Created in COMSOL Multiphysics 6.2

Optimization of a Tesla Microvalve with Transient Flow

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. Note that the objective function is different from Optimization of a Tesla Microvalve, because the objective is defined in terms of the flow rate with an oscillating pressure difference (instead of the other way around with an objective defined in terms of the pressure drop and oscillating flow rate).

Model Definition

The model solves the transient Navier–Stokes equations with an oscillating pressure drop. The Reynolds number is 500 in this example. A measure of the effectiveness of the design is the average flow rate in the x direction during the period T_{period}:

$$U_{\mathrm{average}} = \int_T\int_\Omega \mathbf{u}\cdot\hat{\mathbf{x}}\,d\Omega dt/T_{\mathrm{period}}$$

where T is the simulation time taken as two periods of pressure oscillation. The model uses an ODE to perform the time integration (see Figure 1):

$$\frac{d\varphi}{dt}t_{\mathrm{max}} = f_{\mathrm{step}}U_{\mathrm{average}}/U_{\mathrm{scale}}$$

where $t_{\mathrm{max}} = 1.75T_{\mathrm{period}}$ and f_{step} is defined so that integration occurs over the last period:

$$f_{\mathrm{step}} = \begin{cases} 0, & t < t_{\mathrm{max}} - T_{\mathrm{period}} \\ 1, & \text{otherwise} \end{cases}$$

The fluid flow is described by the incompressible Navier–Stokes equations:

$$\rho\left[(\mathbf{u}\cdot\nabla)\mathbf{u} + \frac{\partial\mathbf{u}}{\partial t}\right] = -\nabla p + \nabla\cdot\eta(\nabla\mathbf{u} + (\nabla\mathbf{u})^T) - \alpha(\theta_f)\mathbf{u}$$

$$\nabla\cdot\mathbf{u} = 0$$

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

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

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

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

where θ_c and θ_f are the control- and filtered material volume factors. 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.

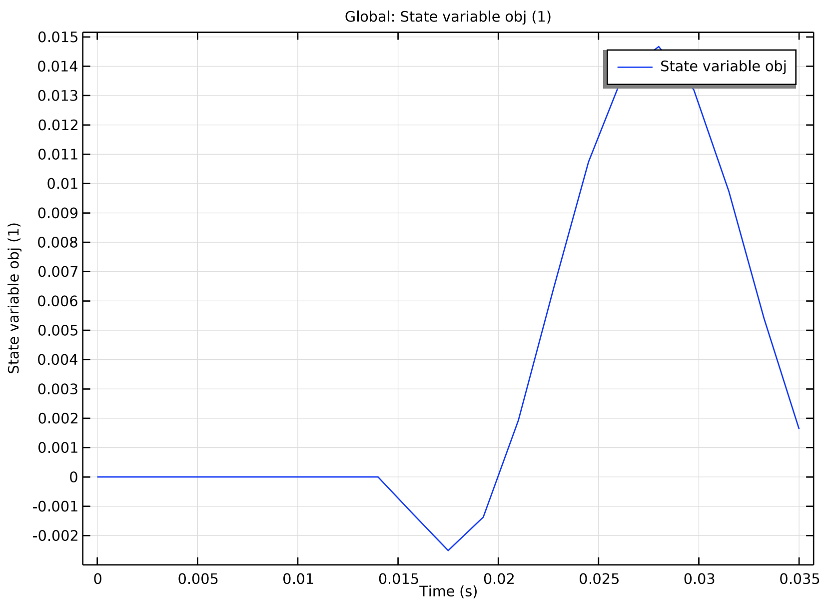

Figure 1: The ODE variable (used to perform the time integration) is plotted for the optimized design.

Results and Discussion

As expected, the design has features similar to that seen in Optimization of a Tesla Microvalve, but the central obstacle is placed further to the right, as seen in Figure 2 and Figure 3. A verification simulation is performed using a body fitted mesh, see Figure 4 and Figure 5. The analysis can be used to ensure that the optimization does not rely on unphysical effects due to the approximate wall descriptions of the optimization.

