Non-Isothermal MEMS Heat Exchanger

Introduction

The following example builds and solves a conduction and convection heat transfer problem using the Non-Isothermal Flow physics.

The example concerns a stainless-steel MEMS heat exchanger, which you can find in lab-on-a-chip devices in biotechnology and in microreactors such as for micro-fuel cells. This model examines the heat exchanger in 3D, and it involves heat transfer through both convection and conduction.

Model Definition—Heat Exchanger

Figure 1 shows the geometry of the heat exchanger. It is necessary to model only one unit cell because they are all almost identical except for edge effects in the outer cells.

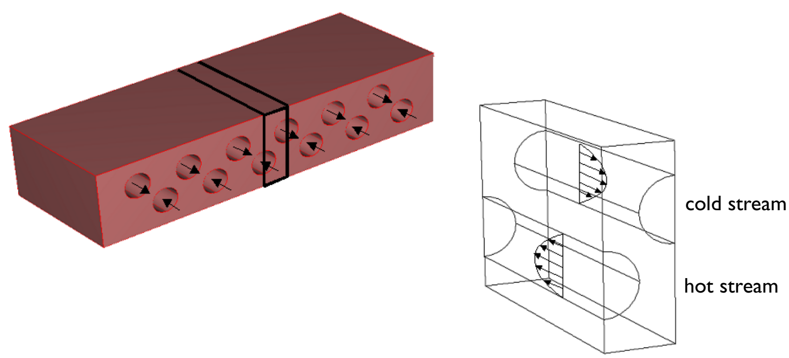


Figure 1: Depiction of the modeled part of the heat exchanger (left).

The governing equation for this model is the heat equation for conductive and convective heat transfer

$$\nabla \cdot (-k\nabla T) = Q - \rho C_p \mathbf{u} \cdot \nabla T \qquad (1)$$

where C_p denotes the specific heat capacity (J/(kg·K)), T is the temperature (K), k is the thermal conductivity (W/(m·K)), ρ is the density (kg/m^3), $\mathbf{u}$ is the velocity vector (m/s), and Q is a sink or source term (which you set to zero because there is no production or consumption of heat in the device).

In the solid part of the heat exchanger the velocity vector, $\mathbf{u} = (u, v, w)$, is set to zero in all directions. In the channels the velocity field is defined by an analytical expression that approximates fully-developed laminar flow for a circular cross section. For both the hot and cold streams, you set the velocity components in the x and z directions to zero.

For the hot stream, the expression

$$v = v_{\mathrm{max}}\left(1-\left(\frac{r}{R}\right)^2\right) \tag{2}$$

gives the y-component of the velocity where

- v_{max} is the maximum velocity (m/s), which arises in the middle of the channel
- r is the distance from the center of the channel (m)
- R is the channel radius (m)

You describe velocity in the cold stream in the same manner but in the opposite direction

$$v = -v_{\mathrm{max}}\left(1-\left(\frac{r}{R}\right)^2\right) \tag{3}$$

In an extended approach, instead of using the analytical expression for the velocity field, the fluid in the channels can be simulated using the Non-Isothermal Flow physics interface. Here the density is defined as

$$\rho = \rho_{\mathrm{m}}\left(1-\frac{T-T_{\mathrm{m}}}{T_{\mathrm{m}}}\right) \tag{4}$$

where ρ_{m} the mean density (kg/m^3), T is the temperature (K), and $T_{\mathrm{m}} = (T_{\mathrm{cold}} + T_{\mathrm{hot}})/2$ is the mean fluid temperature.

The boundary conditions are insulating for all outer surfaces except for the inlet and outlet boundaries in the fluid channels. At the inlets, you specify constant temperatures for the cold and hot streams, T_{cold} and T_{hot}, respectively. At the outlets, convection dominates the transport of heat so you apply the convective flux boundary condition:

$$-k\nabla T\cdot\mathbf{n} = 0$$

Results and Discussion

Figure 2 shows the temperature isosurfaces and the heat flux streamlines for the conductive heat flux in the device. The temperature isosurfaces clearly show the convective term's influence in the channels. Figure 3 displays the corresponding results for the extended model. As the plot shows, the temperature distribution is very similar to that in the first study, which can therefore be concluded to be a good approximation of the extended case.

