Created in COMSOL Multiphysics 6.1

Acoustic Streaming in a Microchannel Cross Section

Introduction

Recent advances in the fabrication of microfluidic systems require handling of live cells and other microparticles. One possibility is to use acoustics and exploit the acoustic radiation force to focus particles and/or separate them based on their acoustical properties. When using acoustics in a microchannel, an acoustic streaming flow will be generated and also affect the particles with a viscous drag force; see Ref. 1.

The particle trajectories are determined by the acoustic radiation force and the viscous drag force. The acoustic radiation force is an effect where momentum is transferred from an acoustic field to particles due to nonlinear terms in the governing equations. This results in a net force acting on the particles — the acoustic radiation force.

Due to the nonlinear terms in the Navier–Stokes equations, harmonic perturbation of the flow will lead to a net time-averaged flow called acoustic streaming. Acoustic streaming is a second-order (nonlinear) acoustic effect. The acoustic streaming influences the viscous drag force on the particles. The trajectory of particles in devices are governed by the balance between the viscous drag force (from the streaming flow) and the acoustic radiation force.

There are losses in the acoustic field that produce a heat source. This can be frictional losses in the viscous boundary layers where acoustic energy is transformed into heat, which can result in heating of a microfluidic system.

The model is of a 2D cross section of a microfluidic channel which can, for example, be used for upconcentrating or separating particles in biological fluid samples. This model is based on pressure acoustics and uses effective boundary conditions (the thermoviscous boundary layer impedance or BLI condition) to include the effects of the viscous boundary layers; see Ref. 2.

Model Definition

The model consists of a rectangular fluid domain actuated by vibrating walls; see Figure 1. It is assumed that the surrounding solid is acoustically hard. The model first computes the acoustic field in a **Frequency Domain** study; then the time-averaged second-order fields, the acoustic streaming flow and acoustic heating in a **Stationary** study; and lastly the particle trajectories in a **Time-dependent** study.

The acoustic field is modeled with **Pressure Acoustics, Frequency Domain** and the **Thermoviscous Boundary Layer Impedance** is used to account for the damping in the thin viscous boundary layers.

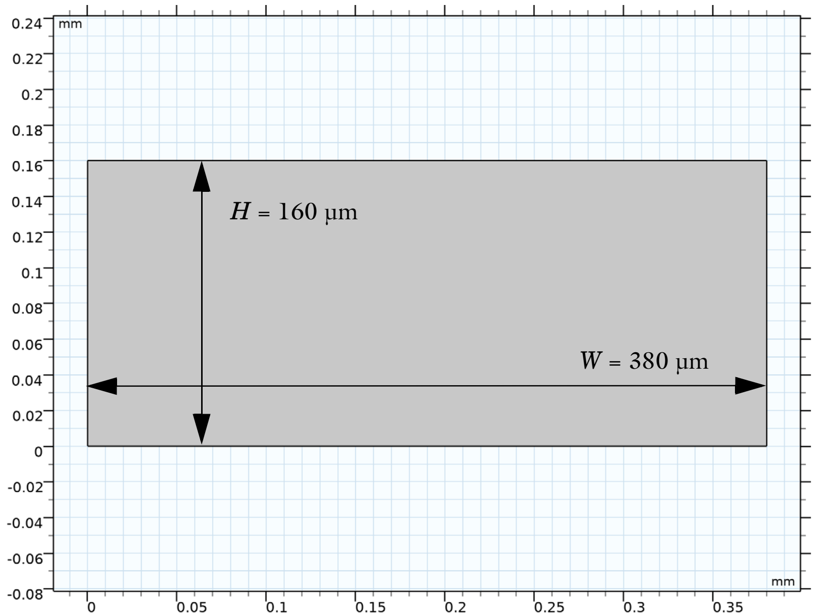


Figure 1: Geometry of the 2D cross section of the microfluidic channel.

The acoustic streaming flow is modeled using the multiphysics couplings **Acoustic Streaming Domain Coupling** and **Acoustic Streaming Boundary Coupling**, which from the acoustic field compute and apply the acoustic source terms to the fluid flow interface. The source terms consist of a domain force and a slip velocity on the boundary. The slip velocity includes the contribution from the thin viscous and thermal boundary layers.

The heating from the acoustic field primarily occurs in the viscous boundary layer. The heat generated in the viscous boundary layer is computed analytically (it exists as a predefined variable) and imposed as a boundary layer heat source. To mimic a typical silicon chip with a glass lid, the bottom and the two sides of the rectangle have a constant temperature (due to silicon being a very good heat conductor), while the top is thermally insulating (glass transports heat much less than silicon). Therefore, the temperature will increase at the glass lid, and there will be a temperature gradient across the microfluidic channel.

