Created in COMSOL Multiphysics 6.2

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.

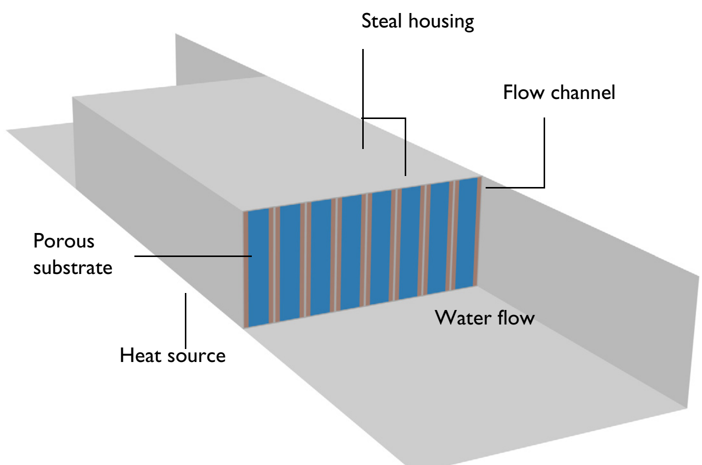


Figure 1: Geometry and operating conditions of the porous MCHS.

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.

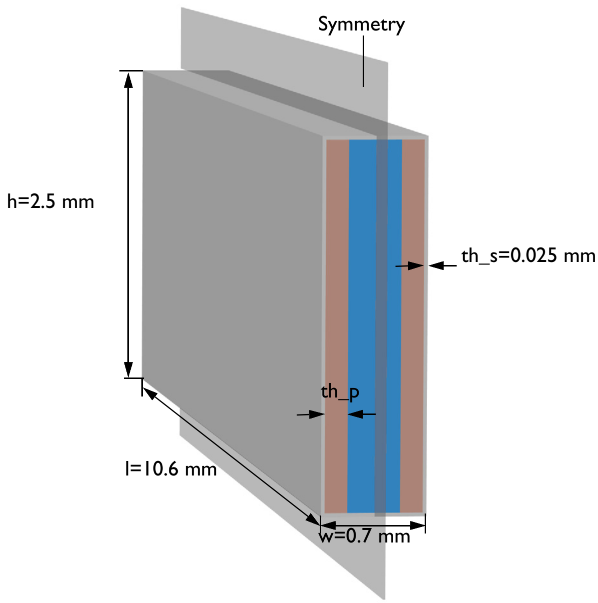


Figure 2: Geometry of the modeling domain. The free flow domain used to provide the inflow profile for the MCHS is not shown.

The flow channels contain sintered porous metal blocks with a porosity of $\varepsilon = 0.404$ on each side. A heat source with $q_{in} = 100\ \mathrm{W/cm^2}$ is attached to the bottom. Water with an inlet velocity of $u = 0.2\ \mathrm{m/s}$ and a temperature of $T_{in} = 300\ \mathrm{K}$ 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 $\mathbf{u}$ 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:

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

The index `ref` 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 $1 \leq \mathrm{Re} \leq 1000$. An estimation of the Reynolds number (Equation 3) results in $\mathrm{Re} \sim 300$ 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.