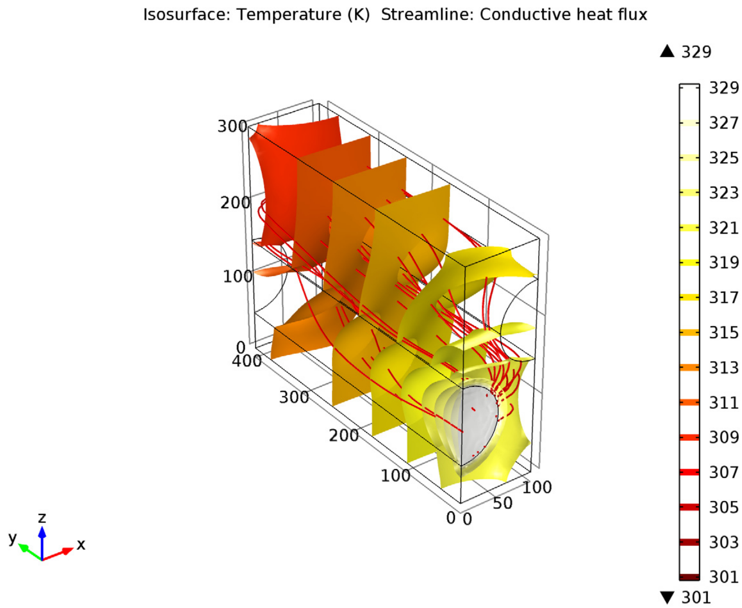


Figure 2: Isotherms and conductive heat flux streamlines in the cell unit's geometry.

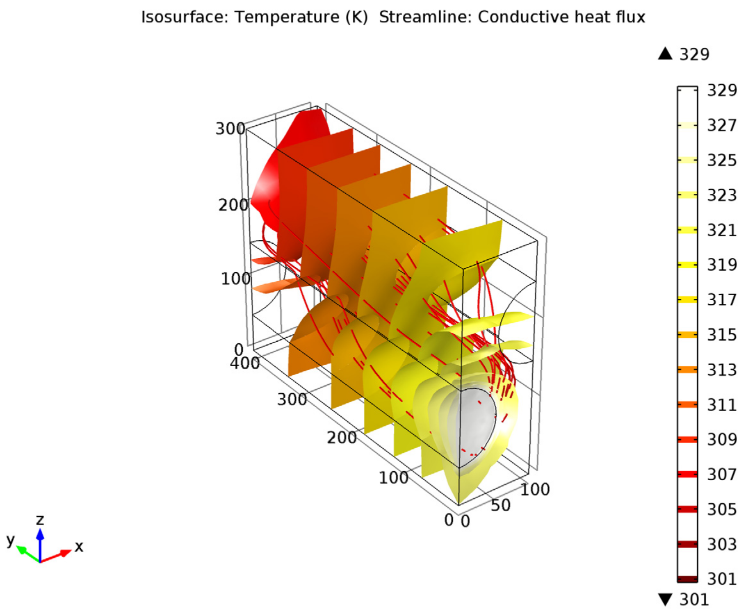


Figure 3: Extended model results; isotherms and conductive heat flux streamlines in the cell unit's geometry.

Model Library path: `Heat_Transfer_Module/Heat_Exchangers/heat_exchanger_ni`

Modeling Instruction

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

NEW

1 In the **New** window, click the **Model Wizard** button.

MODEL WIZARD

1 In the **Model Wizard** window, click the **3D** button.

2 In the **Select physics** tree, select **Fluid Flow>Non-Isothermal Flow>Laminar Flow (nitf)**.

3 Click the **Add** button.

4 Click the **Study** button.

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

6 Click the **Done** button.

GLOBAL DEFINITIONS

Parameters

1 On the **Home** toolbar, click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
R	50[um]	5.000E-5 m	Channel radius
v_mean	2.5[mm/s]	0.002500 m/s	Mean velocity
T_hot	330[K]	330.0 K	Temperature, hot channel
T_cold	300[K]	300.0 K	Temperature, cold channel
T_mean	(T_hot+T_cold)/2	315.0 K	Mean temperature
rho_mean_w	1000[kg/m^3]	1000 kg/m^3	Fluid mean density

GEOMETRY 1

1 In the **Model Builder** window, under **Component 1** click **Geometry 1**.

2 In the **Geometry** settings window, locate the **Units** section.

3 From the **Length unit** list, choose **µm**.

Work Plane 1

1 On the **Geometry** toolbar, click **Work Plane**.

2 In the **Work Plane** settings window, locate the **Plane Definition** section.

3 From the **Plane** list, choose **zx-plane**.

Rectangle 1

1 In the **Model Builder** window, under **Component 1>Geometry 1>Work Plane 1** right-click **Plane Geometry** and choose **Rectangle**.

