MNIS – Physical models for micro and nanosystems

Exercise 8: Heat transfer in an air-cooled heatsink

WHAT ARE WE GOING TO LEARN:

-typical boundary conditions encountered heat transfer modeling

-combining heat transfer with convective fluid flow

In this model, we will cover an example of heat transfer around a heat sink cooled by laminar airflow.

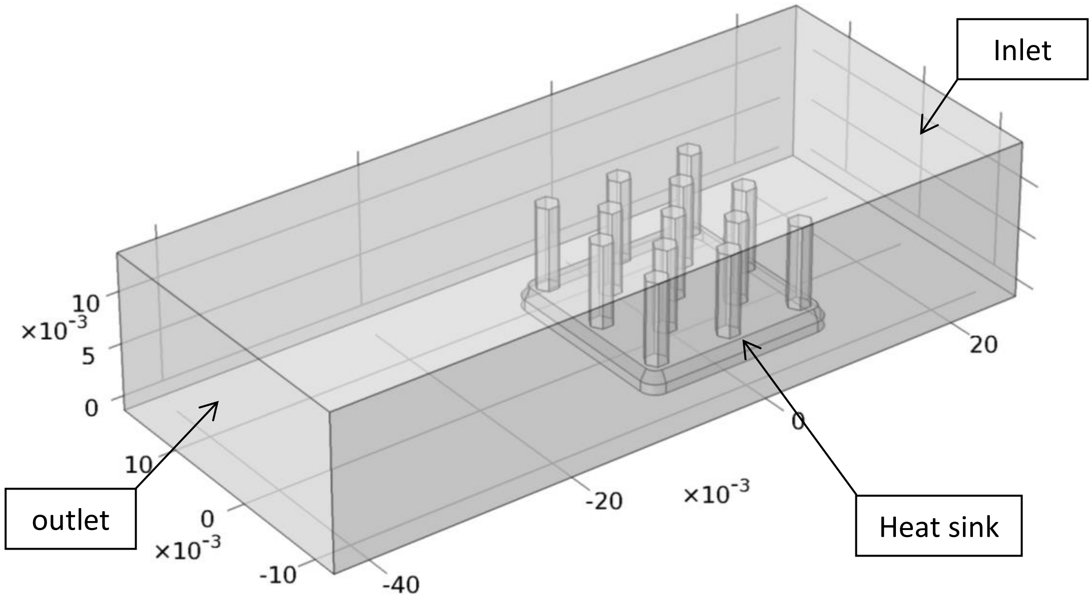


Fig. 1. The model.

The model finds the thermal balance between the heat sink and the air flowing through the channel. Heat is transported via conduction in the heat sink and through conduction and convection in the cooling air. The temperature is set at the inlet and the outlet. The base of the heat sink receives a 1 W heat flux. The transport of thermal energy at the outlet is dominated by convection. The airflow velocity is defined by a parabolic velocity profile (fully developed Poiseuille flow). We have a constant pressure at the outlet. At all solid surfaces we have the no-slip and no penetration boundary conditions.

1. BUILDING THE MODEL

We will first start Comsol and define the type of geometry as **3D** and choose the physics package **Heat Transfer ► Conjugate Heat Transfer ► Laminar Flow.** Select **stationary study**. Choose **done.**

Let us define the following parameters:

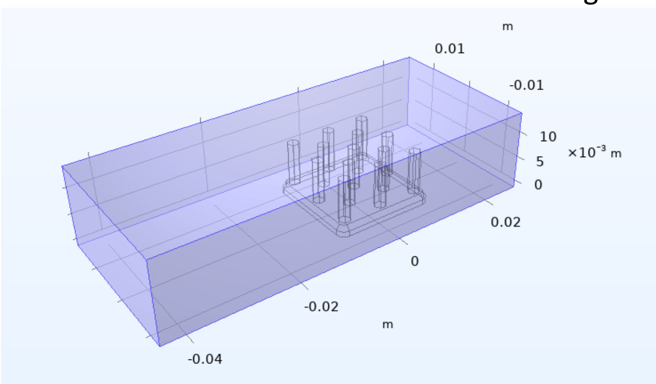
Name	**Expression**	**Description**
L_channel	7 [cm]	channel length
W_channel	3 [cm]	channel width
H_channel	1.5[cm]	channel height
Uu	5	
U0	Uu*1 [cm/s]	mean inlet velocity
Heating_pwr	1 [W]	heating power

We will now import the binary file with the description of the heatsink. In the **Model Builder**, right-click **Model 1 ► Geometry 1** and choose **Import** . Go to the Settings window for Import. Locate the Import section. Click the Browse button. Browse to the model's Model Library folder and double-click the file heat_sink_n19.mphbin. Click the Build Selected button.
This generates the heat sink part of the model.

We should now build the channel.

