Created in COMSOL Multiphysics 6.2

Shape 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 up to 100. This model sets up a shape optimization model with inspiration from the model Optimization of a Tesla Microvalve, which finds a design for a Tesla valve using topology optimization. The optimization problem tends to favor thin walls, and therefore the model uses interior walls. The result of the optimization is verified by remeshing in the deformed configuration and varying the Reynolds number.

Model Definition

The physics of this model is identical to that of Optimization of a Tesla Microvalve, so the main difference is in the setup of the parameterized geometry. The result of the topology optimization is shown in Figure 1, while the initial geometry for this model is shown in Figure 2.

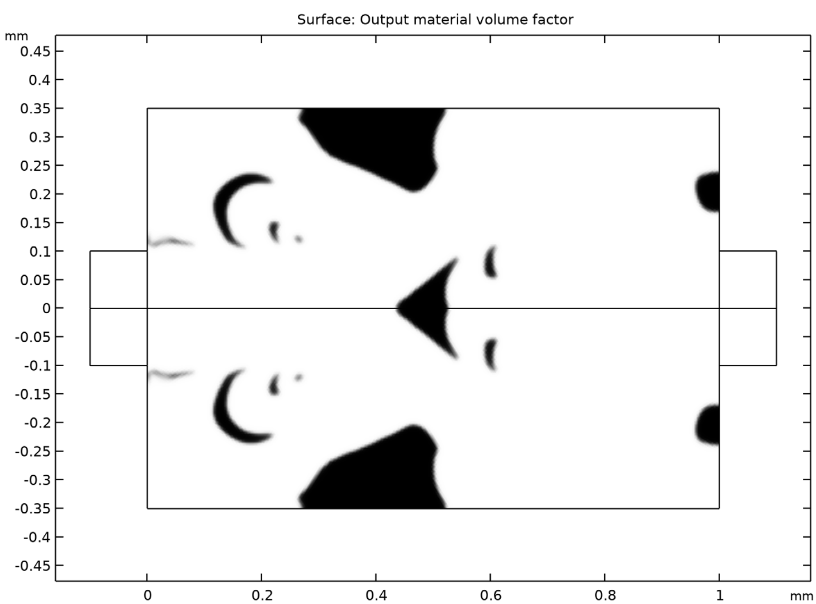


Figure 1: The topology optimized design computed in the model Optimization of a Tesla Microvalve.

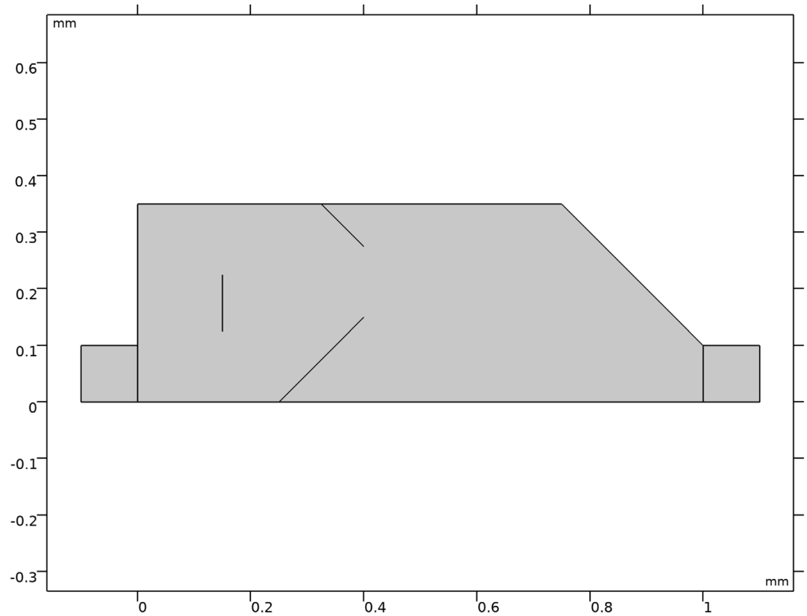


Figure 2: The initial geometry used for this model uses interior walls to prevent inverted elements and topological changes.

