Model created in COMSOL Multiphysics 6.4

Minimizing the Flow Velocity in a Microchannel

Introduction

Topology optimization of the Navier–Stokes equations is encountered in different branches and applications, such as in the design of ventilation systems for cars and optimal reactors. A common technique applicable to such problems is to let the distribution of porous material vary continuously. In this model, the objective is to find the optimal distribution of a porous material in a microchannel such that the horizontal velocity component at the center of the channel is minimized.

The model is inspired by Ref. 1.

Model Definition

The model geometry (Figure 1) consists of three regions: the inlet channel, the design domain, and the outlet channel. A prescribed pressure drop between the inlet and the outlet drives the flow.

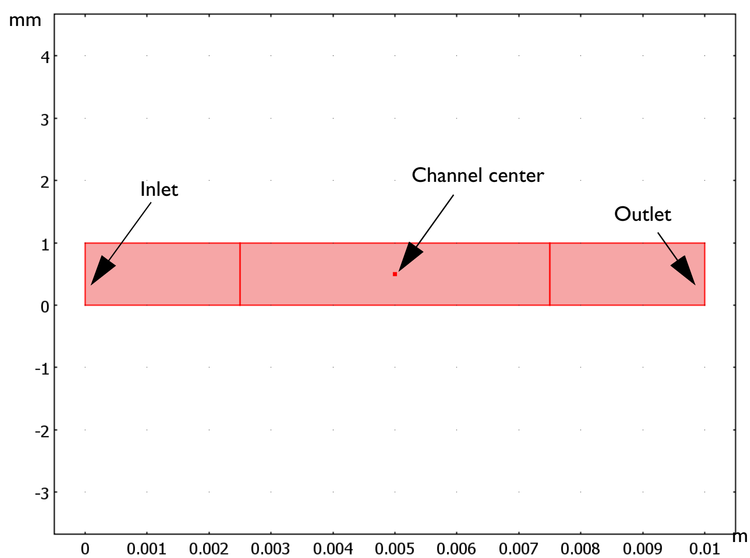


Figure 1: The model geometry.

The fluid flow in the channel is described by the Navier–Stokes equations

$$\rho(\mathbf{u}\cdot\nabla)\mathbf{u} = -\nabla p + \nabla\cdot\eta(\nabla\mathbf{u} + (\nabla\mathbf{u})^T) - \alpha(\gamma)\mathbf{u}$$
$$\nabla\cdot\mathbf{u} = 0$$

where $-\alpha(\gamma)\mathbf{u}$ is a force term in which the coefficient

$$\alpha(\gamma) \equiv \alpha_{max}\frac{q(1-\gamma)}{q+\gamma} \qquad (1)$$

characterizes the flow in a porous medium. In Equation 1, α is interpreted as a continuous mapping determined by the function γ: $\Omega \rightarrow [0, 1]$, which in the limit of decreasing Darcy number and decreasing mesh size should be a discrete-valued function. When γ equals 1, α is zero, corresponding to free flow. Conversely, for $\gamma = 0$, $\alpha = \alpha_{max}$, where α_{max} is related to the dimensionless Darcy number, Da, according to

$$\mathrm{Da} = \frac{\eta}{\alpha_{max}L^2}$$

The convergence of the optimization process depends on three important factors: the Darcy number, the mesh size, and the coefficient q. Rewriting Equation 1 it is easily seen that $\alpha/\alpha_{max} \rightarrow 1-\gamma$ in the limit $q \rightarrow \infty$. In this limit, γ can be interpreted as the local porosity, ranging between 0 (filled) and 1 (open channel).

Results and Discussion

Figure 2 displays the design variable, γ, which represents the distribution of porous material. As the plot shows, γ is either 0 or 1 in most of the domain, with a narrow transition zone in between. The width of this transition zone is mesh dependent; you can reduce it by decreasing the mesh-element size. Alternatively, decreasing the Darcy number also gives harder boundaries at the interface between porous and open domains.

