Created in COMSOL Multiphysics 6.1

Acoustic Microfluidic Pump

Introduction

Microfuidic devices are becoming increasingly popular in the field of biomedical research and diagnostics, especially for preparing and analyzing fluid samples. To create functional microfluidic systems, pumps, valves, mixers, and so on, are needed also in macrofluidic systems. This model investigates an acoustically driven microfluidic pump which can drive a flow in a closed microfluidic system. The model geometry is inspired by, but not identical to, the one investigated experimentally and numerically by Huang *et al.* in Ref. 1.

The acoustic microfluidic pump is driven by the nonlinear acoustic phenomenon known as acoustic streaming. Acoustic streaming is a steady fluid flow created by an acoustic field due to nonlinear terms in the Navier–Stokes equations. In a mathematical sense, where the acoustic field is the linearized first-order field of the Navier–Stokes equations, the acoustic streaming field is the time-averaged second-order field in the perturbation scheme.

In the acoustic microfluidic pump, the streaming is driven by nonlinear contributions arising close to sharp edge structures in the microfluidic channel.

Model Definition

The model is of a 2D acoustic microfluidic pump consisting of a water-filled microfluidic channel of width W_c = 0.6 mm and the channel is closed. One section of the channel includes an array of sharp edge structures on the boundary, as seen in Figure 1. It is assumed that the surrounding solid is acoustically hard and therefore only the fluid domain is modeled.

The acoustic streaming is induced by an acoustic body force that is induced due to nonlinearities in Navier-Stokes equation; see Ref. 2. The acoustic body force, $\mathbf{f}_{\mathrm{aco}}$, is given as

$$\mathbf{f}_{\mathrm{aco}} = -\nabla \cdot \langle \rho_0 \mathbf{v}_{\mathrm{aco}} \mathbf{v}_{\mathrm{aco}} \rangle \tag{1}$$

The angle brackets represent the time-average of the oscillating acoustic fields. To model the thin viscous and thermal boundary layers and the acoustic body force correctly, it is necessary to use a fine mesh around the sharp edge structures. The sharp edge structures has a thickness of 0.2 μm at the tip. In the region around the sharp edge structure, a full thermoviscous acoustic model is used, and in the large domain a pressure acoustic model is used to limit the computational requirements. It is necessary to use the **Thermoviscous Acoustics, Frequency Domain** interface around the sharp edge structures because the geometry features are of a length scale comparable to the viscous and thermal boundary layers in the fluid.

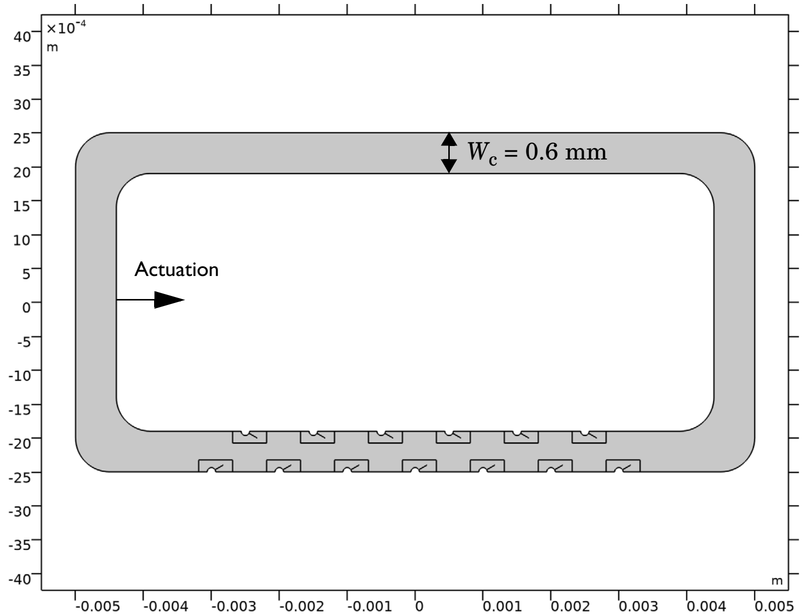


Figure 1: Geometry of the microfluidic pump.

