15th International Research/Expert Conference
"Trends in the Development of Machinery and Associated Technology"
TMT 2011, Prague, Czech Republic, 12-18 September 2011

A LAGRANGIAN APPROACH FOR MODELING DISCHARGE OF PARTICLES IN TWO-PHASE FLOWS

Orhan Gokcol
Bahçeşehir University
Department of Mechatronics Engineering,
Besiktas 34349, Istanbul, Turkey

Mutlu Tunc
Mutlu Tunc Consultancy
Istanbul, Turkey

ABSTRACT
In this study, the interaction between two-phase turbulent jet flow and a bluff body is investigated by using a purely Lagrangian computational model. The flow field is modeled by using discrete vortex elements. The solid particles (powders) which are injected into the jet flow are acted as the second phase. The whole system is considered as a model for the discharge of powders in fluidized beds. Viscous effects are taken into consideration by using Random Walk approach. The instantaneous particle distributions are investigated for ranges of different particle diameters. It is shown that the flow field characteristics are a strong function of particle diameter.
Keywords: Two-Phase Jet Flow, Vortex Dynamics, Fluidized Beds, Two-Way Coupling,

1. INTRODUCTION
Flow and discharge of fine particles or powders as fluid-solid mixtures in fluidized bed systems find many applications such as nuclear reactors, fossil fuel combustors, food powder processing for granulation, milling, air-lift pumps, and metal coating [1]. Study of such systems requires modeling of solid and fluid phases with a coupled manner. In this investigation, a well-known Vortex Element Method (VEM) so called Discrete Vortex Method is used to simulate the two-phase turbulent jet flow interacting with a downstream bluff-body. VEMs have been used successfully for a wide variety of flow problems [2,3,4]. The outstanding success of VEMs in simulating free shear layer flows such as jets, wakes and mixing layers enables us to use them in more sophisticated flow simulations, such as two-phase flows. The solid particles act as a second phase. A free and confined jet flow interacting with a downstream body is studied.

2. MODELING OF THE FLUID PHASE
The fluid-structure interactions are very common in many engineering applications. Such kinds of interactions are the sources of acoustic noise and also structural random vibrations. There are important applications of two-phase shear flows interacting with downstream structures such as the cases in combustion chambers and the fluidized bed applications. The interaction between the jet and the bluff body involves the unsteady interaction of vortices caused by free jet and vortices generated from the sharp edges of the bluff body as shown in Fig. 1. The interaction is essentially a function of the downstream body position and also the free jet Reynolds number, $u_\infty D/\upsilon$. Figure 2 shows the research geometry for this study. The jet flow is generated from a nozzle exit by using source elements inside the nozzle. The source distribution is chosen to be parabolic so as to provide a correct initial velocity profile at the nozzle exit. Solid walls of the nozzle, chamber and bluff body are modeled by

using surface bounded vortex singularities.

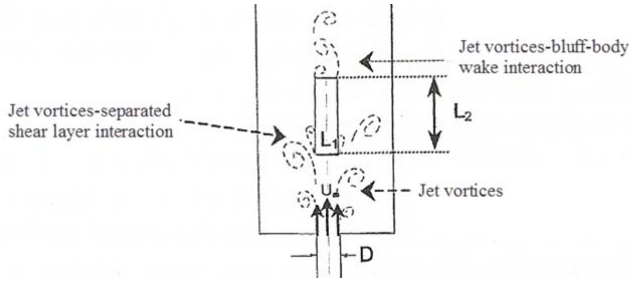

Figure 1. Flow features of the jet-bluff body interaction

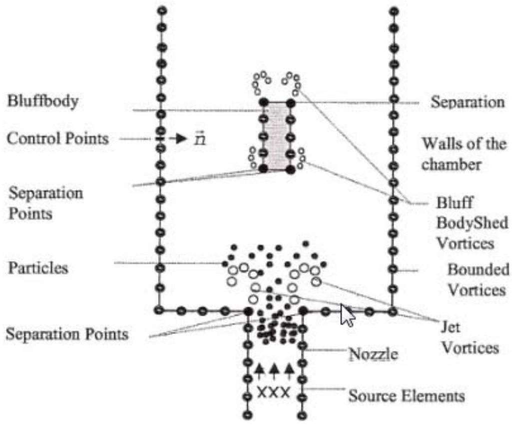

Figure 2. Research geometry

Control points which are chosen in the middle of the adjacent vortices are used to apply the necessary flow boundary conditions. The source strengths can be adjusted easily to obtain the desired Reynolds numbers. At high Reynolds numbers, jet flows produce side vortices which are periodically shedding towards downstream. In this study, the formation of these jet vortices are modeled by using an inviscid, 2-D discrete vortex model [5].

