

Pinched Flow Fractionation

Introduction

Pinched flow fractionation (PFF) is a purely hydrodynamic method of separating particles by size or mass, using a microfluidic device. The separation of particles is critical in many biomedical applications ([Ref. 1](#)). One of the popular areas of interest is the separation of cells from whole blood such as red blood cells, white blood cells, and platelets. A clear blood plasma is also utilized in cancer detection.

This is a tutorial model of a microfluidic device with two inlets. Inlet 1 delivers a sample of fluid containing particles to separate while inlet 2 delivers pure fluid with no particles at a significantly higher flow rate than the first. When the two inlets converge at a narrow microchannel, the stream carrying the sample is pinched into a narrow region near the wall opposite the high-velocity flow. The smaller particles are pushed closer to the wall. After entering the microchannel, the wall-induced lift force pushes the particles toward the center of the channel. Under the combination of lift and drag forces, the particles become sorted by size, with larger particles migrating closer to the center of the channel.. The sample then enters a wider section that branches into many outlets. The particles exit through different branches based on their lateral position when exiting the microchannel, effectively sorting the particles by size.

The outlet at the end close to inlet 2 is called the drain. When the drain is shorter and/or broader than the other outlets, then the method is called asymmetrical pinched flow fractionation (AsPFF) ([Ref. 2](#)). AsPFF has lesser resistance to the flow in the drain and hence allows more liquid to flow. This in turn allows the wider spread of streamlines along the outlets and hence the separation utilizes more of the outlet branches and the range of sizes of particles collected by each outlet becomes smaller. Thus AsPFF allows better fractionation of particles compared to PFF. The model used in this example can be used to simulate both PFF and AsPFF by simply changing the value of ratio of drain length to branch length in the geometry.

Model Definition

The geometry consists of two inlets each of width 0.1 mm and length 0.5 mm fused with each other at an angle of 45°. The inlets converge to a microchannel of width 20 μm and length 120 μm . The microchannel opens to a wider opening which is divided into twelve outlets and one drain. The distance of each outlet from the midpoint of opening of microchannel is 1 mm while the distance to the drain outlet is 0.5 mm. The dimensions of inlets, microchannel, outlets, and drain and the number of outlets can be changed by editing appropriate parameters of the geometry.

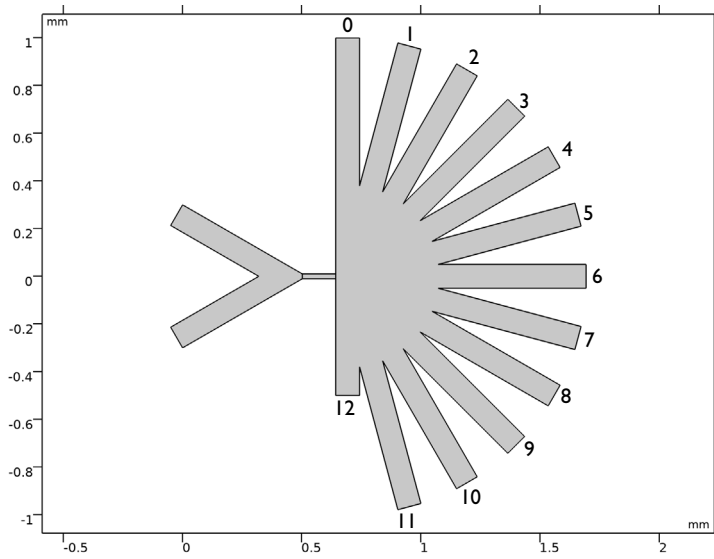


Figure 1: Model geometry of asymmetric pinched flow fractionation device. Fluid enters through two inlets on the left and exits through the 13 numbered outlets.

