Created in COMSOL Multiphysics 5.6

Optimization of a Tesla Microvalve

Introduction

A Tesla valve inhibits backward flow on a fixed geometry by utilizing friction forces instead of moving parts, Ref. 1. This means fluid can flow freely in one direction but not in the reverse direction. Typically the Reynolds number of the flow in microfluidics is between 1 and 100.

Model Definition

The model solves two instances of the Navier-Stokes equations, one for the forward flow and one for the reverse. The Reynolds number is 100 in this example. A measure of the effectiveness of the design is the ratio of the pressure drop between the inlet and outlet for the forward and reverse flow. The pressure drop is defined as:

$$L\Delta p_{\text{forward}} = \int_{\text{inlet}} p dS - \int_{\text{outlet}} p dS$$

where L is the length of the inlet and outlet. For the reverse flow the same expression is used, except the inlet and outlet correspond to different boundaries in the model. The ratio of the pressure drop between the reverse and forward flow is then:

$$D_i = \frac{\Delta p_{\text{backward}}}{\Delta p_{\text{forward}}}$$

Unfortunately this expression does not form a well posed objective function, so an alternative expression is required for the optimization problem. According to Ref. 2, the energy dissipation is a well posed objective function for topological optimization:

$$\text{obj} = \int_{\Omega} (\tau{:}\mathbf{S} + \alpha(\mathbf{u}\cdot\mathbf{u}))dV$$

where τ is the viscous stress and $\mathbf{S}$ is the strain rate tensor:

$$\mathbf{S} = \frac{1}{2}(\nabla\mathbf{u} + (\nabla\mathbf{u})^T)$$

The fluid flow is described by the Navier-Stokes equations:

$$\begin{aligned} \rho(\mathbf{u}\cdot\nabla)\mathbf{u} &= -\nabla p + \nabla\cdot\eta(\nabla\mathbf{u} + (\nabla\mathbf{u})^T) - \alpha(\theta)\mathbf{u} \\ \nabla\cdot\mathbf{u} &= 0 \end{aligned}$$

where the coefficient $\alpha(\theta)$ depends on the distribution of material which impedes the flow within the device. In this example, $\alpha(\theta)$ is given by:

$$\alpha(\theta_p) = \alpha_0\theta_p, \quad \theta_p = \frac{\alpha_0(q+\theta)}{q+\theta}$$

$$\theta = \frac{(\tanh(\beta(\theta_f - \theta_\beta)) + \tanh(\beta\theta_\beta))}{(\tanh(\beta(1-\theta_\beta)) + \tanh(\beta\theta_\beta))}$$

$$\theta_f = R_{\min}^2\nabla^2\theta_f + \theta_c$$

where θ_c and θ_f, θ and θ_p are the control- and filtered material volume factor. To avoid the effect of grayscale the filtered field is projected to construct the material volume factor, θ, which is related to the damping term using a convex function, see Ref. 3. This method is used to avoid grayscale and spurious holes as seen in Figure 1.

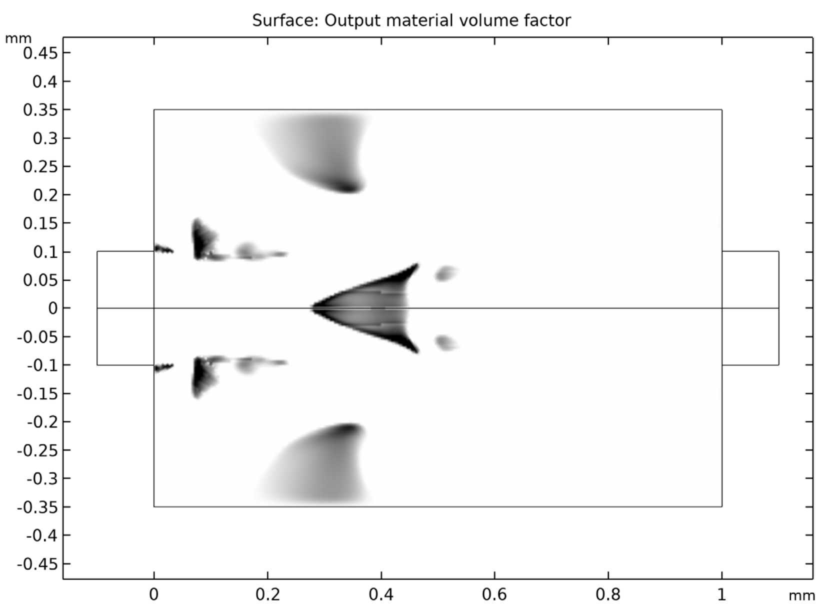

Figure 1: Optimizing without a filter gives a design with interpretation issues.

