Created in COMSOL Multiphysics 5.6

Particle Trajectories in a Laminar Static Mixer

Introduction

In static mixers, also called motionless or in-line mixers, a fluid is pumped through a pipe containing stationary blades. This mixing technique is particularly well suited for laminar flow mixing because it generates only small pressure losses in this flow regime. This example studies the flow in a twisted-blade static mixer. It evaluates the mixing performance by calculating the trajectory of suspended particles through the mixer.

Model Definition

This model studies the mixing of particles (the *discrete phase*) in a channel containing water (the *continuous phase*) at room temperature. The geometry consists of a tube with three fixed, twisted blades of alternating rotations (Figure 1).

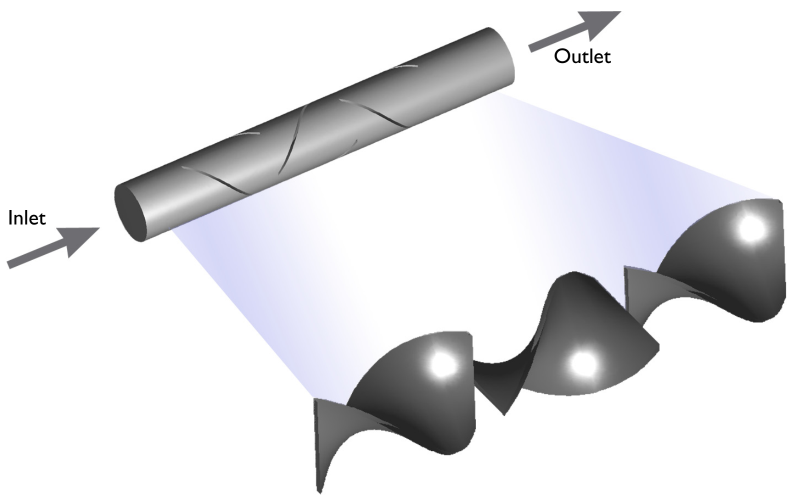

Figure 1: Depiction of a laminar static mixer containing three blades with alternating rotations.

The tube's radius, r_a, is 3 mm, the length is $14r_a$, and the length of each blade is $3r_a$. At the **Inlet**, a Poiseuille flow is specified with an average velocity of 1 cm/s. At the **Outlet**, a uniform pressure of 0 Pa (relative to atmosphere) is specified.

The Laminar Flow interface is used to solve for the fluid velocity and pressure:

$$\rho(\mathbf{u}\cdot\nabla)\mathbf{u} = \nabla\cdot[-p\mathbf{I}+\mu(\nabla\mathbf{u}+(\nabla\mathbf{u})^T)]$$
$$\nabla\cdot\mathbf{u} = 0$$

where

- μ is the dynamic viscosity (SI unit: $\mathrm{kg/(m{\cdot}s)}$),
- $\mathbf{u}$ is the fluid velocity (SI unit: $\mathrm{m/s}$),
- ρ is the fluid density (SI unit: $\mathrm{kg/m^3}$), and
- p is the pressure (SI unit: Pa).

The motion of small particles in a fluid is governed by Newton's second law,

$$\frac{\mathrm{d}\mathbf{q}}{\mathrm{d}t} = \mathbf{v}$$

$$\frac{\mathrm{d}}{\mathrm{d}t}(m_\mathrm{p}\mathbf{v}) = \mathbf{F}_\mathrm{t}$$

where

- $\mathbf{q}$ is the particle position (SI unit: m),
- $\mathbf{v}$ is the particle velocity (SI unit: $\mathrm{m/s}$),
- m_p is the particle mass (SI unit: kg), and
- $\mathbf{F}_\mathrm{t}$ is the total force (SI unit: N).

This equation can be solved using a set of second-order equations for the position vector components, or two sets of coupled first-order equations for the position and velocity components. However, in this example, a simplifying assumption is used to solve a single set of first-order equations instead.