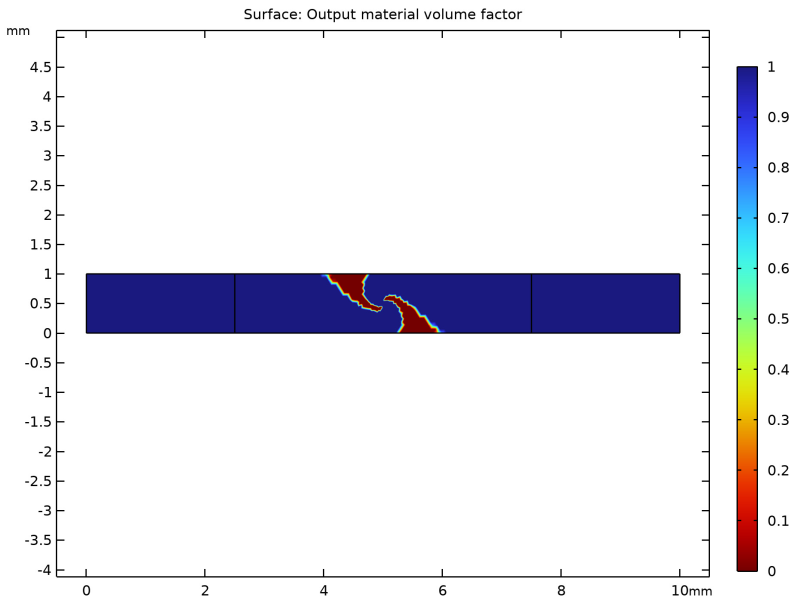


Figure 2: Distribution of porous material; the red areas represent open channel.

A question that naturally arises in this type of problems concerns uniqueness of the solution. In this case, there is at least one more solution that gives exactly the same result; because the channel has no upside or downside, a solution mirrored around the axis $y = 0.5$ mm would give exactly the same flow.

Figure 3 contains a surface plot of the horizontal velocity component and a streamline plot of the velocity field resulting from the optimization process. In addition, the contour $\gamma = 0.5$ indicates the border between the open channel and filling material. The plots reveal how the flow turns around, with a negative horizontal velocity at the center of the channel. Note also that the x-velocity has a minimum of roughly -14 mm/s at the design point.

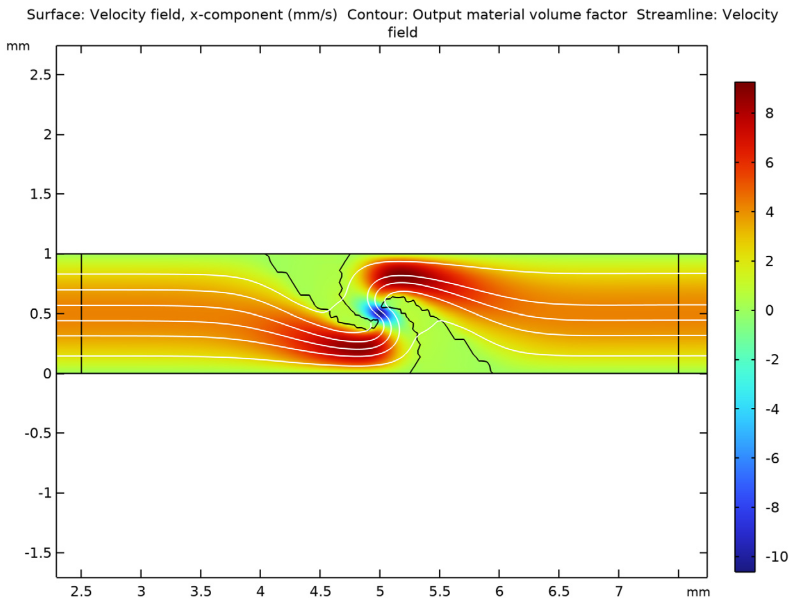


Figure 3: The horizontal velocity (surface plot) and velocity field (streamlines) after optimization. In addition, the contour $\gamma = 0.5$ *indicates the border between open channel and filling material.*

If you were to increase the streamline density in the above plot, some streamlines passing through the barriers would appear. This effect is due to a small amount of leakage, which can be reduced further by increasing the mesh resolution.