In the **Model Builder**, right-click **Geometry 1** and choose **Work Plane**. In the **Model Builder**, right-click **Plane Geometry** (below the **Work Plane 1** node) and choose **Rectangle**. Go to the Settings window for Rectangle. Locate the Size and Shape section. In the **Width** field, enter **L_channel**. In the **Height** field, enter **W_channel**. Locate the Position section. In the **xw** field, enter -4.5e-2. In the **yw** field, enter **-W_channel/2**. Click the Build Selected button.

We now need to extrude the rectangle to form the 3D channel. In the Model Builder, right-click **Work Plane 1** (wp1) and choose **Extrude**. Go to the Settings window for **Extrude**. Locate the **Distances from Work Plane** section. In the table under **Distances** enter **H_channel**. Click the **Build Selected** button. Click the **Transparency** button on the Graphics toolbar  , next to the scene light icon. Your model should now look something like this:



2. DEFINING SELECTIONS

Here, we are going to define selections, these will help us select different characteristic parts of the model when assigning materials and boundary conditions. To do this, right-click on **definitions ► selections,** choose **explicit** from the menu. In the settings window, you can choose **domain** or **boundary**. Let us create domain selections called **tube** for the part of the model containing the fluid and **heatsink** for the heatsink. We should also define the boundaries **inlet** (boundary 115), **outlet** (boundary 1) and **heatsink bottom** (boundary 8)

3. DEFINING MATERIAL PROPERTIES

The fluid contained in the rectangular channel will be air, while the heat sink will be made of aluminum. Locate the built-in material **air** in the material browser and assign it to the channel. Similarly, add the **Aluminum 3003-H18** material and assign it to the heatsink.

4. DEFINING BOUNDARY CONDITIONS

Fluid transport

Here, we need to first define the boundary conditions for the fluid flow. In the **Model Builder**, select **Laminar Flow (spf)** under **Component 1 (comp1)** and keep Domain 1 (air channel) only under **Domain selection**. This specifies that domain 1 is made of a fluid, in this case air. We now have to define the inlet and outlet boundary condition for the fluid flow. In the **Model Builder**, right-click **Laminar Flow (spf)** and choose the boundary condition **Inlet**. Select **Inlet** from the **Boundary selection** dropdown menu (Boundary 115, vertical place in the back of the channel, closer to the heatsink). Go to the **Settings** window for **Inlet**. Locate the Boundary Condition section. From the Boundary condition list, select **Fully developed flow**.

Locate the **Fully developed flow** section. In the $\mathbf{U_{av}}$ field, enter **U0**. In the Model Builder, right-click **Laminar Flow (spf)** and choose the boundary condition **Laminar Flow ► Outlet** Select Boundary 1 only and choose **Pressure** under **Boundary Condition** (should already be selected by default).

Heat transport

In the **Model Builder**, right-click **Heat Transfer (ht)** and choose the boundary condition **Heat Transfer ▸ Temperature**. Select the boundary that corresponds to the inlet. The default temperature, corresponding to 20 degrees applies at the inlet.

Still under **Heat Transfer (ht)**, make sure that under **Solid 1** you only have the heatsink selected (domain 2), while under **Fluid 1** you have the tube (domain 1). Rename these two conditions accordingly.

In the **Model Builder**, right-click **Heat Transfer (ht)** and choose the boundary condition **Heat Transfer ▸ Heat Flux.** Select Boundary 8 (bottom of the heatsink) only. Go to the Settings window for **Heat Flux**. Locate the **Heat Flux** section. Click the **Heat Rate** selection button. In the $\mathbf{P_0}$ field, enter **Heating_pwr**. This corresponds to 1W of heating being generated under the heatsink with the initial parameters.

In the **Model Builder**, right-click **Heat Transfer (ht)** and choose the boundary condition **Flow conditions ▸ Outflow.** Select Boundary 1 (pipe outlet) only. This defines the boundary through which the heat is transported out of the system.

5. MESH

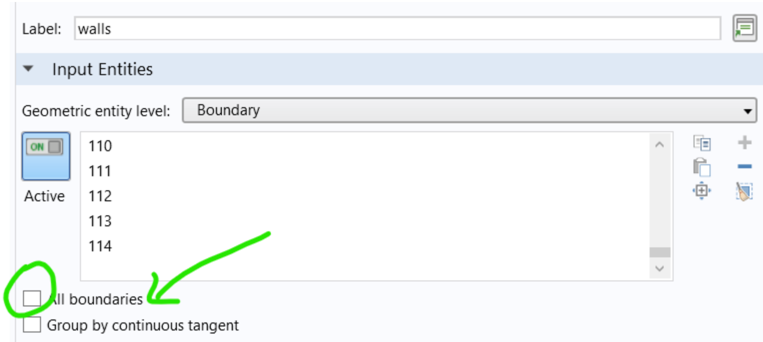
Because out model has both an imported and a manually created domain, we need to explicitly create the mesh. For this, select **Mesh 1, keep it as physics-controlled mesh** and select Extremely **Coarse** under **Element size**. Select **build all**.

6.