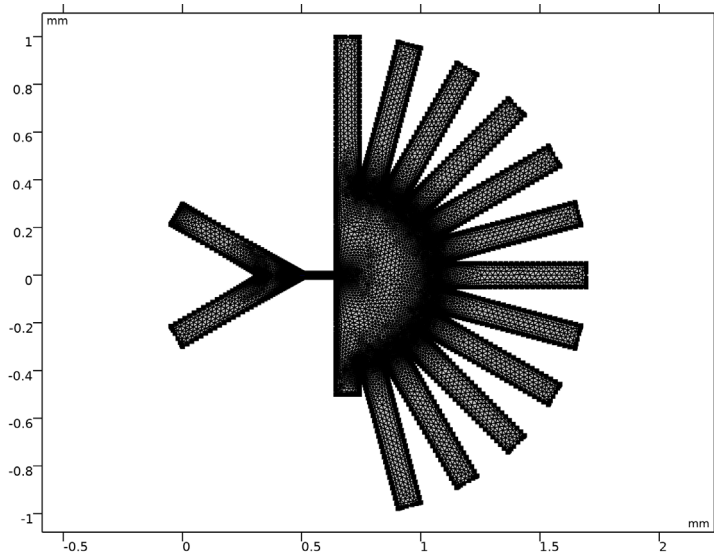


Figure 2: The mesh is refined around the entrance of microchannel and near corners.

The sample used here is water containing spherical particles of density 1300 kg/m^3 , with diameters uniformly distributed between $1.5 \text{ }\mu\text{m}$ and $11 \text{ }\mu\text{m}$. The inlet in the upper left corner of Figure 1 delivers the sample at the rate of $25 \text{ }\mu\text{l/hr}$. The inlet in the lower left corner delivers pure water at the rate of $1200 \text{ }\mu\text{l/hr}$. The outlet branches are numbered from 0 to 12 in a clockwise direction, with 12 being the drain in Figure 1. The microchannel should be meshed at higher resolution relative to the rest of the geometry. When the particle enters the microchannel there is sudden change in the velocity, thus the entrance is manually meshed to finer sizes as shown in Figure 2.

The geometry is divided into three domains; the first domain consists of inlets, second domain is the microchannel, and the third domain includes the outlets. The lift force is most significant in the microchannel where the particles will be in very close proximity to the channel walls. Thus the lift force is applied only in the second domain using the **Lift Force** feature. The drag force is applied in all domains using the **Drag Force** feature.

Notes on COMSOL Implementation

The model is solved in two steps. First, the fluid velocity and pressure are solved using the Laminar Flow interface and **Stationary** study step. Then the particle trajectories are created using the Particle Tracing for Fluid Flow interface and **Time Dependent** study step.

A total of 100 particles are released, one at every 5 ms. The released particles have random diameter between $1.5 \text{ }\mu\text{m}$ and $11 \text{ }\mu\text{m}$ and random position along the boundary. The particles cannot be released very close to the walls of inlet because the wall induced lift force gives nonphysical values if distance between wall and particle is less than the particle radius, so only the middle 80% of the inlet boundary is used to generate the particles. To release particles with random diameters, **Specify particle diameter** needs to be chosen from the **Particle size distribution** options in the **Additional Variables** setting of the Particle Tracing for Fluid Flow interface.

The asymmetric flow of fluid in the drain outlet is easily captured using an **Outlet** feature in the Laminar Flow interface. A boundary condition of zero gauge pressure is applied to each outlet. The pressure drop along the drain channel is faster because of its smaller length which results in a higher flow rate compared to the other branches.

For the smallest particles in this model, the relaxation time is approximately $0.16 \text{ }\mu\text{s}$, which is about three million times smaller than the maximum output time. However, the default time-dependent solver for most particle tracing models is a second-order implicit method that handles numerically stiff problems rather well, even when taking time steps that are larger than the relaxation time. Nevertheless, the smallest particles are still the main driver of the computational cost of transient inertial particle tracing simulations.

Results and Discussion

The velocity field and the streamlines are shown in Figure 3. The streamlines closer to the upper wall of the microchannel lead to the upper branches whereas those closer to the lower wall lead toward lower branches or into the drain. The effect of AsPFF is seen in the greater density of streamlines in the drain where the flow rate is higher due to its lower viscous losses compared to the other branches.