In this example the total force is dominated by the **Drag Force** $\mathbf{F}_\mathrm{D}$ (SI unit: N). Because the particles are very small and the particle velocity relative to the fluid is not too large, the Stokes drag law is applicable,

$$\mathbf{F}_\mathrm{D} = 3\pi\mu d_\mathrm{p}(\mathbf{u} - \mathbf{v})$$

where

- $\mathbf{u}$ (SI unit: $\mathrm{m/s}$) is the fluid velocity,
- μ (SI unit: Pa s) is the fluid dynamic viscosity, and
- d_p (SI unit: m) is the particle diameter.

In this example, the particle diameter is $0.5\ \mu\mathrm{m}$ and the surrounding fluid has a dynamic viscosity of $\mu = 10^{-3}$ Pa s.

TIME SCALES IN INERTIAL PARTICLE TRACING

The characteristic time scale for a particle to accelerate due to the Stokes drag is

$$\tau_p = \frac{\rho_p d_p^2}{18\mu}$$

In this example, the particle density is $\rho_p = 2200\ \mathrm{kg/m^3}$. Thus the characteristic time scale, sometimes called the particle velocity response time or the Lagrangian time scale, is about $\tau_p = 30$ ns. In comparison, the fastest particles in this model take about 2 s to reach the outlet. Furthermore, the time at which a given particle might encounter a significant fluid velocity gradient is not known *a priori*, and this time is likely to differ for all particles. It would therefore take something on the order of 100 million timesteps to fully resolve the acceleration of particles in the nonuniform velocity field. This would be a very time-consuming way to model such a simple particle-laden flow.

A less computationally expensive approach is to use the first-order formulation called **Newtonian, ignore inertial terms** in the Particle Tracing for Fluid Flow interface. In this formulation, the particle velocity is assigned at each timestep such that the net force on each particle is zero. In this example, the only force on each particle is the **Drag Force**, so the particle velocity is automatically defined such that $\mathbf{F}_D = \mathbf{0}$. This recovers the trivial solution of tracer particles that follow fluid streamlines, $\mathbf{u} = \mathbf{v}$. If the model were later extended to include other forces, such as gravity, electromagnetic, or thermophoretic forces, then this formulation would assign the particle velocity so that the drag force on each particle was perfectly counterbalanced by the other applied forces.

Notes on COMSOL Implementation

There are 3000 particles released. The number density of released particles at the **Inlet** boundary is proportional to the fluid velocity at the inlet. This means that there are more particles released at the center of the boundary where the inlet velocity magnitude is highest and fewer particles released near the edges where the velocity magnitude is low.

The model is solved in two stages. First, the fluid velocity and pressure are solved for using a **Stationary** study step. Then the particle trajectories are computed using a **Time Dependent** study step. The solution from the **Stationary** study is used to define the fluid velocity for the purpose of exerting a **Drag Force** on the particles, but the presence of particles is not considered when modeling the fluid. This constitutes a unidirectional coupling, which is valid for sparse flows where the volume fraction of particles in the fluid is very small and the momentum imparted onto the fluid by the particles can be neglected.

Also note the following assumptions of the particle tracing implementation:

- Particles do not displace the fluid in the volume they occupy.
- Interaction between model particles is neglected, so the distance between particles can be less than the particle diameter.
- When solving the equations of motion of each particle, the coordinates of the particle center are used. A particle is considered to hit a surface (such as a mixing blade, the outer wall of the mixer, or the outlet) when the trajectory of its center intersects the surface.

Results and Discussion

The particle trajectories are plotted in Figure 2. The color along the trajectories indicates the total shear rate at the particle position. Because of discretization error in the fluid velocity field, and because the **Time Dependent** solver to compute the particle positions uses discrete time steps, particles may occasionally hit one of the mixer blades or the outer boundary of the flow. In addition, some particles pass close enough to the mixing blades that the fluid velocity becomes extremely slow, so the particles don't reach the outlet even by the final time in the study.

The transmission probability is the ratio of the number of particles that reach the outlet to the total number of particles released. In this example the transmission probability is computed by the **Particle Counter** feature, which records the number of particles that reach the **Outlet** and expresses it as a fraction of the total number of released particles.

For this specific configuration the transmission probability is about 0.80. This means that about 20% of the particles remain trapped in the mixer. If the study were run for more than 5 seconds then this transmission probability would gradually increase.