The acoustic domain force is computed and applied to the **Laminar Flow** interface by the **Acoustic Streaming Domain Coupling** multiphysics feature that couples the **Thermoviscous Acoustics, Frequency Domain** interface with the **Laminar Flow** interface. The complementary **Acoustic Streaming Boundary Coupling**, available for both **Pressure Acoustics, Frequency Domain** and **Thermoviscous Acoustics, Frequency Domain**, is not used since its contributions are negligible in this setup (it can be included but will be dominated by the other feature). The boundary coupling from thermoviscous acoustics is only important if the boundary vibrates. Thus, if the vibrations of the thin flaps were modeled, the **Acoustic Streaming Boundary Coupling** should be included for modeling the resulting steady fluid flow.

Results and Discussion

The acoustic field is modeled as a **Pressure Acoustics, Frequency Domain** in the big domain with a thermoviscous fluid model (viscous bulk losses), and with a **Thermoviscous Acoustics, Frequency Domain** in the 13 small domains surrounding the sharp edge structures.

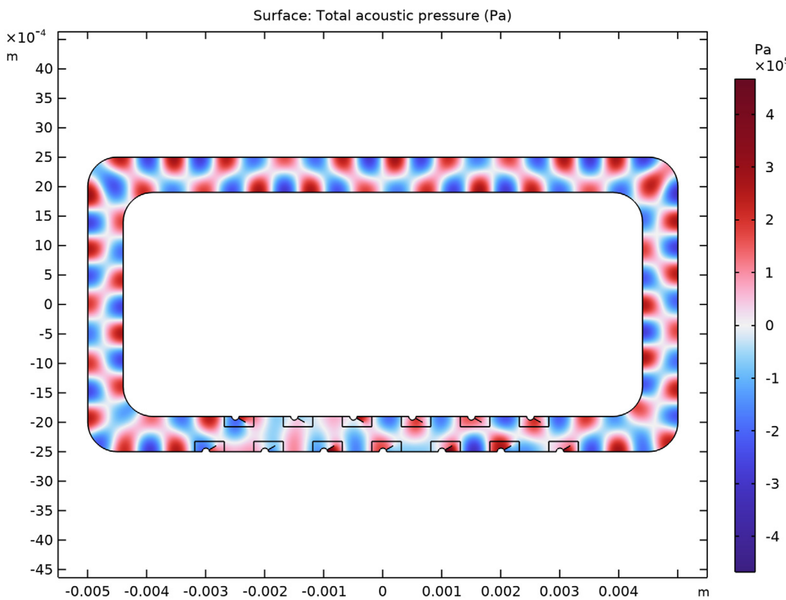


Figure 2: Acoustic pressure field in the microfluidic channel.

The acoustic field is actuated on the four vertical boundaries in the large pressure acoustic domain; see Figure 1. To get the correct pressure levels of the standing acoustic field, the dissipation in the thermal and viscous boundary layers in the pressure acoustics domain is included by using the boundary condition **Thermoviscous Boundary Layer Impedance** on the boundaries.

In Figure 2, the pressure field for an actuation at 2 MHz and actuation displacement of 1 nm is shown. The steady acoustic-streaming flow is solved in a stationary study step using the solved acoustic fields. The streaming is induced by an acoustic body force $\mathbf{f}_{\mathrm{aco}}$ (see Equation 1) near the sharp edge structures (added by the **Acoustic Streaming Domain Coupling feature**). The acoustic body force around one of the sharp edge structures is shown in Figure 3. Note that the acoustic body force is located at the tip of the thin edge structure.

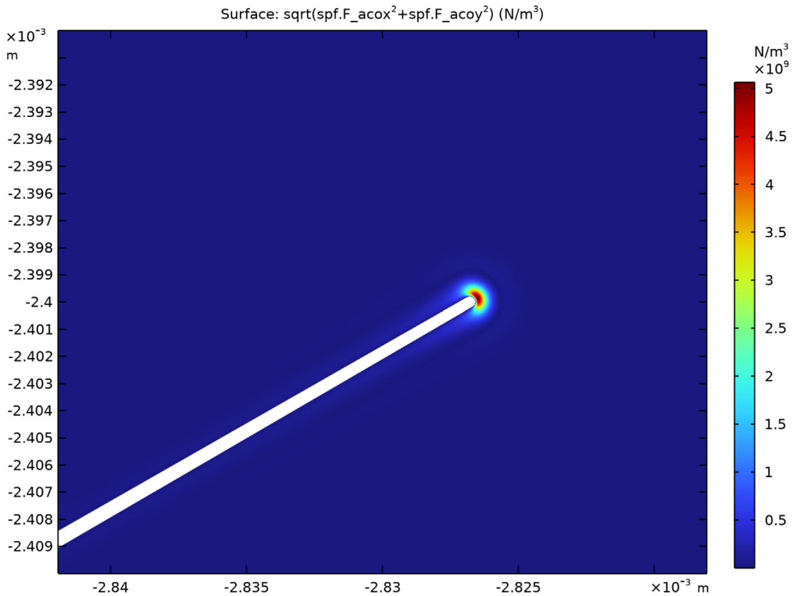


Figure 3: Acoustic body force generated at the tip of the sharp edge structure.

