Created in COMSOL Multiphysics 6.0

Dispersion of Heavy Particles in a Turbulent Channel Flow

Introduction

In this benchmark model, solid particles are released in a fully developed turbulent channel flow. The total force acting on particles in a fluid comprises a large number of physical phenomena, including but not limited to the drag force, gravity force, buoyancy force, pressure gradient force, added mass effect, lift force, and Brownian force. In this example, the drag force is assumed to be the dominant factor in determining the particle trajectories.

If the flow in the turbulent channel flow is modeled using the Reynolds-averaged Navier-Stokes (RANS) equations, then the fluid velocity is treated as the sum of a deterministic mean flow term and a random velocity perturbation that represents the eddies.

The advantage of the RANS equations over Direct Numerical Simulation (DNS) is that the RANS equations can be solved without requiring a mesh that is sufficiently fine to resolve all of the eddies in the flow, which may be impractically computationally expensive. Furthermore, by treating the chaotic aspect of turbulent flow in a statistical or time-averaged sense, the RANS equations permit stationary solutions to turbulent flow problems, while in reality the flow field is constantly evolving as eddies of all sizes are created, transported, and destroyed.

This example provides an overview of the essential considerations when modeling the motion of particles in a turbulent channel flow. Among the factors considered in this example are:

- The treatment of the drag force as containing a deterministic (advective) and random (diffusive) part,
- The effect of anisotropy of the turbulent eddies on particle motion in wall-bounded flows, and
- the effect of particle inertia on the duration of particle-eddy interactions.

While this example couples the Particle Tracing for Fluid Flow interface to the fluid velocity field and turbulent kinetic energy based on a RANS model, the resulting particle distributions are compared to DNS data compiled by Marchioli et al. (Ref. 1), which show reasonably good agreement. As in Ref. 1, the particles are observed to cluster near the channel wall for an intermediate range of Stokes numbers where they have sufficient inertia to cross some eddies in the flow but not enough to consistently be reflected at the channel wall.

Model Definition

The model geometry is a 2D vertical channel containing air. The model parameters are the same as those used in the DNS simulation in Ref. 1:

- $\nu = 15.7 \times 10^{-6}\ \mathrm{m^2/s}$ is the air kinematic viscosity,
- $\rho = 1.3\ \mathrm{kg/m^3}$ is the air density,
- $h = 0.02\ \mathrm{m}$ is the channel half-width,
- $v_\mathrm{a} = 1.65\ \mathrm{m/s}$ is the average fluid velocity, and
- $\rho_\mathrm{p} = 769\rho$ is the particle density.

These parameters yield a value of $\mathrm{Re} = v_\mathrm{a}h/\nu = 2100$ for the Reynolds number of the flow. The authors predict a shear velocity of $u_\tau = 0.11775\ \mathrm{m/s}$ for a shear Reynolds number of approximately $\mathrm{Re}_\tau = u_\tau h/\nu = 150$.

TURBULENT FLOW

This example solves the Reynolds-averaged Navier-Stokes (RANS) equations and uses the standard k-ε turbulence model (Ref. 2), one of the most frequently used turbulence models in computational fluid dynamics. The k-ε model introduces dependent variables and transport equations for two new quantities:

- The turbulent kinetic energy k (SI unit: $\mathrm{m^2/s^2}$) represents the energy per unit mass associated with eddies in the flow.
- The turbulent dissipation rate ε (SI unit: $\mathrm{m^2/s^3}$) indicates the rate at which turbulent kinetic energy in the eddies is converted to thermal energy.

These new dependent variables provide insight into the size and lifetime of eddies in the flow:

- The ratio k/ε has units of time. The average eddy lifetime is often estimated by multiplying this ratio by a dimensionless constant of order unity.
- The ratio $k^{3/2}/\varepsilon$ has units of length and indicates the length scale of the largest eddies in the flow.