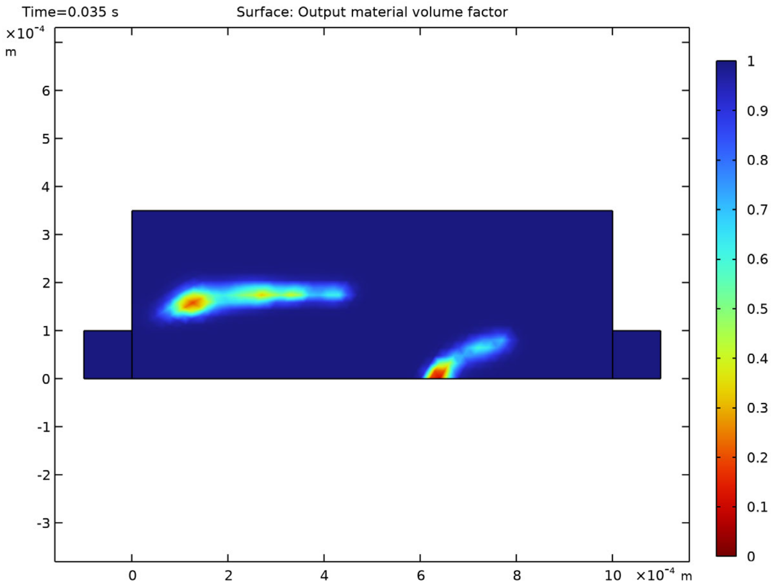

Figure 2: The filtered material volume factor is plotted for the optimized design.

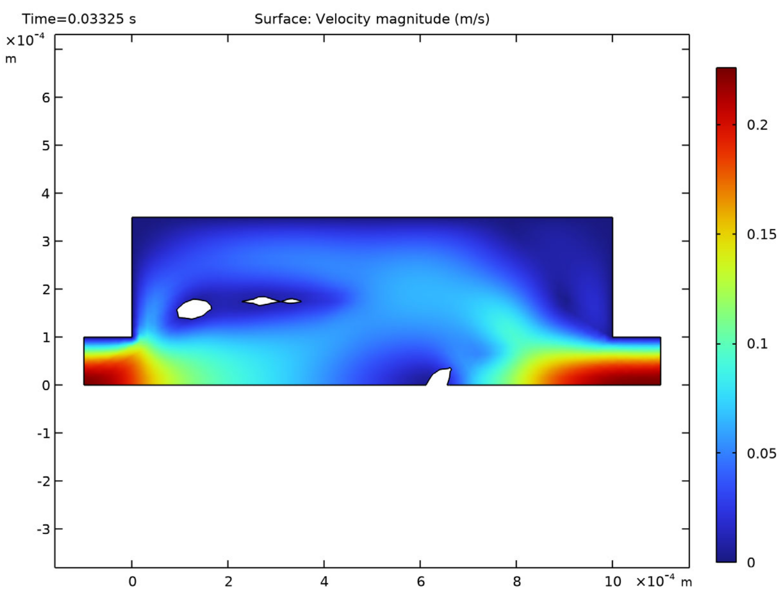

Figure 3: The flow velocity is plotted, while the pressure drop is driving the fluid backward.

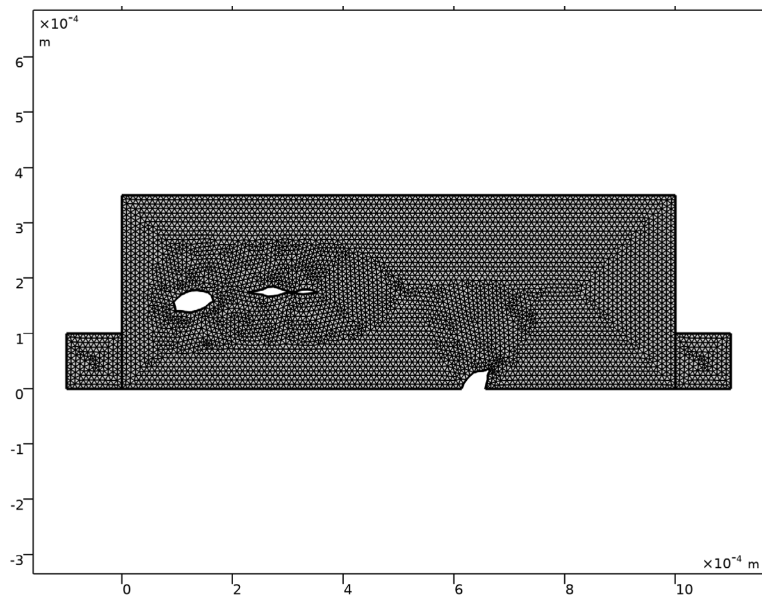

Figure 4: The body fitted mesh used for the verification allows for no-slip boundary conditions.