Results and Discussion

As expected, the forward and reverse flow are identical but in opposite directions for the initial topology; see Figure 2 and Figure 3. The forward flow in the optimized design after 200 iterations can be seen in Figure 4. Material has been added close to the outlet in a triangular shape that makes the forward flow bend smoothly around it. This smooth diversion of the flow from the point of impingement results in an overall low pressure drop between the inlet and the outlet. The reverse flow, shown in Figure 5, is far more interesting. The triangular shaped obstacle has a flat edge normal to the incident fluid, which means the velocity is redirected upward and downward toward the exterior walls. The redirected flow is then directed toward additional obstacles that further impede the flow path.

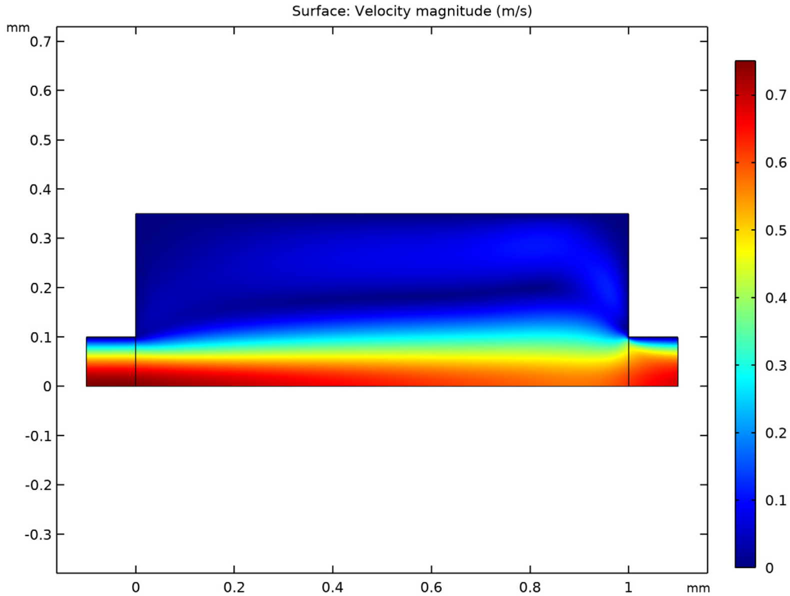

Figure 2: Forward flow velocity field, initial geometry.

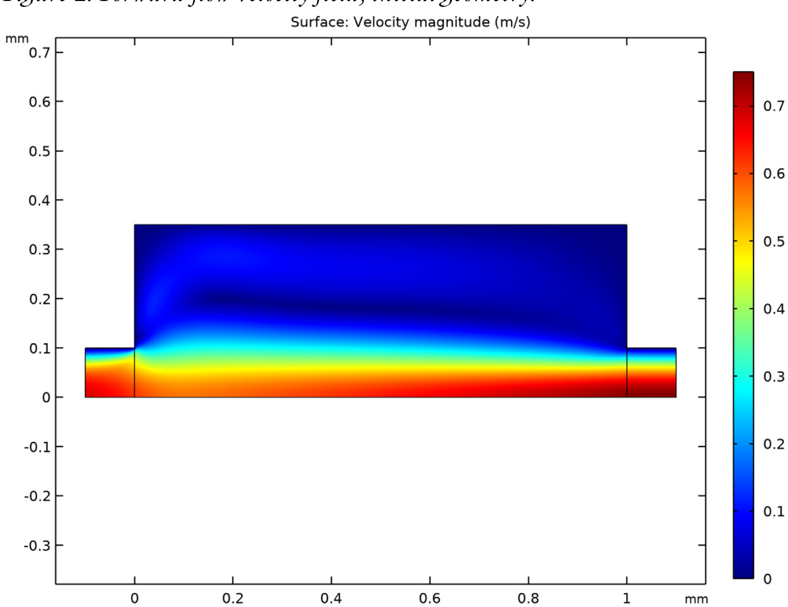

Figure 3: Reverse flow velocity field, initial geometry.