The model uses 2nd order Bernstein polynomials to deform the geometry. This functionality is available via the Free Shape Domain and Polynomial Boundary features. Reducing the Maximum displacement is guaranteed to reduce the tendency for inverted elements or clashing of boundaries. Three lines will be deformed and two of them will be allowed to slide along Roller boundaries. The flow rates are fixed, so it is the pressure drop which is taken as the objective function. The optimization uses the adjoint method for the sensitivity analysis and therefore the model can be solved in approximately 10 minutes.

Results and Discussion

The optimization leads to significant distortion of the mesh as shown in Figure 3. Surprisingly, it is actually possible to improve on the result of topology optimization with shape optimization. This can be attributed to the fact that the optimization problem favors thin walls, which are trivial to realize with interior walls in shape optimization, but in topology optimization they require high damping and small elements.

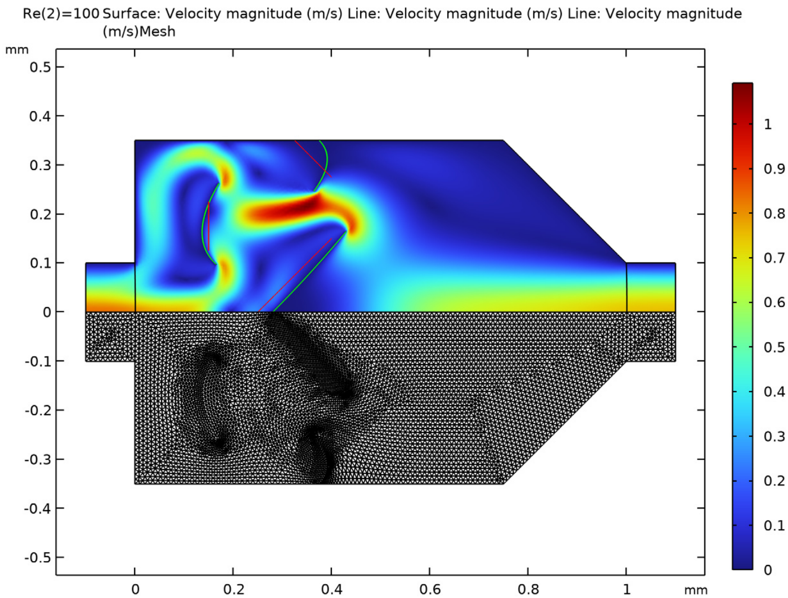


Figure 3: The result of the optimization is shown in terms of the mesh and the backward flow velocity. Significant distortion of the mesh can be observed in the areas that are close to boundaries with larger deformations. The red lines indicate the initial geometry, while the green lines represent the optimal geometry.

The design is verified by remeshing in the deformed frame and varying the Reynolds number as plotted in Figure 4.