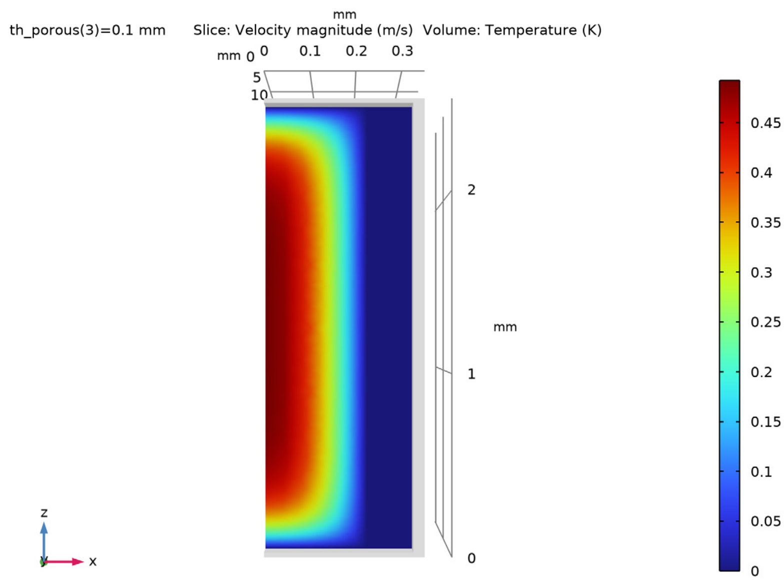


Figure 3: Cross section of the velocity field. One can easily recognize the area of the porous medium on the right side by the blue color which implies low velocities.

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

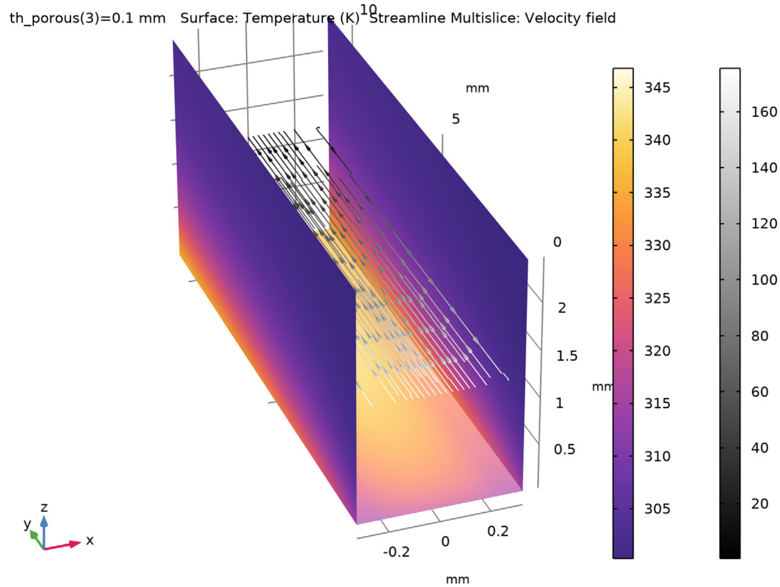


Figure 4: Temperature distribution (color) and velocity field (arrows) with the gray scale indicating the pressure.

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.

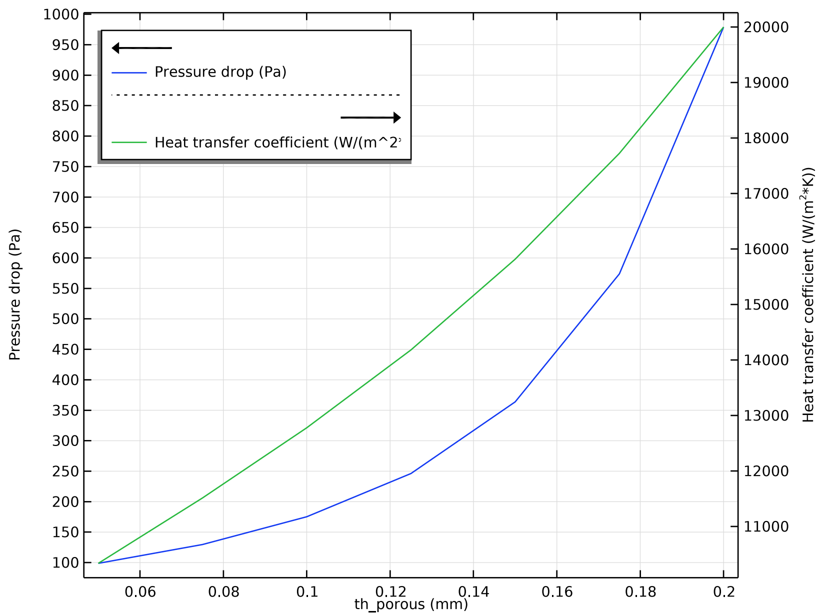


Figure 5: Pressure drop and average heat transfer coefficient.