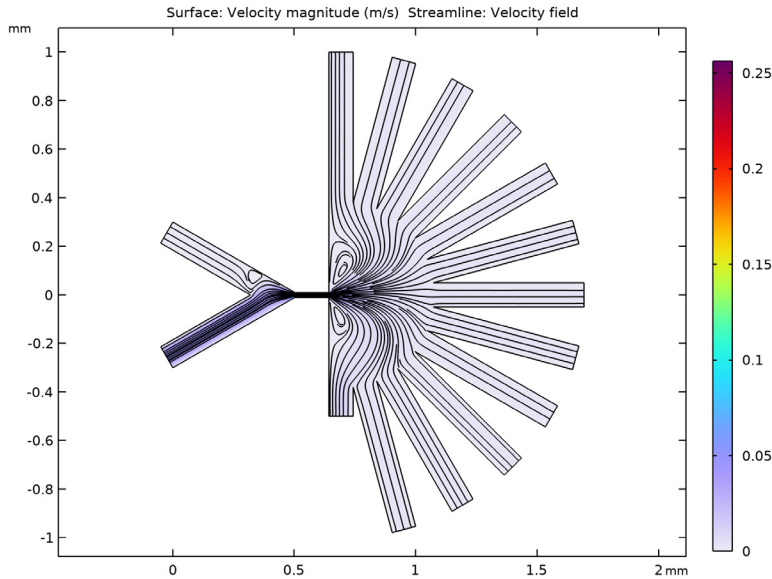


Figure 3: Velocity field and streamlines for the fluid flow

When the fluid from two inlets are squeezed through the microchannel, the particles are sorted based on their size as the lift force pushes the larger particle farther from the wall in perpendicular direction to the flow. When the particles exit the microchannel, they flow with the fluid toward different outlets based on their lateral position in the channel and their inertia (Figure 4).

When zooming in very close to the ends of the microchannel, most notably the right end, some particle trajectories may appear to pass through the channel walls instead of exiting through the end of the microchannel. This is just an artifact of the way that the lines in particle trajectory plots are displayed, with interpolation between a discrete number of output times. If the number of output times were significantly increased, the particles would appear to exit the channel normally, but the file size would be much larger.

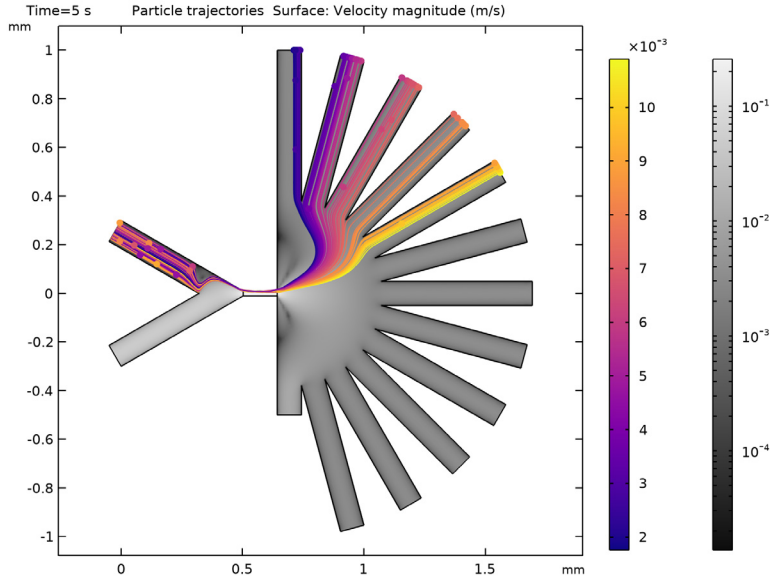


Figure 4: Particle trajectories colored by particle size top of fluid velocity field. The background shows the fluid velocity magnitude on a logarithmic scale

Figure 5 is a histogram that shows the number of particles exiting from each outlet. In this simulation, the particles exit from first 5 outlet branches. In addition, Figure 6 is a 2D histogram that shows the particle sizes exiting from each outlet.

These two histograms give a clear quantitative picture of sorting of particles using the AsPFF method. Because the model is fully parameterized, a natural extension would be to use a **Parametric Sweep** or some optimization functionality to utilize a larger number of outlet branches, while simultaneously reducing the range of particle diameters that exit through any single branch.