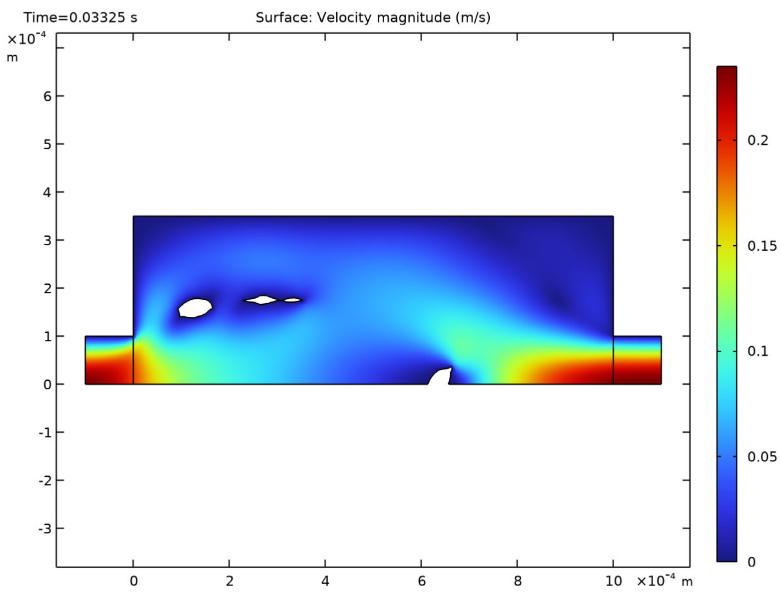

Figure 5: The flow velocity is plotted, while the pressure drop is driving the fluid backward.

A quantitative comparison with the optimization results reveals that objective is slightly worse for the verification, as opposed to the behavior in Optimization of a Tesla Microvalve.

Notes About the COMSOL Implementation

This example uses an objective function defined as a time integral, but the evaluation of transient objectives is always performed at the last time step. Therefore the time integration is implemented as an ODE.

A fixed time step is combined with a segregated solver and recycling of the forward time steps to ensure robustness of the adjoint problem.

References

1. S. Lin, "Topology Optimization of Micro Tesla Valve in low and moderate Reynolds number," Chinese Academy of Sciences, China, September 27, 2011.

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. Methods Eng.*, 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 uses an ODE for the time integration, and the original MMA optimization solver (which is not globally convergent) is used. Finally, fixed time steps and a segregated solver is used to ensure robustness of the adjoint problem.

Application Library path: `Microfluidics_Module/Fluid_Flow/tesla_microvalve_transient_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)**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Mathematics>ODE and DAE Interfaces>Global ODEs and DAEs (ge)**.

5 Click **Add**.

6 Click **Study**.

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

8 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 Click **Load from File**.

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

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`.

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

Square 1 (sq1)

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

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

3 In the **Side length** text field, type `L1`.

4 Locate the **Position** section. In the **x** text field, type `-L1`.

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

Move 1 (mov1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Move**.

2 In the **Settings** window for **Move**, locate the **Input** section.

3 From the **Input objects** list, choose **Square 1**.

4 Select the **Keep input objects** check box.

5 Locate the **Displacement** section. In the **x** text field, type `L+L1`.

Symmetry

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

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

3 From the **Level** list, choose **Boundary**.

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

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

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

7 Right-click **Symmetry** and choose **Duplicate**.

Outlet

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Symmetry 1 (boxsel2)**.

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

3 Locate the **Box Limits** section. In the **x maximum** text field, type `-L1*0.999`.

4 In the **y maximum** text field, type `Inf`.

5 Right-click **Outlet** and choose **Duplicate**.

Inlet

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Outlet 1 (boxsel3)**.

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

3 Locate the **Box Limits** section. In the **x minimum** text field, type `L+L1*0.999`.

4 In the **x maximum** text field, type `Inf`.

Inlet/Outlet

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

2 In the **Settings** window for **Union Selection**, type `Inlet/Outlet` in the **Label** text field.

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

4 Locate the **Input Entities** section. Click **Add**.

5 In the **Add** dialog box, in the **Selections to add** list, choose **Outlet** and **Inlet**.

6 Click **OK**.

Pressure Reference Point

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

2 In the **Settings** window for **Disk Selection**, type `Pressure Reference Point` in the **Label** text field.

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

4 Locate the **Size and Shape** section. In the **Outer radius** text field, type `L/1000`.

MATERIALS

Water

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

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

3 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Density	rho	rho0	kg/m³	Basic
Dynamic viscosity	mu	mu0	Pa·s	Basic

COMPONENT 1 (COMP1)