The particle trajectories are modeled using the **Particle Tracing for Fluid Flow** interface, which computes the particle trajectories based on the two contributing forces: the acoustic radiation force and the viscous drag force. These two forces depend differently on the size of the particles: for small particles the viscous drag force dominates. whereas the acoustic radiation force dominates for large particles. In most applications, the acoustic radiation force is used to focus particles. This is therefore not possible for particles below a critical particle size, which depends on the material properties of the particle.

Results and Discussion

The system is actuated at the resonance frequency $f_0 = 1.9652$ MHz for the horizontal half-wave resonance. This results in the acoustic field in Figure 2 and an acoustic energy density which is typical for these types of microfluidic devices. The main damping in the system is due to losses in the viscous boundary layers, which therefore determine the Q-factor of the system and the width of the resonance peak. In an actual microfluidic chip there will also be significant damping in the piezoelectric transducer used to actuate the system and in the glue layer. This can widen the resonance peak and lower the resonance frequency.

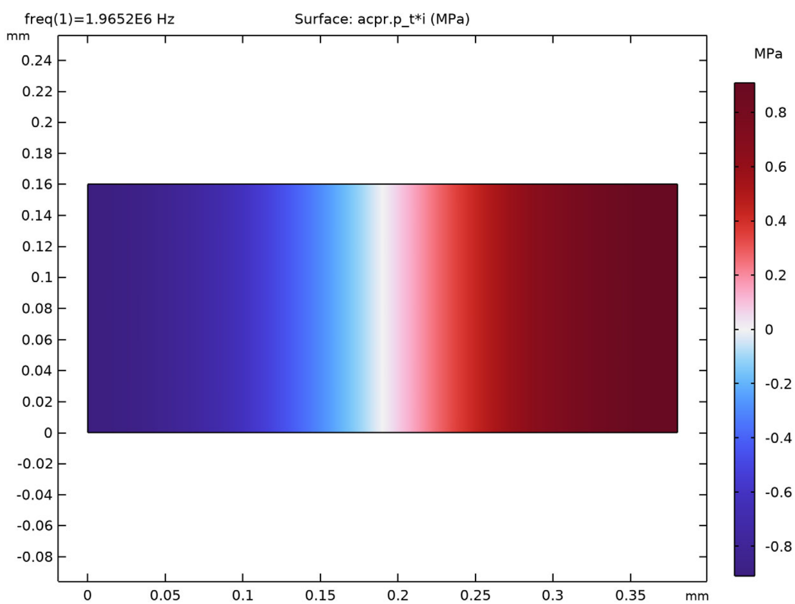


Figure 2: Acoustic pressure field in the rectangular channel at the horizontal half-wave resonance.

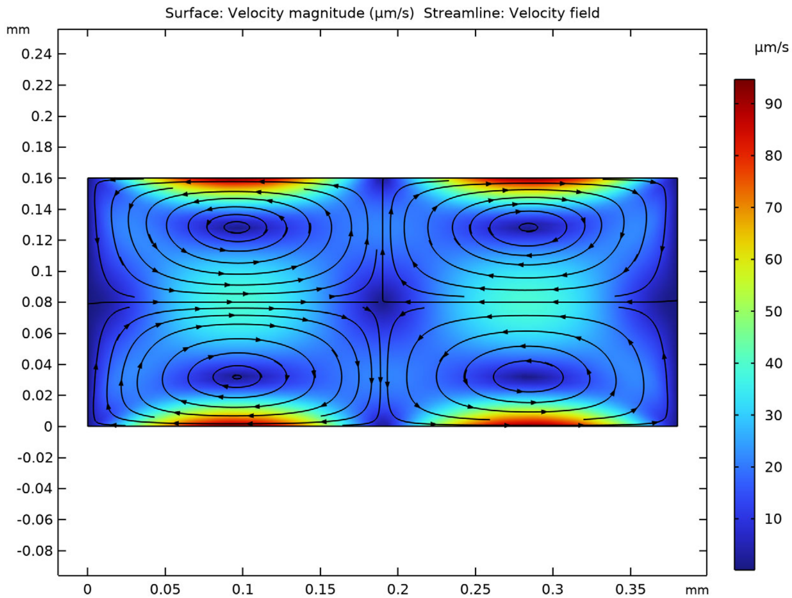


Figure 3: Acoustic streaming flow in the microchannel cross section. The color scale shows amplitude of the fluid flow velocity while the black streaming lines show the direction of the flow field. The streaming consists of the classical four Rayleigh Streaming rolls.

Figure 3 shows the acoustic streaming induced by the acoustic field; the color plot shows the amplitude of the flow field and the black streamlines show the direction of the fluid flow. The streaming flow forms classical Rayleigh streaming rolls, which are typical for boundary-driven streaming. The Rayleigh streaming is induced by the stresses and forces in the viscous boundary layers. In this model, the contributions from the viscous boundary layers are applied as slip velocity and therefore the boundary layers do not have to be numerically resolved.

