Created in COMSOL Multiphysics 6.2

Optimizing Band Dispersion in an Electroosmotic Flow Through a Curved Microchannel

This model is licensed under the COMSOL Software License Agreement 6.2.
All trademarks are the property of their respective owners. See www.comsol.com/trademarks.

Introduction

This model uses the Optimization Module to minimize the turn induced dispersion of a chemical species in an electroosmotic flow through a curved channel (Ref. 1, Ref. 2, and Ref. 3). The dispersion is mainly caused by two factors: the difference in path length between the inner and outer boundaries of the curved channel, and the difference in electric field magnitude along these two boundaries. The curve induced dispersion may impede the ability to detect the species. The downstream concentration profile produced by the optimized geometry has a maximum concentration almost three times larger than that produced by the original geometry.

Model Definition

The original geometry of the device is shown in Figure 1. The width of the channel is 100 μm and the outer radius r_0 for the curved channel is 150 μm. The length L for the straight section of the microchannel is 1 mm. The chemical species band is introduced at $x = -600$ μm and modeled as the Gaussian pulse with a peak concentration of 1 mol/m^3.

The solute band is carried by electroosmosis through the curved geometry. The steady state electroosmotic flow is modeled by solving the Creeping-flow equations with the Helmholtz–Smoluchowski slip boundary condition applied at the channel walls:

$$u_{\mathrm{eo}} = -\frac{\varepsilon_0 \varepsilon_r \zeta}{\mu} E_t$$

where u_{eo} is the electroosmotic slip velocity, ε_0 is the permittivity of free space, ε_r is the relative permittivity of the solution, μ is the dynamic viscosity of the fluid, ζ is the zeta potential, and E_t is the tangential component of the electric field as the wall. The electric field required to estimate the slip velocity is computed by solving Laplace's equation.

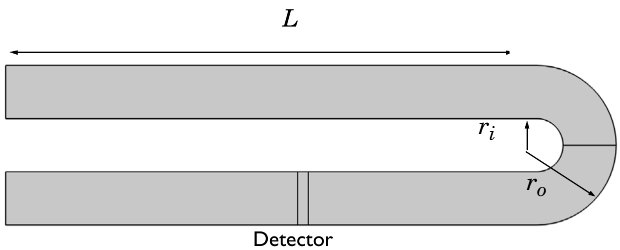


Figure 1: The initial model geometry is a constant width microchannel with a 180-degree turn.

The solute transport through the curved channel is modeled using the transient convection-diffusion equation. The diffusivity, D, of the species is taken as $1 \cdot 10^{-11}\ \mathrm{m}^2/\mathrm{s}$, which results in a Peclet number of roughly 6100, based on the slip velocity and width, W, of the channel ($\mathrm{Pe} = u_{\mathrm{eo}}W/D$).

The first study models the dispersion in the solute band through the constant radius geometry with a 180-degree turn. In the second study the Optimization Module is used to optimize the geometry for minimal solute band dispersion. The inner channel curve is represented by a Bézier curve (Ref. 4). Five parameters are chosen to optimize the geometry including the inner radii, control points for the Bézier curve and their corresponding weights. A more complicated geometric discretization could be achieved by employing multiple Bézier curves. The Bézier curve parameters are used as the optimization variables with the objective of minimizing the solute dispersion. The solute dispersion for the given Peclet number can be related to the difference in time taken by the solute molecules to traverse along inner and outer edges of microchannel. Hence the following objective function is utilized to arrive at the optimal design:

$$\text{Objective} = \min(\mathrm{abs}(t_{\mathrm{in}} - t_{\mathrm{out}}))$$

where, t_{in} and t_{out} are the times taken by the solute to move around the curve along the inner and outer walls respectively. Using this simple objective function means that only the steady state solution of the velocity field is required for the optimization step.

