Created in COMSOL Multiphysics 5.6

Performance of a Porous Microchannel Heat Sink

Introduction

This model studies the performance of a microchannel heat sink (MCHS) with a porous block structure and compares its performance with that of a conventional MCHS. Todays demands on electronic components to become smaller and more efficient at the same time place equally high demands on the corresponding cooling devices. The use of porous material along the flow channels can enhance the heat transfer, by increasing the heat transfer surface area. At the same time, the pressure drop is also increased, requiring more pumping power. With a parametric study over the thickness of the porous substrate an optimized design of the porous MCHS can be found.

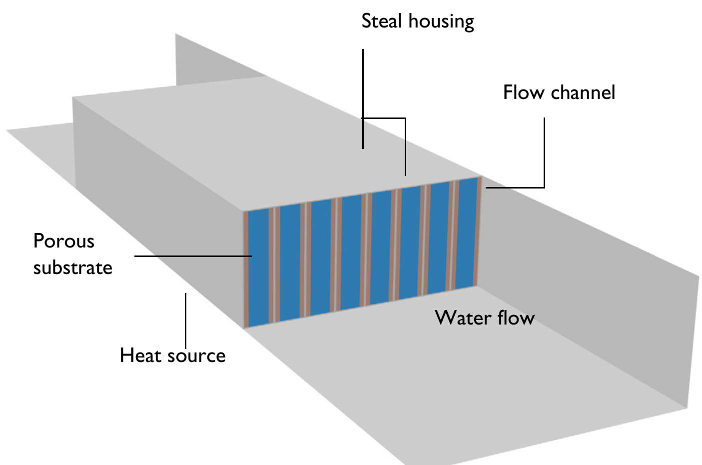


Figure 1: Geometry and operating conditions of the porous MCHS.

Model Definition

The design and operating conditions are taken from Ref. 1 and are illustrated in Figure 1. The problem can be reduced to modeling only one half of a single channel. This is sufficient, because the performance is mainly influenced by the pressure drop and heat transfer from the bottom boundary to the water in the channel. The geometry of the modeled domain is shown in Figure 2.

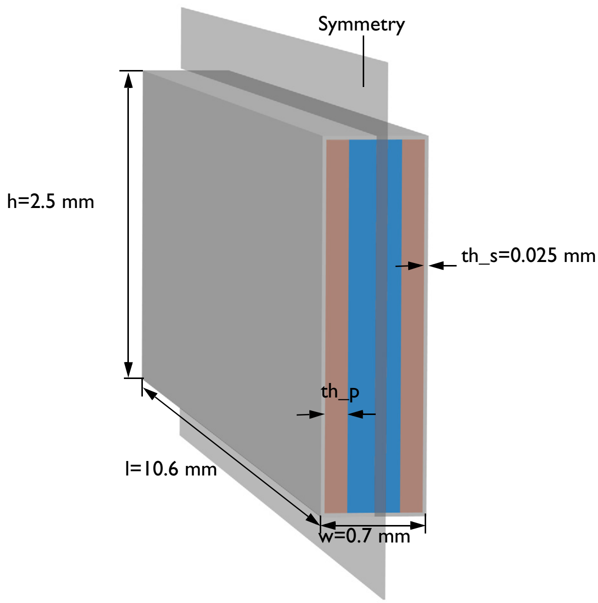


Figure 2: Geometry of the modeling domain. The free flow domain used to provide the inflow profile for the MCHS is not shown.

The flow channels contain sintered porous metal blocks with a porosity of $\varepsilon = 0.402$ on each side. A heat source with $q_{\mathrm{in}} = 100\ \mathrm{W/cm}^2$ is attached to the bottom. Water with an inlet velocity of $u = 0.2\mathrm{m/s}$ and a temperature of $\mathrm{T}_{\mathrm{in}} = 300\ \mathrm{K}$ is used as cooling fluid. The flow is assumed to be laminar, incompressible and stationary. The flow properties are also independent of the temperature field. Inside the porous domains the governing equation is the Brinkman equation with a Forchheimer correction term (also known as the Brinkman-Forchheimer or Darcy-Brinkman-Forchheimer equation). The pressure drop depends on the velocity field **u** as

$$-\nabla p = \frac{\mu}{\kappa}\mathbf{u} + \frac{c_F}{\sqrt{\kappa}}\rho\mathbf{u}|\mathbf{u}| \tag{1}$$

where μ (Pa·s) is the fluid viscosity, ρ (kg/m^3) the density, and κ (m^2) the permeability of the porous substrate.

To evaluate the performance of the porous MCHS over the conventional MCHS, the first computation solves the model assuming only free flow. Then, a second study performs a parametric sweep over the thickness of the porous substrate ($\mathrm{th_p}$). The following performance parameters are evaluated:

1 Pressure drop, that is the pressure difference between inlet and outlet of the porous MCHS

2 Average heat transfer coefficient of the MCHS, given by

$$h_{\mathrm{mchs}} = \frac{q_{\mathrm{in}}}{\overline{T}_{\mathrm{w}} - T_{\mathrm{in}}} \tag{2}$$

with the average wall temperature at the bottom centerline $\overline{T}_{\mathrm{w}}$.