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.

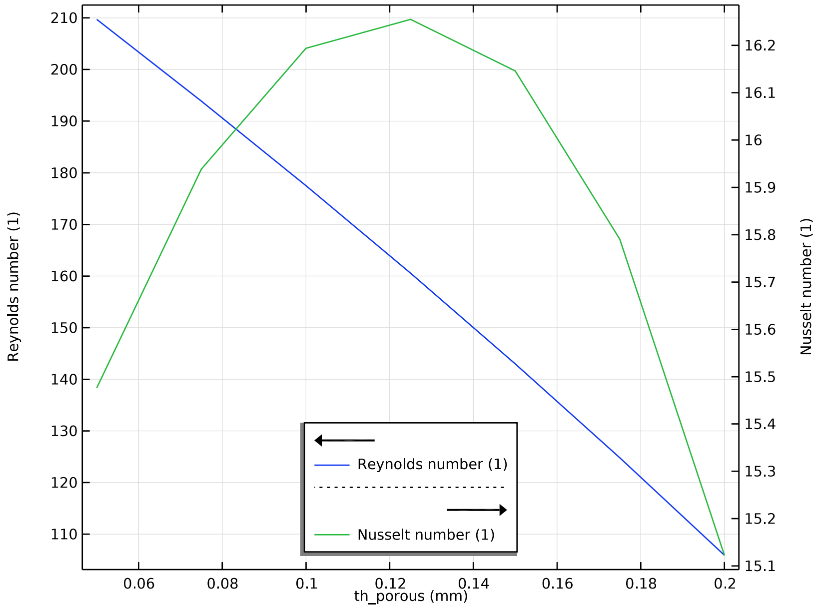


Figure 6: Reynolds and Nusselt numbers.

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

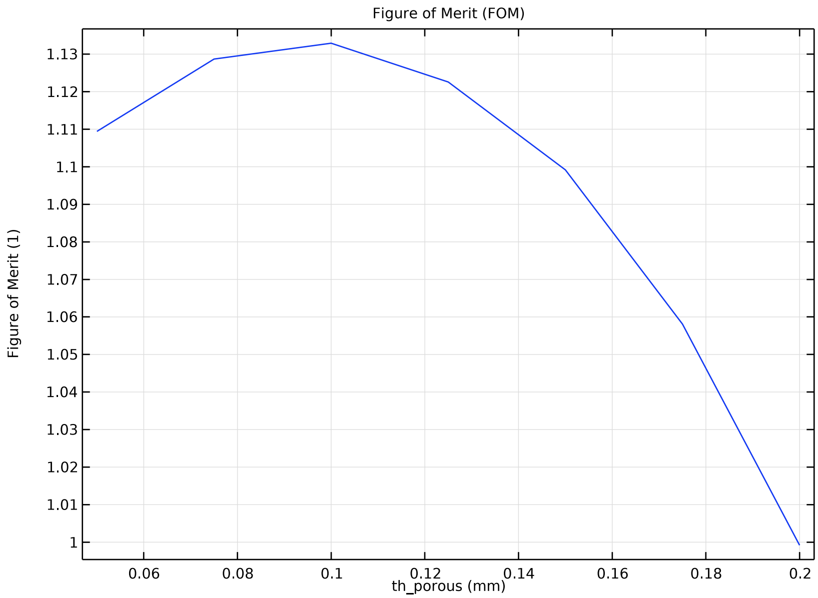


Figure 7: The Figure of Merit comparing the performances of the porous and the conventional MCHS.

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 parameterized, 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 Transf.*, vol. 122, pp. 1313–1326, 2018.

