10/10

Dalhousie University

ECED 6240

CMOS-MEMS

Lab. 2

Report

Professor: Dr. Yuan Ma

Student:

March/2014

Summary

This report presents the simulation results of a thermal actuator model using COMSOL Multiphysics. In order to do the simulations, a COMSOL tutorial (attached to this report) of a 3D thermal actuator was used. The actuator consists of a 3 beam structure, 2 of them are thin while the other one is ticker (Fig. 1). Because of the electrical potential applied between the two thin beams, current passes through them heating them up and; therefore, making them expand. The expansion makes the thin beams bend over the other one that doesn't expand since its temperature does not increase. Eventually, the structure extremity experiences a vertical displacement output bigger than the beams thermal expansion (Fig. 1). The simulations provided 3D filled contour plots of these physical parameters: temperature (Fig. 2), voltage (Fig. 3), displacement (Fig. 4), heat flow (Fig. 5) and current (Fig. 6). The data list of heat flow and current were saved in excel files. As expected, simulations pointed out that temperature and current density are higher in the thin beams. However, the current density is even higher in the extremities of the fine beams where also happens the maximum value of heat flow. In summary, the simulation results of the 3D thermal actuator model were coherent and expected.

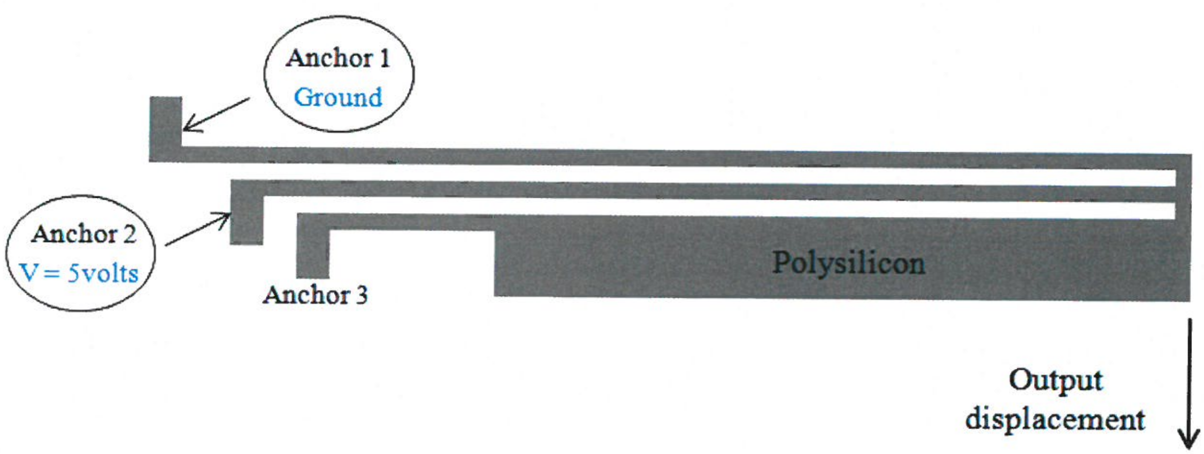

Fig. 1: Sketch of the thermal actuator.