Results and Discussion

The velocity field magnitude (surface plot) and the electric potential (contours) are plotted for the original geometry in Figure 2. The equipotential lines are closer along the inner curve which results in a nonuniform electric field and velocity field in the curved section.

The solute band traverses inner edge more quickly and consequently there is substantial dispersion (see Figure 3).

The velocity magnitude is almost uniform across the cross section of the microchannel in the optimized geometry shown in Figure 4. Additionally the tapered section in the optimized geometry increases the path length along the inner microchannel edge. The concentration surface plot in Figure 5 shows the reduction in solute dispersion in the optimized geometry.

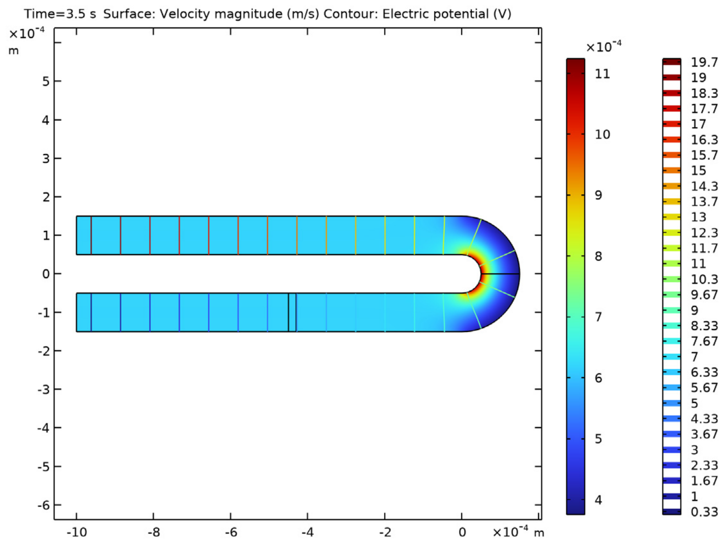


Figure 2: Velocity magnitude (surface) and electric potential (contour) for the original geometry.

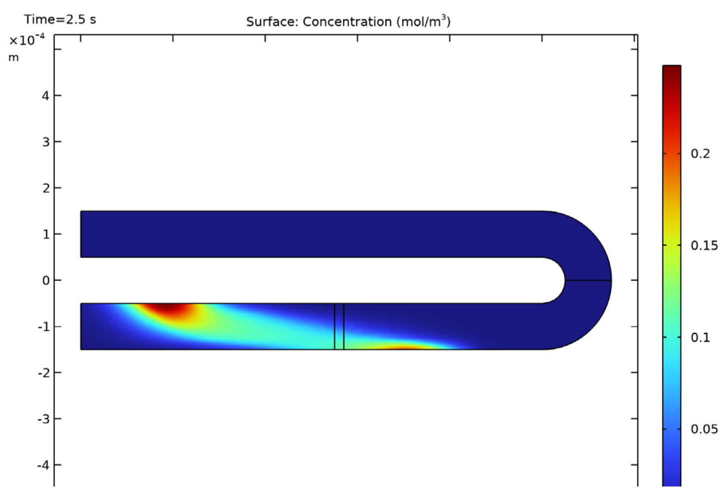


Figure 3: Concentration surface plot in the original geometry showing dispersion due to curved channel.