Density Model 1 (dtopo1)

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

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

3 From the **Selection** list, choose **Rectangle 1**.

4 Locate the **Filtering** section. From the $R_{\min}$ list, choose **User defined**.

5 In the text field, type `meshsz`.

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

7 In the β text field, type `beta`.

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

9 In the q_{Darcy} text field, type `q`.

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

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

Prescribed Material 1

1 In the **Topology Optimization** toolbar, click **Prescribed Material**.

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

3 From the **Selection** list, choose **Square 1**.

DEFINITIONS

Velocity Average

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

2 In the **Settings** window for **Domain Probe**, type `Velocity Average` in the **Label** text field.

3 In the **Variable name** text field, type `uAvg`.

4 Locate the **Source Selection** section. From the **Selection** list, choose **Rectangle 1**.

5 Locate the **Probe Type** section. From the **Type** list, choose **Integral**.

6 Locate the **Expression** section. In the **Expression** text field, type `u/L/H`.

LAMINAR FLOW (SPF)

Symmetry 1

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

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

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

Periodic Flow Condition 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Periodic Flow Condition**.

2 In the **Settings** window for **Periodic Flow Condition**, locate the **Boundary Selection** section.

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

4 Locate the **Flow Condition** section. In the Δp text field, type `dp*cos(2*pi*t/Tperiod)`.

Pressure Point Constraint 1

1 In the **Physics** toolbar, click **Points** and choose **Pressure Point Constraint**.

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

3 From the **Selection** list, choose **Pressure Reference Point**.

Volume Force 1

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

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

3 From the **Selection** list, choose **Rectangle 1**.

4 Locate the **Volume Force** section. Specify the **F** vector as

`-alpha*dtopo1.theta_p*u`	x
`-alpha*dtopo1.theta_p*v`	y

GLOBAL ODES AND DAES (GE)

Global Equations 1 (ODE1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Global ODEs and DAEs (ge)** click **Global Equations 1 (ODE1)**.

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

3 In the table, enter the following settings:

Name	f(u,ut,utt,t) (1)	Initial value (u_0) (1)	Initial value (u_t0) (1/s)	Description
obj	objt*tmax-(tmax-Tperiod < t)*uAvg/U0	0	0	

The boolean expression filters out initialization effects in the first period.

MESH 1

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 size** text field, type `meshsz`.
5 Click **Build All**.

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 In the **Study** toolbar, click **Optimization** and choose **Topology Optimization**.
2 In the **Settings** window for **Topology Optimization**, locate the **Optimization Solver** section.
3 In the **Maximum number of iterations** text field, type `25`.
4 Select the **Move limits** check box. In the associated text field, type `0.2`.
5 Click **Add Expression** in the upper-right corner of the **Objective Function** section. From the menu, choose **Component 1 (comp1)>Global ODEs and DAEs>comp1.obj - State variable obj - 1**.
6 Locate the **Objective Function** section. From the **Type** list, choose **Maximization**.
7 From the **Objective scaling** list, choose **Manual**.
8 In the **Scale** text field, type `1.5e-3`.
9 Locate the **Output While Solving** section. From the **Probes** list, choose **None**.

Step 1: Time Dependent

1 In the **Model Builder** window, click **Step 1: Time Dependent**.
2 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
3 In the **Output times** text field, type `range(0,tmax/20,tmax)`.
4 In the **Study** toolbar, click **Get Initial Value**.

RESULTS

Output material volume factor

In the **Model Builder** window, expand the **Results>Topology Optimization** node.

Surface 1

1 In the **Model Builder** window, expand the **Output material volume factor** node, then click **Surface 1**.

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

3 From the **Color table transformation** list, choose **Reverse**.

OPTIMIZATION

Solver Configurations

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

Solution 1 (sol1)

In this case the original MMA (1987) converges to the correct topology with fewer model evaluations.

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

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

3 Clear the **Globally Convergent MMA** check box.

The sensitivity analysis only happens to be robust when the time steps from the forward solution are recycled for the adjoint solution.

4 From the **Gradient method** list, choose **Adjoint**.

5 Click to expand the **Backward Time Stepping** section. From the **Backward time stepping** list, choose **From forward**.

We need to fix the time stepping to ensure robustness of the adjoint problem.

6 In the **Model Builder** window, expand the **Optimization>Solver Configurations> Solution 1 (sol1)>Optimization Solver 1** node, then click **Time-Dependent Solver 1**.