In addition to being a very common turbulence model in its own right, the k-ε model is relatively easy to combine with Lagrangian particle tracking methods because the turbulence variables immediately give estimates of the amplitude of the velocity perturbations due to turbulent eddies (proportional to $\sqrt{k}$) and the average eddy lifetime (proportional to k/ε). For a stationary, incompressible flow, the transport equations solved when using the k-ε turbulence model are

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

$$\rho\nabla\cdot\mathbf{U} = 0$$

$$\rho(\mathbf{u}\cdot\nabla)k = \nabla\cdot\left[\left(\mu+\frac{\mu_T}{\sigma_k}\right)\nabla k\right]+P_k-\rho\varepsilon$$

$$\rho(\mathbf{u}\cdot\nabla)\varepsilon = \nabla\cdot\left[\left(\mu+\frac{\mu_T}{\sigma_\varepsilon}\right)\nabla\varepsilon\right]+C_{\varepsilon 1}\frac{\varepsilon}{k}P_k-C_{\varepsilon 2}\rho\frac{\varepsilon^2}{k}$$

$$\mu_T = \rho C_\mu\frac{k^2}{\varepsilon}$$

$$P_k = \mu_T[\nabla\mathbf{u}:(\nabla\mathbf{u}+(\nabla\mathbf{u})^T)]$$

where the dependent variables are the fluid velocity $\mathbf{u}$ (SI unit: m/s), pressure p (SI unit: Pa), and the aforementioned transport variables k and ε.

The dimensionless constants in these equations have the following default values:

$$C_\mu = 0.09 \qquad C_{\varepsilon 1} = 1.44 \qquad C_{\varepsilon 2} = 1.92 \qquad \sigma_k = 1.0 \qquad \sigma_\varepsilon = 1.3$$

For more details on the different turbulence models that are available, see the Single Phase Flow Interfaces chapter in the *CFD Module User's Guide*. The k-ε model is usually a convenient choice when coupling the turbulent flow to a particle tracing simulation because of its relatively low memory requirements and fast convergence, and because it directly allocates degrees of freedom for k and ε, which are necessary to accurately model turbulent dispersion of the particles.

LAGRANGIAN PARTICLE TRACKING

Marchioli et al. (Ref. 1) provide a wide range of DNS results for simulations with and without the lift and gravity forces. In this example, only the drag force is considered.

Because the particle density is several orders of magnitude greater than the air density, the buoyant force and added-mass effect can safely be neglected. The particles are also large enough that the Brownian Force can be ignored.

The equation of motion for each particle is therefore

$$\frac{d}{dt}\left(m_p\frac{d\mathbf{q}}{dt}\right) = \mathbf{F}_D$$

where

- m_p (SI unit: kg) is the particle mass,

- $\mathbf{q}$ (SI unit: m) is the particle position, and
- $\mathbf{F}_D$ (SI unit: N) is the drag force.

In general, the drag force is defined as

$$\mathbf{F}_D = \frac{1}{\tau_p} m_p(\mathbf{u} - \mathbf{v}) \tag{1}$$

where $\mathbf{u}$ (SI unit: m/s) is the fluid velocity at the particle's position and $\mathbf{v}$ (SI unit: m/s) is the particle velocity. The particle relaxation time τ_p (SI unit: s) is defined as

$$\tau_p = \frac{4}{3} \frac{\rho_p d_p}{\rho C_D |\mathbf{u} - \mathbf{v}|} \tag{2}$$

where

- ρ_p (SI unit: kg/m^3) is the particle density,
- d_p (SI unit: m) is the particle diameter, and
- C_D (dimensionless) is the drag coefficient.

The choice of drag law, which determines the appropriate definition of C_D, depends on the relative Reynolds number Re_r (dimensionless) of the particle in the fluid. For a spherical particle the relative Reynolds number is

$$\mathrm{Re}_r = \frac{\rho |\mathbf{u} - \mathbf{v}| d_p}{\mu}$$

For $\mathrm{Re}_r \ll 1$ the Stokes drag law is applicable,

$$C_D = \frac{24}{\mathrm{Re}_r}$$

so that Equation 2 reduces to

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

PARTICLE TRACING WITH TURBULENT DISPERSION

In the Reynolds-Averaged Navier-Stokes (RANS) approach, the turbulent eddies are only solved for in a statistical sense, by estimating their energy and dissipation rate. A Direct Numerical Simulation (DNS) could resolve the individual eddies but is often too memory-intensive and time-consuming for many practical applications. Therefore, in Equation 1 the fluid velocity $\mathbf{u}$ is not given deterministically. Instead, the fluid velocity is treated as a linear combination of a deterministic part based on the mean flow (which is the field $\mathbf{u}$ solved for by the RANS equations) and a turbulent perturbation term $\Delta\mathbf{u}$,