Application Library path: `Porous_Media_Flow_Module/Heat_Transfer/porous_microchannel_heat_sink`

Modeling Instructions

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

NEW

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

MODEL WIZARD

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

2 In the **Select Physics** tree, select **Heat Transfer>Conjugate Heat Transfer>Laminar Flow**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GEOMETRY 1

1 In the **Geometry** toolbar, click **Insert Sequence** and choose **Insert Sequence**.

2 Browse to the model's Application Libraries folder and double-click the file `porous_microchannel_heat_sink_geom_sequence.mph`.

3 In the **Geometry** toolbar, click **Build All**.

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

GLOBAL DEFINITIONS

Geometry Parameters

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, type `Geometry Parameters` in the **Label** text field.

Material Properties and Operating Conditions

1 In the **Home** toolbar, click **Parameters** and choose **Add>Parameters**.

2 In the **Settings** window for **Parameters**, type `Material Properties and Operating Conditions` in the **Label** text field.

3 Locate the **Parameters** section. 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
T_in	300[K]	300 K	Inlet temperature
u_in	0.2[m/s]	0.2 m/s	Inlet velocity

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

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 (pmat1)

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**.

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.

HEAT TRANSFER IN SOLIDS AND FLUIDS (HT)

Porous Medium 1

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

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**.

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**.

4 Locate the **Porous Medium** section. From the **Flow model** list, choose **Non-Darcian flow**.

This enables the Forchheimer pressure drop.

Porous Matrix 1

1 In the **Model Builder** window, click **Porous Matrix 1**.

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

3 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**.

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

MATERIALS

Porous Material 1 (pmat1)

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Materials>Porous Material 1 (pmat1)** node, then click **Porous Material 1 (pmat1)**.

2 In the **Settings** window for **Porous Material**, locate the **Phase-Specific Properties** section.

3 Click **Add Required Phase Nodes**.

Solid 1 (pmat1.solid1)

1 In the **Model Builder** window, click **Solid 1 (pmat1.solid1)**.

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`.

Porous Material 1 (pmat1)

1 In the **Model Builder** window, click **Porous Material 1 (pmat1)**.

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

3 From the **Material** list, choose **Water (mat1)**.

This **Homogenized Material** is used for all physics features that are not related to a porous medium feature in any of the physics interfaces.

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
Heat capacity at constant pressure	Cp	`Cp_f`	J/(kg·K)	Basic
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	`k_f`	W/(m·K)	Basic
Density	rho	`rho_f`	kg/m³	Basic
Dynamic viscosity	mu	`mu_f`	Pa·s	Basic

Complete the physics setup by adding the boundary conditions.

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 3 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**.

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 **Boundary Condition** section. From the list, choose **Fully developed flow**.

5 Locate the **Fully Developed Flow** section. In the U_{av} 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**.

DEFINITIONS (COMP1)

The performance and efficiency of a heat sink can be described by Equation 2 to Equation 5. Load the variables from a text file. Before, 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.

Average: Inlet

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

5 In the **Label** text field, type `Average: Inlet`.

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 2 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 Click **Load from File**.

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

MESH 1

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 Right-click **Component 1 (comp1)>Mesh 1** and choose **Edit Physics-Induced Sequence**.

Corner Refinement 1

In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** right-click **Corner Refinement 1** and choose **Build Selected**.

Free Triangular 1

1 In the **Mesh** toolbar, click **More Generators** and choose **Free Triangular**.

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

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

4 Click **Build Selected**.

5 Click the **Go to XZ View** button in the **Graphics** toolbar.

Free Tetrahedral 1

In the **Model Builder** window, right-click **Free Tetrahedral 1** and choose **Delete**.