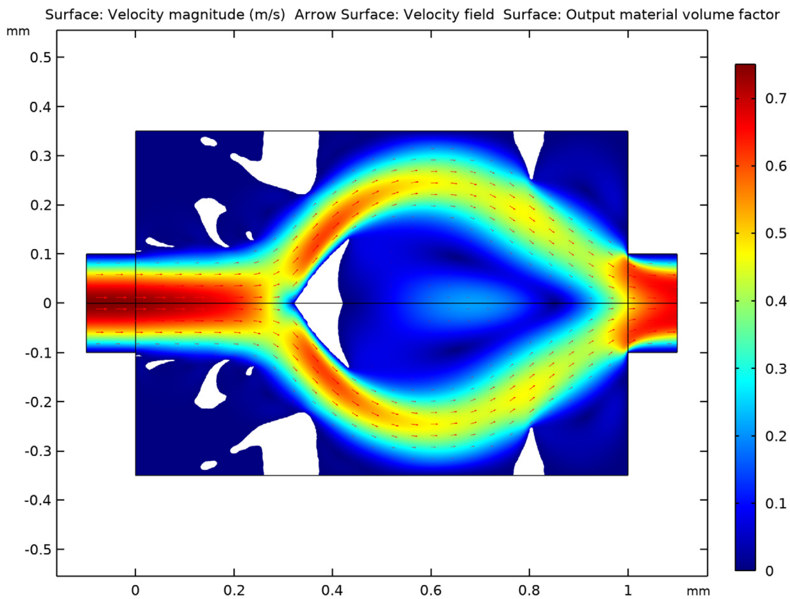

Figure 4: Forward flow, optimized topology.

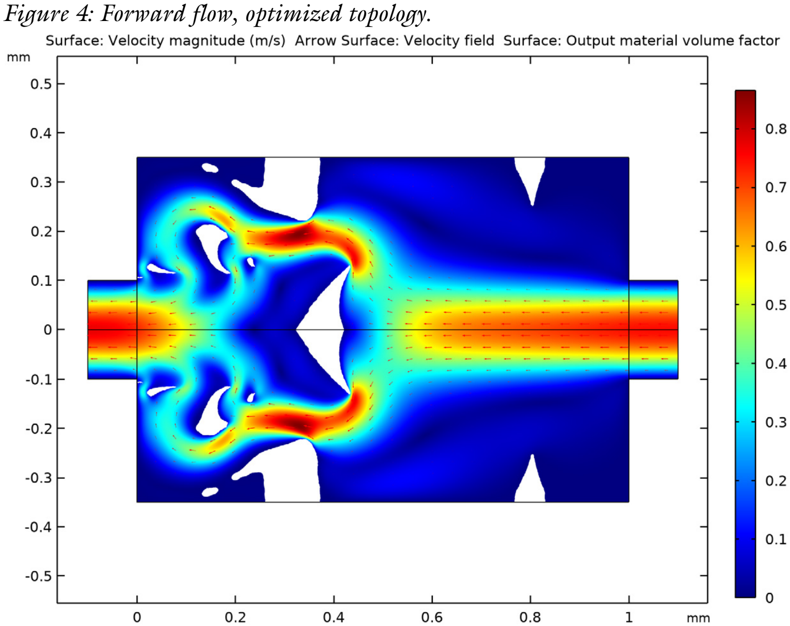

Figure 5: Reverse flow, optimized topology.