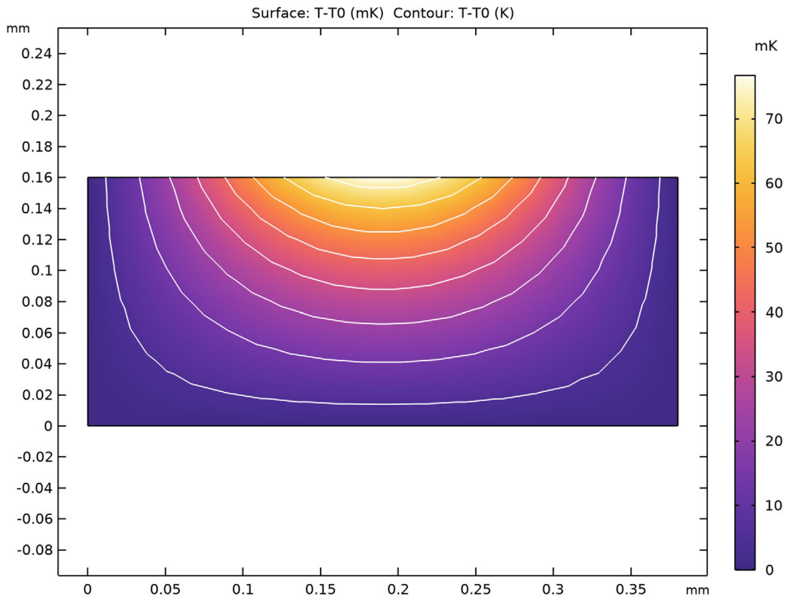


Figure 4: The temperature field created due to the acoustic heat source in the viscous boundary layers. The white lines represents contour lines of the temperature field.

The heating from the acoustic field induces a temperature gradient across the microchannel. The temperature increase due to the acoustic field is shown in Figure 4. The boundary condition on the acoustic field, **Thermoviscous Boundary Layer Impedance**, computes the heat generated in the viscous boundary layer in the variable `acpr.tvb1.Q_tot`. This boundary heat source is used to model the temperature increase in the microfluidic channel. In this example, the acoustic field results in a small temperature increase of the order mK. The temperature increase in an actual microfluidic device will depend on the acoustic field, but also on the thermal properties of the surrounding solid.

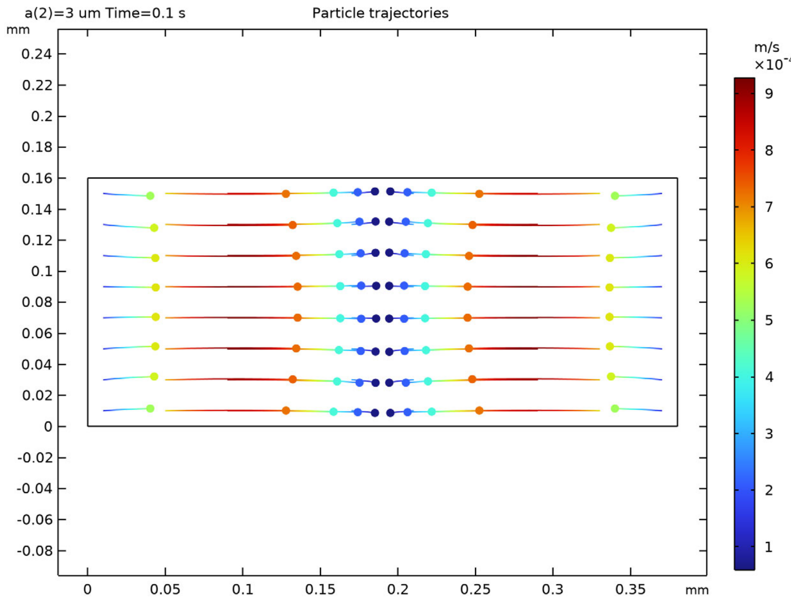


Figure 5: The position of particles with radius a = 3 µm at a specific time step. The color of the particles represent the amplitude of their velocity and the lines their trajectory.

The particle trajectory is determined by the acoustic radiation force and the viscous drag force. In Figure 5, the positions of polystyrene particles can be seen at time t = 0.1 s. The color of the particles represents the velocity amplitude and the lines represent the particle trajectory. In Figure 5 the particles have radius a = 3 µm and their trajectories are dominated by the acoustic radiation force focusing the particles in the pressure node. For smaller polystyrene particles with radius a = 0.4 µm, the particle trajectories are shown in Figure 6 at t = 3 s. For the small particles, the viscous drag force dominates; the particles are dragged by the fluid flow, and are not focused in the pressure nodes. The velocities of the small particles are an order of magnitude smaller than those of the large particles.

