluride - Bi<sub>2</sub>Te<sub>3</sub> in the **Label** text field.
- 3 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	1.6[W/(m*K)]	W/(m·K)	Basic
Density	rho	7740[kg/m^3]	kg/m <sup>3</sup>	Basic
Heat capacity at constant pressure	Cp	154.4[J/(kg*K)]	J/(kg·K)	Basic
Electrical conductivity	sigma_iso ; sigma_ii = sigma_iso, sigma_ij = 0	1.1e5[S/m]	S/m	Basic
Relative permittivity	epsilon_nr_iso ; epsilon_rii = epsilon_nr_iso, epsilon_rij = 0	1	I	Basic
Seebeck coefficient	S_iso ; Sii = S_iso, Sij = 0	2e-4[V/K]	V/K	Basic

## 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>Copper**.
- 4 Click **Add to Component** in the window toolbar.
- 5 In the **Materials** toolbar, click  **Add Material** to close the **Add Material** window.



## MATERIALS


*Copper (mat2)*

- 1 Select Domains 1 and 3 only.
- 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
Seebeck coefficient	$S_{iso}$ ; $S_{ii} = S_{iso}$ , $S_{ij} = 0$	$6.5e-6$ [V/K]	V/K	Basic

## HEAT TRANSFER IN SOLIDS (HT)


*Temperature 1*

- 1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Heat Transfer in Solids (ht)** and choose **Temperature**.
- 2 Click the  **Wireframe Rendering** button in the **Graphics** toolbar.
- 3 Select Boundary 3 only.
- 4 In the **Settings** window for **Temperature**, locate the **Temperature** section.
- 5 In the  $T_0$  text field, type  $T_0$ .


## ELECTRIC CURRENTS (EC)

In the **Model Builder** window, under **Component 1 (comp1)** click **Electric Currents (ec)**.

*Ground 1*

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Ground**.
- 2 Select Boundary 3 only.



*Normal Current Density 1*

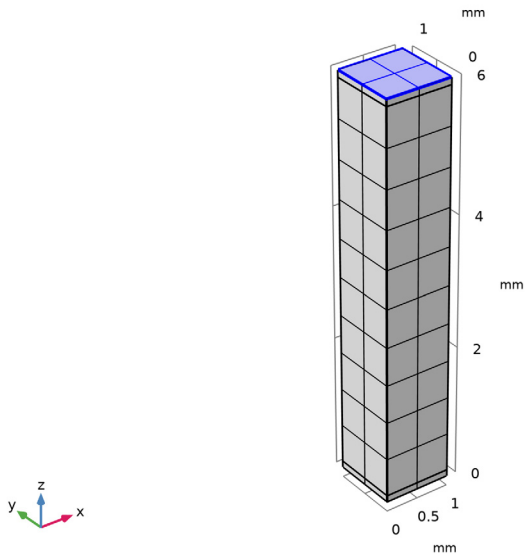
- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Normal Current Density**.
- 2 Select Boundary 10 only.
- 3 In the **Settings** window for **Normal Current Density**, locate the **Normal Current Density** section.
- 4 In the  $J_n$  text field, type  $J_0$ .

Due to the geometrical properties, replacing the default tetrahedral mesh by a hexahedral sweep mesh is more suited.


## MESH I

*Swept I*

- 1 In the **Mesh** toolbar, click  **Swept**.
- 2 In the **Settings** window for **Swept**, click to expand the **Source Faces** section.
- 3 Select Boundary 10 only.  
Now visualize the mesh and compare it with the figure below.
- 4 Click  **Build All**.



## STUDY I

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


## RESULTS

*Temperature (ht)*

The first default plot shows the temperature field; compare with [Figure 2](#).

*Volume I*

- 1 In the **Model Builder** window, expand the **Temperature (ht)** node, then click **Volume I**.
- 2 In the **Settings** window for **Volume**, locate the **Expression** section.
- 3 From the **Unit** list, choose **degC**.


- 4 In the **Temperature (ht)** toolbar, click  **Plot**.

#### *Electric Potential (ec)*

The second default plot group shows the electric potential distribution as in [Figure 4](#).

Add a predefined isothermal contours plot and enhance it with heat flux arrows to obtain a figure similar to [Figure 3](#).

### **ADD PREDEFINED PLOT**


- 1 In the **Home** toolbar, click  **Windows** and choose **Add Predefined Plot**.
- 2 Go to the **Add Predefined Plot** window.
- 3 In the tree, select **Study 1/Solution 1 (sol1)>Heat Transfer in Solids>Isothermal Contours (ht)**.
- 4 Click **Add Plot** in the window toolbar.

### **RESULTS**

#### *Isosurface 1*

- 1 In the **Model Builder** window, expand the **Isothermal Contours (ht)** node, then click **Isosurface 1**.
- 2 In the **Settings** window for **Isosurface**, locate the **Expression** section.
- 3 From the **Unit** list, choose **degC**.

#### *Arrow Volume 1*

- 1 In the **Model Builder** window, right-click **Isothermal Contours (ht)** and choose **Arrow Volume**.
- 2 In the **Settings** window for **Arrow Volume**, locate the **Arrow Positioning** section.
- 3 Find the **x grid points** subsection. In the **Points** text field, type 2.
- 4 Find the **y grid points** subsection. In the **Points** text field, type 2.
- 5 Locate the **Coloring and Style** section. From the **Arrow length** list, choose **Logarithmic**.
- 6 In the **Isothermal Contours (ht)** toolbar, click  **Plot**.