$$\mathbf{F}_{\mathrm{D}} = \frac{1}{\tau_{\mathrm{p}}} m_{\mathrm{p}}(\mathbf{u}' - \mathbf{v})$$

$$\mathbf{u}' = \mathbf{u} + \Delta\mathbf{u}$$

The amplitude and direction of the velocity perturbation $\Delta\mathbf{u}$ is derived from the turbulence variables k and ε that are solved for by the k-ε turbulence model.

The Particle Tracing for Fluid Flow interface supports two different formulations for the turbulent dispersion term:

- The Discrete Random Walk (DRW) model is similar to the modified Eddy Interaction Model of Gosman and Ioannides (Ref. 3). In the DRW model, unique velocity perturbations are sampled and added to the mean velocity field at discrete times based on the estimated eddy lifetime.
- The Continuous Random Walk (CRW) model, or Continuous Filter White Noise (CFWN) model (Ref. 4). In the CRW model, the velocity perturbations are integrated over time. The eddy velocity and lifetime are built into the time derivatives of the evolving velocity perturbation components.

In this example, the CRW model will be used. A brief overview of this model is given below. For more comprehensive details on both turbulent dispersion models, see the Particle Tracing for Fluid Flow chapter of the *Particle Tracing Module User's Guide*.

The classical Langevin-equation model for homogeneous isotropic stationary turbulence (HIST) is (Ref. 5)

$$du_i = -u_i(t)\frac{dt}{\tau_i} + \sigma_i\sqrt{\frac{2}{\tau_i}}d\xi_i \tag{3}$$

where the subscript $i \in \{1, 2, 3\}$ indicates a component of the fluid velocity field. In isotropic turbulence, the RMS fluid velocity fluctuation in any direction is equal,

$$\sigma = \sigma_1 = \sigma_2 = \sigma_3 = \sqrt{\frac{2k}{3}}$$

Because the turbulence is isotropic, the velocity perturbations can be aligned with any orthonormal coordinate system. In the Particle Tracing for Fluid Flow interface they are simply aligned with the Cartesian coordinates.

Inhomogeneous and Anisotropic Turbulence

In wall-bounded flows the assumption of homogeneous isotropic turbulence no longer applies. The turbulence becomes *inhomogeneous* because the turbulent kinetic energy is heavily damped close to the walls, giving it a nonzero gradient in this region. The turbulence is *anisotropic* because the eddies in the region close to the wall are not equally likely to point in any direction; the instantaneous velocity component normal to the wall is more heavily damped than those in the streamwise and spanwise directions.

In the Particle Tracing for Fluid Flow interface, anisotropic and inhomogeneous turbulent dispersion are modeled by selecting the **Include anisotropic turbulence in boundary layers** check box in the settings for the **Drag Force** node. This check box is only available when the Continuous Random Walk model for turbulent dispersion is used. When this check box is selected, corrections for inhomogeneous and anisotropic turbulence are applied in the region $y^+ < 100$, where y^+ (dimensionless) is the wall distance in viscous units,

$$y^+ = x_2 \frac{u_\tau}{\nu}$$

where x_2 (SI unit: m) is the normal distance to the nearest wall, ν (SI unit: m^2/s) is the kinematic viscosity of the fluid, and u_τ (SI unit: m/s) is the friction velocity,

$$u_\tau = \frac{\tau_\mathrm{w}}{\rho}$$

where τ_w (SI unit: N/m^2) is the wall shear-stress. The friction velocity is usually defined on the **Wall** boundaries by one of the turbulent flow interfaces. In the region of inhomogeneous, anisotropic turbulence, the components of the turbulent velocity perturbation are defined in the coordinate system given by

$$\begin{aligned} u_1 &= \text{streamwise component} \\ u_2 &= \text{wall normal component} \\ u_3 &= \text{spanwise component} \end{aligned}$$