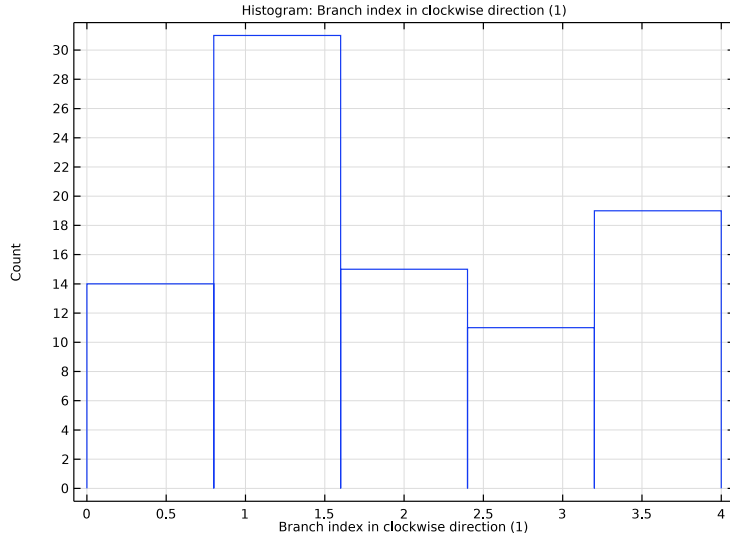


Figure 5: Histogram of number of particles exiting through each outlet branch.

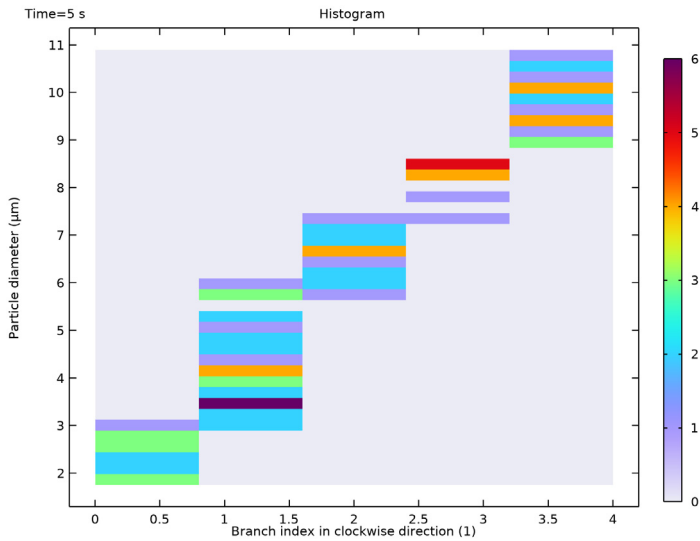


Figure 6: 2D histogram showing the range of particle size exiting through each outlet branch.

References

1. D.R. Gossett, W.M. Weaver, A.J. Mach, S.C. Hur, H.T.K. Tse, W. Lee, H. Amini, and D. Di Carlo, “Label-free cell separation and sorting in microfluidic systems,” *Anal. Bioanal. Chem.*, vol. 397, p. 3249–3267, 2010.
2. J. Takagi, M. Yamada, M. Yasuda, and M. Seki, “Continuous particle separation in a microchannel having asymmetrically arranged multiple branches,” *Lab Chip*, vol. 5, p. 778–784, 2005.

Application Library path: Particle_Tracing_Module/Fluid_Flow/
pinched_flow_fractionation

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  **Study**.
- 5 In the **Select Study** tree, select **General Studies>Stationary**.
- 6 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 In the table, enter the following settings:

Name	Expression	Value	Description
V1	25[u1/h]	6.9444E-12 m³/s	Flow rate in inlet 1
V2	1200[u1/h]	3.3333E-10 m³/s	Flow rate in inlet 2
rho_p	1300[kg/m³]	1300 kg/m³	Particle density
d_min	1.5[um]	1.5E-6 m	Minimum particle diameter
d_max	11[um]	1.1E-5 m	Maximum particle diameter

GEOMETRY 1

Insert the prepared geometry sequence from file. You can read the instructions for creating the geometry in the appendix.

