MAE 355 – Fundamentals of Heat Transfer
Project
Due Date: **Wednesday**, **May 19, 2021**
(by 11:59 PM on Blackboard, late submissions will be penalized)

Project Goal

Please create a COMSOL model as shown in the attached tutorial. The dimensions and values
are mentioned in the tutorial file as well as below:
n_fins 4 "Number of pins, x direction"
L_chip 40[mm] "Chip size"
H_chip 4[mm] "Chip height"
P0 10[W] "Total power dissipated by the electronic chip"

Please run the simulation for the following 3 cases, with varying velocity for each case:
1) As in the tutorial, with 4 fins and following air flow velocities: 0 m/s, 0.5 m/s, 1 m/s and 2 m/s.
2) With 8 equally-spaced fins and following air flow velocities: 0 m/s, 0.5 m/s, 1 m/s and 2 m/s.
3) With no fins (only flat surface) and following air flow velocities: 0 m/s, 0.5 m/s, 1 m/s and 2 m/s.

Please prepare one figure of maximum domain temperature (on y-axis) vs. air flow velocities (on x-axis) for all 3 cases (i.e. there should be 3 curves in the figure).

Files to Submit on Blackboard:
1. Figure
2. Three COMSOL run files for each case (.mph)
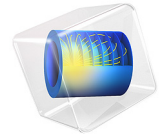
Electronic Chip Cooling

Introduction

This model is an introduction to simulations of device cooling. A device (here a chip associated to a heat sink) is cooled by a surrounding fluid, air in this case. This tutorial demonstrates the following important steps:

- Defining a heat rate on a domain using automatic volume computation.
- Modeling the temperature difference between two surfaces when a thermally thick layer is present.
- Including the radiative heat transfer between surfaces in a model.

In addition, this tutorial compares two approaches for modeling the air cooling. First, only the solid is represented and a convective cooling heat flux boundary condition is used to account for the heat transfer between the solid and the fluid. In a second step, the air domain is included in the model and a nonisothermal flow model is defined.

Model Definition

The modeled system describes an aluminum heat sink used for the cooling of an electronic chip, as shown in Figure 1.
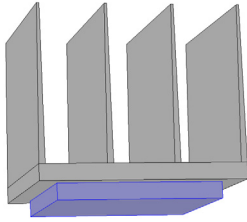
Figure 1: Geometry of the heat sink and the electronic chip.

The heat sink represented in gray in Figure 1 is mounted inside a channel with a rectangular cross section. Such a setup is used to measure the cooling capacity of heat sinks. Air enters the channel at the inlet and exits the channel at the outlet. Thermal grease is used to improve the thermal contact between the base of the heat sink and the top surface of the electronic component. All other external faces are thermally insulated. The heat dissipated by the electronic component is equal to 5 W and is distributed through the chip volume.

The cooling capacity of the heat sink can be determined by monitoring the temperature in the electronic component.

The model solves a thermal balance for the electronic component, heat sink, and air flowing in the rectangular channel. Thermal energy is transferred by conduction in the electronic component and the aluminum heat sink. Thermal energy is transported by conduction and advection in the cooling air. Unless an efficient thermal grease is used to improve the thermal contact between the electronic component and the heat sink, the temperature field varies sharply there. The temperature is set at the inlet of the channel. The transport of thermal energy at the outlet is dominated by convection.

Initially, heat transfer by radiation between surfaces is neglected. This assumption is valid as the surfaces have low emissivity (close to 0), which is usually the case for polished metals. In a case where the surface emissivity is large (close to 1), the surface-to-surface radiation should be considered. This is done later in this tutorial, where the model is modified to account for surface-to-surface radiation at the channel walls and heat sink boundaries. Assuming that the surfaces have been treated with black paint, the surface emissivity is close to 1 in this second case.

The flow field is obtained by solving one momentum balance relation for each space coordinate (x, y, and z) and a mass balance equation. The inlet velocity is defined by a parabolic velocity profile for fully developed laminar flow. At the outlet, the normal stress is equal to the outlet pressure and the tangential stress is canceled. At all solid surfaces, the velocity is set to zero in all three spatial directions.