3. MODELING OF THE SOLID PHASE

Solid particles which represent the solid phase move under the effect of various number of force components in the flow field. These forces are mainly due to; drag force, virtual mass effect, Basset history terms, Saffman lift forces and gravity [1,5,6,7]. As proposed by [5], only drag force and gravity are taken into consideration and other terms are considered to be small. Therefore, the acceleration of the particle can be written as

$$\frac{d\vec{U}_P}{dt} = F_1(\vec{U}_g - \vec{U}_p) + (1 - \frac{\rho_g}{\rho_p})\vec{g} \qquad (1)$$

where $\vec{U}_g$ is the fluid velocity, $\vec{U}_p$ is the particle velocity, ρ_g is the fluid density, ρ_p is the particle density and F_1 is the drag coefficient which is given in Tunc (2000). In the calculation procedure, vortices and particles are shed in prescribed time intervals. Drag force and particle Reynolds number is calculated at each time step. As an initial condition, particles' velocities are set equal to fluid velocity at the same location. Particles are advanced to their new positions by solving $dx_p/dt=u_p$ and $dy_p/dt=v_p$.

In modeling the particle-fluid interaction, a two-way coupling model [5] is used where total force imposed on the fluid by particles as a flow body force is taken into consideration by apportioning the forces to the grids in Eulerian type calculations. The body force can be obtained from the particle equations of motion. The model follows a similar approach of the vortex-in-cell method [8]. The flow field is divided into cells. At each instant of time, a disturbance velocity due to fluid-particle interaction is computed inside the cells. This flow component is mapped to the grids of the cells by area weighting. At each grid point the disturbance vorticity is computed. Then by using inverse interpolation, this vorticity value is converted into circulation and added to discrete vortex strengths inside each cells. Finally the vortices are advanced at each time step.

4. RESULTS AND DISCUSSION

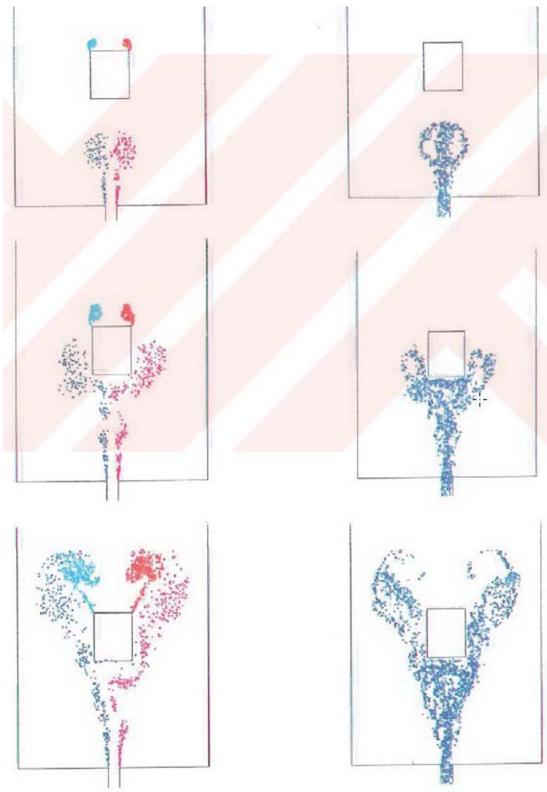

Figure 3. Interaction between two phase flow field and solid boundary (left: vorticity distribution, right: particle distribution) at non-dimensional time, T=50

Figure 3 shows the vorticity distribution of the flow phase and particle distribution of the solid phase. The interaction between the solid boundaries and fluid/solid mixture is clearly seen. Separation points are chosen as the locations where flow has a tendency to form the well-known Karman Vortex street.

The effect of particle diameter to axial and horizontal turbulent fluctuation velocities are shown in Figure 4. The maximum values are obtained in downstream regions. As particle diameters increase axial fluctuations also increase. It can be concluded that the selection of particle diameter plays an essential role for a successful mixing between the phases. The efficiency of the VEM to model fluid-solid mixture is also clear.