- 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 `pinched_flow_fractionation_geom_sequence.mph`.
- 3 In the **Geometry** toolbar, click  **Build All**. The geometry should look like [Figure 1](#).

DEFINITIONS

Variables 1

- 1 In the **Home** toolbar, click  **Variables** and choose **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
dphi	$\pi[\text{rad}]/N_b$	rad	Angle between adjacent branches
phi0	$d\phi i/2$	rad	Half of the angle
dx	$x - C_r$	m	Distance from center of rotation along x-axis
phi	$\text{atan2}(dx, y)$	rad	Angle with positive y-axis around center of rotation
exit	$\text{floor}((\phi - \phi_0)/d\phi i) + 1$		Branch index in clockwise direction

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.

LAMINAR FLOW (SPF)

First the Laminar Flow interface is used to calculate the velocity and pressure fields resulting from fluid entering through two inlets at different flow rates.

Inlet 1

- 1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Laminar Flow (spf)** and choose **Inlet**.
- 2 Select Boundaries 3, 5, and 6 only, the edges in the upper left corner of the model geometry.
- 3 In the **Settings** window for **Inlet**, locate the **Boundary Condition** section.
- 4 From the list, choose **Fully developed flow**.
- 5 Locate the **Fully Developed Flow** section. Click the **Flow rate** button.
- 6 In the V_0 text field, type V1.
- 7 In the D_z text field, type Wi.

The channel has a square cross section so the out-of-plane thickness in this case equals the in-plane width.

Inlet 2

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Inlet**.
- 2 Select Boundary 1 only, the edge in the lower left corner of the model geometry.
- 3 In the **Settings** window for **Inlet**, locate the **Boundary Condition** section.
- 4 From the list, choose **Fully developed flow**.
- 5 Locate the **Fully Developed Flow** section. Click the **Flow rate** button.
- 6 In the V_0 text field, type V2.
- 7 In the D_z text field, type Wi.

Outlet 1

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

- 2 Select Boundaries 15, 19, 28, 29, and 46–54 only. This selection includes the ends of all of the outflow branches on the right side of the geometry.

MESH 1

- 1 In the **Model Builder** window, under **Component 1 (comp1)** click **Mesh 1**.
- 2 In the **Settings** window for **Mesh**, locate the **Sequence Type** section.
- 3 From the list, choose **User-controlled mesh**.

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 **Predefined** list, choose **Extra fine**.

Size 1

- 1 In the **Model Builder** window, click **Size 1**.
- 2 In the **Settings** window for **Size**, locate the **Element Size** section.
- 3 From the **Predefined** list, choose **Extra fine**.

Size 2

- 1 In the **Mesh** toolbar, click  **Sizing** and choose **Size**.
- 2 Drag and drop below **Size 1**.
- 3 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.
- 4 From the **Geometric entity level** list, choose **Boundary**.
- 5 Select Boundary 11 only, the entrance of the microchannel.
- 6 Locate the **Element Size** section. 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 0.1 [um].
- 9 Click  **Build All**. The mesh should look like [Figure 2](#).

STUDY 1

Step 1: Stationary

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

RESULTS

Surface

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

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

3 Click  **Change Color Table**.

4 In the **Color Table** dialog box, select **Rainbow>Prism** in the tree.

5 Click **OK**.

Velocity (spf)

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

Streamline I

1 In the **Velocity (spf)** toolbar, click  **Streamline**.

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

3 From the **Positioning** list, choose **Magnitude controlled**.

4 In the **Minimum distance** text field, type 0.001.

5 In the **Maximum distance** text field, type 0.01.

6 In the **Velocity (spf)** toolbar, click  **Plot**. The plot should look like [Figure 3](#).

ADD PHYSICS

Next, calculate the particle trajectories using the Particle Tracing for Fluid Flow interface.

1 In the **Home** toolbar, click  **Add Physics** to open the **Add Physics** window.

2 Go to the **Add Physics** window.

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

4 Click **Add to Component I** in the window toolbar.

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

PARTICLE TRACING FOR FLUID FLOW (FPT)

1 In the **Settings** window for **Particle Tracing for Fluid Flow**, locate the **Additional Variables** section.

2 From the **Particle size distribution** list, choose **Specify particle diameter**.

Particle Properties I

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

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 ρ_{p_p} .