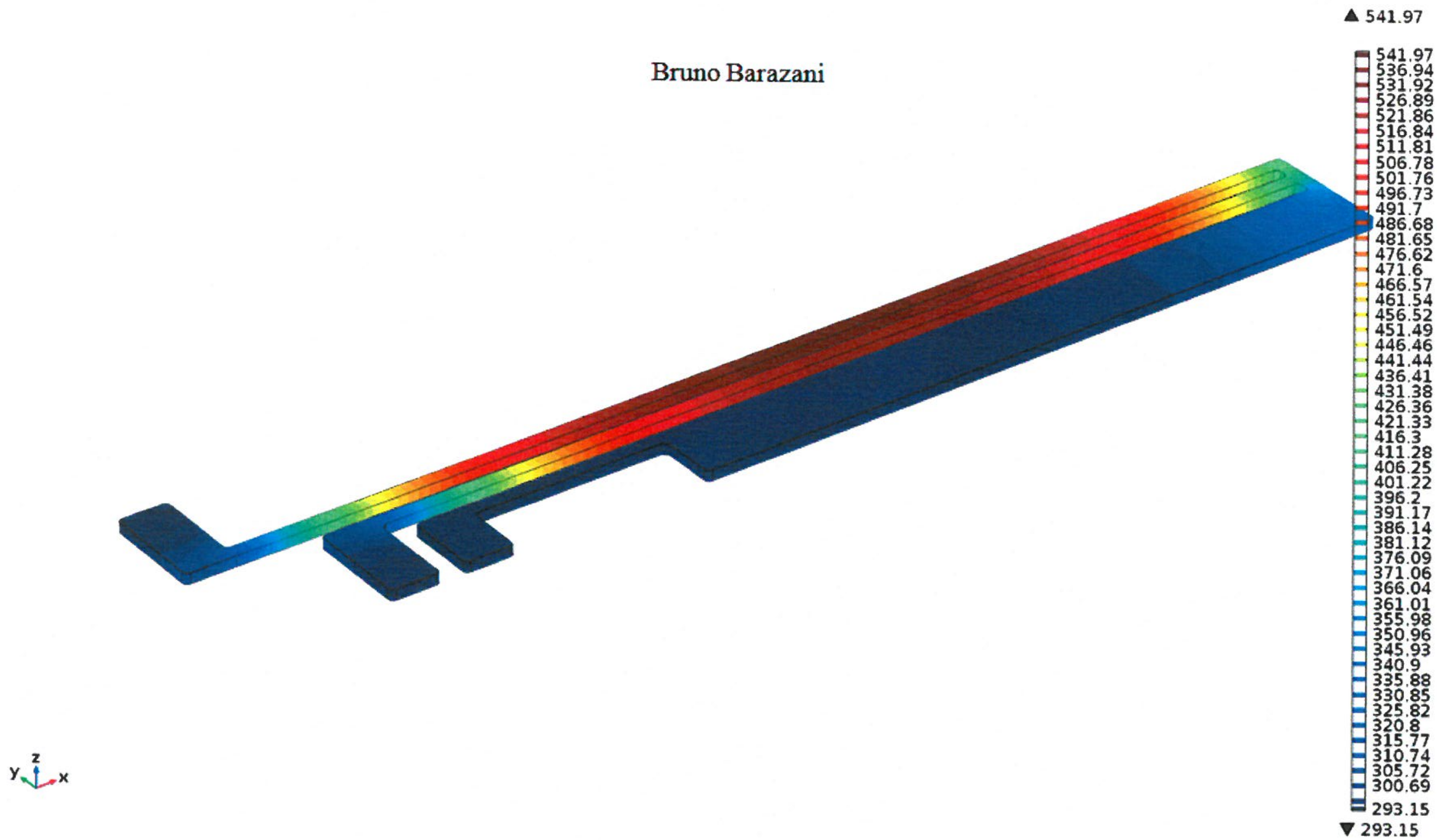

Fig. 2: Contour plot of the temperature in Kelvin degrees.