Figure 4 shows the pressure distribution, verifying the prescribed pressure drop through the channel length. The pressure drop is naturally concentrated to the region with porous material.

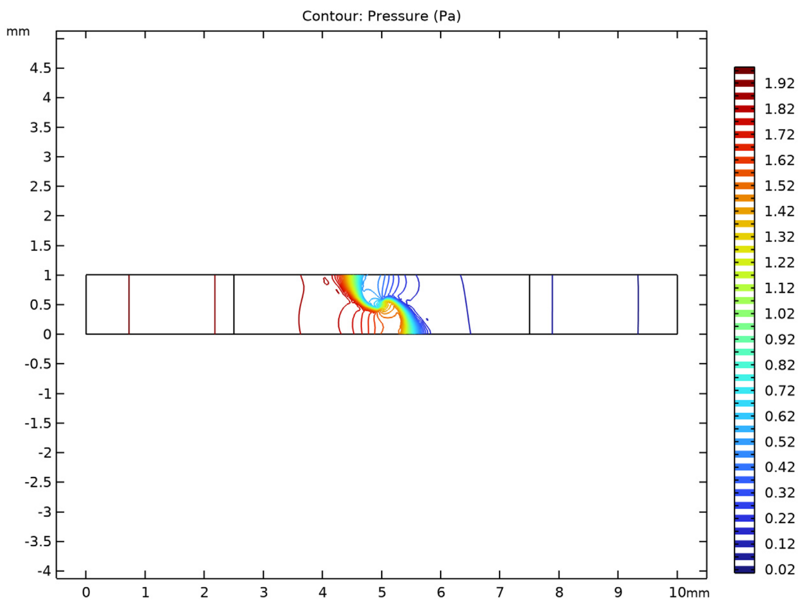


Figure 4: Pressure distribution in the channel, the pressure drop is concentrated to the porous domain.

Reference

1. L. Højgaard Olesen, F. Okkels, and H. Bruus, "A High-level Programming-language Implementation of Topology Optimization Applied to Steady-state Navier–Stokes Flow," *Int. J. Numer. Methods Eng.*, vol. 65, pp. 975–1001, 2005.

Notes About the COMSOL Implementation

This model combines a **Density Model** feature with a Laminar Flow interface. First, you calculate the solution for the flow in an empty channel (that is, with no porous material). You then solve the optimization problem. In each iteration, the software calculates a solution for the flow problem and feeds it to the optimization routine, which updates the design variables.

If the specified convergence criterion is fulfilled, the solution process terminates; otherwise the new design-variable values are used in the next calculation of the flow problem. However, for design problems such as this one, there is a tradeoff between computation time and convergence. The solution may be sufficiently improved long before convergence is reached in the mathematical sense. Therefore, it is useful to limit the number of steps taken by the optimization algorithm after which the solution can be evaluated and restarted if not yet satisfactory.

Setting up this kind of model with a general optimization routine requires quite a bit of work, but as you will discover, solving this problem with the built-in tools for optimization in COMSOL Multiphysics is easy.

Application Library path: `Optimization_Module/Topology_Optimization/reversed_flow`

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 **2D**.

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

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

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

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
q	1	1	Optimization parameter
meshsz	20[um]	2E-5 m	Center point mesh size
L	1[mm]	0.001 m	Inlet height

The purpose of the constant u0 is to introduce a rough velocity scale that you can use to make the objective function dimensionless with a value of the order 1.

GEOMETRY 1

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

2 In the **Settings** window for **Geometry**, locate the **Units** section.

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

Rectangle 1 (r1)

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type 2.5.

Rectangle 2 (r2)

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type 5.

4 Locate the **Position** section. In the **x** text field, type 2.5.

Rectangle 3 (r3)

1 In the **Geometry** toolbar, click **Rectangle**.

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type 2.5.

4 Locate the **Position** section. In the **x** text field, type 7.5.

Point 1 (pt1)

1 In the **Geometry** toolbar, click **Point**.

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

3 In the **x** text field, type 5.

4 In the **y** text field, type 0.5.

5 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.

6 Click **Build All Objects**.

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

The geometry should now look like that in Figure 1.