Inlet 1

- 1 In the **Physics** toolbar, click  **Boundaries** and choose **Inlet**.
- 2 Select Boundary 5 only.
- 3 In the **Settings** window for **Inlet**, locate the **Release Times** section.
- 4 Click  **Range**.
- 5 In the **Range** dialog box, type 0 in the **Start** text field.
- 6 In the **Step** text field, type 0.05.
- 7 In the **Stop** text field, type 5.
- 8 Click **Replace**.
- 9 In the **Settings** window for **Inlet**, locate the **Initial Position** section.
- 10 From the **Initial position** list, choose **Random**.
- 11 Locate the **Initial Velocity** section. From the **u** list, choose **Velocity field (spf)**.
- 12 Locate the **Initial Particle Diameter** section. From the **Distribution function** list, choose **Uniform**.
- 13 From the **Sampling from distribution** list, choose **Random**.
- 14 In the $d_{p,min}$ text field, type d_{min} .
- 15 In the $d_{p,max}$ text field, type d_{max} .

The number of particles per release and the number of particle diameter values are both 1 because these numbers multiply with each other and with the number of release times. So in this case, only one particle is released per release time. If a large number is used in any one or both of the values, then it could result in an inordinate number of particles being released.

Drag Force 1

- 1 In the **Physics** toolbar, click  **Domains** and choose **Drag Force**.
- 2 In the **Settings** window for **Drag Force**, locate the **Domain Selection** section.
- 3 From the **Selection** list, choose **All domains**.
- 4 Locate the **Drag Force** section. From the **u** list, choose **Velocity field (spf)**.

Lift Force 1

- 1 In the **Physics** toolbar, click  **Domains** and choose **Lift Force**.
- 2 Select Domain 2 only.
- 3 In the **Settings** window for **Lift Force**, locate the **Lift Force** section.
- 4 From the **Lift law** list, choose **Wall induced**.

- 5 Locate the **Parallel Boundary 1** section. Click to select the  **Activate Selection** toggle button.
- 6 Select Boundary 13 only.
- 7 Locate the **Parallel Boundary 2** section. Click to select the  **Activate Selection** toggle button.
- 8 Select Boundary 12 only.
- 9 Locate the **Lift Force** section. From the **u** list, choose **Velocity field (spf)**.

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> Time Dependent**.
- 4 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Laminar Flow (spf)**.
- 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: Time Dependent

- 1 In the **Settings** window for **Time Dependent**, locate the **Study Settings** section.
- 2 Click  **Range**.
- 3 In the **Range** dialog box, type 1e-3 in the **Step** text field.
- 4 In the **Stop** text field, type 5.
- 5 Click **Replace**.
- 6 In the **Settings** window for **Time Dependent**, click to expand the **Values of Dependent Variables** section.
- 7 Find the **Values of variables not solved for** subsection. From the **Settings** list, choose **User controlled**.
- 8 From the **Method** list, choose **Solution**.
- 9 From the **Study** list, choose **Study 1, Stationary**.
- 10 In the **Home** toolbar, click  **Compute**.

RESULTS

Particle Trajectories I

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

Color Expression I

- 1 In the **Model Builder** window, expand the **Particle Trajectories I** node, then click **Color Expression I**.
- 2 In the **Settings** window for **Color Expression**, locate the **Expression** section.
- 3 In the **Expression** text field, type `fpt.dp`.
- 4 Locate the **Coloring and Style** section. Click  **Change Color Table**.
- 5 In the **Color Table** dialog box, select **Thermal>Plasma** in the tree.
- 6 Click **OK**.

Particle Trajectories (fpt)

In the **Model Builder** window, under **Results** click **Particle Trajectories (fpt)**.