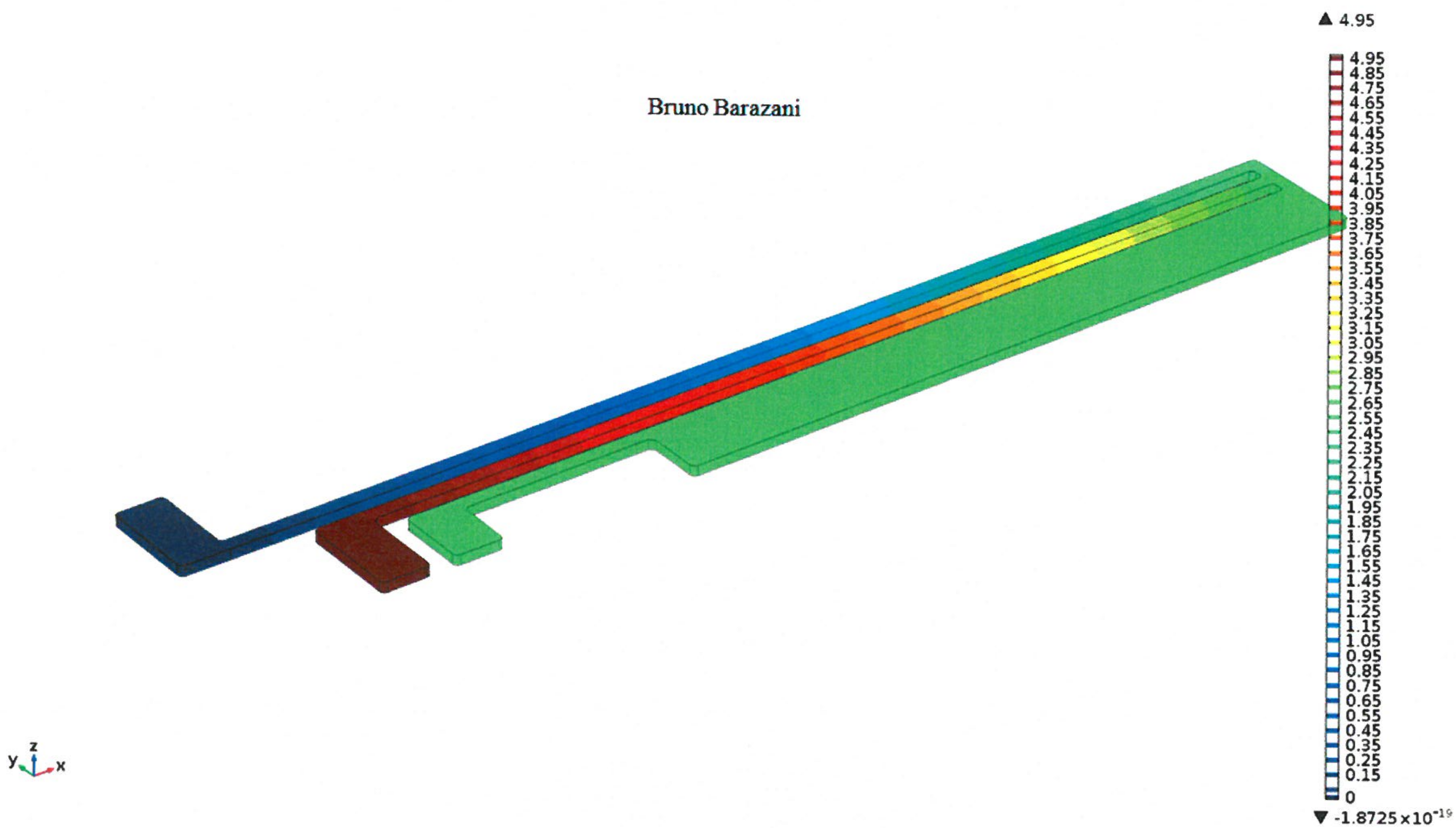

Fig. 3: Contour plot of the voltage in volts (V).