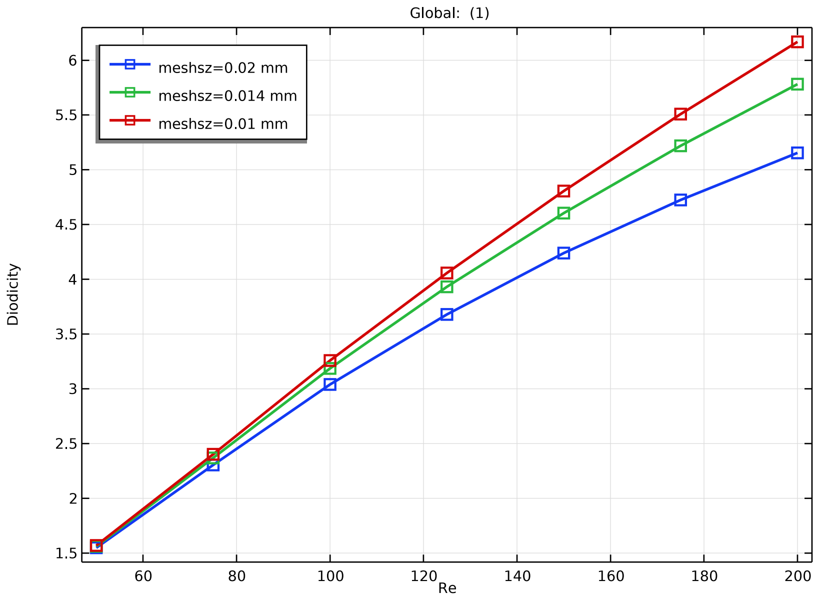


Figure 4: The pressure drop ratio is plotted as a function of the Reynolds number for three different mesh sizes.

Reference

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

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: `Microfluidics_Module/Fluid_Flow/tesla_microvalve_shape_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 Click **Add**.

5 Click **Study**.

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

7 Click **Done**.

GEOMETRY 1

Create the geometry. To simplify this step, insert a prepared geometry sequence.

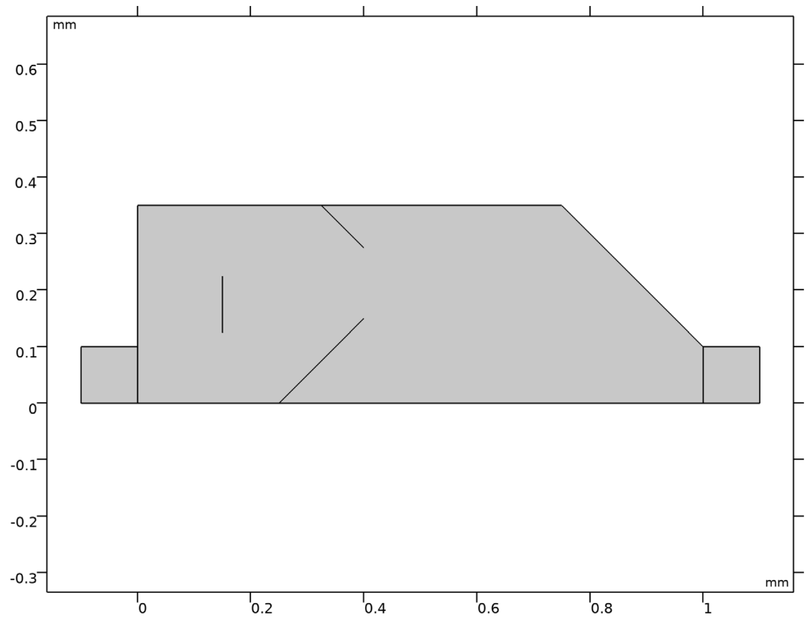
1 In the **Geometry** toolbar, click **Insert Sequence** and choose **Insert Sequence**.

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

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

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

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



The geometry should now look like that in Figure 1. Note that the inserted geometry is parameterized and that the parameters used are automatically added to the list of global parameters in the model.

6 In the **Model Builder** window, collapse the **Geometry 1** node.

GLOBAL DEFINITIONS

Geometrical Parameters

Add a new parameter group for calculating the average inlet velocity as a function of the Reynolds number.

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

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

Parameters 2

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

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
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	0.01[mm]	1E-5 m	Mesh size

MATERIALS

Add a blank material for rho0 and mu0.

Material 1 (mat1)

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

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

3 In the table, enter the following settings:

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

LAMINAR FLOW (SPF)

Set up the boundary conditions for the two flow directions.

Interior Wall 1

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

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

3 From the **Selection** list, choose **Interior Walls**.

Symmetry 1

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

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

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

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 **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**.