3 Reynolds number is defined as

$$\mathrm{Re} = \frac{\rho u_{\mathrm{in}} D_{\mathrm{h}}}{\mu} \tag{3}$$

with the hydraulic diameter D_h (m) that is defined based on the length and width of the free flow channel, l_f and w_f respectively, as follows:

$$D_{\mathrm{h}} = \frac{2 l_{\mathrm{f}} w_{\mathrm{f}}}{l_{\mathrm{f}} + w_{\mathrm{f}}}$$

4 The Nusselt number describes the ratio of convective to conductive heat transfer according to

$$\mathrm{Nu} = \frac{D_{\mathrm{h}} h_{\mathrm{mchs}}}{k_{\mathrm{f}}} \tag{4}$$

where k_f is the fluids thermal conductivity.

5 The Figure of Merit (FOM) compares the performance of two different designs with the following expression:

$$\mathrm{FOM} = \frac{h_{\mathrm{mchs}} / h_{\mathrm{mchs,\,base}}}{(\Omega / \Omega_{\mathrm{base}})^{1/3}} \tag{5}$$

The index `base` refers to the values for the MCHS without the porous structure and $\Omega = u_{\mathrm{in}} l_{\mathrm{f}} w_{\mathrm{f}} \Delta p$ is the pumping power.

[Equation 1](#) is valid for $1 \leq Re \leq 1000$. An estimation of the Reynolds number ([Equation 3](#)) results in $Re \sim 300$ such that the choice of the Brinkman-Forchheimer equation is valid.

Results and Discussion

Figure 3 shows the velocity field in a cross section of the channel. The velocity magnitude inside the porous structure is small (dark blue) compared to that of the free flow channel.

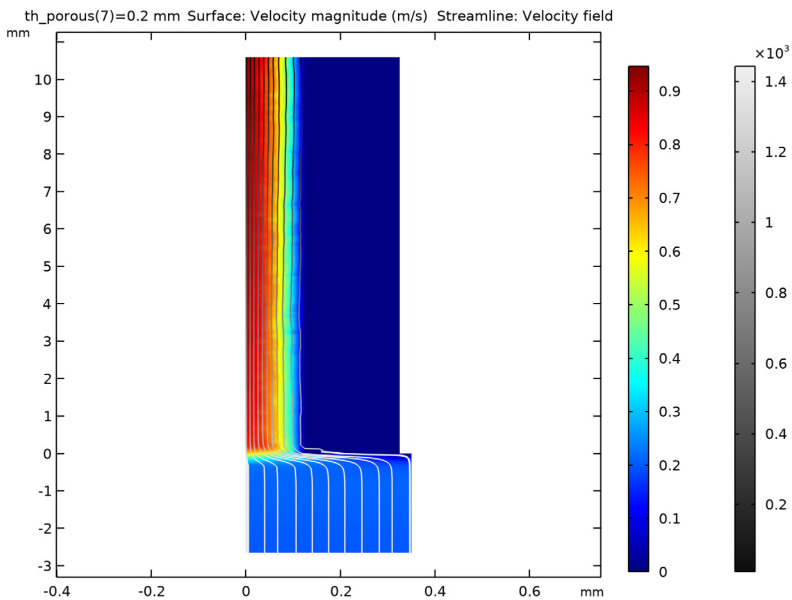


Figure 3: Cross section of the velocity field along the channel (scaled view). The dark blue color indicates the porous structure, because the velocity magnitude is small in this area.

The temperature distribution is shown together with the velocity profile in Figure 4.

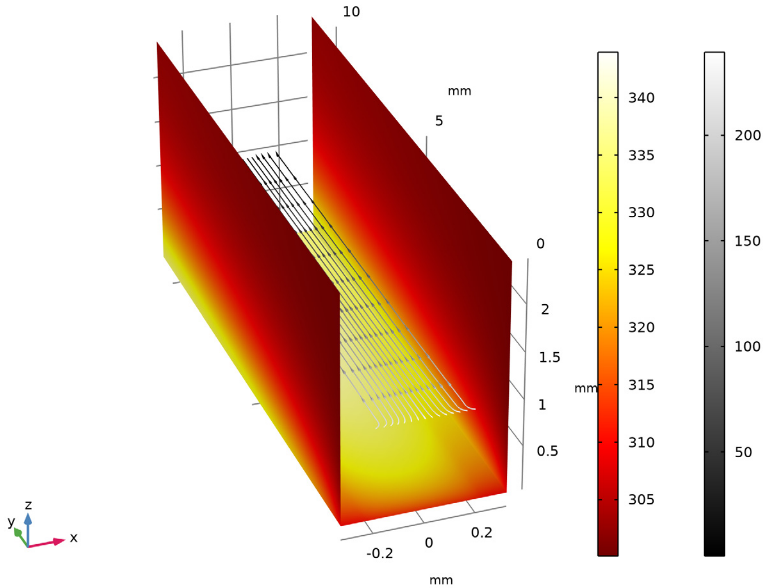


Figure 4: Temperature distribution (color) and velocity field (arrows) with the gray scale indicating the pressure.