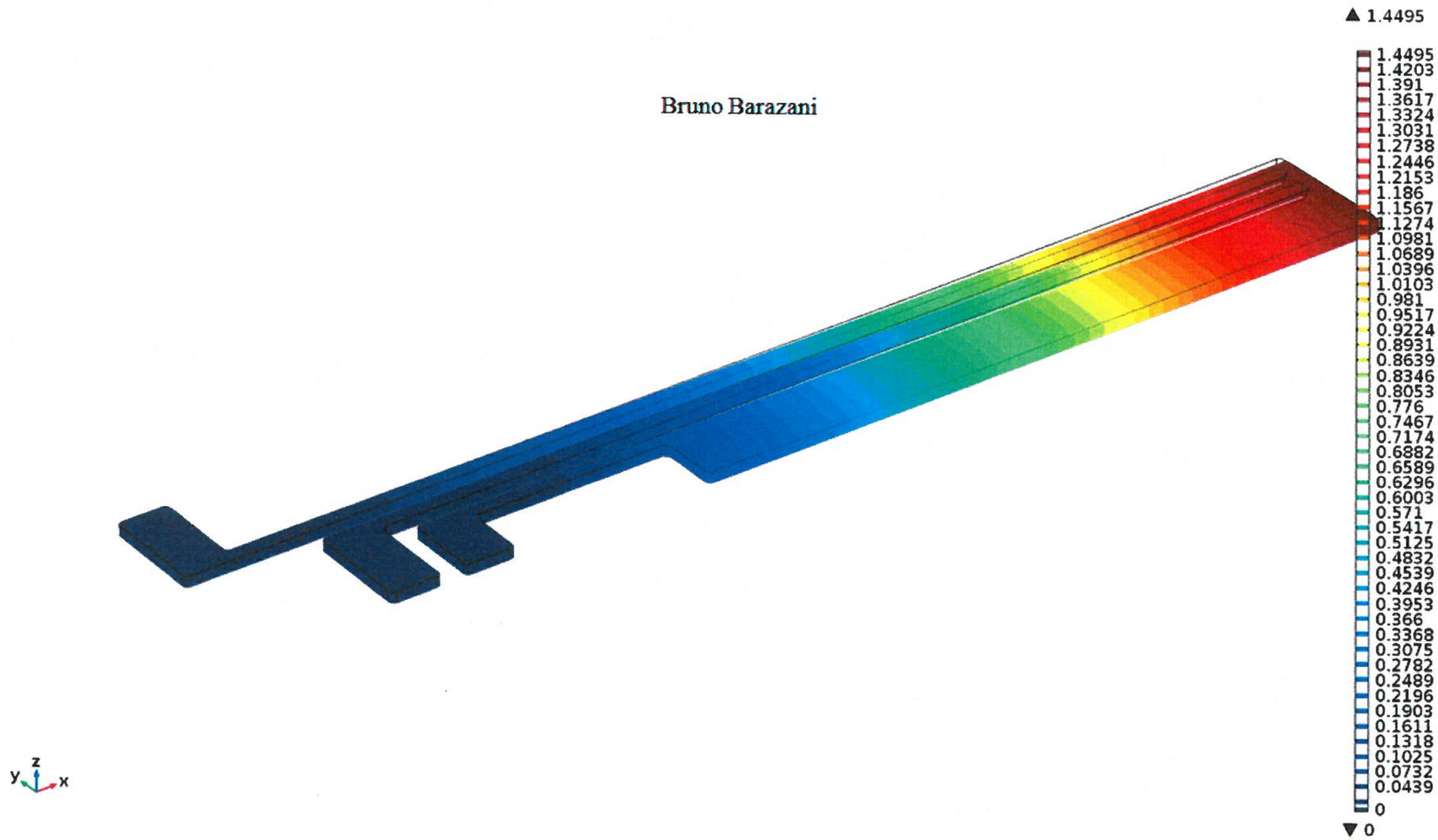

Fig. 4: Contour plot of the displacement in micrometers (μm).