Boundary Layers 1

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

2 In the **Settings** window for **Boundary Layers**, locate the **Geometric Entity Selection** section.

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

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

Boundary Layer Properties 1

1 In the **Model Builder** window, expand the **Boundary Layers 1** node, then click **Boundary Layer Properties 1**.

2 Select Edges 6, 9, 21, 23, and 27 only.

3 In the **Settings** window for **Boundary Layer Properties**, click **Build Selected**.

Swept 1

1 In the **Mesh** toolbar, click **Swept**.

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

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

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

5 Click **Build Selected**.

Swept 2

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

Distribution 1

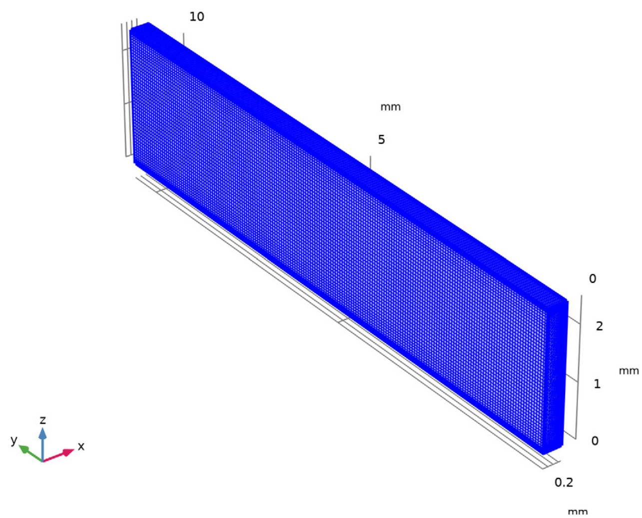
1 Right-click **Swept 2** and choose **Distribution**.

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

3 In the **Number of elements** text field, type 2.

4 Click **Build All** and compare with the image below.

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



LAMINAR FLOW (SPF)

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.

Porous Medium 1

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

HEAT TRANSFER IN SOLIDS AND FLUIDS (HT)

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**.

DEFINITIONS (COMP1)

Next, calculate the MCHS design with a porous substrate and conduct a parametric sweep on the substrate thickness in a second study. Although a parametric sweep can be done on multiple parameters, a single parameter will suffice to demonstrate the fundamental approach for this demo model.

To compare the different designs in terms of overall performance, the figure of merit can be calculated according to Equation 5. To obtain the reference values for the heat sink without a porous material, the result from the first study is used by employing the `withsol` operator. Complete the variable list.

Variables 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Definitions** click **Variables 1**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
`h_ref`	`withsol('sol1', h_mchs)`	W/(m²·K)	`Reference heat transfer coefficient`
`Omega_ref`	`withsol('sol1', Omega)`	W	`Reference pumping power`
`FOM`	`h_mchs/h_ref/(Omega/Omega_ref)^(1/3)`		`Figure of Merit`

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

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

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

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`

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

RESULTS

Arrange the plots into groups to facilitate their association with the different studies.

Pressure (spf), Temperature (ht), Temperature and Fluid Flow (nitf1), Velocity (spf)

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Temperature (ht)**, **Velocity (spf)**, **Pressure (spf)**, and **Temperature and Fluid Flow (nitf1)**.

2 Right-click and choose **Group**.

Reference MCHS

In the **Settings** window for **Group**, type `Reference MCHS` in the **Label** text field.

Pressure (spf) 1, Temperature (ht) 1, Temperature and Fluid Flow (nitf1) 1, Velocity (spf) 1

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Temperature (ht) 1**, **Velocity (spf) 1**, **Pressure (spf) 1**, and **Temperature and Fluid Flow (nitf1) 1**.

2 Right-click and choose **Group**.

Parametric MCHS

In the **Settings** window for **Group**, type `Parametric MCHS` in the **Label** text field.

Global Evaluation 1

Evaluate the performance of the porous microchannel heat sink.

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

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

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

4 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Definitions>Variables>dp - Pressure drop - Pa**.

5 Repeat this step to add the other variables as well. Alternatively, you can enter their names directly into the list of expressions. These are `Omega`, `h_mchs`, `Nu`, and `Re`.

6 Click **Evaluate**.

TABLE 1

1 Go to the **Table 1** window.

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

RESULTS

Table Graph 1

1 In the **Model Builder** window, under **Results>1D Plot Group 9** 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.

6 Right-click **Results>1D Plot Group 9>Table Graph 1** and choose **Duplicate**.

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 **Heat transfer coefficient (W/(m^2*K))**.