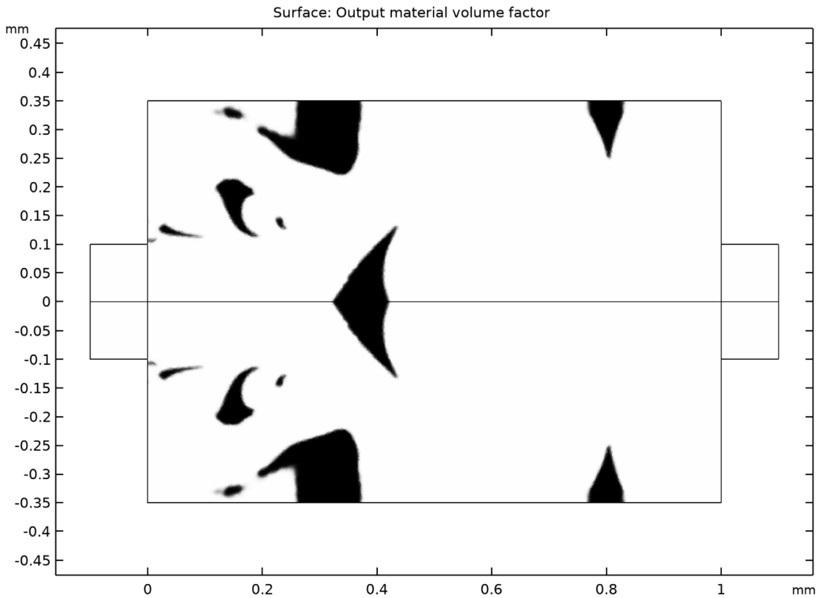

Figure 6: Control variable after optimization.

Finally, a global evaluation is used to compute the ratio of the pressure drop between the forward and reverse flow for a damping 100 times the value used for the optimization. The computed value should be approximately 2.34.

References

1. S. Lin, "Topology Optimization of Micro Tesla Valve in low and moderate Reynolds number," Chinese Academy of Sciences, China, September 27, 2011 http://senlin.weebly.com/uploads/6/6/1/4/6614199/sen_lin_topology_optimization_of_micro_tesla_valve.pdf.

2. 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. Meth. Engng*, vol. 65, pp. 975–1001, 2006.

3. T. Borrvall and J. Petersson, "Topology optimization of fluids in Stokes flow," *Int. J. Numer. Meth. Fluid*, vol. 41, pp. 77–107, 2003.

Notes About the COMSOL Implementation

The model is set up using two Laminar Flow interfaces, one for the forward flow and one for the reverse.

Application Library path: `Optimization_Module/Topology_Optimization/tesla_microvalve_optimization`

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)**.

Add two **Laminar Flow** interfaces, one for the forward flow and one for the backward flow.

3 Click **Add**.

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

5 Click **Add**.

6 Click **Study**.

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

8 Click **Done**.

GEOMETRY 1

The geometry is on the order of millimeters, so change the geometry unit.

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**.

Add some parameters to compute the average inlet velocity for the flow, based on a chosen Reynolds number, in this case, 100.

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
Re	100	100	Reynolds number
D	0.2[mm]	2E-4 m	Characteristic dimension
L	5*D	0.001 m	Length of channel
H	1.75*D	3.5E-4 m	Width of channel
mu0	1E-3[Pa*s]	0.001 Pa·s	Dynamic viscosity
rho0	1E3[kg/m^3]	1000 kg/m³	Density
Uin	Re*mu0/(rho0*D)	0.5 m/s	Average inlet velocity
meshsz	0.005*L	5E-6 m	Mesh size

GEOMETRY 1

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 `L`.

4 In the **Height** text field, type `H`.

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 `D/2`.

4 In the **Height** text field, type `D/2`.

5 Locate the **Position** section. In the **x** text field, type `-D/2`.

6 Click **Build Selected**.

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 `D/2`.

4 In the **Height** text field, type `D/2`.

5 Locate the **Position** section. In the **x** text field, type `L`.

Symmetry

1 In the **Geometry** toolbar, click **Selections** and choose **Box Selection**.

2 In the **Settings** window for **Box Selection**, type `Symmetry` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Box Limits** section. In the **y maximum** text field, type `1e3*eps`.

5 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Left

1 In the **Geometry** toolbar, click **Selections** and choose **Box Selection**.

2 In the **Settings** window for **Box Selection**, type `Left` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Box Limits** section. In the **x maximum** text field, type `-D/2+1e3*eps`.

5 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

Right

1 In the **Geometry** toolbar, click **Selections** and choose **Box Selection**.

2 In the **Settings** window for **Box Selection**, type `Right` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Box Limits** section. In the **x minimum** text field, type `L+D/2-1e3*eps`.