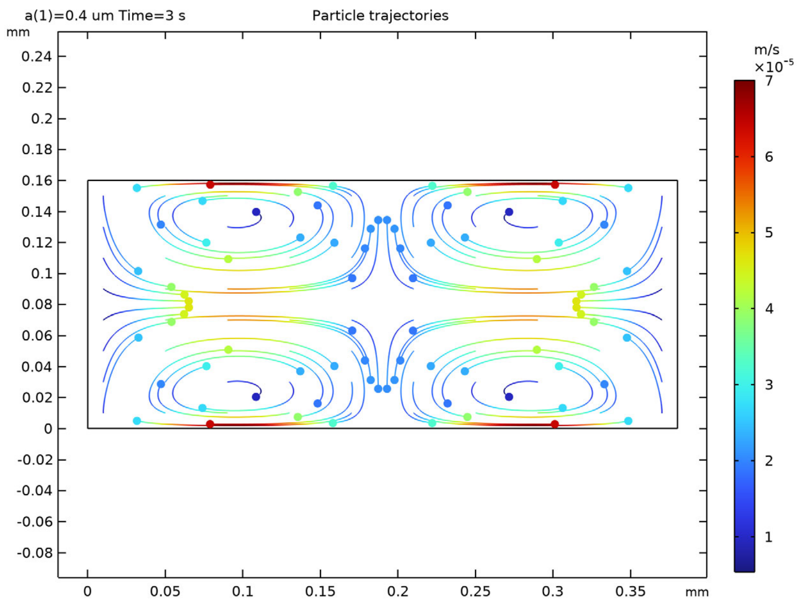


Figure 6: The position of particles with radius a = 0.4 µm at a time step. The color of the particles represent the amplitude of their velocity and the lines their trajectory.

Notes About the COMSOL Implementation

The implementation is based on pressure acoustics, meaning that the viscous boundary layers are not resolved numerically. Therefore, effective boundary conditions are used to represent the impact of the boundary layers (the **Thermoviscous Boundary Layer Impedance** condition). This is used for the damping of the acoustic field, slip velocity for the fluid flow, and boundary heat source for the acoustic heating. These are all analytical expressions that are valid when the viscous boundary layer thickness is a lot smaller than the acoustic wavelength and the geometrical length scales; see Ref. 2. Some of the analytical expressions depend on the derivatives at the boundary. Therefore, a thin boundary-layer mesh element is used to improve the accuracy of the normal derivative at the boundary.

References

1. P.B. Muller, R. Barnkob, M.J. Herring Jensen, and H. Bruus, "A numerical study of microparticle acoustophoresis driven by acoustic radiation forces and streaming-induced drag forces," *Lab. Chip.*, vol. 12, pp. 4617–4627, 2012.

2. J.S. Bach and H. Bruus, “Theory for pressure acoustics with viscous boundary layers and streaming in curved elastic cavities,” *J. Acoust. Soc. Am.*, vol. 144, no. 2, pp. 766–784, 2018.

Application Library path: `Acoustics_Module/Nonlinear_Acoustics/acoustic_streaming_microchannel_cross_section`

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>Acoustic Streaming>Acoustic Streaming from Pressure Acoustics**.

3 Click **Add**.

4 In the **Select Physics** tree, select **Heat Transfer>Heat Transfer in Fluids (ht)**.

5 Click **Add**.

6 In the **Select Physics** tree, select **Fluid Flow>Particle Tracing>Particle Tracing for Fluid Flow (fpt)**.

7 Click **Add**.

8 Click **Study**.

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

10 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_streaming_microchannel_cross_section_parameters.txt`.

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)

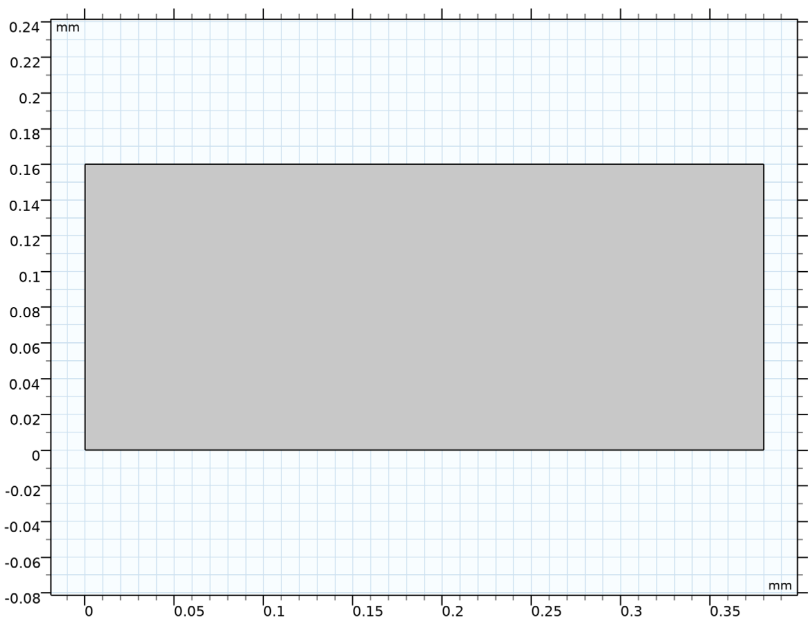
1 In the **Geometry** toolbar, click **Rectangle**.