The pressure drop and average heat transfer coefficient as a function of the thickness of the porous structure are shown in Figure 5. With increasing thickness, both values also increase.

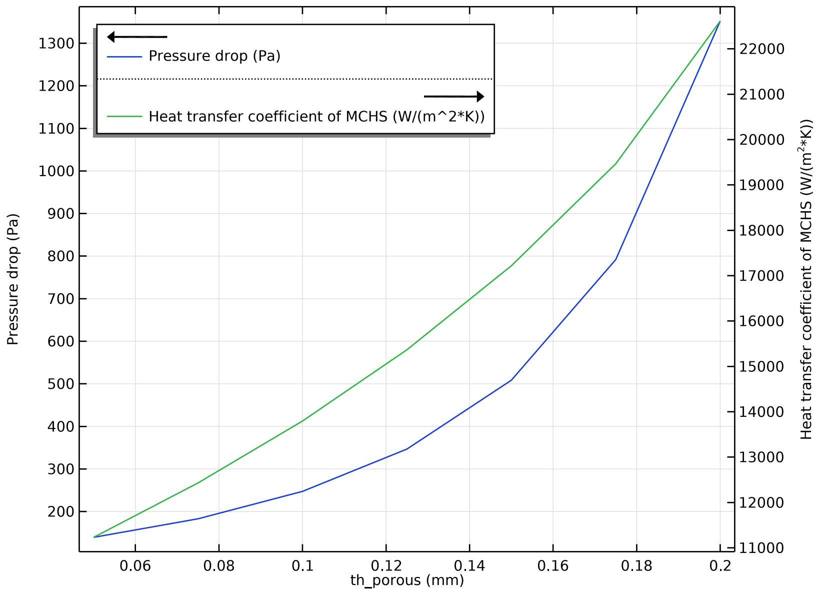


Figure 5: Pressure drop and average heat transfer coefficient.

Figure 6 shows how the dimensionless Reynolds and Nusselt numbers depend on this thickness. The Reynolds number decreases with increasing th_p and varies in the range from

100 to 210, meaning that the choice of the Brinkman-Forchheimer equation is justified. The Nusselt number has a maximum at th_p = 0.125 mm.

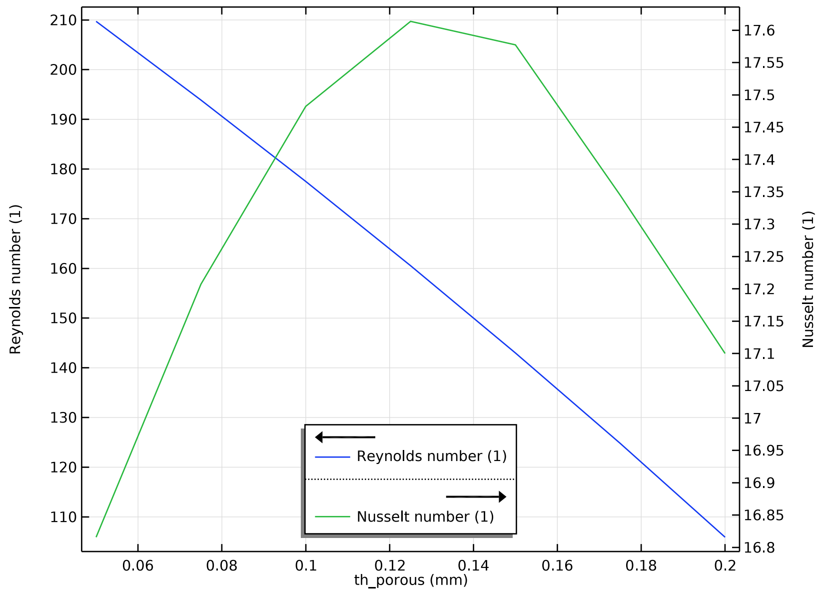


Figure 6: Reynolds and Nusselt numbers.

The Figure of Merit (Figure 7) shows that the optimal performance is achieved for th_p = 0.1 mm.

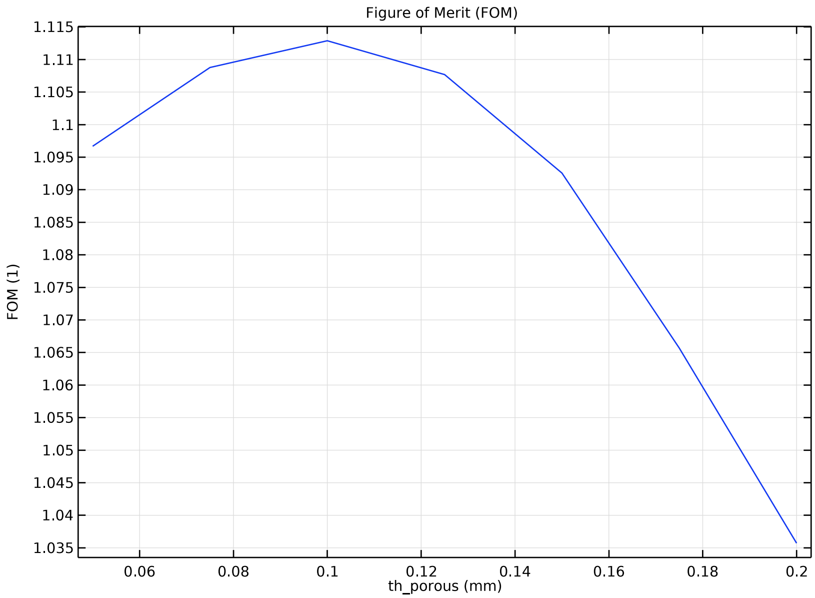


Figure 7: The Figure of Merit comparing the performances of the porous and the conventional MCHS.