LAMINAR FLOW 2 (SPF2)

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

Interior Wall 1

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

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

3 From the **Selection** list, choose **Interior Walls**.

Symmetry 1

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

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

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

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**.

MESH 1

Define the mesh using the `meshsz` parameter.

Size 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 1** and choose **Size**.
2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
3 From the **Geometric entity level** list, choose **Boundary**.
4 From the **Selection** list, choose **Interior Walls**.
5 Locate the **Element Size** section. From the **Predefined** list, choose **Extremely fine**.
6 Click the **Custom** button.
7 Locate the **Element Size Parameters** section.
8 Select the **Maximum element size** check box. In the associated text field, type `meshsz/2`.

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**, click to expand the **Element Size Parameters** section.
3 Locate the **Element Size** section. From the **Predefined** list, choose **Extremely fine**.
4 Locate the **Element Size Parameters** section. In the **Maximum element size** text field, type `meshsz`.
5 In the **Minimum element size** text field, type `meshsz/2`.
6 In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**.

DEFINITIONS

Define the objective function as a variable using average operators.

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**.

Variables 1

1 In the **Model Builder** window, right-click **Definitions** and choose **Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
Di	(aveop2(p2)-aveop1(p2))/(aveop1(p)-aveop2(p))		Diodicity

INITIAL DESIGN

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

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

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

RESULTS

Pressure (spf), Pressure (spf2), Velocity (spf), Velocity (spf2)

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

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

Initial Design

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

Velocity (spf2)

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

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

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

Evaluate the objective function for the initial design.

Global Evaluation 1

1 In the **Results** toolbar, click **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
Di	1	

4 Click **Evaluate**.

DEFINITIONS

Define the shape optimization problem using **Polynomial Boundary** features.

COMPONENT 1 (COMP1)

Free Shape Domain 1

In the **Physics** toolbar, click **Optimization** and choose **Shape Optimization**.

Polynomial Boundary 1

1 In the **Shape Optimization** toolbar, click **Polynomial Boundary**.

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

3 From the **Selection** list, choose **Interior Walls**.

4 Locate the **Control Variable Settings** section. In the text field, type `L3/2`.

Symmetry/Roller 1

1 In the **Shape Optimization** toolbar, click **Symmetry/Roller**.

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

3 From the **Selection** list, choose **Top and Bottom**.

Add a second Study to perform the optimization. Initialize the solution, so that a plot of the deformed geometry and mesh can be created.

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

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

OPTIMIZATION

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

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

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

RESULTS

Pressure (spf) 1, Pressure (spf2) 1, Velocity (spf) 1, Velocity (spf2) 1

1. In the **Model Builder** window, under **Results**, Ctrl-click to select **Velocity (spf) 1**, **Pressure (spf) 1**, **Velocity (spf2) 1**, and **Pressure (spf2) 1**.
2. Right-click and choose **Group**.

Optimized

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

Line 1

1. In the **Model Builder** window, right-click **Velocity (spf2) 1** and choose **Line**.
2. In the **Settings** window for **Line**, locate the **Coloring and Style** section.
3. From the **Coloring** list, choose **Uniform**.
4. From the **Color** list, choose **Green**.

Selection 1

1. Right-click **Line 1** and choose **Selection**.
2. In the **Settings** window for **Selection**, locate the **Selection** section.
3. From the **Selection** list, choose **Interior Walls**.

Line 2

1. In the **Model Builder** window, right-click **Velocity (spf2) 1** and choose **Line**.
2. In the **Settings** window for **Line**, locate the **Data** section.
3. From the **Dataset** list, choose **Initial Design/Solution 1 (sol1)**.
4. Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

Selection 1

1. Right-click **Line 2** and choose **Selection**.
2. In the **Settings** window for **Selection**, locate the **Selection** section.
3. From the **Selection** list, choose **Interior Walls**.