ADD MATERIAL

1 In the **Materials** toolbar, click **Add Material** to open the **Add Material** window.

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

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

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

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

MATERIALS

Water, liquid (mat1)

Next use the **Density Model** feature to set up the design variable.

COMPONENT 1 (COMP1)

Density Model 1 (dtopo1)

1 In the **Physics** toolbar, click **Optimization** and choose **Topology Optimization**.

Only the center part of the channel geometry is needed in the optimization, so you only have to define the control variable there.

2 Select Domain 2 only.

3 In the **Settings** window for **Density Model**, locate the **Filtering** section.

4 From the **Filter type** list, choose **None**.

5 Locate the **Interpolation** section. From the **Interpolation type** list, choose **Darcy**.

6 In the q text field, type `q`.

7 Locate the **Control Variable Initial Value** section. In the θ_0 text field, type `1`.

This defines the shape functions for the design variable used in the optimization. The initial value 1 corresponds to a channel free from porous material.

DEFINITIONS

Variables 1

1 In the **Definitions** toolbar, click **Local Variables**.

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

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

4 Select Domain 2 only.

The maximum damping is limited by the mesh size, because the mesh has to resolve the transition between marginal flow in the porous material and free flow in fluid region.

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

Name	Expression	Unit	Description
`alpha_max`	`4*spf.mu/meshsz^2`	Pa·s/m^2	`Volume force coefficient, max value`
`alpha`	`alpha_max* dtopo1.theta_p`	Pa·s/m^2	`Volume force coefficient`

Point Probe 1 (point1)

1 In the **Definitions** toolbar, click **Probes** and choose **Point Probe**.

2 In the **Settings** window for **Point Probe**, type `obj` in the **Variable name** text field.

3 Locate the **Source Selection** section. From the **Selection** list, choose **Point 1**.

4 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Laminar Flow** > **Velocity and pressure** > **Velocity field - m/s** > **u - Velocity field, x-component**.

This defines the objective function to be proportional to the x-component of the velocity at the center.

LAMINAR FLOW (SPF)

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

2 In the **Settings** window for **Laminar Flow**, click to expand the **Discretization** section.

3 From the **Discretization of fluids** list, choose **P2+P1**.

This setting gives quadratic elements for the velocity field.

Volume Force 1

1 In the **Physics** toolbar, click **Domains** and choose **Volume Force**.

In the next steps, you specify the pressure drop along the channel length by prescribing the pressure at the inlet and at the outlet. The resulting pressure gradient drives the flow in the channel.

2 Select Domain 2 only.

3 In the **Settings** window for **Volume Force**, locate the **Volume Force** section.

4 Specify the **F** vector as

`-alpha*u`	x
`-alpha*v`	y

Inlet 1

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

2 Select Boundary 1 only.

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

4 From the list, choose **Pressure**.

5 Locate the **Pressure Conditions** section. In the p_0 text field, type `2`.

Outlet 1

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

2 Select Boundary 10 only.

Keep all other boundaries at the default condition, which is the no-slip condition.

MESH 1

This model is naturally highly dependent on the mesh size. In this example, choose a dense mesh; note, however, that this mesh can be further improved.

Free Triangular 1

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

Size

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

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 Click the **Custom** button.