The normalized Langevin equations in these directions are

$$\begin{aligned} d\left(\frac{u_1}{\sigma_1}\right) &= -\left(\frac{u_1}{\sigma_1}\right)\frac{dt}{\tau_1} + \sqrt{\frac{2}{\tau_1}}d\xi_1 + \frac{\partial(\overline{u_1 u_2/\sigma_1})}{\partial x_2}\frac{dt}{1+\mathrm{Stk}} \\ d\left(\frac{u_2}{\sigma_2}\right) &= -\left(\frac{u_2}{\sigma_2}\right)\frac{dt}{\tau_2} + \sqrt{\frac{2}{\tau_2}}d\xi_2 + \frac{\partial\sigma_2}{\partial x_2}\frac{dt}{1+\mathrm{Stk}} \\ d\left(\frac{u_3}{\sigma_3}\right) &= -\left(\frac{u_3}{\sigma_3}\right)\frac{dt}{\tau_3} + \sqrt{\frac{2}{\tau_3}}d\xi_3 \end{aligned}$$

To account for the anisotropy of the flow, the following definitions of the σ_i terms are given. These expressions are taken from Dehbi (Ref. 5), wherein they are attributed to DNS fits of channel flow as described by Dreeben and Pope (Ref. 6).

$$\sigma_1^+ = \frac{\sigma_1}{u_\tau} = \frac{0.40y^+}{1+0.0239(y^+)^{1.496}}$$
$$\sigma_2^+ = \frac{\sigma_2}{u_\tau} = \frac{0.0116(y^+)^2}{1+0.203y^+ + 0.00140(y^+)^{2.421}}$$
$$\sigma_3^+ = \frac{\sigma_3}{u_\tau} = \frac{0.19y^+}{1+0.0361(y^+)^{1.322}}$$

In the boundary layer, the Lagrangian time scale τ_i from Equation 3 is approximately the same in all directions:

$$\tau_\mathrm{L} = \tau_1 = \tau_2 = \tau_3$$

Kallio and Reeks Ref. 7 give the following polynomial fit for the time scale:

$$\tau_\mathrm{L}^+ = \tau_\mathrm{L}\frac{u_\tau^2}{\nu} = \begin{cases} 10 & y^+ \le 5 \\ 7.122 + 0.5731y^+ - 0.00129(y^+)^2 & 5 \le y^+ \le 100 \end{cases}$$

where ν (SI unit: m^2/s) is the fluid kinematic viscosity. Away from the wall, the Lagrangian time scale is simply

$$\tau_\mathrm{L} = \frac{C_\mathrm{L}k}{\varepsilon}$$

where C_L is a dimensionless constant.

EFFECT OF PARTICLE INERTIA

To investigate the effect of particle inertia on anisotropic turbulent dispersion in the channel, a **Parametric Sweep** is performed over the Stokes number St (dimensionless) of the particle:

$$\mathrm{St} = \frac{\tau_\mathrm{p}u_\tau^2}{\nu}$$

where τ_p (SI unit: s) is the particle relaxation time,

$$\tau_\mathrm{p} = \frac{\rho_\mathrm{p}d_\mathrm{p}^2}{18\rho\nu}$$

Marchioli et al. (Ref. 1) use six values of the Stokes number: 0.2, 1, 5, 15, 25, and 125. For a given value of St, the corresponding particle diameter is

$$d_{\mathrm{p}} = \frac{\nu}{u_\tau}\sqrt{18\frac{\rho}{\rho_{\mathrm{p}}}\mathrm{St}}$$

Results and Discussion

The results of the turbulent flow simulation are shown in Figure 1 and Figure 2. The channel has an extremely high aspect ratio, so to better visualize the geometry, **Automatic** has been selected from the **View scale** list in the **Axis** settings. This allows the coordinate axes to be scaled independently of each other so that the plot fits the Graphics window.