Surface I

- 1 In the **Particle Trajectories (fpt)** toolbar, click  **Surface**.
- 2 In the **Settings** window for **Surface**, locate the **Data** section.
- 3 From the **Dataset** list, choose **Study I/Solution I (sol1)**.
- 4 Locate the **Coloring and Style** section. Click  **Change Color Table**.
- 5 In the **Color Table** dialog box, select **Linear>GrayPrint** in the tree.
- 6 Click **OK**.
- 7 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.
- 8 From the **Scale** list, choose **Logarithmic**.
- 9 In the **Particle Trajectories (fpt)** toolbar, click  **Plot**. The plot should look like [Figure 4](#).

Animation I

- 1 In the **Particle Trajectories (fpt)** toolbar, click  **Animation** and choose **Player**.
- 2 In the **Settings** window for **Animation**, locate the **Frames** section.
- 3 In the **Number of frames** text field, type 100.

- 4 Click the  **Play** button in the **Graphics** toolbar.

1D Histogram

- 1 In the **Home** toolbar, click  **Add Plot Group** and choose **1D Plot Group**.
- 2 In the **Settings** window for **1D Plot Group**, type 1D Histogram in the **Label** text field.
- 3 Locate the **Data** section. From the **Dataset** list, choose **Particle I**.
- 4 From the **Time selection** list, choose **Last**.

Histogram 1

- 1 In the **1D Histogram** toolbar, click  **Histogram**.
- 2 In the **Settings** window for **Histogram**, locate the **Expression** section.
- 3 In the **Expression** text field, type `exit`.
- 4 Locate the **Bins** section. In the **Number** text field, type 5.
- 5 Locate the **Output** section. From the **Function** list, choose **Discrete**.

Filter 1

- 1 In the **1D Histogram** toolbar, click  **Filter**.
- 2 In the **Settings** window for **Filter**, locate the **Point Selection** section.
- 3 In the **Logical expression for inclusion** text field, type `x>Pb`.
- 4 In the **1D Histogram** toolbar, click  **Plot**. The plot should look like [Figure 5](#).

2D Histogram

- 1 In the **Home** toolbar, click  **Add Plot Group** and choose **2D Plot Group**.
- 2 In the **Settings** window for **2D Plot Group**, type 2D Histogram in the **Label** text field.
- 3 Locate the **Data** section. From the **Dataset** list, choose **Particle I**.

Histogram 1

- 1 In the **2D Histogram** toolbar, click  **More Plots** and choose **Histogram**.
- 2 In the **Settings** window for **Histogram**, locate the **x-Expression** section.
- 3 In the **Expression** text field, type `exit`.
- 4 Locate the **y-Expression** section. In the **Expression** text field, type `fpt.dp`.
- 5 From the **Unit** list, choose **μm**.
- 6 Locate the **Bins** section. Find the **x bins** subsection. In the **Number** text field, type 5.
- 7 Find the **y bins** subsection. In the **Number** text field, type 40.
- 8 Locate the **Output** section. From the **Function** list, choose **Discrete**.
- 9 Locate the **Coloring and Style** section. Click  **Change Color Table**.

10 In the **Color Table** dialog box, select **Rainbow>Prism** in the tree.

11 Click **OK**.

Filter 1

1 In the **2D Histogram** toolbar, click  **Filter**.

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

3 In the **Logical expression for inclusion** text field, type $x > Pb$.

4 In the **2D Histogram** toolbar, click  **Plot**. The plot should look like [Figure 6](#).

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

Appendix: Geometry 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

Geometry parameters

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

2 In the **Settings** window for **Parameters**, type Geometry parameters 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 `pinched_flow_fractionation_geom_sequence_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 L_i .
- 4 In the **Height** text field, type W_i .
- 5 Locate the **Position** section. In the **y** text field, type $-\sin(HA_i) * L_i$.
- 6 Locate the **Rotation Angle** section. In the **Rotation** text field, type HA_i .
- 7 Click  **Build Selected**.

Mirror 1 (mir1)

- 1 In the **Geometry** toolbar, click  **Transforms** and choose **Mirror**.
- 2 Select the object **r1** 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.
- 7 Click  **Build Selected**.
- 8 Click the  **Zoom Extents** button in the **Graphics** toolbar.