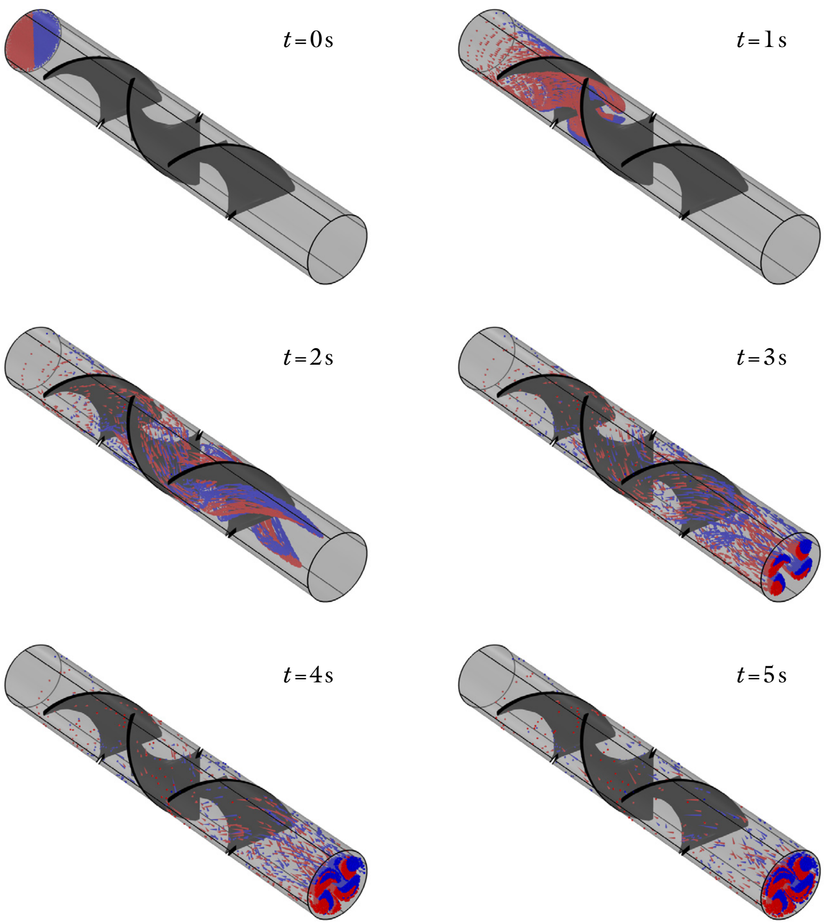

Figure 2: Particle positions in the mixer at different solution times. The comet tails indicate the particle velocity. The particles are colored red if they were released with initial position $x < 0$; otherwise they are colored blue.

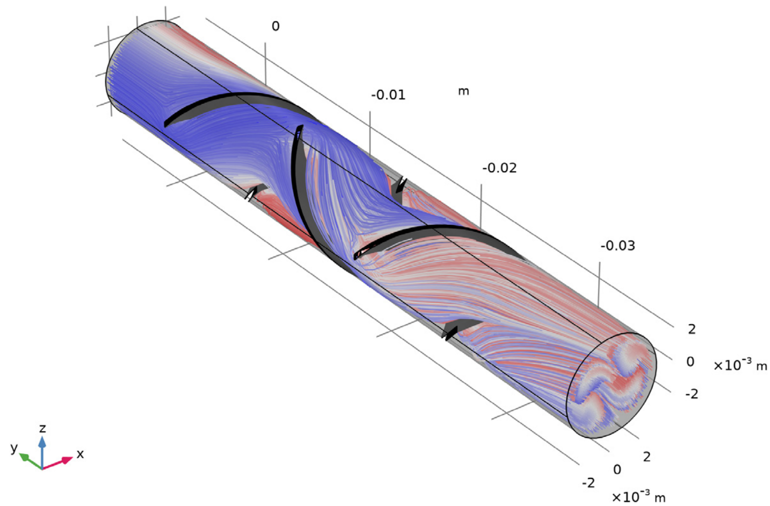

Figure 3: Plot of the particle trajectories inside the laminar mixer. The color is the shear rate.

