Inertial Focusing Between Two Parallel Walls

Introduction

As neutrally buoyant particles are carried by a fluid in a cylindrical pipe, they are focused to a ring with a radius of approximately 0.6 times the pipe radius R. The ring of particles, which is caused by a combination of lift and drag forces, is sometimes called the Segré-Silberberg annulus in honor of Segré and Silberberg's oft-cited observations of this effect (Ref. 1). The phenomenon by which particles migrate to this equilibrium position is called inertial focusing. Ho and Leal (Ref. 2) have derived expressions for the forces that cause a similar migration in a two-dimensional parabolic flow between two parallel walls.

In this example, the Particle Tracing for Fluid Flow interface is used to compute the trajectories of particles in a two-dimensional Poiseuille flow. This example uses built-in corrections to the lift force and drag force to account for the presence of the walls. A time dependent study demonstrates that the particles converge toward distances of $0.2D$ from either wall, where D is the distance between the walls, regardless of their initial positions.

Model Definition

The total force on the neutrally buoyant particles in a creeping flow consists of a drag force and lift force, since by definition the gravitational and buoyant forces cancel each other out. The particles experience a drag force given by Stokes' law.

$$\mathbf{F}_D = 6\pi\mu r_p(\mathbf{u} - \mathbf{v})$$

Where r_p (SI unit m) is the particle radius, μ (SI unit Pa s) is the fluid viscosity, $\mathbf{u}$ (SI unit m/s) is the fluid velocity, and $\mathbf{v}$ (SI unit m/s) is the particle velocity. The lift force is given by Ref. 2:

$$\mathbf{F}_L = \rho\frac{r_p^4}{D^2}\beta(\beta G_1(s) + \gamma G_2(s))\mathbf{n}$$

$$\beta = |D(\mathbf{n} \cdot \nabla)\mathbf{u}_p|$$

$$\gamma = \left|\frac{D^2}{2}(\mathbf{n} \cdot \nabla)^2\mathbf{u}_p\right|$$

$$\mathbf{u}_p = (I - (\mathbf{n} \otimes \mathbf{n}))\mathbf{u}$$

where

- I (dimensionless) is the identity matrix,

- **n** (dimensionless) is the wall normal at the nearest point on the reference wall,
- D (SI unit: m) is the distance between the channel walls,
- s is the normalized distance from the particle to the reference wall, divided by D so that $0 < s < 1$ for particles in the channel
- G_1 and G_2 are dimensionless functions of the normalized wall distance, given in Ref. 2. These functions are plotted in Figure 1 and Figure 2.

The lift force only acts in the direction perpendicular to the velocity of the fluid. The spherical particles are further assumed to be small compared to the channel width and rotationally rigid.

The velocity field **u** is first computed using the Laminar Flow physics interface, then coupled to the Particle Tracing for Fluid Flow physics interface using the **Drag Force** node. The **Laminar inflow** boundary condition is used to automatically compute a fully developed fluid velocity profile at the **Inlet** boundary. It is well-known that the fully developed velocity profile for the laminar flow of a Newtonian fluid between two parallel walls is parabolic, so in this example it would have also been possible to enter the analytic expression for the fluid velocity directly. However, the velocity has instead been computed using the Laminar Flow physics interface to demonstrate a workflow that is appropriate for a more general geometry.
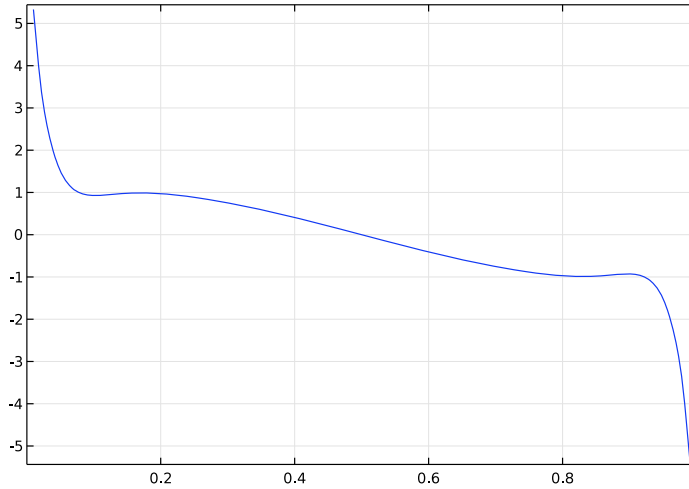
Figure 1: The function $G_1(s)$ used to define the wall-induced lift force.
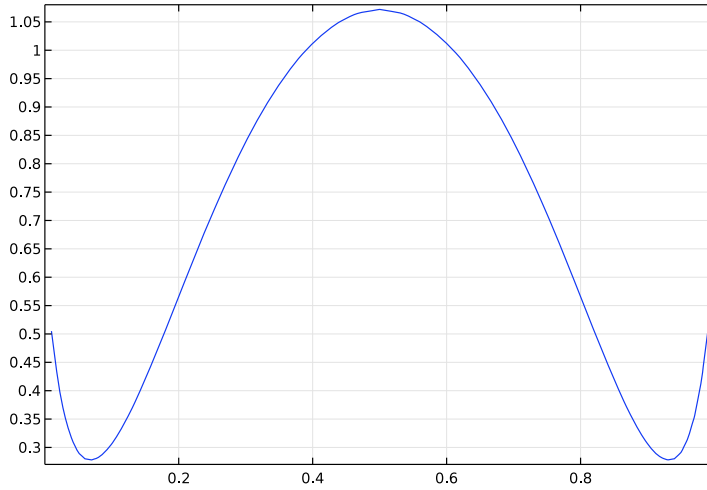
Figure 2: The function $G_2(s)$ used to define the wall-induced lift force.