5 Locate the **Output Entities** section. From the **Include entity if** list, choose **Entity inside box**.

6 Click **Build All Objects**.

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

GLOBAL DEFINITIONS

Add a blank material for `rho0` and `mu0`.

Material 1 (mat1)

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

2 In the **Materials** toolbar, click **User-Defined Property Group**.

3 In the **Settings** window for **Property Group**, locate the **Output Properties** section.

4 Click **Select Quantity**.

5 In the **Physical Quantity** dialog box, type `density` in the text field.

6 Click **Filter**.

7 In the tree, select **General>Density (kg/m^3)**.

8 Click **OK**.

9 In the **Settings** window for **Property Group**, locate the **Output Properties** section.

10 In the table, enter the following settings:

Property	Variable	Expression	Unit	Size
Density	rho	rho0	kg/m³	1x1

11 Click **Select Quantity**.

12 In the **Physical Quantity** dialog box, type `viscosity` in the text field.

13 Click **Filter**.

14 In the tree, select **Transport>Dynamic viscosity (Pa*s)**.

15 Click **OK**.

16 In the **Settings** window for **Property Group**, locate the **Output Properties** section.

17 In the table, enter the following settings:

Property	Variable	Expression	Unit	Size
Dynamic viscosity	mu	mu0	Pa·s	1x1

MATERIALS

Material Link 1 (matlnk1)

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

LAMINAR FLOW (SPF)

Inlet 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Laminar Flow (spf)** and choose **Inlet**.

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

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

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 `Uin`.

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 **Right**.

Wall 2

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

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

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

4 Locate the **Boundary Condition** section. From the **Wall condition** list, choose **Slip**.

LAMINAR FLOW 2 (SPF2)

In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow 2 (spf2)**.

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 **Right**.

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 `Uin`.

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 **Left**.

Wall 2

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

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

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

4 Locate the **Boundary Condition** section. From the **Wall condition** list, choose **Slip**.

The advancing front mesher propagates the directions of the geometry into the interior, so use a Delaunay mesh to avoid favored directions in the mesh.

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 **Extremely fine**.

Free Triangular 1

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

2 In the **Settings** window for **Free Triangular**, click to expand the **Tessellation** section.

3 From the **Method** list, choose **Delaunay**.

Size

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

2 In the **Settings** window for **Size**, click to expand the **Element Size Parameters** section.

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

4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `meshsz`.

5 Click **Build All**.

STUDY 1

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

RESULTS

Mirror 2D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Mirror 2D**.

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

3 In row **Point 2**, set **x** to `1` and **y** to `0`.

Velocity (spf)

1 In the **Model Builder** window, click **Velocity (spf)**.

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

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

Velocity (spf2)

1 In the **Model Builder** window, click **Velocity (spf2)**.

2 In the **Velocity (spf2)** toolbar, click **Plot**.

Now define two average operators on the inlet and outlet of the modeling domain. These will be used to compute the pressure ratio between the inlet and outlet for the forward and reverse flow.

DEFINITIONS

Average 1 (aveop1)

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 **Left**.

Average 2 (aveop2)

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 **Right**.

Integration 1 (intop1)

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

2 Select Domain 2 only.

Define the variable corresponding to the ratio of the pressure difference between the forward and backward flow.

Variables 1

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

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
dP_forward	aveop1(p)-aveop2(p)	Pa	Pressure difference, forward direction
dP_backward	aveop2(p2)-aveop1(p2)	Pa	Pressure difference, backward direction
Di	dP_backward/dP_forward		Ratio of pressure differences

In order to evaluate the ratio of the pressure differences, the model needs updating. Note that the model does not need to be solved again.

STUDY 1

In the **Study** toolbar, click **Update Solution**.

RESULTS

Diodicity

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

2 In the **Settings** window for **Evaluation Group**, type `Diodicity` in the **Label** text field.

Global Evaluation 1

1 Right-click **Diodicity** and choose **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)> Definitions>Variables>Di - Ratio of pressure differences**.

3 In the **Diodicity** toolbar, click **Evaluate**.

TABLE

1 Go to the **Table** window.

Observe that the ratio of the pressure differences is very close to 1. This concludes solving of the forward problem, now the optimization problem needs to be set up.