One useful way to visualize how the particles mix is to use a **Poincaré Map** plot. The **Poincaré Map** places a colored dot where each particle passes through a cut plane (also known as a Poincaré section). The **Cut Plane** dataset can easily be used to define a single Poincaré section or multiple parallel planes.

In Figure 4, the location of the particles at 6 Poincaré sections are shown. The color represents the location of the particle at its initial position. Particles colored red had an initial position of $x < 0$ and particles colored blue had an initial position of $x > 0$. The `at` operator is used to mark the particles with the color of their initial position. As the particles move downstream (in the negative y-direction), they begin to mix together.

By the end of the mixer, the particles have not yet mixed completely; there are still significant pockets of only red and only blue particles.

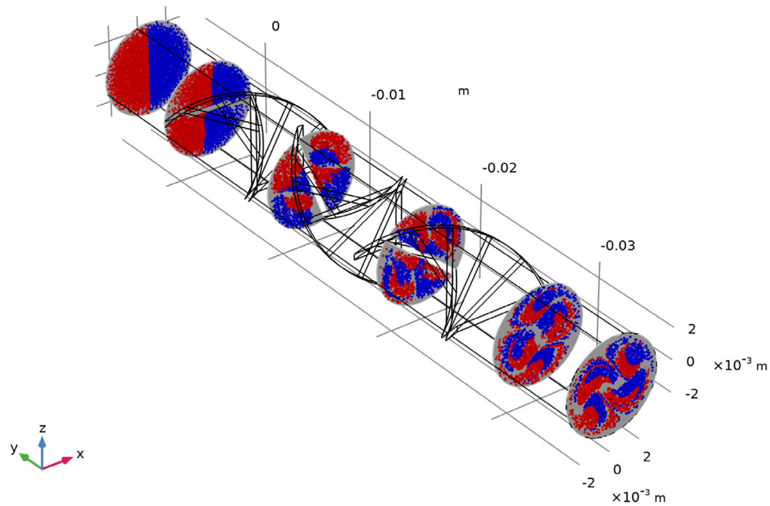

Figure 4: Poincaré maps of the particle trajectories at different Poincaré sections. The color is a logical expression indicating which particles had an initial position at $x < 0$.

A **Phase Portrait** plot can also be used to visualize the mixing performance. A single **Phase Portrait** gives a snapshot of all particles at the same time whereas the **Poincaré Map** shows the intersection points of particles with a surface, even if these intersections occur at different times. The two axes in the phase portrait can be assigned any particle variables, and are often used to plot velocity or momentum versus position. In Figure 5 the horizontal axis represents the x-coordinates of the particles, and the vertical axis represents the z-coordinates of the particles. Phase portraits are given at 6 different solution times. At the final time (t = 5 s) there are still pockets of exclusively blue or red particles, indicating the particles are not yet completely mixed.