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

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

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

5 Click **Build All Objects**.



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)

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 **Model Input** section.

3 In the T text field, type `T0`.

Thermoviscous Boundary Layer Impedance 1

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

2 In the **Settings** window for **Thermoviscous Boundary Layer Impedance**, locate the **Fluid Properties** section.

3 From the **Fluid material** list, choose **Water, liquid (mat1)**.

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

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

d0*acpr.iomega	x
0	y

The system is actuated by a boundary velocity in the **Thermoviscous Boundary Layer Impedance** boundary condition.

LAMINAR FLOW (SPF)

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

2 In the **Settings** window for **Laminar Flow**, locate the **Physical Model** section.

3 In the T_{ref} text field, type `T0`.

To resolve the fluid-flow field on the coarse mesh used for the acoustics, change the discretization.

4 Click to expand the **Discretization** section. From the **Discretization of fluids** list, choose **P2+P1**.

Pressure Point Constraint 1

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

Fluid Properties 1

1 In the **Model Builder** window, click **Fluid Properties 1**.

2 In the **Settings** window for **Fluid Properties**, locate the **Model Input** section.

3 From the T list, choose **User defined**. In the associated text field, type `T0`.

Pressure Point Constraint 1

1 In the **Model Builder** window, click **Pressure Point Constraint 1**.

2 Select Point 2 only.

HEAT TRANSFER IN FLUIDS (HT)

In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Fluids (ht)**.

Temperature 1

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

2 Select Boundaries 1, 2, and 4 only.

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

4 In the T_0 text field, type `T0`.

Boundary Heat Source 1

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

2 In the **Settings** window for **Boundary Heat Source**, locate the **Boundary Heat Source** section.

3 From the Q_b list, choose **Total thermoviscous power dissipation in boundary layers (acpr/tvb1)**.

4 Select Boundary 3 only.

The boundary heat source `acpr.tvb1.Q_tot` is the heat source from the acoustically thin boundary layers applied as a boundary heat source.

PARTICLE TRACING FOR FLUID FLOW (FPT)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Particle Tracing for Fluid Flow (fpt)**.

2 In the **Settings** window for **Particle Tracing for Fluid Flow**, locate the **Particle Release and Propagation** section.

3 From the **Formulation** list, choose **Newtonian, ignore inertial terms**.

Drag Force 1

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

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

3 From the **u** list, choose **Velocity field (spf)**.

4 Locate the **Model Input** section. In the T text field, type `T0`.

5 Select Domain 1 only.

Acoustophoretic Radiation Force 1

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

2 Select Domain 1 only.

3 In the **Settings** window for **Acoustophoretic Radiation Force**, locate the **Model Input** section.

4 In the T text field, type `T0`.

5 Locate the **Acoustic Fields** section. From the p list, choose **Pressure (acpr)**.

6 From the **u** list, choose **Total acoustic velocity (acpr/fpam1)**.

7 Locate the **Particle Material Properties** section. In the $c_{\mathrm{p,p}}$ text field, type `cp_p`.

8 In the $c_{\mathrm{s,p}}$ text field, type `cs_p`.

Particle Properties 1

1 In the **Model Builder** window, click **Particle Properties 1**.

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

3 From the ρ_{p} list, choose **User defined**. In the associated text field, type `rho_p`.

4 In the d_{p} text field, type `2*a`.

Release from Grid 1

1 In the **Physics** toolbar, click **Global** and choose **Release from Grid**.

2 In the **Settings** window for **Release from Grid**, locate the **Initial Coordinates** section.

3 Click **X Range**.

4 In the **Range** dialog box, choose **Number of values** from the **Entry method** list.

5 In the **Start** text field, type `10[µm]`.

6 In the **Stop** text field, type `W-10[µm]`.

7 In the **Number of values** text field, type `10`.

8 Click **Replace**.

9 In the **Settings** window for **Release from Grid**, locate the **Initial Coordinates** section.

10 Click **Y Range**.

11 In the **Range** dialog box, type `10[µm]` in the **Start** text field.

12 In the **Stop** text field, type `H-10[µm]`.

13 From the **Entry method** list, choose **Number of values**.

14 In the **Number of values** text field, type `8`.

15 Click **Replace**.

MESH 1

Free Triangular 1

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

Boundary Layers 1

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

2 In the **Settings** window for **Boundary Layers**, click to expand the **Transition** section.

3 Clear the **Smooth transition to interior mesh** check box.

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 Locate the **Layers** section. In the **Number of layers** text field, type `1`.