The acoustic body force at each thin edge structure creates a complicated fluid flow at the bottom half of the microfluidic channel. But because all the thin edge structures have the same direction, a nonzero flow field is created in the entire channel. The resulting fluid flow is shown on a logarithmic scale in Figure 4. The acoustic microfluidic pump creates a counterclockwise fluid flow. The logarithmic scale is chosen to capture both the complicated and fast flow around the thin edge structures and the fluid flow in the rest of the microfluidic channel.

The example demonstrates how an acoustic field can induce a steady fluid flow in closed microfluidic systems, also demonstrated experimentally by the research group of Tony J. Huang in Ref. 1.

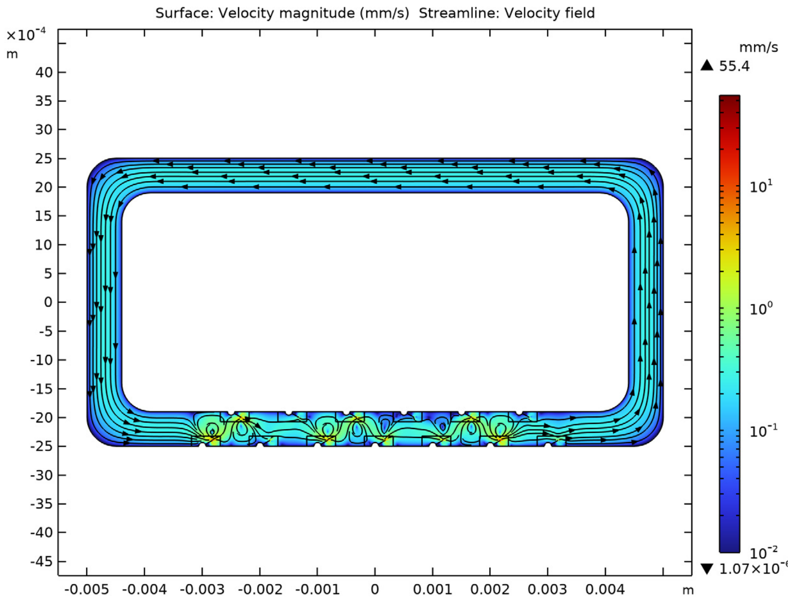


Figure 4: Steady streaming flow in the microfludic channel driven by the acoustic source terms.

Notes About the COMSOL Implementation

The force and stresses implemented on the fluid domain depend on the derivatives of the acoustic fields. It is therefore recommended to increase the element order of the dependent variables of the Thermoviscous Acoustics, Frequency Domain interface to quadratic serendipity for the acoustic pressure and cubic serendipity for the acoustic velocity and temperature.

References

1. P.-H. Huang and others, “A reliable and programmable acoustofluidic pump powered by oscillating sharp edge structures,” *Lab Chip*, vol. 14, pp. 4319–4323, 2014.

2. P.B. Muller and H. Bruus, “Numerical study of thermoviscous effects in ultrasound-induced acoustic streaming in microchannels,” *Phys. Rev. E*, vol. 90, p. 043016, 2014.

Application Library path: `Acoustics_Module/Nonlinear_Acoustics/acoustic_microfluidic_pump`

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 **Acoustics>Pressure Acoustics>Pressure Acoustics, Frequency Domain (acpr)**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Acoustics>Thermoviscous Acoustics>Thermoviscous Acoustics, Frequency Domain (ta)**.