Figure 1 shows the fluid velocity magnitude and velocity streamlines. Figure 2 shows the ratio of turbulent kinetic energy to turbulent dissipation rate in the modeling domain. This is greatest near the left boundary, which is the axis of symmetry. It decreases near the wall, suggesting that eddy lifetimes are much shorter there. It is useful to plot this ratio before proceeding with the particle tracing simulation because it shows the minimum resolution in time needed to accurately capture the particle-eddy interactions.

The particle trajectories for the greatest simulated value of the Stokes number, St = 125, are shown in Figure 3. The particles were released uniformly along the cross section halfway down the channel, to prevent them from hitting any **Inlet** or **Outlet** boundaries as a result of the turbulent diffusion. For intermediate values of the Stokes number, the particles cluster near the wall, as shown by the histograms in Figure 4. At very low Stokes number, St = 0.2, the effect of anisotropic turbulence on the number density of particles is less pronounced because such particles don’t have enough inertia to frequently cross the eddies. At very high Stokes number, St = 125, the effect is less pronounced compared to intermediate values because the inertia of such particles is so high that they often reflect off the wall and back into the bulk medium.

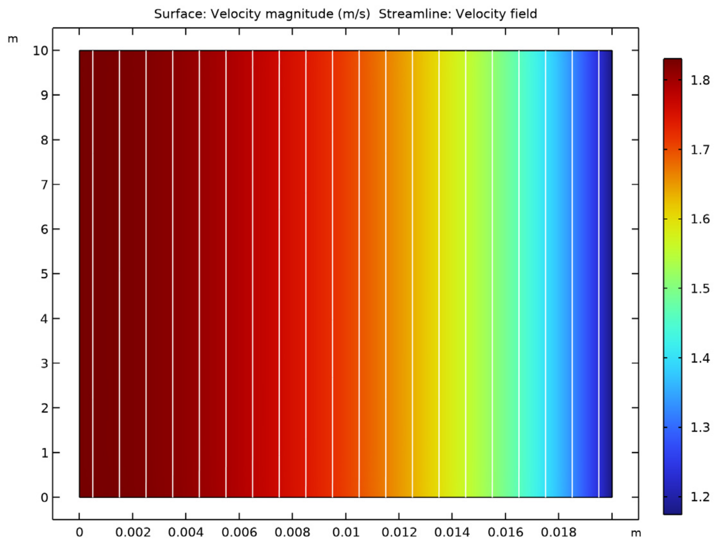

Figure 1: Fluid velocity in the channel. Velocity streamlines are shown as white lines.

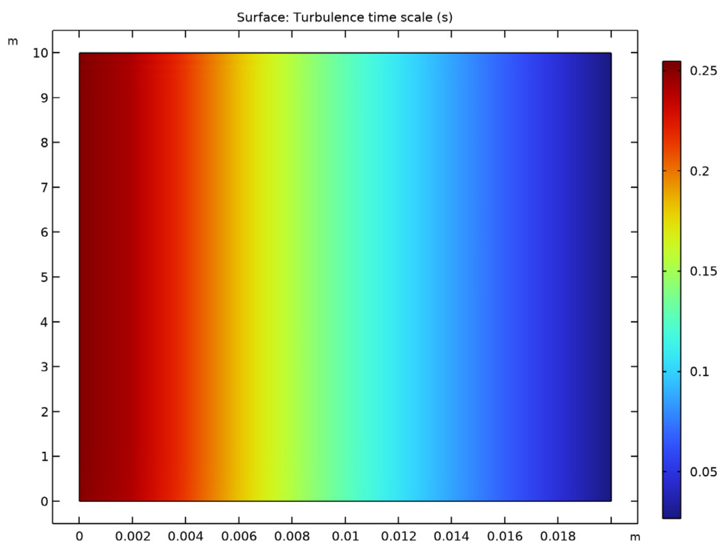

Figure 2: Turbulence time scale, or ratio of turbulent kinetic energy to turbulent dissipation rate, in the channel.