   A single boundary layer ensures accurate normal derivatives on the boundary used for the **Acoustic Streaming Boundary Coupling**.

5 Click **Build All**.

STUDY 1: ACOUSTIC FIELD

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

2 In the **Settings** window for **Study**, type `Study 1: Acoustic field` in the **Label** text field.

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

Step 1: Frequency Domain

1 In the **Model Builder** window, under **Study 1: Acoustic field** 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`.

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 **Preset Studies for Some Physics Interfaces>Stationary**.

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

5 In the **Select Study** tree, select **Preset Studies for Some Physics Interfaces> Time Dependent**.

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

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

STUDY 2: STATIONARY FIELDS

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

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

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

Step 1: Stationary

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

2 In the **Settings** window for **Stationary**, click to expand the **Values of Dependent Variables** section.

3 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 **Study 1: Acoustic field, Frequency Domain**.

STUDY 3: PARTICLE TRACING

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

2 In the **Settings** window for **Study**, type `Study 3: Particle Tracing` in the **Label** text field.

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
a (Particle radius)	`0.4 3`	`um`

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

Set the tolerance manually to avoid taking small time steps when the particles are at the pressure node.

4 In the **Relative tolerance** text field, type `1e-3`.

5 In the **Output times** text field, type `range(0,0.02,3)`.

6 Locate the **Physics and Variables Selection** section. In the table, enter the following settings:

Physics interface	Solve for	Equation form
Pressure Acoustics, Frequency Domain (acpr)		Automatic (Frequency domain)
Laminar Flow (spf)		Automatic (Stationary)
Heat Transfer in Fluids (ht)		Automatic (Stationary)
Particle Tracing for Fluid Flow (fpt)	√	Automatic (Time dependent)

7 In the table, enter the following settings:

Multiphysics couplings	Solve for	Equation form
Acoustic Streaming Domain Coupling 1 (asdc1)		Automatic (Stationary)
Acoustic Streaming Boundary Coupling 1 (asbc1)		Automatic (Stationary)

8 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**.

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

10 From the **Study** list, choose **Study 2: Stationary fields, Stationary**.

STUDY 1: ACOUSTIC FIELD

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

STUDY 2: STATIONARY FIELDS

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: Acoustic field/Solution 1 (sol1)>Pressure Acoustics, Frequency Domain>Acoustic Pressure (acpr)**.
3 Click **Add Plot** in the window toolbar.
4 From the **Home** menu, choose **Add Predefined Plot**.

RESULTS

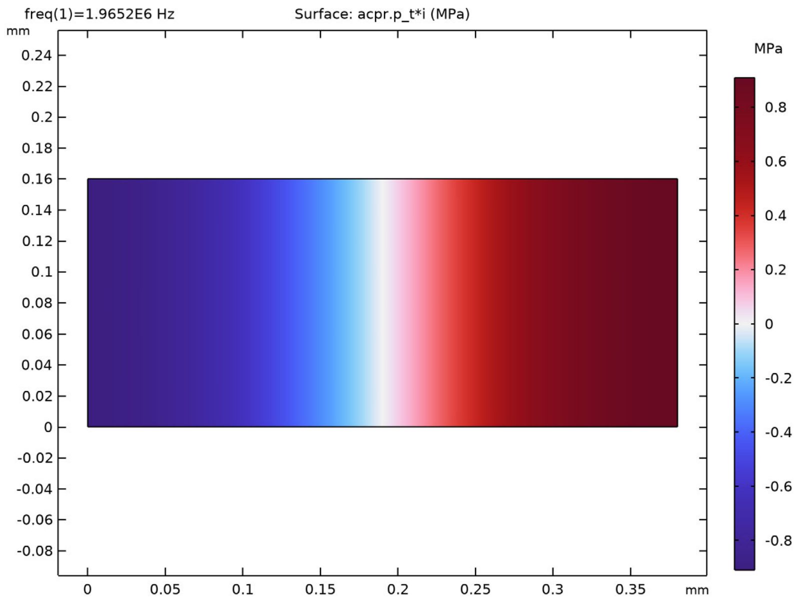
Acoustic Pressure (acpr)

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

Surface 1

1 In the **Model Builder** window, expand the **Acoustic Pressure (acpr)** node, then click **Surface 1**.
2 In the **Settings** window for **Surface**, locate the **Expression** section.
3 In the **Expression** text field, type `acpr.p_t*i`.
4 From the **Unit** list, choose **MPa**.