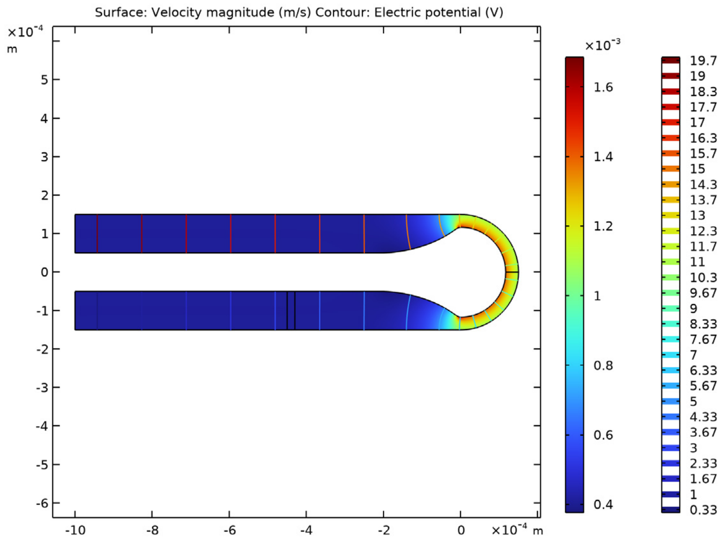


Figure 4: Velocity magnitude (surface plot) and electric potential (contour plot) in the optimized geometry.

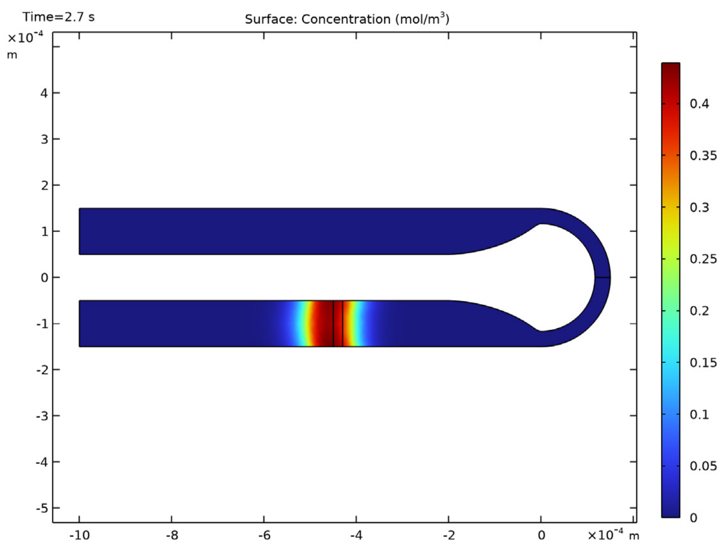


Figure 5: Concentration surface plot illustrating the optimized design for minimized dispersion.

Figure 6 compares the average concentration passing through the detector (domain 3) for original as well as optimized geometry. The obtained average concentration intensity peak is almost three times higher for the optimized geometry compared to the original geometry for the given Peclet number.

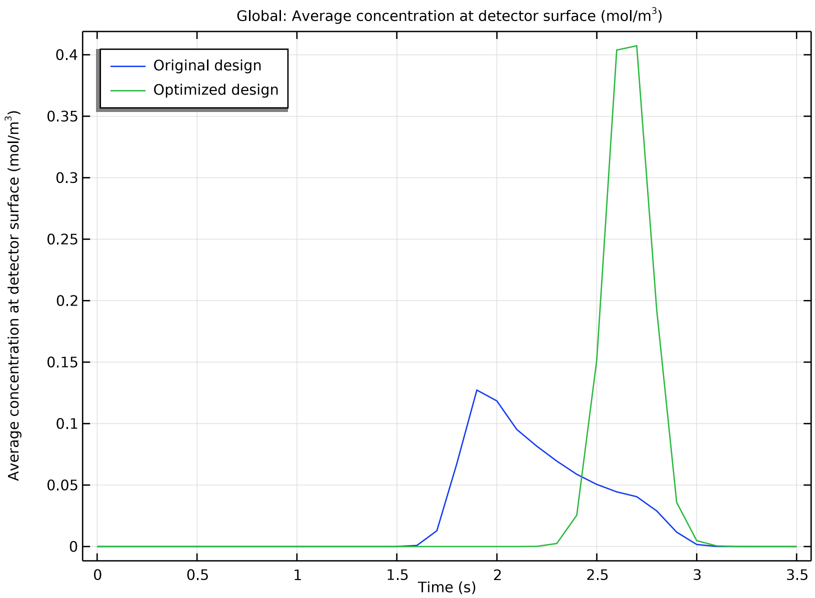


Figure 6: The average concentration through the detector for the original and optimized geometries.