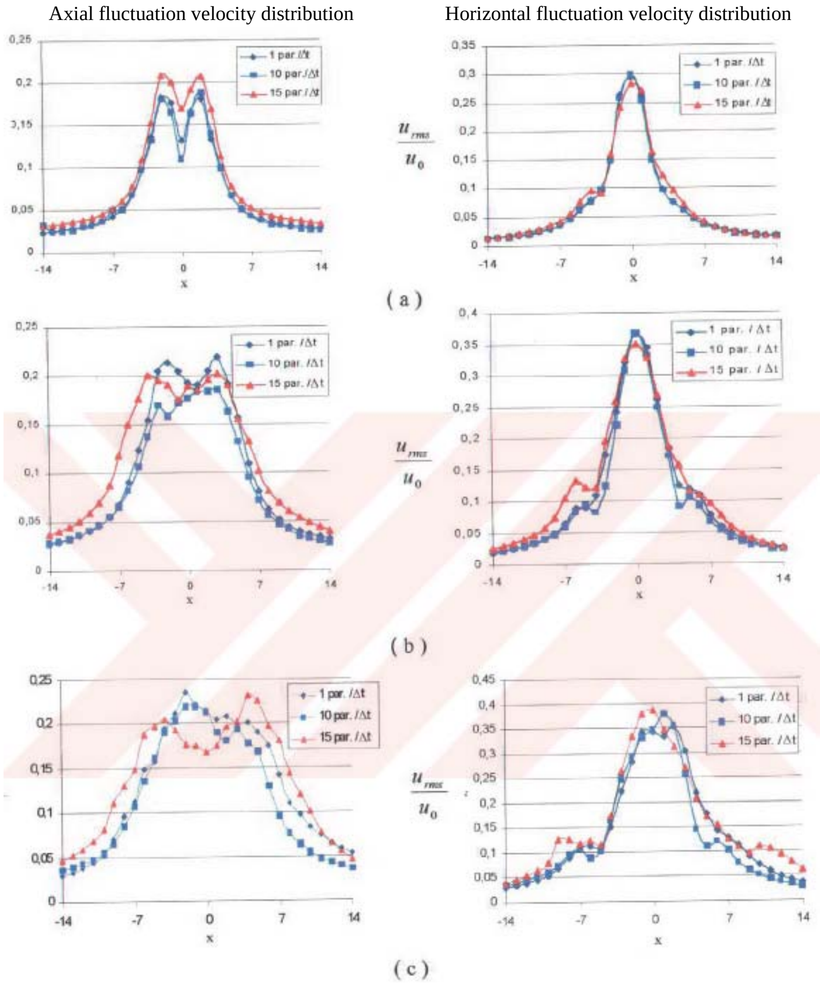

Figure 4. The variation of fluctiation velocities (u' and v') for different number of particles (1, 10 and 15)for x/d=5 (a), x/d=10 (b), x/d=15 (c).

5. REFERENCES

[1] Ishii, M. And Hibiki, T. (2006). Thermo-Fluid Dynamics of Two-Phase Flow, Springer Science.

[2] Lewis, R.I. (2005). Vortex Element Methods for Fluid Dynamic Analysis of Engineering Systems, (Cambridge Engine Technology Series).

[3] Sarpkaya, T. (1994). Vortex Element Methods for Flow Simulation. Adv. Appl. Mech., 31, pp. 113-247.

[4] Kamemoto, K. (2004). On Contribution of Advanced Vortex Element Methods Toward Virtual Reality of Unsteady Vortical Flows in the New Generation of CFD. J. of the Braz. Soc. of Mech. Sci. & Eng., XXVI, 4, pp. 368-378.

[5] Tunc, M. (2000) Modeling Two-Phase Flows by Using Discrete Vortices (PhD Thesis), Istanbul University, Institute of Science, Istanbul, Turkey.

[6] Crowe, C.T., Trout, T.R. and Chung, J.N. (1996). Numerical Model for Two-Phase Turbulent Flows, Annu. Rev. Fluid Mech., 2, pp. 11-43.

[7] Drew, D. (1988). Mathematical Modeling of Two-Phase Flow, Annu. Rev. Fluid Mech., 15, 261.

[8] Christiansen, J.P. (1973). Numerical Simulation of Hydrodynamics by the method of Point Vortices, J. of Computational Physics, 13, pp 253-265.