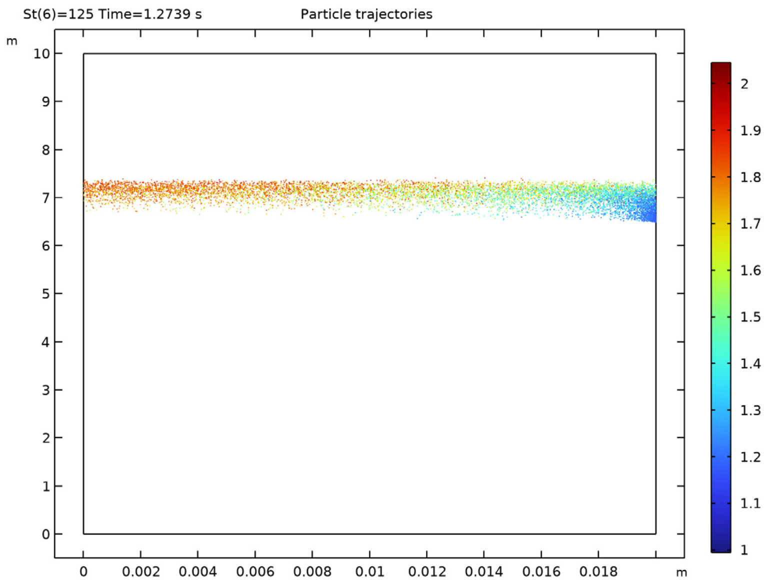

Figure 3: Particle trajectories in the channel.

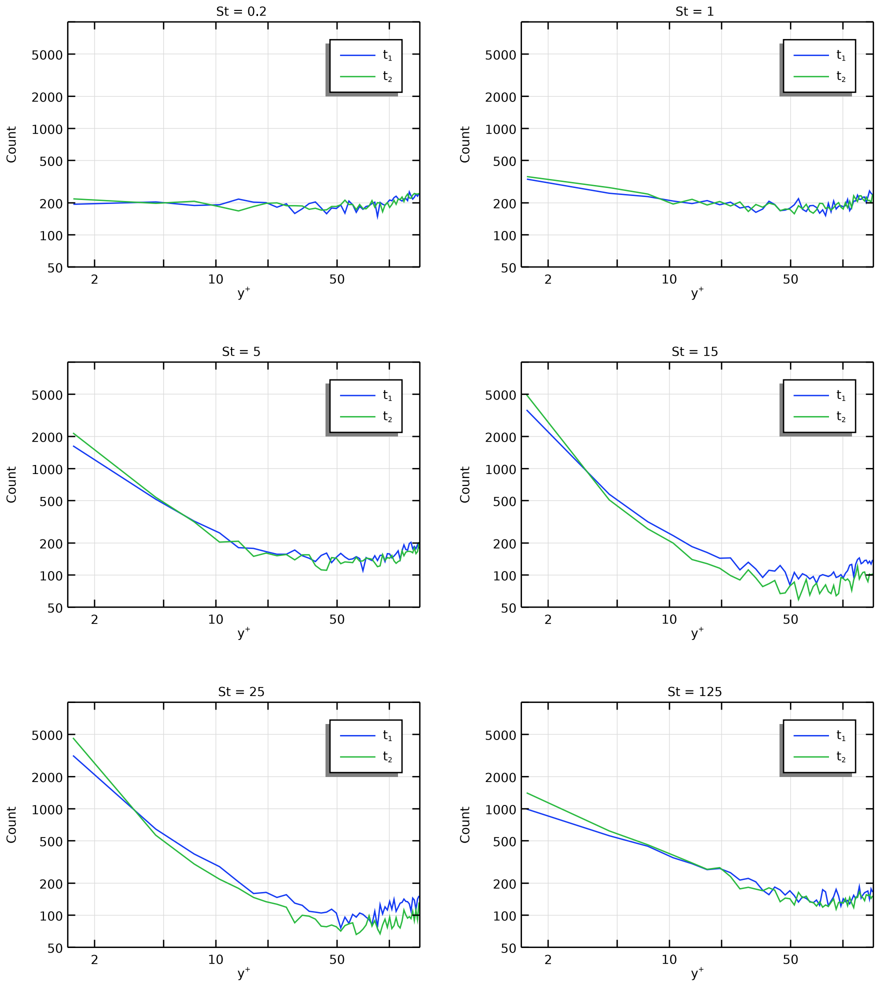

Figure 4: Comparison of histograms of particle position for different values of the Stokes number.