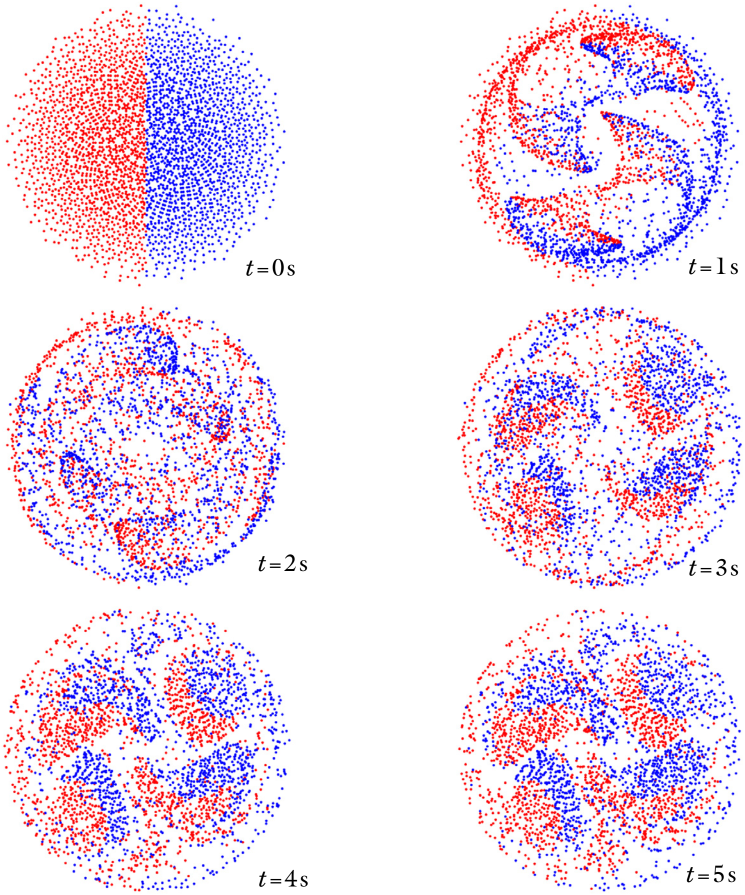

Figure 5: Plot of the particle position in the xz-plane at different times. The red particles had an initial position of $x < 0$ and the blue particles had an initial position of $x > 0$.

Reference

1. R. Perry and D. Green, *Perry's Chemical Engineering Handbook*, 7th ed., McGraw-Hill, 1997.

Application Library path: `Particle_Tracing_Module/Fluid_Flow/laminar_mixer_particle`

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 **3D**.

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**.

GEOMETRY 1

The mixer geometry is quite complicated so start by importing it from a file.

Import 1 (imp1)

1 In the **Home** toolbar, click **Import**.

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

3 Click **Browse**.

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

5 Click **Import**.

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
Ra	3[mm]	0.003 m	Tube radius
u_av	1[cm/s]	0.01 m/s	Mean velocity

Add some explicit selections on the geometry. These will be useful during results processing.

DEFINITIONS

Outer Surfaces

1. In the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for **Explicit**, type `Outer Surfaces` in the **Label** text field.
3. Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4. Select the **Group by continuous tangent** check box.
5. Select Boundaries 1, 8, 19–23, 48–50, 57, and 58 only. That is, select the inlet, outlet, and outer cylindrical walls. This is much easier while the **Group by continuous tangent** check box is selected.

Blade Surfaces

1. In the **Definitions** toolbar, click **Explicit**.
2. In the **Settings** window for **Explicit**, type `Blade Surfaces` in the **Label** text field.
3. Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.
4. Select Boundaries 2–7, 9–18, 24–47, and 51–56 only. Instead of clicking so many outer boundaries individually, an easier approach is to use the **Select box** button on the Graphics toolbar. Selecting the **Group by continuous tangent** check box may also be helpful.

MATERIALS

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	1000	kg/m³	Basic
Dynamic viscosity	mu	1e-3	Pa·s	Basic

LAMINAR FLOW (SPF)

Now add an expression for the inflow velocity which is parabolic.

Inlet 1

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

2 Select Boundary 23 only.

3 In the **Settings** window for **Inlet**, locate the **Velocity** section.

4 In the U_0 text field, type `2*(1-(x^2+z^2)/Ra^2)*u_av`.

The boundary condition which was just added was rather complicated but necessary to get a fully developed flow profile. The CFD, Microfluidics, and Plasma modules all have a special **Laminar inflow** boundary condition which ensures a fully developed flow profile at the inlet. It is not necessary to enter a complicated expression for the velocity profile, just the average velocity or flowrate.

Outlet 1

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

2 Select Boundary 20 only.

MESH 1

The mesh needs to be quite fine to ensure that the particle motion is accurate through the modeling domain. In this case, take care to ensure that the mesh is fine on the mixing blades.

Free Triangular 1

1 In the **Mesh** toolbar, click **Boundary** and choose **Free Triangular**.

2 Click the **Wireframe Rendering** button in the **Graphics** toolbar.

3 Select Boundaries 5, 16–18, and 53–55 only.

Size 1

1 Right-click **Free Triangular 1** and choose **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 **Extremely fine**.