The thermal conductivity of air, heat capacity of air, and air density are all temperature-dependent material properties. You can find all of the settings in the physics interface for Conjugate Heat Transfer in COMSOL Multiphysics. The material properties, including their temperature dependence, are available in the Material Browser.

Results and Discussion

MODELING USING CONVECTIVE COOLING BOUNDARY CONDITION

In this part of the model, only the solid domains are represented. Instead of computing the flow velocity, pressure, and temperature in the air channel, a convective cooling boundary condition is used at the heat sink boundaries. The approach enables very quick computations, but its accuracy relies on the heat transfer coefficient that is used to define the convective cooling condition. In this configuration, an empirical value, $10 \ \mathrm{W/(m^2 \cdot K)}$, is used.

Next, to model the thermal contact between the heat sink and the chip three hypotheses are considered. In a first simulation an ideal contact is assumed.
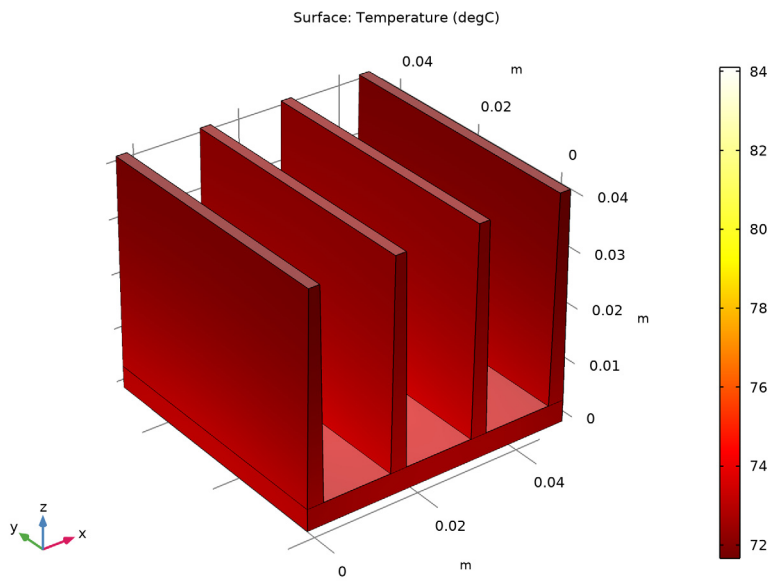
Figure 2: Temperature plot of the heat sink for the first configuration.

Under ideal conditions, the maximum temperature in the chip is about 84°C.

first case in Figure 5. This confirms that radiative heat transfer is not negligible when the surface emissivity is close to 1.
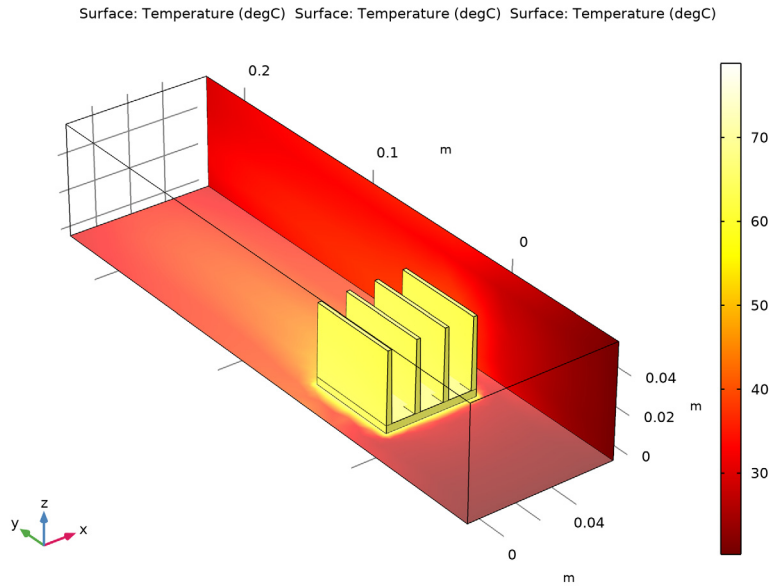
Surface: Temperature (degC) Surface: Temperature (degC) Surface: Temperature (degC)

Figure 6: The effects of surface-to-surface radiation on temperature. The surface plot shows the temperature field on the channel walls and the heat sink surface.

