Model created in COMSOL Multiphysics 6.4

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 (Bi_2Te_3). It is capped by two thin copper electrodes, 0.1 mm thick.

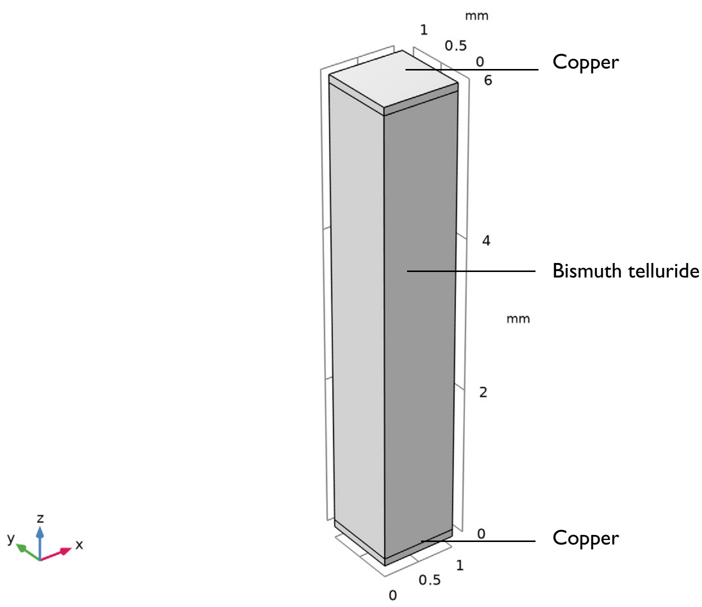


Figure 1: Thermoelectric leg geometry.

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/m³
Heat capacity at constant pressure	154.4 J/(kg·K)
Electric 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.

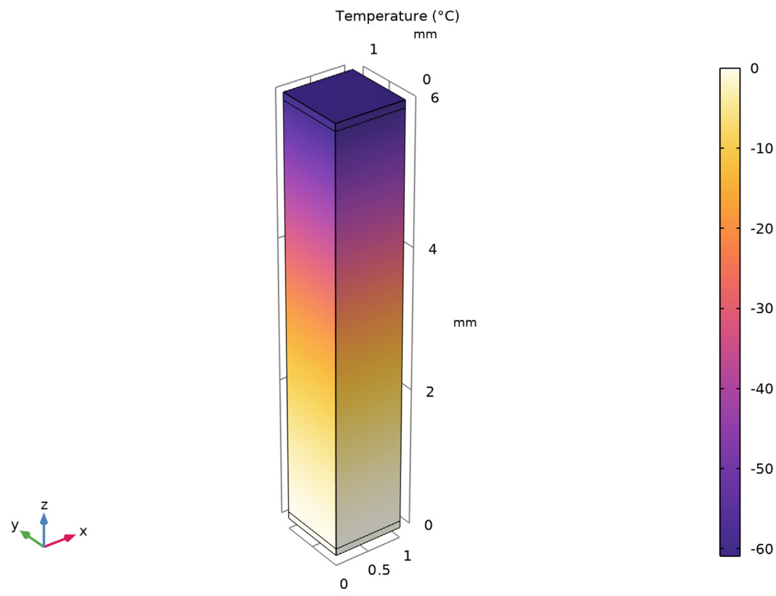


Figure 2: Temperature field on the thermoelectric leg surface.

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).

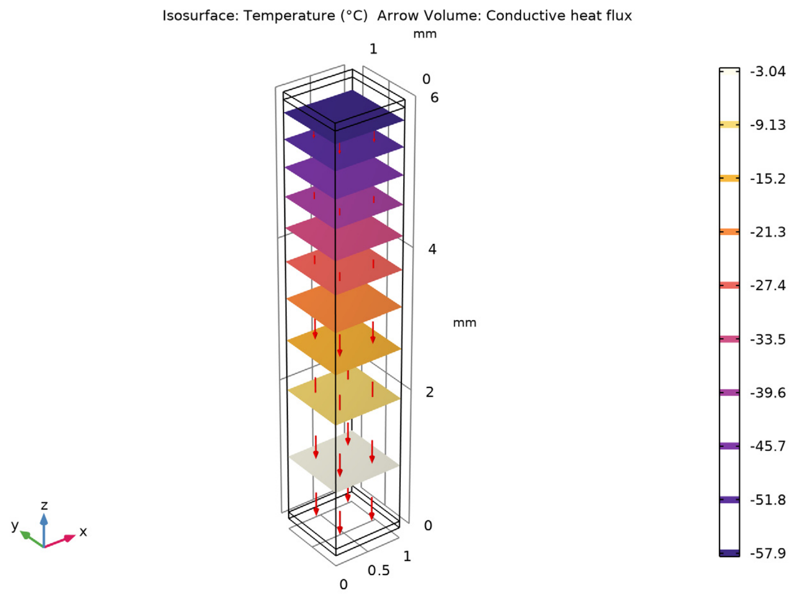


Figure 3: Isothermal surfaces and heat flux in the thermoelectric leg.