2 In the **Rectangle** settings window, locate the **Size** section.

3 In the **Width** edit field, type 300.

4 In the **Height** edit field, type 100.

5 Click the **Build Selected** button.

6 Click the **Zoom Extents** button on the Graphics toolbar.

Circle 1

1 Right-click **Plane Geometry** and choose **Circle**.

2 In the **Circle** settings window, locate the **Size and Shape** section.

3 In the **Radius** edit field, type R.

4 In the **Sector angle** edit field, type 180.

5 Locate the **Position** section. In the **xw** edit field, type 100.

6 Click the **Build Selected** button.

Circle 2

1 Right-click **Plane Geometry** and choose **Circle**.

2 In the **Circle** settings window, locate the **Size and Shape** section.

3 In the **Radius** edit field, type R.

4 In the **Sector angle** edit field, type 180.

5 Locate the **Position** section. In the **xw** edit field, type 200.

6 In the **yw** edit field, type 100.

7 Locate the **Rotation Angle** section. In the **Rotation** edit field, type 180.

8 Click the **Build Selected** button.

9 On the **Work Plane** toolbar, click **Close**.

Extrude 1

1 On the **Geometry** toolbar, click **Extrude**.

2 In the **Extrude** settings window, locate the **Distances from Plane** section.

3 In the table, enter the following settings:

Distances (µm)
400

4 Click the **Build Selected** button.

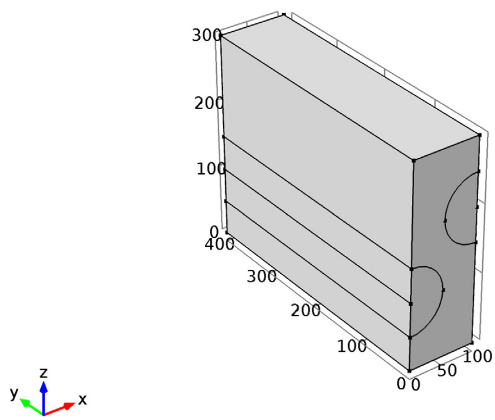
5 Click the **Zoom Extents** button on the Graphics toolbar.

Ignore Edges 1

1 On the **Geometry** toolbar, click **Virtual Operations** and choose **Ignore Edges**.

2 On the object **fin**, select Edges 8 and 33 only.

3 Click the **Build Selected** button.



DEFINITIONS

Explicit 1

1 On the **Definitions** toolbar, click **Explicit**.

2 In the **Model Builder** window, under **Component 1>Definitions** right-click **Explicit 1** and choose **Rename**.

3 Go to the **Rename Explicit** dialog box and type `solid` in the **New name** edit field.

4 Click **OK**.

5 Select Domain 1 only.

Explicit 2

1 On the **Definitions** toolbar, click **Explicit**.

2 Select Domains 2 and 3 only.

3 Right-click **Component 1>Definitions>Explicit 2** and choose **Rename**.

4 Go to the **Rename Explicit** dialog box and type `channels` in the **New name** edit field.

5 Click **OK**.

Explicit 3

1 On the **Definitions** toolbar, click **Explicit**.

2 Select Domain 2 only.

3 Right-click **Component 1>Definitions>Explicit 3** and choose **Rename**.

4 Go to the **Rename Explicit** dialog box and type `hot channel` in the **New name** edit field.

5 Click **OK**.

Explicit 4

1 On the **Definitions** toolbar, click **Explicit**.

2 Select Domain 3 only.

3 Right-click **Component 1>Definitions>Explicit 4** and choose **Rename**.

4 Go to the **Rename Explicit** dialog box and type `cold channel` in the **New name** edit field.

5 Click **OK**.

Explicit 5

1 On the **Definitions** toolbar, click **Explicit**.

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

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

4 Select Boundaries 6, 8, 12, and 13 only.

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

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

7 Click **OK**.

Variables 1

1 Right-click **Definitions** and choose **Variables**.

2 In the **Variables** settings window, locate the **Geometric Entity Selection** section.

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

4 From the **Selection** list, choose **channels**.

5 Locate the **Variables** section. In the table, enter the following settings:

Name	Expression	Unit	Description
`rho_w`	`rho_mean_w*(1-(T-T_mean)/T_mean)`		`Fluid density`

MATERIALS

On the **Home** toolbar, click **Add Material**.

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>Steel AISI 4340**.