Mesh 1

1. In the **Model Builder** window, right-click **Velocity (spf2) 1** and choose **Mesh**.
2. In the **Settings** window for **Mesh**, locate the **Coloring and Style** section.
3. From the **Element color** list, choose **None**.

Deformation 1

1. Right-click **Mesh 1** and choose **Deformation**.

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

3 In the **x-component** text field, type `0`.

4 In the **y-component** text field, type `-2*y`.

5 Locate the **Scale** section.

6 Select the **Scale factor** check box. In the associated text field, type `1`.

OPTIMIZATION

Add a **Shape Optimization** study step and apply a continuation in the Reynolds number to increase stability. Use an `if` statement to avoid contributions to the objective function from the intermediate Reynolds number.

Shape Optimization

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

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

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

4 In the **Maximum number of iterations** text field, type `20`.

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

Modify the expression, so that the objective function becomes zero for the first Reynolds number.

6 Locate the **Objective Function** section. In the table, enter the following settings:

Expression	Description
`if(Re==100,comp1.Di,0)`	`Diodicity`

7 From the **Type** list, choose **Maximization**.

8 Locate the **Output While Solving** section. Select the **Plot** check box.

9 From the **Plot group** list, choose **Velocity (spf2) 1**.

Step 1: Stationary

Use continuation in the Reynolds number to improve the robustness of the nonlinear solver.

1 In the **Model Builder** window, click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.

3 Select the **Auxiliary sweep** check box.

4 Click Add.

Perform a verification analysis by remeshing the deformed geometry and verifying the results for various meshes and Reynolds numbers.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
Re (Reynolds number)	50 100	

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

RESULTS

Global Evaluation 1

1 In the **Model Builder** window, under **Results>Derived Values** click **Global Evaluation 1**.

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

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

4 From the **Parameter selection (Re)** list, choose **Last**.

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

Optimization/Solution 2 (sol2)

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

2 Right-click **Results>Datasets>Optimization/Solution 2 (sol2)** and choose **Remesh Deformed Configuration**.

ADD STUDY

1 In the **Study** 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 Click **Add Study** in the window toolbar.

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

STUDY 3

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 box for **Deformed geometry (Component 1)**.

3 Click to expand the **Values of Dependent Variables** section. Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.

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

5 From the **Study** list, choose **Optimization, Stationary**.

6 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

7 Click **Add**.

8 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
Re (Reynolds number)	`range(50,25,200)`	

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

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

11 Locate the **Study Settings** section. Clear the **Generate default plots** check box.

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
meshsz (Mesh size)	`0.02 0.014 0.01`	mm

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

Copy the plot from the second study.

RESULTS

Velocity (spf2) 1

In the **Model Builder** window, under **Results>Optimized** right-click **Velocity (spf2) 1** and choose **Duplicate**.

Verification

1 In the **Model Builder** window, under **Results>Optimized** click **Velocity (spf2) 1.1**.

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

3 Locate the **Data** section. From the **Dataset** list, choose **Verification/ Parametric Solutions 1 (sol4)**.

4 From the **Parameter value (Re)** list, choose **100**.

5 In the **Verification** toolbar, click **Plot**.

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

7 Move out the **Verification** plot out of the group.

Global Evaluation 1

Evaluate the diodicity and plot it versus the Reynolds number.