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.

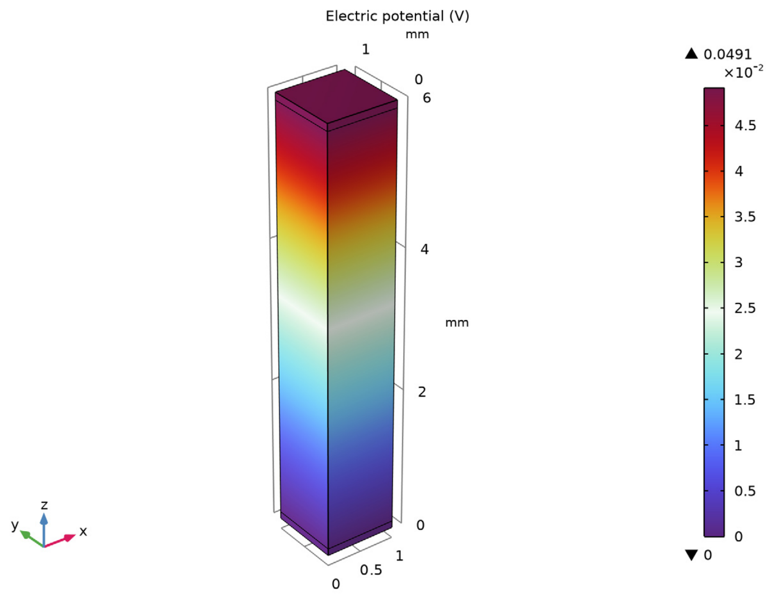


Figure 4: Electric potential in the thermoelectric leg.

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 1

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** checkbox.
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/m²	`Inward current density`

MATERIALS

Bismuth Telluride - Bi2Te3

1 In the **Materials** toolbar, click **Blank Material**.

2 In the **Settings** window for **Material**, type `Bismuth Telluride - Bi2Te3` 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/m³	Basic
Heat capacity at constant pressure	Cp	`154.4[J/(kg*K)]`	J/(kg·K)	Basic
Electric conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	`1.1e5[S/m]`	S/m	Basic
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	`1`	1	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 the **Add to Component** button 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 ; Sii = S_iso, Sij = 0	6.5e-6[V/K]	V/K	Basic

HEAT TRANSFER IN SOLIDS (HT)

Temperature 1

1. In the **Physics** toolbar, click **Boundaries** 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 `T0`.

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 `J0`.

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

MESH 1

Swept 1

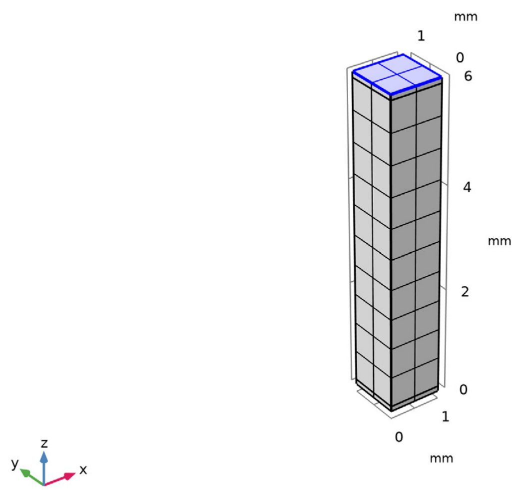
1 In the **Mesh** toolbar, click **Swept**.

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 1

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

RESULTS

Temperature (ht)

The first default plot shows the temperature field; compare with Figure 2.

Volume 1

1 In the **Model Builder** window, expand the **Temperature (ht)** node, then click **Volume 1**.

2 In the **Settings** window for **Volume**, locate the **Expression** section.

3 From the **Unit** list, choose **°C**.

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 isothermal contours plot from **Result Templates** and enhance it with heat flux arrows to obtain a figure similar to Figure 3.

RESULT TEMPLATES

1 In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

3 In the tree, select **Study 1/Solution 1 (sol1)** > **Heat Transfer in Solids** > **Isothermal Contours (ht)**.

4 Click the **Add Result Template** button in the window toolbar.

5 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

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 **°C**.

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**.