3 In the **Add material** window, click **Add to Component**.

MATERIALS

Steel AISI 4340

1 In the **Model Builder** window, under **Component 1>Materials** click **Steel AISI 4340**.

2 In the **Material** settings window, locate the **Geometric Entity Selection** section.

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

ADD MATERIAL

1 Go to the **Add Material** window.

2 In the tree, select **Built-In>Water, liquid**.

3 In the **Add material** window, click **Add to Component**.

MATERIALS

Water, liquid

1 In the **Model Builder** window, under **Component 1>Materials** click **Water, liquid**.

2 In the **Material** settings window, locate the **Geometric Entity Selection** section.

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

NON-ISOTHERMAL FLOW

Fluid 1

1 In the **Model Builder** window, expand the **Component 1>Non-Isothermal Flow** node, then click **Fluid 1**.

2 In the **Fluid** settings window, locate the **Thermodynamics** section.

3 From the ρ list, choose **User defined**. In the associated edit field, type `rho_w`.

4 From the γ list, choose **User defined**. In the associated edit field, type `1`.

Heat Transfer in Solids 1

1 On the **Physics** toolbar, click **Domains** and choose **Heat Transfer in Solids**.

2 In the **Heat Transfer in Solids** settings window, locate the **Domain Selection** section.

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

Inlet 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Inlet**.

2 Select Boundary 5 only.

3 In the **Inlet** settings window, locate the **Boundary Condition** section.

4 From the **Boundary condition** list, choose **Laminar inflow**.

5 Locate the **Laminar Inflow** section. In the U_{av} edit field, type `v_mean`.

Inlet 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Inlet**.

2 Select Boundary 15 only.

3 In the **Inlet** settings window, locate the **Boundary Condition** section.

4 From the **Boundary condition** list, choose **Laminar inflow**.

5 Locate the **Laminar Inflow** section. In the U_{av} edit field, type `v_mean`.

Outlet 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Outlet**.

2 Select Boundaries 11 and 14 only.

Symmetry, Flow 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Symmetry, Flow**.

2 Select Boundaries 4 and 17 only.

Temperature 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

2 Select Boundary 5 only.

3 In the **Temperature** settings window, locate the **Temperature** section.

4 In the T_0 edit field, type `T_hot`.

Temperature 2

1 On the **Physics** toolbar, click **Boundaries** and choose **Temperature**.

2 Select Boundary 15 only.

3 In the **Temperature** settings window, locate the **Temperature** section.

4 In the T_0 edit field, type `T_cold`.

Outflow 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Outflow**.

2 Select Boundaries 11 and 14 only.

MESH 1

Free Triangular 1

1 In the **Model Builder** window, under **Component 1** right-click **Mesh 1** and choose **More Operations>Free Triangular**.
2 In the **Free Triangular** settings window, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **channel walls**.

Size 1

1 Right-click **Component 1>Mesh 1>Free Triangular 1** and choose **Size**.
2 In the **Size** settings window, locate the **Element Size** section.
3 Click the **Custom** button.
4 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
5 In the associated edit field, type `10[um]`.
6 In the **Model Builder** window, right-click **Mesh 1** and choose **Free Tetrahedral**.
7 Right-click **Mesh 1** and choose **Build All**.

STUDY 1

Solver 1

1 On the **Study** toolbar, click **Show Default Solver**.
2 In the **Model Builder** window, expand the **Study 1>Solver Configurations** node.
3 In the **Model Builder** window, expand the **Solver 1** node, then click **Dependent Variables 1**.
4 In the **Dependent Variables** settings window, locate the **Scaling** section.
5 From the **Method** list, choose **None**.
6 On the **Study** toolbar, click **Compute**.

RESULTS

Velocity (nitf)

The first default plot shows the velocity magnitude on slices.

Temperature (nitf)

The second default plot shows the temperature on channel inner surfaces.

To reproduce Figure 3, proceed as follows:

3D Plot Group 3

1 On the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Model Builder** window, under **Results** right-click **3D Plot Group 3** and choose **Isosurface**.

3 In the **Isosurface** settings window, locate the **Expression** section.

4 Verify that the **Temperature (T)** is defined as default expression.

5 In the **Levels** edit field, type `range(301,2,329)`.

6 Locate the **Coloring and Style** section. From the **Color table** list, choose **ThermalLight**.

7 In the **Model Builder** window, right-click **3D Plot Group 3** and choose **Streamline**.

8 In the **Streamline** settings window, locate the **Expression** section.

9 Click **Conductive heat flux (nitf.dfluxx,...,nitf.dfluxz)** in the upper-right corner of the section. Locate the **Streamline Positioning** section. From the **Positioning** list, choose **Start point controlled**.

10 Locate the **Coloring and Style** section. From the **Line type** list, choose **Tube**.

11 Select the **Radius scale factor** check box.

12 On the **3D plot group** toolbar, click **Plot**.

13 Right-click **3D Plot Group 3** and choose **Rename**.

14 Go to the **Rename 3D Plot Group** dialog box and type `Temperature isosurfaces and Conductive heat flux streamlines` in the **New name** edit field.

15 Click **OK**.