Notes About the COMSOL Implementation

This model shows how to analyze the performance of the porous MCHS for varying porous substrate thickness. The model geometry and operating conditions are fully parametrized, such that you can easily extend the model for various parameters, as for example the inlet velocity or other channel dimensions.

Reference

1. A. Ghahremannezhad and K. Vafai, “Thermal and hydraulic performance enhancement of microchannel heat sinks utilizing porous substrates,” *Int. J. Heat Mass Transfer*, vol. 122, pp. 1313–1326, 2018.

Application Library path: `Porous_Media_Flow_Module/Heat_Transfer/porous_microchannel_heat_sink`

Modeling Instructions

ROOT

Start by loading the model file that contains the geometry and selections used throughout the modeling process.

1. From the **File** menu, choose **Open**.
2. Browse to the model's Application Libraries folder and double-click the file `porous_microchannel_heat_sink_geom_sequence.mph`.

GLOBAL DEFINITIONS

Parameters 1

The geometry parameters are already present after loading the file. Add a few more parameters for the material properties and operating conditions.

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
rho_f	998[kg/m^3]	998 kg/m³	Density, fluid
mu_f	8.55e-4[Pa*s]	8.55E-4 Pa·s	Viscosity, fluid
k_f	0.6[W/(m*K)]	0.6 W/(m·K)	Thermal conductivity, fluid
Cp_f	4182[J/(kg*K)]	4182 J/(kg·K)	Heat capacity, fluid
por	0.404	0.404	Porosity
d_p	20[um]	2E-5 m	Pore size
kappa	d_p^2/150*por^3/(1-por)^2	4.9502E-13 m²	Permeability
q_in	50[W/cm^2]	5E5 W/m²	Heat load

Name	Expression	Value	Description
T_in	300[K]	300 K	Inlet temperature
u_in	0.2[m/s]	0.2 m/s	Inlet velocity

ADD PHYSICS

1 In the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.

2 Go to the **Add Physics** window.

3 In the tree, select **Fluid Flow>Single-Phase Flow>Laminar Flow (spf)**.

4 Click **Add to Component 1** in the window toolbar.

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

6 Click **Add to Component 1** in the window toolbar.

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

ADD MULTIPHYSICS

1 In the **Home** toolbar, click **Add Multiphysics** to open the **Add Multiphysics** window.

2 Go to the **Add Multiphysics** window.

3 In the tree, select **Heat Transfer>Conjugate Heat Transfer>Laminar Flow**.

4 Click **Add to Component** in the window toolbar.

5 In the **Home** toolbar, click **Add Multiphysics** to close the **Add Multiphysics** window.

This adds the coupling between the **Laminar Flow** and **Heat Transfer in Solids and Fluids** interfaces.

Next, add the materials. For the fluid use a user-defined material with the parameters defined above. Load steel from the Material Library.

GLOBAL DEFINITIONS

Water

1 In the **Model Builder** window, under **Global Definitions** right-click **Materials** and choose **Blank Material**.

2 In the **Settings** window for **Material**, type `Water` in the **Label** text field.

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>Steel AISI 4340**.

4 Click **Add to Global Materials** in the window toolbar.

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

MATERIALS

Material Link 1 (matlnk1)

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **More Materials>Material Link**.

Material Link 2 (matlnk2)

1 Right-click **Materials** and choose **More Materials>Material Link**.

2 In the **Settings** window for **Material Link**, locate the **Link Settings** section.

3 From the **Material** list, choose **Steel AISI 4340 (mat2)**.

4 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Solid**.

Porous Material 1 (poromat1)

1 Right-click **Materials** and choose **More Materials>Porous Material**.

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

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

4 Locate the **Nonporous Material Setting** section. From the **Material** list, choose **Water (mat1)**.

Fluid 1 (poromat1.fluid1)

Right-click **Porous Material 1 (poromat1)** and choose **Fluid**.

Solid 1 (poromat1.solid1)

1 In the **Model Builder** window, right-click **Porous Material 1 (poromat1)** and choose **Solid**.

2 In the **Settings** window for **Solid**, locate the **Solid Properties** section.

3 From the **Material** list, choose **Steel AISI 4340 (mat2)**.

4 In the θ_s text field, type `1-por`.

Now, set up the domain conditions. This determines which material properties are required and you can fill in the missing materials afterward. For this step the selections are helpful.

LAMINAR FLOW (SPF)

Assume incompressible flow.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.