Application Library path: Heat_Transfer_Module/Tutorials, _Forced_and_Natural_Convection/chip_cooling

From the **File** menu, choose **New**.

NEW

In the **New** window, click ![icon] **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click ![icon] **3D**.

2 In the **Select Physics** tree, select **Heat Transfer>Heat Transfer in Solids and Fluids (ht)**.

3 Click **Add**.

4 Click ➡ **Study**.

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

6 Click ☑ **Done**.

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 Click 📂 **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file
   `chip_cooling.txt`.

GEOMETRY 1

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 **Width** text field, type L_chip.

4 In the **Depth** text field, type L_chip.

5 In the **Height** text field, type H_chip.

6 Locate the **Position** section. In the **z** text field, type -H_chip.

7 Click 🔲 **Build Selected**.

PART LIBRARIES

1 In the **Geometry** toolbar, click △ **Parts** and choose **Part Libraries**.

2 In the **Model Builder** window, click **Geometry 1**.

3 In the **Part Libraries** window, select **Heat Transfer Module>Heat Sinks>
   heat_sink_straight_fins** in the tree.

4 Click 📥 **Add to Geometry**.

GEOMETRY 1

Heat Sink - Straight Fins 1 (pi1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Heat Sink -
   Straight Fins 1 (pi1)**.

2 In the **Settings** window for **Part Instance**, locate the **Input Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
n_fins_x	n_fins	4	Amount of fins in x direction
X_fins_bottom	3[mm]	0.003 m	Fin dimension in x direction, bottom
X_fins_top	2[mm]	0.002 m	Fin dimension in x direction, top

4 Locate the **Position and Orientation of Output** section. Find the **Coordinate system in part** subsection. From the **Work plane in part** list, choose **Work plane for heat sink base (wp11)**.

5 Find the **Displacement** subsection. In the **xw** text field, type -5[mm].

6 In the **yw** text field, type -5[mm].

7 Click to expand the **Domain Selections** section. In the table, enter the following settings:

Name	Keep	Physics	Contribute to
Heat sink base		√	None
Step domain		√	None
Array of fins		√	None
All	√	√	None

8 Click to collapse the **Domain Selections** section. Click to expand the **Boundary Selections** section. In the table, enter the following settings:

Name	Keep	Physics	Contribute to
Bottom boundary		√	None
Exterior		√	None
Exterior boundaries without heat sink base	√	√	None
All		√	None

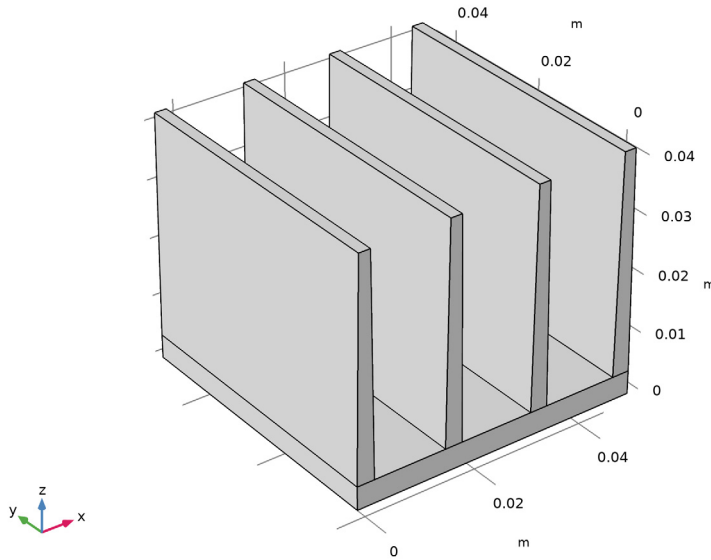**9** Click to collapse the **Boundary Selections** section. In the **Geometry** toolbar, click **Build All**.

10 Click the ⊕ **Zoom Extents** button in the **Graphics** toolbar.

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>Aluminum**.