4 Locate the **Element Size Parameters** section. In the **Maximum element growth rate** text field, type `1.1`.

Size 1

1 In the **Model Builder** window, right-click **Free Triangular 1** and choose **Size**.

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

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

4 Select Domain 2 only.

5 Locate the **Element Size** section. Click the **Custom** button.

6 Locate the **Element Size Parameters** section.

7 Select the **Maximum element size** checkbox. In the associated text field, type `0.08`.

Size 2

1 Right-click **Free Triangular 1** and choose **Size**.

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

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

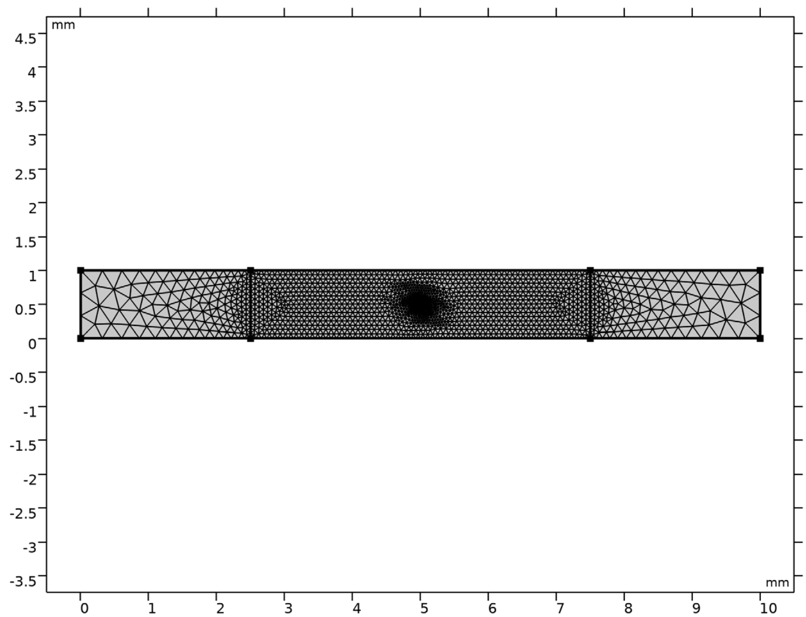
4 Select Point 5 only.

5 Locate the **Element Size** section. Click the **Custom** button.

6 Locate the **Element Size Parameters** section.

7 Select the **Maximum element size** checkbox. In the associated text field, type `meshsz`.

8 Click **Build All**.



Although you can choose to solve the optimization problem directly, it can be good to check that the initial solution looks reasonable.

STUDY 1

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

RESULTS

Probe Plot Group 1

The resulting plots show the magnitude of the velocity field, the pressure, and the design variable distribution in the channel. Proceed to solve the optimization problem.

STUDY 1

Topology Optimization

1 In the **Study** toolbar, click **Optimization** and choose **Topology Optimization**.

2 In the **Settings** window for **Topology Optimization**, click **Add Expression** in the upper-right corner of the **Objective Function** section. From the menu, choose **Component 1 (comp1)** > **Definitions** > **comp1.obj - Point Probe 1 - m/s**.

3 Locate the **Objective Function** section. Find the **Objective settings** subsection. From the **Objective scaling** list, choose **Initial solution based**.

4 Click to expand the **Output** section. Select the **Plot** checkbox.

5 In the table, enter the following settings:

Plot group	Plot window
Velocity (spf)	Graphics

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

RESULTS

Velocity (spf)

The default plot shows the magnitude of the velocity field in the channel. Generate Figure 3 with the following instructions:

Surface

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

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Laminar Flow** > **Velocity and pressure** > **Velocity field - m/s** > **u - Velocity field, x-component**.

3 Locate the **Expression** section. From the **Unit** list, choose **mm/s**.

Contour 1

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

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

3 In the **Expression** text field, type `dtopo1.theta`.

4 Locate the **Levels** section. From the **Entry method** list, choose **Levels**.

5 In the **Levels** text field, type `0.5`.

6 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

7 From the **Color** list, choose **Black**.

8 Clear the **Color legend** checkbox.

Streamline 1

1 Right-click **Velocity (spf)** and choose **Streamline**.

2 In the **Settings** window for **Streamline**, 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 - Velocity field**.

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

4 In the **Minimum density level** text field, type `0`.

5 In the **Maximum density level** text field, type `10.9`.

6 Locate the **Coloring and Style** section. Find the **Point style** subsection. From the **Color** list, choose **White**.

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

8 Click the **Zoom Box** button in the **Graphics** toolbar and then use the mouse to zoom in.

Pressure (spf)

The second default plot shows the pressure distribution Figure 4.

Output material volume factor

The last plot shows the material distribution in the channel Figure 2.

Objective Function History

1 In the **Model Builder** window, expand the **Results** > **Topology Optimization** node, then click **Results** > **Probe Plot Group 1**.

2 In the **Settings** window for **1D Plot Group**, type `Objective Function History` in the **Label** text field.