Add a density topology feature, which can be used to distinguish between free flow and solid regions. This variable will be coupled back to the **Laminar Flow** interfaces later. The filter radius should not be smaller than the mesh element size, so the default will work, but a fixed value has to be chosen to make the result of the optimization mesh independent.

DEFINITIONS

Density Model 1 (dtopo1)

1 In the **Definitions** toolbar, click **Optimization** and choose **Density Model**.

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

2 Select Domain 2 only.

We use a filter to prevent the checkboard instability for the control variable `theta` which may otherwise occur.

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

4 From the **Interpolation type** list, choose **Darcy**.

5 Locate the **Projection** section. From the **Projection type** list, choose **Hyperbolic tangent projection**.

6 Locate the **Interpolation** section. In the q_{Darcy} text field, type `1`.

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

8 Locate the **Control Variable Discretization** section. From the **Element order** list, choose **Constant**.

Now the design variable used in the optimization is defined. The initial value 1 corresponds to a channel free from porous material.

Now define the friction force to be used in the **Laminar Flow** interfaces, the viscous and friction dissipation which can be integrated over the domain to obtain a suitable objective function.

Variables 1

1 In the **Model Builder** window, 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
phi_forward	spf.Qvd+alpha*(u^2+v^2)	W/m³	Dissipation density, forward flow
phi_backward	spf2.Qvd+alpha*(u2^2+v2^2)	W/m³	Dissipation density, backward flow
phi_total	phi_backward+phi_forward	W/m³	Total dissipation

Name	Expression	Unit	Description
`E_forward`	`intop1(phi_forward)`	W/m	`Energy dissipation, forward flow`
`E_backward`	`intop1(phi_backward)`	W/m	`Energy dissipation, backward flow`
`obj`	`E_backward/E_forward`		`Objective function`
`alpha`	`16.*mu0*dtopo1.theta_p/meshsz^2`	Pa·s/m²	`Friction force`

LAMINAR FLOW (SPF)

Add the friction force to the **Laminar Flow** interfaces so that the fluid flows around regions where `theta` is 0 and through regions where `theta` is 1.

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

Volume Force 1

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

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

LAMINAR FLOW 2 (SPF2)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow 2 (spf2)**.

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

Volume Force 1

1 Select Domain 2 only.

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

3 Specify the **F** vector as

`-alpha*u2`	x
`-alpha*v2`	y

OPTIMIZATION

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

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

Topology Optimization

1 Right-click **Optimization** and choose **Optimization>Topology Optimization**.

2 In the **Settings** window for **Topology Optimization**, locate the **Optimization Solver** section.

3 From the **Method** list, choose **SNOPT**.

4 In the **Optimality tolerance** text field, type `1E-9`.

5 In the **Maximum number of iterations** text field, type `200`.

6 Click **Add Expression** in the upper-right corner of the **Objective Function** section. From the menu, choose **Component 1 (comp1)>Definitions>Variables>comp1.obj - Objective function**.

7 Locate the **Objective Function** section. From the **Type** list, choose **Maximization**.

8 In the **Study** toolbar, click **Get Initial Value**.

Solver Configurations

In the **Model Builder** window, expand the **Optimization>Solver Configurations** node.

Solution 1 (sol1)

1 In the **Model Builder** window, expand the **Optimization>Solver Configurations> Solution 1 (sol1)>Optimization Solver 1>Stationary 1** node, then click **Fully Coupled 1**.

2 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.

3 From the **Nonlinear method** list, choose **Constant (Newton)**.

RESULTS

Mirror 2D 1

Use the mirrored dataset to plot the value of `theta` during the optimization.

Topology

1 In the **Home** toolbar, click **Add Plot Group** and choose **2D Plot Group**.

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

3 Locate the **Data** section. From the **Dataset** list, choose **Mirror 2D 1**.

Surface 1

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

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

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

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

5 Clear the **Color legend** check box.

6 Click to expand the **Range** section. Select the **Manual color range** check box.

7 In the **Maximum** text field, type `1`.

8 In the **Topology** toolbar, click **Plot**.

OPTIMIZATION

Topology Optimization

1 In the **Model Builder** window, under **Optimization** click **Topology Optimization**.

2 In the **Settings** window for **Topology Optimization**, locate the **Output While Solving** section.

3 Select the **Plot** check box.

4 From the **Plot group** list, choose **Topology**.

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

RESULTS

Forward Flow

The forward and backward flow pattern computed using the optimization solver can now be visualized.