Results and Discussion

The fluid velocity magnitude is plotted in Figure 3. Note that the plots are not drawn to scale; the channel length is much greater than the channel width. As expected for a laminar flow between two parallel walls, the velocity profile is parabolic.

The trajectories of the neutrally buoyant particles are plotted in Figure 4. The color expression is the y-component of the particle velocity in $\mathrm{mm/s}$. All of the particles seem to approach equilibrium positions at distances of about $0.3D$ on either side of the channel center, where D is the channel width. Particles that are released near the center of the channel take more time to approach these equilibrium positions because they are released where the velocity gradient has the lowest magnitude, so the initial lift force is weaker.
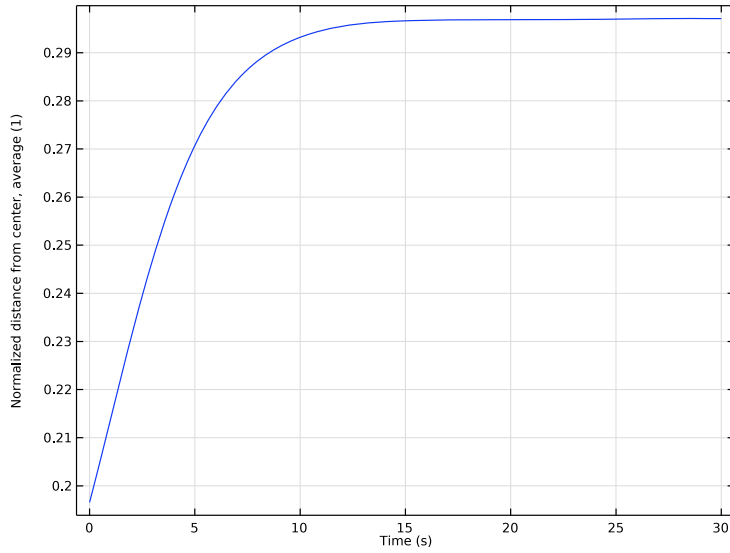
The average and standard deviation of the normalized distance from the particles to the channel center are plotted in Figure 5 and Figure 6, respectively. This confirms that the equilibrium distance from the channel center is about $0.3D$.
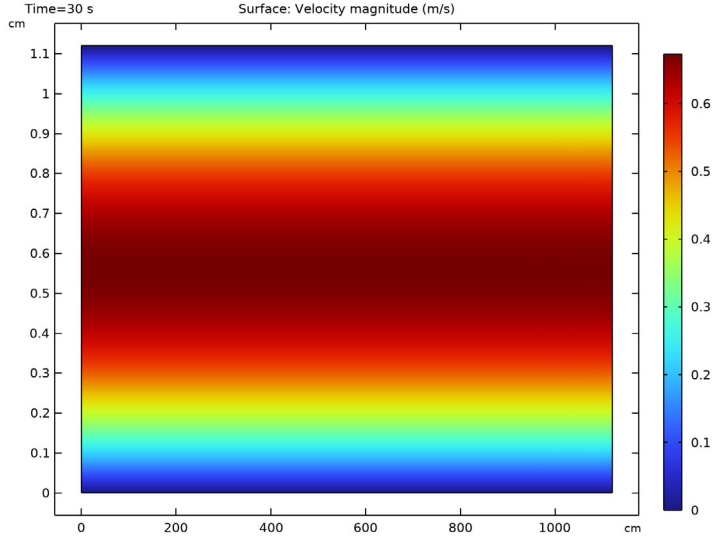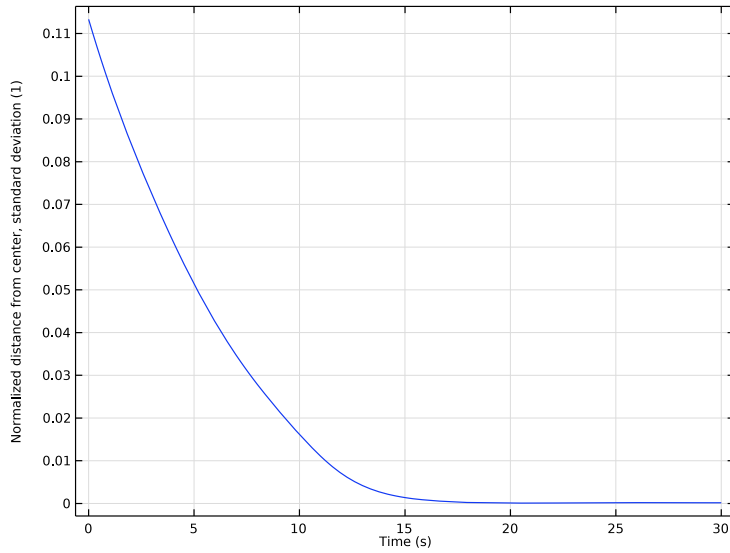
Figure 3: Parabolic fluid velocity profile in a channel bounded by two parallel walls.
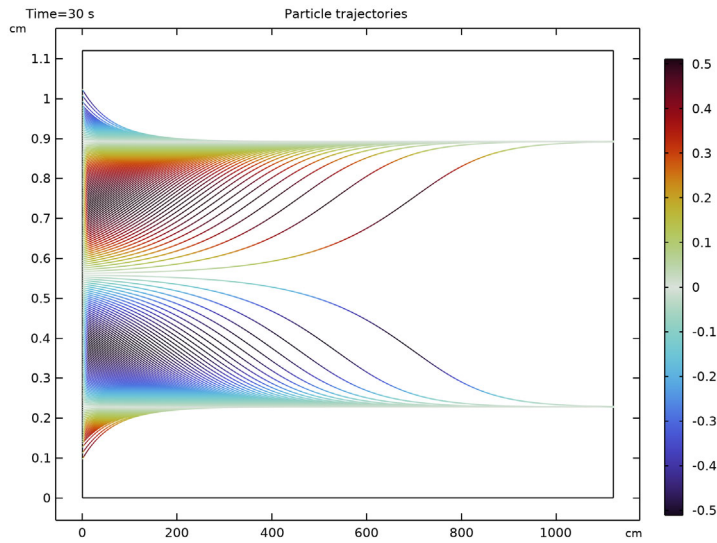
Figure 4: Particle trajectories in the channel. The color expression is the y-component of the particle velocity in mm/s.