References

1. C. Marchioli, M. Picciotto, and A. Soldati, "Influence of gravity and lift on particle velocity statistics and transfer rates in turbulent vertical channel flow", *International Journal of Multiphase Flow*, vol. 33, no. 3, pp. 227–251, 2007.

2. D.C. Wilcox, *Turbulence Modeling for CFD*, 2nd ed., DCW Industries, 1998.

3. A. D. Gosman and E. Ioannides, "Aspects of computer simulation of liquid-fueled combustors", *Journal of Energy*, vol. 7, no. 6, pp. 482–490, 1983.

4. L. Tian and G. Ahmadi, "Particle deposition in turbulent duct flows–comparisons of different model predictions", *Aerosol Science*, vol. 38, 2007, pp. 377–397.

5. A. Dehbi, "Turbulent particle dispersion in arbitrary wall-bounded geometries: A coupled CFD-Langevin-equation based approach", *International Journal of Multiphase Flow*, vol. 34, no. 9, pp. 819–828, 2008.

6. T. D. Dreeben and S. B. Pope, "Probability density function and Reynolds-stress modeling of near-wall turbulent flows", *Physics of Fluids*, vol. 9, no. 1, pp. 154–163, 1997.

7. G. A. Kallio and M. W. Reeks, "A numerical simulation of particle deposition in turbulent boundary layers", *International Journal of Multiphase Flow*, vol. 15, no. 3, pp. 433–446, 1989.

Application Library path: `CFD_Module/Particle_Tracing/flow_channel_turbulent_dispersion`

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>Turbulent Flow>Turbulent Flow, k-ε (spf)**.

3 Click **Add**.

4 Click **Study**.

5 In the **Select Study** tree, select **General Studies>Stationary**.

6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

Load the model parameters from a file.

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 `flow_channel_turbulent_dispersion_parameters.txt`.

GEOMETRY 1

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 `halfWidth`.

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

5 Click **Build All Objects**.

The geometry has a very high aspect ratio. Adjust the View settings to make it easier to see.

DEFINITIONS

View 1

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

Axis

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

4 Click **Update**.

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

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

TURBULENT FLOW, K-ε (SPF)

Inlet 1

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

2 In the **Settings** window for **Inlet**, locate the **Boundary Condition** section.

3 From the list, choose **Fully developed flow**.

4 Select Boundary 2 only.

5 Locate the **Fully Developed Flow** section. In the U_{av} text field, type `va`.

Outlet 1

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

2 Select Boundary 3 only.

Symmetry 1

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

2 Select Boundary 1 only.

MESH 1

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.

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

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

Distribution 2

1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.

2 Select Boundaries 2 and 3 only.

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 `nelemWidth`.

6 In the **Element ratio** text field, type `15`.

7 Click **Build All**.

STUDY 1

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

RESULTS

Velocity (spf)

Add some streamlines to the default plot of the fluid velocity.

Streamline 1

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

2 Select Boundary 2 only.

3 In the **Settings** window for **Streamline**, locate the **Coloring and Style** section.

4 Find the **Point style** subsection. From the **Color** list, choose **White**.

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

Compare the resulting plot to Figure 1.

Turbulence Time Scale

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

2 In the **Settings** window for **2D Plot Group**, type `Turbulence Time Scale` in the **Label** text field.

Surface 1

1 Right-click **Turbulence Time Scale** and choose **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Turbulent Flow, k-ε>Turbulence variables>spf.tauT - Turbulence time scale - s**.

3 In the **Turbulence Time Scale** toolbar, click **Plot**.

Compare the resulting plot to Figure 2.

Next, solve for the particle trajectories in the turbulent flow.

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 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 **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 At the bottom of the **Add Study** section, clear the check box next to the Turbulent Flow, k-ε interface, which will not be solved for in the **Time Dependent** study step.

4 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies> Time Dependent**.

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)

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 `rhop`.

4 In the d_p text field, type `dp`.

Assign the **Outlet** condition to both the top and bottom boundaries. Assign the **Symmetry** condition at the symmetry axis. The distance from the remaining **Wall** boundary will be used to compute the anisotropic turbulent velocity perturbations.