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

2 From the **Dataset** list, choose **Mirror 2D 1**.

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

4 In the **Label** text field, type `Forward Flow`.

Arrow Surface 1

Right-click **Forward Flow** and choose **Arrow Surface**.

Arrow Surface 1

1 In the **Model Builder** window, expand the **Results>Forward Flow** node, then click **Arrow Surface 1**.

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

3 Find the **x grid points** subsection. In the **Points** text field, type `30`.

4 Find the **y grid points** subsection. In the **Points** text field, type `30`.

5 In the **Forward Flow** toolbar, click **Plot**.

Surface 2

1 In the **Model Builder** window, right-click **Forward Flow** and choose **Surface**.

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

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

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

5 From the **Color** list, choose **White**.

6 Locate the **Range** section. Select the **Manual data range** check box.

7 In the **Maximum** text field, type `0.5`.

8 In the **Forward Flow** toolbar, click **Plot**.

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

Now follow a similar procedure for the backward flow.

Arrow Surface 1, Surface 2

1 In the **Model Builder** window, under **Results>Forward Flow**, Ctrl-click to select **Arrow Surface 1** and **Surface 2**.

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

Backward Flow

1 In the **Model Builder** window, under **Results** click **Velocity (spf2)**.

2 In the **Settings** window for **2D Plot Group**, type `Backward Flow` in the **Label** text field.

Arrow Surface 1

1 Right-click **Backward Flow** and choose **Paste Multiple Items**.

2 In the **Settings** window for **Arrow Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Laminar Flow 2>Velocity and pressure>u2,v2 - Velocity field**.

3 In the **Backward Flow** toolbar, click **Plot**.

Backward Flow

1 In the **Model Builder** window, click **Backward Flow**.

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

3 From the **Dataset** list, choose **Mirror 2D 1**.

4 In the **Backward Flow** toolbar, click **Plot**.

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

6 In the **Backward Flow** toolbar, click **Plot**.

Topology

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

2 In the **Topology** toolbar, click **Plot**.

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

Backward Flow, Forward Flow, Pressure (spf), Pressure (spf2)

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Forward Flow**, **Pressure (spf)**, **Backward Flow**, and **Pressure (spf2)**.

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

Optimized Design

In the **Settings** window for **Group**, type `Optimized Design` in the **Label** text field.

Lets evaluate the damping with respect to the relative amount of dissipation in the solid region to see, if the optimization result can be trusted.

Dissipation in Solid Regions

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

2 In the **Settings** window for **Evaluation Group**, type `Dissipation in Solid Regions` in the **Label** text field.

Global Evaluation 1

1 Right-click **Dissipation in Solid Regions** and choose **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**
`intop1(phi_total*(1-dtopo1.theta))/intop1(phi_total)`	1	Relative dissipation in solid

4 In the **Dissipation in Solid Regions** toolbar, click **Evaluate**.

There is significant power loss in the solid regions, which is unphysical. Therefore it is a good idea to setup a verification analysis in a new component.

The **Density Model** triggers a **Filter Dataset**, when the default plots are created, but we added it later, and therefore we create the **Filter Dataset** manually.

Filter 1

1 In the **Results** toolbar, click **More Datasets** and choose **Filter**.
2 In the **Settings** window for **Filter**, locate the **Expression** section.
3 In the **Expression** text field, type `dtopo1.theta`.
4 Locate the **Filter** section. In the **Lower bound** text field, type `0.5`.
5 Locate the **Evaluation** section. Clear the **Use derivatives** check box.
6 Click **Plot**.
7 Right-click **Filter 1** and choose **Create Mesh Part**.

MESH PART 1

Adapt 1

1 In the **Mesh** toolbar, click **Modify** and choose **Elements>Adapt**.
2 In the **Model Builder** window, right-click **Mesh Part 1** and choose **Build All**.
3 Right-click **Mesh Part 1** and choose **Create Geometry from Mesh**.
4 In the **Settings** window for **Adapt**, locate the **Adaptation** section.
5 In the **Size expression** text field, type `meshsz/2`.
6 In the **Maximum coarsening factor** text field, type `Inf`.