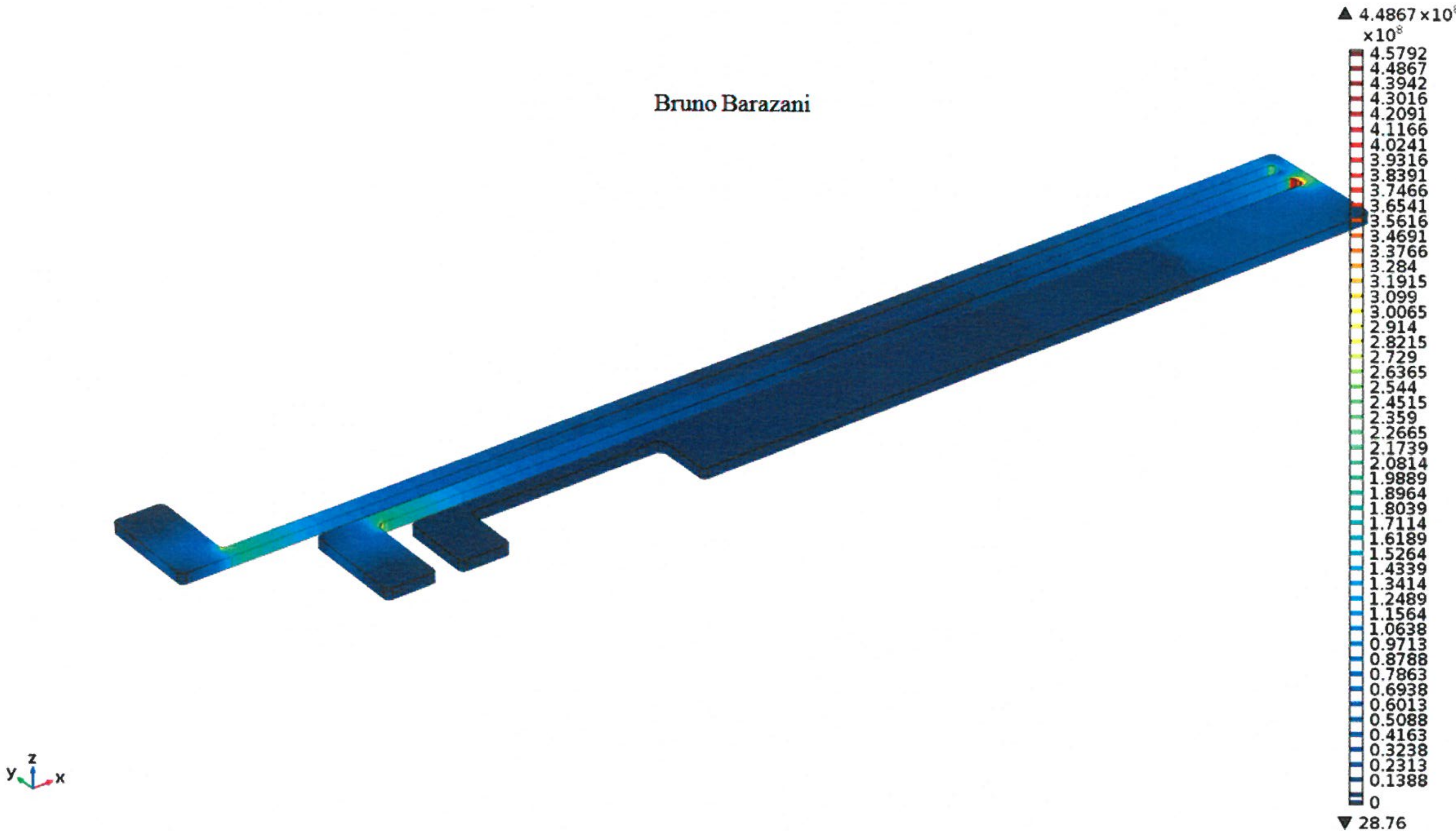

Fig. 5: Contour plot of the heat flow in [W/m^2].