References

1. S.K. Griffiths and R.H. Nilson, "Low-Dispersion Turns and Junctions for Microchannel Systems," *Analytical Chemistry*, vol. 73, no. 2, pp. 272–278, 2001.

2. S.K. Griffiths and R.H. Nilson, "Band Spreading in Two-Dimensional Microchannel Turns for Electrokinetic Species Transport," *Analytical Chemistry*, vol. 72, no. 21, pp. 5473–5482, 2000.

3. J.L. Molho, A.E. Herr, B.P. Mosier, J.G. Santiago, T.W. Kenny, R.A. Brennen, G.B. Gordon, and B. Mohammadi, "Optimization of Turn Geometries for Microchip Electrophoresis," *Analytical Chemistry*, vol. 73, no. 6, pp. 1350–1360, 2001.

4. M. Jain, A. Rao, and K. Nandakumar, "Study on Groove Shape Optimization for Micromixers", presented at COMSOL Conference 2012, Boston.

Application Library path: `Microfluidics_Module/Fluid_Flow/microchannel_dispersion_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 **AC/DC>Electric Fields and Currents>Electric Currents (ec)**.
3 Click **Add**.
4 In the **Select Physics** tree, select **Fluid Flow>Single-Phase Flow>Creeping Flow (spf)**.
5 Click **Add**.
6 In the **Select Physics** tree, select **Chemical Species Transport>Transport of Diluted Species (tds)**.
7 Click **Add**.
8 Click **Study**.
9 In the **Select Study** tree, select **General Studies>Stationary**.
10 Click **Done**.

GLOBAL DEFINITIONS

Geometry parameters

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 `microchannel_dispersion_optimization_geom_parameters.txt`.

5 In the **Label** text field, type `Geometry parameters`.

Model parameters

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

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 `microchannel_dispersion_optimization_model_parameters.txt`.

5 In the **Label** text field, type `Model parameters`.

GEOMETRY 1

Cubic Bézier 1 (cb1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Cubic Bézier**.

2 In the **Settings** window for **Cubic Bézier**, locate the **Control Points** section.

3 In row **1**, set **y** to `r2`.

4 In row **2**, set **x** to `kk*r2` and **y** to `r2`.

5 In row **3**, set **x** to `r2` and **y** to `kk*r2`.

6 In row **4**, set **x** to `r2`.

Line Segment 1 (ls1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

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

3 From the **Specify** list, choose **Coordinates**.

4 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.

5 Locate the **Starting Point** section. In the **x** text field, type `r2`.

6 Locate the **Endpoint** section. In the **x** text field, type `P1`.

Cubic Bézier 2 (cb2)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Cubic Bézier**.

2 In the **Settings** window for **Cubic Bézier**, locate the **Control Points** section.

3 In row **1**, set **y** to `P1`.

4 In row **2**, set **x** to `kk*P1` and **y** to `P1`.

5 In row **3**, set **x** to `P1` and **y** to `kk*P1`.

6 In row **4**, set **x** to `P1`.

Cubic Bézier 3 (cb3)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Cubic Bézier**.

2 In the **Settings** window for **Cubic Bézier**, locate the **Control Points** section.

3 In row **1**, set **y** to `P1`.

4 In row **2**, set **x** to `-20e-6` and **y** to `P1`.

5 In row **3**, set **x** to `-40e-6` and **y** to `P2`.

6 In row **4**, set **x** to `-RL` and **y** to `r1`.

7 Locate the **Weights** section. In the **1** text field, type `P3`.

8 In the **2** text field, type `P4`.

9 In the **3** text field, type `P5`.

Polygon 1 (pol1)