Size

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

2 In the **Settings** window for **Size**, locate the **Element Size** section.

3 From the **Predefined** list, choose **Extremely fine**.

4 Click the **Custom** button.

5 Locate the **Element Size Parameters** section. In the **Curvature factor** text field, type `0.15`.

Free Triangular 2

1 In the **Mesh** toolbar, click **Boundary** and choose **Free Triangular**.

2 Select Boundary 23 only.

Size 1

1 Right-click **Free Triangular 2** and choose **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**.

Free Tetrahedral 1

1 In the **Mesh** toolbar, click **Free Tetrahedral**.

2 In the **Settings** window for **Free Tetrahedral**, click **Build All**.

STUDY 1

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

RESULTS

Velocity (spf)

Now that the flow field has been computed, add the interface to compute the particle trajectories.

ADD PHYSICS

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 Find the **Physics interfaces in study** subsection. In the table, clear the **Solve** check box for **Study 1**.

5 Click **Add to Component 1** in the window toolbar.

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

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.

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**.

The drag force feature should get the fluid velocity field and viscosity from the **Laminar flow** interface.

Drag Force 1

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

2 Select Domain 1 only.

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

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

5 From the μ list, choose **Dynamic viscosity (spf/fp1)**.

6 Locate the **Additional Terms** section. Select the **Include virtual mass and pressure gradient forces** check box.

The goal is to release particles with a number density proportional to the magnitude of the fluid velocity.

Inlet 1

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

2 Select Boundary 23 only.

3 In the **Settings** window for **Inlet**, locate the **Initial Position** section.

4 From the **Initial position** list, choose **Density**.

5 In the N text field, type `3000`.

6 In the ρ text field, type `spf.U`.

Particle Counter 1

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

2 Select Boundary 20 only.

3 In the **Settings** window for **Particle Counter**, locate the **Particle Counter** section.

4 From the **Release feature** list, choose **Inlet 1**.

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 d_{p} text field, type `5E-7[m]`.

STUDY 2

Step 1: Time Dependent

1 In the **Model Builder** window, under **Study 2** click **Step 1: Time Dependent**.

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

3 Click **Range**.

4 In the **Range** dialog box, type `0.2` in the **Step** text field.

5 In the **Stop** text field, type `5`.

6 Click **Replace**.

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

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

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

10 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**.

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

12 From the **Study** list, choose **Study 1, Stationary**.

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

RESULTS

Particle Trajectories (fpt)

1 In the **Settings** window for **3D Plot Group**, locate the **Color Legend** section.

2 Clear the **Show legends** check box.

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

Surface 1

1 Right-click **Particle Trajectories (fpt)** and choose **Surface**.

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

3 In the **Expression** text field, type `1`.

4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

5 From the **Color** list, choose **Black**.

Selection 1

1 Right-click **Surface 1** and choose **Selection**.

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

3 From the **Selection** list, choose **Blade Surfaces**.

Surface 2

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

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

3 In the **Expression** text field, type `1`.

4 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

5 From the **Color** list, choose **Gray**.

Selection 1

1 Right-click **Surface 2** and choose **Selection**.

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

3 From the **Selection** list, choose **Outer Surfaces**.

Transparency 1

In the **Model Builder** window, right-click **Surface 2** and choose **Transparency**.

Particle Trajectories 1

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

2 Find the **Point style** subsection. From the **Type** list, choose **Comet tail**.

Color Expression 1

1 In the **Model Builder** window, expand the **Particle Trajectories 1** node, then click **Color Expression 1**.

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

3 In the **Expression** text field, type `at(0,qx<0)`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **WaveLight**.

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

6 Click the **Go to Default View** button in the **Graphics** toolbar.

Particle Trajectories (fpt)

Select different values from the **Time (s)** list and then click the **Plot** button to display the particle positions at different times. The positions at six different output times are shown in Figure 2.