5 Click **Add**.

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

7 Click **Add**.

8 Click **Study**.

9 In the **Select Study** tree, select **Preset Studies for Some Physics Interfaces>Frequency Domain**.

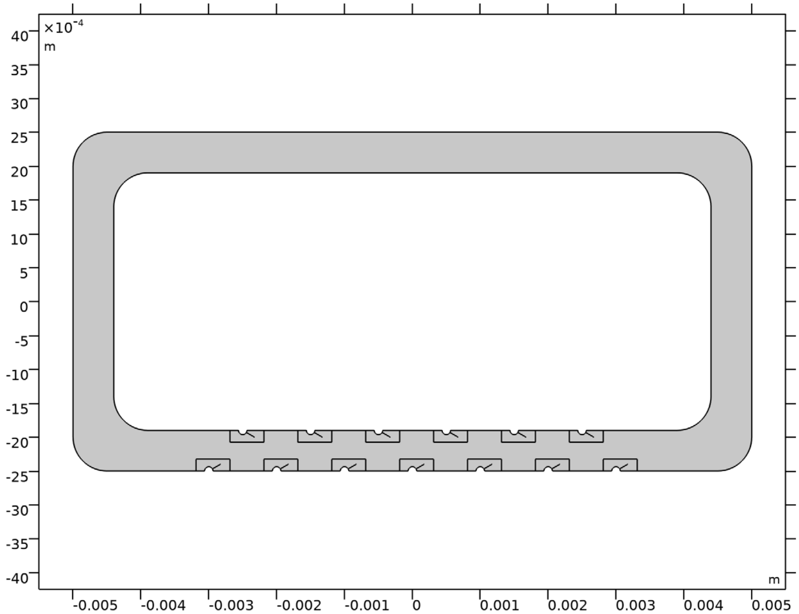
10 Click **Done**.

GEOMETRY 1

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 `acoustic_microfluidic_pump_geom_sequence.mph`.

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



Having imported the geometry, it can easily be modified as it is parameterized. Simply change the value of a dimension in the parameters list; this will update the geometry automatically.

GLOBAL DEFINITIONS

Parameters - Geometry

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

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

Parameters - Model

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

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

3 Locate the **Parameters** section. Click **Load from File**.

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

ADD MATERIAL

1 In the **Home** toolbar, click **Add Material** to open the **Add Material** window.
2 Go to the **Add Material** window.
3 In the tree, select **Built-in>Water, liquid**.
4 Click **Add to Component** in the window toolbar.
5 In the **Home** toolbar, click **Add Material** to close the **Add Material** window.

PRESSURE ACOUSTICS, FREQUENCY DOMAIN (ACPR)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Pressure Acoustics, Frequency Domain (acpr)**.
2 In the **Settings** window for **Pressure Acoustics, Frequency Domain**, locate the **Domain Selection** section.
3 Click **Clear Selection**.
4 Select Domain 1 only.

Pressure Acoustics 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Pressure Acoustics, Frequency Domain (acpr)** click **Pressure Acoustics 1**.
2 In the **Settings** window for **Pressure Acoustics**, locate the **Pressure Acoustics Model** section.
3 From the **Fluid model** list, choose **Thermally conducting and viscous**.

Thermoviscous Boundary Layer Impedance 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Thermoviscous Boundary Layer Impedance**.
2 In the **Settings** window for **Thermoviscous Boundary Layer Impedance**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **All boundaries**.
4 Locate the **Fluid Properties** section. From the **Fluid material** list, choose **Water, liquid (mat1)**.

Thermoviscous Boundary Layer Impedance - Actuation

1 In the **Physics** toolbar, click **Boundaries** and choose **Thermoviscous Boundary Layer Impedance**.
2 In the **Settings** window for **Thermoviscous Boundary Layer Impedance**, type `Thermoviscous Boundary Layer Impedance - Actuation` in the **Label** text field.
3 Select Boundaries 1, 4, 111, and 112 only.

4 Locate the **Fluid Properties** section. From the **Fluid material** list, choose **Water, liquid (mat1)**.

5 Locate the **Mechanical Condition** section. From the **Mechanical condition** list, choose **Velocity**.

6 Specify the $\mathbf{v}_0$ vector as

act_v0	x
0	y

The boundary condition is used to actuate the system by using the velocity boundary condition.

THERMOVISCOUS ACOUSTICS, FREQUENCY DOMAIN (TA)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Thermoviscous Acoustics, Frequency Domain (ta)**.

2 Select Domains 2–14 only.

Thermoviscous acoustics is used in all the domains with sharp edges to capture the effect in the viscous boundary layers.

3 In the **Model Builder** window, click **Thermoviscous Acoustics, Frequency Domain (ta)**.

4 In the **Settings** window for **Thermoviscous Acoustics, Frequency Domain**, click to expand the **Discretization** section.