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

2 In the **Settings** window for **Polygon**, locate the **Object Type** section.

3 From the **Type** list, choose **Open curve**.

4 Locate the **Coordinates** section. From the **Data source** list, choose **Vectors**.

5 In the **x** text field, type `-RL -L -L 0`.

6 In the **y** text field, type `r1 r1 r2 r2`.

Convert to Curve 1 (ccur1)

1 In the **Geometry** toolbar, click **Conversions** and choose **Convert to Curve**.

2 Click in the **Graphics** window and then press Ctrl+A to select all objects.

Convert to Solid 1 (csol1)

1 In the **Geometry** toolbar, click **Conversions** and choose **Convert to Solid**.

2 Select the object **ccur1** only.

Mirror 1 (mir1)

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

2 Select the object **csol1** only.

3 In the **Settings** window for **Mirror**, locate the **Input** section.

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

5 Locate the **Normal Vector to Line of Reflection** section. In the **x** text field, type `0`.

6 In the **y** text field, type `1`.

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 `-IL`.

4 In the **y** text field, type `r1`.

Point 2 (pt2)

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 `-IL`.

4 In the **y** text field, type `r2`.

Point 3 (pt3)

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 `-IL`.

4 In the **y** text field, type `-r2`.

Point 4 (pt4)

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 `-IL`.

4 In the **y** text field, type `-r1`.

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 `2e-5`.

4 In the **Height** text field, type `r2-r1`.

5 Locate the **Position** section. In the **x** text field, type `-4.5e-4`.

6 In the **y** text field, type `-r2`.

Form Union (fin)

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

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

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**
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	`1e-3`	S/m	Basic
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	`er_w`	1	Basic
Density	rho	`rho0`	kg/m³	Basic
Dynamic viscosity	mu	`mu0`	Pa·s	Basic

ELECTRIC CURRENTS (EC)

Ground 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Electric Currents (ec)** and choose **Ground**.

2 Select Boundary 1 only.

Electric Potential 1

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

2 Select Boundary 4 only.

3 In the **Settings** window for **Electric Potential**, locate the **Electric Potential** section.

4 In the V_0 text field, type `V0`.

CREEPING FLOW (SPF)

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

Open Boundary 1

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

2 Select Boundaries 1 and 4 only.

Wall 1

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

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

3 From the **Wall condition** list, choose **Electroosmotic velocity**.

4 From the $\mathbf{E}$ list, choose **Tangential electric field (ec/cucn1)**.

5 From the **Electroosmotic mobility** list, choose **Built-in expression**.

6 In the ζ text field, type `zeta`.

7 In the ε_r text field, type `er_w`.

TRANSPORT OF DILUTED SPECIES (TDS)

Transport Properties 1

1 In the **Model Builder** window, under **Component 1 (comp1)> Transport of Diluted Species (tds)** click **Transport Properties 1**.

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

3 From the $\mathbf{u}$ list, choose **Velocity field (spf)**.

4 Locate the **Diffusion** section. In the D_c text field, type `D`.

Initial Values 1

1 In the **Model Builder** window, click **Initial Values 1**.

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 In the c text field, type `cini`.