1 In the **Model Builder** window, under **Results>Derived Values** click **Global Evaluation 1**.

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

3 From the **Dataset** list, choose **Verification/Parametric Solutions 1 (sol4)**.

4 From the **Parameter selection (meshsz)** list, choose **Last**.

5 From the **Parameter selection (Re)** list, choose **Manual**.

6 In the **Parameter indices (1-7)** text field, type `3`.

7 Click next to **Evaluate**, then choose **New Table**.

Diodicity vs. Re

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

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

3 Locate the **Plot Settings** section.

4 Select the **y-axis label** check box. In the associated text field, type `Diodicity`.

5 Locate the **Legend** section. From the **Position** list, choose **Upper left**.

Global 1

1 Right-click **Diodicity vs. Re** and choose **Global**.

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

3 From the **Dataset** list, choose **Verification/Parametric Solutions 1 (sol4)**.

4 Locate the **y-Axis Data** section. In the table, enter the following settings:

Expression	Unit	Description
Di	1	

5 Locate the **x-Axis Data** section. From the **Axis source data** list, choose **Re**.

6 In the **Diodicity vs. Re** toolbar, click **Plot**.

7 Click to expand the **Coloring and Style** section. From the **Width** list, choose **2**.

8 Find the **Line markers** subsection. From the **Marker** list, choose **Square**.

9 In the **Diodicity vs. Re** toolbar, click **Plot**.

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

Arrow Line 1

1 In the **Model Builder** window, expand the **Results>Shape Optimization** node, then click **Arrow Line 1**.

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

3 In the **Number of arrows** text field, type 50.

4 In the **Shape Optimization** toolbar, click **Plot**.

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

Geometry Modeling Instructions

If you want to create the geometry yourself, follow these steps.

ADD COMPONENT

In the **Home** toolbar, click **Add Component** and choose **2D**.

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
D	0.2[mm]	2E-4 m	Characteristic length
L	5*D	0.001 m	Channel length
H	1.75*D	3.5E-4 m	Channel width
LT	150[µm]	1.5E-4 m	First line x and y range
XT	250[µm]	2.5E-4 m	First line position
LT2	LT/2	7.5E-5 m	Second line x and y range
XT2	XT+LT2	3.25E-4 m	Second line position
X3	150[µm]	1.5E-4 m	Third line x position

Name	Expression	Value	Description
Y3	H/2	1.75E-4 m	Third line y position
L3	100[µm]	1E-4 m	Third line length
phi1	-pi/2 [rad]	-1.5708 rad	Third line orientation

GEOMETRY 1

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

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

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

Rectangle 1 (r1)

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

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

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

Rectangle 3 (r3)

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

2 In the **Settings** window for **Rectangle**, click **Build Selected**.

3 Locate the **Size and Shape** section. 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`.

Polygon 1 (pol1)

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

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

3 In the table, enter the following settings:

x (mm)	y (mm)
L-H+D/2	H
L	D/2
L	H

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 In the **x** text field, type `XT`.

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

6 In the **x** text field, type `XT+LT`.

7 In the **y** text field, type `LT`.

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

Line Segment 2 (ls2)

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 In the **x** text field, type `XT2`.

5 In the **y** text field, type `H`.

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

7 In the **x** text field, type `XT2+LT2`.

8 In the **y** text field, type `H-LT2`.

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

Line Segment 3 (ls3)

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 In the **x** text field, type `X1+L1*cos(phi1)/2`.

5 In the **y** text field, type `Y3+L3*sin(phi1)/2`.

6 In the **x** text field, type `X3+L3*cos(phi1)/2`.

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

8 In the **x** text field, type `X3-L3*cos(phi1)/2`.

9 In the **y** text field, type `Y3-L3*sin(phi1)/2`.

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

Difference 1 (dif1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **r1** only.

3 In the **Settings** window for **Difference**, locate the **Difference** section.

4 Click to select the **Activate Selection** toggle button for **Objects to subtract**.

5 Select the object **pol1** only.

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**.

Top

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

2 In the **Settings** window for **Box Selection**, type `Top` 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 minimum** text field, type `H*0.999`.

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

Interior Walls

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

2 In the **Settings** window for **Union Selection**, type `Interior Walls` 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 **Line Segment 1**, **Line Segment 2**, and **Line Segment 3**.

6 Click **OK**.

Top and Bottom

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

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

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

4 In the **Label** text field, type `Top and Bottom`.

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

6 In the **Add** dialog box, in the **Selections to add** list, choose **Symmetry** and **Top**.

7 Click **OK**.

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

The model geometry is now complete.