4 In the **1D Plot Group 9** toolbar, click **Plot**.

Heat-Transfer Coefficient and Pressure Drop

1 In the **Model Builder** window, under **Results** click **1D Plot Group 9**.

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.

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

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

Reynolds and Nusselt Numbers

1 In the **Model Builder** window, under **Results** click **Heat-Transfer Coefficient and Pressure Drop 1**.

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 2

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/Parametric Solutions 1 (sol5)**.

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

Expression	Unit	Description
FOM	1	Figure of Merit

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

TABLE 2

1 Go to the **Table 2** window.

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

Results

FOM

1 In the **Model Builder** window, under **Results** click **1D Plot Group 11**.

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 11** 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 13%.

Modify the velocity plot as follows:

Slice

1 In the **Model Builder** window, expand the **Results>Parametric MCHS>Velocity (spf) 1** node, then click **Slice**.

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

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

Volume 1

In the **Model Builder** window, right-click **Velocity (spf) 1** and choose **Volume**.

Selection 1

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

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

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

Material Appearance 1

1 In the **Model Builder** window, right-click **Volume 1** and choose **Material Appearance**.

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

3 From the **Appearance** list, choose **Custom**.

4 From the **Material type** list, choose **Steel**.

To better display the results, create a new view.

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

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

7 Click **OK**.

View 3D 5

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

Camera

1 In the **Model Builder** window, expand the **View 3D 5** 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.

Velocity (spf) 1

1 In the **Model Builder** window, under **Results>Parametric MCHS** click **Velocity (spf) 1**.

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

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

4 Locate the **Plot Settings** section. From the **View** list, choose **View 3D 5**.

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

6 In the **Velocity (spf) 1** toolbar, click **Plot**.

7 Click the **Go to XZ View** button in the **Graphics** toolbar and compare with Figure 3.

The velocity profile clearly shows which part is free flow and which part is porous media flow.

8 Click the **Go to Default View** button in the **Graphics** toolbar to return to the previous view.

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/Parametric Solutions 1 (sol5)**.

3D Plot Group 12

1 In the **Results** toolbar, click **3D Plot Group**.

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

3 From the **Dataset** list, choose **Mirror 3D 1**.

Surface 1

1 Right-click **3D Plot Group 12** and choose **Surface**.

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

3 Click **Change Color Table**.

4 In the **Color Table** dialog box, select **Thermal>HeatCameraLight** in the tree.

5 Click **OK**.

Selection 1

1 Right-click **Surface 1** and choose **Selection**.

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

3 From the **Selection** list, choose **Wall Boundaries**.

4 Select Boundaries 6, 14, and 17 only.

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

3D Plot Group 12

In the **Model Builder** window, under **Results>Parametric MCHS** click **3D Plot Group 12**.

Streamline Multislice 1

1 In the **3D Plot Group 12** toolbar, click **More Plots** and choose **Streamline Multislice**.

2 In the **Settings** window for **Streamline Multislice**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)> Laminar Flow>Velocity and pressure>u,v,w - Velocity field**.

3 Locate the **Multiplane Data** section. Find the **x-planes** subsection. In the **Planes** text field, type `0`.

4 Find the **y-planes** subsection. In the **Planes** text field, type `0`.

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

6 In the **Minimum distance** text field, type `0.002`.

7 In the **Maximum distance** text field, type `0.01`.

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

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

Color Expression 1

1 Right-click **Streamline Multislice 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. Click **Change Color Table**.

5 In the **Color Table** dialog box, select **Linear>GrayScale** in the tree.

6 Click **OK**.

Velocity and Temperature Fields

1 In the **Model Builder** window, under **Results>Parametric MCHS** click **3D Plot Group 12**.

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

4 Locate the **Plot Settings** section. From the **View** list, choose **View 3D 5**.

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

Rotate the geometry to get a similar result as in Figure 4.

6 In the **Velocity and Temperature Fields** toolbar, click **Plot**.