5 From the **Element order for pressure** list, choose **Quadratic serendipity**.

6 From the **Element order for velocity** list, choose **Cubic serendipity**.

7 From the **Element order for temperature** list, choose **Cubic serendipity**.

To accurately model the acoustic body force in the **Acoustic Streaming Domain Coupling**, it is necessary to increase the element order of the acoustic fields.

MULTIPHYSICS

Acoustic-Thermoviscous Acoustic Boundary 1 (atb1)

1 In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Boundary>Acoustic-Thermoviscous Acoustic Boundary**.

2 In the **Settings** window for **Acoustic-Thermoviscous Acoustic Boundary**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **All boundaries**.

Acoustic Streaming Domain Coupling 1 (asdc1)

1 In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Domain>Acoustic Streaming Domain Coupling**.
2 In the **Settings** window for **Acoustic Streaming Domain Coupling**, locate the **Coupled Interfaces** section.
3 From the **Source** list, choose **Thermoviscous Acoustics, Frequency Domain (ta)**.
4 Locate the **Domain Selection** section. From the **Selection** list, choose **All domains**.

LAMINAR FLOW (SPF)

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

Pressure Point Constraint 1

1 In the **Physics** toolbar, click **Points** and choose **Pressure Point Constraint**.
2 Select Point 20 only.

MESH 1

Free Triangular 1

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

Size 1

1 Right-click **Free Triangular 1** and choose **Size**.
2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
3 From the **Geometric entity level** list, choose **Domain**.
4 From the **Selection** list, choose **All domains**.
5 Select Domains 2–14 only.
6 Locate the **Element Size** section. From the **Predefined** list, choose **Extremely fine**.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)>Mesh 1** click **Size**.
2 In the **Settings** window for **Size**, locate the **Element Size** section.
3 From the **Calibrate for** list, choose **Fluid dynamics**.
4 From the **Predefined** list, choose **Extra fine**.

Distribution 1

1 In the **Model Builder** window, right-click **Free Triangular 1** and choose **Distribution**.
2 Select Boundaries 120, 124, 128, 132, 136, 140, 144, 148, 152, 156, 160, 164, and 168 only.

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

4 In the **Number of elements** text field, type `6`.

The distribution node is added at the tip of the sharp edges to ensure that the edge on the tip is well resolved.

Boundary Layers 1

In the **Mesh** toolbar, click **Boundary Layers**.

Boundary Layer Properties

1 In the **Model Builder** window, click **Boundary Layer Properties**.

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

3 From the **Selection** list, choose **All boundaries**.

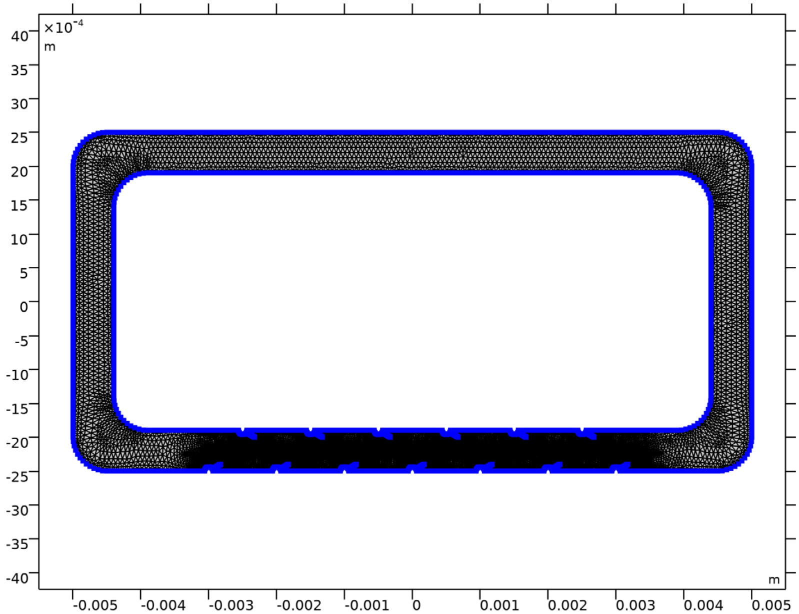
4 Select Boundaries 1–6, 8, 10–12, 14, 17–20, 22, 25–28, 30, 33–36, 38, 41–44, 46, 49–52, 54, 57–60, 62, 65–68, 70, 73–76, 78, 81–84, 86, 89–92, 94, 97–100, 102, 105–108, and 110–172 only.