Figure 5: Average normalized distance of particles from the center of the channel.

Figure 6: Standard deviation of the normalized distance of particles from the center of the channel.

References

1. G. Segré, and A. Silberberg, "Behaviour of macroscopic rigid spheres in Poiseuille flow. Part 1. Determination of local concentration by statistical analysis of particle passages through crossed light beams," *J. Fluid Mech.*, vol. 14, pp. 115–135, 1962.

2. B.P. Ho and L.G. Leal. "Inertial migration of rigid spheres in two-dimensional unidirectional flows," *J. Fluid Mech.,* vol. 65, no. 2, pp. 365–400, 1974.

Application Library path: `Particle_Tracing_Module/Fluid_Flow/` `inertial_focusing`

Modeling Instructions

From the **File** menu, choose **New**.

N E W

In the **New** window, click ◈ **Model Wizard**.

M O D E L   W I Z A R D

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**.

G L O B A L   D E F I N I T I O N S

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 `inertial_focusing_parameters.txt`.

Variables 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Definitions** and choose **Variables**.

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
dn	abs(qy/d-0.5)		Normalized distance from center
dnave	fpt.ave(dn)		Normalized distance, average
dnstd	sqrt(fpt.ave(dn^2)-fpt.ave(dn)^2)		Normalized distance, standard deviation

These variables will be used during postprocessing to observe how the neutrally buoyant particles are focused by lift forces in the channel.

Axis

1 In the **Model Builder** window, expand the **Component 1 (comp1)>Definitions>View 1** node, then click **Axis**.

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

3 From the **View scale** list, choose **Automatic**. This allows the geometry, which has an extremely high aspect ratio, to be viewed more easily.

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 **cm**.

Rectangle 1 (r1)

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

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

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

4 In the **Height** text field, type d.

5 Click ▦ **Build All Objects**.

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	rhof	kg/m³	Basic
Dynamic viscosity	mu	muf	Pa·s	Basic

Use the Laminar Flow interface to compute the fluid velocity profile in the channel.