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 $-0.1 * W_i * \cos(A_i)$.
- 4 In the **y** text field, type $\sin(HA_i) * L_i - 0.1 * W_i * \sin(A_i)$.
- 5 Click  **Build Selected**.

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 $-0.9 * W_i * \cos(A_i)$.
- 4 In the **y** text field, type $\sin(HA_i) * L_i - 0.9 * W_i * \sin(A_i)$.
- 5 Click  **Build Selected**.

Union 1 (uni1)

- 1 In the **Geometry** toolbar, click  **Booleans and Partitions** and choose **Union**.
- 2 Click in the **Graphics** window and then press Ctrl+A to select all objects.
- 3 In the **Settings** window for **Union**, click  **Build Selected**.

Delete Entities 1 (del1)

- 1 In the **Geometry** toolbar, click  **Delete**.
- 2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.
- 4 On the object **unil**, select Boundaries 9–11, 13, 15, and 16 only.
- 5 Click  **Build Selected**.

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 Pc.
- 5 In the **y** text field, type Hwc.
- 6 Locate the **Endpoint** section. From the **Specify** list, choose **Coordinates**.
- 7 In the **x** text field, type Pc.
- 8 In the **y** text field, type -Hwc.
- 9 Click  **Build Selected**.

Partition Objects 1 (par1)

- 1 In the **Geometry** toolbar, click  **Booleans and Partitions** and choose **Partition Objects**.
- 2 Select the object **del1** only.
- 3 In the **Settings** window for **Partition Objects**, locate the **Partition Objects** section.
- 4 Find the **Tool objects** subsection. Click to select the  **Activate Selection** toggle button.
- 5 Select the object **ls1** only.
- 6 Click  **Build Selected**.

Delete Entities 2 (del2)

- 1 In the **Geometry** toolbar, click  **Delete**.
- 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 **par1**, select Domain 2 only.
- 5 Click  **Build Selected**.

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 Lc.
- 4 In the **Height** text field, type Wc.
- 5 Locate the **Position** section. In the **x** text field, type Pc.
- 6 In the **y** text field, type -Hwc.
- 7 Click  **Build Selected**.

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 Wb.
- 4 In the **Height** text field, type Lb.
- 5 Locate the **Position** section. In the **x** text field, type Pb.
- 6 Click  **Build Selected**.

Rotate 1 (rot1)

- 1 In the **Geometry** toolbar, click  **Transforms** and choose **Rotate**.
- 2 Select the object **r3** only.
- 3 In the **Settings** window for **Rotate**, locate the **Rotation** section.
- 4 Click  **Range**.
- 5 In the **Range** dialog box, type 0 in the **Start** text field.
- 6 In the **Step** text field, type -180/Nb.
- 7 In the **Stop** text field, type -180.
- 8 Click **Replace**.
- 9 In the **Settings** window for **Rotate**, locate the **Center of Rotation** section.
- 10 In the **x** text field, type Cr.
- 11 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** check box.
- 12 From the **Show in physics** list, choose **Off**.
- 13 Click  **Build Selected**.
- 14 Click the  **Zoom Extents** button in the **Graphics** toolbar.

Union 2 (uni2)

- 1 In the **Geometry** toolbar, click  **Booleans and Partitions** and choose **Union**.
- 2 In the **Settings** window for **Union**, locate the **Union** section.
- 3 From the **Input objects** list, choose **Rotate 1**.
- 4 Clear the **Keep interior boundaries** check box.
- 5 Click  **Build Selected**.

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 W_b .
- 4 In the **Height** text field, type $L_d + 0.01 \cdot L_b$.
- 5 Locate the **Position** section. In the **x** text field, type P_b .
- 6 In the **y** text field, type $-L_b \cdot 1.01$.
- 7 Click  **Build Selected**.

Difference 1 (dif1)

- 1 In the **Geometry** toolbar, click  **Booleans and Partitions** and choose **Difference**.
- 2 Select the object **uni2** only.
- 3 In the **Settings** window for **Difference**, locate the **Difference** section.
- 4 Find the **Objects to subtract** subsection. Click to select the  **Activate Selection** toggle button.
- 5 Select the object **r4** only.
- 6 Click  **Build Selected**.

Form Union (fin)

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