2 In the **Settings** window for **Laminar Flow**, locate the **Physical Model** section.

3 From the **Compressibility** list, choose **Incompressible flow**.

4 Select the **Enable porous media domains** check box.

5 Locate the **Domain Selection** section. From the **Selection** list, choose **Flow Domain**.

Fluid and Matrix Properties 1

1 In the **Physics** toolbar, click **Domains** and choose **Fluid and Matrix Properties**.

2 In the **Settings** window for **Fluid and Matrix Properties**, locate the **Domain Selection** section.

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

4 Locate the **Fluid Properties** section. From the **Fluid material** list, choose **Water (mat1)**.

5 Locate the **Porous Matrix Properties** section. From the **Porous material** list, choose **Steel AISI 4340 (mat2)**.

6 From the ε_p list, choose **User defined**. In the associated text field, type `por`.

7 From the **Permeability model** list, choose **Non-Darcian**.

8 From the κ list, choose **User defined**. In the associated text field, type `kappa`.

HEAT TRANSFER IN SOLIDS AND FLUIDS (HT)

Fluid 1

1 In the **Model Builder** window, under **Component 1 (comp1)> Heat Transfer in Solids and Fluids (ht)** click **Fluid 1**.

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

3 From the **Selection** list, choose **Flow Domain**.

Porous Medium 1

1 In the **Physics** toolbar, click **Domains** and choose **Porous Medium**.

2 In the **Settings** window for **Porous Medium**, locate the **Domain Selection** section.

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

Porous Matrix 1

1 In the **Model Builder** window, expand the **Porous Medium 1** node, then click **Porous Matrix 1**.

2 In the **Settings** window for **Porous Matrix**, locate the **Matrix Properties** section.

3 From the **Define** list, choose **Solid phase properties**.

You can now specify the values of the missing material properties.

GLOBAL DEFINITIONS

Water (mat1)

1 In the **Model Builder** window, under **Global Definitions>Materials** click **Water (mat1)**.

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
Density	rho	`rho_f`	kg/m^3	Basic
Dynamic viscosity	mu	`mu_f`	Pa·s	Basic
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	`k_f`	W/(m·K)	Basic
Heat capacity at constant pressure	Cp	`Cp_f`	J/(kg·K)	Basic
Ratio of specific heats	gamma	1	1	Basic

Complete the physics setup by adding the boundary conditions.

LAMINAR FLOW (SPF)

In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.

Inlet 1

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

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

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

4 Locate the **Velocity** section. In the U_0 text field, type `u_in`.

Outlet 1

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

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

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

Symmetry 1

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

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

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

HEAT TRANSFER IN SOLIDS AND FLUIDS (HT)

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

Boundary Heat Source 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Boundary Heat Source**.

2 Select Boundary 7 only.

3 In the **Settings** window for **Boundary Heat Source**, locate the **Boundary Heat Source** section.

4 In the Q_b text field, type `q_in`.

Inflow 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Inflow**.

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

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

4 Locate the **Upstream Properties** section. In the T_{ustr} text field, type `T_in`.

Outflow 1

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

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

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

Symmetry 1

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

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

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

To evaluate the performance, define new variables (see Equation 2 to Equation 5).

To evaluate the pressure drop define a nonlocal average coupling at the inlet of the porous MCHS. Use the average temperature of the centerline at the bottom surface to evaluate the heat transfer coefficient.

DEFINITIONS (COMP1)

Average: Inlet of Porous MCHS

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Average**.

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

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

4 Click the **Wireframe Rendering** button in the **Graphics** toolbar.

5 Select Boundaries 9 and 17 only.

6 In the **Label** text field, type `Average: Inlet of Porous MCHS`.

Average: Centerline, Bottom Surface

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Average**.

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

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

4 Select Edge 7 only.

5 In the **Label** text field, type `Average: Centerline, Bottom Surface`.

Variables 1

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

2 In the **Settings** window for **Variables**, locate the **Variables** section.

3 In the table, enter the following settings:

Name	Expression	Unit	Description
Dh	2*width_channel*height_channel/(width_channel+height_channel)	m	Hydraulic diameter
dp	aveop1(p)	Pa	Pressure drop
Tw	aveop2(T)	K	Average wall temperature
Omega	spf.out1.Mflow/rho_f*dp	W	Pumping power
h_mchs	q_in/(Tw-T_in)	W/(m²·K)	Heat transfer coefficient of MCHS
Nu	h_mchs*Dh/k_f		Nusselt number
Re	rho_f*u_in*Dh/mu_f		Reynolds number

MESH 1

In order to solve the model efficiently and achieve high accuracy, two aspects must be considered. The geometry has a high aspect ratio which can lead to an unnecessarily large number of mesh elements. The boundary heat source at the bottom surface leads to a high temperature gradient in z-direction close to this boundary.

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.

2 In the **Settings** window for **Mesh**, locate the **Physics-Controlled Mesh** section.

3 From the **Element size** list, choose **Finer**.

4 In the table, clear the **Use** check boxes for **Heat Transfer in Solids and Fluids (ht)** and **Nonisothermal Flow 1 (nitf1)**. to build a mesh for the flow domains only.

5 Locate the **Mesh Settings** section. From the **Sequence type** list, choose **User-controlled mesh**.

Free Tetrahedral 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Free Tetrahedral 1**.