5 In the **Acoustic Pressure (acpr)** toolbar, click **Plot**.



6 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 2: Stationary fields/Solution 2 (sol2)>Laminar Flow>Velocity (spf)**.

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

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

RESULTS

Surface

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

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

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

Velocity (spf)

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

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

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

Streamline 1

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

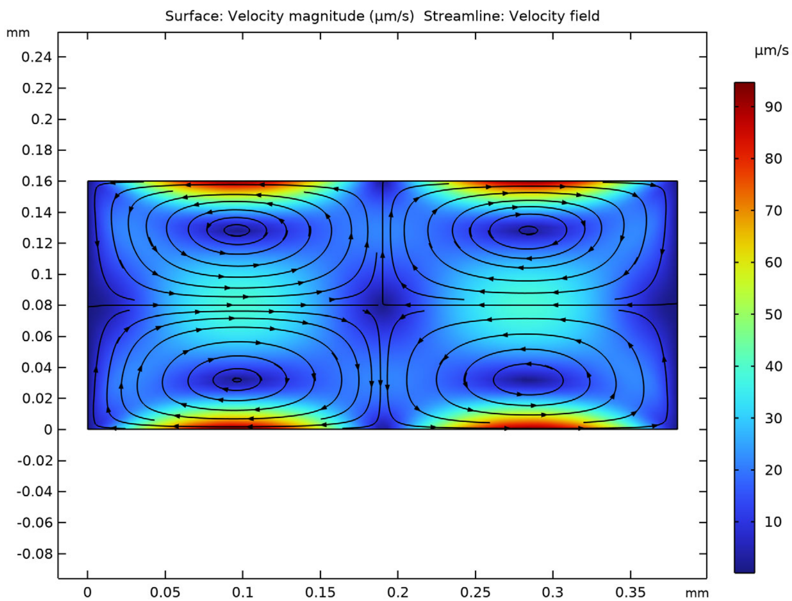
2 In the **Settings** window for **Streamline**, locate the **Streamline Positioning** section.

3 From the **Positioning** list, choose **Uniform density**.

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

5 Locate the **Streamline Positioning** section. In the **Separating distance** text field, type `0.03`.

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



7 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 2: Stationary fields/Solution 2 (sol2)>Heat Transfer in Fluids>Temperature (ht)**.

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

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

RESULTS

Temperature (ht)

1. In the **Model Builder** window, under **Results** click **Temperature (ht)**.
2. In the **Settings** window for **2D Plot Group**, locate the **Color Legend** section.
3. Select the **Show units** check box.

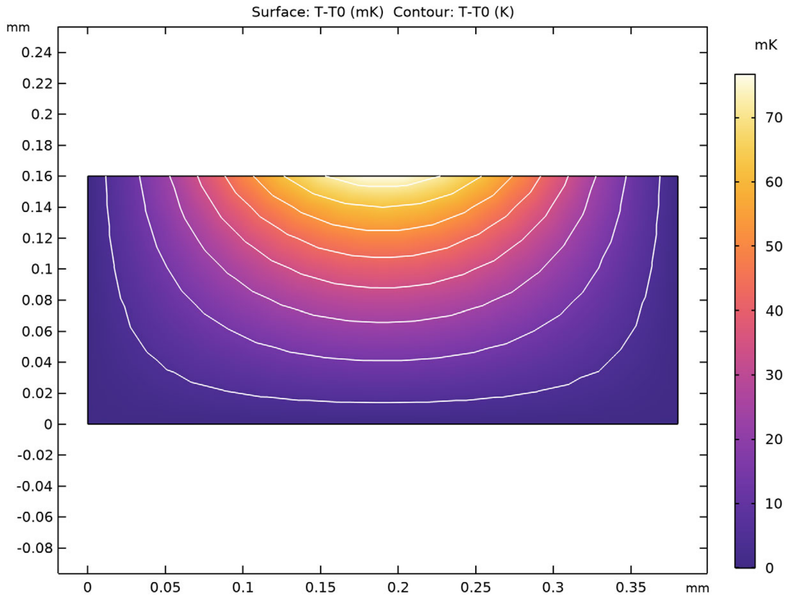
Surface

1. In the **Model Builder** window, expand the **Temperature (ht)** node, then click **Surface**.
2. In the **Settings** window for **Surface**, locate the **Expression** section.
3. In the **Expression** text field, type `T-T0`.
4. From the **Unit** list, choose **mK**.

Contour 1

1. In the **Model Builder** window, right-click **Temperature (ht)** and choose **Contour**.
2. In the **Settings** window for **Contour**, locate the **Expression** section.
3. In the **Expression** text field, type `T-T0`.
4. Locate the **Levels** section. In the **Total levels** text field, type `8`.
5. Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.
6. From the **Color** list, choose **White**.
7. Clear the **Color legend** check box.