7 In the **Settings** window for **Time-Dependent Solver**, click to expand the **Time Stepping** section.

8 From the **Steps taken by solver** list, choose **Manual**.

9 In the **Time step** text field, type `tstep`.

10 Right-click **Time-Dependent Solver 1** and choose **Segregated**.

Switch to a **segregated solver** for increased robustness of the adjoint problem.

11 Right-click **Segregated 1** and choose **Segregated Step** twice.

12 In the **Settings** window for **Segregated Step**, type `Optimization` in the **Label** text field.

13 Locate the **General** section. In the **Variables** list, choose **Pressure (comp1.p)**, **Velocity field (comp1.u)**, and **Global Equations 1 (comp1.ODE1)**.

14 Under **Variables**, click **Delete**.

15 In the **Model Builder** window, under **Optimization>Solver Configurations> Solution 1 (sol1)>Optimization Solver 1>Time-Dependent Solver 1>Segregated 1** click **Segregated Step 1**.

16 In the **Settings** window for **Segregated Step**, type `Fluid Flow` in the **Label** text field.

17 Locate the **General** section. Under **Variables**, click **Add**.

18 In the **Add** dialog box, in the **Variables** list, choose **Control material volume factor (comp1.dtopo1.theta_c)**, **Pressure (comp1.p)**, and **Velocity field (comp1.u)**.

19 Click **OK**.

20 In the **Model Builder** window, under **Optimization>Solver Configurations> Solution 1 (sol1)>Optimization Solver 1>Time-Dependent Solver 1>Segregated 1** click **Segregated Step 2**.

21 In the **Settings** window for **Segregated Step**, type `Objective` in the **Label** text field.

22 Locate the **General** section. Under **Variables**, click **Add**.

23 In the **Add** dialog box, in the **Variables** list, choose **Control material volume factor (comp1.dtopo1.theta_c)** and **Global Equations 1 (comp1.ODE1)**.

24 Click **OK**.

25 In the **Model Builder** window, click **Direct (Merged)**.

26 In the **Settings** window for **Direct**, locate the **General** section.

27 From the **Solver** list, choose **PARDISO** to reduce the computational time.

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 **Output material volume factor**.

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

RESULTS

Objective

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

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

3 In the **Objective** toolbar, click **Plot**.

Output material volume factor

1 In the **Model Builder** window, under **Results>Topology Optimization** click **Output material volume factor**.

2 In the **Output material volume factor** toolbar, click **Plot**.

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

Velocity (spf)

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

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

3 From the **Dataset** list, choose **Filter**.

4 From the **Time (s)** list, choose **0.03325**.

It is obvious to see that there are significant approximation errors for the no-slip boundary condition in the optimization.

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

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

Perform a verification in a new component to ensure that the optimization results does not rely on unphysical effects.

Filter

1 In the **Model Builder** window, expand the **Results>Datasets** node, then click **Filter**.

2 In the **Settings** window for **Filter**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Definitions> Density Model 1>Auxiliary variables>dtopo1.theta_f - Filtered material volume factor**.

3 Right-click **Results>Datasets>Filter** and choose **Create Mesh Part**.

MESH PART 1

Import 1

1 In the **Settings** window for **Import**, locate the **Import** section.

2 Click **Import**.

3 In the **Model Builder** window, right-click **Mesh Part 1** and choose **Create Geometry**.

Copy/paste the physics, materials, and probe from the 1st component and correct the selections.

GLOBAL ODES AND DAES (GE), LAMINAR FLOW (SPF)

1 In the **Model Builder** window, under **Component 1 (comp1)**, Ctrl-click to select **Laminar Flow (spf)** and **Global ODEs and DAEs (ge)**.

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

COMPONENT 2 (COMP2)

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

GLOBAL ODES AND DAES (GE2), LAMINAR FLOW (SPF2)

1 In the **Model Builder** window, under **Component 2 (comp2)**, Ctrl-click to select **Laminar Flow (spf2)** and **Global ODEs and DAEs (ge2)**.

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

MATERIALS

Water (mat1)

In the **Model Builder** window, under **Component 1 (comp1)>Materials** right-click **Water (mat1)** and choose **Copy**.

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

DEFINITIONS (COMP1)

Velocity Average (uAvg)

In the **Model Builder** window, under **Component 1 (comp1)>Definitions** right-click **Velocity Average (uAvg)** and choose **Copy**.

DEFINITIONS (COMP2)

In the **Model Builder** window, under **Component 2 (comp2)** right-click **Definitions** and choose **Paste Domain Probe**.