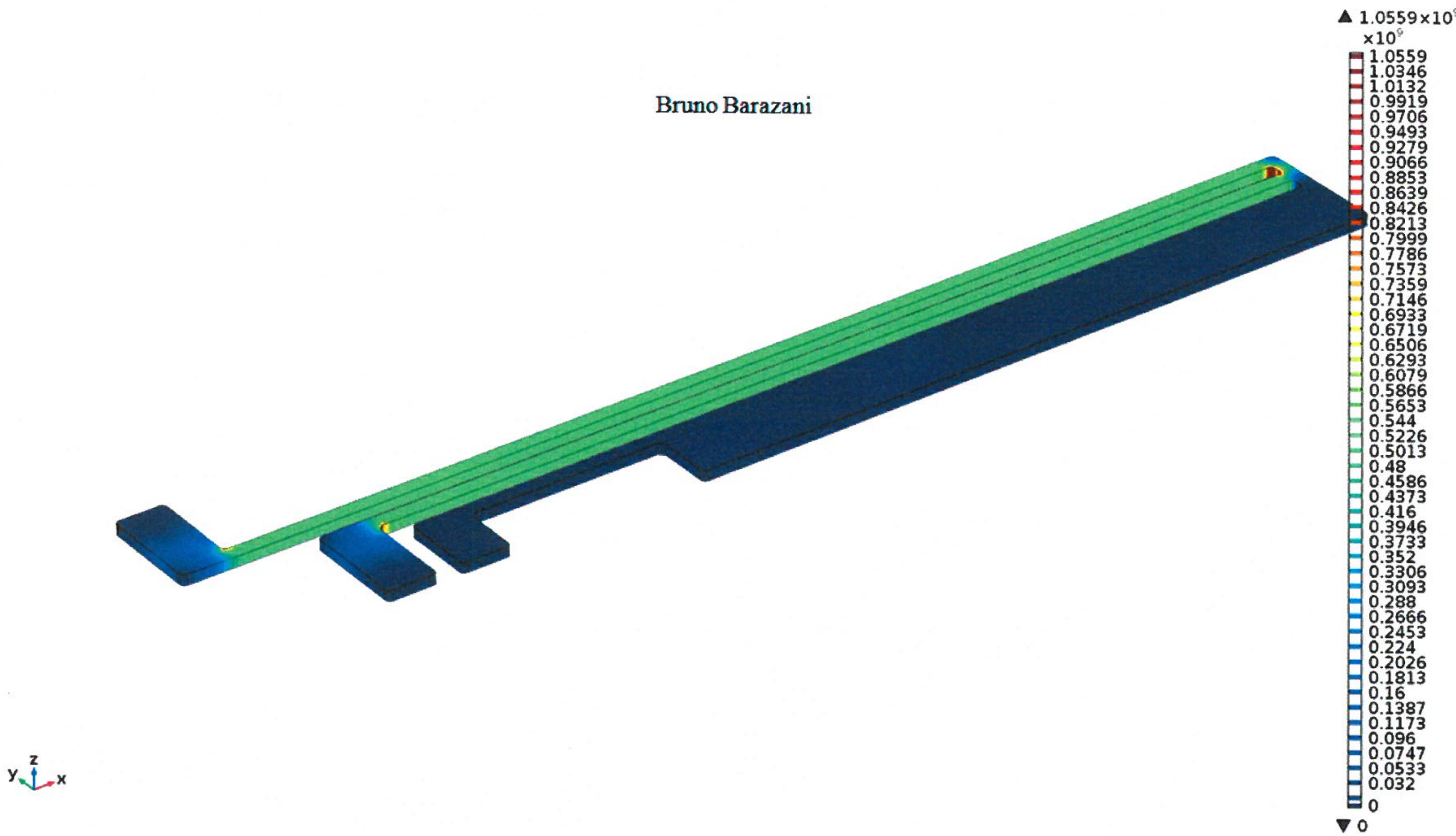

Fig. 6: Contour plot of the current density in [A/m^2].

Thermal Microactuator

Introduction

For a description of this model, see Thermal Microactuator, which describes a version of the same model (called `thermal_actuator_tem_parameterized`) that only differs in the way the geometry is created; while the modeling instructions below describe how you can import the finished geometry from an MPHBIN-file, the instructions in Thermal Microactuator detail the steps required to create the geometry in the COMSOL Desktop.

Model Library path: `MEMS_Module/Actuators/thermal_actuator_tem`

Modeling Instructions

MODEL WIZARD

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

2 Click **Next**.

3 In the **Add physics** tree, select **Structural Mechanics>Joule Heating and Thermal Expansion (tem)**.

4 Click **Next**.

5 Find the **Studies** subsection. In the tree, select **Preset Studies>Stationary**.

6 Click **Finish**.

MODEL 1

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

2 Go to the **Rename Model** dialog box and type `Thermal Actuator` in the **New name** edit field.

3 Click **OK**.

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
htc_s	0.04[W/(m*K)]/2[um]	Heat transfer coefficient
htc_us	0.04[W/(m*K)]/100[um]	Heat transfer coefficient, upper surface
DV	5[V]	Applied voltage

GEOMETRY 1

Import 1

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

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

3 Click the **Browse** button.

4 Browse to the model's Model Library folder and double-click the file `thermal_actuator.mphbin`.

5 Click the **Build All** button.

6 Click the **Go to Default 3D View** button on the Graphics toolbar.

DEFINITIONS

Explicit 1

1 In the **Model Builder** window, under **Thermal Actuator** right-click **Definitions** and choose **Selections>Explicit**.

2 In the **Explicit** settings window, locate the **Input Entities** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 10, 30, 50, 70, 76, and 82 only.