5 Locate the **Layers** section. In the **Number of layers** text field, type `6`.

6 From the **Thickness specification** list, choose **First layer**.

7 In the **Thickness** text field, type `0.2E-6`.

8 Click **Build All**.



STUDY 1

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

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

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

Stationary

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

The study consists of two steps: first a `Frequency Domain` study step to model the acoustic fields and, second, a `Stationary` study step to model the fluid flow.

Step 1: Frequency Domain

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

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

3 In the **Frequencies** text field, type `f0`.

Step 2: Stationary

1 In the **Model Builder** window, click **Step 2: Stationary**.

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

3 In the table, enter the following settings:

Multiphysics couplings	Solve for	Equation form
Acoustic-Thermoviscous Acoustic Boundary 1 (atb1)		Automatic (Frequency domain)

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

RESULTS

From the **Home** menu, choose **Add Predefined Plot**.

ADD PREDEFINED PLOT

1 Go to the **Add Predefined Plot** window.

2 In the tree, select **Study 1/Solution 1 (sol1)>Acoustic-Thermoviscous Acoustic Boundary 1>Acoustic Pressure (atb1)**.

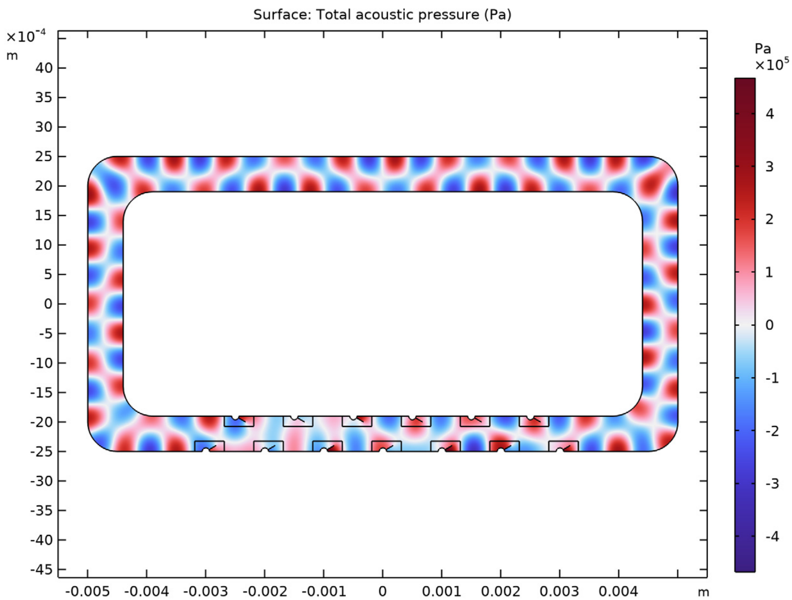
3 Click **Add Plot** in the window toolbar.

4 From the **Home** menu, choose **Add Predefined Plot**.

RESULTS

Acoustic Pressure (atb1)

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



Acoustic Body Force

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

2 Right-click **Results** and choose **2D Plot Group**.

3 In the **Settings** window for **2D Plot Group**, type `Acoustic Body Force` in the **Label** text field.

4 Click to expand the **Selection** section. From the **Geometric entity level** list, choose **Domain**.

5 Select Domain 2 only.

6 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

7 Locate the **Color Legend** section. Select the **Show units** check box.

Surface 1

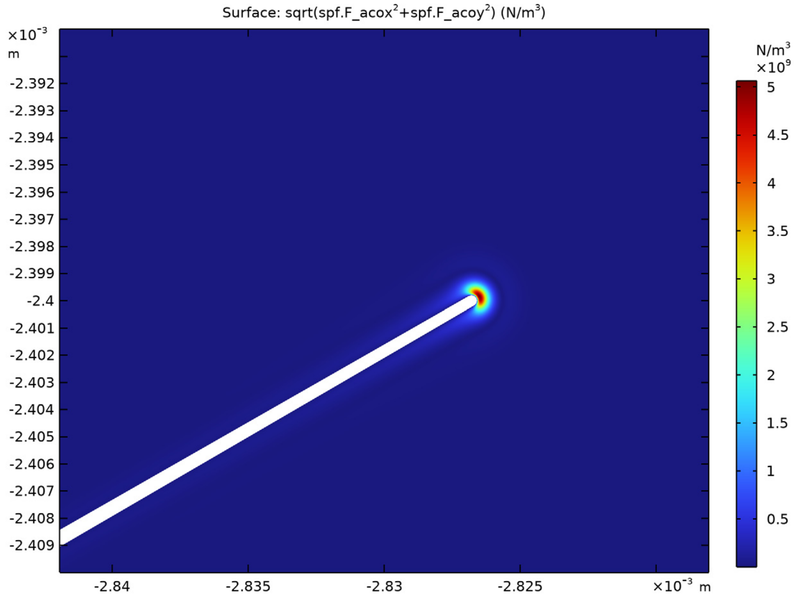
1 Right-click **Acoustic Body Force** and choose **Surface**.