Particle Trajectories (fpt) 1

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

2 Expand the **Particle Trajectories (fpt) 1** node.

Particle Trajectories 1

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

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

3 Find the **Point style** subsection. From the **Type** list, choose **None**.

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

Color Expression 1

1 In the **Model Builder** window, click **Color Expression 1**.

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

3 In the **Expression** text field, type `at(0,qx)`.

4 In the **Particle Trajectories (fpt) 1** toolbar, click **Plot**. The resulting plot should look like Figure 3.

Global Evaluation 1

1 In the **Results** toolbar, click **Global Evaluation**.

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

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

4 From the **Time selection** list, choose **Last**.

5 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Particle Tracing for Fluid Flow>Particle Counter 1>fpt.pcnt1.alpha - Transmission probability**.

6 Click **Evaluate**.

Cut Plane 1

1 In the **Results** toolbar, click **Cut Plane**.

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

3 From the **Dataset** list, choose **Particle 1**.

4 Locate the **Plane Data** section. From the **Plane** list, choose **xz-planes**.

5 In the **y-coordinate** text field, type `0.006`.

6 Select the **Additional parallel planes** check box.

7 In the **Distances** text field, type `0.006 0.016 0.026 0.036 0.042`.

8 Click **Plot**.

Poincaré Maps

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Poincaré Maps` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Particle 1**.

4 Locate the **Title** section. From the **Title type** list, choose **None**.

Poincaré Map 1

1 In the **Poincaré Maps** toolbar, click **More Plots** and choose **Poincaré Map**.

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

3 From the **Cut plane** list, choose **Cut Plane 1**.

4 Locate the **Coloring and Style** section. Select the **Radius scale factor** check box.

5 In the associated text field, type `0.4`.

6 In the **Poincaré Maps** toolbar, click **Plot**.

Color Expression 1

1 Right-click **Poincaré Map 1** and choose **Color Expression**.

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

3 In the **Expression** text field, type `at(0,qx<0)`.

4 Locate the **Coloring and Style** section. From the **Color table** list, choose **WaveLight**.

5 Clear the **Color legend** check box.

Surface 1

1 In the **Model Builder** window, right-click **Poincaré Maps** and choose **Surface**.

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

3 From the **Dataset** list, choose **Cut Plane 1**.

4 Locate the **Expression** section. In the **Expression** text field, type `1`.

5 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

6 From the **Color** list, choose **Gray**.

7 In the **Poincaré Maps** toolbar, click **Plot**.

8 Click the **Go to Default View** button in the **Graphics** toolbar. The resulting plot should look like Figure 4.

Phase Portrait

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

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

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

4 Locate the **Data** section. From the **Dataset** list, choose **Particle 1**.

Phase Portrait 1

1 In the **Phase Portrait** toolbar, click **More Plots** and choose **Phase Portrait**.

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

3 From the **x-axis** list, choose **Manual**.

4 In the **Expression** text field, type `comp1.qx`.

5 From the **y-axis** list, choose **Manual**.

6 In the **Expression** text field, type `comp1.qz`.

Color Expression 1

1 Right-click **Phase Portrait 1** and choose **Color Expression**.

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

3 Clear the **Color legend** check box.

4 Locate the **Expression** section. In the **Expression** text field, type `at(0,qx<0)`.

5 Locate the **Coloring and Style** section. From the **Color table** list, choose **WaveLight**.

6 In the **Phase Portrait** toolbar, click **Plot**.

Phase Portrait

1 In the **Model Builder** window, click **Phase Portrait**.

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

3 From the **View** list, choose **View 2D 2**.

4 Click **Go to Source**.

View 2D 2

By default the **Phase Portrait** plot scales the coordinate axes so that the plot fits in the **Graphics** window. This is to ensure that the phase portrait is shown clearly even if the two axes correspond to quantities with vastly different orders of magnitude, like position and momentum. In the present case, both axes represent position components, so by selecting **View 2D 2** a reasonable-looking 1:1 aspect ratio is enforced.

Phase Portrait

1 In the **Model Builder** window, expand the **View 2D 2** node, then click **Results> Phase Portrait**.

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

3 From the **Time (s)** list, choose **0**.

4 In the **Phase Portrait** toolbar, click **Plot**.

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

Plot the phase portrait at different solution times by selecting values from the **Time (s)** list. The phase portraits at 1-second intervals are shown in Figure 5.