8 In the **Temperature (ht)** toolbar, click **Plot**.



STUDY 3: PARTICLE TRACING

Step 1: Time Dependent

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

RESULTS

Particle Trajectories - Large (fpt)

1 In the **Settings** window for **2D Plot Group**, type `Particle Trajectories - Large (fpt)` in the **Label** text field.

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

3 Locate the **Data** section. From the **Time (s)** list, choose **0.1**.

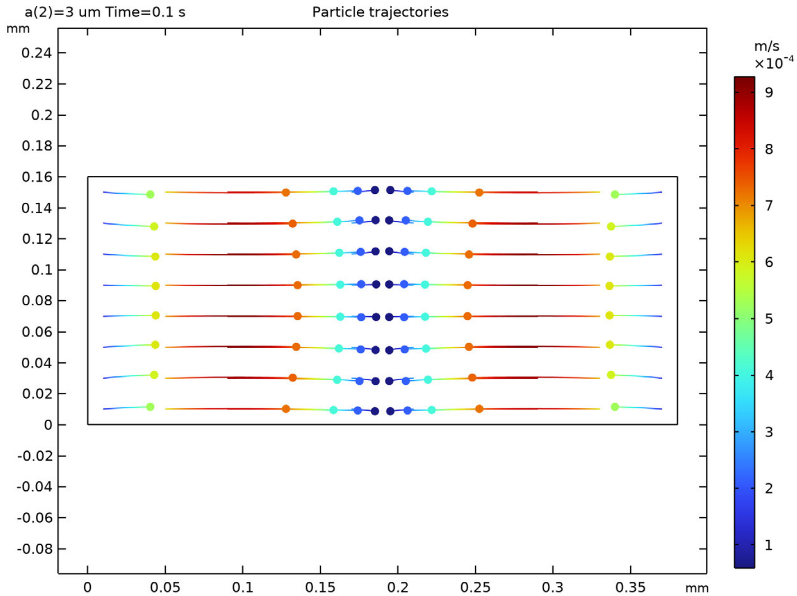
Particle Trajectories 1

1 In the **Model Builder** window, expand the **Particle Trajectories - Large (fpt)** node, then click **Particle Trajectories 1**.

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

3 Find the **Line style** subsection. From the **Type** list, choose **Line**.

4 In the **Particle Trajectories - Large (fpt)** toolbar, click **Plot**.



Particle Trajectories - Small (fpt)

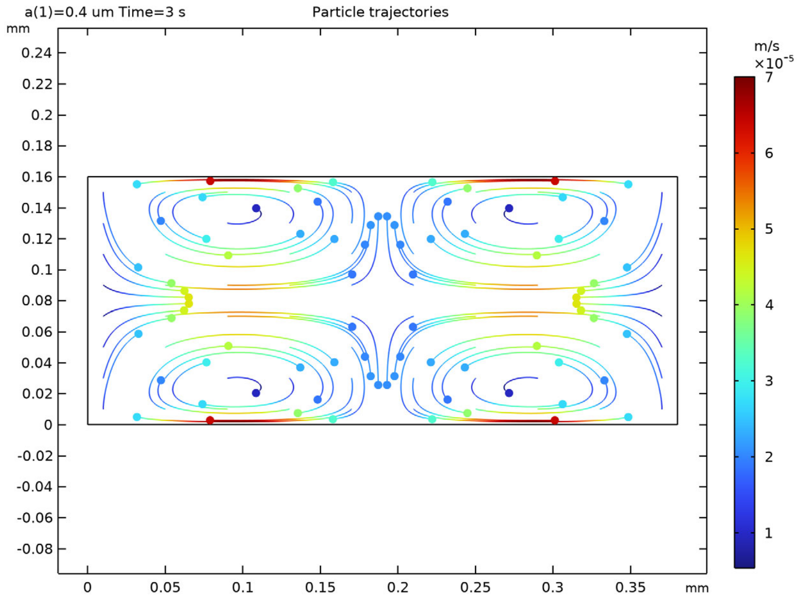
1 In the **Model Builder** window, right-click **Particle Trajectories - Large (fpt)** and choose **Duplicate**.

2 In the **Settings** window for **2D Plot Group**, type `Particle Trajectories - Small (fpt)` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter value (a (um))** list, choose **0.4**.

4 From the **Time (s)** list, choose **3**.

5 In the **Particle Trajectories - Small (fpt)** toolbar, click **Plot**.



Finally, visualize the time-dependent study of the particle trajectories by an animation.

Particle Trajectories

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

2 In the **Settings** window for **Animation**, type `Particle Trajectories` in the **Label** text field.

3 Locate the **Scene** section. From the **Subject** list, choose **Particle Trajectories - Large (fpt)**.