2 In the **Settings** window for **Free Tetrahedral**, click to expand the **Scale Geometry** section.

3 In the **y-direction scale** text field, type `0.3`.

4 In the **z-direction scale** text field, type `0.5`.

5 Click **Build All**.

This mesh is optimized for the calculation of the flow equation. To take the high temperature gradient at the bottom of the porous MCHS into account build a mesh optimized for the heat transport.

MESH 2

In the **Mesh** toolbar, click **Add Mesh**.

Reference 1

1 In the **Mesh** toolbar, click **Modify** and choose **Mesh>Reference**.

2 In the **Settings** window for **Reference**, locate the **Reference** section.

3 From the **Mesh** list, choose **Mesh 1**.

This creates the same mesh. Additionally a mesh for the solid part is required.

Free Tetrahedral 1

In the **Mesh** toolbar, click **Free Tetrahedral**.

Finally, the mesh is refined in the lower area to resolve the large temperature gradient properly.

Refine 1

1 In the **Mesh** toolbar, click **Modify** and choose **Elements>Refine**.

2 In the **Settings** window for **Refine**, locate the **Refine Options** section.

3 From the **Refinement method** list, choose **Regular refinement**.

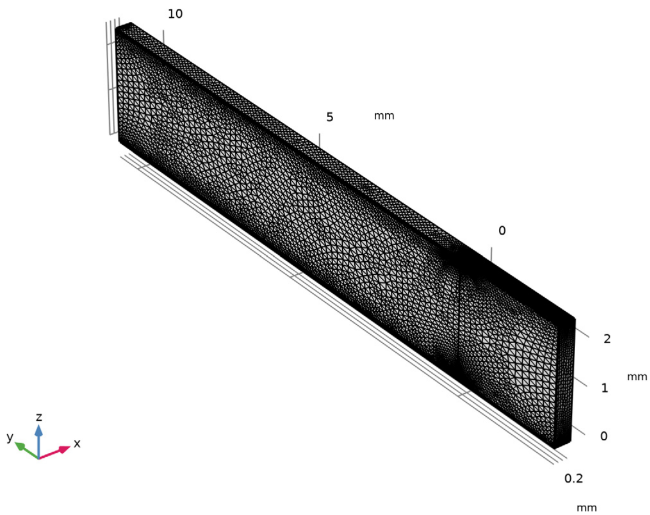
4 Click to expand the **Refine Elements in Box** section. Select the **Specify bounding box** check box.

5 In row **x**, set **Upper bound** to `width`.

6 In row **y**, set **Upper bound** to `length`.

7 In row **z**, set **Upper bound** to `3*th_solid`.

8 Click **Build All**.



ADD STUDY

Now add a study. Use a one-way stationary study where you first solve for the flow field using the first mesh and then use the solution obtained when solving for the heat transfer with the second mesh.

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Selected Multiphysics>Stationary, One-Way NITF**.

4 Click **Add Study** in the window toolbar.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 1

Step 1: Stationary

To compare the performance of the porous MCHS with that of a conventional one, ignore the porous domain in the first study. To do so, deactivate the relevant features.

1 In the **Settings** window for **Stationary**, click to expand the **Mesh Selection** section.

2 In the table, enter the following settings:

Geometry	Mesh
Geometry 1	Mesh 1

3 Locate the **Physics and Variables Selection** section. Select the **Modify model configuration for study step** check box.

4 In the **Physics and variables selection** tree, select **Component 1 (comp1)> Laminar Flow (spf)>Fluid and Matrix Properties 1**.

5 Click **Go to Source**.

LAMINAR FLOW (SPF)

Fluid and Matrix Properties 1

In the **Model Builder** window, under **Component 1 (comp1)>Laminar Flow (spf)** right-click **Fluid and Matrix Properties 1** and choose **Disable in All Studies**.

HEAT TRANSFER IN SOLIDS AND FLUIDS (HT)

Porous Medium 1

In the **Model Builder** window, under **Component 1 (comp1)> Heat Transfer in Solids and Fluids (ht)** right-click **Porous Medium 1** and choose **Disable in All Studies**.

STUDY 1: REFERENCE MCHS

1 In the **Model Builder** window, click **Study 1**.

2 In the **Settings** window for **Study**, type `Study 1: Reference MCHS` in the **Label** text field.

3 In the **Home** toolbar, click **Compute**.

RESULTS

Global Evaluation 1

Next, evaluate the performance parameters of the reference MCHS.

1 In the **Results** toolbar, click 8.5 **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.

3 In the table, enter the following settings:

Expression	Unit	Description
dp	Pa	Pressure drop
Omega	W	Pumping power
h_mchs	W/(m^2*K)	Heat transfer coefficient of MCHS
Nu	1	Nusselt number
Re	1	Reynolds number

4 Click **Evaluate**.

ADD STUDY

Add a second study and perform a parametric sweep over the thickness of the porous substrate. Of course you can run a parametric sweep over many parameters. For this demo model a single parameter is sufficient to demonstrate the principal approach.

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **Preset Studies for Selected Multiphysics>Stationary, One-Way NITF**.