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**, click to expand the **Discretization** section.

3 From the **Discretization of fluids** list, choose **P2+P1**. The lift force depends on the spatial derivatives of the fluid velocity components. Therefore, increasing the discretization order is required for accurate calculation of the particle trajectories.

Inlet 1

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

2 Select Boundary 1 only.

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. In the U_{av} text field, type Vav.

Outlet 1

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

2 Select Boundary 4 only.

To ensure that one unique solution exists, the pressure must be fixed at a point in the geometry.

Pressure Point Constraint 1

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

2 Select Point 3 only.

Now that the boundary conditions for the Laminar Flow interface have been set up, use the Particle Tracing for Fluid Flow interface to compute the trajectories of the particles.

ADD PHYSICS

1 In the **Physics** 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 At the bottom of the **Add Physics** section, clear the check box next to **Study 1**. The particle trajectories are not solved for in the **Stationary** study step.

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

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

PARTICLE TRACING FOR FLUID FLOW (FPT)

The particles are assumed to be neutrally buoyant, so they must have the same density as the fluid.

Particle Properties 1

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

4 In the d_p text field, type dp.

Release particles at the inlet boundary.

Inlet 1

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

2 Select Boundary 1 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 200.

6 In the ρ text field, type y>0.1*d&&y<0.9*dThis density-based expression prevents particles from being released too close to the wall. Because the model uses a lift force based on the distance from each particle to the nearest wall, a particle released less than one particle radius from the boundary will produce nonsensical results.

7 Locate the **Initial Velocity** section. From the **u** list, choose **Velocity field (spf)**.

Outlet 1

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

2 Select Boundary 4 only.

Lift Force 1

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

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

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

4 Locate the **Lift Force** section. From the **Lift law** list, choose **Wall induced**.

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

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

Select the two adjacent walls. The **Lift Force** feature uses the distance and direction from the closest point on each wall to compute the lift force.

7 Locate the **Parallel Boundary 1** section. Click to select the ▭ **Activate Selection** toggle button.

8 Select Boundary 2 only.

9 Locate the **Parallel Boundary 2** section. Click to select the ▭ **Activate Selection** toggle button.

10 Select Boundary 3 only.

Next, set up the drag force. Like the lift force, the drag force on the particles is also affected by the presence of nearby walls.

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)**.

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

6 Select the **Include wall corrections** check box.

MESH 1

Because of the high aspect ratio of the geometry, a structured mesh should be used.

Mapped 1

In the **Mesh** toolbar, click ▦ **Mapped**.

Distribution 1

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundaries 1 and 4 only.

Adjust the mesh element distribution so that the mesh is finer close to the channel walls.

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

4 From the **Distribution type** list, choose **Predefined**.

5 In the **Number of elements** text field, type 20.

6 In the **Element ratio** text field, type 4.

7 Select the **Symmetric distribution** check box.

STUDY 1

Step 2: Time Dependent

1 In the **Study** toolbar, click ⟋ **Study Steps** and choose **Time Dependent>
Time Dependent**.

2 In the **Physics interfaces in study** section, clear the check box next to **Laminar Flow (spf)**,
which will not be solved for in this study step.

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

4 Click � **Range**.

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

6 In the **Stop** text field, type 30.

7 Click **Replace**.

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

RESULTS

Velocity (spf)
The default plot of the velocity norm should look like Figure 3.

Plot the particle trajectories as lines to see how the particles get focused as they move
through the channel.

Particle Trajectories 1

1 In the **Model Builder** window, expand the **Particle Trajectories (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**.

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 fpt.vy.

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

5 Locate the **Coloring and Style** section. Click ▮▮▮ **Change Color Table**.

6 In the **Color Table** dialog box, select **Rainbow>DipoleDark** in the tree.

7 Click **OK**.

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

Set up 1D plots of the average and standard deviation of the distance of the particles from the center of the channel.

Mean

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

2 In the **Settings** window for **1D Plot Group**, type Mean in the **Label** text field.

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

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

5 Locate the **Legend** section. Clear the **Show legends** check box.

Global 1

1 Right-click **Mean** and choose **Global**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
dnave	1	Normalized distance from center, average

4 In the **Mean** toolbar, click ▣ **Plot**. The plot should look like Figure 5.

Standard Deviation

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

2 In the **Settings** window for **1D Plot Group**, type Standard Deviation 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**.

5 Locate the **Legend** section. Clear the **Show legends** check box.

Global 1

1 Right-click **Standard Deviation** and choose **Global**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
dnstd	1	Normalized distance from center, standard deviation

4 In the **Standard Deviation** toolbar, click ▣ **Plot**. The plot should look like Figure 6.