4 Click ⬛ **Add to Component 1 (comp1)**.

5 In the tree, select **Built-in>Silica glass**.

6 Click ⬛ **Add to Component 1 (comp1)**.

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

MATERIALS

Silica glass (mat2)

1 Click the ⊕ **Zoom Extents** button in the **Graphics** toolbar.

2 Click the ⬛ **Wireframe Rendering** button in the **Graphics** toolbar.

3 Select Domain 3 only.

In order to easily reuse the selection of the domain corresponding to the chip, create a dedicated selection for it.

4 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

5 Click $\blacksquare$ **Create Selection**.

6 In the **Create Selection** dialog box, type Chip in the **Selection name** text field.

7 Click **OK**.

DEFINITIONS

Ambient Properties 1 (ampr1)
In the **Physics** toolbar, click $\blacksquare$ **Shared Properties** and choose **Ambient Properties**.

HEAT TRANSFER IN SOLIDS AND FLUIDS (HT)

Heat Source 1
1 In the **Model Builder** window, under **Component 1 (comp1)** right-click
Heat Transfer in Solids and Fluids (ht) and choose **Heat Source**.

2 In the **Settings** window for **Heat Source**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **Chip**.

4 Locate the **Heat Source** section. Click the **Heat rate** button.

5 In the P_0 text field, type P0.

Heat Flux 1
1 In the **Physics** toolbar, click $\blacksquare$ **Boundaries** and choose **Heat Flux**.

2 In the **Settings** window for **Heat Flux**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Exterior boundaries without heat sink base (Heat Sink - Straight Fins 1)**.

4 Locate the **Heat Flux** section. Click the **Convective heat flux** button.

Choose external forced convection.

5 Enter the flow velocity. Enter length of plate .

Then the external temperature is set to the ambient temperature defined in the **Ambient Properties 1** node (the default value is 293.15 K).

6 From the T_{ext} list, choose **Ambient temperature (ampr1)**.

7 In the **Home** toolbar, click $\blacksquare$ **Compute**.

The computation takes a few seconds and about 1 GB of memory.

*Temperature (ht)*

Two default plots are generated automatically. The first one shows the temperature profile on the boundaries, and the second one shows the isothermal surfaces.

Surface

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

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

3 From the **Unit** list, choose **degC**.

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

Surface: Temperature (degC)
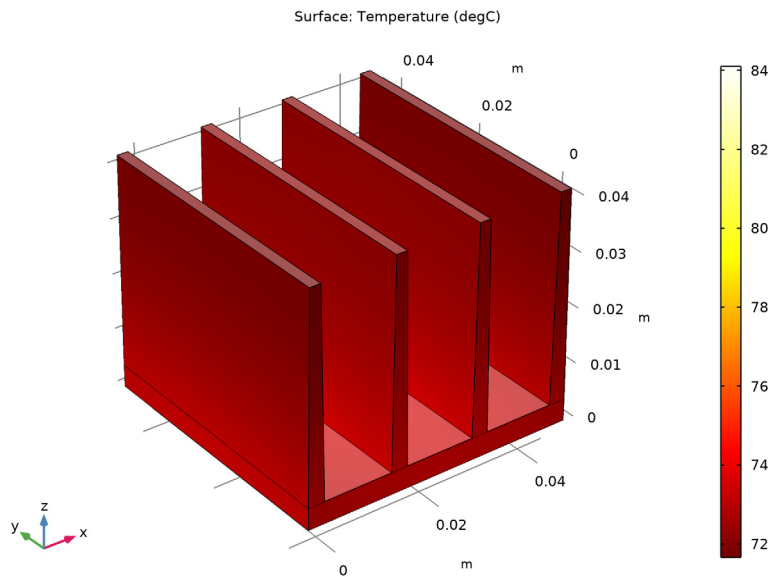
GEOMETRY I

1 Click the ↘ **Go to Default View** button in the **Graphics** toolbar.

2 Click the ⟷ **Zoom Extents** button in the **Graphics** toolbar.

Now update the model to evaluate the effect of the thermal contact between the chip and the heat sink. First assume a poor thermal contact due to a thin film of air between the chip and the heat sink.