Inflow 1

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

2 Select Boundary 4 only.

3 In the **Settings** window for **Inflow**, locate the **Concentration** section.

4 In the $c_{0,c}$ text field, type `cini`.

Outflow 1

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

2 Select Boundary 1 only.

DEFINITIONS

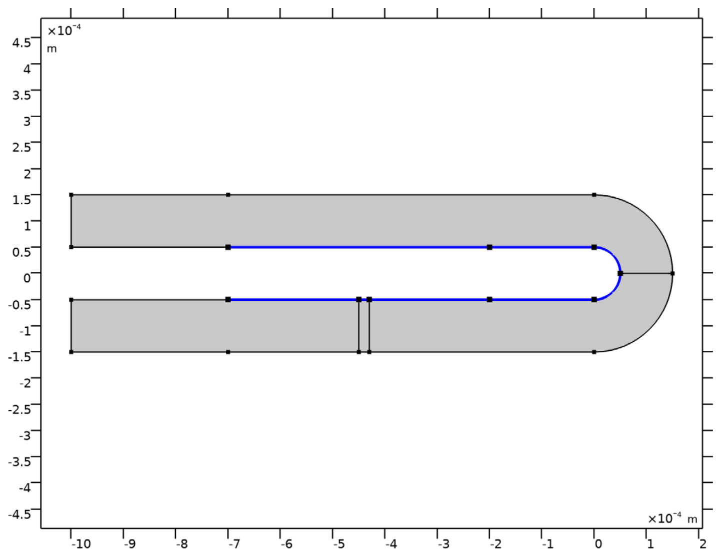
Inner Curve

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

2 In the **Settings** window for **Integration**, type `Inner Curve` in the **Label** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 8, 9, 13, 16, 18, 19, 21, and 22 only.



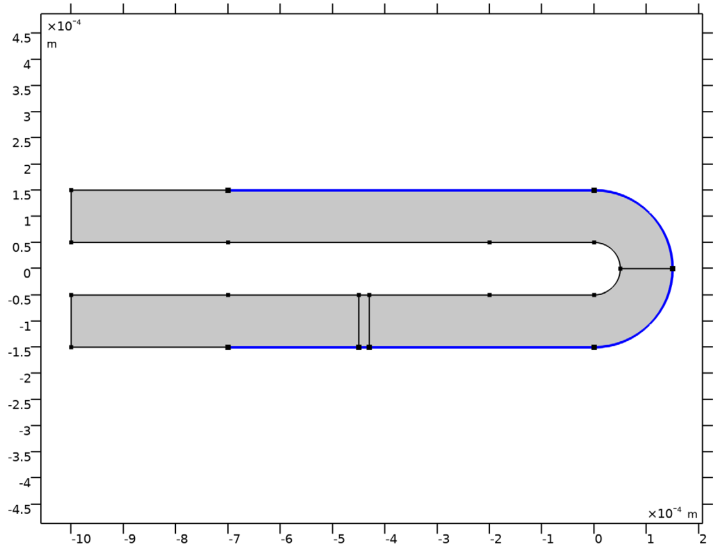
Outer Curve

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

2 In the **Settings** window for **Integration**, type `Outer Curve` in the **Label** text field.

3 Locate the **Source Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 7, 10, 12, 15, 20, and 23 only.



Detector Surface

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

2 Select Domain 3 only.

3 In the **Settings** window for **Average**, type `Detector Surface` in the **Label** text field.

Define a gaussian pulse for the initial value of the concentration.

Gaussian Pulse 1 (gp1)

1 In the **Definitions** toolbar, click f(X) **More Functions** and choose **Gaussian Pulse**.

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

3 In the **Location** text field, type `xm`.

4 In the **Standard deviation** text field, type `sigma`.

5 From the **Normalization** list, choose **Peak value**.

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
cini	c0*gp1(x[1/m])*(y>0)		Concentration Gaussian Pulse
t_in	intop1(1/spf.U)	s	Time taken along inner curve
t_out	intop2(1/spf.U)	s	Time taken along outer curve
c_avg	aveop1(c)	mol/m^3	Average concentration at detector surface

ORIGINAL CURVED CHANNEL STUDY

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

2 In the **Settings** window for **Study**, type Original Curved Channel Study in the **Label** text field.

Run the simulation of the flow in the original channel by solving for the electric field and the fluid flow first, followed by a time-dependent simulation of the transport of diluted species.