4 Click **Add Study** in the window toolbar.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
th_porous (Porous structure thickness)	range(0.05,0.025,0.2)	mm

Step 1: Stationary

1 In the **Model Builder** window, click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Mesh Selection** section.

3 In the table, enter the following settings:

Geometry	Mesh
Geometry 1	Mesh 1

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

Create a cross-section plot of the velocity (Figure 3) as follows:

RESULTS

Cut Plane 1

1 In the **Results** toolbar, click **Cut Plane**.

2 In the **Settings** window for **Cut Plane**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 2/Parametric Solutions 1 (sol5)**.

4 Locate the **Plane Data** section. From the **Plane** list, choose **xy-planes**.

5 In the **z-coordinate** text field, type `height/2`.

Velocity, Cross Section

1 In the **Results** toolbar, click **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Velocity, Cross Section` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Cut Plane 1**.

4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

Surface 1

Right-click **Velocity, Cross Section** and choose **Surface**.

Streamline 1

1 In the **Model Builder** window, right-click **Velocity, Cross Section** and choose **Streamline**.

2 In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.

3 From the **Positioning** list, choose **Magnitude controlled**.

4 In the **Density** text field, type `10`.

Color Expression 1

1 Right-click **Streamline 1** and choose **Color Expression**.

2 In the **Settings** window for **Color Expression**, locate the **Coloring and Style** section.

3 From the **Color table** list, choose **GrayPrint**.

4 Locate the **Expression** section. In the **Expression** text field, type `p`.

5 In the **Velocity, Cross Section** toolbar, click **Plot**.

6 In the **Model Builder** window, expand the **Results>Views** node.

Axis

1 In the **Model Builder** window, expand the **Results>Views>View 2D 6** node, then click **Axis**.

2 In the **Settings** window for **Axis**, locate the **Axis** section.

3 From the **View scale** list, choose **Automatic**.

4 From the **Automatic** list, choose **Anisotropic**.

5 In the **y weight** text field, type 3.

6 Click **Update**.

7 Click the **Zoom Extents** button in the **Graphics** toolbar.

Velocity, Cross Section

1 In the **Model Builder** window, click **Velocity, Cross Section**.

2 In the **Velocity, Cross Section** toolbar, click **Plot**.

Global Evaluation 1

To analyze the performance of the porous MCHS, duplicate the **Global Evaluation 1** node and apply the new dataset.

Global Evaluation 2

1 In the **Model Builder** window, under **Results>Derived Values** right-click **Global Evaluation 1** and choose **Duplicate**.

2 In the **Settings** window for **Global Evaluation**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 2/Parametric Solutions 1 (sol5)**.

4 Click next to **Evaluate**, then choose **New Table**.

TABLE

1 Go to the **Table** window.

2 Click **Table Graph** in the window toolbar.

RESULTS

Table Graph 1

1 In the **Model Builder** window, under **Results>1D Plot Group 10** click **Table Graph 1**.

2 In the **Settings** window for **Table Graph**, locate the **Data** section.

3 From the **Plot columns** list, choose **Manual**.

4 In the **Columns** list, select **Pressure drop (Pa)**.

5 Click to expand the **Legends** section. Select the **Show legends** check box.

Table Graph 2

1 Right-click **Results>1D Plot Group 10>Table Graph 1** and choose **Duplicate**.

2 In the **Settings** window for **Table Graph**, locate the **Data** section.

3 In the **Columns** list, select **Heat transfer coefficient of MCHS (W/(m^2*K))**.

4 In the **1D Plot Group 10** toolbar, click **Plot**.

Heat-Transfer Coefficient and Pressure Drop

1 In the **Model Builder** window, under **Results** click **1D Plot Group 10**.

2 In the **Settings** window for **1D Plot Group**, type `Heat-Transfer Coefficient and Pressure Drop` in the **Label** text field.

3 Locate the **Plot Settings** section. Select the **Two y-axes** check box.

4 In the table, select the **Plot on secondary y-axis** check box for **Table Graph 2**.

5 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

6 In the **Heat-Transfer Coefficient and Pressure Drop** toolbar, click **Plot**, and compare with Figure 5.

Reynolds and Nusselt Numbers

Plot the dimensionless Reynolds and Nusselt numbers in the same way.

1 Right-click **Heat-Transfer Coefficient and Pressure Drop** and choose **Duplicate**.

2 In the **Settings** window for **1D Plot Group**, type `Reynolds and Nusselt Numbers` in the **Label** text field.

Table Graph 1

1 In the **Model Builder** window, expand the **Reynolds and Nusselt Numbers** node, then click **Table Graph 1**.

2 In the **Settings** window for **Table Graph**, locate the **Data** section.

3 In the **Columns** list, select **Reynolds number (1)**.

Table Graph 2

1 In the **Model Builder** window, click **Table Graph 2**.

2 In the **Settings** window for **Table Graph**, locate the **Data** section.

3 In the **Columns** list, select **Nusselt number (1)**.

4 In the **Reynolds and Nusselt Numbers** toolbar, click **Plot**, and compare with Figure 6.

Reynolds and Nusselt Numbers

1 In the **Model Builder** window, click **Reynolds and Nusselt Numbers**.

2 In the **Settings** window for **1D Plot Group**, locate the **Legend** section.

3 From the **Position** list, choose **Lower middle**.

Global Evaluation 3

To compare the different designs in terms of overall performance, the figure of merit can be calculated according to Equation 5 as follows:

1 In the **Results** toolbar, click 8.5 **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 2/Parametric Solutions 1 (sol5)**.