MATERIALS

Material Link 2 (matlnk2)

In the **Model Builder** window, under **Component 2 (comp2)** right-click **Materials** and choose **More Materials>Material Link**.

LAMINAR FLOW (SPF), LAMINAR FLOW 2 (SPF2)

1 In the **Model Builder** window, under **Component 1 (comp1)**, Ctrl-click to select **Laminar Flow (spf)** and **Laminar Flow 2 (spf2)**.

   Copy/paste the physics from the first component and fix the selections.

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

LAMINAR FLOW (SPF3)

In the **Model Builder** window, right-click **Component 2 (comp2)** and choose **Paste Multiple Items**.

LAMINAR FLOW (SPF3), LAMINAR FLOW 2 (SPF4)

1 In the **Model Builder** window, under **Component 2 (comp2)**, Ctrl-click to select **Laminar Flow (spf3)** and **Laminar Flow 2 (spf4)**.

2 In the **Messages from Paste** dialog box, click **OK**.

Inlet 1

1 In the **Model Builder** window, under **Component 2 (comp2)>Laminar Flow (spf3)** click **Inlet 1**.

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

3 From the **Selection** list, choose **Left (Import 1)**.

Outlet 1

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

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

3 From the **Selection** list, choose **Right (Import 1)**.

Wall 2

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

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

3 From the **Selection** list, choose **Symmetry (Import 1)**.

Volume Force 1

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

LAMINAR FLOW 2 (SPF4)

Inlet 1

1 In the **Model Builder** window, under **Component 2 (comp2)>Laminar Flow 2 (spf4)** click **Inlet 1**.

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

3 From the **Selection** list, choose **Right (Import 1)**.

Outlet 1

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

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

3 From the **Selection** list, choose **Left (Import 1)**.

Wall 2

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

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

3 From the **Selection** list, choose **Symmetry (Import 1)**.

Volume Force 1

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

DEFINITIONS (COMP2)

Setup a new average operators, so a new diodicity variable can be defined for the new component.

Average 3 (aveop3)

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 **Left (Import 1)**.

Average 4 (aveop4)

1 Right-click **Average 3 (aveop3)** and choose **Duplicate**.

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

3 From the **Selection** list, choose **Right (Import 1)**.

Variables 2

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
Di	(aveop4(p2)-aveop3(p2))/(aveop3(p)-aveop4(p))		Ratio of pressure differences

MESH 2

Change the mesh settings to avoid element concentrations near irrelevant details of the imported geoemtry.

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 Parameters** section.

3 In the **Maximum element size** text field, type `0.01`.

4 In the **Curvature factor** text field, type `10`.

5 In the **Resolution of narrow regions** text field, type `0.1`.

6 Click **Build All**.

ADD STUDY

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 **General Studies>Stationary**.

4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check boxes for **Laminar Flow (spf)** and **Laminar Flow 2 (spf2)**.

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

6 In the **Model Builder** window, click the root node.

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

OPTIMIZATION

Step 1: Stationary

1 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

2 In the table, clear the **Solve for** check boxes for **Laminar Flow (spf3)** and **Laminar Flow 2 (spf4)**.

VERIFICATION

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

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

3 Select the **Generate default plots** check box.

4 In the **Label** text field, type `Verification`.

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

RESULTS

Velocity (spf3)

In the **Model Builder** window, collapse the **Results>Velocity (spf3)** node.

Pressure (spf3), Pressure (spf4), Velocity (spf3), Velocity (spf4)

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Velocity (spf3)**, **Pressure (spf3)**, **Velocity (spf4)**, and **Pressure (spf4)**.

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

Verification

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

Topology Optimization

In the **Model Builder** window, right-click **Topology Optimization** and choose **Delete**.

Diodicity

Compute the diodicity for the component (without power loss in solid domains).

Global Evaluation 2

1 In the **Model Builder** window, right-click **Diodicity** and choose **Global Evaluation**.

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

3 From the **Dataset** list, choose **Verification/Solution 2 (3) (sol2)**.

4 Click **Add Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 2 (comp2)>Definitions>Variables>Di - Ratio of pressure differences**.

5 In the **Diodicity** toolbar, click **Evaluate**.

The diodicity is actually higher for the verification simulation, so optimization has found a good design topology.