Step 1: Stationary

1 In the **Model Builder** window, under **Original Curved Channel Study** click **Step 1: Stationary**.

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

3 In the table, clear the **Solve for** check box for **Transport of Diluted Species (tds)**.

Step 2: Time Dependent

1 In the **Study** toolbar, click **Study Steps** and choose **Time Dependent>Time Dependent**.

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

3 Click **Range**.

4 In the **Range** dialog box, type 3.5 in the **Stop** text field.

5 Click **Replace**.

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

7 In the table, clear the **Solve for** check boxes for **Electric Currents (ec)** and **Creeping Flow (spf)**.

8 In the **Model Builder** window, click **Original Curved Channel Study**.

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

10 Clear the **Generate default plots** check box.

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

RESULTS

Velocity and Electric Potential (Original Channel)

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

2 In the **Settings** window for **2D Plot Group**, type `Velocity and Electric Potential (Original Channel)` in the **Label** text field.

Surface 1

1 Right-click **Velocity and Electric Potential (Original Channel)** and choose **Surface**.

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

3 In the **Expression** text field, type `spf.U`.

Contour 1

1 In the **Model Builder** window, right-click **Velocity and Electric Potential (Original Channel)** and choose **Contour**.

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

3 In the **Total levels** text field, type `30`.

4 In the **Velocity and Electric Potential (Original Channel)** toolbar, click **Plot**.

Concentration (Original Channel)

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

2 In the **Settings** window for **2D Plot Group**, type `Concentration (Original Channel)` in the **Label** text field.

3 Locate the **Data** section. From the **Time (s)** list, choose **2.5**.

Surface 1

1 Right-click **Concentration (Original Channel)** and choose **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)>Transport of Diluted Species>Species c>c - Concentration - mol/m³**.

3 In the **Concentration (Original Channel)** toolbar, click **Plot**.

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 **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Transport of Diluted Species (tds)**.
4. Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Stationary**.
5. Click **Add Study** in the window toolbar.
6. In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Step 1: Stationary

Minimize the difference in time needed for fluid to travel along the inner and outer edges of the channel.

1. In the **Model Builder** window, click **Study 2**.
2. In the **Settings** window for **Study**, type `Shape Optimization Study` in the **Label** text field.
3. Locate the **Study Settings** section. Clear the **Generate default plots** check box.

Optimization

1. In the **Study** toolbar, click **Optimization** and choose **Optimization**.
2. In the **Settings** window for **Optimization**, locate the **Optimization Solver** section.
3. From the **Method** list, choose **BOBYQA**.
4. Locate the **Objective Function** section. In the table, enter the following settings:

Expression	Description	Evaluate for
`abs(comp1.t_out-comp1.t_in)`		Stationary

5. Locate the **Control Variables and Parameters** section. Click **Add**.
6. In the table, enter the following settings:

Parameter name	Initial value	Scale	Lower bound	Upper bound
P1 (Optimization parameter 1)	`50[um]`	`50[um]`	`50[um]`	`130[um]`

7. Click **Add**.

8 In the table, enter the following settings:

Parameter name	**Initial value**	**Scale**	**Lower bound**	**Upper bound**
P2 (Optimization parameter 2)	`50[um]`	`50[um]`	`50[um]`	`130[um]`

9 Click **Add**.

10 In the table, enter the following settings:

Parameter name	**Initial value**	**Scale**	**Lower bound**	**Upper bound**
P3 (Optimization parameter 3)	`0.5`	`0.5`	`0.01`	`1`

11 Click **Add**.

12 In the table, enter the following settings:

Parameter name	**Initial value**	**Scale**	**Lower bound**	**Upper bound**
P4 (Optimization parameter 4)	`0.5`	`0.5`	`0.01`	`1`

13 Click **Add**.

14 In the table, enter the following settings:

Parameter name	**Initial value**	**Scale**	**Lower bound**	**Upper bound**
P5 (Optimization parameter 5)	`0.5`	`0.5`	`0.01`	`1`

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

RESULTS

Velocity and Electric Potential (Optimized Channel)

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

2 In the **Settings** window for **2D Plot Group**, type `Velocity and Electric Potential (Optimized Channel)` in the **Label** text field.