Velocity Average (dom2)

1 In the **Model Builder** window, under **Component 2 (comp2)>Definitions** click **Velocity Average (dom2)**.

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

3 Locate the **Source Selection** section. From the **Selection** list, choose **Rectangle 1 (Import 1)**.

LAMINAR FLOW (SPF2)

Symmetry 1

1. In the **Model Builder** window, expand the **Component 2 (comp2)>Laminar Flow (spf2)** node, then click **Symmetry 1**.
2. In the **Settings** window for **Symmetry**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Symmetry (Import 1)**.

Periodic Flow Condition 1

1. In the **Model Builder** window, click **Periodic Flow Condition 1**.
2. In the **Settings** window for **Periodic Flow Condition**, locate the **Boundary Selection** section.
3. From the **Selection** list, choose **Inlet/Outlet (Import 1)**.

Pressure Point Constraint 1

1. In the **Model Builder** window, click **Pressure Point Constraint 1**.
2. In the **Settings** window for **Pressure Point Constraint**, locate the **Point Selection** section.
3. From the **Selection** list, choose **Pressure Reference Point (Import 1)**.

Volume Force 1

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

GLOBAL ODES AND DAES (GE2)

Global Equations 1 (ODE1)

1. In the **Model Builder** window, expand the **Component 2 (comp2)> Global ODEs and DAEs (ge2)** node, then click **Global Equations 1 (ODE1)**.
2. In the **Settings** window for **Global Equations**, locate the **Global Equations** section.

3 In the table, enter the following settings:

Name	f(u,ut,utt,t) (1)	Initial value (u_0) (1)	Initial value (u_t0) (1/s)	Description
`obj`	`objt*tmax-(tmax-Tperiod<t)*uAvg/U0`	`0`	`0`	

MESH 2

Free Triangular 1

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

Size

Change the settings to give a uniform mesh with an element size half of that used for the optimization.

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 size** text field, type `meshsz/2`.

5 In the **Maximum element growth rate** text field, type `Inf`.

6 In the **Curvature factor** text field, type `Inf`.

7 Click **Build All**.

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

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 **Empty Study**.

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

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

OPTIMIZATION

Step 1: Time Dependent

In the **Model Builder** window, under **Optimization** right-click **Step 1: Time Dependent** and choose **Copy**.

STUDY 2

In the **Model Builder** window, right-click **Study 2** and choose **Paste Time Dependent**.

1 In the **Model Builder** window, under **Study 2** click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Physics and Variables Selection** section.

3 In the table, enter the following settings:

Physics interface	Solve for	Equation form
Laminar Flow (spf)		Automatic (Stationary)
Global ODEs and DAEs (ge)		Automatic (Time domain)
Topology Optimization (Component 1)		Automatic

4 Click to expand the **Results While Solving** section. From the **Probes** list, choose **None**.

OPTIMIZATION

1 In the **Model Builder** window, under **Optimization** click **Step 1: Time Dependent**.

2 In the **Settings** window for **Time Dependent**, locate the **Physics and Variables Selection** section.

3 In the table, enter the following settings:

Physics interface	Solve for	Equation form
Laminar Flow (spf2)		Automatic (Stationary)
Global ODEs and DAEs (ge2)		Automatic (Time domain)

VERIFICATION

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

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

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

RESULTS

Objective, Pressure (spf), Velocity (spf)

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

Velocity (spf)

Drag and drop above **Topology Optimization>Output material volume factor**.

Topology Optimization 1

In the **Model Builder** window, under **Results** right-click **Topology Optimization 1** and choose **Delete**.

Objective (Verification)

1 In the **Model Builder** window, under **Results** click **1D Plot Group 8**.

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

Objective (Verification), Pressure (spf2), Velocity (spf2)

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Velocity (spf2)**, **Pressure (spf2)**, and **Objective (Verification)**.

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

Verification

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

Use an **Evaluation Group** to compare the objective in the two components.

Evaluation Group 1

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

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

3 From the **Time selection** list, choose **Last**.

Global Evaluation 1

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

2 Right-click **Global Evaluation 1** and choose **Duplicate**.

Global Evaluation 2

1 In the **Model Builder** window, click **Global Evaluation 2**.

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

3 From the **Dataset** list, choose **Verification/Solution 2 (4) (sol2)**.

4 From the **Time selection** list, choose **Last**.

5 In the **Evaluation Group 1** toolbar, click **Evaluate**.

Velocity (spf2)

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

2 In the **Settings** window for **2D Plot Group**, click **Plot Previous**.

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