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

3 In the **Expression** text field, type `sqrt(spf.F_acox^2+spf.F_acoy^2)`.

4 In the **Acoustic Body Force** toolbar, click **Plot**.

The acoustic body force that induces the fluid flow is located at the tip of the thin flaps.



5 In the **Home** toolbar, click **Add Predefined Plot**.

ADD PREDEFINED PLOT

1 Go to the **Add Predefined Plot** window.

2 In the tree, select **Study 1/Solution 1 (sol1)>Laminar Flow>Velocity (spf)**.

3 Click **Add Plot** in the window toolbar.

4 In the **Home** toolbar, click **Add Predefined Plot**.

RESULTS

Velocity (spf)

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

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

3 Select the **Show units** check box.

Surface

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

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

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

Streamline 1

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

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

3 In the **x-component** text field, type `u2`.

4 In the **y-component** text field, type `v2`.

5 Locate the **Streamline Positioning** section. From the **Positioning** list, choose **Uniform density**.

6 In the **Separating distance** text field, type `0.02`.

7 Locate the **Coloring and Style** section. Find the **Point style** subsection. From the **Type** list, choose **Arrow**.

8 Locate the **Streamline Positioning** section. In the **Separating distance** text field, type `0.01`.

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

Velocity (spf) - Logarithmic

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

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

3 In the **Settings** window for **2D Plot Group**, type `Velocity (spf) - Logarithmic` in the **Label** text field.

4 Locate the **Color Legend** section. Select the **Show maximum and minimum values** check box.

Surface

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

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

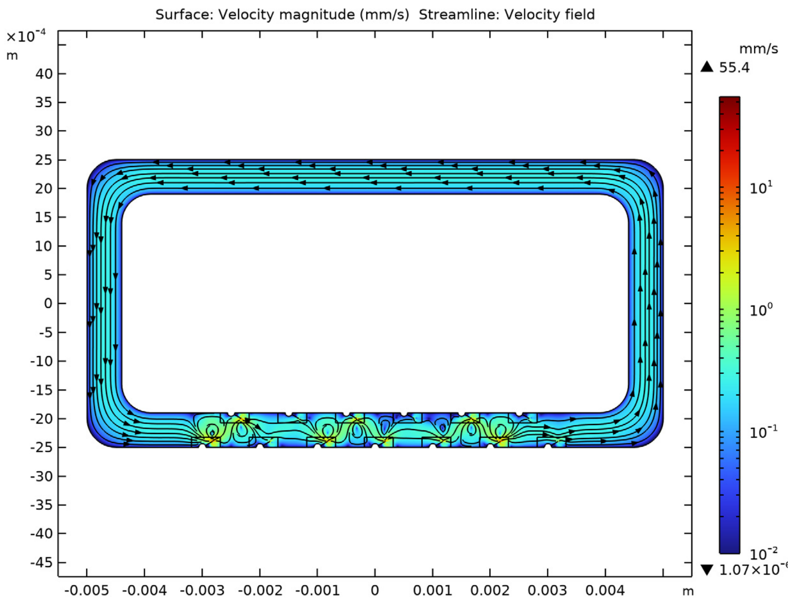
3 From the **Scale** list, choose **Logarithmic**.

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

5 In the **Minimum** text field, type `1e-2`.

6 In the **Velocity (spf) - Logarithmic** toolbar, click **Plot**.

The fluid flow is represented on a logarithmic scale since the velocity amplitude varies by orders of magnitude.



Geometry Sequence 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 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 `acoustic_microfluidic_pump_geom_sequence_parameters.txt`.

GEOMETRY 1

Rectangle 1 (r1)

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Geometry 1** node.