3 Locate the **Plot Settings** section. From the **Frame** list, choose **Spatial (x, y, z)**.

4 Locate the **Data** section. From the **Dataset** list, choose **Shape Optimization Study/ Solution 3 (sol3)**.

Surface 1

1 Right-click **Velocity and Electric Potential (Optimized Channel)** and choose **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)>Creeping Flow>Velocity and pressure>spf.U - Velocity magnitude - m/s**.

Contour 1

1 In the **Model Builder** window, right-click **Velocity and Electric Potential (Optimized Channel)** and choose **Contour**.

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

3 In the **Total levels** text field, type `30`.

4 In the **Velocity and Electric Potential (Optimized Channel)** toolbar, click **Plot**.

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 **Physics interfaces in study** subsection. In the table, clear the **Solve** check boxes for **Electric Currents (ec)** and **Creeping Flow (spf)**.

4 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Time Dependent**.

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

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

OPTIMIZED CHANNEL VERIFICATION

1 In the **Model Builder** window, click **Study 3**.

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

Step 1: Time Dependent

1 In the **Model Builder** window, under **Optimized Channel Verification** 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(0,0.1,3.5)`.

4 From the **Tolerance** list, choose **User controlled**.

5 In the **Relative tolerance** text field, type `0.001`.

6 Click to expand the **Values of Dependent Variables** section. Find the **Initial values of variables solved for** subsection. From the **Settings** list, choose **User controlled**.

7 From the **Method** list, choose **Solution**.

8 From the **Study** list, choose **Shape Optimization Study, Stationary**.

9 Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

10 From the **Method** list, choose **Solution**.

11 From the **Study** list, choose **Shape Optimization Study, Stationary**.

12 In the **Model Builder** window, click **Optimized Channel Verification**.

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

14 Clear the **Generate default plots** check box.

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

RESULTS

Concentration (Optimized Channel)

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

2 In the **Settings** window for **2D Plot Group**, type `Concentration (Optimized Channel)` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Optimized Channel Verification/ Solution 6 (sol6)**.

4 From the **Time (s)** list, choose **2.7**.

Surface 1

1 Right-click **Concentration (Optimized Channel)** and choose **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)> Transport of Diluted Species>Species c>c - Concentration - mol/m³**.

3 In the **Concentration (Optimized Channel)** toolbar, click **Plot**.

Plot the average concentration in the detector for the original and optimized channels.

Average Concentration in Detector

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

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

Global 1

1 Right-click **Average Concentration in Detector** and choose **Global**.

2 In the **Settings** window for **Global**, locate the **y-Axis Data** section.

3 In the table, enter the following settings:

Expression	Unit	Description
c_avg	mol/m^3	Average concentration at detector surface

4 Click to expand the **Legends** section. From the **Legends** list, choose **Manual**.

5 In the table, enter the following settings:

Legends
Original design

6 Right-click **Global 1** and choose **Duplicate**.

Global 2

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

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

3 From the **Dataset** list, choose **Optimized Channel Verification/Solution 6 (sol6)**.

4 Click to expand the **Title** section. From the **Title type** list, choose **None**.

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

Legends
Optimized design

Average Concentration in Detector

1 In the **Model Builder** window, click **Average Concentration in Detector**.

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

3 From the **Position** list, choose **Upper left**.

Compare the original and optimized channels using animation.

Animation 1

1 In the **Results** toolbar, click **Animation** and choose **File**.

2 In the **Settings** window for **Animation**, locate the **Target** section.

3 From the **Target** list, choose **Player**.

4 Locate the **Scene** section. From the **Subject** list, choose **Concentration (Original Channel)**.

5 Click the **Play** button in the **Graphics** toolbar.

Animation 2

1 In the **Results** toolbar, click **Animation** and choose **File**.

2 In the **Settings** window for **Animation**, locate the **Target** section.

3 From the **Target** list, choose **Player**.

4 Locate the **Scene** section. From the **Subject** list, choose **Concentration (Optimized Channel)**.

5 Click the **Play** button in the **Graphics** toolbar.