Outlet 1

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

2 Select Boundaries 2 and 3 only.

Symmetry 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Symmetry**.
2 Select Boundary 1 only.

Wall 1

1 In the **Model Builder** window, click **Wall 1**.
2 In the **Settings** window for **Wall**, locate the **Wall Condition** section.
3 From the **Wall condition** list, choose **Bounce**.

Add the drag force, using the fluid velocity and turbulence variables computed in the previous study.

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 $\mathbf{u}$ list, choose **Velocity field (spf)**.
5 Locate the **Turbulent Dispersion** section. From the **Turbulent dispersion model** list, choose **Continuous random walk**.
6 From the k list, choose **Turbulent kinetic energy (spf)**.
7 From the ε list, choose **Turbulent dissipation rate (spf)**.
8 Select the **Include anisotropic turbulence in boundary layers** check box.
9 In the u^* text field, type `ustar_exp`.
10 Locate the **Wall Corrections** section. From the **Mesh search method** list, choose **Use tolerance**.
11 In the r text field, type `0.03`.

Release particles from the middle of the channel. Initially, the number density of particles is uniform over the width of the channel.

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 `halfWidth/(2*Np)`.
6 In the **Stop** text field, type `halfWidth*(1-1/(2*Np))`.

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

8 Click **Replace**.

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

10 In the $q_{y,0}$ text field, type `height/2`.

11 Locate the **Initial Velocity** section. Specify the $\mathbf{v}_0$ vector as

u	x
v	y

STUDY 2

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
St (Particle Stokes number)	0.2 1 5 15 25 125	

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 In the **Output times** text field, type `0 t1 t2`.

4 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**.

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

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

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

RESULTS

Particle Trajectories (fpt)

Compare the default trajectory plot to Figure 3.

Number Density, St = 0.2

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.
2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.
3 From the **Dataset** list, choose **Particle 1**.
4 In the **Label** text field, type `Number Density, St = 0.2`.
5 Locate the **Data** section. From the **Parameter selection (St)** list, choose **From list**.
6 In the **Parameter values (St)** list, select **0.2**.
7 From the **Time selection** list, choose **From list**.
8 In the **Times (s)** list, choose **0.76433** and **1.2739**.
9 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.
10 In the **Title** text area, type `St = 0.2`.
11 Locate the **Plot Settings** section. Select the **x-axis label** check box.
12 In the associated text field, type `y<sup>+</sup>`.

Histogram 1

1 Right-click **Number Density, St = 0.2** and choose **Histogram**.
2 In the **Settings** window for **Histogram**, locate the **Expression** section.
3 In the **Expression** text field, type `fpt.df1.yplus`.
4 Locate the **Bins** section. In the **Number** text field, type `50`.
5 Click to expand the **Legends** section. Select the **Show legends** check box.
6 From the **Legends** list, choose **Manual**.
7 In the table, enter the following settings:

Legends
t₁
t₂

8 In the **Number Density, St = 0.2** toolbar, click **Plot**.

Number Density, St = 0.2

The histogram shows that the number density throughout the cross section is nearly uniform. Use a logarithmic scale and manual axis limits to more easily compare such histograms for each value of the Stokes number.

1 Click the **x-Axis Log Scale** button in the **Graphics** toolbar.
2 Click the **y-Axis Log Scale** button in the **Graphics** toolbar.

3 In the **Model Builder** window, click **Number Density, St = 0.2**.

4 In the **Settings** window for **1D Plot Group**, locate the **Axis** section.

5 Select the **Manual axis limits** check box.

6 In the **x minimum** text field, type `1.4`.

7 In the **x maximum** text field, type `150`.

8 In the **y minimum** text field, type `50`.

9 In the **y maximum** text field, type `1e4`.

10 In the **Number Density, St = 0.2** toolbar, click **Plot**.

Duplicate the 1D Plot Group containing the **Histogram** plot and select other values of the Stokes number to observe how particle inertia affects the particle number density in the channel cross section. As the Stokes number increases, the particles begin to accumulate in the boundary layer close to the wall. All six sets of histograms are shown in Figure 4.