6. SOLUTION

In the **Model Builder**, right-click **Study 1** and choose **Compute**. In order to create more useful plots, we will look at the temperature distribution in the channel and velocity distribution in the fluid (air) in order to see the convection effect (change of velocity around the heated element). We will first define a subset of boundaries that we will use for subsequent result visualization. The purpose of this is to display 3D plots for the selection only, allowing us to "see into" the model. This is in general a useful approach when dealing with 3D models.

In the Model Builder, right-click **Component 1 (comp1) ► Definitions** and choose **Selections ► Explicit**. Go to the **Settings** window for **Explicit**. Locate the **Input Entities** section. From the **Geometric entity level** list, select **Boundary**. Right-click **Explicit 6** and choose **Rename**. Go to the **Rename Explicit** dialog box and enter **walls** in the **New name field**. To populate the entity list, select first **All boundaries** (see picture)**.** Keep boundaries 3 and 5–114 only.



In the **Model Builder**, expand the **Data Sets** node under **Results**. Right-click **Results ► Data Sets ► Study 1/Solution 1** and choose **Selection**. Go to the **Settings** window for **Selection**. Locate the **Geometric Entity Selection** section. From the **Geometric entity** level list, select **Boundary**. From the **Selection** list, select **walls**.

Plotting the results

In the **Model Builder**, click **Results ► Temperature (ht)**. Go to the **Settings** window for **3D Plot Group**. Locate the **Data** section. From the **Data set** list, select **Solution 1**. Right-click **Results ► Temperature (ht)** and choose **Arrow Volume**. Go to the **Settings** window for **Arrow Volume**. In the upper-right corner of the **Expression** section, click **Replace Expression**. From the menu, search for **velocity** and choose **u, v, w - velocity field.**

Go to the **Settings** window for **Arrow Volume**. Locate the **Arrow Positioning** section. Find the **x grid points** subsection. In the **Points** field, enter **40**. Find the **y grid points** subsection. In the **Points** field, enter **20**. Find the **z grid points** subsection. From the **Entry method list**, select **Coordinates**. In the **Coordinates** field, enter **5e-3**. In the **Model Builder**, right-click **Arrow Volume 1** and choose **Color Expression**. Go to the **Settings** window **for Color Expression**.

In the upper-right corner of the **Expression** section, click **Replace Expression**. From the menu, search for **velocity** and choose **spf.U - velocity magnitude**.
Click the **Plot** button.

The end result should look something like this:

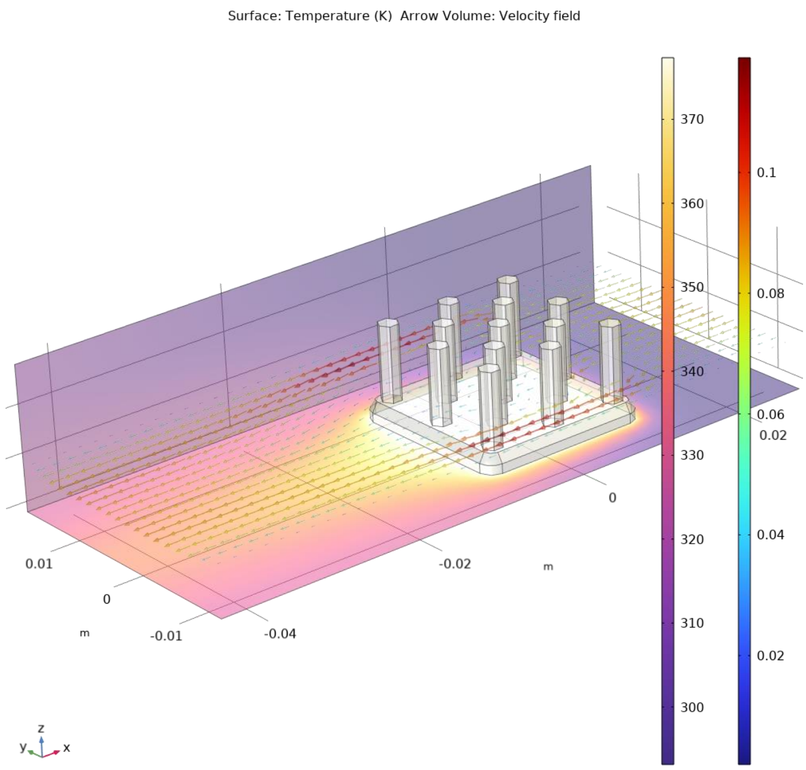


Fig. 2. Finite element solution

You can also measure the temperature at different portions of the heatsink to evaluate the effectiveness of convective cooling. To do this, right-click on **Results -> Data sets** and add **Edge 3D**. Under selection, add edge 148 (one of the edges on the top of the middle fin). Rename it to **top edge**. Under **derived values** add **average –> line average** and choose **top edge** under **selection**. Click evaluate (orange equal sign on top of the settings subwindow). You should be getting around 375K for a mean inlet velocity of 0.05 m/s (defined in the parameters).