2 Right-click **Geometry 1** and choose **Rectangle**.

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

4 In the **Width** text field, type `L`.

5 In the **Height** text field, type `W`.

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 `L-channel_w*2`.

4 In the **Height** text field, type `W-channel_w*2`.

Rectangle 1 (r1)

1 In the **Model Builder** window, click **Rectangle 1 (r1)**.

2 In the **Settings** window for **Rectangle**, locate the **Position** section.

3 From the **Base** list, choose **Center**.

Rectangle 2 (r2)

1 In the **Model Builder** window, click **Rectangle 2 (r2)**.

2 In the **Settings** window for **Rectangle**, locate the **Position** section.

3 From the **Base** list, choose **Center**.

Fillet 1 (fil1)

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

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

3 On the object **r1**, select Points 1–4 only.

4 On the object **r2**, select Points 1–4 only.

5 In the **Settings** window for **Fillet**, locate the **Radius** section.

6 In the **Radius** text field, type `corner`.

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 (m)	**y (m)**
`-10*flap_w`	`-W/2`
`cos(flap_angle)*flap_L`	`-W/2+sin(flap_angle)*flap_L`
`+10*flap_w`	`-W/2`

4 Click **Build Selected**.

Circle 1 (c1)

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

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

3 In the **Radius** text field, type `flap_L/4`.

4 In the **Sector angle** text field, type `180`.

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

Rectangle 3 (r3)

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

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

3 In the **Width** text field, type `2*flap_L`.

4 In the **Height** text field, type `0.7*flap_L`.

5 Locate the **Position** section. In the **x** text field, type `-0.75*flap_L`.

6 In the **y** text field, type `-W/2`.

7 Click **Build Selected**.

Copy 1 (copy1)

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

2 Select the objects **c1**, **pol1**, and **r3** only.

3 In the **Settings** window for **Copy**, locate the **Displacement** section.

4 In the **x** text field, type `range(-3*flap_dist,(3*flap_dist-(-3*flap_dist))/6,3*flap_dist)`.

5 Click **Build Selected**.

Polygon 2 (pol2)

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 (m)	y (m)
-10*flap_w-0.5*flap_dist	-W/2+channel_w
cos(flap_angle)*flap_L-0.5*flap_dist	-W/2+channel_w-sin(flap_angle)*flap_L
+10*flap_w-0.5*flap_dist	-W/2+channel_w

Circle 2 (c2)

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

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

3 In the **Radius** text field, type flap_L/4.

4 In the **Sector angle** text field, type 180.

5 Locate the **Position** section. In the **x** text field, type -0.5*flap_dist.

6 In the **y** text field, type -W/2+channel_w.

7 Locate the **Rotation Angle** section. In the **Rotation** text field, type 180.

Rectangle 4 (r4)

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

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

3 In the **Width** text field, type 2*flap_L.

4 In the **Height** text field, type 0.7*flap_L.

5 Locate the **Position** section. In the **x** text field, type -0.75*flap_L-flap_dist/2.

6 In the **y** text field, type -W/2+channel_w-flap_L*0.7.

7 Click **Build Selected**.

Copy 2 (copy2)

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

2 Select the objects **c2**, **pol2**, and **r4** only.

3 In the **Settings** window for **Copy**, locate the **Displacement** section.

4 In the **x** text field, type range(-2*flap_dist,(3*flap_dist-(-2*flap_dist))/5,3*flap_dist).

5 Click **Build Selected**.

Union 1 (uni1)

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

2 Select the object **fil1(1)** only.

3 Click the **Select All** button in the **Graphics** toolbar.

4 In the **Settings** window for **Union**, click **Build Selected**.

Fillet 2 (fil2)

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

2 In the **Settings** window for **Fillet**, locate the **Points** section.

3 Find the **Vertices to fillet** subsection. Click to select the **Activate Selection** toggle button.

4 On the object **uni1**, select Points 19, 32, 45, 58, 71, 84, 97, 110, 123, 136, 149, 162, and 175 only.

5 Locate the **Radius** section. In the **Radius** text field, type `flap_w`.

6 Click **Build Selected**.

Delete Entities 1 (del1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Delete Entities**.

2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

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

4 On the object **fil2**, select Domains 2, 4–7, 9–12, 14–17, 19–22, 24–27, 29–32, 34–37, 39–42, 44–47, 49–52, 54–57, 59–62, and 64–67 only.

5 Click **Build All Objects**.

Form Union (fin)

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