5 Right-click **Thermal Actuator>Definitions>Explicit 1** and choose **Rename**.

6 Go to the **Rename Explicit** dialog box and type `substrate contact` in the **New name** edit field.

7 Click **OK**.

MATERIALS

Material Browser

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

2 In the **Material Browser** window, locate the **Materials** section.

3 In the tree, select **MEMS>Semiconductors>Poly-Si**.

4 Right-click and choose **Add Material to Model** from the menu.

Poly-Si

By default, the first material you add applies on all domains so you can keep the Geometric Scope settings.

1 In the **Model Builder** window, under **Thermal Actuator>Materials** click **Poly-Si**.

2 In the **Material** settings window, locate the **Material Contents** section.

3 In the table, enter the following settings:

Property	Name	Value
Electric conductivity	sigma	5e4

JOULE HEATING AND THERMAL EXPANSION

Fixed Constraint 1

1 In the **Model Builder** window, under **Thermal Actuator** right-click **Joule Heating and Thermal Expansion** and choose the boundary condition **Solid Mechanics>Fixed Constraint**.

2 Select Boundaries 10, 30, and 50 only.

Roller 1

1 In the **Model Builder** window, right-click **Joule Heating and Thermal Expansion** and choose the boundary condition **Solid Mechanics>Roller**.

2 Select Boundaries 70, 76, and 82 only.

Heat Flux 1

1 Right-click **Joule Heating and Thermal Expansion** and choose the boundary condition **Heat Transfer>Heat Flux**.

This boundary condition applies to all boundaries except the top-surface boundary and those in contact with the substrate. A Temperature condition on the **substrate_contact** boundaries will override this Heat Flux condition so you do not explicitly need to exclude those boundaries. In contrast, because the Heat Flux boundary condition is additive, you must explicitly exclude the top-surface boundary from the selection. Implement this selection as follows.

2 In the **Heat Flux** settings window, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **All boundaries**.
4 In the **Graphics** window, click on the top surface and then right-click to remove it from the selection.
5 Locate the **Heat Flux** section. Click the **Inward heat flux** button.
6 In the h edit field, type `htc_s`.

Heat Flux 2

1 Right-click **Joule Heating and Thermal Expansion** and choose the boundary condition **Heat Transfer>Heat Flux**.
2 Select Boundary 4 only.
3 In the **Heat Flux** settings window, locate the **Heat Flux** section.
4 Click the **Inward heat flux** button.
5 In the h edit field, type `htc_us`.

Temperature 1

1 Right-click **Joule Heating and Thermal Expansion** and choose the boundary condition **Heat Transfer>Temperature**.
2 In the **Temperature** settings window, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **substrate contact**.

Ground 1

1 Right-click **Joule Heating and Thermal Expansion** and choose the boundary condition **Electric Currents>Ground**.
2 Select Boundary 10 only.

Electric Potential 1

1 Right-click **Joule Heating and Thermal Expansion** and choose the boundary condition **Electric Currents>Electric Potential**.
2 Select Boundary 30 only.
3 In the **Electric Potential** settings window, locate the **Electric Potential** section.
4 In the V_0 edit field, type `DV`.

MESH 1

In the **Model Builder** window, under **Thermal Actuator** right-click **Mesh 1** and choose **Free Tetrahedral**.

Size

1 In the **Model Builder** window, under **Thermal Actuator>Mesh 1** click **Size**.
2 In the **Size** settings window, locate the **Element Size** section.
3 From the **Predefined** list, choose **Fine**.

Size 1

1 In the **Model Builder** window, under **Thermal Actuator>Mesh 1** right-click **Free Tetrahedral 1** and choose **Size**.
2 In the **Size** settings window, locate the **Element Size** section.
3 From the **Predefined** list, choose **Finer**.
4 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.
5 Select Boundaries 86–91 only.
6 In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.

STUDY 1

Step 1: Stationary

1 In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.
2 In the **Stationary** settings window, locate the **Study Settings** section.
3 Select the **Include geometric nonlinearity** check box.
4 In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

RESULTS

Temperature (tem)

1 Click the **Go to Default 3D View** button on the Graphics toolbar.

The third default plot shows the combined temperature field and deformation.