4 Locate the **Expressions** section. In the table, enter the following settings:

Expression	Unit	Description
`h_mchs/withsol('sol1',h_mchs)/(Omega/withsol('sol1', Omega))^(1/3)`	1	FOM

With the `withsol` operator, you can use results from other solutions than that of the chosen dataset.

5 Click next to **Evaluate**, then choose **New Table**.

TABLE

1 Go to the **Table** window.

2 Click **Table Graph** in the window toolbar.

RESULTS

FOM

1 In the **Model Builder** window, under **Results** click **1D Plot Group 12**.

2 In the **Settings** window for **1D Plot Group**, click to expand the **Title** section.

3 From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Figure of Merit (FOM)`.

5 In the **1D Plot Group 12** toolbar, click **Plot**, and compare with fig Figure 7.

6 In the **Label** text field, type `FOM`.

With a porous substrate thickness of 0.1 mm the performance has increased by approximately 12%.

Study 2/Parametric Solutions 1 (sol5)

To reproduce Figure 4, proceed as follows.

In the **Model Builder** window, click **Study 2/Parametric Solutions 1 (sol5)**.

Selection

1 In the **Results** toolbar, click **Attributes** and choose **Selection**.
2 In the **Settings** window for **Selection**, locate the **Geometric Entity Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 Select Domains 3 and 4 only.

Mirror 3D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Mirror 3D**.
2 In the **Settings** window for **Mirror 3D**, locate the **Data** section.
3 From the **Dataset** list, choose **Study 2/Parametric Solutions 1 (sol5)**.

Surface 3

1 In the **Results** toolbar, click **More Datasets** and choose **Surface**.
2 In the **Settings** window for **Surface**, locate the **Data** section.
3 From the **Dataset** list, choose **Study 2/Parametric Solutions 1 (sol5)**.
4 Select Boundaries 10, 18, and 21 only.

Mirror 3D 2

1 In the **Results** toolbar, click **More Datasets** and choose **Mirror 3D**.
2 In the **Settings** window for **Mirror 3D**, locate the **Data** section.
3 From the **Dataset** list, choose **Surface 3**.

Cut Plane 2

1 In the **Results** toolbar, click **Cut Plane**.
2 In the **Settings** window for **Cut Plane**, locate the **Data** section.
3 From the **Dataset** list, choose **Mirror 3D 1**.
4 Locate the **Plane Data** section. From the **Plane** list, choose **xy-planes**.
5 In the **z-coordinate** text field, type `1`.

Velocity and Temperature Fields

1 In the **Results** toolbar, click **3D Plot Group**.
2 In the **Settings** window for **3D Plot Group**, type `Velocity and Temperature Fields` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2/ Parametric Solutions 1 (sol5)**.

4 From the **Parameter value (th_porous (mm))** list, choose **0.1**.

Surface 1

1 Right-click **Velocity and Temperature Fields** and choose **Surface**.

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

3 From the **Dataset** list, choose **Mirror 3D 2**.

4 From the **Solution parameters** list, choose **From parent**.

5 Locate the **Expression** section. In the **Expression** text field, type `T`.

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

Velocity and Temperature Fields

In the **Model Builder** window, click **Velocity and Temperature Fields**.

Streamline Surface 1

1 In the **Velocity and Temperature Fields** toolbar, click **More Plots** and choose **Streamline Surface**.

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

3 From the **Dataset** list, choose **Cut Plane 2**.

4 From the **Solution parameters** list, choose **From parent**.

5 Locate the **Streamline Positioning** section. From the **Positioning** list, choose **Magnitude controlled**.

6 In the **Density** text field, type `8`.

7 Locate the **Coloring and Style** section. Find the **Point style** subsection. From the **Type** list, choose **Arrow**.

8 From the **Arrow type** list, choose **Cone**.

Color Expression 1

1 Right-click **Streamline Surface 1** and choose **Color Expression**.

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

3 In the **Expression** text field, type `p`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **GrayScale**.

Velocity and Temperature Fields

1 In the **Model Builder** window, click **Velocity and Temperature Fields**.

2 In the **Settings** window for **3D Plot Group**, locate the **Plot Settings** section.

3 Clear the **Plot dataset edges** check box.

Add a view to get a better impression.

4 Click the **Show More Options** button in the **Model Builder** toolbar.

5 In the **Show More Options** dialog box, in the tree, select the check box for the node **Results>Views**.

6 Click **OK**.

View 3D 7

In the **Model Builder** window, right-click **Views** and choose **View 3D**.

Camera

1 In the **Model Builder** window, expand the **View 3D 7** node, then click **Camera**.

2 In the **Settings** window for **Camera**, locate the **Camera** section.

3 From the **View scale** list, choose **Manual**.

4 In the **x scale** text field, type 5.

5 In the **z scale** text field, type 2.

Rotate the geometry to get a similar image as Figure 4.